Frame Analysis

1 Introduction **7**
1.1 Generally 7

2 Program organization **9**
2.1 Main menu 10
2.1.1 Input geometry 11
2.1.1.1 File 12
2.1.1.2 Edit 13
2.1.1.3 View 13
2.1.1.4 Input 14
2.1.1.5 Lists 15
2.1.1.6 Calculate 17
2.1.1.7 Options 18
2.1.1.8 Window 20
2.1.1.9 Help 20
2.1.2 Input loads 21
2.1.2.1 File 21
2.1.2.2 Edit 21
2.1.2.3 View 22
2.1.2.4 Basic loadcase 22
2.1.2.4.1 Code independent 23
2.1.2.5 Loads 23
2.1.2.6 Loadcase 23
2.1.2.6.1 Code independent 24
2.1.2.7 Calculate 24
2.1.2.8 Option 25
2.1.2.9 Window 26
2.1.2.10 Help 26
2.1.3 Results 26
2.1.3.1 File 26
2.1.3.2 Edit 27
2.1.3.3 View 27
2.1.3.4 Diagrams 27
2.1.3.5 Tables 28
2.1.3.6 Options 29
2.1.3.7 Window 29

2.1.3.8 Help 30
2.1.4 **Design** **30**
2.2 The Toolbar 30
2.2.1 Input geometry 32
2.2.1.1 The geometry of the frame 34
2.2.1.2 Support 35
2.2.1.3 Springs 36
2.2.1.4 Hinges 36
2.2.1.5 Initial bow imperfection 36
2.2.1.6 Sections 37
2.2.1.7 Group 38
2.2.1.8 Tension forses 38
2.2.2 Input loads 38
2.2.2.1 Uniform loads 40
2.2.2.2 Point loads and moments 41
2.2.2.3 Joint displacements 43
2.2.2.4 Temperature loads 43
2.2.2.5 Self weight 44
2.2.3 Results 44
2.3 Input geometry 45
2.3.1 File 45
2.3.1.1 Wizards 46
2.3.1.1.1 Purlin 46
2.3.1.1.2 Gerber beam 47
2.3.1.1.3 Continuous beam 47
2.3.1.1.4 Columns 48
2.3.1.1.5 Truss 48
2.3.1.1.6 Frame truss 51
2.3.1.1.7 W-truss 52
2.3.1.1.8 T-truss 53
2.3.1.1.9 Swedish truss 53
2.3.1.1.10 Rafter 54
2.3.1.1.11 Parallel truss 54
2.3.1.1.12 Reversed saddle truss 55
2.3.1.1.13 Saddle truss 55
2.3.1.1.14 Ridge truss 56
2.3.1.1.15 Frame type 1 56

2.3.1.1.16 Frame type 2 57
2.3.1.1.17 Arch 57
2.3.1.1.18 Gridframe 58
2.3.2 Input 58
2.3.2.1 Title 59
2.3.2.2 Convert code version 59
2.3.2.3 Geometry 59
2.3.2.4 Sections 61
2.3.2.5 Material 64
2.3.2.6 Hinges 64
2.3.2.7 Member type 65
2.4 Results 66
2.4.1 Diagrams 66
2.4.2 Tables 68
2.5 Printout 70
2.5.1 Print Manager 75

3 Methods of calculation 79
3.1 Analysis 79

4 References 81

Information

Frame Analysis 6

Copyright: Structural Design Software

Datum: 090119

Latest information about programs from **WIN-Statik** and **FEM-Design** see

www.strusoft.com

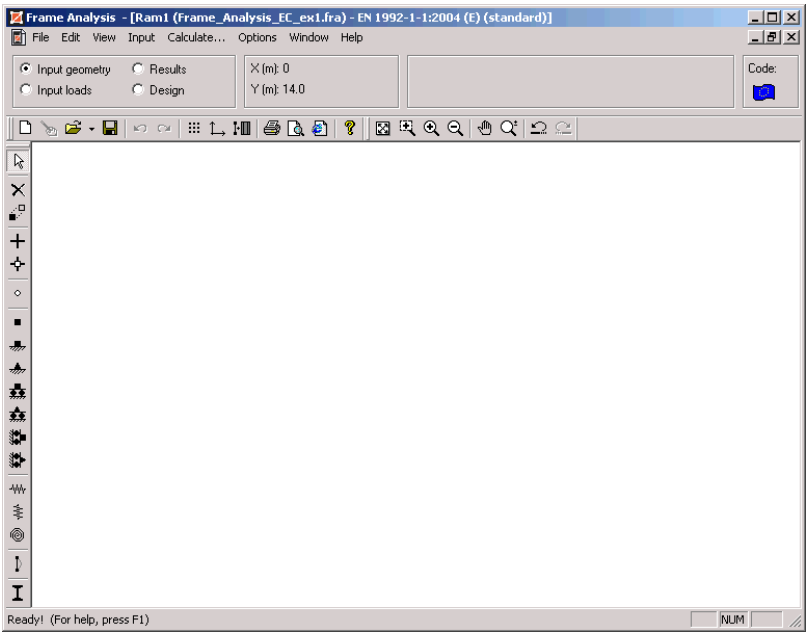
1 Introduction

1.1 Generally

With the software **Frame Analysis** arbitrary plane structures can be analysed according to first- and second order theory. With the program comes a **Steel**-, **Timber**- and **Concrete Module** to design frames made of steel, timber and concrete according to various codes.

2 Program organization

When starting the program the program window is shown as below:



The program window consists of the following fields or windows:

- At the top of the window the so-called title line is showing the name of the program (**Frame Analysis**), the project name (defined in **Title**) and the current file with its information (**Frame 1...** if **Save as** hasn't been used).
- The main menu, below the title line, contains several menu selections. The main menu will change depending on what section is active.
- Below the main menu there is a status bar which contains various fields showing information on: active program mode, the coordinates of the mouse, the active member or joint, active code and basic loadcase - or case loadcase list - depending on active program mode.

- The window has two toolbars, one below the status field and one on the left side of the window (except for the **Design mode**). By clicking on the toolbar buttons one can quickly choose the right command. The toolbars can be hidden, shown and moved.
- The window is dominated by the drawing area, on which the frame is made.
- The message field is the field at the very bottom of the window showing information or an explanation on the active/marked command.

The toolbar at the left side of the window is used for graphical input, while the different menu options are to be use for numerical input.

2.1 Main menu

In the main menu the user chooses a window by clicking on a suitable menu text. This either leads to the opening of the window or makes a sub-menu visible. There can be several levels of sub-menus. By clicking on a sub-menu choice, the window will eventually be opened.

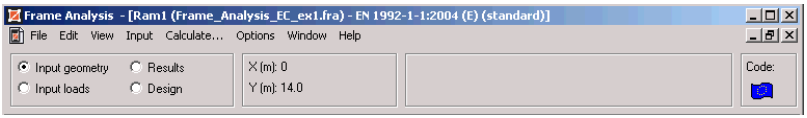
Some menu options can be marked in grey. This indicates that the given input data has made it unsuitable to open the window, and in order to prevent this the menu option is marked in grey.

Some sub-menu options are marked with a tick, which indicates that the activity is ***on***.

The tools in the toolbar can sometimes be used for a quicker choice of windows.

The appearance of the main menu will change depending on what program modeis active. The **Frame Analysis** program contains three different sections: **Input geometry**, **Input loads** and **Results**. If any of the programs **Steel Module**, **Timber Module** or **Concrete Module** have been installed, also the **Design mode** will be available (as seen below):

2.1.1 Input geometry



File Contains commands for different functions such as open saved information, save new information and print. Several of the **File** options are identical to other Windows programs.

Edit Contains editing tools like undo and redo.

View This function is used for controlling the graphic drawing of the frame on the screen. It also makes it possible to show/hide different fields/tools, and to switch from one program mode to another.

Input This menu option helps the user to define the problem in a numerical way. Note that the problems often can be defined graphically by using the graphical menu **Geometry tools**. The user can also mix numerical input data with graphical input data.

Lists Either shows the different sections that have been chosen, the number of members using the section, the total length, weight and area, or the members section, length, weight and area.

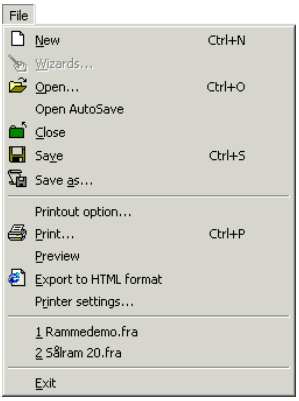
Calculate Starts the calculation.

Options Contains functions to initiate the drawing area in a way that is suited for the current problem.

Windows Shows the current windows.

Help Contains the complete manual of the program.

2.1.1.1 File



New Starts a new calculation. The short cut is Ctrl+N.

Wizards The user can choose from several pre-defined trusses and frames. See 2.3.1.1 for a more thorough description.

Open Retrieves input data from an earlier calculation. A list on saved input data files (files that end with fra) is shown. The short cut is Ctrl+O.

Open Auto Save Retrieves input data that has been saved automatically (in the **TEMP** catalogue). This function is normally not needed.

Close Closes current activity.

Save Saves input data for current calculations in the file with the current file name. **Save** is not to be used until after having used **Save as**. The short cut is Ctrl+S.

Save as Saves input data for current calculations in a file with a filename different from the current.

Printout option Used to choose input data and the result that will be printed.

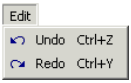
Print Used to print the current document. The short cut is Ctrl+P.

Preview Makes it possible to, on the screen, study the exact outcome of the chosen printout.

Printer settings This function is used to specify the printout when it comes to type of printer, paper width and so on.

Exit Ends **Frame Analysis**.

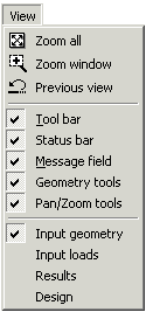
2.1.1.2 Edit



Undo This option is used to cancel a command or an action. It can be repeated several times. The short cut is Ctrl+Z.

Redo This option is used to reverse the effects of the command **Undo**. It can be repeated several times. The short cut is Ctrl+Y.

2.1.1.3 View



Zoom all Presents the frame drawn in a maximum use of the window.

Zoom window A specified area of the frame can be enlarged.

Previous view Makes it possible to revert to the previous zoom view.

Tool bar Makes it possible to hide and restore the tool palette.

Status bar Makes it possible to hide and restore the status bar.

Message field Makes it possible to hide and restore the message field.

Geometry tools Makes it possible to hide and restore the geometry tools.

Pan/zoom tools Makes it possible to hide and restore the zoom functions.

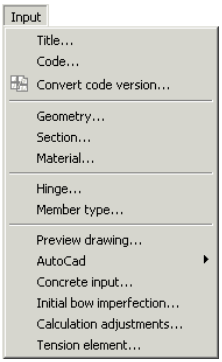
Input geometry This function is used to activate the geometry mode.

Input loads This function is used to activate the load mode.

Results This function is used to activate the result mode.

Design This function is used to activate the design mode. This option will be marked in grey if none of the **Steel**-, **Timber**- or **Concrete Modules** are accessible.

2.1.1.4 Input



Title This function is used to write general information on the current calculation.

Code Switches between a code independent and a **Code** adequate analysis depending on the number of codes available. The latter option is needed if a design is to be executed.

Convert code version With this option the user can convert between different country annexes for EuroCode or between different versions of a code for a country.

Geometry Current geometric data is defined numerically.

Section This function is used to define and edit sections as well as the direction of the sections.

Material This function is used to define and allocate material to sections.

Hinge Members hinges (moment equals zero) are shown.

Member type Displays types of members (Normal, Purlin, Gerber beam).

Concrete input If a codedependent calculation is chosen and concrete members are included then material values for concrete and reinforcement are definded under this option.

Initial bow imperfection With this option initial bow imperfection can be defined chosen members. (Only if Steel, Timber or Concrete Module is available).

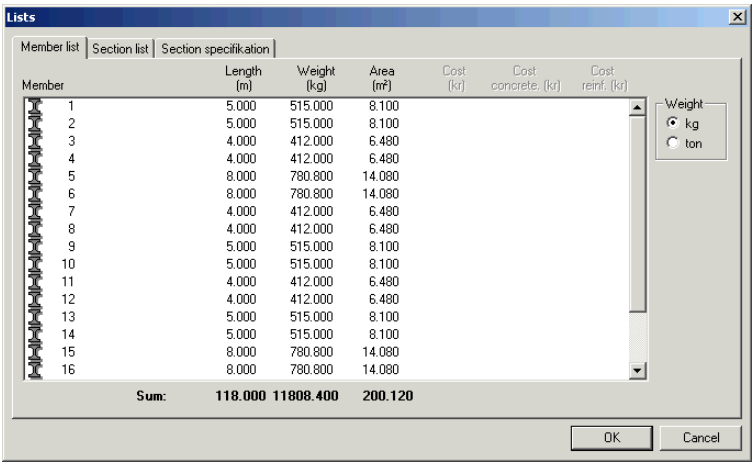
Calculation adjustments With this option necessary calculation accuracy for concrete members can be defined.

Tension element With this option arbitrary members can be declared as tension members meaning members that only resist tension but not compression.

2.1.1.5 Lists

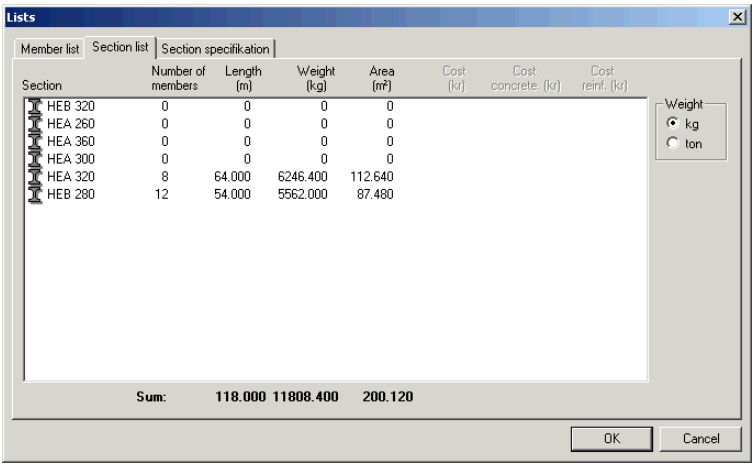
The menu option **Lists** shows a dialogue box with three tabs: **Member list**, **Section list**, **Section specification**, see below:

Member list
Contains a list on member numbers associated section, length, weight and surface.



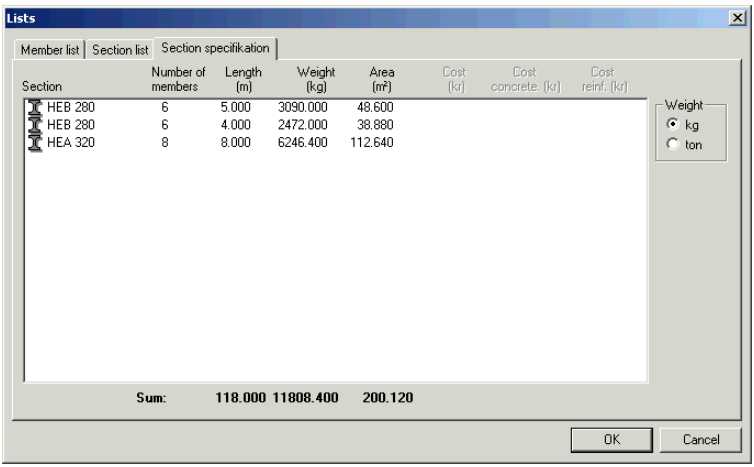
Section list

This function contains a list on used sections considering quantity, length, weight and surface.



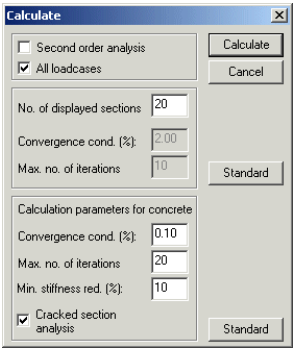
Section specification

This function contains a list where each member with the same section/material and length has been added.

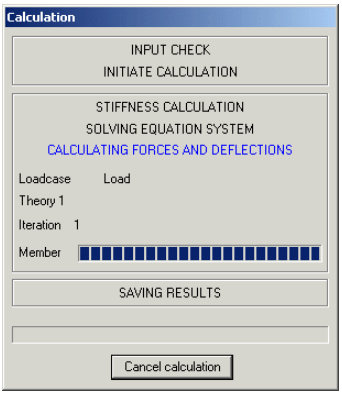


2.1.1.6 Calculate

The calculations are being made with the **Calculate** option. When this option has been activated the following dialogue box will be shown on the screen.



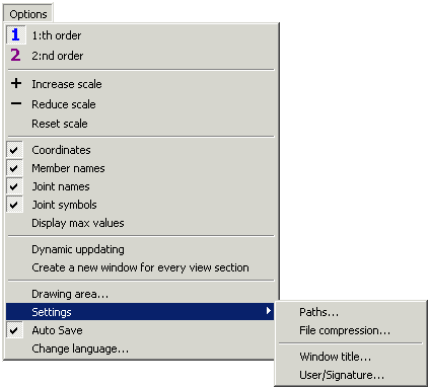
The user must state if the frame will be calculated according to the second order theory and which loadcases are to be analysed. It is, in most cases, wise to use the conditions given as standard. If not all defined loadcases are to be calculated this box will be cleared, and the user can choose what loadcases are to be calculated.



Without a second order analysis a possible design will be performed with consideration to user defined buckling lengths.

When the calculation is completed the result-window will be activated, and the **Result mode** will then be activated in the status field. See chapter 2.4.

2.1.1.7 Options



Coordinates This function shows/ hides the current coordinates of the cursor.

Member names Shows/hides the member names.

Hinges Shows/hides member hinges.

Joint names Shows/hides joint names.

Joint symbols Shows/hides joint symbols.

Section image Shows/hides the image of the member's current section.

Detailed This function shows the members drawn according to scale with consideration to the current section.

Dynamic update This function updates the calculation automatically when a change of input data has been made. This option builds on the assumption that every view section has been assigned a window of its own (see below).

Create a new window for every view section Here view sections means **Input geometry**, **Input loads**, **Results** and **Design** which by this option are assigned windows of their own.

Renumbering ***Automatically***: Renumbers joints and members (creates new joint-/member names, according to current order in the internal lists.

Members: Renumbers members (creates new member names as the members are being marked).

Joints: Renumbers members (creates new joint names as the joints are being marked).

Reassortment

Automatically: Re-sorts the internal lists so that they come in the same order as the member- and joint numbers.

Members: Re-sorts the internal lists so that they come in the same order as the members have been marked.

Joints: Re-sorts the internal lists so that they come in the same order as the joints have been marked.

Drawing area

This function is used to decide drawing-area size, distance between points (grid), origin of co-ordinate axis, resolution of the grid and the snap distance to the joint/ member (pixels).

Settings

Paths: This function is used to define paths to data files such as section database, codes or wizards.

File compression:

Window title: This function is used to specify the text in the window title-line. (**Note** that title or filename has to be included.).

User/signature*:* This function is used to remove users and to show how the automatic generation of the signature is to be performed.

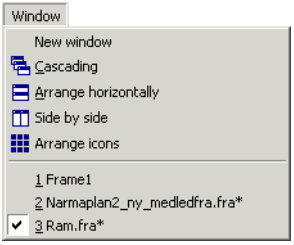
Auto Save

This function is used to create a document restore-file automatically after a 10-minute time interval. The restored file can contain information that hasn't been saved and would have disappeared from the original document otherwise. The restored file doesn't replace the **Save** command. The sample must still be saved when the job is finished.

Change language

This function is used to change between English and Swidish.

2.1.1.8 Window



New window This function opens a new window with the same example where the user can have several images, belonging to the same example, visible at the same time.

Cascading This function is used to arrange the document windows that are shown in a way that makes them cascade each other.

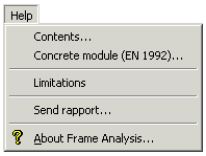
Arrange horizontally This function is used to arrange current document windows so that they all can be shown in full.

Side by side This function places all open document windows side by side, so that they all can be seen at the same time.

Arrange icons This function is used to arrange the different calculation-example icons that might be on the screen.

1 Frame1.fra* The hook shows that the window that represents the **Frame 1** example is active.

2.1.1.9 Help



Contents The **Frame Analysis** manual can be accessed directly in the program by this option.

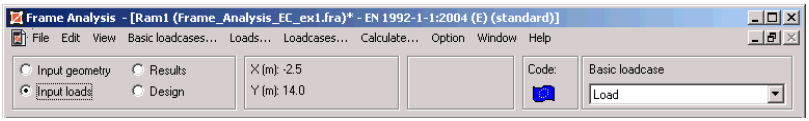
Concrete module (EN 1992)...

Limitations

Send report This function is used to send **Frame Analysis** e-mail to Structural Design Software.

About Frame Analysis This function shows information about the program, for instance version number.

2.1.2 Input loads



File See 2.1.1 above.

Edit See 2.1.1 above.

View See 2.1.1 above.

Basic loadcase This function is used to name different basic loadcases. The user will also state current load type when calculating timber members. Type of load and load value is given in **Loads**.

Loads The numerical input of loads that are a part of the different basic loadcases as well as their value and placement. This input can also be made completely graphically.

Loadcases This function is used to name and describe the different loadcases. The basic loadcases will be combined with the belonging load coefficients to ultimate- accident- and serviceability loadcases if a codedependent calculation will be made.

Calculate See 2.1.1 above.

Option See 2.1.1 above.

Window See 2.1.1 above.

Help See 2.1.1 above.

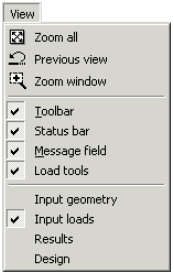
2.1.2.1 File

See 2.1.1.1 above.

2.1.2.2 Edit

See 2.1.1.2 above.

2.1.2.3 View



Zoom all	See 2.1.1.3 above.
Previous view	See 2.1.1.3 above.
Zoom window	See 2.1.1.3 above.
Tool bar	See 2.1.1.3 above.
Status bar	See 2.1.1.3 above.
Message field	See 2.1.1.3 above.

Load tools	Makes it possible to hide and restore the load tools.
Input geometry	See 2.1.1.3 above.
Input loads	See 2.1.1.3 above.
Results	See 2.1.1.3 above.
Design	See 2.1.1.3 above.

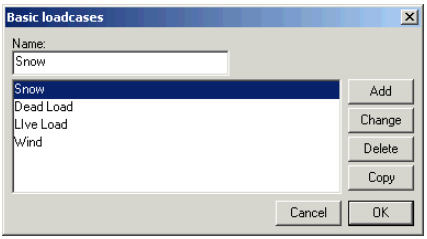
2.1.2.4 Basic loadcase

The **Basic loadcase** menu-option shows a dialogue box. The basic loadcases that will be combined to loadcases are named in this box. It is advisable to state the different basic loadcases that are to be used during the calculation. The basic loadcases are defined by naming them and then press **Add**. Press **OK** when all the basic loadcases have been defined.

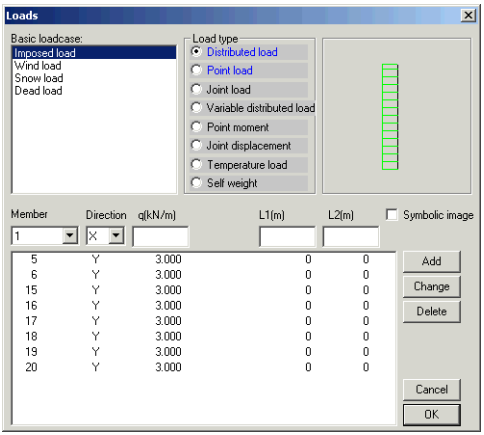
The basic loadcases that have been defined will bee shown in a pull down menu in the status bar right at the top of the screen to the right.

The user must go to **Loads** or to the toolbar at **Input loads** in order to define type, size and direction of loads in the basic loadcases.

2.1.2.4.1 Code independent



2.1.2.5 Loads



The user defines: load type, placement, direction and size of the loads that are to be a part of previously defined basic loadcases.

One of the basic loadcases defined above must be activated in order to state a load that will operate upon the construction. This is done in the top left dialogue box, as seen above.

If the input has been made graphically the numerical value can be confirmed. The **Symbolic image** option in the dialogue box shows more detailed what the load types look like graphically.

Loads can also be stated graphically, see chapter 2.2.2.

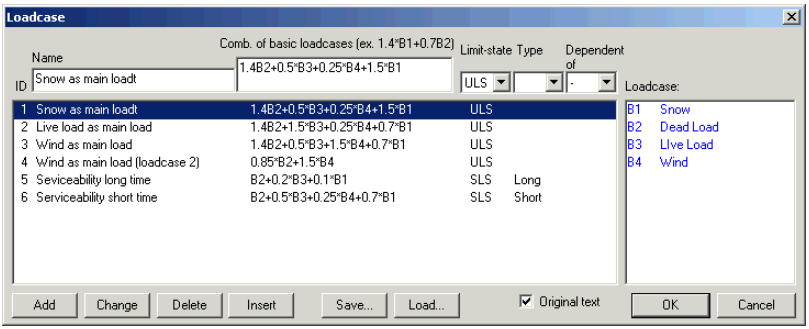
2.1.2.6 Loadcase

Under **Loadcase** the various basic loadcases are combined to loadcases with consideration to current load coefficients. It is also possible to save and retrieve loadcases.

It is advisable to state the various partial-coefficients that belong to the various loadcases, as the loadcases are being defined. By defining several various loadcases with various partial-coefficients it will be rather easy to compare them between themselves, to see what loadcase is the most dangerous. The loadcases are defined in the same way as everything else has been defined earlier i.e. press **Add** for every new loadcase that is to be added, and end with **OK** when all loadcases have been defined.

If the construction has been defined via any of the wizard that contain load generation, and the option **Loadcase** has been activated, the chequer above will contain all loadcases that the **Code** prescribes.

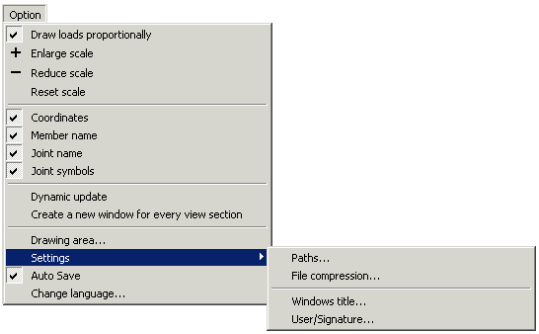
2.1.2.6.1 Code independent



2.1.2.7 Calculate

See 2.1.1.6 above.

2.1.2.8 Option



Draw loads proportionally This function is used to draw loads proportionally.

Enlarge scale This function is used to enlarge the scale on drawn loads.

Reduce scale This function is used to reduce the scale on drawn loads.

Reset scale This function is used to recalculate (reset) the scale on proportional loads.

Coordinates See 2.1.1.7 above.

Member name See 2.1.1.7 above.

Joint name See 2.1.1.7 above.

Joint symbols See 2.1.1.7 above.

Dynamic update See 2.1.1.7 above.

Create a new window for every view section See 2.1.1.7 above.

Drawing area See 2.1.1.7 above.

Settings See 2.1.1.7 above.

Auto Save See 2.1.1.7 above.

Change language See 2.1.1.7 above.

2.1.2.9 Window

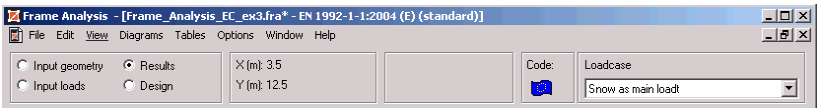
See 2.1.1.8 above.

2.1.2.10 Help

See 2.1.1.9 above.

2.1.3 Results

After the completion of the calculation the **Result** section will be lit and the design of the main menu will change as seen below. The **Result** option in the main menu makes it possible to have the calculation result shown numerically and graphically.



File See 2.1.1 above.

Edit See 2.1.1 above.

View See 2.1.1 above.

Diagrams This function makes it possible to choose what sort of calculation-result one wants to have shown in a graphic way.

Tables Selects what tables are to be written on the screen.

Option Selects to study the result according to the first- and/or second order theory as well as functions to initiate the drawing area in a way that fits the current problem.

Window See 2.1.1 above.

Help See 2.1.1 above.

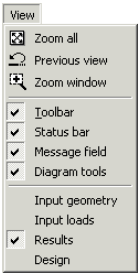
2.1.3.1 File

See 2.1.1.1 above.

2.1.3.2 Edit

See 2.1.1.2 above.

2.1.3.3 View



Zoom all See 2.1.1.3 above.

Previous view See 2.1.1.3 above.

Zoom window See 2.1.1.3 above.

Toolbar See 2.1.1.3 above.

Status bar See 2.1.1.3 above.

Message field See 2.1.1.3 above.

Diagram tools Makes it possible to hide and restore the diagram tools.

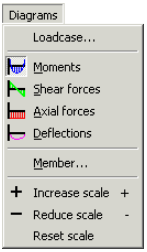
Input geometry See 2.1.1.3 above.

Input loads See 2.1.1.3 above.

Results See 2.1.1.3 above.

Design See 2.1.1.3 above.

2.1.3.4 Diagrams



Loadcase This function is used to choose what loadcase is to be active in the account for a result.

Moment This function displays current moment diagrams.

Shear force This function displays current shear force diagrams.

Axial force This function displays current axial force diagrams.

Deflections This function displays current deflections diagrams.

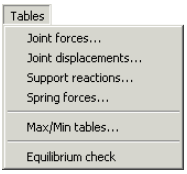
Member This function makes it possible to show diagrams and tables for a separate member.

Increase scale This function is used to increase the scale of the diagrams.

Reduce scale This function is used to reduce the scale of the diagrams.

Reset scale This function is used to recalculate (reset) the scale of the diagrams.

2.1.3.5 Tables



Joint forces This function shows the calculated section forces: **M**, **T** and **A** for each joint, according to first- or second order theory and for each loadcase.

Joint displacement This function shows the calculated joint displacement for each joint, according to first- or second order theory and for each loadcase.

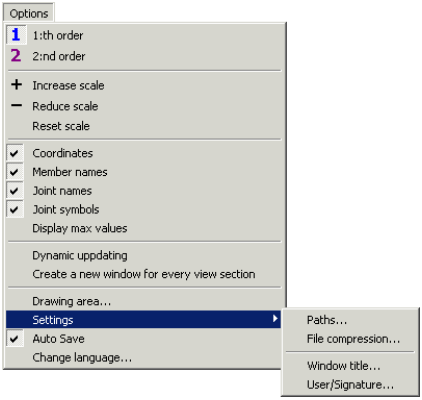
Support reactions This function shows calculated support reactions in **x**- and **y**- directions and moments for supported joints. These support reactions can be chosen for first- or second order theory as well as for each loadcase.

Spring forces This function shows possible spring forces in **x**- and **y**- directions and moments. These support reactions can be chosen for first- or second order theory as well as for each loadcase.

Max/Min tables This function shows the calculated maximum positive- and maximum negative forces as well as stresses for each member with current distances, together with at the same point acting respective moments, shear forces and axial forces. These values can be chosen for first- or second order theory as well as for each loadcase.

Equilibrium check This function shows an equilibrium check for every loadcase and can be chosen for first or second order theory.

2.1.3.6 Options



1st order	Views/hides the result of the 1st theory.
2nd order	Views/hides the result of the 2nd theory.
Increase scale	See 2.1.3.4 above.
Reduce scale	See 2.1.3.4 above.
Reset scale	See 2.1.3.4 above.
Coordinates	See 2.1.1.7 above.
Member names	See 2.1.1.7 above.
Joint names	See 2.1.1.7 above.
Joint symbols	Views/hides joint symbols.
Display max values	This function displays maximum moments, shear- and axial forces.
Dynamic update	See 2.1.1.7 above.
Create a new window for every view section	See 2.1.1.7 above.
Drawing area	See 2.1.1.7 above.
Settings	See 2.1.1.7 above.
Auto Save	See 2.1.1.7 above.
Change language	See 2.1.1.7 above.

2.1.3.7 Window

See 2.1.1.8 above.

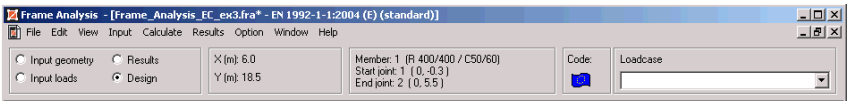
2.1.3.8 Help

See 2.1.1.9 above.

2.1.4 Design

When the calculation has been completed the program mode **Design** will be lit and the appearance of the main menu will change as seen below. The **Design** option in the main menu makes it possible to define possible supplementary information for a design, and to study the outcome of the calculation.

Note that this section will only be available if any of the **Steel**-, **Timber**- or **Concrete Modules** are installed.



2.2 The Toolbar

The toolbar contains the graphic tools with which a large part of the, for the calculation, necessary information can be defined. Each button in the toolbar represents a tool. When using the graphic tools the handling will often be quicker and simpler than it is with numerical input.

The toolbar (the toolbox) is a window itself so it can be moved and changed in size.

The user activates the tools by clicking with the mouse on the chosen button. The cursor's appearance will now change to that of the chosen tools.

The toolbar is in the drawing area's upper part and at the left side of the window. The toolbox in the upper part of the window will look the same through the entire run; see the image below.

This toolbox contains the most frequent and common commands that are accessible in all the program's four program modes.

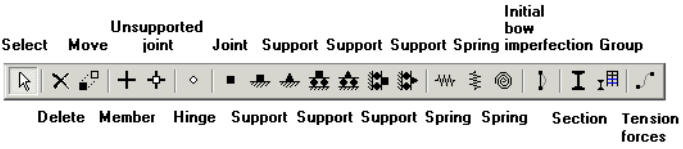
New	Starts a new example.
Wizards	Retrieves pre-defined frames, rafters and trusses with their parameter measure. Also see 2.1.1.1. A more detailed description of **Wizards** is to be found in chapter 2.3.1.1.
Open	Retrieves input data from a previous example and shows a list on saved input data-files (files that end with FRA or RAM).
Save	Saves input data for a current calculation in a file with current file name. Save is not to be used until having used **Save as**.
Undo	This button is used to cancel a command or an action. It can be repeated several times. The short cut is Ctrl+Y.
Redo	This button is used to reverse the effects of the command **Undo**. It can be repeated several times. The short cut is Ctrl+Z.
Grid	Displays/hides grids.
Co-ordinate axis	Displays/hides co-ordinate axis.
Detailed members	This button is used to display section shaped members.
Print	This button is used to print input data and the result that has been chosen in **Printout options** under **File** in the main menu.
Preview current view	Displays a print preview of the current view.
Export to HTML	Exports the current output to a HTML file.
Information	This button shows information about the program, for instance version-number.

Zoom all	With the **Zoom all** function the drawing area will be used to its maximum extent.
Zoom window	A specified area can be enlarged with this tool.
Zoom in	This button is used to enlarge the window gradually.
Zoom out	This button is used to reduce the window gradually.
Panorate	This button is used to activate the panorate function.
Zoom	This button is used to zoom dynamically. After having activated this button the appearance of the cursor will change as it is introduced to the drawing area. If the left mouse button is pressed down at the same time as the cursor is being moved there will be a dynamic zoom in or zoom out depending on the direction of transport.
Previous	Shows the previous view.
Next	Shows the next view.

The toolbox on the left side of the program window changes appearance depending on program mode (see below). The active program mode is marked in the status bar (the upper part of the window). By choosing program mode in the status bar or in the **View** menu the mode's toolbox can be obtained (or concealed).

2.2.1 Input geometry

When the program is in the geometry mode the frame model can also be defined graphically. The following buttons can then be used.



Select	Point tool (Choice of object.).
Delete	Makes it possible to remove joints or members. See 2.2.1.1.

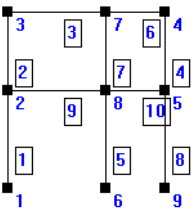
Move	Used to move joints. See 2.2.1.1.
Members	Used to create new members. See 2.2.1.1.
Unsupported joint	This function creates a joint that, at a possible design, isn't automatically laterally supported with consideration to flexural buckling out of the frame plane. The joint is however considered to have a good enough torsional stiffness to assume a support with consideration to lateral-torsional buckling. An unsupported joint doesn't split members, it only allows connection with other members to itself. See 2.2.1.1.
Hinge	Used to create hinges. See 2.2.1.4.
Joint	Used to create joints. See 2.2.1.1.
Support	Used to create a support rigid in all directions. See 2.2.1.2.
Support	Used to create a support rigid in **x**- and **y**-direction. See 2.2.1.2.
Support	Used to create a support rigid in **y**- and **z**-direction. See 2.2.1.2.
Support	Used to create a support rigid in **y**-direction only. See 2.2.1.2.
Support	Used to create a support rigid in **x**- and **z**-direction. See 2.2.1.2.
Support	Used to create a support rigid in **x**-direction only. See 2.2.1.2.
Spring	Used to create a horizontal spring. See 2.2.1.3.
Spring	Used to create a vertical spring. See 2.2.1.3.
Spring	Used to create a moment spring. See 2.2.1.3.

Initial bow imperfection	Used to allocate initial bow imperfections to the members. Only active at a code dependent calculation. See 2.2.1.5.
Section	Point tool for sections. See 2.2.1.6.
Group	Used to group members so that all members in the group can be allocated the same section with one marking.
Tension forces	Used to define members as tension members meaning that they can resist only tension but not compression.

2.2.1.1 The geometry of the frame

Graphic input is made from the toolbar shown at the left side of the screen.

The cross in the toolbar is used to define the geometry. The current coordinates are being shown simultaneously in the message field. By placing the cross at the required coordinates and clicking one time on the left mouse button a joint will be defined. If the mouse button is kept pressed down the member will be created between the coordinates where the mouse button was pressed down and where it was released. If the shift key is pressed at the same time the cross will only be able to move in an orthogonal way. If two joints are to be tied together with a member the user must place the cross on the first joint point and then press the left mouse button. The user will then drag the cross to the second joint point and let go of the button. A structure can look like the structure below. The coordinates and other data are the same as those at the numeric input above.



When the input of joints and members are being done graphically they will automatically be placed in the numeric menu, which has been described above. The arrow above the cross is used to edit joint points. By pointing at the joint point that is to be moved, and then keep the left mouse button pressed down, the joint point will automatically be moved to the point where the arrow is when the mouse button was released. All members that are attached to the current joint will automatically follow. All members attached to the joint must be removed before removing the joint.

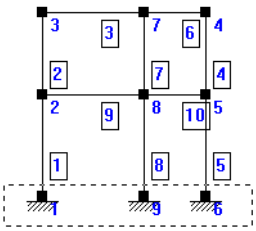
Important! A joint that has been defined with the **Member** tool will automatically be regarded as laterally supported with consideration to flexural instability out off the frame plane at a possible design. If the joint should not be supported against flexural buckling out of the frame plane but only against lateral-torsional buckling the tool **Unsupported joint** can be used if the joint is considered to have a good enough torsional stiffness to assume a support with consideration taken to lateral-torsional instability. An unsupported joint doesn't split members; it only allows connection with other members to itself.

The geometry can be edited numerically through **Input/Geometry** if it has been defined graphically. First graphically draw the geometry roughly, and then edit it numerically. A more thorough description of the numeric input of the geometry is to be found in chapter 2.3.2.3.

2.2.1.2 Support

The supports can also be given graphically by choosing one of the tools above (see 2.2.1), and place them on the joints that are to be defined as supports.

The geometry on the screen can look as below after a graphic or numeric input of the supports; all supports are rigid.



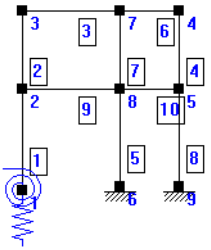
The supports can be created immediately by having the left mouse button pressed down and at the same time drag it over the joints where supports are to be defined. This is marked with a dashed square as seen above. The joints within the square will be allocated the current support.

The joint button can only be used to take away support.

A more thorough description of the so-called numeric input is to be found in chapter 2.3.2.3.

2.2.1.3 Springs

The different springs can be combined. In the figure below a **y**-direction spring and a rotation spring have been combined and replace the previous rigid support. The current supports must be removed before it is possible to replace supports with springs. The supports and springs can be removed the same way as the members and joint points (chapter 2.2.1.1. The geometry of the frame).



The springs can be created immediately by having the left mouse button pressed down and at the same time drag it over the points where springs are to be defined. The user can also point at the current joint and click with the left mouse button.

A more thorough description is to be found in chapter 2.3.2.3.

2.2.1.4 Hinges

The input of the hinges is done with help from the ring in the toolbar. By clicking on the joints or the members the hinge will be applied to the outer ends of the member.

By clicking on the member a hinge on the nearest joint will be defined.

If the user clicks on centre of the joint hinges will be applied on all members that are connected to the current joint.

If the user clicks one more time the hinge will be removed.
A more thorough description is to be found in chapter 2.3.2.6.

2.2.1.5 Initial bow imperfection

To be able to choose **Initial bow imperfection** any of the **Steel- Timber-** or **Concrete Modules** must be installed.

Consideration must be taken to initial bow imperfection when designing axial force loaded members. If flexural buckling in the frame's plane is being calcula-

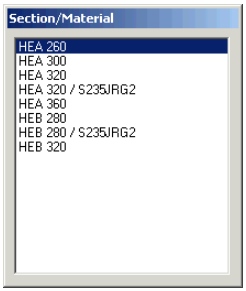
ted with the help from a buckling diagram, or of a corresponding method, the effect is a part of it. If on the other hand the second order theory is used the effect of initial bow imperfection is a part of the calculation. This can either be done by deforming the members, to the required position before calculation or can horizontal loads, and counter directed reaction forces responding to these, be added. The **Frame Analysis** uses a variation of the first method to consider the initial bow imperfection for the members of the user's choice within the solution.

Consideration to initial bow imperfection is being made with help from the tool shown to the left. This button must be activated in order to allocate initial bow imperfection to the members. By keeping the left mouse- button pressed down, and drag it over the members where initial bow imperfection is to be applied, all members within the chequer will be allocated initial bow imperfection. The members that have been allocated this effect will be marked in blue.

2.2.1.6 Sections

The requested section is allocated graphically to the members with help from the tool shown to the left. A chequer will display the preselected sections when the button has been activated. In order to allocate the members correct sections the current section is marked in the chequer after which the members that are to be allocated the sections will be marked.

By keeping the left mouse-button pressed down, and drag it over the members where the current section is to be applied, all members within the chequer will be allocated the current section.



The members that have been allocated the current section will be marked in blue.

The members can also be allocated the sections numerically. A more thorough description is to be found in chapter 2.3.2.3.

2.2.1.7 Group

In order to be able to tie members to a section more quickly several members can be brought together to a group with help from the tool to the left.

All members belonging to a group will be marked in blue when the group number is active. An optional number of groups can be given at this point.

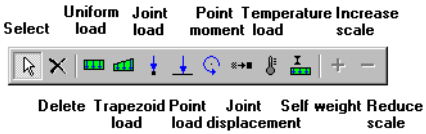
To mark that a member is no longer to be a part of a group the **None** option must be activated after which the current member is being marked.

2.2.1.8 Tension forses

If the structure contains members which are not able to resist compression they can be defined as tension members with this tool. All tension members will be marked in blue when this tool is active.

2.2.2 Input loads

The program is in load mode when loads are to be defined. In this mode the toolbox to the left in the window changes appearance and the following tools are accessible:



Select	Point tool. (Choice of object).
Delete	Makes it possible to delete loads.
Uniform loads	Used to define uniform loads. See 2.2.2.1.
Trapezoid load	Used to define trapezoid loads.
Joint loads	Used to define joint loads.
Point load	Used to define point loads on the member.

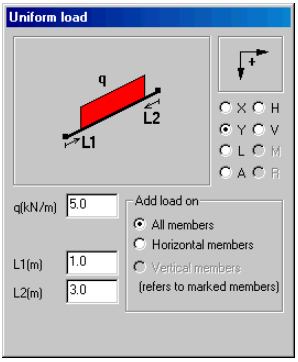
Point moment	Used to define the point moment on the member.
Joint displacement	Used to define joint displacements.
Temperature loads	Used to define temperature loads.
Self weight	Used to define the self-weight of the members.
Increase scale	Used to increase the scale on drawn loads. This option is available when **Draw loads proportionally** is active. Also see 2.1.2.8.
Reduce scale	Used to reduce the scale on drawn loads. This option is available when **Draw loads proportionally** is active. Also see 2.1.2.8.

One of the previously defined basic loadcases must be activated in order to define a load that has effect upon the construction. This is done in the **Basic loadcase** pull down menu at the top right of the status field, also see 2.1.2.4.

When choosing a load-tool an input box for current load type will be presented. In this box the user can choose size, direction and possible spread for the load. It is advisable to choose the size of the loads as the characteristic values considering that for example partial safety factors often are added later on (see **Loadcases** chapter 2.1.2.6).

The choices of direction are as follows:

X X-direction
Y Y-direction
H Horizontal
V Vertical
L Along, perpendicular to the member
A Along, in the same direction as the member
M Moment
R Angular displacement



All load types can, just as the sections, be allocated several members at the same time by keeping the left mouse button pressed down and drag it over the members that are to be allocated the load. The line of action to remove or change a load is the same as when changing joints and members. The arrow in the graphic menu is used to point at the load one wants to remove or change.

All loads that have been defined in the same basic loadcase will be displayed on the screen. The loads that have been allocated a different basic loadcase will not be displayed until this basic loadcase has been activated.

A more thorough description on the numerical input of loads is to be found in chapter 2.1.2.5.

2.2.2.1 Uniform loads

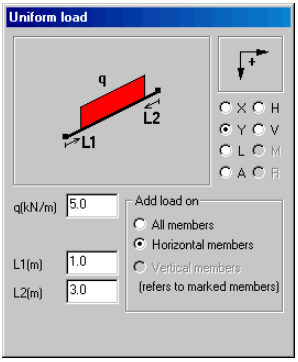
Uniform loads can be defined as loads with constant load intensity and trapezoid loads with linear varying intensity.

Uniform load with constant load intensity

Trapezoid load with variable load intensity

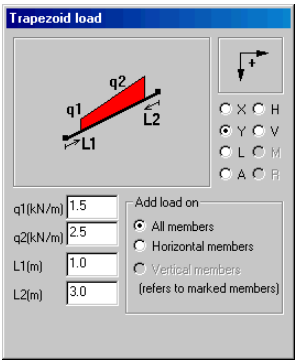
Uniform loads are defined by indicate intensity and direction of the uniform load. The measures L1 and L2 are set to zero as default, which means that the load is assumed to have effect upon the entire member.

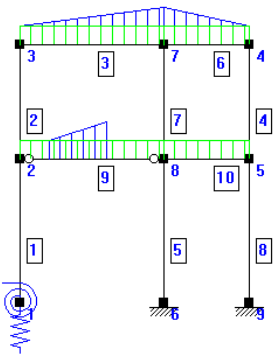
By feeding in the current measures for L1 and/or L2 the user will state if the load only has effect on a part of the member.



Trapezoid loads are defined by stating the intensities at the load's start point and end point and if the load acts perpendicular to the member or in the axial direction. The measures L1 and L2 are set to zero as default, which means that the load is assumed to have effect upon the entire member. By defining the current measures for L1 and/or L2 the user will state if the load only has effect on a part of the member.

When a loadcase has been defined the structure can for example look as the figure below.





2.2.2.2 Point loads and moments

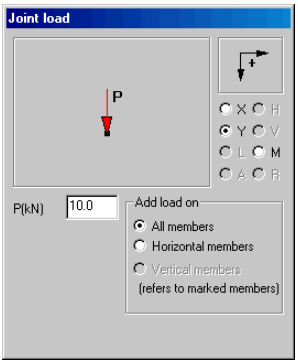
The point loads can be point loads, point moments, joint loads and joint moments. The point loads can be defined with help from some of the following buttons that are in the toolbar. The difference between point loads and joint loads is that point loads acts on the member whereas the joint loads are applied at the joints.

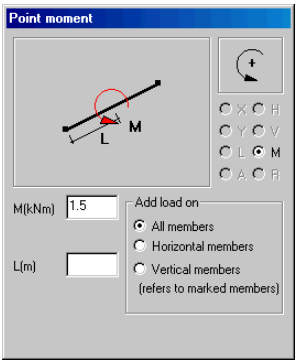
Joint loads and Joint moments

Point loads

Point moments

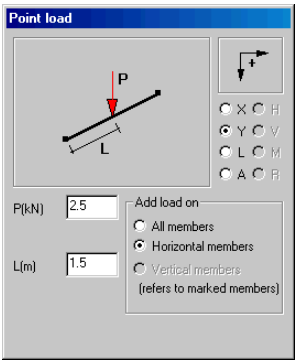
Joint loads and joint moments are being defined by stating size and direction in the input window and then placing them on the joints.

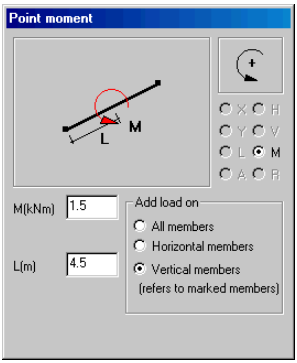




Size, direction and distance from the member's starting joint must be stated in order to define point loads and point moments. The starting joint is the joint that was stated first at the geometry input.

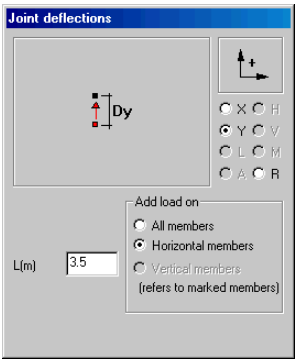
Point loads can't be stated if the distance to a joint is closer than the length/250, in what case joint loads must be used.





2.2.2.3 Joint displacements

This is used to state a prescribed joint displacement. This can for example be the case at support settlings.

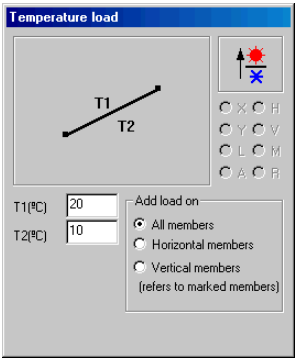


The joint displacement can be defined as a displacement in the **x**-axis, **y**-axis or as a rotation displacement. In order to state a rotation displacement the symbol **R** (see the picture to the left) must be marked. The displacement is given in degrees.

2.2.2.4 Temperature loads

When doing the input on temperature loads it is important that the thermal expansion for the material of the chosen section has been defined. If any of the pre-defined materials are being used the coefficient of heat expansion will also be pre-defined.

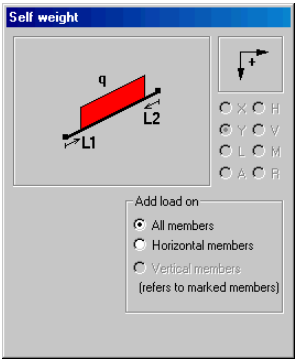
The button seen to the left below is used for defining temperature loads. When the button is activated the following input box will be shown.



The materials will shrink/expand if the user states the same temperature on the upper- as well as the underside. This leads to internal stresses in the member.

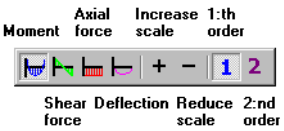
2.2.2.5 Self weight

One or several members can be loaded with self-weight answering to the type of cross section that has been tied to the member.



2.2.3 Results

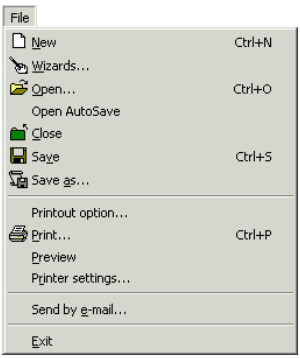
When the calculation is finished the **Results** option in the status bar will be lit. This position's corresponding toolbox contains the following tools that makes it possible to choose what sort of calculation result is desired to be shown graphically.



Moment	Shows the moment distribution over the entire structure for the selected loadcase as a diagram.
Shear force	Shows the shear force distribution over the entire structure for the selected loadcase as a diagram.
Axial force	Shows the axial force distribution over the entire structure for the chosen loadcase as a diagram.
Deflection	Shows the deflections of the structure as a diagram.
Increase scale	This function is used to increase the scale of the diagrams. Also see 2.1.3.4.
Reduce scale	This function is used to reduce the scale of the diagrams. Also see 2.1.3.4.
1:th order	Calculation result for the 1st order theory is shown.
2:nd order	Calculation result for the 2nd order theory is shown.

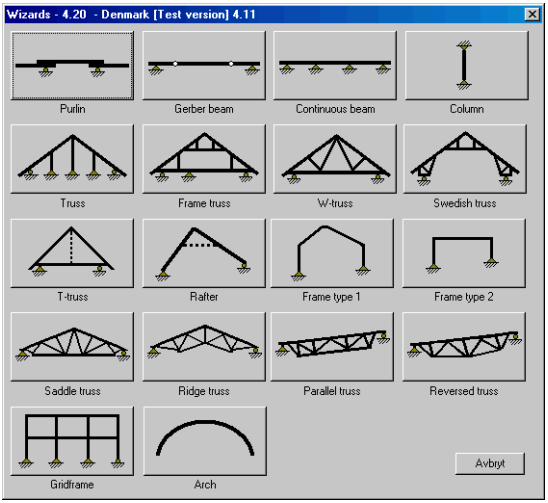
2.3 Input geometry

2.3.1 File



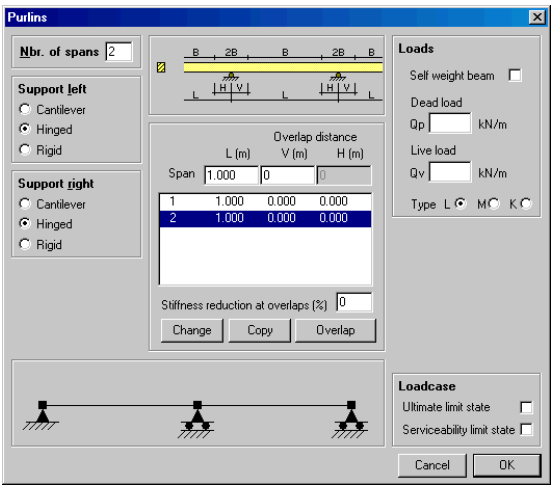
The **File** menu option contains several options that are identical to other Windows programs, except for the **Wizards** option as seen below. **Wizards** is a separate module that can be bought to the **Frame Analysis** program.

2.3.1.1 Wizards



The **Wizards** option shows a dialogue box where various pre-defined beams, frames, rafters and trusses can be stated easily. If codedependent wizards are available, loads and loadcases can be automatically generated for the Swedish, Norwegian and Danish load code.

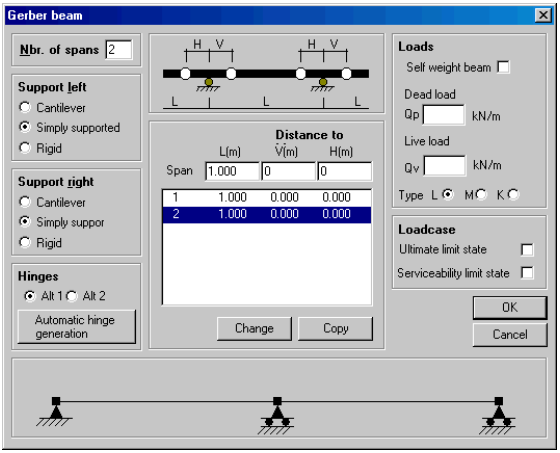
2.3.1.1.1 Purlin



Number of spans, left and right end-support, overlap distance for each span, loads and a potential reduction of stiffness at overlaps, are stated for purlins.

An image of the beam can be previewed in the wizard while defining input data.

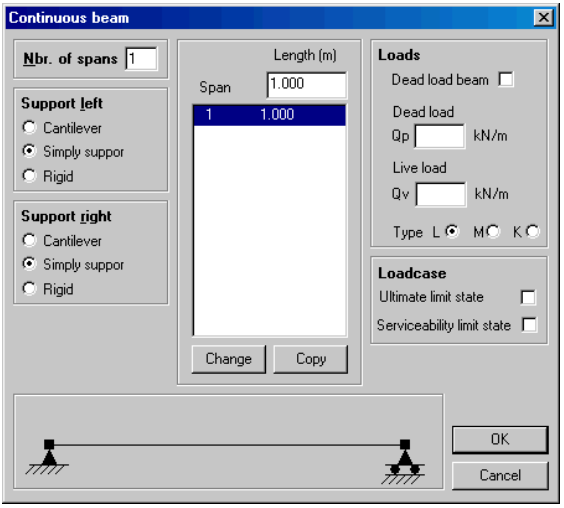
2.3.1.1.2 Gerber beam



Number of spans, left and right end-support, span lengths, loads and distance to hinges for each span is stated for gerber beams.

An image of the beam can be previewed in the wizard while defining input data.

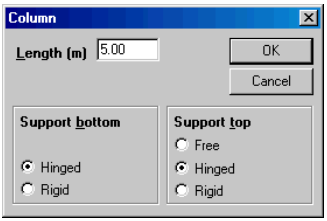
2.3.1.1.3 Continuous beam



Number of spans, left and right end-support, span lengths and loads are stated for continuous beams.

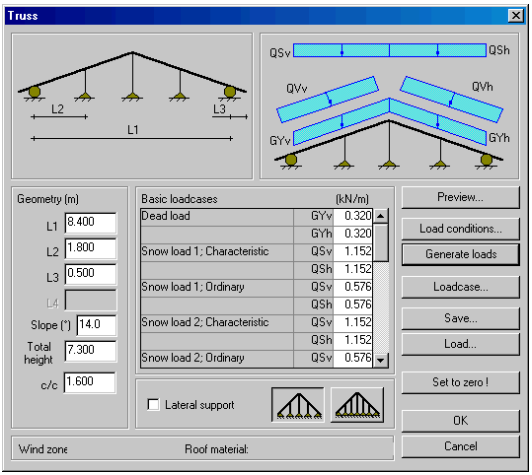
An image of the beam can be previewed in the wizard while defining input data.

2.3.1.1.4 Columns



The length and the upper and lower end-support are stated for columns.

2.3.1.1.5 Truss

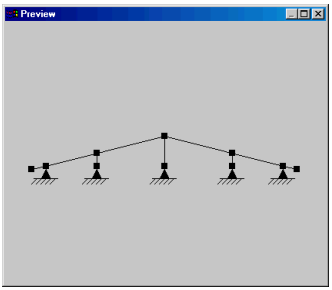


Necessary geometrical data and current loads are stated for truss. Loads and loadcases can in some cases also be generated automatically by the program.

Two types of trusses can be generated from this wizard by activating any of the buttons at the bottom to the right.

Geometry

This is where current slope and the geometrical measures to define the truss are stated. For codedependent wizards also the overall height of the building and distance between trusses has to be stated in order to calculate the load for each truss.



Current geometry can be studied in the wizard with the **Preview** option.

Loads and Loadcases

For codedependent wizards also loads and loadcases can be defined or generated automatically.

Dead load

This is where the dead load of the roof on the left- and the right hand of the roof are stated. Dead load can also be generated automatically via load generation as seen below.

Snow load

This is where the snow load on the left- and the right hand of the roof is stated for different snow situations.

Characteristic load: State the characteristic value of the snow load.
Ordinary load: State the ordinary value of the snow load.

Wind load

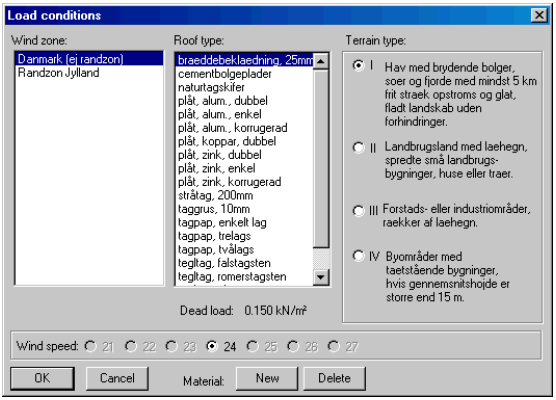
This is where the wind load on the left- and the right hand of the roof is stated for different wind situations.

Characteristic load: State the characteristic value of the wind load.
Ordinary load: State the ordinary value of the wind load.

Load conditions

With this option necessary input are defined to make it possible to automatically generate dead load for the roof together with snow- and wind loads according to the active load code.

Danish code DS410

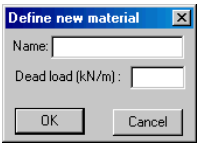


Wind zone

Choose the wind zone the and reference wind speed belonging to it will be shown at the bottom of the box. If more than one wind speed can be in question for the chosen wind zone, the program chooses the one giving the greatest load action. The user can change this if desired.

Roof type

A current type of roof can be chosen from the above displayed list. If none of these are applicable the list can be completed with own roof materials with help from the **New** option. A name and current dead load would be given as seen to the right:



Defined roof materials are being removed by the **Delete** option.

In order to consider also the self weight of the truss itself and purlins and such, the dead load is increased with 0,2 kN/m^2.

Terrain type

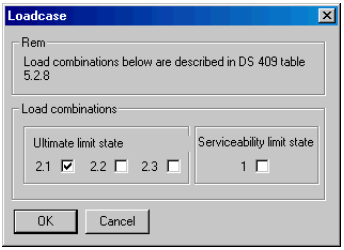
The current type of terrain must be stated in order to calculate the correct wind load.

Generate Loads

Dead load, snow loads and wind loads are in the end generated according to active load code DS410 with the **Generate loads** option. Both the characte-

ristic and the ordinary loads, i.e. the characteristic value multiplied with the current load-reduction factor ψ are generated.

Automatic generation of loadcases



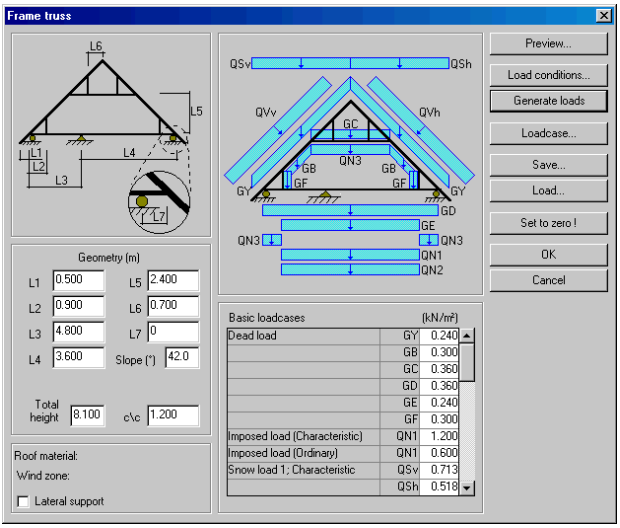
With the **Loadcase** option the program can automatically generate all necessary load combinations depending on what the user chooses from the box to the left:

Possible options are, according to DS409 5.2.8, load combination 2.1-2.3 for the ultimate limit state and load combination 1 for the serviceability limit state.

Lateral support

If this option is activated the top chord is assumed laterally supported the upper edge when a design is performed.

2.3.1.1.6 Frame truss



In addition to what has been described for **Truss** the following loads are stated for codedependent wizards:

Dead load

GB Addition for furnished part.

GC Self weight collar beams.

GD Self weight attic floor.

GE Addition for furnished part.

GF Self weight post wall.

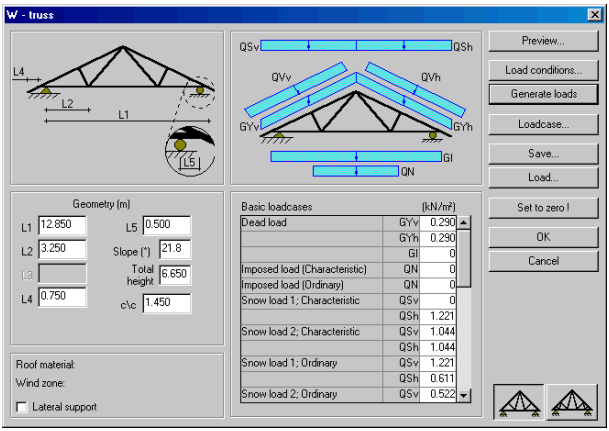
Imposed load

This is where **Imposed load** for rooms in residential buildings, dead and live part and load for attic space, live part are stated.

Generate loads

- **Danish code DS410**
  Imposed load are generated according to active load code DS410 with the **Generate loads** option. For all other loads, see **Truss** above.

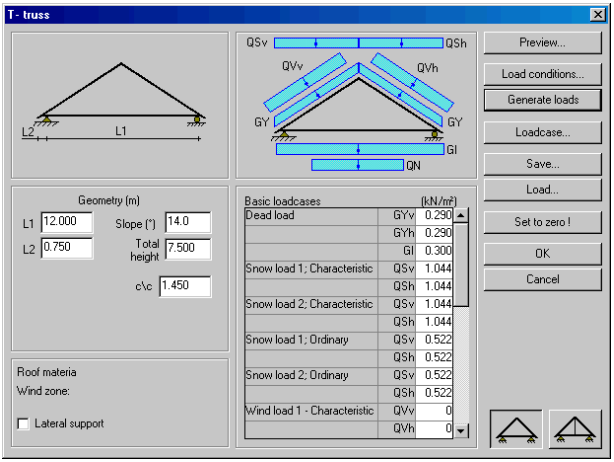
2.3.1.1.7 W-truss



All input is done according to **Truss** and **Frame truss**; see above.

W and WW trusses can be generated from this wizard by activating any of the bottom right buttons.

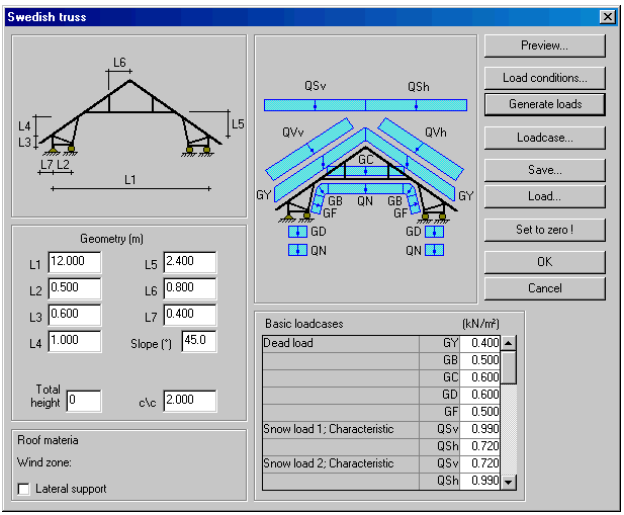
2.3.1.1.8 T-truss



All input is done according to **Truss** and **Frame truss**; see above.

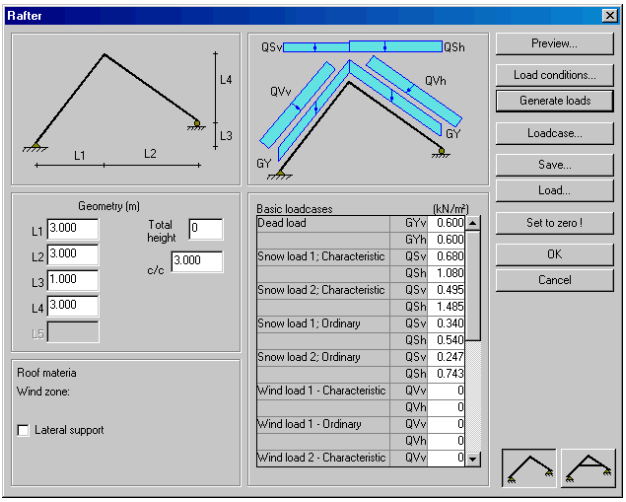
Two types of T-trusses can be generated from this wizard by activating any of the bottom right buttons.

2.3.1.1.9 Swedish truss



All input is done according to **Truss** and **Frame truss**; see above.

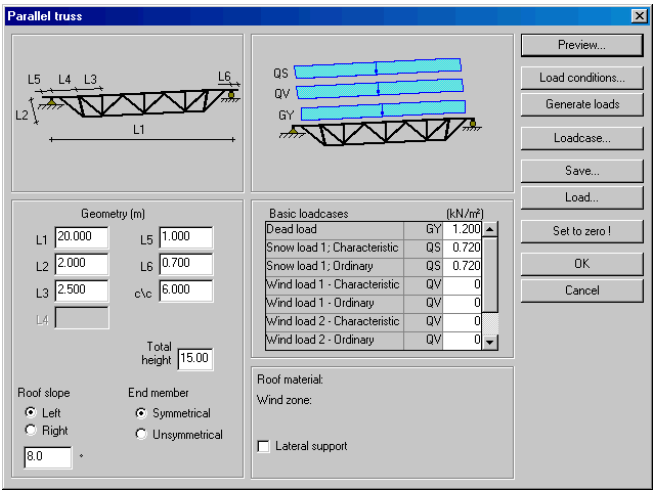
2.3.1.1.10 Rafter



All input is done according to **Truss** and **Frame truss**; see above.

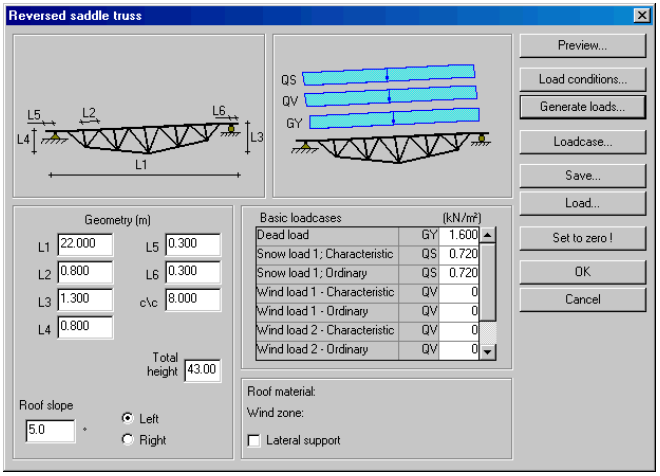
Two types of rafters can be generated from this wizard by activating any of the bottom right buttons.

2.3.1.1.11 Parallel truss



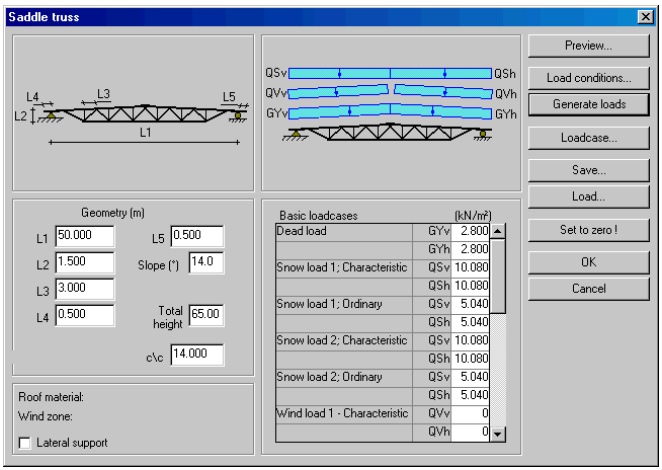
All input is done according to **Truss** and **Frame truss**; see above.

2.3.1.1.12 Reversed saddle truss



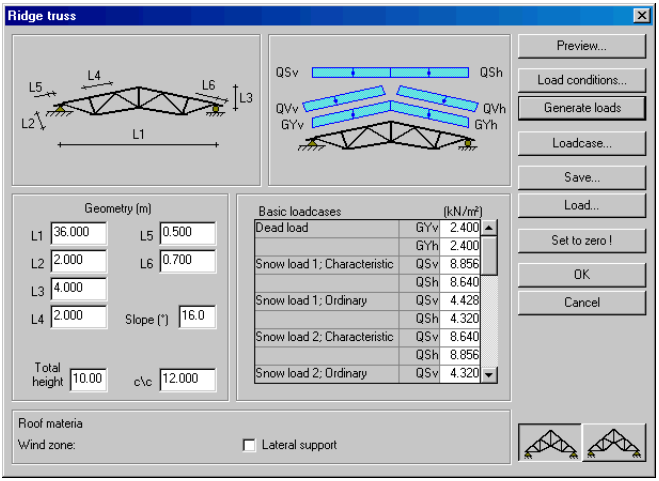
All input is done according to **Truss** and **Frame truss**; see above.

2.3.1.1.13 Saddle truss



All input is done according to **Truss** and **Frame truss**; see above.

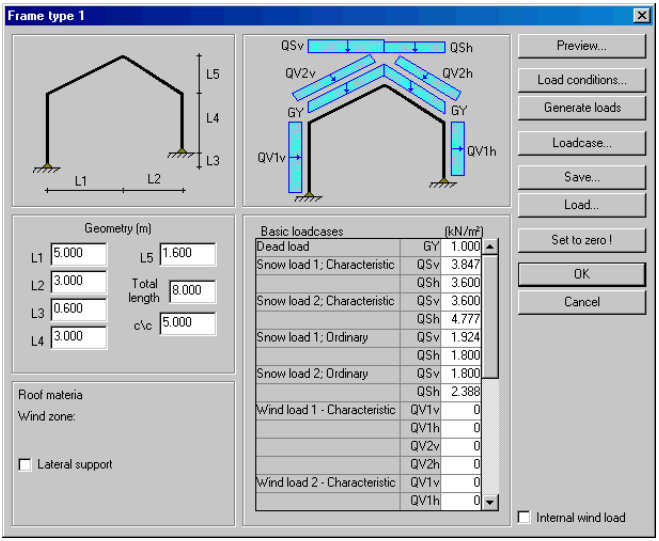
2.3.1.1.14 Ridge truss



All input is done according to **Truss** and **Frame truss**; see above.

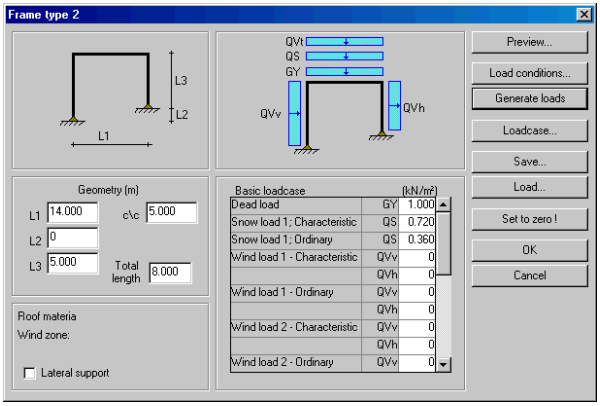
Two types of ridge trusses can be generated from this wizard by activating any of the bottom right buttons.

2.3.1.1.15 Frame type 1



All input is done according to **Truss** and **Frame truss**; see above.

2.3.1.1.16 Frame type 2

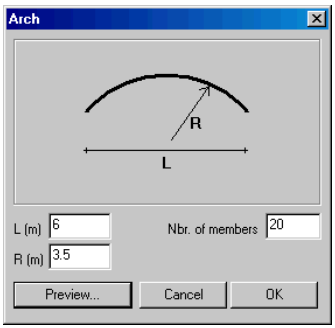


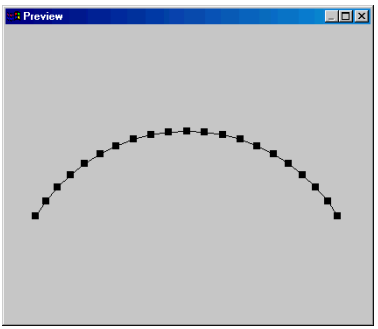
All input is done according to **Truss** and **Frame truss**; see above.

2.3.1.1.17 Arch

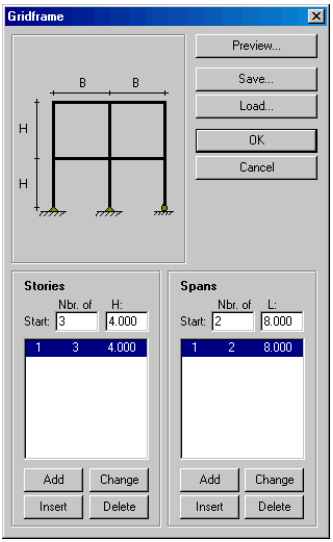
This is where the span and height of the arch and the number of members the arch is to be divided into are stated. The height of the arch has to be more than half of the span.

With the **Preview** option current geometry can be studied in the wizard. See the picture to the right:



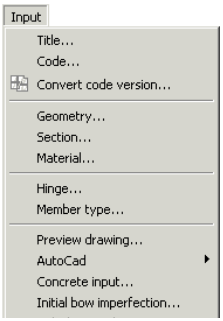


2.3.1.1.18 Gridframe



This is where number of stories and spans are being stated with their measures.

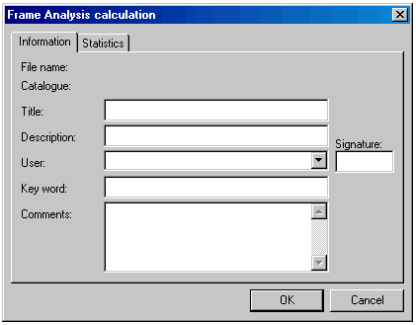
2.3.2 Input



The **Input** menu option makes it possible to state all the necessary input data needed to describe the current example. The options **Title**, **Code**, **Geometry**, **Sections**, **Material**, **Hinge**, **Member-type**, **Concrete input...**, **Initial bow imperfection...** and **Calculation adjustments...** can be found there.

These are being described in the following chapters. Graphic input is possible for geometry, hinges and loads with help from the toolbar, see 2.2.1.

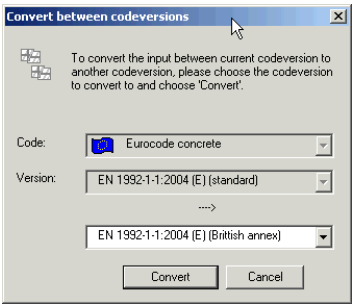
2.3.2.1 Title



The **Title** option displays a dialogue box where general information on the calculation in progress can be given.

The user can also receive statistics on the project.

2.3.2.2 Convert code version



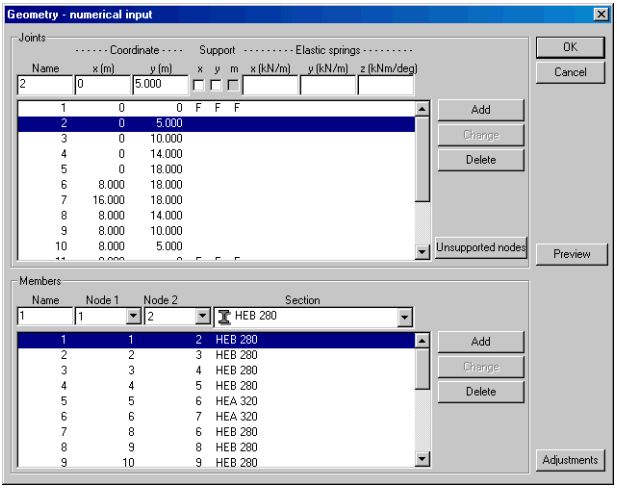
With this option the user can convert between EuroCode standard and the available annexes for different countries.

2.3.2.3 Geometry

The geometry of the construction can be defined in three different ways:

- Numerically, with help from the commands in the main menu, see above,
- Graphically, with help from the tools in the toolbar, see chapter 2.2.1.1,
- With help from predefined trusses, beams and frames that can be chosen in **Wizards** through **File** in the main menu or through the toolbar, see chapter 2.3.1.1 and 2.2.

The numeric input of the construction's geometry is done under **Input\Geometry**; see below.



In the **Geometry** dialogue box, the locations of the joints are defined numerically together with the members. Support conditions (joint information), sections and material (member information) are also stated.

On the input line for joint information, see above, coordinates, possible support conditions and spring stiffness are stated.

The user must first define the joint/member for each new added member and then press **Add**. When changing already defined joints/members the user must press the joint/member that is to be changed. The user must press **Change** to effect the changes. Press **OK** when the construction is defined. The defined topology will then be drawn on the screen. The topology for the above shown values will look as the figure below. Member information can be removed with **Delete**. The user must point at the current line in order to retrieve earlier defined values to the input line.

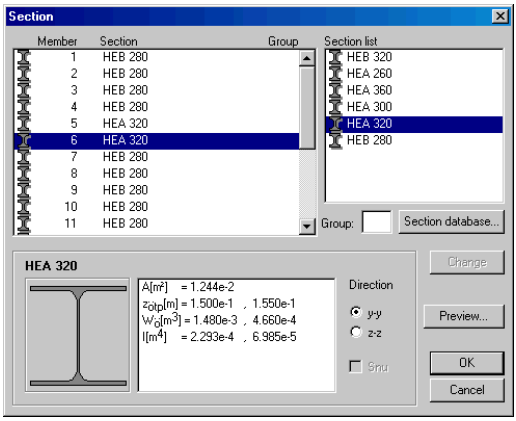
The supports are being defined by stating what directions are to be fixed. This is done by marking the squares under the **Support** headline for the current joint. The square is now marked with a cross and this is marked with an **F** in the geometry list. A prescribed displacement that doesn't equal zero is regarded as a load, see chapter on **Joint displacements**, 2.2.2.3. Either are all joints and members defined before the supports are inserted, or the joint and support are being defined at the same time. In the first case the current joint must be stated before the supports can be stated. This is done by clicking on the joint in the numeric menu and then state in what directions the supports are to be fixed. The

user must press **Change** to effect the changes. If the supports are being defined at the same time as the joint the user must press **Add** instead of **Change**. The support conditions of the construction can, just as the geometry for the frame, be defined graphically, see chapter 2.2.1.2.

The numeric input of springs is basically done the same way as it was for supports. The only difference is that the spring stiffness (in kN/m) has to be stated. Springs can be defined in the X-, Y- and Z directions (rotation spring in kNm/rad). The support has to be removed before a spring is to replace an already defined support. The support is basically removed the same way as it was stated. Spring supports can also be defined graphically, see chapter 2.2.1.3.

All section types defined are shown in the section list.

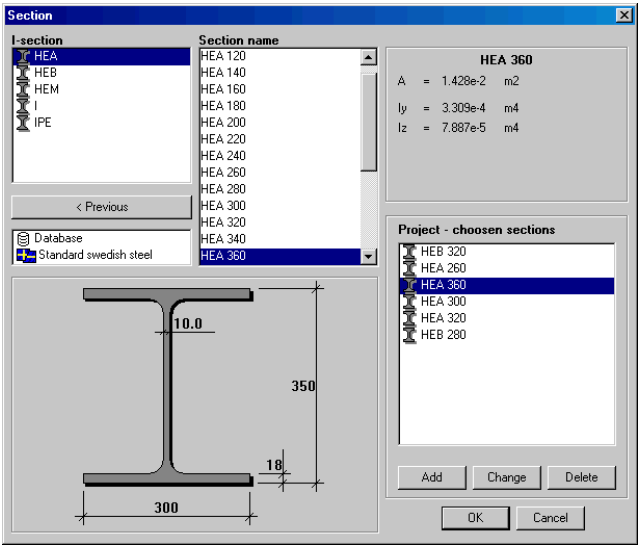
2.3.2.4 Sections



The members are allocated cross sections by choosing from available types in the section list.

Marking the member and then double clicking on the required section in the section list makes the choice.

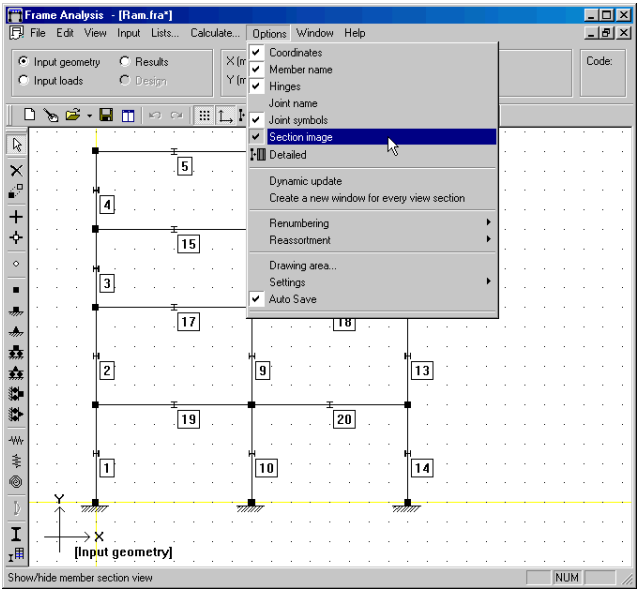
By clicking on the **Preview** button an image of the frame, with the current member marked in blue for a better survey, is displayed.



The section direction is displayed in the list and for the current section in the lower left corner. All sections are being placed in the **y-y** direction, i.e. with its stiff direction in the frame's plane, as default. If the user wants to change the section direction **z-z** can be stated for active sections.

Some sections can be rotated in 180° with the **Turn** option. By choosing **Change** these sections will change direction in the member list above.

The section list is being filled in with help from the **Section database**, which is opened by pressing this button.

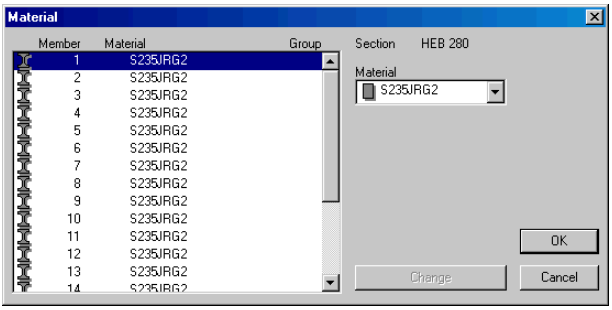


The database contains a variety of standard sections of steel- and timber profiles that are connected to different codes. Furthermore, **Non standard steel**, **Nonstandard timber** and **Non standard concrete** as well as **Numeric** and **Arbitrary** are always available. At the **Non standard** options the sections are being defined by stating applicable measures for the current type of section. At the **Numeric** option required geometrical values are being stated without being connected to a particular type of section. At the **Arbitrary** option, optional sections can be defined with a polygon. The input, to the list, of chosen section names is being done with the **Add** button, removed from the list with the **Delete** button and changed with the **Change** button.

Sections that are to be introduced to **Project-chosen sections**, which are edited by the **Add**, **Delete** and **Change** buttons, can be chosen from the section name list.

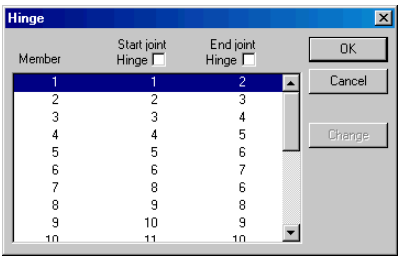
The section that is connected to each member is displayed graphically with the **Section image** option in the **Option** menu; see chapter 2.1.1.7.

2.3.2.5 Material



Materials are defined by choosing from a list of available materials that are dependent of the type of section that has been chosen for the member. For standard sections only code materials are available. Numeric material is also available for non standard, numeric- and arbitrary sections.

2.3.2.6 Hinges

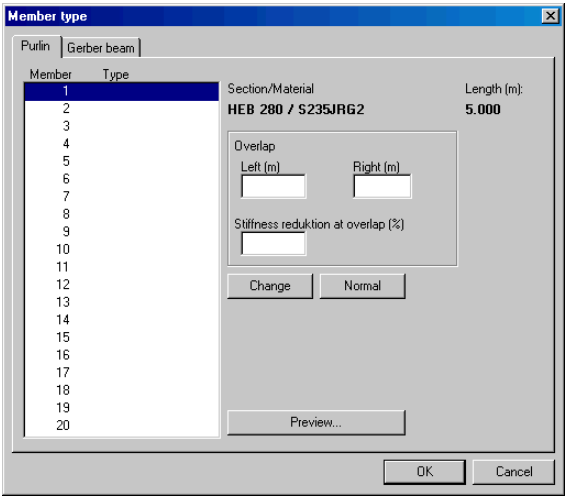


Members with hinge connections, where the hinges can take forces, in the **x**- and **y**-direction, but no moments can be stated numerically and graphically.

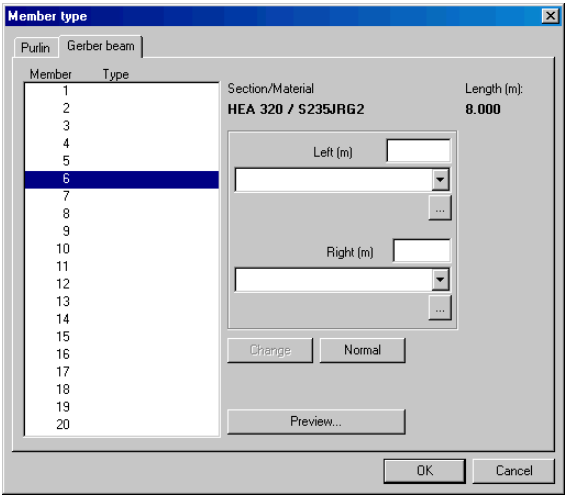
The numeric input is done under the **Input\Hinge** menu; see the dialogue box below. The member is allocated hinges by first marking the current member line, marking the current square with a cross, clicking on **Change** and then **OK**.

The hinges of the frame can also be defined graphically, see chapter 2.2.1.4.

2.3.2.7 Member type



Here it is also possible to define purlins and gerber beams, see 2.3.1.1.2 and 2.3.1.1.1.

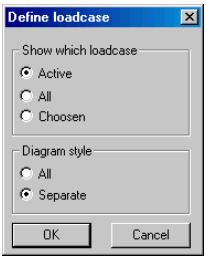


2.4 Results

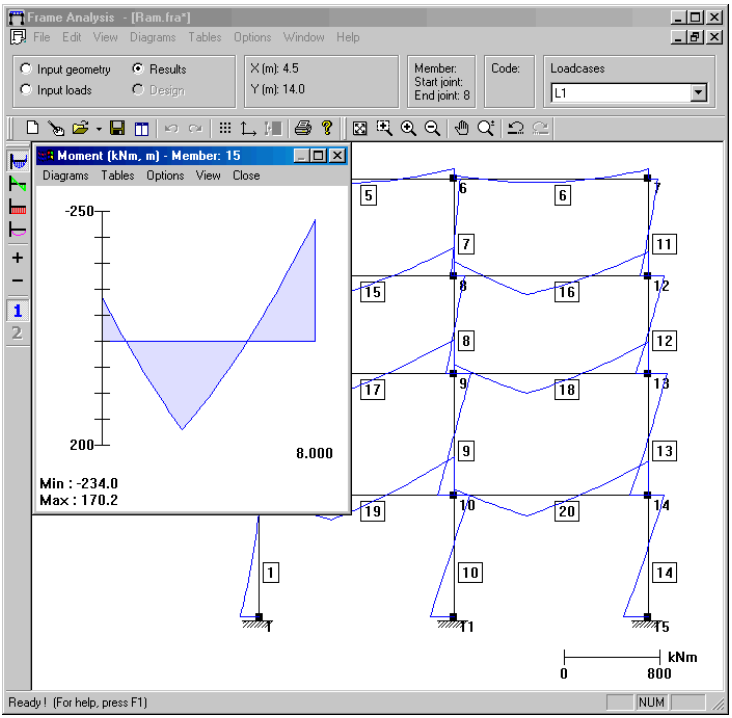
The **Results** option will be lit in the status field when the calculation has been completed. Then the moment diagram according to the first order theory for loadcase 1 will be shown. When the result window is activated the appearance of the main menu as well as that of the toolbox will change, see chapter 2.1.3 and 2.2.3.

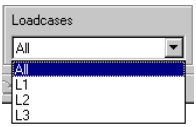
2.4.1 Diagrams

The **Diagrams** menu option in the main menu makes it possible to choose what kind of calculation result should be shown graphically. The **Loadcase**, **Moments**, **Shear forces**, **Axial forces**, **Deflections**, **Member**, **Increase scale**, **Reduce scale** and **Reset scale** options are to be found below **Diagrams**. Diagrams can be drawn for the whole structure as well as for particular members.

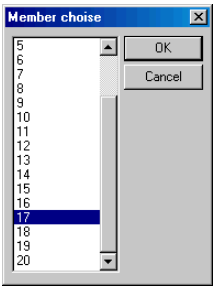


The user chooses what loadcase is to be displayed and if the curves are to be drawn superimposed or separately.



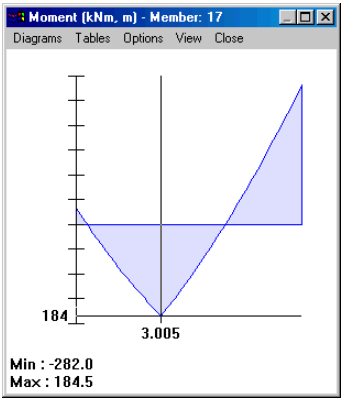


In the above picture, in the program-window's upper right part, there is a **Loadcase** window where the user chooses what loadcase is to be active, see to the left.



One way to show diagrams and tables for a particular member is to choose the **Member** option under **Diagrams**. This option displays the **Member choice** dialogue box where the user can choose a member.

The moment diagram of the selected member will be shown when the option has been made, see below.

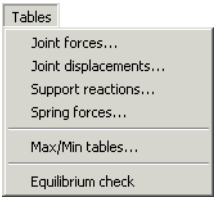


The easiest way to get diagrams for particular members is to double click on the current member. If the left mouse-button pressed down, a cross-air pointer, as well as the current value and co-ordinate, will be visible.

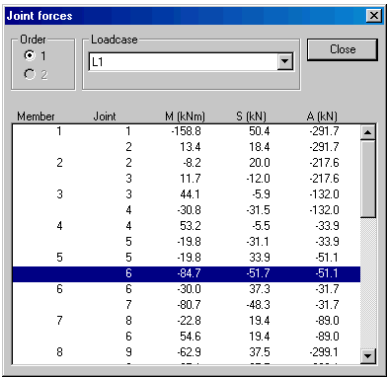
The **Increase scale** and **Reduce scale** options in the **Diagram** menu-option makes it possible to increase or reduce the scale of the diagrams.

Results can also be shown from the toolbar, which is located at the left side of the window. A more thorough description of these tools is to be found in chapter 2.2.3.

2.4.2 Tables



The **Tables** menu-option in the main menu makes it possible to have a calculation-result shown numerically in tabular form. The options **Joint forces**, **Joint displacements**, **Support reactions**, **Spring forces**, **Max/Min tables** and **Equilibrium check** are to be found under **Tables**.

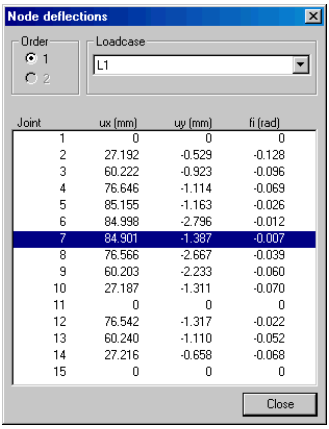


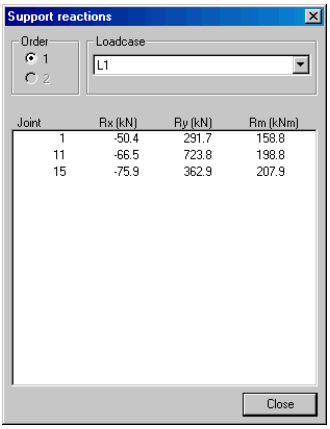
Each table can be chosen for the 1st or 2nd order theory as well as for each loadcase.

In the joint force table, the forces for every joint, i.e. **M** for moment, **S** for and shear force **A** for axial force are given.

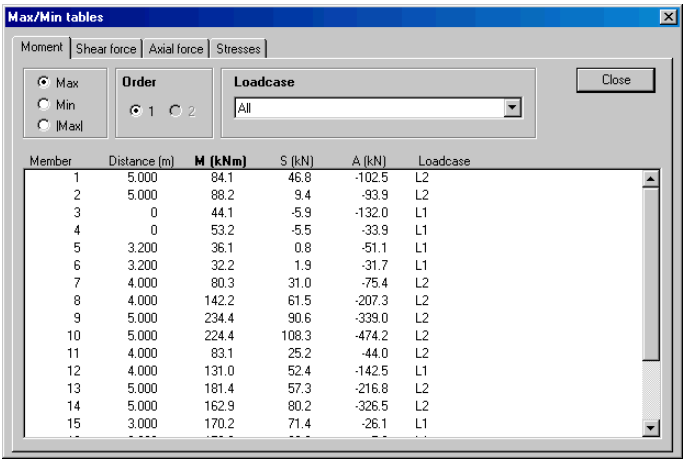
The table for joint displacements can look as below with displacements in mm and the torsion **fi** in radians.

These are given in the **x**- and **y**-direction as well as moments for supported joints in the table for support reactions.

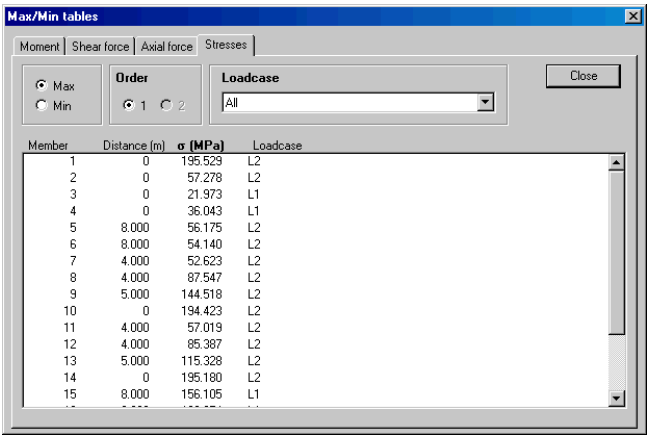


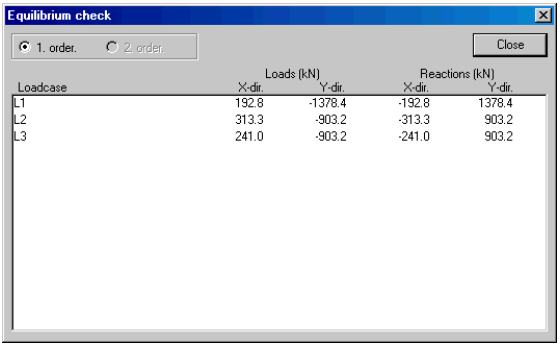


When having calculated several loadcases at the same time the user can receive tables with maximum values for moments, shear force, axial forces and stresses with the **Max/Min tables** option. When it comes to moment the following table is shown.



When it comes to stresses the following table is shown.



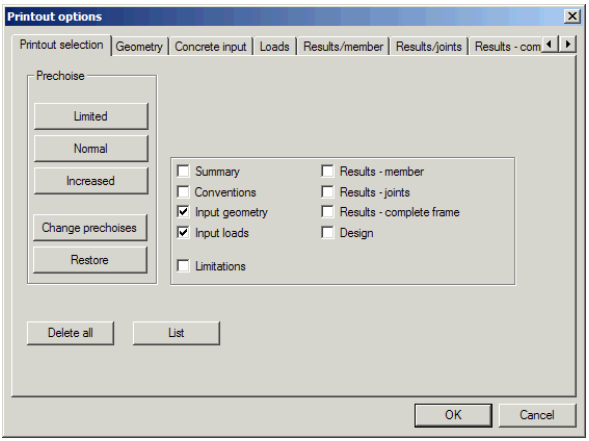


In order to check the calculated results, an **Equilibrium check** can be made for all loadcases. An example is shown below.

2.5 Printout

In order to make printouts the user has to choose **Printout options** under **File**. The **Printout options** menu option displays a dialogue-box with six flaps: **Printout selection**, **Geometry**, **Loads**, **Results/member**, **Results/joint**, **Results-complete frame**.

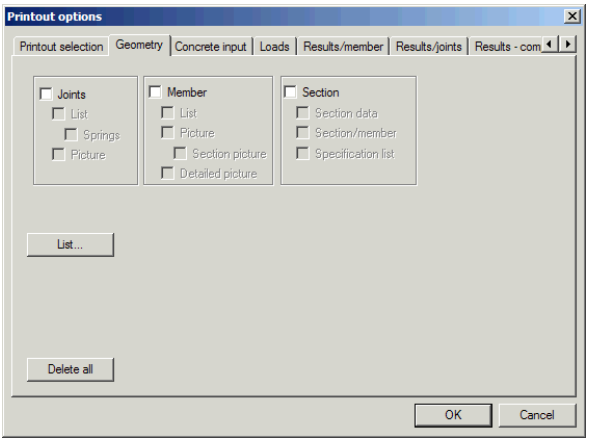
Printout selection



Here a selective choice can be done of the required printout so that the number of pages can be adapted to the user's wishes.

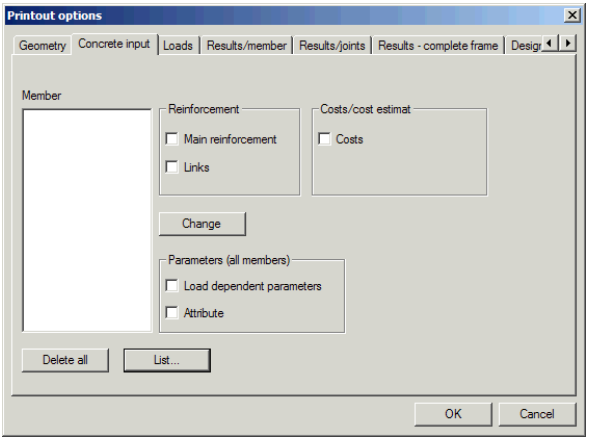
Under each flap there is a **Delete all** option which can be used to ensure that possible earlier options have been removed, and that only the options stated after having pressed on **Delete all** are those that are to be printed.

Geometry



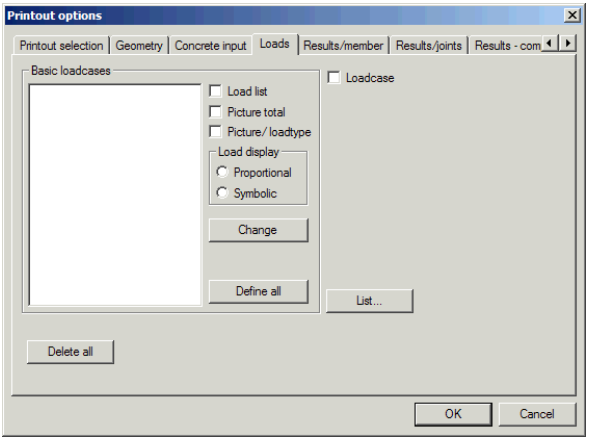
Here the user can choose the geometry-input data that is to be printed.

Concrete input



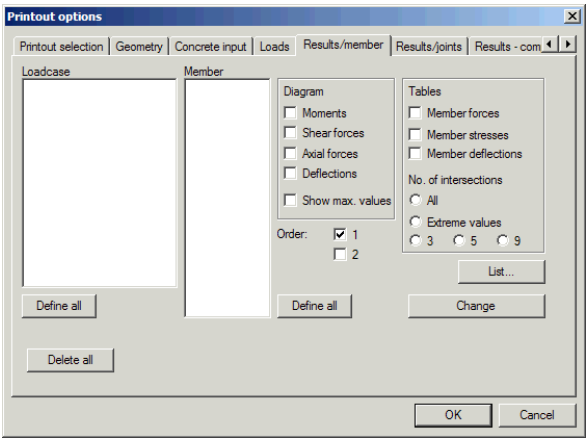
Here the user can choose the concrete input that is to be printed.

Loads



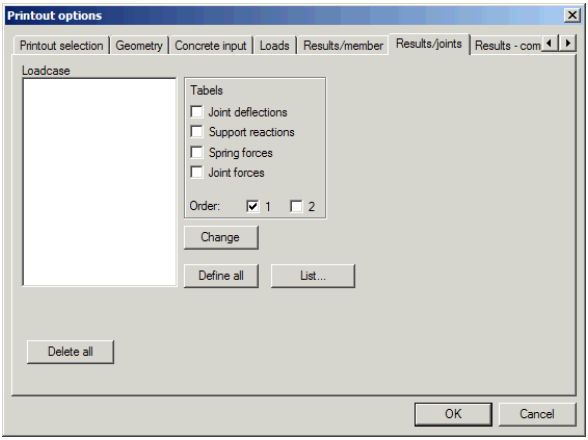
Here the user can choose the load-input data that is to be printed.

Results/member



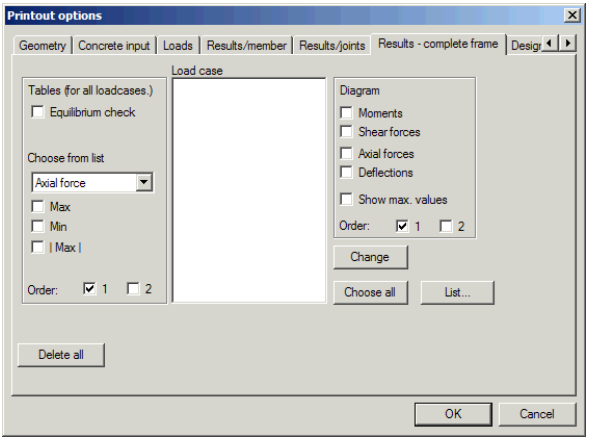
Here the user can choose what diagrams and tables are to be printed for each and every member.

Results/joint



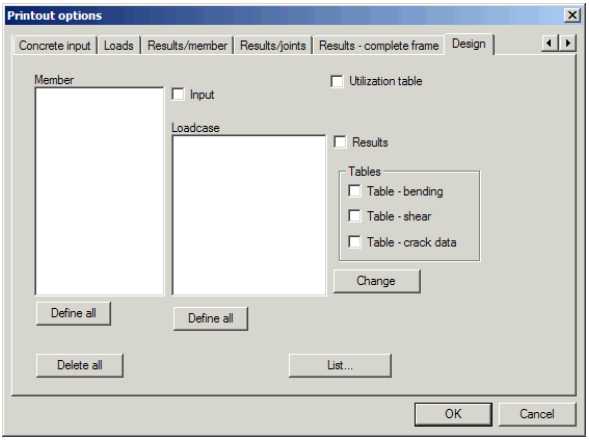
Here the user can choose the joint-result tables that are to be printed.

Results - complete frame



Here the user can choose what diagrams and tables valid for the complete frame are to be printed.

Design



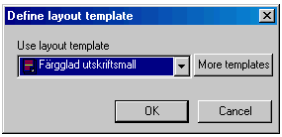
Here the user can choose the result that are to be printed.

Use **File\Print** on the menu line when all results that are to be printed have been chosen.

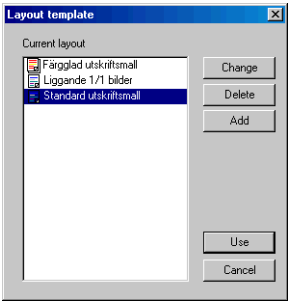
2.5.1 Print Manager

The function of the **Print Manager** is to enable the user to make a personal template for what the printout is to look like, the size of the pictures etc. or, alternatively choose the standard template that already exists within the program.

Choose **Preview** or **Print** from **File** or press the **Printout** button in the main window of the program.

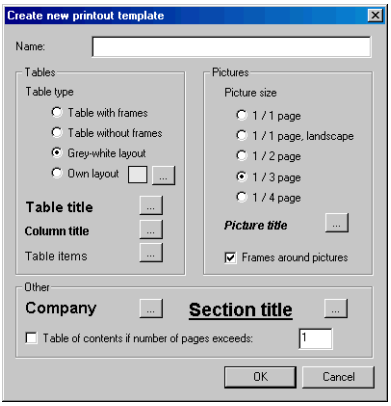


Then choose a printout template from the list. The program comes with a printout template called **Standard printout template**. Press the **More templates…** button to edit current printout template or to create a new printout template.



A list on printout templates that can be used will now be displayed. By clicking on the **Edit** button the chosen template can be changed to the user's requirements.

A printout template decides the layout of the printout, i.e. what font, colour etc. is to be used. In the example to the left the printout template has been named **Standard printout template**.



The printout template consist of the following parts:

Tables

A table can have the following appearances:

- **Tables with frames**
  Frames round the table where each table column is separated from nearby table column/columns with one or several perpendicular lines.

- **Tables without frames**
  Table divided into columns without separation lines.

- **Grey-white layout**
  Table without frames where every other table-post has been marked with a soft grey background colour to make it more distinct.

- **Personal layout**
  This function is the same as that for **Grey-white layout**, but the user can now choose the background colour himself. A suitable colour can be chosen from a colour palette by clicking on the **…** button.

  The user can set the appearance of the contents of the table. By clicking on the Table title button to the right of the text the font of the table title can be chosen from a font-dialogue box. This is also valid for the column title and the table-posts.

  Note that too big fonts can cause tables or any other objects to fail to fit in the page. If this should happen the table columns, or the width of the objects, will automatically be adjusted in breadth without any regards to text editing.

  If the object won't fit the page the **Print Manager** will try and create a new page hoping to find more space. This attempt can fail, depending on what personal adjustments the user has done, and the print manager will then cancel the printout generation and show an error message.

Pictures

The user himself can set the size of the pictures by choosing one of the following alternatives:

- **1/1 Page**
  Choose the **1/1 page** option if you want the pictures to be presented in a whole page size.

- **1/1 Landscape**
  Choose landscape if you want the pictures to be presented lying in a whole page size. Depending on earlier printer settings the **Print Manager** will switch from landscape for pictures to portrait for tables and other text material.

- **1/2 Page**
  Choose the **1/2 Page** option if you want the pictures to be presented in the size of half a page.

- **1/3 Page**
  Choose the **1/3 Page** option if you want the pictures to be presented in the size of a third part of a page.

- **1/4 Page**
  Choose the **1/4 Page** option if you wish the pictures to be presented in the size of a quarter of a page. This option is the most economically when thinking of the total number of generated pages. It is also the most preferable option if the user wants to obtain smaller printouts.

- **Picture title**
  Press the **…** button to the right of the text if you wish to change the font size of the picture title.

- **Frames round pictures**
  Mark **Frames round pictures** with a cross if all picture objects, that are to be printed, are to have a frame round them.

Other

- **Company name**
  Press the button to the right of the company name text if you wish to choose an appropriate font.

- **Section title**
  Press the button to the right of the section title text in order to change the text characteristics of the headings.

- **Table of contents**
  A table of contents can be obtained by marking this option. All generated objects, as well as the number of the page, on which the object has been printed, can be added to the table of contents.

  The user can, just to mention one example, set if he wants to obtain a table of contents for all printouts that have more than 20 pages. The table of contents is added at the end of the generated document.

3 Methods of calculation

This description of the methods of calculation is not meant to show the process of the program in detail, but serves to give enough information on how the calculation-result has been attained.

3.1 Analysis

Frame Analysis executes the frame calculations with the finite element method where the displacement method is used to determine displacements and stresses for the whole structure by using stiffness-relations for the individual members.

The construction is divided into members and joints. The joints are those points where the members are connected. The joint conception can however be perceived in a more strict mathematical meaning, as a discreet point in the pattern that describes the structure.

The members of the construction are first being analysed and general relations between forces and displacements of the joints are being set up, these are the stiffness relations for the members. The demands of continuity and equilibrium in the joints can then be formulated with matrix-expressions and the result will be a connection between joint forces and joint displacements for the whole construction, which is the stiffness-relation of the system. The connection can generally be written in matrix-form as:

$$\mathbf{K}\,\mathbf{a} = \mathbf{f}$$

Where **K** is a global stiffness-matrix, **a** is joint displacements and **f** contains the joint forces.

The course of the calculation can be summarised as:

- Define the problem, divide the structure into beam-members and joints and initiate support as boundary conditions.
- Form member equations, i.e. connections between forces and displacements for separate members.

- Assemble; this means that the member equations are being put into the equilibrium relations for the joints, which results in an equation system for the whole structure.
- The equation system is being solved with consideration to current boundary conditions. At this the joint displacements and the reaction forces will be known.
- Member equations and the now known displacements determine the member forces.

Consideration is taken to the alteration in the forces of the members, caused by deformation of the structure, when using the second order theory.

4 References

1. *Konstruktionsberäkningar med dator*, Studentlitteratur, Lund, 1990.
2. *Handboken Bygg*, Allmänna grunder, Avsnitt A30, Strukturstabilitet, Stockholm, 1985.
3. *Flexibiliteter och styvheter hos prismatiska balkpelare med konstant axialkraft,* B. Edlund, S. Wahlström och B. Åkesson, CTH, Inst. f Byggnadsstatik, Göteborg, 1979.
4. *Formelsamling i hållfasthetslära*, KTH, Inst. F. Hållfasthetslära, Stockholm, 1986.
5. *Theory of elastic stability*, S. P. Timoshenko och J. M. Gere, McGraw-Hill, 1961.
6. *Strength of materials*, S. P. Timoshenko, van Nostrand Company, 1947.
7. *Boverkets Konstruktionsregler BKR99*.
8. *Handboken Bygg*, Konstruktion, Avsnitt K18, Stockholm, 1985.