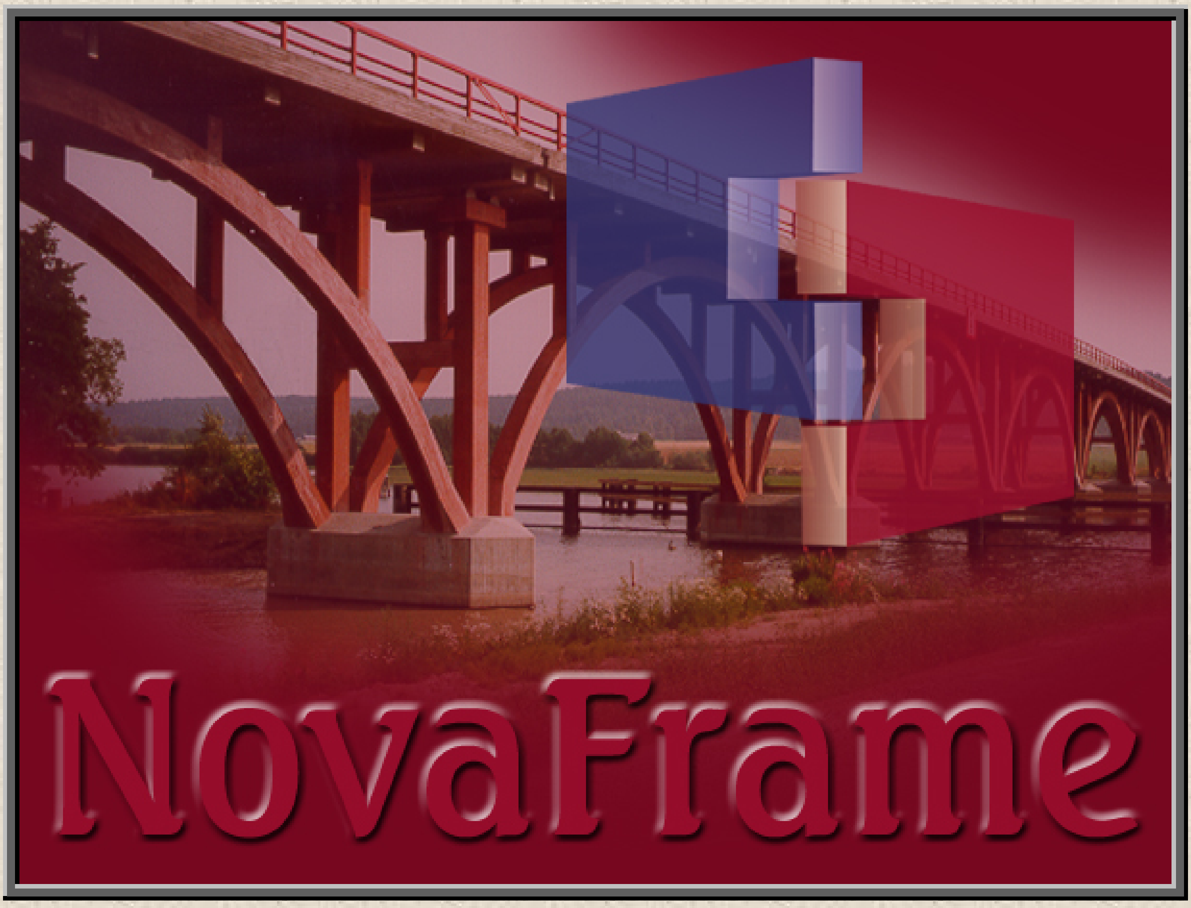
User's Guide

Version 5



AAS-JAKOBSEN

An analysis program for space frames

License agreement

Dr. Ing. A.Aas-Jakobsen A/S will not accept responsibility for technical, editorial or other errors or omissions in NovaFrame or in this NovaFrame User's Guide.

Dr. Ing. A.Aas-Jakobsen A/S will not accept responsibility for damages of any kind resulting from such errors.

Dr. Ing. A.Aas-Jakobsen A/S will not accept responsibility for the results from the analysis performed with NovaFrame. The user should always confirm the analysis results.

You must purchase one license for each of the computers with NovaFrame installed. Unauthorized use of NovaFrame will be prosecuted.

Table of Contents

Appendicies

1 Introduction

Nova Frame is a Windows-program for static and dynamic analysis of three dimensional frame structures. The program is based on the finite element method.

1.1 How to use this manual

This manual will assist you in using NovaFrame as an analysis tool, but it will also give you a more thorough description of the facilities that the program offers.

If this is your first session with NovaFrame, you should read this chapter and Chapter 2 before you start using the program. Chapter 8, Tutorial example 1, is an introductory example to show you how to use NovaFrame. You should work through this example before you start with your first real analysis problem. There are also several examples and tutorials in the appendices, which demonstrates some of the more advanced features.

Chapter 3 - 7 gives you more detailed information about the different built in facilities in NovaFrame. The different parts of this chapter should be read whenever you need more information about the relevant facilities.

1.2 Conventions used in the manual

The following conventions are used in this manual:

Window names and name of dialog boxes are written in ***bold italic*** style.

Menu choices and button names are written in **bold** style.

File names are written in *italic* style.

1.3 Program facilities

This version of NovaFrame offers the following facilities:

- No program limitation on maximum number of loads, nodes or elements, but your available memory could be a limitation. You should, however, not use load, node or element numbers that exceed 32767 (upper limit of integer value).

- Line geometry can be defined, i.e. straight lines, circles and clothoides. Nodes may be generated based on this line geometry. NovaDesign cross sections may be attached to these geometric lines called reference lines.

- General massive and panel sections are supported. Predefined sections and standard steel profiles.

- Sections defined in NovaDesign are available through the common database. Analysis properties of an element can be automatically calculated based on input in NovaDesign. Linear variation between sections at different stations on a reference line is supported.

- Tendons for posttensioned concrete can be modelled with automatic calculation of losses.

- Many types of static loads:
  Self weight
  Temperature
  Distributed load
  Concentrated load
  Concentrated moment
  Prestressing loads
  Trapezoidal load
  Gradient loads
  Hydrostatic pressure and soil pressure

- Traffic loads, which support the load description of various codes and regulations.

- Calculate redistribution of forces due to creep or relaxation.

- Account for 2.order effects by defining the axial force level in the structure.

- Calculate eigenvalues and eigenmodes of vibration.

- Calculate buckling modes and buckling lengths.

- Earthquake analysis.

- Stochastic wind analysis.

- Integrated load combination.

- ASCII-input of all input data including load combinations.

- ASCII-output of all available results.

- The graphic frame model is constantly updated on the screen as you build it.

- Several attribute options to customise the information included in frame model.

- Direct screen plot and printer listing of all available results.

- The program automatically creates an analysis document. You can customise the contents of this document yourself.

- Steel design, both stresses and utilisation including buckling effects.

1.4 Program and file system

Program files
The program is located in the file *NovaFram.exe*.
The on-line help is located in the file *NovaFram.hlp*.
Database functions are included in a file called *quadXX.dll*. (Where: *XX* is the DB version.)

Database files
The program works on a set of database files with the names **.gdd, *.gdh, *.gdi and *.gdn*. All your input data and actual results will be saved in these files when you choose **save** during a session of NovaFrame.

The database is common for several applications. Some of these applications are:

- NovaFrame
- NovaDesign
- NovaVR

Read more about the database in section 1.5.

ASCII- files
You can build your own ASCII input-files to generate input data. This is described in detail in section 2.5.

Each time you perform a save operation, a backup of all ASCII - input is made. The name of the backup file is **.bak*.

The selection of visible elements can also be done by an ASCII-file (**.sel)*.

You can also generate ASCII output-files of results (**.lst*) in NovaFrame. These files can then be used as input to design programs. An automatic generation of a result file containing combined section forces with the syntax needed in NovaDesign is also available (**.frc* -Section forces).

For detailed information on how to import and export ASCII-files, you should study section 2.3 Main program structure.

1.5 Common database

NovaBase
NovaBase is the name of the database system, which is used by NovaFrame. This database is available for other applications and several applications may share the data in this database.

When several applications are to work towards a common database it is necessary with an element of database administration in order to ensure consistency with respect to changes made in the various application.

The programs handle this administration but as a user it is useful to know some basic rules in order to understand the messages which occur when changes are made in one application and a modification is attempted by another application.

1. Several applications retrieve data from the database and no changes are made to the contents of the database: Date-Time (DT) is set in each application to the DT of the database. Database is available for all applications.

2. Application 1 starts to modify contents: Database is read only for all other applications.

3. Application 1 is finished modifying contents of the database and has saved the changes to the database: New DT is set in the database and in application 1 - the database is now available for other applications.

4. Application 2 intend to modify contents of database: Application 2 compares its own DT with the database DT. If application 2 has an older DT than the database, then application 2 must reload all relevant data from the database before changes can be performed.

2 Brief introduction to the user interface

This is only a brief introduction to the user interface. You should work through the example given in Chapter 8 to achieve a better understanding of the program facilities. We recommend, however, that you read through this chapter before you start with the example.

2.1 Main program principle

The first step in running NovaFrame is to open a document. This document is similar to a document you used in e.g. Word or Excel. When you perform an analysis in NovaFrame, information about your structure is included in the document and updated according to the changes you make.

You cannot, however, edit directly in the document, but you can customise the contents and set-up of the document so it fits into your analysis report.

All input data and results are stored in the database files (**.gdd, *.gdh, *.gdi and *.gdn*.) when you choose **save** during a session of NovaFrame.

The database files are common to several applications. These applications can share the information in the database. The main path of information sharing is as follows:

- Geometric cross sections assigned to reference lines in application NovaDesign. See sect. 3.8 for more details concerning reference lines.
- Geometry is given to the reference line in NovaFrame or from road geometry data defined by another application, the road definition will then be available from the database.
- Node co-ordinates can automatically be calculated based on the geometry of the reference line.
- Element stiffness properties are automatically calculated based on cross sections defined on the reference line
- Following an analysis and a load combination these forces are written to the database. When saved they are available for the concrete design in NovaDesign.

There are several methods and levels of information sharing between the two programs NovaFrame and NovaDesign:

- No information sharing
- Sharing cross sections by their numbers (Section on reference line, see sec. 3.8.)
- Sharing cross sections by their stations on a defined line with geometry

Sharing information with other programs is an option. The programs can perfectly well be used as stand-alone applications.

2.2 How to start the program

To start NovaFrame:

- Double-click on program icon

You can also <u>drag</u> the actual file from Explorer to NovaFrame if you have associated the extension first.

2.3 Main program structure

Main menu
Figure 2.3.1 shows the main menu in NovaFrame.

Figure 2.3.1 Main menu in NovaFrame

You will normally work from left to right on the menu during a session of NovaFrame.

File: Use the menu to open new or old documents. You can open several documents during the same run of NovaFrame. Each document is connected to one database-file and you can only work with one structure in each document.

The leftmost tool-button opens a new document. The second button opens an old document and the third button saves the generated input and results to the database-file.

You print the document from the **File** menu by choosing **Print** or you use the tool-button with a printer icon.

The documents are shown in the main window. You can scroll the document up and down and study the contents, but the figures will only be shown as framed blanks. Use **Preview** in the **File** menu to study the figures in the document.

Options: Use this menu to specify your general project data or customise the contents and set-up of your document.

Analysis: Use this menu to open the ***Frame window***, which is the window you work in when you build, solve and postprocess your structure. A more thorough explanation of this window is given below.

Window: This menu gives you different choices of how to arrange your documents in the main window. You don't need to worry about this menu if you don't run NovaFrame with more than one document open at the same time.

Help: This menu gives you information about how to use the program and how to use the help system. See section 2.4.

Frame window

Figure 2.3.2 shows the ***Frame window*** menu. This is the menu you use to build, solve and postprocess your structure.

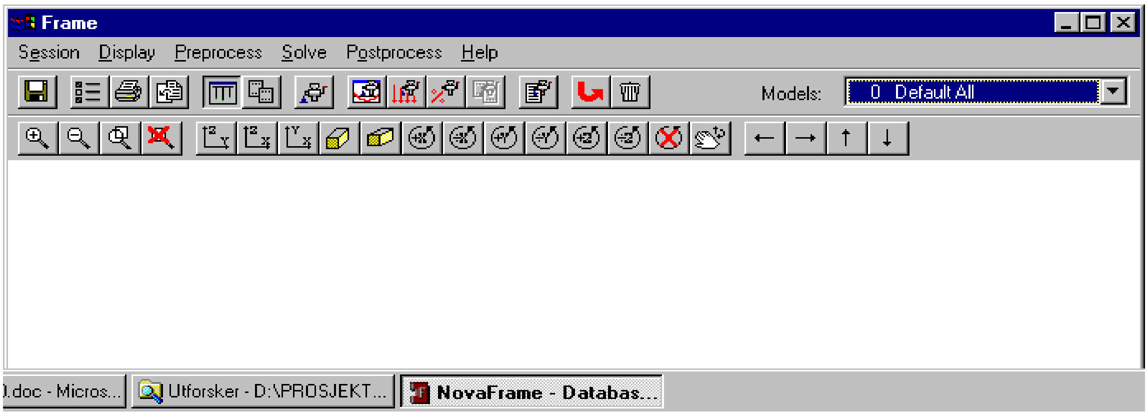


Figure 2.3.2 Frame window menu

Session: Use this menu to close window and save all data and results generated in this window to the database-file.

Display: This menu is shown on figure 2.3.2. Use this menu to select visible elements (**Visible elements...**), zoom or rotate the frame model (**View...**), customise the graphic attributes of the model (**Attributes...**), print the model (**Print...**) or copy the model into the analysis document (**Copy to document...**).

Preprocess: You open the input card file from this menu. You use the input cards to generate model geometry and load input. Input cards for defining reference lines and reference line geometry is also a menu item.

Solve: The actual frame analysis is started from this menu.

Postprocess: Use this menu to specify load combinations, plot analysis results or list analysis results.

Help: You open the help text for the ***Frame window*** from this menu

2.4 Help system

This section gives you a brief introduction to the help system included with NovaFrame. You will find a detailed description of how to use the help system if you choose **Using help** from **Help** in the main menu.

You should be an active user of the help-system the first couple of times you run NovaFrame. This is a quick way of learning to use the program effectively.

There are 3 different ways to use the help system:

1. Choose **Index** from **Help** in the main menu to get a list of the menu choices in the main menu and a list of the most used working tasks in NovaFrame. Click on the green words (hypertext) to get more information on a subject.
2. Press F1 or click on the **Help** button if you want help in the currently active dialog box.

3. If you work in the main window, you can press shift + F1. The cursor will then change to a question mark and you will get help on the next item you click on. You also achieve this by clicking on the rightmost tool button in the main window.

2.5 Text file input vs. dialog input

The model can be generated either interactively through input dialogs or from one, or more, text-input files. The two methods have different advantages:

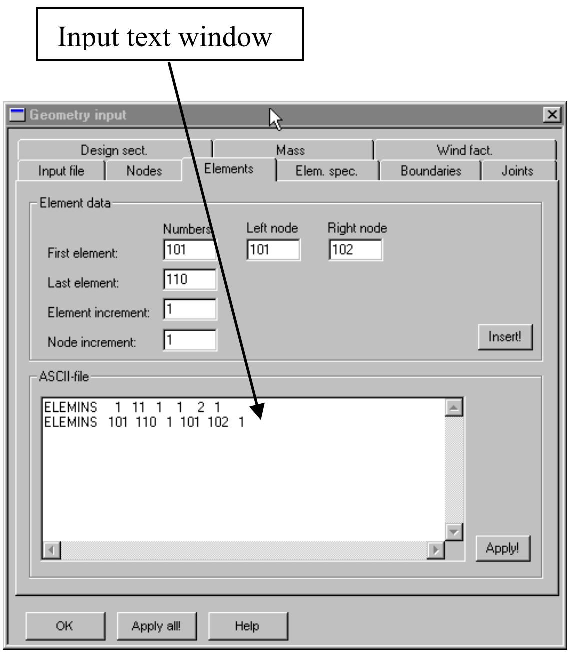
Dialog input:
- Usually faster for creating smaller input models
- The dialog gives an explanation to each input data field.

Text file input:
- Whole model can be generated from a single input file
- Comments can be added to the text-input file to explain the contents
- Aliases, mathematical expressions, DO-loops and IF-statements can be fully utilised.

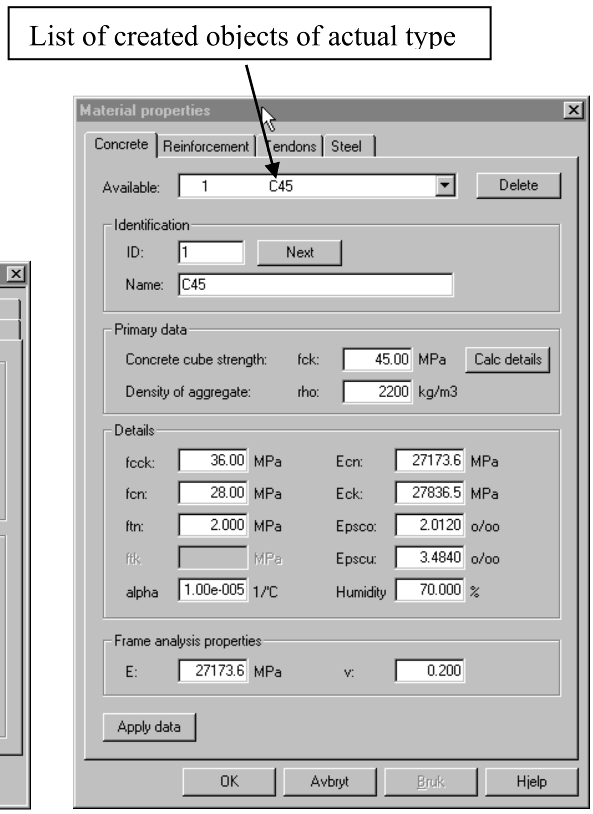
Dialog input:

The input dialogs are of two types (see figure below); one with a text window at the bottom that shows the input commands relevant for the actual input dialog, and one without this text window. In stead of the text window this dialog type has a drop list at the top which shows all objects of the actual type that have been created.

The two dialog types work slightly different. In the dialogs with text window you must first press the **Insert** button in order to load generate input commands from the values in the dialog input. You then press the **Apply** or **Apply all** button to execute. In the other dialog type, you press **Apply data** directly, after entering the correct values in the input fields.



Input dialog with text window



Input dialog without text window

Text file input

This method is available for those who prefer to work with the text input in an external text editor.

In order to execute the text input, select the menu option **Read input file…** which is available from the main windows. This will open the ***Read input file*** dialog (see figure below). The existing model is by default deleted before loading the new input.

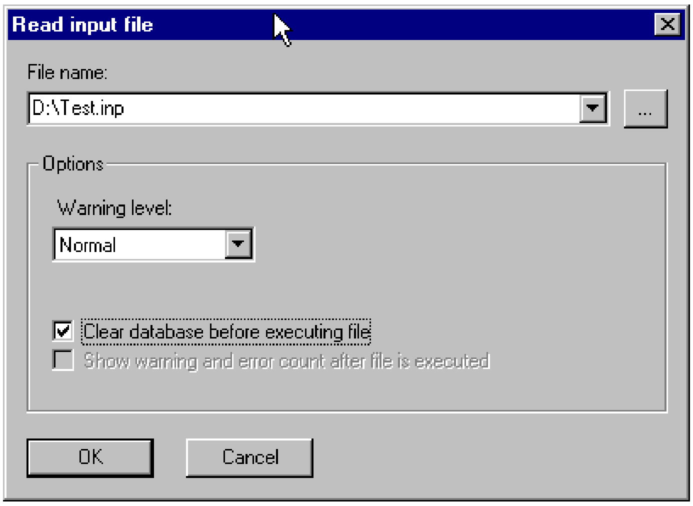


Fig. 2.1 Read input file- dialog

The input may also be loaded into input dialogs with an ***Input file*** card, see figure below. Only the input command lines of the valid type for the actual dialog are loaded from the specified input file. For instance, only reference line commands are loaded into ***the Reference Line*** dialog. The input must be executed by pressing the **Apply All** or **Apply** buttons in these dialogs.

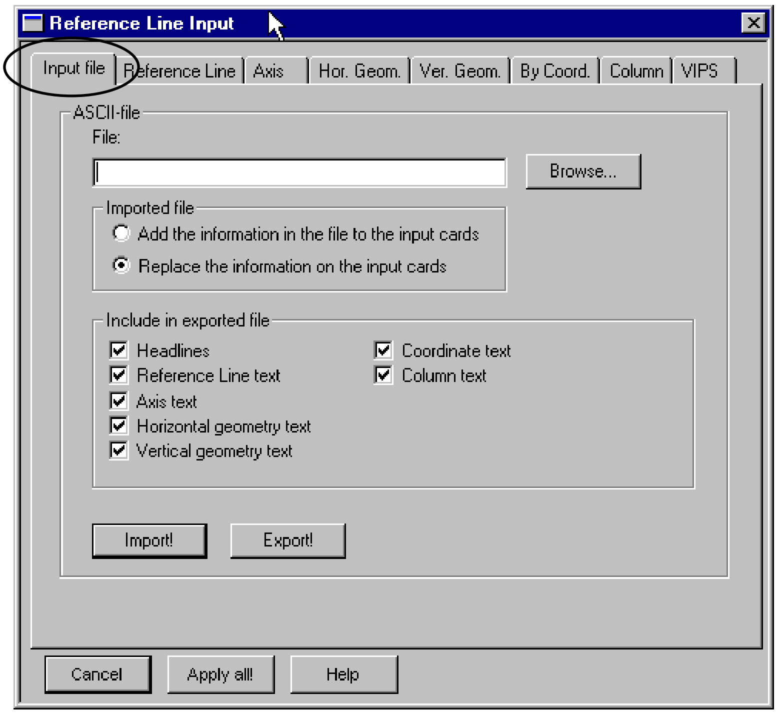


Fig. 2.2 Example of dialog with an Input file *card*

3 Materials and design parameters

3.1 General

In general, all cross sections should be assigned a material type and a corresponding design parameter set. The material data are used in both the frame analysis and in design calculations. Design parameter data are, as the name reveals, primarily used in design calculations. A few cross section types, which are not intended for design calculations, are excepted from this rule. The table below presents what is the required input for the different cross section categories.

CATEGORY	MATERIAL AND DESIGN PARAMETER SET	YOUNG'S MODULUS AND POISSON'S RATIO	NO MATERIAL OR DESIGN PARAMETER
MASSIVE - GENERAL	X		
- PREDEFINED	X		
PANEL - GENERAL	X		
- PREDEFINED	X		
CATALOGUE PROFILE	X		
ANALYSIS SPECIFIC		X	
SPRING			X

The different cross section categories are explained in more detail in chapter 5 - Cross sections.

Valid materials are concrete and steel. In addition reinforcement steel and tendon steel are separate material categories with corresponding design parameter sets.

3.2 Concrete

Material data

A parabolic stress-strain relationship for the stress-strain relationship is applied according to NS 3473, as shown in the figure below. This relationship is also applied for the Swedish design code BBK 94, with some modifications.

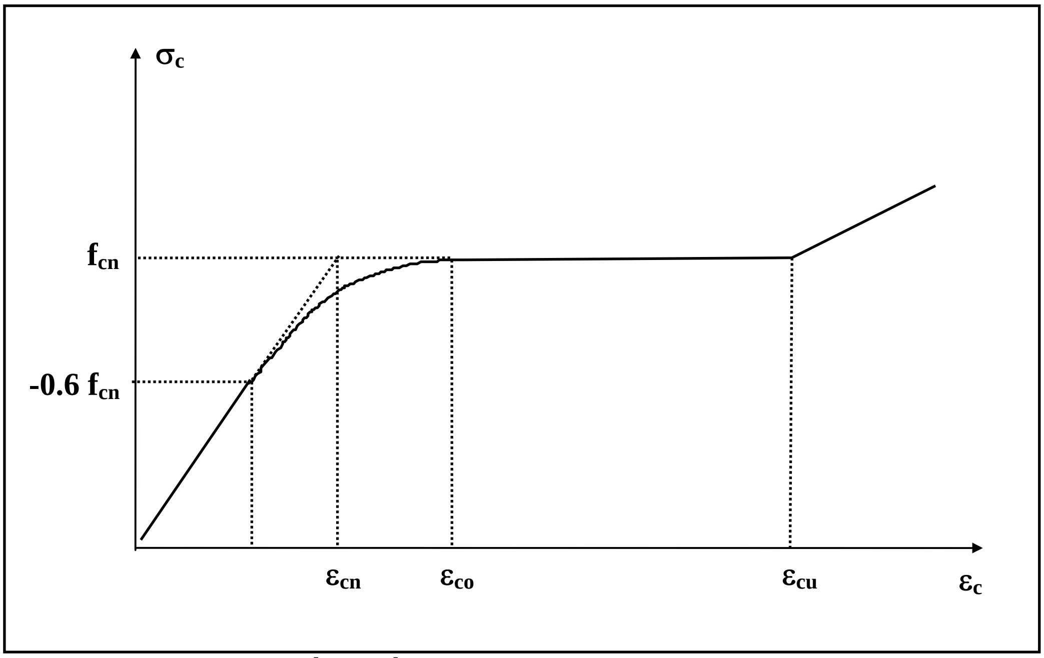


Fig. 3.1 Stress-strain relationship - Concrete

The key input values for giving the concrete material properties are:

NS3473: Compressive cube strength (f_{ck}), and the aggregate density (ρ).
BBK94: Concrete grade (hållfasthetsklass), and the aggregate density (ρ).

Based on these values, all parameters required to define the stress-strain relationship can be calculated.

The effect of aggregate density is included in accordance with. NS 3473 section 9.2.2 and 11.1.1. Normal density concrete aggregate should have a density of 2200 kg/m^3 (NS3473) or 2400 kg/m^3 (BBK94, NS3473 4.th edition), which are the default values.

If the concrete material is assigned to a cross section used in the frame analysis, both a representative Young's modulus and Poisson's ratio are required. These values are by default taken as:

Young's modulus	Eck
Poisson's ratio	0.20

These values can however be changed independent of actual concrete grade.

Design parameters

The design parameters for concrete are total material safety factors and strain limits.

The strain limits are meant as allowable strain in the design of the cross section. They are normally set to the ultimate strain limit (ε_{cu}) of the actual concrete grade.

Note!
If you change concrete grade for a cross section, the strain limits in the design parameter set for this cross section is not updated automatically. Therefore, remember to do this manually. This is especially important if you go to a higher grade, which has a smaller ultimate strain limit. The design strain limit may than become smaller that the ultimate limit, if left unchanged!

The total material safety factors are design code dependent. In BBK94 it consists of three factors; η, γ_n and $\gamma_{m.}$

The safety class factor γ_n (BBK94 section 1.1.1.4) should, however, not be included in the material safety factors. This factor is included automatically based on selected safety class, which is a section design parameter input (the default is safety class 3 which implies γ_n =1.2 in ULS conditions).

Section 2.3 of BBK94 suggests a smaller value for the product $\eta\gamma_m$ for Young's modulus than for concrete strength. You should enter the total safety factor corresponding to concrete strength, and the program will make the appropriate scaling of the safety factor for the Young's modulus.

3.3 Steel

Material data

A linear stress-strain relationship is used for construction steel, as shown in the figure below. The main input is the characteristic yielding strength (f_{sk}) and the Young's modulus Esk

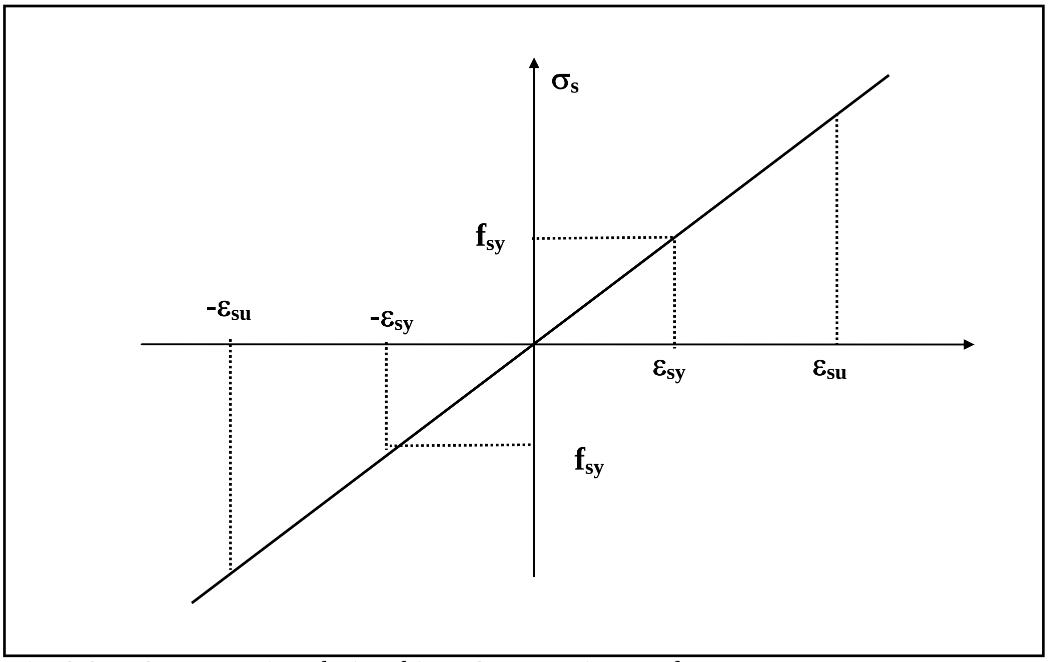


Fig. 3.2 Stress-strain relationship – Construction steel

Several predefined steel grades are available from the input dialog and from text input. The yielding strength is constant, and therefore independent of plate thickness.

Design parameters

The design parameters for steel are total material safety factors and strain limits.

The strain limits are meant as allowable strain in the design of the cross section.

3.4 Section design parameters

Section design parameters for concrete sections:

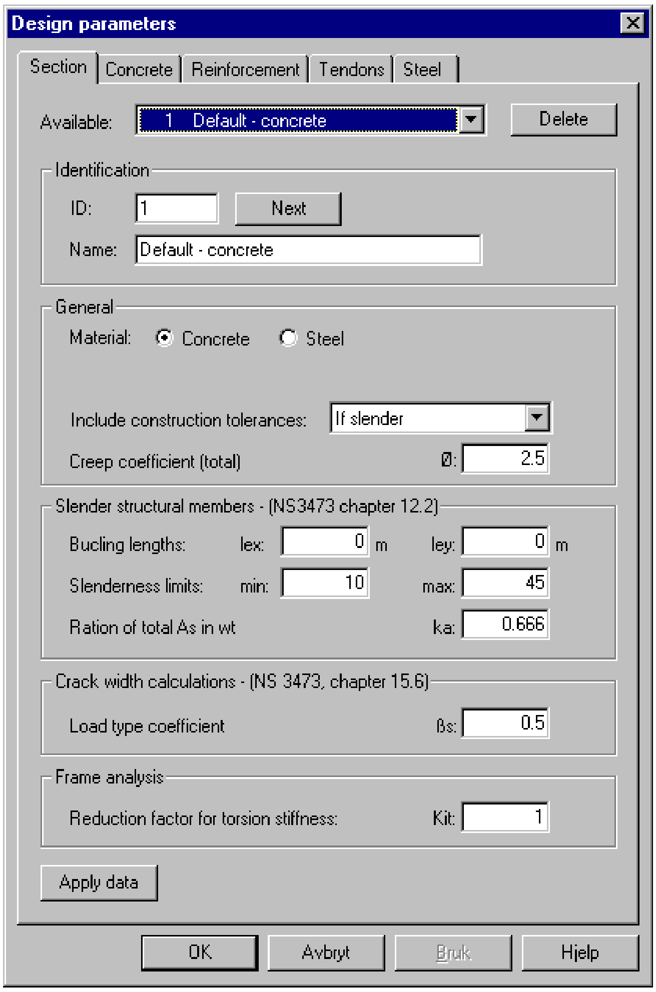


Fig. 3.3 Section design parameter input dialog - concrete

Construction tolerances:
According to NS 3473 construction tolerances should be taken into account for both ULS-(section 12.2.1 and 12.2.3) and PLS limit state (section 14.1.1). If the applied axial force is compressive then the eccentricity e_{0i} due to construction tolerances will give additional bending moments which should be added to the applied bending moments.

In the design calculations the additional moments due to construction tolerances are always included in slenderness calculations, but it is optional to include them for non-slender structural members. You can set your selection for this option in the section design parameter input. In the ***Frame input*** window select **Preprosess...** – **Design par…**

There are two options available;

- 'If slender' Include additional moments due to construction tolerances only if the actual structure is slender (for ULS or PLS, buckling length(s) given)
- 'Always' Include additional moments due to construction tolerances(for ULS or PLS)

Note!
Default value is 'Always'

Slenderness effects:
For slender structure cross sections loaded with a compressive axial force, additional bending moments due to slenderness effects should be included according to NS 3473 section 12.2 and A12.2

The program will calculate and include additional moments in design calculations if;

- The limit state for actual load combination is ULS or PLS
- buckling lengths are given (about one or both cross section axis)
- the axial force is compressive
- the 'Include slenderness' option is activated in the design setup

If you have a frame analysis model, more accurate buckling lengths can be calculated by NovaFrame when performing a buckling analysis. You may however give buckling lengths associated with the actual section by using section design parameters. The buckling lengths and other slenderness specific input data are section design parameter input. In the ***Frame input*** window select **Preprosess** – **Design parameters…**

If you don't have a frame model and wish to perform a buckling check you must therefore evaluate the size of the buckling length based on your understanding of the behavior of the actual structure, or take them from another structural analysis which includes calculation of buckling lengths, before entering them as input in NovaFrame/NovaDesign.

Please refer to the NovaDesign manual for more complete information on the design procedures.

Note!
Reduction factor for torsion stiffness may be entered. Default value is 1.0 which implies that the torsion stiffness will be:

$$1.0 \cdot G \cdot I_t = E \cdot I_t / 2(1+\nu) \approx 0.3 \cdot E \cdot I_t$$

Section design parameters for steel sections

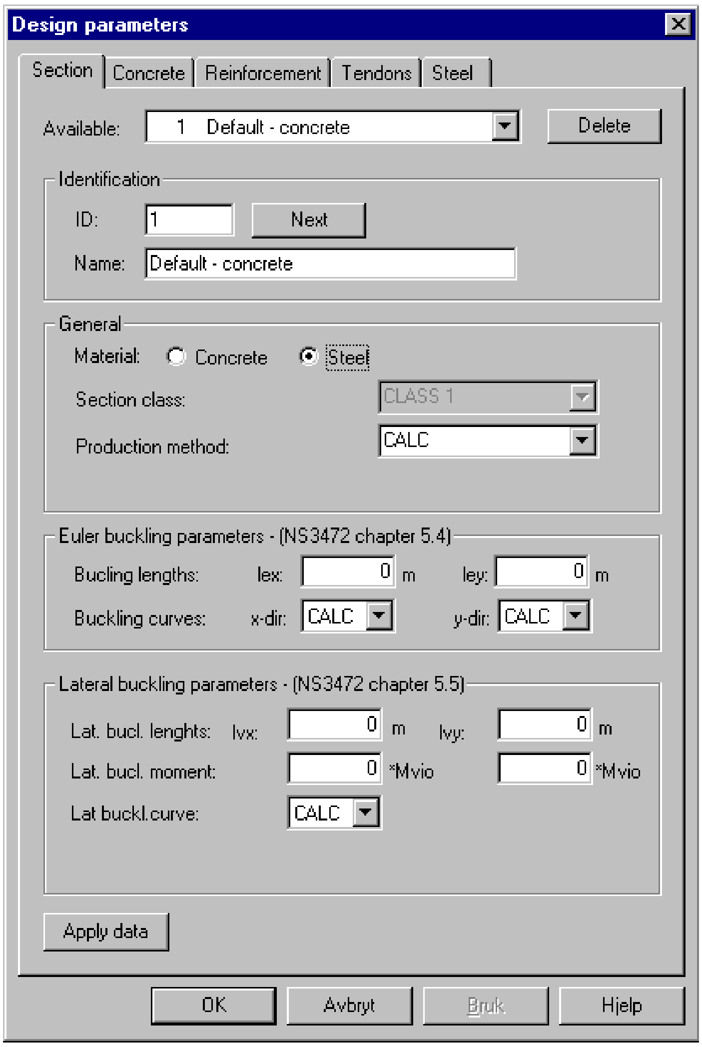


Fig. 3.4 Section design parameter input dialog -steel

The information defined in the section design parameters is mainly associated with the buckling properties of the cross section. You may define buckling lengths and buckling curves associated with the cross section.

Properties both for Euler buckling and lateral buckling is available input.

The buckling curves are calculated based on the cross section type and input "production method". Please refer to table 11 in NS3472:2001.

You may give your specific buckling curve/lateral buckling curve.

The buckling lengths and other slenderness specific input data are section design parameter input. In the ***Frame input*** window select **Preprosess – Design parameters…**

4 Reference lines

4.1 General

Reference lines are important in the communication of data between NovaFrame and NovaDesign. The idea of a reference line is to introduce the concepts of roads and railways more directly into the design process and to reduce the work of generating geometry input data.

Because this subject is extensive, the presentation of reference lines has been moved to a separate appendix: *Using Reference lines* (App2_RefLines.pdf). Here you will find a thorough introduction of the concept and use of reference lines. This appendix also explains how reference lines are used in the interaction between NovaDesign and NovaFrame.

5 Cross sections

5.1 General

You define cross section geometry and springs in the ***Cross section*** input dialog. This dialog is available in both NovaFrame and NovaDesign.

Each cross section or spring a must be given a unique section number. For cross section on reference lines other than reference line 0, this number is also the actual position along the line.

- Reference line 0: Section number = identification
- All other reference lines: Section number = position along reference line [m]

Cross section are connected to the beam elements with the command ELEMSPEC (in the ***Element specify*** input card). The section number in this command refers to those defined on reference line 0. The concept of reference lines and how it links cross sections to the element model is further described in appendix 2; *Using reference lines.*

Before you start modelling cross section geometry you should decide whether you need an exact representation of the cross section for steel or concrete design checks, or if you just need cross section properties for the frame analysis. Several cross section types are available in the program. The different types are presented in the following chapters.

If you are going to build your frame model based on reference lines you must make sure that you know where the intersection point of the reference line is located in the cross section plane. By default the intersection point is the centre of gravity.

5.2 Cross section types

A cross section is created with the XSECT command. In this command you are asked to enter the cross section type. Several cross section types are available. They can be divides into two main categories; massive and panel sections. Both user-defined (general) and predefined shapes are available for both categories.

If no design calculation is required, you may omit the more detailed cross section input by selecting a simple cross section type from the category; ANALYSIS SPECIFIC.

Springs are applied in the frame model as beam elements assigned a special cross section type with defined spring stiffness constants. SPRING is therefore the last cross section category.

The table below gives a summary of the available cross sectional types. Each type is discussed in more detail in the following chapters.

CATEGORY		CAPABILITIES		
		BEAM ELEMENT CROSS SECTION	STEEL DESIGN	CONCRETE DESIGN
MASSIVE - GENERAL	MASSIVE	X		X
- PREDEFINED	MASSIVE	X		X
PANEL - GENERAL	PANEL	X	X	
- PREDEFINED	PANEL	X	X	
CATALOGUE PROFILE	PANEL	X	X	
ANALYSIS SPECIFIC	-	X		
SPRING	-	X		

Table 5.1 Available cross section types

5.2.1 General massive

In order to create a general massive cross section, the value in the 'type' field in the XSECT command must be GENERAL (or the value 13).

The general, user defined, section geometry is entered as a series of section points defining the section surface. A continuous line interconnects the section points. The cross section area is automatically taken as the area enclosed by this continuos line. No additional area input is therefore requires as for the panel sections (except shera or torsion areas which is an optional input). A curved (circular) face can be entered as a radius (radius point) between two consecutive coordinate points. Each section point, except radius points, has an identification number for later reference

The ascii command for entering cross section points is PT

It is required that the section points are entered in the correct order. The basic rules when entering the section points are;

- The first and last section point must have matching coordinate values.
- The section points must be entered in a counterclockwise direction.
- A void in the section may be specified by moving in the opposite direction, i.e. in clockwise direction
- You can not start or end with a radius point
- Corresponding section points when going from outside to inside of the cross section must have matching coordinate values (section point 4 and 10, and section point 5 and 9 in the figure below)

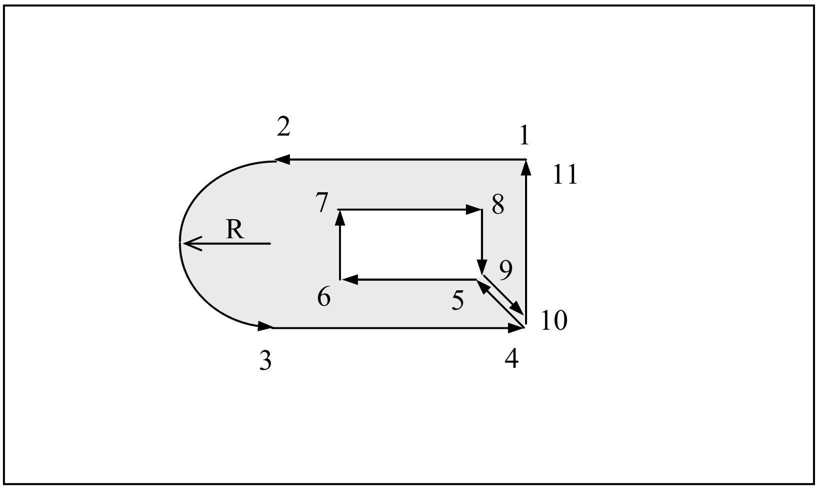


Fig. 5.1 Massive general cross section – section point input direction

Note!
The torsion stiffness is taken as Ix + Iy. Use the scaling factor k_{It} in the section design parameter if this default value is not acceptable.

5.2.2 Predefined massive

Cross section type specified in the XSECT command must be RECT (11) or CIRC (12) .

Available predefined shapes for massive cross sections are rectangular (or box) and circular (or tubular). If flange and web thicknesses are given, the rectangular section becomes a box section. If wall thickness is given for the circular section, it becomes a tubular.

The cross section measures such as width and height etc are entered with the ascii command DIM. All meassures are given in [mm]. The figure below shows the available predefined panel section types

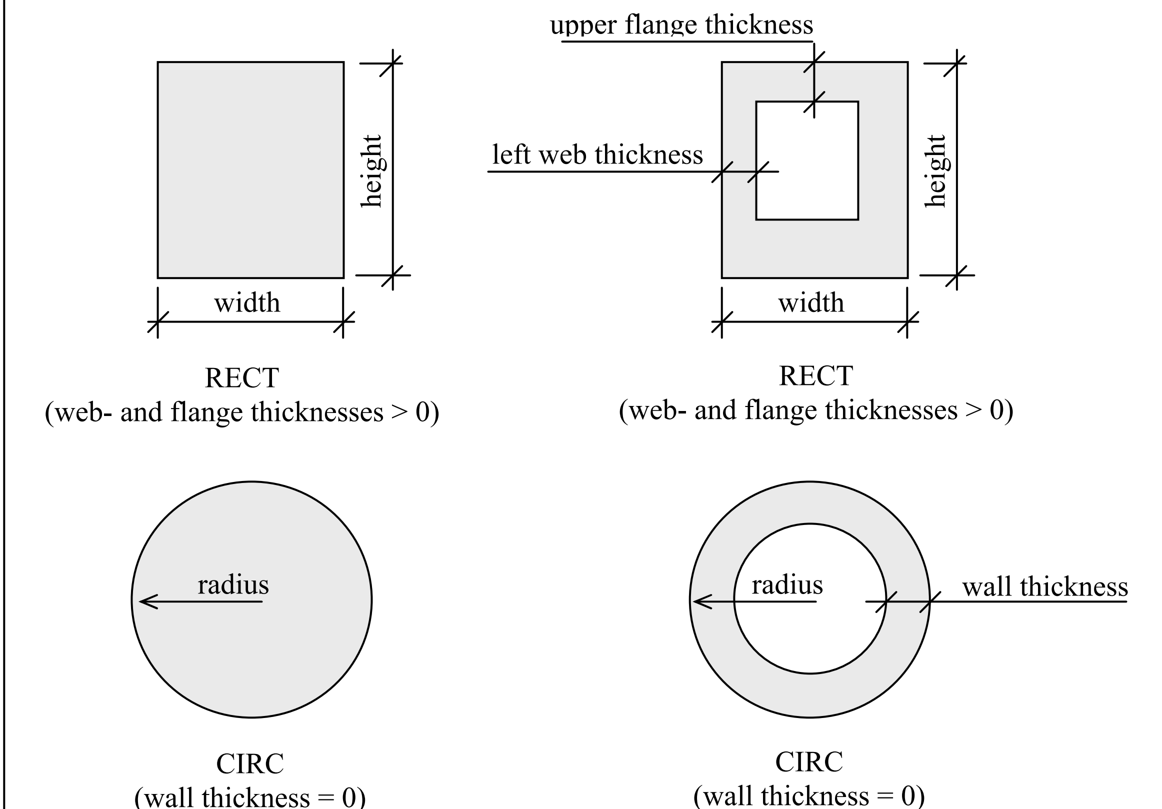


Fig. 5.2Available massive predefined shapes - dimensions

5.3 Panel sections

5.3.1 General panel

In order to create a general panel cross section, the value in the 'type' field in the XSECT command must be PANEL (or the value 14).

Creating general panel sections is divided into two steps:

1. Enter all cross section points required using the PT command
2. Create panel elements between specified section points using the PANEL command

This is analogous to the principle of node-element connection in the frame model. Each panel stretches between two section points.

The panels have a constant thickness. The thickness is given in mm.

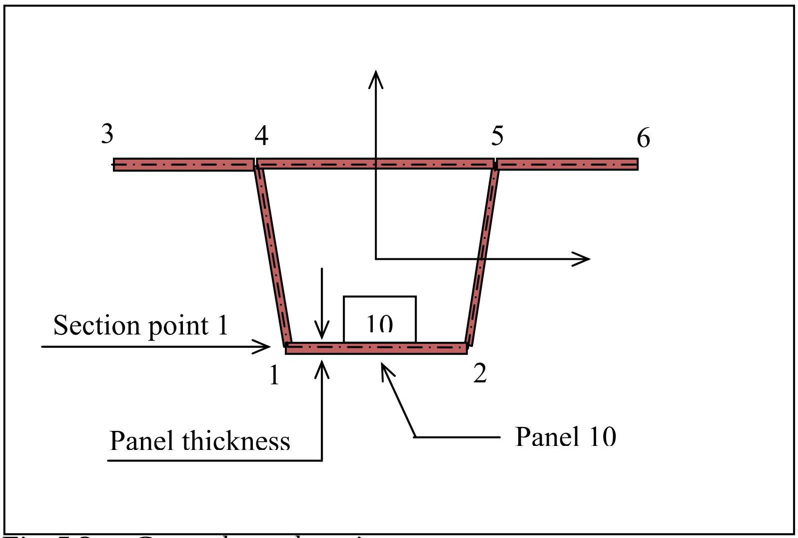


Fig. 5.3 General panel section

Sample input for panel 10 (Section number is set to 99)

```
PT,,99,1,ABS,-500.0,-400.0    ! point 1 coordinates are (-500,-400)
PT,,99,2,ABS, 500.0,-400.0    ! point 2 coordinates are (-500,-400)
PANEL,,99,10,1,2,20.0         ! panel thickness is 20 mm
```

The program will find any closed cells within the panel cross section in order to obtain the correct torsion stiffness and shear stress distribution. The panels stretching between the section points 1-2-5-4 in the figure above defines such a cell. Note that the same section point number must be used at the connection point of two panels. Using different section points with matching coordinate values will be interpreted as a discontinuity (gap).

The program is capable of calculating both axial stresses and in plane shear stresses. Panel elements are not intended for concrete section design.

5.3.2 Predefined panel

Cross section type specified in the XSECT command must be one of the following:

IDS	I-profile double symmetric
ISS	I-profile single symmetric
T	T-section
BOX	Box section
PIPE	Pipe (tubular)
WBOX	Winged box

The cross section measures and plate thicknesses are entered with the ascii command DIM. All measures are given in [mm]. The figure below shows the available predefined panel section types. Note, all meassures (not plate thicknesses) are between plate center lines, as shown for 'height' in the figure below.

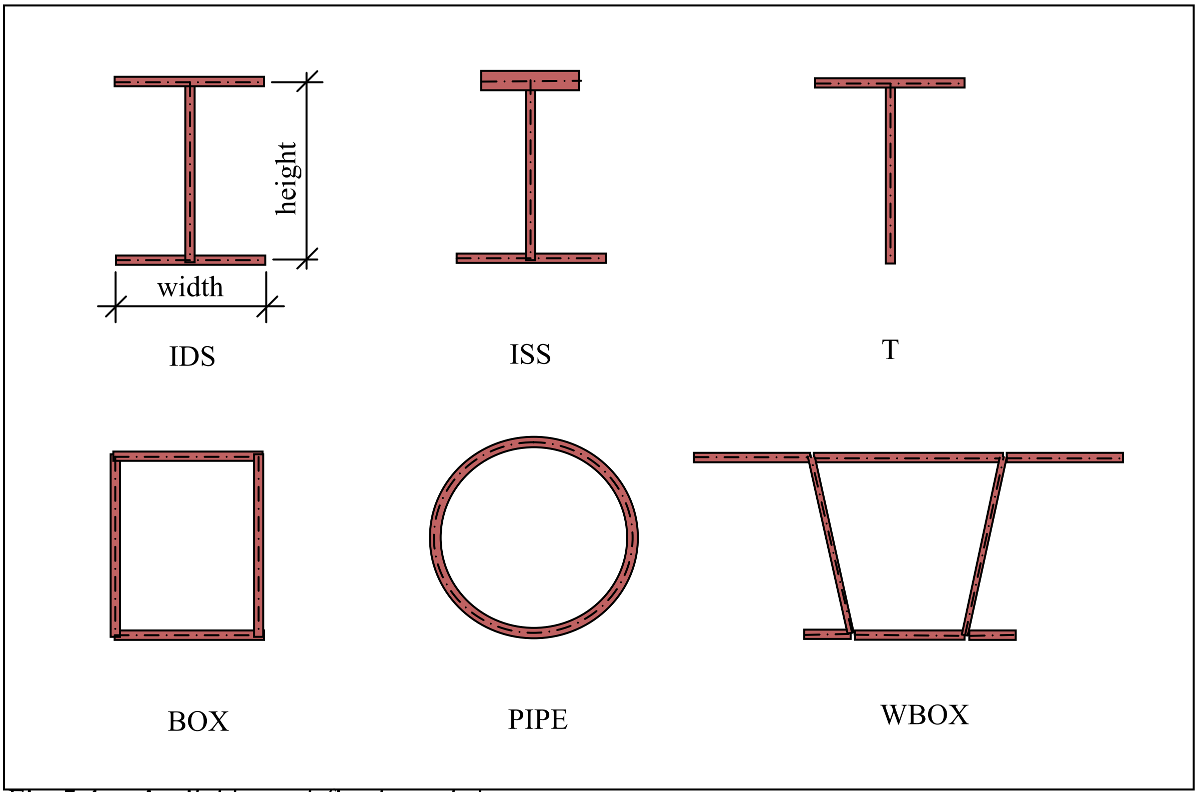


Fig. 5.4 Available predefined panel shapes

5.4 Catalogue profiles

The program offers a wide selection of predefined standard catalogue steel profiles, such as IPE, HEA etc. No section geometry input is required to create a catalogue profile. Just specify the profile name in the 'type' data field in the XSECT command

Sample input that creates a new HEA200 profile for section number 91:

```
XSECT,,91,HEA200,STEEL,1,1,2      ! create a new HEA200 profile
```

All section properties for catalogue profiles are given the exact value according to profile table values.

5.5 Sections with no design capabilities

In most cases, a frame analysis would be followed by a design check of the element cross sections. For those parts of the frame model (or whole) where no such design check is required, you may select the cross section from this category.

Cross section type specified in the XSECT command should be one of the following:

MSTIFF	Section properties given explicit
MRECT	Rectangular cross section
MCIRC	Circular (or tubular) cross section
MBOX	Box cross section
MT	T cross section

The sections in this category are very similar to those you find in the category 'Massive general', such that the cross sectional properties are based on the dimensions input given with the DIM command. The main difference in the input compared to the other cross sections categories (exept springs) is that you don't assign any material type or design parameter sets. In stead you specify the Young's modulus and Poisson's ratio to be used in the frame analysis for the elements using this cross section. The figure below gives a survey of actual sections in the category 'No design'.

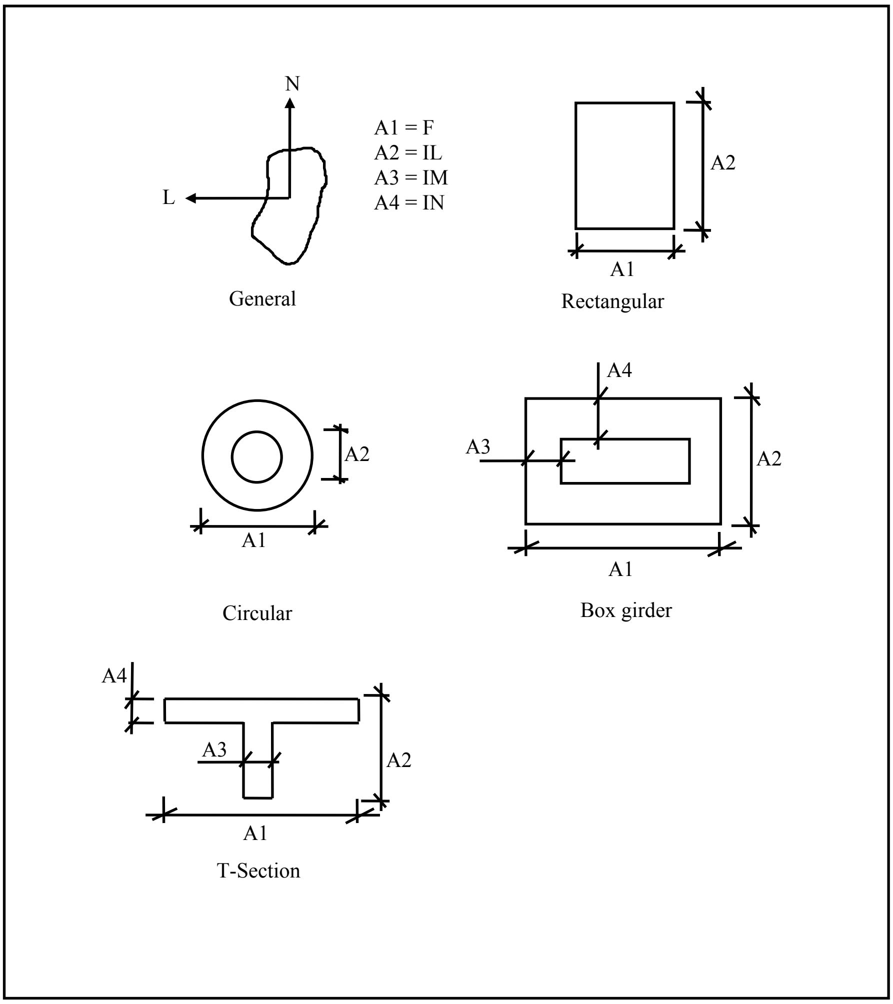


Fig. 5.5 Available sections with no design capabilities

5.6 Springs

Springs are applied in the frame model as beam elements assigned a special cross section type with defined spring stiffness constants.

Cross section type specified in the XSECT command should be one of the following:

MDEFSPR	spring with stiffness for translational DOFs
MROTSPR	spring with stiffness for rotational DOFs

The spring constants units are [kN/m] and [kNm/rad]

A spring is always connected to "the ground" in its node no. 2. The first node of the spring must be connected to an element end, the second node is only used for defining the direction of the spring. The second node can be connected to an element end, however the spring element will not give any contributions to the stiffness matrix in the second node.

A spring has the same local axis system as an ordinary element

6 Frame geometry

This chapter gives you a more thorough introduction to some important aspects of frame analysis with NovaFrame.

6.1 Numbering of nodes, elements and loads

The first step in modelling a frame is to divide the frame into elements and nodes. An element is a straight line between two nodes.

The frame geometry, the section properties and the actual load variation determine how you should model your frame. You must of course be sure that you have enough elements to represent all important effects. You should, however, be aware that section forces (but not displacements) can be calculated in arbitrary sections along an element.

There are no limitations of the number of nodes, elements and loads you can generate. However, the memory in your PC can be a limitation. In addition you should not use numbers higher than 32767 (max. positive integer value).

You should consider the following advice when you choose your numbering system:

- The input should be short and easy (as few input lines as possible).
- The element numbering should follow the structural parts such as columns and beams.
- Do not use all consecutive numbers in your first modelling session, but save some numbers for extra elements, nodes or loads in later runs.
- It is recommended to select load case numbers so they form groups containing the different load categories such as dead weight, temporary loads etc.
- It is possible in NovaFrame to use the same numbers for load combinations as for load cases.

In old versions of frame programs, the node numbering influenced the bandwidth in the stiffness matrix, which again influenced the calculation time. NovaFrame has included an optimiser-function, which renumber nodes internally to make the bandwidth smaller. However, you should always try to get as small node number difference as possible across an element to be sure to minimise the equation system.

6.2 Global and local axis

Two coordinate systems are used in NovaFrame.

- Global coordinate system; XYZ-system
- Local system; LMN-system

Nodes and boundary conditions are given in the global system.
Geometric sections are given in the local system.
You can choose if you want to give your loads in the global or local system.
Displacements are given in the global system.
Section forces are given in the local system.

You can choose among four different methods to describe your local system for the actual elements. This is done in the input card called ***Element specification***. You can use different methods for different elements in the same frame. The methods are:

- Default direction
- N-direction method
- Node 3 method
- Alpha method
- Column method

Default direction
M-axis follows the element axis and is positive from node 1 to node 2 of the element.
L-axis and N-axis follow global axis.
Positive direction of N-axis is determined by the angle between positive Z-axis and positive N-axis. This angle is always less than 90 degrees.
Positive direction of L-axis is determined by the right hand rule for the local LMN-system.

Be aware of the shift in axis direction for different element angles relative to the global system as shown on figure 3.2.1. Because of this "unexpected" shift, you should always check your local axis system by including the graphic attribute for local axis on your plot.

N-direction method
By this method you simply decide in which global direction your local N-axis should point. M-axis always follows element axis as for the default method. L-axis is then given by the right hand rule.

This method is very convenient when your frame has unexpected shifts of local system.

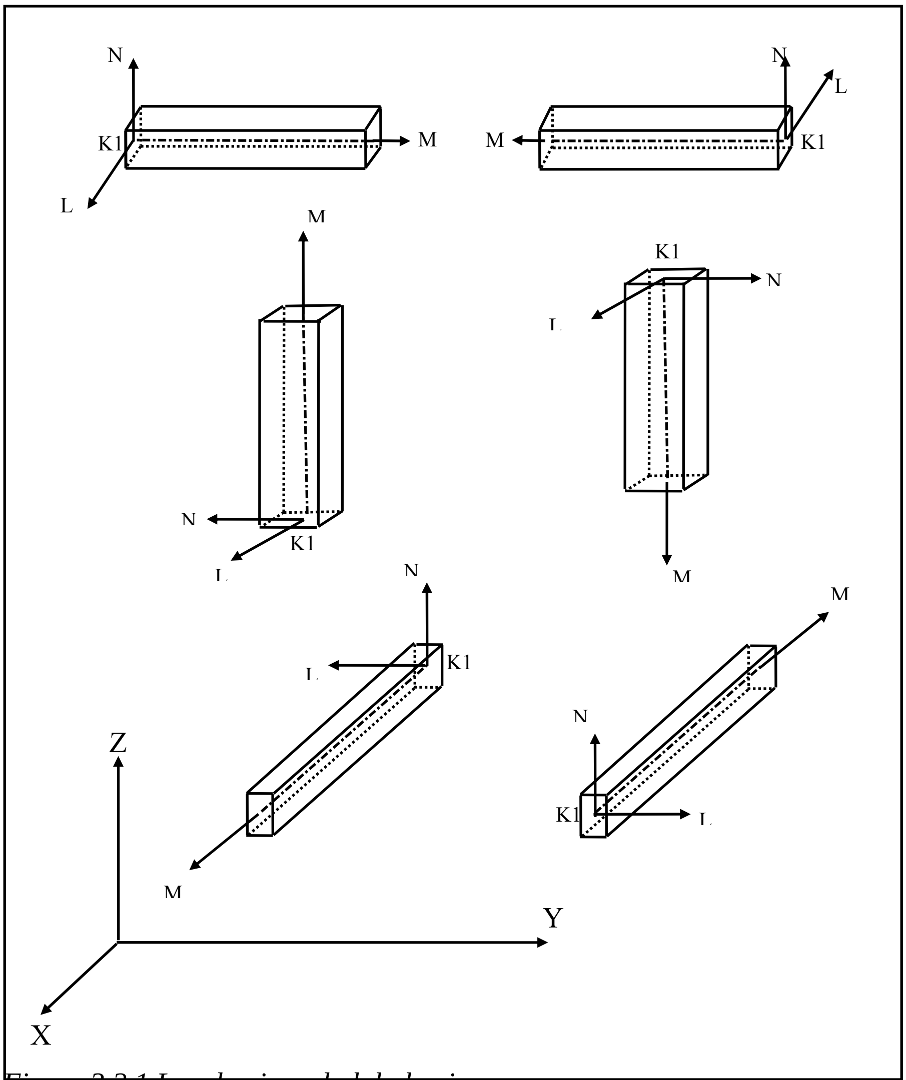


Figure 3.2.1 Local axis and global axis

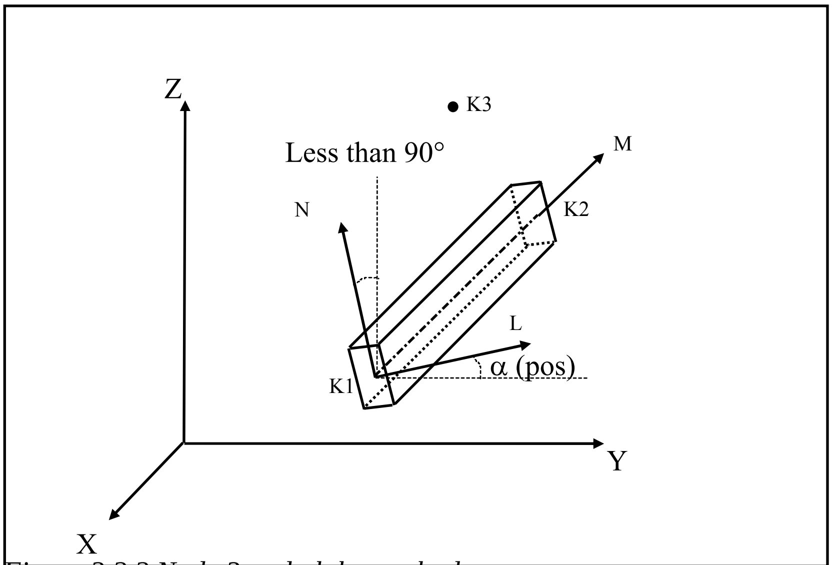


Figure 3.2.2 Node 3 and alpha method

Node 3 method
Node 3 is a node you specify for the actual element and which by definition is placed in the positive part of the local MN-plane. This is illustrated on figure 3.2.2.

You can use nodes you have specified for other elements as node 3 or you can define separate nodes just for this purpose. You don't have to worry about the bandwidth of your stiffness matrix when you introduce the node 3 method. These extra nodes will not influence the equation system size.

Alpha method
This method is difficult to understand and should not be used by inexperienced users. This method is also illustrated on figure 3.2.2.

By this method you can rotate your local system relative to the global system by giving an alpha-angle in degrees. This angle is defined as the angle you must rotate the local system about M-axis to make the L-axis parallel with the XY-plane (horisontal). Positive angle follows the right hand rule.

Special rules apply for vertical elements. An element is defined as vertical if the difference between the element length and projection of the element on Z-axis is less than 0.001. The angle between the L-axis and the X-axis must now be less than 90 degrees. Alpha is the angle you must rotate the M-axis to make L-axis parallel to the XY-plane. Positive angle follows the right hand rule also for this situation.

Column method
This method is used for defined columns, see section 3.8. Columns are vertical reference lines. If nodes and thereby elements are connected to this column line the direction of the local N-axis is set to point in the direction of the road reference line to which the column is connected.

Changing local axis
There is a possibility to change the local axis without changing the cross section associated with an element. Use section number 0 in order perform changes to the local axis only.

6.3 Boundary conditions and joints

You define your boundary conditions on the ***Boundaries*** input card and the actual joint data on the ***Joint*** input card.

Joints can only be specified on the element ends. In a joint one or more of the section forces are equal to zero.

Boundary conditions in NovaFrame are defined as fixed degrees of freedom for selected nodes or a master-slave connection between selected nodes.

You can only define fixed degrees of freedom in the global coordinate system, but it is possible to model them in a local coordinate system if you introduce springs or joints. This is illustrated on figure 3.4.1.

You should, however, try to avoid using inclined springs. The stiffness of the springs are decomposed to the global axis directions leading to large numbers outside the diagonal of the stiffness matrix if the spring is modelled to stiff. This effect can again lead to an inaccurate solution. Check your results by comparing the equilibrium between loads and reaction forces.

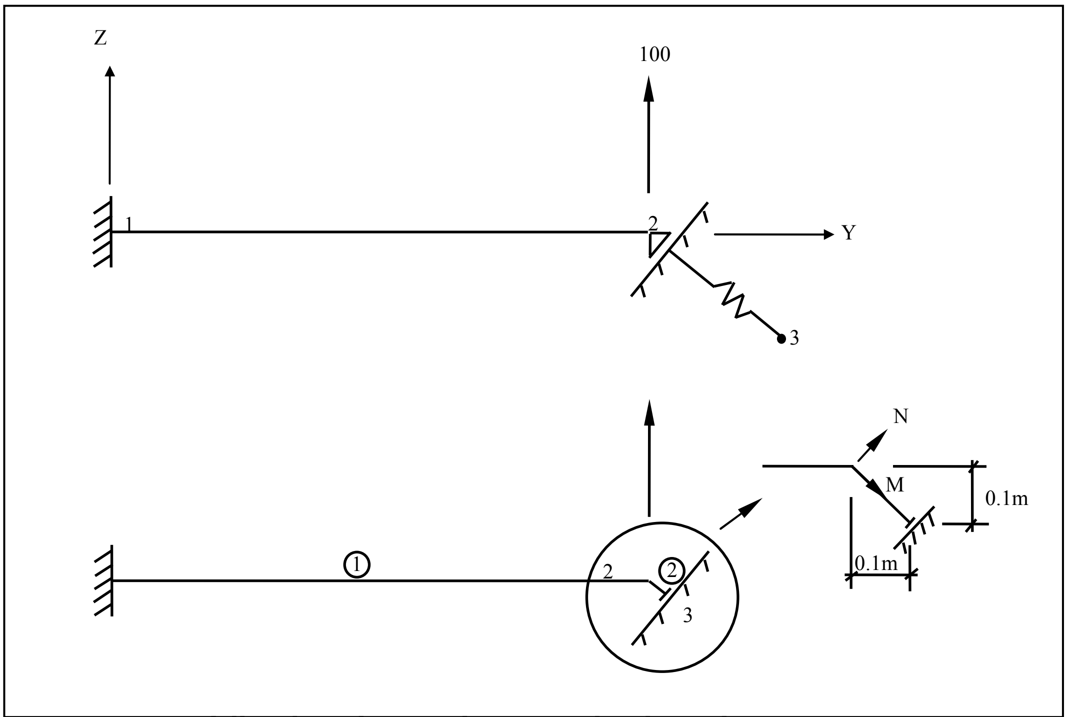


Figure 3.4.1 Modelling boundary conditions in a local coordinate system.

A master-slave connection is an easy method to model a stiff connection in your model. You must define one node as the master and the other nodes as slaves. The rotation of the slave nodes will be equal to the rotation of the master node. The displacement of the slave nodes will be equal to the displacement of the master node plus the rotation of the master node multiplied with the distance between the master and slave nodes.

Both master and slave nodes must be connected to an element end.

To model the system shown on figure 3.4.2, you can define node 2 as the master node and node 4, 7 and 9 as slaves.

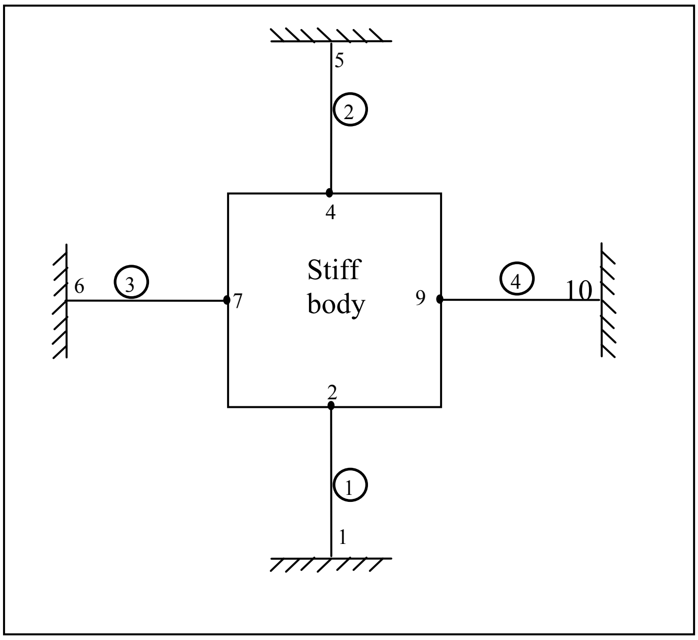


Figure 3.4.2 A typical master-slave connection

6.4 Element mass

NovaFrame has the possibility of calculating eigenfrequencies, i.e. modes of vibration. The results of an eigenvalue analysis can be used in a succeeding dynamic analysis..

The required data in order to perform an eigenvalue analysis is to define the mass properties of the elements. The mass of the elements can be given by these methods:

- Density [ton / m^3], Type = 1.
  Mass per unit length is calculated by multiplying with the element area.
- Distributed mass [ton / m], Type = 3.
- Point mass [ton], Type = 4.
  Distance from first node to the position of the point mass can be given.
- Rotation mass [ton · m^2], Type = 5.
  Distance from first node to the position of the point mass can be given.

Rotation mass can together with point mass be used for the description of a mass with an eccentricity = e. The mass M is moved and a rotation mass of $M \cdot e^2$ is added.

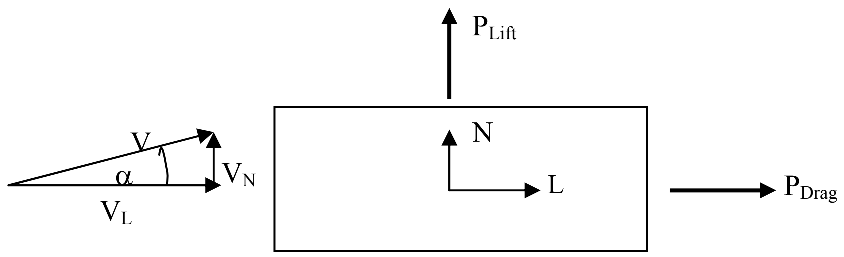
The user must specify each of the global directions the rotation mass is to apply.

Note!
Using master slave connections when calculating eigenfrequencies and modes must be used with care. The reason is that the mass in the slave node is lumped to the master node. Since the mass matrix is a pure diagonal matrix there will be some losses of information when the mass is lumped to the master node.

6.5 Wind areas and shape factors

Wind areas and shape factors of elements can be defined in NovaFrame. These properties are used in a static wind loadcase. For static wind loadcases, see chapter 4.1..

The wind speed at the element location is decomposed in the element local directions, one component in the L-direction and one component in the N-direction. The wind component; V_L gives a drag- and a lift component as shown in fig. 3.6.1.

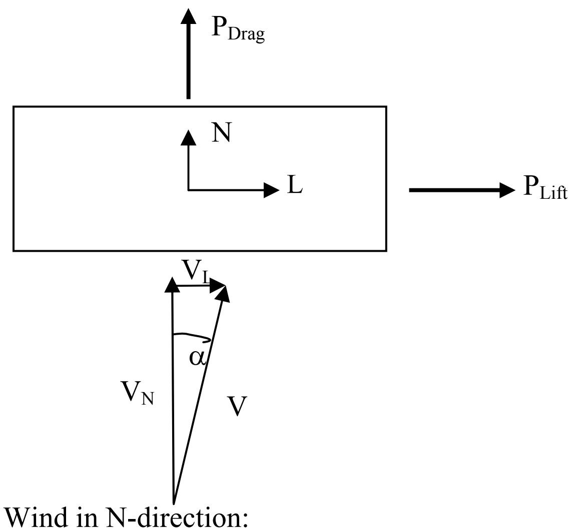


Wind in L-direction:

$$P_{Drag} = 0.5 \cdot \rho \cdot (CA)_D \cdot V_L^2$$

$$P_{Lift} = 0.5 \cdot \rho \cdot \left[(CA)_{L_0} + (CA)_{L_1} \cdot \alpha\right] \cdot V_L^2$$

$$\alpha = \tan(V_N / V_L)$$



Wind in N-direction:

$$P_{Drag} = 0.5 \cdot \rho \cdot (CA)_D \cdot V_N^2$$

$$P_{Lift} = 0.5 \cdot \rho \cdot \left[(CA)_{L_0} + (CA)_{L_1} \cdot \alpha\right] \cdot V_N^2$$

$$\alpha = \tan(V_L / V_N)$$

Fig. 3.6.1 Decomposition of wind speed.

Wind Component in L-Direction:

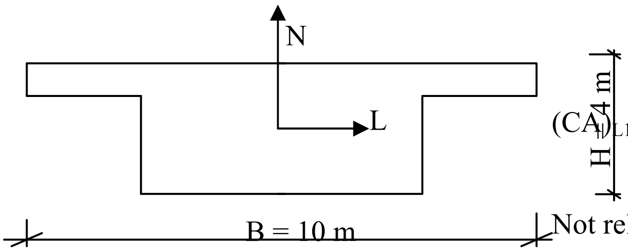
$(CA)_D$ = Drag coefficient
$(CA)_{L0}$ = Lift coefficient
$(CA)_{L1}$ = Change in lift coefficient with respect to angle α.
Contributes to load in N-Direction when multiplied with angle α.
$(CA)_{M0}$ = Torsion coefficient
$(CA)_{M1}$ = Change in torsion coefficient with respect to angle α.

Wind Component in N-Direction:

$(CA)_D$ = Drag coefficient
$(CA)_{L0}$ = Lift coefficient
$(CA)_{L1}$ = Change in lift coefficient with respect to angle α.
Contributes to load in L-Direction when multiplied with angle α.
$(CA)_{M0}$ = Torsion coefficient
$(CA)_{M1}$ = Change in torsion coefficient with respect to angle α.

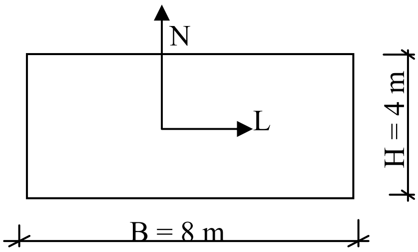
In fig. 3.6.2 examples of shape factors are shown. Reference is made to the following rules and recommendations:

- NS3479
- Danish standard DS410.2
- British Standard BS5400:Part2
- Eurocode1



Wind in L-direction:
$(CA)_D$ = 1.8* x H (drag)
$(CA)_{L0}$ = 0.2 x B (lift, ø = 0)
$(CA)_{L1}$ = 3.5 x B (lift, ø = 0)

Wind in N-direction:
Not relevant.



Wind in L-direction:
$(CA)_D$ = 1.9* x H
$(CA)_{L0}$ = 0.2 x B
$(CA)_{L1}$ = 3.5 x B

Wind in N-direction:
$(CA)_D$ = 2.4* x B
$(CA)_{L0}$ = 0.2 x H
$(CA)_{L1}$ = 3.5 x H

* dependant of H/B.

Fig.3.6.2 Example of shape factors

6.6 Design sections

You can specify as many design sections as you want in each element. Use the ***Design section*** input card to specify the actual design sections.

When an analysis is performed there will only be calculated section force results in the design sections that are defined.

Design sections are defined with relative distances from node 1 of the actual element. NovaFrame calculate section forces, but not displacements, in each design section.

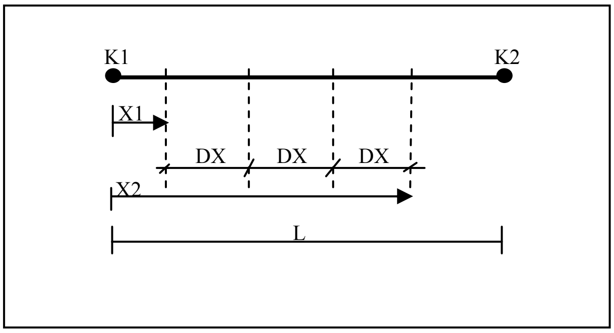


Figure 3.7.1 Design sections where results are calculated

A survey of positive section forces is given in figure 3.7.2.

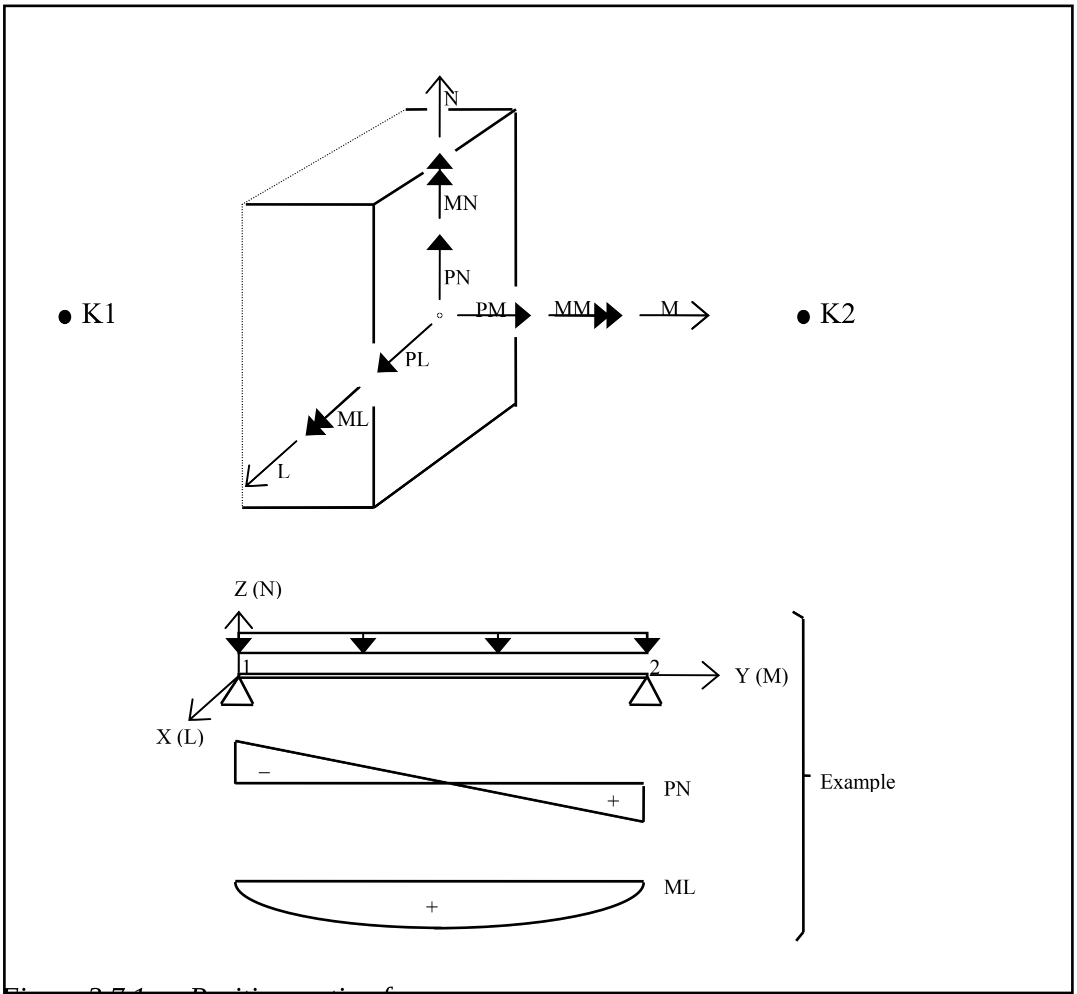


Figure 3.7.1 Positive section forces

6.7 Reference lines

Reference lines are an important subject in the communication of data between NovaFrame and NovaDesign. The idea of a reference line is to introduce the concepts of roads and railways more directly into the design process and to reduce the work of generating geometry input data.

Reference lines are geometric lines. NovaDesign can assign cross sections to reference lines at specific stations along this line. (Station; (no.) = profilnummer). Geometry is defined for the reference line in the ***Frame-preprocessor*** window. Stations along the line can be labelled by defining axis numbers and names.

Note:
Reference line 0 is not a geometric line. It is an abstract line where cross sections are assembled by their number. You can create cross sections in NovaDesign to Reference line 0. All cross sections defined in NovaFrame are assembled in reference line 0. Reference line 0 can not be deleted.

6.7.1 Types of reference lines

The types of reference lines available in NovaFrame are:

- **Horizontal projection**, the station numbering is on the horizontal projection in the XY-plane. This is the usual method of defining road geometry.
  Reference line type = 1. (Use Horisontal Segments and Vertical Segments for assigning geometry to this reference line type.)

- **Vertical projection**, the station numbering is the projection of the line on the Z-axis. This type is used for columns. Station numbering can be choosen to represent elevation. Reference line Type = 2.
  (Use Coords in order to assign geometry to the reference line.)

- **Station by length of line**. This is a more general approach where a line may consist of both vertical and horizontal parts and every station will still be unique. Reference line Type = 3.
  (Use Coords in order to assign geometry to the reference line.)

6.7.2 Cross sections on reference lines

The cross sections are assigned to the different stations on a reference line in NovaDesign.

All cross sections defined in NovaFrame are contained in reference line 0. The cross sections defined on reference line 0 are those who can directly be used for specifying section properties to specific elements in the ***element specification input..***

Cross sections assigned to reference lines other than 0 can be used for automatic calculation of section properties for elements connected to this reference line. Currently linear variation of a cross section between two stations is supported. If a dead-weight loadcase is defined the weight of the element is automatically calculated based on the calculated cross section areas in each end of the element. The weight is applied as a trapezoidal load.

The stiffness properties of an element is calculated as the average stiffness over the element length. The stiffness properties are calculated at each end of the element and then averaged.

For reference lines of type 1 and 2 the cross sections properties are applied to the elements without any transformation. The user must ensure that the angle between the element length axis (M-axis) and the reference line geometry is within acceptable limits.

For reference line of type 1; horizontal projection, this implies that the angle between the element axis and the horizontal plane must be checked.

Local Element axis: For elements with section properties calculated by reference lines, the local N axis will always be in positive global Z-direction for type 1.

Offsets from a reference line:
For cross sections defined on geometric reference lines it is possible to define an offset from the cross section COG to the reference Line. The offset consists of a x-offset and a y-offset in the cross section local axis. Correct signs on the ordinates are found when positioning the section COG or section point relative the reference line position (= "origo").

6.7.3 Assign geometry to a reference line

Geometry to a reference line can be assigned by different methods. These methods are not to be combined for the same reference line.

1) Defining geometry with horizontal segments and vertical segments: This is the usual method for specifying road and railway geometry. Horizontal segments can be straight lines, circles or clothoides in the global X-Y -plane. Vertical segments are straight lines or circles. This is available for Reference line of type 1, i.e. stations by horizontal projection.
2) Defining geometry by co-ordinates. A series of co-ordinates can be defined. The geometry of the reference line is the linear interpolation between these co-ordinates. This is available for Reference line of all types

6.7.4 Columns

Columns can be defined by connecting a vertical reference line to a road reference line. This connection can be made to a station number or an axis. This column will then follow the defined point on the road reference line if the geometry changes.

6.7.5 Create node geometry on a reference line

Node co-ordinates can automatically be calculated by a reference line by assigning nodes to different stations on a reference line. Nodes can also be generated by the use of axis. See ***the node input card*** for details and options.

6.7.6 Plot of reference lines

Select Reference Line in the Attribute dialogue. Choose station area for which the reference line is plotted. Reference line number may be included. Axis names can also be shown on the reference line.

7 Tendons

This version of NovaFrame includes the possibility to define tendons for post-tensioned concrete structures. The effects of tendons are supported both in the NovaFrame analyses and in the concrete design performed by NovaDesign.

7.1 General

You define your tendons in the ***Tendons*** input card.

NovaFrame supports both post- and pre-Tensioned tendons.

7.2 Tendon geometry

In order to define the geometry of a tendon there are sets of information that needs to be considered:

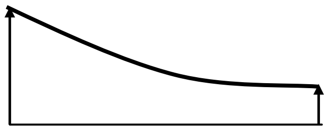
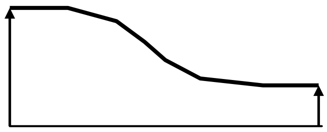
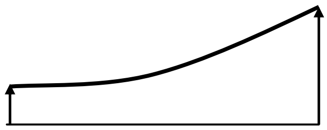
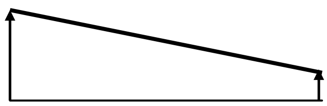
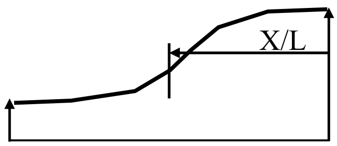
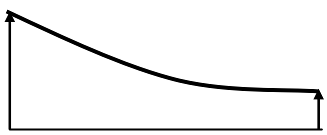
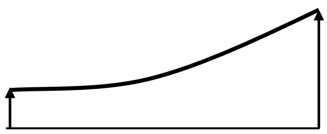
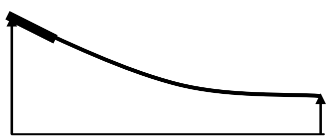
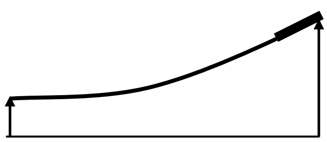
- Type of geometry? Choose between element lengths, stations or axes along a reference line when defining the tendon length. If you use reference lines then design calculations by NovaDesign with automatic cable position is available.
- How are the tendon offsets to be used, what are they relative to?

The geometry of a tendon is built by a sequence of curves consisting of 3. order polynomials. The curves have constraints, which supports the actual starts and ends, inflection points and lower point of a tendon.

Curve lengths:
The curve segment length is defined by either:
The length of an element sequence.
The length defined by stations on a reference line.
The distance between axis on a reference line.

Length definition by reference line and by axes:
For the definition to be unique it is required that there are no double defined elements on the reference line. (This implies that two or more elements with the same nodes are not allowed). The reason is for the application to find which loads are to be applied on each element due to the response imposed by the tendons).
The applications checks all elements defined by their nodes on the specified reference line in order to verify if they are a part of the tendon.
The element numbering must be in positive sequence with the positive station numbering.
It is possible to start or end the curve pieces of tendon on any position along an element.
The tendon must be defined in positive station sequence.

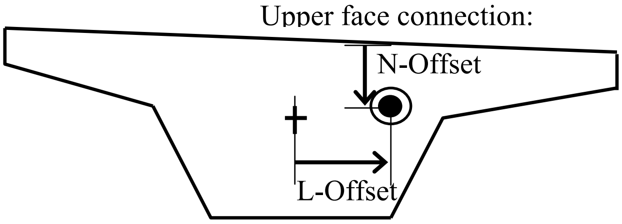
The following curve pieces of geometry are available:

Type	Curve segments	Properties
Type 1 Typically left end of tendon		Length = L Y'' (0) = 0 Y'(L) = 0
Type 2 S-curve		Length = L Y' (0) = 0 Y'(L) = 0 Y''(L/2) = 0
Type 3 Typically right end of tendon		Length = L Y' (0) = 0 Y''(L) = 0
Type 4 Straight line		Length = L Y' (0) = Y'(L) Y''(0->L) = 0
Type 5 S-curve with spec. inflection		Length = L Y' (0) = 0 Y'(L) = 0 Y''(X/L) = 0 (X/L specified, [0<X/L<1.0]
Type 6 Typically left end of tendon		Length = L Y'' (0) = 0 Y'(0) = φ (negative on fig.) Y'(L) = 0
Type 7 Typically right end of tendon		Length = L Y' (0) = 0 Y''(L) = 0 Y' (L) = φ (positive on fig.)
Type 8 Typically left end of tendon Starts with 1m straight part!		Length = L Y'' (0) = 0 Y'(0) = φ (negative on fig.) Y'(L) = 0
Type 9 Typically right end of tendon Ends with 1m straight part!		Length = L Y' (0) = 0 Y''(L) = 0 Y' (L) = φ (positive on fig.)

Curve offsets:
Relative to element axis is the most immediate option. This implies that the position of the tendon is described by in the element local axis system. This method implies that the cables are directly referred relative to the COG of the cross section.

Curve offsets, face connections:
If the section points of a cross section are assigned face properties, it is possible to define the position of the tendon relative to a face rather than relative to the section COG, i.e. element M-axis.
N – geometry can be defined relative to the upper face or relative to the lower face. In this situation the L-offset is still measured from COG to a local X-coordinate which is used for calculating the N-offset.
The advantage of using face offsets rather than COG offsets is that the input defining the cable geometry may be considerably simpler. Example: tendons in top slab of cantilever bridges.
Face associations to section points are achieved by using the SECTFACE command.

Upper face connection:



Curve offsets, relative to reference line:
It is possible to define the position of the tendon relative to the reference line directly. The L- and N-offsets are still in the local X-/Y-coordinate system of the cross section.

Curve offsets, relative to section points:
It is possible to define the position of the tendon relative to a section point defined on the cross section. The L- and N-offsets are still in the local X-/Y-coordinate system of the cross section.

NOTE! The tendon geometry needs to find elements at all locations along a reference line in the interval where the tendon is defined.

NOTE! The N-direction is used as primary direction. The full length of the tendon must be defined by N offsets. As a rule: always start with defining the N offset, even if the offset is zero.

NOTE! The element numbering along a tendon must be positive in the same order as the tendon parts are defined.

7.3 Losses and stressing

The effective force from the stressing of the tendons is reduced due to losses. These losses are both immediate and time dependant. NovaFrame supports the following losses:

- Friction, (immediate).
- Wobble, (immediate).
- Wedge draw in, (immediate).
- Creep, (Time dependant).
- Shrinkage, (Time dependant).
- Relaxation, (Time dependant).

The initial stressing of the cable combined with the sum of these losses gives the current stressing level in the tendon. Each of the losses are described in more detail in the following.

4.3.1 Losses due to friction

The losses due to friction is calculated according to Coulomb's formula

$$P(x) = P_o \cdot e^{-(\mu\alpha + kx)}$$

P(x) = Post-tensioning force at a distance x from the stressing anchorage.
P_o = Post-tensioning force at the stressing anchorage.
μ = Coefficient for friction.
α = Sum of angular deviations (in radians) of the tendon in all planes over the distance x.
k = Wobble factor (inaccuracies in placing) per unit length.

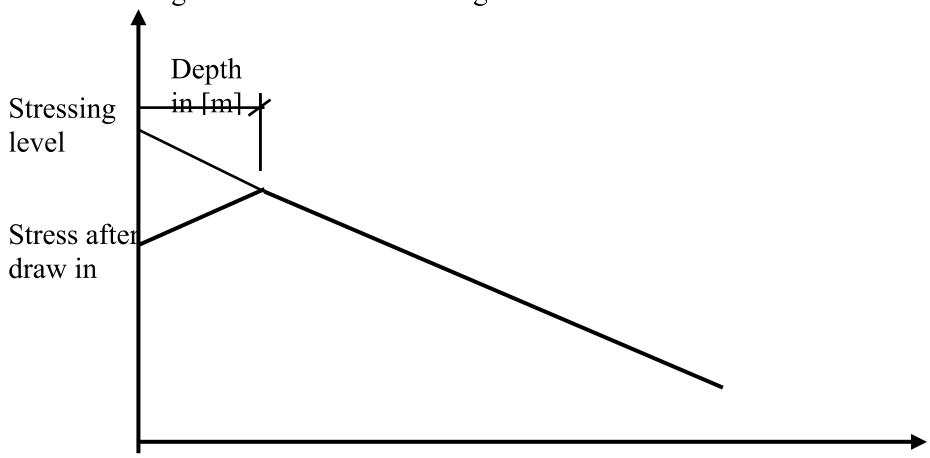
The values for μ and k are obtained from the supplier of tendon products. Some typical values are presented below:

Coefficient μ:
Tendons in standard metal ducts: Range from; 0.16 – 22, mean 0.19.
Tendons threaded through semi-rigid, previously placed ducts; 0.13-0.19, mean 0.16.
The angular deviation is summed along the tendon. The angular deviation is based on a line integral along the tendon. Curvature of the system line to which the tendon geometry is defined relative to is accounted for.

Coefficient k:
Range; (0.6-1.0) $\cdot 10^{-3}$/m, mean 0.8 $\cdot 10^{-3}$/m.

7.3.1 Draw-in of wedges

When the strands are locked off in the anchorage the wedges move through a fixed distance of Δx, which is supplier dependent. The effect of the wedge draw-in is a reduction of the tendon force close to the anchorage. This is illustrated in figure 4.4.1 :



The depth of the effect is dependant of the wedge draw in and the friction losses. NovaFrame calculates this depth by iteration until compatibility is met.

4.3.2 Creep

Creep is a time dependant change in strain of the concrete. The effect of creep is dependent of the stress level. For more details on the background for calculation creep factors and changes in strain, please refer to /2/ - NS3473 pr. A.9.3.

The loss in the tendon due to creep is accounted for by specifying a strain in the tendon. The user calculates this strain.

4.3.3 Shrinkage

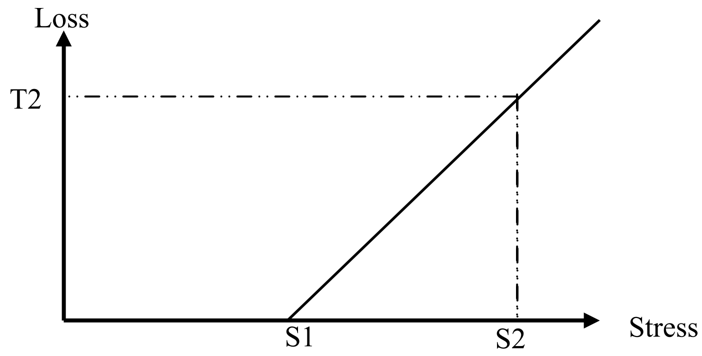
Shrinkage is also a time dependant change in strain of the concrete. The effect is however not dependant of the stress level, and represents a change in the volume/length of the concrete. For more details on the background for calculation shrinkage strain, please refer to //2/ - NS3473 pr. A.9.3.

The loss in the tendon due to shrinkage is accounted for by including a strain in the tendon. The user calculates this strain.

4.3.4 Relaxation

Relaxation is also a time dependant change in strain of the prestressing steel. The effect is dependent of the stress level, and temperature. Curves are available from the tendon material supplier.

The loss in the tendon due to relaxation is accounted for by defining a simplified curve, which represents the relaxation loss as a function of initial stressing level. The curve is bilinear and is represented below:



8 Loads and related items

This version of NovaFrame includes several different load categories:

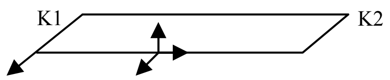
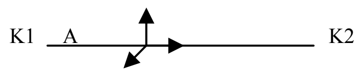
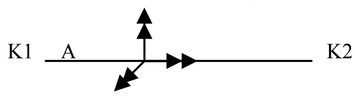
- Static loads with a large number of types to choose among in order to achieve efficient data input.
- Traffic loads with calculation of response and worst position of vehicles and trains.
- Dynamic loads with response of from a dynamic analysis.
- Creep loads with calculation of response in the structure due to creep and relaxation.
- Axial forces in structural members in order to account for stiffness changes due to 2.order effects.

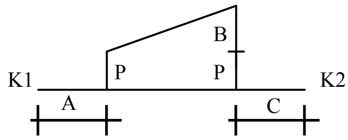
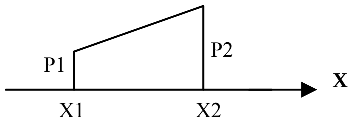
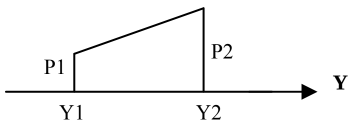
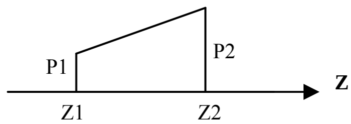
The input cards for load data is found by selecting **Preprocess** in the ***Frame window*** and by selecting the ***Load Data*** menu item.

8.1 Static loads

You define your static loads in the ***Loads*** input card.

Static load specifications include several alternatives for applying distributed pressure, concentrated forces or strains on the element model. The loads are applied on elements of the model. All loads must be assigned to a loadcases. Most loads can be applied in both global and local direction. The static loads are:

Type:	Description:	Dir:	Input:
1	Self weight	Z	P = self weight [kN/m^3].
2	Temperature	M	P = ΔT Temp. increase [°C], $\alpha = 10^{-5}$ m/°C, by default. A = Temp. gradient L, dT/L_{width}, [°C/m]. B = Temp. gradient N, dT/N_{width}, [°C/m]. C = Temp. coefficient factor on α.
3	Distributed load	All	P = distributed load [kN/m]. A = eccentricity in L-direction [m]. B = eccentricity in N-direction [m]. 
4	Concentrated point load	All	P = point load [kN]. A = distance from K1 to load [m], (if A> *l*, then A = *l*). 
5	Concentrated moment	All	P = point load [kN]. A = distance from K1 to load [m], (if A> *l*, then A = *l*). 

7	Prestressing (constrained)	M	P = prestressing load [kN, positive]. A = eccentricity [m] in N-direction at K1 B = increase of eccentricity at K2 C = Number of cables 
8	Trapeze load	All	P = Load intensity closest to K1 [kN/m]. A = Distance [m]. B = Increase in load intensity. (Note !!) C = Distance [m]. 
10	Static wind	N/A	P = Wind speed at 10 m height [m/sec.] A = Wind direction relative to X-axis [deg.] B = Ground level [m] C = Exponent for wind profile (optional; defaults to 0.213 if not given). (Wind loads acc. to NS3491-4, see type 60,61)
11	Prestressing	M	Input as for type 7.
12	Shrinkage	N/A	P = Shrinkage strain in 0/00. A-C = not in use.
15	Tendons	N/A	P = Tendon no. to activate. A = 1 = constrained forces only, 0 = full B = Stressing phase, 0 = default, 1 = first end only C = Time dependant losses. CSR, zero or 1, (000:none, 111:all)
20	Gradient X	All	P = P1 = Value of load at ordinate X1, [kN/m]. A = X1 = Global ordinate. B = P2 = Value of load at ordinate X2, [kN/m]. C = X2 = Global ordinate. (Note: X2 > X1) 
21	Gradient Y	All	P = P1 = Value of load at ordinate Y1, [kN/m]. A = Y1 = Global ordinate. B = P2 = Value of load at ordinate Y2, [kN/m]. C = Y2 = Global ordinate. (Note: Y2 > Y1) 
22	Gradient Z	All	P = P1 = Value of load at ordinate Z1, [kN/m]. A = Z1 = Global ordinate. B = P2 = Value of load at ordinate Z2, [kN/m]. C = Z2 = Global ordinate. (Note: Z2 > Z1) 

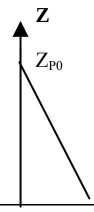
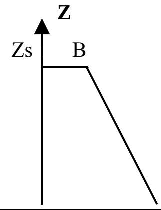
30	Hydrostatic pressure	L, N	P = Z_{P0} = zero water level) A = ρ_w = Density of water [kg/m^3]. 
40	Soil pressure	X,Y	P = Z_s = Reference soil level [m]. A = ρ_s = Density of soil [kg/m^3]. B = Soil pressure at Z_s[kN/m]. C = Soil coefficient k. 
60	Static 10min wind acc. to NS3491-4	N/A	P = Basic wind speed at 10 m height [m/sec.] A = Wind direction relative to X-axis [deg.] B = Terrain category (0,1,2,3,5) C = Ground level [m] Note: Push the "..details" button to get help
61	Static gust wind acc. to NS3491-4	N/A	P = Basic wind speed at 10 m height [m/sec.] A = Wind direction relative to X-axis [deg.] B = Terrain category (0,1,2,3,5) C = Ground level [m] Note: Push the "..details" button to get help

Figure 4.1.1 Available static loads types

A loadcase can consist of as many load input-lines as you need to specify the actual loadcase. You can also include several load types in the same loadcase.

Note!
Remember to check global load resultants for the loadcases you define. For complicated loading situations it can be advantageous to use more than one loadcase to describe a load action in order to ease verification of global load resultants.

8.1.1 Dead weight

Weight per volume unit. The direction of the load is global positive Z, (Dir = 3).
Dead weight applies to the full length of the element. The area of the cross section associated with the element is multiplied with weight per volume unit. The load is then treated as a trapezoidal load.

Normally used values according to NS3491-1 /3/:

Material	[kN/m^3]
Reinforced concrete	25
Steel	77
Aluminium	27

If element properties are calculated based on sections defined by reference lines, the section area is calculated at each end of the element and the load is applied as a trapezoidal load.

8.1.2 Temperature load

This load supports increase of temperature in degrees [Celcius] and temperature gradient for both local axes.

The thermal coefficient is by default $\alpha = 10^{-5}$ m/°C, which is applicable for concrete. The thermal default coefficient can be multiplied with a value if needed to give correct values for other materials. Example: use 1.2 for steel or 2.3 for aluminium.
The load direction is automatically set to M-direction, the element length axis.

Temperature gradient:
L–gradient gives curvature and response around N-axis, temperature varies by width of section in L-direction. Positive gradient will result in negative curvature.
N–gradient gives curvature and response around L-axis, temperature varies by width of section in N-direction. Positive gradient will result in negative curvature.

$$\text{Gradient L} = \frac{(TL+) - (TL-)}{W\text{-}L}$$

$$\text{Gradient N} = \frac{(TN+) - (TN-)}{W\text{-}N}$$

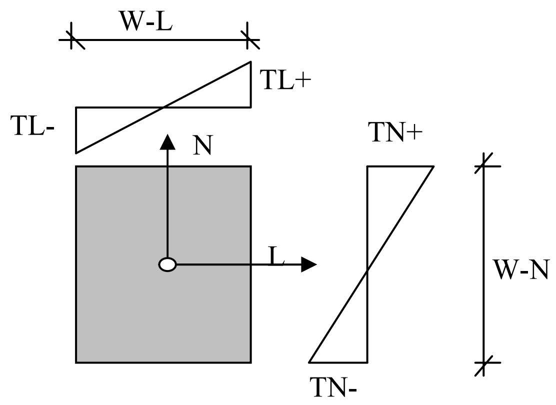


Figure 4.1.2 Temperature gradient

The program calculates constrained (“parasite”) forces due to temperature loads. I.e. for statically determined structures this will result in deflections but no sectional forces.

8.1.3 Distributed load

Evenly distributed load on the entire length of an element. Both eccentricity in L-direction and N-direction can be specified.
The load resultant is P·(length of element), independently of the load direction.

8.1.4 Concentrated point load and moment

Point load or moment can be specified at a given distance from the 1st node of an element. If the distance A is larger than the length of the element then A is automatically corrected to the element length.

It is not possible to place a concentrated load directly on a node, but the same effect is achieved by placing the concentrated load on an element end. If a design section (see chapter 3) and concentrated load are placed at the same point on the element axis, the program will by default place the design section to the right of the concentrated load when calculating shear forces.

8.1.5 Prestressing loads

Longitudinal prestressing force has linear N-eccentricity variation over the element length. The prestressing force P is positive when resulting in compression.

Constrained "parasite" type (type = 7):
The prestressing is treated as applied strains. For statically determined systems this implies only deflections and no section forces. For statically indeterminate structures, only the constrained («parasite») section forces are calculated. When designing the cross section the prestressing force and its eccentricity must be added.

Non constrained type (type = 11):
This type calculates both resulting sectional forces and deflections in the model.
This type is used when the stresses in the model are required in order to perform a creep analysis.

The amount of prestressing can be scaled by using the last term C = number of cables. By default the value is 1.0.

(The term "prestressing" is used somewhat inaccurate. The term is meant to cover both pre- and post- tensioned tendons.)

8.1.6 Trapezoidal loads

Trapezoidal load has uneven distributed load on specified parts of the elements.
Note that the intensity of the load is per unit length of the element independently of the load direction.

8.1.7 Static wind loads

General:
The given wind speed in 10 m height is used for calculating the wind speed dependent of the z-ordinate of the element. This is done according to the formulae:

$$V_z = V_{10} \cdot \left(\frac{z}{10} \right)^{0.213}$$

The units [kN] and [m] must be used. The exponent of 0.213 is default, the value can be overridden.

Remember to place the zero level of the structures at the zero level of the wind fields.
If z < 1.0 [m], z is automatically set to 0.
Wind areas and factors must have been defined in order for this load type to result in loads on a model. See section 3.6.

According to NS3491-4:
The Norwegian standard NS3491-4 defines two levels of the wind speed:

- 10 minutes mean wind speed (load type 60)
  which is used in connection with a stochastic dynamic analysis where the static and dynamic load effects are added.
- Gust wind speed (1-3 sec.) (load type 61)
  which is used when the dynamic effect of the wind is neglected.

The 10 minutes mean basic wind speed v_b 10m above ground level is defined by:

$v_b = C_{DIR}*C_{YEAR}*C_{LEVEL}*C_{PROB}*v_{REF}$

The reference wind speed v_{REF} and correction factors C are defined in NS3491-4. These can also be found by pushing the "..details" button which appear in the load case dialog when load type 60 or 61 is chosen.

The mean wind speed $v_s(z)$ is:

$v_s(z) = v_b *c_t(z)* k_T * \ln(z/z_0)$ for $z> z_{min}$

where the topography coefficient $c_{t(z)}$ and terrain factor k_T are defined in NS3491-4. The topography coefficient $c_t(z)$ is set to 1.0 in NovaFrame.

8.1.8 Shrinkage loads

Shrinkage load consists of an applied (axial) strain in the specified elements.

8.1.9 Tendons

In order to obtain force from a defined tendon it is required to define loadcases which holds the forces that the tendon apply on the structure. Defining loadcases of type «Tendon» does this.

The user must define which elements are to be by activated for this command and which tendon who is activated. In addition the user must specify if the tendon stresses are to be calculated as full forces or as constrained (parasite) force. For the use in concrete design the constrained option is the most common approach.
User can define the stressing phase of the tendon. 0 is default, and all the defined stressing is activated.
Also the user must select which of the time dependant losses who are to be included when calculating strains in the tendon. CSR = Creep, Shrinkage and Relaxation, each with a flag of 0 or 1. 111- implies all time dependant losses. 000- implies no time dependant losses.

8.1.10 Gradient loads

Gradient load types are distributed loads varying with global co-ordinates. Only elements or part of elements between the ordinates 1 and 2 will be subjected to the application of loads.
For loads applied in global direction the values of the loads are scaled. The scaling is based on the projection of the element on a plane normal to the load direction relative to the element length. For loads applied in local direction no projection scaling is performed.

The calculated load on an element is applied as a trapezoidal load. (Type 8).

8.1.11 Hydrostatic water pressure

Water pressure is calculated based on the global ordinate Z and the defined zero water level. For elements or part of elements above this zero level there will be no load applied. The calculated load on an element is applied as a trapezoidal load. (Type 8).

8.1.12 Soil pressure

Soil pressure results in both vertical pressure and horizontal pressure on an element. The projection of the pressure is applied on the element. The calculated load on an element is applied as a trapezoidal load. (Type 8).
A width of 1 m is assumed when calculating the change in soil pressure vs. depth.

Note!
It is important to verify that the result of the automatic load application is as intended.

8.2 Traffic loads

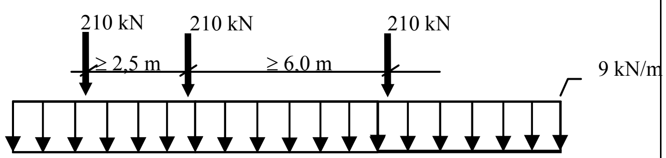
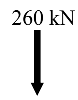
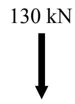
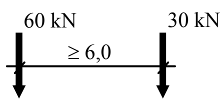
You define your traffic lines in the ***Traffic lines*** input card and your traffic loads in the ***Traffic loads*** input card.

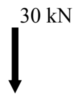
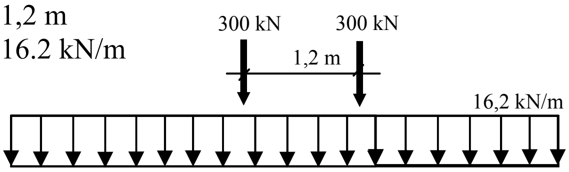
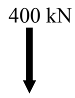
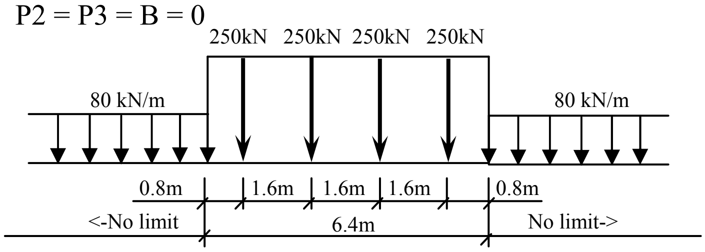
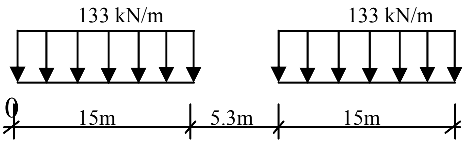
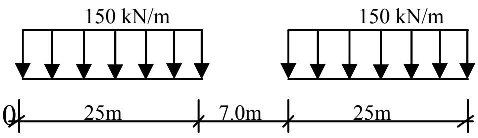
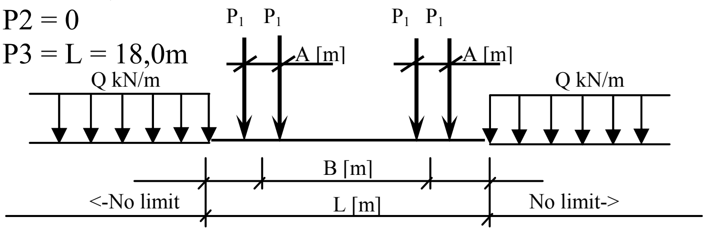
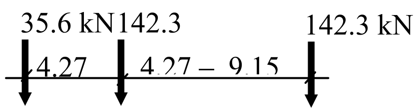
You can define loads or sets of loads with variable position along a specified line of elements. This variable load can be highway, pedestrian or railway specific. The program will calculate the most unfavourable position of the traffic load and the section forces for this position.

The first step of defining a traffic load is to define a traffic line, which consists of a sequence of elements connected with each other. For these elements a number of position are selected. The required number of position will depend on the length of the elements and what type of traffic load which travels along the line. The user must verify that sufficient accuracy is obtained.

Note!:
- *The elements on a traffic line must be defined in the same sequence as the traffic load passes them.*
- *Each traffic load case automatically allocates 12 load case numbers. This means that if you for instance define a traffic load case as no 15, the next free load case number for you to use is no 27.*

There are several predefined traffic loads available. These are described below with reference to the regulations on which they are based.

Type:	Description:	Ecc:	Input
1	Highway V1 Ref. /4/	Max/ Min	Q = 9 kN/m P1 = P2 = P3 = 210 kN A = 2,5 m (dist. P1 $\rightarrow$ P2) B = 6,0 m (dist. P2 $\rightarrow$ P3) 
1	Highway V2 Ref. /4/	Max/ Min	P1 = 260 kN A = B = Q = P2 = P3 = 0 
1	Highway V3 Ref. /4/	Max/ Min	P1 = 130 kN Q = P2 = P3 = 0 A = B = 0 
1	Pedestrian G1 Ref. /4/	Max/ Min	Q = 4 kN/m^2*width (Width must be given by user) P1 = P2 = P3 = 0 A = B = 0
1	Pedestrian G2 Ref. /4/	Max/ Min	P1 = 30 kN P2 = 60 kN A = 6,0 m Q = P3 = B =0 

1	Pedestrian G3 Ref. /4/	Max/ Min	P1 = 30 kN Q = P2 = P3 = 0 A = B = 0 
1	Highway LM1 Eurocode 1 Ref. /12/	Max/ Min	P1 = 300 kN P2 = 300 kN A = 1,2 m Q = 16.2 kN/m 
1	Highway LM2 Eurokode 1 Ref. /12/	Max/ Min	P1 = 400 kN Q = 0 A = B = 0 
2	Railway UIC71 Ref. /5/	Max	Q = 80 kN/m P1 = 250 kN A = 1,6 m P2 = P3 = B = 0 
3	Railway SW/0 Ref. /5/	Max	Q = 133 kN/m A = 15,0 m B = 5,3 m P1 = P2 = P3 = 0 
3	Railway SW/2 Ref. /5/	Max	Q = 150 kN/m A = 25,0 m B = 7,0 m P1 = P2 = P3 = 0 
4	Subway	Max	Q = 27,8 kN/m (= 4*P1/L) A = 2,1m B = 7,0 m P1 = 125,0 kN P2 = 0 P3 = L = 18,0m 
6	AASHTO HS20-44 Truck load Ref. /8/	Max/ Min	A = 4,27 m B = 9,15 m P1 = P2 = 142,3 kN P3 = 35,6 kN Q = 0 

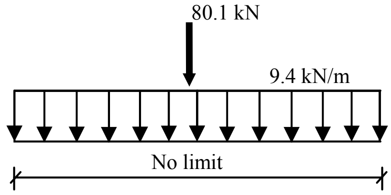
6	AASHTO HS20-44 Lane load Ref. /8/	Max/ Min	Q = 9,4 kN/m P1 = 80,1 kN P2, A = * P3 = B = 0  * : See note regarding maximum negative moment.

Figure 4.2.1 Available traffic load descriptions

The load values are positive in negative Z-direction. Eccentricities are positive in element L-direction.

A traffic load may consist of several tracks (lanes) of loads in order to support the design of multilane highways. Each track can have variable transverse position within a maximum and minimum value, see figure 4.2.2.

There are some restrictions on the use of railway traffic loads:

- Minimum eccentricity is not in used.
- Eccentricity maximum value is used with given sign.
- For load of type UIC71 all the four point loads are assumed to be present.

Be aware of the following items when you are using AASHTO traffic loads:

- Only HS20-44 loads are implemented. You must adjust the values for HS20-44 to model H15-44, H20-44 and HS15-44. Also the value for concentrated load in the lane load configuration used to determine maximum shear must be adjusted manually.

- For lane load you must include an extra concentrated load to find maximum negative moment, ref. 3.11.3 /8/. This load must be included manually. You should consider carefully the actual spacing between the two concentrated loads to make sure that they are always located in separate spans as required.

- To make sure that the spacing between the concentrated axle loads will be within the required accuracy, you should not model your traffic position spacing larger than A/2 = 2.14 m.

- The distances A and B are as follows: A: minimum distance between P1 and P2. B: maximum distance between P1 and P2. A: also the distance between P3 and P1 or P2, depending on what is most unfavourable. If the given value of B is less than A, distance between P1 and P2 will be A.

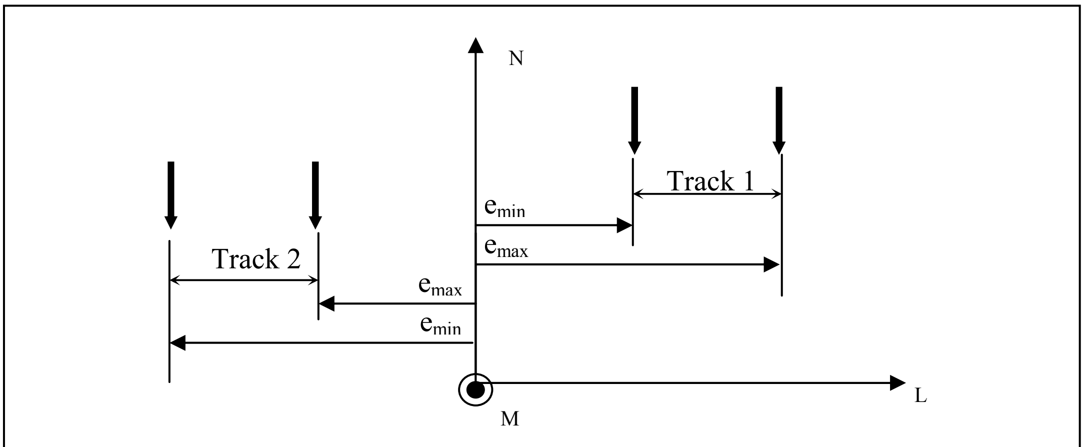


Figure 4.2.2 The definition of tracks end eccentricities.The traffic load is positioned at e_{maks} or e_{min}.

For each design section the results from a traffic analysis is 12 sets of section forces, max and min. for each of the six section force components with corresponding section forces. For each track in the traffic load a line of codes are presented which shows detailed description of the position of the traffic load. These codes are:

For highway and pedestrian bridges:

- 0- No load.
- 1- Q with eccentricity ecc. max..
- 2- Q + P1 with eccentricity ecc. max.
- 3- Q + P2 with eccentricity ecc. max.
- 4- Q + P3 with eccentricity ecc. max.
- 6- Q with eccentricity ecc. min..
- 7- Q + P1 with eccentricity ecc. min.
- 8- Q + P2 with eccentricity ecc. max.
- 9- Q + P3 with eccentricity ecc. max.

Note: If Q = 0, The codes 1 and 6 are not shown.

For railway design:

- 0- No load.
- 1- Distributed load Q is present.
- 2- 1 Point load is present at sub element.
- 3- 2 Point loads are present at sub element.
- 4- 3 Point loads are present at sub element.
- 5- 4 Point loads are present at sub element.

A short description on how the calculations are performed is presented below:
Positions on the elements of the traffic line define the number of sub elements for each element. This choice will therefore affect the accuracy of the calculations. For each position (sub element) on an element there are two unit loadcases generated, a centric point load and a torsion moment load. The unit loads are positioned at the centre of each of the sub elements.
The application performs an ordinary static analysis in order to calculate section forces for all the unit load cases.
The most unfavourable position of a traffic load is then calculated by adding and scaling these unit load cases within the rules that apply to the actual traffic load description. Point loads are always positioned at a specific position, never in-between.

There are a few sources of inaccuracies that need to be considered:

- The actual distributed load is converted to point loads on the middle of each sub element.
- A point load can not be closer to a support than half the length of a sub element.
- The length of sub elements can be too large to describe the distance between the point loads with sufficient accuracy.

It is recommended that the user perform tests in order to find the number of positions, which gives sufficient accuracy. See the appendices for examples on the use of traffic loads.

8.3 Dynamic loads

You define your dynamic loads in the ***Dyn Loads*** input card.

Dynamic loads in NovaFrame relates to the results of:

- Dynamic stochastic wind analysis
- Earthquake analysis

These analyses are based on a modal technique. In order to perform these calculations an eigenvalue analysis must therefore be run.

8.3.1 Dynamic stochastic wind analysis

NovaFrame includes a module for stochastic dynamic wind analysis. This module is used to calculate the dynamic response of a wind sensitive structure to turbulent wind. The method used is often called "Buffeting Analysis" originally proposed by A. G. Davenport in early 1960s., see for example ref. /9/.

The response of the wind is divided into two parts

- Static response (calculated in an ordinary static analysis), section 5.1.7.
- Dynamic response calculated by means of the stochastic wind analysis described in this section.

The stochastic analysis uses the structures eigenfrequencies and mode shapes to to determine the dynamic response. It is therefore necessary to perform an eigenvalue analysis , see section 7.1, before the stochastic analysis is started. In the stochastic analysis the contribution from each mode is assumed to be uncorrelated and are summed up using the RMS (root mean square) method.

In the stochastic analysis all components of the wind turbulence (horizontal in the wind direction, horizontal normal to the wind direction and vertical normal to the wind direction) are taken into account provided that the corresponding wind area and shape factors are defined, see section 3.6.

The input for a stochastic wind analysis is given in the dialog **Preprocess->Load Data->Dyn. Loads.** Choose DYNAMIC WIND in the load type field. The following input is needed:

- The loadcase number where results are to be saved.
- The modes which shall be considered (First mode, Last mode, step is 1)
- The spectrum type (default is the Kaimal spectrum defined in NS3491-4). Note that other spectra or spectra parameters can be defined in the dialog **Preprocess->Spectra ->Wind Spectra**
- The basic wind speed (as defined in NS3491-4). The basic wind speed is the 10min mean wind speed 10m above ground if the terrain was of type 2 (II).
- Horizontal wind direction, positive angle from X-axis towards Y, [deg.].
- Terrain category as defined in NS3491-4.
  0: Rough open sea
  1: Rough sea near shore, smooth flat country
  2: Farmland
  3: Suburban, industrial area or forest

4: Urban areas

- Mechanical damping, (relative to critical dampening);
  - β = 0.008 steel structures and non-cracked concrete
  - β = 0.016 cracked concrete

Note that the aerodynamic damping is automatically calculated by the program.

Reference is made to /9/.

8.3.2 Earthquake excitation

NovaFrame also offer the possibility to perform a dynamic response analysis for seismic loading of bridges. The approach included in NovaFrame is based on the response spectrum method using modal decomposition analysis to represent the structural response for a multi-degree-of-freedom system.

For Norwegian bridges the procedure for seismic response analyses are described in ref. /7/ and in more detail in ref. /10/. Procedures included in Version 4.0 of NovaFrame follow these guidelines.

In general a response spectrum analysis for seismic loading is a rather simple type of analysis. The response spectrum, which define the loading, give in principle the response for a so-called SDOF (single degree of freedom) system. With the use of modal analysis the structures response in terms of deflections and sectional forces are calculated for each mode. The values for each mode are calculated by multiplying the mode unit deflection with the value of the spectrum at the mode eigenfrequency. To obtain the total response for the structure, the response from each mode must be combined. As a basis NovaFrame assume the modes to be non-correlated and the contribution from each excited mode is added with the use of the RMS method. This is not correct for closely spaced modes where the CQC-method should be used. Presently NovaFrame only offers the possibility to inspect the results for closely spaced modes in order to determine whether a CQC-procedure is needed.

An earthquake calculation usually demands a large number of eigenfrequencies and modes in order to give sufficient accuracy. The value of the mode unit deflection is also multiplied with the mass distribution in order to obtain a mass ratio; this is actually a normalised modal mass ratio giving a "modal participation factor". This factor is meant to give an indication on whether sufficient number of modes is included or not. Commonly it is requested that this factor should be above 0.90 for each of the three directions when enough modes are included. One should be aware of that this only gives an indication. A thorough inspection of participating modal shapes is needed to assure that enough modes are included.

The basic start point for a seismic loading analysis is defining the response spectrum. For a structure located directly on the bedrock, two different response spectra are pre-defined in NovaFrame. One is the NPD response spectrum defined in ref. /6/ and commonly used for structures located in the North Sea on the continental shelf. The other is the spectrum defined by "Seismic Zonation of Norway" and also included in ref. /7/ to be used for Norwegian bridges.

After having defined the response spectrum to be used, the next point is to decide the peak ground acceleration for the location of the analysed structure. For the two spectra defined above, this is in principle the acceleration at 40 Hz and is used to scale the above spectra. For Norwegian bridges values of this parameter is defined in ref. /7/ by contour lines on a seismic zonation map for Norway.

In general earthquake loading consist of simultaneous accelerations in three orthogonal directions. The accelerations in each direction may be assumed to be uncorrelated. The horizontal acceleration transverse to the main direction may be assumed to have a lower content of energy than the main direction (approx. 85 %). Likewise may the acceleration in the vertical direction also be assumed to have lower energy content. This may be frequency dependent and ref. /7/ gives a possible definition for this purpose. Thus seismic loading should consist of two load cases, one for a main direction along the bridge axis and one for a main direction transverse to the bridge axis. These loadcases may be constructed from three basic loadcases, one for acceleration in each of the three orthogonal axis directions (x-dir, y-dir, z-dir). The two combined loadcases may then be constructed by RMS-combination of the basic loadcases as follow:

- Earthquake in X-dir: 1.00*x-dir, 0.85*y-dir, 1.0*z-dir
- Earthquake in Y-dir: 0.85*x-dir, 1.00*y-dir, 1.0*z-dir

where it is assumed that in vertical direction a special response spectrum for this direction is used (see ref. /7/ for a definition of this spectrum, also predefined in NovaFrame).

Response spectra give as stated above, the actual response of a SDOF-system. Damping for the system is thus included in the used spectrum. The predefined spectra in NovaFrame are developed for 5 % relative damping. In order to modify this damping two correction factor methods are included in NovaFrame. These give correction factor on the spectra values by the following functions:

$$D = 1.48 - 0.30 \cdot \ln(\beta) \qquad \text{ELOCS 1984 , } \beta[\%]$$

$$D = \sqrt{7/(2+\beta)} \geq 0.7 \qquad \text{Eurocode 8 , } \beta[\%]$$

In order to used this possibility, the used spectrum must define that damping correction should be allowed (and by which method, please refer to the section concerning definition of earthquake response spectra's). Then you may include the damping to be used in the definition of the loadcase.

The definition of a loadcase for seismic loading thus consists of the following items:

- The loadcase number where results are to be saved.
- The modes which are to be calculated (First mode, Last mode, step is 1)
- Type of response spectrum to be used (both predefined and specially defined spectra may here be used, please refer to the section concerning definition of earthquake response spectra's for more information)
- Ground acceleration "Ga", [m/s^2] used to scale selected response spectrum.
- K_x: Scaling of ground acceleration in X-direction.
- K_y: Scaling of ground acceleration in Y-direction.
- K_z: Scaling of ground acceleration in Z-direction.
- Modification of damping, see comments above and in section concerning definition of earthquake response spectra's

Ground acceleration values (i.e. Peak Ground Acceleration, PGA) for bridges in Norway are defined by ref./7/. Actual acceleration at frequency *f* is thus dependent upon three parameters: 1) Ground acceleration value, 2) Spectral normalization value and 3) Spectral frequency value at frequency *f*. Further information on this subject is given in section concerning definition of earthquake response spectra's.

A result from an earthquake analysis is presented for each mode with the following parameters:

- Mode no.
- Loadcase no.

- SVX [m/s]; spectra value in for X-direction.
- SVY [m/s]; spectra value in for Y-direction.
- SVZ [m/s]; spectra value in for Z-direction.
- Maximum displacement for this mode.
- MRX; “Mass ratio” in X-direction, participating mass.
- MRY; “Mass ratio” in Y-direction, participating mass.
- MRZ; “Mass ratio” in Z-direction, participating mass.

The presence of soil or sedimentary rock layers above the bedrock at a construction site will modify the ground motion characteristics. In order to include this effect in the dynamic response analysis some sort of soil-structure interaction analysis should be performed. To include this in a simple manner the structure should be supported on springs, which should simulate the effect of the foundation stiffness, and the response spectrum of the bedrock should be modified to simulate the response of a massless rigid foundation.

Foundation springs must be determined for the actual structure and the actual foundation used (please refer to the section concerning definition of springs for this type of input). Correction terms for the bedrock spectrum is in ref. /7/ given for two types of soil and the resulting spectra are included in NovaFrame. So also for this type of structures, a seismic loading analysis may easily be performed with the current version of the program.

See the appendices for examples.

.

8.4 Creep loads

You define creep loads in the ***Creep Loads*** input card. Prior to defining creep load you must define creep combinations. This is done in the ***Creep combination*** input card.

The possibility of calculating the effect of creep is included in this version of NovaFrame. Redistribution of section forces and displacements can be calculated.

In order to perform a creep calculation the elements of a frame structure must first have a defined stress level, which is obtained from previously defined and analysed loadcases.
Then each element of the active frame structure must have a partial creep factor, which is defined by the material and section properties, and the duration of the creep phase. The section forces are assumed to be constant within the time of the creep phase. At the end of the phase the changes in response is calculated. If the effect of creep redistribution is significant in relation to the total response then a creep phase can, in order to improve accuracy, be subdivided into shorter phases, see fig. 4.4.1.

Total response for a specific load is then calculated by the use of load combinations.

The required input for performing a creep analysis consists of the following input:

- First a stress level is defined by assigning previously defined loadcases to a creep combination. Several loadcases can then be added and multiplied with a load factor.
- A creep loadcase must be defined. This creep loadcase will subsequent to the analysis contain the redistribution of section forces and the changes in deflections. A creep loadcase assigns creep combinations and partial creep factors to the elements. Partial creep factors may be assigned directly by value or indirectly by giving the age of

concrete when concrete was first loaded and the start and end time of the current creep phase.

A creep calculator is included in order to ease the calculation of directly given partial creep factors. Please note that for NovaFrame defined sections of type 1, the creep calculator is unable to compute the efficient thickness h_0. The implemented calculation of creep factors is according to /2/-NS3473 pt. A.9.3.2.

Note!
One should be familiar with the basis of the method in order to understand its limitations and thereby be able to take the necessary steps in order to get reasonably accurate results. The accuracy of the calculations is dependent of the number of design sections of an element. The number of design sections should give a correct description of the actual force distribution along the element.

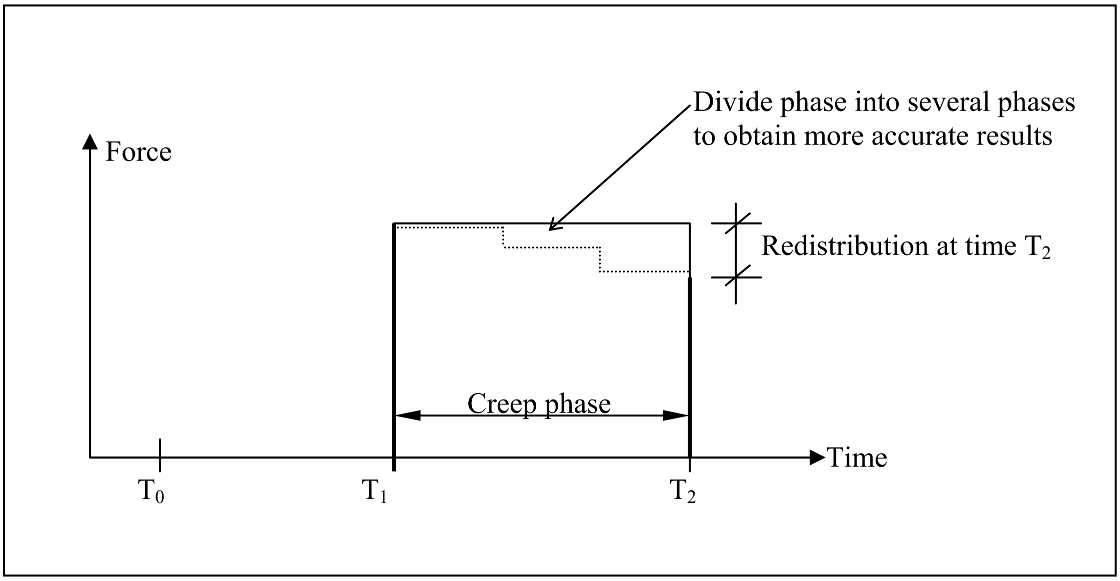


Fig 4.4.1 Creep phases.

See the appendices for various examples.

Note!
When using the automatic partial creep factor calculation option on NovaFrame defined sections (Type = 2 – 5) the associated concrete material is the material parameter set no 1. This set can be modified in NovaDesign. By default this is concrete quality C45. Relative humidity is 70%.

8.5 External forces and displacements

You define your external forces in the ***Displacements*** input card.
You define your external displacements in the ***Sect. Forces*** input card.

You can define the results of external loadcases or external ordinary combination in these input cards. This makes it possible to include results obtained elsewhere, for example from an analysis performed by an another application or based on hand calculations.

9 Buckling- and 2. order analysis

Normally equilibrium is formulated for the undeformed structure. This is termed a 1st order analysis. If equilibrium is formulated for the deformed structure the analysis is termed a 2nd order analysis:

$$([K_1] + [K_2])*\{w\} = \{P\}$$

where:

[K_1] – 1st order stiffness matrix. This matrix depends on the geometry of the frame and E-modulus and the section properties.
[K_2] – 2nd order stiffness matrix (geometric stiffness matrix). This matrix depends on the geometry of the frame and the axial forces in the elements.
{w} – displacement vector
{P} – load vector

The 2nd order effect can be included in all types of analysis.

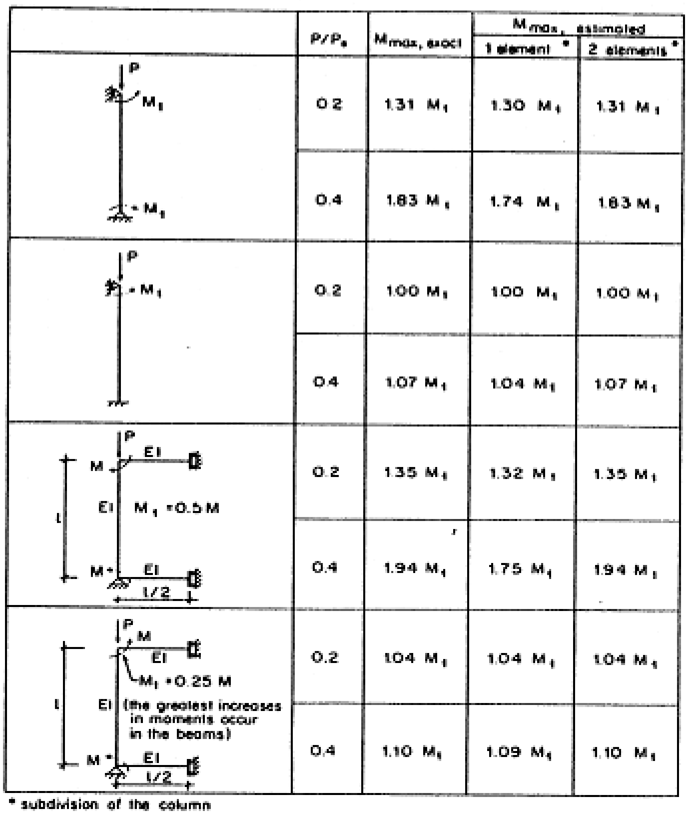
If the load vector is set equal to zero the equation reduces to the eigenvalue problem:

$$([K_1] + \lambda\ [K_2])*\{w\} = \{0\}$$

where
λ - buckling load factor
{w} – buckling mode

NovaFrame includes a module for buckling analysis.

The finite element method is an approximate calculation method where the structure is replaced by a discrete element model with assumed deformation modes. For 1st order calculations this is usually a good idealisation. For 2nd order calculations a coarse element model may give inaccurate results. It is also important that the 1st order moments has the same shape/path as the
a ... ith centric load will not have
a ... there will always be an additional
n ... mples are shown demonstrating
tl
C ... s.



	P/P_e	$M_{max, exact}$	$M_{max, estimated}$ 1 element *	2 elements *
Case 1	0.2	1.31 M_1	1.30 M_1	1.31 M_1
	0.4	1.83 M_1	1.74 M_1	1.83 M_1
Case 2	0.2	1.00 M_1	1.00 M_1	1.00 M_1
	0.4	1.07 M_1	1.04 M_1	1.07 M_1
Case 3	0.2	1.35 M_1	1.32 M_1	1.35 M_1
	0.4	1.94 M_1	1.75 M_1	1.94 M_1
Case 4	0.2	1.04 M_1	1.04 M_1	1.04 M_1
	0.4	1.10 M_1	1.09 M_1	1.10 M_1

* subdivision of the column

Fig. 4.10 Accuracy of computed moments in braced frames

9.

C ... axial force in the elements
n

T ... ıation of defined load cases. In
tl ... s where the 2nd order effects are

Fig 4.5.2 The accuracy of the method.

neglected (**[K$_2$]** = **[0]**). This is usually sufficient accurate since the axial forces are almost independent of the 2nd order effects.

The axial forces are input in the dialog **Preprocess->Buckling and Misc.->Axial force**

Note!

Only elements with design sections will be associated with sectional forces. It is the axial force in the first section of an element that is used.

Only one axial force pattern can be used in each analysis.

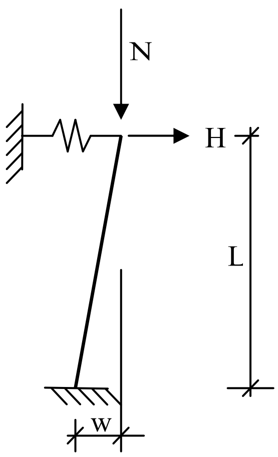
.

9.2 Buckling analysis

As mentioned the only input to the buckling analysis is the axial forces. The buckling analysis is started from the **Solve** dialog.

Output from the buckling analysis is the buckling load P_0 and the corresponding buckling deformation shape (buckling mode). P_0 is the factor all axial forces must be multiplied with in order to reach the buckling load level.

9.3 2. Order analysis



w – displacement
K – spring constant

Equilibrium non-deformed system: $K \cdot w = H$

Equilibrium deformed system: $K \cdot w = H + N \cdot w / L$

Which gives: $(K - N / L) \cdot w = H$

"Geometric stiffness matrix": N / L

With reference to the figure above, the force F in the spring is:

$$F = K * w = K * H / (K - N / L)$$

In a 1. order analysis (N=0) this equation reduces to :

$$F = H$$

In a second order analysis the spring force increases due to the axial force N.

It is important to note that when you include the 2nd order effect in your analysis you get out the total forces including the "additional 2.order moment".

An important presumption in order to calculate 2nd order moments is that the assumed bending stiffness in the analysis (E·I) is less or equal to the actual bending stiffness calculated when taking into account both reinforcement, and the non-linear properties (cracking) of concrete. This applies for sections both in beams and in columns.

9.4 Buckling length

In connection with the buckling analysis the buckling length for each element is determined for each mode. The buckling length is calculated from the formula:

$$l_{critk} = \pi \sqrt{\frac{EI}{\lambda * N}}$$

where

EI – stiffness in the buckling direction
λ – buckling factor
N – axial force in considered element

The program automatically determines the buckling direction from the buckling mode.

It is important to include enough buckling modes in the buckling analysis to determine the buckling length in both L- and N direction. The largest buckling length in the two directions are stored for later use in design. The calculated buckling length are found in the list box **Preprocess->List..-> Miscellaneous->Bucklinglength.**

10 Using models and multiple analysis

For structures which are constructed and loaded in different sequences it is required to run several analyses with different static models. There are facilities included in NovaFrame enabling this to be handled effectively. Choose **Models and Analyses** in the **Preprocess** menu in the ***Frame*** window to locate the appropriate input cards.

10.1 Models

Use the ***Models*** input card to define the different analysis models.

Defining models (= sub models of geometry) gives you the possibility to have different structural models available.

A model consists of a selected set of elements. This selection can be made by:

- adding elements
- adding elements to models
- deleting elements from a model
- adding models to previously defined models
- deleting models from previously defined models

The model will then consist of the elements and their properties and the nodes of each element as defined in section 3.

If a model is derived from a previously defined model, the properties of those elements and nodes are maintained also. This is important to remember if one uses the facilities outlined in the next section.

Note: The "0-Model" is reserved for use by the program. This is the "Default All" model, which contain all the geometry data defined in the geometry cards, i.e. all elements. It is possible to refer to model 0 when defining new models.

10.2 Modifying model properties

Use the ***Model modify*** input card to define your modifications to a model.

The input given in these cards will either change the node or the element properties
Node properties:

- Boundary conditions
- Master slave settings

Element properties:

- Cross section no. associated with element
- Mass properties
- Axial force properties
- Wind areas
- Joints

For detailed input see section 10, ASCII-Input.

10.3 Calculation groups

Use the ***Calc group*** input card to define your calculation groups.

Making a calculation group is a method of selecting loadcases, traffic loads or a calculation into a named group. A calculation group also contains the information concerning the type of analysis, which is to be in the contents of the calculation group.

The following types of analyses are available:

- Ordinary static analysis, contents of calculation group is loadcases.
- Traffic load analysis, contents of calculation group is traffic loads.
- Creep analysis, contents of calculation group is a single creep load (case).
- Eigenvalue analysis, contents of calculation group are number of modes.
- Earthquake analysis, contents of calculation group is dynamic load (cases).
- Dynamic wind analysis, contents of calculation group is dynamic load (cases).
- Buckling analysis, contents of calculation group are number of modes

10.4 Analysis

Use the ***Analysis*** input card to define your analysis.

The input in these cards will define an analysis consisting of a model and a calculation group.

When solving this analysis the geometry of the selected model will be analysed for the contents of the selected calculation group.

See the examples in the appendices, especially the "MyBridge" tutorial, which shows the use of several models and multiple analyses.

Analysis set-up can be defined if the user wish to include geometric stiffness matrix to specific analyses. Please refer to the ANSETUP command in appendix 1.

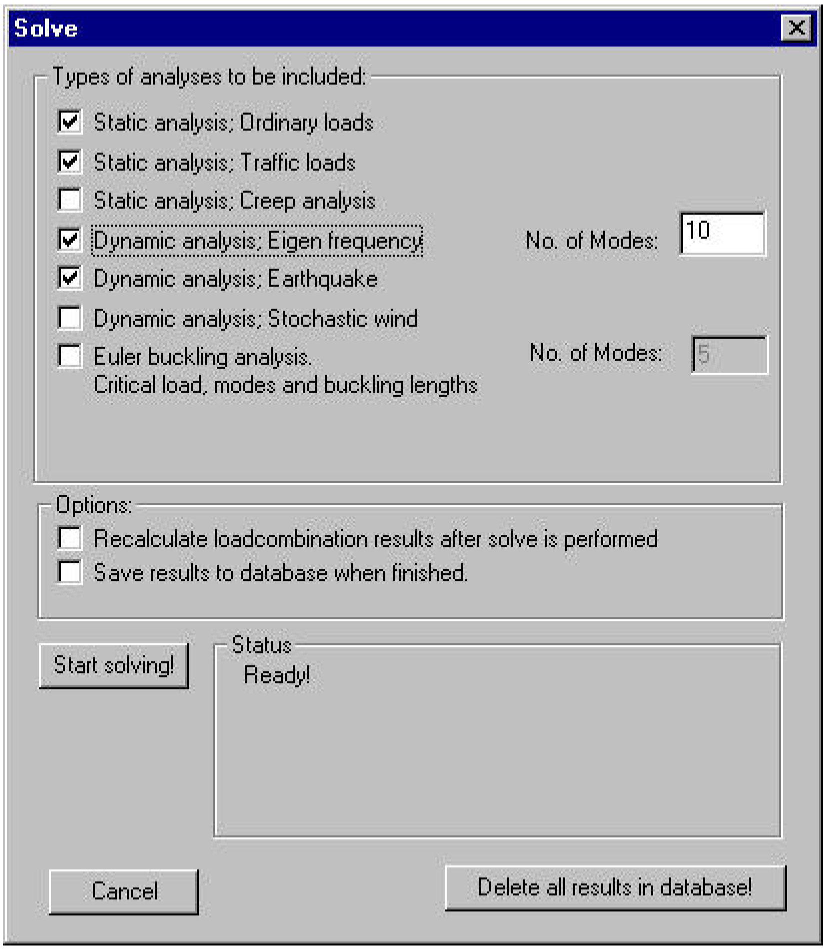
11 Starting analyses

When choosing the **Solve** menu in the frame window, a dialog box will appear and enable you to start the analyses.

The appearance of the dialog box will depend on whether or not additional models have been defined. If you use sub models, your analysis statements must define the contents of your analysis. See section 5.

11.1 Single analysis

If there are no additional models defined (default) then the solve dialog box will look like this:

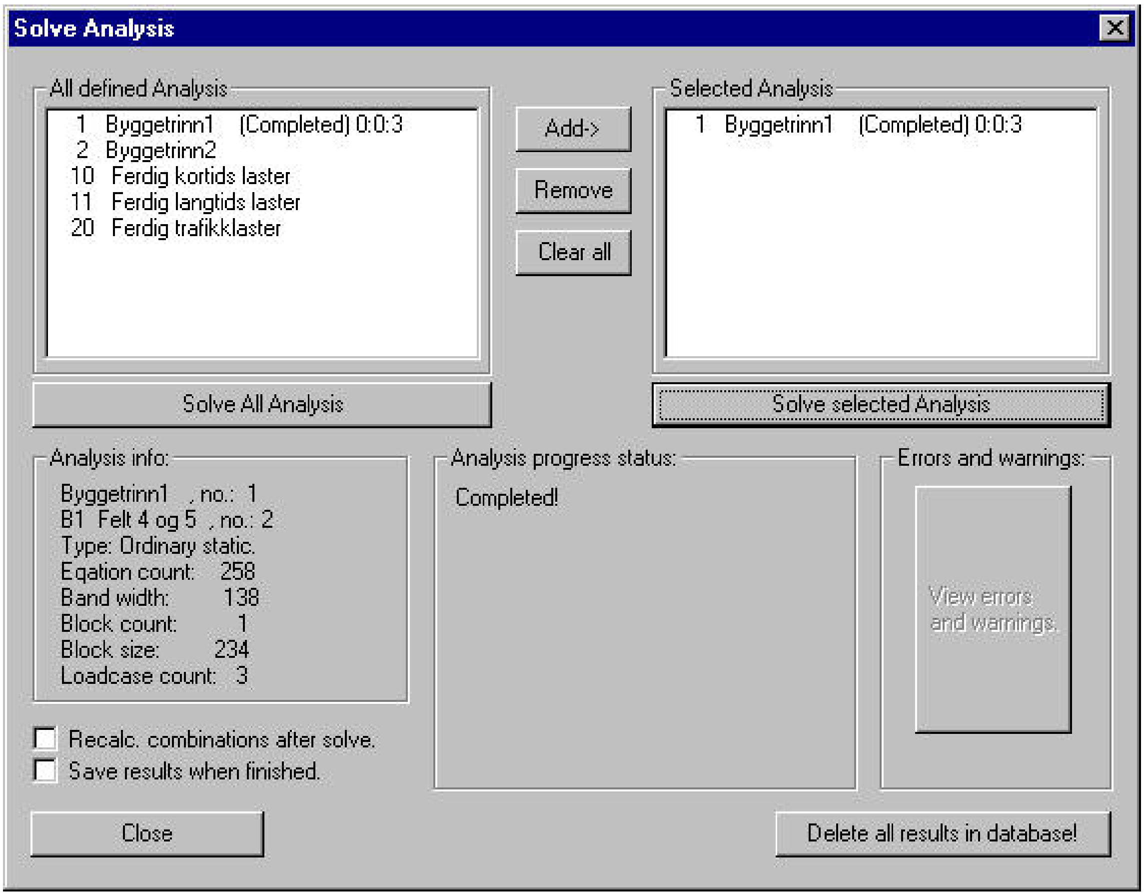


Select the types of analyses and then press the start button. If no errors are found then the analyses will be performed. The status window will show the progress of the current analysis.

Geometric stiffness matrix is only included in the buckling analysis.

11.2 Multiple analyses

If additional models are defined then the dialog box will look like this:



If all analyses are to be performed then press the "Solve all analysis" button. A selection of analysis can alternatively be run. Errors and warnings can be viewed by pressing the button as shown.

Geometric stiffness matrix is always included in the buckling analyses. The user may however include the geometric stiffness matrix to other types of analyses when using multiple analyses. In order to achieve this it is required to define ANSETUP commands.

12 Post-processing and load combinations

Depending on the type of analyses which is performed there will be various results available for plotting and printing. These are displacements, section forces, equilibrium checks, support reactions, accelerations and eigenmodes.

Displacements and section forces can be combined into load combinations. In this version of NovaFrame, both ordinary load combinations and sorted load combinations are supported.

Choose **Loadcomb…** in the **Postprocess** menu in the ***Frame*** window to find the relevant input card.

12.1 Results

Types of available results are:

- Node displacements in global coordinates.
- Section forces in the local element coordinate system, in the defined design sections.
- Support reactions in global coordinates.
- Equilibrium checks in global coordinates.
- Node accelerations in global coordinates, as a result of a dynamic analysis.
- Eigenfrequencies and eigenmode shapes with nodal displacements in global co-ordinates.
- Stress calculation. Calculates normal stresses in selected section points. The section points are selected and associated with faces by using the SECTFACE command.

Equilibrium checks are performed by adding all active loads acting on the model and adding all node reaction forces in the model. These values should of course balance, i.e. with opposite signs for each main direction in the global co-ordinate system.

Use the ***Listing of results*** dialog box menu to list the actual results for both load cases and combinations. This dialog box is opened from the ***Frame*** window by choosing **List results...** in the **Postprocess** menu. The results can be copied to an ASCII-file, printed directly on a printer or copied to the document.

Node displacements and section forces can be plotted on the frame model. This is also possible for eigenmodes, which can also be animated. Plots are requested from the ***Results to plot*** dialog box in the ***Frame*** window. The plots can then be copied to the document by choosing **Copy to document...** in the **Display** menu in ***Frame*** window or printed directly by choosing **Print...** in the same menu.

For traffic loads it is possible to plot the most unfavourable position of the described load. One must select: traffic load no., which tracks, element no., design section and the section force component in order to get the results of one position. This is done in the **Postprocess.. plot traffic pos..** menu.

Definition of positive section forces is shown in figure 7.1.1.

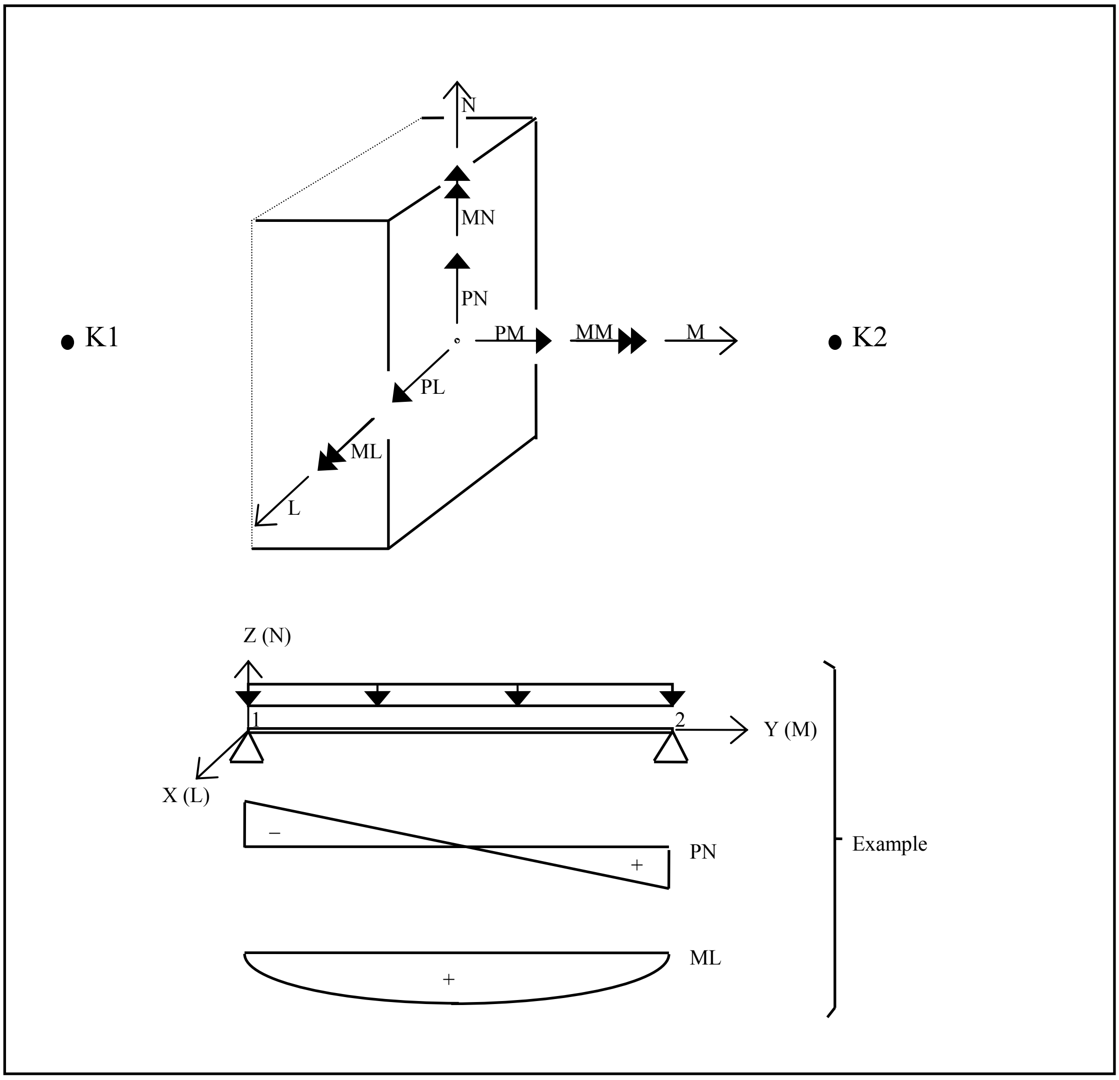


Figure 7.1.1 Positive section forces

12.2 Ordinary load combination

Use the ***Ordinary loadcombination*** input card to establish the ordinary load combinations.

The ordinary load combination gives you the possibility to combine actual load cases with specified load factors. You can specify as many loadcase-factor pairs as you want to for each combination. You can also include already defined combinations in a new load combination.

You can define load cases and load combinations using the same numbers.

12.3 Sorted combinations

Use the ***Sort. loadcombination*** input card to establish sorted combinations.

Sorted load combination gives you the possibility to find maximum/minimum force components amongst a large number of combinations. There are 6 components of element forces: PL, PM, PN, ML, MM and MN. A sorted load combination consists of 12 sets of forces, maximum and minimum of each load component with corresponding forces.
A sorted combination can be a combination of load cases, ordinary load combinations, traffic loads or an already defined sorted combination, with specified factors.
You can specify as many load cases/ ordinary combinations/ traffic loads/ sorted combination - factor pairs as you want for each combination.

There are five methods by which loads can be combined:

1- Adds all forces.
2- The most unfavourable for each of the specific load component is selected.
3- The most unfavourable for each of the specific load component is selected, the sign can be positive or negative.
4- Adds the forces if they are unfavourable for the actual component.
5- Adds the forces if they are unfavourable for the actual component, the sign can be positive or negative.

The method can not change within a sorted combination if several lines of input is used.

Limit state is not in use for sorted combinations. The option is however included as a part of the input. Limit state can be used in order to logically group sorted combinations.

Tutorial examples in appendices contains a detailed description on the use of sorted combinations.

12.4 Sorted combination lines

Use the ***Sort line loadcombination*** input card to calculate results for sorted combinations.

Sorted combination lines gives you the possibility to find the largest set of force components for each design section for the elements that you specify. The largest forces are calculated among the sorted combinations that are included in the definition of the sorted combination line. You can limit the amount of results to be listed by choosing the specific force components that are to be calculated.

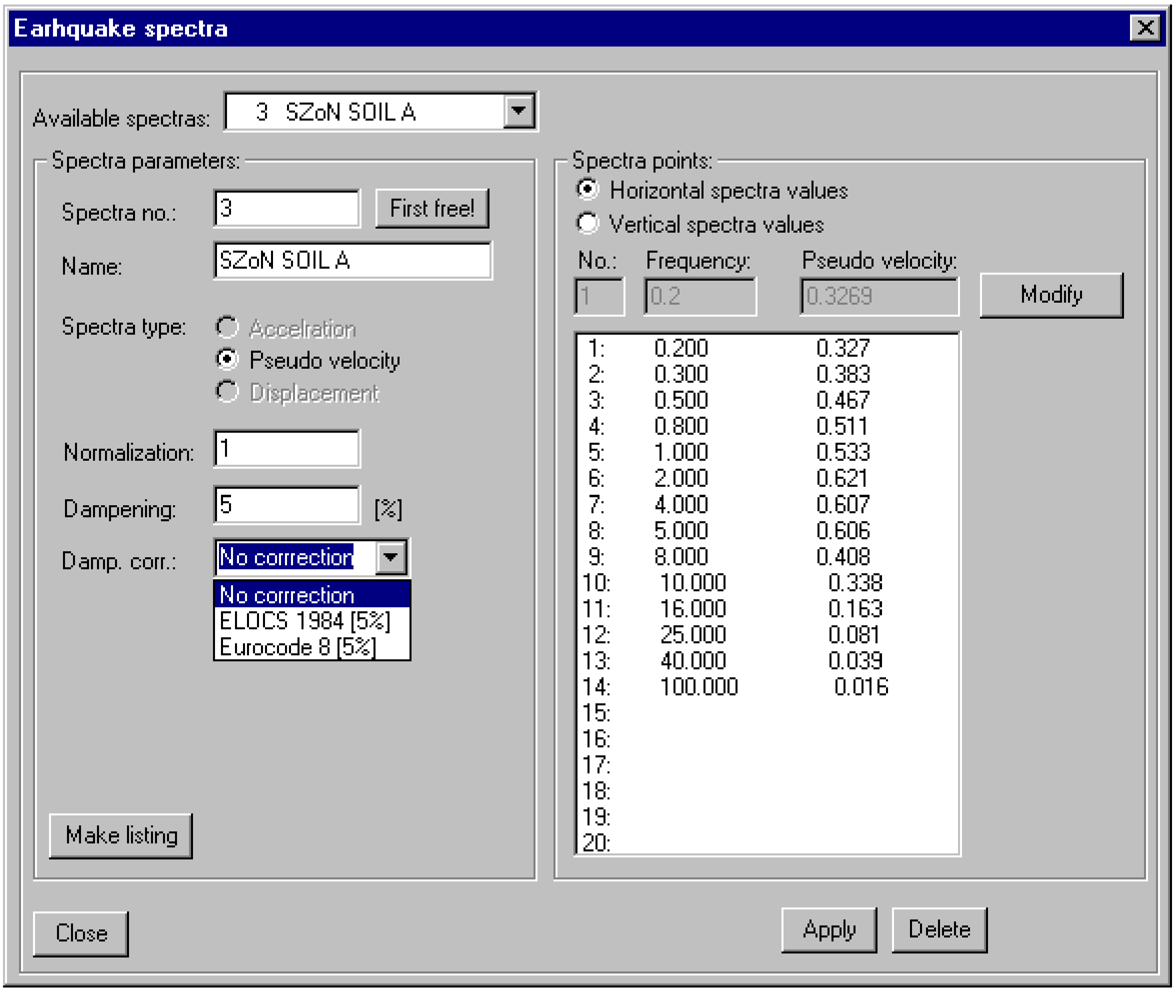
13 Miscellaneous

13.1 Earthquake spectres

NovFrame includes several default earthquake response spectra's. Most of the spectra's are related to Norwegian rules and regulations.

The user may however define their own response spectra's customized to their own needs.

From the **Frame->Preprocess->Spectra menu**, this user dialog is available:



The user defines the response spectra by inserting points in the right part of the dialog. Currently only pseudo velocity is supported as spectra value. Due care should be made in when specifying the normalization factor.

Both horizontal and vertical response spectra values can be given. If no vertical spectra is given, then the horizontal spectra will be used.

Damping used for calculating the response spectra. The spectra can be modified when specifying a damping correction method. The included methods are both related to an initial damping of 5% in the response spectra.

Correction factor on the spectra values are given by these functions:

$$D = 1.48 - 0.30 \cdot \ln(\beta) \qquad \text{ELOCS 1984 , } \beta[\%]$$
$$D = \sqrt{7/(2+\beta)} \geq 0.7 \qquad \text{Eurocode 8 , } \beta[\%]$$

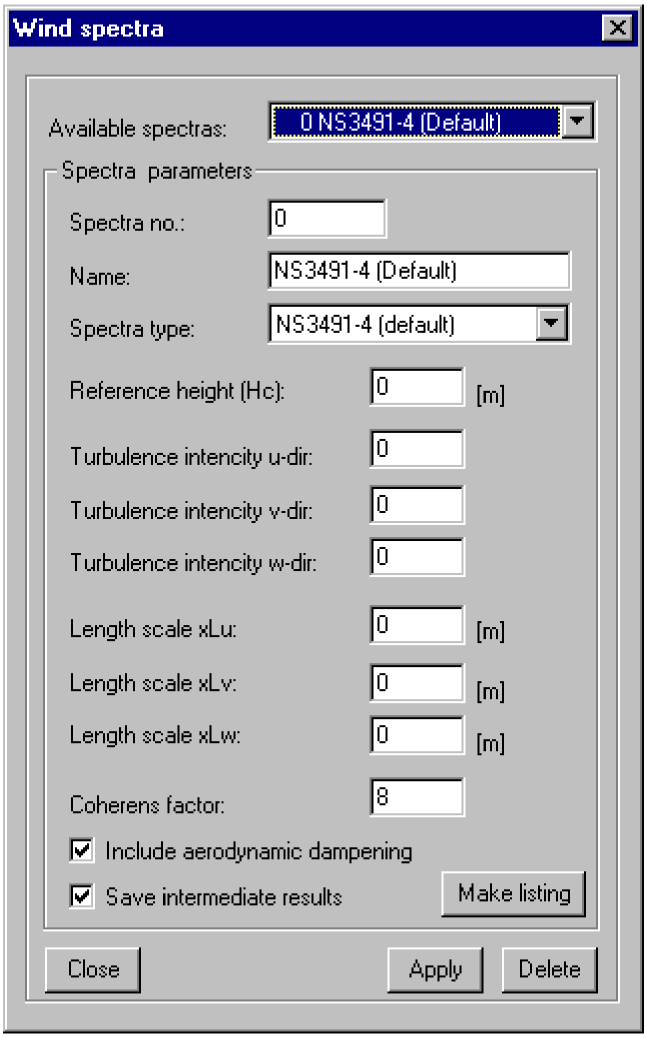
The earthquake spectra can be represented by ASCII commands, please refer to the EQSPEC and EQSPECPT commands.

13.2 Wind spectres

NovFrame includes a few default wind spectra's. Most of the spectra's are related to Norwegian rules and regulations.

The user may however define their own spectra's customized to their own needs.

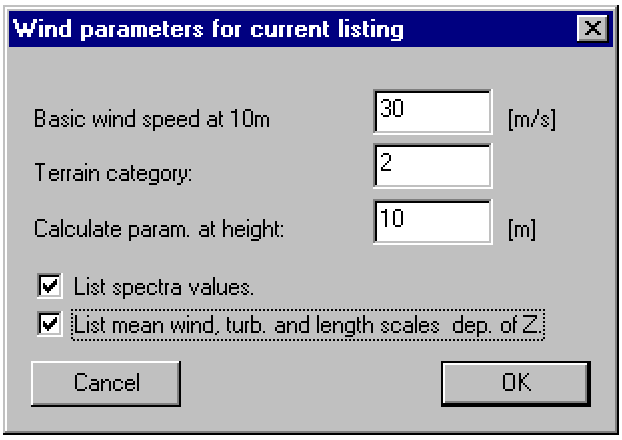
From the **Frame->Preprocess->Spectra menu**, this user dialog is available:



The user defines (modifies) the wind spectra by inserting a reference height. The spectra parameters such as length scales and turbulence intensities will be calculated accordingly at this height. By default if no reference height is entered the maximum node Z-value for the structural model will used as the reference height.

Each of the length scales and turbulence intensities can overridden with user given values

Listing of wind spectra values:



The listing will look like this:

```
WIND SPECTRAS:

WIND SPECTRA NO.          :  0
-------------------------------------------------------
Wind spectra name         :  NS3491-4 (Default)
Spectra type              :  NS3491-4 (Default)

Spectra values            :
Basic wind speed at 10m   :    30.000 [m/s]
Terrain category          :     2
Reference height          :    10.000 [m]

Wind speed at ref. height:    30.200 [m/s]
Turbulence intencity Iu   :     0.189
Turbulence intencity Iv   :     0.147
Turbulence intencity Iw   :     0.104
Length scale xLu          :   100.000 [m]
Length scale xLv          :    23.729 [m]
Length scale xLw          :     8.320 [m]
Coherence factor          :     8.000
Include aero. dampening   :       Yes
Include intermediate res.:       Yes


     Freq          w    lg(Freq)    Suu*Freq    Svv*Freq    Svv*Freq
     [Hz]    [rad/s]   [lg(Hz)]     sigu**2     sigv**2     sigw**2
------------------------------------------------------------------
    0.010      0.063     -2.000       0.137       0.032       0.011
    0.020      0.126     -1.699       0.188       0.065       0.022
    0.030      0.188     -1.523       0.208       0.102       0.033
    0.040      0.251     -1.398       0.214       0.139       0.045
    0.050      0.314     -1.301       0.214       0.174       0.057
    0.060      0.377     -1.222       0.210       0.205       0.069
    0.070      0.440     -1.155       0.206       0.230       0.082
    0.080      0.503     -1.097       0.200       0.250       0.095
    0.090      0.565     -1.046       0.195       0.265       0.108
    0.100      0.628     -1.000       0.189       0.276       0.121
    0.100      0.628     -1.000       0.189       0.276       0.121
    0.200      1.257     -0.699       0.146       0.269       0.231
    0.300      1.885     -0.523       0.120       0.228       0.280
    0.400      2.513     -0.398       0.103       0.195       0.288
    0.500      3.142     -0.301       0.090       0.171       0.279
    0.600      3.770     -0.222       0.081       0.153       0.265
    0.700      4.398     -0.155       0.074       0.139       0.250
    0.800      5.027     -0.097       0.068       0.128       0.235
    0.900      5.655     -0.046       0.064       0.118       0.222
    1.000      6.283      0.000       0.060       0.110       0.210
    1.000      6.283      0.000       0.060       0.110       0.210
    2.000     12.566      0.301       0.039       0.070       0.139
    3.000     18.850      0.477       0.030       0.053       0.107
    4.000     25.133      0.602       0.025       0.044       0.089
```

5.000	31.416	0.699	0.021	0.038	0.076
6.000	37.699	0.778	0.019	0.034	0.068
7.000	43.982	0.845	0.017	0.030	0.061
8.000	50.265	0.903	0.016	0.028	0.056
9.000	56.549	0.954	0.014	0.026	0.052

Z [m]	V(z) [m/s]	Iu	Iv	Iw	xLu [m]	xLv [m]	xLw [m]
10.000	30.200	0.189	0.147	0.104	100.000	23.729	8.320
20.000	34.151	0.167	0.130	0.092	123.114	29.218	10.247
30.000	36.462	0.156	0.122	0.086	139.039	33.005	11.581
40.000	38.102	0.150	0.117	0.082	151.572	35.992	12.637
50.000	39.374	0.145	0.113	0.080	162.066	38.501	13.529
60.000	40.413	0.141	0.110	0.078	171.177	40.687	14.312
70.000	41.292	0.138	0.108	0.076	179.279	42.640	15.017
80.000	42.053	0.136	0.106	0.075	186.607	44.415	15.664
90.000	42.725	0.133	0.104	0.074	193.318	46.050	16.266
100.000	43.325	0.132	0.103	0.073	199.526	47.573	16.832
110.000	43.868	0.130	0.102	0.072	205.314	49.002	17.371
120.000	44.364	0.128	0.100	0.071	210.744	50.354	17.888
130.000	44.821	0.127	0.099	0.070	215.865	51.640	18.386
140.000	45.243	0.126	0.099	0.070	220.718	52.869	18.870
150.000	45.636	0.125	0.098	0.069	225.334	54.050	19.342
160.000	46.004	0.124	0.097	0.069	229.740	55.188	19.805
170.000	46.350	0.123	0.096	0.069	233.956	56.290	20.260
180.000	46.676	0.122	0.096	0.068	238.003	57.359	20.709
190.000	46.984	0.121	0.095	0.068	241.895	58.399	21.154

14 Tutorial example 1

Use this chapter to work through a very simple example. By the time you have finished this chapter you will be familiar with the main capabilities included in NovaFrame.

14.1 The Tutorial frame

The frame to analyse in this chapter is shown in figure 8.1.1.

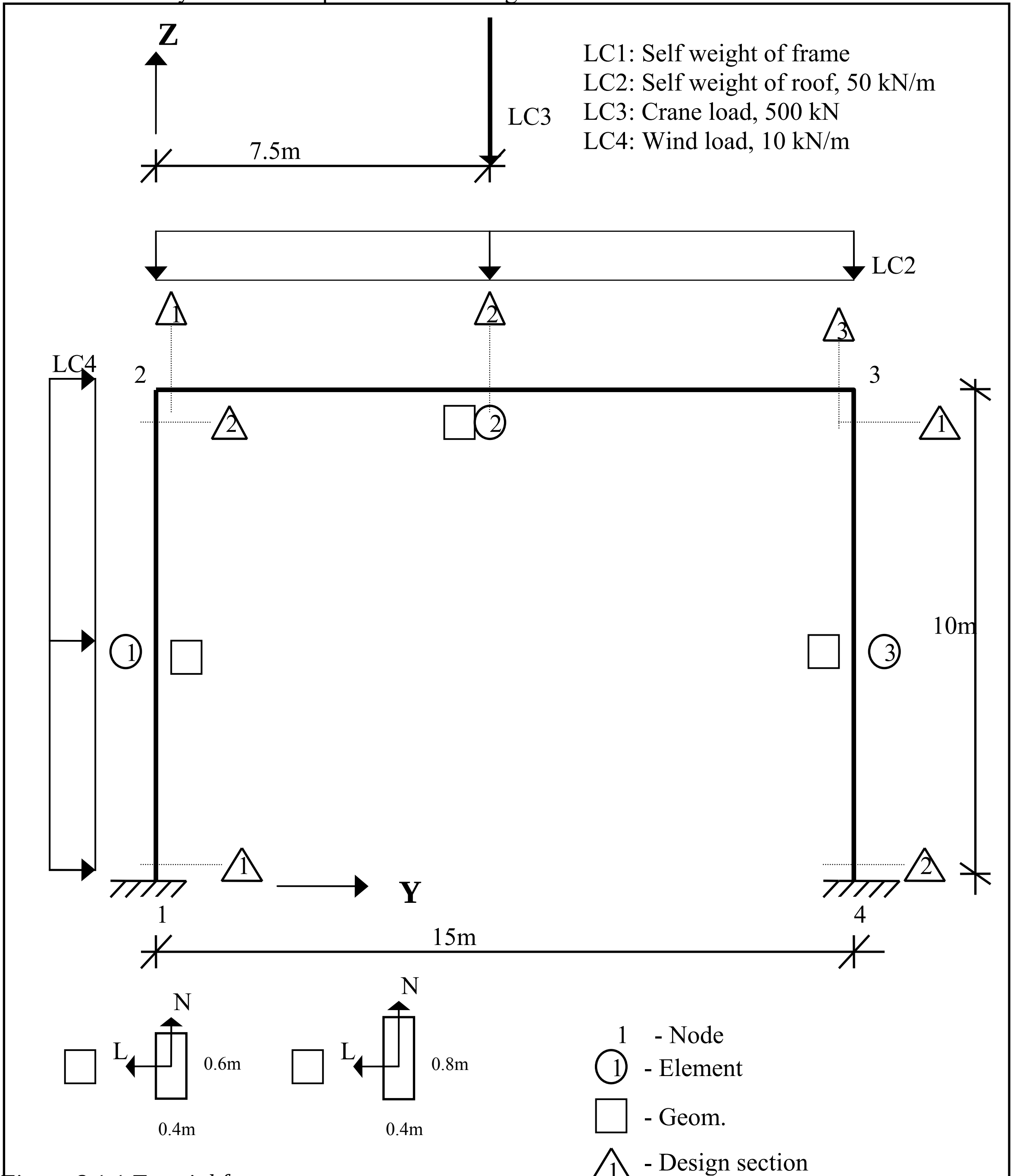


Figure 8.1.1 Tutorial frame

14.2 Input data

The first step in the analysis process is to define the geometry. It is important to decide how the frame should be modelled before you start to submit your geometry data. You should read chapter 3.1 to learn more about this aspect if you are an inexperienced user of frame analysis programs.

First step

- Start the program by clicking on the program icon.
- Open a new document and connect it to a new database by clicking on the leftmost tool button.
- Open the ***Frame*** window by choosing **Frame...** in the **Analysis** menu.
- Choose **Build model...** in the **Preprocess** menu.

You have now opened the stack of input cards in which you specify all your geometry and load input. You should access the cards in same order as they are shown in the stack. Access the ***Node*** card first.

Input cards

All the cards except the first one are equally designed. You use the edit fields and control buttons in the upper part of the cards to build an ASCII-input line. The values from these controls are converted to an input line in the ASCII-editor on the lower part of the input card when you click on **Insert** or **Delete**. You must click on **Apply** to transform the ASCII-line to actual model items.

Click on the **Help** button to get information on how to use the ***Input*** cards. You will also find information on how to copy, paste and cut in the editor.

If you click on the **Apply all** button, the **Apply** buttons on all ***Input*** cards will be processed in the same order as the cards are shown.

The new model items will be shown in the ***Frame*** window when you click on **Apply**, but the data is not saved to the database before you choose **Save** from the **Session** menu in the ***Frame*** window.

Actual in-data

Write the following input lines in the respective ***Input*** cards to model the Tutorial frame, see fig 8.2.1.

```
*** NODE INPUT ***
NODEINS  1  2  1  0.000   0.000   0.000  0.000  0.000   10.000
NOEDINS  3  4  1  0.000  15.000  10.000  0.000  0.000  -10.000


*** ELEMENT INPUT ***
ELEMINS  1  3  1  1  2  1


*** GEOMETRY SECTION INPUT ***
SECTINS  1  3  30000  0.2  0.400  0.600  0.000  0.000  Section 1
SECTINS  2  3  30000  0.2  0.400  0.800  0.000  0.000  Section 2


*** ELEMENT SPECIFICATION INPUT ***
ELSPINS  1  3  2  1  0  4
ELSPINS  2  2  0  2  0  4


*** BOUNDARY CONDITION INPUT ***
BOUNDINS  1  4  3  1  1  1  1  1  1


*** DESIGN SECTION INPUT ***
DESIGINS  1  3  2  0.200  0.800  0.600
DESIGINS  2  2  1  0.200  0.800  0.300
```

Figure 8.2.1 Actual input lines for the Tutorial frame geometry

```
*** LOAD INPUT ***
LOADINS  1  1  3  1  1  3   -25.000  0.000  0.000  0.000  Loadcase 1
LOADINS  2  2  2  1  3  3   -50.000  0.000  0.000  0.000  Loadcase 2
LOADINS  3  2  2  1  4  3  -500.000  7.500  0.000  0.000  Loadcase 3
LOADINS  4  1  1  1  3  2    10.000  0.000  0.000  0.000  Loadcase 4
```

Figure 8.2.2 Actual input lines for the Tutorial frame loads

Figure 8.2.1 shows the ASCII-file you can make from the first card in the stack, called ***Input file***. Use this card to import ASCII-files with input data or build (Export) ASCII-files.

Viewing the model

When you have finished the modelling work with the ***Input*** cards, you must be sure that all your data is correct. You can do this easily in NovaFrame by studying the frame model in the ***Frame*** window.

Choose **View...** from the **Display** menu to open the ***Frame view*** dialog box. Use this dialog box to zoom and rotate the frame figure You can move the figure sideways or up and down with the scrollbars at the edges of the ***Frame*** window.

Open the ***Attribute*** dialog box by choosing **Attributes...** from the **Display** menu. You should try out this dialog box carefully. Try to include the different attributes one by one. You should also inspect the dialog boxes you open by clicking on **Specify**. The drawing attributes included in NovaFrame give you the possibility to inspect all your geometry and load-input.

Try also to open the ***Visible elements*** dialog box by choosing **Visible elements...** from the **Display** menu. In this dialog box you can decide which elements you want to be visible on the frame model figure. Note that all elements (visible or not) are included in the solution process.

Both the ***Frame view*** and the ***Attribute*** dialog box can be minimised by clicking on the arrow in upper right corner of the dialog boxes. Use this facility if you want the dialog boxes as icons in the ***Frame*** window.

Figure 8.2.2 shows the Tutorial frame in NovaFrame with the actual loads.

Figure 8.2.2 Tutorial frame in NovaFrame

14.3 Solving the system

The next step is to solve the equation system. You do this simply by choosing **Static...** from the **Solve** menu in the ***Frame*** window. You must then click on **Start solving!** in the ***Solve static analysis*** dialog box. You get feedback on the process in this dialog box while the process is running.

14.4 Postprocessing

NovaFrame offers many possibilities for postprocessing your analysis results.

Loadcombination
The first step in your postprocessing work is to perform loadcombination. Choose **Loadcomb...** in the **Postprocess** menu in the ***Frame*** window. Now you will see a stack of ***Loadcombination*** cards. These cards work in a similar manner as the ***Input*** cards.

Choose the ***Ordinary loadcombination*** card. You should write the following combination lines to perform a simple combination of the results of the Tutorial frame.

```
*** ORDINARY COMBINATION INPUT ***
ORDCOMB   10  1  1  1   1000  2  1.000   Self weight

ORDCOMB  100  1  2  10  1.000  Self weight and crane load
ORDCOMB  100  1  1  3   1.600  Self weight and crane load

ORDCOMB  101  1  2  10  1.000  Self weight and wind load
ORDCOMB  101  1  1  4   1.600  Self weight and wind load
```

Figure 8.4.1 Loadcombination input

Figure 8.4.1 is the exported ASCII-file you produce by clicking on the **Export** button on the ***Input file*** card in the ***Loadcombination*** stack.

As you learn by examining the ***Ordinary loadcombination*** card, it is possible to combine both load cases and old combinations in a new loadcombination. However, this must be done on separate lines as the example above shows.

The limit state you choose is of no importance in this program, but will be used in a future version of the concrete design program called NovaDesign.

Click on **Apply** and the combination is performed. If you get a message stating that the program is unable to complete the combination, you should try to solve the system once more and then perform the loadcombination.

Listing of results

Choose **List results...** from the **Postprocess** menu to open the ***Listing of results*** dialog box. This is the dialog box you use to list all actual results from your frame analysis. Choose to list **Section forces** from **Loadcases** and choose all elements. Click on **View listing**.

You can now see the actual results in the editor. When you are satisfied with the results listed, you can add the results to the document or send them to a printer or copy them to an ASCII-file.

An automatic generation of section forces as ASCII-input to NovaDesign is included in the **Postprocess** menu. Choose **Customized export...** to open the ***ASCII export to NovaDesign*** dialog box. In this dialog box you can generate an ASCII-file containing all loadcombination section forces supported by NovaDesign. This approach is an alternative that is not used when the applications are sharing the database.

Plot of results

Choose **Plot results...** from the **Postprocess** menu to open the ***Results to plot*** dialog box. This is the dialog box you use to order the results you want to plot. You must then use the ***Attribute*** dialog box to customise which results to plot and how they are shown on the frame model figure.

Choose Loadcombination 100; Self weight and crane load, and click on **Load results**. You will now see the moment diagram for moments about L-axis for combination no. 100. This diagram will, however, not be as exact as the diagram shown on figure 4.5 because you have not chosen sufficient number of design sections on each element.

To revise the number of design section, choose the ***Design section*** input card and change the distance increment value to 0.1. Click on **Apply all**, solve the system and click on **Apply all** on the ***Loadcombination*** cards. If you order a plot of combination no. 100 again, your plot should look like the plot shown on figure 4.5.

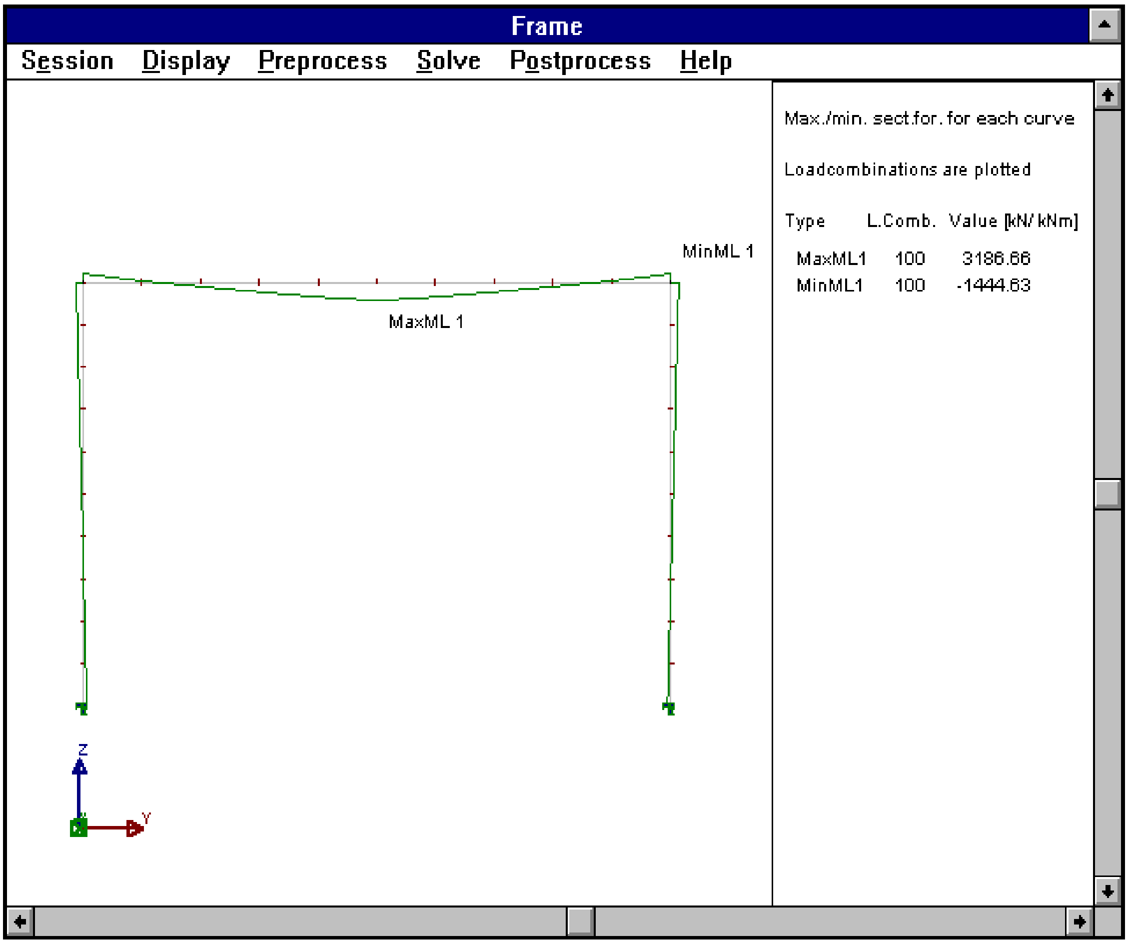


Figure 4.5 Moment diagram for loadcombination no. 100

Try out the different attribute choices you find by pressing **Specify** for **Section forces** and **Displacements** in the the ***Attribute*** dialog box.

14.5 Customise your document

Before you leave NovaFrame you should customise your document to suit your needs.

If you want to include plots of the model with or without results, you simply choose **Copy to document...** in the **Display** menu in the ***Frame*** window. You should study the help text for this dialog box to find out how the plots are treated in the document.

If you want a listing of results in your document, you must open the ***Listing of results*** dialog box, as explained in section 4.4, and copy the listing into the document.

Now, close the ***Frame*** window and choose **Document...** from the **Option** menu. In this dialog box you can customise the contents of your document. Use a couple of minutes to study the possibilities carefully. Note that you can both include and exclude listing and plots you have ordered from the ***Frame*** window or you can delete the plots completely.

You should also study the ***Document page*** dialog box. Choose **Page...** from the **Option** menu. Make yourself familiar with the possibilities in this dialog box.

It is a good idea to preview the contents of your document before you print it. You can scroll the document up and down by using the scrollbars in the main window, but the figures are only shown as white boxes. A better solution is to choose **Print preview...** from the **File** menu.

When you are satisfied with your document, choose **Print...** from the **File** menu and the document is ready to be included in your analysis report.

15 References

References to codes, regulations and literature:

/1/ - NS3472:2001, 3. ed. September 2001
Steel structures – Design rules.

/2/ - NS3473:1998, 5. ed. November 1998
Concrete structures – Design rules.

/3/ - NS3491-1, 1. ed. December 1998.
“Basis of design and actions on structures,
Part 1: Densities , self weight and imposed loads”
(replaces the NS3479)

/4/ - Norwegian Road Authority;
“Lastforskrifter for bruer og ferjekaier i de offentlige vegnett. Normal 184
Vegdirektoratet 1995”.

/5/ - “Jernbaneverket. JD-525”

/6/- NPD, Norwegian Petroleum Directorate:
“Regulations concerning loadbearing structures in the petroleum activities” - 1992

/7/- Norwegian Road Authority;
“Prosjekteringsregler for bruer. Normal 185 Vegdirektoratet Mars 1996”

/8/- AASHTO
Standard specifications for highway bridges, 15. Edition, 1992

/9/- Claës Dyrbye and Svend O. Hansen:
WIND LOADS ON STRUCTURES
John Wiley & Sons, 1999

/10/- Norwegian Road Authority;
“Veiledning i jordskjelvsanalyse for bruer” Rapport 2000-09, Vegdirektoratets
Bruavdeling, April 2000

/11/ - NS3491-4, (Preliminary).
“Basis of design and actions on structures,
Part 4: Wind loads”
(soon to replace this part in NS3479)

/12/- Standard Norge.
NS-EN 1991-2:2003 +NA:2010. “Eurokode 1: Laster på konstruksjoner. Del 2:
Trafikklast på bruer”