

**Table of contents**

<b>1. General .....</b>	<b>4</b>
<b>2. Structural Wizard.....</b>	<b>5</b>
<i>Structural Data for prototypes .....</i>	<i>5</i>
<b>3. Steel profiles.....</b>	<b>6</b>
<b>4. Coordinate system .....</b>	<b>6</b>
<b>5. Units .....</b>	<b>7</b>
<b>6. Nodal points.....</b>	<b>7</b>
<b>7. Element cross sections .....</b>	<b>7</b>
<b>8. Elements.....</b>	<b>8</b>
<b>9. Supports .....</b>	<b>9</b>
<b>10. Nodal loads.....</b>	<b>9</b>
<b>11. Nodal masses .....</b>	<b>9</b>
<b>12. Element distributed loads.....</b>	<b>10</b>
<b>13. Element distributed masses.....</b>	<b>10</b>
<b>14. Static analysis .....</b>	<b>11</b>
<b>15. Dynamic Analysis .....</b>	<b>11</b>
<b>16. Displacements .....</b>	<b>11</b>
<b>17. Internal forces.....</b>	<b>11</b>
<b>18. Diagrams .....</b>	<b>12</b>
<b>19. Natural frequencies .....</b>	<b>12</b>
<b>20. Mode shapes.....</b>	<b>12</b>
<b>21. Reports.....</b>	<b>13</b>
<i>Report example.....</i>	<i>13</i>

**22. Graphic parameters ..... 13**

**23. References..... 14**

**FRAME2Dexpress**, Version 11/08, User's guide. Copyright © RUNET®software & expert systems.

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 implied. 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: [support@runet-software.com](mailto: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.

### 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

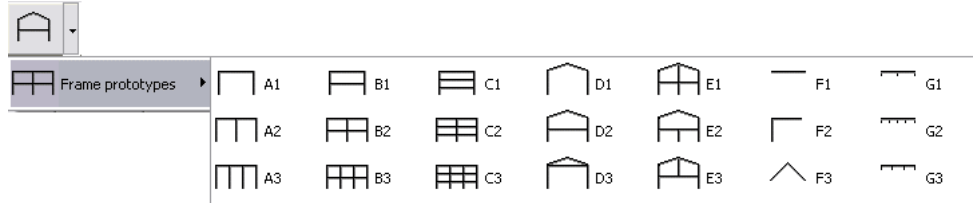
Metric units.

### **Input data steps**

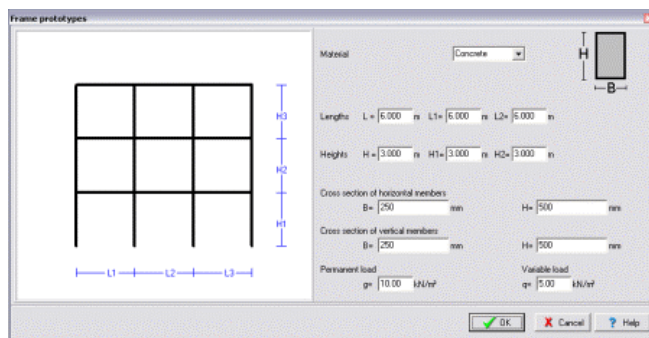
- Nodal coordinates
- Element cross sections
- Element data
- Supports
- Nodal loads and nodal masses
- Distributed element loads and distributed masses.

## 2. Structural Wizard

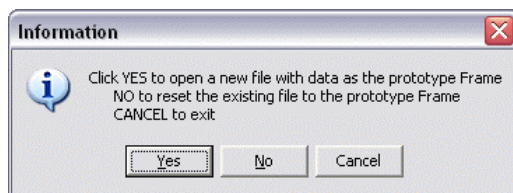
If you want to create a new structure with help from the structural wizard, go to [File/Frame Prototypes] and choose from a menu with 21 ready types of structures.



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.

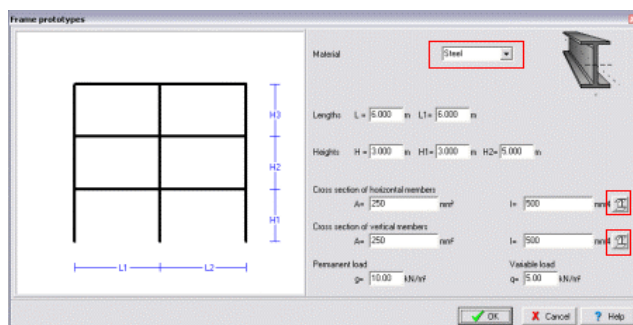
### Structural Data for prototypes

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.

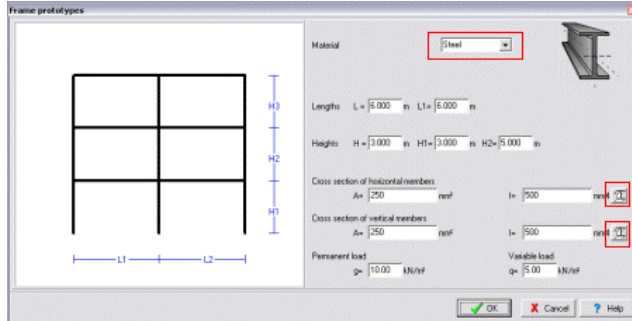
If you choose general material, you enter cross section area and moment of inertia in mm<sup>2</sup> and mm<sup>4</sup>

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.

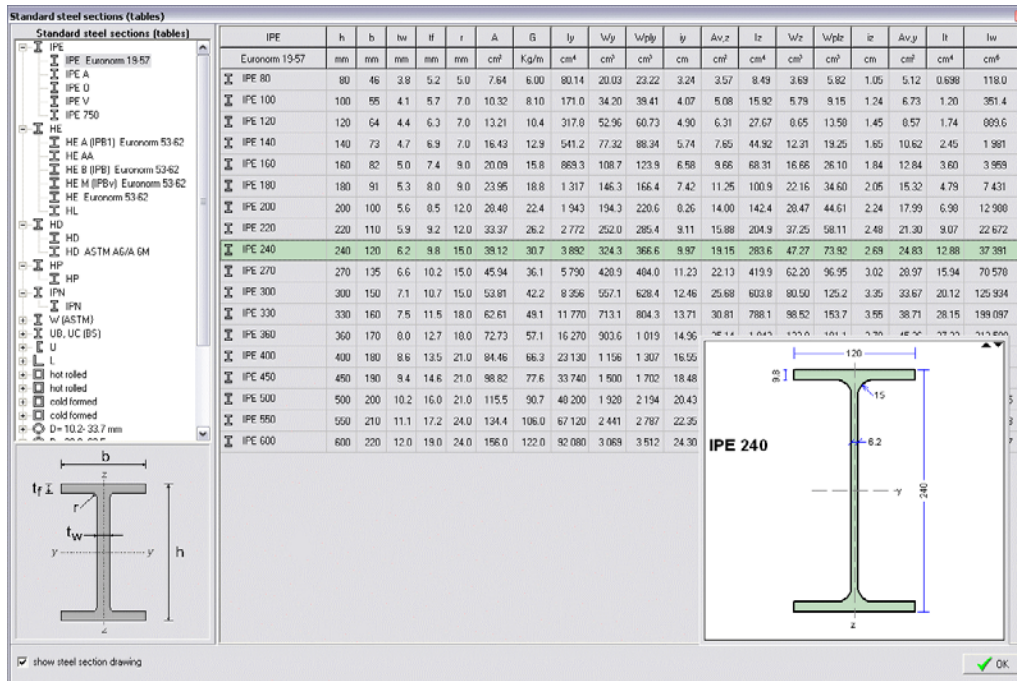


### 3. Steel profiles

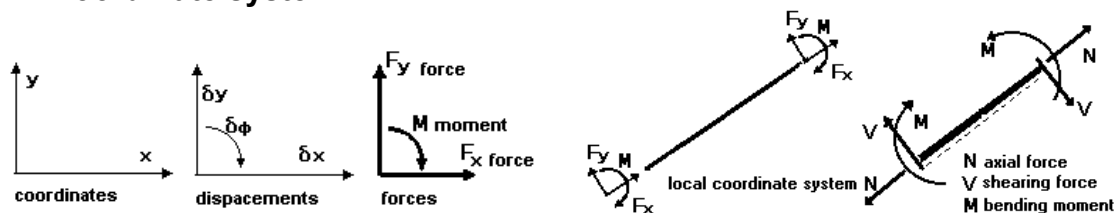
If you choose steel for structural material, then by clicking on the button  you can choose the steel profile from a complete list, and the cross section area and moment of inertia are set.



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:

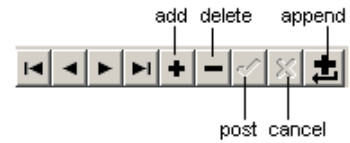
**Length**, coordinates, in meters (m).

**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  $g=9.81 \text{ m/sec}^2$ .

**Modulus of elasticity**, in GPa ( $\text{kN/mm}^2$ ).

**Specific weight**, in  $\text{kN/m}^3$ .

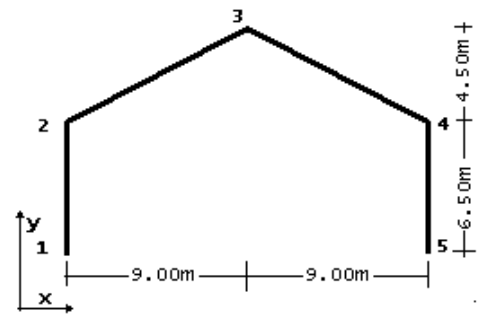
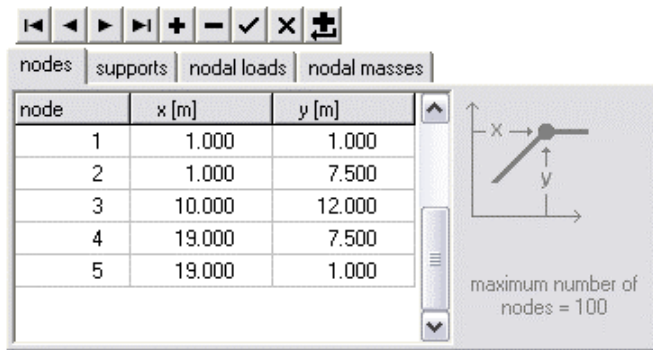


### 6. Nodal points

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.

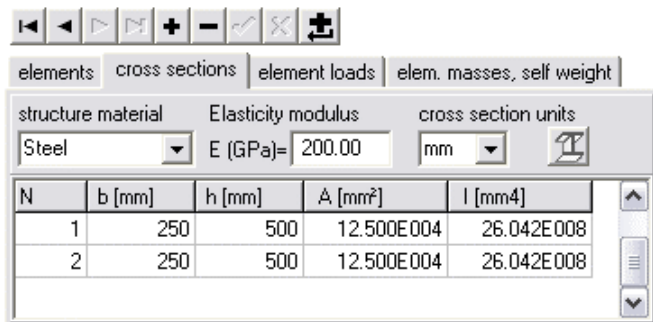
You must not have node numbers missing. This means if you have a structure with 24 nodes you have to input the coordinates of all the nodes from 1 to 24.



### 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 ( $\text{kN/mm}^2$ ), (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.

N	cr. s	b [cm]	h [cm]	b1 [cm]	h1 [cm]	A [cm <sup>2</sup> ]	I [cm <sup>4</sup> ]
1		25.0	50.0	0.0	0.0	12.500E002	26.042E004
2		25.0	50.0	0.0	0.0	12.500E002	26.042E004

### 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](#).

element	node A	node B	cr. sect.
1	1	2	2
2	2	3	1
3	3	4	1
4	4	5	2

Diagram labels: load on element, nodal point number, element number, section group number.

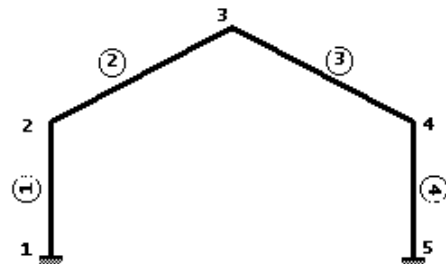
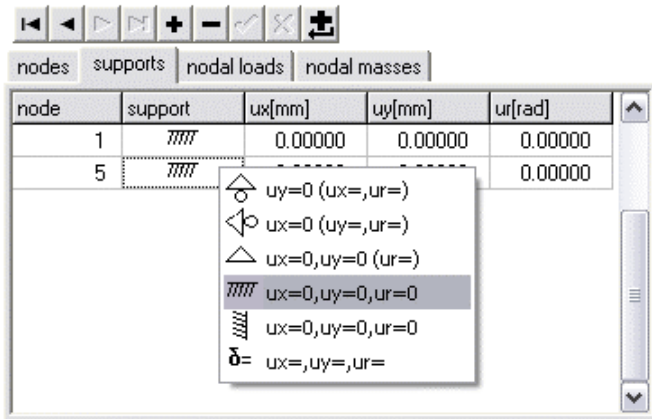
### 9. Supports

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  $u_x=3$ . ( 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  $u_x$ ,  $u_y$  and rotation  $u_r$ .

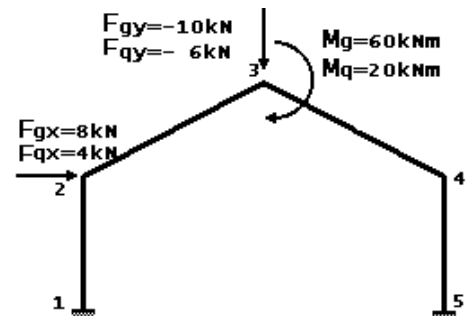
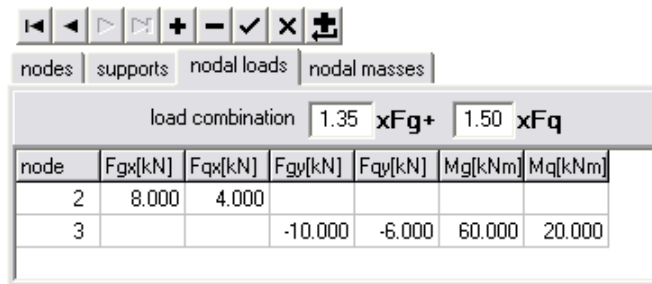


Support conditions for the structure.

### 10. 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  $C_g \times F_g + C_q \times F_q$ , 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.

### 11. 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>.

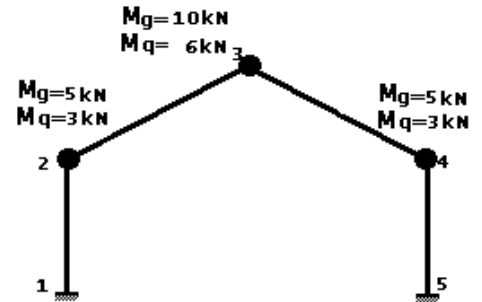
The masses in the dynamic analysis are taken as  $C_g \times M_g + C_q \times M_q$ . Default values are  $C_g=1.00$  and  $C_q=0.30$  (Eurocode 1, EN 1991 1-1:2003).

nodes supports nodal loads nodal masses

mass combination 1.00 xMg + 0.30 xMq

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

Input nodal weights in kN, nodal masses are computed dividing by  $g=9.81\text{m/sec}^2$



### 12. 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.

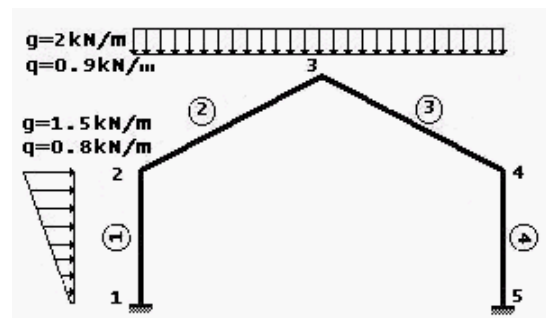
The loading direction, perpendicular to element axis, vertical or horizontal, is selected from the menu appearing by clicking the last column (direction).

The loading in the static analysis is taken as  $C_g \times F_g + C_q \times F_q$ , 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).

elements cross sections element loads elem. masses, self weight

load combination 1.35 xG + 1.50 xQ

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	



### 13. 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  $9.81\text{m/sec}^2$ .

The mass in the dynamic analysis are taken as  $C_g \times M_g + C_q \times M_q$ . 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.

elements cross sections element loads elem. masses, self weight

Weight density kN/m<sup>3</sup> 25.00 include self weight in loads and masses

Mass combination 1.00 xMg + 0.30 xMq

element	Gg [kN/m]	Gq [kN/m]
3	3.000	1.000
4	3.000	1.000

Input line weights kN/m, element line masses are computed dividing by  $g=9.81\text{m/sec}^2$

### 14. 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  $3N \times 3B$ , where N is the number of nodes and B is the nodal bandwidth.

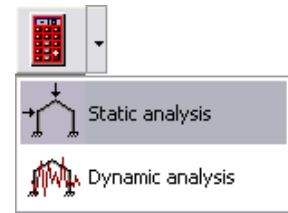
The element distributed loads are converted to equivalent nodal loads, that are added to the existing nodal loads, and the load matrix in the equilibrium equations is obtained.

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.

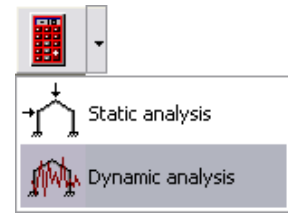
From the element end forces the diagrams of bending moments, shear forces and axial forces are obtained.



### 15. Dynamic Analysis

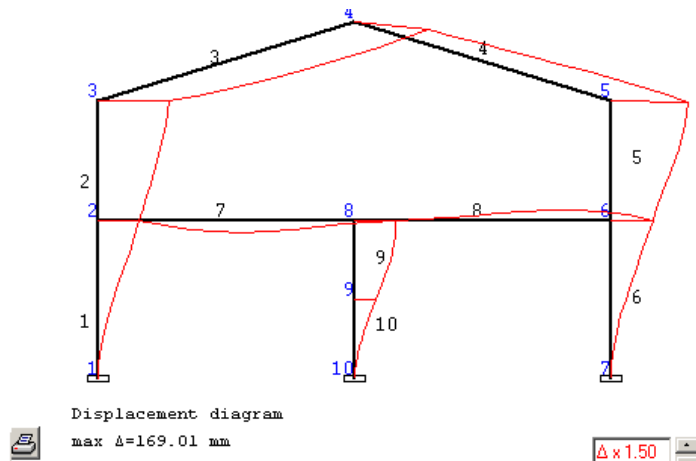
The dynamic analysis starts with the corresponding menu. The analysis steps are shown on the screen with their corresponding duration.

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  $3N \times 3B$ , where N is the number of nodes and N the nodal bandwidth. The matrix eigenvalue equation is solved using the generalized Jacobis method.



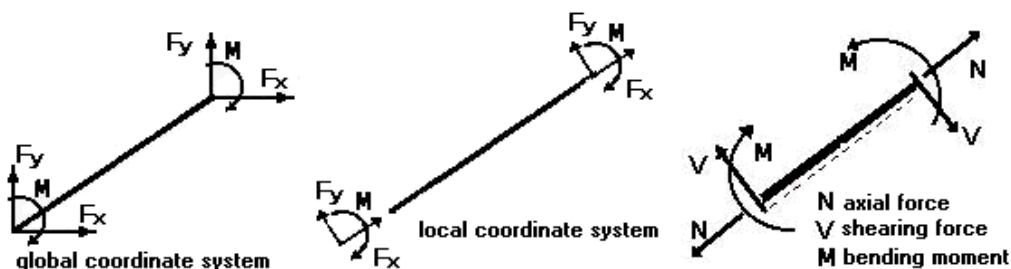
### 16. Displacements

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.



### 17. Internal forces

Member end forces



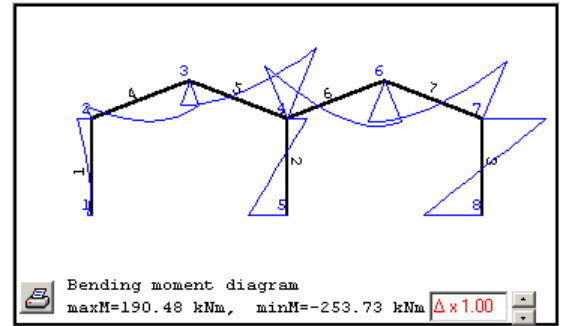
### 18. Diagrams

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



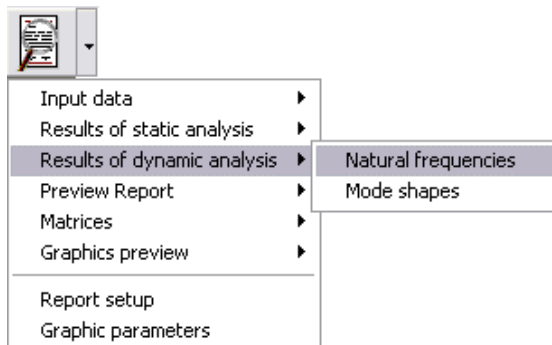
The diagram size is regulated by clicking at

Quick print of the diagram by clicking at

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

### 19. Natural frequencies

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



Natural frequencies		
$\alpha/\alpha$	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

### 20. Mode shapes

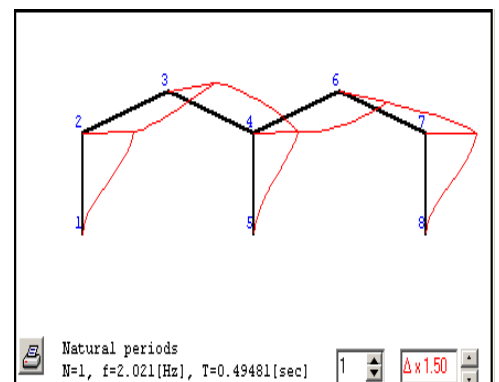
The natural frequencies and the mode shapes are obtained after the dynamic analysis.

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



## 21. 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.

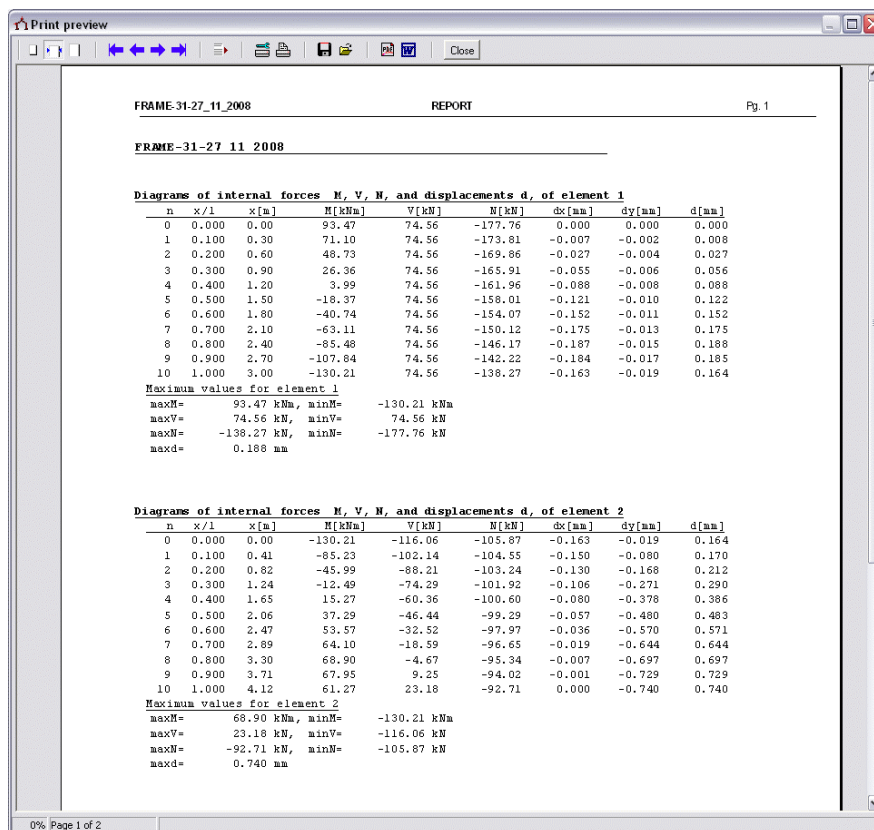
The font, and the margins are adjusted from the menu.

**Diagrams of internal forces M, V, N, and d**

n	x/l	x[m]	M[kNm]	V[kN]
0	0.000	0.00	-130.21	-116.06
1	0.100	0.41	-85.23	-102.14
2	0.200	0.82	-45.99	-88.21
3	0.300	1.24	-12.49	-74.29

print   save to file   sent to WORD   enter report

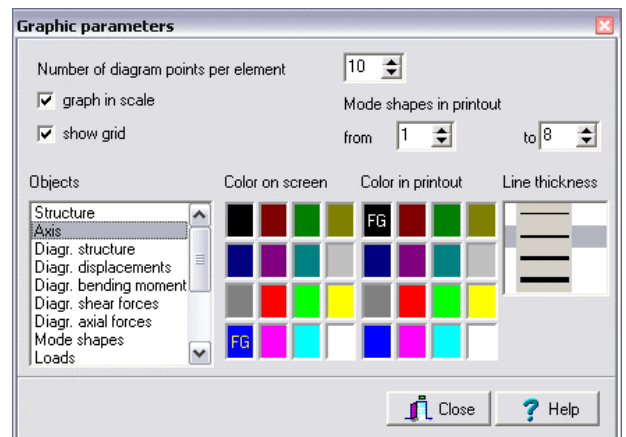
## Report example



## 22. 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.

## 23. References

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