
User's Guide

Version 5



 **AAS-JAKOBSEN**

For design of reinforced concrete sections

License agreement

Dr. Ing. A.Aas-Jakobsen A/S will not accept responsibility for technical, editorial or other errors or omissions in NovaDesign or in this NovaDesign 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 design performed with NovaDesign. The user should always confirm the design results.

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

Table of Contents:

| | |
|---|-----------|
| 1. Introduction | 3 |
| 1.1. Program facilities..... | 3 |
| 1.2. Conventions used in this User's guide | 3 |
| 2. General | 4 |
| 2.1. Design codes..... | 4 |
| 2.2. Program file system..... | 4 |
| 2.3. Common database | 5 |
| 2.4. Help system | 6 |
| 3. Brief introduction to the user interface..... | 7 |
| 3.1. How to start the program..... | 7 |
| 3.2. Database and document..... | 7 |
| 3.3. Creating a new file..... | 8 |
| 3.4. Document window..... | 9 |
| 3.5. Customizing the document | 10 |
| 3.6. Cross section design window | 12 |
| 3.7. Design calculation window | 13 |
| 3.8. Capacity chart window | 14 |
| 4. Material data..... | 16 |
| 4.1. Concrete..... | 17 |
| 4.2. Reinforcement steel | 19 |
| 4.3. Tendon steel | 20 |
| 4.4. Assigning and modifying material grades | 21 |
| 5. Design parameter data..... | 23 |
| 6. Reference lines | 24 |
| 6.1. General | 24 |
| 7. Cross section geometry and springs | 25 |
| 7.1. General | 25 |
| 7.2. Cross section types | 26 |
| 7.3. Massive sections..... | 27 |
| 7.3.1. General massive | 27 |
| 7.3.2. Predefined massive..... | 28 |
| 7.4. Panel sections | 29 |
| 7.4.1. General panel | 29 |
| 7.4.2. Predefined panel..... | 30 |
| 7.5. Catalogue profiles..... | 31 |
| 7.6. No design sections | 32 |
| 7.7. Springs..... | 33 |
| 7.8. Reference line intersection | 34 |
| 7.9. Cross section faces | 35 |
| 8. Cross sections | 37 |
| 8.1. Cross section coordinate systems | 38 |
| 8.2. Creating a new cross section | 39 |
| 8.3. Cross section geometry..... | 40 |
| 8.4. Modify cross section | 43 |
| 8.5. Copying cross sections | 43 |
| 8.6. Delete cross sections..... | 44 |
| 8.7. Reference line intersection point | 45 |
| 8.8. Interpolation of cross section geometry | 46 |
| 8.9. Cross sections orientation relative to element axis in NovaFrame | 46 |
| 9. Reinforcement..... | 47 |

| | | |
|------------|--|-----------|
| 9.1. | Reinforcement groups | 48 |
| 9.2. | Tendon groups | 49 |
| 9.3. | Shear reinforcement..... | 50 |
| 9.4. | Reinforcement and shear reinforcement types | 53 |
| 10. | Cross section forces | 55 |
| 10.1. | Sign conventions..... | 56 |
| 10.2. | Sign conventions using section forces from NovaFrame analysis..... | 57 |
| 11. | Cross section sub areas | 59 |
| 11.1. | Introduction..... | 59 |
| 11.2. | Sub areas | 60 |
| 11.3. | Connecting shear reinforcement to sub area faces | 63 |
| 11.4. | Shear areas | 65 |
| 11.5. | Torsion areas..... | 66 |
| 12. | Design calculations | 69 |
| 12.1. | Introduction..... | 69 |
| 12.2. | Design calculation setup | 70 |
| 12.3. | Iteration method | 73 |
| 12.4. | Crack width calculations..... | 75 |
| 12.5. | Slenderness calculations | 78 |
| 12.6. | Construction tolerances..... | 80 |
| 13. | Capacity charts | 81 |
| 13.1. | Capacity chart setup..... | 81 |
| 13.2. | Modifying chart and display options | 82 |
| 13.3. | Maintenance of charts | 82 |
| 13.4. | Viewing chart results | 82 |
| 14. | Tutorial..... | 83 |
| 14.1. | Tutorial 1 – Part A | 84 |
| 14.2. | Tutorial 1 – Part B | 98 |

1. Introduction

NovaDesign is a program for design and capacity check of reinforced and prestressed concrete beam structures. The program supports the Norwegian concrete design code NS3473 4th and 5th edition and Eorocode-2 with Norwegian national annex.

With this program you can easily create and check capacity of simple beam cross sections, or you can create cross sections with complex geometry and any reinforcement and tendon configuration.

An important program facility is the connection to the frame analysis program NovaFrame through a common database. The database also has an administration system that ensures consistency between model and results. This relieves you from all manual exchange of data such as section forces and stiffness values from the frame analysis program to the design program.

1.1. Program facilities

The following main items are included in the program:

- The cross-section may have arbitrary geometry.
- Includes both tendons, ordinary reinforcement and shear reinforcement.
- Includes all beam section forces (axial force, biaxial moment, biaxial shear and torsion moment).
- Concrete and steel stresses are calculated as well as crack widths.
- Includes shear and torsion calculations according to NS 3473
- The program can automatically find the required reinforcement amounts. This also includes shear reinforcement
- LWA concrete may be defined.
- Slenderness effects may be accounted for.
- Capacity charts may be calculated, e.g. Moment-axial force and moment-curvature.

1.2. Conventions used in this User's guide

This user's guide applies some typographical conventions to make it easier to understand.

- Names of dialog boxes are written with ***Bold italic*** style
- Menu- and toolbar items are written in **Bold** style
- File names are written in *italic* style

2. General

2.1. Design codes

The program supports the Norwegian concrete design code NS3473 4th and 5th edition and Eurocode-2.

Appropriate design code is selected when creating a new project (creating a new file, see section 3.3). You can not change selected design code interactively in the user interface. If you wish to run your project using an other design code use the following procedure:

1. Export all data to an ascii file
2. Modify the code specific data according to the description in the commands manual (appendix 1) so they correspond to the selected design code.
3. Create a new project file (or use the existing file if you wish to overwrite existing data)
4. Read modified ascii file.

Note! All code specific commands are marked in the commands manual.

2.2. Program file system

Program files

The program is located in the file *novadesign.exe*.

The on-line help is located in the file *novadesign.hlp*.

An online version of this document is available in the file *ND_Users_guide_31.pdf*.

Database functions are included in a file called *quad44.dll*.

Database files

The program works on a set of database files called **.gdd*, **.gdh*, **.gdi* and **.gdn*. All input data and actual results will be saved on these files when you choose **save** during a session of NovaDesign.

The database is common for several applications. These applications are:

- NovaFrame
- NovaDesign
- NovaViewVR

Read more about the database in the next section.

ASCII- files

All model data can be imported and exported to/from an ascii file. The ascii command syntax is further described in Appendix 1 'Ascii Command Input'

You can also export all calculated results from NovaDesign to ascii-files, including listing of capacity chart results as tables. Design calculation results (*.lst)

2.3. Common database

NovaBase

NovaBase is the name of the database system, which is used by NovaDesign. 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 applications.

The program handles 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 closed for all other applications.
3. Application 1 is finished modifying contents of 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 intends 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.

2.4. Help system

NovaDesign offers an on-line help system (hlp-file), which includes most of the contents of this user's guide and some other topics as well.

You should be an active user of the help-system the first couple of times you run NovaDesign. This is an effective of learning a new program.

There are different ways of activating the help system:

1. Choose **Help - Help Topics** from the menus to get a overview of available help functions
2. Press F1 or click 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 related to the next item you click on. You also achieve this by clicking on the rightmost tool button in the main window.

A pdf-version of this user's guide and the different appendices are also available from the program menus under **Help – Users Guides...**

3. Brief introduction to the user interface

This is only a brief introduction to the user interface. You should work through the tutorial in Chapter 12 to achieve a better understanding of the program facilities. We recommend, however, that you read through this chapter before you start with the example.

3.1. How to start the program

The program executable file is *NovaDesign.exe*. If the installation of the program was successful, a shortcut to this file should be available from the Start button under *NovaProg*.

To start the program, select *NovaDesign31* under *Start – NovaProg*. You may also create a shortcut from your desktop.

If you have associated the database file extension (.gdd) to NovaDesign, you may also start the program by double-clicking an existing database file in *Explorer*.

3.2. Database and document

The first step in running NovaDesign is to create a new database file (gdd-file). The database will have a corresponding document window in the user interface. This document is similar to a document you used in e.g. Word or Excel. When you perform an analysis in NovaDesign, information about your structure is included in the document and updated according to the changes you make.

You cannot, however, edit the text 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 NovaDesign.

3.3. Creating a new file

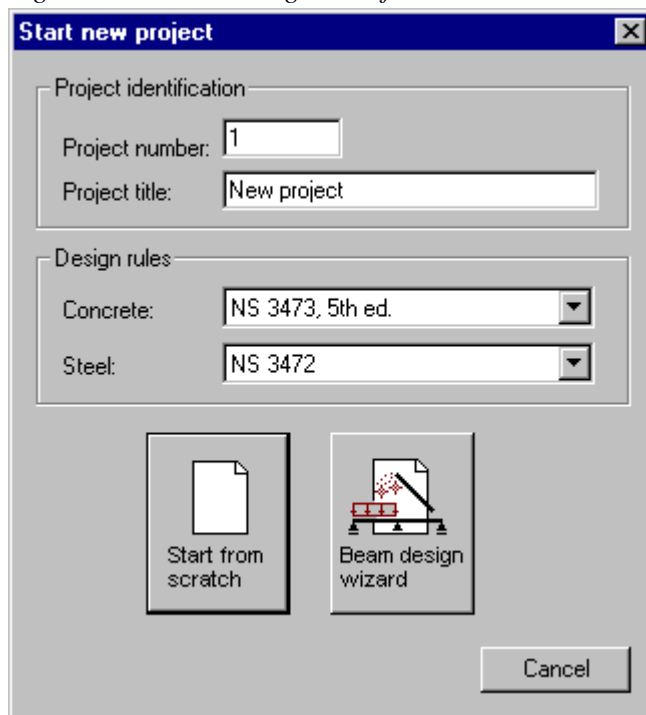
Create a new file by selecting **New** from the **File** menu. The dialog window in figure 3.1 will then be displayed. This dialog gives you the choice to start from scratch or to run the Beam Design Wizard.

Enter a project number and a project name (maximum 64 characters), or just use the default values. Project number and name can be changed later from **Project Data...** in the **Option** menu.

Select appropriate concrete design code.

Press the '*Start from scratch*' button if you wish to start with a empty file, or press the '*Beam Design Wizard*' button to run this wizard.

Fig3.1 Creating a new file



The new file will be given a default name, '*Database\$*'. You will be prompted the first time you try to save the file, and asked to give a new file name. We recommend that you do not use the default name.

3.4. Document window

When you create a new- or open an existing database. The database will appear as a document window in the main frame of the program. The name of the database file is shown at the top of the window. The document is actually the same as the database, but the document also serve as a customizable text document. The document can be used as a complete documentation of your program input and all calculated results. Cross section plots and design result listings can be added to the document through the **Copy to Document** option in the **Display** menu

Tool button:  .

From the document window you can either start working on the model data (adding material data, creating cross sections etc.) or customize the contents of the. How to customize the contents of the report is further described in section 3.6.

You can make a print-out of the report at any time. Select **Print...** from the **File** menu

Tool button:  .

You can scroll the document up and down to study the contents. Any plots (cross section plots and capacity charts) will, however, be shown only as empty frames. Drawing of the plots in the report will be included if you select **Print** or **Print Preview...** from the **File** menu.

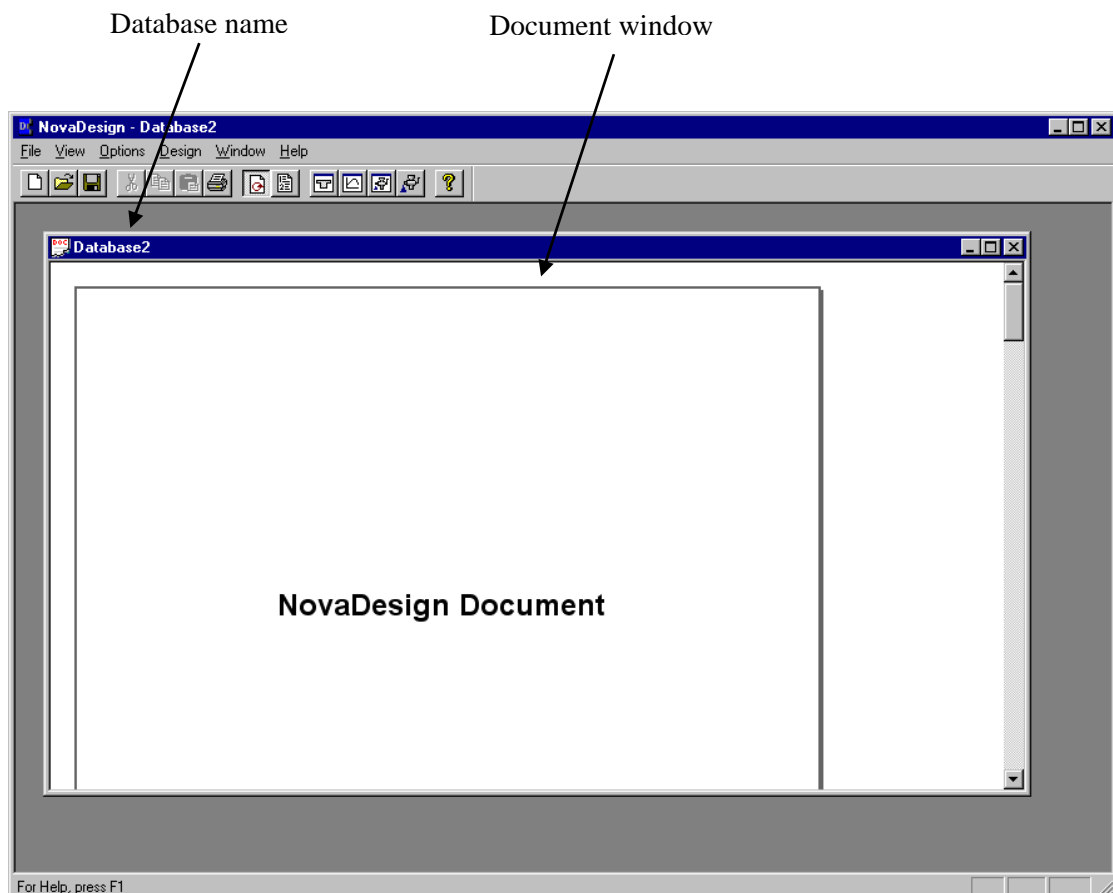


Fig. 3.2 The Document window

3.5. Customizing the document

The contents of the document can be customized in two ways:

1. Include or exclude specified parts of the document from the **Document Setup** dialog.
2. Add cross section plots or design result listings to the document by using the **Copy to document...** options from the file menu.

Document setup

The **Document Setup** dialog will appear when you select **Document Setup...** from the **Options** menu.

Tool button: 

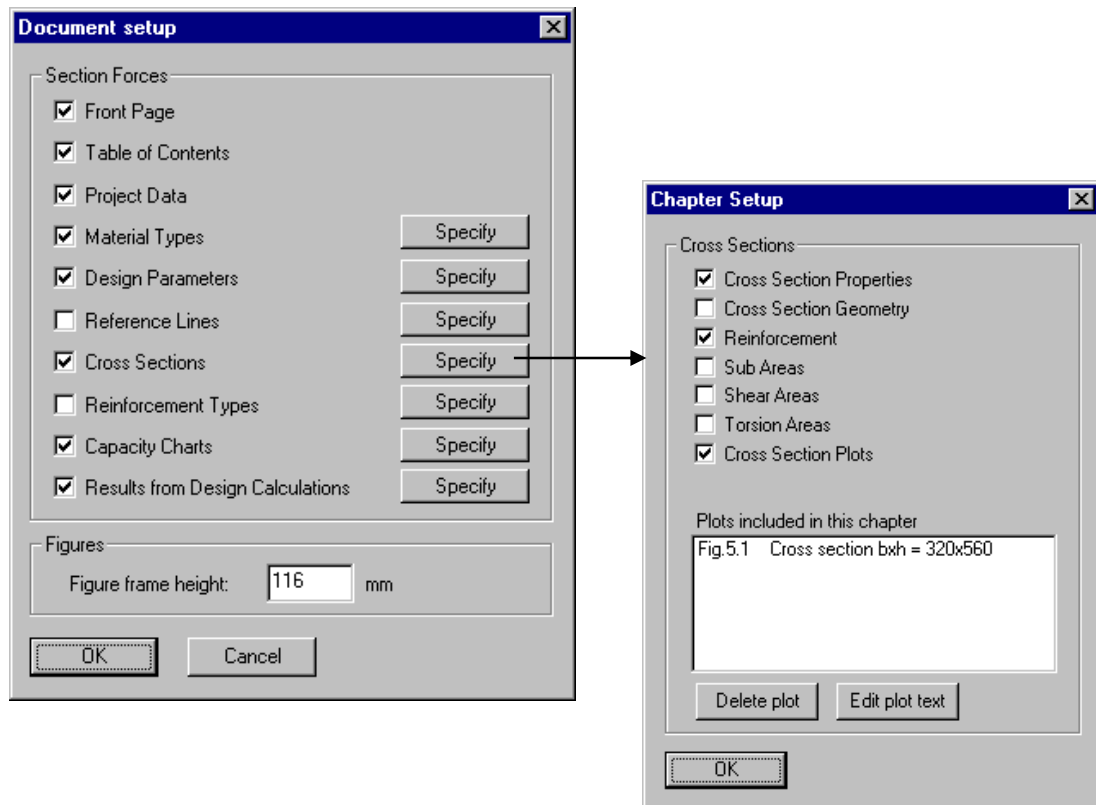


Fig. 3.3 Document Setup and Chapter Setup dialog windows

The document contents is organized in two chapter levels. The headlines of the main chapters are presented in the **Document Setup** dialog. In this dialog you may check or uncheck specific chapters in order to include or exclude these from the document.

To the right of each of the main chapter titles (except the first tree) there is a button with the text 'Specify'. If you press this button, a new dialog, **Chapter Setup**, appears which allows you to customize the subchapters of this main chapter. For those chapters that can contain plots, a list of plots added to the document is also presented in this dialog. You may delete a plot or change the

plot text from this dialog. In order to do so, you must first make a selection in the list of available plots.


Copy to document

You may add specific cross section plots or result listings from design calculations to the document. No such plots or listings are automatically included in the default document setup. In order to add the current cross section plot, or the current design result listing select **Copy to Document** from the **Display** menu.

Tool button: 

3.6. Cross section design window

The **Cross Section design** window can be opened from the **Document** window. Select **Design** and then **Cross Sections...** from the **Document** menu, or use the corresponding tool button.

Tool button: 

In the **Cross section design** window you can create and modify cross section through input dialogs. The different dialogs for creating or modifying the cross section are available from the menu at the top of the window. The plot will be automatically refreshed when you make any modifications to the cross section.

The **Cross section design** window shows a plot of the currently selected cross section. Changing the selection in the drop lists at the top of the window will change the currently selected section, see figure below.

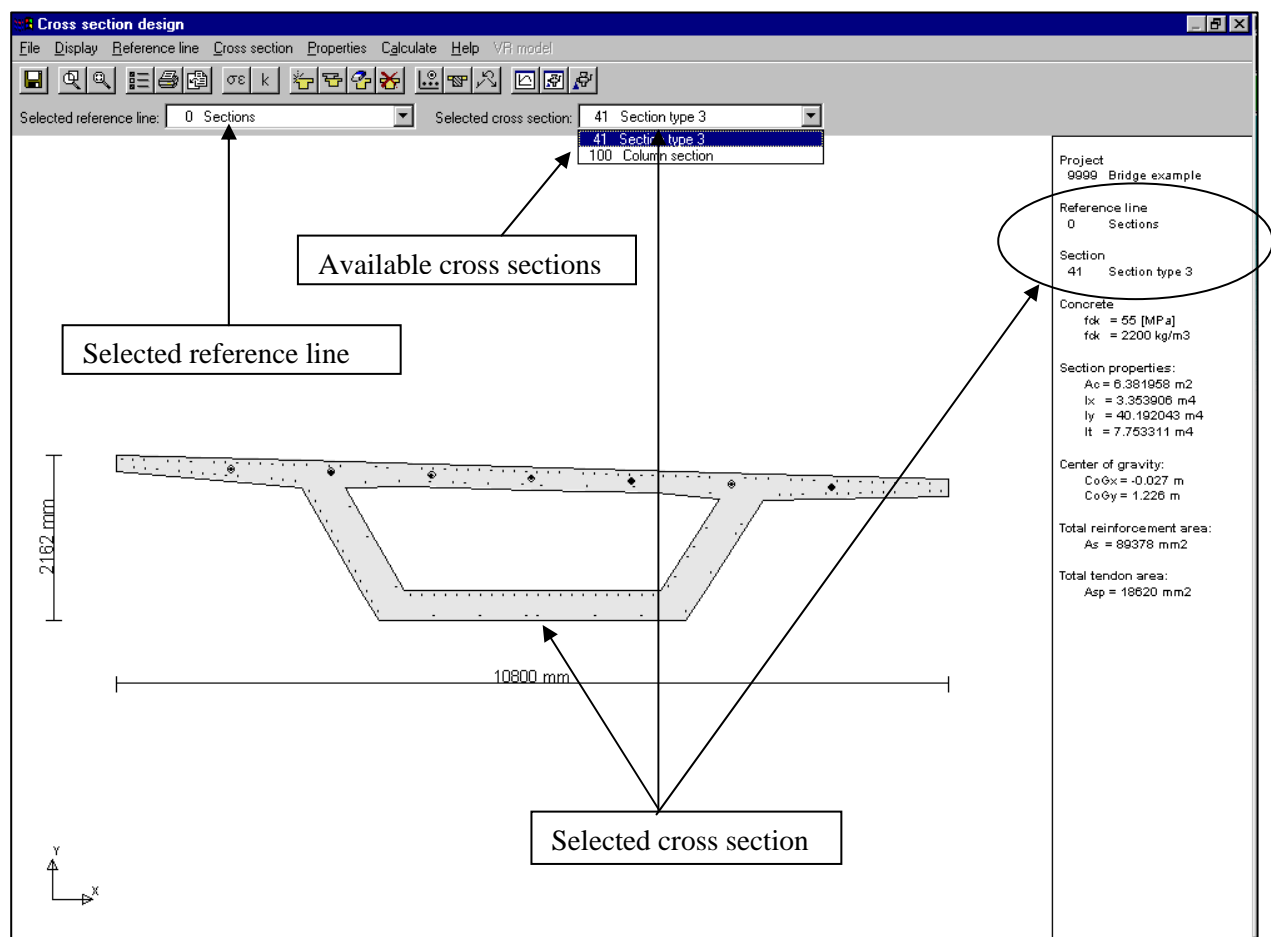


Fig. 3.5 Cross section design window

You should be aware that you can only modify the currently selected cross section in the **Cross section design** window.

The only exception are cross section forces (**Section forces...** from the **Properties** menu). They can be entered for any cross section, regardless of which cross section that is currently selected.

3.7. Design calculation window

The *Design calculation* window can be opened from both the *Cross section design* window, and the *Document* window;

from the *Document* window, select **Design** and then **Design Calculations....**, or
from the *Cross section design* window, select **Calculate** and then **Design Calculations....**

or use the corresponding tool button which is available in both windows.

Tool button: 

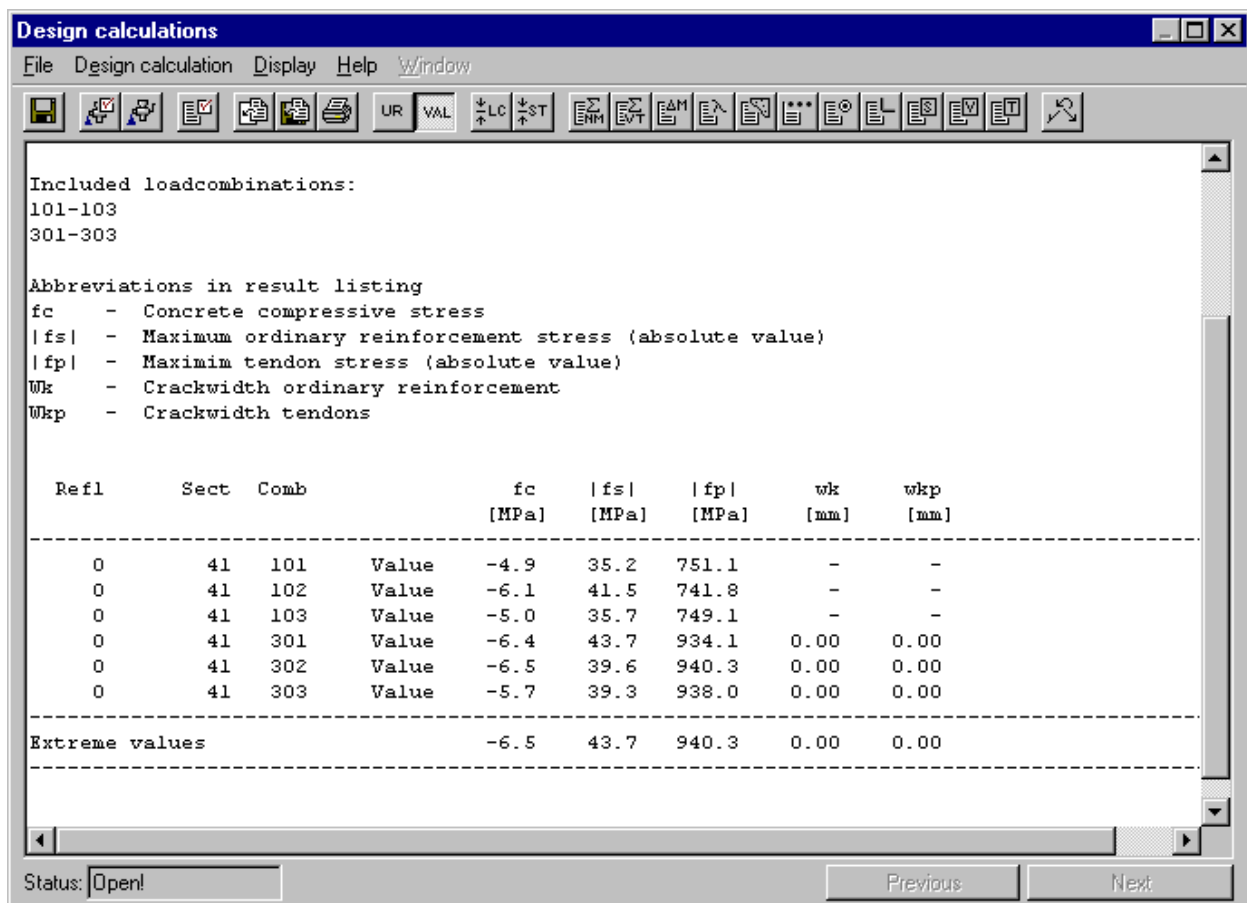







Fig. 3.6 Design calculation window

Design calculations includes calculation of strain, stress, concrete crack width etc. for the applied section forces. The results can be presented as either values , or as utilization ratios .

The procedure for calculating and viewing results is as follows (see section 11 for more details):

1. Create your own customized calculation setup, tool button , or use one of the default setups.
2. Open the calculate dialog, tool button . Then select and run one or more calculation setups.
3. Make a selection for results to list in the Design calculation window, tool button .

3.8. Capacity chart window

The **Capacity Chart** window can be opened from both the **Cross Section design** window, and the **Document** window;

from the **Document** window, select **Design** and then **Capacity charts....** , or
from the **Cross section design** window, select **Calculate** and then **Capacity charts**

or use the corresponding tool button which is available in both windows.

Tool button: 

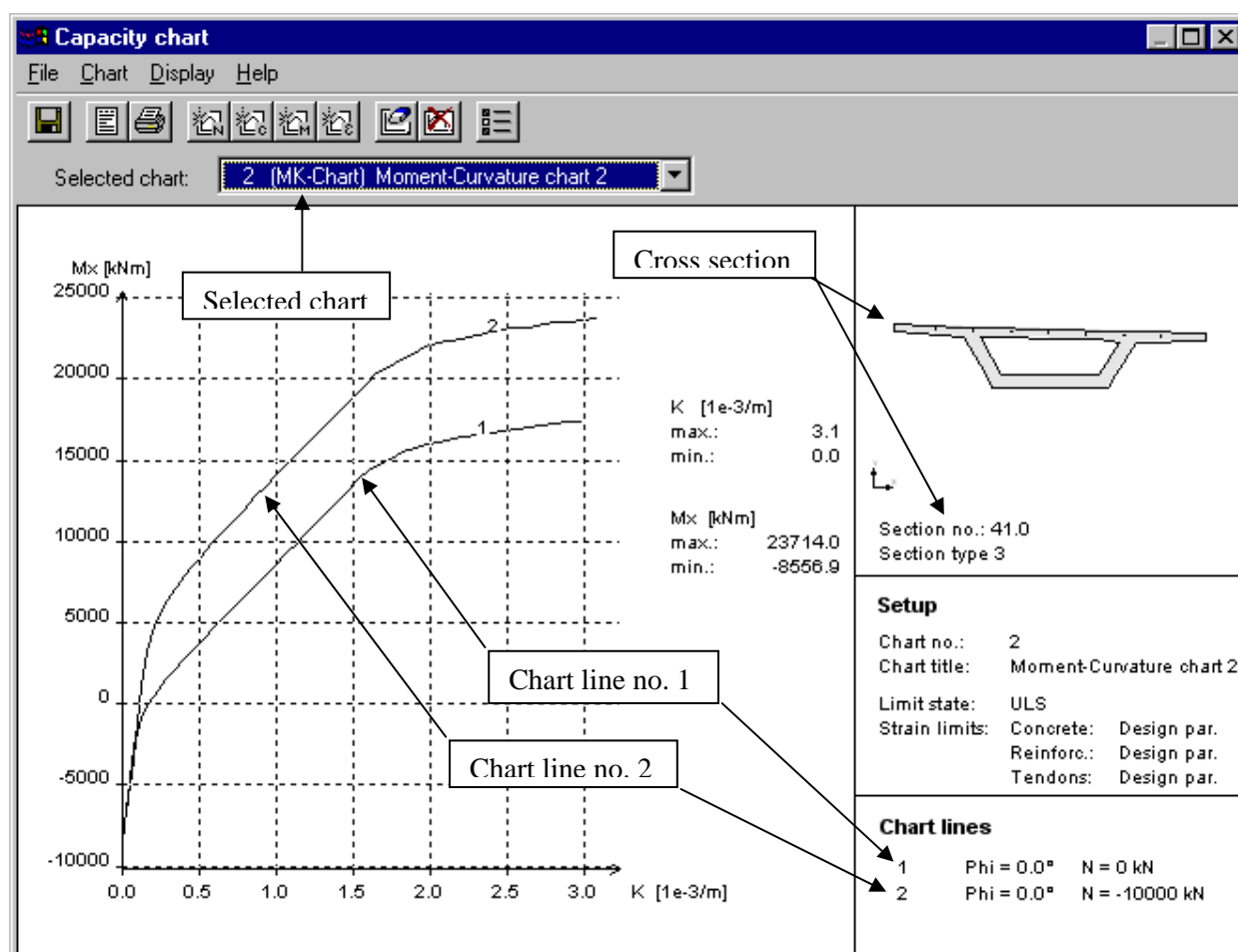


Fig. 3.7 Capacity chart window

The **Capacity Chart** window shows a graphic presentation of a specified cross sections capacity, for a given set of conditions (strain limits, limit state, etc).

You may create any number of charts. Each new chart will be added to the list of available charts. Use the drop list 'Selected chart' to jump between available charts.

An important feature of the program is that the capacity charts are always updated according to the current geometry and properties of the cross section selected for the chart. If you make any changes to the cross section, the chart will be automatically recalculated in the background.

There are four different types of capacity charts:

- MN-chart:** Gives cross section moment capacity for any axial force, within the range of the axial force capacity of the cross section. Each chart line corresponds to a specified angle for the bending axis (ϕ)
- MC-chart:** Gives cross section moment capacity for any curvature within the range of the curvature limited by the selected strain limits. Each chart line corresponds to a specified axial force level (N), and angle for the bending axis (ϕ).
- MM-chart:** Gives cross section bi-axial moment capacity. Each chart line corresponds to a specified axial force level (N).
- ME-chart:** Gives maximum cross section moment capacity for any concrete strain level within a range specified by the user. Each chart line corresponds to a specified axial force level (N), and angle for the bending axis (ϕ).

The capacity charts can be included in the document. Select **Document Setup...** from the **Option** menu in the **Document** window. Then check the '*Capacity chart*' chapter in the **Document Setup** dialog

For more details on capacity chart calculations, see chapter 11.

4. Material data

To open input dialogs for entering material data, select **Material Qualities...** from the **Properties** menu in the *Cross Section design* window, or from the **Design** menu in the *Document* window.

Tool button:



| | |
|-----------------|---------------|
| Ascii commands: | MCONCR |
| | MTEND |
| | MREINF |

Available material types are concrete, reinforcement steel, and tendon steel. You may define any number of material grades of each type. By default one material grade of each material type is created automatically when you create a new database.

Default material grades created when a new database is opened:

| | |
|----------------------|-------|
| Concrete: | C45 |
| Reinforcement steel: | B500C |
| Tendon steel: | 1670 |

4.1. Concrete

Concrete is the only valid material type for cross sections. A parabolic stress-strain relationship for the stress-strain relationship is applied according to NS 3473 section 11, as shown in figure 4.1.

The key input values for giving the concrete material properties is the compressive cube strength (f_{ck}), and the aggregate density (ρ). Based on these values the parameter values required to define the stress-strain relationship can be calculated. If you change one of the key input values (f_{ck} or ρ), in the dialog you should press the 'Calc. details!' button to update the values in the 'Detail'-frame.

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/m³, which is the default value in the input dialog.

A step by step example explaining how to create a new concrete grade from the input dialog is given in the next page.

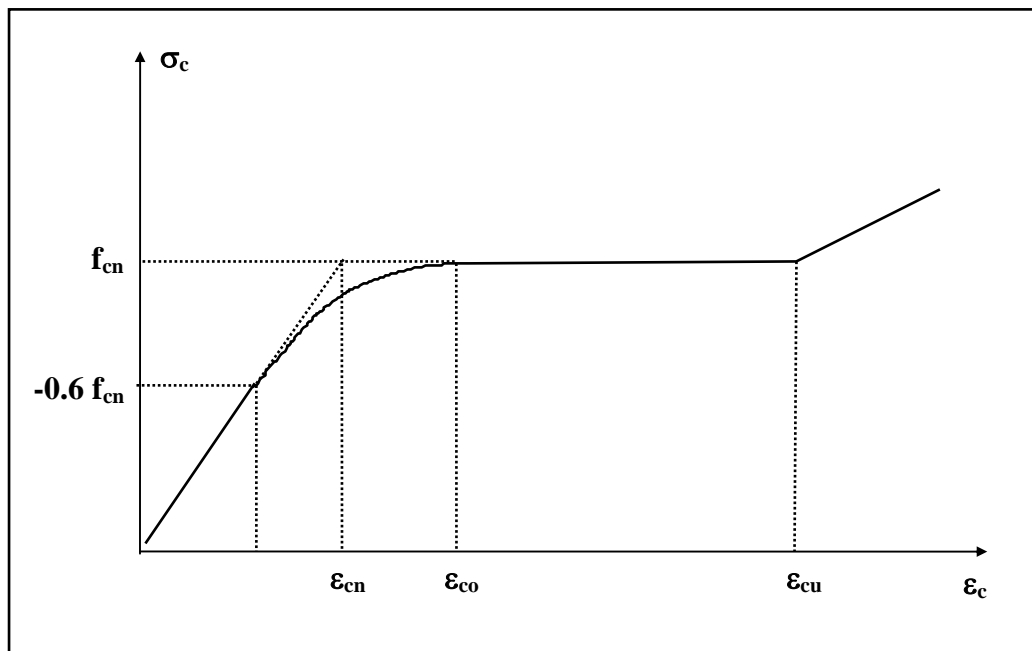


Fig.4.1 Stress-strain relationship for concrete

Note!

The stress-strain relationship continues outside ultimate strain limit (ϵ_{cu}) with a slope equal to $E_c/2$, in order to ensure convergence in design calculations.

Figure 4.2 shows the input dialog for concrete material properties. Step 1 to 5 gives an example on how to create a new concrete material, grade C55:

1. Enter ID for new material grade (or press 'Next' button)
2. Enter any name, for example. 'C55'
3. Enter value for fck, in our case '55.0'
4. Press 'Calc. details' button to calculate values in 'Detail' frame according to NS 3473.
5. Press 'Apply data'

The material grade C55 should now be available in drop list.

The screenshot shows the 'Material properties' dialog box with the 'Concrete' tab selected. The 'Available:' list shows '1 C45' and a 'Delete' button. The 'Identification' section has 'ID: 1' and 'Name: C45', with a 'Next' button. The 'Primary data' section has 'Concrete cube strength: fck: 45.00 MPa' and 'Density of aggregate: rho: 2200 kg/m3', with a 'Calc details' button. The 'Details' section has fields for fck, fcn, ftn, ftk, Ecn, Eck, Epsco, and Epscu. The 'Frame analysis properties' section has 'E: 27173.6 MPa' and 'v: 0.200'. At the bottom are 'OK', 'Cancel', 'Apply', and 'Help' buttons. Arrows point to the following elements:

- 1. 'Next' button
- 2. 'Name' text box
- 3. 'fck' text box in Primary data
- 4. 'Calc details' button
- 5. 'Apply data' button

Fig. 4.2 Concrete material properties dialog

4.2. Reinforcement steel

A bilinear stress-strain relationship is used for reinforcement steel. The input for reinforcement steel is characteristic yielding strength (f_{sk}), Young's Modulus (E_{sk}) and ultimate strain limit (ϵ_{su}). Default values are according to NS 3472.

Young's modulus outside ultimate strain limit (ϵ_{su}) is equal to $E_{sk}/2$, in order to ensure convergence in design calculations.

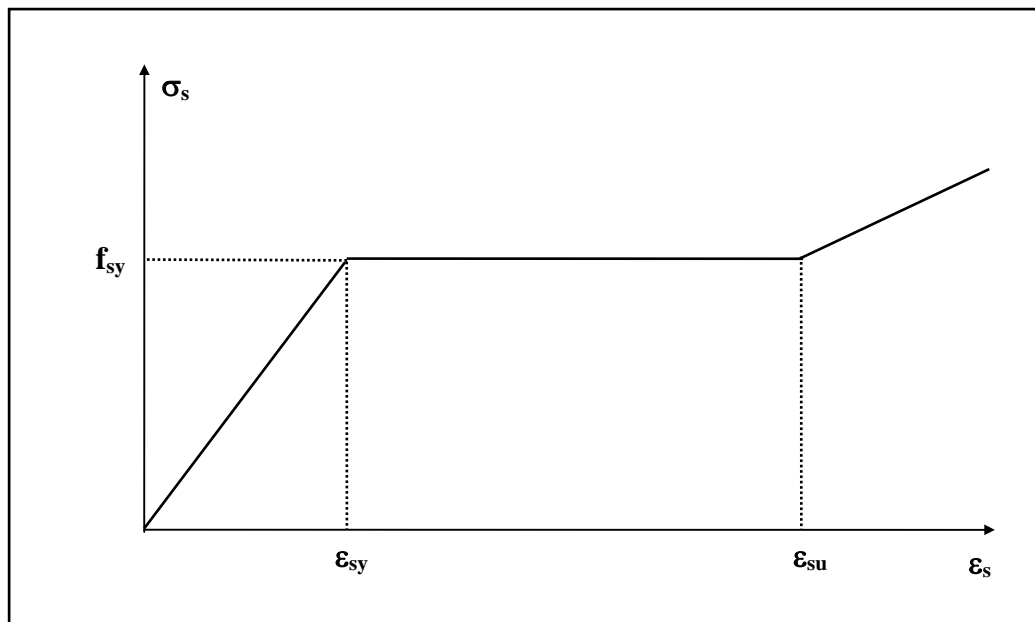


Fig. 4.3 Stress-strain relationship of reinforcement steel

4.3. Tendon steel

The stress-strain relationship for tendon steel is bilinear and similar to reinforcement steel, see figure below. The default Young's modulus is 195,000 MPa, and ultimate strain limit is set to 10.0 o/oo

The prestressing effect is included by specifying an initial strain. This is however not a material property, and is therefore entered for each tendon group..

Young's modulus outside ultimate strain limit (ϵ_{su}) is equal to $E_{sk}/2$, in order to ensure convergence in design calculations.

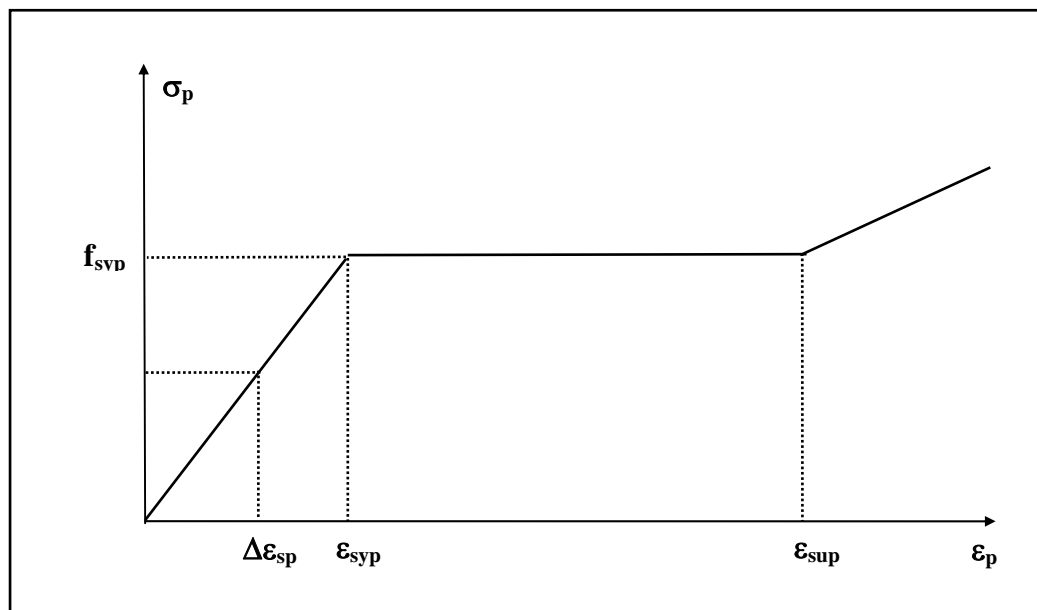


Fig.4.4 Stress-strain relationship for tendon steel

The stress-strain relationship continues with a slope equal to $E_{sk}/2$ for strain values larger than ϵ_{sup} , in order to ensure convergence in design calculations.

4.4. Assigning and modifying material grades

When you create a new cross section, a new reinforcement- or tendon group, you must select which material grade to use.

You select a material grade by giving its identification number (ID). In the input dialogs the available material grades are presented in a drop list. The figure below shows an example.

By default only one material grade of each type is available. If you need other material grades you must either create a new material grade or modify the existing.

The image shows a software dialog box titled "New cross section". It has three tabs: "Rectangular", "Circular", and "General". The "Rectangular" tab is selected. Inside the dialog, there are several sections:

- Identification:** A "Section" field with the value "1" and a "Name" field with the value "Rectangular section".
- Outer dimensions:** "Width" and "Height" fields, both set to "300 mm".
- Thicknesses:** "Left web", "Right web", "Upper flange", and "Lower flange" fields, all set to "0 mm".
- Cross section connection to reference line:** Four radio button options: "In center of gravity.", "Offset to center of gravity.", "In specified section point.", and "Offset to specified section point.".
- Parameter sets:** Three dropdown menus: "Concrete material:" (set to "1 C45"), "Concrete design:" (set to "1 Default values"), and "Section design:" (set to "1 Default values").
- Options:** A checkbox labeled "Create default sub-, shear- and torsion areas" which is checked.

At the bottom are "OK", "Cancel", "Apply", and "Help" buttons. A callout box with an arrow points to the "Concrete material:" dropdown, containing the text "Select material grade in drop-list".

Fig. 4.5 Specifying cross section material grade

Normally, many cross section will use the same material grade, as shown in the figure below. Because the cross sections recognize the material grade only by its ID, it is very easy to change material properties for all these cross sections. Simply modify the properties for the concrete, and all cross sections using this material will be affected.

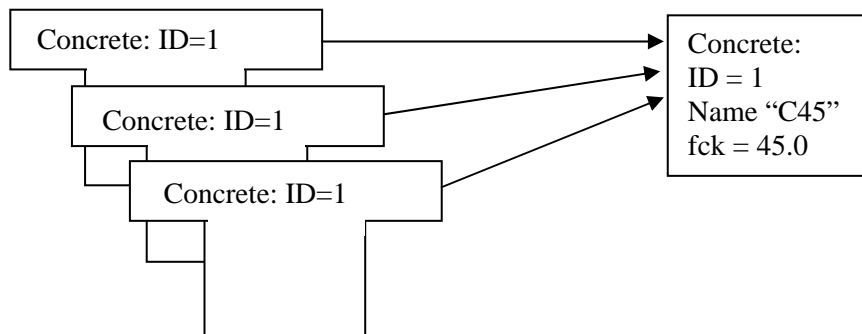


Fig. 4.6 Cross sections using the same material grade

If you wish to change the material properties for only a selection of the cross sections in the example above, you will have to create a new material with a different ID.

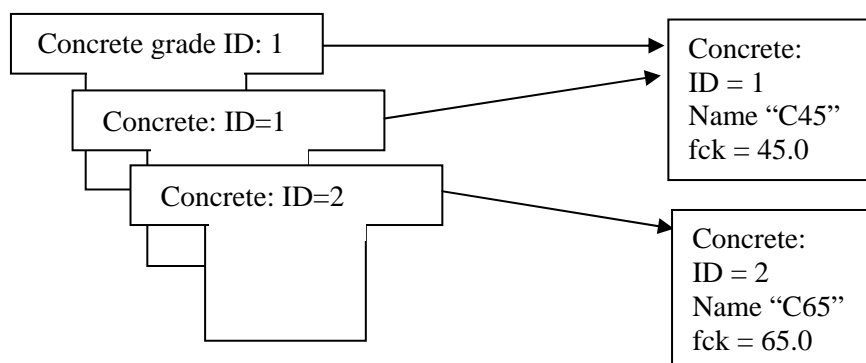


Fig. 4.7 Cross sections using different material grades

The cross section material is used in these examples, but the principles are also valid for reinforcement and tendon materials.

Also note:

- It is OK to create two identical material grades with different identification (ID)
- The name is only for convenience and does not have to reflect the actual material grade. For instance, giving a material the name; 'Project default', is OK.
- Default material ID is always 1.
- Cross sections using a non-existing material will have no stiffness.

5. Design parameter data

To open input dialogs for entering design parameter data, select **Design Parameters...** from the **Properties** menu in the *Cross Section design* window, or from the **Design** menu in the *Document* window.

Tool button:



| | |
|-----------------|--|
| Ascii commands: | DPSECT DPCONCR DPTEND DPREINF |
|-----------------|--|

There are four different types of design parameter sets:

Section design parameters:

Includes overall cross section parameters such as theoretical buckling length and total creep factor.

Concrete design parameters

Includes concrete material safety factors and strain limits.

Reinforcement design parameters

Includes reinforcement steel material safety factors, strain limits and other reinforcement specific design parameters.

Tendon design parameters

Includes tendon steel material safety factors, strain limits, and other reinforcement specific design parameters.

By default one design parameter set of each type is created automatically when you create a new database.

6. Reference lines

6.1. General

| | |
|-----------------|---------------------------------|
| Ascii commands: | REFLINE (and more...) |
|-----------------|---------------------------------|

The reference lines is 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).

See Appendix 2 '*Using Reference Lines*' for a more thorough introduction of the concept of reference lines and how they are used in the interaction between NovaDesign and NovaFrame.

You may add, modify or delete reference lines by selecting **New...**, **Modify...** or **Delete** from the **Reference line** menu in the *Cross section design* window. However, it is recommended that you use NovaFrame to work on the reference line geometry, because it offers both input dialogs for geometry data and plot of reference line geometry.

7. Cross section geometry and springs

7.1. General

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

Tool button: 

Each cross section or spring 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 on 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 3; *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.

7.2. 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 divided 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; NO DESIGN.

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 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 | |
| NO DESIGN | - | X | | |
| SPRING | - | X | | |

Table 7.1 Available cross section types

7.3. Massive sections

7.3.1. 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 continuous line. No additional area input is therefore required as for the panel sections (except shear 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)

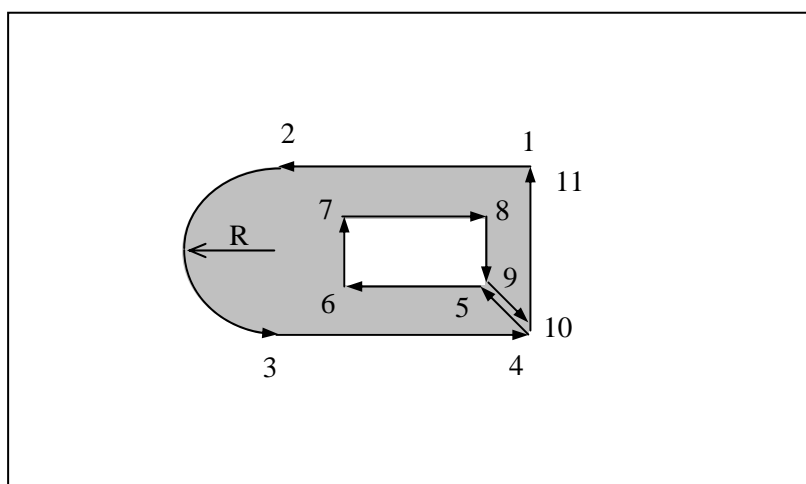


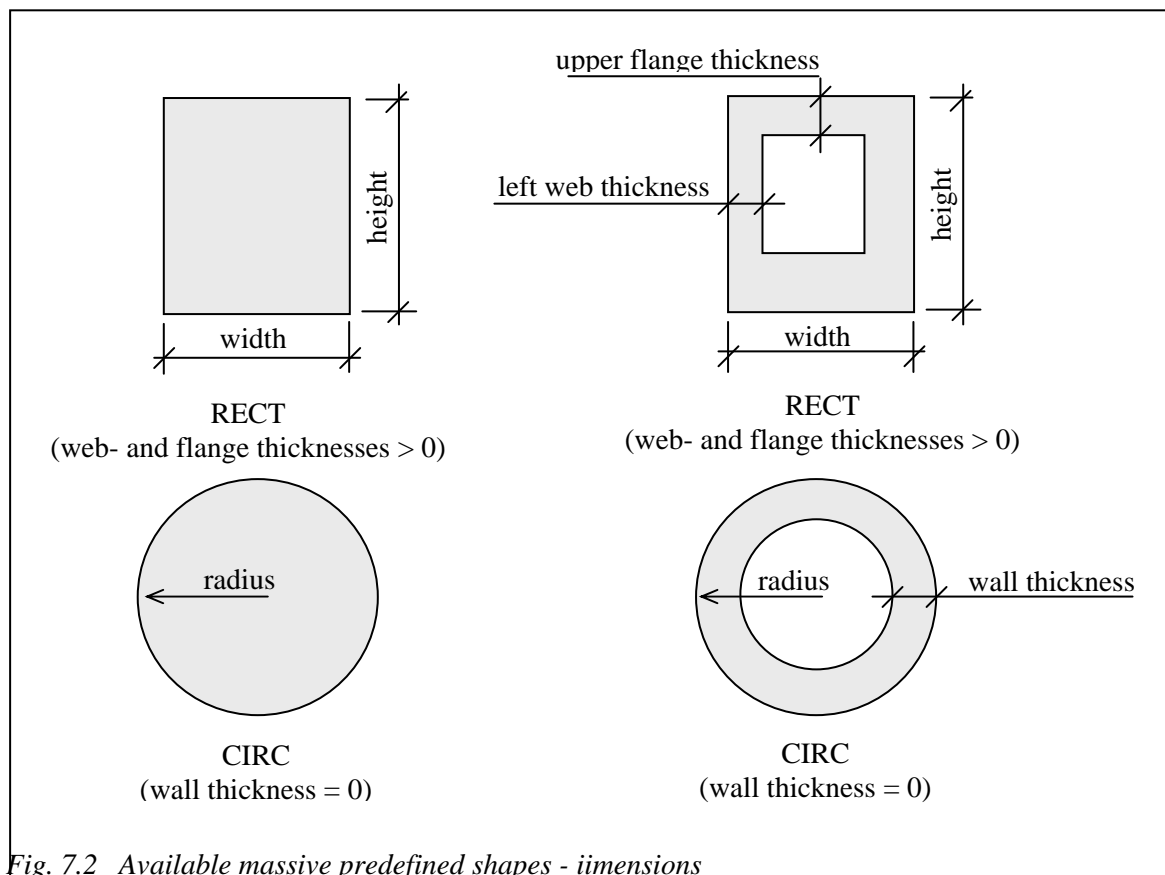
Fig. 7.1 Massive general cross section – section point input direction

7.3.2. 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 measures are given in [mm]. The figure below shows the available predefined panel section types



7.4. Panel sections

7.4.1. 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.

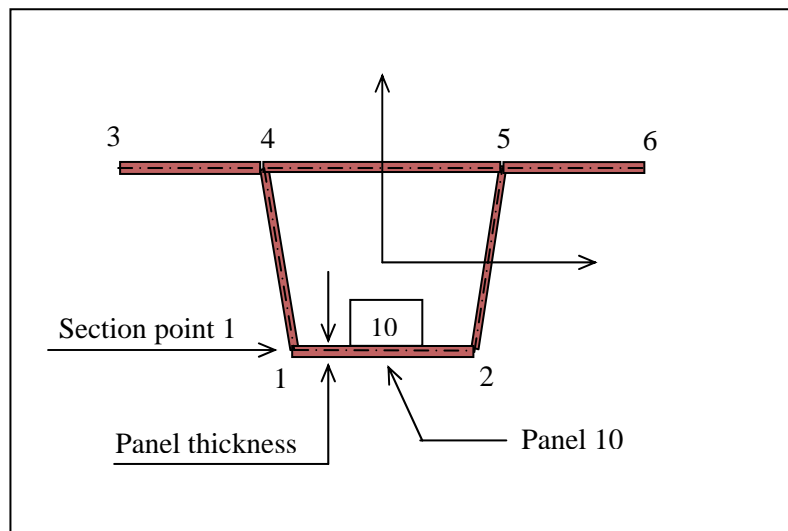


Fig. 7.3 General panel section

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.

7.4.2. 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 measures (not plate thicknesses) are between plate center lines, as shown for 'height' in the figure below.

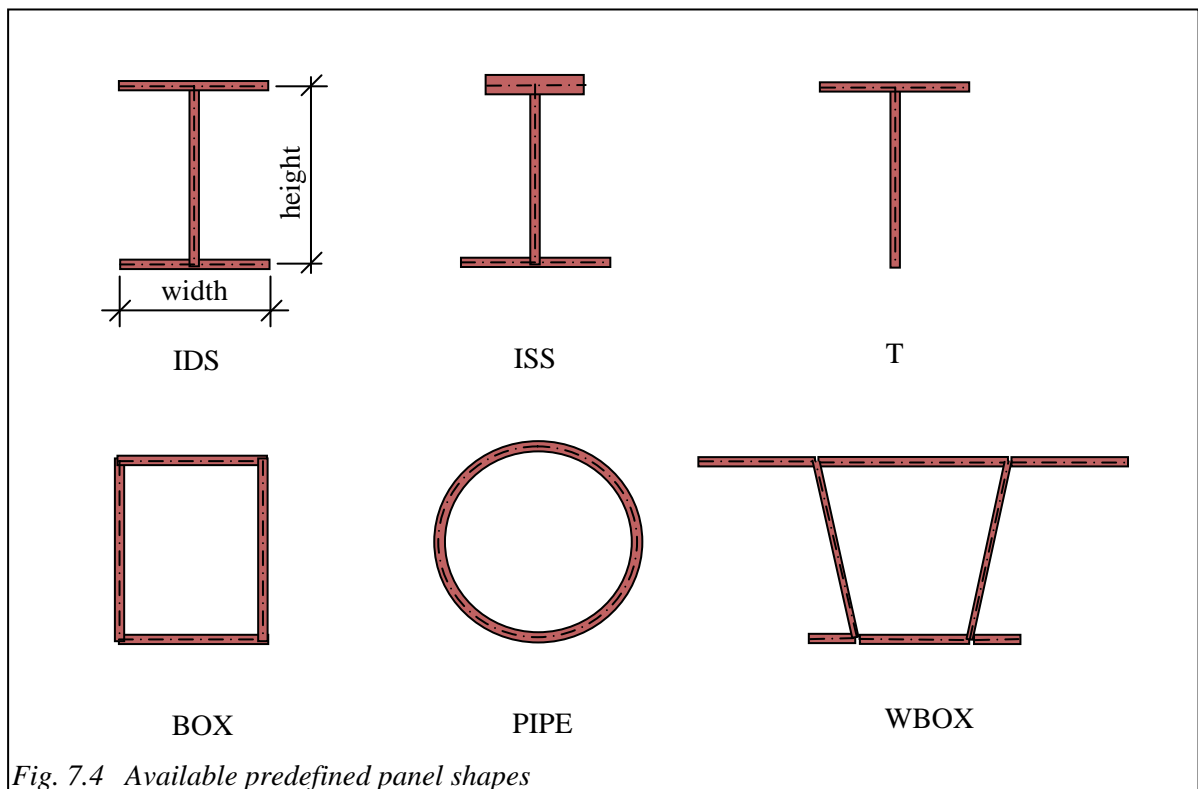


Fig. 7.4 Available predefined panel shapes

7.5. 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, "" ! creates a new HEA200 profile
```

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

| | |
|---------|------------------------------------|
| IPE | I-profile |
| HEA | I-profile |
| HEB | I-profile |
| HEC | I-profile |
| HEM | |
| UNP | |
| UPE | |
| PFC | |
| PFC | |
| UAP | |
| T_PROF | |
| RHS_V_K | Rectangular hollow section, square |
| RHS_V_R | Rectangular hollow section |
| RHS_K_K | |
| RHS_K_R | |
| TUBE | |
| TUBE_E | |
| IPE | |

7.6. No design sections

In most cases a frame analysis would be followed by a design check of the cross sections used in the frame model. For those part of the frame model (or whole) 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 | Curcular (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 (except 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'.

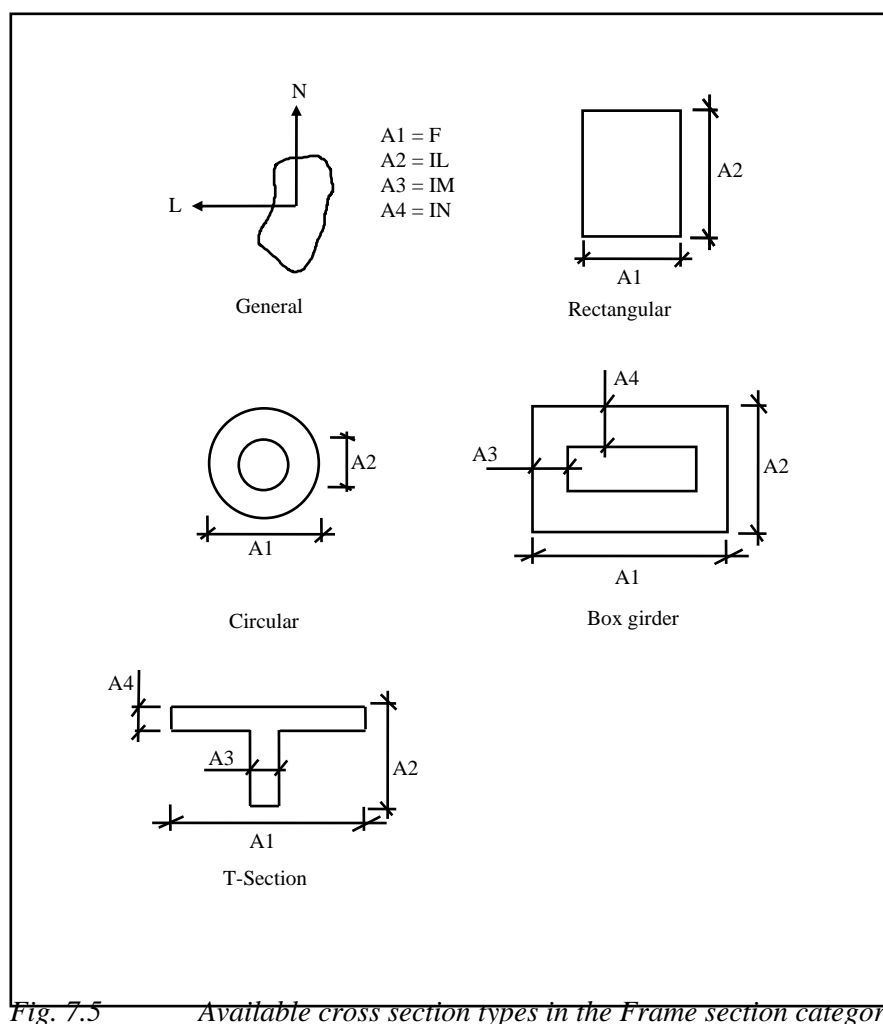


Fig. 7.5 Available cross section types in the Frame section category

7.7. Springs

In the NovaFrame analysis model a spring must be represented as a beam element given a cross section with typical spring stiffness properties. Two cross section types are available for this purpose, in the cross section category 'Springs'. Cross section type specified in the XSECT command should be one of the following:

- MDEFSPR Translational spring
- MROTSPR Rotational spring

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.

Note, a spring has the same local axis system as an ordinary element, i.e. spring stiffnesses are in element LMN-directions.

If you need to specify both translational- and rotational-spring stiffnesses at a node in the frame model, you must create two separate spring elements connected between the actual node and the 'ground' node.

7.8. Reference line intersection

This section is only valid for reference lines other than reference line 0!

The cross section is always oriented perpendicular to the reference line. By default the program assumes that the intersection point for the reference line is the section center of gravity (COG). This is convenient in many cases. For example; a cross section on a column reference line would normally be positioned with its COG at the column axis.

In some situations however, it is advantageous not to position the cross section COG on the reference line. For example; if the reference line were the road centerline, the intersection point would normally be located just above the upper face of the bridge cross section. The program therefore allows you to 'move' the cross section relative to the reference line. The figure below shows the program options.

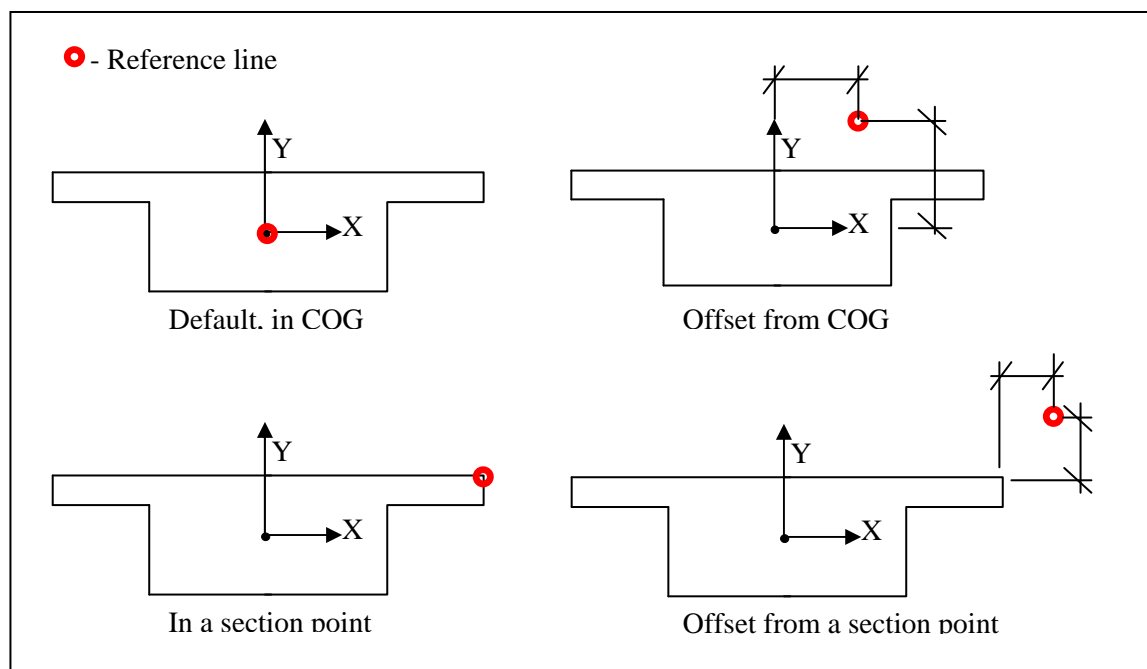


Fig. 7.6 Positioning cross section COG, available options

Please note that correct signs on the ordinates are found when positioning the section COG or section point relative the reference line position (= "local origo").

7.9. Cross section faces

The cross section surface consists of a number of section points. The command SECTFACE makes it possible to associate these section points to upper or lower face of the cross section. This face association can be utilized in elastic stress calculation and for vertical positioning of tendon groups:

- Elastic stress: Output is stresses at upper face and lower face
- Tendon groups: Vertical geometry is given relative to upper and lower face

UPPER and LOWER can be used as alias for face association code in the SECTFACE command

Sample input related to the figure below:

```
SECTFACE, , 91, 1, 2, 1, LOWER
SECTFACE, , 91, 5, 8, 1, UPPER
```

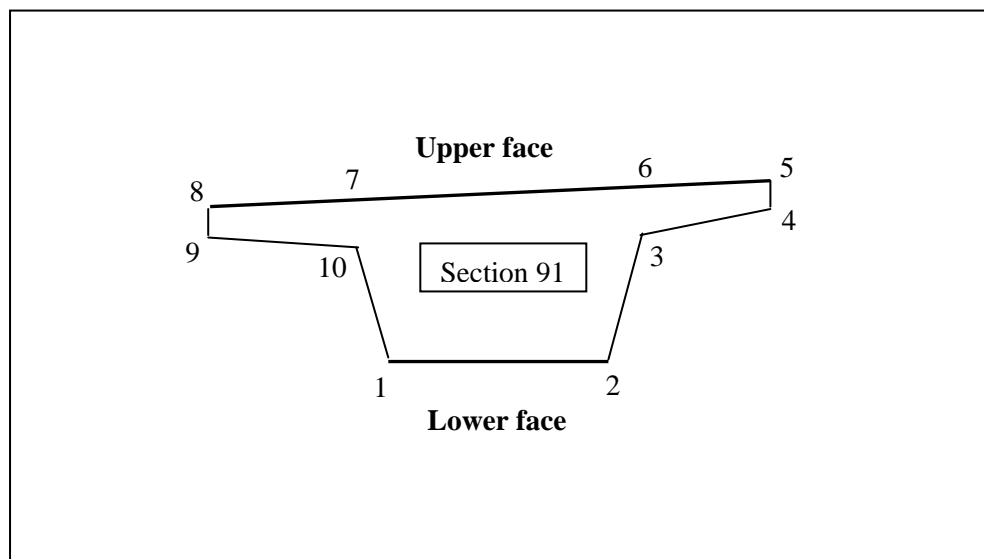


Fig. 7.7 Face association

8. Cross sections

A cross section is always connected to a reference line. Each cross section is identified with a station number. For reference lines other than reference line 0, this station number is also the position along the reference line where the cross section is located. For reference line 0 however this station number is only a number used to identify the cross section (cross section number).

You can add, modify, copy or delete cross sections for all available reference lines by selecting **New...**, **Copy...**, **Modify...** or **Delete** from the **Cross Section** menu in the *Cross section design* window.

When creating an new cross section you must start with the cross section surface geometry. NovaDesign offers two predefined shapes and general (free shape) geometry.

8.1. Cross section coordinate systems

When entering coordinates for cross section points the coordinate values always refer to a in-plane xy-coordinate system. This is the locale cross section coordinate system. You are in free to choose the location of the origin. The x-axis is always shown as horizontal (positive direction to the right), and y-axis vertical (positive direction upwards).

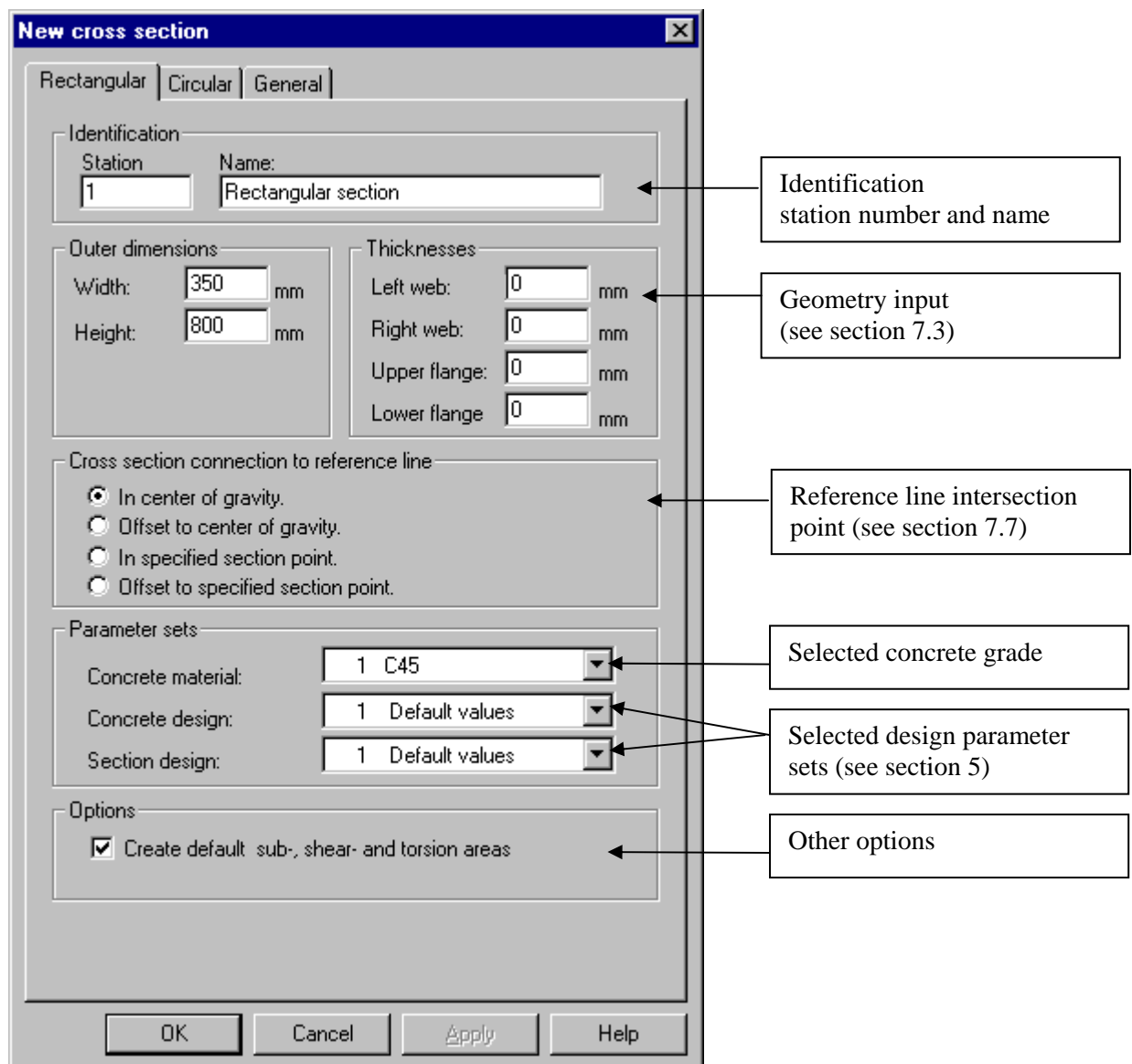
8.2. Creating a new cross section

To create a new cross section for the currently active reference line, select **New...** from the **Cross Section** menu.

Tool button: 

Ascii command: **SECT**

The input dialog for creating a new cross section has three tabs. One for each of the two predefined shapes- (rectangular, circular), and one for free cross section shape (general). Click the correct tab for your cross section, and enter the required data.



The screenshot shows the 'New cross section' dialog box with the 'Rectangular' tab selected. The dialog is divided into several sections with corresponding annotations:

- Identification:** Contains 'Station' (1) and 'Name' (Rectangular section). Annotation: Identification station number and name.
- Outer dimensions:** Contains 'Width' (350 mm) and 'Height' (800 mm).
- Thicknesses:** Contains 'Left web' (0 mm), 'Right web' (0 mm), 'Upper flange' (0 mm), and 'Lower flange' (0 mm). Annotation: Geometry input (see section 7.3).
- Cross section connection to reference line:** Contains four radio button options: 'In center of gravity.' (selected), 'Offset to center of gravity.', 'In specified section point.', and 'Offset to specified section point.'. Annotation: Reference line intersection point (see section 7.7).
- Parameter sets:** Contains three dropdown menus: 'Concrete material' (1 C45), 'Concrete design' (1 Default values), and 'Section design' (1 Default values). Annotations: Selected concrete grade, Selected design parameter sets (see section 5).
- Options:** Contains a checked checkbox 'Create default sub-, shear- and torsion areas'. Annotation: Other options.

At the bottom of the dialog are buttons for 'OK', 'Cancel', 'Apply', and 'Help'.

Figure 7.1 New cross section dialog

8.3. Cross section geometry

| | |
|-----------------|-------------------------|
| Ascii commands: | PT DIM |
|-----------------|-------------------------|

The section surface geometry must be entered for each new cross section. If you wish to modify the geometry for an existing cross section, select **Modify...** from **the Cross section** menu.

The program offers three different geometry types; two predefined shapes and one general (free shape) geometry.

For free shape geometry, you must enter all points on the cross section surface necessary to describe the geometry. For the two predefined shapes you only need to enter key geometric values such as; width, height, etc.

You are free to choose the location of the origin of the section coordinate system in the cross section plane. Selecting location of origin is just a matter of convenience. For the predefined shapes, the origin is automatically placed in center of section.

Rectangular

Measures are explained in the figure below. Enter all web and flange thicknesses to create a box section, or enter 0 (default) to create a massive rectangular section.

Note!

For rectangular sections, you may let the program automatically create sub, shear and torsion area by selecting this option under 'Options' in the dialog window.

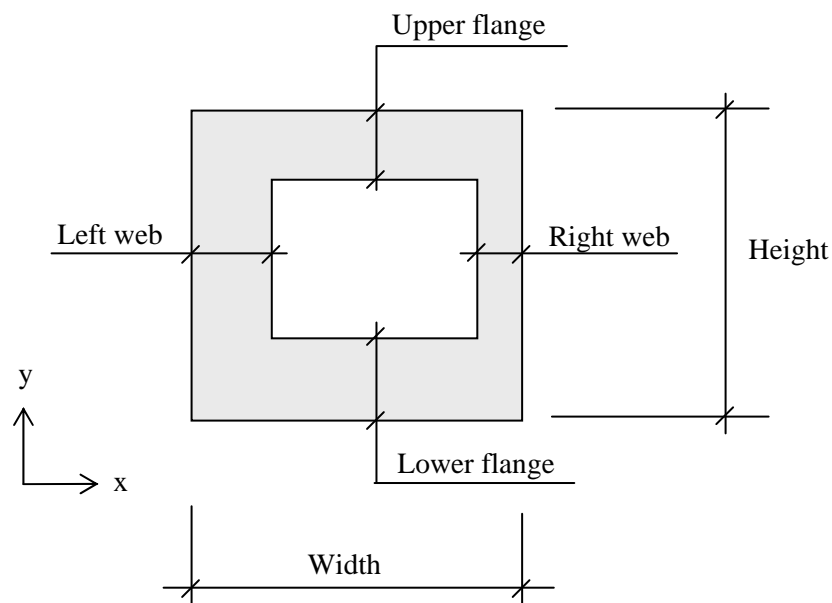


Fig. 7.2 Predefined shape - Rectangular

Circular

Measures are explained in the figure below. Enter wall thickness to create a tubular section, or enter 0 (default) to create a massive circular section.

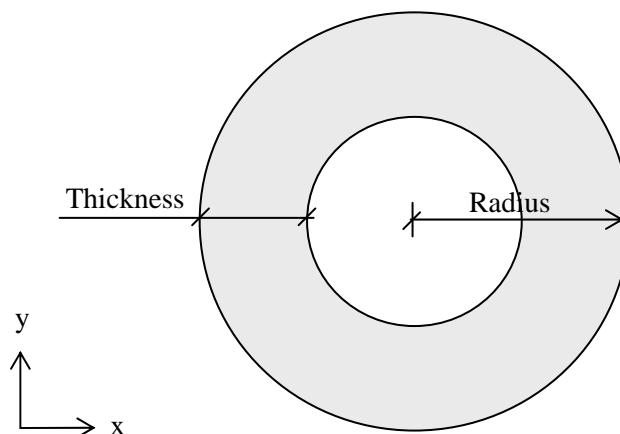


Fig. 7.3 Predefined shape - Circular

General

The free shape section geometry is described by a series of section points interconnected by a continuous line. 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

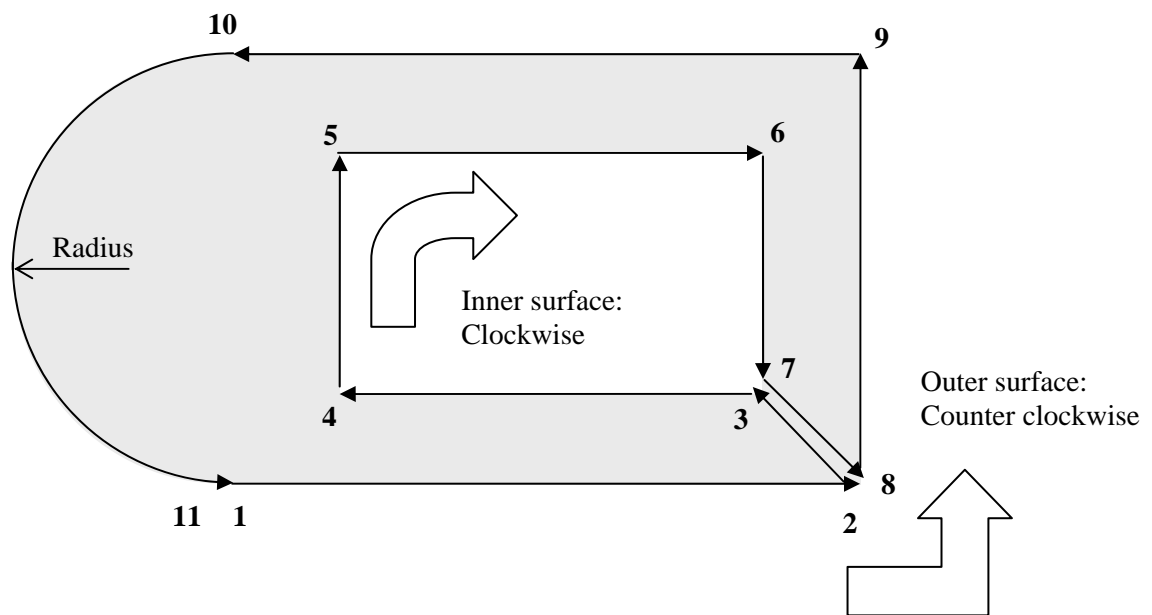


Fig. 7.4 Free section shape - General

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 specified in a counterclockwise direction.
- A hole 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 2 and 8, and section point 3 and 7 in fig 7.4)

8.4. Modify cross section

The cross section you wish to modify must be selected as the active cross section in the **Cross section design** window.

In order to modify the identification, geometry, connected parameter sets or reference line intersection point, select **Modify...** from the **Cross Section** menu.

Tool button: 

In order to modify reinforcement, tendons or sub areas for a cross section, open the appropriate input dialog for these items from the **Properties** menu.

8.5. Copying cross sections

The cross section you wish to make a copy of must be selected as the active cross section in the **Cross section design** window.

The program offers the possibility to make copies of an existing cross section by selecting **Copy...** from the **Cross Section** menu in the **Cross section design** window.

Tool button: 

The following cross section data will be copied:

- Geometry
- Reinforcement
- Connected parameter sets
- Sub, shear and torsion areas

This can be useful if you need to create many sections with the same or similar geometry, reinforcement etc. An existing cross section can be copied to any available reference line, and not only within the same reference line.

You are not allowed to assign the new cross section the same station number or section number as an existing section.

Note!

A copy will not be updated due to changes in the original the copy was created from. Therefore you should complete as much as possible of the input data for the original cross section before you make a copy.

8.6. Delete cross sections

The cross section you wish to delete must be selected as the active cross section in the *Cross section design* window. then select **Delete...** from the **Cross Section** menu.

Tool button:



8.7. Reference line intersection point

This section is only valid if you use a reference line other than reference line 0!

The reference line is always oriented perpendicular to the section plane. By default the program assumes that the intersection point for the reference line is the section center of gravity (CoG). But you may select a different intersection point. The options are shown in Fig. 2.

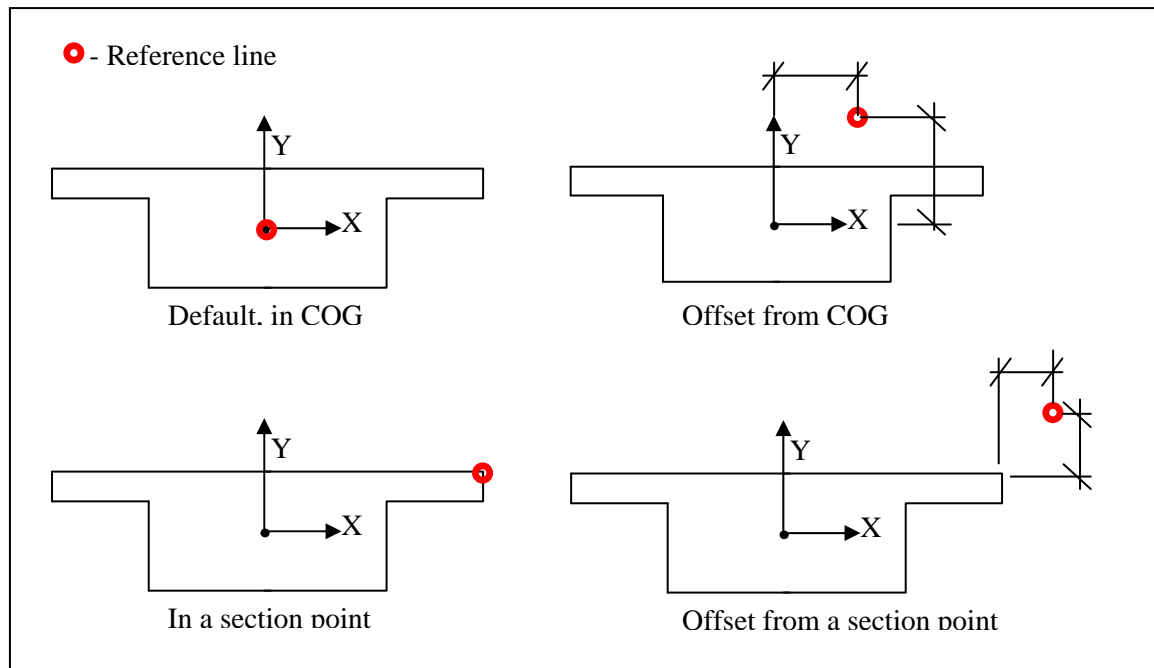


Fig. 7.5 Intersection point for reference line

8.8. Interpolation of cross section geometry

This facility can be useful if you are using reference line in combination with a frame analysis in NovaFrame.

The concept of reference lines and the interaction between NovaDesign and NovaFrame, is described more in detail in appendix 2 *'Using Reference Lines'*

8.9. Cross sections orientation relative to element axis in NovaFrame

If you use cross sections created in NovaDesign to generate stiffness properties in a NovaFrame analysis, you should be aware of how the cross section is oriented when connected to the frame elements.

Cross sections connected to reference line = 0

These are always oriented such that the cross section;

- x-axis points in the direction of local element L-axis
- y-axis points in the direction of local element N-axis

Cross sections connected to other reference lines

If the nodes of the elements in NovaFrame are connected to a reference line, the cross sections are always oriented such that the section as you see it in NovaDesign is as you see it when looking in positive reference line direction in the frame model. The sections are always oriented such that the cross section;

- y-axis points in the direction of local element N-axis

9. Reinforcement

You define reinforcement amount and position the **Reinforcement** input dialog. The dialog is only available in NovaDesign.

Tool button: 

Tendon amount, position and properties may also be taken from the tendon input in the frame model. This requires that you have defined the cross section geometry in the frame model by reference line (node location is generated based on reference line geometry and no override of cross section is made).

Reinforcement input is valid for massive concrete cross sections (see section 7.3 *Massive sections* for an overview of massive section types) There are three types of reinforcement available;

- Ordinary (longitudinal) reinforcement
- Shear reinforcement
- Tendons

Input card for entering reinforcement data are found by selecting **Reinforcement & Tendons...** from the **Properties** menu in the **Cross section design** window,.

Tool button:

The dialog box displayed in this selection contains tabs covering all possibilities for entering both ordinary reinforcement and tendons for the active cross section.

9.1. Reinforcement groups

| | |
|----------------|--------------------------|
| Ascii command: | RE REAS |
|----------------|--------------------------|

You can enter any number of reinforcement groups into each cross section

A reinforcement group consists of one or more reinforcement bundles placed in the cross section based on one single description of how many and where these bundles should be placed.

Note!

The notation 'bundle' is used as a general term throughout the program, even if the bundle only contains one unit and calling it a 'bar' would be more exact.

Location

Location of reinforcement group is defined by selecting which section points the group should be connected to, and values for cover and lateral cover relative to these section points.

For predefined massive section types (rectangular and circular) reinforcement location is not referred to section points, but to 'faces', such as 'left face' or 'inner face'.

Amount

You define the amount by selecting a reinforcement type to associate with the relevant reinforcement group. The definition of a reinforcement type is explained in a separate section in this user's guide. By default no reinforcement amount is entered when you define the first reinforcement group for a cross section. You must therefore modify the list of available reinforcement types. Each reinforcement group can be assigned many reinforcement types, but only one of these are selected (active) at any time.

9.2. Tendon groups

| |
|---|
| Ascii command: TE TEAS |
|---|

You can enter any number of tendon cable groups into each cross section.

Location

Location of tendon group can be entered in two ways. Either by specifying reference section points as described for reinforcement groups, or 'by coordinates'. If you select to give location 'by coordinates' you must enter X- and Y-coordinates for start and end point.

Amount

There is no reinforcement type connected to tendon groups, as is the case for reinforcement groups and shear reinforcement. You only enter one definition of reinforcement amount in one of the following ways;

1. Give number of cables and area of each cable (n)
2. Give total area of all cables in group (As)

If you choose to enter total area for whole group, you should also enter the spacing between the tendons if you are going to calculate crack widths.

Initial strain

Initial strain should be given as strain due to initial stressing of the tendon, minus all losses at the actual position of the cross section.

$$\epsilon_i = \epsilon_P - \epsilon_{loss}$$

Before entering an initial strain you must be certain that the tendon forces (P and P*e) are not already accounted for in the applied cross section forces.

If tendon forces have already been included in the cross section forces, then initial strain should be set to zero. If initial strain is not set to zero, then the stressing of the tendon will be included twice in the cross section design calculations.

If on the other hand, only parasite forces from prestressing of the tendons are included in the applied cross section forces, then stressing of the tendon should be accounted for in NovaDesign by entering an appropriate value for ϵ_i .

Note!

For crack calculations for tendons, the cover value you enter for the tendons reduced by half the duct diameter. You must therefore enter the appropriate value for duct diameter. Also, reduction in section area due to ducts are not included in the calculations. This simplification will in most cases have little effect on the accuracy of calculated results.

9.3. Shear reinforcement

| | |
|----------------|---|
| Ascii command: | SHRE SHREPT SHREAS |
|----------------|---|

NovaDesign offers the possibility to add shear reinforcement to the cross sections and calculate utilization ratios (UR) of the shear reinforcement due to shear and torsion moments.

Note!

You can include shear- and torsion calculations in the design calculations even if you have not added shear reinforcement to your cross sections. It is sufficient that you have defined shear and/or torsion areas in order to calculate required amounts of shear reinforcement.

Geometry

It is assumed that all shear reinforcement is oriented in the cross section plane. Geometry and location of the shear reinforcement units are defined through a series of points, which can be of two types; ordinary points and radius points. There are three types of ordinary points; A, B and C.

Ordinary points are connected to a defined and numbered section point on the cross section surface, and to the cross section surface line before and/or after this section point. Radius points can be placed between two ordinary points to create a curved section of the shear reinforcement bar.

Note!

It is not necessary to enter radius points for required bending of the bar. This will be entered automatically by the program.

Ordinary points

The section points, which the ordinary points are connected to, are also called reference points. In addition to the reference point, you must also enter two cover values, C1 and C2. Location of the reinforcement point is calculated based on reference point, reinforcement point type and cover values C1 and C2 according to figure 9.

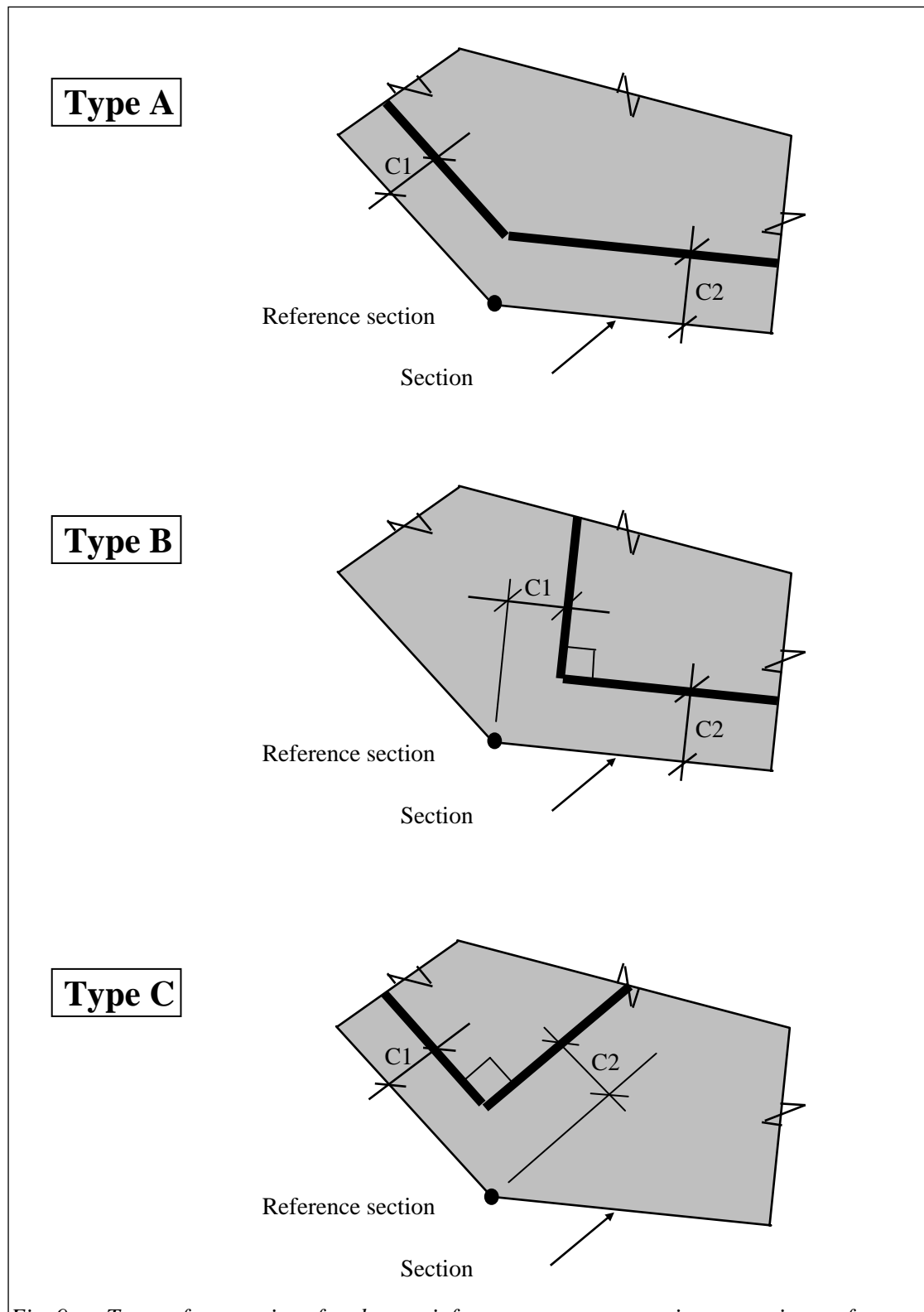


Fig. 9 — Types of connections for shear reinforcement geometry points to section surface

If the two line segments for type A are close, or parallel (angle less than 1 degree), the reinforcement point will be placed a distance C1 from a line perpendicular to the section surface through the reference section point.

If one of the line segments is a curved part of the section surface, the tangent line to the curve at the reference section point is used for finding the location of shear reinforcement point.

Radius point

A radius point must always be inserted between two ordinary points

Note!

If you enter a radius point and the radius is too small to connect the two points it links, the radius point will be removed, and a warning message issued.

Reinforcement amount

In the same way as for ordinary reinforcement, the amount of shear reinforcement is set by selecting a reinforcement type. A reinforcement type contains:

- bar diameter
- number of bars pr. bundle (normally 1)
- spacing between each bundle

End hooks

It is possible to select different types of end hooks for the shear reinforcement bar.

Your selection here will not affect the calculated results. The end hook type will only be used for drawing purposes and when calculating bar length and weight.

It is possible to choose 'markers only'. If you select this option, no contribution from end hooks will be added to bar length or weight.

9.4. Reinforcement and shear reinforcement types

| | |
|----------------|--------------------------------|
| Ascii command: | RETYP SHRETYP |
|----------------|--------------------------------|

A reinforcement type is a description of the amount of reinforcement for a reinforcement group. In the same way shear reinforcement types apply to shear reinforcement.

Note!

Reinforcement types do not apply to tendon groups

Reinforcement and shear reinforcement types are common data for all reference lines and cross sections in your model. Each type has a unique ID. If you create a new reinforcement or shear reinforcement type when working on a cross section, this type will be available for all cross sections and all reference lines. Also the ID assigned to this type can not be used by another type later. However a reinforcement type and a shear reinforcement type may have the same ID.

You can create or edit reinforcement types in a dialog window, which you can open. You open this dialog window by pressing the **Modify list...** button in the dialog windows where you define reinforcement groups and shear reinforcement.

Reinforcement types

There is three ways to specify the amount of reinforcement for a reinforcement type

1. Give number of bundles (n)
2. Give bar spacing (cc)
3. Give total steel area (As)

You should note that type 1 and 3 describes a fixed reinforcement steel amount. For type 2 however the amount depends on the length of the line the reinforcement group is distributed along.

When using type 3 you must also specify bar diameter and spacing. This is due to crack calculations where these are important parameters. Reinforcement type 3 is always shown as one thick line in the cross section drawing.

The reinforcement types you assign to a reinforcement group are assembled in a list. This list also gives the order in which the program will step reinforcement amount if you use the option in the program for stepping reinforcement automatically. You are free to select the order of the assigned reinforcement types in the list.

Shear reinforcement types

The amount for a shear reinforcement type is always given by defining bar spacing (cc).

The shear reinforcement types you assign to shear reinforcement are assembled in a list. This list also gives the order in which the program will step reinforcement amount if you use the option in the program for automatically stepping reinforcement. You are free to select the order of the

assigned reinforcement types in the list. You should note that shear and a torsion calculation results in a value for maximum bar spacing. If you choose to let the program include this requirement when stepping shear reinforcement automatically in design calculations, the final amount may be too large unless you have included enough alternative spacing values.

10. Cross section forces

| | |
|----------------|------------------|
| Ascii command: | SECTFORCE |
|----------------|------------------|

You can enter values and calculate forces for all 6 components of a beam element. Forces can either be entered directly into NovaDesign (called external forces), or loaded from a NovaFrame analysis through the common database.

Note!

External forces can only be applied to sections on reference line 0.

For entering section forces directly, you select **Section forces...** from the **Properties** menu in the **Cross section Design** window.

Tool button:

The force components are indexed according to the cross section xy-coordinate system;

- Axial force N
- In-plane shear forces V_x, V_y
- Bending moments M_x, M_y
- Torsion moment T or M_t

10.1. Sign conventions

Direction of positive section forces are defined as shown in figure 4

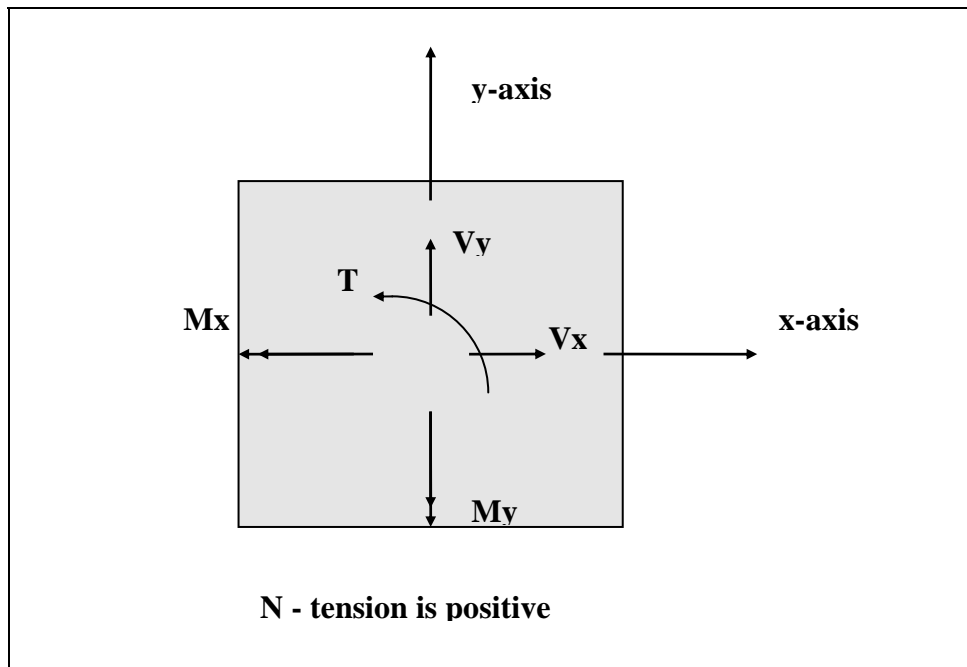


Fig. 9.1 Positive section force directions

10.2. Sign conventions using section forces from NovaFrame analysis

If you are using the interaction between NovaFrame and NovaDesign, and are loading cross section forces directly from the common database the sign conventions are handled by the program.

It is however important that you understand how NovaDesign cross sections are oriented when applied to elements in NovaFrame.

Elements using NovaDesign sections connected to reference line = 0

The figure below shows how the NovaDesign cross section is oriented when applied to a frame element. It also shows force components and directions for NovaFrame and NovaDesign. Note that NovaFrame forces are applied on a section on end 2 as seen towards end 1 (opposite of NovaDesign section)

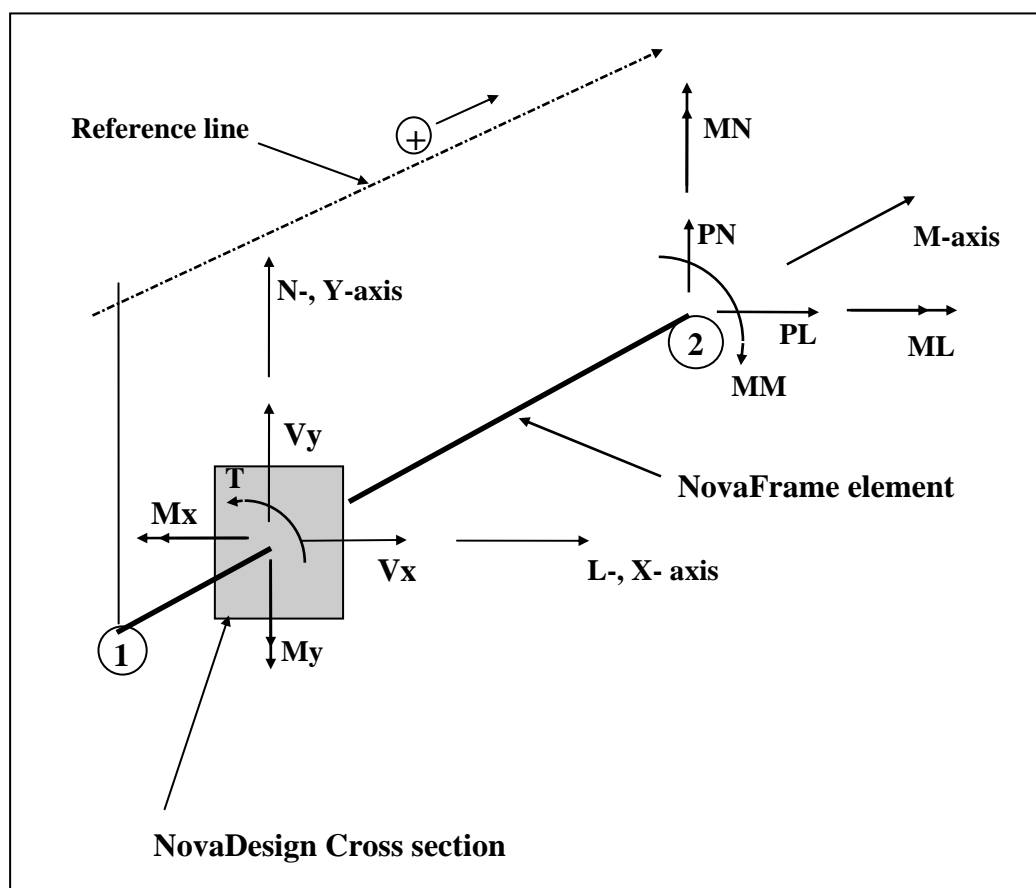


Fig. 9.2 Sign conventions for cross section connected to NovaFrame element

This gives the following relationship in section forces and signs between NovaDesign and NovaFrame;

| | | |
|----------------|---|----|
| N | = | PM |
| V _x | = | PL |
| V _y | = | PN |
| M _x | = | ML |
| M _y | = | MN |
| T | = | MM |

Elements using NovaDesign cross sections connected to real reference lines

The orientation of the cross section shown in the previous figure is also valid for elements connected to a reference line as long as the node at end 2 has a larger station number than the node at end 1.

If not, the x-axis of the section will point in the opposite direction of the element L-axis which gives the following transformation;

| | | |
|----------------|---|-----|
| N | = | PM |
| V _x | = | PL |
| V _y | = | PN |
| M _x | = | ML |
| M _y | = | -MN |
| T | = | MM |

Here the change in sign of bending moment about L-axis is important. This transformation is done automatically by the program. If you list section forces in NovaDesign, you will see that the sign of M_y is changed compared to M_N in the corresponding listing in NovaFrame

In general, shear and torsion calculations in NovaDesign do not depend on the sign of shear or torsion forces (changing signs will not affect calculated results).

11. Cross section sub areas

11.1. Introduction

The cross section area is defined by a series of section points. This area together with reinforcement and tendon amounts are used when finding an equilibrium state between applied section forces (N , M_x and M_y) and internal forces due out-of-plane strain in concrete, reinforcement and tendons.

In-plane forces (V_x , V_y and M_t) however are not included in this equilibrium calculation. The capacity for these section forces are calculated based on simplified calculation methods according to NS 3473. A basic assumption for these methods is that the shear- and torsion areas are rectangular areas. Since most cross sections are not rectangular you must define parts (sub areas) of the cross sections which meet this assumption. You can define any number of sub areas in a cross section. The sub areas in themselves do not carry any of the applied shear- or torsion forces. The sub areas are only area definitions. You must define shear- and torsion areas which refer to the defined sub areas. The same sub area may be used for both shear and torsion areas.

11.2. Sub areas

Ascii command: **SUBAR**

A sub area is defined by selecting a series of cross section points, which defines the extent of the area. The section points should be given in the same order as the section surface points, i.e. in counter clockwise direction.

Note!

You do not have to close the sub area (first and last section point in area must not have the same location). This is done automatically by the program.

Because the sub areas are used in shear and torsion calculations according to simplified methods in NS3473, the sub areas must be close to a rectangular shape. Consequently, all sub areas must have four corner points and four faces extending between corner points. The program automatically finds which of the section points included in the sub area that should be corner points (can be overridden from ascii command input SUBAR). If it can not find suitable corner points, a warning message will be issued, and the program will not accept the area you have suggested.

Conventions used by the program for identification and numbering of corner points and faces for sub areas are shown in the figure below.

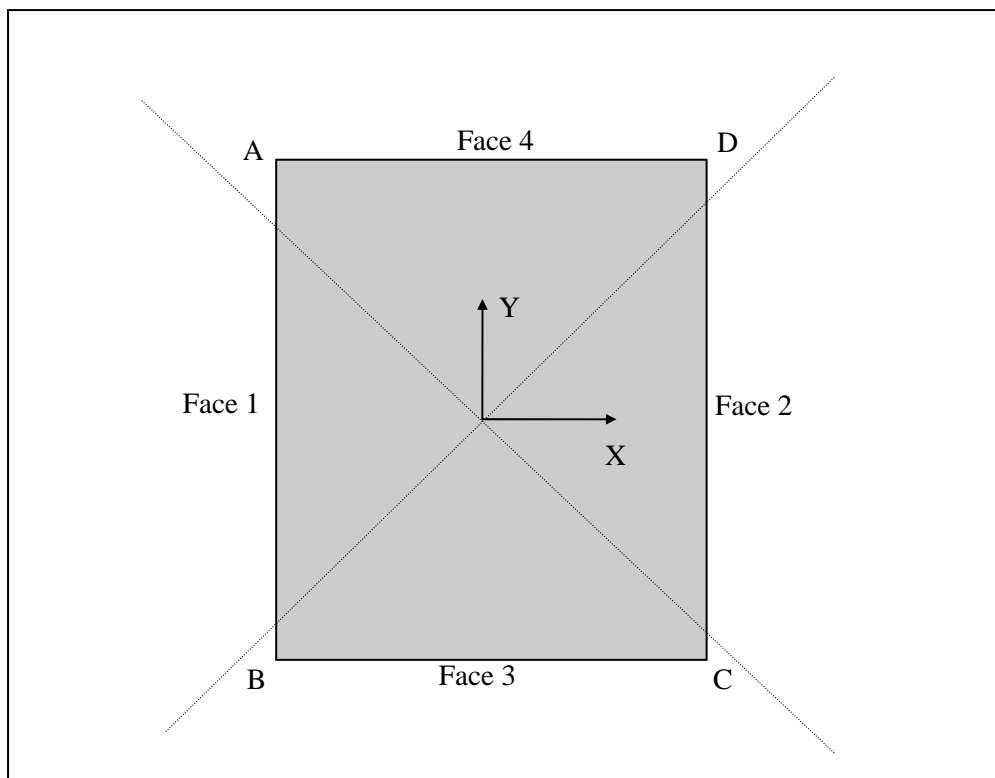


Fig. 10.1 Definition and identification of corner points and faces for sub areas

The program will select corner points by finding the points which have the largest distance to two imaginary lines (dotted lines in the figure above) making a 45° and -45° angle with the x-axis of

the section. If two or more points have the same distance (but different location), the points with the larger distance from X-axis and Y-axis are selected for sub areas with default shear direction in Y-direction and X-direction respectively.

Note!

All points with the exact same location as a corner point are corner points. This is important to remember if you later are defining shear reinforcement and reinforcement groups to be automatically connected to sub area faces as described in the following section.

Section radius points (curves on section surface) can not be included in sub areas, only coordinate points. The sub area will therefore consist of straight-line segments.

Sub areas must be given a type, which describes what type of structure or part of structure the sub area is, according to definitions in NS 3473 section 18. Available types are;

| Type | Default shear force direction |
|--------|-------------------------------|
| BEAM | (Y-direction) |
| COLUMN | (X-direction) |
| WEB | (Y-direction) |
| FLANGE | (X-direction) |
| WALL | (Y-direction) |
| PLATE | (X-direction) |

This type definition is currently used only for defining default shear direction for shear calculations. Default directions are shown in parentheses above.

Properties

The geometrical properties calculated for the sub area are the basis for the properties of the shear- and torsion areas using the sub area.

It is an important facility of the program that the sub areas, and thereby the properties used for shear and torsion calculations, are connected to the actual section geometry through its section points. If you had only a few cross sections and their geometry are not likely to be modified at a later stage, it would be efficient to permit entering of the geometric properties directly. But in many cases you have a variation of section geometry, or make modifications during the design process. A good example is when you have a section geometry that varies along the reference line, such that every design section has a unique section geometry. Then it is efficient to let the program handle calculation of geometrical properties for shear and torsion calculations.

The program will always check the geometry of the sub area before accepting your input.

- **b** or **h** are smaller than or equal to 0
- Average angle between default shear direction and corresponding opposite sub area faces must not exceed values set by the program (see figure below).
- Ratio of width/height must not exceed values set by the program.

| | |
|----------|--|
| A_c | Area of sub area. Area of any holes within the area are subtracted from A_c |
| I_{cx} | Moment of inertia about x-axis for sub area's center of gravity |
| I_{cy} | Moment of inertia about y-axis for sub area's center of gravity |
| S_{cx} | Static moment S_c about y-axis for sub area's center of gravity for shear in x-direction |

$$S_{cx} = (A_c/2) \cdot (b/4)$$

S_{cy} Static moment S_c about x-axis for sub area's center of gravity for shear in y-direction

$$S_{cy} = (A_c/2) \cdot (h/4)$$

b $(l_3+l_4)/2$, where l_3 and l_4 are straight line distance between corner points of face 3 and 4 of connected sub area.

h $(l_1+l_2)/2$, where l_1 and l_2 are straight line distance between corner points of face 1 and 2 of connected sub area

t_{min} Minimum thickness of sub area perpendicular to shear direction

θ_x Angle between section x-axis and centerline between face 3 and 4 of sub area

θ_y Angle between section y-axis and centerline between face 1 and 2 of sub area

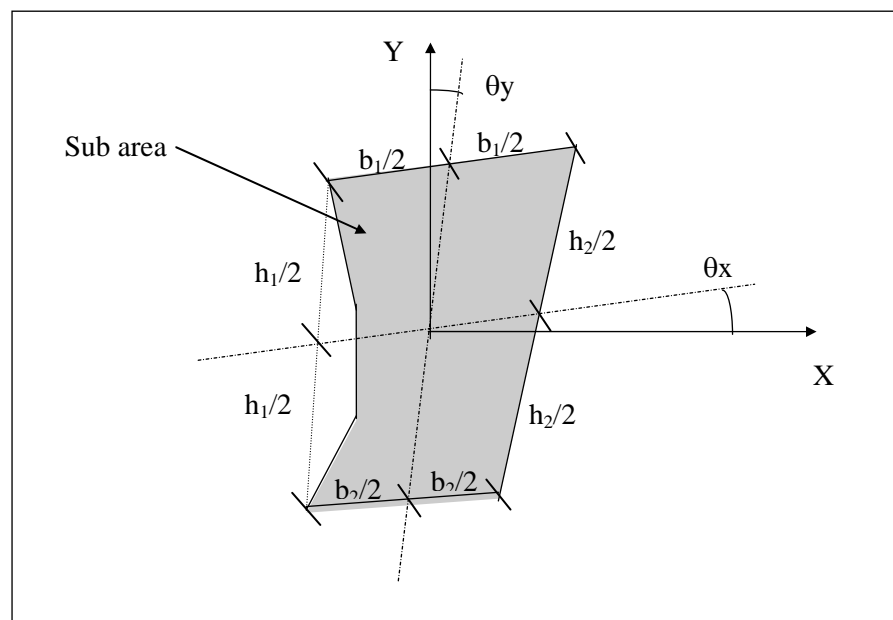


Fig.10.2 Inclination angles of sub areas

11.3. Connecting shear reinforcement to sub area faces

The first step in shear and torsion calculations is to calculate if concrete capacity alone is sufficient. If not, the program will calculate required additional shear reinforcement due to these force components.

You may wish also to let the program compare the calculated required shear reinforcement amounts with the amounts you have in the actual section at the time of calculation. (If the current amount is not sufficient the program can also increase the amounts automatically if you have made this possibility available to the program).

In order to compare required amounts with actual amounts, the program must first be able to find actual amounts connected to each of the faces of the sub areas. This can be done in two ways;

1. Default connection

Let the program find amounts automatically by connecting shear reinforcement to sub area faces.

The program applies some simple rules when finding default connection of shear reinforcement to sub area faces. You should be aware of these rules in order to decide whether they are sufficient for your cross section. The rules used by the program are as follows;

- A shear reinforcement unit is connected to a sub area face if it extends between corner points of the actual face without passing via any other corner points. The term 'extending between' refers to the reference section points for the shear reinforcement geometry definition
- The same shear reinforcement unit can only be connected to the same face once.

2. User specified connection

Make your own selection among defined shear reinforcement units

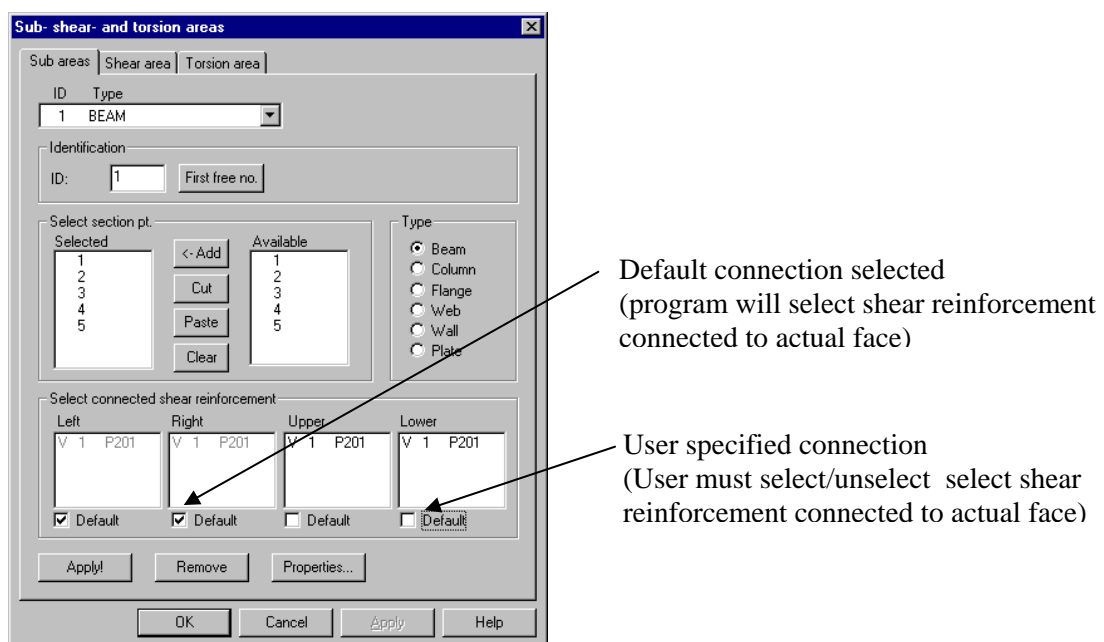


Fig 10.3 Sub area input dialog

An important advantage of using 'default connection' is that if you later add new shear reinforcement to the cross section, or change location of existing shear reinforcement, the program will instantly calculate new shear reinforcement amounts for sub area faces without any action by the user. If you have a 'user specified' connection, you must also remember to include the actual changes in shear reinforcement for affected sub areas.

Note!

The same shear reinforcement unit may be connected to more than one face. Still the actual shear reinforcement will only be assigned the largest value (required A_{sv} or A_{st}) of the contribution from the different faces it is connected to. I.e. the contributions will not be added for the same force component. However, contributions from different force component (V_x , V_y and T) will be added..

A sub area will produce no design calculation results unless it is used in a shear or torsion area. It may seem unnecessary that it is required to go through the job to define sub areas, and then shear and torsion areas. Why not omit sub areas altogether. The reason is that it may be convenient to use the same area again for shear forces in a different direction, or maybe for carrying torsion moments. Then, it is convenient to avoid making the area definition more than once. Another important advantage is that the sub areas serve as a location to collect and combine results from both shear and torsion calculations.

11.4. Shear areas

| |
|-----------------------------|
| Ascii command: SHEAR |
|-----------------------------|

Each cross section can have many shear areas. In order to define a shear area you must enter the shear force component for the shear area (V_x , V_y , or default), and a sub area which serve as the area definition.

If you select 'default' as shear force direction for the shear area, the applied shear force is determined by the type of the connected sub area. See description of default shear directions in the sub area chapter of this User's Guide.

The applied shear force in one direction will be distributed to the defined shear areas in this direction based on the formulas:

$$V_{xi} = V_x \cdot A_{vxi} / \Sigma A_{vx}$$

$$V_{yi} = V_y \cdot A_{vyi} / \Sigma A_{vy}$$

where:

V_{xi} is the shear force applied to shear area i

A_{vxi} is the area of shear area i

ΣA_{vx} is the total shear area in x-direction

11.5. Torsion areas

| | |
|----------------|--------------|
| Ascii command: | TORAR |
|----------------|--------------|

A cross section can have only one torsion area. The torsion area can be one of two types

- Solid area
- Box area

If you select 'solid area' you select one sub area which defines the actual area used for torsion moment.

If you select box area, you must specify four sub areas, one sub area for each wall of the box. You can not select the same sub area more than once within the same torsion area.

Since the box area is defined by four areas, the program must connect them together and select corner points for both inner and outer surfaces. It can only select corner points among the section points that are common for the two intersecting sub areas. It is therefore important that you have included enough section points both when defining cross section geometry and when defining sub area geometry. The corner points that the program searches for are indicated with circles in figure below.

Note!

In the figure below, the top flange sub area may include the whole upper flange. The resulting torsion area would still be as shown as hatched in the figure.

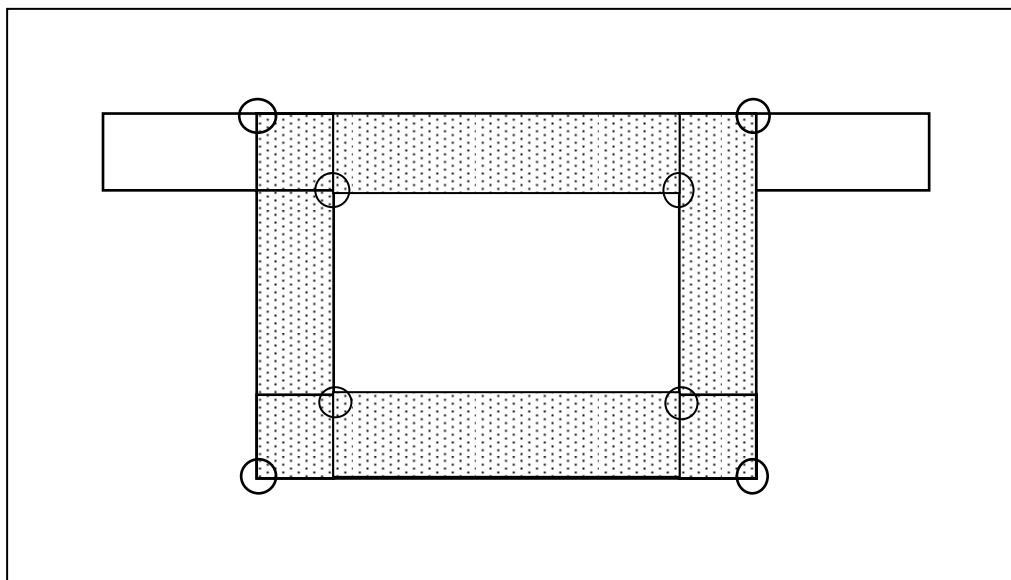


Fig. 10.4 Location of corner points for torsion area of type 'box area'

If there is more than one section point (with different locations) at the intersection of the selected sub areas, the program will select between these points using the method described in section 3.7.

Properties

There are two types of torsion areas in NovaDesign. The way geometrical properties are calculated are the same for some properties, and different for others. Therefore properties calculated the same way for both types are presented first under 'Common'.

General geometrical properties such as A_c and I_c are not included here.

Common

u_c Length along outer surface of torsion area

A_{ct} Area inside outer surface, including area of holes

A_o 'Effective' torsion area

$$A_o = A_{ct} - 0.5 \cdot t_c \cdot u_c, \quad \text{see below for explanation of } t_c \text{ and } u_c$$

Solid area

I_t calculated according to theory for St. Venant's torsion.
 $I_t = k_1 \cdot b \cdot t^3$, k_1 is set to $0.33 - 0.19 \cdot b/h$, where h is always larger than b .
 This is the same formula and k_1 used for calculating torsion stiffness for NovaFrame created rectangular sections in frame analysis

t_c Calculated according to NS 3473 cl 12.4.2
 $t_c = 0.2 \cdot d_{\max}$, where d_{\max} is diameter of the largest circle fitting inside torsion area

u_o $u_o = u_c - 2 \cdot (c_1 + c_2 + c_3 + c_4)$, where c_1 to c_4 are average concrete cover for connected shear reinforcement for face 1 to 4 of sub area. If no shear reinforcement is connected to sub area used as torsion area, c_1 to c_4 is set to default cover value for reinforcement design parameter set with lowest ID.

Note!

Calculation of u_o may be inaccurate if faces of sub area used as torsion area are not straight, or if concrete cover for connected shear reinforcement varies a lot along a face.

Box area

I_t Calculated according to theory for St. Venant's torsion for thin walled sections.

$$I_t = 4 \cdot A_o^2 / (f(ds/t)) \quad (\text{Bredt's 2. formula})$$

For each sub area in torsion area ds/t equals $A_{ci}/2t$, where A_{ci} is the area of actual sub area and t is average thickness of actual sub area

t_c Smallest wall thickness for all sub areas that the torsion area is made up of.
 Concrete outside outer stirrup is not included if:

1. Average concrete cover of shear reinforcement connected to the face on outer surface of each sub area is larger than 50% of average wall thickness of that sub area. The sub areas in mind are those connected to torsion area.

2. Overall shear compression stress is larger than $0.4 \cdot f_{cd}$, calculated for each sub area separately. This is only calculated during design calculations. See note below:

$$\sigma_c = 2 \cdot \tau, \quad \tau = \tau_T + \tau_V$$

$$\tau_T = T / (4 \cdot A_o \cdot t_{ci}),$$

T is applied torsion moment

t_{ci} is average wall thickness (A_c/h) for actual sub area

A_o is area inside centerline of stirrups. If no shear reinforcement is entered, centerline of box area is used

$$\tau_V = V_i / A_{ci},$$

V_i is applied shear force on sub area i which is included in torsion area

A_i is area of sub area i

Note!

t_c is load dependent. Therefore t_c is always recalculated for each load combination in design calculations

$$u_o = u_c - 2 \cdot (c_1 + c_2 + c_3 + c_4),$$

where c_1 to c_4 are average concrete cover for connected shear reinforcement for faces on outer surface for sub area 1 to 4. If no shear reinforcement is connected to sub area used as torsion area, c_1 to c_4 is set to default cover value for reinforcement design parameter set with lowest ID.

Note!

Calculation of u_o may be inaccurate if faces of sub areas used in torsion area are not straight, or if concrete cover for connected shear reinforcement varies a lot along a face.

12. Design calculations

12.1. Introduction

| | |
|-----------------|---|
| Ascii commands: | DCSETUP DCOPT DCSECT DCCOMB DCCONV |
|-----------------|---|

Design calculations can be run for all cross sections created in NovaDesign. The applied cross section forces can either be loaded directly from a NovaFrame analysis through the common database, or you can enter the section forces directly in the **Cross Section Forces** dialog (external load combinations). In both cases the term 'load combinations' is used, even if there is no load combination performed in NovaDesign or NovaFrame for section forces entered in the **Cross Section Forces** dialog.

Design calculations can be run for both ordinary load combinations and sorted combination lines from a NovaFrame analysis (see NovaFrame User's Guide). Results from single NovaFrame load cases can not be used in design calculations because no limit state is assigned to them.

NovaDesign can perform the following in a design calculation;

1. Find required amounts of ordinary reinforcement and shear reinforcement based on different design requirements.
2. Calculate utilization of concrete reinforcement and tendons for applied section forces.

You can perform all necessary operations connected to design calculations from the **Design calculation** window, which is available from both document window and **Cross section design** window.

Tool button: 

From the **Design calculation** window you can;

- Create or modify design calculation setups
- Run design calculations
- View results

There are many options available related to how design calculations should be performed, which effects to include, and which section force components that should be included. You make your selections from the available options in a design calculation setup, which is further described in the next section.

12.2. Design calculation setup

A design calculation setup is a selection of cross section and load combinations to include in a design calculation. A design calculation setup also includes a set of options which can be used to customize the design calculation in detail.

You can create or edit design setups by selecting **Setup..** from the **Design calculation** menu in the *Design calculations* dialog.

Tool button: 

Each design calculation setup has an ID and a name for later reference. You can create any number of setups. The calculation setups can be executed from the *Run Design Calculation* dialog. Select **Design Calculations – Run...**

Tool button: 

Options

A short explanation of the options available for the design calculation setup are given below. The options you set for a calculation setup will apply to all selected sections and load combinations included in the setup.

Include shear forces

Activate this option to include calculation of shear force capacity according to NS 3473 section 12.3. Note, shear areas must have been defined for the actual cross sections!

Include torsion moment

Activate this option to include calculation of torsion moment capacity according to NS 3473 section 12.4. Note, torsion areas must have been defined for the actual cross sections!

Sort max-min forces

If you select this option, the program will pick only the worst 14 load combinations for each limit state among the combinations you have included in the design setup. You can not use this option together with sorted combination lines

The following load combinations are included:

| | |
|---------|-------------------------------|
| Min/Max | N |
| Min/Max | Mx |
| Min/Max | My |
| Max | Mr (resulting bending moment) |
| Min/Max | eccentricity (Mr/N) |
| Min/Max | Vx |
| Min/Max | Vy |
| Max | T |

If there are less than 14 load combinations available for a section, only the actual number of load combinations are included. The max-min sorting routine may not always include the 'worst case', but should in most cases be sufficient.

Include slenderness effects

Activate this option to include calculation of additional moments due to slenderness effects according to NS 3473 section 12.2. Note, slenderness effects are only calculated if you have also assigned buckling lengths for the cross sections through connected section design parameter sets!

Lock current longitudinal reinforcement

Activate this option to lock all reinforcement amounts at current level. If deactivated the program will step reinforcement amount using the reinforcement tables defined for the reinforcement groups

Step to meet crack width requirements

If this option is activated will step reinforcement amount to try to meet the crack width requirement (w_k). This option is only available if the 'Lock current longitudinal reinforcement' option is deactivated.

Step reinforcement with compressive strain

If this option is activated the program is allowed also to step reinforcement groups in the compressive zone of the cross section. The program will still always try to step reinforcement groups in the tensile zone first, until last entry in reinforcement table is reached for these groups. This option is only available if the 'Lock current longitudinal reinforcement' option is deactivated.

Start on first amount in reinforcement step table

If this option is activated the program will reset all reinforcement amounts for included cross section to the first entry in the reinforcement step table before starting design calculations (if not it will start on the currently selected amounts). This option is only available if the 'Lock current longitudinal reinforcement' option is deactivated.

Lock current shear reinforcement

Activate this option to lock all shear reinforcement amounts at current level. If deactivated the program will step shear reinforcement amount using the reinforcement tables defined for the shear reinforcement units.

Step to meet maximum bar spacing requirements

If this option is activated will step reinforcement amount to try to meet the maximum spacing requirements from the torsion calculations (NS 3473 section 12.4.6). This option is only available if the 'Lock current shear reinforcement' option is deactivated.

Start on first amount in reinforcement step table

If this option is activated the program will reset all shear reinforcement amounts for included cross section to the first entry in the reinforcement step table before starting design calculations (if not it will start on the currently selected amounts). This option is only available if the 'Lock current shear reinforcement' option is deactivated.

Sections included in setup

You must select which cross sections or design sections in a NovaFrame analysis that should be included in the setup. If you wish to include design sections from a NovaFrame analysis in your setup, you can do this by either specifying intervals of elements or interval of stations, if you are using section by reference line in NovaFrame. You are not allowed to mix selection by element and reference line within the same setup

Note!

If you specify sections for reference line 0, only external section forces are available, and not section forces from the NovaFrame analysis. If you are using sections from reference line 0 in your frame analysis you can include these sections by specifying the actual NovaFrame elements in your setup.

Load combinations included in setup

Select which load combinations to include in the design calculation setup. The program will search for the specified load combinations for all sections you have included in the setup. No warning message is given if a load combination included in the setup is not found

You can include either ordinary load combinations or sorted combination lines in a setup. The term 'ordinary load combinations' means both ordinary load combinations from NovaFrame, and external forces entered in NovaDesign. 'Sorted combination lines' are a special type of load combinations defined in NovaFrame. See NovaFrame User's Guide for explanation of this term.

12.3. Iteration method

When calculating equilibrium between applied sectional forces (N, M_x and M_y) and internal forces the program uses a Newton-Raphson iteration method. The following relation between strain and forces is used:

$$\mathbf{F}_i = \mathbf{K} \cdot \mathbf{r}$$

| | |
|----------------|--|
| \mathbf{F}_i | internal forces (N, M _x , M _y) |
| \mathbf{K} | Stiffness matrix |
| \mathbf{r} | internal strain (ϵ_o , κ_x , κ_y) |

Normally equilibrium is achieved after only a few (3-5) iterations. Some times however the program fails to find an equilibrium state. The text 'No Convergence' will then appear in the result listing for the actual cross section an load combination. There are two reasons why the iteration can fail:

1. Applied sectional forces are too large
2. Iteration method fails

The first case is very rare, because the stress strain curves for all materials have a small Young's modulus even when ultimate strain limit is exceeded. This modulus is automatically set to half the initial modulus. However, if you have included additional moments due to slenderness effects in your design calculations this could give a convergence problem because the total bending moments are increasing rapidly during the iteration process. This indicates a stability problem for your structure. You can check if this is the problem by listing the additional moments in the **Design Calculation** window to see if they are extremely large.

The second case may occur even if the applied section forces are within the capacity of the cross section. One can easily check if the convergence problem can be due to an overloading of the cross section by creating a moment-axial- or biaxial-moment chart for the actual cross section, and then plot the section forces in the chart.

If the iteration fails you could solve this problem by changing the convergence settings. These settings are available from **Design Calculation – Convergence settings...** in the **Design Calculation** window.

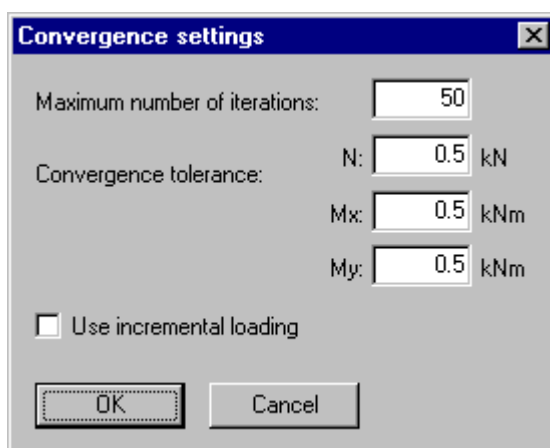


Fig.11.1 Convergence settings for design calculations

The available options for improving convergence of the iteration process are:

- Increase maximum number of iterations (default; 50)
- Increase convergence tolerance (default; 0.5 kN or kNm)
- Use incremental loading of the sectional forces. (default; off)

Increase convergence tolerance is normally the most efficient way to achieve convergence.

Increasing the maximum number of iterations to more than 150-200 will normally have little effect.

12.4. Crack width calculations

Crack widths (w_k) are calculated for SLS load combinations for both ordinary reinforcement and tendons. Calculations are carried out according to NS 3473 section 15.2, 15.6 and A.15.6. The calculation

- Check if concrete section is uncracked
- Calculate crack spacing S_{rk} for each reinforcement and tendon group separately
- Calculate crack width W_k for each reinforcement and tendon group separately

The crack width calculation can be switched off for any reinforcement group (not tendons). This option can be activated from the input dialogs or in the ascii command input. This can be useful if the calculated crack widths obviously are too conservative due to limitations in the program input or calculation method. In particular the ratio between the effective concrete area and reinforcement area may be too large, and thereby giving larger crack widths. This is further explained below. You should however be careful before switching crack width calculation off, and make sure you have fully understood the calculation method and implementation of the crack width calculations before doing so.

Crack load

If the maximum concrete tensile stress for the applied section forces is less than the limits given in NS 3473 section 15.6.1, then the concrete is assumed to be uncracked, and all crack widths are set to zero for the actual cross section and load combination.

Concrete is uncracked if (NS 3473, 15.6.1)

$$(k_w \cdot \sigma_N + \sigma_M) < k_w f_{tn} / k_t \quad (\text{tensile axial section force})$$

$$(\sigma_N + \sigma_M) < k_w f_{tn} / k_t \quad (\text{compressive axial section force})$$

k_t is a reinforcement- and tendon design parameter input. k_w is taken as the maximum cross section height perpendicular to the actual bending (neutral) axis.

Even if concrete can be assumed uncracked in the crack width calculation, the concrete tensile stress capacity is set to zero during calculation of the equilibrium state between applied sectional forces and internal forces.

Crack spacing S_{rk}

Crack spacing is calculated for each reinforcement and tendon group separately (same S_{rk} for all rebars or tendons in the actual group) according to the formula:

$$S_{rk} = 1.7 \cdot \{ s_{ro} + k_c \cdot A_{cef} / \Sigma[\pi \cdot \phi / (f_{tk} \cdot k_b / \tau_{bk})] \} \quad (\text{NS 3473, A.15.6.1a})$$

An important factor in this formula is the ratio between effective concrete area (A_{cef}) and the actual reinforcement amount within this area ($\Sigma[\pi \cdot \phi]$). NovaDesign calculates A_{cef} as the area limited by the distribution line for the reinforcement group (start and end section points) and h_{cef} , perpendicular to this distribution line. See figure below. The height of the effective concrete area h_{cef} is calculated by the program according to NS 3473. The input value $h_{cef, max}$ in NovaDesign (reinforcement group input) is only an upper limit value for h_{cef} . Normally $2.5 \cdot (c + \phi/2)$ or $h/(1 + \varepsilon_{min}/\varepsilon_I)$ will give a smaller value.

| | | |
|-----------------------------|---|-----------------------------|
| h_{cef} = the smaller of: | $h_{cef, max}$ | (design parameter input) |
| | $2.5 \cdot (c + \phi/2)$ | (calculated by the program) |
| | $h/(1 + \varepsilon_{min}/\varepsilon_I)$ | (calculated by the program) |

If two or more reinforcement or tendon groups are connected to the same section face (same section points) it is assumed that the same value for crack spacing S_{rk} can be used for all of them. The smallest value is then used for all of these reinforcement and tendon groups.

When calculating the actual reinforcement amount that contributes to $\Sigma[\pi \cdot \phi]$ for A_{cef} for one reinforcement or tendon group, the program will take only the rebars or cables for the actual reinforcement group. This means, even if there is another reinforcement group with rebars within the calculated A_{cef} , these rebars will not be added to the sum $\Sigma[\pi \cdot \phi]$. This method is slightly conservative if, for example, there is more than one layer of reinforcement.

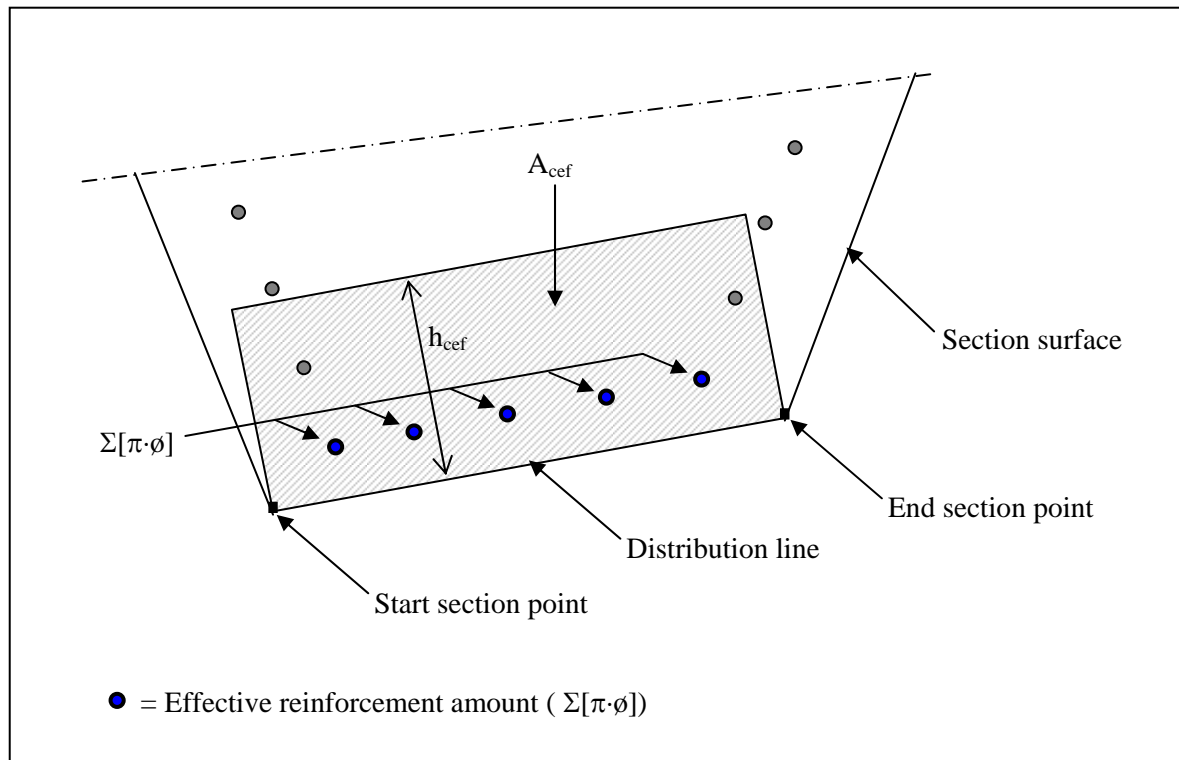


Fig.11.2 Effective concrete area and contributing reinforcement area

Crack widths

Crack widths are calculated for each rebar and tendon separately according to the formula:

$$w_k = S_{rk}[(1 - \beta \cdot \sigma_{sr2}/\sigma_{s2}) \cdot \sigma_{s2}/E_{sk} - \epsilon_{cs}] \quad (\text{NS 3473, A.15.6.2c})$$

β is a design parameter input value. By default $\beta = 0.4$, which applies for permanent or repetitive loads

There is no input in NovaDesign for the shrinkage strain ϵ_{cs} , and the value is automatically set to zero. This is an acceptable simplification if the actual construction is free to move in the direction of the shrinkage and there is no significant build up of internal forces due to shrinkage. If, however, shrinkage induces sectional forces in the construction because it is constrained to deformations (boundary conditions), the effect of shrinkage must be included in the crack width calculation. Suggested methods are:

- Apply shrinkage loads in NovaFrame using an appropriate (cracked) section stiffness. Add shrinkage the shrinkage load case to the load combinations used for design calculations.

or:

- Calculate crack widths due to shrinkage strain alone based on calculated crack spacing S_{rk} : $w_k = S_{rk} \cdot \epsilon_{cs}$. Then, reduce allowed crack widths (w_d) accordingly in the design parameter set for reinforcement and tendons before running design calculations.

These methods also apply for other imposed deformation loads such as temperature loads.

12.5. Slenderness calculations

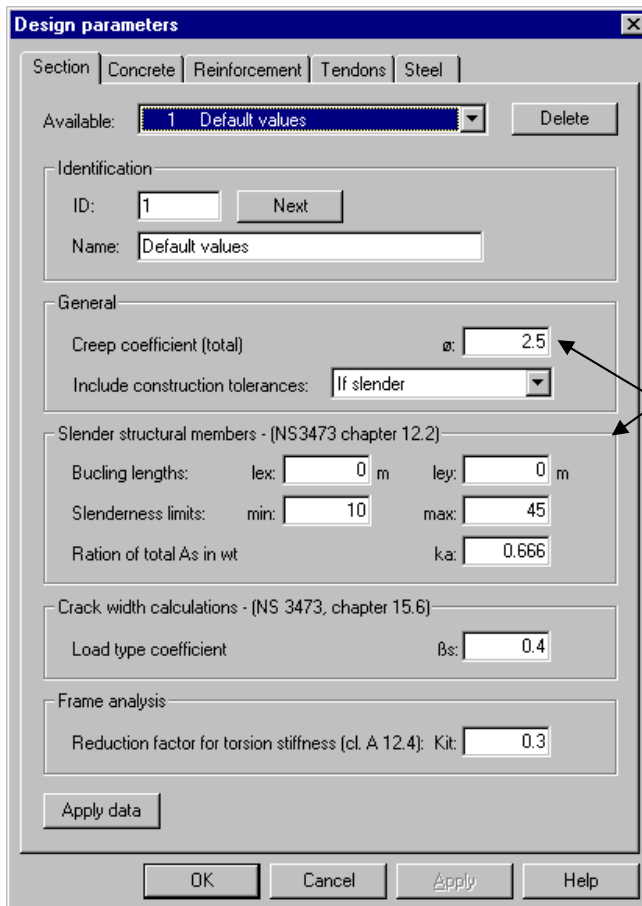
For slender structure cross sections loaded with a compressive axial force, additional bending moments due to slenderness effects should be included according to NS 3473 section 12.2 and A12.2

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

Buckling lengths can not be calculated in NovaFrame. You must therefore evaluate the size of these 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 NovaDesign. The buckling lengths and other slenderness specific input data are section design parameter input. In the **Cross section design** window select **Properties – Design parameters...**

Tool button: 



Design parameters

Section: Concrete Reinforcement Tendons **Steel**

Available: 1 Default values [Delete]

Identification

ID: 1 [Next]

Name: Default values

General

Creep coefficient (total) α : 2.5

Include construction tolerances: If slender

Slender structural members - (NS3473 chapter 12.2)

Buckling lengths: lex: 0 m ley: 0 m

Slenderness limits: min: 10 max: 45

Ratio of total A_s in wt ka: 0.666

Crack width calculations - (NS 3473, chapter 15.6)

Load type coefficient β_s : 0.4

Frame analysis

Reduction factor for torsion stiffness (cl. A 12.4): Kit: 0.3

[Apply data]

[OK] [Cancel] [Apply] [Help]

Input for slender structural members

Fig.11.3 Slenderness calculation input

The axis definition of buckling lengths are:

lex: buckling about cross section x-axis (buckling deformation in y-direction)
 ley: buckling about cross section y-axis (buckling deformation in x-direction)

If buckling length is given about only one axis then additional moments will be included only about this axis (buckling is forced in one direction). If buckling lengths are given for both axis, then the program calculates additional moments for both axis. The main buckling direction is then selected as the direction giving smallest resistance against buckling. Normally, this is one of the main cross section axis (x- or y-axis).

The following effects are included in the slenderness calculations:

- Construction tolerances
- Deflection due to long term effects (creep)
- Deflection due to applied bending moments (and additional moments)

Additional 1. order moments due to construction tolerances (NS 3473 section 12.2.3) are always included about the axis where buckling lengths are given.

If necessary slenderness data is entered for a cross section, the first thing the program does is to calculate the geometric slenderness (λ) and the load dependant slenderness (λ_N). If the geometric slenderness is larger than recommended in NS 3473 section 12.2.4, a warning message will be generated, but the calculation will continue. If the load dependant slenderness about an axis is lower than the limit given in the section design parameter input (see figure in previous page), then the additional bending moments are set to 0.0.

When calculating the geometric slenderness the contribution from the reinforcement and tendons are also included (ϖ_t , NS 3473 section 12.2.4) The contribution is scaled with the factor k_a which is a section design parameter input. The default value for this parameter is 2/3 (see NS 3473 section 12.2.4). The value for f_{sd} in the formula for ϖ_t is set to maximum the stress at 2.5 o/oo strain of the tendon steel.

Results from the slenderness calculation can be listed in the design calculation window.

For listing of total additional moments select;

Display – Section Add. Moments

Tool button



For detailed results select:

Display – Section Slenderness


Tool button

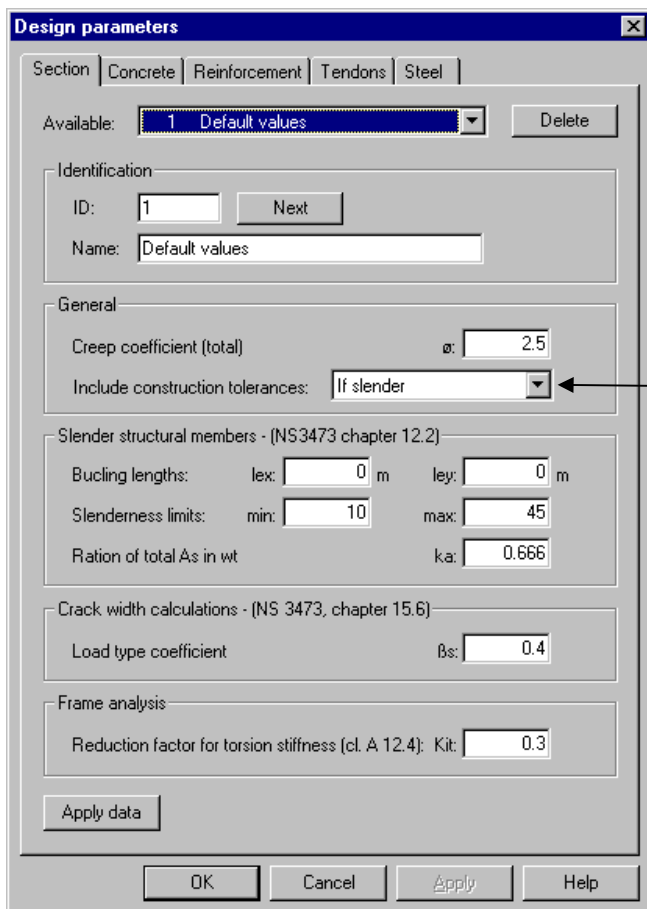


12.6. Construction tolerances

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 in NovaDesign 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 **Cross section design** window select **Properties – Design parameters...**

Tool button: 



Design parameters

Section Concrete Reinforcement Tendons Steel

Available: 1 Default values Delete

Identification

ID: 1 Next

Name: Default values

General

Creep coefficient (total) α: 2.5

Include construction tolerances: If slender ← Option for construction tolerances

Slender structural members - (NS3473 chapter 12.2)

Buckling lengths: lex: 0 m ley: 0 m

Slenderness limits: min: 10 max: 45

Ratio of total As in wt ka: 0.666

Crack width calculations - (NS 3473, chapter 15.6)

Load type coefficient βs: 0.4

Frame analysis

Reduction factor for torsion stiffness (cl. A 12.4): Kit: 0.3

Apply data

OK Cancel Apply Help

Fig. 11.4 Section design parameter input dialog

There are two options available;

- 'If slender' Include additional moments due to construction tolerances only if the actual only if 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'

13. Capacity charts

| |
|---|
| Ascii commands: CHART CHARTLN |
|---|

NovaDesign can calculate four types of capacity charts:

- Moment-axial force
- Moment-curvature
- Biaxial moment
- Moment-strain

All types of charts are calculated in the same dialog window. You open this dialog by selecting **Capacity Charts** from the **Calculate** menu in the *Cross section design* window, or from the document window

Tool button:



The chart dialog window is similar to the cross section design window. Just below the tool bar there is a drop list containing all previously calculated charts. When you open this window for the first time, this list is empty, and you must start with creating a new chart. You do this by selecting one of the **new...** options in the **Chart** menu. The program will then display a chart setup dialog window. Chart setup is described in the next section

All charts you create are stored in the NovaDesign document. If you do not wish to display them in the document, simply uncheck the capacity chart chapter in the document setup.

13.1. Capacity chart setup

The setup gives all necessary input for the program to calculate a new chart. You can also change the setup for an existing chart by selecting **Modify...** in the **Chart** menu.

Strain limits

You can choose between using strain limits in design parameter sets connected to the cross section, or you can enter strain values specifically for the current chart.

If you choose to use strain limits from design parameter sets, you can have different strain values for all reinforcement and tendon groups in the cross section. The program is capable of handling this, but there could be situations when the program produces slightly conservative results.

If you enter strain limits specifically for this chart, all reinforcement groups will have the same strain limit, and all tendon groups will have the same strain limit.

Chart line

You can include any number of chart lines in one chart. Each chart line can have a unique rotation axis (ϕ) and/or axial force (N). All other values in the chart setup are common for all chart lines in the chart.

13.2. Modifying chart and display options

Following first time creation of a chart, you may later change any of the calculation requirements or display options. However you may not change chart type. For example you may not change a Moment-Axial chart into a Moment-Curvature chart. If you wish to modify calculation requirements or cross section for the active chart, you open the dialog box for chart modification as described in the previous section.

You may also make some changes in display attributes of the chart, such as; text to chart, display of axis etc. You do this by selecting **Attributes...** options in the **Display** menu

Toolbar:

13.3. Maintenance of charts

NovaDesign makes sure that all charts are updated. If you change geometry, reinforcement, material parameters or design parameters for a cross section, all charts using this cross section will be updated, also in the document. If you delete a cross section, the charts using this cross section will also be deleted automatically. If material or design parameter sets included in a cross section are deleted such that calculation requirements are no longer available for a chart, the chart will not be deleted but the chart will become blank.

13.4. Viewing chart results

Listing

Normally you will wish to view the calculated capacity tables as charts, because graphically visualized information is usually easier to understand. However, some times you may wish to get a listing of the calculated values. You can do this by selecting **View listing...** from the **Chart** menu. You may print the listing, or save it in an ascii file. You can also copy from the listing (Ctrl + C), and paste the text (Ctrl + V) into any program file that accepts text from the clipboard.

Tool button:

Moment axis

It is important to note that the values in the charts always refer to moments about the section x- and y-axis. If you select an angle for the neutral axis different from 0 for a MN-chart, you will get a display of moment capacity about x-axis (by default). In order to obtain bending about the specified neutral axis, there is a corresponding bending moment about y-axis. You can display this by switching displayed moment axis. If you wish to get the total capacity (M_r), you will find a column for this in the chart listing.

Tool button:

14. Tutorial

This tutorial is split into two parts - A and B.

The text describing the different steps in this tutorial is rather detailed, but we recommend that you follow the procedure carefully in the start, click by click.

14.1. Tutorial 1 – Part A

In this part you will get an introduction to the following subjects:

- Create a new file
- Create cross section geometry
- Add ordinary reinforcement
- Enter cross section forces
- Run a design calculation
- View results from design calculations.

Follow the steps in the tutorial to create a cross section according to the description below. Apply section forces and check if suggested reinforcement is sufficient.

Input data:

| | |
|----------------------|---|
| Cross section: | Column-section |
| Concrete grade: | C45 (default concrete grade, see section 4) |
| Reinforcement steel: | B500C (default steel grade, see section 4) |
| Concrete cover: | $c = 55 \text{ mm}$ |
| Crack width limit: | $w_k = 0.3 \text{ mm}$ |
| Reinforcement: | $\phi 16 \text{ c150}$ |

| | | |
|----------------|------|--|
| Applied forces | ULS: | $N = -2000$, $M_x = 300$, $M_y = 1400$, $V_x = 600$, $V_y = 200$, $T = 300$ |
| | SLS: | $N = -1400$, $M_x = 200$, $M_y = 900$, $V_x = 0$, $V_y = 0$, $T = 0$ |

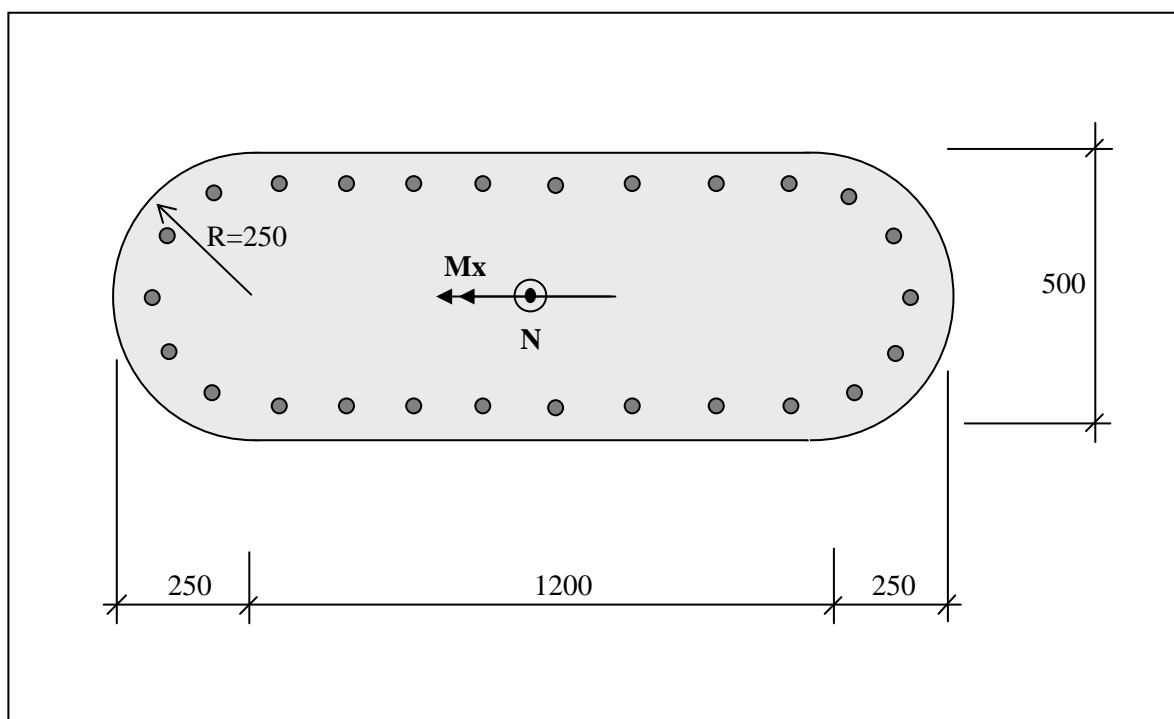
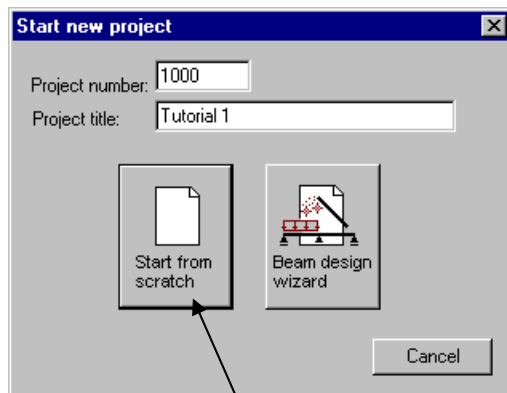


Fig. 13.1 Tutorial cross section

1. Create a new file

- Select **New...** from the **File** menu . The *Start new project* dialog is displayed.
- Enter project number, and set '*Tutorial 1*' as project title.



- Press the '*Start from scratch*' button, and a new file will be opened.

The program window should look like the figure below. The new file has the default name '*Database1*'.

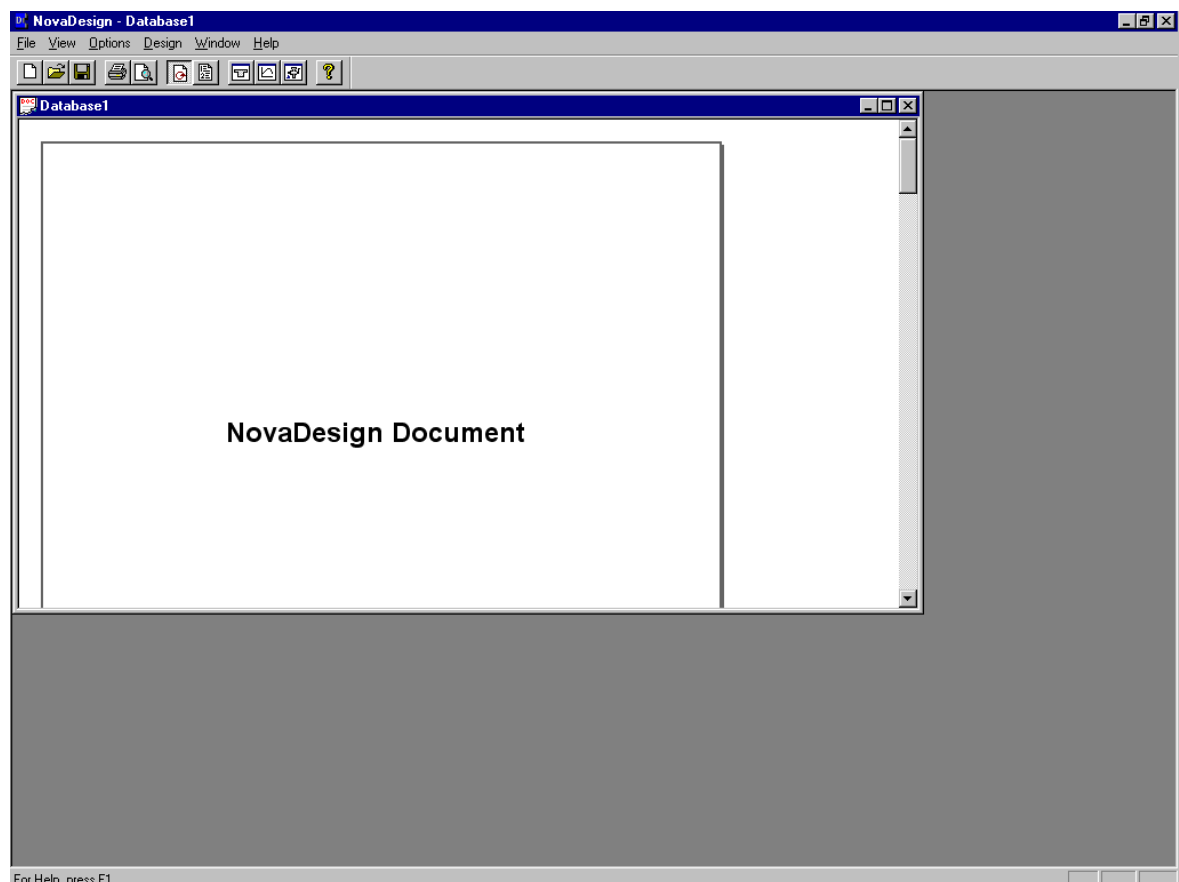


Fig. 13.2

2. Create cross section geometry

- Select **Design - Cross Sections...** from the menu to open the *Cross Section design* window.

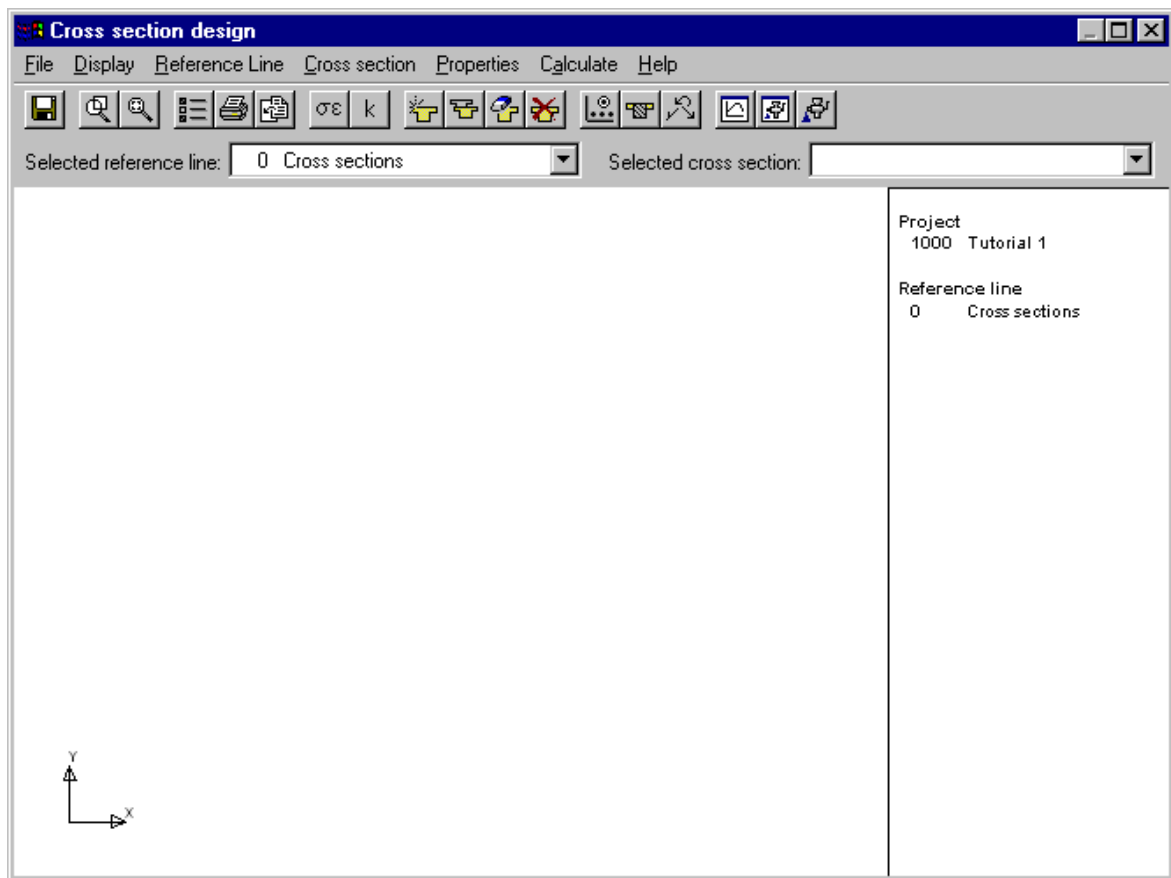


Fig. 13.3

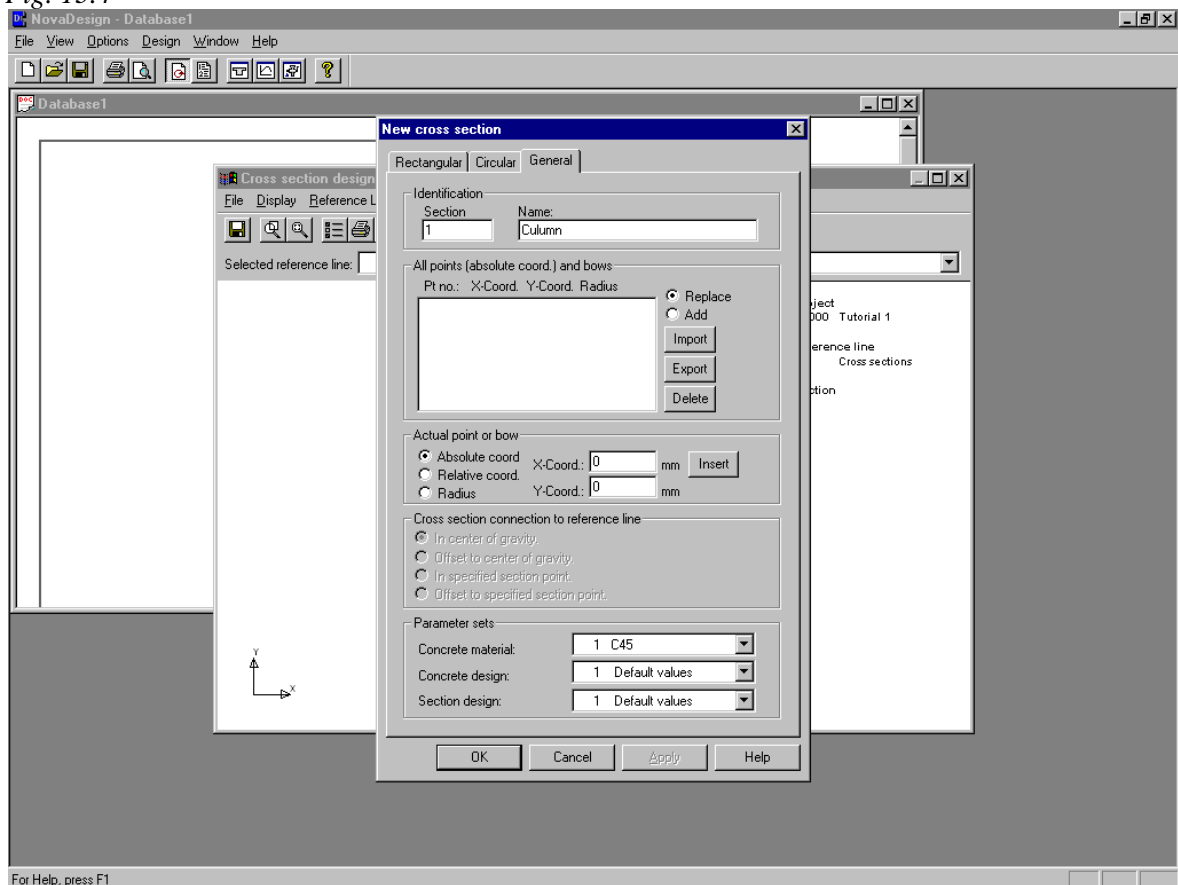
The *Cross section design* window is the main window for building cross sections. A plot of the cross section you are working on is shown in the drawing area of the window. You have not yet created the cross section. Therefore the drawing area is empty, except for the legend to the right. The drop list '*Selected cross section:*' is also empty. It would normally hold the station numbers and names of all available cross sections.

Now, create the new cross section:

- Select **Cross Section -New...** from the menu. The input dialog for creating a new cross section is displayed.
- This dialog has three tabs at the top. Press the *'General'* tab, and the input window for entering section points becomes visible.
- Keep the default section number, *'1'*.
- Give the new section a name, for example. *'Column'*
- Keep the default concrete grade, C45.

The program window should now look like the figure below. It is time to start entering section points defining the surface of the cross section.

Fig. 13.4



Enter the following section point coordinates (unit mm):

| Point | X-coord | Y-coord | Radius |
|-------|---------|---------|--------|
| 1 | 0 | 0 | |
| 2 | 1250 | 0 | |
| - | | | 250 |
| 3 | 1250 | 500 | |
| 4 | 0 | 500 | |
| - | | | 250 |
| 5 | 0 | 0 | |

Enter first section point:

- The coordinates is in our case (0, 0) for the first section point. Make sure 0 is entered in both the 'X-' and 'Y-coord' field.
- Press the 'Insert' button. The section point is added to list. As you can see, the section point number is set automatically.

Enter next section point:

- The coordinates for the next point should be (1250, 0). Enter 1250 in the 'X-coord' field. Press the 'Insert' button.

Enter radius point:

- Press the 'Radius'
- The radius should be 250. Enter 250 in the 'Radius' field. Press the 'Insert' button.

The input dialog should now look like this:

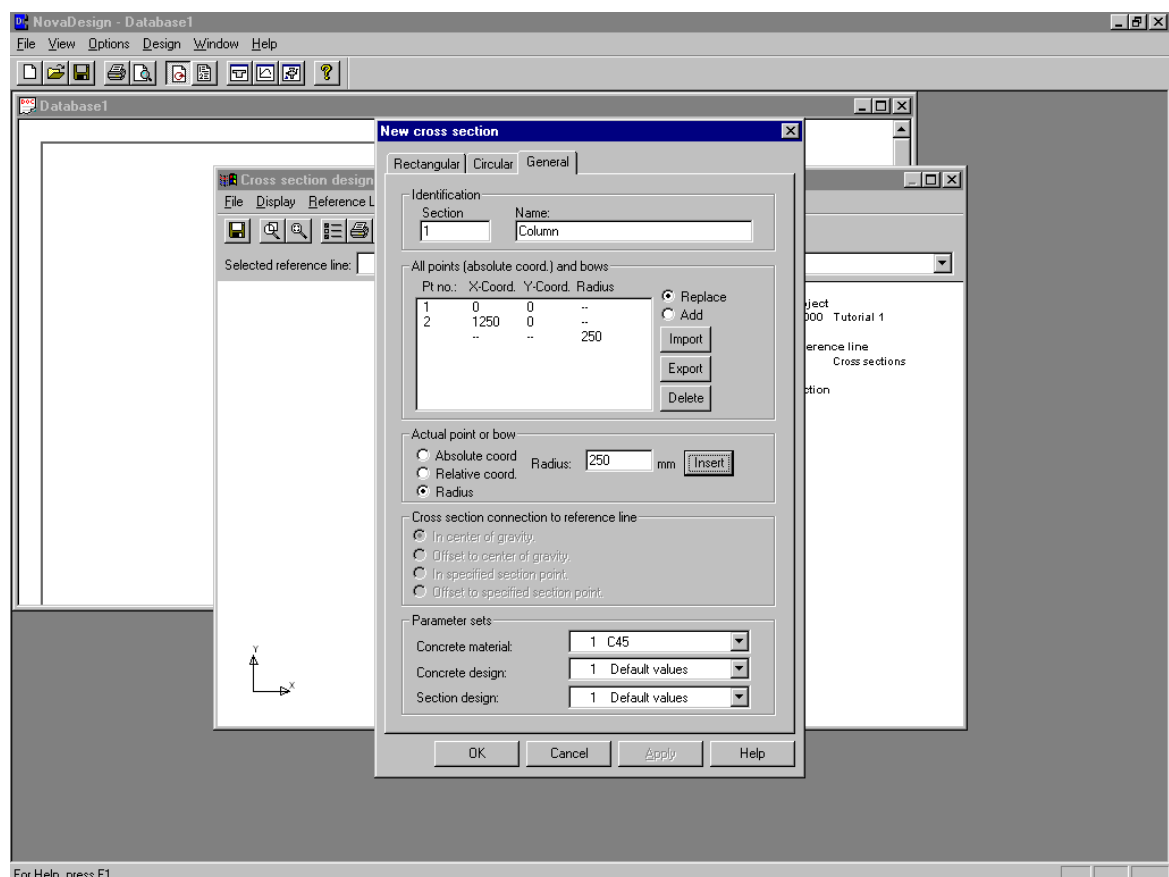


Fig. 13.5

- Enter the remaining section points, (remember to switch back to '*Absolute coord*' before entering next section point)
- When all section points are entered, press the OK button.

If you should enter a wrong coordinate or radius value, you can modify the section point by double-clicking the actual section point in the list. The '*Insert*' button will then become a '*Modify*' button. Enter new coordinate values in the input fields, and press the '*Modify*' button.

The input dialog should now look like this:

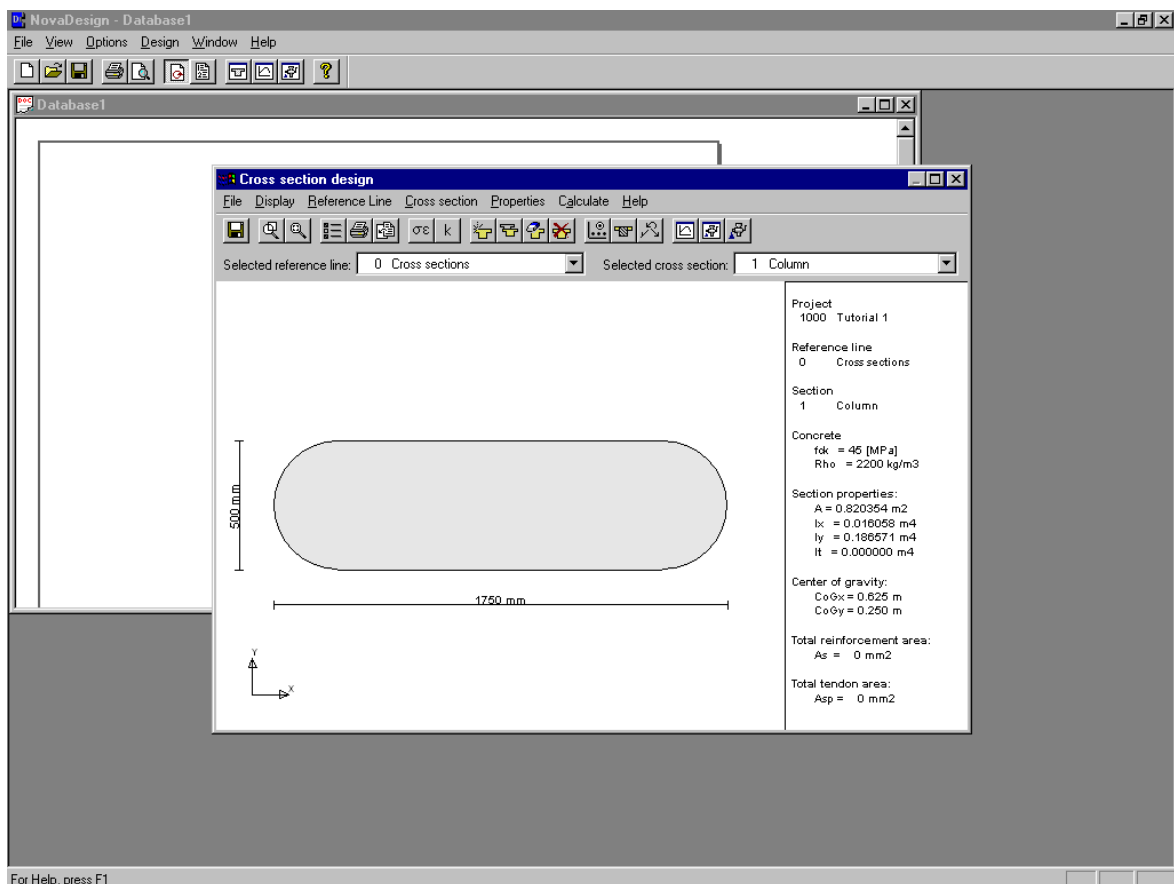


Fig. 13.6

3. Add ordinary reinforcement

- Select **Reinforcement & Tendons ...** from the **Properties** menu. Tool button: 

Enter dialog data as shown in the figure below. The reinforcement is connected to the line between section point 1 and 2. Cover is set to 65 mm to allow space for stirrups. Lateral cover is set to 40 mm (lateral cover is the distance parallel to the section face).

The input value h_{cef} refers to NS 3473 section A.15.6.2.1. Enter the cross section thickness perpendicular to the reinforcement location (or half this distance if opposite face is also reinforced, as in our case).

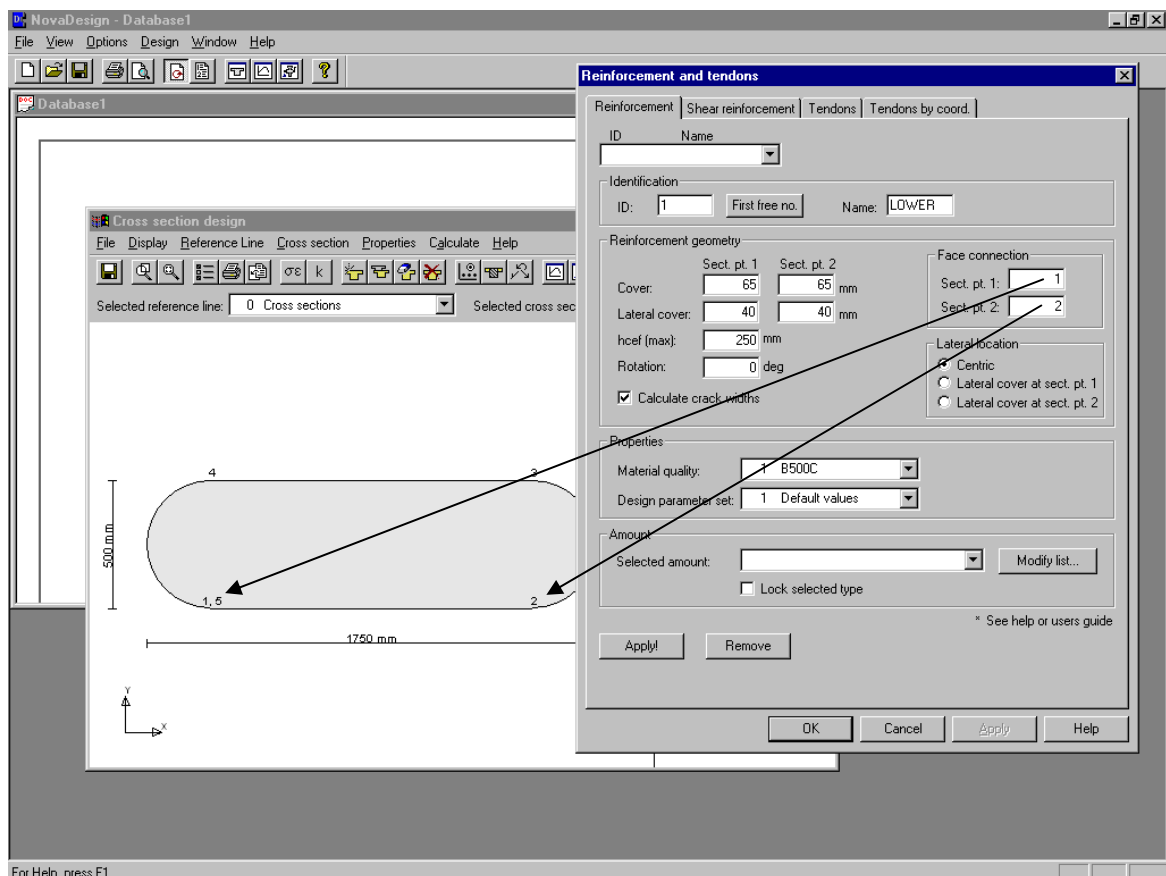
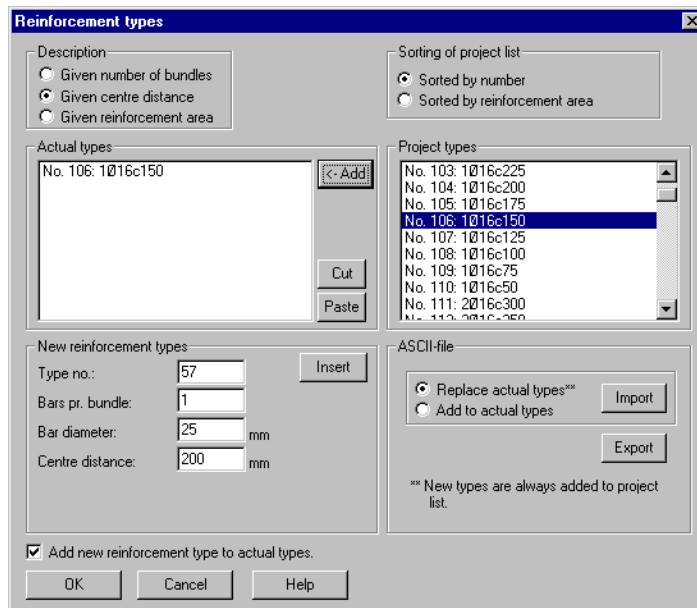


Fig. 13.7

As you can see, the list box 'Selected amount:' in the 'Amount' frame is empty. Therefore, you must press the 'Modify list...' button to open the **Reinforcement types** dialog to specify an amount.

This dialog has a large number of input options. We do not need to go into detail on this now. Just follow the steps in the next page:

- Select 'Given center distance'
- Select (click) the item '1Ø16c150' in the list box under 'Project types'
- Press the '<- Add' button to add this amount to the 'Actual types' list.



- Press the 'OK' button.
- '1Ø16c150' should now appear as the selected amount in the reinforcement input dialog.
- Press the 'Apply' button

The input dialog should now look like this:

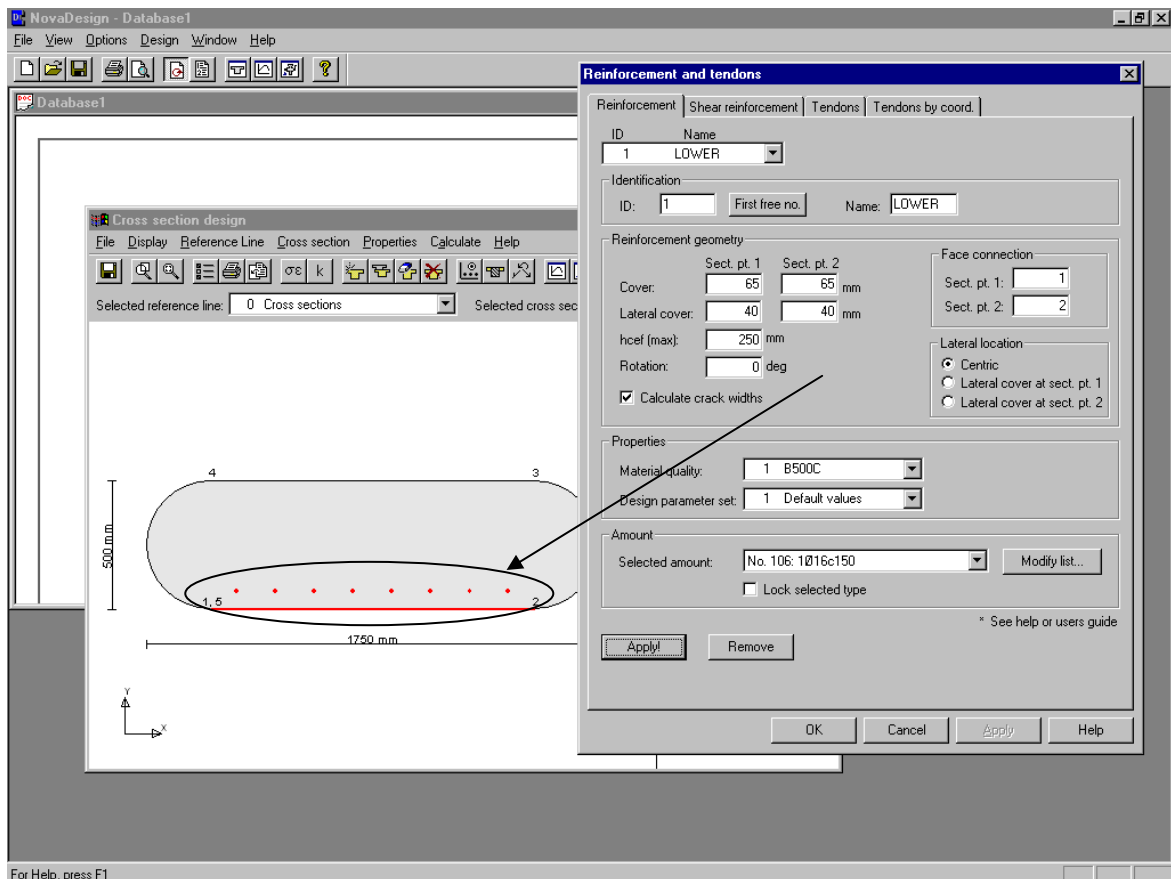


Fig. 13.8

You have now entered the first reinforcement group located at the lower face of the section. We should then add a second group at the upper face:

- Enter a new ID (2) for this new reinforcement group. and change name to 'UPPER'
- Set section points in 'Face connection' to '3' and '4'.
- Press the 'Apply' button, and the upper face reinforcement should be visible in the cross section plot in the background.
- Press the 'OK' button.

Do the same for reinforcement group 3 (RIGHT), section points 2 and 3 and reinforcement group 4 (LEFT), section points 4 and 5.

The input dialog should now look like this:

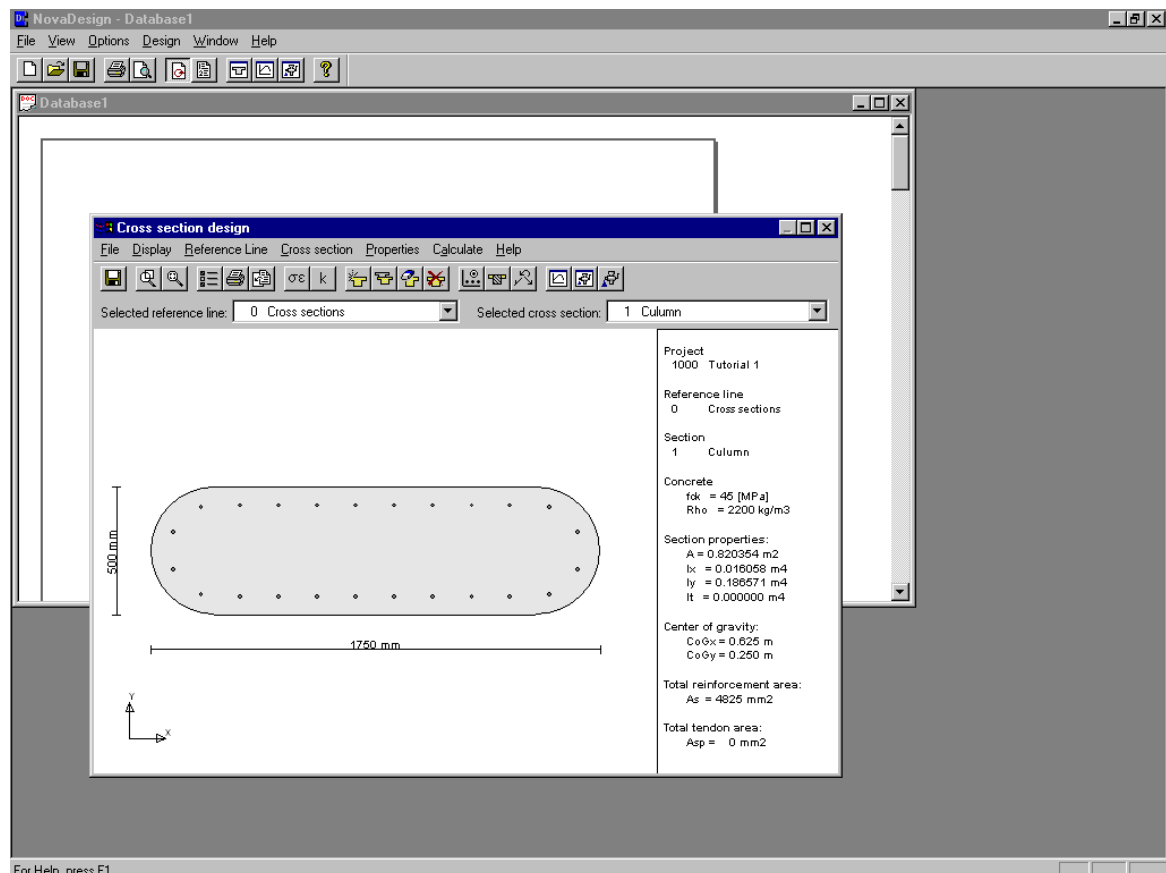


Fig. 13.9

You have now completed input of geometry and reinforcement, and its time to enter the cross section forces.

4. Apply cross section forces

- Select **Section forces ...** from the **Properties** menu. Tool button: 

- Enter the cross section forces for combination 1 (ULS):

$$N = -2000, M_x = 300, M_y = 1400, V_x = 600, V_y = 200, T = 300$$

- Press the '*Insert*' button

- Change the selected limit state to '*SLS*'

- Enter the cross section forces for combination 2 (SLS):

$$N = -1400, M_x = 200, M_y = 900, V_x = 0, V_y = 0, T = 0$$

- Press the '*Insert*' button

The input dialog should now look like this:

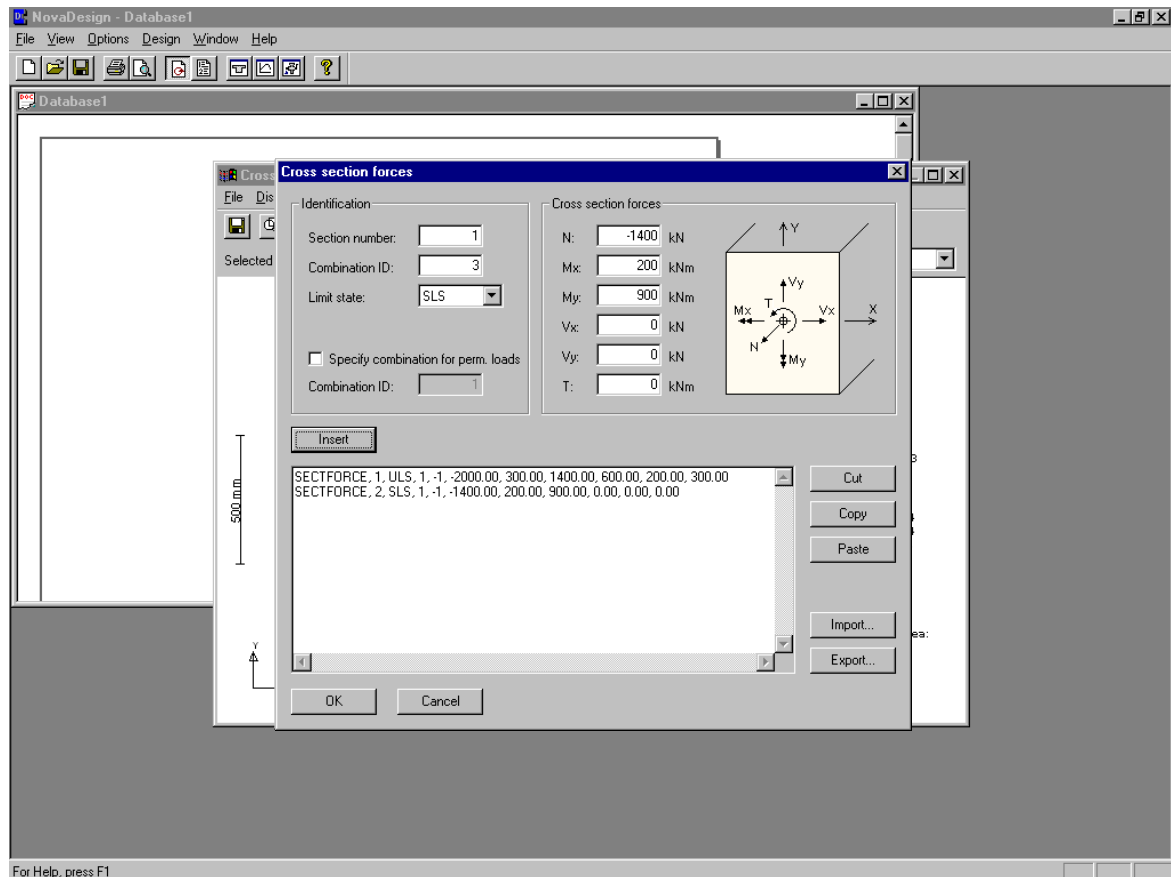



Fig. 13.10

- Press the '*OK*' button to close the cross section forces dialog.

You are now ready to calculate the cross section strain, stress and crack widths for the applied sectional forces.

6. Run design calculations

- Select **Design Calculations...** ... from the **Calculate** menu, to open the *Design calculation* window.

Tool button:  .

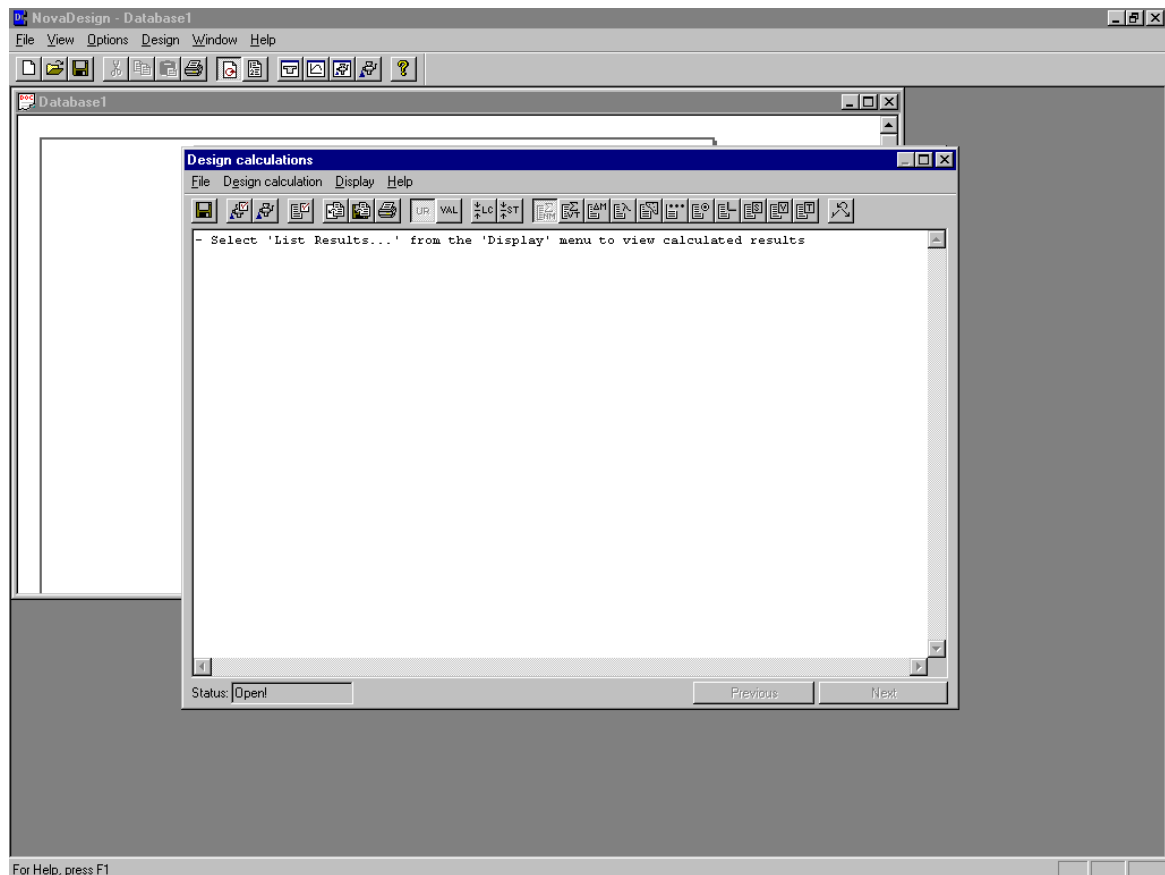


Fig. 13.11

- Select **Calculate Setup...** from the **Design calculation** menu to open the *Calculate* dialog

Tool button:  .

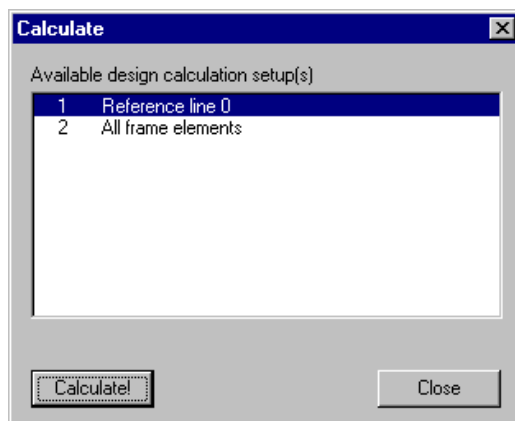



Fig. 13.12

The default selection in this dialog, setup number 1, is OK. How to create a custom setup for a design calculations is further described in section 11.2.

- Just press the '*Calculate*' button, to start the calculations.

The calculation is now completed.But where is the calculation output?....

The answer is; you must first make a selection of which results (cross sections and combinations) to list, because there is no default selection. The program will, however, remember the last selection if you re-enter this window or run calculations again

- Select **List results...** from the **Display** menu. Tool button .
- Select '*1 Column*' in the list of available sections. Combination 1 and 2 will then appear in the list of available combinations.
- Select combination 1 and 2. (use shift- or ctrl- for multiple-selection list boxes)

The dialog window should now look like this:

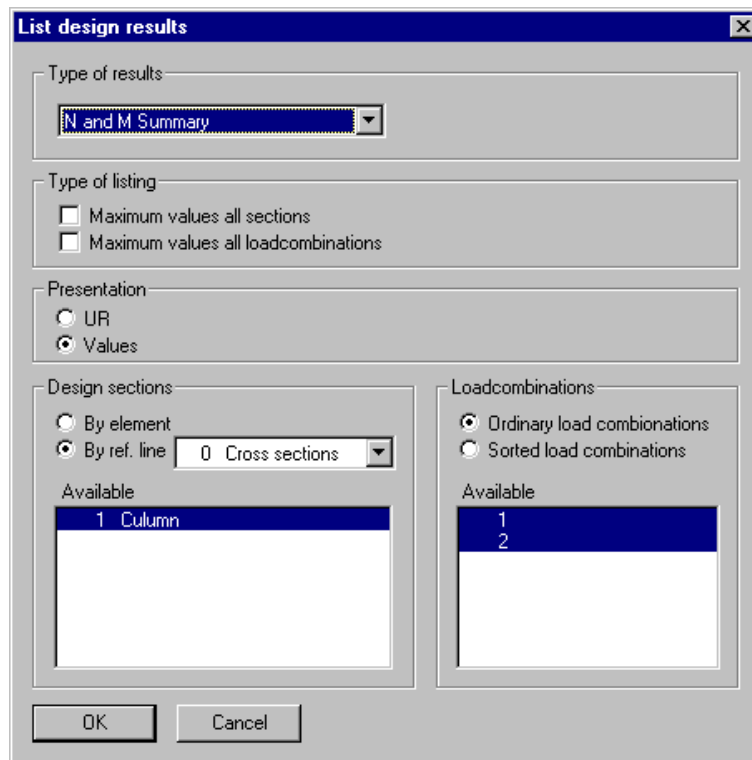


Fig. 13.13

- Press the '*OK*' button.

Moment and axial force result summary will be listed in the window. Note, the listed values are UR (utilization ratios). As you can see all URs are acceptable (< 100%).

| | |
|-----------------------|-----------|
| Concrete stress: | UR = 85 % |
| Reinforcement stress: | UR = 57 % |
| Crack widths: | UR = 84 % |



Conclusion: Reinforcement amounts are sufficient.

You can now select from a large number of result tables for current selection of cross sections and combinations. The different tables are available in the **Display** menu, or use the tool bar at the top of the window

Tool bar:



Also try:

- Switch to listing of values (instead of URs). Tool button: 
- Make a printout. Tool button: 

Close the *Design calculation* window before moving to the next step.

7. Save to file

The data you have entered so far are only in the computer memory. In order to save to file, select **Save** in the **File** menu (this menu is available from all the program windows).

- Enter 'Tutorial 1' as file name.
- Select a folder, and press the 'Save' button

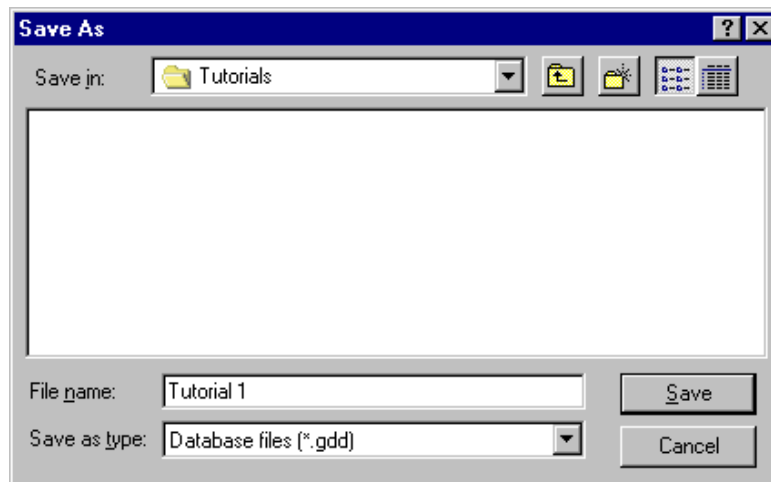


Fig. 13.14

14.2. Tutorial 1 – Part B

In this part you will get an introduction to the following subjects:

- Enter input for calculating slenderness effects
- Modify reinforcement amount
- Create a reinforcement step table
- Customise the design calculation setup

The column section that you created in part A is the cross section of a column with a height $L = 14.0$ m. It is fixed at the bottom and free to rotate at the top ($l_e = 0.7 \cdot L$).


Input data:

| | | |
|-------------------|--|---|
| Cross section: | Column-section | |
| Column height: | 14.0 m | |
| Buckling lengths: | $0.7 \cdot 14.0 \text{ m} = 9.8 \text{ m}$ | (about both axis) |
| Creep factor: | 1.75 | (after 100 years, load applied after 28 days) |

8. Input data for calculating slenderness effects

All required input data for calculating slenderness effects are located at the Section design parameter input dialog.

From the *Cross Section Design* window select **Properties – Design Parameters...**

Tool button: 

The dialog box has five pages. The first page is the section design parameter input ('Section' is the text at the top of this page). As you can see there is already one parameter set available called 'Default values'. This parameter set is one of several default data that are created when you open a new database. This parameter set is already connected to the cross section we created in step A. We will therefore just modify the data for this parameter set, and not create a new set.

- Change the name to '*le = 9.8 m*' (any name can be used)
- Set the creep coefficient to *1.75*
- Set buckling lengths $l_{ex} = 9.8$ m, $l_{ey} = 9.8$ m
- Leave slenderness limits (load dependent slenderness) and k_a unchanged
- Press the '*Apply data*' button to apply the new data to the design parameter set
- A dialog box will appear to inform you that calculated results (from step A) will be deleted. Press the '*Yes*' button to accept.

The dialog should now look like this:

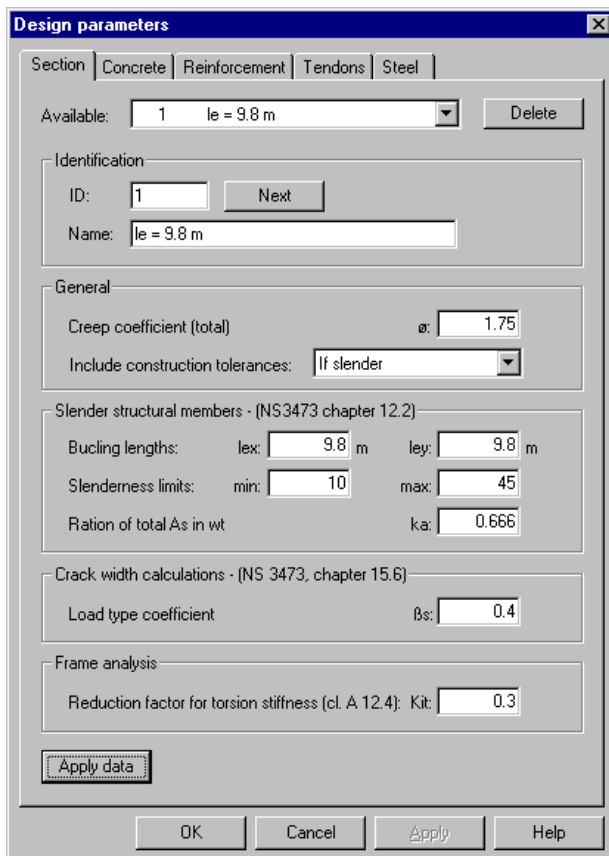



Fig. 13.15

- Press the '*OK*' button to close the dialog

9. Check capacity with slenderness effects included

Now you can re-run the design calculation. Follow the procedure in section 6 in step A of this tutorial.

If you list the axial and bending moment summary, you will see that the calculation did not converge for the ULS load combination. This is due to the large additional moments due to the slenderness effects.

- List the detailed results for slenderness effects. Tool button: 

As you can see, the additional moment due to deflection (Mdefx) is more than four times the applied moment about the weak axis (Mx)

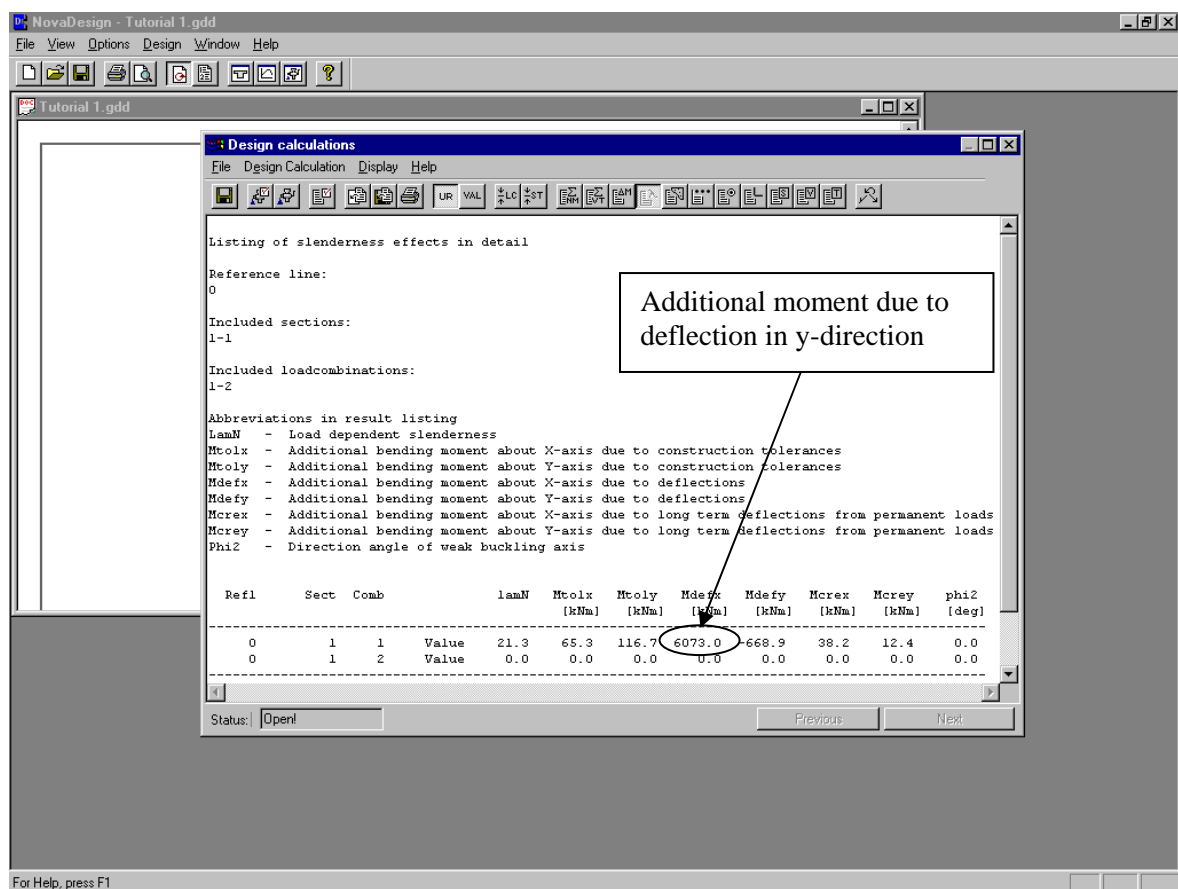


Fig. 13 16

Additional reinforcement is therefore required. In the next step of this tutorial we will create a reinforcement step table for the 'UPPER' and 'LOWER' reinforcement group.

You may wonder why calculation of additional moments due to slenderness effects were suddenly included in the design calculation, even if no changes were made to the design calculation setup before re-running the calculation. The reason is that the calculation setup 'Reference line 0' also includes slenderness effects, but in step A of this tutorial no buckling lengths were given, and therefore the additional moments were zero.

9. Create a reinforcement step table

We will now modify the reinforcement amounts. We will do this by setting up a table of reinforcement types for each reinforcement group. During the design calculation the program can step the reinforcement amount according to these tables until all design requirements are met (or the bottom of all tables is reached).

- Close the **Design Calculation** window

- Select **Properties - Reinforcement & Tendons...** from the menu. Tool button: 

Now, add the reinforcement types '1 ϕ 20c150', '2 ϕ 16c150' and '2 ϕ 20c150' to the reinforcement type list for reinforcement group 1 ('LOWER'), by following the steps below:

- Make sure the reinforcement group 'LOWER' is selected in the drop list
- Press the 'Modify list...' button to open the **Reinforcement types** dialog.
- Select the amount '1 ϕ 20c150' in the 'Project types' list. (You must scroll down the list to find this amount)
- Press the '<- Add' button. The list 'Actual types' should now contain two items.
- Add the amounts '2 ϕ 16c150' and '2 ϕ 20c150' in the same way.

The dialog should now look like in figure below.

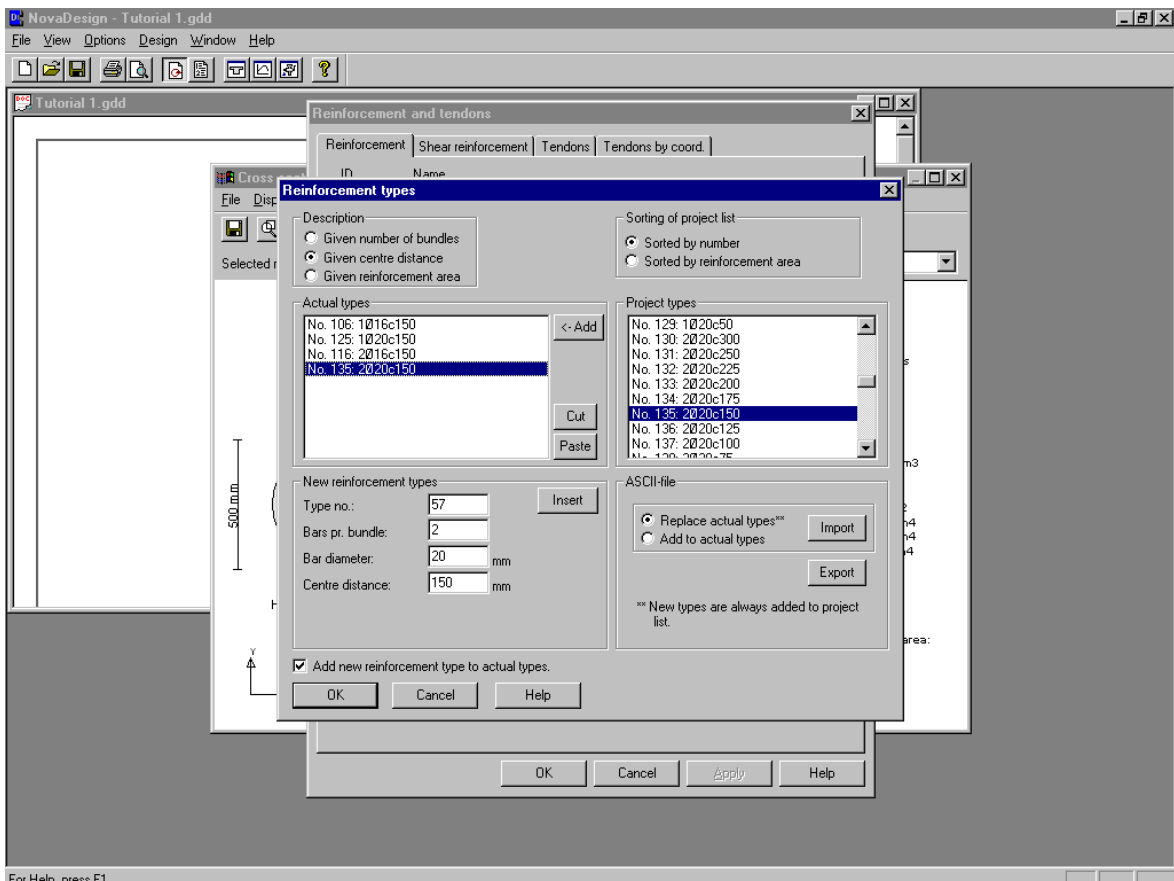


Fig. 13.17

- Press the 'OK' button

Take a look at the '*Selected amount:*' drop list in the reinforcement dialog. As you can see, the selected amount is still '*1 ϕ 16c150*'. But if you open the drop list you can see that you now have also the three other amounts as available choices. Your reinforcement step table for the reinforcement group '*LOWER*' is now:

1 ϕ 16c150
1 ϕ 20c150
2 ϕ 16c150
2 ϕ 20c150

- Select '*1 ϕ 16c150*' in the drop list.
- Press the '*Apply*' button to accept the new input data.

Now, do the same with the reinforcement group '*UPPER*'. (use the drop list at the top of the dialog to change the selected reinforcement group)

- Press the '*OK*' button to end reinforcement input.

The current reinforcement is unchanged for all the reinforcement groups in the cross section, but '*UPPER*' and '*LOWER*' now have a table with increasing reinforcement amounts. The program can use these tables to step reinforcement amounts during design calculations. In the next step we will create a new design calculation setup which allows the program to do this.

10. Create a customized design calculation setup

In step A of this tutorial you ran a design calculation from a default setup, without knowing what this setup included. We will now take a closer look at the design setup, and create a setup which allows the program to step the reinforcement amounts to meet the design requirements.

- Select **Calculate - Design Calculations...** from menu to open the *Design calculation* window.

Tool button: .

- Select **Design calculation - Setup...** from the menu to open the *Design calculation setup* dialog

Tool button: .

The dialog should look like this:

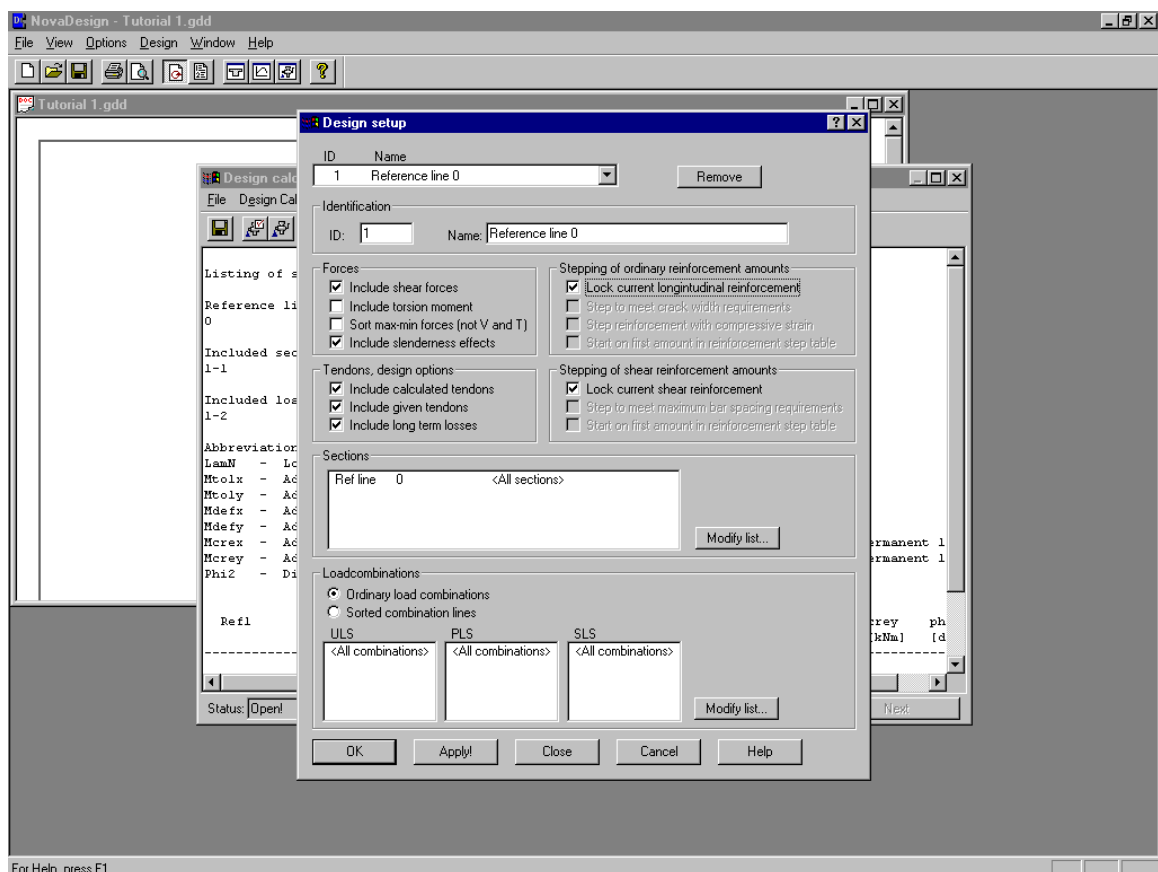


Fig. 13.18.

In a design calculation setup you select which cross sections and load combinations to include in a design calculation. You also select which calculation options that should be activated (this is further explained in section 11.2).

Now create a new setup following the steps below:

- Set setup ID to '3' and setup name to, for example, 'My setup'
- Uncheck the 'Lock current longitudinal reinforcement' option in order to allow automatic stepping of reinforcement amounts.
- Check the 'Step to meet crack width requirements' option.
- Leave the selections within the 'Sections' frame and 'Load combination' frame as they are
- Press the 'Apply!' button

As you can see, the option 'Include slenderness effects' is switched on as default

The dialog should now look like this:

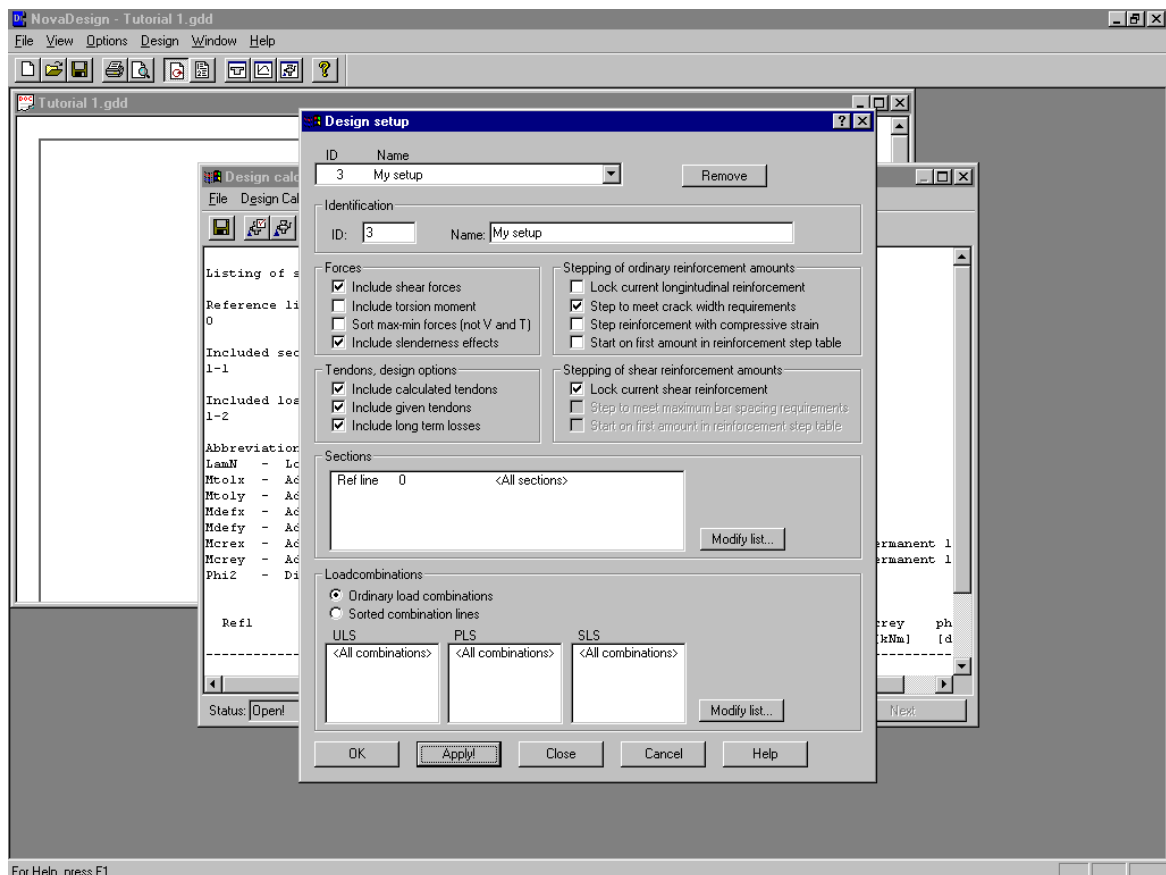


Fig. 13.19

The text in the list in the 'Sections' frame can be somewhat mysterious, if you are not yet familiar with the reference line concept in NovaDesign and NovaFrame. However, your cross section, 'Column', is connected to reference line 0, and is therefore included in 'Ref line 0 <All sections>'. If you wish to learn more about reference lines, see Appendix 2 'Using Reference Lines'.

- Press the 'OK' button to close the dialog window.

Now run design calculations using the setup you have just created:

- Select **Design Calculation – Run...** from the menu to open the *Calculate* dialog.

Tool button: .

- Select setup number 3, '*My setup*'. See figure below.

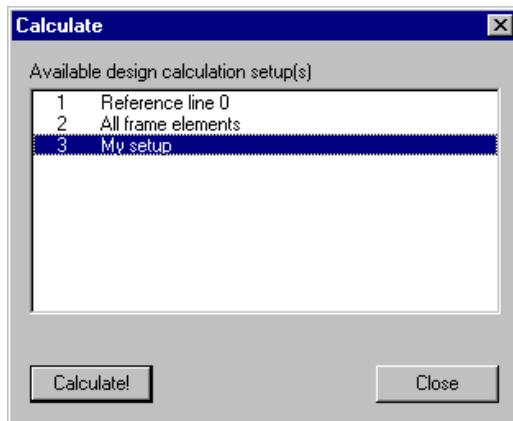



Fig. 13.20

- Press the '*Calculate*' button.

When calculation is completed, list axial and bending summary. As you can see, the calculation has converged and the maximum UR is now 97 %.

List detailed reinforcement results to see the selected reinforcement. Tool button 

The column 'Type' indicates the selected reinforcement type (amount) for each reinforcement group. As you can see, the program has selected type 126 for reinforcement group 1. Type 126 equals '1 ϕ 20c150'.

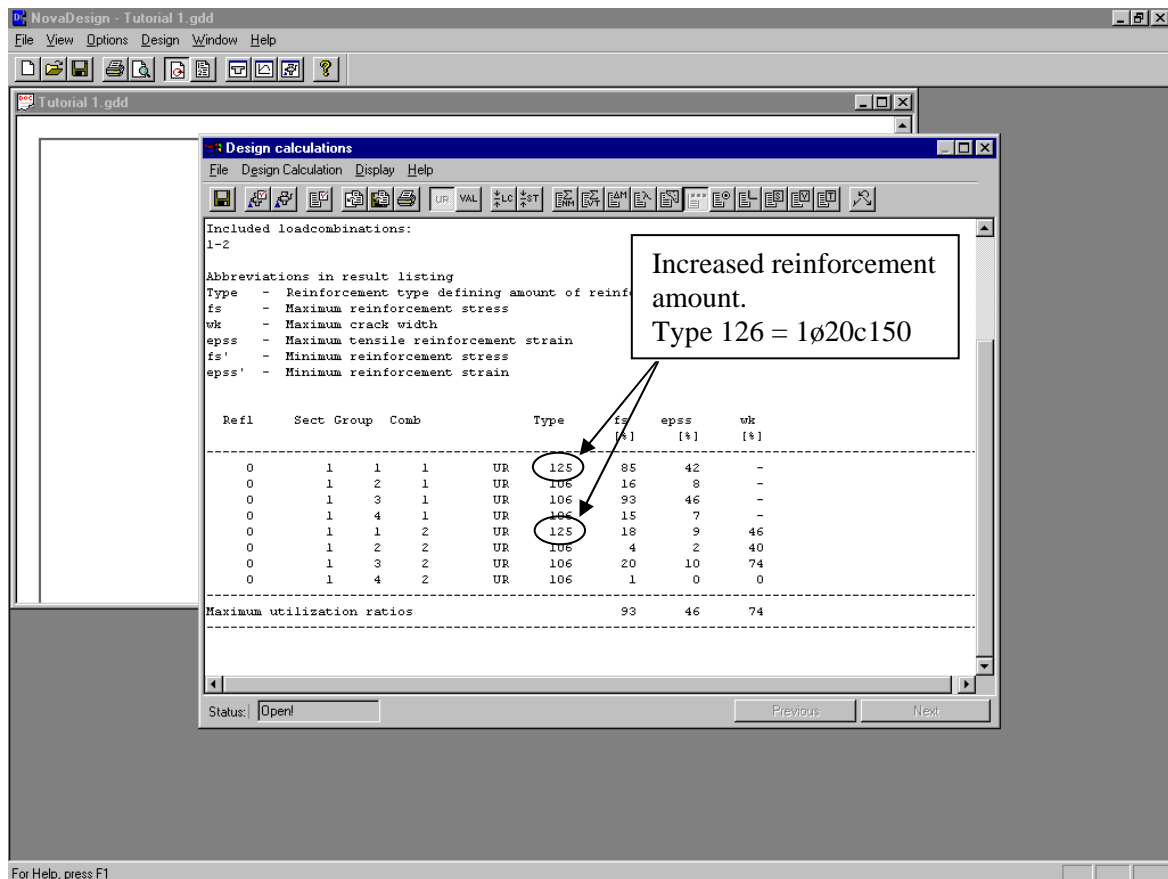



Fig. 13.21

Also, list slenderness results to see that that additional moments due to deflection are much smaller.

Now, close the **Design Calculation** window. It is possible to see that the rebars at the lower face of the cross section is slightly larger than the rest. To display information about bar diameter open the **Attribute** dialog

- Select **Display – Attributes...** from the menu in the **Cross section design** window.
- Check '*Reinf. diam.*' to show bar diameter for reinforcement
- Press the '*Specify*' button to the right of this check box.
- Select '*Do not apply zooming on attribute*'
- Click '*OK*'
- Click '*Close*' in the **Attribute** dialog

Bar diameter is now shown in the cross section plot. The numbers can be difficult to read because they are drawn on top of each other. Zoom in on the lower left corner of the section to make the text easier to read.

- Click the 'Zoom area' tool button .
- Point cursor in the plot window
- Click left mouse button - drag a zooming area – release mouse button

The dialog window should now look something like this:

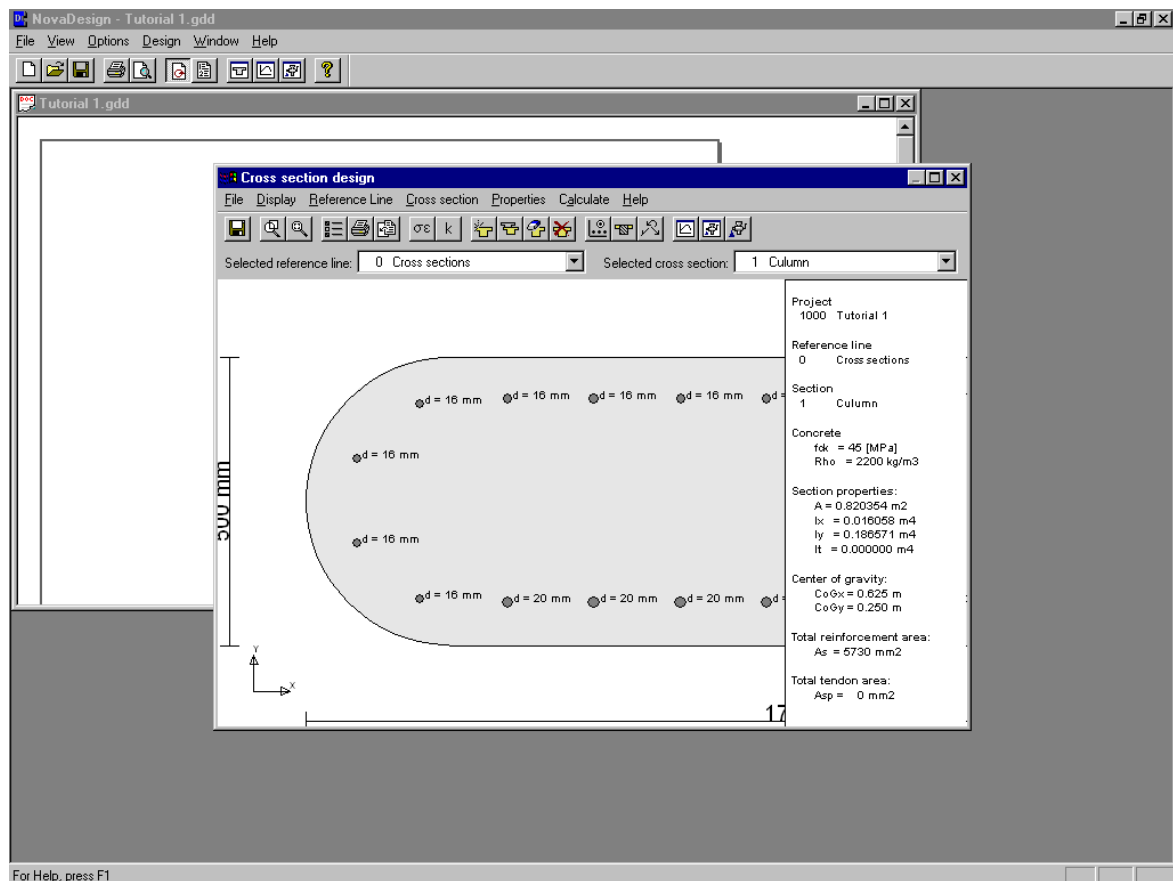


Fig. 13.22

This completes step B of this tutorial.

There are many subjects that are not included in this tutorial, like;

- Capacity charts
- Shear and torsion calculations
- Tendons
- Creating a design report
- Interaction with NovaFrame.

More information about these, and other subjects is available in this User's Guide, in the NovaFrame User's Guide and in the different appendices.