

4C-Temp&Stress

Ver. 2.0 for Windows

User Manual

***DTI Building Technology
March 1998***

DTI Building Technology - The Concrete Centre - Gregersensvej, P.O.Box 141, DK-2630 Taastrup
Telephone 43 50 41 50 - Telefax 43 50 40 69

Contents

1.	Welcome to 4C-Temp&Stress	1-1
2.	Applications of 4C-Temp&Stress	2-1
2.1	Temperature and Stress Simulation in Hardening Concrete Structures	2-2
3.	Hardware and Software	3-1
3.1	Hardware and Software Requirements	3-1
3.2	System Effect	3-1
3.3	Limitations	3-2
3.4	Installation	3-3
4.	4C-Temp&Stress and Windows	4-1
4.1	How to Start 4C-Temp&Stress	4-1
4.2	Mouse Handling	4-1
4.3	Memory	4-1
5.	4C-Temp&Stress Fundamentals and Features	5-1
5.1	Program Structure	5-1
5.2	What is a Project?	5-2
5.3	Project Files	5-2
6.	Project Editor	6-1
6.1.	General	6-1
6.2	Geometry Mode	6-2
6.2.1	View Options	6-2
6.2.1.1	View Settings	6-2
6.2.1.2	Scale Font	6-3
6.2.1.3	View Buttons	6-3
6.2.2	How to Draw Volumes	6-3
6.2.3	How to Edit Volumes and C-pipes/H-wires	6-5
6.2.4	How to Draw More than One Volume	6-8
6.3	Thermal Mode	6-9
6.3.1	Volumes	6-9
6.3.2	Surfaces and Thermal Boundary Conditions	6-13
6.3.2.1	Thermal Boundary Conditions	6-15
6.3.3	Cooling Pipes / Heating Wires	6-23
6.4	Load Mode	6-25
6.4.1	Supports	6-25
6.4.1.1	Definition	6-26
6.4.1.2	How to Add and Edit a Support	6-26
6.4.1.3	Examples	6-28
6.4.2	Self-Weight	6-30
6.4.3	Point Loads	6-30
6.4.3.1	Definition	6-30
6.4.3.2	How to Add and Edit a Point Load	6-32
6.4.4	Line Loads	6-34
6.4.4.1	Definition	6-34
6.4.4.2	How to Add and Edit a Line Load	6-34

6.4.5	Momentum	6-35
6.4.5.1	Definition	6-35
6.4.5.2	How to Add and Edit a Momentum	6-36
6.4.6	Examples	6-37
6.5	Text Mode	6-39
6.5.1	General	6-39
6.5.2	Text on Arrow	6-40
6.5.3	Text in Box	6-42
6.6	Calculation Parameters	6-43
6.6.1	General	6-43
6.6.2	Thermal Analysis	6-44
6.6.3	Stress Analysis	6-44
6.6.4	Dimensions	6-45
6.6.5	Time Specifications	6-45
6.6.6	Mesh Node Generation	6-46
6.6.7	Non-Linear Calculations	6-47
6.6.8	Circles	6-48
6.6.9	Self Weight	6-48
6.7	Project Items	6-49
6.7.1	New Project	6-49
6.7.2	Load Project	6-50
6.7.3	Save Project	6-50
6.7.4	Save Project As	6-51
6.7.5	Setup Project	6-51
6.7.6	Print Project	6-53
7.	Material Database	7-1
7.1	Database Structure	7-1
7.2	Concrete Database	7-2
7.2.1	Mix Proportions	7-2
7.2.2	Properties	7-4
7.2.3	Heat	7-6
7.2.4	E-Modulus	7-9
7.2.5	Poisson Ratio	7-11
7.2.6	Thermal expansion coefficient	7-13
7.2.7	Compressive Strength	7-15
7.2.8	Tensile Strength	7-16
7.2.9	Initial Strain	7-18
7.2.10	Creep	7-19
7.3	Concrete Constituents	7-21
7.4	Materials	7-23
7.5	Soil	7-25
7.6	Insulation	7-25
8.	Project Solver	8-1
8.1.	Generate Finite Element Mesh	8-1
8.2.	Calculate Temperatures	8-1
8.3	Calculate Stresses	8-2
9.	Result Viewer	9-1
9.1	General	9-1
9.2	Load and Print Results	9-2

9.3	Geometry Mode	9-4
9.3.1	Settings	9-5
9.3.2	Points	9-6
9.3.3	Volumes	9-7
9.3.4	Geometry System	9-8
9.4	Diagram Mode	9-9
9.4.1	Temperature Analysis	9-9
9.4.2	Maturity Analysis	9-14
9.4.3	Strength Analysis	9-15
9.4.4	Approximated Stress Analysis	9-17
9.4.5	Stress Analysis	9-18
9.4.6	Diagram System	9-20
9.4.7	Saving in datafile	9-23
9.5	Isocurve Mode	9-24
9.5.1	Temperature Analysis	9-24
9.5.2	Maturity Analysis	9-26
9.5.3	Strength Analysis	9-27
9.5.4	Approximated Stress Analysis	9-29
9.5.5	Stress Analysis	9-30
9.5.6	Isocurve System	9-32
9.6	Windows View	9-33
10.	Tutorials	10-1
10.1	Temperature calculation	10-1
10.2	Temperature and approximated stress calculation	10-3
10.3	Temperature and stress calculation	10-6

1. Welcome to 4C-Temp&Stress

Welcome to 4C-Temp&Stress*, the software that handles thermal, maturity and stress analysis in concrete structures. 4C-Temp&Stress provides the ability to outline your projects, carry out full or approximated calculations and view the results in a graphical interface that provides an intuitive and efficient work environment.

This manual will help you understand the basic elements in 4C-Temp&Stress and how to use the program. It contains both technical information and step by step instructions to help you describe your projects. The following items tell you where to find information in the manual.

Applications of 4C-Temp&Stress

Section 2, *Applications of 4C-Temp&Stress*, describes the practical applications of 4C-Temp&Stress to analyse concrete curing.

Hardware & Software

For information about hardware and software requirements, system effect and how to install and start 4C-Temp&Stress, please see section 3, *Hardware & software*, and section 4, *4C-Temp&Stress and Windows 3.1*.

4C-Temp&Stress Fundamentals and Features

Section 5, *4C-Temp&Stress fundamentals and features* provides a view over fundamentals and features in 4C-Temp&Stress. Each module and its features is introduced in this section.

The Project editor

How to define all parts of a project - like geometry and thermal and static boundary conditions - is described in section 6, *Project Editor*, and 7, *Material Database*.

The Project solver

How to activate the calculations is described in section 8, *Project Solver*.

Result Viewer

How to present the calculated results is described in section 9, *Result viewer*.

Tutorials

A general view of how to define projects in order to make temperature calculations, temperature and approximated stress calculations or temperature and stress calculations respectively is given in section 10, *Tutorials*.

* 4C-Temp&Stress: Concrete, Control, Casting, Curing - Temperature and Stress calculations.

2. Applications of 4C-Temp&Stress

4C-Temp&Stress is a computer program designed to determine how temperatures and stresses vary in a two-dimensional polygonal area (with or without holes).

Perpendicular to the two dimensional area arbitrary plane strain and plane stress situations can be handled. The material properties are constant or develop as a function of maturity in hardening concrete structures.

The following analyses can be performed:

Thermal analysis: Determination of the temperatures as a function of place and time (transient analysis).

Thermal analysis is based on:

- thermal boundary conditions such as external temperature, insulation, cooling pipes, heating wires etc.
- internal heat generation by the materials
- in the case of hardening concrete structures: casting program.

The thermal boundary conditions can be changed according to a prescribed program. This will occur in hardening concrete structures during form work removal and when cooling pipes are switched on or off.

Stress analysis: Determination of the stresses as a function of place and time (transient analysis).

Stress analysis is based on:

- external loads and supports
- thermal expansion obtained by a previous thermal analysis
- how stiffness and strength properties develop as a function of maturity, where the maturity is found from the thermal analysis
- creep and relaxation of materials.

The external loads can be described as functions of time.

Typically, 4C-Temp&Stress can be used in analysing:

- linear-elastic disks in a plane stress or plane strain situation.
- hardening and restrained concrete structures. A further discussion is found in section 2.1.

All computations are based on the finite element method. 4C-Temp&Stress performs automatically all the operations necessary to transform the physical problem into a formulation that can be handled by the finite element method. For example, the user specified geometry is automatically divided into triangular plane elements. These elements are able to describe a quadratic variation of temperature and stress.

2.1 Temperature and Stress Simulation in Hardening Concrete Structures

Introduction

During the hardening process of concrete energy is released as heat. Due to temperature differences the concrete will try to expand or contract. If the concrete surface is colder than the interior of the concrete structure, the contraction at the surface may cause stress larger than the tensile strength and cracks are formed. This situation will typically occur at the maximum temperature difference between the surface and the interior.

When casting takes place upon an existing concrete structure the relative stiffness will have an influence on whether or not cracks are formed during the hardening process. Crack formation caused by the presence of an adjacent concrete structure will typically take place when the newly cast concrete is in the cooling phase, i.e. after the generation of heat of hydration has stopped.

General

The above mentioned phenomena may occur when a wall is cast upon an existing concrete structure (see figure 2.1).

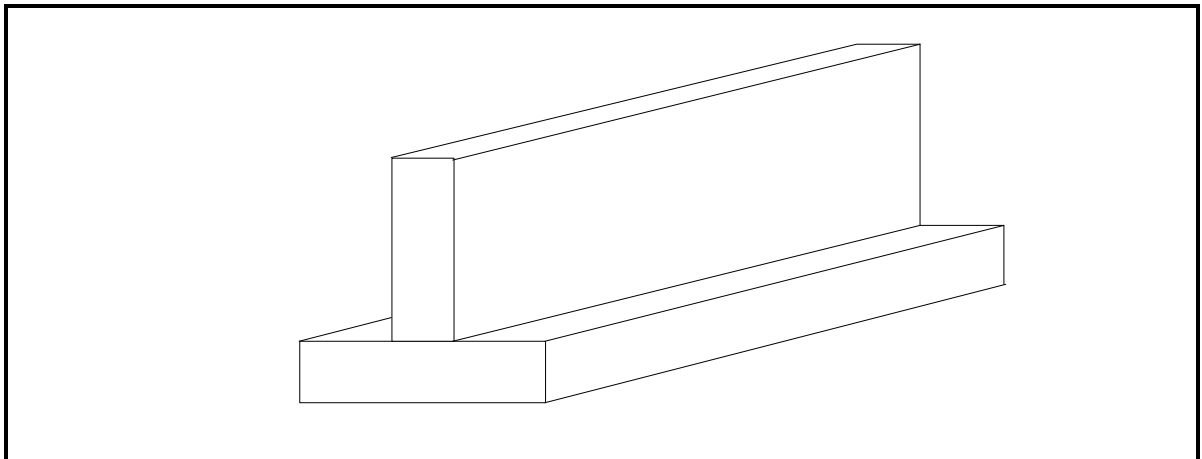


Figure 2.1

Wall cast on existing deck.

The phenomena may also occur in a number of other situations, but the wall/deck example is well-suited for the demonstration of the mechanisms involved.

On basis of the heat of hydration in the hardening concrete and the thermal boundary conditions of the structure (type of form work, insulation, outdoor temperature etc.) the temperature distribution is calculated as a function of time. In a structure similar to the one shown in figure 2.1 the temperature distribution will not change from one cross section to another (except close to the end surfaces). An example of such a temperature distribution calculated with 4C-Temp&Stress is shown in figure 2.2. From figure 2.2 it is seen that the deck cools the bottom part of the wall and that the deck itself is heated.

When the wall is exposed to a temperature rise it will try to expand in all directions. As shown schematically in figure 2.3a it is not possible for the wall to expand freely in the longitudinal direction because it is restrained by the deck.

This means that the wall and the deck are subject to compressive and tensile stresses respectively acting perpendicular to the cross section. In the cross section plane stresses are caused by temperature differences (see figure 2.3b). Due to the heat of hydration a 3-dimensional stress situation is created irrespective of the temperature distribution being 2- or 3-dimensional.

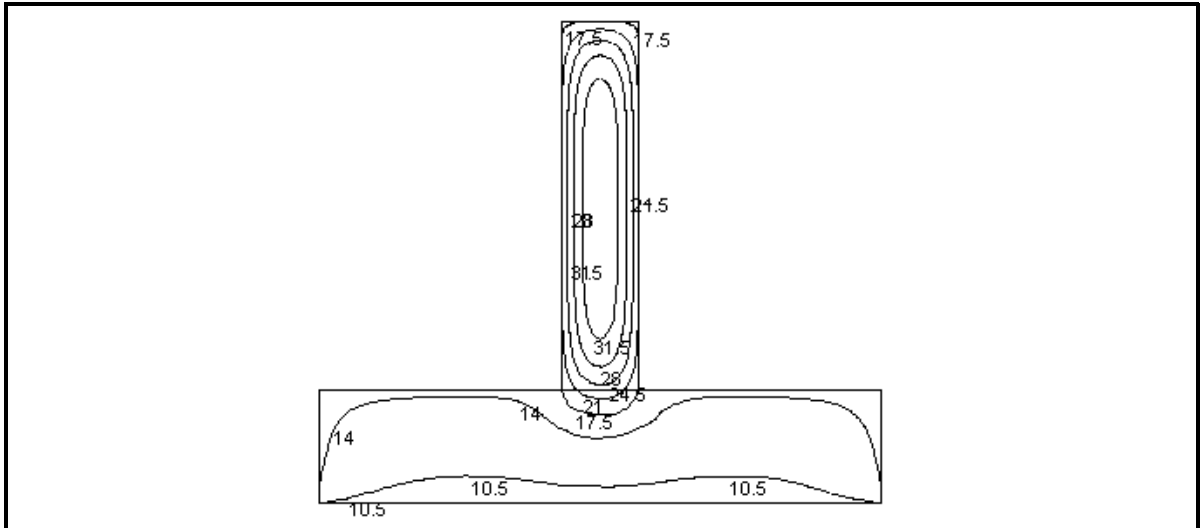


Figure 2.2
Isotherms in the hot wall and the cold deck.

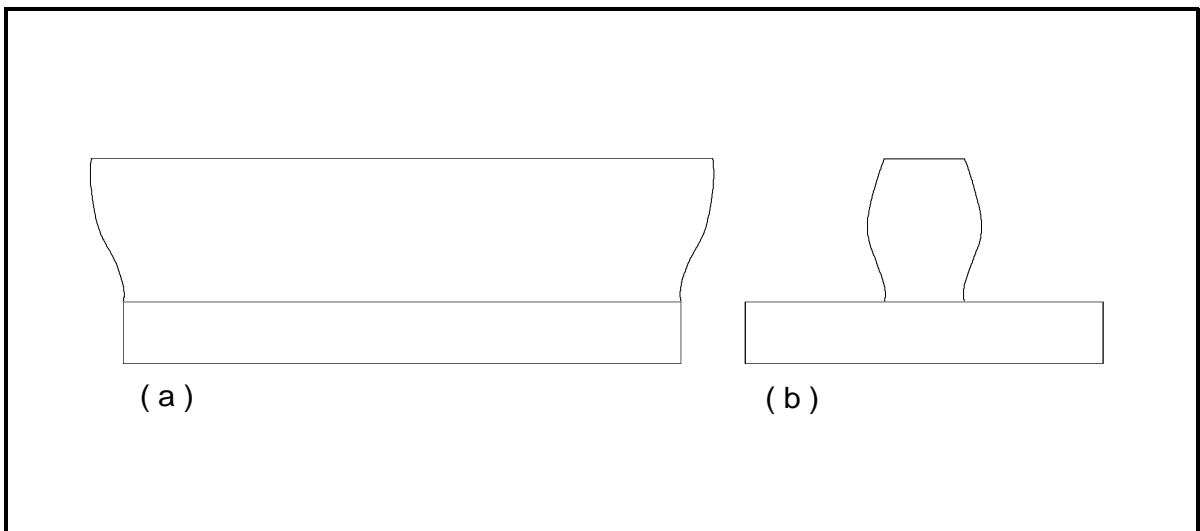


Figure 2.3a - b
Temperature induced deformations.

In order to determine the stress situation and with that the cracking risk it is necessary to know the modulus of elasticity, the creep and relaxation and the tensile strength of the hardening concrete as a function of maturity.

Due to the change in e.g. the modulus of elasticity in the concrete during the hardening period, the stress analysis requires a considerable computational effort. Although a 3-dimensional stress distribution exists, frequently it turns out that this stress distribution is constant from one cross section to the next when moving in the longitudinal direction of the wall/deck structure.

This is the case in the wall/deck situation when a given cross section is situated at such a distance from the ends that these do not influence the stress distribution in the observed cross section. In the end zones the stress situation is changing due to the free movement of the end surfaces (see figure 2.3a).

In most cases this will cause a reduction of the stress level in the end zones. Therefore, the stress distribution at a certain distance from the end zones will be governing.

The use of 4C-Temp&Stress makes it possible to perform the following calculations for hardening concrete structures:

- temperature distribution
- stress distribution
- distribution of the tensile stress/strength ratio.

Thermal Analysis

In the calculation of the temperature distribution it is possible to specify:

- the heat of hydration of the concrete
- thermal boundary conditions (insulation, heating mats, cooling pipes etc.)
- casting schedule (casting time, casting temperature).

The thermal boundary conditions can be specified as functions of time corresponding to form work removal, start/stop of cooling, etc.).

Stress Analysis

As previously stated the temperature development causes a 3-dimensional stress distribution. If it is assumed that the stress distribution is constant in the longitudinal direction the problem can be treated as a disk (see figure 2.4a), where a plane elastic strain distribution originating from the deformation is in equilibrium with the shear forces caused by the temperature induced stress (see figures 4b 1-2-3).

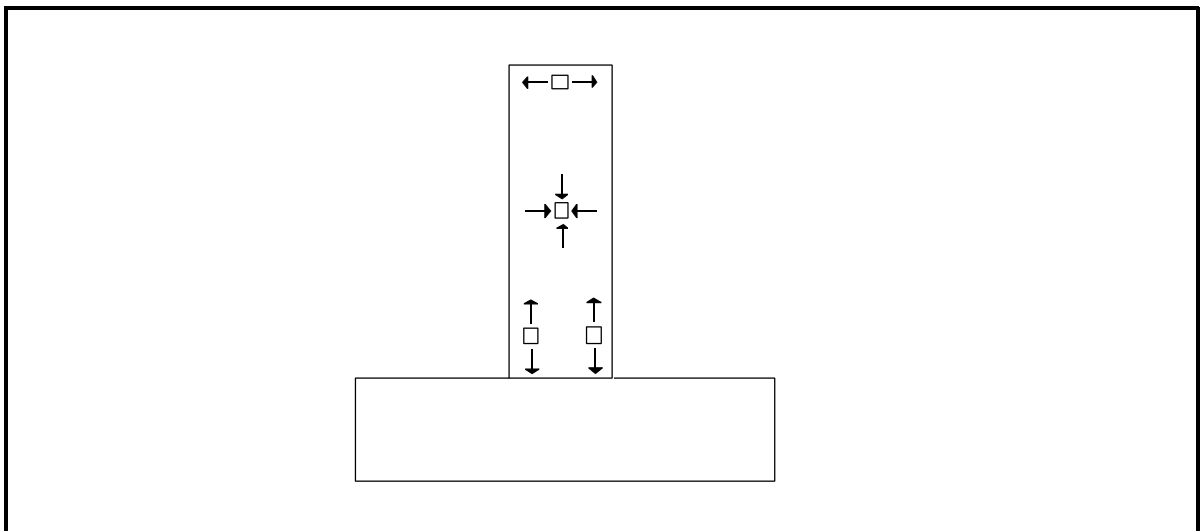


Figure 2.4a
Disk forces.

Figure 2.4a shows how the disk forces are generated due to the temperature differences. Figure 2.4b-1 shows the stress developed if the structure is totally restrained so that no displacements take place in the longitudinal direction. The stress distribution depends directly on the temperature and the stiffness of the concrete (both are functions of time).

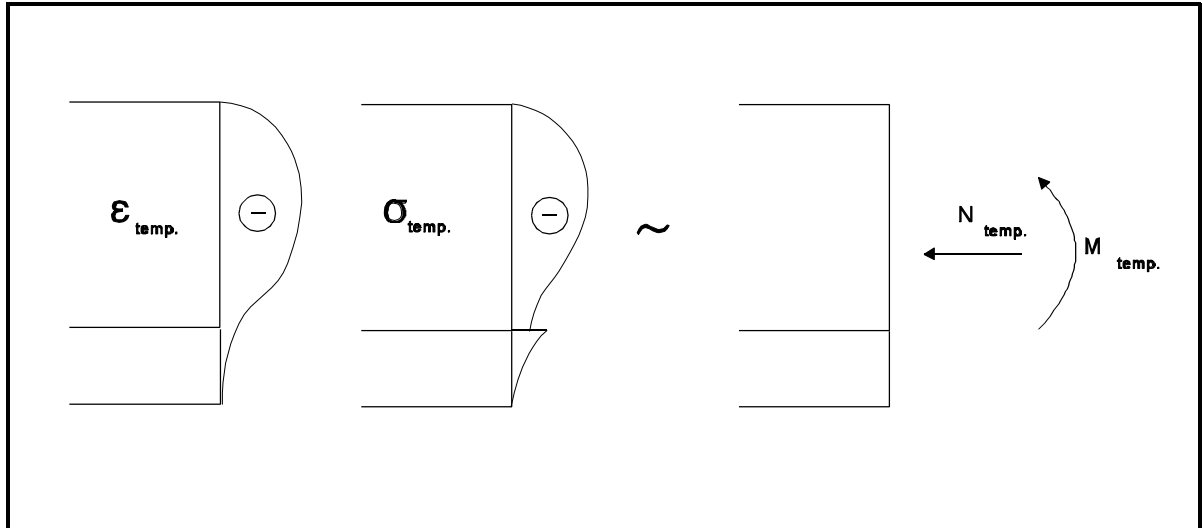


Figure 2.4b-1

Temperature induced shear forces in a totally restrained cross section.

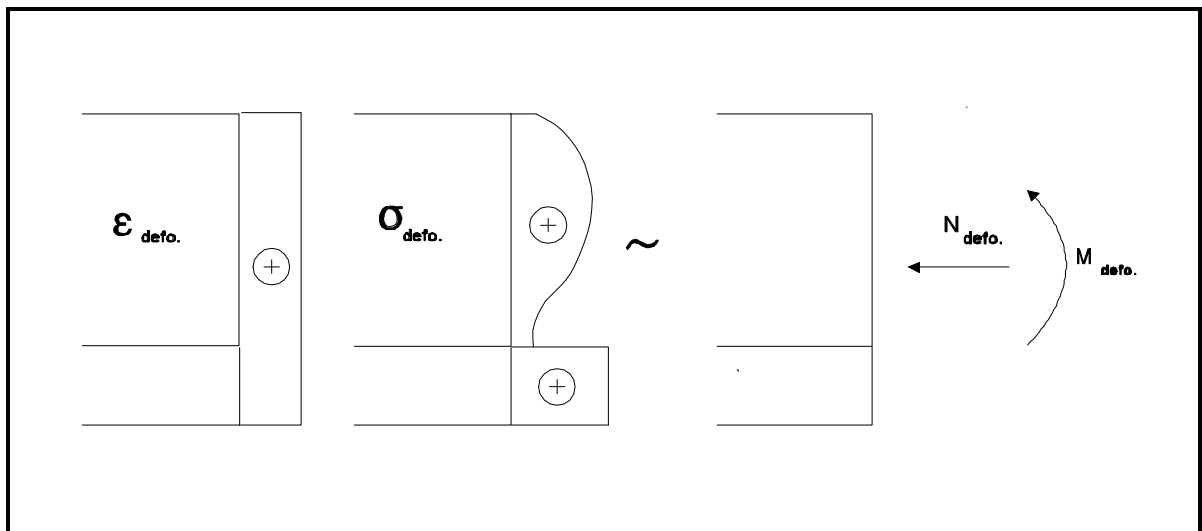


Figure 2.4b-2

Shear forces originating from deformation in cross section with free longitudinal deformation and fixed curvature.

Figures 4b-2 and 4b-3 show the stress distribution caused by the deformation when the cross section is only restrained against curvature (this is for instance the case with long structures on soil or foundation) and when the cross section deforms freely (for instance in the case of short structures).

Equilibrium is obtained when:

$$N_{temp} + N_{defo} = 0$$

and

$$M_{temp} + M_{defo} = 0$$

The stress calculation is based upon:

- the results from the thermal analysis
- the development of stiffness, strength, creep and relaxation properties as a function of maturity
- static and geometrical boundary conditions (load, support conditions etc.).

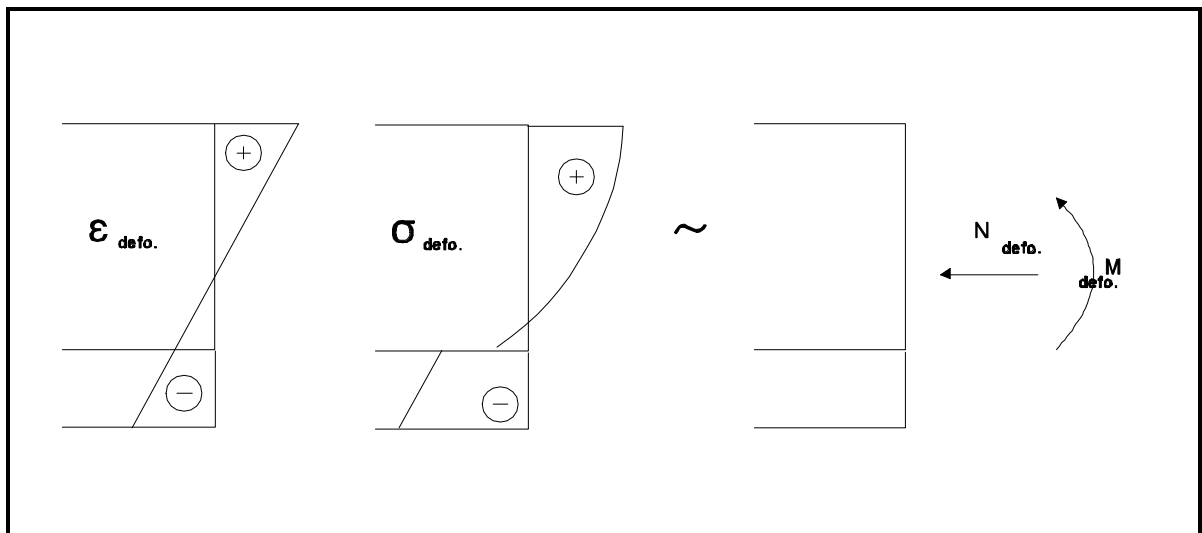


Figure 2.4b-3

Shear forces originating from deformation in free cross section.

The static loads can be specified as functions of time. Loads coplanar with the area can be single forces and trapezoidal line loads acting on the boundary. The loads which are related to the plane strain are a normal force and the components of two bending moments. The forces may for instance originate from the tensioning of tendons or from dead load of the structure.

In addition to the calculation of the stress distribution as a function of time also the degree of utilization of the tensile strength is calculated as a function of time.

Stress Approximation

As stated in the preceding paragraph the stress analysis can be treated similar to the disk calculation of the cross section of the structure with a plane elastic strain distribution acting perpendicular to the disk. If it is assumed that the Poisson ratio is equal to zero, the analysis can be divided into two separate parts. Given these conditions the first part determining the stress distribution perpendicular to the cross section is very simple and is performed together with the temperature calculation.

Because the computational work required by the general stress calculation is rather heavy, the approximate stress calculation is a short cut to obtain an indication of whether or not the general stress calculation will yield an acceptable result.

The approximate stress distribution will indicate primarily whether there is a risk of crack formation due to the presence of adjacent structures with restraining effect.

3. Hardware and Software

This section provides a system guide to 4C-Temp&Stress, which covers hardware & software requirements, the effect of the system configuration and how to install 4C-Temp&Stress.

3.1 Hardware and Software Requirements

Minimum system configuration

Hardware:	PC	486DX 33 MHz
	RAM	8 Mb *
	HDD 4C-Temp&Stress system files	2.5 Mb
	HDD Win32S system files	2.5 Mb
	HDD project files	20 Mb
	Drive	3½" or 5¼"
	Display adapter	VGA
	Display	VGA
	Mouse	IBM compatible

* For temperature calculations 4 Mb RAM may be sufficient.

Software: DOS 5.0 and Windows 3.1 (or later releases)

Recommended system configuration

Hardware:	PC	Pentium
	RAM	8-16 Mb
	HDD 4C-Temp&Stress system files	2.5 Mb
	HDD Win32S system files	2.5 Mb
	HDD project files	Min. 100 Mb
	Drive	3½" or 5¼"
	Display adapter	SVGA
	Display	SVGA
	Mouse	IBM compatible
	Printer	Laser Printer (PostScript)

Software: DOS 6.2 and Windows 3.11 (or later releases)
or Windows NT 3.5 (or later release) or Windows 95

3.2 System Effect

The calculation time depends on the following items

- Windows system resources (see the Windows User guide "Optimizing Windows")
- The project size. Especially the Calculation parameters specified by the user (see the section 6.6 about the calculation parameters).

3.3 Limitations

There are certain limitations in 4C-Temp&Stress about task-size. In the current version

96000 integers
32000 floats

can be utilized in the formulation of a task.

Furthermore 200000 integers are used for the bandwidth-optimization of the equations. The maximum number of nodes which can be handled is 9998. Nodes are found at the corners, in the middle of the edges and in the centre of the element. All these limitation can be adapted, if necessary.

Thermal and stress analyses require a large amount of memory. The memory is used for equilibrium equations.

Figure 3.1 shows a structure which needs about 12 Mb internal memory (RAM). Tasks which need more than 16 Mb RAM have to be executed in a 32 bit WINDOWS-NT environment. Figure 3.2 shows a structure which needs about 32 Mb RAM.

It is recommended to command an amount of memory which can hold the operating system, 4C-Temp&Stress-computer program and the above mentioned equilibrium equations. Otherwise the computer program uses the harddisk as internal memory and this will increase the time of calculation by 100-200 times. The structure in Figure 3.1 can only just be handled in a 16 Mb internal memory. The calculation times are 2 hours and 8 hours respectively for thermal and stress analyses (PC-486, 66 Mhz).

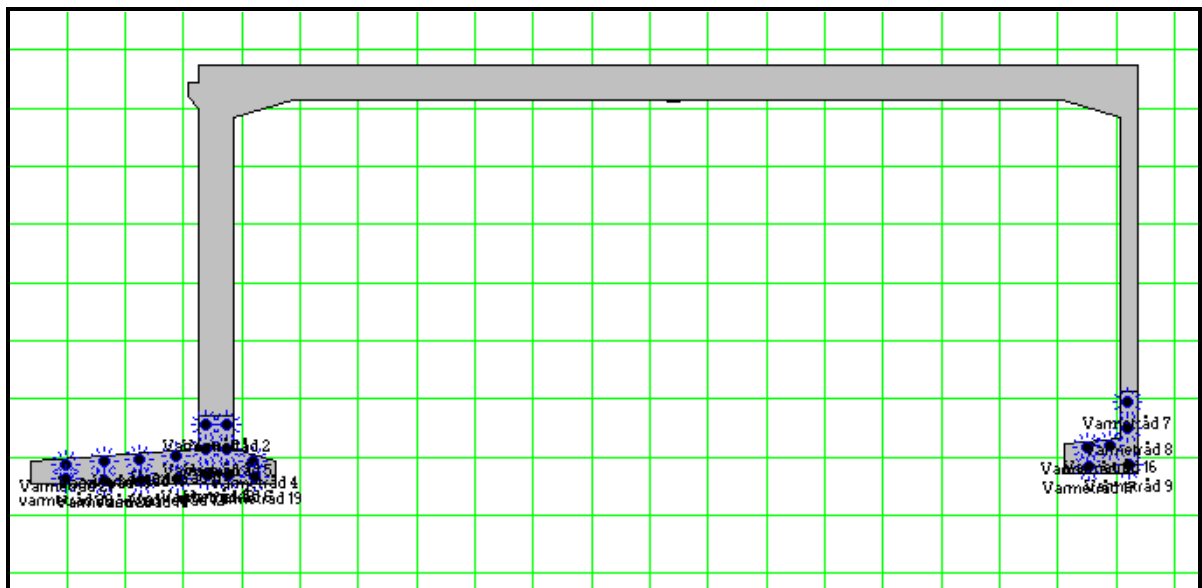


Figure 3.1

Structure which needs about 12 Mb RAM. The cross-section has a width of app. 16 m and contains 22 heating cables.

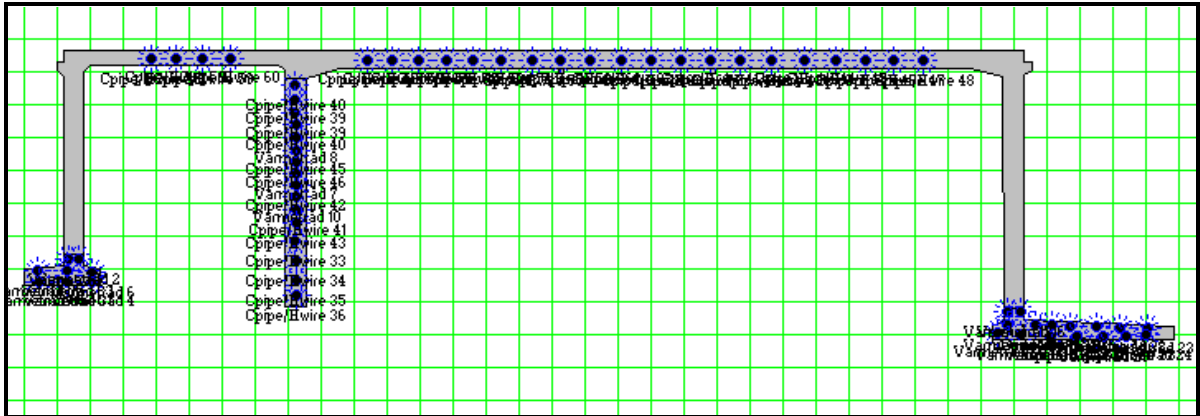


Figure 3.2

Structure with needs of about 32 Mb RAM. The cross-section has a width of app. 29 m and contains 64 heating cables.

3.4 Installation

The installation includes creating the 4C-Temp&Stress directories and copying the system files to the appropriate directories, using the supplied installation program.

Place the 4C-Temp&Stress disk No. 1 in the high density diskette drive of your computer.

In Windows run the SETUP program on that drive. You may do this by typing A:SETUP under the File Run menu in the Program Manager. If running Windows 95 or Windows NT 4.0 select the Start-Run menu item.

The installation program makes a new program group for 4C-Temp&Stress (Default is DTI Applications).

Before starting the program the security key must be placed in one of the parallel ports of the computer. If a printer is connected to the used parallel port the security key must be placed between the computer and the printer and the printer has to be **turned on**.

You can now start 4C-Temp&Stress by clicking the program icon with the DTI logo and the title 4C.

The first time 4C-Temp&Stress is started a warning appears saying that the file DEFAULT.SET is missing. Click the **OK** button to continue. The message can be avoided in future runnings by creating DEFAULT.SET as described in section 6.7.1.

The first time you run 4C-Temp&Stress the system information must be updated in order to make your company name appear on the print-outs.

Place the disk named System information in the high density disk drive of your computer.

Select the menu item [Project] and the sub item [Update system information]. Double click on the relevant drive in the list (normally [a]). Click the OK button.

NB! If you run into problems while working with Windows and 4C-Temp&Stress and want to install 4C-Temp&Stress again remember to backup the database files, otherwise the installation will overwrite the database files that include your concrete records, cement records etc.

4. 4C-Temp&Stress and Windows

This section gives a short introduction of how 4C-Temp&Stress is working in the Windows environment, so that you will be able to start 4C-Temp&Stress from Windows, and how Windows and 4C-Temp&Stress handle mouse movement/click and also how to optimize Windows, if 4C-Temp&Stress needs more memory. For more specific information about the Windows environment and how to optimize Windows see the Windows User Guide.

4.1 How to Start 4C-Temp&Stress

You start 4C-Temp&Stress in Windows 3.1x as follows:

- Double click the 4C-Temp&Stress program icon, or
- Click the 4C-Temp&Stress program icon and press the [Enter] key on the keyboard.

You start 4C-Temp&Stress in Windows 95 and Windows NT as follows:

- Select the Start-Programs-DTI Applications menu
- Select the 4C-Temp&Stress program item.

4.2 Mouse Handling

4C-Temp&Stress is based on mouse handling to make the handling of objects (*volumes* like a base, a wall etc.) and Functions (*toolbar start function buttons* like start Result Viewer or start Generate finite element mesh) fast and easy. The menus and windows can be activated through the mouse or keyboard, like all other Windows application of course. How to draw and define objects or select tool bar buttons, see the sections 6.1 and 6.2. For more general information about mouse- and keyboard handling, see the Windows User Guide section Windows Basic, Basic Mouse and Keyboard Techniques.

4.3 Memory

4C-Temp&Stress is a true Windows application that takes full advantage of the memory features that Windows 3.1 provides. 4C-Temp&Stress requires (recommends) 8-16 Mb free memory to carry out stress calculations depending on the complexity of the project. The more memory in the system - the more complex projects and "equations-heavy" stress calculations can be performed! See section 3.3 for information about the limitations. With the recommended system configuration and the recommended Windows setup there will be no problems performing the temperature and approximated stress calculation on a medium-sized project. If the project is getting large and complex, it might be necessary to optimize the system by increasing the physical amount of memory on the PC. This requires technical assistance.

NB! The window in 4C-Temp&Stress for the temperature and stress calculation shows the required amount of memory in kb (buffersize: equations, temperature and stress) to perform the current calculation. You can compare the required memory for the equations with the installed memory on you PC and remember to deduct the memory that Windows requires itself.

5. 4C-Temp&Stress Fundamentals and Features

This section outlines a view over some of the most important fundamentals that will be used in this manual and describes the features in 4C-Temp&Stress. The features in 4C-Temp&Stress are identified by the three main modules based on the three main tasks in handling temperature, maturity and stress analysis in concrete constructions. The three main tasks and the corresponding modules are *Specifying the Project*, (Project Editor), *Do the Calculations*, (Project Solver) and *View the Results*, (Result Viewer).

5.1 Program Structure

The following table provides a short introduction of each module in 4C-Temp&Stress and shows the abbreviations for each module, that will be used in the rest of this manual.

Modules	Abbreviations	Introduction
Project Editor	PE	<p>The PE is the place where you outline and edit your projects and the concrete database.</p> <p>To outline a project includes drawing the construction (volumes), placing and specifying the cooling pipes, heating wires, boundary conditions, loads and supports. Besides specifying the calculation parameters and activating the calculations. Projects will be stored on the disk in a Project Database, where you can load old projects - edit and rename them - as you wish.</p> <p>The PE is also the place where you handle the concrete database and all the mix-material databases. The databases will be available whenever you enter the PE no matter which project you are outlining.</p>
Project Solver	PS	<p>The PS is the module that takes care of project calculations like Generating the Finite Element Mesh (FEM) and calculating the temperature, maturity, approximated stress and stress developments.</p> <p>Before any calculation of temperature, maturity or stress developments, a generation of the FEM is required. The FEM will be generated on basis of the specified calculation parameters which are specified under the menu item Calculation in the PE module.</p> <p>After generating the FEM calculation of temperature, maturity or stress developments can be executed. All input calculation parameters required for one of these calculations are specified under the menu item Calculation in the PE module.</p> <p>The calculations are executed inside the PE module. This makes it fast and easy to edit calculation parameters and redo a calculation.</p>

Result Viewer	RV	<p>The RV is the module that takes care of displaying the results in either an x/y-diagram or an isocurve view. Results generated by the PS are either temperature, maturity, strength, approximated stress or stress as a function of time and place.</p> <p>All these results can be presented in an x/y-diagram showing the results as a function of time. The diagram can contain either the average values for each cast, the min. and max. values for each cast or the results in specified geometry points.</p> <p>All the calculated results can also be presented as isocurves showing the values in the cross section at a specified time after casting.</p>
---------------	----	--

5.2 What is a Project?

A project covers all the information that is necessary to carry out temperature- or stress calculations and it is divided into the following subjects:

- **Calculation parameters** e.g. Total time process, Time step desired etc.
- **Volumes** e.g. Dimensions, Material type, Start temperature etc.
- **Cooling pipes** e.g. Faces, Diameter, Temperature function etc.
- **Heating wires** e.g. Faces, Diameter, Flux function etc.
- **Surfaces** e.g. Trans. coef. functions, Shield def. etc.
- **Loads** e.g. Factor, Angle (x-axis), Load function etc.
- **Momentum** e.g. Factor, Momentum function etc.
- **Supports** e.g. Point and Line, Fixed displacement etc.
- **Texts** e.g. Comments to the structure
- **View** e.g. Grid size, Scaling mode, Zoom etc.

When you have outlined your project you can save the project in the project database. The project database covers all the projects that you have created and saved on the disk in the directory C:\4C\DAT or another valid directory. Each project in the project database has the extension .prj. You can then carry out the calculations and view the results. For more specific information about *how to outline a project* see section 6 and 7, *how to carry out calculations*, see section 8 and finally *how to view results*, see section 9.

4C-Temp&Stress will be in the Project Editor module at starting point. From inside the PE you can switch to either the project database by clicking the menu item [Project], the material database by clicking the menu item [Database] or the Result Viewer module by clicking the appropriate toolbar button or the menu item [Project]. The PS is activated when you start a calculation by clicking either the appropriate toolbar buttons or menu items. For more information about each program module see the following sections. The project database is introduced in the PE section.

5.3 Project Files

All the information about and results of a project are saved in files with the same project name but with different suffixes. In the following list the content of the project files with different suffixes are described:

- .PRJ The file contains all the input data which you have defined in the PE. This file is created when you have saved the project.

.TXT	The file contains all the information used in the calculations. This file is created when you have generated the finite element mesh.
.JOB / .PRO	The files contains the information about the project name, date etc.
.EXP	The file contains the information about the project which is used in the RV including the generated FEM. This file is created when you have generated the finite element mesh.
.TEM	The file contains all the calculated temperature results. This file is created when the temperatures have been calculated.
.MAT	The file contains all the calculated maturity results. This file is created when the temperatures have been calculated.
.SST	The file contains all the calculated approximated stress results. This file is created when 2½ dimension is selected and the temperatures have been calculated.
.STR	The file contains all the calculated stress results. This file is created when the stresses have been calculated.
.DIS	The file contains all the calculated node displacements. There are no access to these from 4C-Temp&Stress. This file is created when the stresses have been calculated.
.DMP	The file contains information about the calculation in ASCII format. After calculation of approximated stress the file will contain the strain plane given in four columns containing : : time [hours] ϵ_0 [m/m] κ_x [1/m] κ_y [1/m]. Notice that the file is rewritten without the strain plane if a stress calculation is performed later on !
.RV1	The file contains information about RV selections.

All the materials used in the project are saved in the database files which contains the materials of all the defined projects. The database files are listed in the following:

concrete.dbs	Contains the concrete materials
cement.dbs	Contains the cement
aggregat.dbs	Contains the aggregates
minadd.dbs	Contains the mineral additives
chemadm.dbs	Contains the chemical additives
mixother.dbs	Contains other types of concrete constituents
water.dbs	Contains the water
material.dbs	Contains the types of materials which are not concrete
soil.dbs	Contains the soil materials
insulat.dbs	Contains the insulation materials.

For the time dependent concrete properties it is possible to use datafiles with measuring values. These datafiles have the following suffixes depending on the type of data:

.HET	Heat development
.EMO	E-Modulus development
.THX	Thermal expansion coefficient
.POI	Poisson development

.CST	Compressive strength development
.TST	Tensile strength development
.STA	Initial strain development
.VIS	Creep - visco 1 development
.VIS	Creep - visco 2 development
.SPR	Creep - spring 1 development
.SPR	Creep - spring 2 development
.CPO	Creep - creep poisson ratio development.

When the work with a project has been finished and you want to save the project on a disk, you may save all the listed files.

To be able to reconstruct the project you only have to save the .PRJ file, the database files and the used datafiles with measured concrete property data.

NB! If an old project file and its database files are copied into the working directory remember to save the existing database files in another directory. Otherwise these will be overwritten and lost.

6. Project Editor

This section will help you understand the structure in the Project Editor (PE) and the basic elements inside and how to use them. The section is divided into several subsections that describe the general structure in PE and the basic elements in PE. The basic element subsections provide step by step instructions to help you outline and analyse your projects. The term *a project* was introduced in section 5.2.

6.1. General

Figure 6.1 shows the PE just after you have entered 4C-Temp&Stress and with no project loaded. The figure introduces some of the elements in the PE.

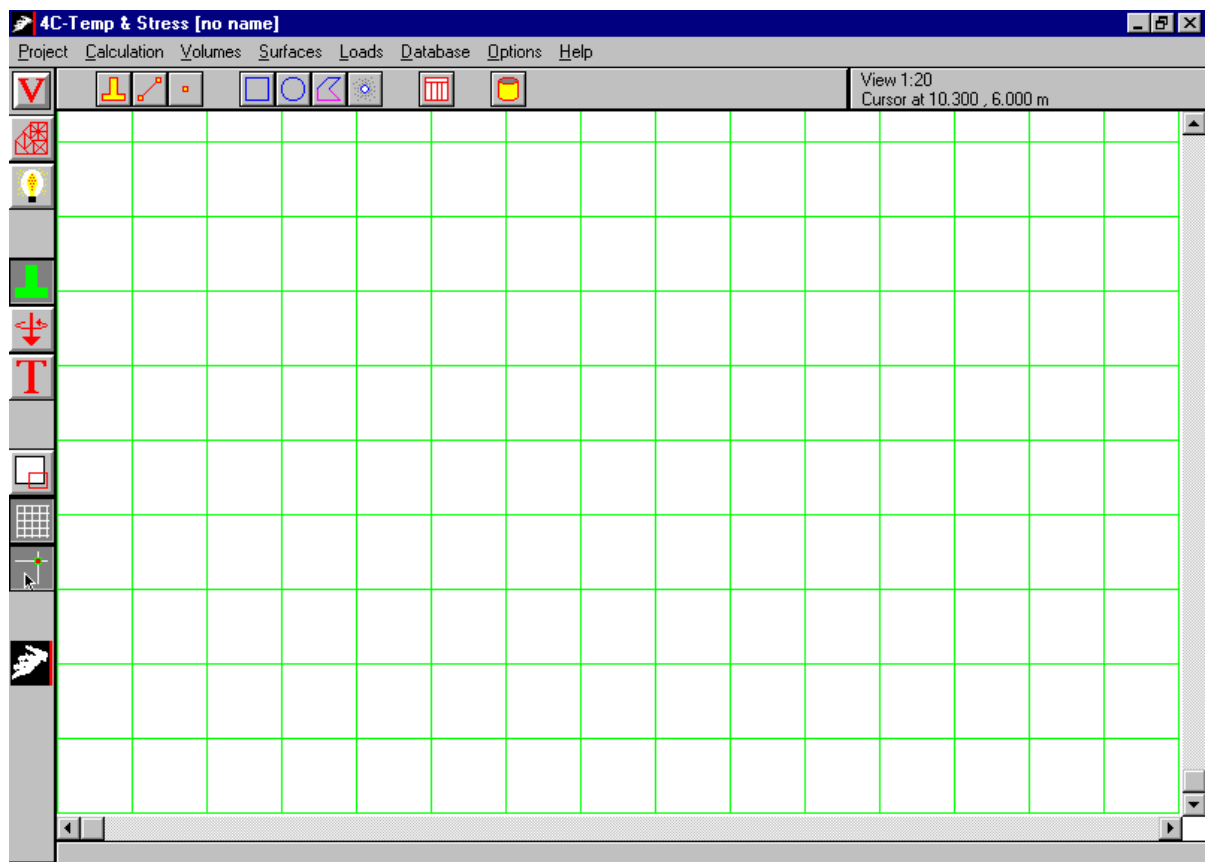


Figure 6.1
4C-Temp&Stress - Project editor.

The PE menu items is placed horizontally in the top of the PE. The PE menu includes all the menu items to be used for outlining your project. Each menu item contains a menu bar and all the subjects in the menu bars can be selected independently of the selected PE mode button.

For more specific information about how to use the menu items and a step by step description of them, see the following sections:

- Project: Section 6.7
- Calculation: Section 6.6 and 8
- Volumes: Section 6.2 and 6.3

Surfaces: Section 6.3
Loads: Section 6.4
Database: Section 7
Options: Section 6.2

6.2 Geometry Mode

This section will help you understand the structure of the Geometry Mode in the project editor. Click the Volume mode button in the vertical tool bar to select the Geometry mode. In Geometry mode it is possible to define the geometry of the construction. Any construction consists of either rectangles, polygons, circles or points or a combination of these geometries. All the geometries are drawn by using the mouse like you do in any other drawing program. This section provides step by step instructions to help you outline volumes, cooling pipes and heating wires in your projects.

6.2.1 View Options

Before you start defining any volumes you may setup the view on the screen.

6.2.1.1 View Settings

Select the menu item [Options] and the sub menu item [View settings] and the view settings dialog box will pop up. The View settings cover information about the current view by defining the Grid step size and the Scaling mode.

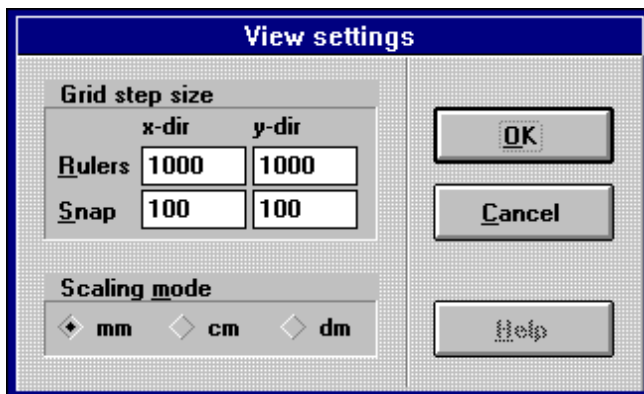


Figure 6.2
View settings dialog box.

Grid step size Identifies the size of the grid (Rulers) and the distance between the increments on the ruler (Snap). The volumes that you draw are forced to adhere to the ruler increments if you have selected snap to grid. See section 6.2.1.3.

Scaling mode Identifies the units of the rulers. The units can either be mm, cm or dm.




The view settings is a part of the project. This option makes your project geometry appear the same way when you retrieve the project as when you closed the project.

NB! Changing **Scaling mode** to a more coarse unit (eg. from mm to cm) during the project will result in loss of precision.

6.2.1.2 Scale Font





In the menu item [Options] you can also select the submenu items [Scale font up] and [Scale font down]. These two options are used to scale the text on the screen up or down so that you always will be able to read the text independently of whether you have zoomed the view in or out. Select the current scale font option and the text on the screen will be changed immediately.

6.2.1.3 View Buttons

Three view buttons are placed in the vertical tool bar: Zoom , Grid  and Snap to grid .

Zoom

Click the zoom button and four buttons will pop up.

These buttons make it possible to zoom in , to zoom out , to zoom in on a specified area by framing the area  and to zoom out to full paper size .

Please notice the view in the upper right corner of the PE. If the view is 1:1 the zoom in option is not available. However, if you have to zoom in and the units in the view settings have been selected to be cm or dm you may change the units to mm or cm. This will make the drawing bigger.

Grid

Click the grid button if you want to show the grid.

Snap to grid

Click the snap to grid button if you want the volumes to be adhered to the ruler snap. NB! The snap to grid is **on** when the button contains a red dot with a green square around in the intersection between the two lines.

6.2.2 How to Draw Volumes

From a geometrical point of view a volume is either a Rectangle, a Circle, a Polygon or a Point. A point illustrates a cooling pipe or a heating wire. The location and the size of the volume is handled automatically by . Automatically means that you do not have to enter the coordinates of the centre or the corners, just place the cursor and click the left mouse button. The following four submode buttons show the different drawing modes that you can select from the horizontal tool bar in the PE. You must select one of these submodes before you can draw a volume.



Rectangle.



Circle.



Polygon.



Point - Cooling pipe or heating wire.

You can only draw volumes in the View area, with the mouse, when the PE is in Geometry & Thermal mode and the Object mode is Volume.

Draw a rectangle

Select the submenu button Rectangle. Locate the position where you want to draw the rectangle. Notice the cursor position in the upper right corner of the PE. Place the cursor at the **upper left corner** of the desired rectangle, click the left mouse button and keep it down, move the cursor (mouse) to the **bottom right corner** of the desired rectangle and release the left mouse button. See figure 6.3.

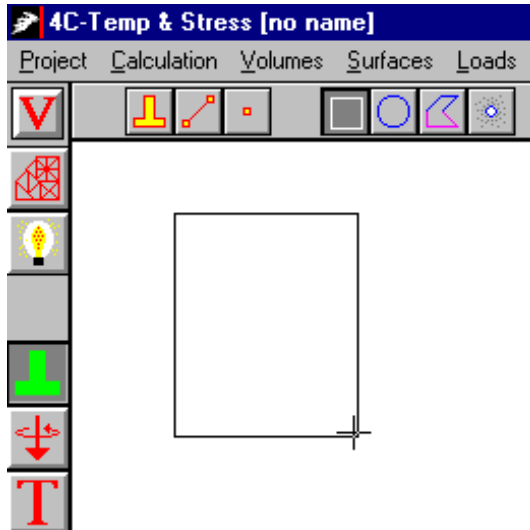


Figure 6.3
Draw a rectangle.

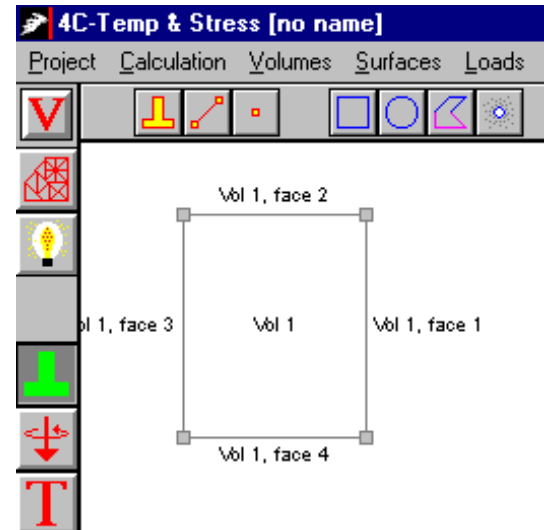


Figure 6.4
Empty rectangle.

The Volume dialog box will pop up immediately after you have drawn a volume. If you want to see the empty volume before you edit the volume parameters, just click the OK button. See Figure 6.4.

Draw a circle

Select the submenu button Circle. Locate the position where you want to draw the circle. Notice the cursor position in the upper right corner of the PE. Place the cursor at the **centre** of the desired circle, click the left mouse button and keep it down, and move the cursor (mouse) to **a corner of the rectangle that touch the desired circle**. When you release the mouse button a circle or an ellipse will be drawn and calculated as a polygon with the desired number of faces. The desired number of faces can be specified in the Calculation parameters as described in section 6.6.8.

Draw a polygon

Select the submenu button Polygon. You may follow the cursor position in the upper right corner of the PE. Place the cursor at the **starting point** of the desired polygon, click the left mouse button and keep it down, move the cursor (mouse) to the **2. point**, click the left mouse button and keep it down, move the cursor (mouse) to the **next point**, click the left mouse button and keep it down etc. To be able to close the polygon the **ending point** must be identical with the **starting point**. The polygon can also be closed by **double clicking** at the last point before the starting point. If two points are identical in a polygon an error will appear when the FEM is generated. It is then necessary to move the two points apart from each other.

Draw a point

Cooling pipes or Heating wires. Select the submenu button Point. The location is identified by the **centre** of the cooling pipe/heating wire and will be handled automatically by . Locate the position where you want to place a c-pipe/h-wire. Notice the cursor position in the upper right corner of the PE. Place the cursor at the **location** of the desired coordinates and click the left mouse button. See figure 6.5 and 6.6. NB! No intersection between the c-pipes/h-wires and the border of the volume is allowed.

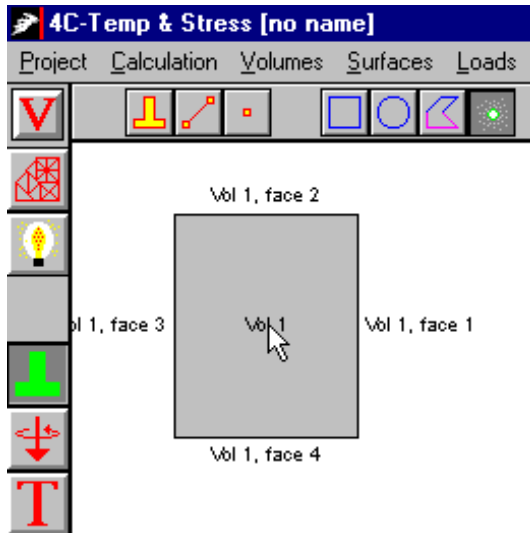


Figure 6.5

Draw a cooling pipe/heating wire.

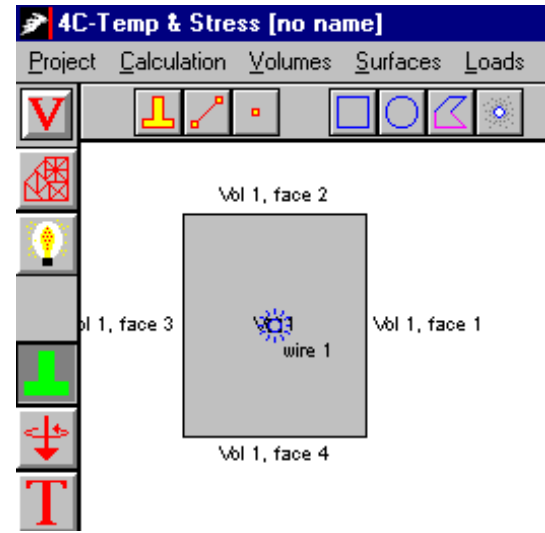


Figure 6.6

Cooling pipe/heating wire in volume.

The Volume dialog box includes a list of all volumes (rectangles, circles and polygons) in the project and all the parameters that you need to specify for each volume. The volume that you have drawn will be highlighted in the list. See section 6.3.1 about the parameters in the Volume dialog box.

A special c-pipe/h-wire dialog box will pop up immediately after you have drawn a c-pipe or a h-wire. If you want to see the location of the c-pipe/h-wire, before you enter the parameters, just click the OK button. See figure 6.6. The c-pipe/h-wire dialog box includes a list of all c-pipes/h-wires in the project and all the parameters that you need to specify for each c-pipe/h-wire. The c-pipe/h-wire that you have drawn will be highlighted in the list. See section 6.3.3 about the parameters in the c-pipe/h-wire dialog box.

6.2.3 How to Edit Volumes and C-pipes/H-wires

There are two ways you can edit a volume or a cooling pipe/heating wire. You can either edit the geometry or the parameters shown in the dialog box for either volumes or c-pipes/h-wires.

Volume geometry

You can edit the geometry of a volume in three ways depending on the object mode, which can be either volume, surface or node mode. Depending on the object mode you can either move the volume, the surface or the node in the view area.

The object mode buttons are as follows:



Volume mode



Surface mode



Node mode

You can select a volume when the Volume mode is selected. Click the left mouse button, when the cursor is above the volume. The nodes in the rectangle that surrounds the volume will become grey after selecting the volume.

You can select a surface when the Surface mode is selected. Click the left mouse button, when the cursor is above the surface. The nodes that extend the line will become grey after selecting the surface.

You can select a node when the Node mode is selected. Click on the left mouse button, when the cursor is above the node. The node will become grey after selecting the node.

Select the volume, the surface or the node, keep the left button down and **move** the cursor to the new location on the displayed view and release the button. If you want to **delete** the volume just select the volume and hit the key [Delete] on the keyboard or click the button Delete object in the horizontal tool bar in the PE:



Delete object.

How to move a volume and edit a volume by moving the right side (surface) and the bottom right corner (node) is shown in figure 6.7 - 6.10.

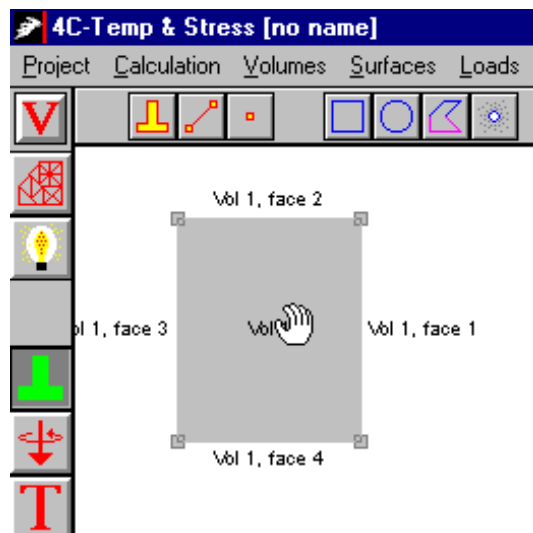


Figure 6.7
Move volume.

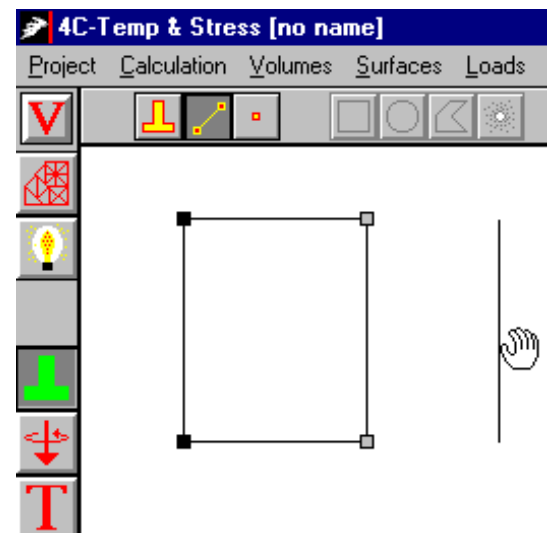


Figure 6.8
Move volume surface.

There are two ways you can **edit a c-pipe/h-wire**, just like volumes. You can either edit the geometry of the c-pipe/h-wire or edit the c-pipe/h-wire parameters shown in the c-pipe/h-wire dialog box.

C-pipe/h-wire geometry

The object mode must be Node mode before you can edit the geometry of a c-pipe/h-wire. You can either delete or move the c-pipe/h-wire around in the view area.

You can select a c-pipe/h-wire with a click on the left mouse button, when the cursor is above the c-pipe/h-wire. The c-pipe/h-wire will become grey after selecting the c-pipe/h-wire.

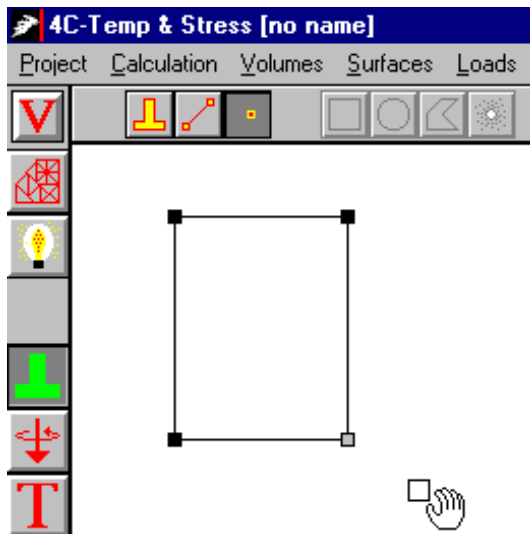


Figure 6.9
Move volume node.

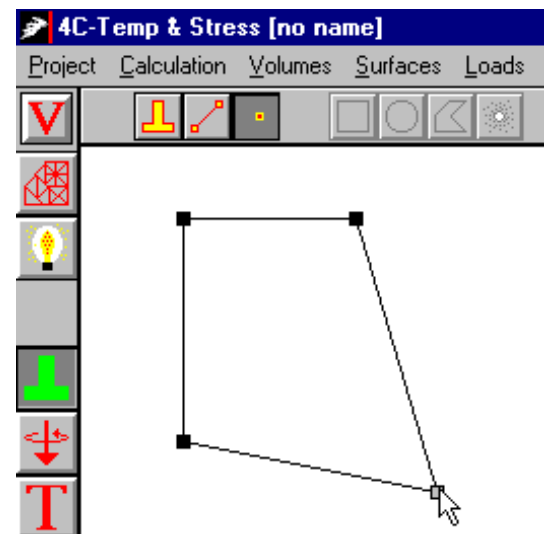


Figure 6.10
New volume object.

If you want to move a c-pipe/h-wire select the c-pipe/h-wire, keep the left button down and **move** the cursor to the new location in the view area. See figure 6.11. If you want to **delete** the c-pipe/h-wire just select the c-pipe/h-wire and press the key [Delete] on the keyboard or click the button [Delete object] in the horizontal tool bar in the PE.



Delete object

Dialog box

Select the volume or the c-pipe/h-wire as described above and then select the PE menu **Volumes** and the menu item **Edit volumes** or **Edit Cooling pipes/h-wires**. This will bring up the dialog box for volumes or for cooling pipes/h-wires. See figure 6.12 where the menu item is selected. Otherwise select the volume/c-pipe/h-wire as described above and then select the **pop up button** or just **double click** the volume/c-pipe/h-wire and the dialog box will pop up directly.

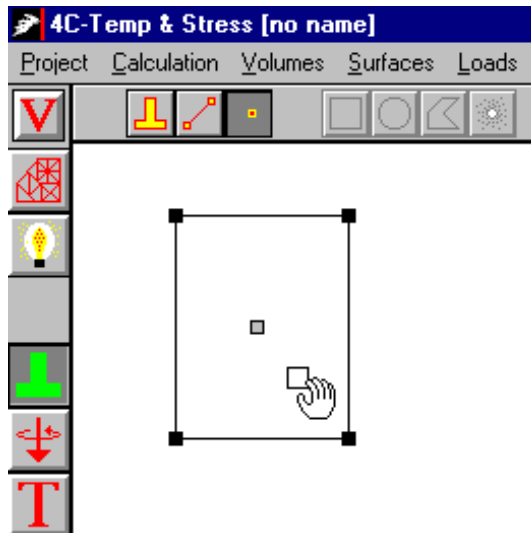


Figure 6.11
Select and move cooling pipe.

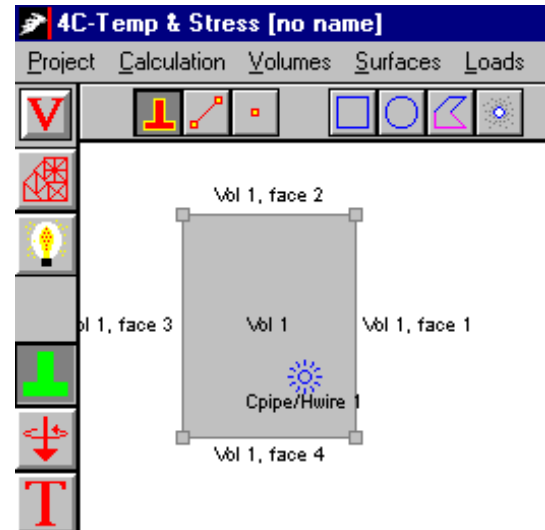


Figure 6.12
Edit volume, cooling pipe or heating wire.

6.2.4 How to Draw More than One Volume

When a structure consists of more than one volume, for instance when a wall is placed on a deck, it is important that the connection between the two volumes is correct.

The construction joint between the wall and the deck have to be well-defined by identical nodes in the deck and the wall. See figure 6.13 and 6.14.

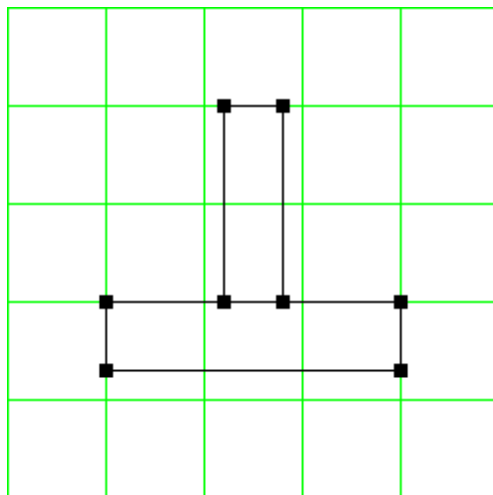


Figure 6.13
The structure consists of two volumes:
Deck and wall.

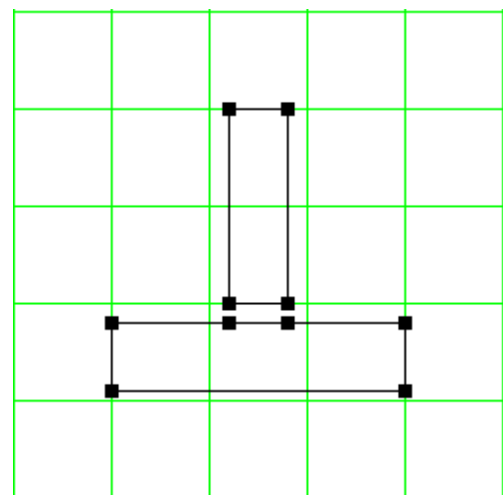


Figure 6.14
The nodes by the construction joint are
defined for both the deck and the wall.

To obtain the identical nodes in the construction joint it is necessary to draw the deck as a polygon with 6 faces and the wall may be drawn as a rectangle or a polygon with 4 faces. See section 6.2.2.

6.3 Thermal Mode

This section will help you understand the structure of the Thermal Mode in the project editor. Click the volume mode button in the vertical tool bar to select the thermal mode. In Thermal mode it is possible to define the type of materials that the structure consists of - like concrete, steel or soil - and to define the thermal boundary conditions for the structure - like ambient temperature, wind velocity, insulation material, cooling pipes and heating wires. This section provides step by step instructions to help you outline the thermal part of your projects.

6.3.1 Volumes

This section defines the volume parameters and describes how to identify a volume and specify the parameters for the volume.

The volume dialog box will pop up immediately after you have drawn the volume. If you have already drawn a volume, you can edit the volume parameters by selecting the volume dialog box as described in section 6.2.3. The volume dialog box includes a list of all volumes in the project and all the parameters that you need to specify for each volume. The volume that you have drawn or selected will be highlighted in the list.

The Volume dialog box is divided into three subareas: The volume list, the current volume definition and the fill material. See figure 6.15. The subareas of the dialog box are described in the following.

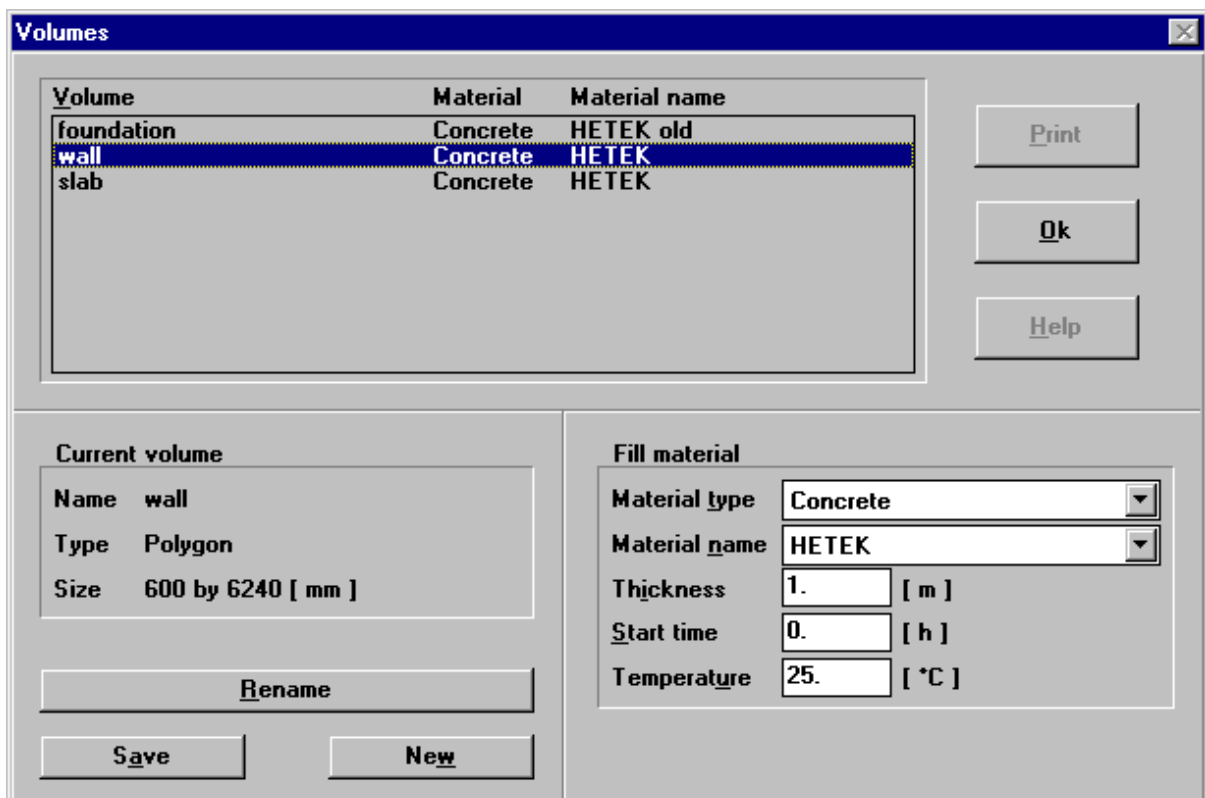


Figure 6.15
Volume dialog box.

Volume list

The volume list includes a list of all volumes in the project identified by the volume name, the material type and the material name. The volume name is described under **current volume**. The material type and the material name are described under **fill material**. The volume that is highlighted in the volume list is the current volume.

Current volume

Name: Identifies the current name of the volume. Click the button **Rename** to change the name of the volume (max. 19 characters).

Type: Identifies the volume type (rectangle or polygon).

Size: Identifies the size of the rectangle that includes the volume. The units are the type selected in the view settings, see section 6.2.1.1.

New: Makes it possible to define a volume by width/height, radius or coordinates in a datafile. When New is selected the New Volume dialog box will pop up. Enter the desired name of the new volume. The type can be Rectangle, Circle or Polygon. Click the down arrow button and select the desired type. Depending of the selected type different parameters must be entered.

Rectangle: Enter the centre coordinates of the desired rectangle together with the size given by width (x) and height (y). The units are the type selected in the view settings, see section 6.2.1.1.

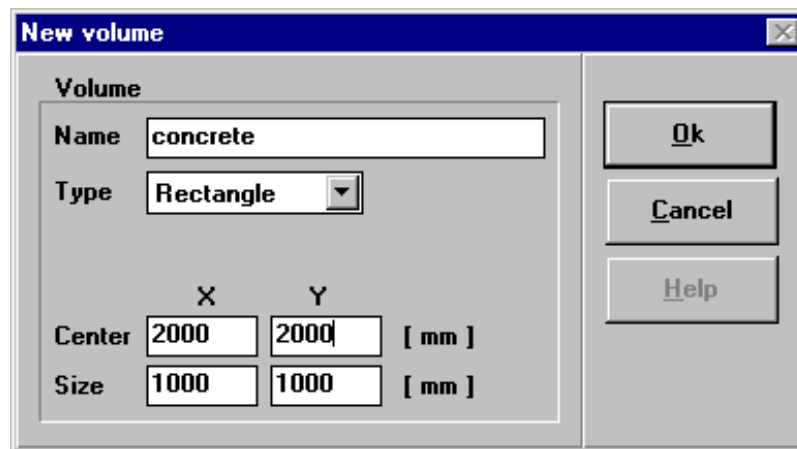


Figure 6.16

New volume dialog box corresponding to the type Rectangle.

It must be ensured that the **entire** rectangle is within the area limited by $0 < x < 32$ m and $0 < y < 32$ m.

Circle: Enter the centre coordinates of the desired circle/ellipse together with the radius in the x- and y-direction. (Equal x- and y-radius gives a circle). The units are the type selected in the view settings, see section 6.2.1.1.

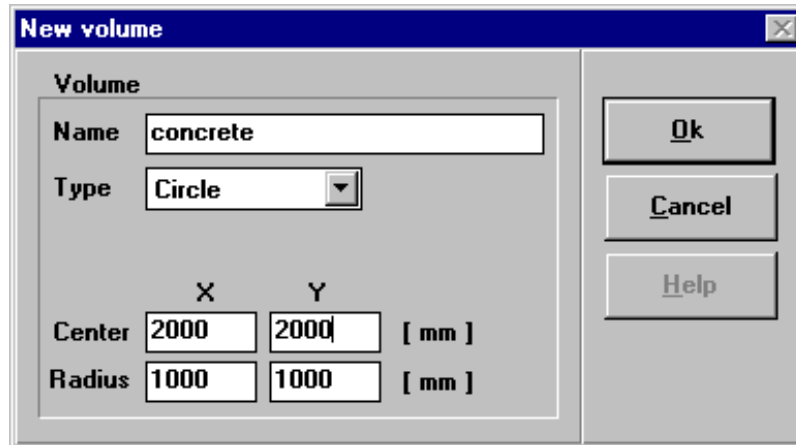


Figure 6.17
New volume dialog box corresponding to the type Circle.

It must be ensured that the **entire** circle/ellipse is within the area limited by $0 < x < 32$ m and $0 < y < 32$ m.

Polygon: A polygon can be defined by corresponding values of x- and y-coordinates to the corners. Click the down arrow button and select the desired datafile.

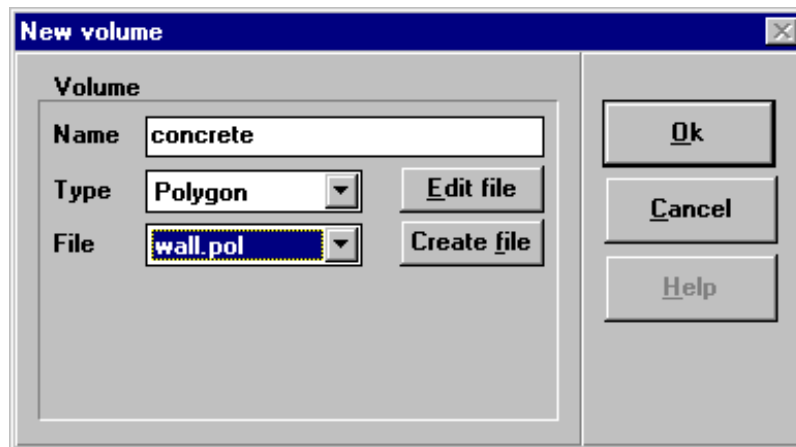


Figure 6.18
New volume dialog box corresponding to the type Polygon.

A new datafile can be created by clicking the Create file button. This will make the Datafile editor appear. Enter the corresponding values of x- and y-coordinates. There is a limit of 50 set of coordinates. The cursor is moved with the mouse or the [Tab] key. If necessary the list can be scrolled by the up- or down arrow button to the right of the list. When the OK button is pressed the filename can be entered. The file must have the suffix *.POL. The units are the type selected in the view settings, see section 6.2.1.1. The coordinates are saved in metres.

NB! Files created by use of other editors can be used. The format of the datafile must be two columns containing (x [m], y [m]) values. st en blank. The file must have the suffix *.POL.

Important! No matter if the datafile is created by 4C-Temp&Stress or by other editors each point must be defined only once. (4C-Temp&Stress will automatically close the polygon). Double points may lead to errors during the FEM-mesh generation. It must be ensured that each point is within the area limited by $0 < x < 32$ m and $0 < y < 32$ m.

A datafile can be edited by clicking the **Edit file** button This will make the Datafile editor appear. The changes will not affect previous defined volumes. Files with more than 50 points can not be edited with the datafile editor, but the values will be used in the calculations.

When **Ok** is pressed in the New volume dialog box the geometry can only be edited by use of the mouse (see section 6.2.3). E.g. if a polygon is to be changed by means coordinates it must be deleted and a new one must be created.

NB! I order to create a volume by width/height, radius or datafile at least one volume need to be defined earlier with the mouse. (Eventually a rectangle can be drawn by mouse and then deleted later on).

Save: Makes it possible to save the coordinates of the selected volume in a datafile. Enter the desired name of the datafile. The suffix must be *.POL. The geometry is saved as corresponding values of x- and y-coordinates in metres to each corner.

If a polygon created by datafile is changed by mouse, the changes in coordinates will **not** be saved in the file unless Save is selected.

Fill material

Material type: Identifies the type of material connected to the volume. The material type can be either none, concrete, material or soil. Click the down arrow button and select the type that matches the desired type. The type identifies the database from where the current material will be loaded.

Hint! Holes in the structure can be made by drawing a volume with fill material type None. Holes must be placed totally inside the contour of another volume. Otherwise the hole must be drawn as a part of the contour by drawing the volume as a polygon.

Material name: Identifies the name of the current material that is saved in the database for the selected material type. Click the down arrow button and select the name that matches the desired name from the material database.

Thickness: Identifies the thickness of the volume in the direction perpendicular to the view plane. Enter the desired thickness of the volume in metres [m]. If the desired calculation is 2 dimensional the real thickness of the volume must be entered. If the calculation is 2½ dimensional a standard thickness of 1 m is to be recommended. **NB!** None of the external loads which can be specified as described in section 6.4.2 are depending on the thickness.


Start time: Identifies the starting time from which the volume is part of the project calculations. For volumes of fresh concrete the start time is typically the casting time. Enter the desired starting time in hours [h].

NB! There must always be at least one volume with start time = 0 h.

Temperature: Identifies the temperature at the starting time. For volumes of fresh concrete the temperature is typically the mix temperature. Enter the desired temperature in [°C].

6.3.2 Surfaces and Thermal Boundary Conditions

This section defines surface parameters and describes how to identify surfaces and how to specify and edit boundary conditions in the PE.

You can select a surface when the Surface mode  is selected. Click on the left mouse button, when the cursor is above the surface. The nodes that extend the line will become grey after selecting the surface. When a surface is selected you can define the surface parameters by selecting the PE menu [Surfaces] and the menu item [Edit surfaces]. This will bring up the Surface dialog box. Otherwise select the surface as described above and then select the **pop up button** or just **double click** the surface and the surface dialog box will pop up directly.

The Volume surface dialog box includes a list of all volume surfaces in the project. The selected surface will be highlighted in the list. The Volume surface dialog box is divided into three subareas: The surface list, the current face and the boundary condition. See figure 6.19 and the description of the subareas of the dialog box below the figure.

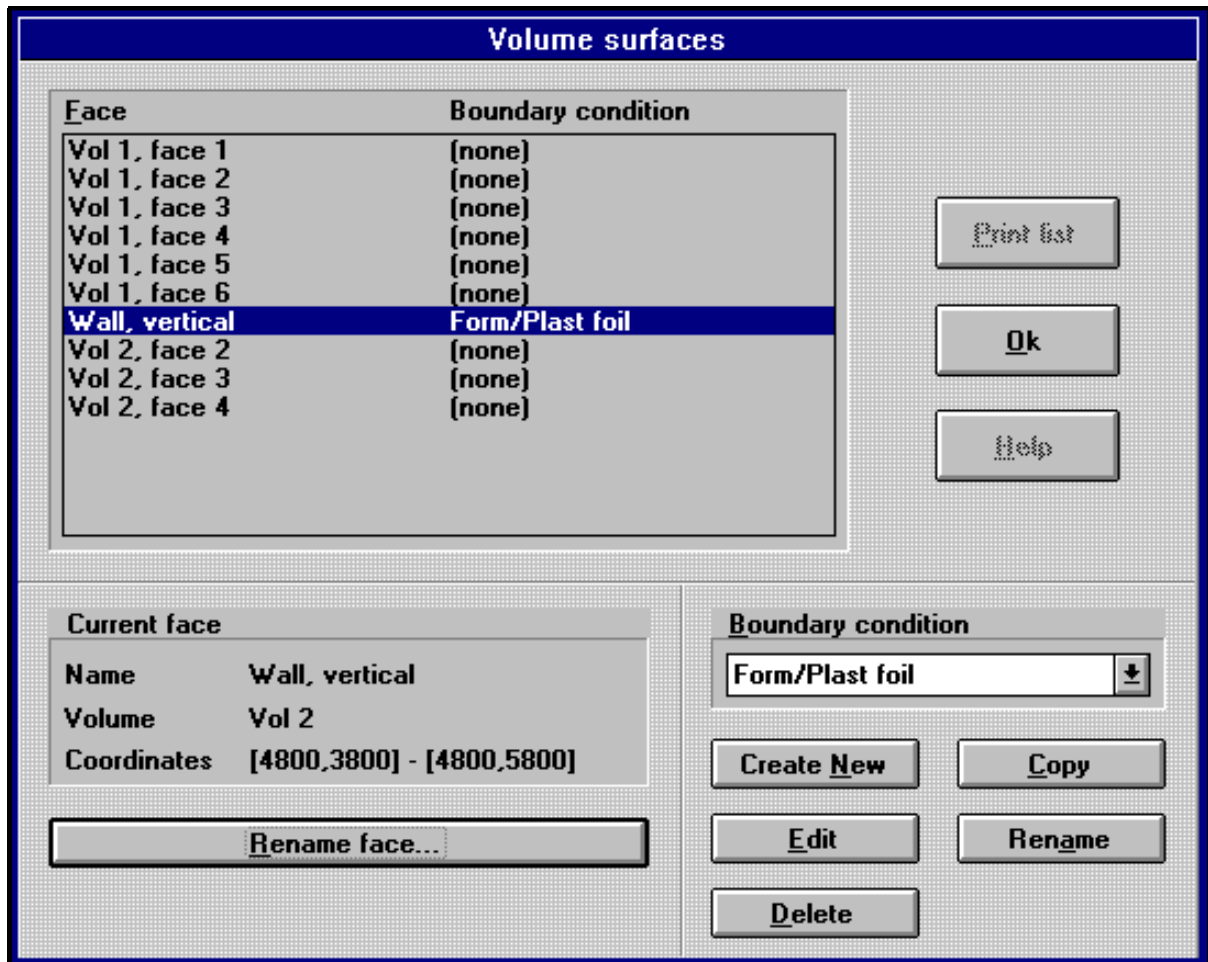


Figure 6.19
Volume surface dialog box.

Surface list

The Surface list includes a list of all surfaces in the project identified by the face name and the connected boundary condition. Select a face by a **click** on the desired face in the list and the specified parameters will pop up in the Current face area and the connected Boundary condition will pop up as well.

Current face

Name: Identifies the current name of the surface. The surfaces are automatically named, so that the name illustrates the connection to the volume. E.g. Vol 1, face 1 is surface number 1 on volume number 1. This name can be **renamed** with a more practical name, that perhaps illustrates the location of the surface - like *Wall, vertical*.

Volume: Identifies the name of the current volume that the surface is connected to.

Coordinates: Identifies the global coordinates for the current surface by the coordinates of the starting and the ending point.

Boundary condition Click the down arrow button to select the desired boundary condition for the current surface. If no boundary condition is created - click the **Create new** button to create a new boundary condition and enter the name of the boundary condition in the dialog box that appears. How to define and edit the boundary condition, see section 6.3.2.1.

If several surfaces are supposed to have the same boundary condition **select** the boundary condition for the first surface, press the **[Shift]** button on the keyboard down and **click** the other surfaces to copy the selected boundary condition.

6.3.2.1 Thermal Boundary Conditions

A thermal boundary condition may describe:

- A convection heat flow out of (or into) the volume. The heat transfer to the surroundings from the surface can be expressed in terms of a temperature difference and a heat transfer coefficient, k , so that the heat flow, q , is:

$$q = k \cdot (T - T_{ext})$$

where T_{ext} is the external temperature and T is the surface temperature of the volume.

The external temperature must always be specified when the thermal boundary condition describes convection heat flow.

The heat transfer coefficient may be specified directly or alternatively specified by definition of the form work, the insulation and the wind velocity. Based on these parameters the heat transfer coefficient, k , is determined as:

$$k = \left[\frac{1}{\alpha_C} + \left(\frac{e}{\lambda} \right)_{ins} + \left(\frac{e}{\lambda} \right)_{form} \right]^{-1}$$

where

k	=	overall transmission coefficient	[kJ/m ² h°C]
α_C	=	convective surface transmission coefficient	[kJ/m ² h°C]
e	=	thickness of insulation and of form work, respectively	[m]
λ	=	thermal conductivity	[kJ/mh°C]

An approximate value for α_C by forced convection can be calculated as a function of the wind velocity:

$$\alpha_C \cong 20 + 14 \cdot v \quad \text{for } v \leq 5 \text{ m/sec}$$

$$\alpha_C \cong 25,6 \cdot v^{0,78} \quad \text{for } v > 5 \text{ m/sec}$$

where v is the wind velocity by forced convection [m/sec].

- Radiation. The radiation can be expressed as:

$$q = C_s \cdot [(T_{ext} + 273)^4 - (T + 273)^4]$$

Where

T	=	surface temperature	[°C]
T _{ext}	=	external temperature	[°C]
C _s	=	radiation coefficient	[kJ/m ² /h/°C ⁴]

- A prescribed temperature on a surface, not depending on internal and external temperatures.
- A prescribed heat flux, not depending on internal and external temperature.
- A combination of convection heat flow, a prescribed heat flux and radiation.

The thermal boundary condition is defined as a function of time which makes it possible to define the thermal boundary condition according to a prescribed curing schedule.

The boundary condition dialog box will appear when you create a new boundary condition or when you want to edit the current one. The boundary condition dialog box is divided into three subareas: The name, the functions and the current function. See figure 6.20 and the description of the subareas of the dialog box below the figure.

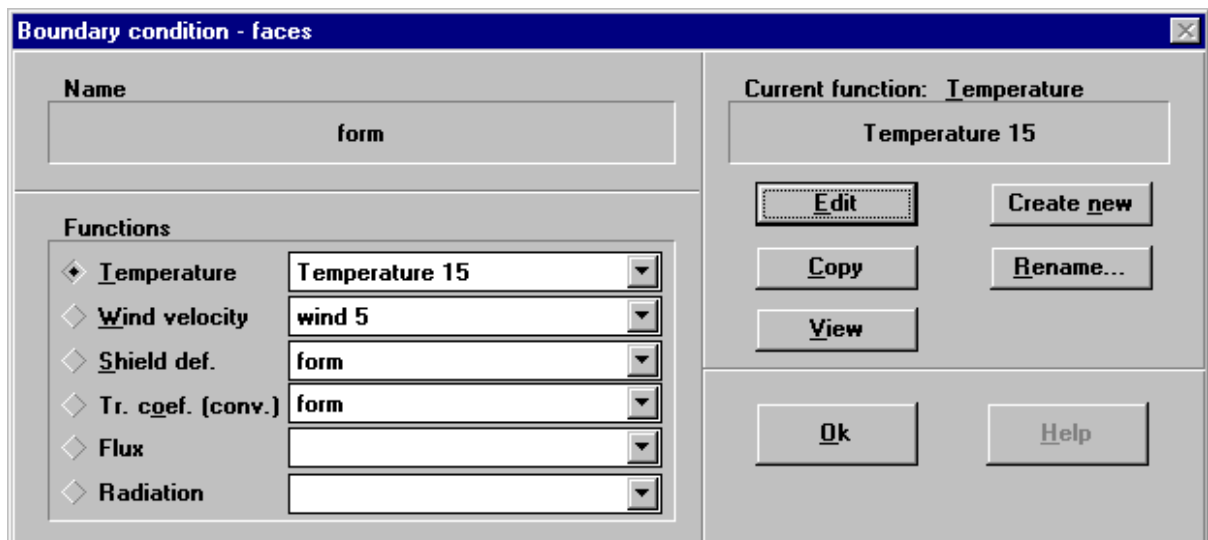


Figure 6.20
Boundary condition dialog box.

Name Identifies the name of the current boundary condition.

Functions Identifies the references to functions that is valid for the current boundary condition. A function can be either a Temperature -, Wind velocity -, Shield def. -, Tr. coef. -, Flux -, or a Radiation-function. The current function is marked in the radio buttons on the left side of the area. Click the **radio** button to select a new current function. Click the **down arrow** button on the right side to select a reference to another function.

Current function Identifies the selected function which is marked in the radio button. The four buttons: **Edit**, **Create new**, **Copy**, **Rename** and **View** are connected to the current function. If no function is created - click the **Create new** button to create a new function and enter the name of the function in the dialog box that appears. Click the **Edit** button to edit the current function. To visualize the current function click the **View** button. The five different types of functions are described in the following.

Temperature

A temperature function can be either a **linear** or a **sine** function of time.

Linear The linear function is described as a parameter function with a number of (Time [h], Temperature [$^{\circ}$ C]) values. The limit of (Time, Temperature) values is 50 values, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6.5 about the total process time. In figure 6.21 an example of a linear temperature curve as a function of time is shown.

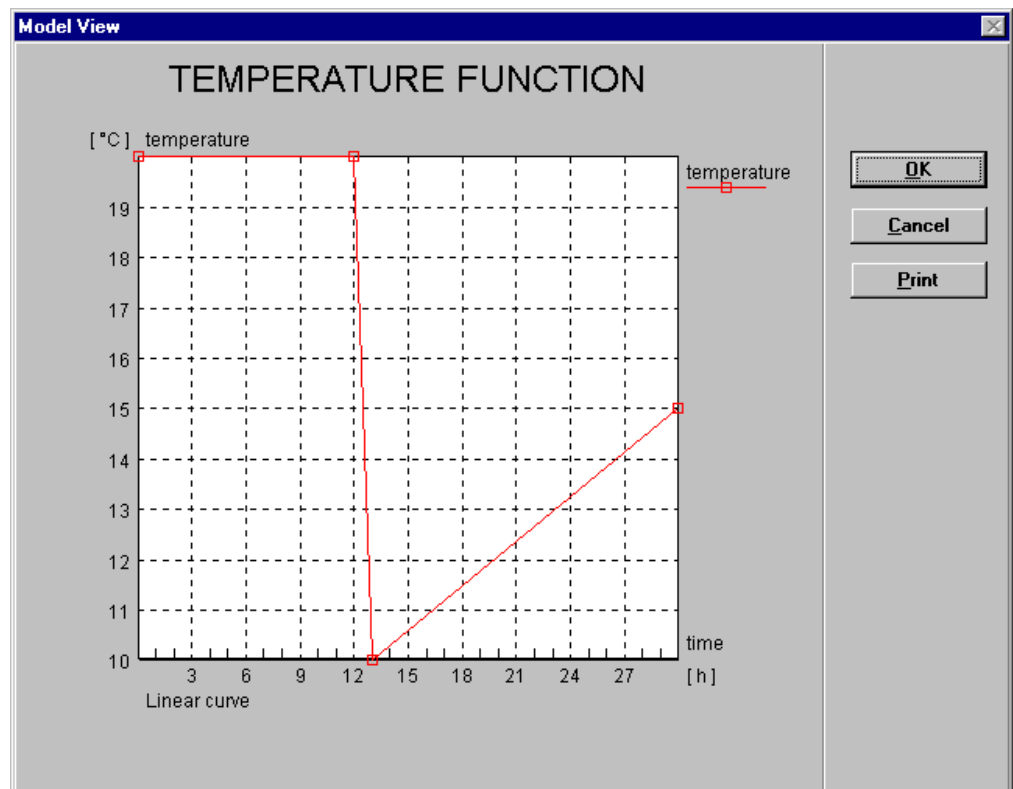


Figure 6.21
A linear temperature function

When you select a linear temperature function the Linear temperature function dialog box will pop up as shown in figure 6.22. Enter the time values in the left column and the temperature values in the right column. Place the cursor in the desired field (either with the mouse or the Tab key) and enter the desired value for either time or temperature.

Scroll the list with a click on the up- or down arrow button to the right of the list to see the entire list of time-temperature values - if necessary !

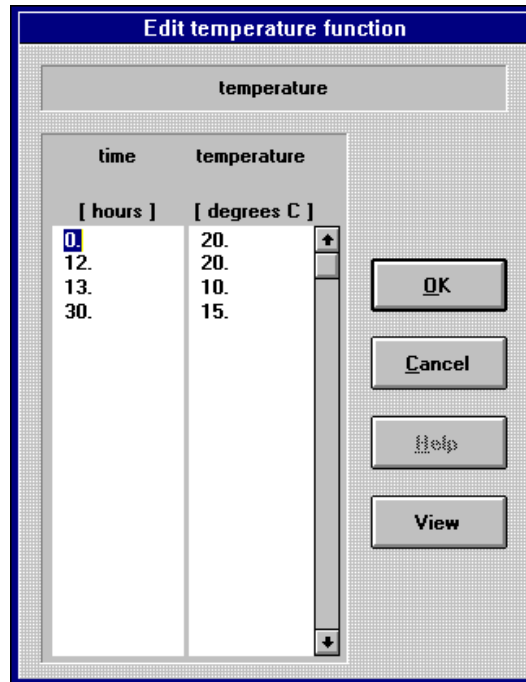


Figure 6.22
Linear temperature function dialog box.

Sine

The sine function is described by three parameters: Start time, Minimum and Maximum. With these three parameters it is possible to define a temperature function with a cycle of 24 hours, where the sine curve can be displaced along the time axis. In figure 6.23 the sine curve as a function of time is shown together with an example of how to define the three parameters.

Start time: Identifies the start time in hours [h] in the 24 hours notation for the sine function. The start time defines where on the time axis of the sine curve the concrete work begins. E.g. if the maximum temperature is reached at 2 p.m. and the minimum temperature is at 2 a.m. the time zero will be at 8 a.m., and if the start time of the concrete work is at 10 a.m. the **start time** on the sine function must be $(10-8) = 2$ h. See figure 6.23.

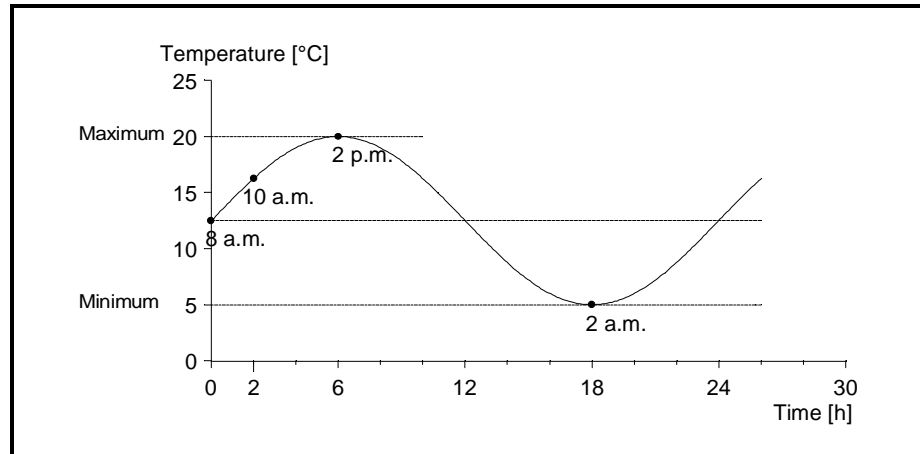


Figure 6.23
A sine temperature function.

Minimum: Identifies the minimum temperature value in [°C] for the sine function e.g. 5°C. See figure 6.23.

Maximum: Identifies the maximum temperature value in [°C] for the sine function e.g. 20 °C. See figure 6.23.

When you select a sine temperature function the Temperature function dialog box will pop up as shown in figure 6.24. Place the cursor in the desired field (either with the mouse or the Tab key) and enter the desired start time value, the minimum and the maximum temperature in the three fields.

Figure 6.24
Sine temperature function dialog box.

Wind Velocity

A wind velocity function is a piecewise constant function of time. The Wind velocity function is described as a parameter function with a number of (Time [h], Wind velocity [m/sec.]) values. The limit of (Time, Wind velocity) values is 50 values, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6.5 about the total process time. In figure 6.25 an example of a piecewise constant wind velocity curve as a function of time is shown.

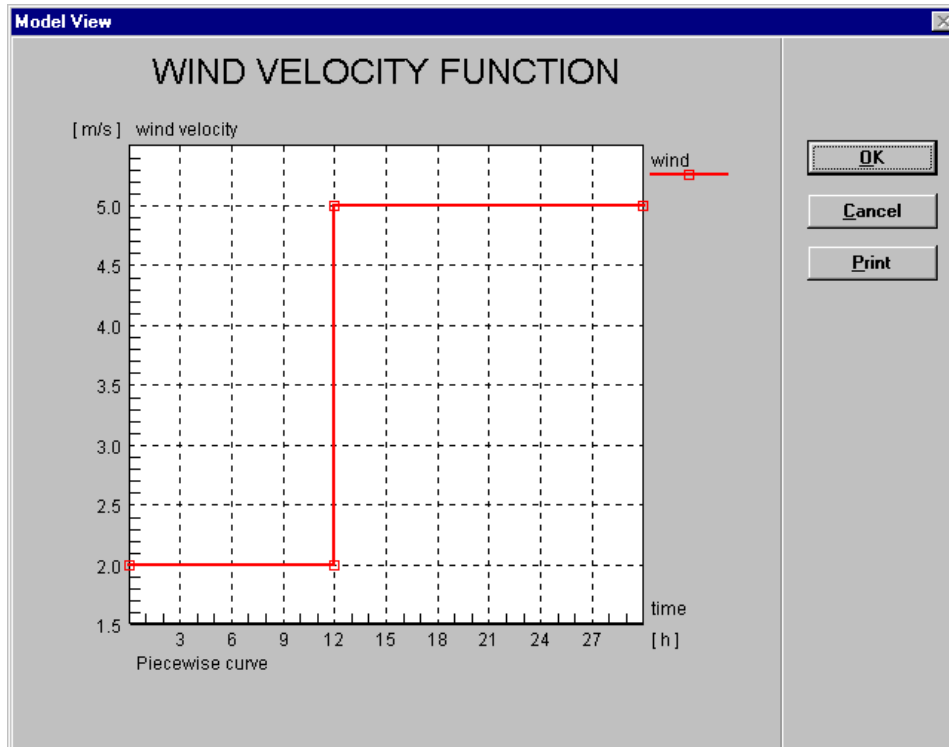


Figure 6.25
A Wind velocity function.

When you Edit the wind velocity function the wind velocity function dialog box will pop up similar to the Temperature function dialog box shown in figure 6.22.

Shield Def.

A shield definition function is a piecewise constant function of time. The shield definition function is described as a parameter function with a number of (Time interval [h], Shield def.) values. The limit of (Time interval, Shield def.) values is 50 values, and the last Time value (t_n) does **not** have to be greater than or equal to the total process time specified in the calculation parameters. If the last Time value is smaller than the total process time the surface will be handled as an unprotected surface in the rest of the process time.

When you edit the shield definition function the Shield definition dialog box will pop up as shown in figure 6.26. The Shield definition dialog box is divided into four subareas: Name, list of insulation materials, database and current selection.

The screenshot shows a dialog box titled "Shield definition" with a blue header bar. It is divided into several sections:

- Name:** A text field containing "Form / unprotected".
- Database:** A dropdown menu showing "Insulation" and "Etha foam, 6 mm" with a downward arrow button.
- Buttons:** "Add to list" and "Create new ..." buttons are located below the database dropdown.
- List of insulation materials:** A list box containing one entry: "0.00 48.00 Form bord, 19 mm".
- Current selection:** A sub-section showing details for the selected material:
 - Name: Form bord, 19 mm
 - Thickness: 0.019 [m]
 - Thermal cond. 0.36 [kJ/m/h/°C]
 - Time interval: 0. [h] to 48. [h]
- Buttons:** "Copy from database" and "Remove from list" buttons are located below the current selection details.
- Bottom Buttons:** "Sort list by time interval", "OK", "Cancel", and "Help" buttons are located at the bottom of the dialog.

Figure 6.26
Shield definition dialog box.

Name Identifies the name of the current shield definition function.

List of insulation materials

The list includes all the selected insulation materials together with the time interval where the insulation is placed on the surface. More than one insulation material can be placed on the surface in the same time interval, and the time intervals are also allowed to overlap each other. When the surface is unprotected the list of insulation materials will be empty.

Database

Identifies the type of insulation material to be selected from the record in the insulation database. Click the down arrow button to view and select the desired insulation material. Click the **Add to list** button to select the current material. The cursor will then be placed in the first field of the time interval in the Current selection subarea. See the section below. If you want to define a new insulation material click the **Create new** button and the Create new insulation dialog box from the database will pop up. For further details, see section 7.6 about the insulation database.

Current selection

Identifies the selected insulation material that has been added to the list. The identification includes the name, the thickness and the thermal conductivity of the insulation as it has been defined in the insulation database. Further the current time interval where the insulation is placed on the surface is defined in this subarea. Place the cursor in the left field (either with the mouse or the Tab key) to enter the time of placing the insulation on the surface and move to the right field (with the Tab key) to enter the end time when to remove the insulation.

The selected type of insulation in a time interval can be changed when it is highlighted in the list of insulation materials. Select the desired new insulation material in the database record and click the **Copy from database** button. This will change the current insulation material in the list.

Click the **Remove from list** button to remove the current insulation from the list of insulation materials

Tr. Coef.

A Heat transfer coefficient function is a piecewise constant function of time similar to the wind velocity function as shown in figure 6.25. The heat transfer coefficient function is described as a parameter function with a number of (Time [h], Heat tr. coef. [kJ/m²h°C]) values. The limit of (Time, Heat tr. coef.) values is 50 values, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6.5 about the total process time.

The heat tr. coef. function can either be generated automatically based on the defined wind velocity- and shield definition functions, or the heat tr. coef. function can be defined by typing the values directly.

Generated automatically

When a wind velocity function **and** a shield definition function have been defined the heat tr. coef. function will be generated automatically when you click the **OK** button in the Wind velocity or the Shield definition dialog box. The generated heat tr. coef. function will be named with the name of the boundary condition and it will contain a parameter function with a number of (Time, Heat tr. coef.) values. The Heat tr. coef. values are calculated as described in the beginning of this section 6.3.2.1.

When you **Edit** the heat tr. coef. function the Heat tr. coef. function dialog box will pop up similar to the Temperature function dialog box shown in figure 6.22, and it is then possible to see the calculated heat tr. coef. values. If you want to **change** the calculated values place the cursor in the current field and type a new value. When you click the **OK** button the changed function will be saved and the selection of wind velocity- and shield definition functions will be cleared.

These functions are no longer the basis for the selected heat tr. coef. function. If you regret the change of the heat tr. coef. function you can just select the wind velocity- and the shield definition functions again.

If you do not change the calculated values the wind velocity- and the shield definition functions are still the basis for the heat tr. coef. function. To **keep** the wind velocity- and the shield definition functions selected you must leave the heat tr. coef. function dialog box by clicking the **Cancel** button instead of the OK button.

Typed directly

Instead of defining a wind velocity- and a shield definition function from which the heat tr. coef. function will be generated you can define the heat tr. coef. function directly. If you have selected a current heat tr. coef. function click the **Edit** button otherwise click the **Create new** button, the heat tr. coef. dialog box will pop up and you can type the (Time, Heat tr. coef.) values directly.

It is possible to control the temperature on a surface directly not depending on internal and external temperatures. The desired surface temperature must be defined in a temperature function and a heat tr. coef. function must be defined where the heat transfer coefficient is 10^6 kJ/m²h°C or greater. This is used for cooling pipes of steel or for steel form work in water.

NB! When the surface is a construction joint between two volumes the heat transfer through the surface to the surroundings will be zero. This condition is defined by setting the heat transfer coefficient to zero.

Flux

A flux function is a piecewise constant function of time similar to the wind velocity function as shown in figure 6.25. The flux function is described as a parameter function with a number of (Time [h], Flux [kJ/m²h]) values. The limit of (Time, Flux) values is 50 values, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6.5 about the total process time.

When you **Edit** the flux function, the flux function dialog box will pop up similar to the Temperature function dialog box shown in figure 6.22.

Radiation

A radiation is a piecewise constant function of time similar to the wind velocity function as shown in figure 6.25. The radiation function is described as a parameter function with a number of (Time [h], Radiation [kJ/m²h/°C⁴]) values. The limit of (Time, Radiation) values is 50 values, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6.5 about the total process time.

When you **Edit** the radiation function, the radiation function dialog box will pop up similar to the Temperature function dialog box shown in figure 6.22.

6.3.3 Cooling Pipes / Heating Wires

This section defines the cooling pipe/heating wire parameters and describes how to identify cooling pipes/heating wires and how to specify the parameters.

The c-pipe/h-wire dialog box will pop up immediately after you have drawn the c-pipe/h-wire. If you have already drawn a c-pipe/h-wire you can edit the parameters by selecting the c-pipe/h-wire dialog box as described in section 6.2.3. The c-pipe/h-wire dialog box includes a list of all c-pipes/h-wires in the project and all the parameters that you need to specify for each c-pipe/h-wire. The c-pipe/h-wire that you have drawn or selected will be highlighted in the list.

The c-pipe/h-wire dialog box is divided into three subareas: The current list, the current definition and the boundary condition. See figure 6.27 and the description of the three subareas below the figure.

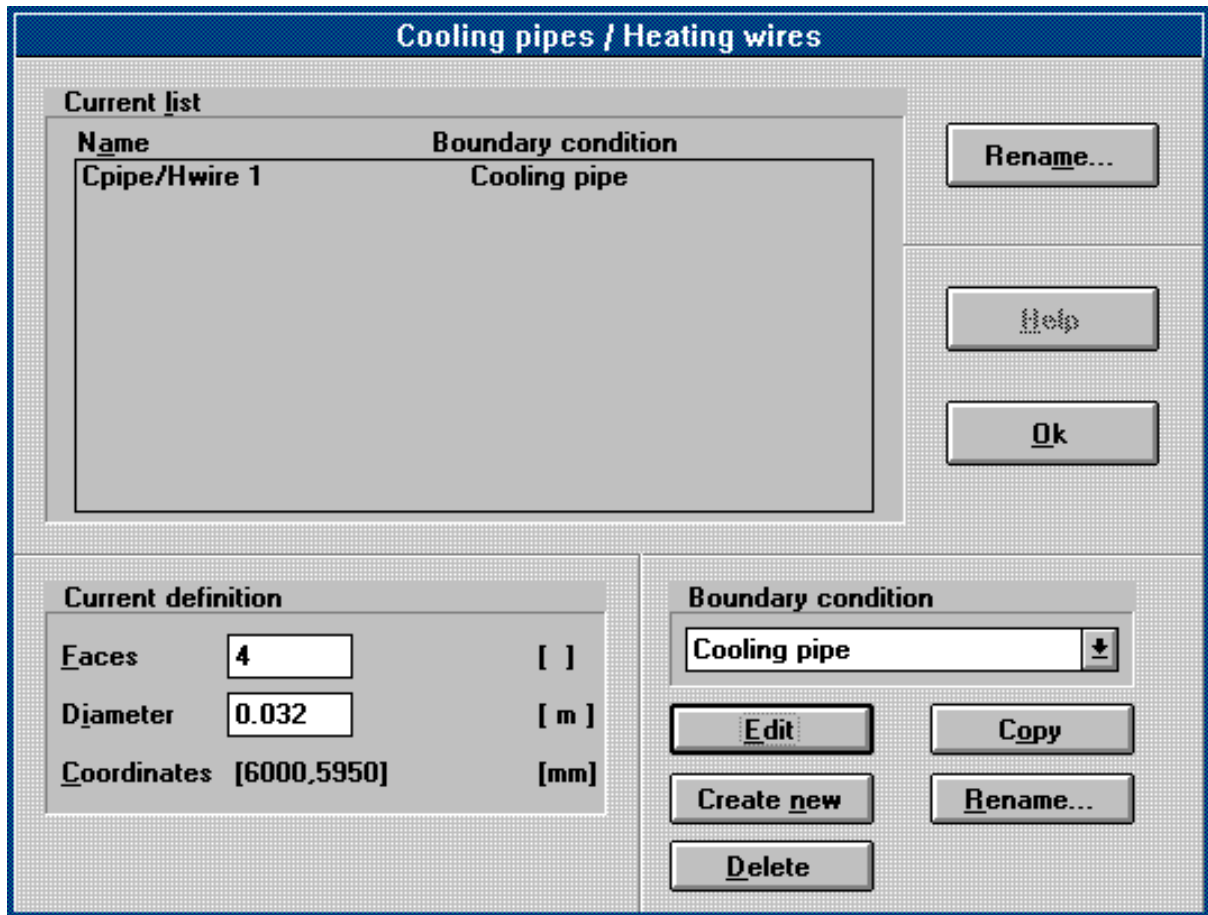


Figure 6.27
Cooling pipe / heating wire dialog box.

Current list

The Current list includes a list of all c-pipes and h-wires in the project identified by the c-pipe/h-wire name and the connected boundary condition. Select a c-pipe/h-wire by a **click** on the desired c-pipe/h-wire in the list and the specified parameters will pop up in the Current definition area and the connected Boundary condition will pop up as well.

Name: Identifies the current name of the c-pipe/h-wire. Click the **Rename** button to change the name of the c-pipe/h-wire.

Current definition

Faces: Identifies the number of faces used to describe the geometry of the c-pipe/h-wire. In 4C-Temp&Stress c-pipes/h-wires are drawn and calculated as polygons, where the number of faces can be defined by the user. E.g. 4 faces means that the c-pipes/h-wires are drawn as squares.

Diameter: Identifies the outer diameter of the cooling pipe in metres [m]. E.g. 0.032 m.

Coordinates: Identifies the coordinates for the centre of the current c-pipe/h-wire.

Boundary condition Click the down arrow button to select the desired boundary condition for the current c-pipe/h-wire. If no boundary condition is created - click the **Create new** button to create a new boundary condition and enter the name of the boundary condition in the dialog box that appears.

The thermal boundary condition for a cooling pipe does always consist of a linear temperature function and a directly typed heat tr. coef. function. The heat tr. coef. for a cooling pipe with full water flow can be calculated as:

$$k = \left(\frac{e}{\lambda} \right)^{-1}$$

where

k	=	heat transfer coefficient	[kJ/m ² h°C]
e	=	wall thickness of the cooling pipe	[m]
λ	=	thermal conductivity of the c-pipe material	[kJ/mh°C]

NB! Similar to the construction joints the heat transfer will also be zero from the surface of a cooling pipe where the cooling is switched off. Then the heat tr. coef. must be zero.

The thermal boundary condition for a heating wire does always consist only of a flux function [kJ/mh].

How to define and edit the boundary conditions see section 6.3.2.1.

If several c-pipes/h-wires are supposed to have the same dimension and boundary condition, **select** the boundary condition and **define** the dimension for the first c-pipe/h-wire, press the **[Shift]** button on the keyboard down and **click** the other c-pipes/h-wires to **copy** the selected boundary condition and the defined dimension.

6.4 Load Mode

This section will help you understand the structure of the Load Mode in the project editor. Click the load mode button in the vertical tool bar to select the mode. In Load mode it is possible to define how the structure is supported and to define several types of static loads - like point loads, line loads, x- and y-momentums. This section provides step by step instructions to help you outline the static part of your projects.

6.4.1 Supports

Supports are used in stress calculations. Supports are not used in approximated stress calculations.

The supports selected must ensure that the structure is unable to perform rigid body displacement. If the structure is able to perform rigid body displacement the stress calculations will be interrupted with an error message saying : "Singular matrix system".

6.4.1.1 Definition

In 4C-Temp&Stress supports can be added as point supports or line supports. A point support is added to a single structure node. A line support is added at a structure surface. Adding a line support actually means that identical supports are added to all FEM-nodes at the surface selected.

In each of the corner nodes of the FEM-element the displacement field is described by 6 parameters. These are: The displacement in x- and y-direction (u_x and u_y), the strain in x- and y-direction ($\epsilon_x=u_{x,x}$ and $\epsilon_y=u_{y,y}$) and the deviation in x- and y-direction ($u_{x,y}$ and $u_{y,x}$). One or more of these parameters can be fixed in order to create supports.

Fixed displacement:	x-direction:	No displacement of the selected FEM-node(s) in the x-direction. ($u_x = 0$).
	y-direction:	No displacement of the selected FEM-node(s) in the y-direction. ($u_y = 0$).
Fixed strain:	x-direction:	No strain in x-direction ($\epsilon_x=u_{x,x}=0$).
	y-direction:	No strain in y-direction ($\epsilon_y=u_{y,y}=0$).
Fixed deviation:	x-direction:	The displacement in the x-direction is held constant in the y-direction ($u_{x,y}=0$).
	y-direction:	The displacement in the y-direction is held constant in the x-direction ($u_{y,x}=0$).

Hint! Sometimes the shear stress τ_{xy} is required to be zero. The shear stress is given as:




$$\tau_{xy} = G \cdot (u_{x,y} + u_{y,x})$$

where

G	=	shear modulus of elasticity	[MPa]
$u_{x,y}$	=	deviation in x-direction	
$u_{y,x}$	=	deviation in y-direction	

If τ_{xy} should be fixed the deviations in both directions must be fixed.

6.4.1.2 How to Add and Edit a Support

1. Click the Point  or Line  support button.
2. Select the node or the surface to be supported.
3. Click the Add Support  button. This makes the support dialog box appear. The support dialog box is divided into two subareas: The current list and the current selection. See figure 6.28 and the description below the figure.

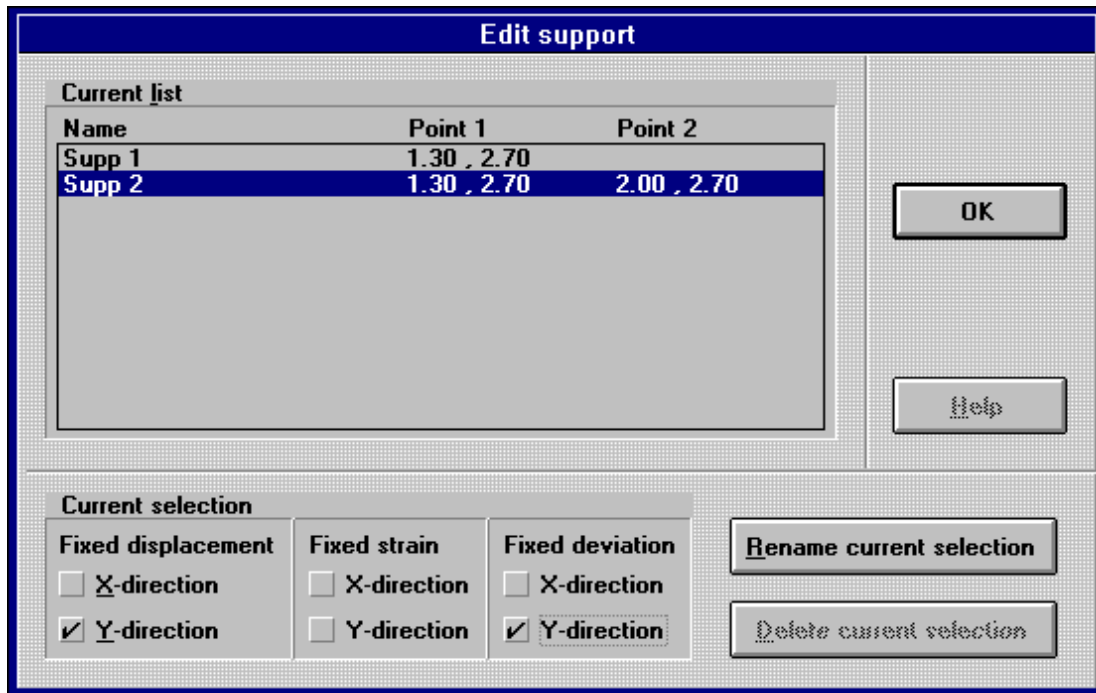


Figure 6.28
Support dialog box.

Current list Identifies all the supports in the project by the name and the position of each support.

The position of a point support is given by the global coordinates of the selected point.

The position of a line support is given by the global coordinates of the two end points of the selected surface.

The support can be renamed by clicking the **Rename** button.

Current selection The support highlighted in the list is the current support. One or more of the parameters described in section 6.4.1.1 can be fixed by marking the parameter. See the examples in section 6.4.1.3.

Supports can be **selected** by clicking the Point or Line support submode button and then clicking the desired point or surface. Note that the structure point or the end points of the surface respectively are greyed.

Supports can be **deleted** by selecting the current support and press the key [Delete] on the keyboard or click the button [Delete object] in the tool bar.

The parameters of a support already added can be **edited** in three different ways:

1. Double click the support to be edited. This will bring up the dialog box immediately.
2. Select the PE menu [Loads] and then the menu item [Edit supports]. If a support has been selected first this will be the current support.

3. Click the Point or Line support submode button and click the **dialog pop up** button. If a support has been selected first this will be the current support.

6.4.1.3 Examples

Example 1: Structure Placed on Totally Stiff Ground.

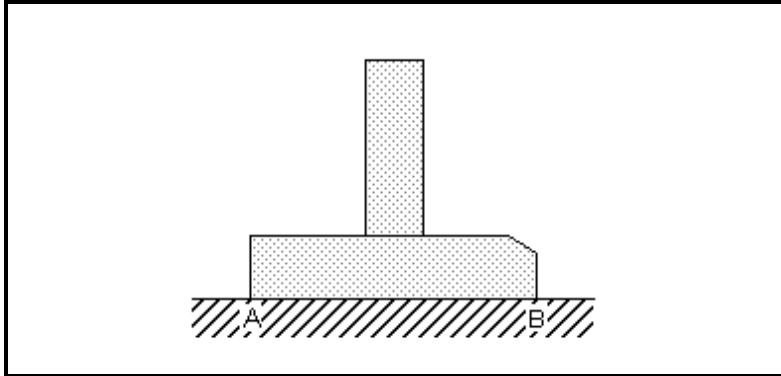


Figure 6.29
Structure placed on totally stiff ground.

The supports chosen to the structure shown in figure 6.29 are:

- Line support at surface from A to B
Fixed displacement in y-direction.
Fixed deviation in y-direction.
- Point support in A (or B)
Fixed displacement in x-direction

Comments:

If deviation in y-direction is not fixed at surface AB, the boundary will be able to "bulge" between FEM-nodes. Usually the distance between the FEM-nodes is so small that this is without importance.

If the point support is left out the structure can perform rigid displacement.

Example 2: Symmetrical Structure.

On a structure that is symmetrical in all conditions, calculations only need to be performed on half of the structure.

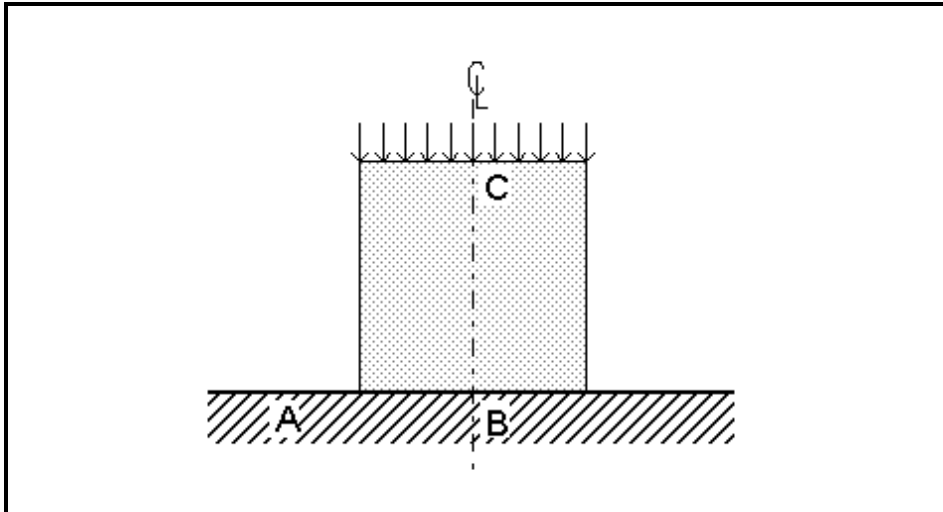


Figure 6.30

Symmetrical structure. The structure is placed on totally stiff ground.

In the symmetry line there will be no displacements in the x- direction. Also the shear stress τ_{xy} will be zero.

The supports chosen to the structure shown in figure 6.30 are:

- Line support at surface from B to C
Fixed displacement in x-direction.
Fixed deviation in x-direction.
Fixed deviation in y-direction.
- Line support at surface from A to B
As described in example 1.

Comments:

The structure can not perform rigid displacement as the displacement in B is fixed in both directions through the line supports.

Example 3: Symmetrical Structure with Antimetrical Load.

Calculations only need to be performed on half of the structure of this type.

Stresses, strains as well as displacements will be antimetrical in this type of structure. This means that displacements in y-direction and strains become zero in the symmetry line.

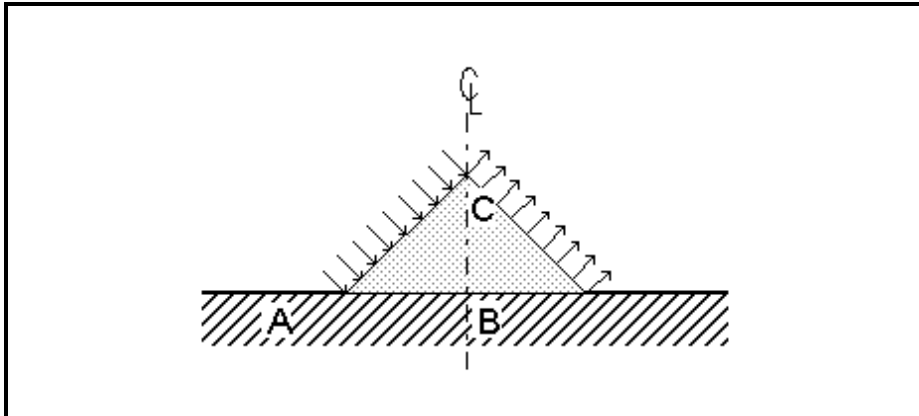


Figure 6.31
Symmetrical structure with antimetrical load. The structure is placed on totally stiff ground.

The supports chosen to the structure shown in figure 6.31 are:

- Line support at surface from B to C
Fixed strain in x-direction.
Fixed strain in y-direction.
Fixed displacement in y-direction.
Fixed deviation in y-direction.
- Line support at surface from A to B
As described in example 1.

Comments:

The structure can not perform rigid displacement as the displacement in B is fixed in both directions through the line supports.

6.4.2 Self-Weight

The self-weight can be turned on and off and the direction of the self-weight can be defined. How to do this is described in section 6.6.9. In default self-weight is turned off.

6.4.3 Point Loads

6.4.3.1 Definition

In 4C-Temp&Stress a point load can be added as either a XY-point load or a Z-point load. A point load can be moved around on the structure in the same way as a cooling pipe, see section 6.2.3.

The **XY-point load** is a load in the xy-plane working over the total thickness of the structure. It is given in [MN]. A XY-point load can be placed at an arbitrary point inside or at the surface of the structure. XY-point loads placed outside the structure are neglected. The direction of the XY-point load is given by the angle with the x-axis taken positive anticlockwise. A XY-point load is positive in the direction defined by the angle with the x-axis. See figure 6.32.

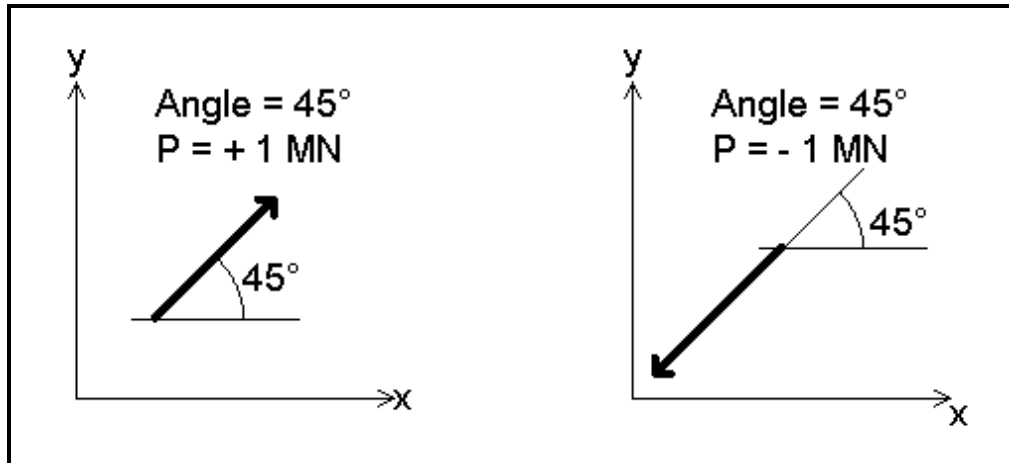


Figure 6.32
Examples of XY-loads.

The **Z-point load** is a point-load in the z-direction (perpendicular to the cross section). Adding a Z-point load corresponds to adding a normal force and two components of a bending moment to the cross section. A Z-point load can be placed anywhere inside or outside the structure. A Z-load is positive corresponding to tension and negative corresponding to pressure at the cross section. It is given in [MN].

The values of both types of point-loads at a given time are calculated as the product of a Factor and a load function. The **load function** is a piecewise constant function of time. The load function is described as a parameter function with a number of (Time [h], Load [MN]) values. The limit of (Time, Load) values is 50, and the last Time value (t_n) must be greater than or equal to the total process time specified in the calculation parameters. See section 6.6 about the total process time. In figure 6.33 an example of a piecewise constant load curve as a function of time is shown.

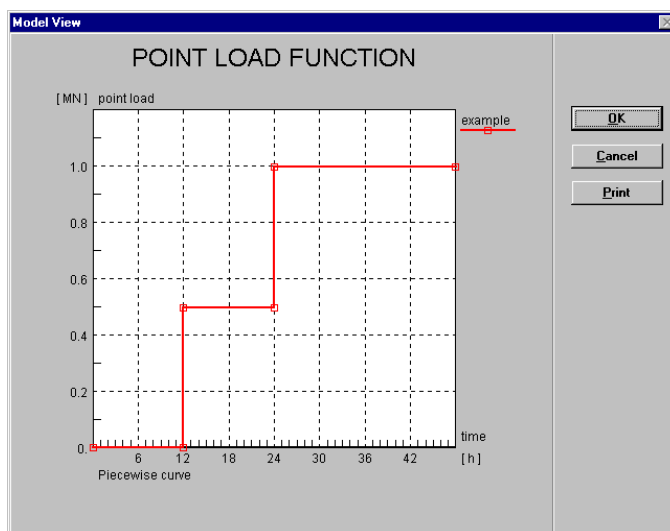





Figure 6.33
A load function.

6.4.3.2 How to Add and Edit a Point Load

1. Click the Point load  button.
2. Choose between XY-load or Z-load by clicking the XY  or the Z  button.
3. Click at the point where the load shall be placed. This brings up the Point load dialog box. The Point load dialog box is divided into four subareas: The current list, the current definition, the function and the coordinates. See figure 6.34.

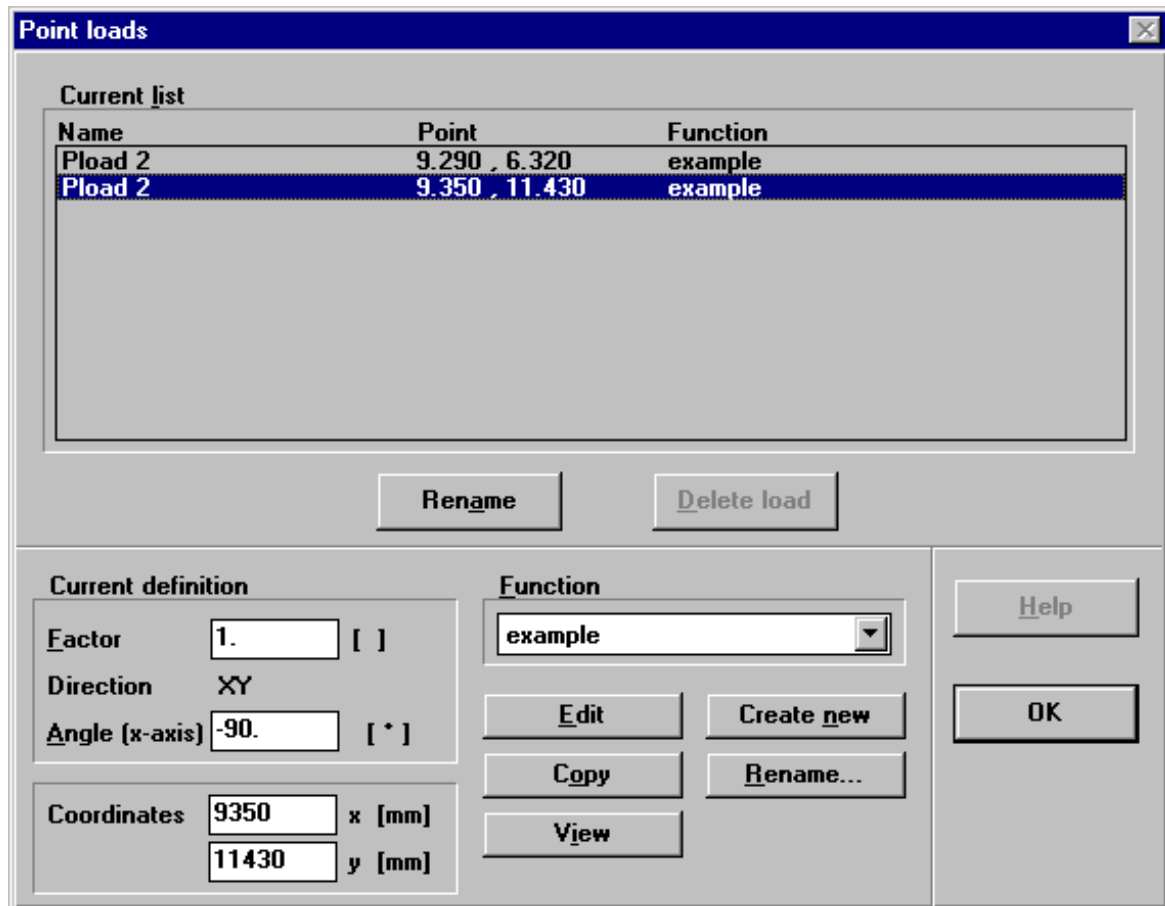


Figure 6.34
Point load dialog box.

Current list Identifies all the point loads in the project by the name, the position and the load function of each load.

The position of a point load is given by the global coordinates of the selected point.

The load can be renamed by clicking the **Rename** button.

Current definition The point load highlighted in the list is the current point load.

Factor: Identifies the factor to be multiplied by the load function.

Direction: Identifies whether the point load is in the XY-plane or in the Z-direction.

Angle: Identifies the angle with the x-axis for XY-point loads.

Function

Click the down arrow button to select the desired load function. If no suitable function is created - click the **Create new** button to create a new function and enter the name of the function in the dialog box that appears.

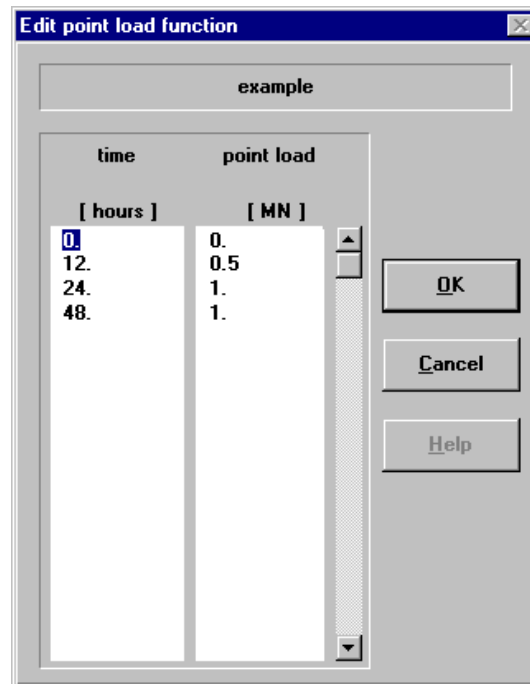


Figure 6.35
Load function dialog box.

When you create a new function or when you want to **edit** the current one the load function dialog box will pop up as shown in figure 6.35. Enter corresponding values of time and load.

The definition of the load function is described in section 6.4.3.1. The function is saved when clicking the **OK** button.

NB! Changing a load function will influence all loads using this function.

The name of the function can be changed by clicking the **Rename** button. The new name will automatically be used for all loads connected to the function as soon as the Point Load dialog box is left.

Coordinates

Defines the position of the point load. The position can be changed by changing the coordinates. The unit corresponds to the unit of the View settings (see section 6.2.1).

The parameters of a Point load already added can be edited in the same way as editing a Point support. Point loads can be deleted in the same way as deleting a support. (see section 6.4.1.2).

Furthermore Point loads can be moved by changing the coordinates or by clicking the point to be moved, keeping the left mouse button down, moving the cursor to the new location and then releasing the button.



6.4.4 Line Loads

6.4.4.1 Definition

A line load is a load in the xy-plane working over the total thickness of the structure. It is given in [MN/m]. Line loads can only be placed at a structure surface. The direction of the line load is given by the angle with the x-axis taken positive anticlockwise. A line load is positive in the direction defined by the angle with the x-axis. See figure 6.32 in section 6.4.3.1.

The value of a line load at a given time is calculated as the product of a Factor and a load function in MN/m as described in section 6.4.3.1.

6.4.4.2 How to Add and Edit a Line Load

1. Click the Line load  button.
2. Select the structure surface to be loaded.
3. Click the add line load  button. This brings up the Line load dialog box. The Line load dialog box is divided into three subareas: The current list, the current definition and the function. See figure 6.36.

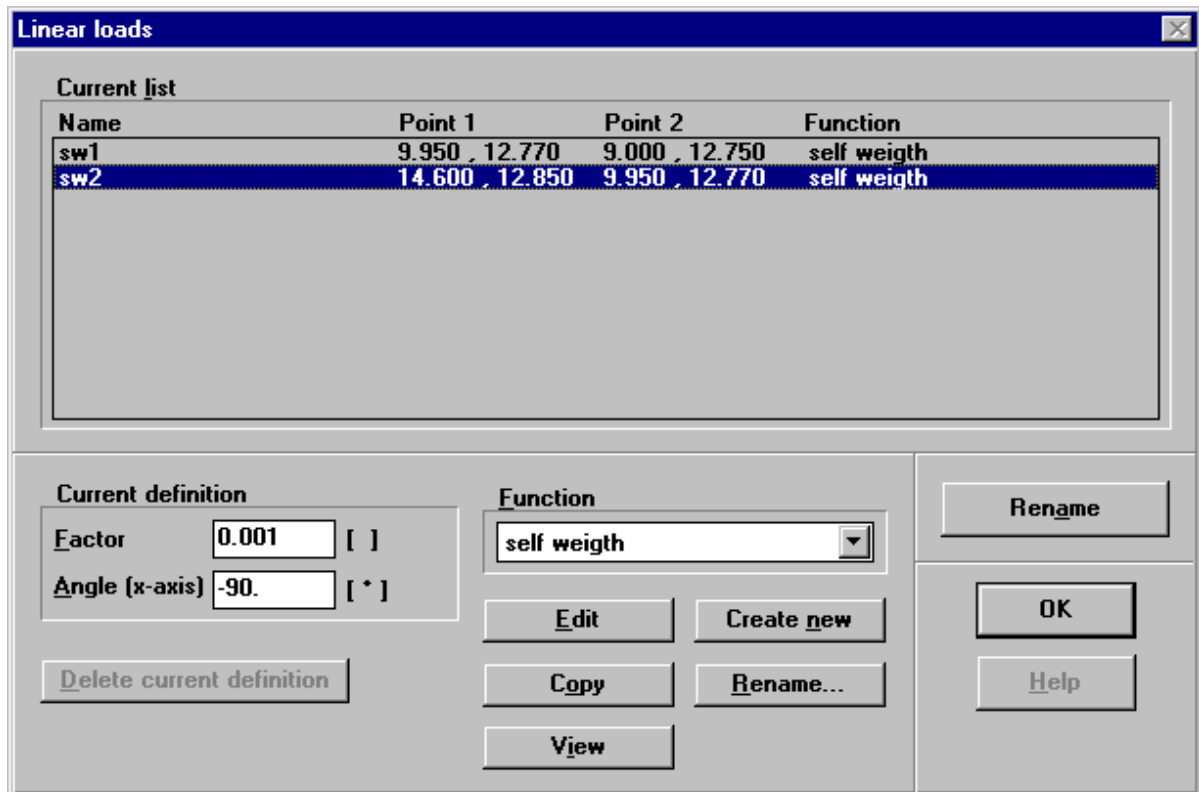


Figure 6.36
Line load dialog box

The Line load dialog box is almost identical to the XY-point load dialog box except that the position of a line load is given by the global coordinates of the two end points of the selected surface. See the description in section 6.4.3.2.

The parameters of a Line load already added can be edited in the same way as editing a Line support. Line loads can be deleted in the same way as deleting a support. (see section 6.4.1.2).

6.4.5 Momentum

6.4.5.1 Definition

The structure can be added a momentum around an axis in the xy-plane working on the total cross section. The momentum must be given as the momentum around the x-axis and the momentum around the y-axis (see figure 6.37).

The value of a X-momentum and Y-momentum respectively at a given time is calculated as the product of a load function in [MNm] and a Factor. See the description of load functions in section 6.4.3.1.

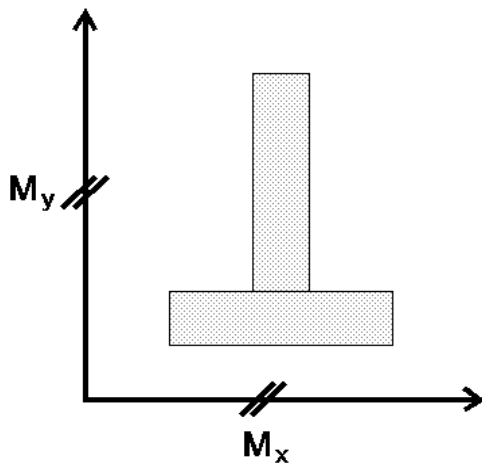




Figure 6.37

6.4.5.2 How to Add and Edit a Momentum

1. Click the X-momentum  or Y-momentum  button. This brings up the X-momentum or Y-momentum dialog box as shown in figure 6.38.

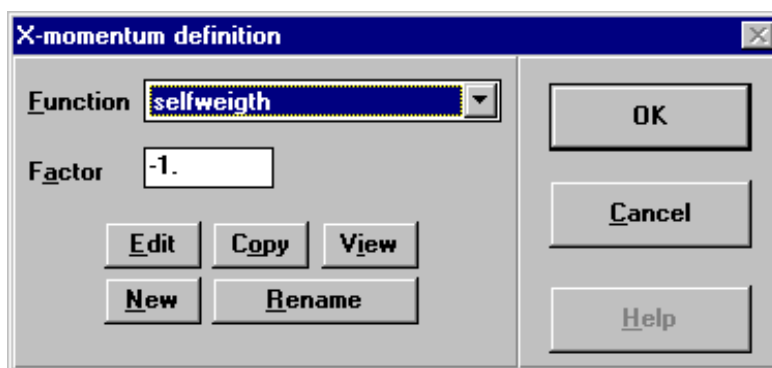


Figure 6.38

X-momentum dialog box.

Enter the Factor and select a load function as described in section 6.4.3.2.

The parameters of a X- or Y-momentum can be edited in the same way as adding a momentum. Otherwise select the PE menu [Loads] and then the menu item [Edit X-momentum] or [Edit X-momentum]. A general X- or Y-momentum can be deleted by setting the Factor to zero.

6.4.6 Examples

Example 1: Prestressed Beam

The stresses in the middle cross section of a simply supported beam are desired. The dimensions of the beam are 0.5 x 1.0 x 5.0 m. In 4C-Temp&Stress the structure is modelled as a rectangular cross section with dimensions 0.5 x 1.0 m.

The beam is prestressed with 2 MN at 48 hours after casting. At 72 hours after casting the form work is removed. This releases a momentum M_g due to self-weight. The density of the concrete is 2300 kg/m³. In the middle cross section M_g is calculated as:

$$M_g = \frac{1}{8} \cdot p_g \cdot l^2 = \frac{1}{8} \cdot \rho \cdot w \cdot h \cdot g \cdot l^2$$

$$= \frac{1}{8} \cdot (2300 \text{ kg/m}^3 \cdot 0.5 \text{ m} \cdot 1.0 \text{ m} \cdot 10 \text{ m/s}^2) \cdot (5 \text{ m})^2 = 0.0359 \cdot 10^6 \text{ Nm}$$

M_g is added as a X-momentum with Factor = -1. The load function is shown in figure 6.39. The X-Momentum is negative in order to get tension in the underside.

Notice that the self-weight included in 4C-Temp&Stress must be turned off.

The Prestress force is added as a Z-Point-Load with Max value = -1 and load function as shown in figure 6.40.

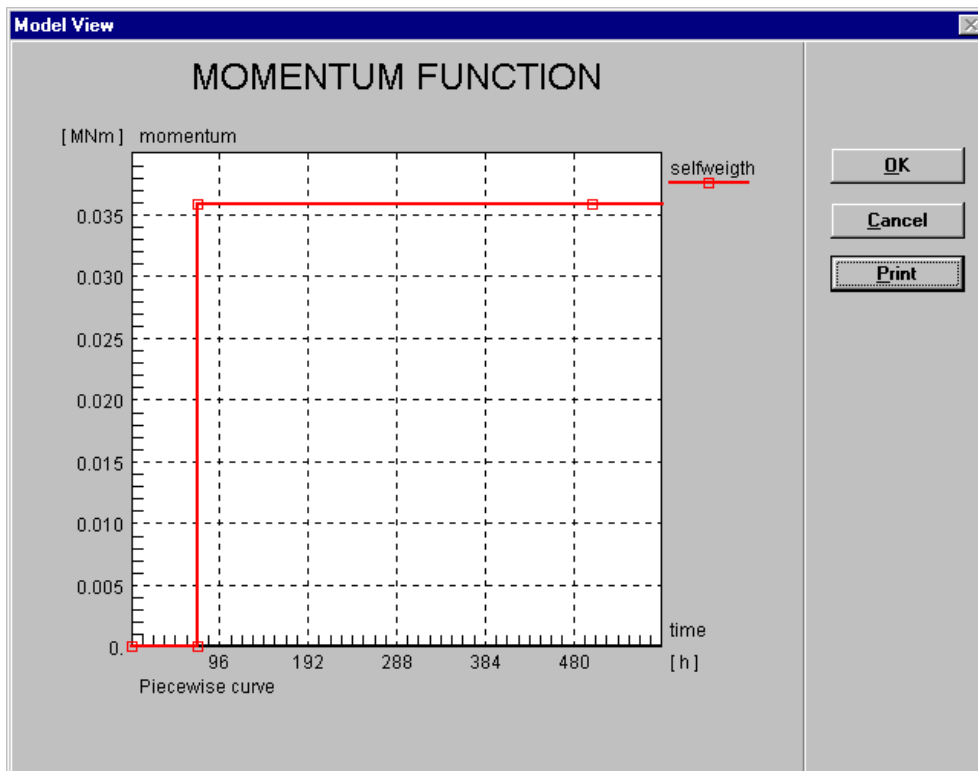


Figure 6.39
X-Momentum function

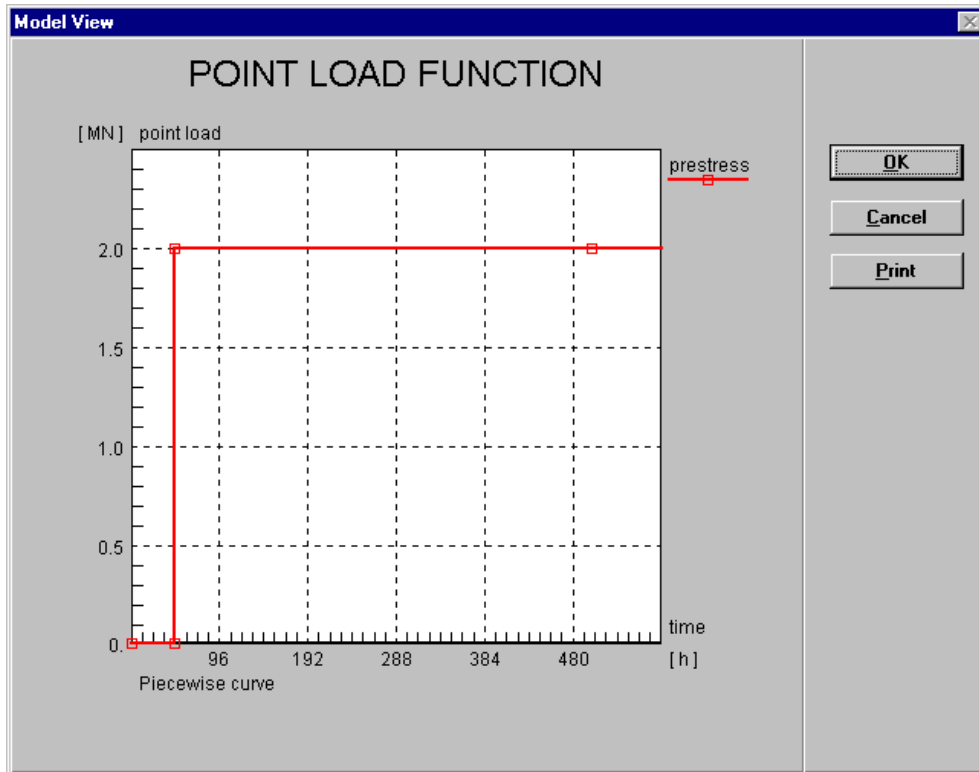


Figure 6.40
Load function for Z-point load.

6.5 Text Mode

This section will help you understand the structure of the Text Mode in the project editor. Click the text mode button in the vertical tool bar to select the mode. In Text mode it is possible to add text on arrows and text in boxes to the drawing shown in Volume mode. This section provides step by step instructions to help you outline the text part of your project.

6.5.1 General

The Text mode contains two different writing modes that may be selected from the horizontal toll bar in the PE. You must select one of these submodes before you can write a text.



Text on arrow



Text in box.

With these two writing modes it is possible to add text to the drawing of a structure as shown in figure 6.41. The possibilities in the two writing modes will be described in the following sections.

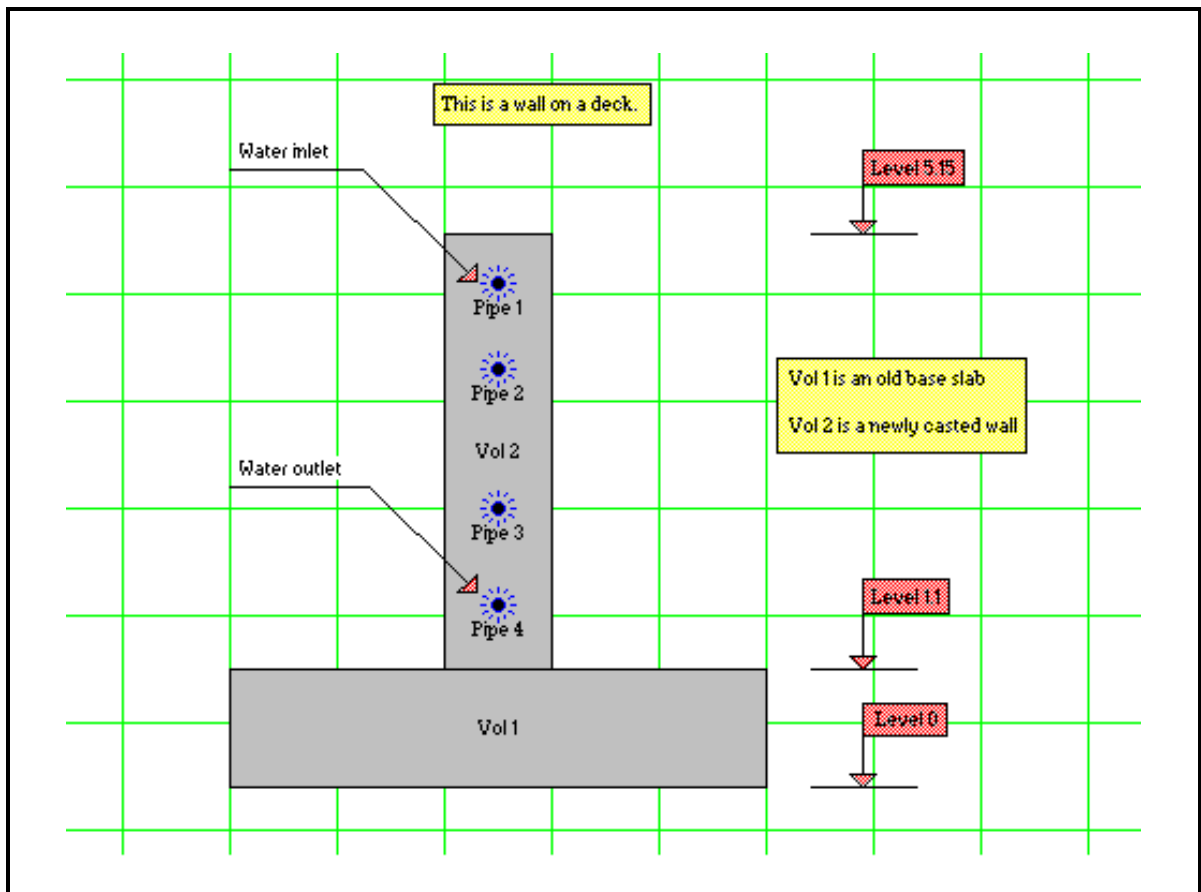



Figure 6.41
Example of the possibilities in Text Mode.

6.5.2 Text on Arrow

This writing mode makes it possible to write text in connection with a drawn line. The line may be with or without an arrow in the end. How to create such a text is described in the following:

1. **Select** the submode button Text on arrow .
2. **Draw the line/arrow** by placing the cursor at the starting point of the desired line/arrow. The arrow will be drawn at the starting point. Click the left mouse button and keep it down, move the cursor to the ending point of the line/arrow and release the left mouse button. This makes the Edit Project Text dialog box appear.
3. **Write the desired text** in the project text field and define the type and placement of the text and the line/arrow. See the Edit Project Text dialog box in figure 6.42 and the description below.

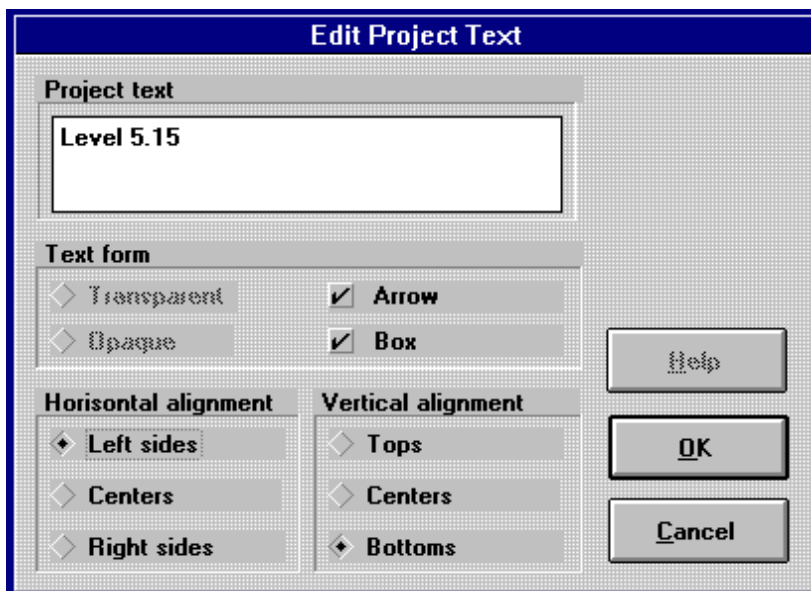


Figure 6.42
Edit Project Text dialog box.

The Edit Project Text dialog box is divided into three subareas: Project text, Text form and Horizontal/vertical alignment.

Project text Identifies the text on the line/arrow. Place the cursor in the field and write the desired text (max 30 characters).

Text form Identifies the form of the drawn line and the written text.

Arrow: A tickmark in the square identifies that an arrow will be drawn at the starting point of the line. If no tickmark is visible no arrow will be drawn on the line.

Box: A tickmark in the square identifies that the text will be written in a rectangular box. On the screen the background colour in this box is red. If no tickmark is visible the text will be written without any box.

Alignment Identifies the horizontal and vertical placement of the text at the end point of the line/arrow. Horizontally the text can be placed so that the end point of the line/arrow is placed either on the left side, on the centre or on the right side of the rectangle that includes the text. Similar the text can be placed so that the end point of the line/arrow vertically is placed either on the top, on the centre or on the bottom of the rectangle that includes the text. See figure 6.43.

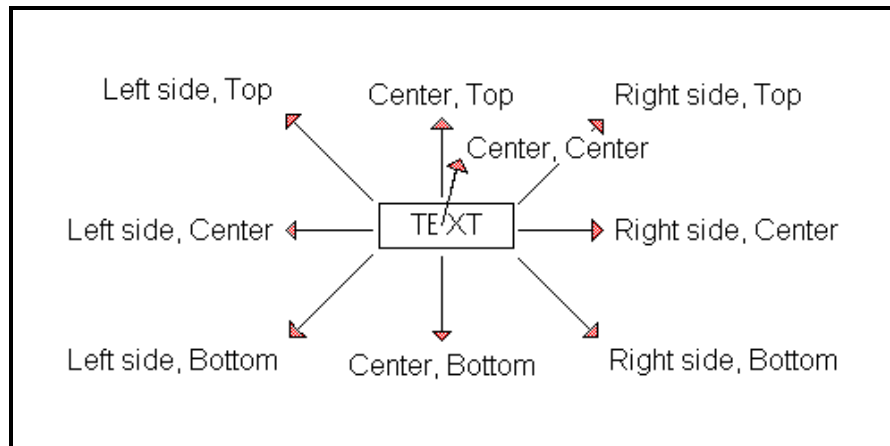


Figure 6.43
The combinations of horizontal and vertical alignments to connect the end point of the line/arrow with the text field.

As shown in figure 6.41 this writing mode can be used to write texts like levels of the structure and in- and outlet of the cooling water. In this case the possibility of drawing a line without any text or arrow together with the facilities in text form and horizontal/vertical alignments have been used in different combinations.

All written text in text mode can be **scaled up and down** by selecting the menu item [Options] and the submenu items [Scale font up] or [Scale font down]. The box around the text will automatically be scaled up together with the text but the line/arrow connected to the text will keep unchanged.

A drawn line/arrow can be **moved** in the same way as a volume. Select the line/arrow with a click on the left mouse button when the cursor is above the line/arrow. The nodes that extend the line/arrow will become grey after selecting the line/arrow. Keep the mouse button down and move the cursor to the new location and release the button. When a line/arrow is selected the text can be **edited** by **double clicking** on the line/arrow. This will bring up the Edit Project Text dialog box.


The length and direction of a drawn line/arrow can not be changed. In these cases it is necessary to draw a new line/arrow and delete the old one. The text from the old line/arrow can be **copied** to the new line/arrow as described in the following:

- Double click on the old line/arrow so that the Edit Project Text dialog box appears.
- Highlight the text with the mouse, keep the [Ctrl] key on the keyboard down and press the [Insert] key on the keyboard. This will copy the selected text to the Clipboard. Leave the dialog box.
- Draw the new line/arrow and place the cursor in the Project text field in the Edit Project Text dialog box.

- Keep the [Shift] key on the keyboard down and press the [Insert] key on the keyboard. This will copy the text from the Clipboard into the project text field.
- Click the OK button and a new line/arrow with the old text has been created.
- Select the old line/arrow and press the [Delete] key on the keyboard.

6.5.3 Text in Box

This writing mode makes it possible to write text without any line or arrow connected to the text. The text may be placed in a box or it may be without the box. How to create such a text is described in the following:

1. **Select** the submode button Text in box .
2. **Draw the rectangle** that surrounds the desired text by placing the cursor at the upper left corner of the desired rectangle, click the left mouse button and keep it down, move the cursor to the bottom right corner of the desired rectangle and release the left mouse button. This makes the Edit Project Text dialog box appear.
3. **Write the desired text** in the project text field and define whether the text shall be in a box or not. See the Edit Project Text dialog box in figure 6.44 and the description below.

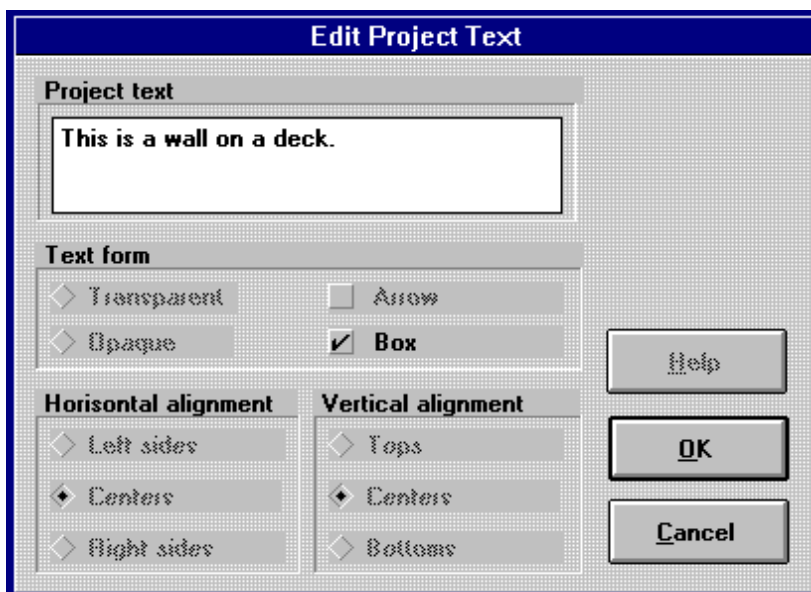


Figure 6.44
Edit Project Text dialog box.

The Edit Project Text dialog box is divided into three subareas: Project text, Text form and Horizontal/vertical alignment.

Project text Identifies the written text. Place the cursor in the field and write the desired text (maximum 80 characters). The text may be written in more than one line. To change line you must keep the [Ctrl] key on the keyboard down and press the [Enter] key on the keyboard. The size of the box around the text will automatically be generated from the amount of written text. The text can be copied to other boxes in the same way as the text on arrows. See section 6.5.2.

Text form Identifies whether the text is in a box or not.

Box: A tickmark in the square identifies that the text will be written in a rectangular box. On the screen the background colour in this box is yellow. If no tickmark is visible the text will be written without any box.

Alignment Identifies the horizontal and vertical placement of the text in the box. The placement of the text is always in the centre of the box.

All written text in text mode can be **scaled up and down** by selecting the menu item [Options] and the submenu items [Scale font up] or [Scale font down]. The drawn box around the text will keep unchanged which means that the box can be too small for the written text if the text is scaled up. Then the text will not be visible on the screen.

A written text can be **moved** in the same way as a volume. Select the text with a click on the left mouse button when the cursor is above the text. The nodes that extend the rectangle around the text will become grey after selecting the text. Keep the mouse button down and move the cursor to the new location and release the button. When a text is selected the text can be **edited** by **double clicking** on the text. This will bring up the Edit Project Text dialog box.

6.6 Calculation Parameters

The geometry, the thermal and the static conditions of a project describe the type of project on which 4C-Temp&Stress is calculating. Further, it is necessary to describe how 4C-Temp&Stress are going to execute the calculations and the required accuracy of the results. This is described by some calculation parameters. This section describes the calculation parameters and how to setup the parameters in different types of calculations.

6.6.1 General

The calculation parameters include all the specific parameters that are required by the calculations. All calculation parameters are placed in one dialog box called calculation parameters dialog box. To open this dialog box click the menu item [Calculation] in the PE menu and the menu item [Parameters] in the menu bar. This will bring up the calculation parameters dialog box as shown in figure 6.45.

Figure 6.45
Calculation parameters dialog box.

The calculation parameters dialog box is divided into eight areas that separate different groups of parameters. The eight groups of parameters include *Thermal analysis*, *Stress analysis*, *Dimensions*, *Time specifications*, *Mesh node generation*, *Non-linear calculations*, *Circles* and *Self weight*.

The following subsections describe each group and the parameters.

6.6.2 Thermal Analysis

Click one of the items Transient or Stationary to select the desired thermal analysis form of the calculations. If no temperature calculation is desired the selected analysis form will be ignored. The basic idea of the thermal analysis form is described in section 2.

Transient Identifies that the calculated temperatures are as a function of place and time. In this case conditions are allowed to vary as a function of maturity.

Stationary Identifies that the calculated temperatures correspond to the temperatures which will appear after a very long time. In this case all conditions must be constant. This analysis form is not implemented in 4C-Temp&Stress yet.

6.6.3 Stress Analysis

Click one of the items Based on thermal results or Not based on thermal results to select the desired stress analysis form of the calculations. If no stress calculation is desired the selected analysis form will be ignored. The basic idea of the stress analysis form is described in section 2.

Based on thermal results	Identifies that the calculated stresses will be based on results from a previous thermal calculation including thermal expansion and material properties which are functions of maturity.
Not based on thermal results	Identifies that the calculated stresses will be based on constant material properties and not on results from a previous analysis. Only external loads are applied.

6.6.4 Dimensions

Click one of the items 2-dimensional or 2½-dimensional to select the desired dimension of the calculations. The basic idea of the dimensions is described in section 2.1.

2 dimension	Identifies that the geometry is treated like a disk.
Plane stress	The geometry is treated like a disc in a plane stress situation.
Plane strain	The geometry is treated like a disc in a plane strain situation.
2½ dimension	Identifies a plane strain situation where it is possible to specify cross-sectional load and support conditions.
	If "2½-dimension" is specified an approximated stress analysis - as described in section 2.1. - is performed together with the temperature analysis. This facility is primarily used in analysing hardening concrete structures.
	Select the desired specifications which concern requirements to the strain plane:
No strain in z-direction	The strain contribution arising from deformation is zero at origo.
No rotation around x-axis	The curvature around the x-axis is zero.
No rotation around y-axis	The curvature around the y-axis is zero.

NB! The requirements to the strain plane concerns the total cross-section.

If for instance a tunnel cross-section is considered the separate foundations will be forced to work together also at the time where topslab and walls have not yet been cast. Unless the foundations are identical and placed at the same level this is not correct. In this case the two foundations should be calculated separately.

6.6.5 Time Specifications

Enter the desired time specifications that will be used in the calculations.

Total process time	The time period in which the calculation will take place [h].
---------------------------	---

Time step, intended The "total process time" is divided into a number of intervals. The parameter specifies the length of the interval [h]. In the case of thermal analysis the intended time step will automatically be subdivided when modifications in the thermal boundary conditions are observed.

In the case of hardening concrete structures a one-hour time step will usually be sufficient.

In calculations which are time-independent the time step should be set equal to the total process time.

Time step factor Identifies the "Integration method". Depending on the time step factor, λ , different integration methods can be performed:

$\lambda = 0.0$ forward difference

$\lambda = 0.5$ central difference

$\lambda = 1.0$ backward difference

Usually $\lambda = 0.5$ will be convenient.

If the structure is exposed to thermal shock, then oscillations may occur. Numerical damping can be introduced by using an λ slightly different from 0.5.

6.6.6 Mesh Node Generation

The structure is automatically divided into triangular elements. These elements are able to describe a quadratic temperature- and stress-field. The validity of the results depends on the element mesh. The field which can be described by a certain mesh, has to be able to describe the solution of the specified problem.

For a given physical problem a minimum density of the element-mesh is required. Five parameters control the density of the element-mesh. If two calculations are performed on different element-meshes, equal results indicates that the, most rough mesh is fine enough, in the present case. If the results are different no conclusions can be made.

The validity of the calculation can also be demonstrated by an evaluation of the obtained results. The calculated temperature and stress isocurves have to be continuous and differentiable.

When computing stress calculations in connection with mesh generation as well as result evaluation pay attention to singular points. Singular points occur in connection with:

- a single-force or support affecting the structure
- re-entrant corners

In a singular point the stresses are locally extremely high. The finite element method cannot handle such stresses and they will not occur in practice. In practice the stresses are reduced by a non-linear behaviour of the material (e.g. formation of cracks). When evaluating the results do not attach to great importance to the results in the singular point areas, but concentrate on the stresses in other areas. These results are independent of the local stress concentrations in the singular points. (Saint Venant's principle.)

If the stress result to be evaluated is situated close to a singular point, the size of the area affected by stress concentration in the singular point has to be examined in each individual case. If an increased element graduation around a singular point does not change the stress result in a given point, such point is outside the area affected by the stress concentration.

When searching for extreme results by showing minimum and maximum results pay attention to that the stress concentration of the singular points can be handled as being either minimum or maximum results.

In volumes of hardening concrete a fine mesh is generated along the border of the volumes. This makes it possible with a rougher mesh in the rest of the structure to describe the temperature- and stress field in volumes of hardening concrete, when these are exposed to thermal shocks such as a sudden removal of the insulation.

NB! Although a structure is clearly symmetrical, the automatically generated finite element mesh might not be symmetrical.

The following five parameters are all defined as percentage of the largest extent of the total structure.

Min. distance to border	Nodes are not generated at a distance from the border of the structure smaller than the specified value.
Density, internal nodes	Density of the mesh which is basis for the generation of internal nodes.
Density, border nodes	Nodes are generated at the border by the specified density.
Density around c-pipes	In a circular area around cooling-pipes, heating wires and concentrated loads the density can locally be increased to the specified value.

Radius around c-pipes This parameter describes the extent of the area with an increased node-density.

NB! The distance between the nodes of the elements must be greater than 1 mm.

6.6.7 Non-Linear Calculations

In calculations including creep a convergence criterion must be defined.

In a given time step the creep strains are changed by $\Delta\epsilon_{\text{creep}}$. In each time step the size of $\Delta\epsilon_{\text{creep}}$ is determined by an iterative solution of the non-linear equations described in section 7.2.10. A satisfying result is said to be reached when the norm of the changes in $\Delta\epsilon_{\text{creep}}$ among two iterations relative to the norm of $\Delta\epsilon_{\text{creep}}$ is lower than the convergence criterion defined. The norm is defined as the square root of the sum of squares of $\Delta\epsilon_{\text{creep}}$ over the cross section.

If the convergence criterion chosen is too low it never will be fulfilled on a given time step. 4C-Temp&Stress will then ask you to use a higher criterion. Before changing the convergence criterion it is recommended to redo the calculation using a smaller time-step and/or a finer element-mesh.

The validity of the results depends on the convergence criterion. If the convergence criterion chosen is too high the calculated creep stresses in a given time step will not be correct. This makes the total stresses wrong. Furthermore the error will sum up during time as the creep stresses depends on the absolute level

of the total stresses. If two calculations are performed with different convergence criteria, identical results indicate that the highest criterion is low enough, in the present case.

Enter the desired convergence criterion that will be used in the calculations including creep. If no calculations including creep are desired the entered convergence criterion will be ignored.

As a start value the convergence criterion may be set at 10^{-3} .

6.6.8 Circles

As described in section 6.2.2 circular volumes will be calculated as polygons with a certain number of faces. The necessary number of faces of the polygon depends on the required accuracy of the results.

Notice that the cross section area is always smaller than the drawn circle but with a number of faces larger than 12 the difference between the areas will be acceptable in most types of calculations.

Enter the desired number of faces to be used on circles. If no circles are included in the calculations the entered number of faces will be ignored.

As a start value the number of faces may be set at 12.

6.6.9 Self Weight

Self weight can be taken into account or neglected. In default it is neglected. When a tick mark is placed self weight is taken into account. In this case the direction of the self weight must be specified by a directional vector.

Direction x The x-component of the directional vector.

Direction y The y-component of the directional vector.

6.7 Project Items

All the items necessary to handle the projects in the project editor are included in the menu item [Project]. In this section the menu item [Project] will be described and all the items to handle the projects like loading, saving and printing the project will be introduced.

Project	Calculation	Volumes	Surf
<u>N</u>ew project...			
<u>L</u>oad project...			
<u>S</u>ave project			
<u>S</u>ave project <u>a</u>s...			
<u>S</u>etup project			
<u>P</u>rint subject			
<u>P</u>rint functions			
<u>P</u>rint calculation parameters			
<u>P</u>rint project...			
<u>R</u>esult viewer...			
<u>E</u>xit			Alt-X

Figure 6.46

Menu item Project.

Figure 6.46 shows the menu bar that appears when you click the menu item [Project] in the PE menu. Each of the menu items is a part of the project handling in 4C-Temp&Stress, except the two last menu items: [Result viewer] which opens the RV and [Exit] which makes you exit the 4C-Temp&Stress program. In the following sections each of the remaining menu items in the Project menu bar will be described.

6.7.1 New Project

When you want to create a new project in the PE, select the menu item [New project].

This will load the project called DEFAULT.SET which is a default project containing the desired standard view and options in 4C-Temp&Stress. The loaded project will be unnamed when it is loaded into the PE which protects the default project from being overwritten.

The standard view and options which are loaded when you select [New project] can be changed as often as you wish by doing the following:

- **select** the menu item [New project]
- define the standard view and options like in an ordinary project
- **select** the menu item [Save project as], see the dialog box in figure 6.46
- place the cursor in the project file field, delete *.prj, type **default.set** and click the **OK** button
- **select** the menu item [New project] to protect the default project from being overwritten.

6.7.2 Load Project

When you want to load an old project into the PE, select the menu item [Load project].

This will bring up the load project dialog box as shown in figure 6.47. Load the current project in one of the following ways:

- **double click** on the project name in the project list, or
- **click** on the project name in the project list and then click the **OK** button, or
- **type** the name of the project file - with the suffix .PRJ - that you want to load and click the **OK** button.

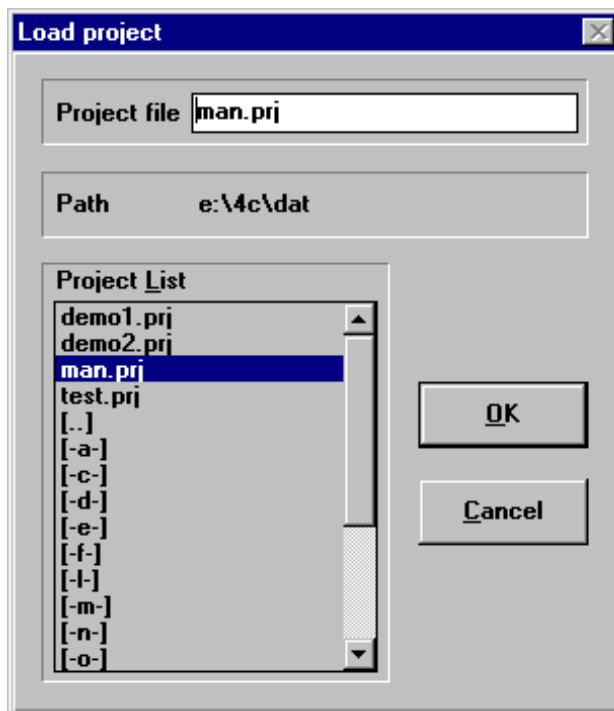


Figure 6.47
Load project dialog box.

6.7.3 Save Project

When an already existing project have been changed you may save the changes by selecting the menu item [Save project]. This will overwrite the existing project with the changed project. If you do not want to overwrite the existing project you may select the menu item [Save project as] instead.

6.7.4 Save Project As

When you want to save a project under a new name select the menu item [Save project as].

This will bring up the save project as dialog box as shown in figure 6.48. Save the current project in one of the following ways:

- **double click** on one of the project names in the project list - this will bring up a dialog box that asks you if you want to overwrite the already existing file, or
- **click** on one of the project names in the project list and then click the **OK** button - this will bring up a dialog box that asks you if you want to overwrite the already existing file, or
- **type** the new name of the project file - with the suffix .PRJ - that you want to save and click the **OK** button.

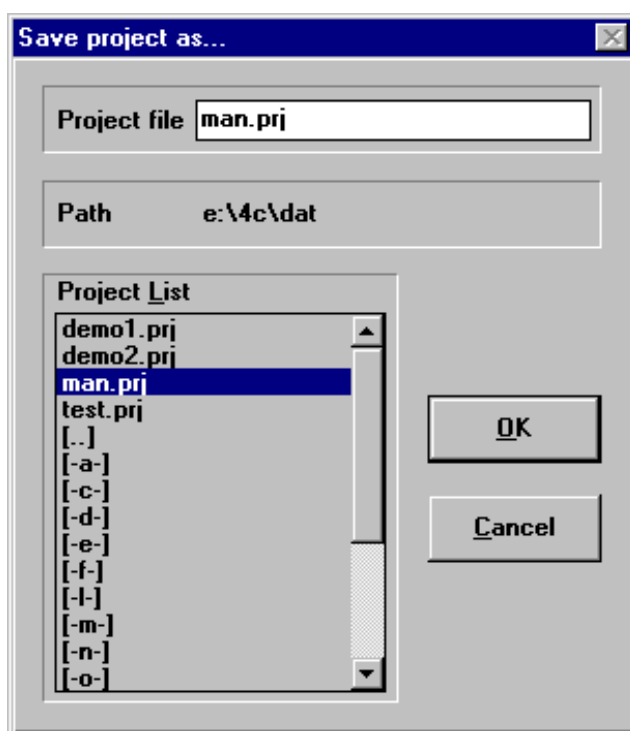


Figure 6.48
Save project as dialog box.

NB! The project name may consist of maximum 8 characters and it must have the suffix .PRJ. The 8 characters may be letters, numbers and other symbols like hyphen (-) and percent (%).

The characters may not be the letters: æ, ø and å or symbols like space (), full stop (.), question mark (?) or asterisk (*).

6.7.5 Setup Project

Before you start the calculations or print out any of the information about a project you may setup some information about the project. Such information is printed on all documentation sheets connected to the current project. Select the menu item [Setup project]. This will bring up the Project setup dialog box as shown in figure 6.49.

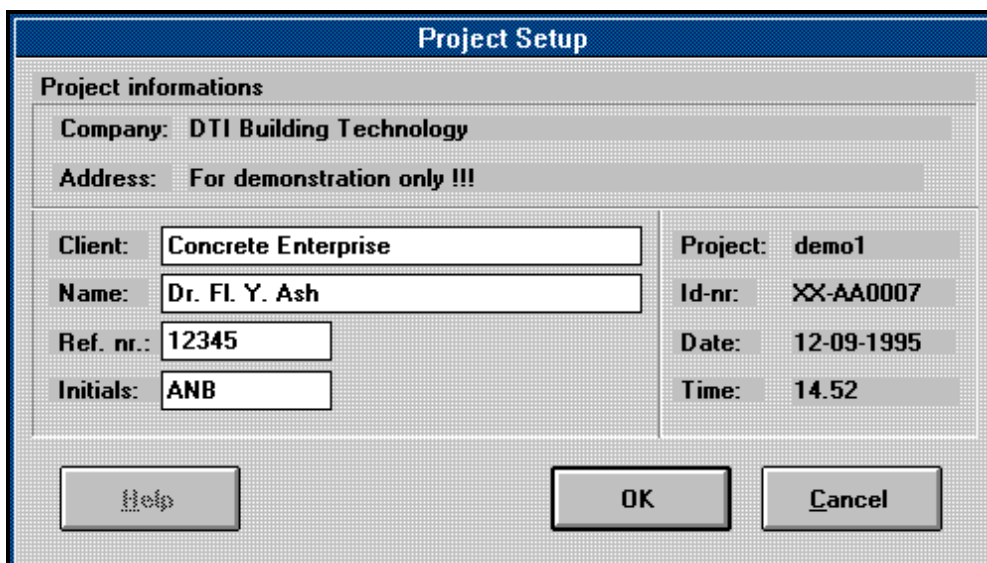
Parts of the project information are generated automatically, other types of information have to be typed in.

Company Identifies the name of the company who is the owner of the 4C-Temp&Stress program. This information is generated automatically.

Address Identifies the address of the company who is the owner of the 4C-Temp&Stress program. This information is generated automatically.

Client Identifies the name of the company who is the client in the case. This information you may type in.

Name Identifies the name of the contact person at the client company. This information you may type in.



The screenshot shows a 'Project Setup' dialog box with the following fields and values:

Field	Value
Company	DTI Building Technology
Address	For demonstration only !!!
Client	Concrete Enterprise
Name	Dr. Fl. Y. Ash
Ref. nr.	12345
Initials	ANB
Project	demo1
Id-nr.	XX-AA0007
Date	12-09-1995
Time	14.52

Figure 6.49
Project setup dialog box.

Ref. no. Identifies the reference number in the current case. This information you may type in.

Initials Identifies the initials of the person who has created the current project. This information you may type in.

Project Identifies the name of the file in which the current project is saved. This information is generated automatically.

Id-no. Identifies the identification number of the current project. This number is a security number to ensure that all printed documentation sheets are connected to the same version of the project. The number is automatically updated each time the project is saved, calculated or printed if something in the project have been changed.

Date Identifies the date where the documentation sheet has been printed. This information is generated automatically.

Time Identifies the time where the documentation sheet has been printed. This information is generated automatically.

6.7.6 Print Project

All elements of the described project may be printed out as documentation for the temperature and stress calculations.

The three menu items in the [Project] menu bar: [Print subject], [Print functions] and [Print calculation parameters] are used to print all types of information about the current project except the used materials from the material database which are printed out directly from the database as described in section 7.2, 7.4, 7.5 and 7.6.

[Print subject] With this menu item you may print out the subject which is shown on the screen at the moment where you select this item. This means that a documentation sheet may be printed for each of the different object modes in Volume, Load and Text mode. The different types of documentation sheets are:

- Volumes, including size, material type, material name, thickness, start time and start temperature
- Faces, including name of boundary condition and name of functions
- Nodes, including faces, diameter and boundary condition for cooling pipes and heating wires
- Supports, including fixed displacements, fixed strains and fixed deviations for point and line supports.
- Loads, including max. load, angle and load function for point and line loads and momentum
- Texts, including volumes with size, material type, material name, thickness, start time and start temperature.

[Print functions] With this menu item you may print out all the different types of functions used in the project. All types of functions will be printed out independent of the selected object mode. The different types of functions are:

- temperature
- wind velocity
- shield
- heat transfer coefficient
- flux
- radiation
- load factor

[Print calculation parameters] With this menu item you may print out all the specified calculation parameters in the project.

When you select one of the three printing items the Print dialog box will pop up on the screen as shown in figure 6.50. The Print dialog box is a standard windows dialog box which is used in a lot of different programs. Therefore it is very general and some of the items are greyed because they are not used in 4C-

Temp&Stress. Select always the Print range **All** because neither Selection nor Pages are used in 4C-Temp&Stress.

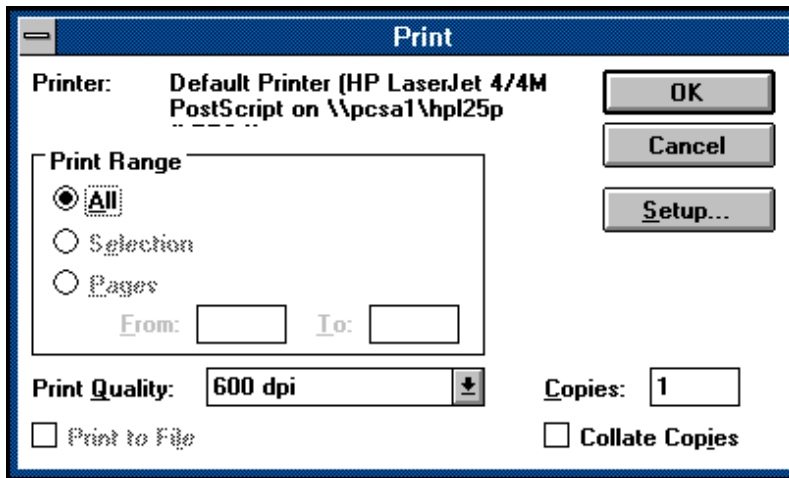


Figure 6.50
Print dialog box.

The current printer type can be changed and specified by selecting the **Setup** button. Further, the printing quality can be selected by clicking on the down arrow button and select the current quality.

7. Material Database

This section describes the structure of the material database in the PE and how to define, edit and update a material in the database.

7.1 Database Structure

The database which is common to all the defined projects contains all necessary information about the used materials in the project. The database consists of several sub-databases that together covers almost all types of materials. You can select each sub-database by clicking the menu item [Database] and then click the menu item with the name of the sub-database: [Concrete], [Material], [Soil], [Insulation] or the concrete constituents: [Cement], [Aggregate], [Mineral additive.], [Chem. Admixture.], [Others] or [Water].

The databases are described in the following sections.

7.2 Concrete Database

Select the menu item [Concrete] in the menu bar. The concrete database dialog box will then pop up as shown in figure 7.1. The concrete database dialog box includes a list of all the defined concretes in the database and a list of property buttons for all the necessary properties you may define for each concrete. The selected concrete is highlighted in the current list. To the left of each property button there is a square button that shows whether the property is defined for the current concrete or not. If the checkmark is visible the property is defined.

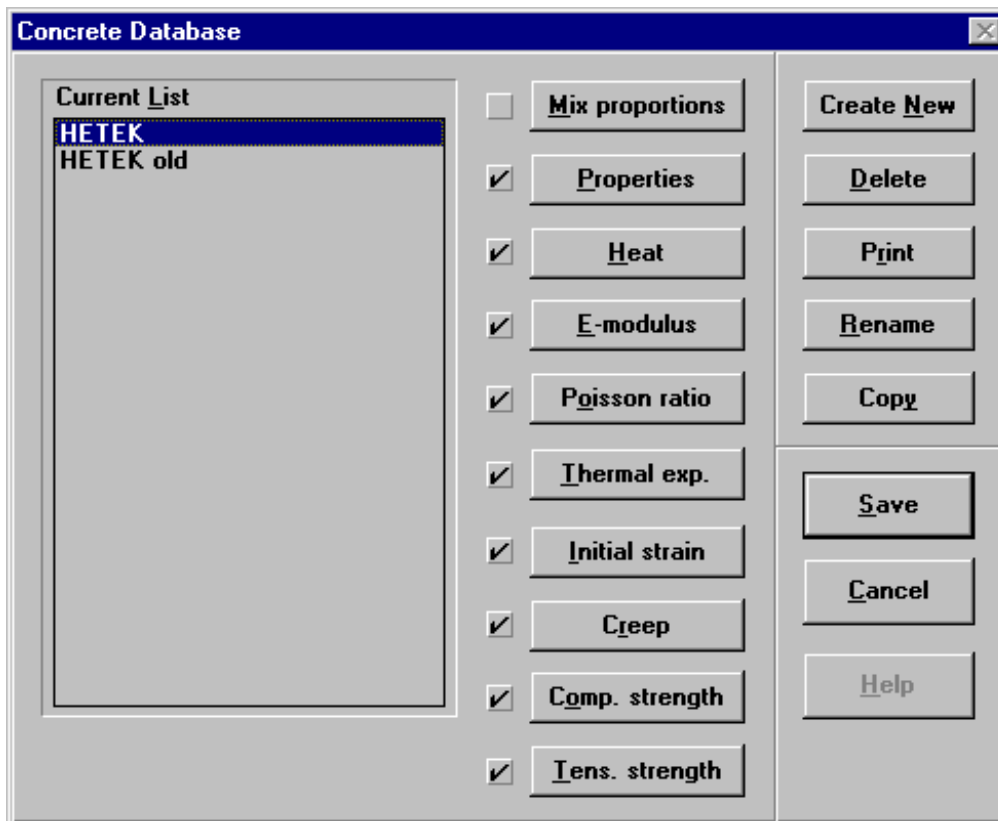


Figure 7.1
Concrete database dialog box.

A new concrete may be defined by clicking the **Create new** button and enter the name of the new concrete in the dialog box that appears. The already existing concretes may be **Renamed**, **Copied**, **Deleted** or **Printed** on paper by selecting the current buttons.

The properties of the selected concrete may be defined and edited by clicking the button of the current property and the dialog box of the current property will pop up. In the following each of the property dialog boxes will be shown and described.

7.2.1 Mix Proportions

Select the **Mix Proportions** button to define what kind of materials and matching contents that go into the current concrete. **NB!** Mix Proportions do not need to be defined.

When you have selected the **Mix proportions** button the Mix proportions dialog box will pop up as shown in figure 7.2. The Mix proportions dialog box is divided into three areas: The current list, the database and the current constituent.

Concrete constituents	Content
PC (R/IS/MA/G)	371.1
Rønne granit 0/4	669.2
Granit 4/8	340.1
Granit 8/25	797.3
Public supply	153.1
Sika Aer 15	0.9
Plastiment A40	2.2

Figure 7.2
Mix proportions dialog box.

Current list

The current list includes all the selected concrete constituents together with the specified content [kg] of each constituent. When you select a concrete constituent in the database and click the **Add to current list** button it will be added to the current list. A concrete constituent can be deleted from the current list by clicking the **Remove constituent** button.

Database

The database is based on selections from the Concrete constituents database. Anyhow, it is possible to create a new concrete constituent inside the concrete database by clicking the **Create new** button, but you have to enter the appropriate concrete constituent database to edit an existing constituent. See section 7.3.

First you have to select the **Type** of concrete constituent you want to add to the current list: Click the down arrow button to get a list of all the Concrete constituent sub-databases and select the database that you want to pick a constituent from. Click for instance the cement database.

Secondly you have to select the **Name** of the concrete constituent you want to add to the current list: Click the down arrow button to get a list of the concrete constituent names in the sub-database and select the name you want to add.

When you have selected both the Type and the Name of the concrete constituent click the **Add to current list** button and the concrete constituent will be shown in the current list. The cursor will be placed in the **Content** field in the current constituent area where you have to type the desired content of the concrete constituent in [kg].

Current constituent Identifies the current highlighted concrete constituent. The identification includes the density [kg/m³] that has been defined in the concrete constituent database and the content [kg] of the current material which you have to type directly in the field. You can edit the content of the material in this field, but you have to enter the appropriate database to edit the density. After adding a concrete constituent and specifying the content you can continue with another constituent until the current list of concrete constituents shows the complete composition of the current concrete.

7.2.2 Properties

Select the **Properties** button to define the general properties of the concrete. These properties are all characterized by a constant value independent of the age of the concrete.

When you have selected the **Properties** button the Concrete properties dialog box will pop up as shown in figure 7.3. The Concrete properties dialog box is divided into three areas: The concrete name, the typed properties and the calculated properties.

Concrete: HETEK	
Slump	70. [mm]
W/C ratio	0.4 [kg/kg]
Air content	5.5 [%]
Specific heat	1.02 [kJ/kg/°C]
Thermal cond.	8. [kJ/m/h/°C]
Density, measured	2328. [kg/m ³]
Act. energy factor 1	33500. [J/mol]
Act. energy factor 2	1470. [J/mol/°C]
Total volume	1.003 [m ³ /m ³]
Cement	284.3 [kg/m ³]
Cement + min. add.	356.1 [kg/m ³]
Density, calculated	2336.0 [kg/m ³]

Figure 7.3
Concrete properties dialog box.

Concrete name Identifies the current concrete selected in the current list in the concrete dialog box. See figure 7.1.

Typed properties Identifies all the properties that you have to define by placing the cursor in the field (with either the mouse or the Tab key) and typing the current values.

Slump: Identifies the measured slump of the concrete [mm]. The slump is not used in the calculations.

W/C-ratio: Identifies the equivalent water-cement ratio of the concrete defined as:

$$W/C\text{-ratio} = \frac{\text{Water content [kg/m}^3]}{\text{Eqv. cement content [kg/m}^3]}$$

where

$$\text{Eqv. cement content} = C + 2 \cdot MI + 0.5 \cdot FA$$

where

C	= Cement content	[kg/m ³]
MI	= Micro silica content	[kg/m ³]
FA	= Fly ash content	[kg/m ³]

The w/c-ratio is not used in the calculations.

Air content: Identifies the measured air content in the fresh concrete [%]. If the mix proportion is specified the air content is used otherwise it is not.

Specific heat: Identifies the specific heat capacity of the concrete [kJ/kg°C]. If the Mix proportion of the concrete is specified the specific heat capacity will be calculated automatically as:

$$c_{p,conc.} = \frac{\sum_{i=1}^n m_i \cdot c_{p,i}}{\sum_{i=1}^n m_i}$$

where

$c_{p,conc.}$	= specific heat capacity of the concrete	[kJ/kg°C]
m_i	= content of concrete constituent no. i	[kg]
$c_{p,i}$	= specific heat capacity of concrete const. no. i	[kJ/kg°C]
n	= number of concrete constituents	

If the mix proportion of the concrete is not specified you may type the value of the specific heat capacity directly [kJ/kg°C].

Thermal cond.: Identifies the thermal conductivity of the concrete [kJ/mh°C]. If no value is known the following values may be used:

- fresh and hardening concrete: 8 kJ/mh°C
- hardened concrete: 6 kJ/mh°C
- hardened light weight concrete: 3 kJ/mh°C

Density measured: Identifies the measured density of the fresh concrete [kg/m³]. If the mix proportion of the concrete is specified the calculated density will be used in the calculations otherwise the measured density will be used.

Act. energy factor 1 and Act. energy factor 2: Identifies the two factors describing the activation energy of the concrete. The activation energy is used to calculate the maturity of the concrete.

The maturity is defined as equivalent age at 20°C as follows:

$$M_{20} = \int_0^t H(\theta) dt$$

where $H(\theta)$ is the temperature function, i.e the relative rate of hardening at temperature θ compared to the rate at 20°C, defined as:

$$H(\theta) = \exp \left[\frac{E}{R} \cdot \left(\frac{1}{293} - \frac{1}{273 + \theta} \right) \right]$$

where

- R = gas constant = 8.314 J/mol°C
 θ = concrete temperature [°C]
 E = activation energy [J/mol], defined as:

$$E = f_1 + f_2 (20 - \theta) \quad \text{for } \theta < 20^\circ\text{C}$$

$$E = f_1 \quad \text{for } \theta \geq 20^\circ\text{C}$$

where

- f_1 = activation energy factor 1, default value = 33500 J/mol
 f_2 = activation energy factor 2, default value = 1470 J/mol°C

Calculated properties Identifies the calculated properties based on the specified mix proportions.

Total volume: Identifies the total volume of the specified concrete constituents including the air content [m³].

Cement: Identifies the content of cement in 1 m³ concrete [kg/m³]. The calculation is based on the specified cement content in Mix proportions and the calculated total volume.

Cement + min. add.: Identifies the content of cement plus mineral additives in 1 m³ concrete [kg/m³]. The content in 1 m³ is calculated on basis of the specified contents of cement and mineral additives in Mix proportions and the calculated total volume.

Density, calculated: Identifies the density of the concrete [kg/m³] calculated on basis of the contents of the specified materials in Mix proportions and the total volume. If the calculated density is present it will be used in the temperature and stress calculations otherwise the measured density will be used.

7.2.3 Heat

Select the **Heat** button to define the adiabatical heat development of the current concrete.

The heat development can either be described by measured values saved in a datafile or by the three parameters Q_∞ , τ_c and α that describe the following mathematical exponential approximation to the heat development:

$$Q(M) = Q_{\infty} \cdot \exp \left[- \left(\frac{\tau_e}{M} \right)^{\alpha} \right]$$

where

$Q(M)$	=	heat development at the maturity M	[kJ/kg]
Q_{∞}	=	total heat development	[kJ/kg]
τ_e	=	time constant	[h]
α	=	curvature parameter	[-]
M	=	maturity	[h]

A graphical presentation of the approximated exponential heat development is shown in figure 7.4.

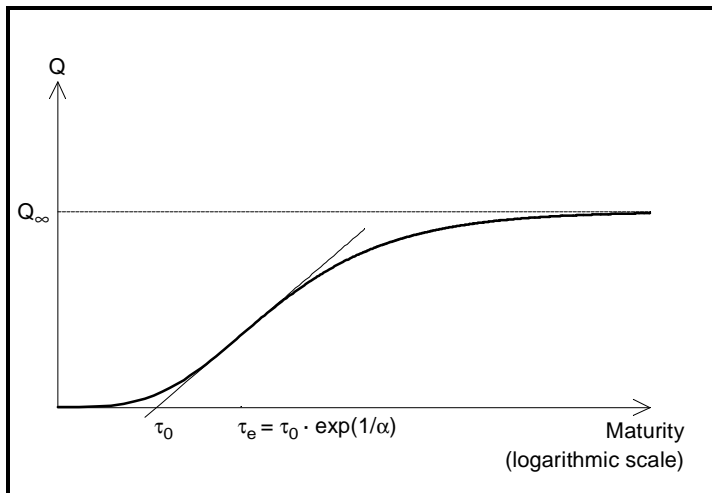


Figure 7.4

Approximated exponential heat development.

When you have selected the **Heat** button the Heat development dialog box will pop up as shown in figure 7.5. The heat development dialog box is divided into three areas: The datafile, curve parameters and the basis for the heat development.

Figure 7.5

Heat development dialog box.

Datafile

Identifies the current datafile with measured values of the heat development. **Click the circle** to the left of [Filename] if the datafile values shall be used directly in the temperature calculations. The format of the datafile must be two columns containing (Maturity [h], Heat development [kJ/kg]) values for the total process time specified in the calculation parameters and the name of the datafile must have the suffix .HET. See section 6.6.5 about the total process time.

Click the down arrow button to select the desired datafile. If no suitable datafile is created click the **Create file** button to create a new datafile. When you click the create file button the Datafile editor will pop up as shown in figure 7.6 and you may enter corresponding values of maturity and heat development. When corresponding values have been specified you may click the OK button and a dialog box will appear where the name of the new datafile must be specified.

The new datafile can now be selected in the datafile list. If you want to edit or view an existing datafile select the file in the list and click the **Edit file** button. This will make the Datafile editor appear. Notice that the limit of (maturity, heat development) values in the Datafile editor is 50. Files with more than 50 corresponding values can not be edited or viewed in the Datafile editor but of course the values may be used in the calculations.

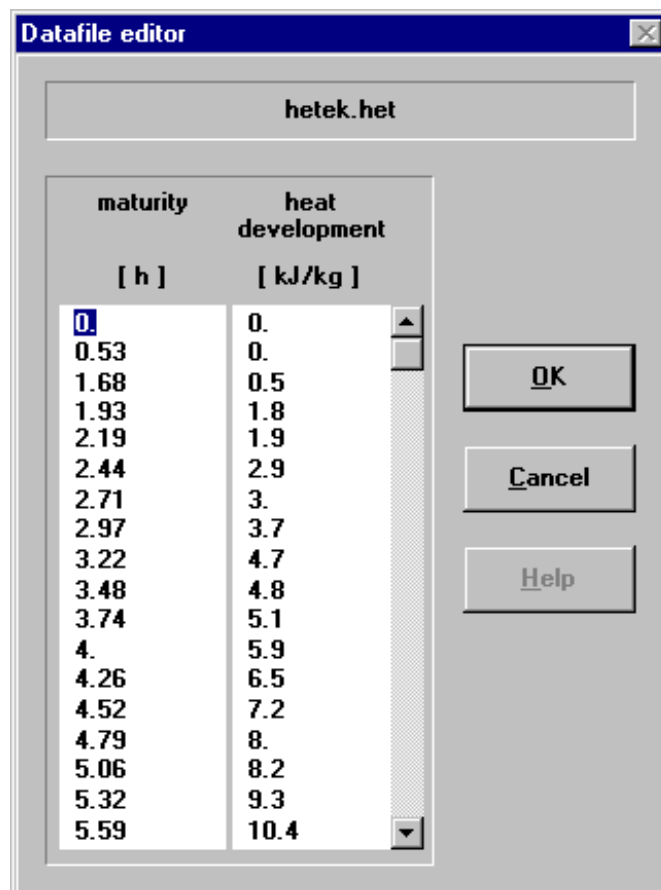


Figure 7.6
Datafile editor dialog box.

Curve parameters Identifies the three parameters Q_{∞} [kJ/kg], τ_c [h] and α [-]. The three parameters can either be typed directly or they can be generated automatically from the measuring values in the datafile. **Click the circle** to the left of [Curve parameters] if the temperature calculations shall be based on an approximated heat development instead of a datafile.

Each of the three parameters can be typed directly in the current fields. If a datafile is selected and the circle to the left of [Curve parameters] is selected, the three parameters may be generated automatically when you click the **Generate** button. Notice the automatic generating needs three start values for the parameters which you must type in the fields before generating.

If you want to fix any of the three parameters in the generating, click the square so a tickmark is shown at the current parameter.

In the calculation message box a message will pop up when the generating is finished to tell you whether it has been possible to generate the three parameters or not. If the generating does not succeed you may try again with some new start values and perhaps one of the parameters fixed.

Click the **View** button to see the approximated exponential heat development based on the curve parameters together with the values specified in the datafile. If no datafile is selected only the approximated development is shown.

The curve parameters are shown on the graph together with the parameters of the linear model described as

$$Q(M) = Q_o \cdot \ln \left(\frac{M}{\tau_o} \right)$$

NB! For an old concrete with high hydration degree there will be no more heat development. In this case the total heat development must be identified with a zero.

Basis for the heat development

Identifies whether the heat development is defined as heat development in proportion to the content of cement or in proportion to the content of cement plus mineral additives in the concrete. **Click the square** to the left of the current basis and if the mix proportions of the concrete are specified the content of cement or cement plus mineral additives in 1 m³ concrete will be identified in the Content field. If the mix proportion of the concrete is not specified, **type** the current content in the content field [kg/m³].

7.2.4 E-Modulus

Select the **E-modulus** button to define the development of the modulus of elasticity of the current concrete.

The modulus of elasticity development can either be described by measured values saved in a datafile or by the three parameters E_{∞} [MPa], τ_e [h] and α [-] that describe a mathematical exponential approximation to the E-modulus development similar to the one for the heat development described in section 7.2.3. Notice that the parameters τ_e and α do not have to be identical with the τ_e and α defined for the heat development.

When you have selected the **E-modulus** button the E-modulus development dialog box will pop up as shown in figure 7.7. The E-modulus development dialog box is divided into two areas: The datafile and the curve parameters.

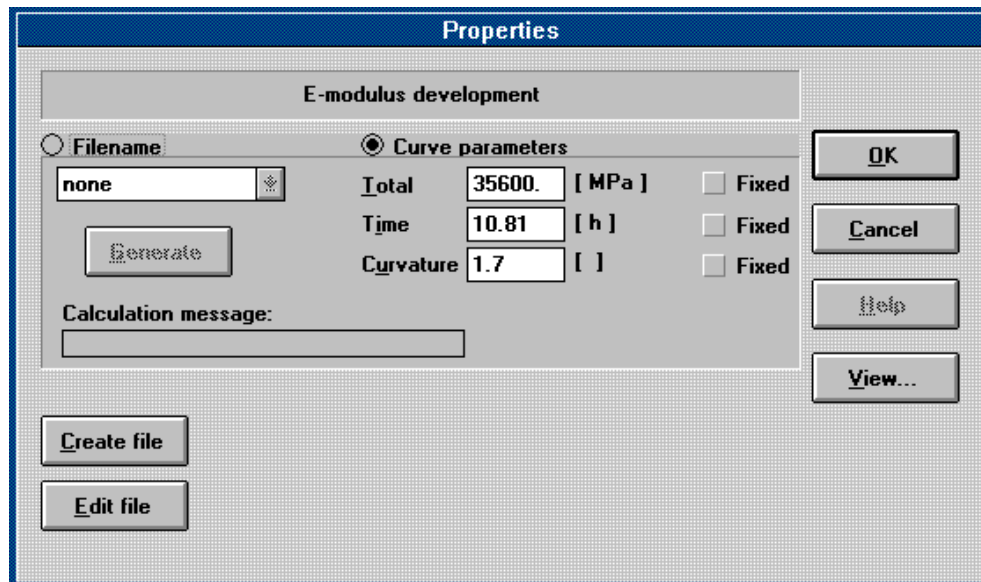


Figure 7.7
E-modulus dialog box.

Datafile

Identifies the current datafile with measured values of the E-modulus development. **Click the circle** to the left of [Filename] if the datafile values shall be used directly in the calculations. The format of the datafile must be two columns containing (Maturity [h], E-modulus [MPa]) values for the total process time specified in the calculation parameters and the name of the datafile must have the suffix .EMO. See section 6.6.5 about the total process time.

Click the down arrow button to select the datafile. Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Curve parameters

Identifies the three parameters E_{∞} [MPa], τ_e [h] and α [-]. The three parameters can either be typed directly or they can be generated automatically from the measuring values in the datafile. **Click the circle** to the left of [Curve parameters] if the calculations shall be based on an approximated development instead of a datafile.

Each of the three parameters can be typed directly in the current fields. If a datafile is selected and the circle to the left of [Curve parameters] is selected, the three parameters may be generated automatically when you click the **Generate** button. Notice the automatic generating needs three start values for the parameters which you may type in the fields before generating.

If you want to fix any of the three parameters in the generating, click the square so a tickmark is shown at the current parameter.

In the calculation message box a message will pop up when the generating is finished to tell you whether it has been possible to generate the three parameters or not. If the generating does not succeed you may try again with some new start values and perhaps one of the parameters fixed.

Click the view button to see the approximated exponential E-modulus development based on the curve parameters together with the values specified in the datafile. If no datafile is selected only the approximated development is shown.

The curve parameters are shown on the graph together with the parameters of the linear model described similar to the one for the heat development.

NB! For an old concrete with high hydration degree the E-modulus will be fully developed from the beginning of the calculations. In this case the time constant must be identified with a zero.

7.2.5 Poisson Ratio

Select the **Poisson ratio** button to define the poisson ratio development of the current concrete.

The poisson ratio development can either be described by measured values saved in a datafile or by the four parameters v_0, v_∞, τ_e and α that describes the following mathematical exponential approximation to the poisson ratio development:

$$v(M) = v_0 - (v_0 - v_\infty) \cdot \exp \left[- \left(\frac{\tau_e}{M} \right)^\alpha \right]$$

where

$v(M)$	=	poisson ratio development at the maturity time M	[-]
v_0	=	poisson ratio of the fresh concrete	[-]
v_∞	=	poisson ratio of the hardened concrete (Total)	[-]
τ_e	=	time constant	[h]
α	=	curvature parameter	[-]
M	=	maturity time defined as equivalent age at 20°C	[h]

A graphical presentation of the approximated exponential poisson ratio development is shown in figure 7.8.

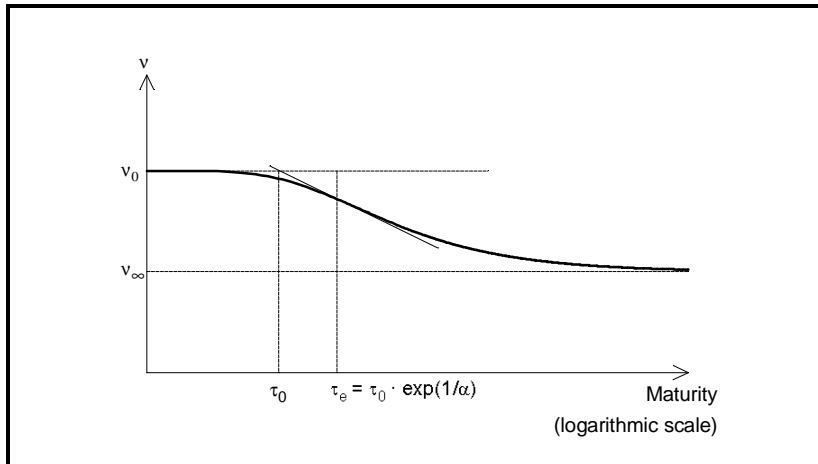


Figure 7.8

Approximated exponential poisson ratio development.

When you have selected the **Poisson ratio** button the poisson ratio development dialog box will pop up as shown in figure 7.9. The poisson ratio development dialog box is divided into two areas: The datafile and the curve parameters.

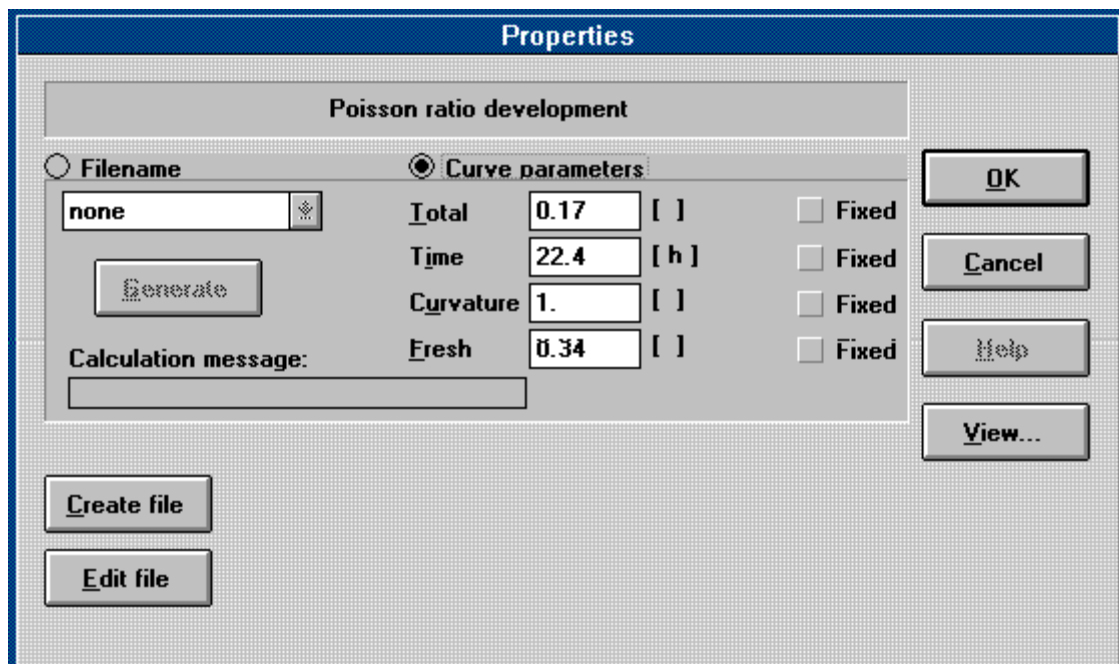


Figure 7.9

Poissons ratio development dialog box.

Datafile

Identifies the current datafile with measured values of the poisson ratio development. **Click the circle** to the left of [Filename] if the datafile values shall be used directly in the calculations. The format of the datafile must be two columns containing (Maturity [h], Poisson ratio [-]) values for the total process time specified in the calculation parameters and the name of the datafile must have the suffix .POI. See section 6.6.5 about the total process time.

Click the down arrow button to select the datafile. Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Curve parameters

Identifies the four parameters v_0 (Fresh), v_∞ (Total), τ_c (Time) and α (Curvature). **Click the circle** to the left of [Curve parameters] if the calculations shall be based on an approximated development instead of a datafile and type the four parameters in the current fields.

Click the **View** button to see the approximated exponential poisson ratio development based on the curve parameters together with the values specified in the datafile. If no datafile is selected only the approximated development is shown.

The curve parameters are shown on the graph together with the parameters at the linear model described as

$$v(M) = v_o - (v_o - v'_o) \cdot \ln \left(\frac{M}{\tau_o} \right)$$

NB! For an old concrete with high hydration degree the poisson ratio will be fully developed from the beginning of the calculations. In this case the time constant must be identified with a zero and the fresh value must be identical with the total value.

7.2.6 Thermal expansion coefficient

Select the **Thermal exp.** button to define the thermal expansion coefficient development of the current concrete.

The thermal expansion coefficient development can either be described by measured values saved in a datafile or by a constant.

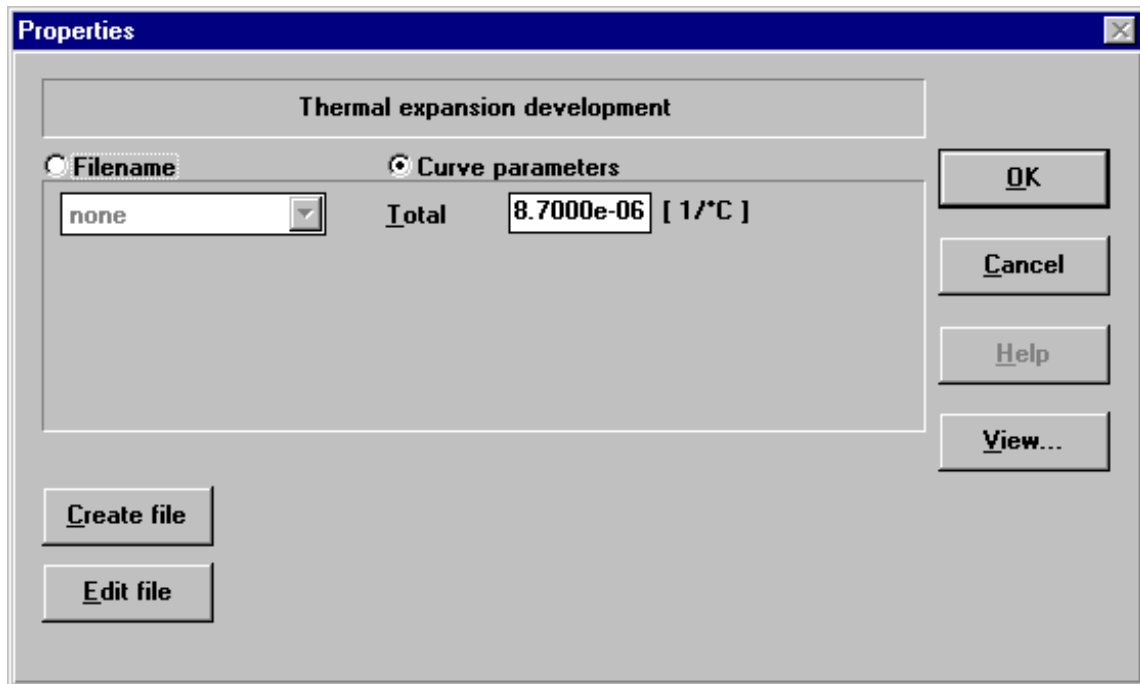


Figure 7.10
Thermal expansion development dialog box.

When you have selected the **Thermal exp.** button the thermal expansion coefficient development dialog box will pop up as shown in figure 7.10. The thermal expansion coefficient development dialog box is divided into two areas: The datafile and the curve parameters.

Datafile

Identifies the current datafile with measured values of the thermal expansion coefficient development. **Click the circle** to the left of [Datafile] if the datafile values shall be used in the calculations. The format of the datafile must be two columns containing (Maturity [h], Thermal exp. [1/°C]) values for the total process time specified in the calculation parameters, and the name of the datafile must have the suffix .THX. See section 6.6.5 about the total process time.

Click the down arrow button to select the datafile. Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Curve parameters

Identifies a constant value of the thermal expansion coefficient. **Click the circle** to the left of [Curve parameters] if the calculations shall be based on a constant thermal expansion coefficient .

Click the **View** button to see the thermal expansion coefficient development defined by datafile or a constant.

7.2.7 Compressive Strength

Select the **Comp. strength** button to define the compressive strength development of the current concrete.

The compressive strength development can either be described by measured values saved in a datafile or by the three parameters $S_{c,\infty}$ [MPa], τ_e [h] and α [-] that describe a mathematical exponential approximation to the compressive strength development similar to the one for the heat development described in section 7.2.3.

When you have selected the **Comp. strength** button the compressive strength development dialog box will pop up as shown in Figure 7.11. The compressive strength development dialog box is divided into two areas: The datafile and the curve parameters.

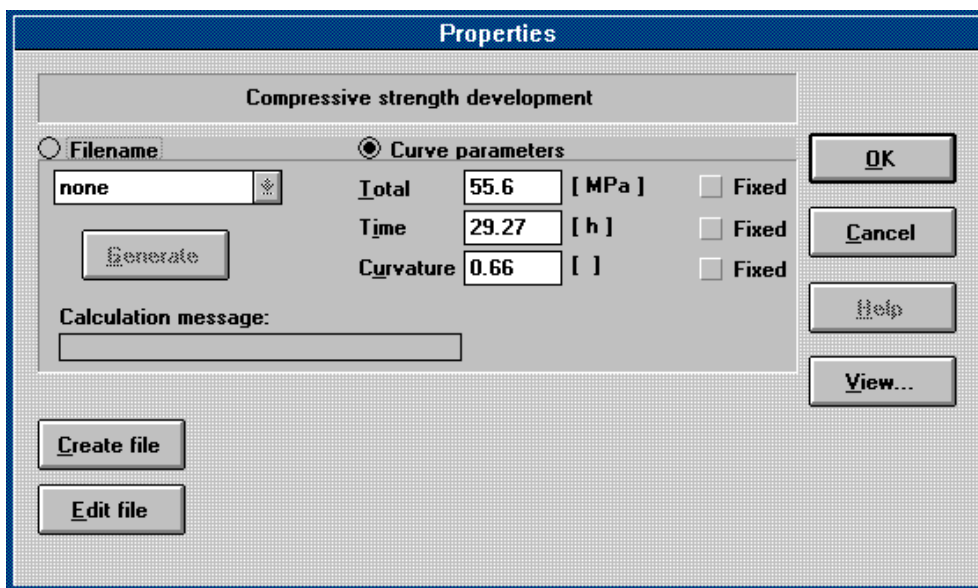


Figure 7.11
Compressive strength development dialog box.

Datafile

Identifies the current datafile with measured values of the compressive strength development. **Click the circle** to the left of [Datafile] if the datafile values shall be used directly in the calculations. The format of the datafile must be two columns containing (Maturity [h], Comp. strength [MPa]) values for the total process time specified in the calculation parameters, and the name of the datafile must have the suffix .CST. See section 6.6.5 about the total process time.

Click the down arrow button to select the datafile. Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Curve parameters

Identifies the three parameters $S_{c,\infty}$ [MPa], τ_e [h] and α [-]. The three parameters can either be typed directly or they can be generated automatically from the measuring values in the datafile. **Click the circle** to the left of [Curve parameters] if the calculations shall be based on an approximated development instead of a datafile.

Each of the three parameters can be typed directly in the current fields. If a datafile is selected and the circle to the left of [Curve parameters] is selected the three parameters may be generated automatically when you click the **Generate** button. Notice the automatic generating needs three start values for the parameters which you may type in the fields before generating.

If you want to fix any of the three parameters in the generating, click the square so a tickmark is shown at the current parameter.

In the calculation message box a message will pop up when the generating is finished to tell you whether it has been possible to generate the three parameters or not. If the generating does not succeed, you may try again with some new start values and perhaps one of the parameters fixed.

Click the **View** button to see the approximated exponential compression strength development based on the curve parameters together with the values specified in the datafile. If no datafile is selected only the approximated development is shown.

The curve parameters are shown on the graph together with the parameters of the linear model described similar to the one for the heat development.

NB! For an old concrete with high hydration degree the compressive strength will be fully developed from the beginning of the calculations. In this case the time constant must be identified with a zero.

7.2.8 Tensile Strength

Select the **Tens. strength** button to define the tensile strength development of the current concrete. The tensile strength can either be an uniaxial tensile strength, a tensile splitting strength or a tensile bending strength.

The tensile strength development can either be described by measured values saved in a datafile or by the three parameters $S_{t,\infty}$ [MPa], τ_c [h] and α [-] that describe a mathematical exponential approximation to the tensile strength development similar to the one for the heat development described in section 7.2.3.

When you have selected the **Tens. strength** button the tensile strength development dialog box will pop up as shown in figure 7.12. The tensile strength development dialog box is divided into two areas: The datafile and the curve parameters.

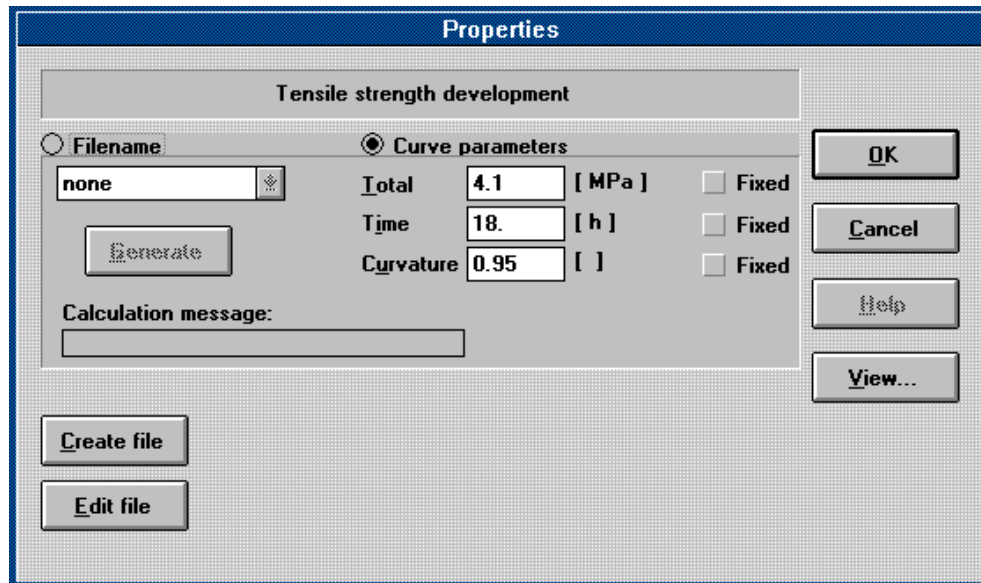


Figure 7.12
Tensile strength development dialog box.

Datafile

Identifies the current datafile with measured values of the tensile strength development. **Click the circle** to the left of [Filename] if the datafile values shall be used directly in the calculations. The format of the datafile must be two columns containing (Maturity [h], Tens. strength [MPa]) values for the total process time specified in the calculation parameters and the name of the datafile must have the suffix .TST. See section 6.6.5 about the total process time.

Click the down arrow button to select the datafile. Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Curve parameters

Identifies the three parameters $S_{t,\infty}$ [MPa], τ_c [h] and α [-]. The three parameters can either be typed directly or they can be generated automatically from the measuring values in the datafile. **Click the circle** to the left of [Curve parameters] if the calculations shall be based on an approximated development instead of a datafile.

Each of the three parameters can be typed directly in the current fields. If a datafile is selected and the circle to the left of [Curve parameters] is selected, the three parameters may be generated automatically when you click the **Generate** button. Notice the automatic generating needs three start values for the parameters which you may type in the fields before generating.

If you want to fix any of the three parameters in the generating, click the square so a tickmark is shown at the current parameter.

In the calculation message box a message will pop up when the generating is finished to tell you whether it has been possible to generate the three parameters or not. If the generating does not succeed, you may try again with some new start values and perhaps one of the parameters fixed.

Click the **View** button to see the approximated exponential tensile strength development based on the curve parameters together with the values specified in the datafile. If no datafile is selected only the approximated development is shown.

The curve parameters are shown on the graph together with the parameters at the linear model described similar to the one for the heat development.

NB! For an old concrete with high hydration degree the tensile strength will be fully developed from the beginning of the calculations. In this case the time constant must be identified with a zero.

7.2.9 Initial Strain

Select the **Initial Strain** button to define the initial strain development of the current concrete. The initial strain can for instance be the measured autogenous shrinkage of the concrete. It can not be any type of strain that requires calculations of combined heat and moisture transportation. The initial strain to which a structure is exposed will result in an expansion or a contraction of the structure in all directions. Initial strain resulting in expansion is calculated positive. This means that shrinkage must be defined as a negative initial strain.

The initial strain development can only be described by measured values or generated values saved in a datafile.

When you have selected the **initial strain** button the initial strain development dialog box will pop up as shown in figure 7.13. The initial strain development dialog box is divided into two areas: The datafile and the data based on area.

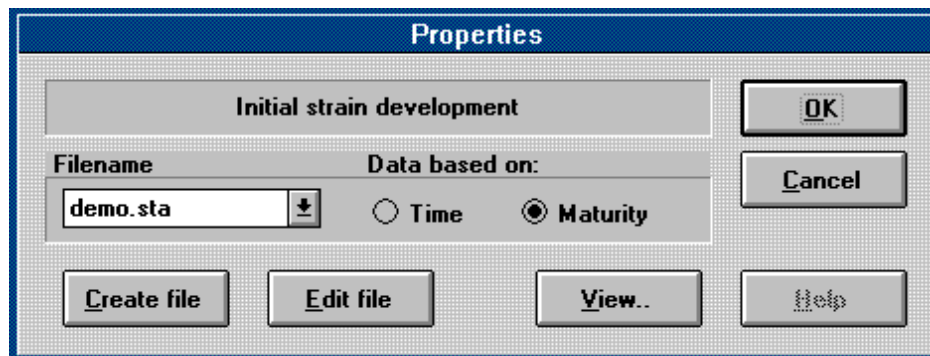


Figure 7.13
Initial strain development dialog box.

Datafile

Identifies the current datafile with measured or generated values of the initial strain development. Select the name of the datafile in the list. The format of the datafile must be two columns containing (Time or Maturity [h], Initial strain [m/m]) values for the total process time specified in the calculation parameters, and the name of the datafile must have the suffix .STA. See section 6.6.5 about the total process time.

Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. See further description in section 7.2.3.

Data based on Identifies whether the values of initial strain are as a function of time [h] or maturity [h]. Click the current circle to select the basis.

7.2.10 Creep

Select the **Creep** button to define the creep parameters of the current concrete.

When creep is included in the project a linear visco elastic material model will be used. In 4C-Temp&Stress this is described by a rheological model as shown in figure 7.14. The model consists of two Kelvin-elements in serial connection.

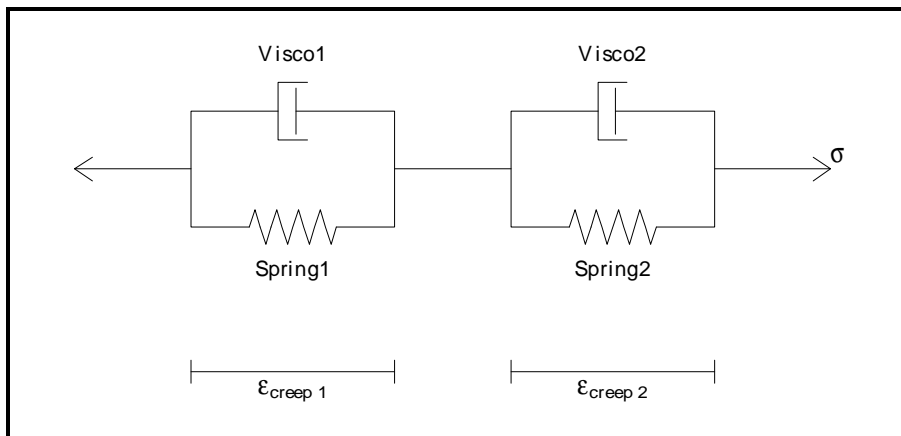


Figure 7.14
Creep model.

Visco1 and Visco2 are the viscosity of the pistons. Spring1 and Spring2 are the spring constants of the springs. These properties can change as a function of maturity.

The creep strain velocity is calculated from

$$\frac{d \epsilon_{\text{creep}}}{dt} = \sum_{i=1}^2 \left(\frac{C}{Visco_i} \cdot \epsilon_{\text{creep},i} - \frac{Spring_i}{Visco_i} \cdot \epsilon_{\text{creep},i} \right)$$

Where $\epsilon_{\text{creep},i}$ is the total creep strains of Kelvin element no. i . $C_{\text{creep},i}$ and $\epsilon_{\text{creep},i}$ contain

$$\text{creep}i = \begin{bmatrix} x \\ y \\ xy \\ z \end{bmatrix}$$

$$= \begin{bmatrix} x \\ y \\ \tau_{xy} \\ z \end{bmatrix}$$

$$C = \begin{bmatrix} 1 & -v_c & 0 & -v_c \\ -v_c & 1 & 0 & -v_c \\ 0 & 0 & 2(1+v_c) & 0 \\ -v_c & -v_c & 0 & 1 \end{bmatrix}$$

where v_c is the creep poisson ratio.

When you have selected the **Creep** button the creep development dialog box will pop up as shown in figure 7.15. Then you can define in which datafiles the development of the four parameters of the Kelvin elements and the creep poisson ratio are described.

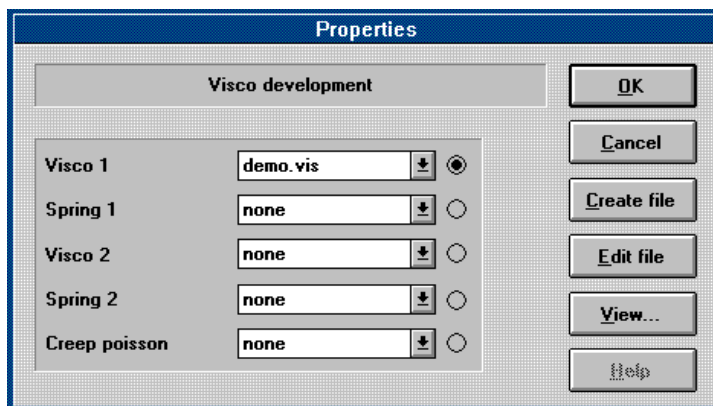


Figure 7.15
Creep development dialog box.

Visco1

Select the name of the datafile in the list. The format of the datafile must be two columns containing (Maturity [h], Visco1 [MN•h/m²]) values for the total process time specified in the calculation parameters. The name of the datafile must have the suffix .VIS. See section 6.6.5 about the total process time. If *none* is selected the piston is left out corresponding to $\text{Visco1}(M) = \infty$. This is the default selection.

- Spring1** Select the name of the datafile in the list. The format of the datafile must be two columns containing (Maturity [h], Spring1 [MN/m²]) values for the total process time specified in the calculation parameters. The name of the datafile must have the suffix .SIR. See section 6.6.5 about the total process time. If *none* is selected the spring is left out corresponding to Spring1(M) = 0. This is the default selection.
- Visco2** Analogous with Visco1
- Spring2** Analogous with Spring1
- Creep poisson** Select the name of the datafile in the list. The format of the datafile must be two columns containing (Maturity [h], ϵ_{creep}) values for the total process time specified in the calculation parameters. The name of the datafile must have the suffix .CPO. See section 6.6.5 about the total process time. If *none* is selected the creep poisson is left out corresponding to $\epsilon_{creep} = 0$. This is the default selection.

Similar to the properties of heat development new datafiles can be created and the existing datafiles can be edited by clicking the **Create file** and the **Edit file** buttons respectively. To show which of the five types of datafiles you will create or edit click the circle to the right of the current type of file before clicking the **Create file** or **Edit file** button. See further description in section 7.2.3

NB! When creep is taken into account a convergence criteria must be defined as described in section 6.6.7.

7.3 Concrete Constituents

Each type of concrete constituents has its own sub-database. The types of concrete constituents are: Cement, Aggregate, Mineral additive, Chemical admixtures, Water and Others.

You can select each sub-database separately from the menu item [Database] and then **click** on the sub-database with the type of concrete constituents you want to select. The procedure to follow is the same for all types of concrete constituents. In the following the procedure will be described for the Cement sub-database.

Select the menu item [Cement] in the menu bar. The cement database dialog box will then pop up as shown in figure 7.16. The cement database dialog box includes a list of all the defined cements in the database. A new type of cement may be defined by clicking the **Create new** button and enter the name of the new cement in the dialog box that appears. The already existing cements may be **Edited** or **Deleted** by selecting the current button.

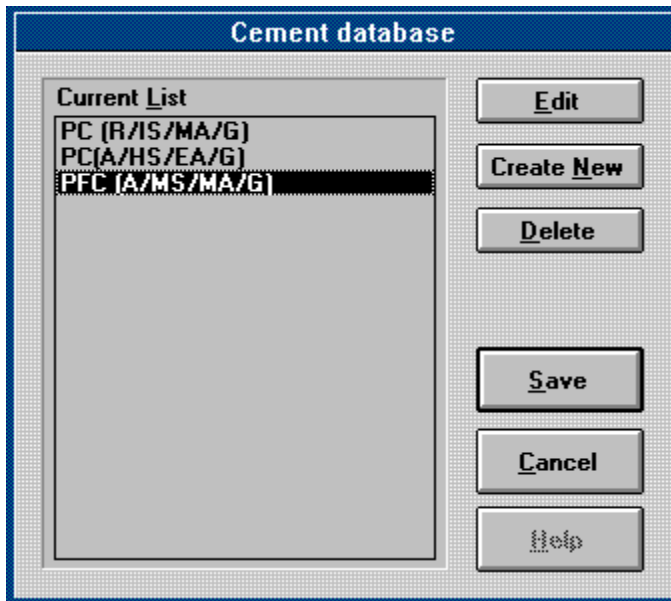


Figure 7.16
Cement database dialog box.

When you **Edit** an existing or **Create a new** type of cement, the cement dialog box will pop up as shown in figure 7.17. In the cement dialog box you may define the necessary properties for the current cement type: Identification, density, specific heat and thermal cond.. To accept the new cement click the **OK** button, otherwise click the **Cancel** button.

Identification Identifies the cement type with a name of maximum 19 characters. When you edit a cement type the name cannot be changed. If you want to change the name **Create** a new cement type and **Delete** the old one.

Density Identifies the density of the current cement type [kg/m^3].

Specific heat Identifies the specific heat capacity of the current cement type [$\text{kJ/kg}^\circ\text{C}$].

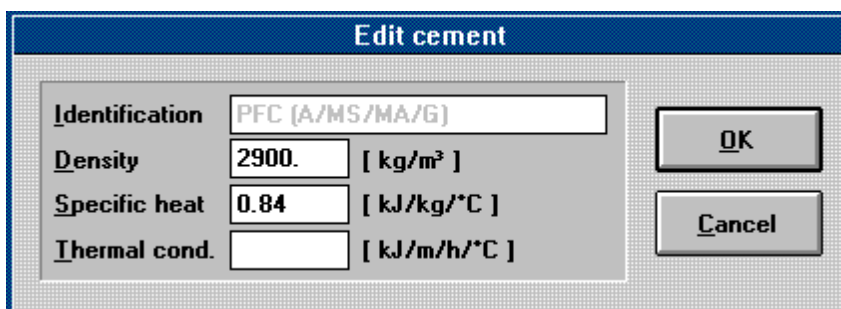


Figure 7.17
Cement dialog box.

NB! If no value is known the following values may be used for the different types of concrete constituents:

Cement, aggregates, mineral additives and other solid constituents:	0.84 $\text{kJ/kg}^\circ\text{C}$
Chemical additives, water and other liquid constituents:	4.18 $\text{kJ/kg}^\circ\text{C}$

Thermal**cond.**

Identifies the thermal conductivity of the current cement type [kJ/mh°C]. It is not necessary to define the thermal conductivity of the constituents and in this version of the program the specified values will not be used in any of the calculations.

7.4 Materials

All types of materials besides concrete and soil may be defined in the Material sub-database. Select the menu item [Materials] in the menu bar. The material database dialog box will then pop up as shown in figure 7.18. The material database dialog box includes a list of all the defined materials in the database - like steel, glass, air and polystyrene. The selected material is highlighted in the current list.

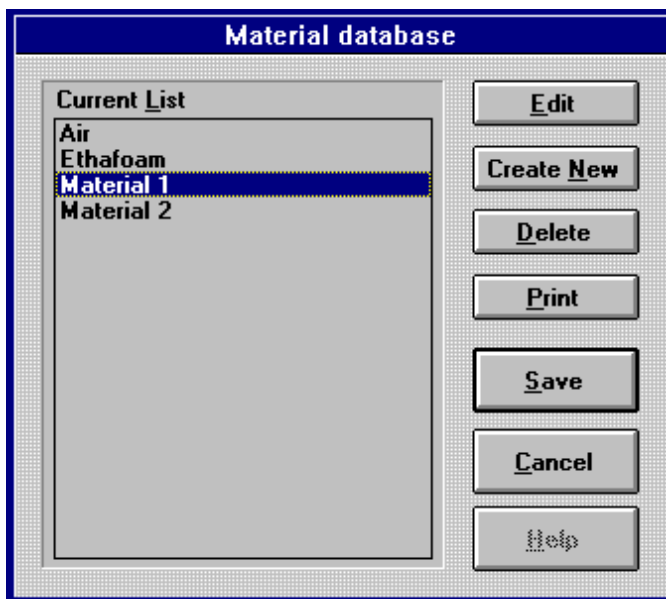


Figure 7.18
Materials database dialog box.

A new material may be defined by clicking the **Create new** button and entering the name of the new material in the dialog box that appears. The already existing materials may be **Edited**, **Deleted** or **Printed** by selecting the current button.

Property	Value	Unit
Identification	Material 1	
Density	2140.	[kg/m ³]
Specific heat	1.1	[kJ/kg/°C]
Thermal conduct.	0.14	[kJ/m/h/°C]
Thermal expansion	0.	[1/°C]
E-modulus	120.	[MPa]
Poisson ratio	0.3	[]
Comp. strength	120.	[MPa]
Tensile strength	120.	[MPa]

Figure 7.19
Material dialog box.

When you **Edit** an existing or **Create a new** type of material, the material dialog box will pop up as shown in figure 7.19. In the material dialog box you may define the necessary properties for the current material type: Identification, density, specific heat, thermal conduct., thermal expansion, E-modulus, poisson ratio, comp. strength and tensile strength. All the properties of the material are characterized by a constant value independent of the age of the material. To accept the new material click the **OK** button, otherwise click the **Cancel** button.

Identification	Identifies the material with a name of maximum 19 characters. When you Edit a material the name cannot be changed. If you want to change the name Create a new material and Delete the old one.
Density	Identifies the density of the current material [kg/m ³].
Specific heat	Identifies the specific heat capacity of the current material [kJ/kg °C].
Thermal conduct.	Identifies the thermal conductivity of the current material [kJ/mh °C].
Thermal expansion	Identifies the thermal expansion coefficient of the current material [°C ⁻¹].
E-modulus	Identifies the modulus of elasticity of the current material [MPa].
Poisson ratio	Identifies the poisson ratio of the current material [-].
Comp. strength	Identifies the compressive strength of the current material [-].
Tensile strength	Identifies the tensile strength of the current material [-]. The tensile strength can either be an uniaxial tensile strength, a tensile splitting strength or a tensile bending strength.

7.5 Soil

Select the menu item [Soil] in the menu bar. The soil database dialog box will then pop up similar to the material database dialog box described in section 7.4 and shown in figure 7.19. The soil database dialog box includes a list of all the defined soils in the database and the procedures to follow when you want to edit, delete or create a new soil material are the exactly the same as described in section 7.4 for the material database.

7.6 Insulation

Select the menu item [Insulation] in the menu bar. The insulation database dialog box will then pop up as shown in figure 7.20. The insulation database dialog box includes a list of all the defined insulation materials in the database. A new type of insulation may be defined by clicking the **Create new** button and entering the name of the new insulation in the dialog box that appears. The already existing insulation materials may be **Edited**, **Deleted** or **Printed** by selecting the current button.

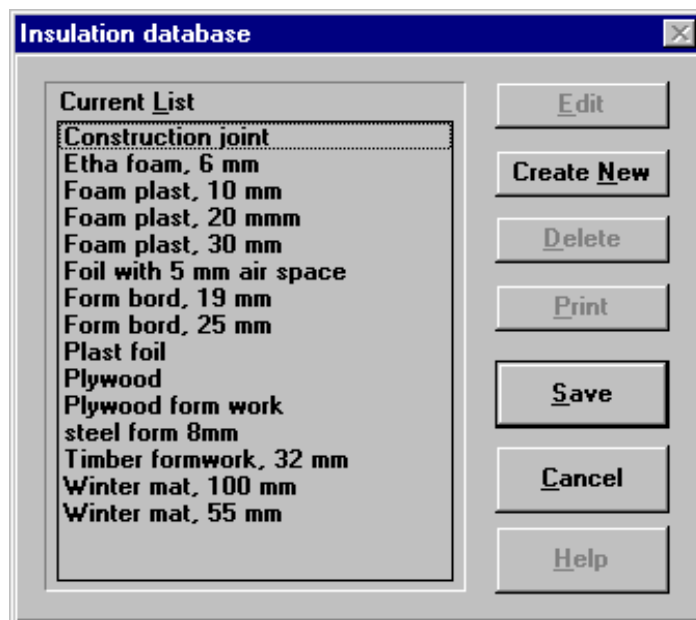


Figure 7.20
Insulation dialog box.

When you **Edit** an existing or **Create a new** type of insulation, the insulation dialog box will pop up as shown in Figure 7.21. In the insulation dialog box you may define the necessary properties for the current insulation material: Identification, thickness and thermal cond.. To accept the new insulation material click the **OK** button, otherwise click the **Cancel** button.

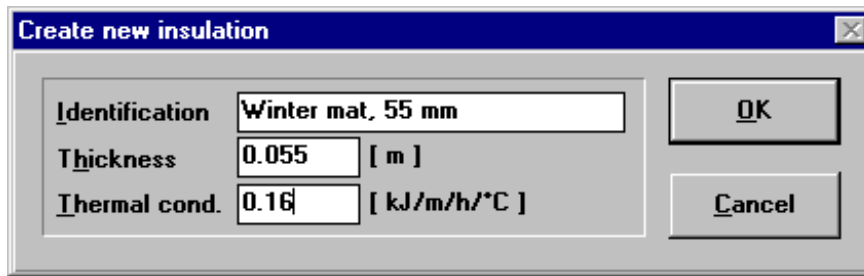


Figure 7.21
Insulation dialog box.

Identification Identifies the insulation type with a name of maximum 19 characters. When you edit an insulation type the name cannot be changed. If you want to change the name **Create** a new insulation type and **Delete** the old one.

Thickness Identifies the thickness of the current insulation type [m].


Thermal cond. Identifies the thermal conductivity of the current insulation type [kJ/mh°C].

8. Project Solver

This section will help you understand how to use the Project solver (PS) which is a separate module of 4C-Temp&Stress. In this section it is described how to start the three different calculations in 4C-Temp&Stress: The FEM generation, the temperature calculation and the stress calculation. In the following sections each of these calculation types are introduced including the necessary parameters to each of the calculation types.

8.1. Generate Finite Element Mesh

When all elements of a project including the project setup and the calculation parameters have been defined and saved in a file you may activate the finite element mesh generation.

To activate the FEM generation **click** the generate FEM button  or select the menu item [Calculation] and then the menu item [Generate]. This will activate the PS and a message box will pop up showing the number of generated elements. If a calculation of a project with the current file name has been executed earlier, a dialog box will pop up first and ask you if you want to continue the calculation. If any changes have been made in the project a new FEM must be generated before further calculations.

When the FEM generation has been executed, the generated FEM may be shown and printed out from the RV.

8.2. Calculate Temperatures


You have to generate a FEM before you can activate the temperature calculation. Otherwise 4C-Temp&Stress will make a beep sound. If the Calculation parameters are set for 2½-dimensions the temperature calculation will include calculations of temperature, maturity and approximated stress. Otherwise it will include only calculations of temperature and maturity.

To be able to execute the **temperature** and **maturity** calculations at least the following material properties must be defined for all used materials in the project:

Concrete	Properties (specific heat capacity, thermal conductivity and density) and Heat development
Material	Density, specific heat capacity and thermal conductivity
Soil	Density, specific heat capacity and thermal conductivity.

To be able to execute the **approximated stress** calculations at least the following material properties **besides** those mentioned above must be defined for all used materials in the project:

Concrete	Thermal expansion development, E-modulus development and Tensile strength development
Material	Thermal expansion, E-modulus and Tensile strength
Soil	Thermal expansion, E-modulus and Tensile strength.

To activate the temperature calculation click the calculate button  or select the menu item [Calculation] and then the menu item [Temperature]. This will activate the PS and a message box will pop up showing the current time step and the processing of the temperature calculation.

When the temperature calculation has been executed, 4C-Temp&Stress will make a beep sound and the results of the temperature calculation may be shown and printed out from the RV.

8.3 Calculate Stresses

You have to generate a FEM before you can activate the stress calculation. Otherwise 4C-Temp&Stress will make a beep sound. If the stress calculation is supposed to be based on thermal results, the temperature calculation has to be executed before you activate the stress calculation. If any changes have been made in the project since the last temperature calculation a new FEM generation and a new temperature calculation must be executed before further stress calculations.

To be able to execute **stress** calculations based on thermal results at least the following material properties must be defined for all used materials in the project:

Concrete	Properties (specific heat capacity, thermal conductivity and density), Heat development, E-modulus development, Thermal expansion development, Poisson ratio development and Tensile strength development
Material	Density, specific heat capacity, thermal conductivity, thermal expansion, E-modulus, Poisson ratio and Tensile strength
Soil	Density, specific heat capacity, thermal conductivity, thermal expansion, E-modulus, Poisson ratio and Tensile strength.

Besides supports to ensure that the structure is unable to perform rigid body displacements must be defined.

To be able to execute **stress** calculations not based on thermal results at least the following material properties must be defined for all used materials in the project:

Concrete	E-modulus development, Poisson ratio development and Tensile strength development
Material	E-modulus, Poisson ratio and Tensile strength
Soil	E-modulus, Poisson ratio and Tensile strength.

Besides supports to ensure that the structure is unable to perform rigid body displacements must be defined.

To activate the stress calculation select the menu item [Calculation] and then the menu item [Stress]. This will activate the PS and a message box will pop up showing the current time step and the processing of the stress calculation.

When the stress calculation has been executed, 4C-Temp&Stress will make a beep sound and the results of the stress calculation may be shown and printed out from the RV.


9. Result Viewer

This section will help you understand the structure in the Result Viewer (RV), the basic elements in it and how to use them. The section is divided into several subsections that describe the general structure in RV and the basic elements in RV. The basic element subsections provide step by step instructions to help you present and analyse the results of your projects.

9.1 General

The result viewer is a separate module in 4C-Temp&Stress which you may activate from the project editor. When a project has been defined in the PE and the calculations of either temperature or stress or both have been executed all the results of the calculations can be presented in the RV.

You can activate the RV in the following ways:

- click the RV button  in the vertical tool bar of the PE or
- select the menu item [Project] in the horizontal tool bar of the PE and then select the menu item [Result viewer].

When you have selected the RV it will be opened and pop up on the screen. If no results are loaded in the RV the menu bar in the RV will only include the menu items: Project, Window and Help. When results from a project are loaded in the RV, the menu bar will include the menu items: Mode and Analysis as well. Each menu item contains a menu bar. The menu item [Analysis] contains three different menu bars depending on which mode you have selected in the menu item [Mode]. Figure 9.1 shows the result viewer when no results are loaded.

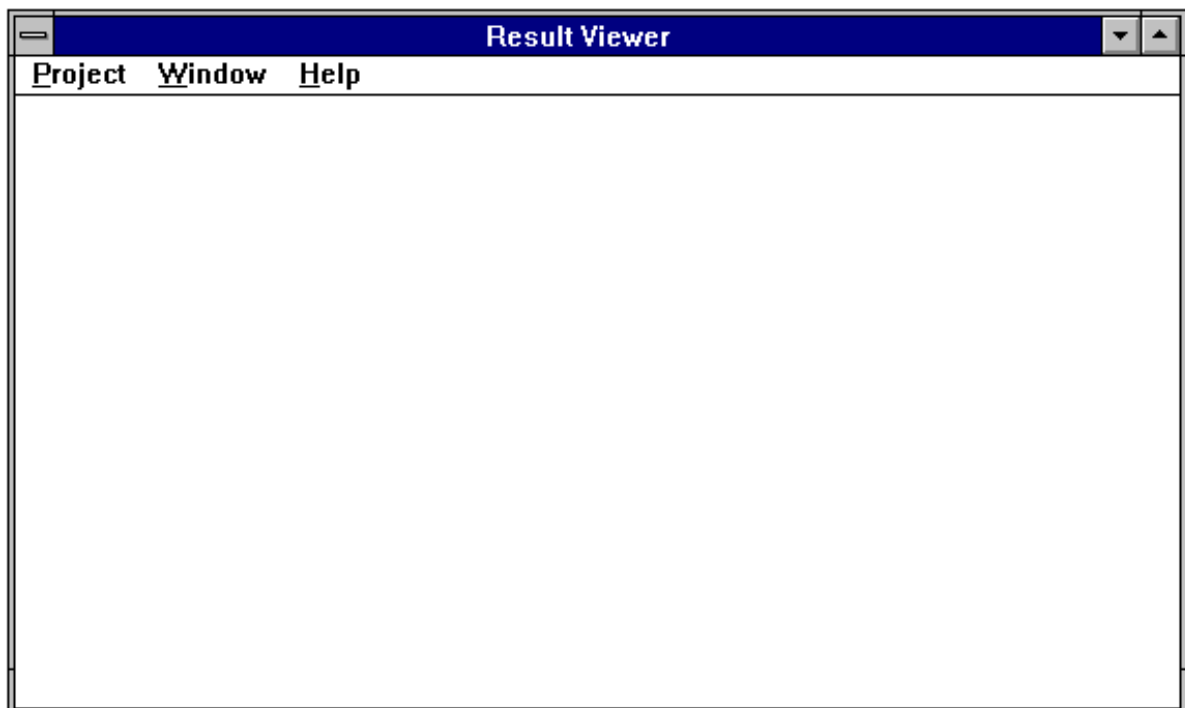


Figure 9.1
4C-Temp&Stress - Result Viewer.

The results of the calculations of your project can be presented in several ways in the RV. The different types of results which can be presented are the following:

- The geometrical contour of the cross section (disk) including specified points
- The generated finite element mesh including specified points
- An x/y-diagram showing either the temperature, the maturity, the strength, the approximated stress or the stress results in specified points as a function of time
- An x/y-diagram showing either the min./max. temperature, the min./max. maturity, the min./max. strength, the min./max. approximated stress or the min./max. stress results of each cast as a function of time
- An x/y-diagram showing either the average temperature, the average maturity, the average strength, the average approximated stress or the average stress results of each cast as a function of time
- An x/y-diagram showing either external or internal temperature difference as function of time
- Isocurves showing either temperature, maturity, strength, approximated stress or stress results in the cross section at any specified time after casting.

How to select these different ways of presenting the results will be described in section 9.3, 9.4 and 9.5.

9.2 Load and Print Results

If a project where the calculations have been executed is loaded into the PE before the RV is selected, the results of the project will automatically be loaded when you open the RV and the geometrical contour of the disk defined in the project together with the generated FEM will be shown at once.

If a project with no calculated results is loaded into the PE before opening the RV you may load the results from another project directly into the RV as follows:

- select the menu item [Project] and in the menu bar select the menu item [Load results]

This will bring up the Load project results dialog box which is shown in figure 9.2.

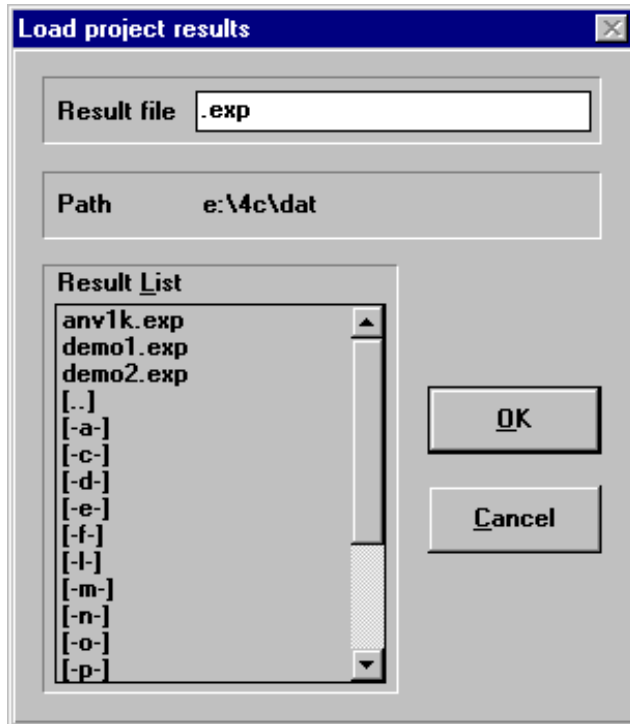


Figure 9.2
Load project results dialog box.

The information about the calculation results is saved in a file with the same name as the project, but with the suffix .EXP. The results will be loaded when you:

- **double click** on the result name in the result list, or
- **click** on the result name in the result list and then click the **OK** button, or
- **type** the name of the result file - with the suffix .EXP - that you want to load and then click the **OK** button.

In 4C-Temp&Stress it is possible to show results from several projects in the RV at the same time. For example if you want to compare the calculated results from two different projects you can do so by loading both files into the RV and then tile the window as described in section 9.6.

All the ways of presenting the results can be **printed** out by selecting the menu item [Project] and the menu item [Print view]. The dialog box Print will then pop up as shown in figure 9.3. The Print dialog box is a standard windows dialog box which is used in a lot of different programs. Therefore it is very general and some of the items are greyed because they are not used in 4C-Temp&Stress. Select always the Print range **All** because neither Selection nor Pages are used in 4C-Temp&Stress.

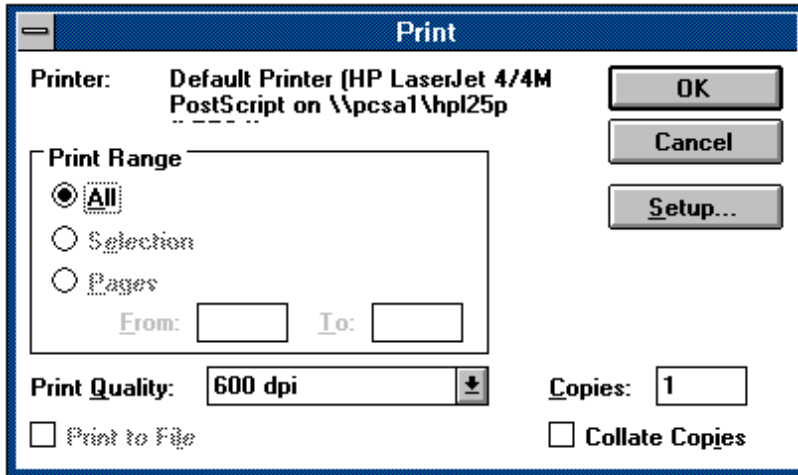


Figure 9.3
Print dialog box.

The current printer type can be changed and specified by selecting the **Setup** button. Further, the printing quality can be selected by clicking on the down arrow button and select the current quality.

9.3 Geometry Mode

There are three different modes in the RV: Geometry mode, Diagram mode and Isocurve mode. This section describes the elements in geometry mode and the types of results which you are able to present by using the geometry mode. Diagram mode and isocurve mode are described in section 9.4 and 9.5 respectively.

Select the menu item [Mode] and the menu item [Geometry] to activate geometry mode in the RV.

In geometry mode it is possible to see the geometrical contour of the disk and the generated finite element mesh. Further, you may define up to ten analysis points in which the calculated values can be shown as a function of time in diagram mode. How to use the RV in geometry mode will be described in the following.

You can **Zoom in** on the view by pressing the [Shift] button on the keyboard down and at the same time with the mouse draw a **rectangle** around the part of the view that you want to zoom in. You can zoom in several times. You can **zoom out** to full view by clicking the left mouse button.

You can select up to ten **analysis points** inside the structure by pressing the [Ctrl.] button on the keyboard down and at the same time move the cursor to the place inside the structure where the analysis point is and then **click** the left mouse button. Notice that the x,y coordinates of the cursor position are shown in the upper right corner of the screen. The coordinates of the selected analysis points are written in the note area to the right of the view. If you want to change the coordinates or remove a selected point, see section 9.3.2.

The different types of geometry view can be selected in the menu item [Analysis]. In the menu bar of Analysis there are four menu items: Settings, points, volumes and geometry system. These menu items will be described in the following sections.

9.3.1 Settings

In this section the menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Settings]. This will make the Geometry settings dialog box appear as shown in figure 9.4.

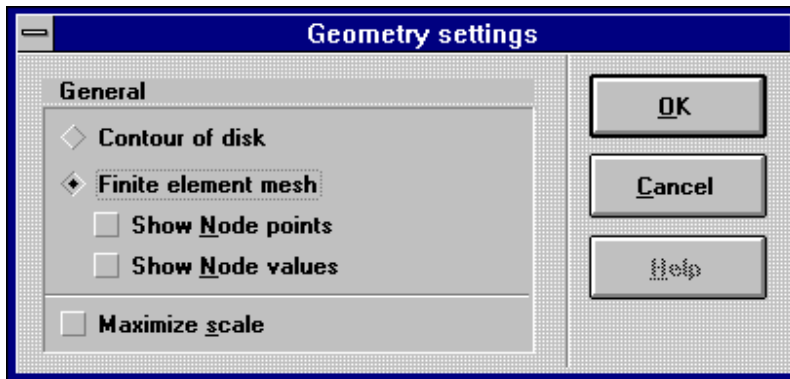


Figure 9.4
Geometry setting dialog box.

In the dialog box you can select whether you want to show only the contour of the disk (the cross section) or you want to show the generated FEM in the disk.

You can also select to maximize the scale which means that the cross section will be zoomed out as much as possible in both x- and y-direction individually. Notice that the geometrical shape of the cross section may be changed when you select maximize scale.

If you choose to show the FEM it is possible also to show the node points in each finite element and the numbers of the node points used in the calculations. In figure 9.4 the FEM is selected, and in figure 9.5 an example of a tunnel structure with this selected view is shown.

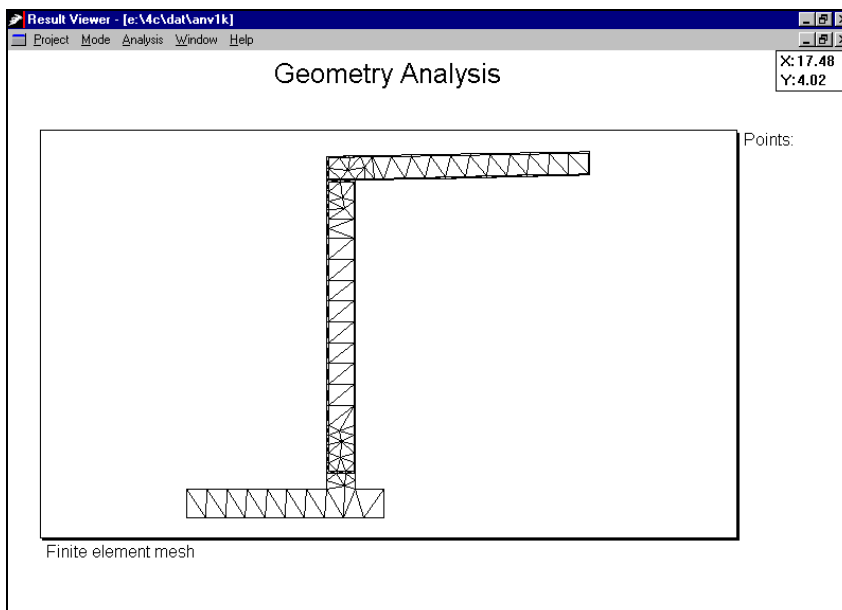


Figure 9.5
Geometry analysis - Finite element mesh.

9.3.2 Points

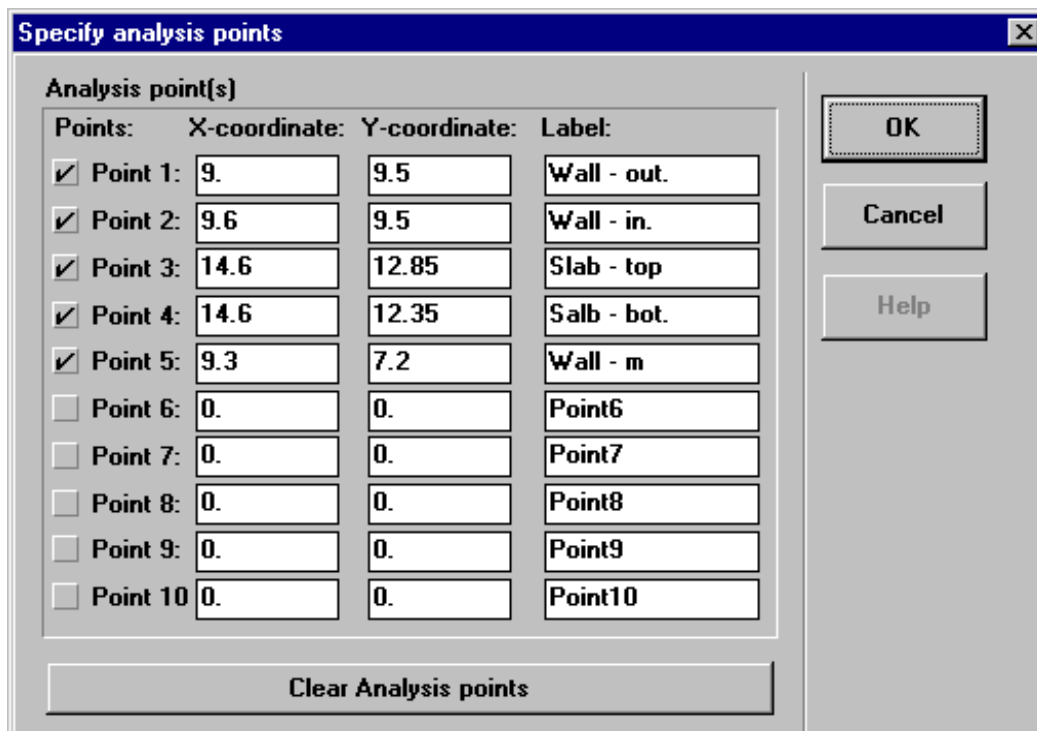
This section describes the menu item [Points]. Select the menu item [Analysis] and the menu item [Points]. This will make the Specify analysis points dialog box appear as shown in figure 9.6. As described in section 9.3 you can select up to ten individual analysis points inside the disk, and in diagram mode you are able to see the calculated values in these points as a function of time.

The analysis points can be pointed out as described in section 9.3 by using the [Ctrl.] button and the mouse, but you can also type the coordinates of the analysis points directly into the Specify analysis points dialog box.

To define an analysis point you may place the cursor in the x-coordinate field and type the global x-coordinate of the current analysis point. Move the cursor to the y-coordinate field (either with the Tab key or the mouse) and type the global y-coordinate of the current point. It is also possible to change the label of the point: Just move the cursor to the label field and write the desired text.

To select the current point you may click the square to the left of the current point so a tickmark is shown.

All ten points can be deleted by clicking the Clear Analysis points button.



The dialog box titled "Specify analysis points" contains a table with columns for "Points:", "X-coordinate:", "Y-coordinate:", and "Label:". There are 10 rows, each representing a point. The first five points are selected (checked), and the last five are not. The "Clear Analysis points" button is at the bottom left, and "OK", "Cancel", and "Help" buttons are on the right.

Points:	X-coordinate:	Y-coordinate:	Label:
<input checked="" type="checkbox"/> Point 1:	9.	9.5	Wall - out.
<input checked="" type="checkbox"/> Point 2:	9.6	9.5	Wall - in.
<input checked="" type="checkbox"/> Point 3:	14.6	12.85	Slab - top
<input checked="" type="checkbox"/> Point 4:	14.6	12.35	Slab - bot.
<input checked="" type="checkbox"/> Point 5:	9.3	7.2	Wall - m
<input type="checkbox"/> Point 6:	0.	0.	Point6
<input type="checkbox"/> Point 7:	0.	0.	Point7
<input type="checkbox"/> Point 8:	0.	0.	Point8
<input type="checkbox"/> Point 9:	0.	0.	Point9
<input type="checkbox"/> Point 10:	0.	0.	Point10

Figure 9.6
Specify analysis points dialog box.

In the dialog box on figure 9.6 five points have been selected. When you click the OK button the five selected points will appear as shown in the example in figure 9.7.

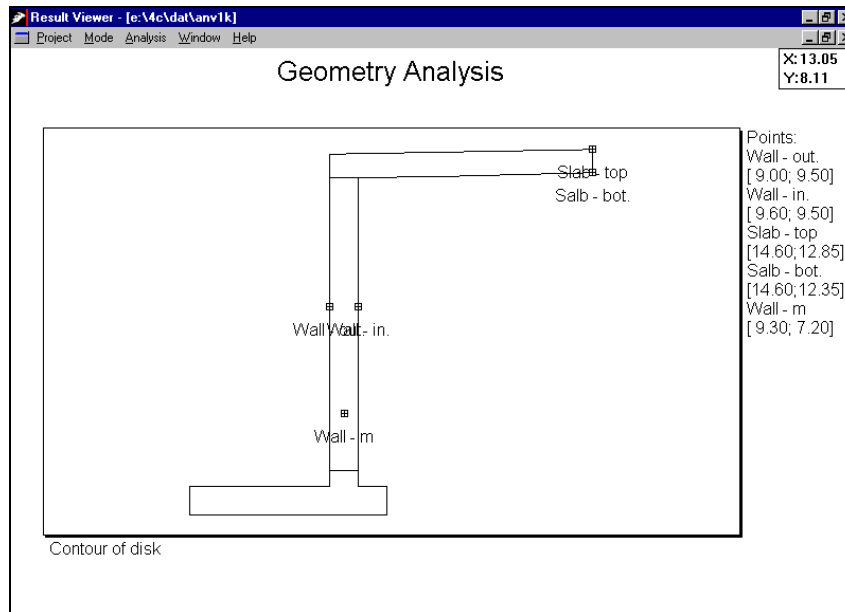


Figure 9.7
Geometry mode - Contour of disk with analysis points.

9.3.3 Volumes

This section describes the menu item [Volumes]. Select the menu item [Analysis] and the menu item [Volumes]. This will bring up the Specify analysis volumes dialog box as shown in figure 9.8. You may select up to five individual volumes in the project for which you want to show the results in diagram and isocurve mode.

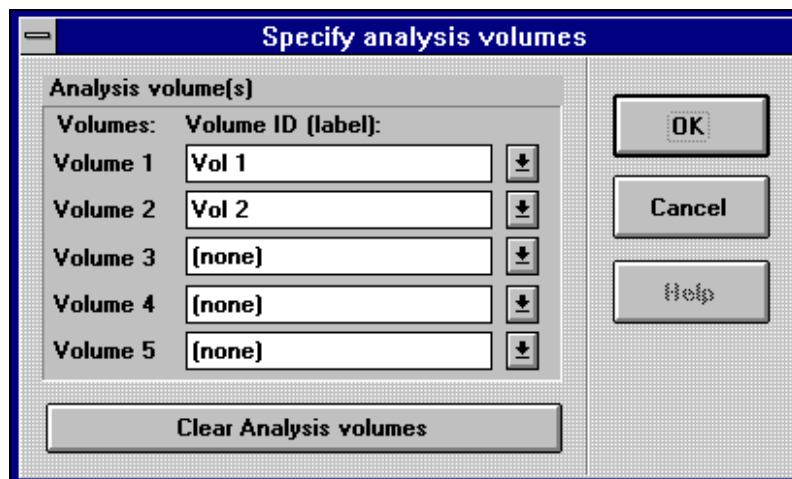


Figure 9.8
Specify analysis volumes dialog box.

Click the down arrow button to the right of the list and select the current volume in the list. The names in the list are the same as the names of the volumes in PE. The five first names in the list will automatically be selected when you open the RV.

The selection of all five volumes can be deleted by clicking the Clear Analysis volumes button.

9.3.4 Geometry System

This section describes the menu item [Geometry system]. Select the menu item [Analysis] and the menu item [Geometry system]. This will make the Geometry system dialog box appear as shown in figure 9.9.

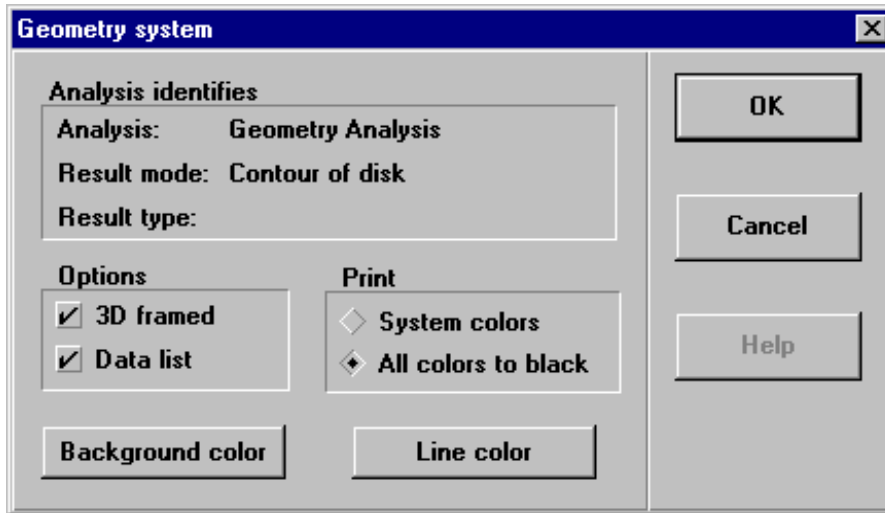


Figure 9.9
Geometry settings dialog box.

The geometry system dialog box contains the identification of the selected view and the general options of the view.

Analysis identification

Identifies the type of analysis and the selected result mode. The result type is never used in geometry mode.

Options

Identifies the general options of the view. When both tickmarks are shown the frame around the view area will be 3-dimensional and the coordinates of the selected analysis points will be shown in the note area. To remove these options just click on the current square so the tickmark disappears.

Print

Identifies the if the system colours are to be used when printing the view or if all colours shall be printed as black. How to print is described in section 9.2. **NB!** System colours will be printed as gray tones unless the printer is able to print colours. Some gray tones may be difficult to see at the print. If this is the case select All colours to black.

Background colour

Identifies the background colour of the view area. To change the colour just click on the background colour button and select the new colour in the colour dialog box that pops up.

Line colour

Identifies the line colour in the view area. To change the colour just click on the line colour button and select the new colour in the colour dialog box that pops up.

9.4 Diagram Mode

This section describes the elements in Diagram mode and the types of results which you are able to present by using the Diagram mode.

Select the menu item [Mode] and the menu item [Diagram] to activate Diagram mode in the RV.

The calculated results are either temperatures, maturities, strengths, approximated stresses or stresses. In Diagram mode it is possible to show all these types of results as a function of time. Further the x/y-diagram can contain either the results in up to ten specified analysis points, the min./max. values for each cast or the average values for each cast. For temperature results temperature differences can be shown as well. How to use the RV in Diagram mode will be described in the following for each of the five types of results: Temperatures, maturities, strength, approximated stresses and stresses.

The different types of diagram views can be selected in the menu item [Analysis]. The menu bar of Analysis includes the items [Points] and [Volumes] which are identical with the items in Geometry mode. See the description of the two items in the sections 9.3.2 and 9.3.3. The menu item [Settings] is connected to the selected analysis type in such a way that there exists a menu item [Settings] for each of the five analysis types. The layout of all the diagram views may be changed as described in section 9.4.6.

9.4.1 Temperature Analysis

In this section the temperature analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Temperature]. This will make the x/y-diagram showing the min./max. temperatures for each cast as a function of time appear. This view is the default view of the temperature analysis. An example of this view is shown for a tunnel structure in figure 9.10.

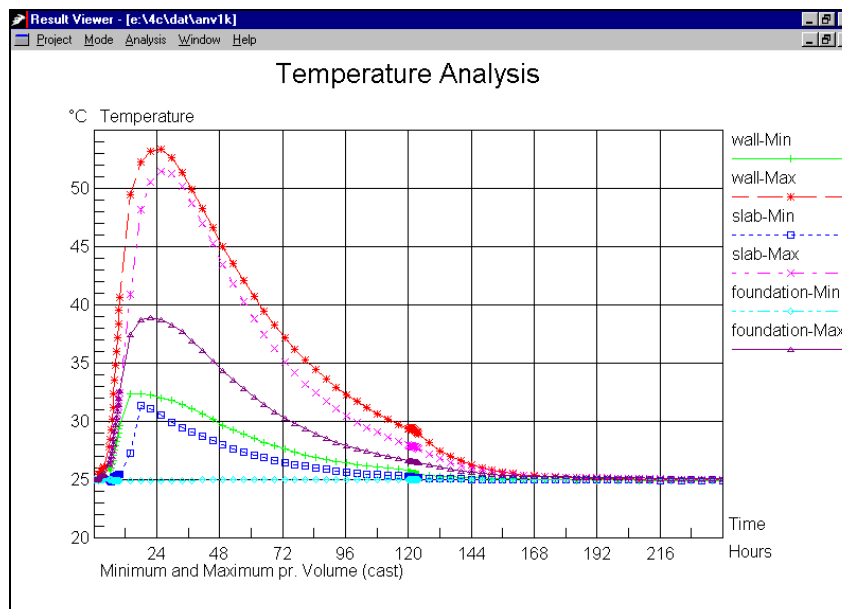


Figure 9.10

Diagram mode - Temperature analysis - Min./max. values for each cast.

To show another view of the temperature analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Diagram settings - temperature analysis dialog box appear as shown in figure 9.11.

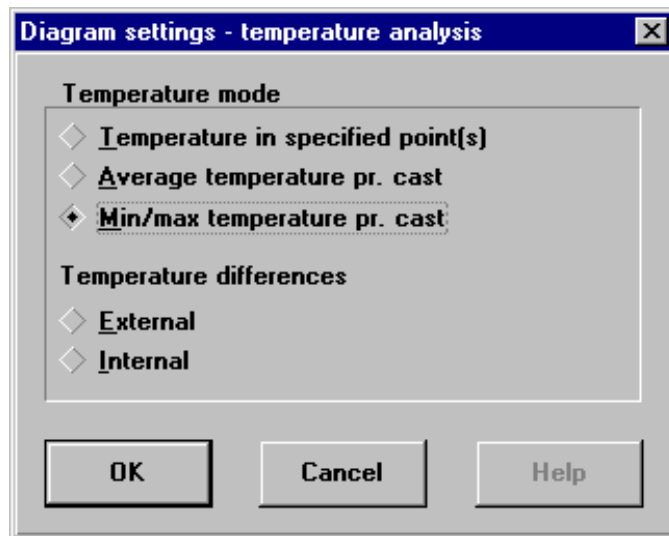


Figure 9.11
Diagram settings - temperature analysis dialog box.

In this dialog box you can select whether you want to show the temperatures in up to ten specified analysis points - see section 9.3 about how to specify the analysis points - the average temperatures for each cast, the min./max. temperatures for each cast, the external or internal temperature differences. To change the selected view, just click the square to the left of the current view.

When selecting the first three temperature modes clicking the OK button will pop up the new view on the screen. Selecting the external or internal difference will bring up new dialog boxes as shown in figure 9.13 and 9.15.

External temperature difference

The external temperature difference D_{ext} is calculated as

$$D_{ext} = \Delta_{NEW} - \Delta_{OLD}$$

where

Δ_{NEW} is the difference between maximum average temperature (observed at time t_{max}) and the actual average temperature in the structural component for which the risk of cracking is assessed (cf figure 9.12)

t_{max} is the time of maximum average temperature in the structural component for which the risk of cracking is assessed

Δ_{OLD} is the difference between the average temperature at time t_{max} and the actual average temperature in the previously cast concrete (cf. figure 9.12)

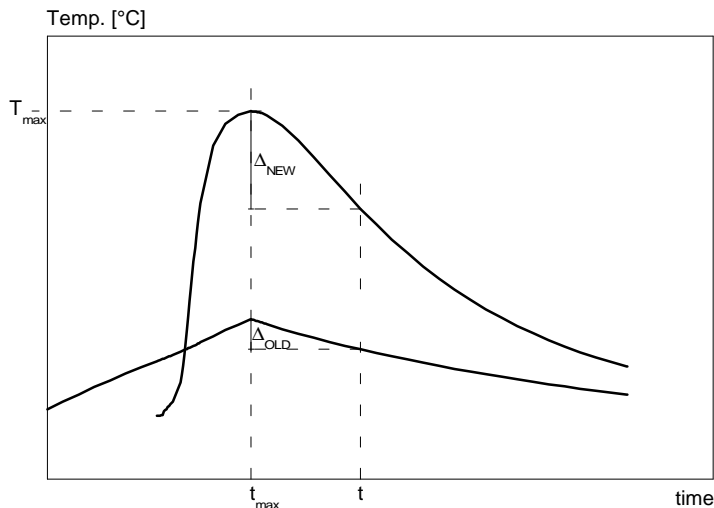


Figure 9.12
The development of the average temperature in adjacent structures

The external temperature difference is specified in the external temperature dialog box as shown in figure 9.13.

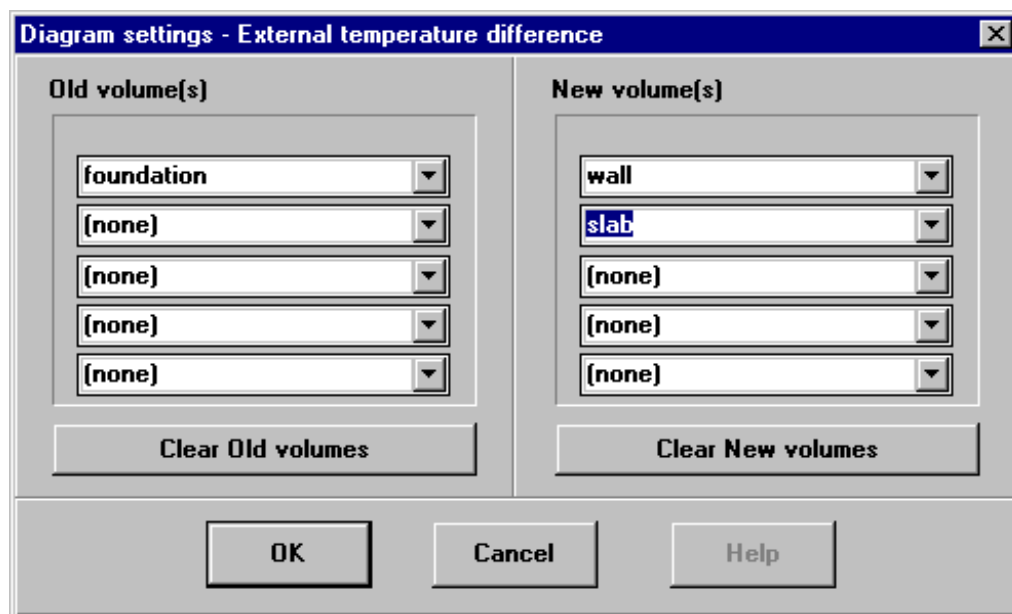


Figure 9.13
External temperature dialog box.

In this dialog box you can select the structural component for which the risk of cracking is assessed and the previously cast concrete. Click the down arrow button to the right of the lists and select the current volume in the list. The names in the lists are the same as the names of the volumes in PE.

Old volume(s): Select the volumes representing previously cast concrete. The average temperature is calculated as the average temperature of the selected volumes. You can select up to five Old volumes.

The selection of all five Old volumes can be deleted by clicking the Clear Old volumes button.

New volume(s): Select the volumes representing the structural component for which the risk of cracking is assessed. The average temperature is calculated as the average temperature of the selected volumes. You can select up to five New volumes.

The selection of all five New volumes can be deleted by clicking the Clear New volumes button.

When you click the OK button the external temperature diagram will pop up on the screen.

In figure 9.13 the external temperature difference between superstructure and foundation of a tunnel structure is selected. This view is shown in figure 9.14.

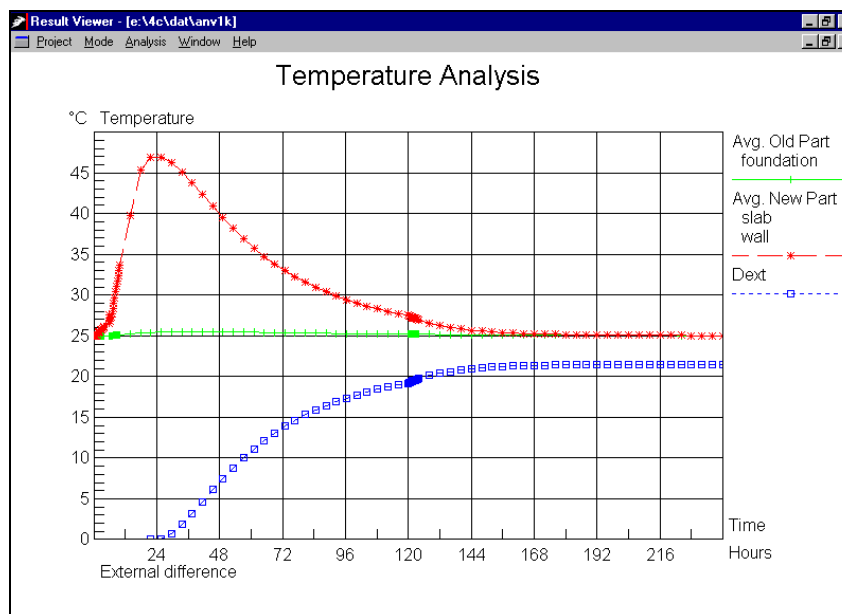


Figure 9.14

Diagram mode - Temperature analysis - External difference

Internal temperature difference

The internal temperature difference in a section is defined as the difference between two of the following temperatures: Average temperature in the section, minimum temperature in the section, maximum temperature in the section and temperature in selected points. The internal temperature difference is specified in the internal temperature dialog box as shown in figure 9.15.

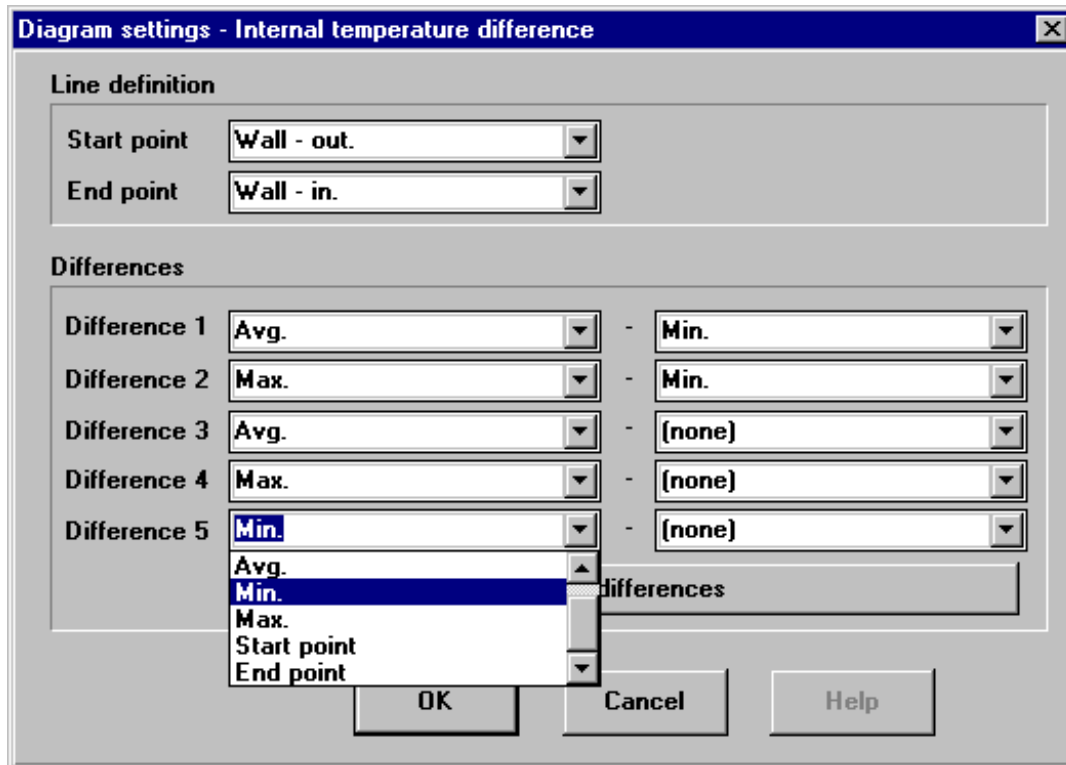


Figure 9.15
Internal temperature dialog box.

The section in which the internal temperature difference is analysed is specified by the line from the start point selected to the end point selected. Click the down arrow button to the right of the lists and select the analysis points to be used.

NB! At least two analysis points must be specified before the internal temperature difference mode can be selected. See section 9.3 about how to specify analysis points.

You can select up to five differences to be viewed. Click the down arrow button to the right of the lists and select the temperature to be used in the difference:

Avg.	average temperature in the section
Min.	minimum temperature in the section
Max.	maximum temperature in the section
Start point	temperature in the start point selected
End point	temperature in the end point selected
(none)	zero temperature

When you click the OK button the internal temperature diagram will pop up on the screen.

In figure 9.15 the internal temperature difference in the wall of a tunnel structure is defined as the Avg.-Min temperature and Max.-Min. respectively. Also the average, minimum and maximum temperatures are selected to be shown. This view is shown in figure 9.16.

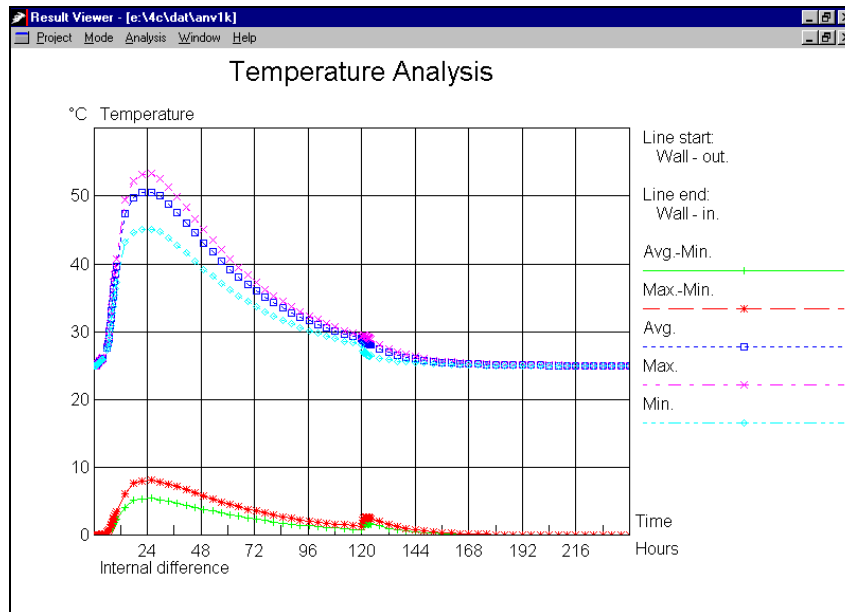


Figure 9.16
Diagram mode - Temperature analysis - Internal differences

9.4.2 Maturity Analysis

In this section the maturity analysis view and its menu item [Settings] will be described. As a part of the temperature calculation 4C-Temp&Stress calculates the developed maturity in every point of the cross section. Similar to the temperatures the maturities can be shown as a function of time.

Select the menu item [Analysis] and the menu item [Maturity]. This will bring up the x/y-diagram showing the min./max. maturities for each cast as a function of time. This view is the default view of the maturity analysis.

To show another view of the maturity analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Diagram settings - maturity analysis dialog box appear as shown in figure 9.17.

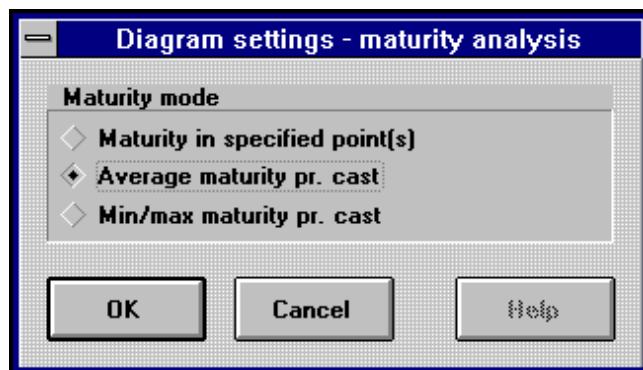


Figure 9.17
Diagram settings - maturity analysis dialog box.

You can select whether you want to show the maturities in up to ten specified analysis points, the average maturities for each cast or the min./max. maturities for each cast. To change the selected view, just click the square to the left of the current view. When you click the OK button the new view will pop up on the screen.

In figure 9.17 the average maturities for each cast are selected and for a tunnel structure this selected view is shown in figure 9.18.

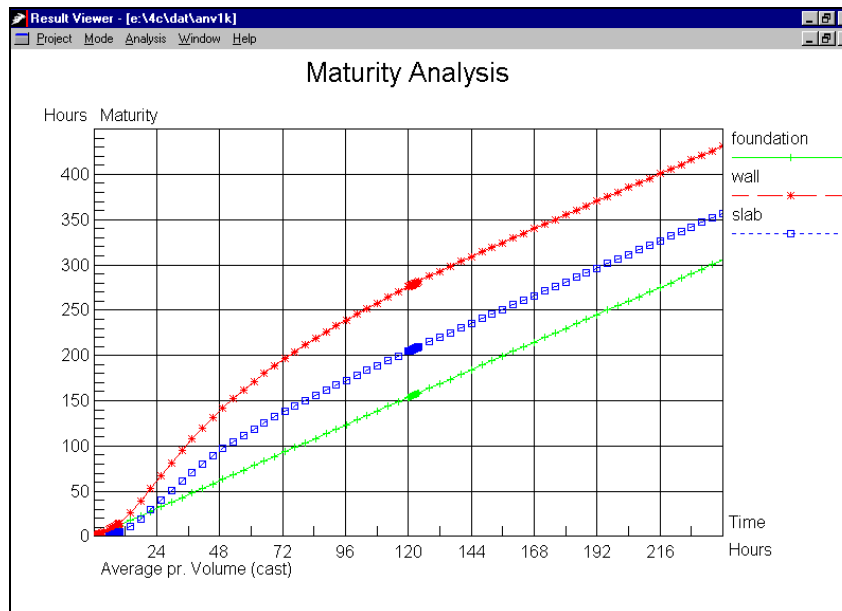


Figure 9.18

Diagram mode - Maturity analysis - Average values for each cast.

9.4.3 Strength Analysis

In this section the strength analysis view and its menu item [Settings] will be described. As a part of the temperature calculation 4C-Temp&Stress calculates the developed strength in every point of the cross section. Similar to the temperatures the strengths can be shown as a function of time.

Select the menu item [Analysis] and the menu item [Strength]. This will bring up the x/y-diagram showing the min./max. compressive strength for each cast as a function of time. This view is the default view of the strength analysis.

To show another view of the strength analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Diagram settings - strength analysis dialog box appear as shown in figure 9.19.

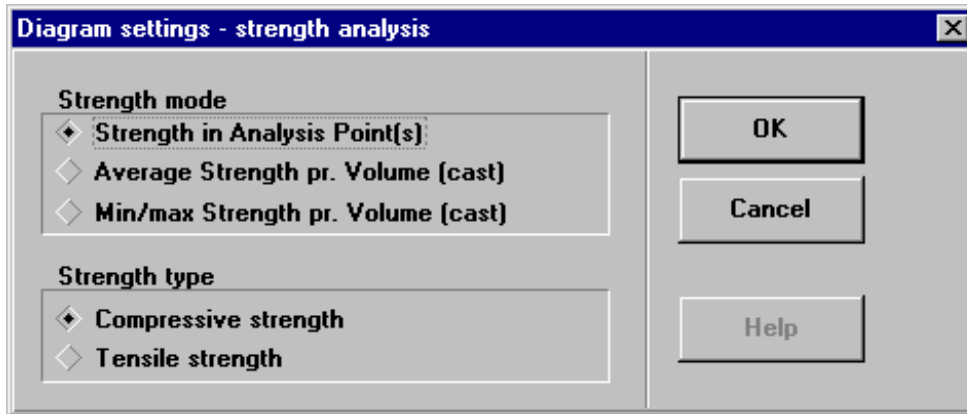


Figure 9.19
Diagram settings - strength analysis dialog box

You can select whether you want to show the strength in up to ten specified analysis points, the average strength for each cast or the min./max. strength for each cast. To change the selected view, just click the square to the left of the current view.

For each of the three modes you can select whether the shown strength shall be the compressive or the tensile strength.

When you click the OK button the new view will pop up on the screen.

In figure 9.19 the compressive strength in specified points is selected and for the tunnel structure this selected view is shown in figure 9.20.

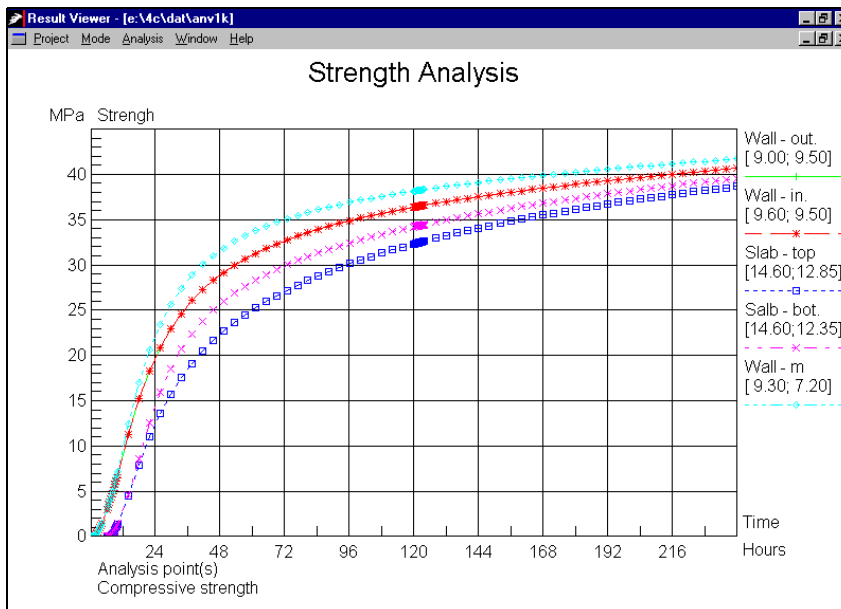


Figure 9.20
Diagram mode - Strength analysis - Compressive strength in selected points.

9.4.4 Approximated Stress Analysis

In this section the approximated stress analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Stress Approximation]. This will make the x/y-diagram showing the min./max. approximated normal stresses, σ_z , for each cast as a function of time appear. This view is the default view of the approximated stress analysis.

To show another view of the approximated stress analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Diagram settings - approximated stress analysis dialog box appear as shown in figure 9.21.

In this dialog box you can select whether you want to show the approximated stresses in up to ten specified analysis points - see section 9.3 about how to specify the analysis points - the average approximated stresses for each cast or the min./max. approximated stresses for each cast. To change the selected view, just click the square to the left of the current view.

For each of the three modes you can select whether the shown approximated stresses shall be the normal stresses, σ_z , or the ratio between the main tensile stress and the current tensile strength.

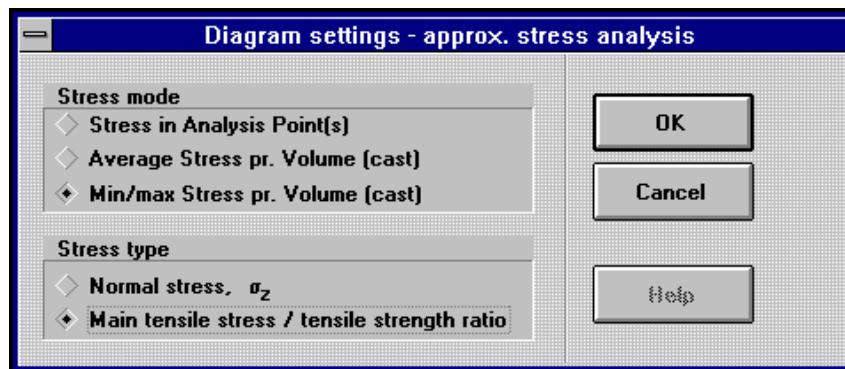


Figure 9.21
Diagram settings - approx. stress analysis dialog box.

When you click the OK button the new view will pop up on the screen.

In figure 9.21 the min./max. ratio between the main tensile approximated stresses and the current tensile strength for each cast are selected and for a tunnel structure this selected view is shown in figure 9.22.

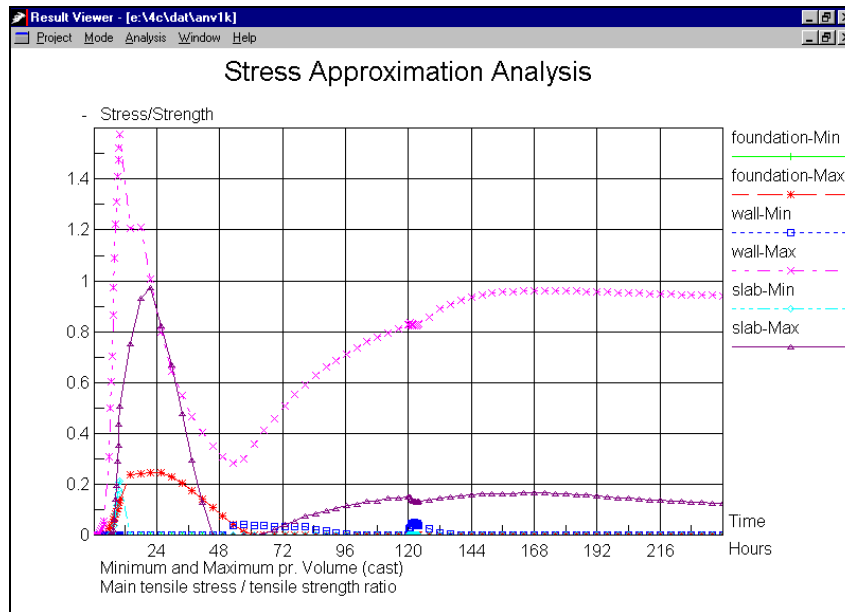


Figure 9.22

Diagram mode - approx. stress analysis - stress/strength ratio.

9.4.5 Stress Analysis

In this section the stress analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Stress]. This will make the x/y-diagram showing the min./max. normal stresses, σ_z , for each cast as a function of time appear. Similar to the approximated stress analysis, this view is the default view of the stress analysis.

To show another view of the stress analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Diagram settings - stress analysis dialog box appear as shown in figure 9.23.

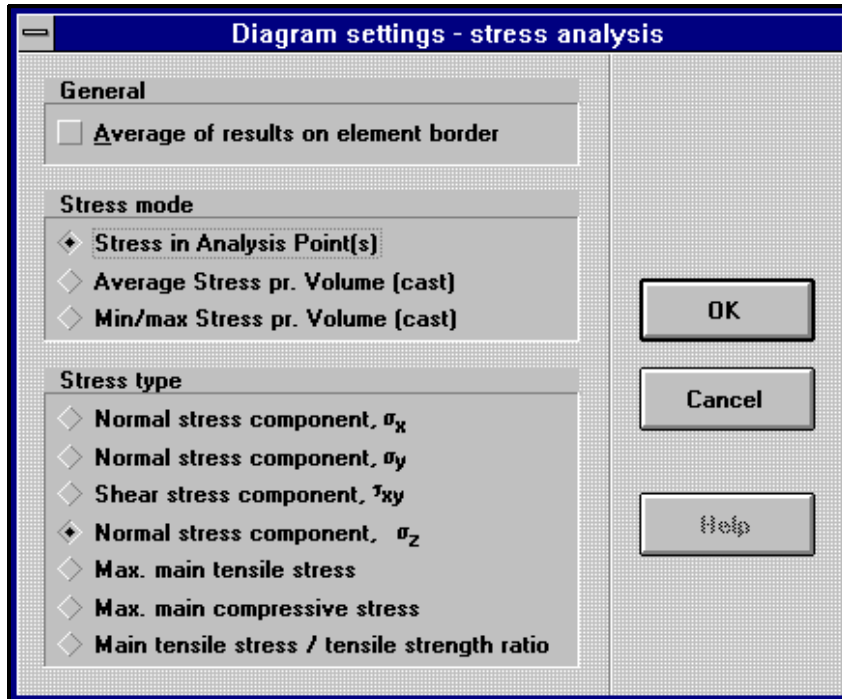


Figure 9.23

Diagram settings - stress analysis dialog box.

In this dialog box you can select whether you want to show the stresses in up to ten specified analysis points - see section 9.3 about how to specify the analysis points - the average stresses for each cast or the min./max. stresses for each cast. To change the selected view, just click the square to the left of the current view.

When the x/y-diagram shows minimum and maximum stresses pay attention to singular points as described in section 6.6.6.

For each of the three modes you can select whether the shown stresses shall be one of the three normal stress components, σ_x , σ_y or σ_z , the shear stress component, τ_{xy} , the maximum main tensile or compressive stresses or the ratio between the main tensile stresses and the current tensile strength.

For all views you can select to average the calculated results on the border of the finite elements. This option is used to obtain more smooth curves.

When you click the OK button the new view will pop up on the screen.

In figure 9.23 the normal stress component, σ_z , in up to ten specified points is selected and for the tunnel structure this selected view is shown in figure 9.24. The specified point is the point near the bottom of the wall shown in figure 9.7.

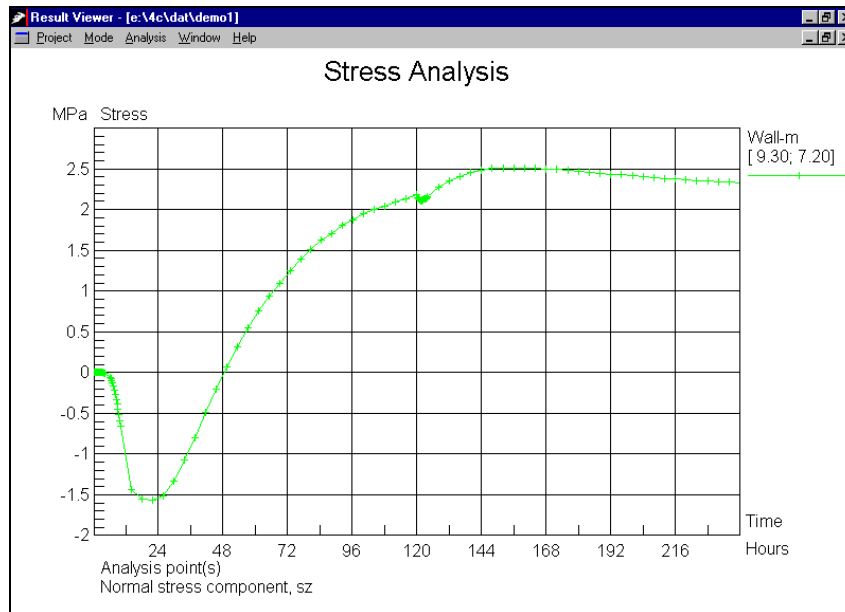


Figure 9.24

Diagram mode - Stress analysis - Normal stress component, σ_z in specified analysis point.

9.4.6 Diagram System

In all the different types of diagrams which are described in the sections above it is possible to change the layout of the diagram such as the grid and the scaling on the axes.

This section describes the menu items [Diagram system], [X-axis] and [Y-axis].

Select the menu item [Analysis] and the menu item [Diagram system]. This will bring up the Diagram system dialog box as shown in figure 9.25.

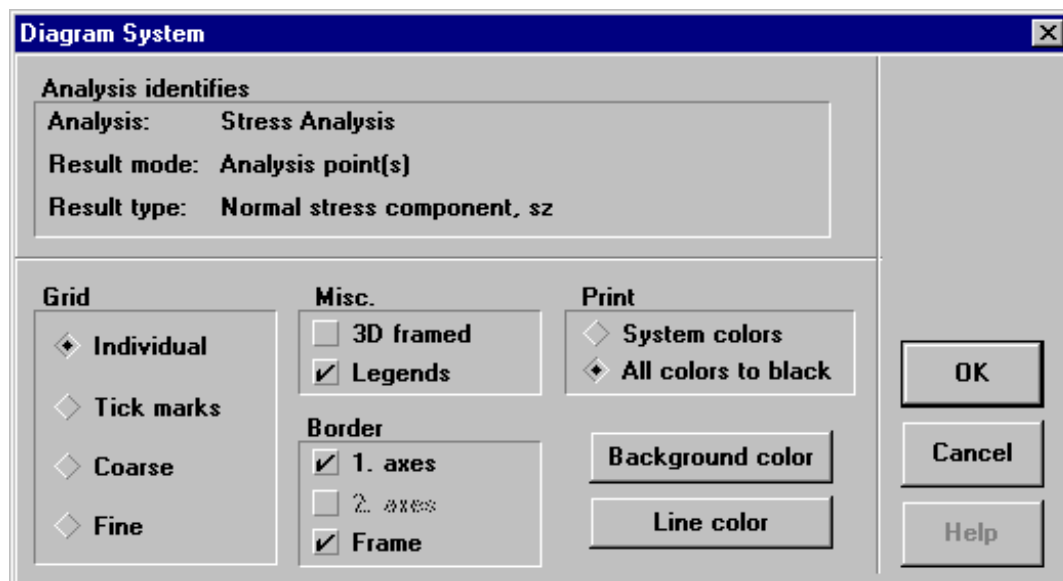


Figure 9.25

Diagram system dialog box.

The diagram system dialog box contains the identification of the selected view and the general options of the view.

Analysis**identification**

Identifies the type of analysis and the selected result mode. The result type is used only for approximated stresses and stresses.

Grid

Identifies the type of grid on both axes.

Individual: The type of grid is defined for each of the axes in each of their dialog boxes

Tickmarks: No grid, only the tickmarks will be shown

Coarse: The grid will follow the coarse tickmarks

Fine: The grid will follow the fine tickmarks.

Misc.

Identifies whether the 3-dimensional frame around the view area and the identification of the curves in the note area (legends) shall be shown or not. When the tickmark is visible the selection will be shown.

Border

Identifies whether the frame and the axes shall be shown or not. When the tickmark is visible the selection will be shown.

Print

Identifies the if the system colours are to be used when printing the view or if all colours shall be printed as black. How to print is described in section 9.2. **NB!** System colours will be printed as gray tones unless the printer is able to print colours. Some gray tones may be difficult to see at the print. If this is the case select All colours to black.

Background colour

Identifies the background colour of the view area. To change the colour just click on the background colour button and select the new colour in the colour dialog box that pops up.

Line colour

Identifies the line colour in the view area. To change the colour just click on the line colour button and select the new colour in the colour dialog box that pops up.

To set up the layout of the x-axis select the menu item [Analysis] and the menu item [X-axis]. This will make the Time axis dialog box appear as shown in figure 9.26.

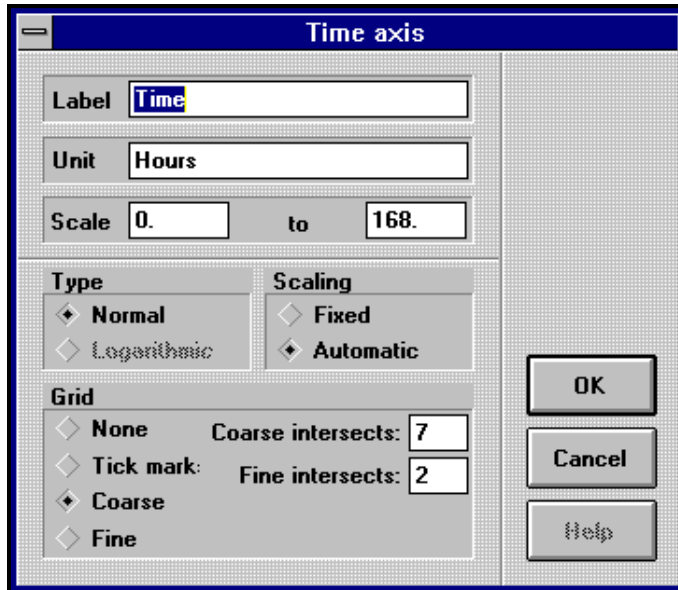


Figure 9.26
Time axis dialog box.

The dialog box contains the identification and the layout of the x-axis.

Label	Identifies the name of the x-axis.												
Unit	Identifies the unit of the values on the x-axis.												
Scale	Identifies the start and end value on the x-axis. If the scaling is fixed the start and end value of the scale may be changed.												
Type	Identifies whether the x-axis is logarithmic or normal. The logarithmic type is not implemented in 4C-Temp&Stress yet.												
Scaling	Identifies whether the number of coarse and fine intersects and the scale are generated automatically or are fixed.												
Grid	Identifies the type of grid on the x-axis. If the scaling is fixed the coarse and fine intersects may be changed.												
	<table> <tr> <td><i>None:</i></td> <td>No vertical grid lines will be shown</td> </tr> <tr> <td><i>Tickmark:</i></td> <td>No grid, only the tickmarks will be shown</td> </tr> <tr> <td><i>Coarse:</i></td> <td>The grid will follow the coarse tickmarks</td> </tr> <tr> <td><i>Fine:</i></td> <td>The grid will follow the fine tickmarks</td> </tr> <tr> <td><i>Coarse intersects:</i></td> <td>The number of coarse intersects</td> </tr> <tr> <td><i>Fine intersects:</i></td> <td>The number of fine intersects.</td> </tr> </table>	<i>None:</i>	No vertical grid lines will be shown	<i>Tickmark:</i>	No grid, only the tickmarks will be shown	<i>Coarse:</i>	The grid will follow the coarse tickmarks	<i>Fine:</i>	The grid will follow the fine tickmarks	<i>Coarse intersects:</i>	The number of coarse intersects	<i>Fine intersects:</i>	The number of fine intersects.
<i>None:</i>	No vertical grid lines will be shown												
<i>Tickmark:</i>	No grid, only the tickmarks will be shown												
<i>Coarse:</i>	The grid will follow the coarse tickmarks												
<i>Fine:</i>	The grid will follow the fine tickmarks												
<i>Coarse intersects:</i>	The number of coarse intersects												
<i>Fine intersects:</i>	The number of fine intersects.												

To set up the layout of the y-axis select the menu item [Analysis] and the menu item [Y-axis]. This will make a dialog box similar to the one for the x-axis shown in figure 9.26 appear. All the set up options are also the same as described for the x-axis.

9.4.7 Saving in datafile

The results corresponding to the different types of diagrams described in the sections above can be saved in a datafile (e.g. temperatures in specified points) .

Specify the diagram to be saved as described in section 9.4.1-9.4.5. Select the menu item [Analysis] and the menu item [Create datafile]. This will bring up the Datafile dialog box as shown in figure 9.27.

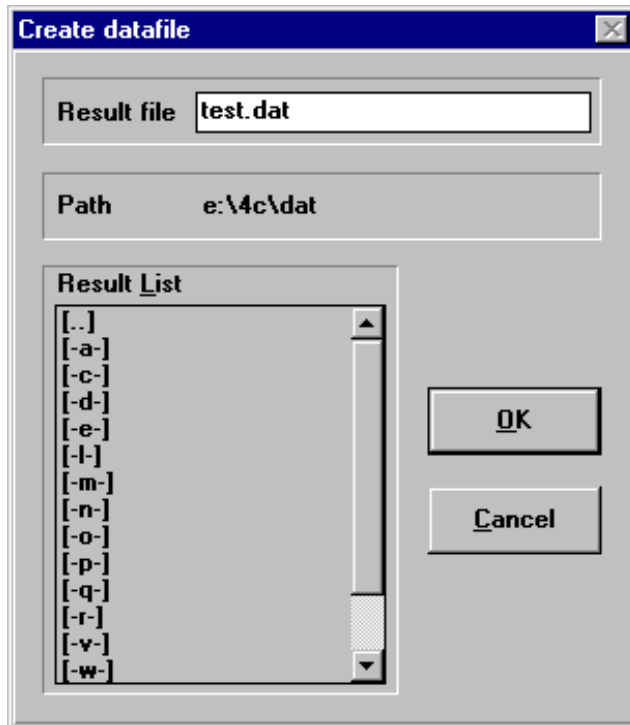


Figure 9.27
Datafile dialog box.

Specify the name of the datafile including the suffix.

NB! The datafile name may consist of maximum 8 characters. The suffix may consist of maximum 3 characters. The characters may be letters, numbers and other symbols like hyphen (-) and percent (%).

The characters may not be the letters: æ, ø and å or symbols like space (), full stop (.), question mark (?) or asterix (*).

When you click the OK button the results corresponding to the selected diagram will be printed in an ASCII-datafile. The first row will contain headings of the columns corresponding to the labels of the diagram. The first column will always contain the time.

NB! If a volume is cast at a time greater than 0, the result value saved in the file will be 99999 for all time steps until casting of the volume independent of the type of diagram.

9.5 Isocurve Mode

This section describes the elements in Isocurve mode and the types of results which you are able to present by using the Isocurve mode.

Select the menu item [Mode] and the menu item [Isocurve] to activate Isocurve mode in the RV.

The calculated results are either temperatures, maturities, strengths, approximated stresses or stresses. In Isocurve mode it is possible to show all these types of results as isocurves at any specified time after casting. How to use the RV in Isocurve mode will be described in the following for each of the five types of results: Temperatures, maturities, strengths, approximated stresses and stresses.

In the same way as in Geometry mode you can **Zoom in** on the view by pressing the [Shift] button on the keyboard down and at the same time with the mouse draw a **rectangle** around the part of the view that you want to zoom in. You can zoom in several times. You can **zoom out** to full view by clicking the left mouse button.

When the calculated values are the same in a whole finite element, isocurves can not be drawn. Instead the finite element mesh will be drawn.

Similar to the Diagram mode the different types of isocurve views can be selected in the menu item [Analysis]. The menu bar of Analysis includes the menu item [Volumes] which is identical with the item in Geometry mode. See the description of the item in section 9.3.3. Similar to the Diagram mode the menu item [Settings] is connected to the selected analysis type in such a way that there exists a menu item [Settings] for each of the five analysis types. The layout of all isocurve views may be changed as described in section 9.5.6.

9.5.1 Temperature Analysis

In this section the temperature analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Temperature]. If no isocurve analysis has been shown before, this will bring up the temperature isocurves at the time zero hours after start calculations. At time zero there are no temperature gradients over the cross section and that is why the whole FEM is shown at time zero. This view is the default view of the temperature analysis. If an isocurve view has been shown before, the temperature isocurves will be shown at the same time as the previous one.

To show another view of the temperature analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Isocurve settings - temperatures dialog box appear as shown in figure 9.28.

In this dialog box you can specify the desired type of view.

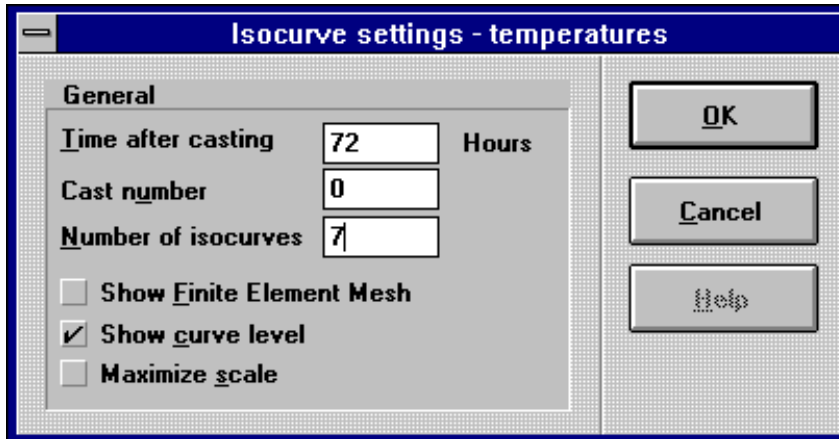


Figure 9.28
Isocurve settings - temperatures dialog box.

- Time after casting** Identifies time after start calculations where the isocurves will be shown [h].
- Cast number** Identifies the number of the casting (volume) in which the isocurves will be shown. The number corresponds to the succession shown in the list in the Specify analysis volumes dialog box. See figure 9.8. The value zero will cause isocurves to be shown in the selected volumes. See section 9.3.3. If no volumes are selected isocurves will be shown in the hole cross section.
- Number of isocurves** Identifies the number of isocurves to be shown.
- Show FEM** When a tickmark is visible in the square the generated finite element mesh will be shown together with the isocurves.
- Show curve level** When a tickmark is visible in the square the values of the isocurves will be written on the curves.
- Maximize scale** When a tickmark is visible in the square the view of the cross section will be zoomed out as much as possible in both x- and y- direction individually. This means that the geometrical shape of the cross section may be changed.

When you have specified the new type of view just click the **OK** button and the new view will pop up on the screen.

In figure 9.28 seven temperature isocurves together with curve levels at the time 72 hours after start calculations are selected and this selected view for a tunnel structure is shown in figure 9.29.

All the important data of the view such as time after start calculations and min. and max. temperatures for each casting are written in the note area to the right of the view area.

Some of the values of the isocurves can be very difficult to read, but you can either select a lower number of isocurves or you can zoom in on the interesting part of the view.

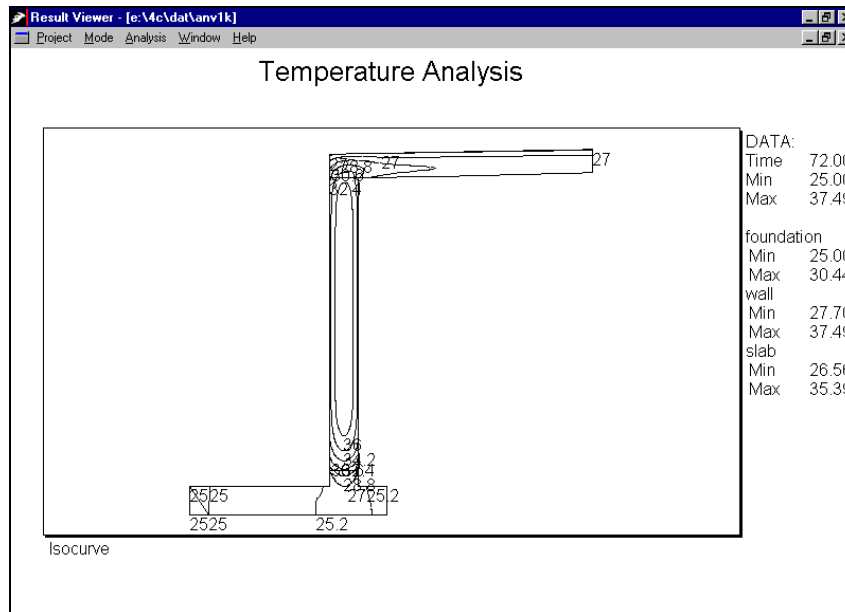


Figure 9.29
Isocurve mode - Temperature analysis.

9.5.2 Maturity Analysis

In this section the maturity analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Maturity]. If no isocurve analysis has been shown before, this will bring up the maturity isocurves at the time zero hours after start calculations. Similar to the temperature analysis there are no maturity gradients over the cross section at time zero and that is why the whole FEM is shown at time zero. This view is the default view of the maturity analysis similar to the temperature analysis. If an isocurve view has been shown before, the maturity isocurves will be shown at the same time as the previous one.

To show another view of the maturity analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Isocurve settings - maturity dialog box appear as shown in figure 9.30.

This dialog box is similar to the dialog box for temperature analysis shown in figure 9.28.

When you have specified the new type of view, just click the OK button and the new view will pop up on the screen.

In figure 9.30 seven maturity isocurves together with curve levels at the time 72 hours after start calculations are selected and this selected view for a tunnel structure is shown in figure 9.31.

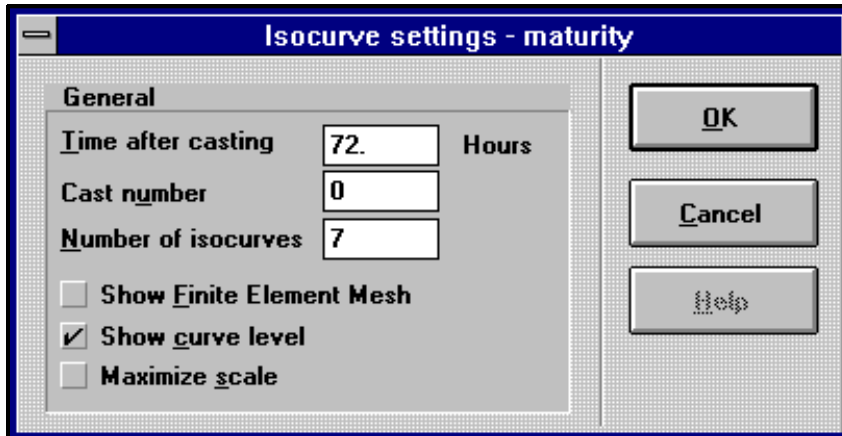


Figure 9.30
Isocurve settings - Maturity dialog box.

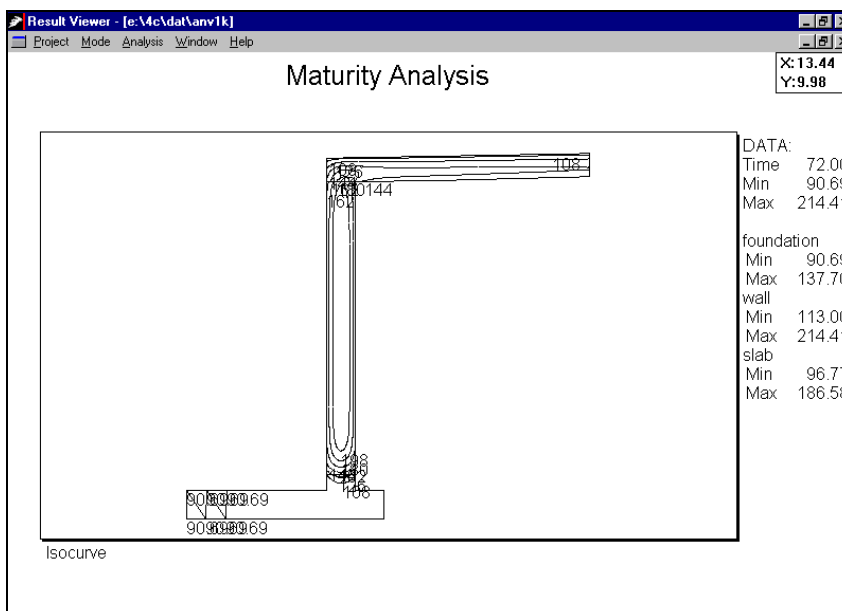


Figure 9.31
Isocurve mode - Maturity analysis.

9.5.3 Strength Analysis

In this section the strength analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Strength]. If no isocurve analysis has been shown before, this will bring up the isocurves at the time zero hours after start calculations.

If an isocurve view has been shown before, the strength isocurves will be shown at the same time as the previous one.

To show another view of the strength analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Isocurve settings - strength dialog box appear as shown in figure 9.32.

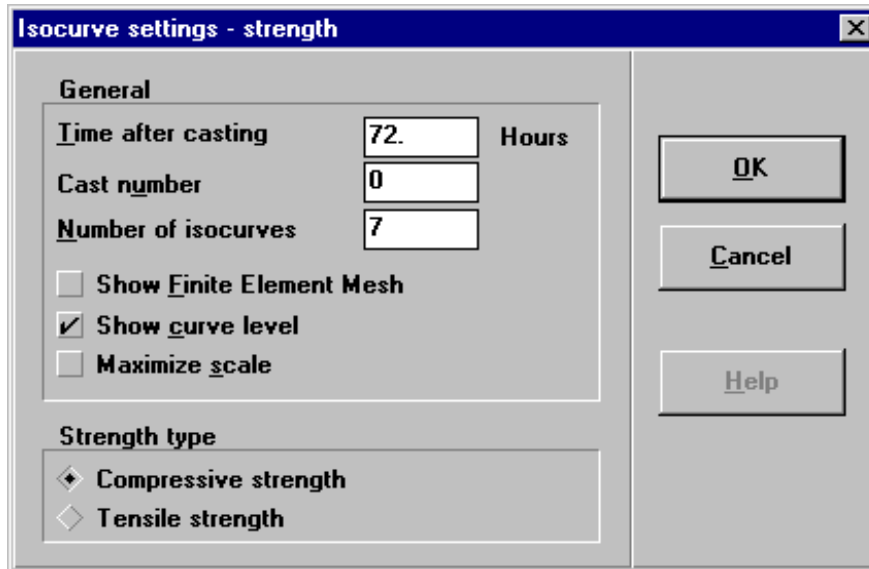


Figure 9.32
Isocurve settings - strength dialog box.

This dialog box contains the same general settings as the dialog boxes for temperature and maturity. Further it is possible to select whether the isocurves are going to show the compressive or the tensile strength.

In figure 9.32 seven compressive strength isocurves together with curve levels at the time 72 hours after start calculations are selected and this selected view for the tunnel structure is shown in figure 9.33.

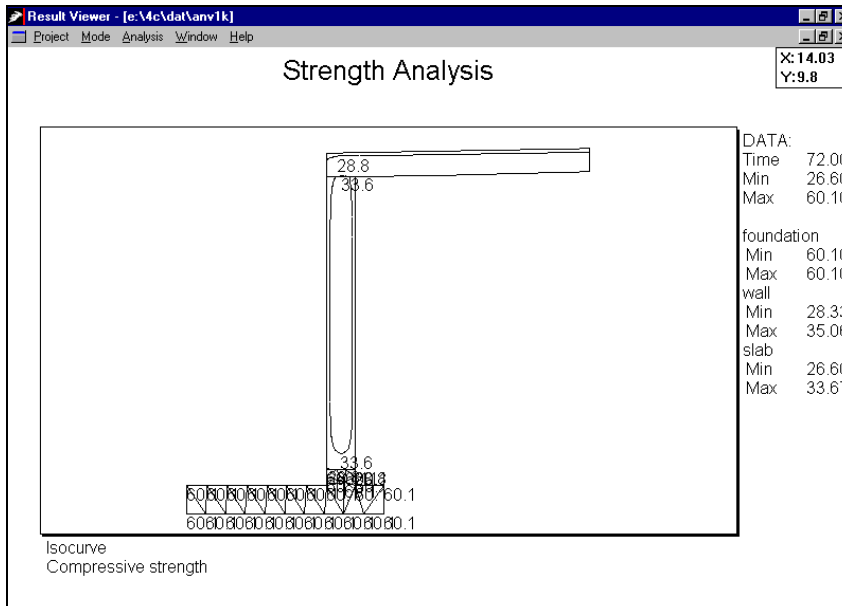


Figure 9.33
Isocurve mode - Strength analysis - compressive strength

9.5.4 Approximated Stress Analysis

In this section the approximated stress analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Stress approximation]. If no isocurve analysis has been shown before, this will bring up the isocurves at the time zero hours after start calculations.

If an isocurve view has been shown before, the approximated stress isocurves will be shown at the same time as the previous one.

To show another view of the approximated stress analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Isocurve settings - approximated stresses dialog box appear as shown in figure 9.34.

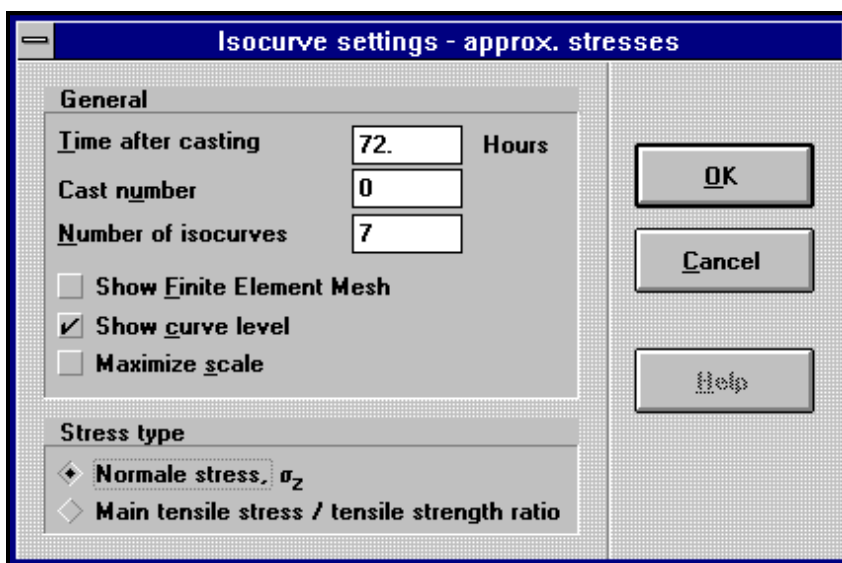


Figure 9.34
Isocurve settings - approximated stresses dialog box.

This dialog box contains the same general settings as the dialog boxes for temperature and maturity. Further it is possible to select whether the isocurves are going to show the approximated normal stresses, σ_z , or the ratio between the main tensile approximated stresses and the current tensile strength.

In figure 9.34 seven approximated normal stress isocurves together with curve levels at the time 72 hours after start calculations are selected and this selected view for a tunnel structure is shown in figure 9.35.

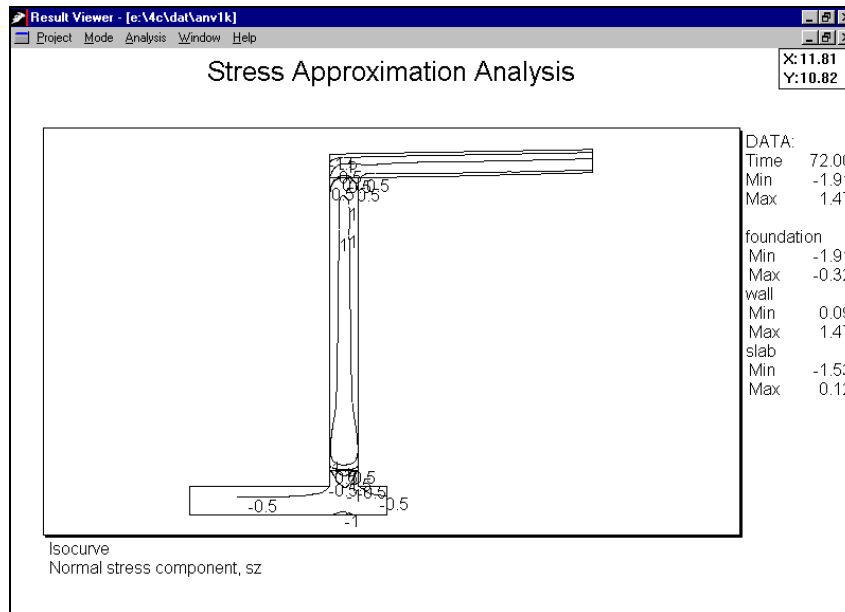


Figure 9.35
 Isocurve mode - Approximated stress analysis - Normal stress component, σ_z .

9.5.5 Stress Analysis

In this section the stress analysis view and its menu item [Settings] will be described.

Select the menu item [Analysis] and the menu item [Stress]. If no isocurve analysis has been shown before, this will bring up the isocurves at the time zero hours after start calculations. If an isocurve view has been shown before, the stress isocurves will be shown at the same time as the previous one.

To show another view of the stress analysis select the menu item [Analysis] and the menu item [Settings]. This will make the Isocurve settings - stresses dialog box appear as shown in figure 9.36.

This dialog box contains the same general settings as the dialog boxes for temperature, maturity and approximated stress. See section 9.5.1.

Besides, you can select to average the calculated results on the borders of the finite elements. This option is used to obtain continuous isocurves. If you do not select to average the results on the borders of the finite elements, you can see whether the generated FEM is too rough to describe the calculated distribution of the stresses. If the isocurves are very discontinuous a finer FEM is required.

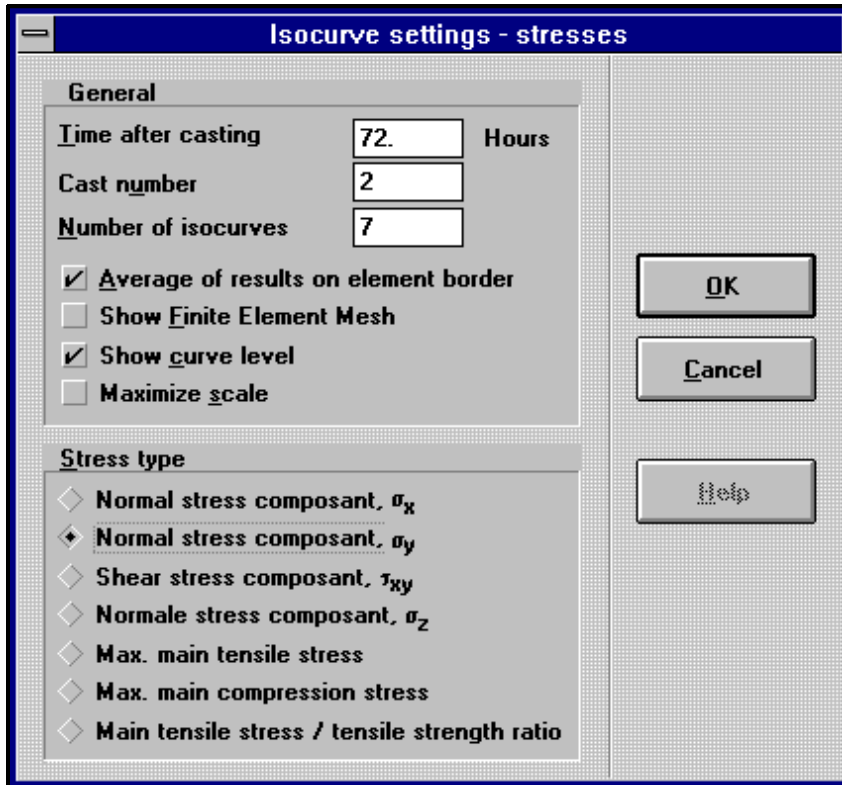


Figure 9.36
Isocurve settings - stresses dialog box.

You can select whether the isocurves are going to show one of the three normal stress components, σ_x , σ_y or σ_z , the shear stress component, τ_{xy} , the maximum main tensile or compressive stresses or the ratio between the main tensile stresses and the current tensile strength.

When you click the OK button the new view will pop up on the screen.

In figure 9.36 seven normal stress component, σ_y , isocurves in volume No. 2 together with curve levels and average results on element borders at the time 72 hours after start calculations are selected and this selected view for a tunnel structure is shown in figure 9.37.

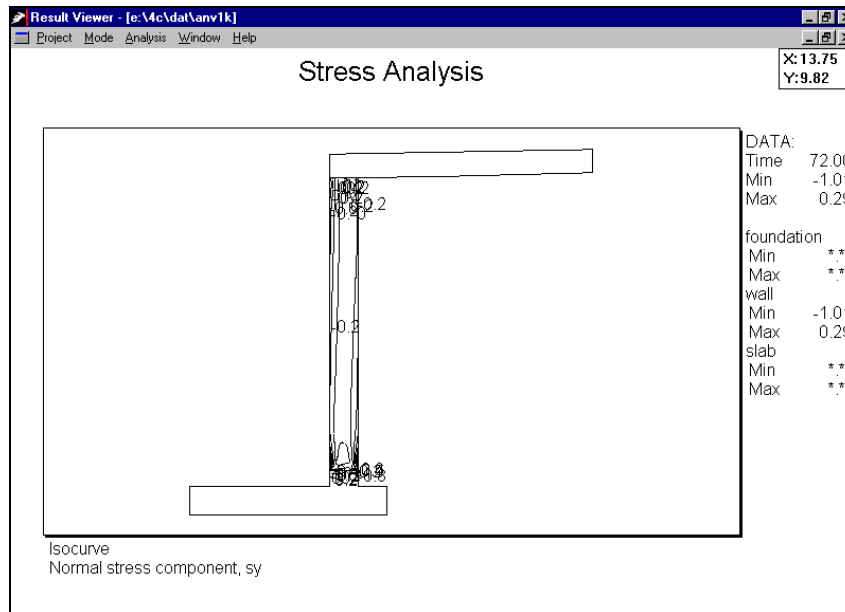


Figure 9.37

Isocurve mode - Stress analysis - Normal stress component, σ_y .

9.5.6 Isocurve System

In all the different types of isocurve views described in the sections above it is possible to change the general layout of the view.

Select the menu item [Analysis] and the menu item [Isocurve system]. This will make the Isocurve system dialog box appear as shown in figure 9.38.

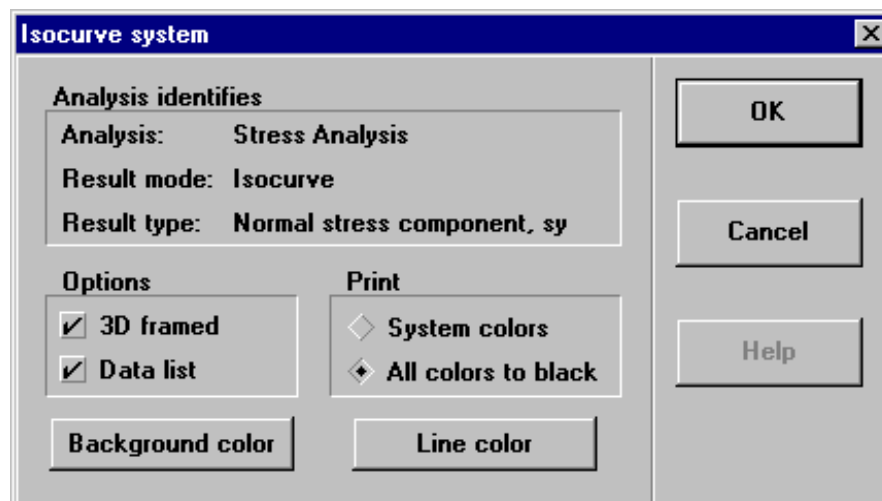


Figure 9.38

Isocurve system dialog box.

The Isocurve system dialog box contains the identification of the selected view and the general options of the view.

Analysis identification	Identifies the type of analysis and the selected result mode. The result type is used only for approximated stresses and stresses.
Options	Identifies whether the 3-dimensional frame around the view area and the identification of the isocurves in the note area shall be shown or not. When the tickmark is visible the selection will be shown.
Print	Identifies the if the system colours are to be used when printing the view or if all colours shall be printed as black. How to print is described in section 9.2. NB! System colours will be printed as gray tones unless the printer is able to print colours. Some gray tones may be difficult to see at the print. If this is the case select All colours to black.
Background colour	Identifies the background colour of the view area. To change the colour just click on the background colour button and select the new colour in the colour dialog box that pops up.
Line colour	Identifies the line colour in the view area. To change the colour just click on the line colour button and select the new colour in the colour dialog box that pops up.

9.6 Windows View

As described in section 9.2 it is possible to have results from several projects in the RV at the same time. When several result files are loaded into the RV some of the result windows are hidden behind the others. To arrange the windows select the menu item [Window].

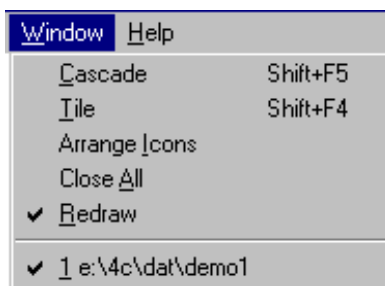


Figure 9.39
Menu items in [Window].

Cascade	Click the Cascade item and the loaded result windows will be resized and arranged overlapped with each title bar visible.
Tile	Click the Tile item and the loaded result windows will be resized and arranged side by side.

If you load another result window after you use the Cascade or Tile item, it will overlap the arranged windows. To include it among the arranged windows, select the Cascade or Tile item again.

Arrange Icons If you have minimized some of the result windows by clicking the minimize button an icon will pop up on the screen. If you have many icons you can arrange the icons by selecting the menu item [Arrange Icons].

The minimized result windows will be maximized again if you click on the current icon and select the menu item [Restore] or if you double click on the icon.

Close All Click the [Close all] item and all the result files will be closed and the RV will be empty.

Redraw When a tickmark is shown at the [Redraw] item the window in the RV will be redrawn every time one of the menu bars have been opened. If no tickmark is shown the window in the RV will be empty until the right mouse button is double clicked or the [Redraw] item is selected again.

This item makes it possible much faster to define all options of a view, such as mode, analysis type, settings and axes.

List of files All the windows with loaded results can be selected in the list of files by clicking the current file. This will alter the selected window into the active window and the view in the window can then be changed.

10. Tutorials

10.1 Temperature calculation

This section describes how to define projects in order to make temperature calculations. If approximated stress and/or stress calculations are to be performed later on read section 10.2 or 10.3 instead.

1. Define material properties

Select the menu item [Database] and the relevant sub item [Concrete], [Soil] and/or [Materials]. The properties to be defined are shown in table 10.1.

Property	Concrete	Soil	Material
Specific heat and density	+	+	+
Thermal conductivity	+	+	+
Heat development	+	-	-
Compressive strength	(+)	(+)	(+)
Tensile strength	(+)	(+)	(+)
See section	7.2	7,5	7,4

Table 10.1

Properties needed for temperature calculations

+ *Required.*

(+) *Optional. Strength values are only needed if you want the development of the strength in the structure to be calculated. The strength does not influence the temperature calculation.*

You may be prompted for other properties when leaving a dialog box. You can type any value as these do not influence the temperature calculation.


2. Define cross-section

Click  to select volume mode.


Draw the geometry as described in section 6.2.2 - 6.2.4. If necessary the view of the screen can be changed as described in section 6.2.1.

Select material type (e.g. concrete) and material name and define start time and temperature as described in section 6.3.1. The thickness can be set to 1 (does not influence the temperature calculation).

3. Define thermal boundary conditions

Click  to select surface mode, then double click on the desired surface. Define the thermal boundary condition as described in section 6.3.2.1.



4. Define cooling pipes/heating wires

Click  and then click at the desired position of the cooling pipe / heating wire. Define the boundary condition as described in section 6.3.3.


5. Define calculation parameters

Select the menu item [Calculation] and the sub item [Parameters]. Define the parameters as described in section 6.6. Select Dimensions to 2-dimensional, otherwise approximated stresses will be calculated (if allowed by the hardware-key). Define Time specifications and change Mesh Node generation if necessary.

6. Calculate

Click  to generate the FEM mesh. Click  to calculate temperatures.

7. View results

Click  to view results as described in section 9.

8. Documentation

Material properties

Print out the material properties by using the menu item [Database] and the relevant sub-item [Concrete], [Soil] or [Materials]. Click on the desired concrete/soil/material and select **Print**.

Geometry, boundary conditions etc.

Print the subject shown on screen by selecting the menu item [Project] and the sub item [Print subject].

Print out the functions used (temperature, wind, etc.) by using the menu item [Project] and the sub item [Print functions].

Print out the calculation parameters by using the menu item [Project] and the sub item [Print calculation parameters].

Results

Print out the results shown in the Result Viewer. Give the desired result type (see sect.9). Select the menu item [Project] and the sub item [Print view].

10.2 Temperature and approximated stress calculation

This section describes how to define projects in order to make temperature and approximated stress calculations. If stress calculations are to be performed later on read section 10.3 instead.

1. Define material properties

Select the menu item [Database] and the relevant sub item [Concrete], [Soil] and/or [Materials]. The properties to be defined are shown in table 10.2.

Property	Concrete	Soil	Material
Specific heat and density	+	+	+
Thermal conductivity	+	+	+
Heat development	+	-	-
E-modulus	+	+	+
Thermal expansion coefficient	+	+	+
Initial strain (autogenous shrinkage)	(+)	-	-
Creep	(+)	-	-
Compressive strength	(+)	(+)	(+)
Tensile strength	+	+	+
See section	7.2	7.5	7.4

Table 10.2

Properties needed for temperature calculations

+ *Required.*

(+) *Optional. Compressive strength values are only needed if you want the development of the strength in the structure to be calculated. Neither the compressive nor the tensile strength influence the temperature or stress calculation.*

You may be prompted for other properties when leaving a dialog box. You can type any value as these do not influence the calculations.


2. Define cross-section

Click  to select volume mode.


Draw the geometry as described in section 6.2.2 - 6.2.4. If necessary the view of the screen can be changed as described in section 6.2.1.

Select material type (e.g. concrete) and material name and define start time and temperature as described in section 6.3.1. The thickness can be set to 1 (does not influence the temperature or approximated stress calculation).

3. Define thermal boundary conditions



Click  to select surface mode, then double click on the desired surface. Define the thermal boundary condition as described in section 6.3.2.1.



4. Define cooling pipes/heating wires

Click  and then click at the desired position of the cooling pipe / heating wire. Define the boundary condition as described in section 6.3.3.

5. Define loads

Click  to select Load Mode.

Click  and  to add Z-point loads as described in section 6.4.3. XY-point loads and line-loads will not influence the calculated stresses.

Click  to add a X-momentum or  to add an Y-momentum as described in section 6.4.5.


6. Define calculation parameters

Select the menu item [Calculation] and the sub item [Parameters]. Define the parameters as described in section 6.6. Select Dimensions to 2½-dimensional and define the requirements to the strain plane (section 6.6.4). Define Time specifications and change Mesh Node generation if necessary.

7. Calculate

Click  to generate the FEM mesh. Click  to calculate temperatures and approximated stresses.

8. View results

Click  to view results as described in section 9.

9. Documentation

Material properties

Print out the material properties by using the menu item [Database] and the relevant sub-item [Concrete], [Soil] or [Materials]. Click on the desired concrete/soil/material and select **Print**.

Geometry, boundary conditions etc.

Print the subject shown on screen by selecting the menu item [Project] and the sub item [Print subject].

Print out the functions used (temperature, wind, etc.) by using the menu item [Project] and the sub item [Print functions].

Print out the calculation parameters by using the menu item [Project] and the sub item [Print calculation parameters].

Results

Print out the results shown in the Result Viewer. Give the desired result type (see sect.9). Select the menu item [Project] and the sub item [Print view].

10.3 Temperature and stress calculation

This section describes how to define projects in order to make temperature and stress calculations.

1. Define material properties

Select the menu item [Database] and the relevant sub item [Concrete], [Soil] and/or [Materials]. The properties to be defined are shown in table 10.3.

Property	Concrete	Soil	Material
Specific heat and density	+	+	+
Thermal conductivity	+	+	+
Heat development	+	-	-
E-modulus	+	+	+
Thermal expansion coefficient	+	+	+
Poisson ratio	+	+	+
Initial strain (autogenous shrinkage)	(+)	-	-
Creep	(+)	-	-
Compressive strength	(+)	(+)	(+)
Tensile strength	+	+	+
See section	7.2	7.5	7.4

Table 10.3

Properties needed for temperature calculations

+ Required.

(+) Optional. Compressive strength values are only needed if you want the development of the strength in the structure to be calculated. Neither the compressive nor the tensile strength influence the temperature or stress calculation.

You may be prompted for other properties when leaving a dialog box. You can type any value as these do not influence the calculations.


2. Define cross-section

Click  to select volume mode.


Draw the geometry as described in section 6.2.2 - 6.2.4. If necessary the view of the screen can be changed as described in section 6.2.1.

Select material type (e.g. concrete) and material name and define start time and temperature as described in section 6.3.1. The thickness influence calculated stresses in disk calculations. If disk calculations are not to be performed the thickness can be set to 1.

3. Define thermal boundary conditions

Click  to select surface mode, then double click on the desired surface. Define the thermal boundary condition as described in section 6.3.2.1.



4. Define cooling pipes/heating wires



Click  and then click at the desired position of the cooling pipe / heating wire. Define the boundary condition as described in section 6.3.3.

5. Define loads

Click  to select Load Mode.




Click  and  or  to add point loads as described in section 6.4.3.

Click  and  to add line loads as described in section 6.4.4.

Click  to add a X-momentum or  to add an Y-momentum as described in section 6.4.5.

6. Define supports



Click  to select Load Mode.

Click  or  to select point or line supports. Select the node or the surface to be supported and click  to add the support. Define the support as described in section 6.4.1.


7. Define calculation parameters

Select the menu item [Calculation] and the sub item [Parameters]. Define the parameters as described in section 6.6. Select Dimensions to 2-dimensional for disk calculations or 2½-dimensional for plain strain situation. If 2-dimensional select plane stress or strain. For 2½-dimensional define the requirements to the strain plane (section 6.6.4). Define Time specifications and change Mesh Node generation if necessary.

8. Calculate

Click  to generate the FEM mesh. Click  to calculate temperatures and approximated stresses. Select the menu item [Calculate] and the sub item [Stress] to calculated stresses.

9. View results

Click  to view results as described in section 9.

10. Documentation

Material properties

Print out the material properties by using the menu item [Database] and the relevant sub-item [Concrete], [Soil] or [Materials]. Click on the desired concrete/soil/material and select **Print**.

Geometry, boundary conditions etc.

Print the subject shown on screen by selecting the menu item [Project] and the sub item [Print subject].

Print out the functions used (temperature, wind, etc.) by using the menu item [Project] and the sub item [Print functions].

Print out the calculation parameters by using the menu item [Project] and the sub item [Print calculation parameters].

Results

Print out the results shown in the Result Viewer. Give the desired result type (see sect.9). Select the menu item [Project] and the sub item [Print view].