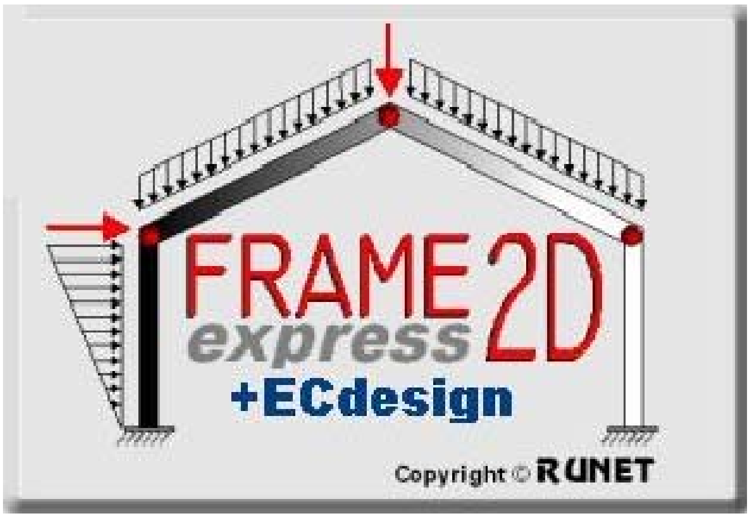


USER'S Manual

Table of contents

RUNET NORWAY as, Tennfjord 6264-N, Norway
e-mail: support@runet-software.com
Internet: http://www.runet-software.com

1. General

Frame2Dexpress is a finite element program, for the static and dynamic analysis of two (2) dimensional frame structures.

The program has been designed to be simple and fast in use.

- The structure is defined by giving the nodal coordinates and the element connectivity or from a Frame Wizard.
- The graph of the structure appears simultaneously with the defined data.
- The support conditions are defined for the supported nodes easy from graphical menu.
- The cross sections are defined either with the area and inertia properties, or by their dimensions (rectangular, T or L type sections).
- Nodal loads are dead loads and live loads.
- Element distributed loads (uniform, triangular, parabolic), dead and live loads can be specified. The self weight may or may not be included. Element distributed load direction, vertical, horizontal or perpendicular to the elements.
- The load combination coefficients can be defined by the user.
- Material properties can be picked up from menu.
- Nodal masses corresponding to dead or live loading.
- Mass combination coefficients for nodal masses or element distributed masses due to dead and live weight can be defined by the user.

<u>Results of static analysis</u>:
- Nodal displacements
- Displacements along each elements.
- Internal forces at element ends in global and local coordinate system.
- Internal forces (bending moments, shear and axial forces) along each element.
- Diagrams of displacements.
- Diagrams of bending moments, shear forces and axial forces.

<u>Result of dynamic analysis</u>:
- Natural frequencies
- Diagrams of mode shapes

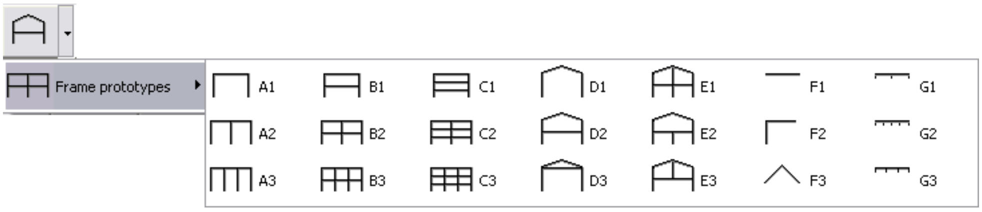
<u>Additional printouts</u>
- Stiffness and mass matrices.
- Equilibrium equations
- Eigenvalue equations

<u>Units</u>
Metric units.

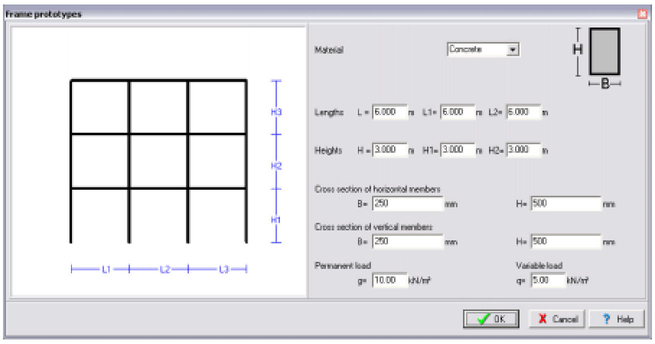
Input data steps
- Nodal coordinates
- Element cross sections
- Element data
- Supports
- Nodal loads and nodal masses
- Distributed element loads and distributed masses.

2. Structural Wizard

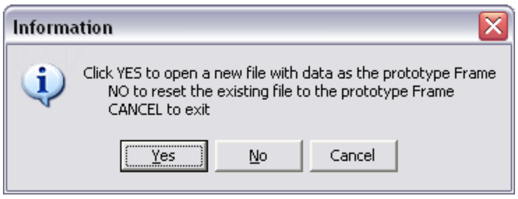
If you want to create a new structure with help from the structural wizard, go to [File/Frame Prototypes] and choose from a menu with 21 ready types of structures.



If you want to open and change the prototypes, you can enter the main menu/Frame prototypes and choose a structure which are quite similar to the one you want to make.



You can edit the different properties of structural material and cross sections.

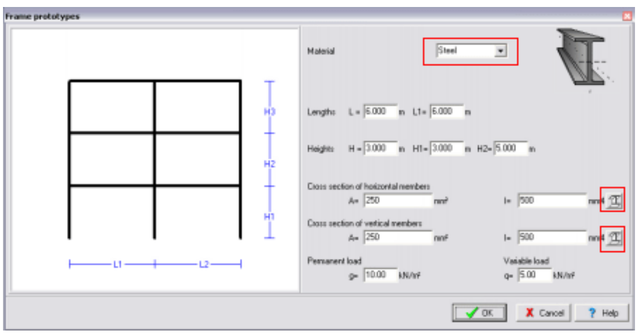


Click [No] if you want to edit the prototype.
Click [Yes] if you want to make a new file.

Structural Data for prototypes

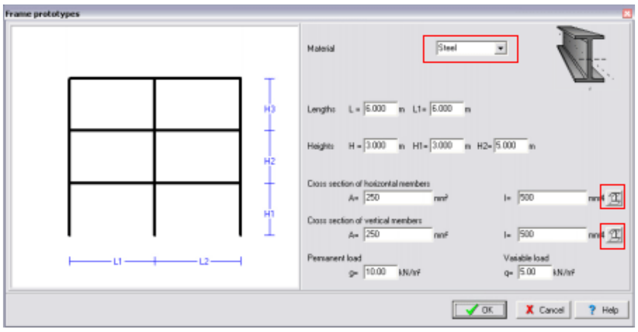
After choosing a prototype, you enter the screen from where you set the basic dimensions of the structure according to the drawing on the left.
If you choose concrete or timber for structural material, edit the cross section width and height in mm.
If you choose general material, you enter cross section area and moment of inertia in mm^2 and mm^4

If you choose steel for structural material, then by clicking on the button  you can choose the steel profile from a complete list, and the cross section area and moment of inertia are set.

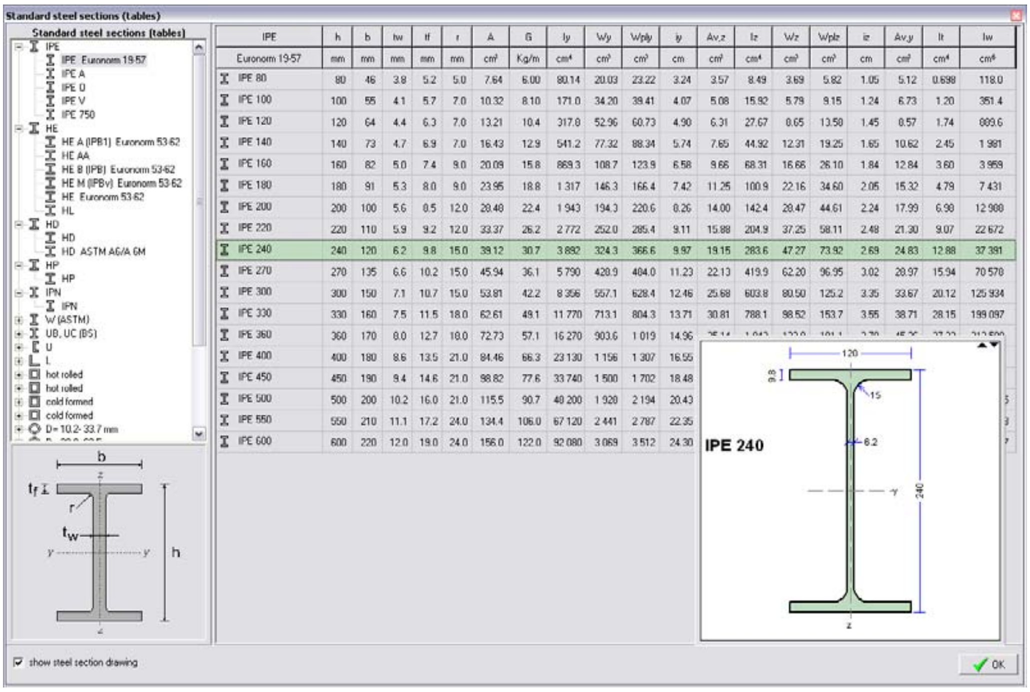


3. Steel profiles

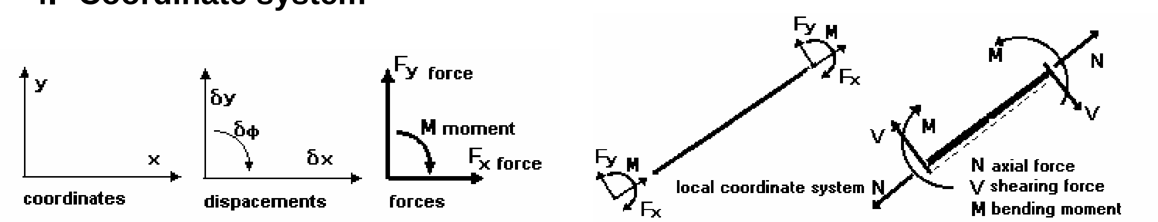
If you choose steel for structural material, then by clicking on the button you can choose the steel profile from a complete list, and the cross section area and moment of inertia are set.



To see the different Steel profiles available in the program, go to [Frame prototypes/Steel profiles] and choose from the list to see the properties of each steel profile.



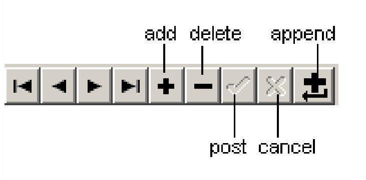
4. Coordinate system



Global coordinate system Local coordinate system (member end forces)

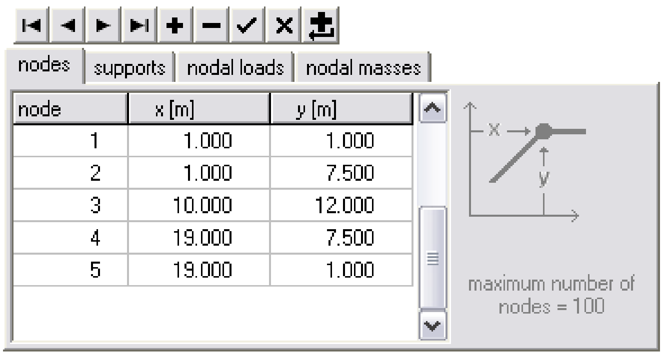
5. Units

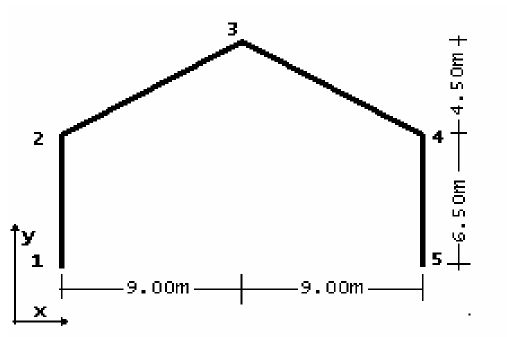
The units used in the program are:
Length, coordinates, in meters (m).
Loads, forces in kN, moments in kN.m, distributed loads in kN/m.
Masses, (lumped or distributed), instead of masses you give the corresponding weights (concentrated or distributed), in kN or kN/m, and the masses are computed dividing with the acceleration of gravity g=9.81 m/sec².
Modulus of elasticity, in GPa (kN/mm²).
Specific weight, in kN/m^3.



6. Nodal points

Give the nodal coordinates in meters [m] in the global coordinate system.
The nodal point numbers must be unique. Two nodes cannot have the same number.
You must not have node numbers missing. This means if you have a structure with 24 nodes you have to input the coordinates of all the nodes from 1 to 24.

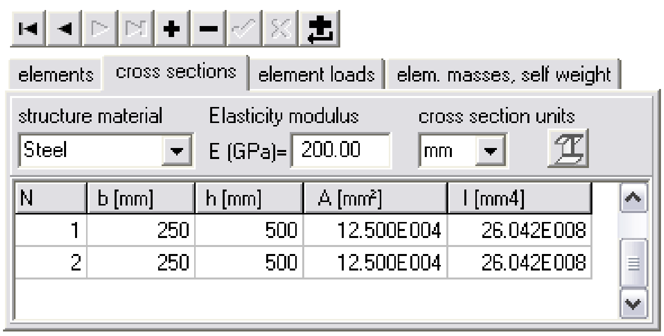




7. Element cross sections

Data for the element cross sections. You group the cross sections and you number the groups, e.g. column sections=1, first floor beam sections=2, second floor beam sections=3. For every cross section group you give the necessary data.

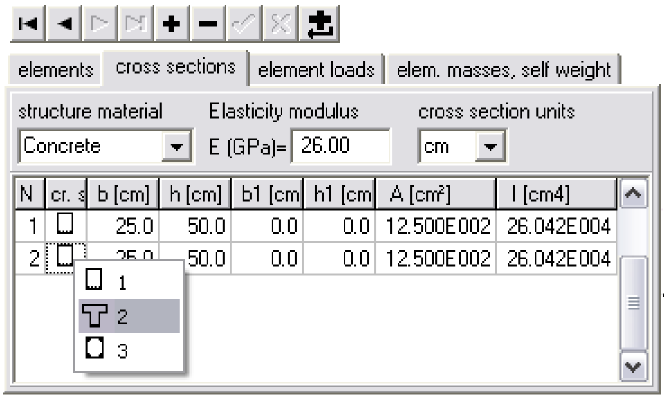
- Select structure material (general, concrete, steel, timber). You give the modulus of elasticity in GPa (kN/mm²), (in case of choosing material concrete, steel or timber an average modulus of elasticity is automatically shown)
- Select the units for the cross section dimensions (mm, cm, m).

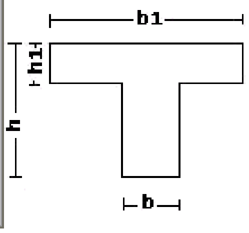


If the cross section is rectangular give the width (**b**) and the height (**h**) of the cross section. The cross section area and moment of inertia are automatically computed.

If you select concrete as material, by clicking at the second column (sect.), you can select orthogonal or T type cross section, and you give the corresponding dimensions (**b**), (**h**), (**b1**), (**h1**) as in the drawing .
For non rectangular cross section give the area and the moment of inertia. In the area and moment of inertia data the symbol E is the exponential factor and means power of ten, e.g. E002 means 10^2, and E-002 means $1/10^2$. This symbol can be used in the data you give.

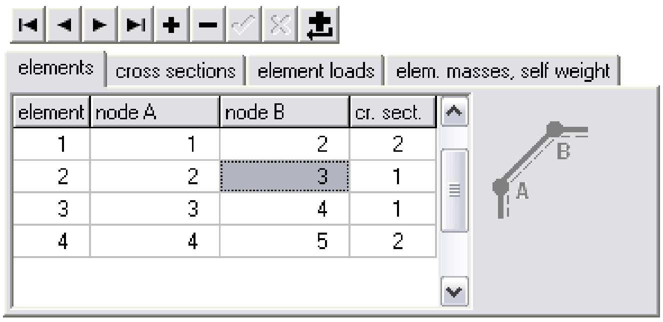
In case of selecting concrete as material, you can select from the second column (sect.), orthogonal or T type cross section, and you give the corresponding dimensions.

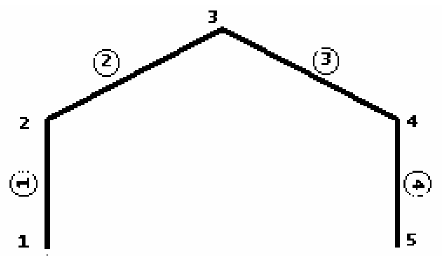


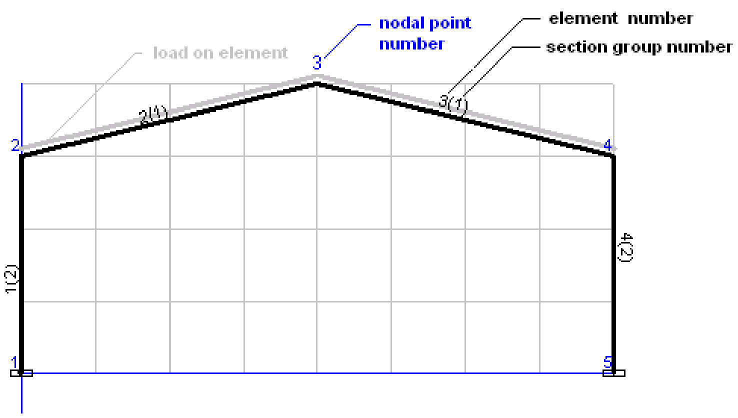


8. Elements

Give the connectivity and topology of the structure. The element numbers must be unique. Two elements cannot have the same number. For every element, give the end nodal points and the cross section number that has been defined in the element cross section data. The two end nodes A and B define the element orientation. The element is oriented with the node A at left end to the node B at the right end, and this way the bottom of the beam element is defined. The positive bending moment is defined when the beam bottom is in tension. See internal forces chapter 15.

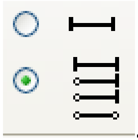


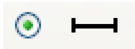


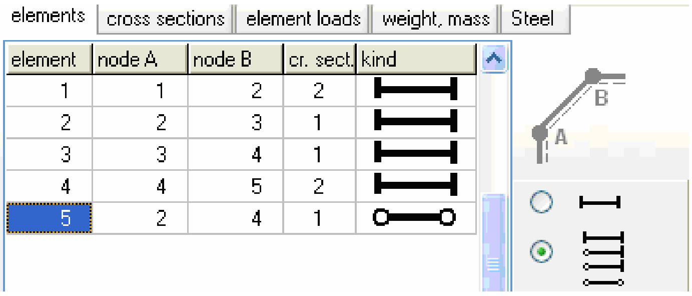


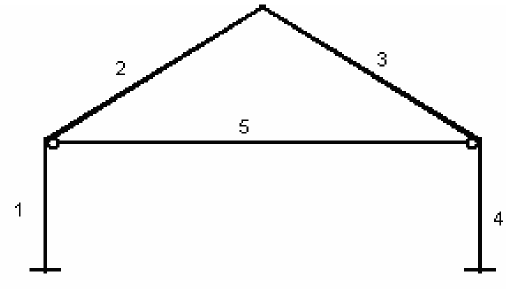
9. Elements with hinges

In some cases you may want to use elements with hinge connection on the left or the right side.

First check . Then a new column (kind) appears in the element section.
Click on this for some element and select the element you want.

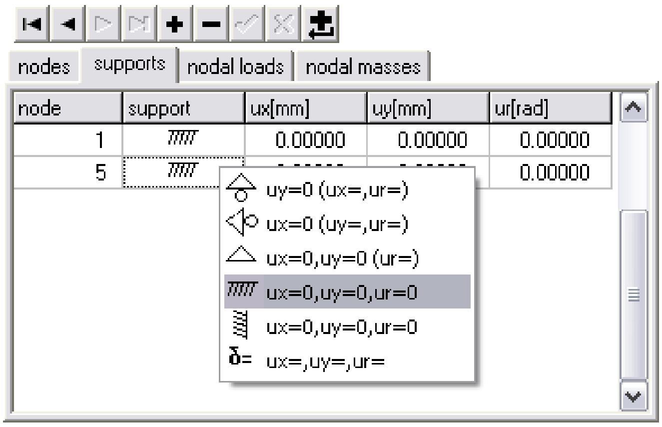
If (no matter how the element kind is) you click  then the solution will be for regular stiff connected on both ends elements.

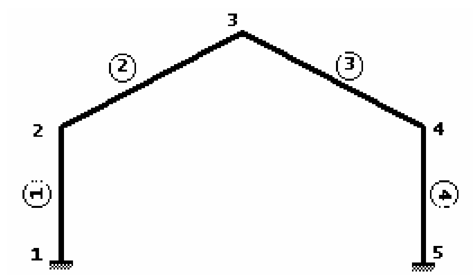




10. Supports

With click at the second (<u>support</u>) column the support choices are shown.
For applied displacements, choose the appropriate support condition and give the applied displacements in [mm] and rotation in radians.
Example: If at a nodal point is applied only horizontal displacement 3 mm, and no vertical, you choose the first menu choice for rolled support, and give ux=3. (if you do not give any number for the rotation but 0 the rotation will be considered free).
With the last menu choice u=, you can specify all the nodal displacements ux, uy and rotation ur.

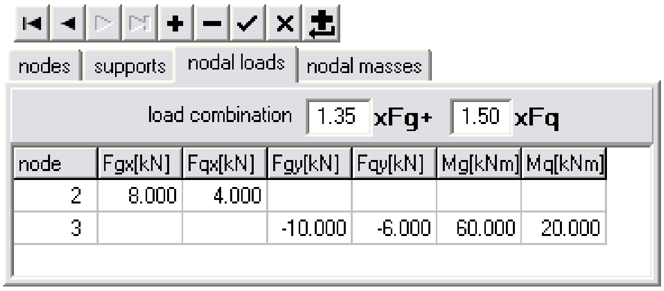




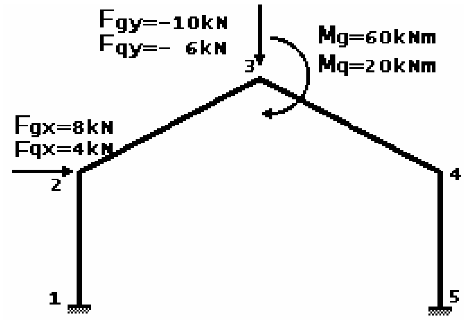
Support conditions for the structure.

11. Nodal loads

Nodal forces in the x, y direction in kN, and nodal moments in kNm. The positive direction of forces and moments is according to the positive direction of the global coordinate system.
The loading in the static analysis is taken as **C_gxF_g+C_qxF_q**, where **C_g** and **C_q** are the load combination coefficients. Default values are C_g=1.35 and C_q=1.50 (Eurocode 1, EN 1991 1-1:2003).



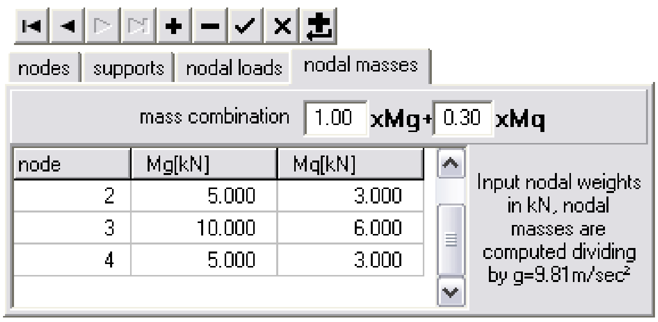
node	Fgx[kN]	Fqx[kN]	Fgy[kN]	Fqy[kN]	Mg[kNm]	Mq[kNm]
2	8.000	4.000				
3			-10.000	-6.000	60.000	20.000



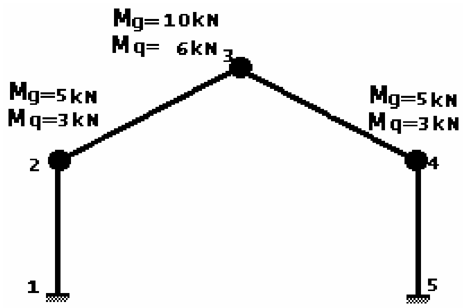
Fgx,Fqx, nodal forces, in kN, in x-x direction, for permanent and live loads, positive from left to right.
Fgy,Fqy, nodal forces, in kN, in y-y direction, for permanent and live loads, positive from down to up.
Mg, Mq, nodal moments, in kNm, for permanent and live loads, positive clockwise.

12. Nodal masses

Lumped nodal masses due to dead (Mg) and live (Mq) load.
To avoid confusion over the mass units, instead of masses you must give the corresponding weights in [kN], and the masses are computed dividing the weights by 9.81 m/sec^2.
The masses in the dynamic analysis are taken as **C_gxM_g+C_qxM_q**. Default values are C_g=1.00 and C_q=0.30 (Eurocode 1, EN 1991 1-1:2003).

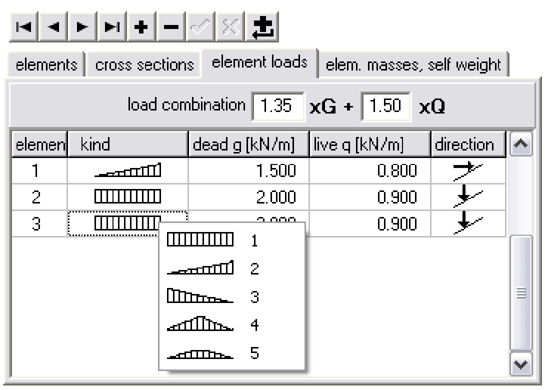


node	Mg[kN]	Mq[kN]
2	5.000	3.000
3	10.000	6.000
4	5.000	3.000

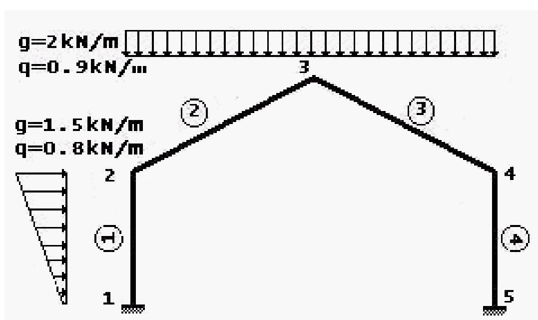


13. Element distributed loads

Element distributed loads, dead (g) and live (q) in [kN/m]. You can select among five different types of distributed loads, from the menu that appears by clicking at the (kind) (second column). The kind of the distributed loads are: uniformly distributed, triangular with maximum value at the right or the left element end, symmetric triangular, and symmetric parabolic.
The loading direction, perpendicular to element axis, vertical or horizontal, is selected from the menu appearing by clicking the last column (direction).
The loading in the static analysis is taken as **C_gxF_g+C_qxF_q**, where **C_g** and **C_q** are the load combination coefficients. Default values are C_g=1.35 and C_q=1.50 (Eurocode 1, EN 1991 1-1:2003).

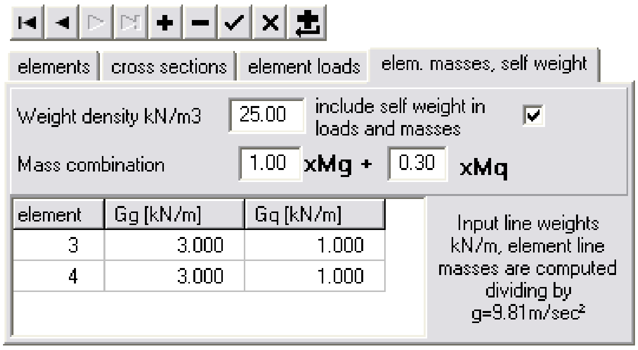


elemen	kind	dead g [kN/m]	live q [kN/m]	direction
1		1.500	0.800	
2		2.000	0.900	
3		2.000	0.900	

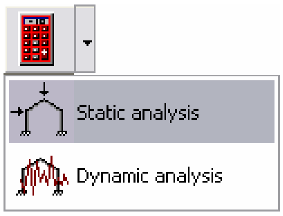


14. Element distributed masses

Distributed masses over the beam elements due to dead (Mg) and live (Mq) load.
To avoid confusion over the mass units, instead of masses you must give the element distributed weights in [kN/m], and the masses are computed dividing the weights by 9.81 m/sec^2.
The mass in the dynamic analysis are taken as $\mathbf{C_g x M_g + C_q x M_q}$. Default values are C_g=1.00 and C_q=0.30 (Eurocode 1, EN 1991 1-1:2003)
If the **self weight** is checked, the self weight of the elements is taken into account in the vertical loads of the static analysis, and the masses in the dynamic analysis. The self weight is computed by multiplying the cross section area of each element with the **specific weight** of the structure material.

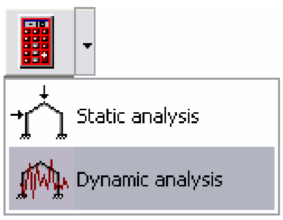


15. Static analysis



The computational steps appear on the screen. First the 6x6 element stiffness matrices are evaluated, using linear elastic theory, and each one is assembled in the global stiffness matrix of the structure, that has dimensions 3Nx3B, where N is the number of nodes and B is the nodal bandwidth.
The element distributed loads are converted to equivalent nodal loads, that are added to the existing nodal loads, and the load matrix in the equilibrium equations is obtained.
The support conditions are applied at the stiffness matrix as well as the load matrix.
The final system of equations is solved with the Gauss elimination method, and the nodal displacements are obtained.
From the nodal displacements the element end displacements are obtained. These nodal displacements multiplied by the element stiffness matrix have as a result the internal forces at the element ends.
From the element end forces the diagrams of bending moments, shear forces and axial forces are obtained.

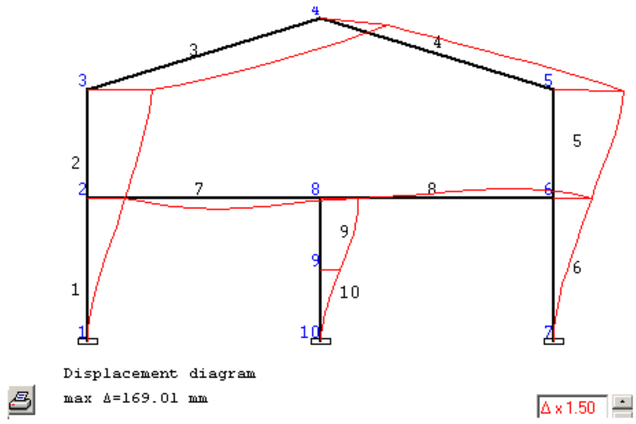
16. Dynamic Analysis



The dynamic analysis starts with the corresponding menu. The analysis steps are shown on the screen with their corresponding duration.
First the 6x6 stiffness and mass matrices are constructed. For the stiffness matrix linear elastic theory is used, and for the mass matrix consistent formulation is used. The element matrices are assembled in the global stiffness and mass matrices that have dimensions 3Nx3B, where N is the number of nodes and N the nodal bandwidth. The matrix eigenvalue equation is solved using the generalized Jacobis method.

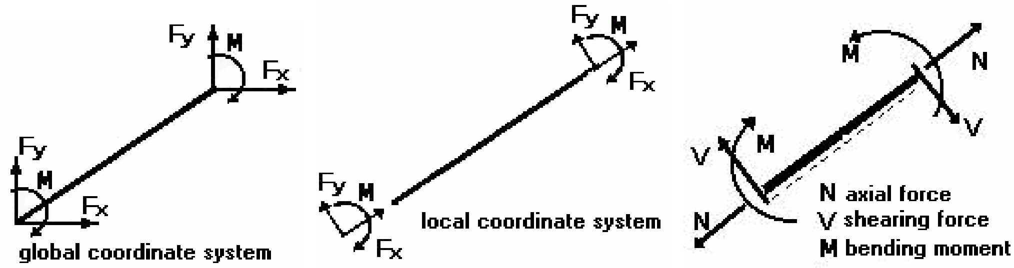
17. Displacements

The adjustment of the colours, the line thickness, the diagram subdivisions and the scaling of the diagrams, can be done with the menu Graphic parameters.
If you select white colour for the node or element numbers they will not appear in the diagrams.



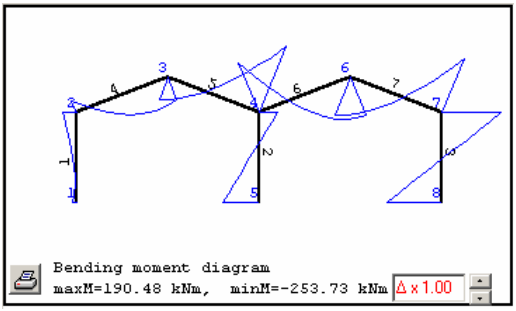
18. Internal forces

Member end forces



19. Diagrams

Use the menu Graphic parameters, to adjust the colour and thickness of the lines of the diagrams. If you select white colour for the colour of the node or element numbers the numbers will not show on the diagrams.
Diagrams of **bending moments**, **shearing forces** and **axial forces**.

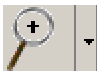


From Graphic parameters :

- adjust the number of subdivisions at which the diagrams are computed for each element.
- adjust the colour and the line thickness of the diagrams. Selecting white colour for the colour of the node or element numbers the numbers will not show on the diagrams.
- adjust the scale and the grid appearance

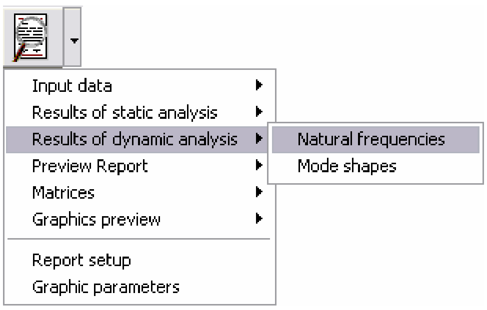
The diagram size is regulated by clicking at 

Quick print of the diagram by clicking at 

Zoom the diagram (x1, x2, x3, x4 times) by clicking at 

20. Natural frequencies

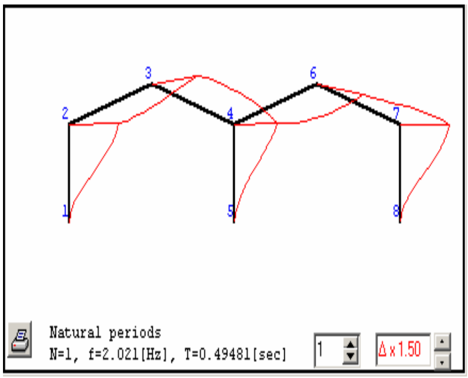
The natural frequencies in (Hz), and the mode shapes are obtained after the dynamic analysis.



Natural frequencies

α/α	Frequency[Hz]	Period[sec]
1	2.02097	0.49481
2	4.36340	0.22918
3	4.59963	0.21741
4	20.35550	0.04913
5	24.93403	0.04011
6	37.34861	0.02677
7	54.38084	0.01839
8	59.47920	0.01681
9	77.59237	0.01289
10	94.43947	0.01059
11	96.52521	0.01036
12	110.57478	0.00904
13	127.48753	0.00784
14	165.85268	0.00603
15	175.34224	0.00570

21. Mode shapes



The natural frequencies and the mode shapes are obtained after the dynamic analysis.

The mode shape number is selected by clicking at

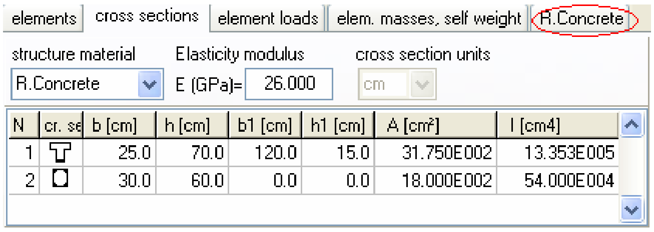
The diagram size is regulated by clicking at

Quick print of the diagram by clicking at

Zoom the diagram (x1, x2, x3, x4 times) by clicking at

22. 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.

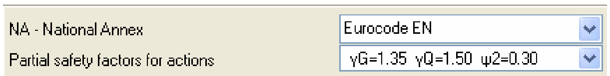


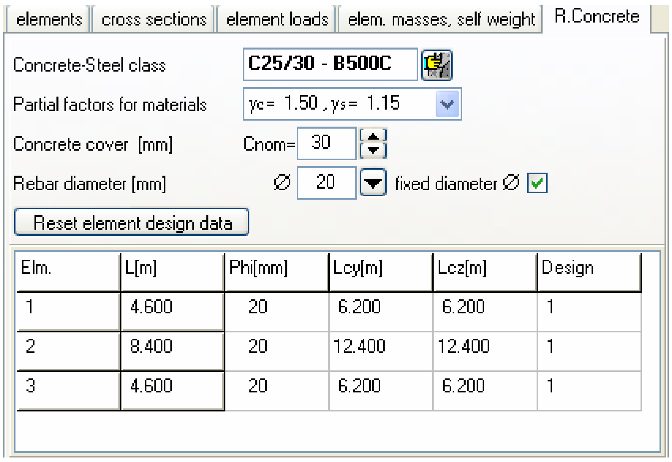
22.1 Design parameters for reinforced concrete

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:
$\gamma c = 1.50$, $\gamma s = 1.15$.

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.

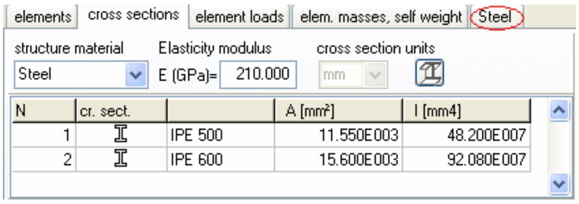
6. Design: = 1 The reinforced concrete design of this element is performed.
= 0 This element is skipped in the design.

22.2 Reinforced concrete design according to Eurocode 2

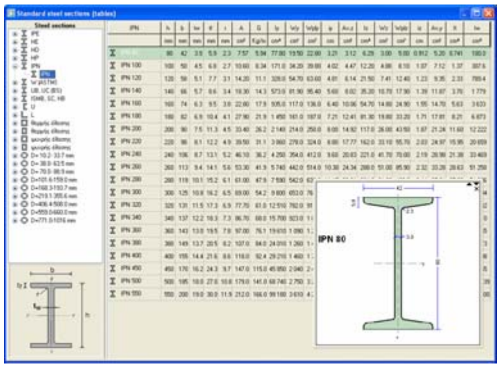
Click Reinforced Concrete Design. All the marked with Design = 1 elements will be verified according to Eurocode 2, §6, for axial force, shear and bending moment in ultimate limit state. The design for reinforcement is performed for mid span, left end and right end of each element. The vertical elements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

23. 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.



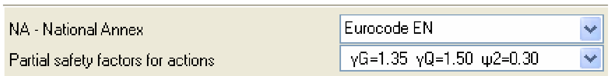
If you click on a line of the table on cross section or click the table with cross sections shows up and you select a standard profile for this element.
In order to proceed with the steel design you have to select standard profiles for all the elements.



23.1 Design parameters for steel

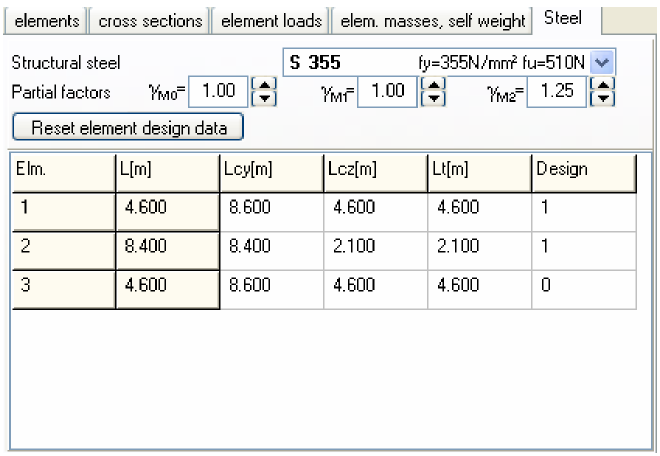
In page Steel you define parameters for the steel design.

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: γM0=1.00, γM1=1.00, γM2=1.25



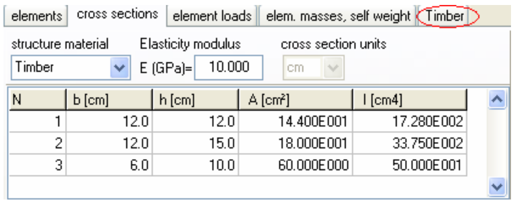
3. For every element you can defile:
Lcy: Buckling length for in-plane flexural buckling (meters), usually the length of the member.
Lcz: Buckling length for out-of-plane flexural buckling (meters), usually the distance of lateral supports as purlins for rafters.
Lt: Buckling length for lateral torsional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

23.2 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.

24. Designing timber members according to Eurocode 5

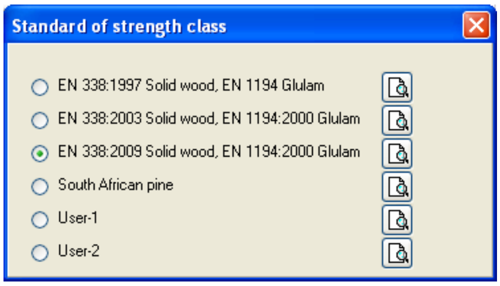


After you select material *Timber* an additional page Timber shows in the material session where you define the additional parameters for timber design.

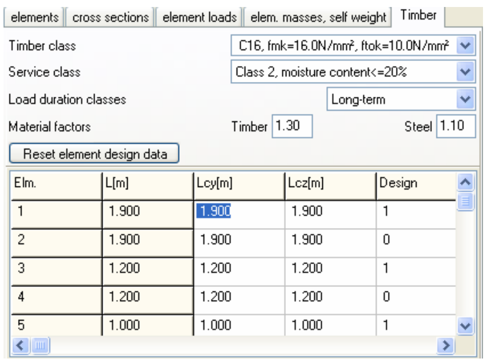
24.1 Design parameters for timber

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).



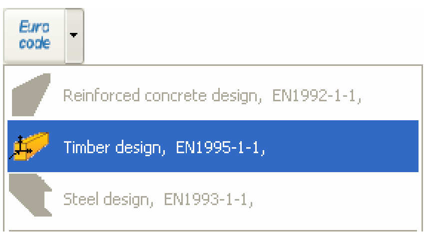
2. Select service class.
Select load duration class. Usually self weight is permanent, snow load and live load is long term, wind load is short term.

3. Partial factors for materials. This is defined according to National annex, usual values: γM=1.30, for timber and γM=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.

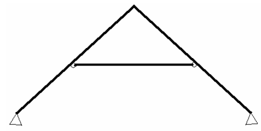
5. Design: = 1 The timber design of this element is performed.
= 0 This element is skipped in the design.

24.2 Timber design according to Eurocode 5

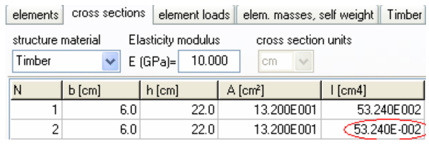


Click Timber Design. All the marked width Design = 1 elements will be verified according to Eurocode 5, §6, for axial force, shear and bending moment in ultimate limit state and according to §6.3 for stability. The buckling critical lengths are defined in the timber design page.
The checks are performed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

24.3 Design notes



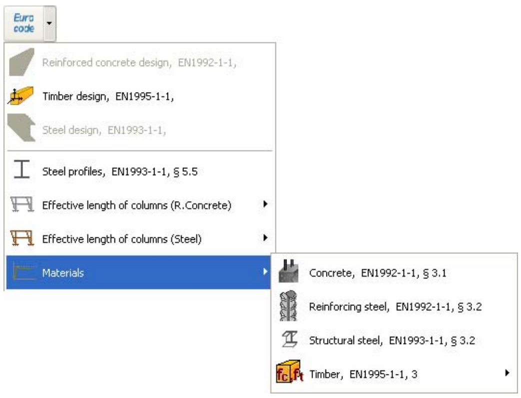
The connections in frame are fixed connections. In case you have to define some timber elements which they are pin-connected to other elements then for these elements define a separate material-section group and after you define the b and h of the cross section, change the moment of inertia to a small 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.

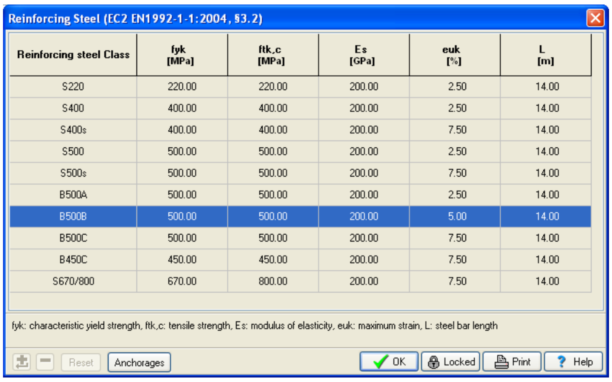


N	b [cm]	h [cm]	A [cm²]	I [cm4]
1	6.0	22.0	13.200E001	53.240E002
2	6.0	22.0	13.200E001	53.240E-002

25. Materials for Reinforcing Concrete, Structural Steel and Timber

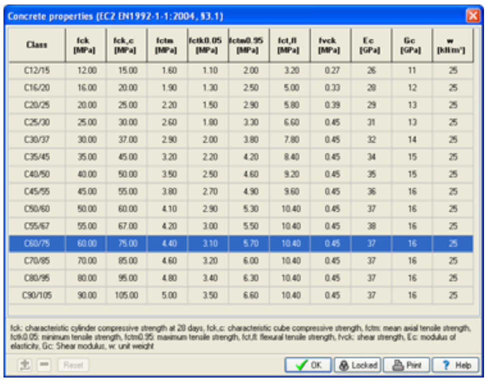
The materials for concrete, reinforcing steel, structural steel and timber are adjusted according to the selected National Annex.
You can change material properties from Design/Materials.
In order to change values for materials first you have to unlock the tables with properties.
Click Closed.
For timber 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.





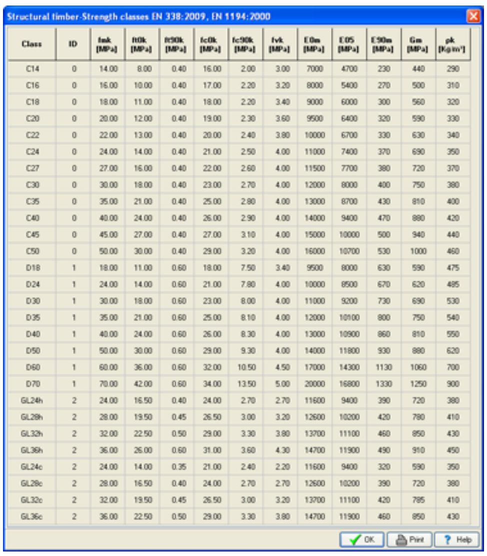
Reinforcing steel Class	fyk [MPa]	ftk,c [MPa]	Es [GPa]	euk [%]	L [m]
S220	220.00	220.00	200.00	2.50	14.00
S400	400.00	400.00	200.00	2.50	14.00
S400s	400.00	400.00	200.00	7.50	14.00
S500	500.00	500.00	200.00	2.50	14.00
S500s	500.00	500.00	200.00	7.50	14.00
B500A	500.00	500.00	200.00	2.50	14.00
B500B	500.00	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

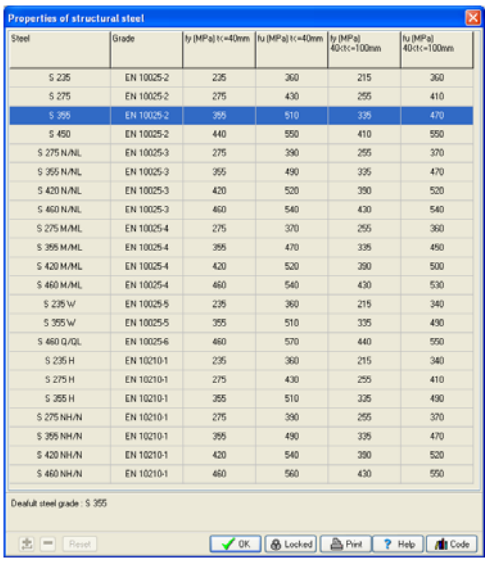


Class	fck [MPa]	fck,c [MPa]	fctm [MPa]	fctk0.05 [MPa]	fctm0.95 [MPa]	fct,fl [MPa]	fvck [MPa]	Ec [GPa]	Gc [GPa]	w [kN/m³]
C12/15	12.00	15.00	1.60	1.10	2.00	3.20	0.27	26	11	25
C16/20	16.00	20.00	1.90	1.30	2.50	5.00	0.33	28	12	25
C20/25	20.00	25.00	2.20	1.50	2.90	5.80	0.39	29	13	25
C25/30	25.00	30.00	2.60	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



Class	ID	fmk [MPa]	ft0k [MPa]	ft90k [MPa]	fc0k [MPa]	fc90k [MPa]	fvk [MPa]	E0m [MPa]	E05 [MPa]	E90m [MPa]	Gm [MPa]	ρk [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

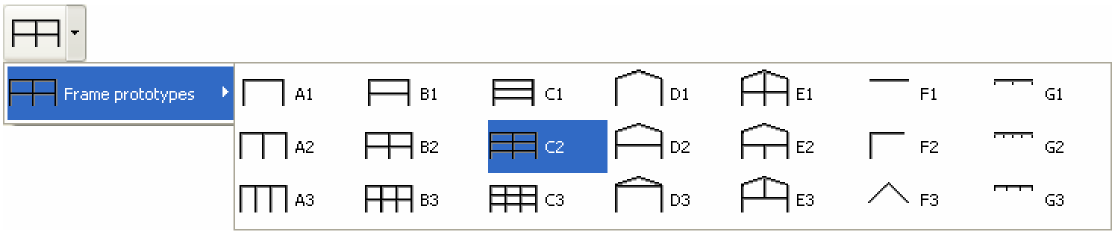


Steel	Grade	fy (MPa) t<=40mm	fu (MPa) t<=40mm	fy (MPa) 40<t<=100mm	fu (MPa) 40<t<=100mm
S 235	EN 10025-2	235	360	215	360
S 275	EN 10025-2	275	430	255	410
S 355	EN 10025-2	355	510	335	470
S 450	EN 10025-2	440	550	410	550
S 275 N/NL	EN 10025-3	275	390	255	370
S 355 N/NL	EN 10025-3	355	490	335	470
S 420 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

26. Frame prototypes

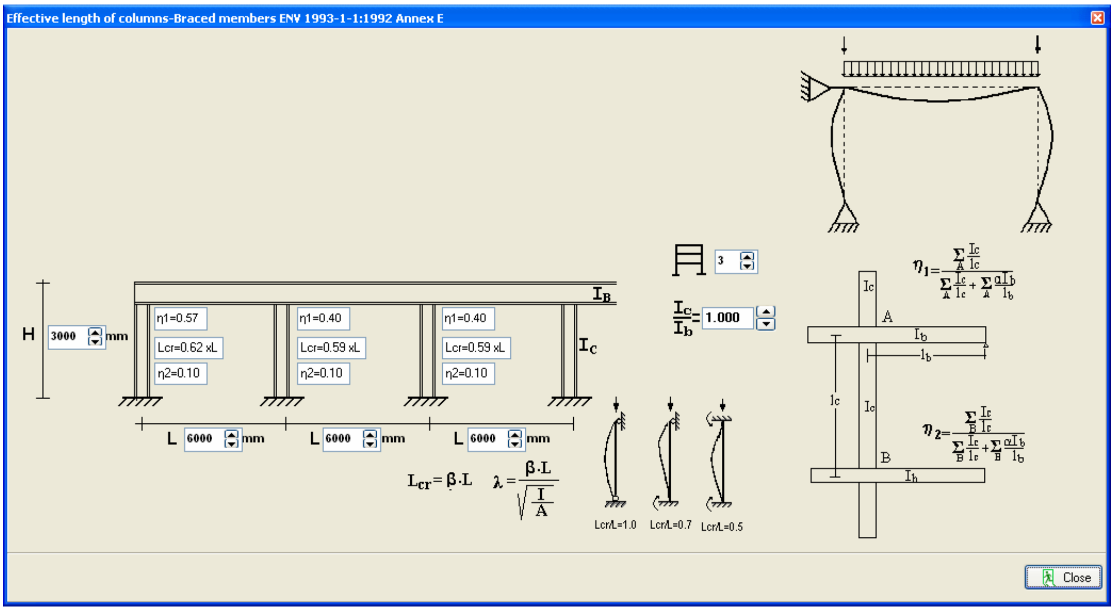
Selecting a Frame prototypes the program defines the nodal coordinates, support conditions and element properties and connectivity.



26.1 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



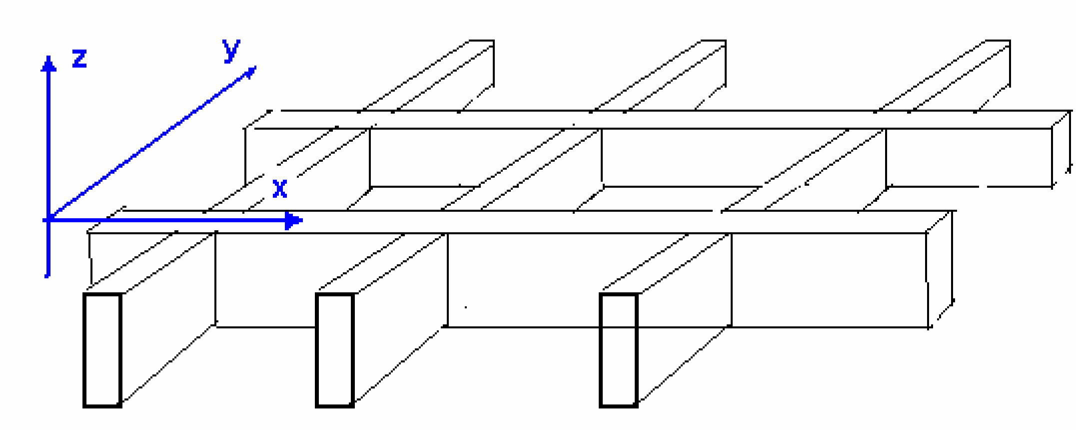
27. 2-D Grillages

Solution and design of 2-D Grillages

New file or Open file from the main menu File\File [Grillage].
Extension of files of grillages is *.GRID2DexpressData.

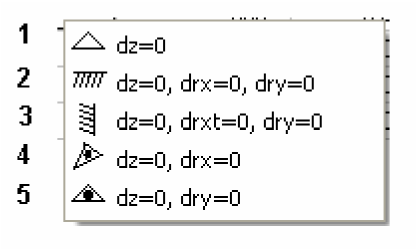
X,Y coordinates and element data as for frame structures.

Coordinate system as in figure, X,Y coordinated at the plane of the grillage, Z coordinate perpendicular to the grillage plane.



Supports

1 : Z movement restricted dz = 0
2,3 : All movements and rotations are zero.
4 : Vertical movement and rotation around the x-x axis restricted. dz = 0, drx = 0. Specify this for beams in the x-x direction if the
twisting deformations are blocked at the supports.
5 : Vertical movement and rotation around the y-y axis restricted. dz = 0, dry = 0. Specify this for beams in the y-y direction if the twisting deformations are blocked at the supports.



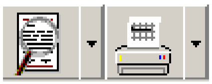
For the solution of the Grillage the shear modulus is taken as 0.40 of the modulus of elasticity, G=0.40xE

For **Nodal loads** in the Z directions downwards specify minus (-) sign.
Element loads with plus sign are considered downwards.

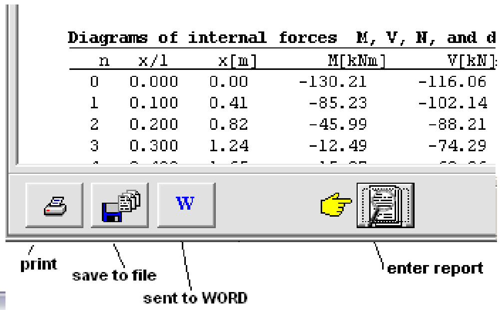
Design for reinforced concrete, timber or steel includes design for twisting (torsional) moments.

28. Reports

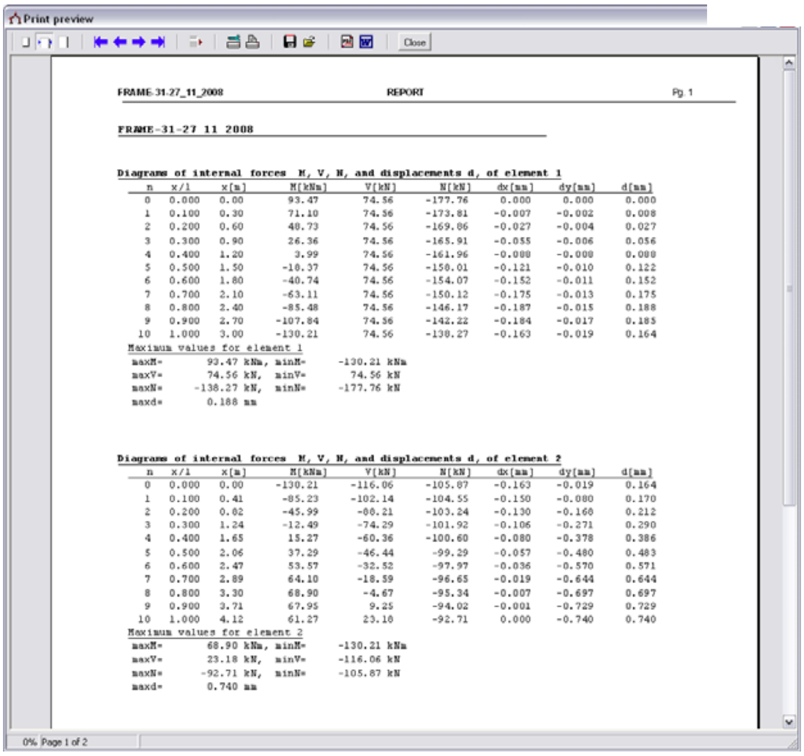
The report appears with the preview or print menu commands.



With the buttons at the bottom left you can **print** the report, **save** as an RTF file, or activate word and load the report in it.
The fourth button when pressed, unlocks the ability to edit the report.
The font, and the margins are adjusted from the menu.



Report example



FRAME-31-27_11_2008 REPORT Pg. 1

FRAME-31-27 11 2008

Diagrams of internal forces M, V, N, and displacements d, of element 1

n	x/l	x[m]	M[kNm]	V[kN]	N[kN]	dx[mm]	dy[mm]	d[mm]
0	0.000	0.00	93.47	74.56	-177.76	0.000	0.000	0.000
1	0.100	0.30	71.10	74.56	-173.81	-0.007	-0.002	0.008
2	0.200	0.60	48.73	74.56	-169.86	-0.027	-0.004	0.027
3	0.300	0.90	26.36	74.56	-165.91	-0.055	-0.006	0.056
4	0.400	1.20	3.99	74.56	-161.96	-0.088	-0.008	0.088
5	0.500	1.50	-18.37	74.56	-158.01	-0.121	-0.010	0.122
6	0.600	1.80	-40.74	74.56	-154.07	-0.152	-0.011	0.152
7	0.700	2.10	-63.11	74.56	-150.12	-0.175	-0.013	0.175
8	0.800	2.40	-85.48	74.56	-146.17	-0.187	-0.015	0.188
9	0.900	2.70	-107.84	74.56	-142.22	-0.184	-0.017	0.185
10	1.000	3.00	-130.21	74.56	-138.27	-0.163	-0.019	0.164

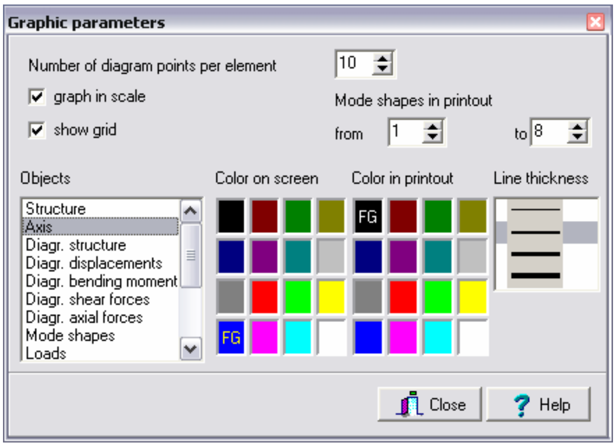
Maximum values for element 1
maxM= 93.47 kNm, minM= -130.21 kNm
maxV= 74.56 kN, minV= 74.56 kN
maxN= -138.27 kN, minN= -177.76 kN
maxd= 0.188 mm

Diagrams of internal forces M, V, N, and displacements d, of element 2

n	x/l	x[m]	M[kNm]	V[kN]	N[kN]	dx[mm]	dy[mm]	d[mm]
0	0.000	0.00	-130.21	-116.06	-105.87	-0.163	-0.019	0.164
1	0.100	0.41	-85.23	-102.14	-104.55	-0.150	-0.080	0.170
2	0.200	0.82	-45.99	-88.21	-103.24	-0.130	-0.168	0.212
3	0.300	1.24	-12.49	-74.29	-101.92	-0.106	-0.271	0.290
4	0.400	1.65	15.27	-60.36	-100.60	-0.080	-0.378	0.386
5	0.500	2.06	37.29	-46.44	-99.29	-0.057	-0.480	0.483
6	0.600	2.47	53.57	-32.52	-97.97	-0.036	-0.570	0.571
7	0.700	2.89	64.10	-18.59	-96.65	-0.019	-0.644	0.644
8	0.800	3.30	68.90	-4.67	-95.34	-0.007	-0.697	0.697
9	0.900	3.71	67.95	9.25	-94.02	-0.001	-0.729	0.729
10	1.000	4.12	61.27	23.18	-92.71	0.000	-0.740	0.740

Maximum values for element 2
maxM= 68.90 kNm, minM= -130.21 kNm
maxV= 23.18 kN, minV= -116.06 kN
maxN= -92.71 kN, minN= -105.87 kN
maxd= 0.740 mm

29. Graphic parameters



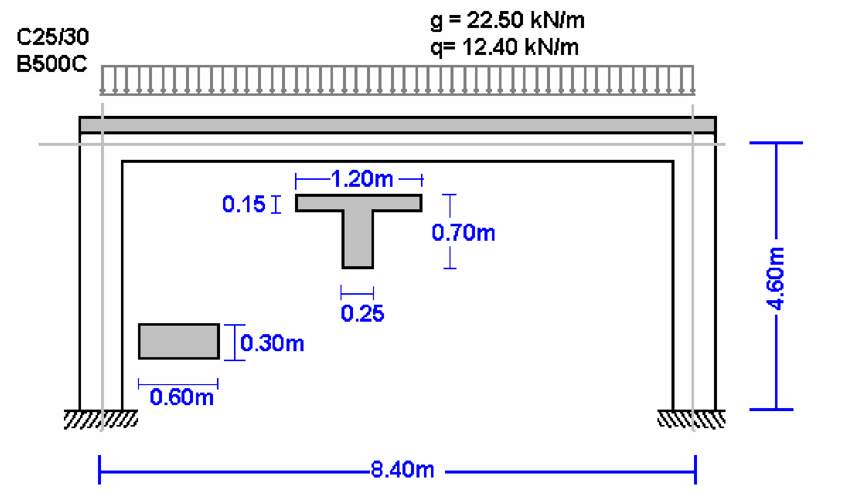
Parameters for graphics on the screen or in the report.
Number of diagram points per elements. The number of subdivision points for each element, at which the diagrams are computed and plotted.
For the graphic appearance you select the graphic object at the left list box and then you click and select from the colour boxes the colour you prefer, and the line thickness. If you select white colour for the colour of the node or element numbers the numbers will not show on the diagrams.

The range of mode shapes in printout is selected with the two values beginning (from) and end (to) mode shape number.
Checking graph in scale the graph will appear with the same scale in horizontal and vertical direction. Otherwise both directions will expand to the available screen or report size. In many cases it is preferable to have the graph out of scale in order to appear more clear.

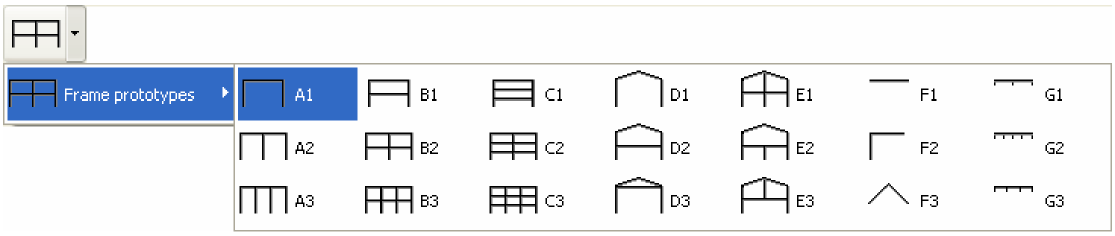
30. Examples

30.1 Example 1

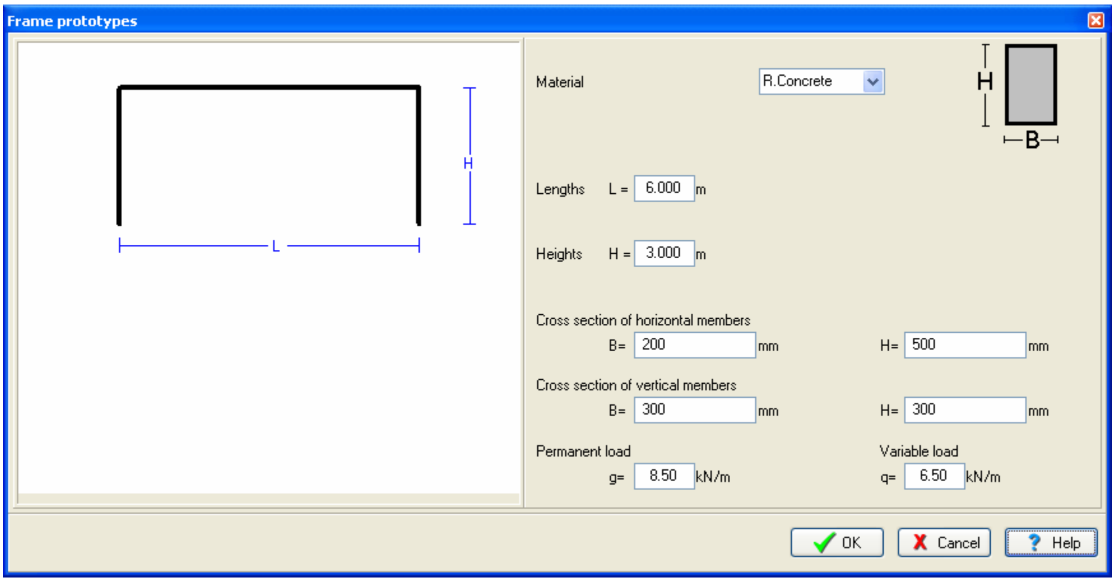
Frame of reinforced concrete 8.40 m x 4.60 m C25/30 B500C



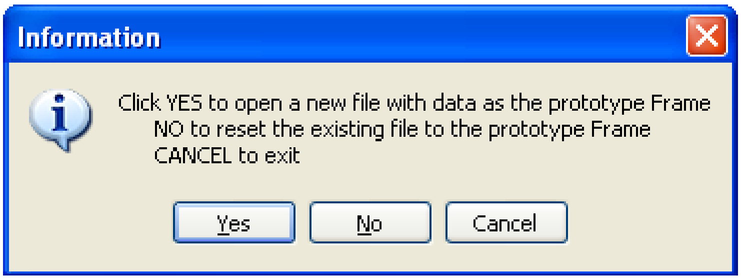
Select a frame from File/Frame prototypes:



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		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

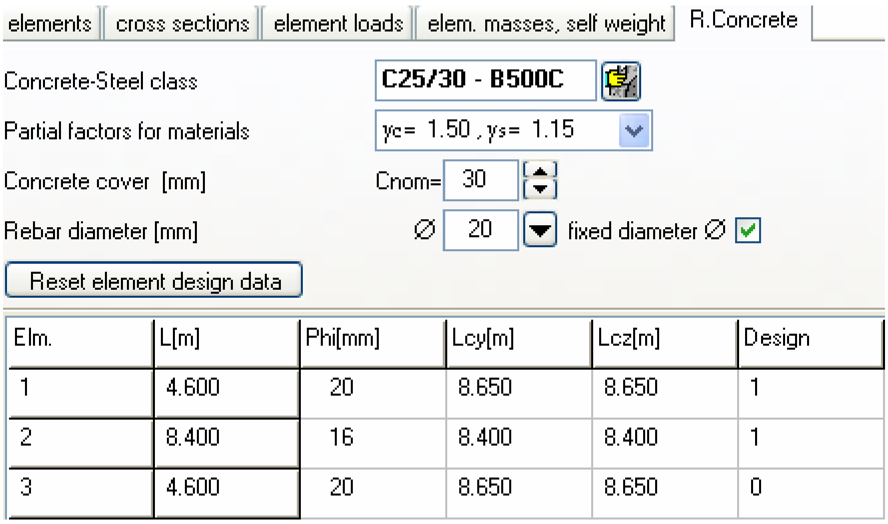
Weight density kN/m3 25.00 — include self weight in loads and masses ☑

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

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

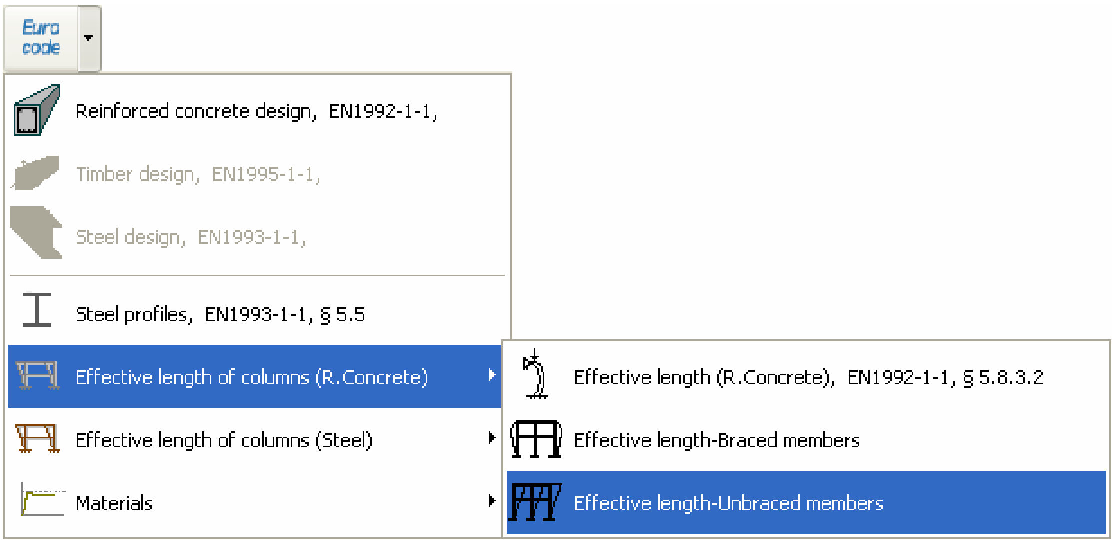
- Concrete. You specify the basic data for the design of reinforced concrete according to Eurocode 2. Select Concrete and Steel class. Partial factors for materials, according to the National Annex, for ULS (ultimate limit state) γc=1.50, γs=1.15 and SLS (serviceability limit state) γc=1.00, γs=1.00.
  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

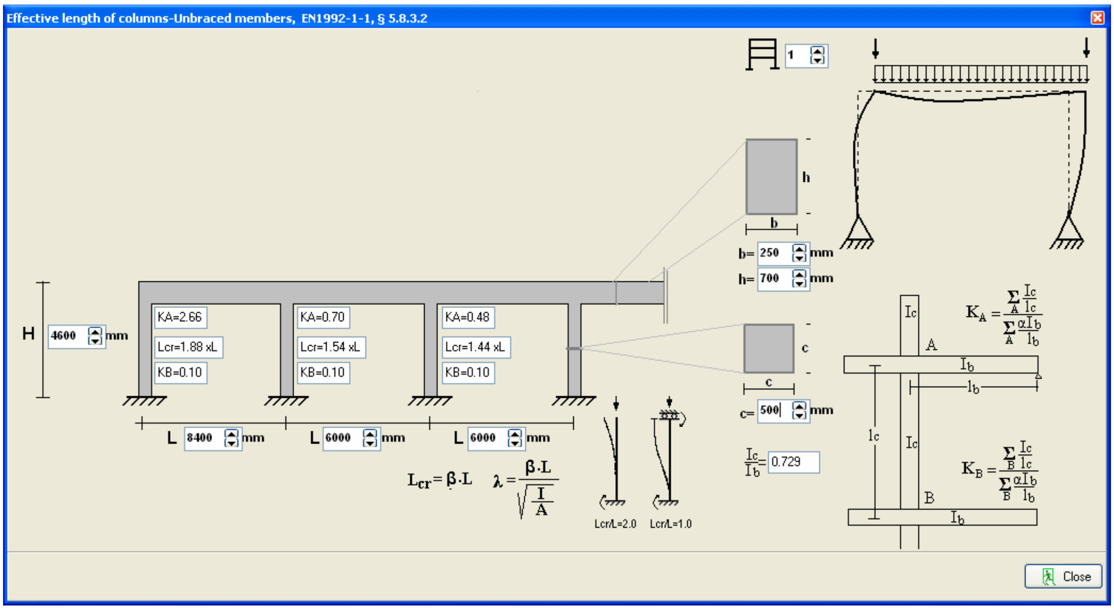
second order effects for the columns, according to Eurocode 2 §5.8.3.
In the column Design, mark with 1 the elements which you want to be included in the design of reinforced concrete. In this example the elements 1 and 2 are marked with (1) and element 3 with (0), as there is no need because of symmetry to include element 3 (right column) in the reinforced concrete design.



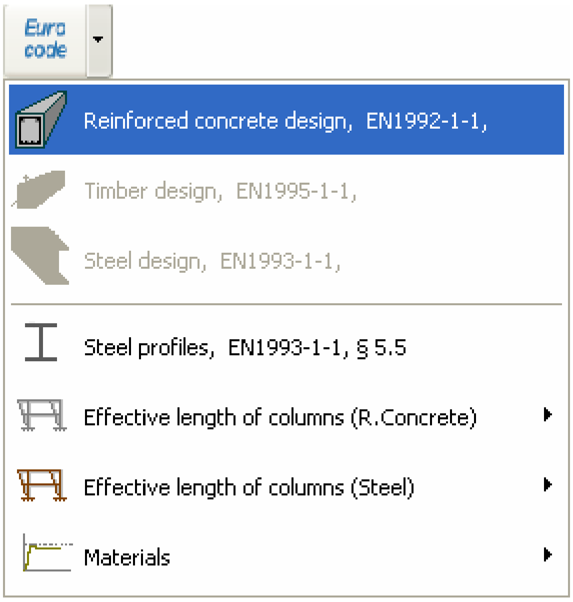
Elm.	L[m]	Phi[mm]	Lcy[m]	Lcz[m]	Design
1	4.600	20	8.650	8.650	1
2	8.400	16	8.400	8.400	1
3	4.600	20	8.650	8.650	0

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

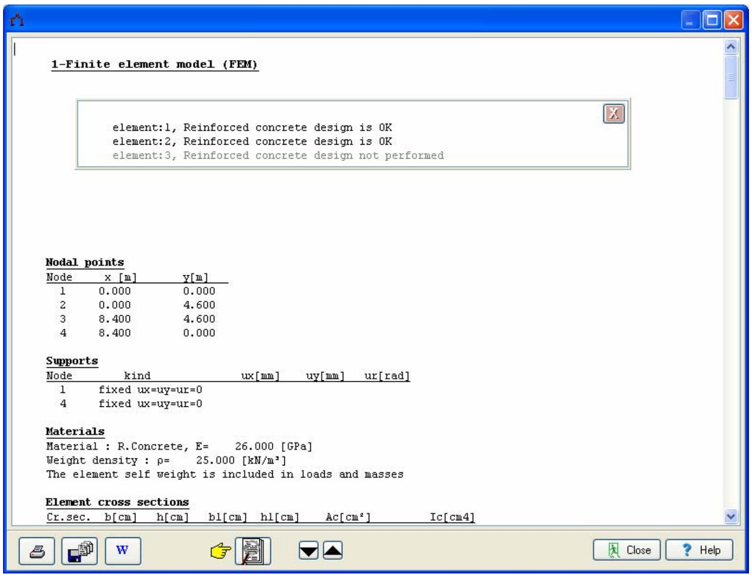




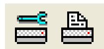
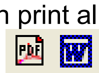
After you give all the data the reinforced concrete design is performed according to Eurocode 2.

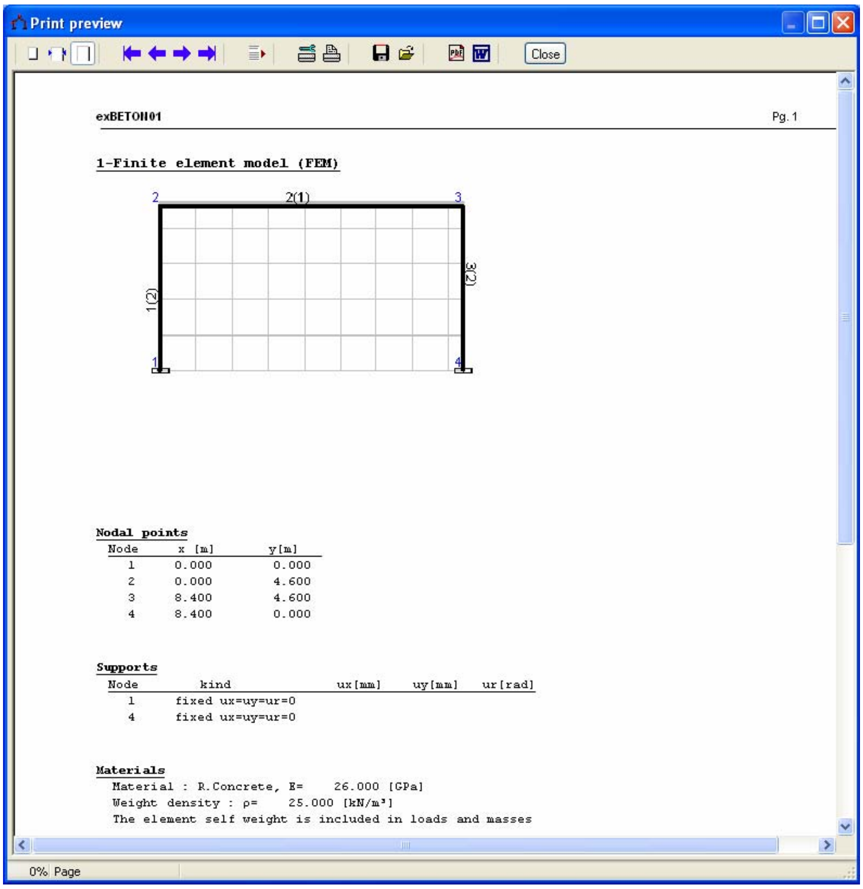


Check if every element is verified in the design.



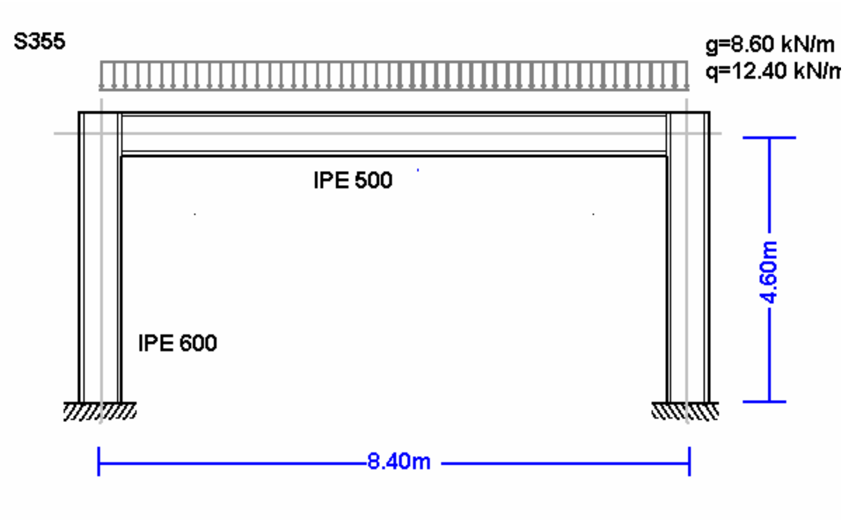
Click  for complete formatted report.

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 .

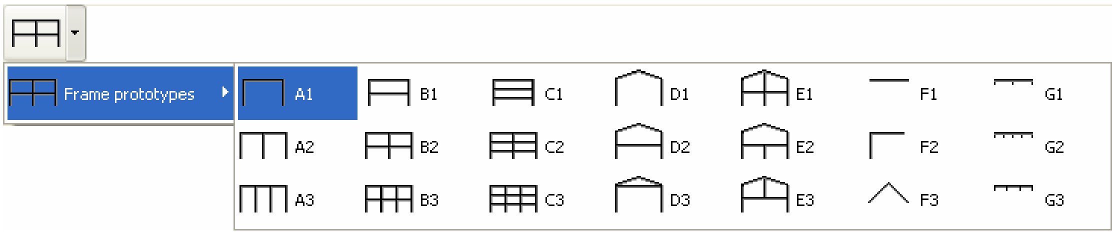


30.2 Example 2

Steel frame 8.40 x 4.60 S355

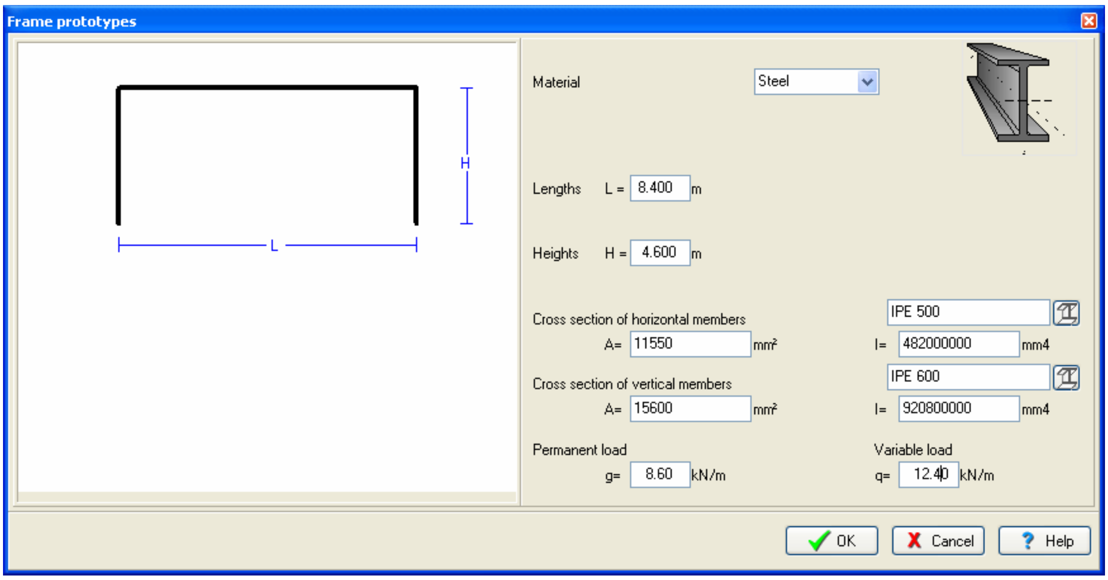


Select a frame from File/Frame prototypes:

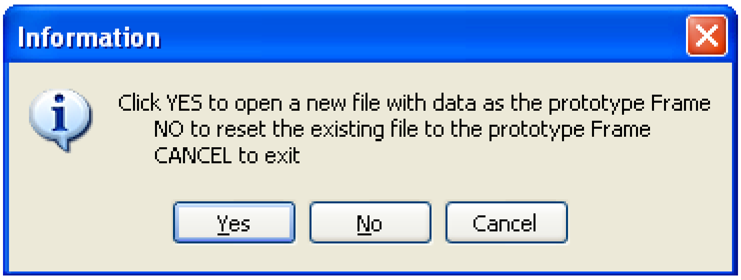


Select material Steel and give the basic dimensions, cross sections and loads.
You can always change and adjust these values afterwards.

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.

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. 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

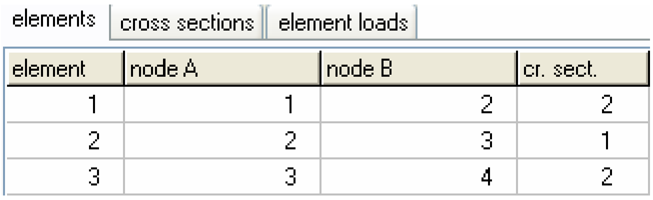
load combination 1.35 **xFg+** 1.50 **xFq**

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.

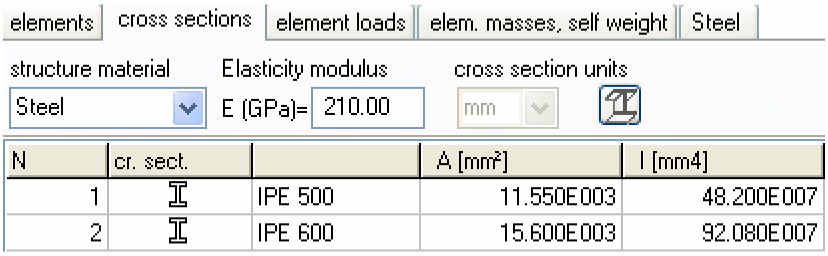
- Elements. The element numbering is displayed on the drawing of the structure.
  Nodes A and B are the left and right nodes of each element. Cross section is the number in

parenthesis next to each element and represents the number of the section group which properties defined in the page cross sections.



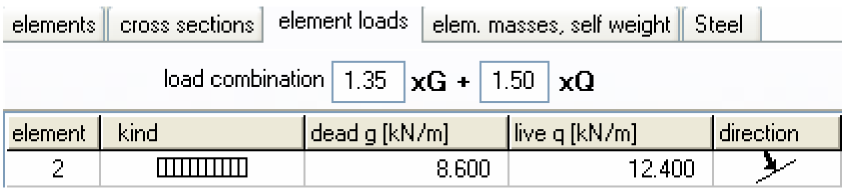
element	node A	node B	cr. sect.
1	1	2	2
2	2	3	1
3	3	4	2

- 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.



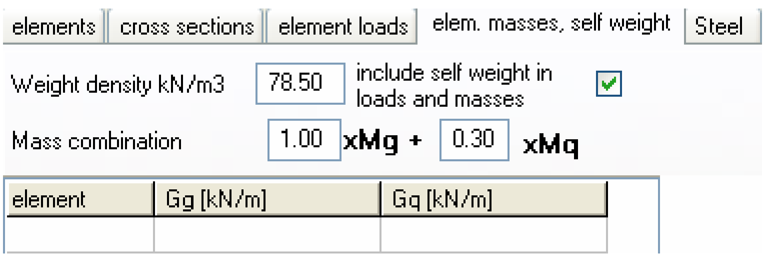
N	cr. sect.		A [mm²]	I [mm4]
1		IPE 500	11.550E003	48.200E007
2		IPE 600	15.600E003	92.080E007

- 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	

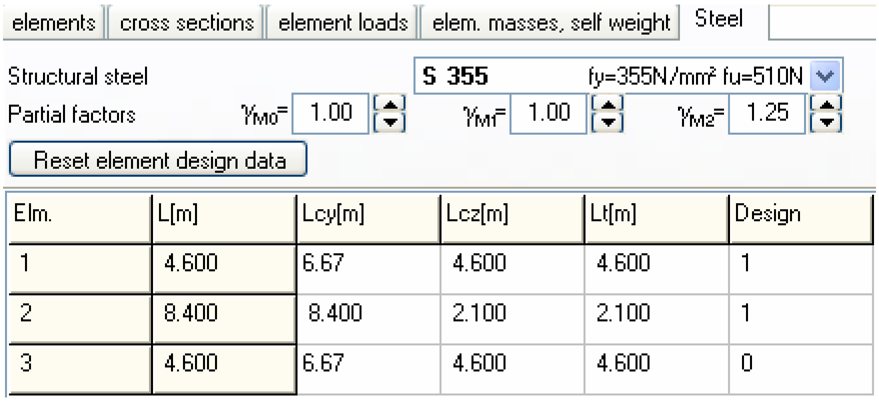
- Element masses, self weight. The weight density of the material. If you check to include the self weight in the load and masses, the program adds in dead loads the self weight of each element (unit weight) x (cross-section area). The weight density is set automatically by the program (steel 78.50 kN/m^3).



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

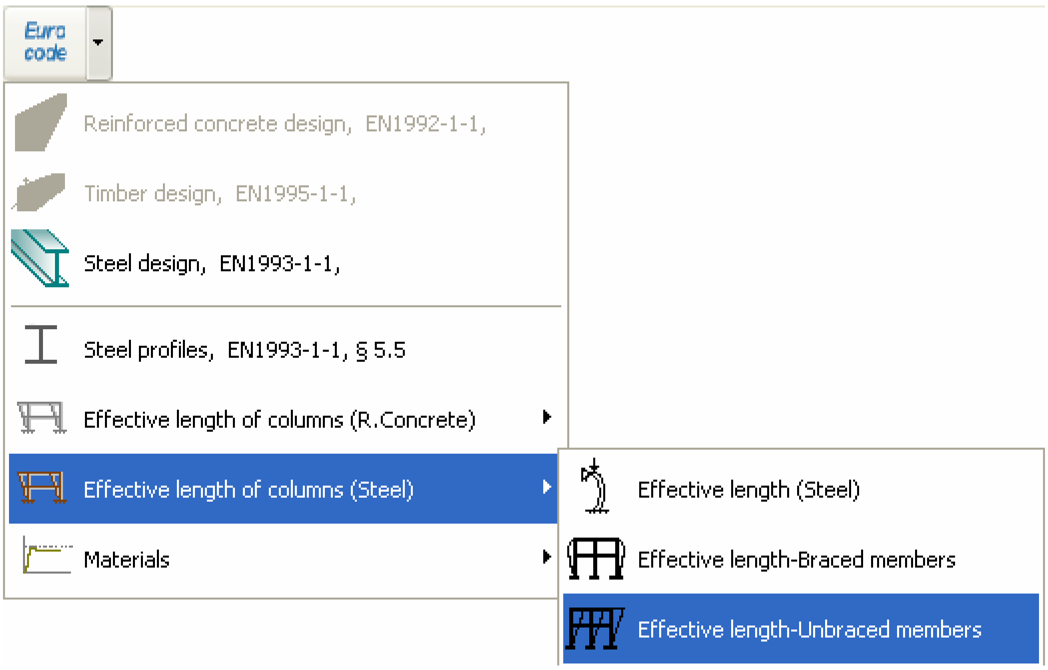
- Steel. You specify the basic data for the design of steel according to Eurocode 3.
  Select Steel grade. Partial factors for materials, according to National annex, for ULS (ultimate limit state) $\gamma M0 = 1.00$, $\gamma M1 = 1.00$, $\gamma M2 = 1.25$. You have to define the buckling lengths. Lcy buckling length for in plane buckling. For braced frames this is less or equal to the member length, for

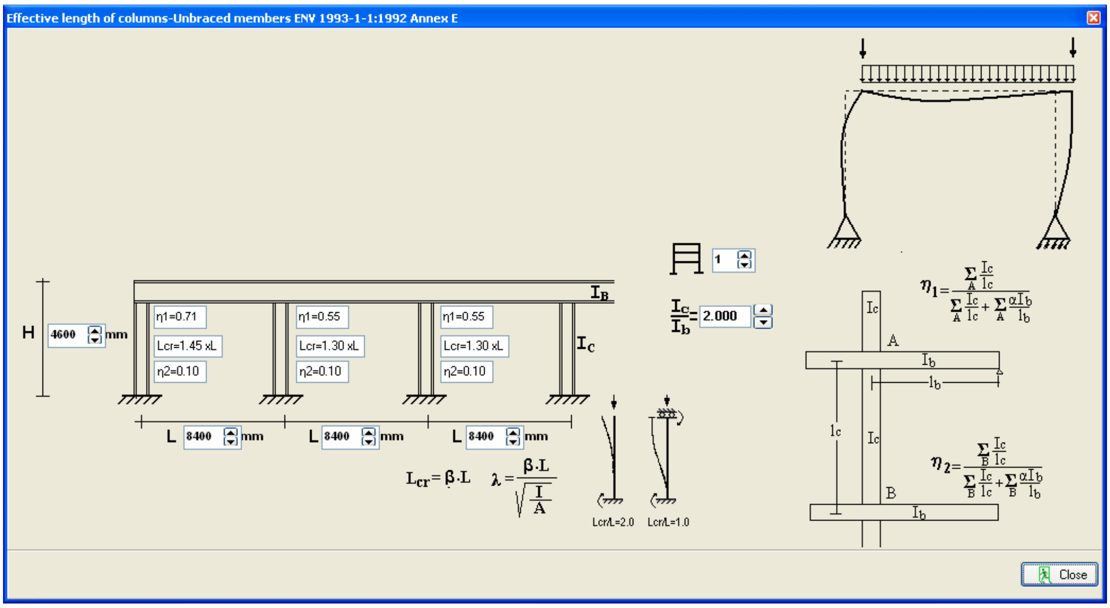
unbraced members is grater. Lcz buckling length for out of plane buckling and it is defined from the lateral supports. For horizontal beams it is usually the distance of the lateral beams or the purlins. In the column Design mark with 1 the elements which you want to be included in the design of steel 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,



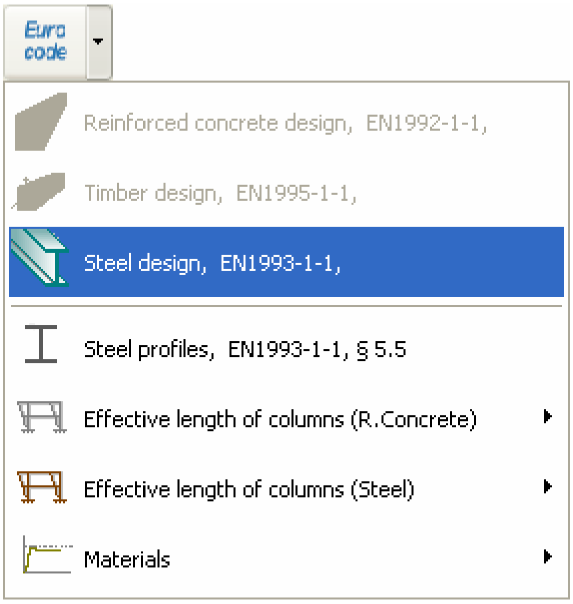
Elm.	L[m]	Lcy[m]	Lcz[m]	Lt[m]	Design
1	4.600	6.67	4.600	4.600	1
2	8.400	8.400	2.100	2.100	1
3	4.600	6.67	4.600	4.600	0

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

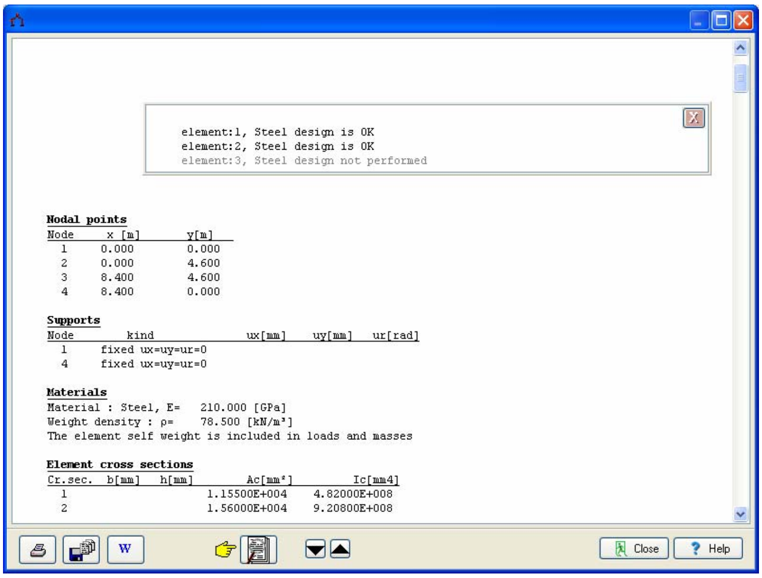




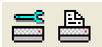
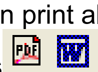
After you give all the data the steel design is performed according to Eurocode 3.

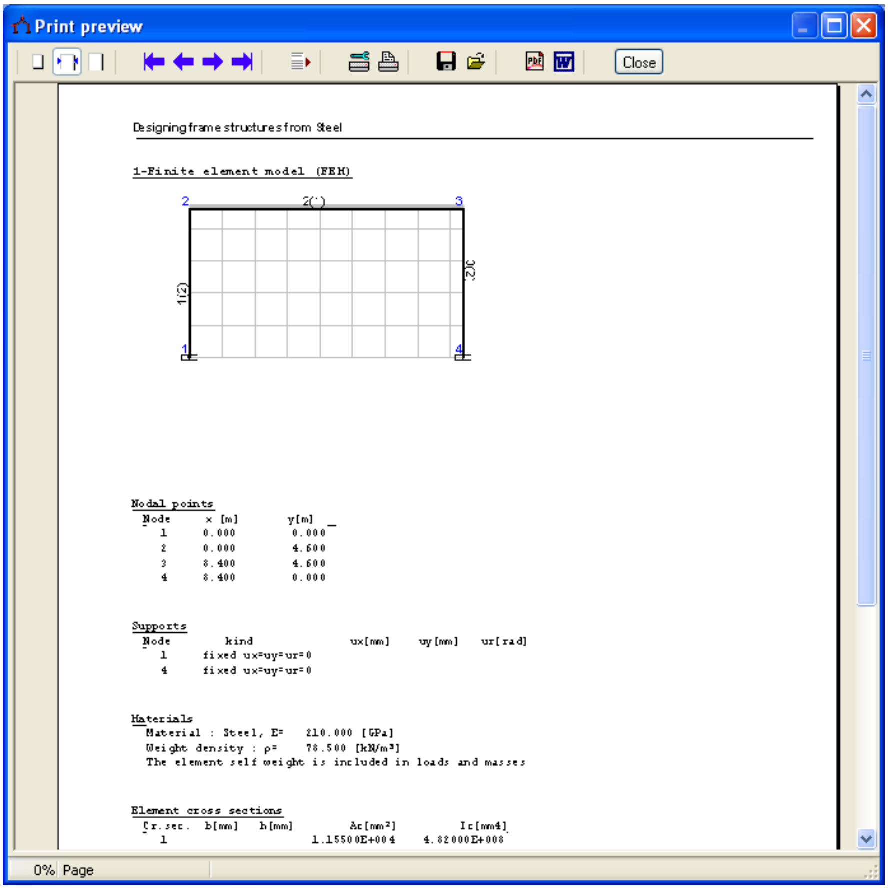


Check if every element is verified in the design,



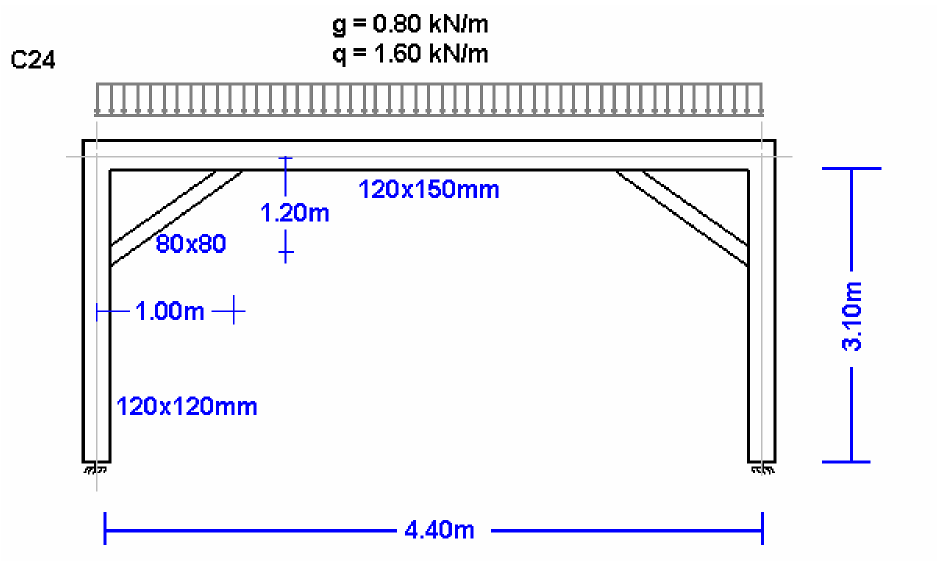
Click  for complete formatted report.

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 .

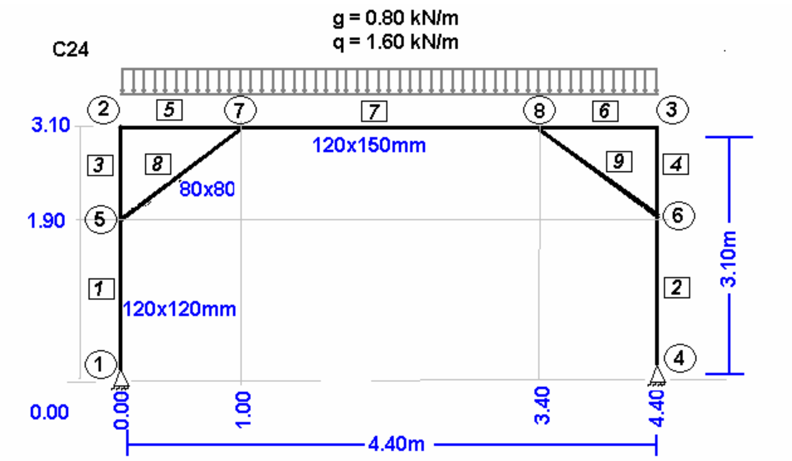


30.3 Example 3

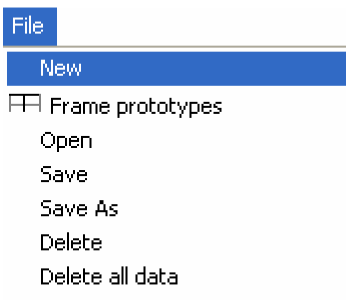
Timber structure 6.40 x 4.60 C24



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) $\gamma G=1.35$, $\gamma Q=1.50$ and SLS (serviceability limit state) $\gamma G=1.00$, $\gamma Q=1.00$.

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

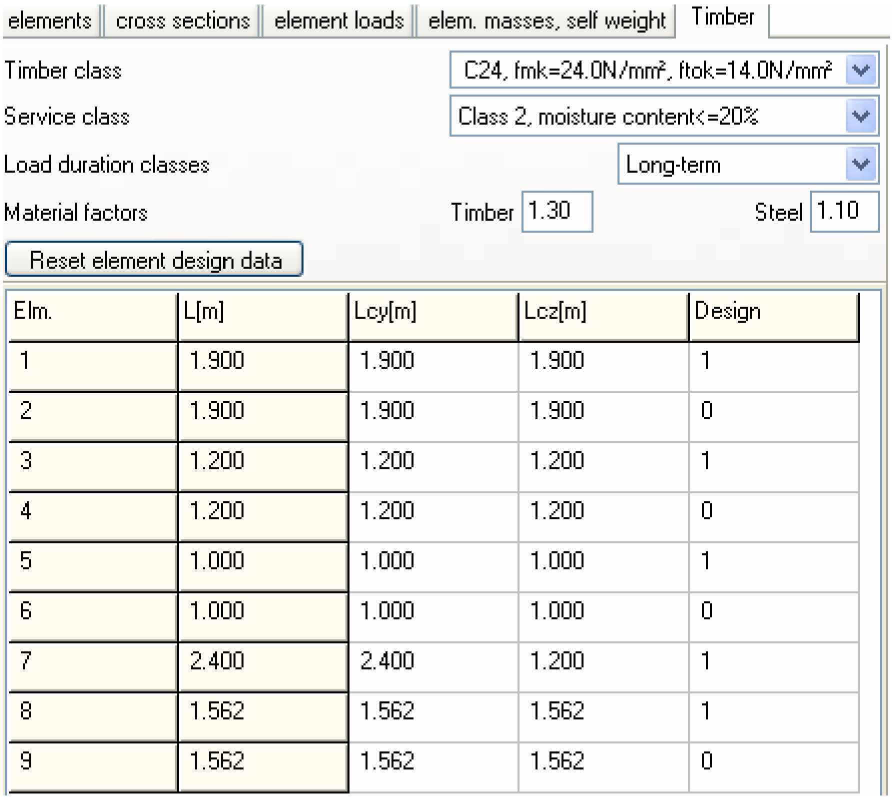
load combination 1.35 **xG +** 1.50 **xQ**

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).

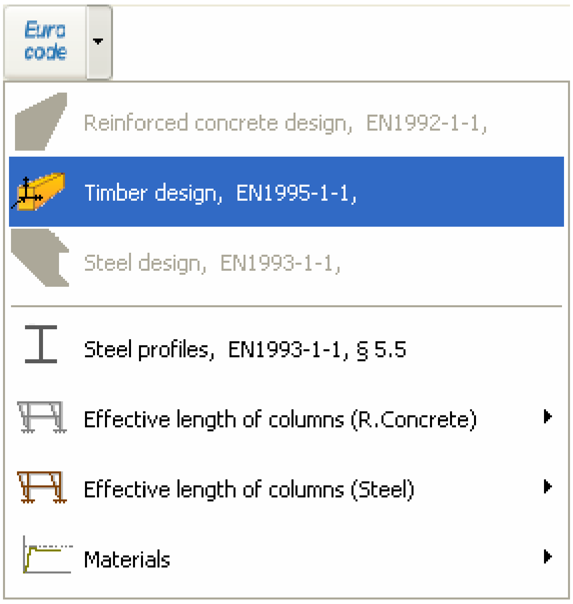
- Timber. You specify the basic data for the design of the timber members according to Eurocode 5. Select timber class (C24), service class and load duration class.

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).

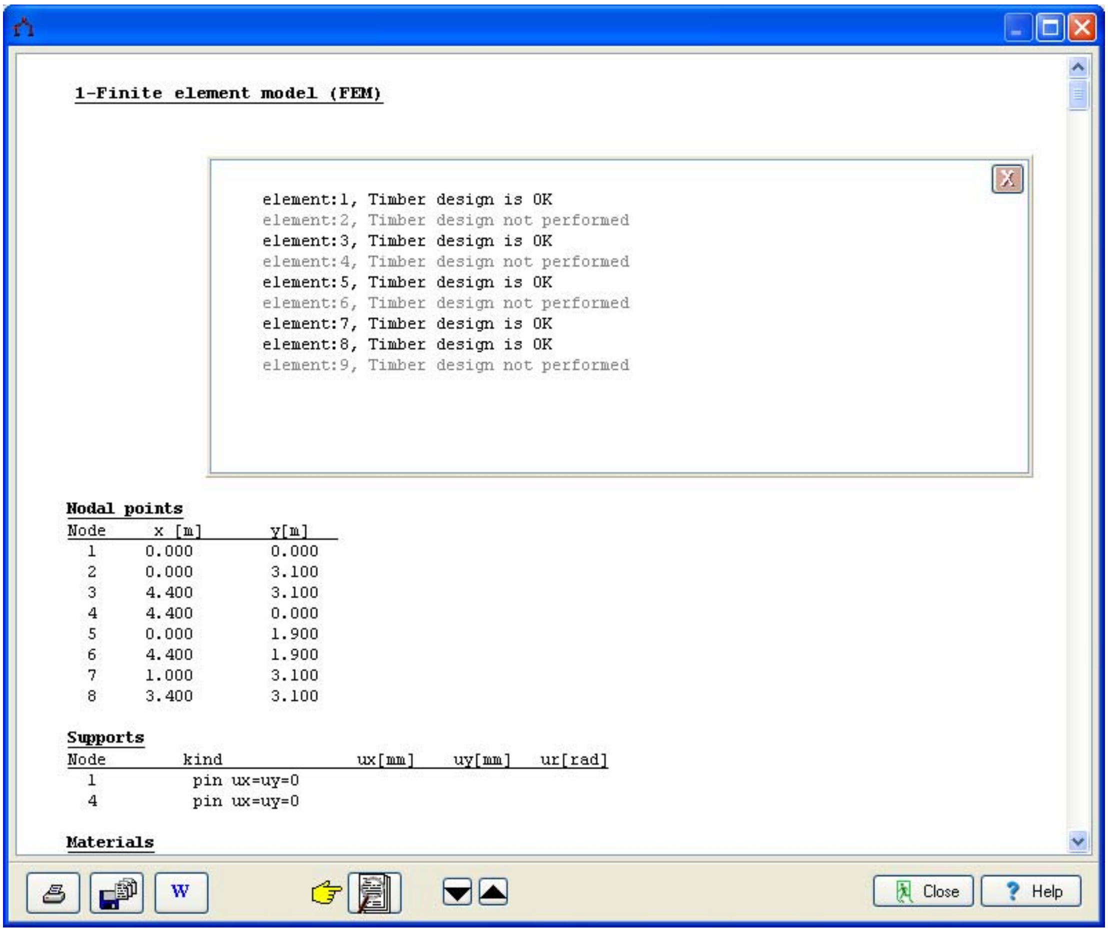


Elm.	L[m]	Lcy[m]	Lcz[m]	Design
1	1.900	1.900	1.900	1
2	1.900	1.900	1.900	0
3	1.200	1.200	1.200	1
4	1.200	1.200	1.200	0
5	1.000	1.000	1.000	1
6	1.000	1.000	1.000	0
7	2.400	2.400	1.200	1
8	1.562	1.562	1.562	1
9	1.562	1.562	1.562	0

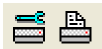
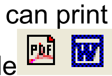
After you give all the data the timber design is performed according to Eurocode 5.



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 .

31. References

Bathe K. J., Wilson E. L.,"Numerical methods in finite element analysis", Prentice-Hall, Inc., 1976.
Gallagher R. H., "Finite element analysis, fundamentals", Prentice-Hall, Inc.,1975.
Ghali A., Neville A. M., "Structural analysis a unified classical and matrix approach", Chapman and Hall, 1978.
Huebner K.H, "The finite element method for engineers", John Wiley and Sons, 1975.
Jennings A., "Matrix computation for engineers and scientists", John Wiley and Sons, 1977.
Meek J. L., "Matrix structural analysis", McGraw-Hill, 1971.
Zienkiewicz O.C., " The finite element method in engineering science". 2nd Ed. McGraw-Hill, 1971.