

© 2000-2016 RUNET Norway as

# USER'S Manual

### Table of contents

1.	General 4
2.	Structural Wizard5
3.	Steel profiles
4.	Coordinate system
5.	Units7
6.	Nodal points7
7.	Element cross sections7
8.	Elements
9.	Elements with hinges9
10.	Supports9
11.	Nodal loads9
12.	Nodal masses10
13.	Element distributed loads10
14.	Element distributed masses11
15.	Static analysis11
16.	Dynamic Analysis11
17.	Displacements
18.	Internal forces12
19.	Diagrams
20.	Natural frequencies12
21.	Mode shapes13
22.	Designing elements from reinforced concrete according to Eurocode 2 13
	22.1 Design parameters for reinforced concrete

	22.2 Reinforced concrete design according to Eurocode 2	14
23.	Designing steel members according to Eurocode 31	14
	<ul><li>23.1 Design parameters for steel</li></ul>	14 15
24.	Designing timber members according to Eurocode 5 1	15
	<ul> <li>24.1 Design parameters for timber</li></ul>	15
25.	Materials for Reinforcing Concrete, Structural Steel and Timber	16
26.	Frame prototypes 1	17
	26.1 Effective lengths for columns	17
27.	2-D Grillages1	8
28.	Reports1	19
29.	Graphic parameters 1	19
30.	Examples2	20
	30.1       Example 1       2         30.2       Example 2       2         30.3       Example 3       2	26
31.	References	36

#### FRAME2Dexpress User's guide. Copyright © RUNET ® software.

The software **FRAME2Dexpress**, described in this users manual, is furnished under a license agreement. The software may be used only in accordance with the terms of the license agreement. Information in this document is subject to change without notice.

#### License and copyright

If you do not agree with the terms of the following Disclaimer and License Agreement, return the program disks before you install and activate it, to RUNET Norway AS, within 30 days of purchase for a full refund of software cost and sales tax.

#### **Disclaimer**

This software should be used only from experienced and licensed professional engineers. The software must be considered as a helping tool for the designer engineer, and can never replace the knowledge, the experience and the judgment of a professional engineer. The user of this software must understand that no matter how advanced and well checked this software is, he should carefully check the results and take responsibility of their use.

#### **Copyright**

This software is owned by RUNET Norway AS, and it is protected by EC (European Community) Copyright Laws and International Treaty Provisions. This software and the accompanying materials, must be treated like any other copyrighted material (e.g. book). It is allowed although to make one copy of the Software for backup or archive purposes. You may not copy and distribute the accompanying materials. It is strictly prohibited by law unauthorized reproduction or resale of this software product and the accompanying materials.

#### Software License

This is a legal agreement between the legal user of this software and RUNET Norway AS. By installing this software you agree to be bound by the terms of this agreement. If you do not agree to the terms of this agreement then do not install this software and return within 30 days after purchase, for a fully refund of your payment.

#### Scope of License

Each licensed copy of **FRAME2Dexpress** must be used either on a single computer, or installed on a single workstation used non-simultaneously by multiple people, but not both. This is not a concurrent use license. You may not rent or lease this software. You may not modify, adapt, translate, reverse engineer, decompose, or disassemble the software. Any violation of this agreement terminates your right to use this software.

#### **Liability Limitations**

**FRAME2Dexpress**, in no event shall be liable for any damages whatsoever (including without limitations, damages for loss of business profits, business interruption, or any other loss) arising of the use of this software. RUNET Norway AS makes no warranties, either expressed or implied, as to the quality or performance of this software, that the results and calculations of this software will meet your requirements, or that the operation of this software will be error free.

This software is a helping tool to aid you in the design. The results of this software must be reviewed and interpreted from experienced licensed engineers, and by no means constitute an acceptable engineering design. **FRAME2Dexpress** and related documentation are provided "AS IS" and without warranties as to performance or merchantability or any other warranties whether expressed or complied. Because of the various hardware and software environment into which this software may be put, no warranty of fitness for a particular purpose is offered. Under no circumstances shall RUNET Norway AS and its personal be liable for any direct or indirect, incidental special or consequential damages resulting from the use or inability to use of this software or related documentation, even if RUNET has been advised of the possibility of such damages.

This agreement shall be governed by EC (European Community) laws. If for any reason a court or competent jurisdiction finds any provision of this agreement, or portion thereof, to be unenforceable, that provision of the agreement shall be enforced to the maximum extend permissible so as to effect the intent of the parties, and the remainder of this agreement shall continue in full force effect.

If this license is too restrictive with the laws of your country, do not use this software and return within 30 days after purchase, for a fully refund of your payment.

RUNET NORWAY as, Tennfjord 6264-N, Norway e-mail: <u>support@runet-software.com</u> 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.
- Results of static analysis:
- 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.

#### Result of dynamic analysis:

- Natural frequencies
- Diagrams of mode shapes

#### Additional printouts

- Stiffness and mass matrices.
- Equilibrium equations
- Eigenvalue equations

#### Units

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.

<u>A</u> ·							
Frame prototypes	• 🕅 A1	┣ В1	<b>—</b> C1	D1	Ē	— F1	<sub>G1</sub>
	∏ A2	В2			FT E2	F2	G2
	∏∏ A3	<b>П</b> ∎з	⊞≣ാ		ʶ E3	∕∧ <sub>F3</sub>	G3

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.

Frame prototypes	Material Cancels x H
на	Lengths L = 6000 m L1= 6000 m L2= 6000 m L2= 6000 m
	Cross rection of horizontal members $B*\left[\frac{50}{20}\right] mm \qquad H*\left[\frac{50}{20}\right] mm$
	Gress section 4 verical remotes:         B ≈ [250]         mm         He [500]         mm           Permanent load         Variable load         Variable load         Variable load         Variable load
1	V DK X Cancel ? Help

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

Informa	ation										
Click YES to open a new file with data as the prototype Frame NO to reset the existing file to the prototype Frame CANCEL to exit											
	<u>Yes</u> <u>N</u> o Cancel										

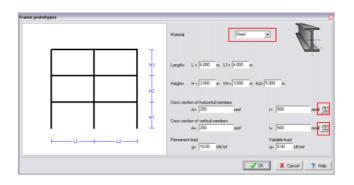
Click [No] if you want to edit the prototype. Click [Yes] if you want to make a new file.

#### **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<sup>2</sup> and mm<sup>4</sup>

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

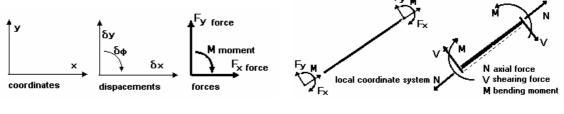
If you choose steel for structural material, then by clicking on the button you can choose the steel profile from a complete list, and the cross section area and moment of inertia are set.

Frame prototypes	
	Malesial Titled T
ю	Length: L = [5.000 n L1+ [5.000 n
H2	Height: H = 3000 n H1= 3000 n H2= 5000 n
н1	Closs section of hosizontal members A+ 250 mm <sup>2</sup> I+ 500 mm <sup>4</sup>
	Coss section of vertical members A= 250 nmF I= 500 nmf
H	Permanent load Valiable load g= 10.00 KN/NF q= 5.00 KN/NF
	VIC X Cancel Y Help

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.

X         FCFA         X           Y         FCFA         X           Y         FCFA         X           Y         FCFA         X           Y         FCFA         X           X         FCA         X	IPE 120 IPE 140 IPE 160 IPE 180 IPE 200 IPE 220 IPE 240	mm 80 100 120 140 160 180 200 220	mm 46 55 64 73 82 91 100	mm 38 41 44 4.7 5.0 5.3	mm 52 57 63 69 74	mm 5.0 7.0 7.0 7.0 9.0	cm <sup>2</sup> 7.64 10.32 13.21 16.43	Kg/m 6.00 8.10 10.4	cm <sup>4</sup> 80.14 171.0 317.8	cm <sup>2</sup> 20.03 34.20 52.96	cm <sup>2</sup> 23.22 39.41 60.73	cm 3.24 4.07 4.90	cm <sup>2</sup> 3.57 5.08 6.31	cm <sup>4</sup> 8.49 15.92	cm <sup>2</sup> 3.69 5.79	cm <sup>3</sup> 5.82 9.15	cm 1.05 1.24	cm² 5.12 6.73	cm4 0.698 1.20	cm <sup>6</sup> 118.0 351.4
Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0           Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0         Image: Type 0	IPE 100 IPE 120 IPE 140 IPE 160 IPE 180 IPE 200 IPE 220 IPE 240	100 120 140 160 180 200	55 64 73 82 91	41 44 4.7 5.0	57 6.3 6.9 7.4	7.0 7.0 7.0	10.32 13.21	8.10 10.4	171.0	34.20	39.41	4.07	5.08	15.92	5.79	9.15	1.24	6.73		
Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V           Image: Term V         Image: Term V         Image: Term V	IPE 120 IPE 140 IPE 160 IPE 180 IPE 200 IPE 220 IPE 240	120 140 160 180 200	64 73 82 91	4.4 4.7 5.0	6.3 6.9 7.4	7.0 7.0	13.21	10.4											1.20	351.4
IF HE         I         I           IF HE A (JPR1) Euronom 53.62         I         I           IF HE A (JPR1) Euronom 53.62         I         I           IF HE M (JPR) Euronom 53.62         I         I           IF HE M (JPR0) Euronom 53.62         I         I           IF HE Function 53.62         I         I           IF HD         I         I      IF HD	IPE 140 IPE 160 IPE 180 IPE 200 IPE 220 IPE 240	140 160 180 200	73 82 91	4.7 5.0	6.9 7.4	7.0			317.8	52.96	60.73	4.90	6.31							
I HE & JIPB1   Euronom 53.62         I           I HE & JIPB1   Euronom 53.62         I           I HE & JIPB1   Euronom 53.62         I           I HE # (JIPB1)   Euronom 53.62         I           I HD         I           I HP         I           I IN         X	IPE 160 IPE 180 IPE 200 IPE 220 IPE 240	160 180 200	82 91	5.0	7.4		16.43						N.91	27.67	0.65	13.50	1.45	8.57	1.74	889.6
THE # ()FP()         Eurocom 3242         I           THE # ()FP()         Eurocom 3242         I           THE Eurocom 3242         I         I           THO         I         I           THO ASTM AGA GM         I         I           THP         I         I           THP         I         I           THO	IPE 180 IPE 200 IPE 220 IPE 240	180 200	91			9.0		12.9	541.2	77.32	88.34	5.74	7.65	44.92	12.31	19.25	1.65	10.62	2.45	1 981
I HE M (#P6) (Euronom 33.62         I           HE Function 33.62         I           I HE Function 33.62         I           I HD          I	IPE 200 IPE 220 IPE 240	200		5.3			20.09	15.8	869.3	108.7	123.9	6.58	9.66	68.31	16.66	28.10	1.84	12.84	3.60	3 959
X HL         X H           X HD         X H	IPE 220 IPE 240		100		8.0	9.0	23.95	18.8	1 317	146.3	166.4	7.42	11.25	100.9	22.16	34.60	2.05	15.32	4.79	7 431
IF HD         I           IF HD         I           IF HD ASTM AG/A GM         I           IF HP         I           IF HP         I           IF HP         I           IF HP         I	IPE 240	220		5.6	8.5	12.0	28.48	22.4	1 943	194.3	220.6	8.26	14.00	142.4	28.47	44.61	2.24	17.99	6.98	12 900
T HD ASTM A6/A GM			110	5.9	92	12.0	33.37	26.2	2772	252.0	285.4	9.11	15.88	204.9	37.25	58.11	2.48	21.30	9.07	22 672
	IFE 270	240	120	62	9.8	15.0	39.12	30.7	3 892	324.3	366.6	9.97	19.15	283.6	47.27	73.92	2.69	24.83	12.88	37 391
I IPN I I		270	135	6.6	10.2	15.0	45.94	36.1	5 790	428.9	484.0	11.23	22.13	419.9	62.20	96.95	3.02	28.97	15.94	70 578
T IPN	IPE 300	300	150	7.1	10.7	15.0	53.91	42.2	8 356	557.1	628.4	12.46	25.68	603.8	80.50	125.2	3.35	33.67	20.12	125 934
I W (ASTM)	IPE 330	330	160	7.5	11.5	18.0	62.61	49.1	11 770	713.1	804.3	13.71	30.81	788.1	98.52	153.7	3.55	38.71	28.15	199 097
	IFE 360	360	170	8.0	12.7	18.0	72.73	57.1	16 270	903.6	1 019	14.96	10.14	1.045	1000	101.1	170	10.00	17.11	
C U	IPE 400	400	180	8.6	13.5	21.0	84.46	66.3	23 130	1 156	1 307	16.55			-	12	20			••
L hot roled	IPE 450	450	190	9.4	14.6	21.0	98.82	77.6	33 740	1 500	1 702	18.48			1	_	-			
cold formed	IPE 500	500	200	10.2	16.0	21.0	115.5	90.7	49 200	1 920	2194	20.43				)	15	_		
C cold formed	IPE 550	550	210	11.1	17.2	24.0	134.4	106.0	67 120	2 4 4 1	2 787	22.35								
Q D-10.2-33.7 mm	IPE 600	600	220	12.0	19.0	24.0	156.0	122.0	92 080	3 069	3 512	24.30	IPE	240			-6.2			
b														240			1000			
tri T																		J		- 1
															1.1			-240		
tw																				
yy h																				
																				- 1
																	1.			
1																	-			
															_		-			
4																2	2			- 1
													10							_
how steel section drawing																				V 08

### 4. Coordinate system



Global coordinate system

Local coordinate system (member end forces)

append

post cancel

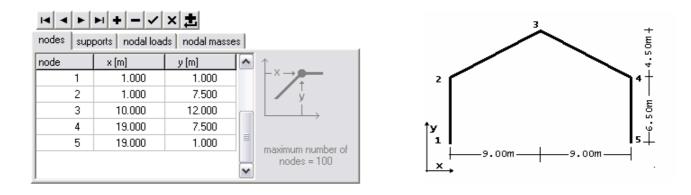
### 5. Units

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/sec<sup>2</sup>. Modulus of elasticity, in GPa (kN/mm<sup>2</sup>). Specific weight, in kN/m<sup>3</sup>.



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.

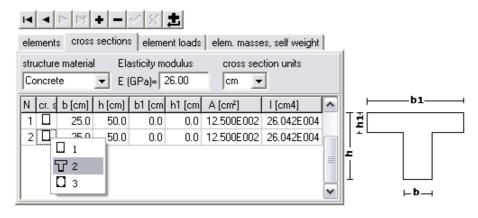
- Select structure material (general, concrete, steel, timber). You give the modulus of elasticity in GPa (kN/mm<sup>2</sup>), (in case of choosing material concrete, steel or timber an average modulus of elasticity is automatically shown)
- Select the units for the cross section dimensions (mm, cm, m).

elements cross sections element loads elem. masses, self weight											
structur Steel	structure material Elasticity modulus cross section units Steel ▼ E (GPa)= 200.00 mm ▼ ①										
N	b [mm]	h [mm]	A [mm²]	l [mm4]	^						
1	250	500	12.500E004	26.042E008							
2	250	500	12.500E004	26.042E008	≡						
					$\overline{\mathbf{v}}$						

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.

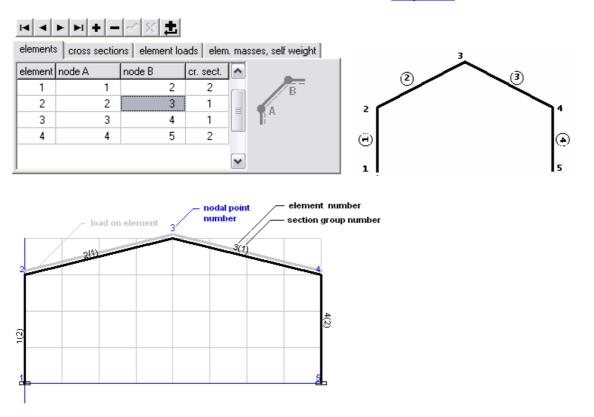
If you select concrete as material, by clicking at the second column (<u>sect.</u>), you can select orthogonal or T type cross section, and you give the corresponding dimensions (**b**), (**h**), (**b**1), (**h**1) as in the drawing . For non rectangular cross section give the area and the moment of 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<sup>2</sup>, and E-002 means 1/10<sup>2</sup>. 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.

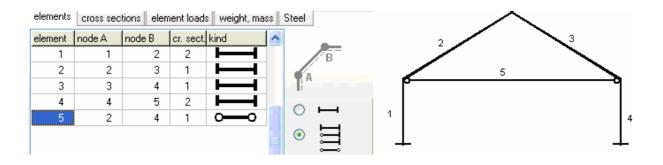


#### 9. Elements with hinges

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



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



### 10. Supports

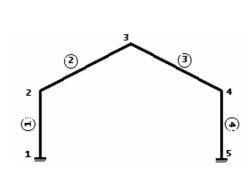
With click at the second (support) column the support choices are shown.

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.

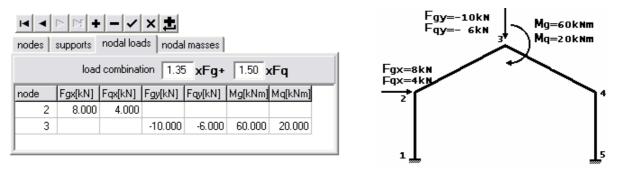
•	$\triangleright$		1 × ±			
nodes	sup	ports nodal	loads 🛛 nodal r	nasses		
node		support	ux[mm]	uy[mm]	ur[rad]	^
	1	TTTT	0.00000	0.00000	0.00000	
	5	Z	₃ → uy=0 (ux=,	.ur=)	0.00000	
		<	р их=0 (uy=,	ur=)		
		4	∆ ux=0,uy=0	(ur=)		
		Π	‴ ux=0,uy=0	l,ur=0		≡
			🛢 ux=0,uy=0	l,ur=0		
		δ	⊨ ux=,uy=,u	r=		
						~



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  $C_{gx}F_{g+}C_{qx}F_{q}$ , 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).



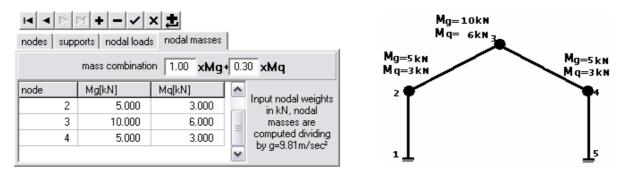
**Fgx,Fqx**, nodal forces, in kN, in x-x direction, for permanent and live loads, positive from left to right. **Fgy,Fqy**, nodal forces, in kN, in y-y direction, for permanent and live loads, positive from down to up. **Mg, Mq**, nodal moments, in kNm, for permanent and live loads, positive clockwise.

### 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<sup>2</sup>.

The masses in the dynamic analysis are taken as  $C_{gxMg+CqxMq}$ . Default values are  $C_{g=1.00}$  and  $C_{q=0.30}$  (Eurocode 1, EN 1991 1-1:2003).

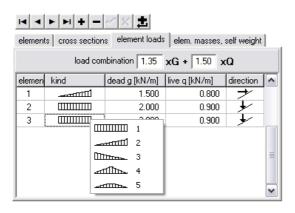


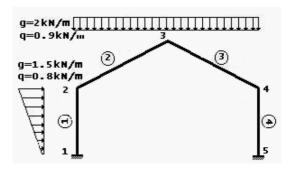
### 13. Element distributed loads

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 (<u>kind</u>) (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  $C_{gx}F_{g+}C_{qx}F_{q}$ , 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).





### 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<sup>2</sup>.

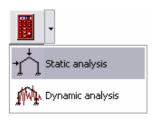
The mass in the dynamic analysis are taken as  $C_{gxMg+CqxMq}$ . Default values are  $C_{g}=1.00$  and  $C_{q}=0.30$  (Eurocode 1, EN 1991 1-1:2003)

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.

	⊠ + − ✓												
elements cross sections element loads elem. masses, self weight													
Weight density kN/m3 25.00 include self weight in loads and masses													
Mass com	Mass combination 1.00 xMg + 0.30 xMq												
element	Gg[kN/m]	Gq [kN/m]	Input line weights										
3	3.000	1.000	kN/m, element line										
4	3.000	1.000	masses are computed dividing by										
			g=9.81m/sec <sup>2</sup>										

### 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.

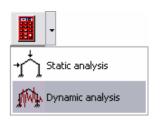
The support conditions are applied at the stiffness matrix as well as the load matrix. The final system of equations is solved with the Gauss elimination method, and the nodal displacements are obtained.

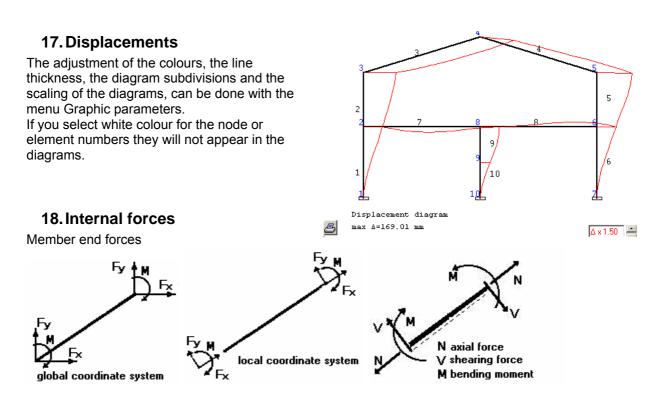
From the nodal displacements the element end displacements are obtained. These nodal displacements multiplied by the element stiffness matrix have as a result the internal forces at the element ends.

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.



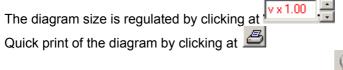


### 19. Diagrams

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

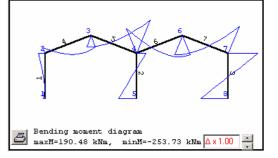


Zoom the diagram (x1, x2, x3, x4 times) by clicking at /

### 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]
▲ 14/18 2				1	2.02097	0.49481
				2	4.36340	0.22918
				з	4.59963	0.21741
(nput data				4	20.35550	0.04913
Results of static analysis	•			5	24.93403	0.04011
Results of dynamic analysis	Þ	Natural frequencies	]	6	37.34861	0.02677
	_	•		7	54.38084	0.01839
Preview Report		Mode shapes		8	59.47920	0.01681
Matrices	•1		1	9	77.59237	0.01289
Graphics preview				10	94.43947	0.01059
al apriles preview	·			11	96.52521	0.01036
Report setup				12	110.57478	0.00904
				13	127.48753	0.00784
Graphic parameters				14	165.85268	0.00603
				15	175.34224	0.00570



preferable.

22.1

### 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

Design parameters for reinforced concrete

In page Concrete you define parameters for the reinforced design.

### 22. Designing elements from reinforced concrete according to Eurocode 2

structure material

×

25.0

30.0

R.Concrete

1 🔽

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. To select concrete and reinforcing stee	l class, clic	k <mark>لا</mark>							
The concrete and reinforcing steel classes are adjusted according to the selected National Annex.	NA - National A Partial safety fac			Eurocode El γG=1.35 γ	Ν Q=1.50 ψ2=0	.30 💌			
You can change strength properties for the concrete and reinforcing steel from	elements cross sections element loads elem. masses, self weight R.Concrete								
Design/Materials/Concrete or Design/Materials/Reinforcing steel.	Concrete-Steel class     C25/30 - B500C       Partial factors for materials     yc= 1.50, ys= 1.15								
2. Partial factors for materials. This is defined according to National	Concrete cov Rebar diamet		Cnom= 3 Ø 2		d diameter Ø	i 🔽			
annex, usual values:	Reset element design data								
γc = 1.50, γs = 1.15.	Elm.	L[m]	Phi[mm]	Lcy[m]	Lcz[m]	Design			
3. Concrete Cnom, cover in mm.	1	4.600	20	6.200	6.200	1			
	2	8.400	20	12.400	12.400	1			
4. Preferable rebar diameter. If you check <i>fixed</i> the selected diameter will be used. If not, some optimum	3	4.600	20	6.200	6.200	1			

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

15.0

0.0

cross section units

31.750E002

18.000E002

Elasticity modulus

E (GPa)= 26.000

120.0

0.0

N cr. se b [cm] h [cm] b1 [cm] h1 [cm] A [cm<sup>2</sup>]

70.0

60.0

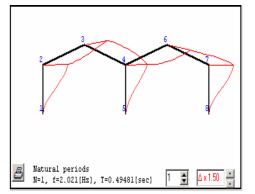
5. For every element you can defile:

If you check fixed the selected diameter will be used. If not, some optimum diameter will be used, around the

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.



[ [cm4]

13.353E005

54.000E004

#### 22.2 Reinforced concrete design according to Eurocode 2

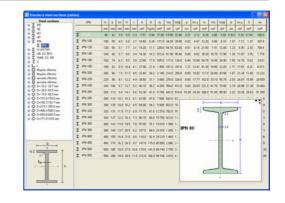
Click Reinforced Concrete Design. All the marked with Design = 1 elements will be verified according to Eurocode 2, §6, for axial force, shear and bending moment in ultimate limit state. The design for reinforcement is performed for mid span, left end and right end of each element. The vertical elements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

elem	ents	cross sectio	ns element load	ls 🛛 elem. masses, s	elf weight Steel	
struc	ture	material E	lasticity modulus	cross section	units	
Stee	el	✓ E	(GPa)= 210.0	00 mm 🗸	I	
N		cr. sect.		A [mm²]	l [mm4]	^
	1	I	IPE 500	11.550E003	48.200E007	
	2	I	IPE 600	15.600E003	92.080E007	
						~



Eurocode EN

γG=1.35 γQ=1.50 ψ2=0.30

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

NA - National Annex

Partial safety factors for actions

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.

el from Design/Materials/Structural steel.	
elements cross sections element loads elem. masses, self weight Steel	
Structural steel S 355 fy=355N/mm² fu=510N 🗸	
Partial factors γ <sub>M0</sub> = 1.00 🖨 γ <sub>M1</sub> = 1.00 🖨 γ <sub>M2</sub> = 1.25 🖨	
Reset element design data	

Elm.	L[m]	Lcy[m]	Lcz[m]	Lt[m]	Design
1	4.600	8.600	4.600	4.600	1
2	8.400	8.400	2.100	2.100	1
3	4.600	8.600	4.600	4.600	0

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

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.

#### 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.

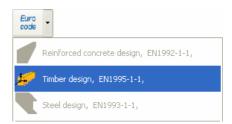
Euro code	-
	Reinforced concrete design,EN1992-1-1,
	Timber design, EN1995-1-1,
	Steel design, EN1993-1-1,

elements	cross section	s element loa	ds 🛛 elem. masses, :	self weight Timber	>
structure i	material El	asticity modulus	cross section	n units	
Timber	🛩 E	(GPa)= 10.00	)0 cm 🗸		
N	b (cm)	h [cm]	A [cm²]	l [cm4]	^
1	12.0	12.0	14.400E001	17.280E002	
2	12.0	15.0	18.000E001	33.750E002	
3	6.0	10.0	60.000E000	50.000E001	
					~

Standard of strength class	
<ul> <li>EN 338:1997 Solid wood, EN 1194 Glulam</li> <li>EN 338:2003 Solid wood, EN 1194:2000 Glulam</li> <li>EN 338:2009 Solid wood, EN 1194:2000 Glulam</li> <li>South African pine</li> <li>User-1</li> <li>User-2</li> </ul>	d d d d d d d d d d d d d d d d d d d

5 (Eq. 6	5.13a).
----------	---------

	s sections eleme	elem	masses self weir	ht Timber	
Timber class		_	<=16.0N/mm², ftc	<b>3</b> 1.	2 🗸
Service class			moisture content		~
Load duration cl	asses		Long-te	m	~
Material factors		Timber 1	.30	Steel 1	.10
Reset elemer	it design data				
Elm.	L[m]	Lcy[m]	Lcz[m]	Design	^
1	1.900	1.900	1.900	1	۳
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	~
<				3	



#### 24.3 Design notes

National Annex.

Design/Materials.

defined by the user.

material properties.

Click Closed.

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  $10^4$ .

The materials for concrete, reinforcing steel, structural

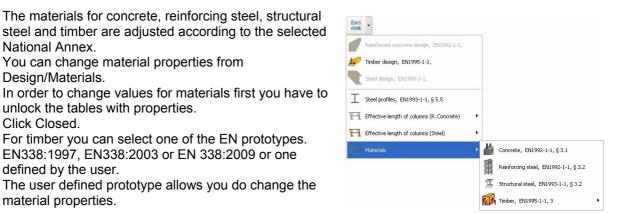
For timber you can select one of the EN prototypes.

The user defined prototype allows you do change the

EN338:1997, EN338:2003 or EN 338:2009 or one

You can change material properties from

unlock the tables with properties.



structure material

Timber

N

### 25. Materials for Reinforcing Concrete, Structural Steel and Timber

einforcing steel Class	fyk [MPa]	ftk,c [MPa]	Es [GPa]	euk [%]	L [m]
\$220	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
\$500s	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
characteristic yield strength,		n, Es: modulus of elasti	city, euk: maximum stra		≜ Print <b>?</b> He

Class	fck [MPa]	fck,c [MPa]	fctm [MPa]	fctk0.05 [MPa]	fctm0.95 [MPa]	fct,fl [MPa]	fvck [MPa]	Ec [GPa]	Gc [GPa]	
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	1.90	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

Class	ID	fink [MPa]	ft0k [MPa]	ft90k [MPa]	fcOk [MPa]	fc90k [MPa]	fvk [MPa]	E0m [MPa]	E05 [MP4]	E90m [MPa]	Gm [MPa]	pk [Kgim']
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	890	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	890	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
GL29h	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	3.90	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

N 10025-2 N 10025-2 N 10025-2 N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-4	225 275 395 440 275 395 420 460 275	360 430 510 550 390 490 520 540	215 255 335 410 255 335 335 390	360 410 550 370 470 520
N 10025-2 N 10025-2 N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-4	355 440 275 355 420 460	510 550 390 490 520	335 410 255 335 390	470 550 370 470
N 10025-2 N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-4	440 275 355 420 460	550 390 490 520	410 255 335 390	550 370 470
N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-3 N 10025-4	275 355 420 460	390 490 520	255 335 390	370 470
N 10025-3 N 10025-3 N 10025-3 N 10025-4	355 420 460	490	335	470
N 10025-3 N 10025-3 N 10025-4	420	520	390	
N 10025-3 N 10025-4	460			520
N 10025-4		540		
	0.00		430	540
	275	370	255	360
N 10025-4	365	470	335	450
N 10025-4	420	520	390	500
N 10025-4	460	540	430	530
N 10025-5	235	360	215	340
N 10025-5	355	510	335	490
N 10025-6	460	570	440	550
N 10210-1	235	360	215	340
N 10210-1	275	430	255	410
N 10210-1	355	510	335	490
N 10210-1	275	390	255	370
N 10210-1	355	490	335	470
N 10210-1	420	540	390	520
N 10210-1	460	560	430	550
	N 10025-5 N 10025-5 N 10025-6 N 10210-1 N 10210-1 N 10210-1 N 10210-1 N 10210-1 N 10210-1 N 10210-1	N 100255 225 N 100255 355 N 100256 460 N 102101 225 N 102101 255 N 102101 255 N 102101 255 N 102101 355 N 102101 420	N 100255         225         360           N 100256         255         510           N 100256         460         570           N 100256         265         360           N 102101         275         430           N 102101         275         510           N 102101         275         510           N 102101         275         510           N 102101         275         490           N 102101         365         490           N 102101         420         540	N 100255         235         360         215           N 100255         395         510         335           N 100256         460         570         460           N 102151         225         430         225           N 102101         275         430         225           N 102101         325         510         325           N 102101         325         510         325           N 102101         395         430         255           N 102101         395         430         335           N 102101         420         540         330

cross section units

I [cm4]

53.240E-002

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

22.0

A [cm²]

22.0 13.200E001

13.200E001

Elasticity modulus

🖌 E (GPa)= 10.000

b [cm] h [cm]

6.0

6.0

### 26. Frame prototypes

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

FF						
Frame prototypes  A	1 🕅 В1	<b>—</b> C1	D1	Ēι	—— F1	<sub>G1</sub>
A	2 🖽 в2	C2		FT E2	F2	<sub>G2</sub>
	з 🖽 вз	∰ III III III		ĒΒ	∕∧ <sub>F3</sub>	<sub>G3</sub>

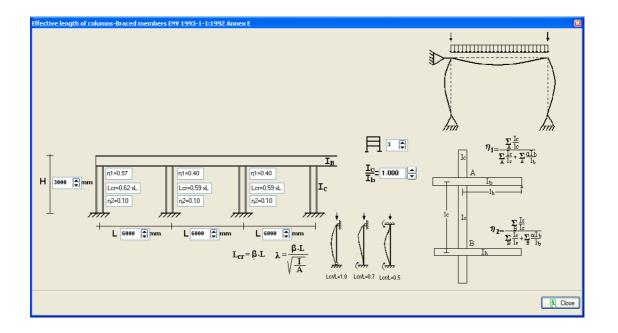
#### 26.1 Effective lengths for columns

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.

Euro -		
Reinforced concrete design, EN1992-1-1,		
🖅 Timber design, EN1995-1-1,		
Steel design, EN1993-1-1,		
T Steel profiles, EN1993-1-1, § 5.5		
Effective length of columns (R.Concrete)	• 1	Effective length (R.Concrete), EN1992-1-1, § 5.8.3.2
Effective length of columns (Steel)	٠ff	Effective length-Braced members
Materials	・歴	Effective length-Unbraced members

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 =  $0.62 \times L$ ,  $0.59 \times L$ 



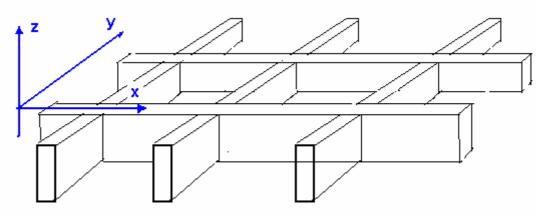
### 27.2-D Grillages

#### 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.

1 -		÷
2	77777 dz=0, drx=0, dry=0	- 1
3	📓 dz=0, drxt=0, dry=0	- 1
4	🌶 dz=0, drx=0	
5	🛥 dz=0, dry=0	
		the second se

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.

enter report

### 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.

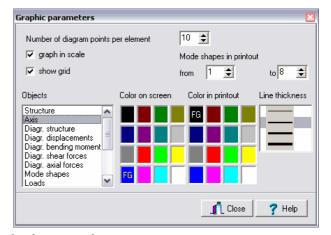
#### **Report example**

FRAME-31	27_11	2008		REPO	RI				Fg. 1
FRME-3	31-27	11 2008					-		
Diagram	of i	nternal fo:	cces K, V,	N, and displa	acements d,	of elemen	t 1		
n	x/1	×[m]	M[kNn]	V(kN)	N[kN]	dx[mm]	dy[mm]	d[nn]	
0	0.000		93.47	74.56	-177.76	0.000	0.000	0.000	
1	0.100		71.10	74.56	-173.81	-0.007	-0.002	0.008	
2	0.200		48.73	74.56	-169.86	-0.027	-0.004	0.027	
3	0.300		26.36	74.56	-165.91	-0.055	-0.006	0.056	
4	0.400		3.99	74.56	-161.96	-0.088	-0.008	0.088	
5	0.500		-18.37	74.56	-158.01	-0.121	-0.010	0.122	
6	0.600		-40.74	74.56	-154.07	-0.152	-0.011	0.152	
7	0.700		-63.11	74.56	-150.12	-0.175	-0.013	0.175	
8	0.800		-85.48	74.56	-146.17	-0.187	-0.015	0.188	
9	0.900		-107.84	74.56	-142.22	-0.184	-0.017	0.185	
10	1.000		-130.21	74.56	-138.27	-0.163	-0.019	0.164	
	P VAIN	tes for ele		100 01 10					
naxH- naxV-		93.47 kNm 74.56 kN,		-130.21 kNm 74.56 kN					
naxv=		138.27 kN.		-177.76 kN					
naxd=		0.188 nm	manna-	-1//./0 KB					
Diagram	of i	nternal fo	cces H, V,	N, and displ	acements d,	of elemen	t 2		
n	x/1	×[m]	M[kNm]	V(kN)	N[kN]	dx[mm]	dy[mm]	d[mm]	
0	0.000		-130.21	-116.06	-105.87	-0.163	-0.019	0.164	
1	0.100		-85.23	-102.14	-104.55	-0.150	-0.080	0.170	
2	0.200		-45.99	-88.21	-103.24	-0.130	-0.168	0.212	
3	0.300		-12.49	-74.29	-101.92	-0.106	-0.271	0.290	
4	0.400		15.27	-60.36	-100.60	-0.080	-0.378	0.386	
5	0.500		37.29	-46.44	-99.29	-0.057	-0.480	0.483	
6	0.600		53.57	-32.52	-97.97	-0.036	-0.570	0.571	
7	0.700		64.10	-18.59	-96.65	-0.019	-0.644	0.644	
8	0.800		68.90	-4.67	-95.34	-0.007	-0.697	0.697	
9 10	0.900		67.95	9.25 23.18	-94.02 -92.71	-0.001	-0.729	0.729	
		4.12 les for ele		23.18	-92.71	0.000	-0.740	0.740	
maxH=		68.90 kNm		-130.21 kNm					
naxV=		23.18 kN,		-116.06 kN					
naxV=		-92.71 kN,		-105.87 kN					
naxd=		0.740 mm		20010/ 88					

### 29. Graphic parameters

Parameters for graphics on the screen or in the report.

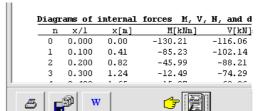
Number of diagram points per elements. The number of subdivision points for each element, at which the diagrams are computed and plotted. For the graphic appearance you select the graphic object at the left list box and then you click and select from the colour boxes the colour you prefer, and the line 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 <u>graph in scale</u> 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.



WORD

sent to

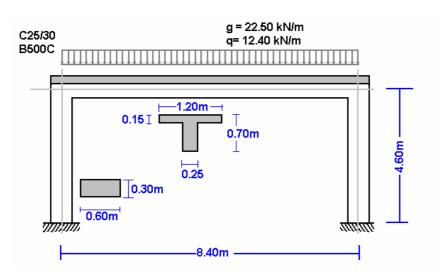
print

save to file

### 30. Examples

### 30.1 Example 1

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



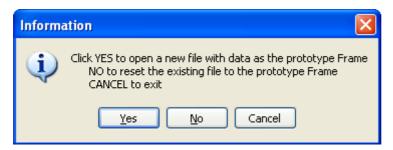
#### Select a frame from File/Frame prototypes:

FTT ·							
Frame prototypes	• A1	— В1	<b>—</b> C1	D1	E1	—	<sub>G1</sub>
	A2	<b>ГП</b> В2	<b>E</b> C2		FT E2	F2	G2
	ПП АЗ	ВЗ	⊞ द3	<b>□</b> D3	ĒЗ	∕_ F3	<del>~~</del> <sub>сз</sub>

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

Frame prototypes	
	Material R.Concrete V H
	Lengths L = 6.000 m
L	Heights H = <u>3.000</u> m
	Cross section of horizontal members
	B= 200 mm H= 500 mm
	Cross section of vertical members
	B= 300 mm H= 300 mm
	Permanent load Variable load
	g= 8.50 kN/m q= 6.50 kN/m
<u>/</u>	VOK X Cancel Rep

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) yG=1.35, yQ=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	~

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	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	m	0.00000	0.00000	0.00000	
4	77777	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 <b>xFg+</b> 1.50 <b>xFq</b>									
node	Fgx[kN]	Fqx[kN]	Fgy[kN]	Fay[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

struc	cture m	aterial	Elasticity n	nodulus	cross section units		
R.Concrete 🗸 E (GPa)= 26.00 cm 🗸							
N	cr. se	b [cm]	h [cm]	b1 [cm]	h1 [cm]	A [cm²]	l [cm4]
1	Г	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 <b>xG +</b> 1.50 <b>xQ</b>							
element	element kind dead g [kN/m] live q [kN/m] direction						
2	2 IIIIIIII 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<sup>3</sup>, steel 78.50 kN/m<sup>3</sup>, timber 9kN/m<sup>3</sup>).

elements cros	s sections	element load	s elem. mas:	ses, self we	eight	R.Concrete
Weight density	kN/m3		ide self weight s and masses	in 🔽		
Mass combinati	on	1.00 <b>xMg</b>	+ 0.30 ,	сMq		
element	Gg [kN/m	i]	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.

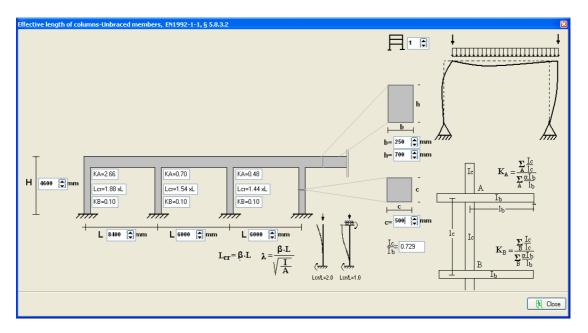
The rebar diameter is used as the optimum desired by the program. If you check next to the rebar diameter then is only this diameter selected by the program. For every element you may specify in the column Phi [mm] the desired rebar diameter eg. 20 mm for columns and 16 mm for beams. The buckling lengths 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.

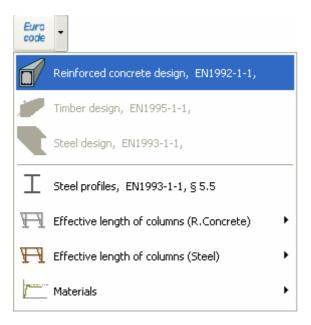
elements Cross sections element loads elem. masses, self weight R.Concrete							
Concrete-Steel	class	C25/3	30 - B500C	4			
Partial factors fo	or materials	ye= 1.	yc= 1.50, ys= 1.15 🛛 🗸				
Concrete cover	[mm]	Cnom=	30 🗲				
Rebar diameter	[mm]	Ø	20 🔽 fix	ed diameter Ø	✓		
Reset elemer	nt design data	)					
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 =  $1.88 \times L = 1.88 \times 4.60 = 8.65 \text{ m}$ 

Euro code	•		
	Reinforced concrete design,EN1992-1-1,		
	Timber design, EN1995-1-1,		
T	Steel design,EN1993-1-1,		
I	Steel profiles, EN1993-1-1, § 5.5		
T ₩	Steel profiles, EN1993-1-1, § 5.5 Effective length of columns (R.Concrete)	2	Effective length (R.Concrete), EN1992-1-1, § 5.8.3.2
₽₹ V=₹		1 П	Effective length (R.Concrete), EN1992-1-1, § 5.8.3.2 Effective length-Braced members



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.

				X	
		l, Reinforced concrete desig			
		<ol> <li>Reinforced concrete designations</li> <li>Reinforced concrete designation</li> </ol>			
	element	o, Reiniulueu Concrete desig	n not betrouged		
Nodal	points				
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			
Suppor					
Node	kind	ux[mm] uy[mm]	ur[rad]		
Node 1	kind fixed ux=	y=ur=0	ur[rad]		
Node	kind	y=ur=0	ur[rad]		
Node 1 4	kind fixed ux=1 fixed ux=1	y=ur=0	ur[rad]		
Node 1 4 Mater:	kind fixed ux=1 fixed ux=1 ials	y=ur=0 y=ur=0	ur[rad]		
Node 1 4 Mater: Mater:	kind fixed ux= fixed ux= ials ial : R.Conc:	y=ur=0 y=ur=0 ete, E= 26.000 [GPa]	ur[rad]		
Node 1 4 Mater: Mater:	kind fixed ux= fixed ux= ials ial : R.Conc:	y=ur=0 y=ur=0	ur[rad]		
Node 1 4 Mater: Nater: Veigh:	kind fixed ux=1 fixed ux=1 ials ial : R.Conc: t density : 1	y=ur=0 y=ur=0 ete, E= 26.000 [GPa]			
Node 1 4 Mater: Nater: Veigh:	kind fixed ux=1 fixed ux=1 ials ial : R.Conc: t density : 1	y=ur=0 y=ur=0 ete, E= 26.000 [GPa] = 25.000 [kN/m*]			
Node 1 4 Mater: Nater: Veigh The e	kind fixed ux=0 fixed ux=0 ials ial : R.Conc: t density : p lement self 0	y=ux=0 y=ux=0 ete, E= 26.000 [G₽a] = 25.000 [kW/m³] eight is included in loads a			
Node 1 4 Mater: Nater: Veigh The e. Elemen	kind fixed ux=1 fixed ux=1 ials ial : R.Conc: t density : p lement self 1 nt cross sect	y=ux=0 y=ux=0 ete, E= 26.000 [G₽a] = 25.000 [kW/m³] eight is included in loads a	nd masses		

Click

for complete formatted report.

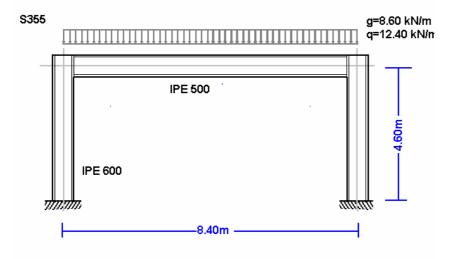
From the report preview you can print all or part (from page to page) of the report can export to PDF or Word files



exBETON	01			
				Pg
<u>1-Fini</u>	te element	model (FEM)		
	2	2(1)	3	
			N	
	2			
	4			
	and the second s			
Nodal p				
Node	x [m]	<u>y[m]</u> 0.000		
	x [m] 0.000 0.000	0.000 4.600		
Node 1 2 3	x [m] 0.000 0.000 8.400	0.000 4.600 4.600		
Node 1 2	x [m] 0.000 0.000	0.000 4.600		
Node 1 2 3 4	x [m] 0.000 0.000 8.400 8.400	0.000 4.600 4.600		
Node 1 2 3 4 Support Node	x [m] 0.000 0.000 8.400 8.400 8.400	0.000 4.600 4.600 0.000 ux (mm)	uy[mm] ux[rad]	
Node 1 2 3 4 <b>Support</b> <u>Node</u> 1	x [m] 0.000 0.000 8.400 8.400 8.400 ***********************************	0.000 4.600 4.600 0.000 ux [mm]	uy[mm] ur[rad]	
Node 1 2 3 4 Support Node	x [m] 0.000 0.000 8.400 8.400 8.400	0.000 4.600 4.600 0.000 ux [mm]	uy[mm] ur[rad]	
Node           1           2           3           4           Support           Node           1           4	x [m] 0.000 0.000 8.400 8.400 8.400 kind fixed ux= fixed ux=	0.000 4.600 4.600 0.000 ux [mm]	uy[mm] ur[rad]	
Node 1 2 3 4 Support Node 1 4 Materia	x [m] 0.000 8.400 8.400 8.400 8.400 8.400 fixed ux= fixed ux=	0.000 4.600 4.600 0.000 ux [mm]		
Node 1 2 3 4 Support Node 1 4	x [m] 0.000 0.000 8.400 8.400 8.400 kind fixed ux= fixed ux=	0.000 4.600 4.600 0.000 ux [mm]	uy[mm] ur[rad]	

#### 30.2 Example 2

#### Steel frame 8.40 x 4.60 S355



#### Select a frame from File/Frame prototypes:

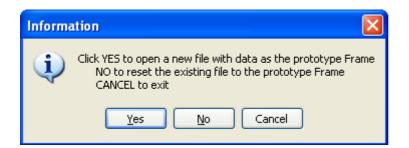
Frame prototypes	• 🗌 A1	⊟ в1	<b>—</b> C1	D1		— F1	<sub>G1</sub>
	A2	н В2	<b>E</b> C2		FT E2	F2	<sub>G2</sub>
	ПП АЗ	ВЗ	Ща		ĒЗ	∕\ <sub>F3</sub>	<sub>G3</sub>

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 $\square$ .

Frame prototypes		×
	Material Steel	
	Lengths L = 8.400 m Heights H = 4.600 m	·
	Heights H = 4.600 m Cross section of horizontal members A= 11550 mm <sup>2</sup>	IPE 500
	Cross section of vertical members A= 15600 mm²	IPE 600
	Permanent load g= 8.60 kN/m	Variable load q= 12.40 kN/m
		K X Cancel ? Help

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)  $\gamma$ 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	~

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 m	hasses
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	

n	ode	support	ux[mm]	uy[mm]	ur[rad]
	1	m	0.00000	0.00000	0.00000
	4	77777	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 com	bination 1.3	35 <b>xFg+</b>	1.50 <b>xF</b> c	1	
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 is 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 Elasticity modulus			cross section units		
Steel 🛛 🖌 E (GPa)= 210.		GPa)= 210.00	mm 🗸 🕺	<u> </u>	
N	cr. sect.		A [mm²]	l [mm4]	
				. []	
1	I	IPE 500	11.550E003	48.200E007	

• 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 el	ement loads elem. m	asses, self weight 🛛 SI	teel	
load combination 1.35 $\mathbf{xG}$ + 1.50 $\mathbf{xQ}$					
element	kind	dead g [kN/m]	live q [kN/m]	direction	
2		8.600	12.400	Y	

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<sup>3</sup>).

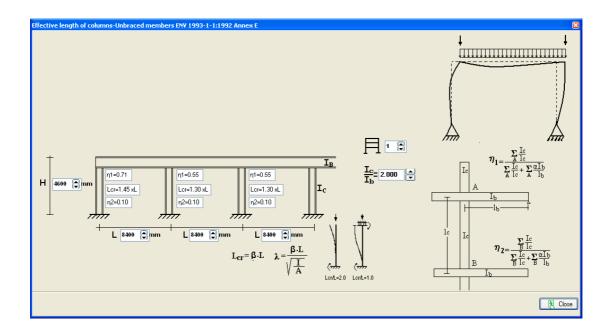
elements cro:	ss sections 🛛 element lo	ads elem. masses, self weight	Steel						
Weight density kN/m3 78.50 include self weight in loads and masses									
Mass combination 1.00 xMg + 0.30 xMq									
element	Gg [kN/m]	Gq[kN/m]							

• 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)  $\gamma M0 = 1.00$ ,  $\gamma M1 = 1.00$ ,  $\gamma M2 = 1.25$ . 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,

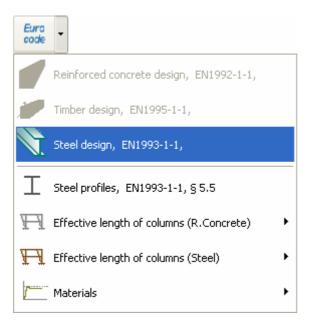
elements Cross sections element loads elem. masses, self weight Steel							
Structural steel	Structural steel S 355 fy=355N/mm² fu=510N 🗸						
Partial factors	Үмо=	1.00 🗲	γ <sub>M</sub> r≡ 1.00	🗧 үм2=	1.25 🚔		
Reset eleme	nt design data						
Elm.	L[m]	Lcy[m]	Lcz[m]	Lt[m]	Design		
1	4.600	6.67	4.600	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 =  $1.45 \times L = 1.45 \times 4.60 = 6.67 \text{ m}$ 

Euro code	•		
	Reinforced concrete design,EN1992-1-1,		
	Timber design,EN1995-1-1,		
J	Steel design,EN1993-1-1,		
Ι	Steel profiles, EN1993-1-1, § 5.5		
Ħ	Effective length of columns (R.Concrete)		
Ħ	Effective length of columns (Steel)	2	Effective length (Steel)
<u>/</u>	Materials	Ħ	Effective length-Braced members
		₩¥	Effective length-Unbraced members



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



Check if every element is verified in the design,

		·
	element:1, Steel design is OK	X
	element:2, Steel design is OK	
	element:3, Steel design not performed	
	carementoro, socca debaga noo personaed	
Nodal	points	
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	
Suppor	rts	
Node	kind ux[mm] uy[mm] ur[rad]	
1	fixed ux=uy=ur=0	
4	fixed ux=uy=ur=0	
Materi		
	ial : Steel, E= 210.000 [GPa]	
	t density : ρ= 78.500 [kN/m³]	
The el	lement self weight is included in loads and masses	
	nt cross sections	
	c. b[mm] h[mm] Ac[mm <sup>2</sup> ] Ic[mm4]	
1	1.15500E+004 4.82000E+008	
2	1.56000E+004 9.20800E+008	
- 20		
zt	🔊 🐨 👉 👰 🖢 🛋	Close ? H

Click

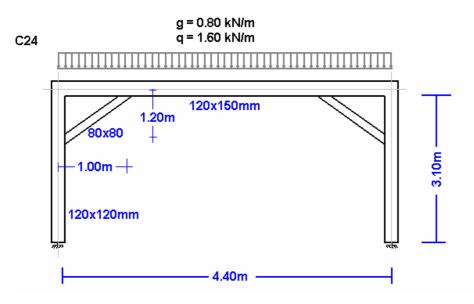
for complete formatted report.

From the report preview you can print all or part (from page to page) of the report can export to PDF or Word files 📴 👿 .

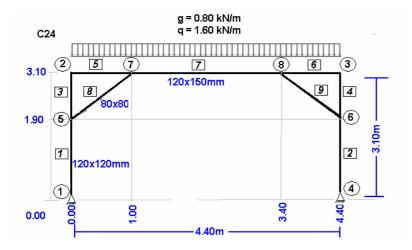
☆ Print preview	
Designing frame structures from Seel	
Modal points           Node         x [m]         y[m]           1         0.000         0.000           2         0.000         4.600           3         5.400         4.600           4         5.490         0.000	
Supports Node kind ux[nm] uy[nm] ur[rid] 1 fixed ux=uy=ur=0 4 fixed ux=uy=ur=0	
<u>Ma</u> terials Material : Steel, E= 210.000 [GDa] Geight density : p= 78.500 [kRM/m <sup>3</sup> ] The element self weight is included in loads and masses	
Element cross sections         δε[nm²]         Iε[nm4]           1         1.15500E+004         4.82000E+008	~
0% Page	

#### 30.3 Example 3

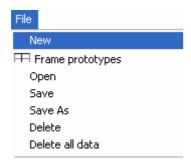
#### Timber structure 6.40 x 4.60 C24



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	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	$\bigtriangleup$	0.00000	0.00000	0.00000

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

nodes	supports nod	al loads 🛛 n	odal masses			
	load con	nbination	1.35 <b>xFg+</b>	1.50 <b>xF</b> (	1	
node	Fgx[kN]	Fqx[kN]	Fgy[kN]	Fay[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.

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

elements cross sections element loads

• 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 v	veight Timber
structure n	naterial Elas	ticity modulus	cross section unit	s
Timber	🔽 E (G	Pa)= 10.000	cm 🗸	
N	b [cm]	h [cm]	A [cm²]	l [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 ele	ement loads elem. m	asses, self weight 🛛 T	imber
	load combinati	on 1.35 <b>xG +</b> 1	50 xQ	
element	kind	dead g [kN/m]	live q [kN/m]	direction
5		0.800	1.600	<b></b>
6		0.800	1.600	<u>۲</u>
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<sup>3</sup>).
- 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)  $\gamma$ M=1.30 and for SLS (serviceability limit state)  $\gamma$ M=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).

elements cross :	sections element	loads ele	em. masses, sel	fweight Tim	ber
Timber class			C24, fmk=24.	0N/mm², ftok=	=14.0N/mm² 💌
Service class			Class 2, moistu	ure content<=;	20% 🔽
Load duration clas	ses			Long-term	*
Material factors			Timber 1.30		Steel 1.10
Reset element (	design data				
Elm.	L[m]	Lcy[m]	Lcz[m]	De	sign
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.

code	•	
	Reinforced concrete design,EN1992-1-1,	
*	Timber design, EN1995-1-1,	
	Steel design,EN1993-1-1,	
Т		
-	Steel profiles,EN1993-1-1,§ 5.5	
Ħ	Effective length of columns (R.Concrete)	٠
Ħ		• •

Check if every element is verified in the design.

		element:1, T	mber desi	gn is OK				
		element:2, T	mber desi	.gn not p	erformed			
		element:3, T						
		element:4, T						
		element:5, T						
		element:6, T						
		element:7, T						
		element:8, T:						
		element:9, T	mber desi	.gn not p	errormed			
Nodal	points							
Node	× [m]	Υ[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						
Suppor	te							
Node	kind	uxl	mm] uy	'[mm] u	r[rad]			
1		ux=uy=0	C 1996) - 199	400 - 70	10401 20			
4	pin	ux=uy=0						
	-1-							
Watowi								1
Materi	.415							
Materi		¢ [				(	R Close 7 H	elp

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