RM2000

Static and Dynamic Analysis

of Spaceframes

USER GUIDE

TDV Ges.m.b.H.

October 2003

Disclaimer and Copyright

Disclaimer

Much time and effort have gone into the development and documentation [i]of *RM2000* and *GP2000*. The programs have been thoroughly tested and used.

The user accepts and understands that no warranty is expressed or implied by the developers or the distributors on the accuracy or the reliability of the program.

The user must understand the assumptions of the program and must apply engineering knowledge and skill to independently verify the results.

Copyright

Contents

1 Program Structure and Functionality

1.1 Program Data File Structure

The program files are established in the "program directory" during the installation process. Additional authorization files (licence files – provided when the program/ module is purchased) that act together with a specific hardlock security device are also necessary for using the program. The installation procedure and the authorization procedure for *RM2000* are described in detail in the Installation Guide.

The installation procedure generates a directory TDV2000 as a subdirectory of the selected installation path. This directory contains the general TDV configuration directory *ETC*, the resource directory *RES* and the Program Directory *RM8*. The Installation Guide document is part of the program and is located in *DOC* the RM8 subdirectory:

TDV2000INST.PDF	Installation Guide (in English)
TDV2000INSTG.PDF	Installation Guide (in German)

1.1.1 Program Data

The Program Directory contains the following files:

RM2000.EXE	Executable Program	
RM2000.TXD	Text-Database (for dialogue and output listings)	
RM2000.TXI	Index files for the text-database	
*.RMD	Provided tables for data import (TCL-Files)	
MAT-*.RMD	Material tables for different design codes	
	MAT-BS.RMD	British Standard BS5400
	MAT-DIN1.RMD	DIN 1045
	MAT-DIN2.RMD	DIN(18800, EC3)
	MAT-HUNG.RMD	Hungarian Code
	MAT-JAP.RMD	Japanese Norm - JIS
	MAT-NOR.RMD	Norwegian Norm - NS
	MAT-OE1.RMD	OENORM B4200
	MAT-OE2.RMD	OENORM B4700
	MAT-POR.RMD	Portuguese Code
	MAT-USA.RMD	American Standard AASHTO

CS-*.RMD	Standard tables for Creep Variables definition:	
	CS-AS96.RMD	AASHTO Model Code 96
	CS-B54.RMD	
	CS-BS54.RMD	BS5400 Model Code
	CS-CEB78.RMD	CEB-FIP Model Code 78
	CS-CEB90.RMD	CEB-FIP Model Code 90
	CS-DI45.RMD	DIN1045 Model Code
	CS-H54.RMD	Hong Kong Model Code
	CS-HS54.RMD	
	CS-HUNG.RMD	Hungarian Code
	CS-NOR.RMD	Norwegian Standard
	CS-OE47.RMD	OENORM B4700 Model
	CS-RSM90.RMD	Revised Summation Model

PL-*.RM	Standard parameters for the graphic presentation (Plots)
……	
PROF.DAT	Cross–section table for standardized steel profiles

The documentation, which can be read directly from the screen and/or printed out, is stored in PDF format in the subdirectory *DOC*. Sketches and pictures referenced and used in the Help-System and in the documentation are also located in this subdirectory in bitmap format (HINT*.BMP). The documentation comprises the following:

RM8E_GStart.PDF	Getting Started (in English)
RM8E_TDescription.PDF	Technical Description (in English)
RM8E_UGuide.PDF	User Guide (in English)
RM8E_Pguide.PDF	Procedure Guide (in English)
RM8G_*.PDF	appropriate German documents.

The configuration file HOST.INI is located in the directory *ETC*. It contains basic configuration data for the GUI (language, colour settings, etc.) and a list of recently used project directories. This file is created by the program when it is started for the first time, and it is adapted during the program run, when the configuration data are changed by using the GUI function  . The original configuration may be restored by deleting this file.

1.1.2 Project Data

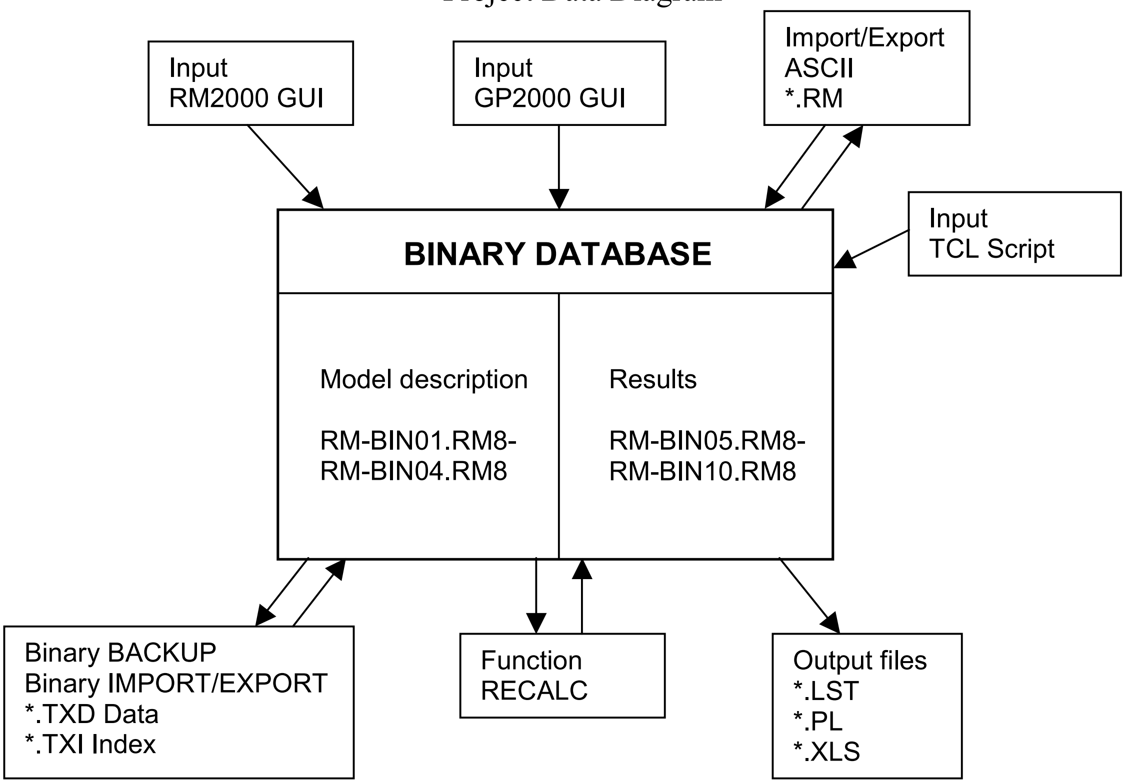
1.1.2.1 Database

The project data is stored in the Project Directory - generally as a binary database. The Project Directory is normally chosen via the start screen displayed on opening *RM2000*, it can also be chosen via ⇧FILE ⇨NEW – to create a new project or ⇧FILE ⇨OPEN to open another existing project. All the project files created without assigning a full path name to them will, by default, be saved in the currently open Project Directory.

The database consists of a set of binary files named RM-BIN01.RM8 to RM-BIN10.RM8 and a set of ASCII files for the graphic presentation named PL-*.RM. The database is unique, i.e. the file set cannot have other names and it can only contain the data for **one** project. A separate working directory must be established for each new project – even for any parallel work on different project variations.

The file set RM-BIN01.RM8 to RM-BIN04.RM8 contains all input (model and loading description) data and will be created the moment that a new project is started. These files and filled and modified during the input process. The file set RM-BIN05.RM8 to RM-BIN10.RM8 contains all the result (output) data and are created/modified when the project is re-calculated (⇧RECALC).

Project Data Diagram



GUI……….GRAPHIC USER INTERFACE

1.1.2.2 Import/Export/Backup

Import means to retrieve data from any directory and file structure including the Project directory and place this data inside the *RM2000* Program data base for the Project.
Data may be imported in any of the following 3 formats:

a) a pair of files xxx.txd and xxx.txi which are stored in binary format.
b) a set of files *.rm which are in ASCII format.
c) a set of script files *.rmd which are in the TCL format.

Binary-import:
The binary file xxx.txd can only be imported once it has been created and it can only be created using the binary export function. The import function additionally requires the appropriate index file xxx.txi for retrieving the data from xxx.txd.

ASCII-import:
It is possible to import the complete set of *.rm files describing the whole database, or to selectively import certain files containing specific data, such as the material properties, or the variable definitions. The file set to be imported may either have been created by a previously performed export procedure, or with any text editor (in the required format!).

TCL-import:
It is possible to import script files stored in the TCL-format. The imported files may either have been created by a previously performed export procedure, or created using any text editor (in the required format).

Note: *Some Standard Data files like material tables for different design codes or Variable definitions are part of the program package. These files are stored in the TCL format and are located in the Program Directory (*.RMD).*

Binary-export:
The function for binary export creates a file set xxx.txd and xxx.txi (being a condensed data set that defines the whole database (model description, loading and construction schedule part). This function is usually used for saving data for later use, or for transferring data to other directories, e.g. for the investigation of different variations etc.

ASCII-export:
The data for the whole database or only certain selected files may be written to a set of ASCII files *.rm. These files may be used for data import later. Only the input data (model description, loading and construction schedule part) of the database may be exported (no results).

TCL-export:
The data for the whole database may be written in a TCL-script file format which generally have the extension .TCL. They are in ASCII format and can be edited using any text editor. See below for detailed description. TDV recommends this type of data transfer.

Backup:
The backup function is more or less the same as the binary export, except that the name of the files to be created cannot be defined by the user. The created file set will be named backup.txd and backup.txi in the project directory.

1.1.2.3 Generating the Database with TCL scripts

A script is a simple text file **without formatting constraints** (ASCII – text file) containing a sequence of commands. TCL script files should be named with the extension '.tcl' – such as 'filename.tcl'.
A script file can be generated using any text editor - open a text editor (e.g.: by selecting the 'editor' button from the icons at the top of the *RM2000* screen), write the sequence of commands and save it as 'filename.tcl'.

The summary and the syntax of the commands to be specified and used in the TCL script files is described in detail in the chapter "Scripts" of this manual.

Note: *Script files can not only be used for generating or updating the Database, but also for specifying a sophisticated Result Action command sequences. These script files can be started interactively in ⇧RESULTS ⇨SCRIPT or automatically in ⇧RECALC by specifying them in the Action Schedule. This option is described in detail in the chapter "Results".*

1.1.3 Setup of a Standard Database

A Standard Database is created in the Program Directory when the program is started for the first time after the installation. The user can re-establish this initial condition by deleting the existing Standard Database in the Program Directory (RM-BIN*.RM8).

Subsequent to starting the program for the first time after the program installation, the user is asked to select one or more Standard TCL Data Files (*.RMD) provided by TDV. Once selected, these files will be included in the Standard Database. The existing *.RMD files are shown in the selection window. The user must highlight the files required to be included and confirms with <ok>.

The Standard Database will **NOT** be created if the selection dialogue is terminated with <cancel>. The initial conditions will remain valid and the program will again ask for the Standard TCL file selection when it is restarted.

The initial Standard Database setup function can not be used for changing, deleting or adding data in the Standard Database. If the data in the Standard Database must be changed, the user can either delete the Standard Database and make a new initial setup, or Modify, Delete, Insert, the data in the Standard Database by starting a Project in the Program Directory, modifying the data and backing-up the project – usually by "exiting the project with backup". The Standard Database will now be **permanently** changed unless the bins RM-BIN01.RM8 to RM-BIN04.RM8 (inclusive) are deleted and the defaults re-established.

Note: *The actual cursor position (per default the first line) in the selection menu is automatically identified as marked, therefore, if the selection dialogue is terminated with <ok> prior to having selected anything, the initial Standard Database will never be completely empty. The user must use general data manipulation techniques (deleting all data after opening it or opening it as "New"), if the Standard Database must be completely empty.*

1.1.4 Copying Standard Data to the Project Database

1.1.4.1 General

The function ⇧FILE ⇨DEFAULTS is used to copy standard data into the Project Database. The data source may be the Standard Database in the Program Directory or any Project Database previously set up when analysing a structure.

The data that may be copied from an external database to the Project database are:

- Materials
- Cross Sections
- Variables

The Copy function input pad is displayed on selection of ⇧FILE ⇨DEFAULTS. A choice must be made between copying Materials, Cross Sections or Variables - the appropriate table from the "Source Database" and from the Project Database will be displayed immediately following the choice selection. Any or all of the items in the Source Database table may be selected and copied to the project Directory by marking them and clicking the -> Copy -> - button.

1.1.4.2 Changing the Source Database

The default "Source Database" is the Standard Database in the Program Directory. Copying data from other projects is often used for Cross Sections, which are not normally available in the Standard Database. This may be done by assigning an arbitrary other project database as Source Database.

The Source Database can be changed by selecting the “Default Database”-button in the function ⇧FILE ⇨DEFAULTS and entering the file name and path of the new directory or by selecting the new file and directory via the “Explorer directory/file tree” that is opened when the “Pull-down menu” arrow is selected.

1.1.4.3 Data Transfer

It is not possible to transfer data of different types (e.g. Materials and Variables) at the same time i.e. if both Materials and Variables need to be copied, it is necessary to select “Materials” first, and to copy the required materials, and then to select “Variables” and to copy the required variables.

1.1.4.4 Copy Data into the Standard Database

It is also possible to add data (e.g. Cross Sections) to the Standard Database (or any other source database). This is done by using the “backward copy” button in the ⇧FILE ⇨DEFAULTS pad.

1.1.5 Demo Examples

A set of demonstration examples is generally delivered together with the program. An overview of these examples is in the demonstration example manual. It is possible to start any of these examples using ⇧FILE ⇨DEMO.

1.1.6 Hardware Requirements

The program system is designed for micro- and mini computers. The required amount of mass storage depends on the size of the structure to be analysed as well as on the number of loading cases and loading combinations considered. Typically a small problem may only require 1MB whereas a large problem could require as much as 800MB of storage capacity and more.

The required RAM capacity depends on the operating system and on the work to be done in parallel with the program. It can be generally said that 128 Mbytes will be sufficient for Windows95/98/Me installations, whereas 256 Mbytes are recommended for WindowsNT/2000/XP environments.

There are no special program requirement for the output devices - all printers and plotters which can operate under standard windows programs can be used for the presentation of results, the model and the input data.

1.2 Structure of the Project Database

1.2.1 Database principles – Objects and Attributes

The *RM2000* database is designed in accordance with the rules for an object oriented database. Data consists of **objects** and **attributes**. Objects may be named or unnamed. Named objects are referenced and sorted by a number or a name, unnamed objects are referenced by their location in the object list. Attributes are directly assigned to the objects.

Whenever an object has a number and a name, the number will be the basic reference term. The name will, in this case, only be an attribute i.e. a descriptive text.

It is possible to input, change and delete data in any order with some restrictions:

- An appropriate object has to be created before any attributes can be entered. E.g. a material has to be created, before the material parameters can be entered.
- An object cannot be referred to before it has been created. E.g. an element can not be allocated to certain nodes if the nodes have not yet been defined.
- An object cannot be deleted if it is referred to by another object. E.g. a node can not be deleted if an element has been allocated to be connected to the specified node.
- It is not possible to rename an object (the new object has to be defined – possibly by copying the attributes of the old object – and then the old object may be deleted).

Note: The program will not allow the user to attempt to carry out illegal operations.

Three types of objects may be distinguished:

a) Named objects (defined by name or number), where the name or the number is unique in the whole database
b) Named objects, where the name or number is not unique in the whole database (it is only unique in the appropriate object table)
c) Unnamed objects, created by reference

Named objects are created with their attributes in separate tables prior to being referenced from other (higher order) objects by name or number.

Unnamed objects are created by reference, this means that they are established in the database when they are referenced. They are identified internally by their location in the reference list, but they may not be referenced directly by the user.

An example for unnamed objects are the Actions. They are listed in the Action Schedule List in the sequence they are applied to the structure, but they have no name or number to be referenced.

1.2.2 Dependency Relationships

Dependency relationships exist between different objects which influence the data manipulation possibilities. These relationships may be:

a) relational

or

b) hierarchical

1.2.2.1 Relational Dependency

Dependencies are called "relational", if the objects are related to other objects in accordance with the principles of a relational database, i.e. they are stored with their attributes in separate independent lists. The relationship is established by pointers assigned to the dependent (higher order) object. E.g. the element geometry is dependent on the nodal point coordinates, therefore the element list contains pointers to the nodal point list. The element is therefore a higher order (dependent) object with respect to the nodes.

Objects are called "relational objects" when they are related to each other in that manner. The names of "relational objects" are unique in the whole database. The rules for the manipulation of such objects are:

- Deleting a higher order (dependent) element does not affect the list of lower order objects. E.g. deleting an element will cause the deletion of the information about connected nodes, but all nodes will remain unchanged in the nodal point list.
- A lower order object cannot be deleted if it is referred to by another (dependent) object. E.g. a node can not be deleted if an element has been allocated to be connected to it.
- Changes of the attributes of a lower order object will also be immediately valid for the dependent higher order objects. E.g. changing nodal coordinates will change the element geometry, loads depending on the element geometry, loading cases depending on these loads, etc.

Examples of relational objects are:

• Materials	
• Cross Sections	
• Nodes	
• Structural elements	dependent on Nodes, Mat., CS, etc.
• Load Sets	dependent on Elements or Nodes, maybe Mat., CS
• Load Cases	dependent on Load Sets

1.2.2.2 Weak Relational Dependency

There also exists a weak form of relational dependencies, where pointers on non-existing objects are allowed, i.e. the dependency is related to the attributes of the lower order objects only if these exist. A typical example of such a relationship is the dependency of loads from a series of elements or nodes. The program allows the user to allocate the elements to the loads even if they (possibly partially) do not exist. The loads applied to non-existing elements will not be considered in the analysis process, only the loads applied to existing elements will be used.

1.2.2.3 Hierarchical Dependency

Objects are called "hierarchical", if they are directly connected to the dependent object. Their names are not unique in the whole database, but only in the list related to the higher order object.

A typical example for these objects are cross section elements and nodes. The cross section element and node tables are directly related to the cross section. Separate element and node tables belong to every different cross section. e.g. the element 1 of cross section CS1 does not necessarily have anything in common with the element 1 of CS2.

The management rules for such objects are essentially different from those of the relational objects:

- Deleting a higher order (dependent) element invokes deleting the whole tree of hierarchically lower ordered objects. E.g. deleting a cross section will delete all related CS-element and CS-node tables.
- A lower order object can always be deleted, except when it is also relationally allocated to a higher order object. E.g. CS-elements can always be deleted from the CS-element table, this action directly affects the cross section geometry. CS-nodes, however, may only be deleted, when they are not referenced by an existing CS-element in the related CS-element table.
- There is no difference to relational objects with respect to attribute changes: Changes to a lower order object will also be valid for the dependent higher order objects.

1.2.2.4 Unnamed Objects

Unnamed objects are necessarily hierarchically related to the higher order (dependent) objects. I.e. they may be deleted without restrictions and they will automatically be deleted if the higher order object is deleted (e.g. all related Actions will be deleted, when a Construction Stage is deleted).

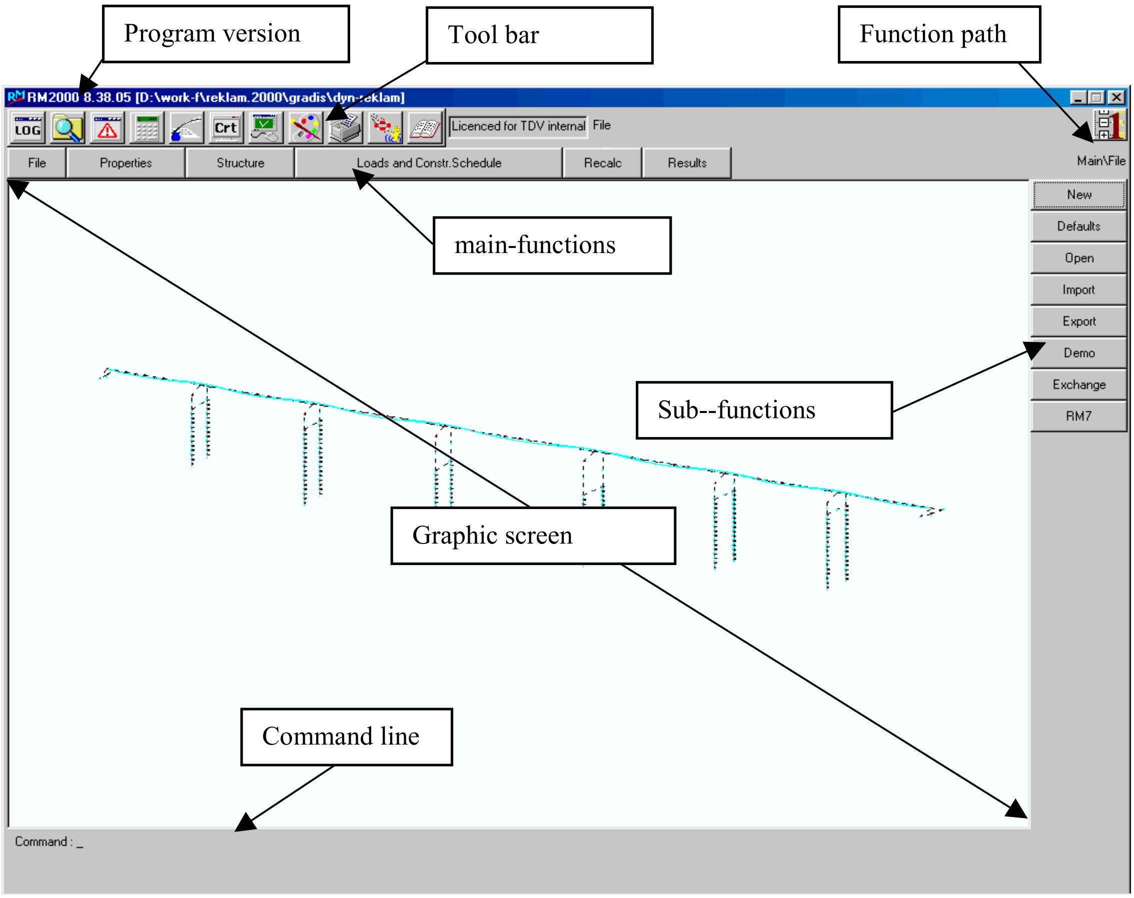
1.2.2.5 Table of Object Relationships

(R) = relational, (H) = hierarchical, (W) = weak, (U) = unnamed

Object	Dependent on			
Node	-			
Element	Nodes (R)	Material (R)	CrossSection(R)	
Material	-			
Add. Group				
CrossSection	CS-elements(H)			
CS-element	CS-nodes (R)			
CS-node				
CS-Add. Point				
Composite CS	Cross-Section			
Tendon				
Load Case	Load Sets (R)			
Load Set	Elements(W)	Nodes(W)	Material(R)	
Lane				
Load Train				
Seismic Case				
Load Info				
Envelope				
Constr. Stage	Activation(H)	Actions(H)		
Action (U)				
Activation (U)	Elements(R)			
Tendon Action				
Grp.File				
Script				
Variables	-	Other Variables		

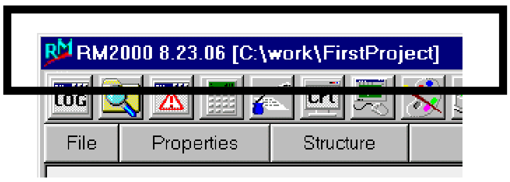
1.3 The *RM2000* Graphical User Interface (GUI)

The *RM2000* main screen, shown below, is similar in design to most Windows programs.

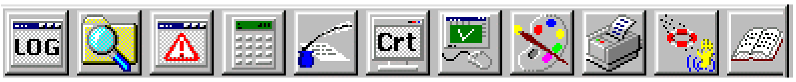


1.3.1 Description of the main user interface parts

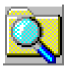
The program version number and the current project path are shown in the top left hand corner of the screen.

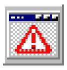


1.3.2 Tool bar



 Opens a window listing the recorded actions.

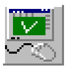
 Opens the Windows-Explorer program starting in the current project directory.

 Lists the errors from the most recent calculations.

 Opens the Windows Calculator program.

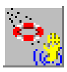
 Opens the default editor program (Textpad or Notepad)

 Opens a program for plotting graphical results.

 Lists all freehand symbols for zooming functions.

 Opens a dialogue window for program parameters.

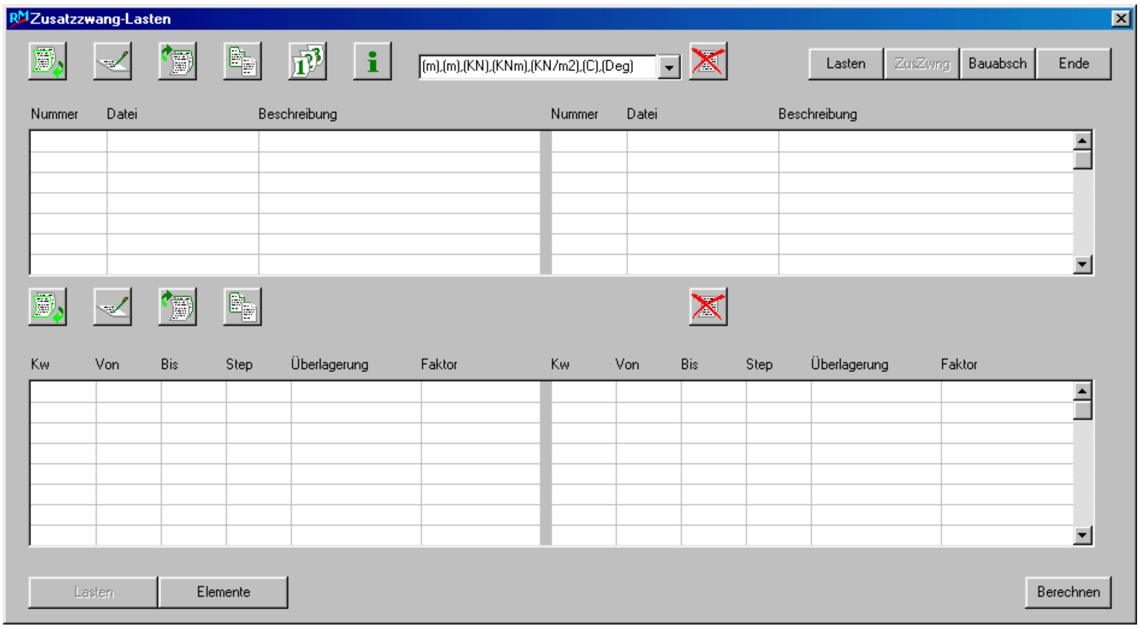
 Prints plot files and other result listings.

 Opens the *RM2000* help files.

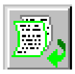
 Opens the *RM2000* online books.

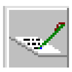
1.3.3 Tables of Database Objects and Parameters

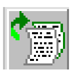
Data are entered in RM2000 by editing object and parameter tables in the GUI. The windows related to the different input functions mostly show an upper object table (for the type of objects to be defined), and a parameter table presenting the parameters related to the selected object below.



Used Icons:


"Insert before" Insert line before the selected object or parameter line.


"Modify" Modify the selected object or parameter line.


"Insert after" Insert a line after the selected object or parameter line.


Copy the selected object or parameter line to the end of the list.


Sort and renumber the entries of the table.


Delete the selected object or parameter line.

1.4 Program Functions

1.4.1 Main functions

The Main function list remains the same at every stage of the program. The sub-function lists on the right side of the screen change with the main function selection.

⇧FILE	Project management (open, create, ...) and import/export.
⇧PROPERTIES	Definition of material properties, cross section properties and variables.
⇧STRUCTURE	Definition of the structural system (nodes, elements, tendon geometry).
⇧LOADS AND CONSTR.SCHEDULE	Definition of loading and construction stages.
⇧RECALC	Definition of calculation parameters and start of the calculation.
⇧RESULTS	Viewing of results and creating of output files (plots and listings).

Note: *The 'up-arrow' symbol ('⇧') will be used in this document to identify a main function, e.g.: ⇧STRUCTURE.*

1.4.2 Sub-functions

On selection of ⇧FILE, the following sub-functions list will be displayed on the right hand side of the screen.

Note: *The 'right-arrow' symbol ('⇨') will be used in this document to identify a sub-function, i.e.: ⇨IMPORT.*

⇨NEW	Start a new project (with empty database).
⇨DEFAULTS	Setup and import template data
⇨OPEN	Open an existing project or start a new one.
⇨IMPORT	Import a saved project (or a part of it).
⇨EXPORT	Export (save) the current project (or a part of it).
⇨DEMO	Select an *RM2000* demo example to be loaded.

⇨EXCHANGE	Change the project information into the desired format.
⇨RM7	Import the RM7 steel cross section table for *RM2000*.
⇨OPTIMIZE	Several options to accelerate the calculation

On selection of ⇧PROPERTIES, the following sub-functions list will be displayed on the right hand side of the screen.

⇨MATERIAL	Modification of materials and material properties.
⇨ADDGRP	Modification of reinforcement/stress groups.
⇨CS	Modification of cross-sections and cross section properties.
⇨VARIABLE	Modification of variables.
⇨AERO CL	Modification of the Aero classes

On selection of ⇧STRUCTURE, the following sub-functions list will be displayed on the right hand side of the screen.

⇨NODE	Definition of nodes and their attributes.
⇨ELEMENT	Definition of elements and their attributes.
⇨TENDON	Definition of tendons and their attributes.
⇨SPECIAL	Comparison of elements and nodes, subdivision of beam or cable elements

On selection of ⇧LOADS AND CONSTR.SCHEDULE, the following sub-functions list will be displayed on the right hand side of the screen.

⇨LOADS	Definition of load cases.
⇨ADDCON	Definition of Additional Constraints (see chap. 6.11)
⇨STAGE	Definition of constructions stages.

On selection of ⇧RECALC, a dialogue box is opened. Several computation options can be selected and general parameters can be set in this pad. On selection of the only sub-function ⇩RECALC, the calculation will be started.

On selection of ⇧RESULTS, the following sub-functions list will be displayed on the right hand side of the screen.

⇨LCASE	Load case results in list form for nodes and elements.

⇨ENVELOPE	Envelope results in list form for nodes and elements.
⇨PLSYS	File editor for the creation of plot-files.
⇨PLCRSH	Graphical presentation of creep and shrinkage coefficients.
⇨PLINFL	Screen Plot of influence lines for all degrees of freedom.
⇨REPORT	Results report for selected elements/nodes and loading cases/envelopes.
⇨SCRIPT	Start a script.

1.5 The *RM2000* Help System

On-line help texts describing what data is to be input and where to input it are available at all times in *RM2000*.

Note:
The Help-Pad with the text appropriate to the actual input pad will be opened the moment that the **Special Function Key F1 on the keyboard** is selected.
The help text can also be opened by clicking the mouse on the help icon of the tool bar – this, however, only works with the help texts for the function menu’s.

The help text generally provides the following information:

- short general description of the current input pad or the current function
- description of the sub-functions to be selected
- description of the variables to be input
- information about default settings
- special hints where necessary
- information about the required next steps after closing the current pad

The INDEX-button on the help pad toolbar gives access to an index of all the available help subjects. Any subject can be selected and shown in the help pad without closing the current input pad.

All manuals and guide documents are available online in addition to the help text.

1.6 Variables as Formulas or Tables

Variables can be defined for any part of the structural analysis and design code checks. These variables can be defined in the form of a formula or as a table.
The program will automatically retrieve the variable information from the data bank when the variable name is referred to as the data information.

Typical Items that are stored under 'Variable' include:

Material Characteristic variations

- Creep factor variation of the material with time
- Shrinkage factor variation of the material with time
- E-modulus variation of the material with time
- Non-linear material behaviour under load

Load variations

- Live load intensity variation with loaded length
- Load spectrum related to time
- Response spectrum for earthquake analysis

Variables can be directly input into the database by using the function ⇧PROPERTIES ⇨VARIABLES. Chapter 3.5 provides full information on the use and application of variables.

Variables can be imported into the program from variable tables that are either part of the installed program package or were prepared as standard tables by the user. The importing is done via ⇧FILE ⇨IMPORT. A list of standard variable tables that are a part of the program package is given in chap. 1.1.1.

Note: *If a variable is imported into the database and it has the same name as an existing variable then the original variable will be overwritten- irrespective of whether the original variable is of completely different form to the new variable (i.e. a table as opposed to a single item or formula) – This is true for all imports within their own type – i.e. for materials and cross sections as well – but a material with a certain name will not be overwritten by a variable or cross section with the same name.*

1.7 Other Help Functions

1.7.1 Macros

Macros are program functions simplifying otherwise complicated input procedures. They generate extensive sets of input data from a few parameters. The input parameters for the macros are not stored in the database - only the generated data is stored. These input parameters may therefore not be subsequently changed by the user. The only way to change this data is to delete the generated data in the database and then to re-generate it.

A typical example of the application of macros is the generation of the Finite Element mesh for the computation of the cross-section properties of cross-sections with standard shapes. The macro will, in this case, generate the whole mesh for a series of cross-sections by entering a few geometric parameters such as depth and width. The generated nodes and elements for each cross-section are stored in the database and are subsequently used in the analysis for the computation of the cross-section properties and for the design checks.

1.7.2 Scripts

Scripts are command sequences. The script language is based on TCL version 7.3. All TCL commands, i.e. loops, conditions, etc. can be used in scripts.

Two different command groups for interfacing RM with the script are provided:

- Input commands
- Result analysis commands

Input commands:
The user can simplify the input of multiple tasks that are similar with the help of input commands. An export command is also available, producing a script file from the RM database. Such exported script files may be modified by using any text editor program, this yields the possibility of easily performing analyses of slightly changed systems or variant investigations without interactive manipulations in the *RM2000* GUI.

Result analysis commands:
The user can produce individual list files or general output file containing data from input database (geometry, …) and result data (forces, stresses, …) with the help of these commands.

A description of these commands is provided in chapter "Scripts"

Interface commands:
The user can implement individual dialogues interfacing with the RM database using input and/or result command with the help of interface commands.

2 General Properties

2.1 General

An essential task for the design engineer is to create a mathematical model of the structure such that the model behaviour simulates the behaviour of the actual structure under various different loading conditions with sufficient accuracy.

The modelling process consists of

- The choice of the basic parameters (e.g. the unit system to be used)
- The approximation of the physical properties of the structure within this basic mathematical system.

The approximation procedure may be sub-divided into 4 categories:

- Modelling the geometric properties
- Modelling the resistance behaviour
- Modelling the impacts on the structure
- Modelling the time domain

These 4 modelling categories, related to the input process for *RM2000*, are described in the next 4 chapters of this user guide:

a) Structural properties (definition of the resistance parameters such as the material behaviour and the cross-section definitions) (Chapter 3)
b) Structure (definition of the geometry of the model and the interaction conditions of the different parts of the model) (Chapter 4)
c) Loading (definition of the impacts on the structure such as external loads, temperature effects, etc.) (Chapter 6)
d) Construction Schedule (definition of the time dependent behaviour of the model) (Chapter 7).

2.2 Analysing a Structure

A brief description of the required procedure for analysing a typical structure is given below.

In order to successfully analyse a structural system of any kind and review the results, the following must be defined:

- The Structural Model
- The cross-section of the various elements in the structure and the materials making up the elements
- The material properties
- The individual loading to the structural model and the loading combinations.
- The time of application of the loading and the time of any structural modification.
- The type of output for the results.

The data preparation for a structural system using *RM2000* is grouped under 5 main headings:

- Properties
- Structure
- Loads and Construction Schedule
- Recalc
- Results

A logical sequence for defining the structure, the loading and the results is listed below in a concise format:
It should be noted that the sequences given below are not the only way that the structure and loading etc. can be defined. The prepared sequence is just a suggestion. The file structure showing where the interactive input Pads for the input data preparation can be found is also given.

Define the structure

Step 1)	Define (import) the material properties	**Properties** → **Material** or **File** → **Import**
Step 2)	Define the required cross section properties	**Properties** → **CS** or **File** → **Import**
Step 3)	Define the structural nodes and their attributes	**Structure** → **Node** or **File** → **Import**
Step 4)	Define the structural Elements (BEAM, SPRING, CABLE, ...), user defined ECC, hinges, beta angle etc ...	**Structure** → **Element** or **File** → **Import**

Step 5)	Assign material properties and cross sections to the elements;	**Structure** → **Element** → **Mat CS** **or** **File** → **Import**
Step 6)	Define additional element attributes if required, (e.g. reinforcement, creation time, ….)	**Structure** → **Element** → **…..** **or** **File** → **Import**
Step 7)	Define PRE-STRESSING CABLE geometry and assign properties to the tendons	**Structure** → **Tendon** → **Geometry** **or** **File** → **Import**

Define the loading

Step 1)	Split the applied loads into logical sets of loads.	**LOADS AND CONSTR. SCHEDULE** → **Loads** → **LSet**
Step 2)	Combine any number of Load Sets to compose the Loading Cases including the definition of load factors.	**LOADS AND CONSTR. SCHEDULE** → **Loads** → **LCase**
Step 3)	Establish the load management system (rules for combining the load cases during the stages of the construction schedule)	**LOADS AND CONSTR. SCHEDULE** → **Loads** → **LManage**

Define the construction schedule

Create all the necessary construction stage activations, actions and durations.

N.B.: The only time that the structure can be changed (modified) is at the beginning of the construction stage (i.e. add a new element or a new cross section unit).

Step 1)	Define elements to be activated/deactivated in the construction stages	**LOADS AND CONSTR. SC HEDULE**	→	**Stage**	→	**Activation**
Step 2)	Define the actions which take place during each stage (Loading Cases, Pre-stressing, Creep & Shrinkage, earthquakes, …)	**LOADS AND CONSTR.SC HEDULE**	→	**Stage**	→	**Action**
Step 4)	Define the actions that take place to the pre-stressing tendons during each construction stage (stress, wedge slip, re-stress etc.)	**LOADS AND CONSTR. SC HEDULE**	→	**Stage**	→	**Tendon**

Recalc

Use ⇧RECALC (re-calculate) to analyse the structure once all the input data is complete.

⇧RECALC can be used at any time during the input preparation as a check on the status of the structural input – all the data does not need to be complete before using it!
Actions for which the required data is not yet complete will not be calculated and a corresponding message will be given.

2.3 Units

2.3.1 General

The data input can be defined in any desired unit-system combination.
The output can also be viewed and printed in any desired unit-system combination.

The unit system internally used in *RM2000* (for the calculation process and data storage in the binary database) is a modified SI system (SI = Système International d'Unités) with:

- **[m] (metres) for the length**
- **[kN] (kilo-Newton) for forces**
- **[°C] (degrees centigrade) for the temperature**
- **[s] (seconds) for the time**
- **and directly derived (consistent) other units.**

All input values entered into the program in special units are immediately transformed internally into the standard system, all output values are transformed back to the output units just before the output action, but internally all values in the database will always remain in the standard system.

Although in principal the user is free to work in an arbitrary unit system, or with different units in different stages, it is recommended that the standard units [kN], [m], [s], [°C], were used, or at least another consistent unit system specified at the beginning, and remaining the same over the whole analysis process.

The main reason for using a consistent unit system is to ensure a clear understanding of the results. Where non-consistent units are used, the user must always to be aware that the derived units may be strange quantities and he must always take this into account when interpreting the results.

Typical consistent input/output units would be:

Force in kN,	Length in m	Moments in kNm	Stress in kN/m^2 (kPa)
Force in MN	Length in m	Moments in MNm	Stress in MN/m^2 (MPa)
Force in kips	Length in feet	Moments in kipft	Stress in Kips/ft^2 (ksf)
Force in kips	Length in inches	Moments in kipins	Stress in Kips/in^2 (ksi)

Another reason for using the standard units is, that the format of output listings is designed to suit the magnitude of the result values arising in the calculation of typical civil engineering structures. The use of strange units may lead to listings, where the results cannot be properly identified due to too few or too many digits being presented. Only in a few cases will the result values be such that they may be bad for presentation purposes when using the standard units.

RM2000 has a special feature for overcoming these presentation problems.

Apart from the option of changing the units, the user can define output factors for the result presentation to get more readable numbers in the tables. The multiplication factor used is displayed in the table header to avoid confusion. A typical example is the. displacements that are multiplied by 1000 (default output factor for deformations) and printed in mm and 1/1000-rad when metres and radians are used in the analysis. The default value for the force multiplication value is 1, but may be set by the user to any other value.

Note: *The will also be applied to all values directly related to forces, such as moments and stresses in the **result** listings. N.B. The force multiplication factor is not applied to input – only to results.*

2.3.2 Viewing, setting and changing active units

The current units for input and output can be viewed and optionally edited in the ⇧RECALC dialogue screen which is opened on selection of ⇧RECALC. Any or all of these units can be changed by choosing the desired units from the displayed pad following selection of the pull-down menu arrow to the right of the Unit window.

Some units can be arbitrarily specified by the user by specifying a unit name and the factor relating this new unit to the appropriate default unit.
These arbitrary user-defined units can be applied to the length and force units.

A concise list of the active units is also displayed in most input pads. The units can be changed via the pull-down menu arrow to the right of this concise Unit window list – as described above – instead of using the ⇧RECALC dialogue screen.

The following units can be changed:

Quantity	Internal	Default I/O
• Length (structure)	[m]	[m]
• Length (Cross Section)	[m]	[m]
• Force	[kN]	[kN]
• Moment	[kNm]	[kNm]
• Stress	[kN/m^2]	[kN/m^2]
• Temperature	[°C]	[°C]
• Angle (general)	[rad]	[deg]

Note: *The more common term “ton” is used as a force unit instead of the unit “Megapond”. It characterises the weight of a mass of 1 ton in the standard earth gravity field.*

The following units are prescribed and may not be changed by the user:

- Time (general) [s] (seconds)
- Time (construction schedule - creep analysis) [d] (days)
- Angle for rotations and angular velocities [rad] (radians)

All other units are consistent to the specified basic units, and may not be directly changed by the user.
E.g.

- Surface load kN/m^2 if [m] is the unit for length(structure)
- Specific weight kN/m^3 if [m] is the unit for length(structure)
- Cross section area cm^2 if [cm] is the unit for length(CS)
- Wobble factor Deg/m if [deg] is the angle and [m] the length unit
- Velocities m/s if [m] is the unit for length(structure)
- Accelerations m/s^2 if [m] is the unit for length(structure)

Special dependencies:

Length:
Length(CS) only influences the following quantities:

- Cross-section lengths used as input values for the cross section definition macros, such as width, height, thickness of cross-section components
- Coordinates of the nodes of the cross-section elements
- Computed Cross sectional areas and moments of inertia
- Tendon areas
- Duct areas

All other quantities related to length are related to the unit specified in "Length(structure)", except the quantities directly defined by the user (moments, stresses).

Note: *This is especially applicable to eccentricities of the cross-section centroid with respect to the system line and surface loads related to the cross-section height or width.*

Material parameters:

- Young's modulus kN/cm^2 if this is the specified stress unit
- Shear modulus equivalent
- Thermal expansion coeff. 1/°C if [°C] is the temperature unit

Note: *The Young's modules are defined in the specified stress unit and are not derived directly from the active length(structure) and force units, such as the unit for the surface loads.*

2.3.3 Results Multiplication Factors

The factor for modifying the result output may only be changed in the ⇧RECALC dialogue pad. The active factor is shown at the top of the ⇧RESULTS ⇨LCASE or ⇧RESULTS ⇨ENVELOPE pads respectively, and is written into the header of the output listings.

2.3.4 Exceptions – Internal Variables with Prescribed Units

Some constants and some variables are specified in the program in default units and can not be subjected to transformations during the input and output processes These constants and variables are:

- The gravity constant constant 9.81 [m/s^2]
- Angular velocities Omega variable [rad/s]
- Node rotations variable [rad]

2.3.5 Percentage Values

Certain values (particularly code related ones) are partial values related to a total amount or to a limit value – they are, for instance, given in percent or per mille in design codes or literature.
Unless specifically noted otherwise, these partial values must be entered in *RM2000* as "absolute values.
E.g. if the damping constant is 5% of the critical damping, the value 0.05 must be entered.

As implied above, where percent [%] or per mille [%o] is required as input, the required unit is explicitly mentioned in the input dialogue and in the input description.
E.g. the relative humidity RH on the construction site, used in some creep laws for the creep and shrinkage coefficients, is entered in [%] and strain values for the stress-strain diagrams defined in the material definition function are entered in per mille [%o].

2.4 Coordinate Systems

2.4.1 General

Every structural model is located within a Global Coordinate system. The position of every part of the structure as well as the directions of loads, displacements, internal forces and stresses are referenced to the chosen coordinate system.
All coordinates in the model are defined with respect to a single, global X-Y-Z coordinate system. Each part of the model (joint, element, or constraint) has its own local coordinate system and all these local coordinate systems are three-dimensional rectangular (Cartesian) systems.

2.4.2 Global Coordinate System

The **global coordinate system** is a three-dimensional rectangular coordinate system. The three axes denoted X_G, Y_G, and Z_G or simply X, Y, and Z, are mutually perpendicular. The location and orientation of the global system are in principle arbitrary.

There is, however, a considerable practical advantage in having a coordinate system with **the global Y direction being oriented opposite to the direction of gravity,** because all default rules for building local coordinate systems are based on this assumption (**XZ-plane = horizontal projection plane -** see chap. 2.4.3). A considerable amount of effort would be required by the user in re-defining the beta angles and the local axes for all the elements in order to correctly specifying the principal inertia planes, if the global Y axis was oriented in an arbitrary other direction.

The coordinate system with the axes X_G, Y_G, and Z_G, as well as a beam element in a general position, along with its associated default local coordinate system (axes x_L, y_L=y', z_L=z') is shown in Figure 2.1.

The global coordinate system as shown in figure 2.1 below is a **left-hand system** (X_G sideways to the right, Y_G upwards, Z_G into the paper). This default setting may actually not be changed by the user. There is however the plan to offer in future the option for working in a right-hand system by setting the appropriate switch in ⇧RECALC. The X_G- and Y_G-directions respectively will then remain unchanged and the Z_G-direction for the **right hand system** will be in the opposite direction to that shown in figure 2.1. This convention will also be valid for internally created local coordinate systems (such as the one shown in figure 2.1 below).

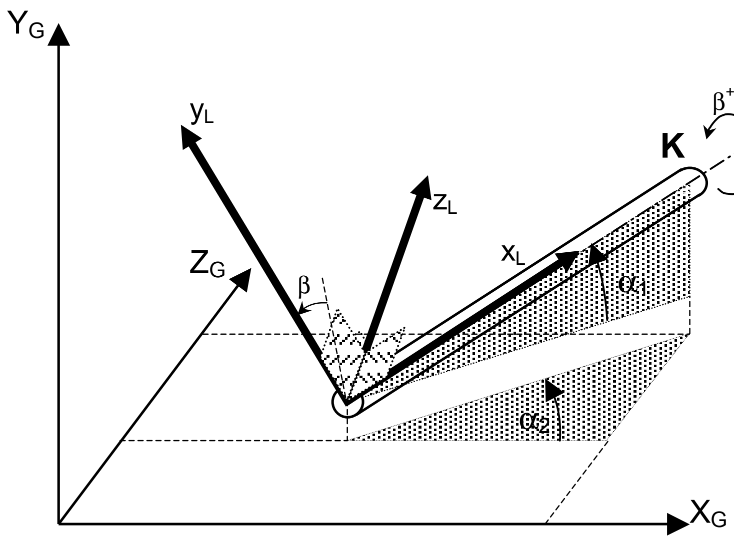


Fig.:2.1 Global coordinate system (left-handed) and default local system

2.4.3 Local Coordinate System for Beam Elements

The coordinates of the nodal points at the element begin I and the element end K and the orientation from I to K define the **local x coordinate direction x_L**.

The angle **α_2 (plan angle)** is defined as the angle between the global X-axis and the normal projection of the element in the XZ-plane (horizontal projection plane), and the angle **α_1 (elevation angle)** is measured in the **"upright projection plane" x_L-Y_G** and is defined as the angle between the XZ-plane and the element axis **x_L**.

α_1 is positive if the local x axis x_L has a positive Y_G-component, α_2 is positive from the X_G axis to the horizontal projection of the local x axis x_L).

The default orientation of the principal axes **y_L and z_L** of the element is defined in the program in accordance with the following rules **(default rules x_L and x_L):**

- The **default local y axis y_L = y'** is perpendicular to the local x axis in the plane built by the **local** x axis and a vector in **global** Y direction **(upright projection plane)**. The direction vector has per definition always a positive Y_G-component, resulting from the definition range of **α_1** being from **–90° to +90°.**
- The **local z axis z_L = z'** is normal to the upright projection plane and defined by the cross product $z_L = y_L \times x_L$ (for a left-hand system) or $z_L = x_L \times y_L$ (for a right-hand system) respectively. The angle between the global Z axis and the axis z' is

equal to the **plan angle α_2** (angle between the global axis X_G and the upright projection plane. It's definition range is 0° to 360°.

This initial local system x_L, y', z' may then be changed by defining a rotation angle β around the local x axis, resulting in the final local system x_L, y_L, z_L.

ß describes the angle of twist that the member has and is defined as the angle between the two planes defined by the local x and y axes on the one hand (1^{st} (main) principal inertia plane) and the local x and global Y_G-axis on the other hand (upright projection plane). If the angle is zero then it needs not be defined.

ß is positive if left-hand rotating (clockwise) around the x_L axis!
Figure 2.2 (drawn for the special case where the direction of x local and X global are the same) shows the general sign convention for the angle ß.

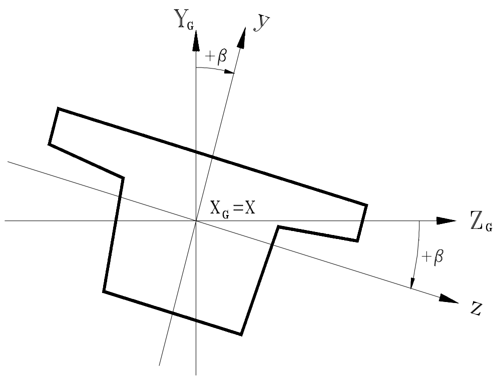


Fig.:2.2 Sign convention for the angle ß for x=X_G (looking against the x direction)

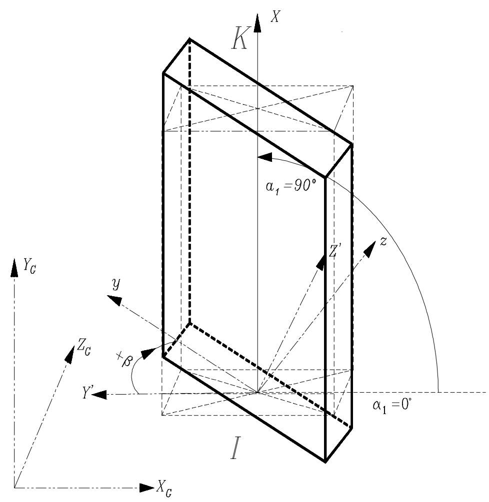


Fig.:2.3 Definition of the local system for x = Y_G

Figure 2.3 shows the convention for defining the local coordinate system in the special case where the element is vertical (x_L=Y_G). The upright projection plane built by x_L and Y_G is then undefined. The global X-Y plane is then taken, i.e. the angle α_2 is set to zero, the angle α_1 is set to 90° or –90° respectively. If ß is zero, then the principal inertia planes will be defined by the global axes X_G and Y_G; or by Y_G and Z_G respectively.

Note: *These sign and direction conventions are also valid for 1-dimensional elements such as spring elements etc.*

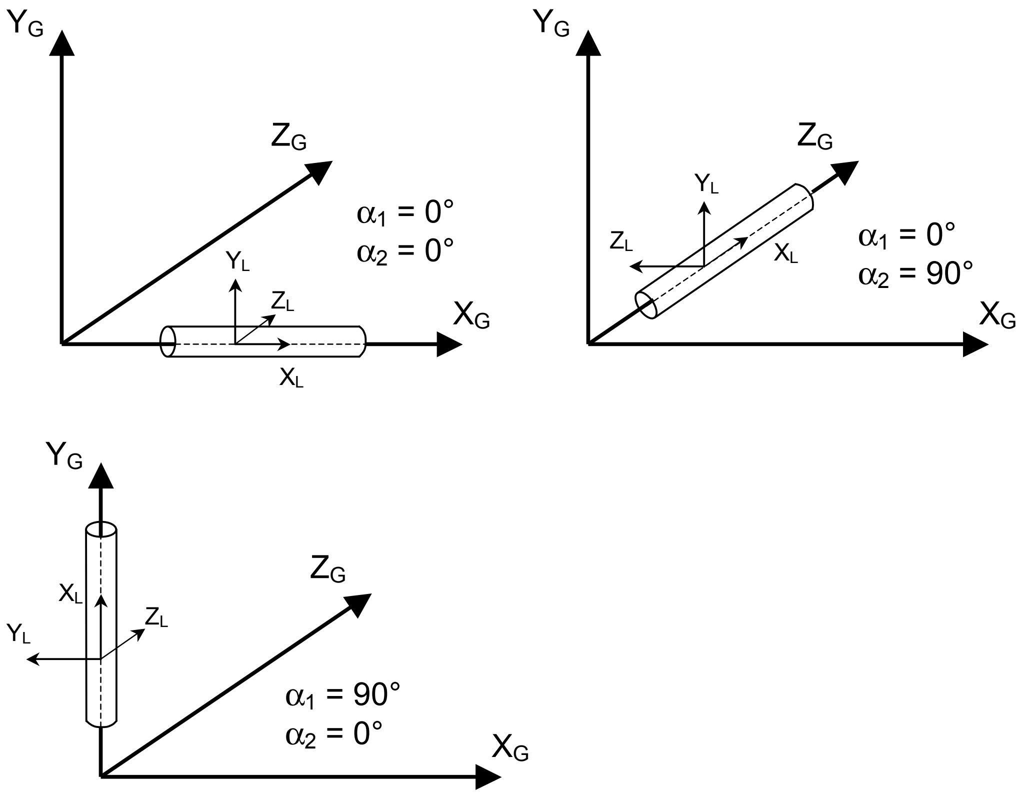


Fig. 2.4: Default local system for the special cases of x-local being in global axes directions

2.4.4 Sign Conventions for Deformations and Internal Forces

2.4.4.1 Deformations

Displacements are positive in the positive axis directions.
Rotations are positive if **right-hand** (clockwise) turning around the axes.

These conventions are the same than those for external forces and moments. They are shown in figure 2.5 and 2.6.

Note: *Attention must be drawn to the fact, that right-hand (clockwise) turning node rotations are positive, although the global system is left-handed and system angles like the β angle defining the rotation of the principal inertia plane are left-hand turning (anti-clockwise).*

2.4.4.2 Internal forces (normal forces, shear forces, bending moments)

The internal forces and moments are related to the local element coordinate system. The sign conventions are graphically presented in figure 2.4.

The sign conventions correspond basically to those generally used in the theory of elasticity (based on a right-handed system!). These conventions define tensile stresses being positive and compression stresses being negative. Shear stresses are positive, if the positive element edge (element end) is moved into the positive transverse direction.

The conventions valid in RM2000 correspond in principal to these basic definitions. Moments are positive in this context if a so-called “tension fibre” in the cross-section is tensioned. This “tension fibre” is in the general approach defined to be at cross-section edge being at the “positive” side with respect to the local coordinate system.

The only 2 exceptions in RM2000 from these general rules are

a) **the definition of the “tension fibre” for the moment M_z.** Whereas the tension fibre for the transverse moments M_y is on the positive z-side – in accordance with the general approach – the tension fibre for the main bearing moment M_z is in RM2000 assumed to be at the negative y-side. The reason is the general civil engineering statics convention, where a moment causing tension at the bottom side is defined to be positive, and a moment causing tension on the top side is negative.

b) **the sign definition of the torsion moment.** Due to the used left-hand system a fully consistent definition based on the sign of shear stresses is not possible for the torsion moment. A torsion moment creates negative shear stresses in z_L direction and positive shear stresses in y_L direction or vice versa. The convention to be established can therefore consider the shear stress rule only either for the y_L or for the z_L direction. The sign of the torsion moment is chosen in RM2000 such, that at the element end a positive moment creates at the positive z-edge of the cross-section positive shear stresses in y direction.

The following table shows the sign conventions for the different internal force components:

	Element begin	Element end
+N (normal force)	$-x_L$	$+x_L$
+Qy (shear force in y-dir.)	$-y_L$	$+y_L$
+Qz (shear force in z-dir.)	$-z_L$	$+z_L$
$+M_T$ (Mx)	left-handed (anti-clockwise)	right-handed (clockwise)
+My	right-handed (clockwise)	left-handed (anti-clockwise)
+Mz	right-handed (clockwise)	left-handed (anti-clockwise)

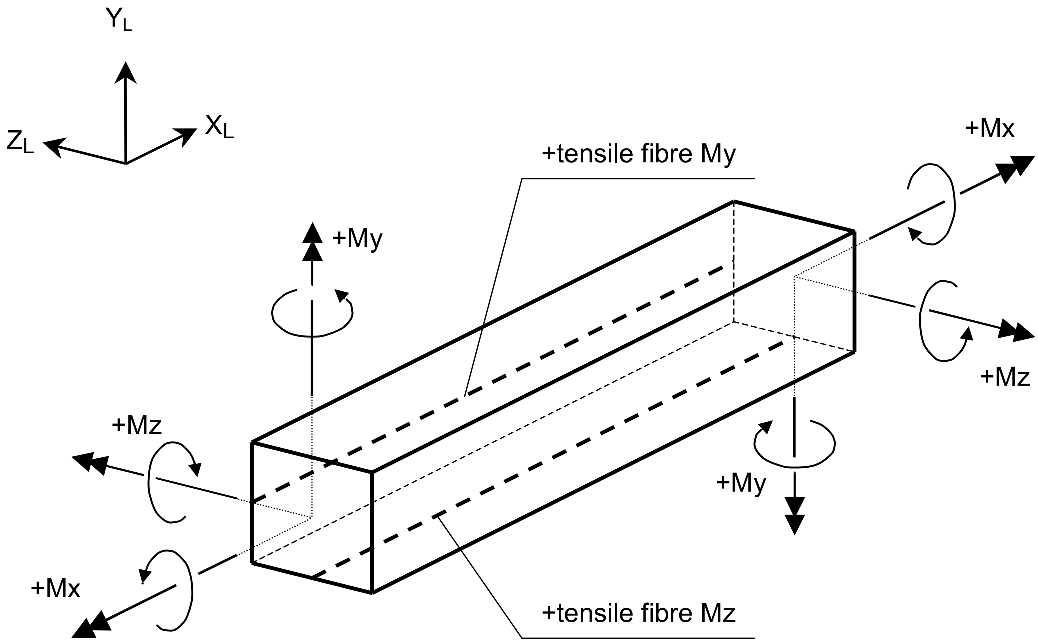


Fig. 2.5: Sign conventions for the internal moments (left: element beginning - right: element end)

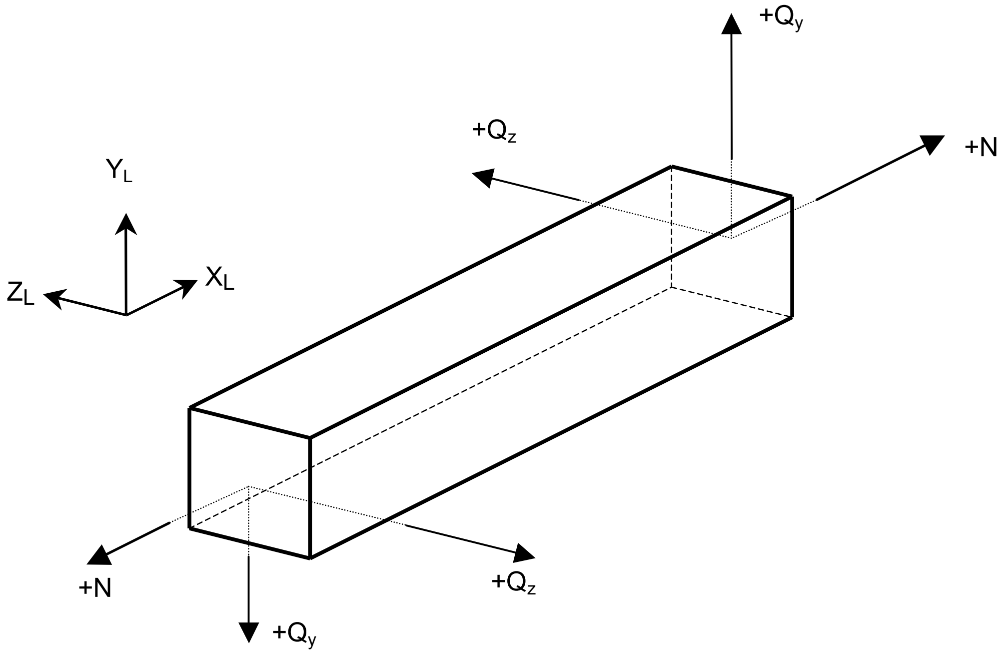


Fig. 2.6: Sign conventions for the internal forces (left: element beginning - right: element end)

2.4.5 Sign Conventions for External Nodal Forces and Moments

The sign convention for the nodal forces is shown in Fig 2.7. for the left-hand coordinate system (default system).

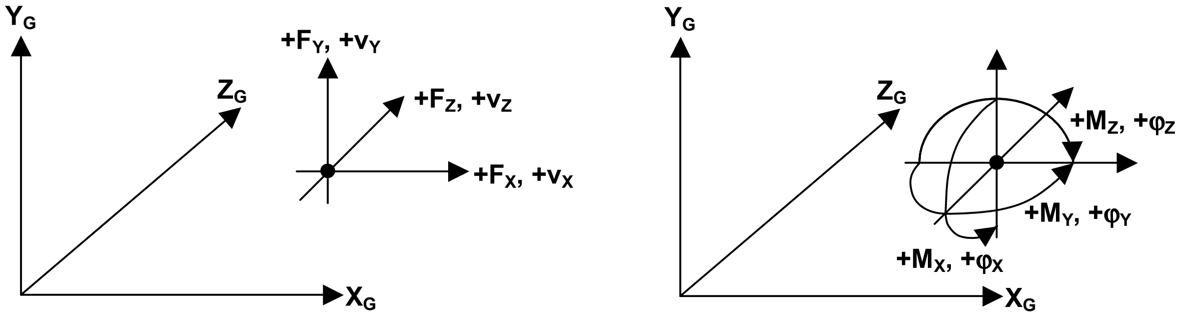


Fig. 2.7: Sign convention for nodal forces and deformations

Note:	Positive forces act in the direction of the global axes and the positive moments act in a clockwise (turning towards the right) direction about the respective global axes – clockwise in terms of looking from the origin (0,0,0) along the axis - (E.g. M_z is positive if acting clockwise about the Z-axis). The same sign convention is valid for the deformations. Note that the convention for moments and rotations is independent of the coordinate system (left-hand or right hand) and is always "clockwise positive".

2.4.6 Sign Conventions for Local External Element Forces and Moments

The sign convention for the **local** forces on a single element is shown in Fig 2.7. for the left-hand coordinate system (default system).

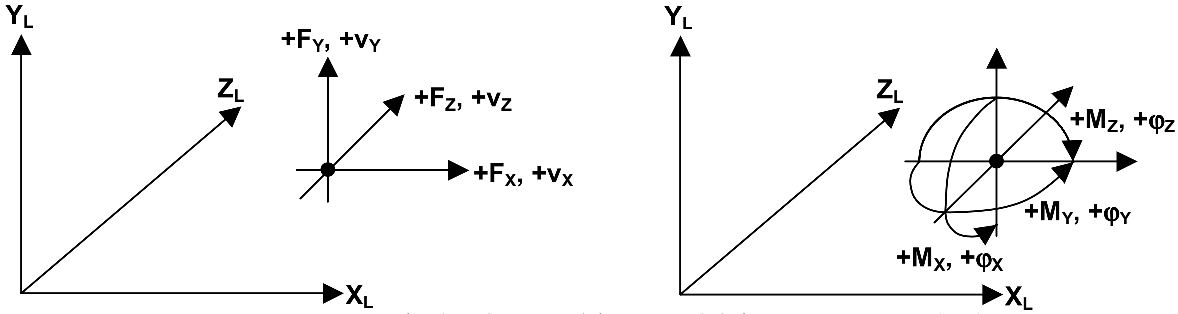


Fig. 2.7: Sign convention for local external forces and deformations on single elements

Note:	Positive forces act in the direction of the local axes and the positive moments act in a clockwise (turning towards the right) direction about the respective local axes – clockwise in terms of looking from the origin (0,0,0) along the axis - (E.g. M_Z is positive if acting clockwise about the Z-axis). The same sign convention is valid for the deformations. Note that the convention for moments and rotations is independent of the coordinate system (left-hand or right hand) and is always “clockwise positive”.

2.5 Transformations

Transformations are a general means for allowing easier input data generation or for generating “special result values”.

Transformations of the structural model include processes such as using local user defined coordinate systems during the input process, using polar or spherical coordinate systems, translation, mirroring or rotation with or without copying of model parts, etc. *RM2000* does not contain any of these special generation facilities at the moment and the user is advised to use the geometric pre-processor *GP2000*, which offers a great variety of facilities for complex geometry, when analysing a complicated structure.

2.6 Design Codes

2.6.1 General

The static and dynamic analysis procedures used in the program are generally standard and are independent of special design code rules. There are however 3 topics which are affected by the design codes and require different treatment in different countries:

- The Material Parameters (Stiffness, Strength, Stress limits)
- The Loads to be applied on the structure
- The Formulae used for the Design Code Checks

The following Design Codes can be selected in the function ⇧RECALC to be considered in the analysis and design:

• OENORM(B4200)	former Austrian code
• OENORM(B4700) (Eurocode EC2)	new Austrian code
• DIN 1045	German code (concrete)
• DIN(18800,EC3)	German code (steel constructions)
• Portuguese Code	
• Norwegian Norm-NS	
• Japanese Norm-JIS	
• BS5400	British Standard
• AASHTO	American Standard

Note: **The design code chosen/specified in the function ⇧RECALC is only used for selecting the appropriate formulae in the design code checks. The material parameters and load assumptions are not directly affected and must be specified by the user in accordance with the required Design Code.**

2.6.2 Design Code dependent Material Properties

Standard material tables for the above Design Codes are provided as part of the program and may be imported into the standard database and also into the actual project database. (Refer: chapters 1.1 and 3.1).

2.6.3 Design Code dependent Loading Assumptions

The program does **not** automatically select the correct materials or Loading Cases or Envelopes/combinations when carrying out the Design Code checks. The selection of a specific Design Code in ⇧RECALC only defines the formulas and limits to be used – it is incumbent on the user to ensure that the correct materials, loading cases, envelopes etc for the design code checks are defined and/or produced in the calculation process.

All the functions required to combine and factorise the results to suit the design code checks are provided in the program.

2.6.4 Design Code Checks

The formulae for any Design Code Check such as 'Ultimate Shear Design', 'Fibre Stress Check', 'Ultimate Load Carrying Capacity Check', 'Principal Stress Check', etc. are used in the program in accordance with the selected Design Code.

2.7 General Program Options

2.7.1 Optimising the Calculation Performance

On selection of ⇧FILE ⇨OPTIMIZE it is allowed to set some options to accelerate the further calculation.

The following options can be set active:

ELEMENTS	NODES	LOAD CASE
HG Open/Close	C+S variable direct	TOTAL SPACE (MB)

If "ELEMENTS", "NODES" and "LOAD CASE" are set active, the program keeps all the information of elements, nodes and load cases in the main memory.

If "HG Open/Close" is set active, the program opens the internal database only at the begin and end of each construction stage, and not at the begin and end of each action.

If "C+S variable direct" is set active, the program takes the variables for creep and shrinkage from the internal database. The variables defined under ⇧PROPERTIES ⇨VARIABLE are not used in this case.

A definition for "TOTAL SPACE (MB)" reserves memory space for the further calculation. Don't allocate the maximum allowable memory space (50-100 MB should remain free for the operating system).

3 Structural Properties

3.1 Standard Data

The term „Structural Properties“ covers all necessary parameters for describing the structural behaviour due to different impacts, besides the discretized geometry.
The most important structural properties are

- Material data
- Cross section data
- Variables

These data may be imported from the Standard Database or other existing Databases as described in chap. 1.

These data and all other structural properties described below may also be imported from export or backup files. Or they are directly defined in the *RM2000* GUI.

3.2 Material

3.2.1 Material Properties

The material properties for any structural component can be specified by reference to a previously defined material, identified by its name or number and stored with the attached parameters in the material list of the database. The material list may be generated by importing data from existing material data files or directly in the GUI in the function ⇧PROPERTIES ⇨MATERIAL.

An individual assignment for the basic elasticity parameters E-Modl, G-Mod, ALFA-T and Gamma (explanation see below) can also be done directly in the system definition function ⇧STRUCTURE ⇨ELEMENT ⇩MAT without using the material table. This function is restricted to standard static analyses without design code checks, prestressing, creep & shrinkage etc.. These extended functions require parameters which can only be defined via a named material in the material list.

The set of parameters to be specified in the material list may be grouped according to their needs in the different program functions.

<u>Basic properties for structural elements (used for Static Analysis)</u>

E-Modl	E Modulus (Young’s Modulus) for the longitudinal direction.
E-Modt	E Modulus (Young’s Modulus) in the transverse direction.

Poiss.	Poisson's ratio
G-Mod	Shear Modulus.
ALFA-T	Coefficient of Thermal expansion/contraction
Gamma	Specific Weight

Properties of Reinforcement Steel

E-Modl	Reinforcement Steel E-Modulus

Properties used for Pre-stressed Steel

E-Modex	Pre-stressing Steel E-Modulus for extension calculation.
SIG-allow-pr	Allowable tendon stress for the tensioning process (reference value).
SIG-allow-SA	Allowable tendon stress for the final state after creep and shrinkage for the maximum loading state.

Properties used for Creep Analysis

Fc 28	28 day concrete cylinder strength
CF	Coefficient of consistency
ZF	Degree of cement hardening (1-3)
WCR	Water cement ratio
CECO	Cement content in concrete

Time dependent functions used for time dependent analysis

PHI(t)	Creep coefficient variation with time.
EPS(t)	Shrinkage coefficient variation with time.
RHO(t)	Relaxation factor variation with time.
EMOD(t)	E-Modulus variation with time.

Material properties for design checks for steel (and similar) structures
Stress limits and material safety factors as described in 3.2.6

Material properties for concrete reinforcement design
Concrete strength values and stress limits as described in 3.2.6

Stress limits for (pre-stressed concrete) fibre stress checks
Concrete stress limits for different load combinations as described in 3.2.6

Material properties for principal stress check (shear capacity)
Principal stress limits and check limits as described in 3.2.6

Material properties for ultimate load check (stress and strain limits, σ-ε diagram)
Stress-Strain diagram data for pre-stressing steel material as described in 3.2.6

3.2.2 Material Groups

The defined materials in the *RM2000* database are assigned to material groups specifying the kind of their usage in the structural system.

The allowed material group names are:

- Concrete
- Steel (Structural)
- Reinforcement
- Pre-stressing Steel
- Wood
- Aluminium
- Other

The material group is not a database object but an attribute of the material, i.e. there cannot be materials with the same material name in different groups. The group name is simply an additional information for the user and used to narrow the selection field.

Note: *The assignment of a special group name to a material does not actually affect any computation procedure. The program does not prevent the user from using misleading group names in the analysis. (E.g. a material designated as "concrete" could be assigned to a pre-stressing tendon without an error message).*

3.2.3 Basic Physical Parameters

Needed for: All structural elements (except spring elements)
All parts of composite structural elements

Not needed for: Reinforcement (except E-Modl)
Pre-stressing tendons (except E-Modl)

Parameter	Other Design Code Notations	Description
E-Modl	E_c (concrete), E_s (steel), etc.	E-Modulus (Young's Modulus) for the longitudinal direction.
E-Modt	E_t	E-Modulus (Young's Modulus) for the transverse direction.
Poiss.	ν	Poisson ratio (transverse expansion coefficient)
G-Mod	G	Shear modulus
ALFA-T	α_t	Thermal expansion/contraction coefficient
Gamma	γ	Specific weight

E-Modl (E28), G-Mod

The basic parameters needed for any structural analysis of beams are the modulus of elasticity (Young's modulus) and the shear modulus. For isotropic materials, these parameters are related to each other via the Poisson ratio (transverse strain coefficient).

The Young's modulus (E-Modl) is stored internally on the intrinsic variable E28 to be used for the definition of other user defined variables.

Poiss.

Poisson's ratio itself is not needed for the analysis of beam structures, but is useful for calculating the shear modulus – Poisson ratio is more commonly tabulated in Design Codes and Design Guidelines than the shear modulus. The shear modulus will be automatically calculated using the modulus of elasticity and Poisson's ratio.

E-Modt

The database also contains an elasticity modulus for the transverse direction, which is actually not used in the calculation process.

ALFA-T

This coefficient is used for the analysis of temperature loading cases. The value of this coefficient is approximately 1.0E-5 [1/°C] for all steel and concrete types, but some design codes require the use of slightly different values.

Note: _The value here defined as a material parameter in the material table is actually only used for analysing non-linear temperature distributions (Action TempVar, see chap. 6.3.7.2), but not for loadings specified by Load Type T (temperature load). ALFA-T is there directly entered as a load parameter. The value of the material table is not considered in this case, even not, if the load parameter Alpha is not defined (=0.0). No loading is applied in this case._

Gamma

The value Gamma is used in certain Load Types for the calculation of self weight and/or masses The value assigned to the material is used, if no other value „Gam“ is directly specified for the Load Type „Self weight“ (Gam=0.0), otherwise the directly specified value will be taken. (see chap. 6.3.2.3, Self Weight).

3.2.4 Properties of Reinforcement and Pre-stressing Steel

Needed for: Reinforcement
Pre-stressing Tendons

Not needed for: Structural elements

Parameter	**Design Code Notations**	**Description**
E-Modex		Pre-stressing steel E-modulus for extension calculation.
SIG-allow-pr	σ_{pm0} (~ $0.7*f_{pk}$) (ON B4750)	Allowable tendon stress after the tensioning process. (f_{pk} = characteristic tensile strength of the pre-stressing steel)
SIG-allow-SA	zulσ_p=$0.7*f_{pk}$ (ÖNB4750)	Allowable tendon stress for the maximum loading conditions after creep and shrinkage.
XI	k_b	Factor for considering reduced adhesion in the crack propagation check (ON B4750).

These parameters are used for the analysis of the tendon tensioning process and special design checks related to pre-stressed concrete.

The basic parameter E-Modl described in the previous section is also required for pre-stressing and reinforcement steel materials, although they are not assigned to structural elements. It is used in the program for establishing the composite cross-section values of reinforced or pre-stressed beams. All other basic parameters are not used for these materials.

<u>E-Modex</u>
E-Modex is a fictitious elasticity modulus of the tendons used for the calculation of the elongation of the tendons. The basic elasticity modulus of the pre-stressing steel (Emodl) is sometimes reduced by 5 to 10 % for this elongation analysis, considering additional flexibilities caused by transversal movement of the tendons in the duct and other influences.

<u>SIG-allow-pr, SIG-allow-SA, XI</u>
These stress limit values are used for the design of the tendon tensioning process (see chap. 5, Pre-stressing) and for special design checks related to pre-stressed concrete analysis.

3.2.5 Properties used for Creep Analysis and Time Dependency

Parameter	Design Code Notations	Description
Fc28	σ_p, f_c',	design value of concrete compressive strength.
CF	-	Consistency parameter of the fresh concrete.
ZF	-	Cement hardening parameter.
PHI(t,t0)	$\varphi(t,t_0)$	Creep coefficient variation with time.
EPS(t,t0)	$\varepsilon_{cs}(t,t_s)$	Shrinkage coefficient variation with time.
RHO(t)	$\rho(t)$	Relaxation factor variation with time.
EMOD(t)	$E_c(t)$	E-Modulus variation with time.

Fc28
Design value of concrete compressive strength. This value is defined in most design codes as the compressive strength of a cylindrical probe at an age of 28 days. Some creep and shrinkage models us it, to determine the creep coefficient and the time dependent elastic modulus.

The parameter Fc28 is assigned internally to the intrinsic variable Fc28 to be used for further user defined variable definitions.

CF
The parameter CF characterises the consistency of the fresh concrete at casting time. Three fresh concrete consistency groups are distinguished in accordance with EN V 206:

CF = 1	stiff	(small water-cement ratio)
CF = 2	plastic	(medium water-cement ratio)
CF = 3	semi-fluid	(high water-cement ratio)

The parameter CF is assigned internally to the intrinsic variable CF to be used for further user defined variable definitions.

This parameter is used in several creep& shrinkage model codes to determine the creep or shrinkage coefficients, e.g. in the improved MC 78 shrinkage prediction model. Intermediate values are allowed to be entered, the related coefficients will then be determined by interpolation procedures.

ZF

The cement hardening parameter ZF is also used for the determination of creep and shrinkage coefficients. It characterizes the type of cement used for the concrete. Three standard cement quality types are on the market:

ZF = 1	slowly hardening cement (SL)
ZF = 2	normal and rapid hardening cement (N, R)
ZF = 3	rapid hardening high strength cement (RS)

The parameter ZF is assigned internally to the intrinsic variable ZF to be used for further user defined variable definitions.

This parameter is also used in several creep & shrinkage models for the determination of the creep and shrinkage coefficients. Intermediate values are allowed, resulting in interpolated related coefficients.

Note: *The definition of CF and ZF is only required for creep & shrinkage models based on them. They are not necessary, when user defined creep and shrinkage coefficients that do not depend on them are used (for further details chap. 7.4.4.1. "Material Parameters").*

WCR

The water-cement-ratio (WCR) is the ratio between the content of water and the content of cement in the fresh concrete. It defines the amount of water per 1 kg cement). This ratio governs essentially the quality of the concrete. The strength of the concrete increases with decreasing values of WCR (0.4-0.7).

CECO

CECO describes the content of cement in the concrete. It defines the amount of cement (weight) in the volume unit of the concrete (kN/m^3 when default units are used).

PHI(t, t_0,…)

Creep coefficient, describing the ratio between the creep strain and the appropriate elastic strain. The creep coefficient must be specified as a user defined variable, if creep and shrinkage should be considered in the analysis. Predefined variable sets covering standard creep model codes (CEB MC 78, CEB MC 90, …), may be used, if the required parameters (Fc28, CF, ZF, RH, …) for evaluating the creep coefficient have been defined. The required data are described in detail in chap. 7.4, Creep & Shrinkage.

EPS(t, t_0, …)

Shrinkage coefficient, describing the shrinkage/swelling strain within a certain time interval. The shrinkage coefficient must be specified as a user defined variable, if creep and shrinkage should be considered in the analysis. Predefined variable sets covering standard creep model codes (CEB MC 78, CEB MC 90, …), may be used, if the required parameters (Fc28, CF, ZF, RH, …) for evaluating the shrinkage coefficient have been defined. The required data are described in detail in chap. 7.4, Creep & Shrinkage.

RHO(t, t_0)
Relaxation factor for the pre-stressing tendons as a function of the time interval. A user defined variable has to be defined if the relaxation of the pre-stressing steel should be taken into account.
This functionality is not yet implemented!

EMOD(t)
E-Modulus variation with time. A user defined variable may be defined, describing the time dependency of the elasticity modulus E-Modl of the structure. This might be a function related to the basic modulus via the intrinsic variable E28, or some other definition (e.g. related to the design compressive strength as proposed in the CEB90 model code). Note that the time dependent modulus is not automatically used in the LOADS AND CONSTR. SCHEDULE analysis, but must be activated by the update function in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION.

3.2.6 Properties for Design Code Checks

Material properties for design checks for steel (and similar) structures

Parameter	**Design Code Notations**	**Description**
Sigma-F	σ_y, f_y	Yield stress
Sigma-F*	σ_y^*, f_y^*	Fictitious Yield stress
Sigm-V1		Stress limit for “ordinary case”
Sigm-V2		Stress limit for “extraordinary case”
Gamma1	γ_1	Material Safety Factor
Gamma2	γ_2	Material Safety Factor
Gamma3	γ_3	Material Safety Factor
Sigma1		Allowable axial stress – “ordinary case”
TAU1		Allowable shear stress “ordinary case”
NY*1		Fictitious buckling safety “ordinary case”
Sigma2		Allowable axial stress „extraordinary case“
TAU2		Allowable shear stress „extraordinary case“
NY*2		Fict. buckling safety “extraordinary case”

Material properties for structural concrete design

Parameter	Design Code Notations	Description
Sig-p		Design value of concrete compressive strength
W28		28 day Concrete Cube Strength
Sig-allow-ch	f_{ctk}	Characteristic concrete tensile strength
Sig-allow-m	f_{ctm}	Mean concrete tensile strength
TAU1	$\tau_{d,1}$	Shear stress limit - zone 1
TAU2	$\tau_{d,2}$	Shear stress limit - zone 2
TAU3	$\tau_{d,3}$	Shear stress limit - zone 3
TAUB		Shear stress limit for beams

Stress limits for (pre-stressed concrete) fibre stress checks

Parameter	Design Code Notations	Description
Sigma-F		Yield stress
Tension stress limits		
General		Tensile stress limit for group 1
Grp2		Tensile stress limit for group 2
Grp3		Tensile stress limit for group 3

Grp4		Tensile stress limit for group 4
Grp5		Tensile stress limit for group 5
Grp6		Tensile stress limit for group 6
Compressive stress limits		
General		Compressive stress limit for group 1
Grp2		Compressive stress limit for group 2
Grp3		Compressive stress limit for group 3
Grp4		Compressive stress limit for group 4
Grp5		Compressive stress limit for group 5
Grp6		Compressive stress limit for group 6

Attention: **Group 6 (without assignment by the user) is taken automatically for checks in acc. with the new Austrian Standard ON B4750 (FIBII for cracked tensile zone).**

Group 1 (General) is taken for ordinary stress check as well as for stress checks for pre-stressing tendons (action FIBCHK and TENDCHK)

<u>Material properties for principal stress check (shear capacity)</u>

Parameter	Design Code Notations	Description
Principal Stress Limits		
SIGG-Q		Serviceability state shear force only
SIGG-Q+MT		Serviceability state shear force and torsion
SIGX		Longitudinal stress limit (zone B)
SIG1-Q-ST		U.L.S shear force only (for webs)
SIG1-Q-PL		U.L.S shear force only (for slabs)
SIG1-Q+MT		U.L.S shear force and torsion (flange-web)
SIG2-Q+MT		U.L.S shear force and torsion (maximum)
SIG2-G		U.L.S compressive stress in flange plates
Check limits		
SIG1-ST		U.L.S shear force only (for webs)
SIG1-PL		U.L.S shear force only (for slabs)
SIG1-MT-M		U.L.S torsion (maximum)
SIG1-Q+MT-M		U.L.S shear force and torsion (maximum)

Material properties for ultimate load capacity check
(stress and strain limits, σ-ε diagram)

Parameter	Design Code Notations	Description
EPS-PL		Proportionality limit of the stress-strain curve of pre-stressing steel
EPS-*		Yield strain
SIG-0.2		Stress for a strain value of 0.2%
SIGMA*		Maximum allowable stress
SIG-0.2/E		Strain related to SIG-0.2 (=0.2%)
SIG-ZUS		Allowable additional stress of un-bonded tendons
X1		Coefficient defining the Tendon type
EPS1-8		Up to 8 strain ordinates for the stress/strain curves in [‰]
SIG1-8		Up to 8 stress ordinates for the stress/strain curves related to the above strain ordinates

EPS-PL, EPS-*, SIG-0.2, SIGMA*, SIG-0.2/E, SIG-ZUS, X1
These parameters are actually not used.

EPS1-8, SIG1-8
Up to 8 pairs of strain/stress values describing the stress strain diagram of the material. This diagram must be specified for the element material (concrete) as well as for the pre-stressing steel and (if existing) the reinforcement steel. The strains are entered in [‰] (per mille), the stresses in the selected stress unit.

The diagram must be specified for the whole allowable strain range. Negative values (compression) must be entered with negative sign.

3.2.7 Definition of Material Data

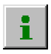
Material data can be directly input into the database by using the function ⇧PROPERTIES ⇨MATERIAL.

Alternatively, material property data can be imported into the program from the Standard Database or any other Project Database using the function ⇧FILE ⇨DEFAULTS, or from previously exported or user prepared material tables in the ASCII or TCL format using the function ⇧FILE ⇨IMPORT. A list of standard material tables delivered as a part of the program package is given in chap. 1.1.1.

Note: *The subsequent importing more than one material table will store all materials of all tables in the database. If the materials in the tables have the same names, materials imported at a later stage will overwrite the previously stored data.*
Materials already assigned to the system may not be deleted from the table, but their data can be changed.

The material input pad consists of two tables:

- The Material table (upper table) shows all existing material types. A new material can be created by using the 'Insert before' or 'Insert after' buttons, existing material types can be modified using the 'Edit' button (at the top of the Material Table).
- The Material Parameter Table (lower table) lists all material properties of the currently active (selected) Material. Only the 'Modify' button is applicable for this table.

The  - button at the top of the Material Table opens an overview window of the currently active (selected) material of the upper table. This overview shows all material properties (same as in the lower table) and allows the definition and modification of each value.

- The **first block** of this overview is inactive for the user. Name, Number etc. of the material are displayed. This block remains always the same.
- The **second block** allows the input and modification of several properties. The requested input is dependent from the selected property group. The property group is selected in the
- **Third block**. Clicking the arrow symbol at the right of the property group name modifies the second block.

All values are directly input in the corresponding field next to the property title. Exceptions are the 'Time Dependency Functions' and the 'Design check parameters'. The Variables describing the time dependency (Creep, Shrinkage, Relaxation, Stiffness changes) may be directly entered if they are user defined, or selected from the TDV provided Variables by clicking the arrow button and selecting one of the offered Variables. For details see chap. 3.2.5 or chap. 7.4, Creep & Shrinkage.

The design check parameters are grouped according to the checks they are used for. Clicking the arrow right of the group specification opens a new window, where the parameters appropriate to the selected design code check may be entered.

The buttons at the bottom of the overview allow to switch between the available materials of the upper table. The <OK> button confirms the input (a second confirmation is required). The material parameter table is automatically updated after confirmation of the modification.

3.3 Reference Point Groups

3.3.1 General

Reference Points are points in a cross-section (and directly related to it). They can be generated interactively either with the graphic definition of cross-sections in *GP2000* or with the function ⇧PROPERTIES ⇨CS ⇩ADDPNT.

Note: *The definition of Reference Points in RM2000 can be very extensive (especially at varying geometry of the CS). It is therefore recommended to define all Reference points already in the geometric preprocessor GP2000.*

Reference points can be used to define:

1. the position of the bending reinforcement in the cross-section
2. points where stresses should be evaluated (e.g. fibre stress check)
3. points for defining a non-linear variation of the temperature over the cross-section
4. points for the description of cuts and characteristic lines (Perimeter) used for the shear dimensioning
5. connection points to the sub-structure (position of the bearings) (**only in *GP2000***)

When defining a reference point the user is asked to input the “type description” and a “group name”, both assigned to the point in order to unite Reference Points with the same properties. The type descriptions define the purpose of the point and are selected out of a predefined list. This list is visualised by pressing the “pull down arrow” at the definition window (see chap. 3.3.3, Types of Reference Points).

The group names are entered by the user (⇧PROPERTIES ⇨ADDGRP) together with the related properties. These group definition is used to distinguish Reference Points of the same type but different purpose (e.g. points for upper and lower layers of bending reinforcement). These group names together with the related parameters are called “Reference Point Groups”. “Additional Group” is an other term used in the program for the “Reference point groups”.

3.3.2 Definition of Reference Point Groups

The Reference Point Groups are sets of parameters describing properties of the Reference Points.

Note: *If Reference Points are defined the in the geometric preprocessor* GP2000*, then related groups are also defined there.*

The table of the “Reference point groups” contains the following definitions:

- Name — Name of the Reference Point Group
- Material — Name of the material
- StrGrp — Stress Limit Group
- Description — Descriptive text (max. 80 characters)

The Reference Point Groups must be defined before they can be assigned to Reference Points. It is recommended to consider before defining the cross-section, which groups are in total needed, and to specify the corresponding table already at the beginning.

The Type of the Reference Points is not a group property but related to the single Reference Points as described in chap. 3.3.3. This allows to group together also Reference Points of different types. In later functions, where the group name is assigned, only the points of the appropriate type will be considered.

But in order to keep the calculation transparent the points of different types ought also to be related to different groups (e.g. a group, describing the reinforcement, should not include a point for a fibre stress check, etc.).

3.3.3 Types of Reference Points

Reinforcement points:(Type REIxxx) to define the distribution of the reinforcement in the cross-section used for the Ultimate Load check and the dimensioning of the reinforcement (see chap. 8, Design Code Checks)

Stress points: (Type FIBPOI) to define where the stresses within a CS should be evaluated (see chap. 8, Design Code Checks)

Temperature points: (Type TEMPPOI) to describe - if necessary - the distribution of the temperature in the CS (see chap. 6.3.7, Temperature Loads)

Characteristic shear points: (Type PERxxx, LINxxx) These points define characteristic lines of the cross-section (border line between web and flanges) in order to calculate the shear stresses or the needed shear reinforcement. The program function “ShChk” (Shear capacity check) uses these points as a reference.

Note: *GP2000 also provides the type “Connection point”, which does not exist in RM2000. GP2000 transforms the points of this type directly to spring elements and eccentric connections and transfers this data RM2000.*

3.3.4 Definition of Reference Points in RM2000

3.3.4.1 General

⇧PROPERTIES ⇨CS ⇩ADDPNT supports the definition of a reinforcement distribution in a cross-section as well as the definition of stress points and diagrams for non-linear temperature variation in the cross-section (as it is the case for AASHTO code temperature loading).

Note: *All Reference Points (reinforcement, stress and temperature points etc.) are defined in relation to cross-sections. The work required for the input increases quickly if lots of different cross-sections exist in the structure. It is therefore strongly recommended to define any reinforcement and stress point in GP2000, where it can be done interactively for a series of cross-sections at once.*

After calling the function ⇧PRPOPERTIES ⇨QS ⇩ADDPNT the cross-section table (upper table on the screen) and the Reference Point table for the active cross-section (lower on the screen) are shown.

Any new reinforcement points or stress points etc. may be created, existing reference point definitions can be deleted or modified by using the appropriate buttons above the lower table.

The following data is asked for, when the 'Insert before', 'Insert after' or 'Modify' button is pressed:

- Point name — Each Reference point must be named by a string

  Note: Only the first 4 characters of the point name will be considered in the fibre stress check. Therefore it is recommended to use only names within this limit.

- Type — Type of the point that is to be defined (an interactive selection is offered when pressing the arrow key next to the input field).

The following Reference Point Types can be chosen:

Types for the reinforcement:

REIPSI	Single point for concentrated reinforcement
REIPOB	Start point of a polygonal line for a distributed reinforcement
REIPOM	Intermediate point of the polygonal line
REIPOE	End point of the polygonal line
REICUM	Mid-point of a curved section of the polygon (REIPOB – REICUM – REIPOE if the total polygon is described by one curve, or REIPOB – REICUM – REIPOM or REIPOM – REICUM – REIPOE).

Note: *A curved section is again approximated by a polygon. In order to get a good approximation of curves with a big aperture angle, they should be divided into sections. E.g. a curve with an opening angle of 270° has to be split up at least into three parts.*

Type for fibre stress checks:

FIBPOI	Point inside the cross-section for stress evaluation

Type for temperature distribution:

TEMPPOI	Point, for specifying a non-linear temperature distribution

Types to describe the characteristic lines for the shear evaluation (characteristic perimeter for shear flow, cut lines between webs and flange plates):

PERPOI	Perimeter point
PERCRC	Mid-point of a curved perimeter section (PERPOI-PERCRC-PERPOI) (polygonal approximation: see REICUM)
LINPOB	Start point of a line (definition of a cut line between web and flange plate)
LINPOE	End point of a line (definition of a cut line between web and flange plate)

- Group — Assigned “Reference Point Group”. This group name is selected from the existent groups shown in the pull-down menu. All available groups must have been previously defined with ⇧PROPERTIES ⇨ADDGRP in the Reference point group table. Stress points (FIBPOI) and temperature points (TEMPPOI) do not necessarily need an assigned group name.

3.3.4.2 Parameters to define the position of the reference point

There are two alternative ways to define the position of the reference point:

- related to the system axis (global reference point of the cross-section, see chap. 3.4.3) and
- related to the intersection of two lines each defined by two nodes in the cross-section.

1.) Related to the system axis

In this case neither node numbers nor angles need to be specified. The distances of the reference points from the system axis are defined in the coordinate system of the cross-section.

➢ Node1/Node2 (left)	0 (not used)
➢ Node1/Node2 (right)	0 (not used)
➢ Dist/z	Horizontal distance from the system axis (in z-direction of the coordinate system of the CS) (Mind units and sign!)
➢ Dist/y	Horizontal distance from the system axis (in y-direction of the coordinate system of the CS (Mind units and sign!)
➢ Angle (left)	0 (not used)
➢ Angle (right)	0 (not used)

2.) Related to two edges

The intersection of two edges of the cross-section and the given distances/angles describe the exact position of the reference point.

The 2 vectors built by the specified nodes (both orientated from the first to the second node) are used to define a local coordinate system (not necessarily rectangular) for defining the distances and angles respectively.

➢ Node1/Node2 (left)	Two nodes of the cross-section defining the first edge
➢ Node1/Node2 (right)	Two nodes of the cross-section defining the second edge
➢ Dist/z	Distance from the first edge (Sign: positive at the left and negative at the right if looking in the direction of the first vector from Node1 to Node2!)
➢ Dist/y	Distance from the second edge (Sign: same sign convention as Dist/z)
➢ Angle (left)	Angle to convert Dist/z
➢ Angle (right)	Angle to convert Dist/y

Attention:	**The angle entered in the left row is used to convert the distance Abst/y (in the right row), the angle entered in the right row is used for converting the distance Abst/z (in the left row). The user specified distance is multiplied by the sine of the related angle and then interpreted as distance from the other line.**

3.3.4.3 Parameter to define the temperature

- TMP — Temperature difference with respect to the initial state in this point (positive when the temperature increases, negative when cooling down)

3.3.4.4 Control view and graphic interface to define the position

A graphic window appears when pressing the INFO-button at the upper edge of the lower list (instead of the 'Insert before' -, 'Insert after' - or 'Modify'-buttons). This window shows the selected cross-section together with the defined Reference Points. An input pad (firstly inactive) showing the actual input values of the selected reference point) is opened on the left side of this control view. This input pad can be activated by the appropriate buttons. in order to enable changing the data or inserting a new point. When pressing the ‚Apply'-button the modified values are entered into the reference point table.

- Apply — The definition is added into the lower table
- Cancel — delete definitions without confirmation

The cross-section view can be manipulated by any ‘Zoom’ function as well as by selecting the buttons at the top of the view:

- ☑ TxtFact — Text size of all numbers in the graphic (higher factor – bigger label size)
- ☑ Elem — Element are shown/not shown
- ☑ Nod — Nodes are shown/not shown
- ☑ El-Numb — Element numbers are displayed/not displayed
- ☑ Nod-Numb — Node numbers are displayed/not displayed
- ☑ Reinf — Reinforcement definitions are displayed/not displayed
- ☑ Strp — Stress points are shown/not shown

By using the zoom function (with key “Ctrl” + drawing a diagonal line) together with node labelling switched on, the required node numbers describing the related edges can be identified on the screen. These values can be entered manually into the corresponding input fields of the input pad.

The data (node numbers) can also be input interactively by selecting the node from a pull-down menu. A short free-hand line quickly drawn over the node position opens a pull-down menu, showing the node number(s) at that position. The appropriate number can be selected to be entered in the edge definition field.

3.3.5 Definition of the Reinforcement (Reinforcement Points)

The term „reinforcement" used in this section indicates the longitudinal reinforcement carrying the forces induced by bending moment and normal force. The required input data for the shear reinforcement is described in section 3.3.8.

In order to specify the reinforcement of a reinforced concrete structure or to prepare the determination of the required reinforcement by the program, the user has to perform following tasks:

1. Definition of the Reference point groups in ⇧PROPERTIES ⇨ADDGRP
2. Setting the required Reference Points in ⇧PROPERTIES ⇨CS ⇩ADDPNT
3. Definition of the fixed and the variable reinforcement in ⇧STRUCTURE ⇨ELEMENT ⇩REINF.

3.3.5.1 Reference point groups for the reinforcement (Reinforcement Groups)

Reference point groups related to reinforcement points are also called Reinforcement Groups. The definition of "Reinforcement Groups" is the main application field for Reference Point Groups. The purpose is to distinguish the various reinforcement layers related to different stress distribution states. This can not be done by using the Reference Point types description. The types only describe the distribution of the reinforcement in the cross-section (e.g. concentrated in a single point, distributed over a line or even over a given perimeter).

Attention: It is not allowed to mix point reinforcements and line reinforcements in the same group, because no proper distribution of the whole reinforcement is possible in this case. RM2000 does not check if point and line reinforcements are mixed up in the same group! The calculation is done with an un-controlled distribution.

The material number of the reinforcement steel is assigned to the Reference Point Group. It is possible to use different steel qualities for different reinforcement layers.

The Stress Limit Group is not used in this case and needs not to be defined for Reference Point Groups specifying a Reinforcement Group.

3.3.5.2 Specifying the required Reference Points (reinforcement points)

The Reference Points for the definition of the reinforcement describe the position of the reinforcement in a cross-section. The Reference Point Type (REIPSI, REIPOB, REIPOM, REIPOE or REICUM; see chap. 3.3.4) assigned to the different points, only defines the distribution of the reinforcement in the CS.

The Reference Points belonging to the considered Reinforcement Group have to be defined at the start and end cross-sections of the element to enable the assignment of a reinforcement to an element. It is therefore recommended to define all reinforcement points in all cross-sections of a continuous element series, even if some reinforcement layers are not needed in a certain section of the superstructure.

3.3.5.3 Specifying the reinforcement in an element

The assignment of the actual reinforcement values to the elements is done in ⇧STRUCTURE ⇨ELEM ⇩REINF. The different Reinforcement Groups are automatically assigned to the elements, using the specified reinforcement points at the start and end cross-sections of each element.

Per default all reinforcement areas are set to 0. A specific amount of reinforcement can be assigned to a certain group by modifying the related line in the Reinforcement Groups table (lower list in the ⇧STRUCTURE ⇨ELEM ⇩REINF pad).

For each assigned Reinforcement Group can be defined:

- Whether it is allocated throughout the entire length of the element or just over a part of it
- whether the total reinforcement is split up in a constant and a variable amount (button “variable”) or fixed by as a constant value (button “fix”)
- a fixed amount A1 (cross-section area of the reinforcement)
- an additional amount A2, fixed or variable – depending on the button “fix/variable”

When pressing the “Modify”-button an input pad appears, asking:

➢ El-from	First element of the series
➢ El-to	Last element of the series for the reinforcement definition
➢ El-step	Element step within the series
⊙ Fix	Both reinforcement areas (A1 and A2) are fixed
⊙ Var	Only A1 is fixed, A2 is variable
➢ Group	Name of the Reinforcement Group (can be selected in the pull-down-menu)
➢ X1/l	Start point of the reinforcement in the element (default : 0.0)
➢ X2/l	End point of the reinforcement in the element (default: 1.0)
➢ A1	Fixed amount of the reinforcement
➢ A2	Variable (or another fixed) amount of the reinforcement as described in the precedent paragraph

The total reinforcement area is always equally distributed to the different parts of the Reinforcement group (i.e. for point reinforcements to all points, for line reinforcements to the total length of the line reinforcements).

The button “fix” is generally used if no additional reinforcement in this group is allowed. The value A2 = 0 is then never changed. In case of a variable value A2 the design module of RM2000 increases the value until the Ultimate Load Check indicates the required safety.

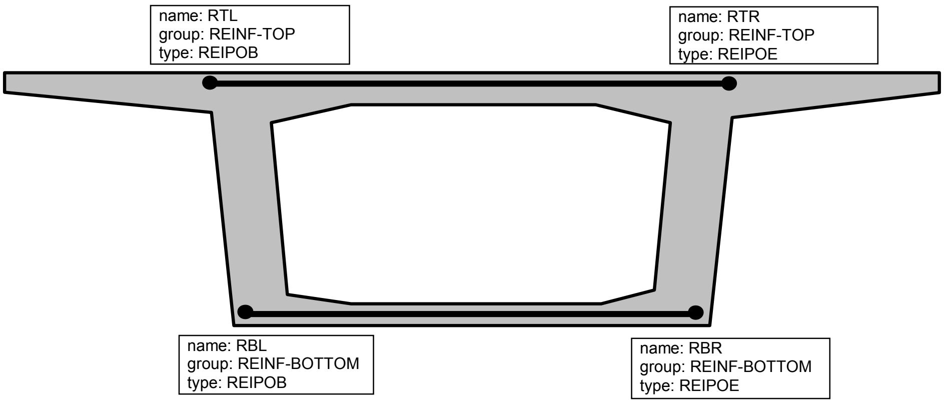
Note: *After a design action the user can change the button from “variable” to “fix” in order to avoid a further increase of A2.*

If more than one Reinforcement Groups with a variable reinforcement A2 exist, the program increases the reinforcement (during the design action) independently from each other. RM2000 always tries to reach a minimum of total necessary reinforcement. In general therefore the reinforcement with the largest distance from the neutral axis will be increased. If more layers of reinforcement are intended to be increased at the same time (e.g. for special symmetric reinforcement) they must be grouped together in one Reinforcement Group. A facility for prescribing a special sequence for increasing the different reinforcement layers (possibly up to a certain limit value) is actually not yet implemented in the program.

A reinforcement area A2 once calculated for a certain loading state is taken into consideration in later design actions for other loading states. If this effect is not desired the user has the choice to add the action ReinIni in the Construction schedule to re-initialise A2 and set it to the value 0 before .

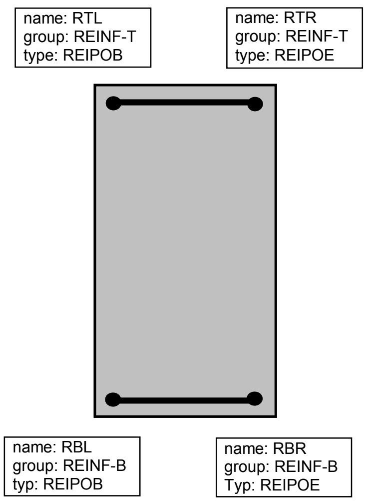
3.3.5.4 Example 1: “Box girder”

2 separate reinforcement layers (distributed over a line) are applied in this example on the top and on the bottom of the box girder.
2 Reference Point Groups (REINF-BOTTOM, REINF-TOP) are therefore specified.

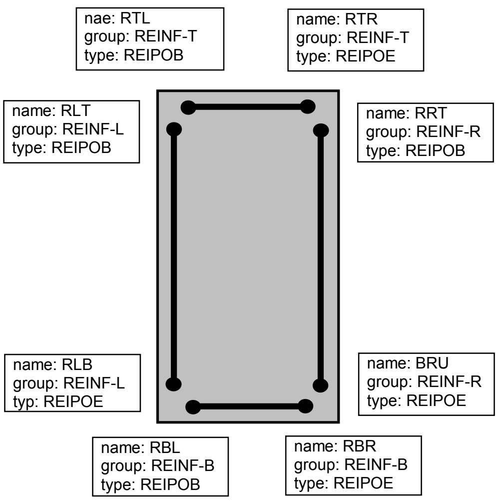


Each Reinforcement Group consists of two reinforcement points (REIPOB...start point of the polygon, REIPOE...end point of the polygon). The names can be chosen ad libitum (here RTL...reinforcement top left, RTR...reinforcement top right, RBL...reinforcement bottom left, RBR...reinforcement bottom right).

3.3.5.5 Example 2: Rectangular cross-section



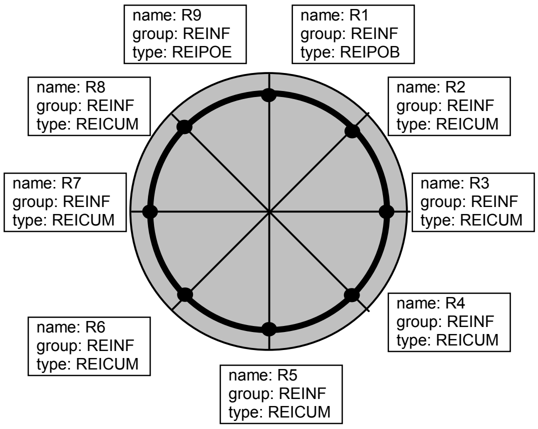
Example a: uni-axial design



Example b: bi-axial design

Two and four Reinforcement Groups are defined in this example (REINF-TOP, REINF-BOTTOM, REINF-LEFT, REINF-RIGHT). Each group consists of two reinforcement points (REIPOB...start point of the polygon, REIPOE...end point of the polygon). The names can be chosen ad libitum (here RTL...reinforcement top left, RTR...reinforcement top right, RBL...reinforcement bottom left, RBR...reinforcement bottom right, etc.). The points in the corners (e.g. RBR and BRU) may have the same position (same coordinates), but they must have different names. It is not possible to assign more than one reference group to the same point.

3.3.5.6 Example: Column



This example gives an idea how to place a Reinforcement Group (REINF) in a circular cross-section. This group is described by nine reinforcement points (REIPOB...start point of the polygon, REICUM…intermediate points of the polygon, REIPOE...end point of the polygon). The names can be arbitrarily chosen (R1-R9). Because of the fact that the same group is assigned to all points, the design module will determine a uniformly distributed reinforcement.

3.3.6 Definition of Stress Evaluation Points

Reference Points for Stress Evaluation are used to check the fibre stresses in the cross-section. The allocation to a Reference Point Group is not necessarily required if a standard Fibre Stress Check is to be made, where the stresses are calculated without any knowledge about the material properties just by using the internal forces and the cross-section values.

Note: *In RM2000 the Stress Evaluation Points can be defined without assigning a Reference Point Group. GP2000 however requires the assignment of a Group*

The assignment of a Reference Point Group to the Stress Evaluation Points may be required:

a) if the stress is calculated from the strain by using the material properties (e.g. if the stress in the reinforcement should be calculated).

b) if a Stress Limit Group should be assigned to the Stress Point. This can only be done by specifying the Stress Limit Group as parameter of the Reference point group. This option allows to assign different stress limits to different Stress Evaluation Points.

The material assignment of the Reference Point Groups is used to calculate the stresses in the Stress Evaluation Points and to check if they are within given limits. If no Group is assigned RM2000 uses the material parameter of the structural element (steel, concrete). By assigning the material “reinforcement steel” to a Reference Point Group it is possible to calculate the stresses in the reinforcement.

Note: *The program also allows to assign the name of a Reinforcement Group to Stress Points. Attention must be paid to the fact, that in those points the stress is calculated for the reinforcement steel (which may not exist there). Points for evaluating the stresses in the concrete have to be assigned a separate Group.*

Per default RM2000 uses the limits of the Stress Limit Group 1 of the structural material for the comparison of the stresses with the limits. The Stress Limit Group and the material assigned to the Reference Point Group is taken if a Reference Point Group is assigned.

The used Stress Limit Groups must have been defined as material parameters, see chap. 3.2.6 and 8.1).

All stress points are automatically numbered by the program. This stress point number can be used instead of the point name, when producing stress plots in ⇧RESULTS ⇨PLSYS.

3.3.7 Definition of a Temperature Distribution (Temperature points)

Reference Points of the type TMPPOI (Temperature Point) are used for the specification of a non-linear temperature distribution over the cross section. As the national codes which require to consider a non-linear distribution confine themselves to a non-linearity in vertical direction, RM2000 is also limited to investigate a non-linear temperature distribution only in the local Y-direction (see chap. 6.3.7, Temperature Loads).

The Temperature Points are generally defined along the local Y-axis. The points must be specified in the right order beginning with the highest point (upper surface) of the cross-section in the negative Y-direction. The Points need not necessarily lie on the axis because only the y-coordinate will be considered (the distribution in z-direction is assumed to be constant).

If several different distributions have to be considered (most codes require investigating a warming up and a cooling down case) the Temperature Points of the same load case must get assigned the same group. A new series of Temperature Points gathered in a separate group must be defined for each further non-linear temperature load case. Thus the assignment of a Reference Point Group can only be omitted when only one distribution case is investigated.
The Parameters “Material” and “Stress group” (StrGrp) are not considered in the temperature calculation. The function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION “TempVar” uses all points of the type TMPPOI which belong to the assigned group.

3.3.8 Characteristic Lines for the Shear Capacity Check

The theoretical background and the requirements for the preparation of the data for a shear design are given in chap. 8.4.

In order to make a shear design it is needed to define characteristic lines in the cross-section which describe the behaviour on shear loading, or the position where the shear stress values ought to be checked.

These lines are defined by the related Reference Points and Reference Groups. Two tasks of the shear check require the definition of Reference Points of the type PERPOI, PERCRC, LINPOB or LINPOE:

a) Perimeter and characteristic web thickness
b) Checks for the cuts between flange plates and webs

3.3.8.1 Substitute Box Cross-section and Web Width

The calculation of the shear flow due to torsion requires the definition of the characteristic perimeter. That is a closed line which describes a thin-walled substitute box to determine the shear flow. In the case of hollow box cross-sections this is – with sufficient accuracy - the centre line of the webs and flange plates forming the hollow box of the cross-section. In the case of solid cross-sections it is mostly assumed that the main part of the shear flow will be taken by the reinforcement. The characteristic perimeter is therefore defined as the line representing the position of shear reinforcement.

The perimeter is defined by points of the type PERPOI and PERCRC. The Perimeter Points must be given in a closed sequence. Curved sections are defined by points of the type PERCRC between the start and end point. An arbitrary point can be selected to be the first point (start point). The perimeter is not automatically closed, a separate end point has to be defined at the position of the start point.

The web width is automatically determined by the program using cuts perpendicular to the perimeter line. The decisive web width is needed for the shear capacity check. This is determined by the program by searching in the total web region for the minimum width. Therefore, the web region has to be specified by the user. This is done by specifying the 2 horizontal cut lines between webs and flange plates using Reference Points of the types LINPOB and LINPOE.

These two cut line definitions have to be assigned to the same Reference Group as the Perimeter Points.

Many design codes require for the shear capacity check for pre-stressed concrete cross-sections to decrease the web width by the sum of the diameters of the ducts arranged side by side in the web, or by a certain part of it. This characteristic duct width to be subtracted from the web width is determined automatically by the program if the tendon group to be considered in the actual cut contains only one tendon. The duct diameter, reduced in accordance with the rules of the selected design code, is subtracted in this case.

If there exists at least one tendon group in the considered cut which contains more than one tendon, this characteristic width cannot be determined by the program, because the information about how the tendons are arranged is missing (side by side or on top of each other). The user is asked to use the function **⇧STRUCTURE ⇨ELEMENT ⇩TIME (input values b-anf, b-end)** to enter the (possibly reduced) characteristic duct widths to be subtracted from the web width. These values are related to the start and end cross-sections of each element.

3.3.8.2 Cut Lines between Webs and Flange Plates

The vertical cut lines describing the connection faces between webs and flange plates are to be grouped together in a 2nd Reference Point Group. These cut lines are defined with Reference Points of the types LINPOB and LINPOE. The cut lines have to be defined in closed sequence, so that the flange plates are arranged between 1st and 2nd cut, 3rd and 4th cut, etc., i.e. left and right boundaries of the flange plates are defined in an alternating sequence.

It is strongly recommended to give names (4 characters) to the Reference Points (created using LINPOB and LINPOE), because these names are used to identify the cut line. The start point end the end point of a cut line may have the same name. If they have different names the name of the cut line will be one of the 2 names.

Points of different cut lines should have different names in order to allow a correct identification of the cut lines in the results listings.

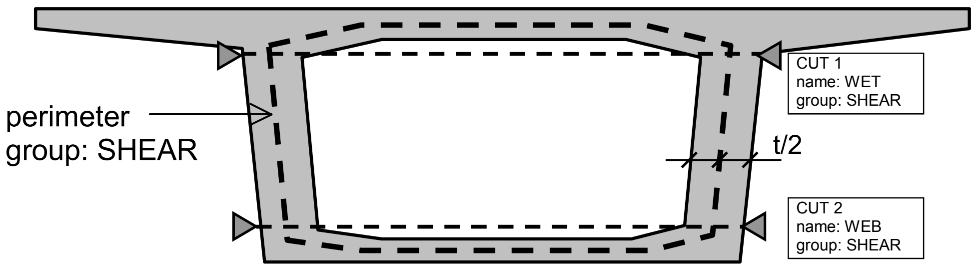
3.3.8.3 Results (amount of necessary shear reinforcement)

The results of the shear capacity checks (required amount of shear reinforcement) are not stored in the general database as database objects. They are only output in the result lists. Therefore they can not be viewed in the system tables of the GUI like the bending reinforcement values A1 and A2. And a function to use these results to be considered in the structural model (e.g. to adapt the stiffness) is not provided in RM2000.

The names of the cut lines (defined by the names of the Reference Points) are used in the result lists to identify the assigned values of the necessary reinforcement area.

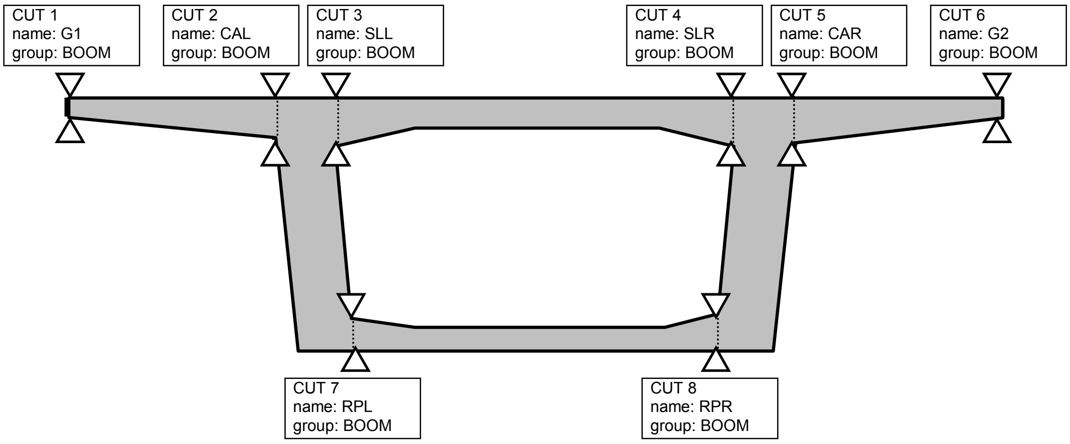
3.3.8.4 Example: Hollow box girder

Reference point group: SHEAR (for the definition of the perimeter and the web thickness)



Definition of the substitute box girder via PERPOI (Perimeter Points)
Definition of the webs (the diameter of the tendon is subtracted from the web thickness) using LINPOB (start point of a line) and LINPOE (end point of a line) i.e. any cut is defined by two points (LINPOB und LINPOE).

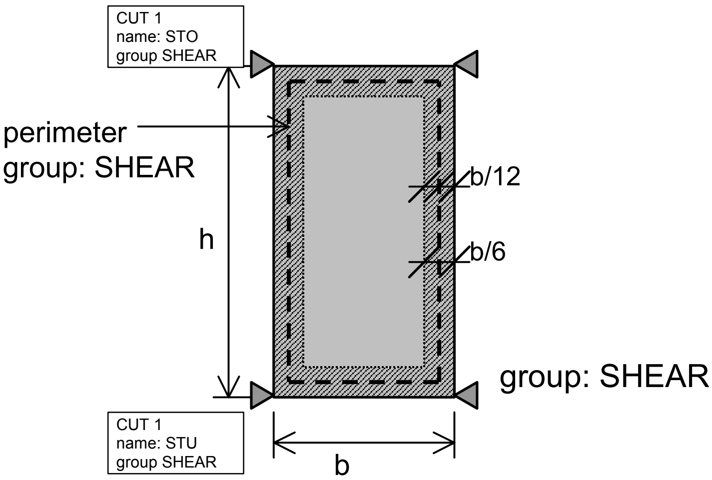
Reference point group: BOOM
(to define the cuts between the flange plates and the webs)



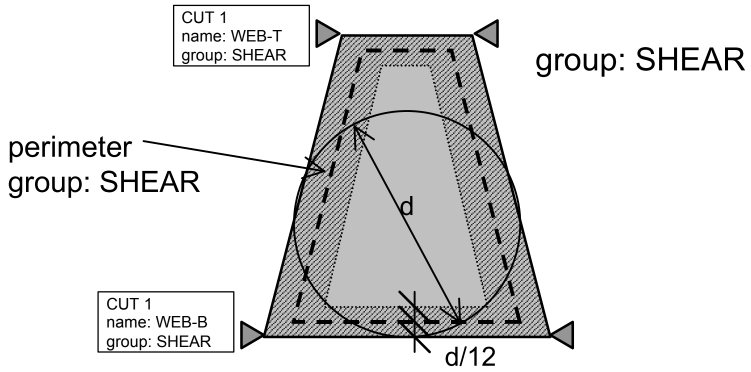
CAL....CANTILEVER-LEFT, CAR....CANTILEVER-RIGHT
SLL....SLEP-LEFT, SLR....SLEP-RIGHT
BPL....BOTTOM PLATE LEFT, BPR....BOTTOM PLATE RIGHT

The border lines G1 and G2 define the outside ends of the cantilevers.

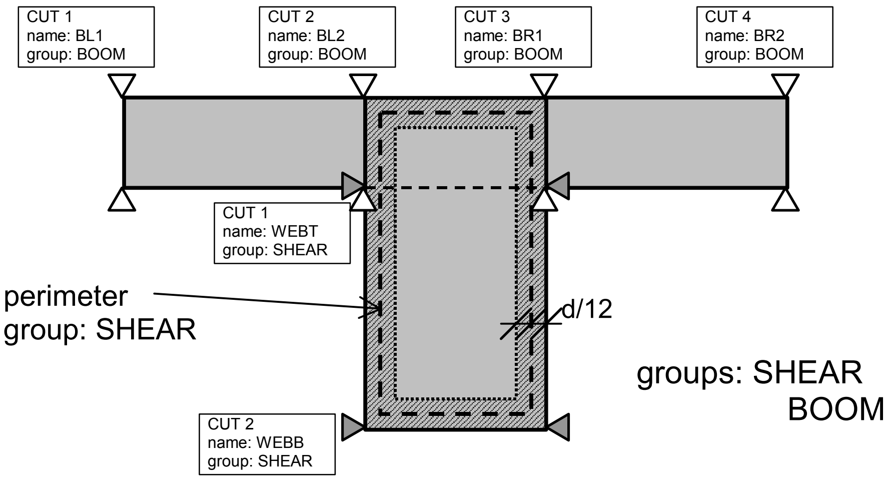
3.3.8.5 Example: rectangular cross-section



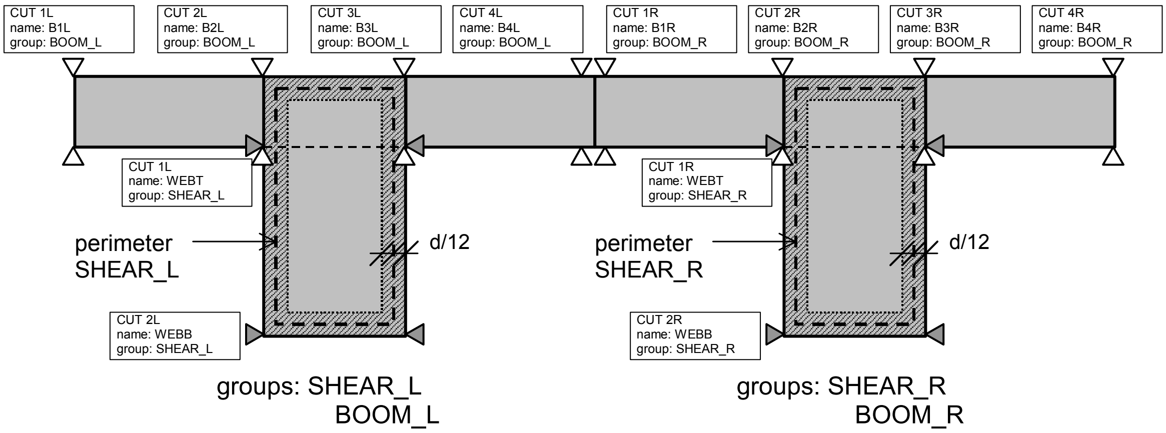
3.3.8.6 General cross-section:



3.3.8.7 Example: T-beam



3.3.8.8 Example: double T-beam



3.4 Cross Section Properties - CS

3.4.1 General

The most important precondition for structural analyses using beam elements is that the cross-sections of the beams remain plane in any state. This may be assumed to be approximately true for most of the bridge superstructures and columns, as the spans are mostly big compared to the cross-section height and width. However, there are many cases in bridge engineering, where this assumption is not valid and this approximation is not allowed (e.g. multiple tee-beam girders). In this case, the cross-section has to be divided into several independent parts building separate main-beams being connected by fictitious cross-beams simulating the connection (see chap. 4, Structure Modelling).

The cross section values for bending and normal force are usually computed differently to those for the shear resistance (shear forces Qy and Qz, torsion moment). RM2000 however uses a consistent approach for normal force, bending, shear forces, St.Venant torsion and warping resistance. There is therefore no need for the user to distinguish between „thick walled“ and „thin walled“ cross-sections and between „open“ and „closed“ (box girder) cross-sections.

The differential equation for the shear stress function for Qy, Qz and Mt is solved in RM2000 numerically using the Finite Element Method. The related cross-section parameters (shear area in Y- and Z-directions, torsion moment of inertia, and position of the shear centre) are directly computed from the solution of the differential equation. The warping resistance of the cross-section is computed using a similar F.E. approach. The cross-section values for Nx, My and Mz (area, moments of inertia and position of the gravity centre) are also computed in *RM2000* in this F.E. analysis. The values computed with this procedure are identical to those usually used for calculating the area and the moments of inertia. This – complete and consistent – calculation of the cross-section values presumes however, that the whole cross-section geometry is defined as a Finite Element mesh.

RM2000 offers 2 methods for defining cross-section parameters:

1. by entering the required cross-section parameters directly into the data base (no calculation by the program), or
2. by fully describing the geometry of the cross-section. The properties used for the stiffness calculation are in this case calculated in the function ⇧RECALC, option Cross-Section Calculation.

Directly defined cross-section properties are entered in the function ⇧STRUCTURE ⇨ELEMENT ⇩CS as element properties. These cross-sections are not themselves database objects. Cross-sections with fully described geometry are named database objects. They are created in the function ⇧PROPERTIES ⇨CS and they will be assigned to the elements by referencing the cross-section name later on in the function ⇧STRUCTURE ⇨ELEMENT ⇩CS.

Note: *Most design code checks, especially for reinforced and pre-stressed concrete, require the full geometric cross-section definition. The direct definition of the cross-section properties suffices only in the case of a standard structural analysis giving only displacements and internal forces as results. In the general case it will be necessary to describe the whole cross-section geometry.*

3.4.2 How to Model the Cross Section Geometry

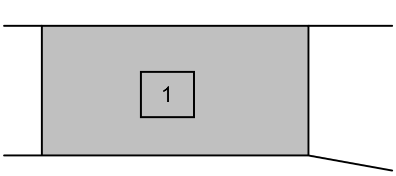
The calculation of the cross-section parameters of cross-sections with fully described geometry is done in RM2000 by using the Finite Element Method. 9-noded iso-parametric elements (Lagrange elements) are used for the computation. This requires the definition of all coordinates of the nodes of the elements as well as the incidents between nodes and elements to be defined, in order to describe the cross-section geometry.

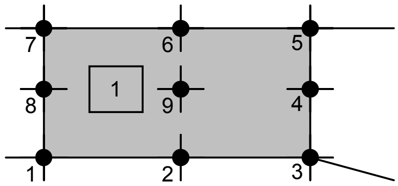
The element mesh used must be sufficiently fine to get accurate answers for the stiffness parameters. The quadratic shape function used in the calculation guarantees a good behaviour even with a very coarse mesh. It can be said, that in general for hollow cross-sections only one element over the thickness of the different cross-section parts will be sufficient. In the case of solid cross-sections it will be necessary to have 3 elements over the thickness to present sufficiently accurately the shear flow in the cross-section.

Note: *The FE solution is a general approach, using the exact shear stress distribution for calculating the torsion moment of inertia and shear centre. Therefore there is no difference any more for computing the cross-section parameters of thin-walled cross-sections (welded steel) and thick-walled cross-sections (concrete) as it was the case in the predecessor program RM7 (module RMQWOST for thin walled, module RMQWORE for thick-walled).*

The following sketch shows the element numbering scheme: the order of the nodes assigned to the elements in the cs-element table is first counter-clockwise along the border, the 9th node being the centre point.

Each node is defined by the coordinates and each element is defined by the 9 node numbers (in the right order).



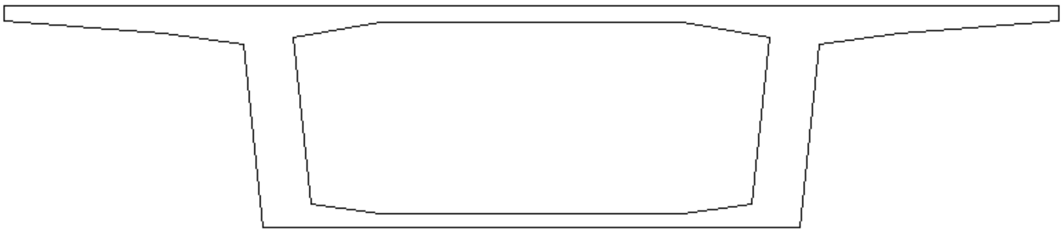


Cross section element sequence of element nodes in RM2000

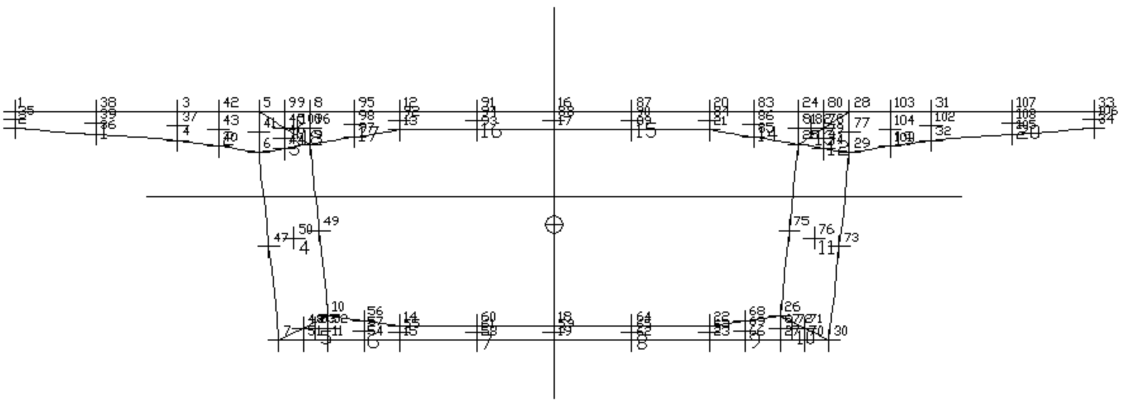
Example above: Element 1 is defined by the nodes 1, 2, 3, 4, 5, 6, 7, 8, 9 (in this sequence!)

The following examples show typical superstructure cross-sections and how to model them for RM2000:

Example 1:



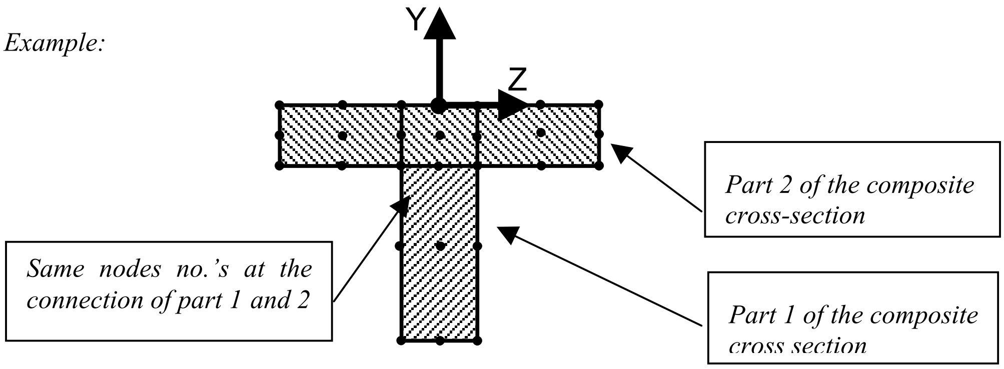
Simple hollow box cross-section



Element mesh for computing the cross-section values in RM2000

Note: *If you define parts of a composite cross-section (see chap. 3.4.7), all node coordinates have to be defined in a common coordinate system. The connection points of two parts needs to have the same number of elements, the same number of nodes and the same node numbers in order to create the composite cross section.*

Example:



The above cross-section and any other arbitrary cross-section may be defined in RM2000 by designing a suitable Finite Element mesh and entering the nodal coordinates and element incidences into the database. This may be done by using the function ⇧PROPERTIES ⇨CS ⇩NODE and ⇧PROPERTIES ⇨CS ⇩ELEMENT or by preparing ASCII files to be imported with the function ⇧FILE ⇨IMPORT.

To prepare a FE-mesh without graphics support is however a very tedious process for any but the simplest cross-sections and **it is therefore strongly recommended that the user should define non-standard cross-sections via the geometric pre-processor GP2000** which is available for use together with RM2000.

RM2000 additionally offers a set of macro functions for standard cross-section shapes described by few geometric parameters. These macros automatically create a suitable FE-mesh. The available standard cross-section types are described in chap. 3.4.3.

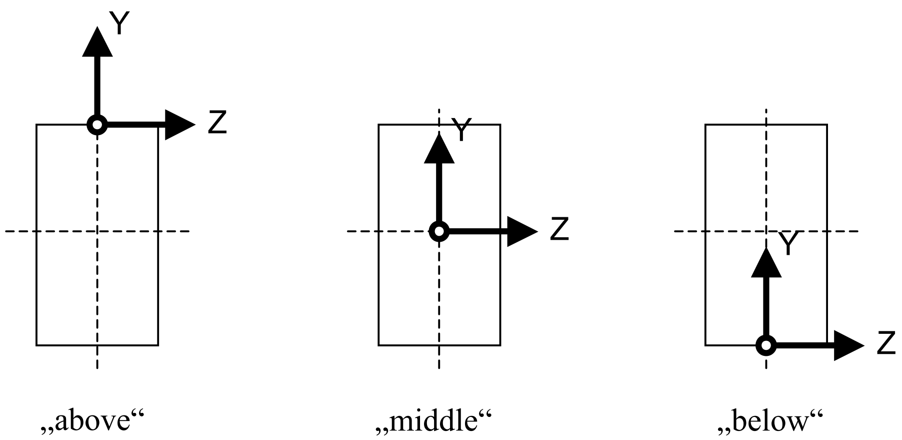
3.4.3 Standard Cross-section Types

Cross-sections can be defined in the form of “Standard Shapes” where a standard section profile (such as a “Tee Section” or an “I-Section”) is selected and the user simply inputs the required dimensions and thickness in the table provided.

The section sizes can also be defined as varying in dimension and thickness between a series of cross-sections ⇧PROPERTIES ⇨CS ⇩CS.

Also it is necessary to define the position of the coordinate origin in the cross section (reference to the system axis). Three types are allowed.

1. “above” The structural system axis will be set to the top of the cross section
2. “middle” The structural system axis will be set to the centre of the cross section
3. “below” The structural system axis will be set to the bottom of the cross section



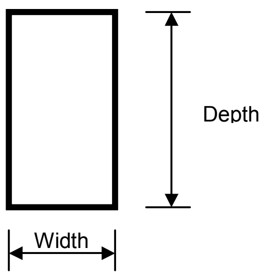
Chap. 3.5.1 shows more in detail, how to define general cross-sections and cross-sections with standardised shapes directly in RM2000, and how to modify cross-sections created in GP2000 or imported from external files.

The following standardised cross-section types are available:

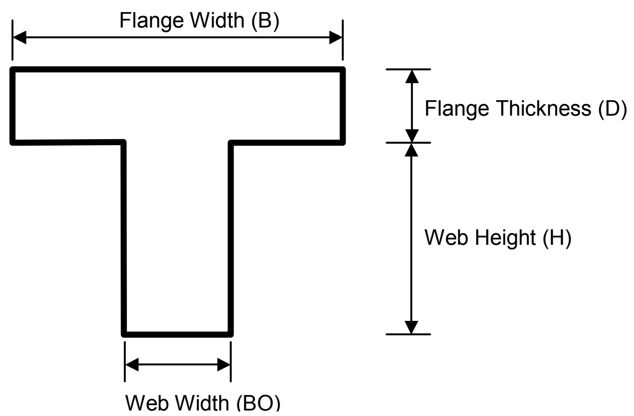
- Rectangular
- Tee-beam
- Tee-beam with haunched flange
- Tee-beam with variable flange thickness
- I-beam with haunches
- I-steel profile
- Channel section (steel)
- L-steel profile (angle)
- Z-steel profile

- T-steel profile
- C-steel profile
- Rectangular hollow section
- Circular hollow section
- Solid Circular Section
- Half I-steel profile
- Back-to-back Channel
- Double L-profile
- Double L-profile-butterfly

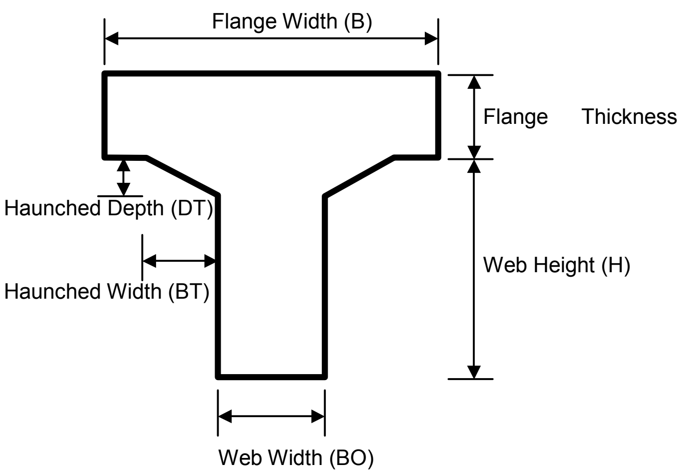
Rectangular cross section



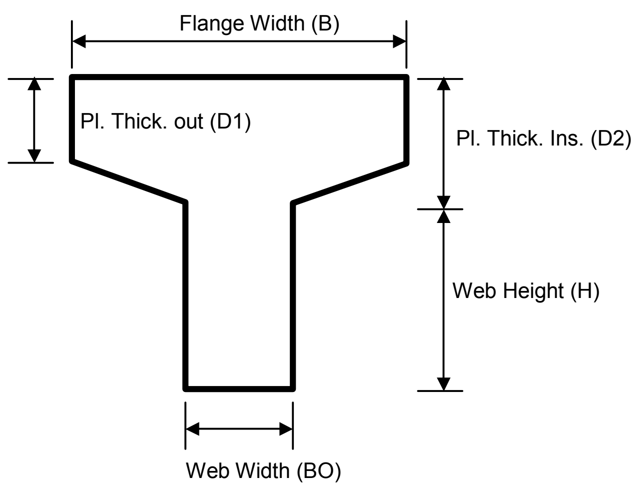
Tee-beam



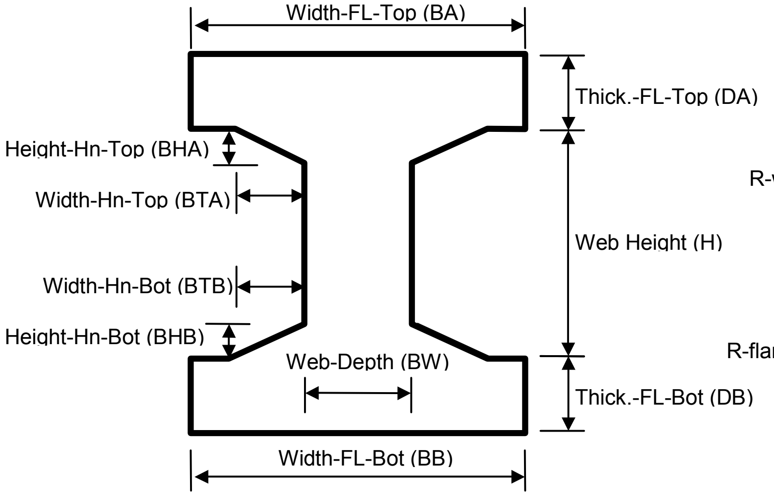
Tee-beam with haunched flange



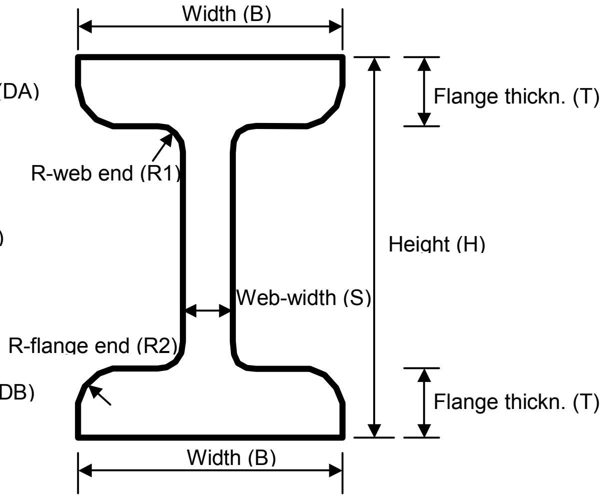
Tee-beam with variable flange thickness



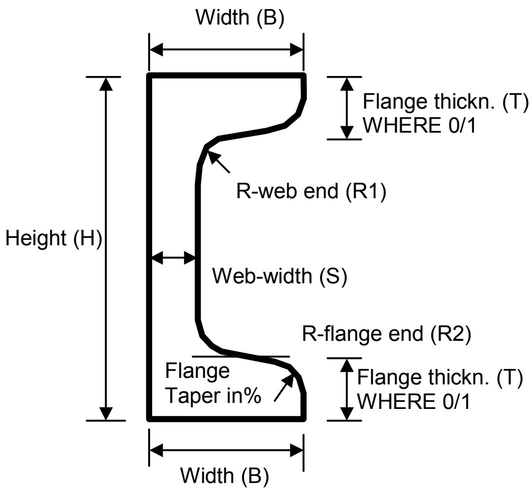
I-beam with haunches



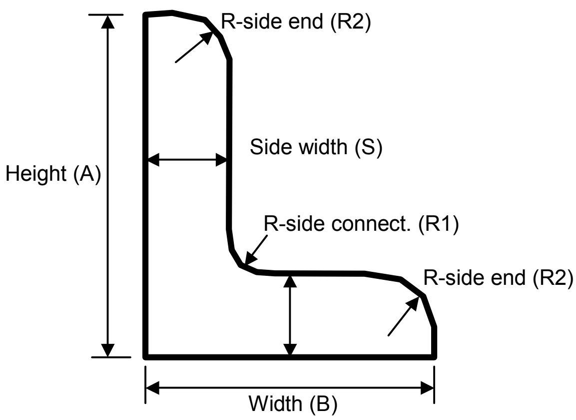
I-steel profile



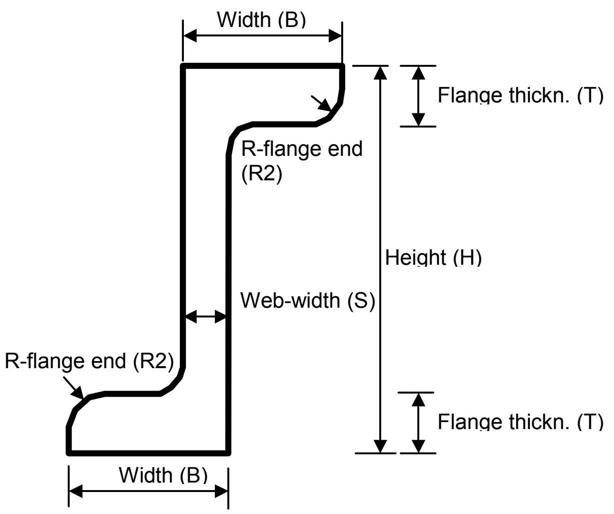
Channel section (steel)



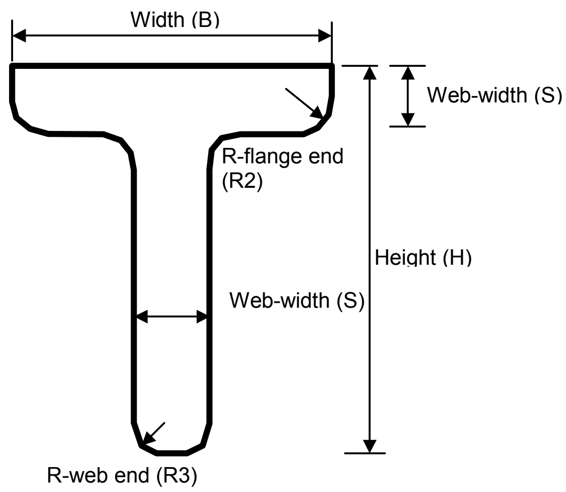
L-steel profile (angle)



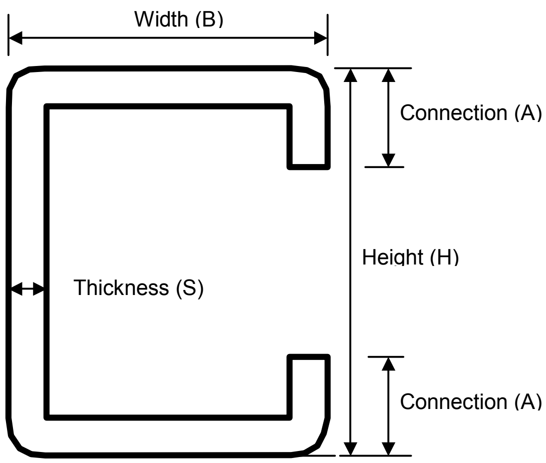
Z-steel profile



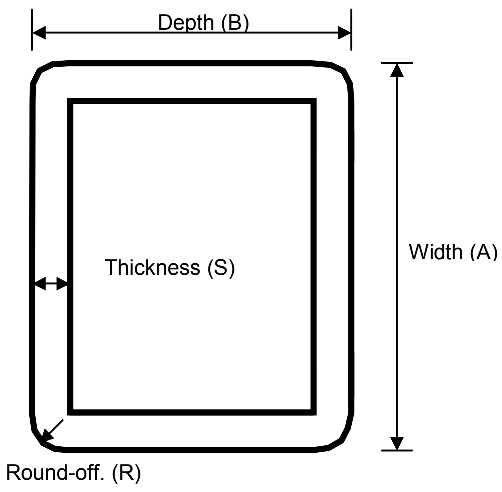
T-steel profile



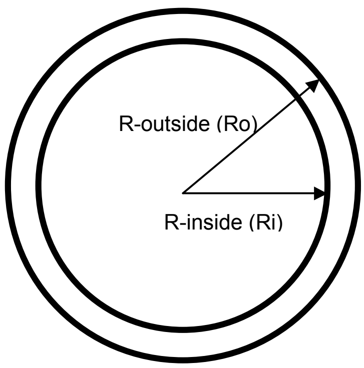
C-steel profile



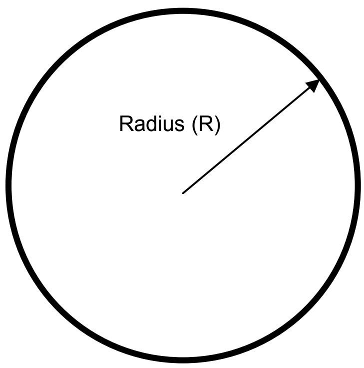
Rectangular hollow section



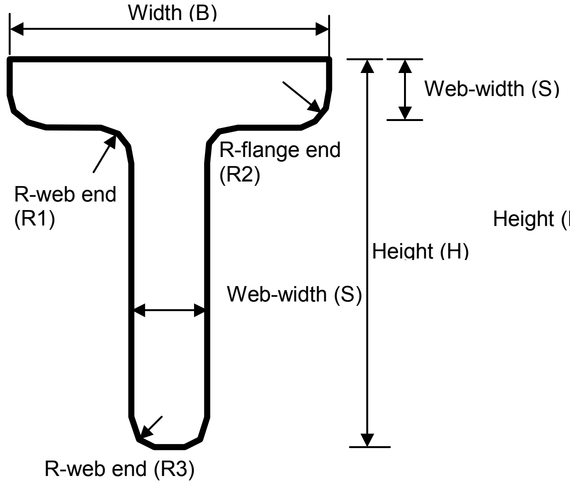
Circular hollow section



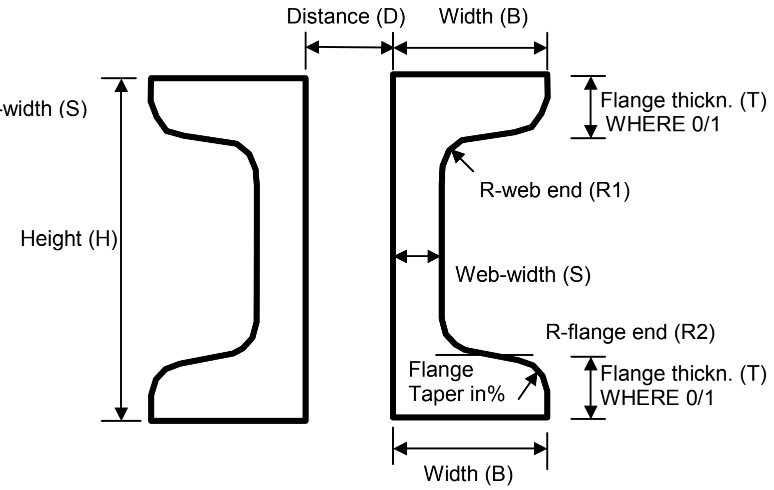
Solid circular section



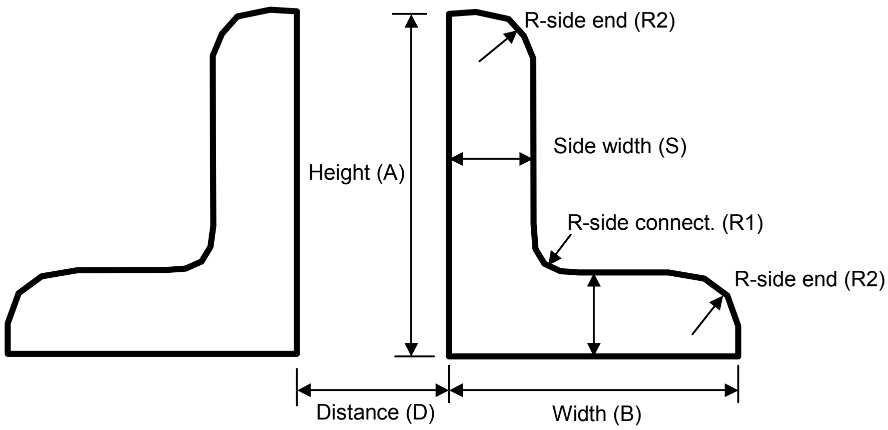
½ I-steel profile



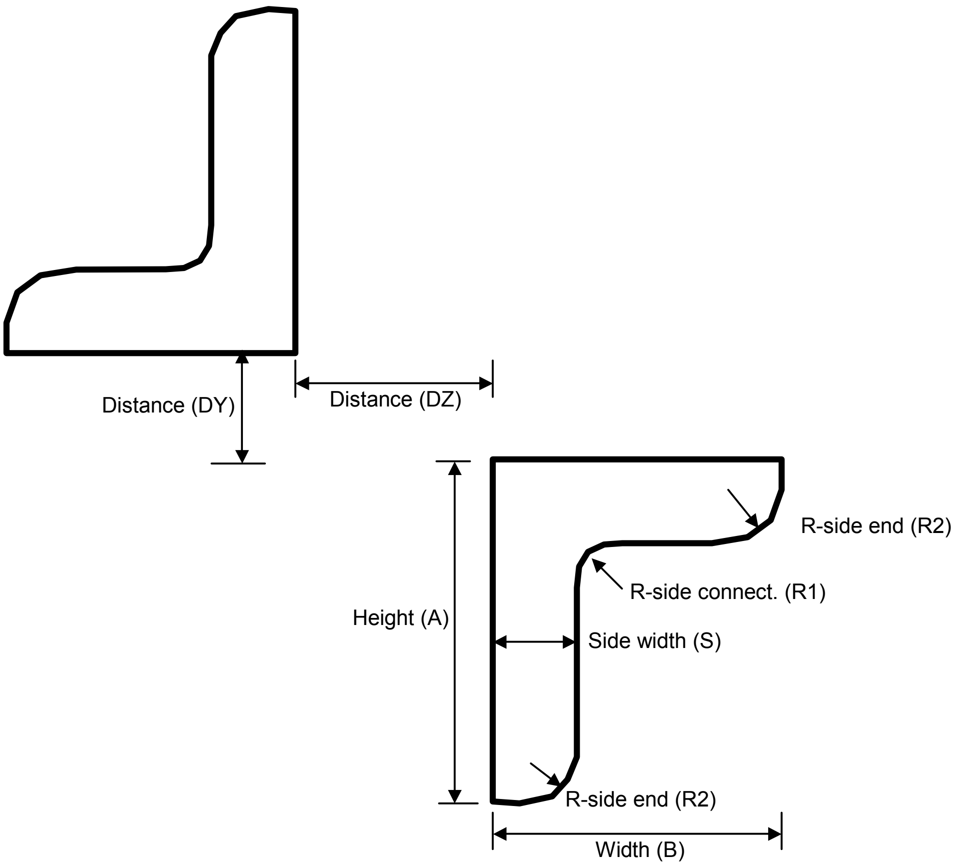
Back-to-back channel



Double L-profile



Double L-profile-butterfly



3.4.4 Section Properties Considered

The following basic geometric properties are used, together with the material properties, to generate the stiffness of the Section. These properties are:

- Ax Cross-sectional area.
- Ay Shear area in the local 'y' direction (bending about the local z-axis).
- Az Shear area in the local 'z' direction (bending about the local y-axis).
- Ix Torsion Moment of Inertia.
- Iy Moment of Inertia about the local 'y' axis.
- Iz Moment of Inertia about the local 'z' axis.
- Iw Warping Moment of Inertia.(*)
- ey Eccentricity e_y of the centre of gravity with respect to the system axis
- ez Eccentricity e_z of the centre of gravity with respect to the system axis
- Beta Deviation angle of the principal inertia axes (from the local axes)
  (calculated using the formula: $tg(2\beta) = \frac{2I_{yz}}{I_z - I_y}$)
- Ey(s) Eccentricity e_{ys} of the shear centre with respect to the system axis
- Ez(s) Eccentricity e_{zs} of the shear centre with respect to the system axis
- $Beta_s$ Deviation angle of the principal shear axes (from the local axes)

Other cross-section values used for loading application or design code checks are:

- y1 Y-below: distance between the gravity centre and the lowermost cross-section point
- y2 Y-above: distance between the gravity centre and the uppermost cross-section point
- z1 Z-left: distance between the gravity centre and the leftmost cross-section point
- z2 Z-right: distance between the gravity centre and the rightmost cross-section point
- U Outside perimeter
- UIN Inside perimeter

Note: *The warping moment of inertia Iw is stored in the database when input, and will be calculated by the program when the cross-section is defined geometrically. It is however not used by the program for the structural analysis, i.e. no warping DOF is considered. Only the torsion moment Ix describing the pure St.Venant torsion is used for computing the torsion stiffness.*

3.4.5 Import Cross-sections

Cross section properties that have been previously prepared can be imported from any directory and file structure including the Project directory and placed inside the RM2000 Program data base for the Project. Files can be imported in binary or ASCII format.

Different section property files containing sets of cross section properties could have been previously prepared by the user (say for another project). The user then has the option to choose between the different available files for the set of section properties for the new Project.

3.4.6 Standard Cross-section Tables

A set of standard structural steel cross sections can be imported into RM2000 for use in the definition of the structural model. Once imported, any of these standard sections can be selected and allocated to certain elements or groups of elements.

The format of these “profile tables” to be imported is an ASCII format and described in detail in the RM7 manual (RM7 is the predecessor program of RM2000). These tables are imported in the function ⇧FILE ⇨RM7.

A model profile table (Austrian or German steel profiles) is available at TDV upon re-quest.

3.4.7 Composite Cross-sections

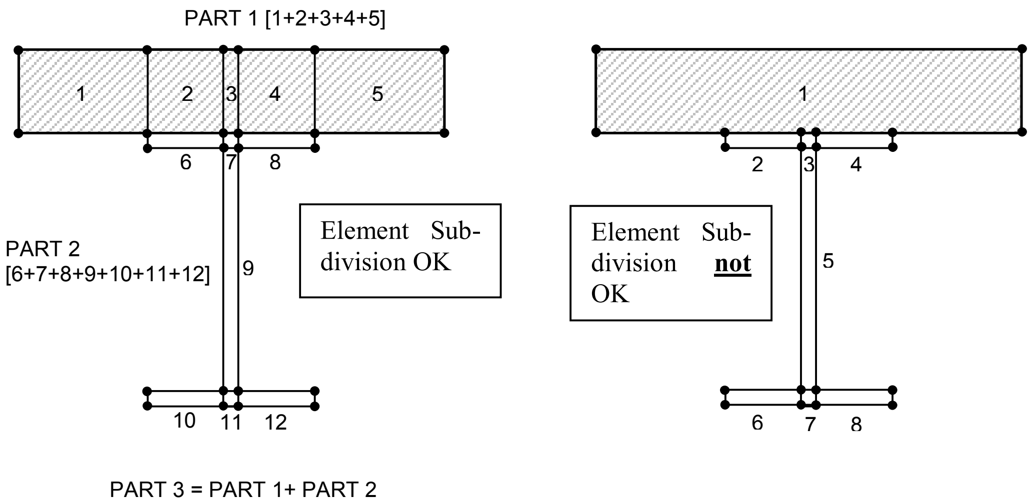
Composite cross sections can be defined and can consist of several different materials.
A composite cross section is:

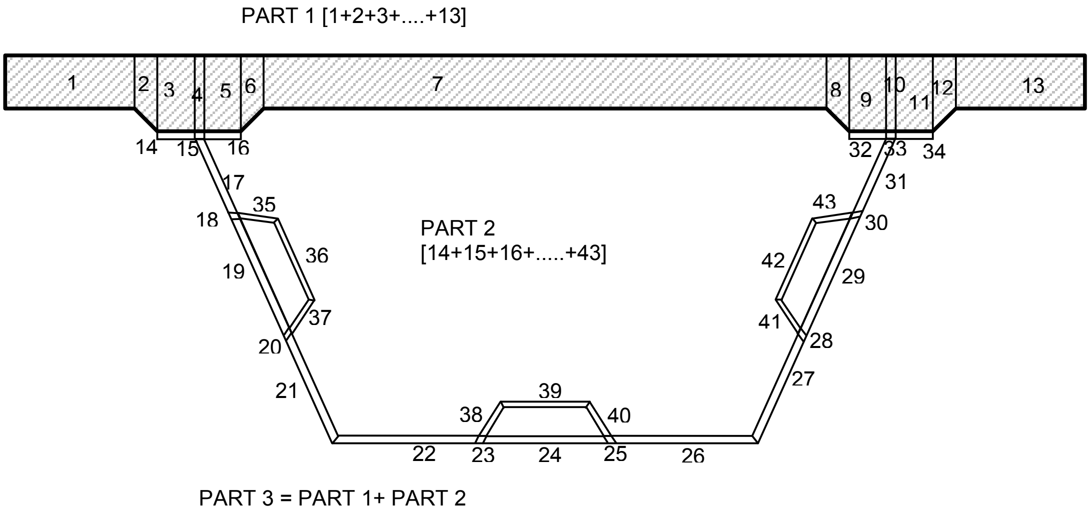
- a section that consists of more than one material type or
- a section that is completed in several steps

A section consisting of only concrete is also considered to be composite when different parts of the section are cast at different times or are made from concrete that has different physical properties.

It is possible to define composite sections directly, but it is recommended that they should be defined via the geometric pre-processor GP2000.

The geometry of a composite cross section needs to be continuous. The subdivision of all parts of a composite cross section must fit together.




PART 1 [1+2+3+....+13]
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
PART 2
[14+15+16+.....+43]
PART 3 = PART 1+ PART 2

3.5 Cross-section Management

3.5.1 Creating and Viewing Cross-sections

A list of all existing cross sections can be viewed under ⇧PROPERTIES ⇨CS ⇩CS. New cross sections can be defined here, existing ones can be modified and/or completed. The Function pad contains the Cross-section table and a graphic presentation of the selected cross-section.

The cross-section table shows the names of the cross-sections, the names of the ASCII files containing the cross section data and a descriptive text for each cross-section. The total cross-section names consist of 2 parts, a preceding character string indicating in general the group of similar cross-sections (e.g. all cross-sections generated from the same cross-section type by varying variable parameters) and a number to distinguish the different cross-sections of the same type.

The graphic presentation details can be governed by using the 'zoom'-functions, or adding or removing information by selecting the corresponding buttons at the top of the graphic window:

- ☑ TxtFact: Text size factor
- ☑ Elem: Cross section Elements are presented
- ☑ Nod Cross section Nodes are presented
- ☑ El-Numb Cross section Elements are labelled
- ☑ Nod-Numb Cross section Nodes are labelled
- ☑ AddPnt: Reference points in the cross-section are presented

New cross-sections are added by selecting the appropriate line in the table and selecting either the 'Insert before' or 'Insert after' button. The opened selection window allows to select one of the macros for standard cross-section shapes or to define a general cross-section. If a macro is selected, the required geometric parameter have to be input. Immediately after the confirmation of the parameter set of the macro, the FE-mesh for the calculation of the cross-section values created and element and node tables for the created cross-section(s) are established in the database.

Note: *The different parameters of the cross-section macros are not stored in the database. Cross-sections, once fully established, may only be modified be modifying the related element or node tables respectively. Another possibility is deleting the whole cross-section and re-defining it by entering the macro data anew.*

The type 'General' allows the user to create a cross section with an arbitrary shape which cannot be described by the geometry parameters of the standard shapes (e.g. a hollow box cross-section must be created in this way). The input for defining general cross-sections in RM2000 is rather complicated (all nodes and elements have to be defined) and not explained in detail. **It is strongly recommended to use the geometric facilities of GP2000 for the generation of general cross sections!**

3.5.2 Cross-section Nodes

The Function ⇧PROPERTIES ⇨CS ⇩NODES shows the table of all existing nodes of the selected cross-section. Existing nodes can be modified, new nodes can be defined. The '+Z' – direction is the horizontal axis inside the cross section (→), The '+Y' – direction is the vertical direction (↑).

3.5.3 Cross-section Elements

The Function ⇧PROPERTIES ⇨CS ⇩ELEM shows the table of all existing elements of the selected cross section. Existing elements can be modified, new elements can be defined.

3.5.4 Cross-section Values

The Function ⇧PROPERTIES ⇨CS ⇩VALUES shows the table of the calculated cross section properties of the selected cross section. The values can not be modified.

Note: *The values are displayed only after having run the calculation with the ⇧RECALC button!*

3.6 Variables

3.6.1 General

Variables are named objects, representing numbers which might vary throughout the analysis process or be dependent on other varying calculation parameters. Variables may also be used for simplification of the input, using short names instead of complicated numbers*, but more often and typically they are used for defining values depending on other parameters. The program will automatically retrieve the variable information from the data bank and evaluate the variable expression when the variable name is referred to as the data information.

* _Note:_ _Variables can actually only be passed to the RM2000 calculation unit in some input situations, where entering variable names is explicitely required (definition of functions and diagrams). Entering Variable names instead of digital numbers in general parameter input fields is not possible._

The evaluation of the variables is done recursively, i.e. it is not necessary to define variables in the right order.

Attention:	**Variable names are not case sensitive. Names written in small letters or mixed letters are equivalent to terms in capital letters. Within this guide variable names are generally written in capital letters for better understanding.**

Besides the **user defined variables** _RM2000_ uses **intrinsic variables** with predefined names. These names are reserved and the user may not use them for user defined variables (e.g. T for the time).

User defined variables may either be **formulae** (mathematical expressions), where a constant is a special case, or **tables** characterized by a set of value pairs and an interpolation rule.

Examples:

- A table can present a diagram for a response spectrum or a diagram relating loads to time (e.g. moving masses or time history).
- A formula can be used to relate tables and/or variables that will be used in the calculation (e.g. user defined creep function).
- Intrinsic variables are setup and/or used by the program internally in the calculation process (e.g. the age of concrete T). They may be used in the variable evaluation process for computing actual values of user defined variables (e.g. creep coefficient $\varphi = \varphi(T)$).

3.6.2 Intrinsic Variables and Functions

3.6.2.1 Intrinsic Variables

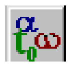
Intrinsic Variables are variables used in RM2000, which may be referenced by the user as independent (abscissa) values or function arguments for computing other (user defined) variables. Some of these variables are variables setup and used in the calculation process (such as the natural frequency OMEGA used in earthquake analyses for evaluating a response spectrum), others are simply input in the RM2000 GUI and stored in the database to be passed to the variable evaluation process when needed.

Attention:	**The values of intrinsic Variables are not transformed to the user defined unit system, when they are passed from the calculation unit of RM2000 to the variable evaluation process. I.e. they are always defined in the default units (kN, m, sec, days), as specified below in detail.**

Table of intrinsic Variables:

t	Age of the material (concrete) of the considered element at the actual time. Used for creep analysis. **[days]**
t0	Age of the material (concrete) of the considered element at the application time of the actual Load Case. Used for creep analysis. **[days]**
ts	Age of the material (concrete) of the considered element when shrinkage theoretically starts (this is not the start of the consideration of shrinkage!). Used for creep analysis. **[days]**
tstart	Absolute time (on the global time axis, i.e. relative to day ‘0’ of the construction schedule) when a certain action takes place. **Actually not used. [days]**
E28	Basic value of the Young’s Modulus used in the calculation. (Usually for concrete the value at an age of 28 days). **[kN/m^2]**
Ax	(Average) cross-section area of the considered element **[m^2]**
Fc28	Design value of compressive strength of the material of the considered element. (Usually for concrete at an age of 28 days). **[kN/m^2]**
CF	Consistency coefficient of the fresh concrete for the considered element. Used for creep analysis. **[-]**
U	The perimeter of the cross section exposed to drying (for the element being considered). Used for creep analysis. **[m]**

UIN	Summation of the internal (hollow box cross section) perimeter(s) of the cross section for the element being considered. Used for creep analysis. **[m]**
RH	Average relative humidity (**in %**) at the construction site for the considered element. Used for creep analysis.
ZF	Cement hardening parameter of the material of the considered element. Used for creep analysis. **[-]**
TMP	Average temperature of the environment of the considered element. Used for creep analysis. **[°C]**
omega	Natural (or angular) frequency for dynamic calculations [omega = **(2*PI)/Period**].
GAMMA	Specific weight of the material of the considered element **[kN/m^3]**.
WCR	Water cement ratio **[-]**
CECO	Cement content in concrete **[Cement weight per concrete volume]**
qlen	Effective length of UDL traffic loading for the considered result point (used in the evaluation of influence lines for defining the line load of load trains as a function of the loaded length **[m]**
RPR	Product of the reinforcement content (A_s/A_c) and the ratio of the moduli of elasticity (E_s/E_c) – equivalent to the ratio of the normal force stiffness of steel and concrete (A_s*E_s/A_c*E_c). Used for creep analysis (Hongkong Standard and British Standard) **[-]**
RelSig	Pre-stressing force utilization level (actual PS-force as a percentage of allowable force (ratio SIG-pr / SIG-allow-pr * 100)). Used for steel relaxation analysis. **[Consideration actually not implemented!]**

All available intrinsic variables can be listed clicking the  button at the right top of the lower table. The listing shows the

- Variable name
- the current value (‘0’ if it is unset, any other value if it is set)
- the unit
- and a brief description

This view can be used for checking the actual value of intrinsic variables throughout the analysis progress. Some values may also be modified to check the influence of the modification on other variables or parameters (e.g. the influence of modified material parameters on the creep and shrinkage coefficients). Attention must be paid to the fact, that such modifications are performed only locally, i.e. the changes are not passed to the database. Values from parameter tables of the database (e.g. material parameters like water-cement ratio, etc.) are not changed in the parameter table. The variables will in a recalculation process again get the original values, unless the parameters have been changed directly in the parameter table.

3.6.2.2 Intrinsic Functions:

- Basic trigonometric functions “cos()”, “sin()”, “tan()”, “acos()”, “asin()”, “atan()”.

Please note that all angles used as arguments of intrinsic functions must be given in radians! (not in the active I/O angle unit!)

- Basic exponential and logarithmic functions “sqr()”, “ln()”, “log()”, “exp()” with (exp(2.5) = $e^{2.5}$).

- Logic functions “abs()”, “min()”, “max”, “hright()”, “hleft()”, “dirac()”, “diract()”.
  abs(a) gives the absolute value of the argument.
  min(a,b) gives the smaller value of the two arguments.
  max(a,b) gives the greater value of the two arguments.
  hright(a,b) = 1 if a>b, else = 0
  hleft(a,b) = 1 if a<b, else = 0
  dirac(a,b,eps) =1 if b-eps<a<b+eps, else =0 (Figure 1a).
  diract(a,b,eps1, eps2) = triangular interpolation (Figure 1b).

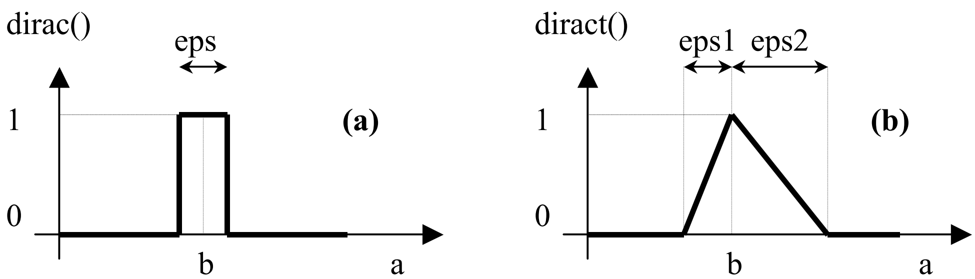


Figure 1. Logical functions dirac() and diract().

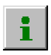
3.6.3 User Defined Variables

The function ⇧PROPERTIES ⇨VARIABLE in the *RM2000* GUI is used to insert new formulae and tables and to view and/or modify existing variables.

Attention:	**The values of user defined Variables are not transformed to the user defined unit system, when they are passed to the calculation unit of RM2000. I.e. they are always assumed to be defined in the program internal standard unit system (kN, m, sec).**

The input pad consists of two lists:
The upper list (variable list) shows all existing user defined variable definitions. New variables (tables/formulae) can be defined using either the 'Insert before' or the 'Insert after' button, existing tables/formulae can be modified and/or deleted using the 'Modify' and/or 'Delete' button. The list contains the following parameters:

- The first column ('VarName') shows the user defined name of the table/formula.
- The second column ('Formula') shows the user defined expression for the formula, or the number of value pairs in case of a table (#5 means 5 value pairs).
- The third column ('Value') shows the actual value of the variable or '###' in case of an undetermined state.
- The fourth column ('Description') shows a user defined descriptive text.

The  button is disabled if the selected variable is a formula and presents graphically the specified diagram if it is a table.

Creating a new variable:
Selecting the 'Insert before' or the 'Insert after' button at the top of the variable list activates the variable input window. The user is asked for the following parameters:

⊙ Formula	Switch, that the variable is a formula
⊙ Table	Switch, that the variable is a table
➢ Name	Name of new variable
➢ Expression	Mathematical expression in case of a 'Formula' (not entered in case of a table)
➢ Description	Descriptive text (max. 80 characters)

The input field for 'expression' is not active, if a table is created. No value can therefore be entered.

Modifying a variable:
Selecting the 'Modify' gives the user the possibility to change the expression or the descriptive text of a defined variable. It is not possible in this function to change the

switch marking the variable as being a formula or as a table. In this case, the variable has to be deleted and entered again as a new variable.

Variable as a table:
The lower list of the input pad is activated if the variable is a table. The variable is in this case defined as the ordinate value of a diagram specified by a set of value pairs and appropriate interpolation rules. The table of value pairs of the selected variable is listed in the lower list of the input pad and can be entered and modified by using the appropriate buttons at the top of the lower list (copy-button is inactive).

The first column (‘VarA’ = abscissa value) shows the value on the horizontal axis of the user defined diagram, the second column (‘VarB’ = ordinate value) shows the related value on the vertical axis. The third column (‘Interpol.’) shows the selected transition rule used for the interpolation between two value pairs.

Note: _The abscissa values of the diagram must be given in monotonic (ascending or descending) order. The transition rule is always related to the interpolation between the actual line and the ensuing line of the list._

The following interpolation rules are available:

- const No interpolation between succeeding values (the ordinate value remains constant until the next abscissa value (diagram with ‘steps’).
- linear Linear interpolation between two value pairs.
- Par T0 Parabolic interpolation (2 second order parabolas from the the start point and from the end point to the centre point, with horizontal tangents at the begin and at the end) (S-shape).
- Par T1 Parabolic (2^{nd} order) interpolation with a horizontal tangent at the begin.
- Par T2 Parabolic (2^{nd} order) interpolation with a horizontal tangent at the end.

4 Structure Modelling

4.1 General Modelling Rules

A complex engineering system is analysed by regarding it as an assembly of elements, the properties of the system being determined from the properties of the individual parts. The junction point between the elements is termed “nodes” or “joints”. Displacement compatibility and stress continuity between the elements must be provided at the nodes.

The location of the joints and elements is critical in determining the accuracy of the structural model. Some of the factors that should be considered when defining the elements (and hence joints) for the structure are:

- The number of elements should be sufficient to describe the geometry of the structure.
- Element boundaries should be located at points, lines, and surfaces of discontinuity:
  - ⇒ Structural boundaries, e.g., corners and edges
  - ⇒ Changes in material properties
  - ⇒ Changes in thickness and other geometric properties
  - ⇒ Support points (Restraints and Springs)

Nodes (or joints) are a fundamental part of every structural model - they perform a variety of functions:

- All elements are connected to the structure (and hence to each other) at the nodes
- The structure is supported at the nodes using Restraints and/or Springs
- All loads and masses applied to the elements are actually transferred to the nodes
- Nodes are the primary locations in the structure at which the displacements are known (the supports) or are to be determined

There are six displacement degrees of freedom at every node - three translatory movements and three rotations. These displacement components are aligned along the local coordinate system of each element.

Note: *No warping DOF’s are actually considered in the program, therefore flexibility terms due to warping effects of the cross-sections cannot be taken into account at the moment.*

The above listed criteria for the subdivision of the structure into elements are principally sufficient for the standard statics of frames, because the stiffness matrices and nodal forces for beam elements are calculated “exactly” in accordance with the deformation method. An additional subdivision to get a better approximation - as usually required for Finite Element procedures – is not necessary in this case. This is true even if primary results (displacements and internal forces) in intermediate points between start and end are required.

However, there are some functions, which use approximations on the element level or are available only for nodal points. An appropriate subdivision of the structure into small elements is required in these cases, to get the required answers. These functions are:

- Varying cross-sections
- Dynamical analyses

Fixing the numbering and element scheme is therefore the most important primary task in every modelling process. It has to be done very thoroughly, considering all requirements regarding accuracy as well as amount and density of results. Practical modelling hints for typical bridge structures will be found later in this guide (Super structure, sub structure, …).

4.2 Definition of Structural Data

4.2.1 Data Input

4.2.1.1 General

The basic structural database objects are nodes and elements, the nodes being described by their coordinates and some additional properties like restraints or nodal masses. The elements are basically described by their start and end nodes, and element properties like material, cross-sections, connection details, etc.

The nodes are numbered using a positive numbering series. Their position is defined by their coordinates in the global coordinate system. They may have or may not have additional attributes in the node table. The nodal coordinates are specified in the function ⇧STRUCTURE ⇨NODE ⇩NODES.

The elements are also numbered with positive numbers. They are basically defined by their (commonly two) nodes, the start point and the end point. Element type, Cross-section and material are necessary attributes, other attributes stored in the element table are arbitrary. The basic element data are specified in the function ⇧STRUCTURE ⇨ELEMENTS ⇩ELEM.

The following element types are available at the moment:

- Beam elements
- Cable elements (Suspension or stay cables)
  Note: Pre-stressing cables are called tendons and are not structural elements!
- Spring elements
- Stiffness matrices
- Flexibility matrices

The special case, where one of the two node numbers is 0 is used when one end of the element is connected to a node, whose coordinates are not specified and whose DOF's are all rigidly restrained. In this case the length and the direction of the element has to be specified by the user, because they cannot be determined by the program from the nodal coordinates. This is performed in the function ⇧STRUCTURE ⇨ELEMENT ⇩BETA.

There are 3 ways to input the structural data in the database:

- By the function ⇧STRUCTURE in the *RM2000* GUI
- By importing them from ASCII or binary external files
- By generating them in the graphic pre-processor *GP2000*

4.2.1.2 Direct Definition in the *RM2000* GUI

There are primarily 3 database objects for describing the structural model:

- Nodes
- Elements
- Tendons

Nodal coordinates are entered and changed in the function ⇧STRUCTURE ⇨NODE. Element data is defined in the function ⇧STRUCTURE ⇨ELEMENT. Tendons are not structural elements but a separate object category. They are defined in ⇧STRUCTURE ⇨TENDON and related to the structural elements. A detailed description of the definition of pre-stressing tendons is found in chap. 5, Pre-stressing.

4.2.1.3 Import of the Structure Data

Complete Structural Systems that have been previously prepared can be imported from any directory and file structure including the Project directory and placed inside the *RM2000* Program data base for the Project. Files can be imported in binary or ASCII format (see chap. 1.1.2)

4.2.1.4 Structure Modelling in *GP2000*

The geometric pre-processor *GP2000* offers the possibility for creating the geometric data by using interactive graphic tools. *GP2000* is fully connected to *RM2000* and writes node and element tables directly to the *RM2000* database. Hence, it is also possible to change or supplement existing data without extra import/export procedures.

4.2.2 Model Parameters – General Remarks

The deflection of the structural model is governed by the displacements of the joints. Every joint of the structural model may have up to six displacement components:

- The joint may translate along its three local axes.
- The joint may rotate about its three local axes.

These six displacement components are known as the **degrees of freedom** of the joint.

Each degree of freedom in the structural model must be one of the following types:

- Active - the displacement is computed during the analysis
- Restrained - the displacement is specified, and the corresponding reaction is computed during the analysis
- Constrained - the displacement is determined from the displacements at other degrees of freedom
- Null - the displacement does not affect the structure and is ignored by the analysis
- Unavailable - the displacement has been explicitly excluded from the analysis

The joint releases are specified in *RM2000* in the form of Element end releases and may be defined in the global or the local coordinate system directions.

4.2.3 Global Degrees of Freedom (DOF's)

4.2.3.1 Available and Unavailable Degrees of Freedom

The set of global degrees of freedom that are available to every joint in the structural model may be explicitly specified. By default, all six degrees of freedom are available to every joint. This default should generally be used for all three-dimensional structures.

It is possible to restrict the available degrees of freedom – this has an advantage for certain 2-.D structures such as a plane truss in the X-Y plane. A plane truss in the X-Y plane only needs VX and VY; a plane frame in the X-Y plane only needs VX, VY, and RZ.

The degrees of freedom that are not specified as being available are called **unavailable** degrees of freedom. Any stiffness, loads, mass, Restraints, or Constraints that are applied to the unavailable degrees of freedom are ignored by the analysis.

4.2.3.2 Restrained Degrees of Freedom

If the displacement of a joint along any one of its available degrees of freedom is specified, such as at a support point, that degree of freedom is restrained. The known value of the displacement may be zero or non-zero, and may be different in different Load Cases. The force along the restrained degree of freedom that is required to impose the specified restraint displacement is called the reaction, and is determined by the analysis.

Unavailable degrees of freedom are essentially restrained. However, they are excluded from the analysis and no reactions are computed, even if they are non-zero.

4.2.3.3 Constrained Degrees of Freedom

Any joint that is part of a Constraint may have one or more of its available degrees of freedom constrained
A degree of freedom ***can not*** be both constrained and restrained.

4.2.3.4 Null Degrees of Freedom

The available degrees of freedom that are not restrained, constrained, or active, are called the null degrees of freedom. Because they have no load or stiffness, their displacements and reactions are zero, and they have no effect on the rest of the structure. The program automatically excludes them from the analysis.

4.2.4 Nodal points

All defined nodal points are stored in the database in the nodal point table. Every node is described by it's number and it's position specified by the coordinates in the global coordinate system. Additional node attributes my be support conditions and/or a nodal mass.

The definition of the nodal points is done in the function ⇧STRUCTURE ⇨NODE. The node table appears on the screen after invoking this function, showing the numbers of the already existing nodes, their coordinates and information labels about support conditions and activation status.

New nodes are created by using the 'Insert before' or 'Insert after' functions. The input pad allows to define a whole series of nodes by using a „from, to, step" specification if the distances between the nodes are constant. The coordinates of the first point of the series and the distance vector from one node to the next are in this case entered. A modification of the node parameters may be done by using the function 'Info'.

Sopport conditions are assigned to the nodes in the function ⇧STRUCTURE ⇨NODE ⇩SUPPORT (or ⇧STRUCTURE ⇨NODE ⇩ECC and ⇧STRUCTURE ⇨NODE ⇩BETA respectively) (Details see chap. 4.5). Nodal mass are defined ⇧STRUCTURE ⇨NODE ⇩MASS (Details see chap. 6 and chap. 9).

4.2.5 Elements

4.2.5.1 Overview

The definition of elements is done in the function ⇧SYSTEM ⇨ELEMENT. By calling this function a part of the element table is shown, containing the numbers of all defined elements, their element type, the numbers of the start- and end-nodes, the status (active/inactive) and the element subdivision for the result presentation.

All defined elements are basically specified by the assigned nodal points. The nodes are the start- and end points of the elastic length of the elements if no eccentric connections are specified (see chap. 4.6). The connection line between start node and end node of an element is called “system axis”. If no eccentricities exist, this axis is identical with the “centre line” of the element, being the straight connection between the centres of gravity of the start- and end-cross-sections.

The system axis and the centre line (possibly different due to eccentric connections) describe the element geometry (length, local axes). The rules for the establishment of the local axes are described in detail in chap. 2.4.

There is principally also the possibility to let one of the two node numbers undefined. This is done by assigning the node no. 0 to this element node. This node does not exist in the node table, and it has no degrees of freedom and no coordinates. The system axis and element centre line are not fully specified in such a case by the node assignment. Additional information is required (length, axis direction) to describe fully the system geometry. This is done in the function ⇧SYSTEM ⇨ELEMENT ⇩BETA.

New elements may be inserted in the table by using the ‘Insert before’ or ‘Insert after’ buttons. Series of elements may be easily defined in the case of a regular numbering. This is done by specifying the element numbers using a ‘From, to, step’-statement and entering the start- and end-node of the first element together with the node number increments for these nodes while proceeding from one element of the series to the next. A modification of the element parameters may also be done for single elements by using the function button ‘Info’.

Different further element parameters are necessary to be input, depending on the type of the element.

4.2.5.2 Element Type „Beam“ – Beam Elements

Apart from the system geometry, beam elements are further described by the cross-section(s) and the assigned material. The following requirements are necessary to allow to calculate the stiffness matrix used for the analysis:

- The (elastic) length of the element must be greater than zero (l > 0)
- The cross-sections must have been assigned (⇧STRUCTURE ⇨ELEMENT ⇩CS)
- Material properties must have been assigned (⇧STRUCTURE ⇨ELEMENT ⇩MAT).

The local coordinate system is automatically built, based on the direction of the element centre line. A further rotation of the principal inertia planes with respect to the automatically built system is done by specifying an angle β in the function ⇧SYSTEM ⇨ELEMENT ⇩BETA. Changing other geometric parameters being fully defined by the node coordinates (length, ...) is refused in the modification pad.
A length (>0) has absolutely to be specified if the node 0 is assigned, in order to have a complete element data set. The direction of the centre line is assumed to be the global x-direction, if the angles α_1 and α_2 respectively have not been specified in ⇧SYSTEM ⇨ELEMENT ⇩BETA to be different from 0.

4.2.5.3 Element Type „Cable“ – Cable Elements

Cable elements are described in the same way than beam elements, by the geometry (length, direction), by the cross-section and by the material. They take however only normal forces and no shear forces or moments. Therefore the definition of the cross-section area is sufficiently describes the cross-section, shear areas or moments of inertia need not to be entered, and the cross-section geometry needs not to be specified explicitly and stored under a cross-section name.

Note: *Cable elements are provided for modelling external cables like stay cables or suspension cables of bridges. Pre-stressing cables are called tendons and are no structural elements. They are defined in ⇧STRUCTURE ⇨TENDON.*

Nodes, only connected to cable elements, have only displacement DOF’s and no rotation DOF’s. Only normal forces and no shear forces or moments are transmitted at the connection points of cable elements with beam elements.

4.2.5.4 Spring Elements (Overview)

Sub-types:

- Spring Static spring
- SFrict Friction spring
- SCont Contact spring
- SHinge Hinge spring
- SBilin Bi-linear spring

The stiffness of spring elements is generally described by spring constants, defining a linear relationship between displacement and rotation differences on one side, and forces or moments respectively on the other side. The different sub-types differ with respect to the validity of this linear relation under different loading conditions. Some sub-types define different relations in different cases.

The spring constants (respectively parameters describing the stiffness) are stored in the element table as „Cross-section values“ (⇧STRUCTURE ⇨ELEMENT ⇩CS), where the displacement spring constants replace the cross-section areas and the rotation spring constants replace the moments of inertia.

No material is assigned to spring elements, the places of the material parameters in the element table are not occupied.

The spring constants implicitly contain the element length, thus the element length stored in the element table is not used for the calculation of the stiffness matrix. Theoretically, spring elements must not connect 2 nodes with different coordinates, i.e. the length of the element must be zero. Otherwise stiffness in not correctly modelled, because the relationship between acting forces and the moments resulting from the lever distance between the 2 nodes is not considered.

Spring elements connecting 2 differently located points therefore generally require the connection point to be specified (the position of the point, where the displacement difference occurs). „Eccentric connections“ (see chap. 4.2.7) from this point to the start and end nodes respectively have to be specified. These rigid lever arms transmit the resulting moments, and the elastic element length again becomes zero. The local axes may however not be automatically determined for elements with a zero element length. Therefore they must be defined by the user in the function ⇧STRUCTURE ⇨ELEMENT ⇩BETA, if they do not coincide with the global axes.

Generally it makes sense to choose the local axes system of spring elements such, that the local x-direction describes the main bearing direction (e.g. the vertical direction if the spring element models a bearing, the horizontal direction if it models the bedding of a pile, etc.). Such an alignment of the local axes improves the result interpretation, because of the notation in the result listings. Using the common notation for beam elements, the internal force in the local x-direction is always called „normal force“ and the forces in y- and z-direction respectively are called shear forces. Some sub-types also differently handle the parameter definition for the normal force direction than for the other local directions.

Note: *Spring elements defined in GP2000 via “connection points” get automatically the appropriate eccentric connections. The direction of the spring elements is automatically rotated the vertical direction, yielding an axes system with the local x-direction showing into the global Y-direction and the local y-direction into the negative X_G-direction. The local z-direction remains unchanged (in Z_G-direction).*

In order to avoid an a priori exclusion of special cases, where the definition of a length for a spring element might be allowed and meaningful, or where special effects should be described, the program does not check whether the length of a spring element is zero. If the coordinates of the start and end nodes are different and no eccentric connection yielding a zero element length are defined, the length of the spring element will be calculated in the same manner than for beam elements, but this length is not used for the calculation of the stiffness matrix. The local axes are however not automatically established and must be defined in ⇧STRUCTURE ⇨ELEMENT ⇩BETA.

Note: *Spring elements connecting 2 nodes with different coordinates are for instance allowed, if they represent springs carrying only loads in the direction of the connection line (local x-direction). No moments to be transmitted will arise in this case.*

4.2.5.5 Static Springs (Linear Displacement-Force Relation)

The stiffness of such spring elements is specified by spring constants describing the linear relation between displacement and rotation differences on one side, and forces or moments respectively on the other side. Spring constants may also be zero, this describes a disconnection of the considered degree of freedom (hinge).

Modelling considerably rigid connections with spring elements requires spring constants to be specified, which are by some orders of magnitude higher than the stiffness coefficients of the other elements. Choosing very high spring constants (e.g. 1.E20) may cause numerical problems in the solution of the equation system. This is to be considered especially, when the local axes do not coincide with the global axes, i.e. the stiffness parameters have to be transformed into e skew system.

Note: *Values of 1.E10 kN/m for displacement springs and 1.E10 kNm/rad for rotation springs are suitable for modelling considerable rigid springs in common civil engineering structures.*

Parameter set for the description of static springs:

- CX Spring constant for the displacement in local x-direction
- CY Spring constant for the displacement in local y-direction
- CZ Spring constant for the displacement in local z-direction
- CMX Spring constant for the rotation around the local x-axis
- CMY Spring constant for the rotation around the local y-axis
- CMZ Spring constant for the rotation around the local z-axis

4.2.5.6 Friction Elements

Parameter set for the description of friction elements:

- Ax Cross-section area
- ni-y Friction coefficient for the local y-direction
- ni-z Friction coefficient for the local z-direction
- Ix Moment of inertia um die locale x-axis
- Iy Moment of inertia around the local y-axis
- Iz Moment of inertia around the local z-axis

4.2.5.7 Contact Elements

These elements are described by the specification of displacement-force diagrams. These diagrams must be specified as variables (tables) in the function ⇧PROPERTIES ⇨VARIABLE (see chap. 3.6). The independent variable (variable 1 = abscissa) is the displacement or rotation value, the dependent variable (variable 2 = ordinate value) the related force or moment respectively. The Variables used for the definition of this diagram may only be tables, no mathematical expressions (formulas), because no predefined internal variable name is reserved for the considered deformation value.

The name of the variable (table) has to be assigned to the related cross-section value in the element table. DOF's without assignment of a Variable name remain disconnected. The assigned diagrams are evaluated in the course of the analysis considering the actual deformation values for computing the actual spring constants or stiffness matrix coefficients respectively.

Attention:	**The diagrams must be specified in terms of the internal calculation units m, kN, rad, kNm, because no units are assigned to user defined Variables. The values are taken for the determination of the internally used stiffness parameters without any adaptation to user defined units.**

Parameter set for the description of contact elements:

- Form-x Displacement-force diagram for the local x-direction
- Form-y Displacements-force diagram for the local y-direction
- Form-z Displacement-force diagram for the local z-direction
- Form-rx Diagram for the rotation around the local x-axis
- Form-ry Diagram for the rotation around the local y-Axis
- Form-rz Diagram for the rotation around the local z-Axis

4.2.5.8 Hinge Elements

Parameter set for the description of hinge elements:

- TypBeg Cross-section type at the element start
- TypEnd Cross-section type at the element end
- EccTyp System Eccentricity type (see chap. 3.4.3)
- CX Spring constant for the displacement in local x-direction
- CY Spring constant for the displacement in local y-direction
- CZ Spring constant for the displacement in local z-direction
- CMX Spring constant for the rotation around the local x-axis
- CMY Spring constant for the rotation around the local y-axis
- CMZ Spring constant for the rotation around the local z-axis

4.2.5.9 Bi-linear Spring Elements

Bi-linear spring elements are defined similarly to normal static springs, but 2 sets of spring constants exist, valid for different loading conditions. The bi-linear spring element of RM2000 is restricted to the special case, where the normal force stiffness (stiffness in x-direction) is constant in all cases, and only for the shear force and rotation spring constants 2 different values are considered, dependent on a normal force limit.

Note: _The simulation of tension members, where the normal force stiffness varies considerably between the tension and compression cases, can not be modelled with bi-linear springs. Contact elements have to be used in this case (see chap. 4.2.5.7)._

Input parameters for describing the element stiffness are the basic spring constants for the unloaded state (or the state, where the normal force limit is not exceeded), the normal force limit, and the spring constants for the loaded state for shear in the local y and z-directions as well as for the rotations.

Additionally, limit shear forces are input, specifying the maximum shear forces which can be beard by the spring element. Plastic material flow occurs if the shear forces exceed the limit values (additional deformations without increase of force). The out-of-balance force is decremented in the iteration process until the limit force is reached. No limit is defined for the bending moments, the linear relation is valid without limit.

The bi-linear springs have been developed for the simulation of connections between the superstructure and the track (respectively rails) of of railway bridges. This kind of non-linear springs for modelling this connection was proposed in the „Design Specifications“ of the Taiwan High Speed Rail Corporation. The 2 different stiffness parameter set are to be used for unloaded track and loaded track respectively.

Attention must be paid to the fact, that non-linear calculations principally require working with total loading states to give correct results. If, as it is very common, differential states are investigated, the initial normal force in the spring element has to be specified in order to allow a correct comparison with the limit force.

4.2.5.10 User Defined Stiffness Matrix

In some cases it is useful and desired to have the possibility of directly defining the stiffness matrix of some elements. A typical case is the analysis of a bridge, where the foundations of the different piers are complex structures consisting of several piles below a foundation plate. For efficiency reasons it is often desired to model this whole foundation structure by one single element.

The foundation structure is there often modelled as a separate structural system (either with RM2000 or with an other FE-program) and loaded with unity loads applied at the point, where the pier is connected. The results of these unity load cases allow a stiffness matrix to be determined, which may be used in the analysis of the total bridge. This yields considerable savings in storage space requirements and computation time, considering that a huge amount of load cases (traffic load) have often to be investigated in bridge analyses.

The input of the stiffness matrices is done in the GUI by inserting the elements of the type „stiff“ into the element table and then entering the components of the stiffness matrix. This is done by selecting the element and pressing the „Modify“-button. This opens an input pad where the considered element series may be specified an the button „Stiff“ is offered, to open an input pad for entering the matrix components.

The total stiffness matrix is a 12x12 matrix, thus containing 144 values, but always being symmetric. The input is performed by entering 4 partial matrices (6x6). For the partial matrices K11 and K22 in the diagonal of the total matrix only the values above the diagonal can be entered. The matrices K12 and K21 may completely be entered, but the non-defined matrix is automatically built by mirroring the other specified matrix.

Despite all functions to ease the input, the definition of stiffness matrices in the GUI is a toilsome process. Therefore, stiffness matrices are preferably prepared as ASCII import files or as TCL files, and imported in the function ⇧FILE ⇨IMPORT.

4.2.5.11 User Defined Flexibility Matrices

Flexibility matrices are the inverse matrices of the stiffness matrices. They may in many cases be easier determined and input than the stiffness matrices themselves. They are entered analogously to the stiffness matrices and are inverted in the program to build the stiffness matrices required for the analysis.

4.2.5.12 Damping Elements (for Dynamic Analyses)

Damping elements are only used in dynamic analyses. They are not considered if they are active in a static computation action. Therefore it is possible to perform static and dynamic computation actions in the same construction stage without changing the element activation.

Sub-types:
- VDamp   Viscous damping
- SDamp   Damping springs

4.2.6 Boundary Conditions

Each nodal point of the system may be assigned a rigid or an elastic support condition.

4.2.6.1 Nodal Point Boundaries

Nodal point boundaries are defined in the function ⇧STRUCTURE ⇨NODE ⇩SUPP. Any boundary point may be specified as an elastic support with spring constants assigned to specify the stiffness of the support.

An absolutely rigid support where the appropriate DOF’s are eliminated from the equation system is not provided in *RM2000*. For simulating a rigid support the user has to specify suitable “high” spring constant values. Values of **1.0E9 to 1.0E12** kN/m or kNm/rad have been found to be reasonable values for simulating a rigid support in usual civil engineering structures when working in this unit system.

The usage of higher values is not recommended, because such stiffness values can cause numerical problems in the solution process. This is especially the case when the spring tensor is transformed into skew directions.

The translatory and rotational spring constants defining the support are per default in the Global Coordinate directions. They may however be transformed to a local system by assigning a set of angles α_1, α_2, and β to the node in the function ⇧STRUCTURE ⇨NODE ⇩BETA. The local coordinate system of the spring will then be generated from these angles in the same way as the local system of beams (see Fig. 2.1).

Another transformation of the spring tensor is possible with respect to an eccentric position of the spring. An eccentric position of the support spring is defined in the function ⇧STRUCTURE ⇨NODE ⇩ECC.

Note: *The data entered in ⇧STRUCTURE ⇨NODE ⇩BETA and ⇧STRUCTURE ⇨NODE ⇩ECC are only attached to the support springs, but not to the nodal masses defined in the same function. Nodal masses are always specified concentrically and the moments of inertia are always related to the global coordinate system.*

Nodal point boundaries are displayed in a structural plot with a small symbol superimposed over the nodal point, indicating the set of releases at that node.

4.2.6.2 Spring Element Boundaries

Spring element boundaries are almost the same as nodal point boundaries. The only difference is that they are treated as elements which may be set active or inactive and which may be specified in selection filters for output lists and graphics.

The boundary points may be specified as spring elements that are connected to NODE 0 at one end. The translatory and rotational spring constants defining the spring element are in the local coordinate directions.

The spring element does not have any real dimension but it can be given a nominal length dimension so that it can be plotted and easily seen. The spring element also has the advantage that it can be orientated in any direction and so the restraints can directly represent the on-site condition.

Note: *The spring elements can also be used as a connection between two parts of the structure (e.g. between a supporting column and a main girder of a bridge, or between a machine housing and an axis of the machine. The elastic constants of such supports are also defined by the six spring constants and the three angles α_1, α_2, β for the local coordinate system.*

Eccentricities can be defined for the element beginning and end in the same way as than for beam elements (⇧STRUCTURE ⇨ELEMENT ⇩ECC).

If all degrees of freedom of a point are to be fixed, this condition may also be achieved by assigning this point to node number 0. These degrees of freedom are then not considered in the global system of equations. Length and angles α_1, α_2, β to determine the local coordinate system must be assigned to all elements connected to node number 0, as these nodes do not have coordinates.

4.2.6.3 Contact Element Boundaries

The contact element is similar to the spring element in its application except that it has very different properties:

The boundary points may also be specified as contact elements that are connected to NODE 0 at one end.
The translatory and rotational spring constants defining the contact element are in the local coordinate directions.

The contact element does not have any real dimension but it can be given a nominal length dimension so that it can be plotted and easily seen. The contact element also has the advantage that it can be orientated in any direction and so the restraints can directly represent the on-site condition.

Note: *Contact elements can also be used as a connection between two parts of the structure (e.g. between a supporting column and a main girder of a bridge, or between a machine housing and an axis of the machine.)*

4.2.7 Eccentric Connections

4.2.7.1 Definitions

All the elements are assumed stiffly connected to their associated nodal points at the element ends. Actual structural members do, however, have finite cross-sectional dimensions and when two elements, such as a beam and column, are connected at a joint the two sections overlap. In many structures the length of the overlap can be a significant fraction of the total length of a connecting element. Refer to Figure below for example.

An eccentric connection introduces a rigid connection between a nodal point and an element beginning or end. The three components of the eccentric connection must be defined for the element beginning and end. The direction of the eccentricity vector is **from the (elastic) element end to the node**, the components of the eccentricities are positive, when the components of this direction vector are in the positive axis directions. The sign convention is shown in the figure below.

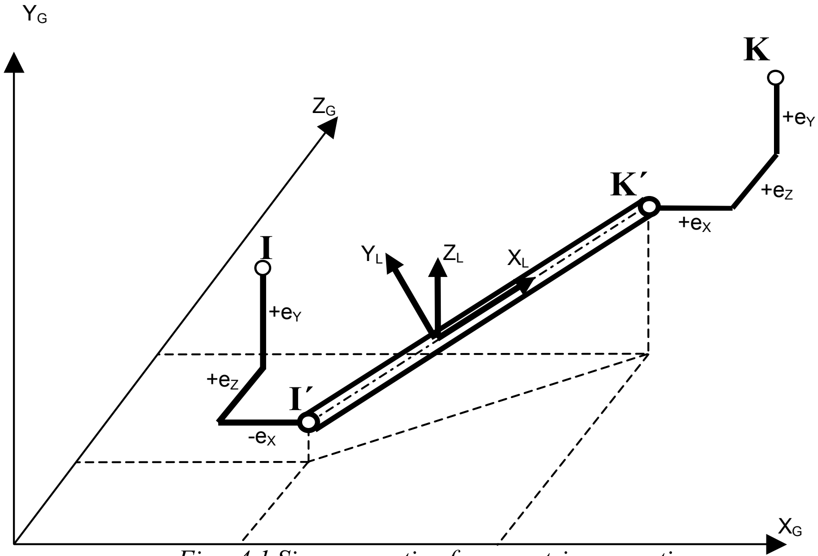


Fig.: 4.1 Sign convention for eccentric connection

The total effective eccentricity vectors consists of 2 parts, the

- **System-Eccentricity** and the
- **Cross-section-Eccentricity**

The first part, in RM2000 called „System-Eccentricity“, must be entered directly by the user in ⇧STRUCTURE ⇨ELEMENT ⇩ECC as a vector in the global coordinate directions. The „Cross-section-Eccentricity“ is related to a reference point in the cross-section and defined in ⇧STRUCTURE ⇨ELEMENT ⇩CS.

Note: *The "System-Eccentricity" can only be entered in terms of direction vectors in the global coordinate system, because the final element axis is not known in the system modelling process before the eccentricities are defined.*

4.2.7.2 Types of Eccentricities

The separation of the total eccentricity vector into the 2 parts described above (the „System-Eccentricity“ and the "Cross-section-Eccentricity") is related to the 2 essentially different types of eccentric connections that may occur in structural models:

a) Connection of an element to a perpendicular series of beams (T-connection)
b) Connection of beam centroids to a "system line" (straight connection line between start and end node of an element), if this system line is different to the element axis.

Connections (a) are directly specified by the user as „System-Eccentricities“ in ⇧STRUCTURE ⇨ELEMENT ⇩ECC. Connections (b) may also be directly specified, but due to the advantageous cross-section related definition possibility they are mostly defined as „Cross-section-Eccentricities“. Both types may be combined, the total eccentricity vector is then the resultant of the 2 parts.

figure to be included

4.2.7.2.1 System-Eccentricities

Basically in structural beam models, all elements are directly connected at their ends to the assigned nodes. Real structures however often have cross-sectional dimensions that must be considered in the geometry definition of the model. If 2 elements are arranged perpendicular to each other (e.g. superstructure and piers of a bridge, flanges and diagonals of a framework girder) generally a region exists at the joint where the sections overlap.

The length of such an overlapping region may be considerable and cause inaccuracies and wrong results. The connection point to the node is in such a case not the position where result values are required. Results are required at the connection face to the cross-section surface of the crossing element series.

The overlapping region may therefore be defined as an „eccentric connection“. The definition of these eccentric connections is done using ⇧STRUCTURE ⇨ELEMENT ⇩ECC. The eccentricities must always be specified as **vectors in the global coordinate directions from the centroid** (or reference point respectively) **to the node**.

4.2.7.2.2 Cross-section-Eccentricities

Another type of eccentricity occurs when the centroid of the element cross-section is eccentric to the connection line between the start and end node (in RM2000 called system line or system axis), i.e. the element axis (described by the centroids of the cross-sections) and the system line do not coincide. Such an eccentricity definition is absolutely necessary in the case of sudden cross-section changes.

Eccentric connections are also often used to ease the definition of the system geometry (e.g. in the case of a varying depth superstructure), where the position of the top of the cross-section is known but the position of the centres of gravity of the different cross-sections is not known. The nodes are in this case arranged along a known reference line, and all cross-sections are defined to be eccentric to this line. The centroid of the cross-sections related to the top surface (which is usually known), is most often used.

The dimensions of this eccentricity need not be specified by the user, but are automatically determined by the program using the “CS reference point”. The **“CS reference point” is the origin for defining the cross-section** in GP2000 (or as general cross-section in RM2000). RM2000 has a switch for placing this reference point for the special internal cross-section types (rectangular, T-beam, etc.) specified by CS macros in ⇧PROPERTIES ⇨CS ⇩CS, either at the top, at the centre or at the bottom of the cross-section (see chap. 3.4.3).

The specification of „Cross-section-Eccentricities“ is done by assigning a specific „Eccentricity-Code“, defining whether or not the eccentricity values of the reference point should be considered (both, the y and z components (YlZl), one of them (YoZl or YlZo) or none (YoZo)). The eccentricity vectors are automatically calculated by the program in accordance with this Eccentricity Code specified in ⇧STRUCTURE ⇨ELEMENT ⇩CS in the input field „EccTyp".

The total Eccentricity Code additionally contains the information about the orientation of the plane used for the calculation of the eccentricity vectors (normal to the element axis (default), vertical (V), horizontal (H)) and whether any angle β should be considered.

Note: *If the cross-section reference point coincides with the centre of gravity, then the definition of the eccentricity code is irrelevant.*

4.2.7.2.2.1 Definition of the position of the cross-section centroid relative to the node[+] by the Eccentricity Code.

YlZl The distance from the centroid to the CS reference point is considered as CS eccentricity (in both, the y and z directions)

YlZo Only the y component of the distance from the centroid to the CS reference point is considered. The z-component is not considered, i.e. the centroid remains in the plane built by the system line and the Y_G-axis.

YoZl Only the z component of the distance from the centroid to the CS reference point is considered. The y-component is not considered, i.e. the centroid remains in the plane built by the system line and the z_L-axis.

YoZo The distance from the centroid to the CS reference point is **not** considered. The cross-section centroid will be placed directly on the node (unless there exists a user defined system eccentricity).

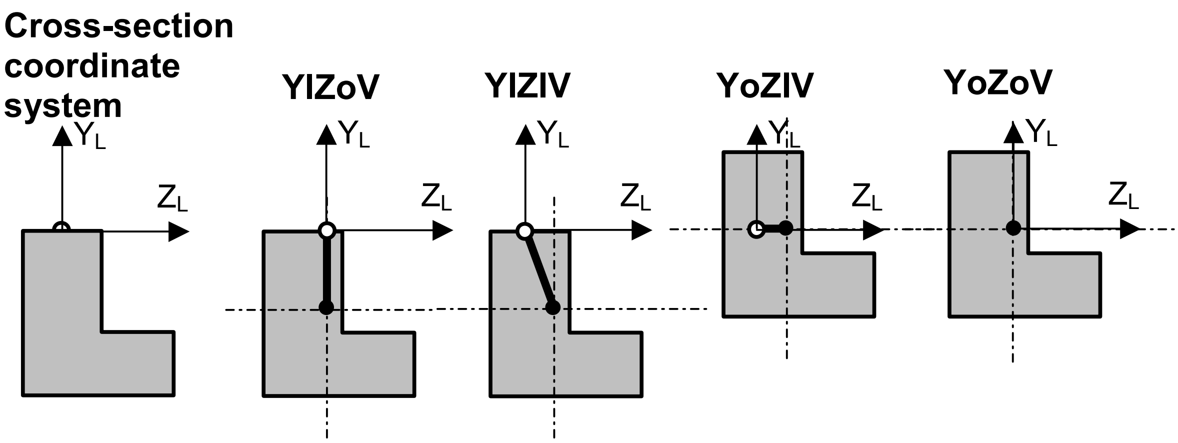


Fig. 4.2 „Eccentricity-Codes" for the centroid position relative to the node (or start point of any user defined eccentricity vector)

[+] Strictly speaking the cross-section eccentricities are related to the start point of any existent user defined system eccentricity vector instead of the node itself.

Attention: Local axes y_L, z_L in accordance with the following description.

4.2.7.2.2.2 Orientation of the cross-section plane in space („CS Position")

In order to consider the cross-section eccentricities as structural eccentricities in the structural system, an **intermediate "local coordinate system"** is built in accordance with the general rules, **using the system line (straight connection line between the nodes) as a fictitious axis x_L.** All references to local axes in the following and previous sections are related to this intermediate system**.**

The switch „CS Position" may get the following settings:
⊙ Normal
⊙ Vertical
⊙ Horizontal

„Normal" means, that the geometry calculation of the eccentricity vector is based on a plane which is normal to the **system line**. „Vertical" means, that this plane is built by vectors in the Y_G and z_L directions**,** and „Horizontal" means, it is built by vectors in the X_G and z_L directions.

In general (default setting) the automatically calculated eccentricities found via cross-section data will be related to the (intermediate) local coordinate system (⊙ **Normal**). The eccentricities will in this case be defined in the "local" system.

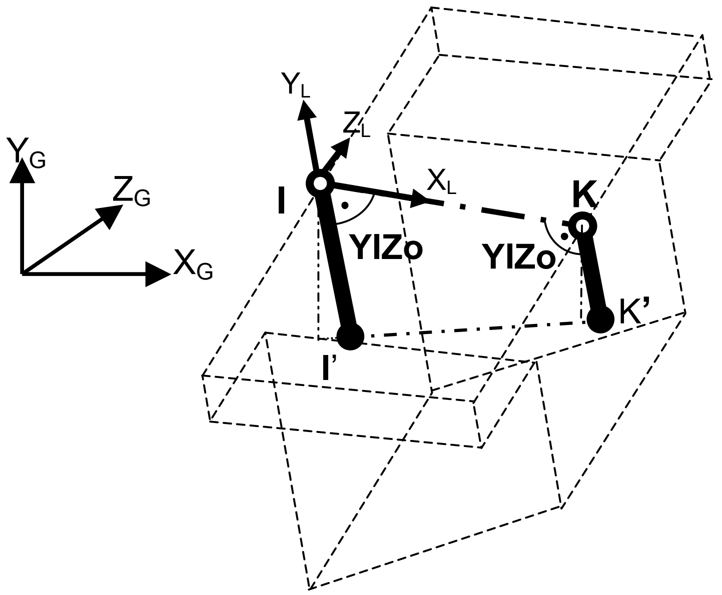


Abb..: 4.3 Local Y eccentricity – the cross-section eccentricity is normal to the system line

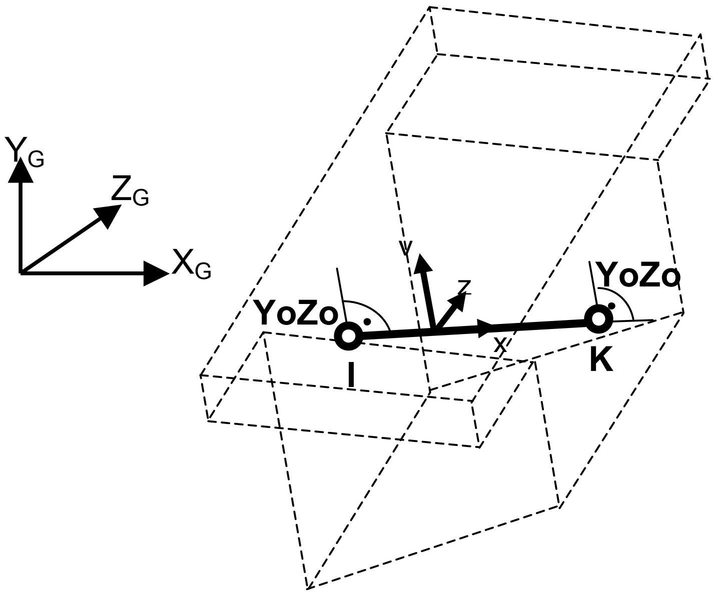


Fig. 4.4 No Y Eccentricity – the cross section centroid coincides with the system line

The options „Vertical“ and „Horizontal“ are provided because cross-sections are often defined in the vertical (and sometimes in the horizontal) plane, as opposed to orthogonal to the element axis. In the case „◉ **Vertical**” the eccentricities are assumed to be defined in a vertical plane. The Y-eccentricities are then always oriented in Y_G-direction (from the centroid to the node (Codes YlZlV etc.).

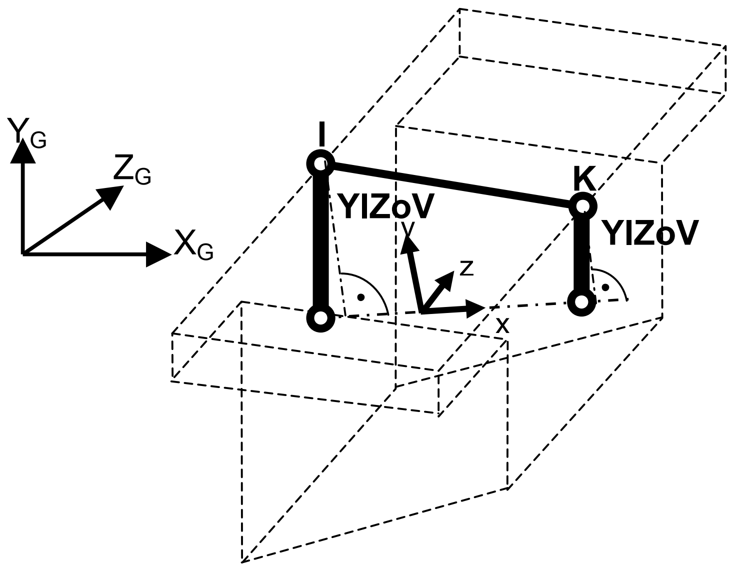


Fig. 4.5 Global Y Eccentricity – the cross section is absolutely vertical

Note: *In practical applications bridge superstructure cross-sections are mostly defined as "vertical sections", even for bridges with a longitudinal inclination or a curved bottom face of the superstructure. A "global" definition of the Y-eccentricity – as described above - is used in this case. The different alignment of the cross-section with respect to the element axis is however not considered for the computation of the stiffness matrix, where the defined cross-section geometry is assumed to be perpendicular to the element axis and not transformed.*

Similarly to the option „Vertical" RM2000 provides the option „Horizontal" (⦿**Horizontal -** Codes YlZlH, etc.). This option is less often used, but it may be helpful for modelling more or less vertical structural parts (columns, pylons, etc.). In this case the cross-section plane is defined as being horizontal, i.e. the CS eccentricities are assumed to be defined in a plane built by vectors in the X_G and z_L directions.

4.2.7.3 Clear Length

The clear length or element length is defined as the length between the eccentric connections (support faces). These are commonly characterised by the intersection point between the cross-section surface of the continuous beam and the element axis of the perpendicular element.

4.2.7.4 Effect of Eccentric Connections on the Output of Internal Forces

The internal forces and moments are output at the faces of the supports. No output is produced within the eccentric connection. An eventually defined element subdivision is also related to the clear length of the beam element. No output is produced within the eccentric connection. Nodal results may however be produced in some result presentation routines by transforming the primary results.

4.2.7.5 Effect of Eccentric Connections on the End Releases

End releases are taken to be at the support faces (at the ends of the clear length of the element) If a moment or shear release is specified, the end offset is assumed to be rigid for bending and shear in that plane at that end.

4.2.8 Element End Releases (Hinges in a general sense)

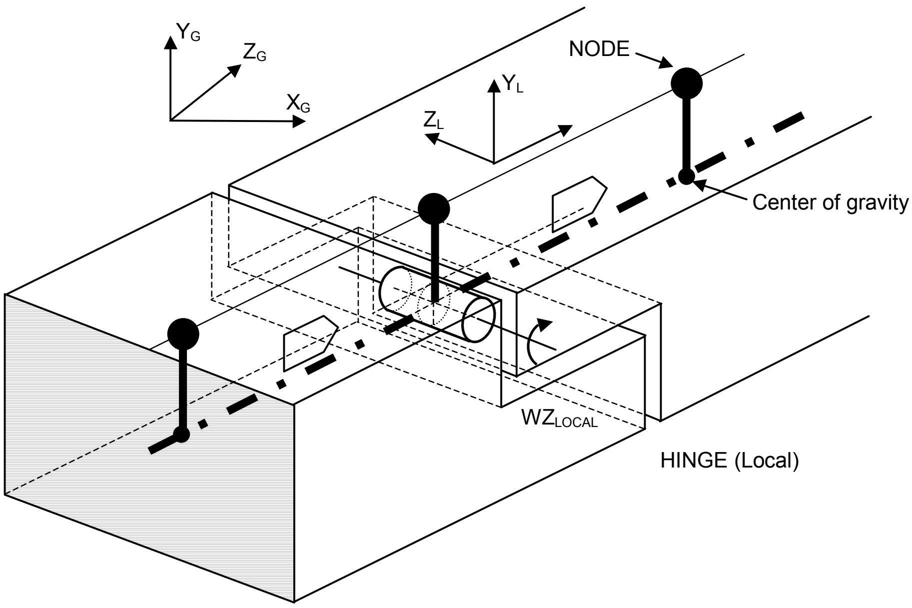
4.2.8.1 General

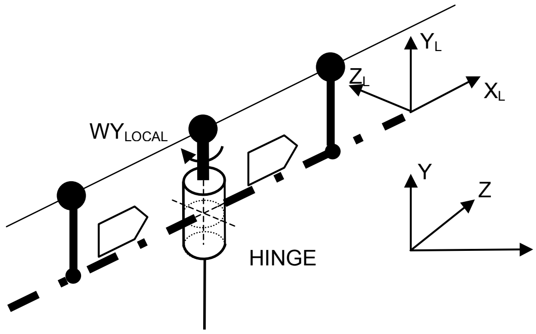
The three translatory and three rotational degrees of freedom at each end of the element are normally continuous with those of the joint.

Any release (disconnection) of one or more of the element degrees of freedom may however be made between the element end and the joint. The releases may be specified in the element local coordinate system or in the global coordinate system. Accordingly, hinges are called to be "global hinges" or "local hinges". Pay attention to the fact that the notation "hinge" is used in RM2000 not only for the disconnection of rotation DOF's, but also for translatory DOF's.

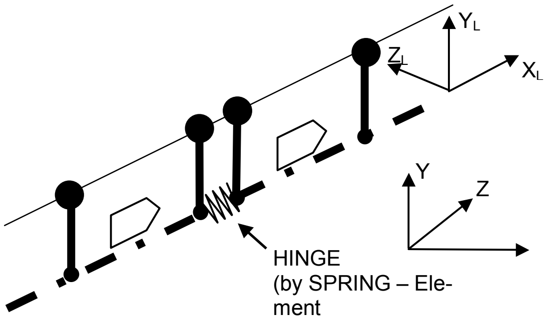
Note: *In the case that eccentric connections are defined, the local hinges are always between the rigid link and the element end, whereas global hinges are always between the rigid link and the node as shown in Fig. 4.3.*

The specified element end release does not affect any other element connected to the joint, i.e. all other elements connected to that node will remain rigidly connected to the node.

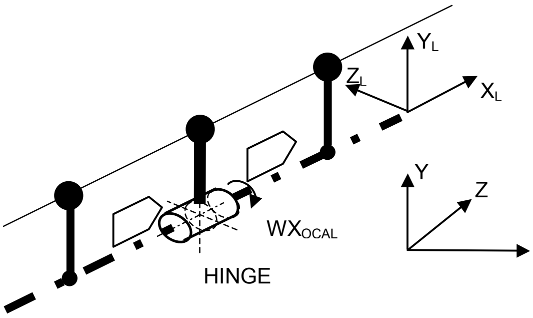




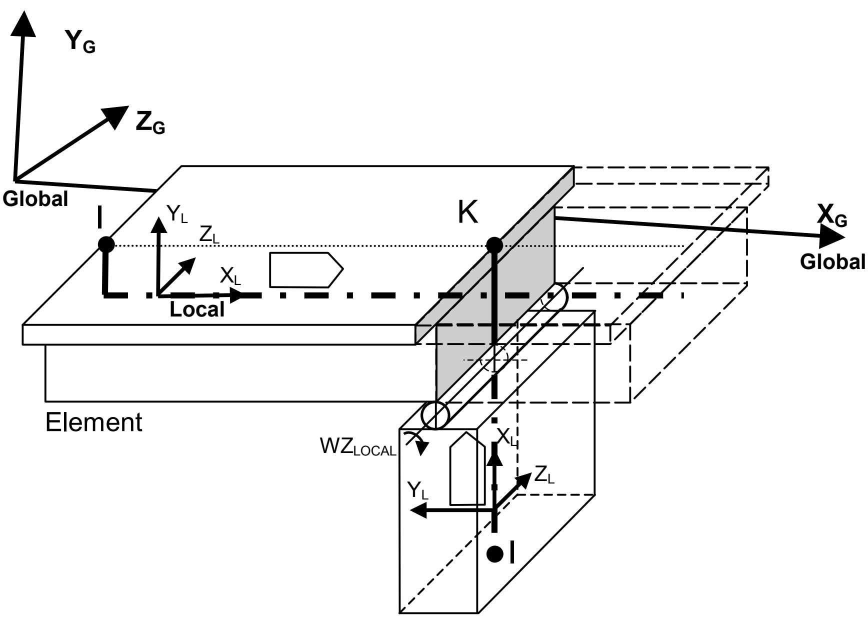
WY_{LOCAL}
HINGE



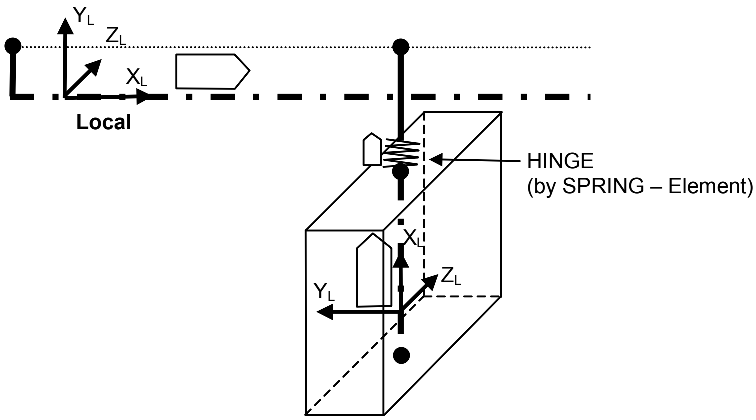
HINGE
(by SPRING – Ele-
ment



WXOCAL
HINGE



Global
Local
Global
Element
WZLOCAL
K



Element end releases are defined in the function ⇧STRUCTURE ⇨ELEMENT ⇩HINGE.

Most release conditions in practical structures can be simulated by this function. Cases, where two or more rigidly connected elements are partially connected to another group of rigidly connected elements require however a different approach. In this case two different nodes with the same coordinates have to be defined. These nodes have to be connected by spring elements whose spring constants are 0 for the released DOF's.

4.2.8.2 Unstable End Releases

Any combination of end releases may be specified for an element provided that the element or the node remain stable.

Some practical examples for restrictions to avoid unstable systems are:

- **All but <u>one</u>** element end connected to a specific node may be released.
- Local torsion hinges must not be defined for both ends of an element or of a straight sequence of elements.
- Global hinges for DOF's contributing components to local torsion must not be defined for both ends of an element or of a straight sequence of elements.
- The whole system must not become unstable due to the release (e.g. rotational hinge in a statically determined beam)

4.3 Modelling of Bridge Structures

4.3.1 General

There are very specific ways in which structures should be modelled for the computer analysis to achieve a correct representation of the real structure. This is particularly true for bridge structures – being more structurally basic, there is little opportunity for the redistribution of loads that occurs in most building structures.

Bridges are generally structures with **one** distinct main bearing direction. This direction is called the longitudinal direction. The geometric dimensions in the longitudinal direction are mostly much greater, than in the lateral or vertical directions. This allows to model the structure as a beam structure, where the assumption is made, that the cross-sections of the different beam elements remain plane throughout the analysis and a linear stress distribution may be assumed.

Bearing effects in lateral direction must not be neglected in some cases (e.g. wide bridges, built as multiple T-beam girders). The assumption that the total cross-section of the superstructure remains plane is not valid anymore in this case. The cross-section has to be taken to pieces, which approximately allow to assume this precondition. The total structure is then modelled as a girder grid.

Sometimes it is necessary to make a separate investigation of the bearing behaviour in the lateral direction for structure with only one main girder (where for the longitudinal bearing behaviour a plane cross-section may be assumed). This is often done on a separate system, where a strip in lateral direction is cut from the superstructure cross-section (e.g. 1 m) and modelled with beam elements representing the different parts of the cross-section. This structure is investigated without considering the longitudinal stresses.

One of the major difficulties in modelling bridge structures is the connection between the different parts of the structure, especially connecting the superstructure to the piers, abutments or other foundation structures. Separate investigations of the bearing behaviour in lateral direction require modelling the indirect support by the main girders in a suitable way by support elements.

4.3.2 Superstructure Modelling

4.3.2.1 Bridges with one single main girder

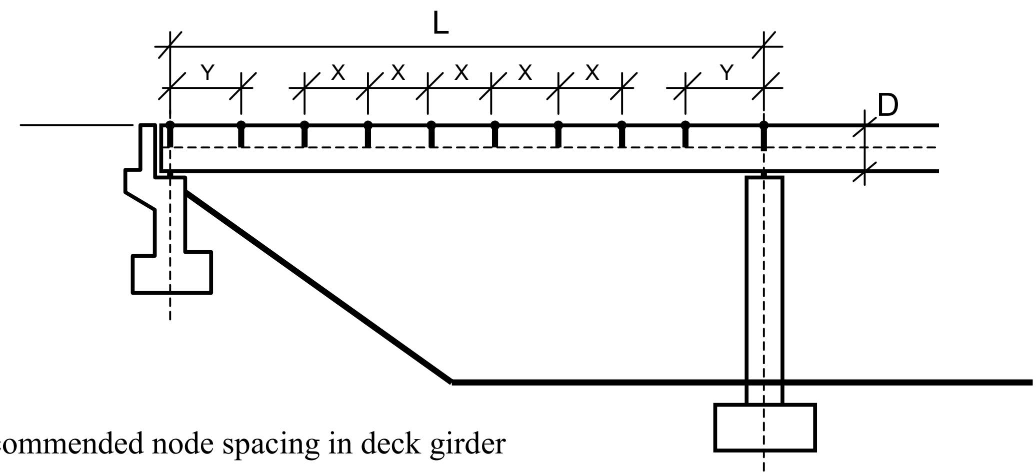
This class contains most hollow box bridges, where it usually may be assumed with sufficient accuracy that the whole cross-section remains plane. Multiple box girders with small cross-section height may however require dividing the cross-section in several parts forming different main girders, and to connect them with cross girders simulating the plates connecting the webs of the cross-section.

Narrow T-beam bridges with one single web or 2 webs with small distance may often also be modelled as bridges with only one main girder. This is also true for narrow plate bridges.

Tablature

Thick Solid Line	Stiff (rigid) connection
Dashed line	Element Centroid
Dot	Nodal point

Certain recommendations are given below for the modelling of super structures:
Longitudinal model



Recommended node spacing in deck girder

X ≤ L/10
Y ≤ D (for elements immediately before and after a support)

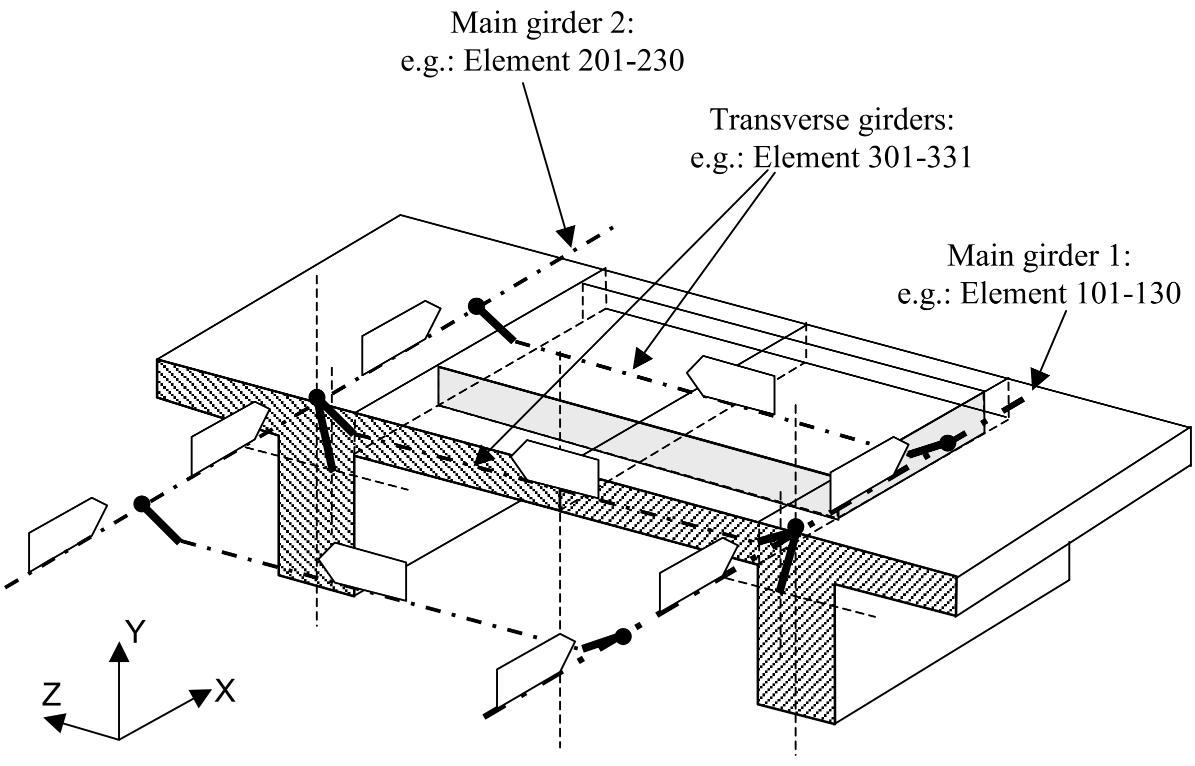
Bridges with pre-fabricated segments – Use 1 node per segment joint – minimum!

4.3.2.2 Bridges with more than one main girder

If a bridge has a wide superstructure cross-section it is not possible anymore to model the superstructure with one single main girder. The cross-section is in this case divided into several parts. The figure below shows a double T-beam divided into 2 parts. The loading of each main girder basically acts on the loaded part, not on the total cross-section.

The connection between the 2 main girders by the upper flange plate is modelled by fictitious cross beams, the beam stiffness being equivalent to the plate siffness. A greater or smaller part – dependent on the stiffness of the connecting plate - of the load acting on one of the main girders will be redistributed by the cross beams to the other girder.

<u>Detail of a double T-beam Super-structure model as a grid of beam elements.</u>
Two main girders are connected with transverse cross-girders



4.3.2.3 Separate investigation of bearing behaviour in transverse direction

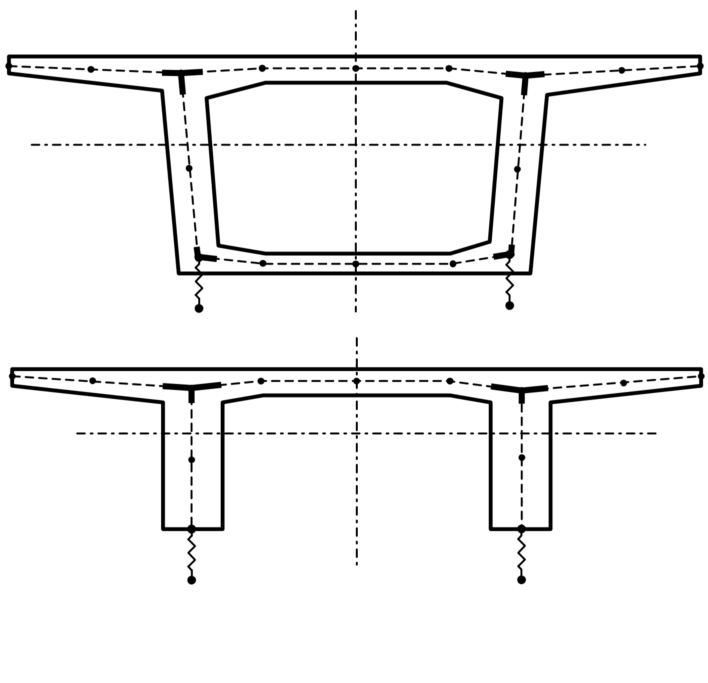
It is often required to investigate locally the bearing behaviour in the transverse direction, even if it is sufficient to model the superstructure for the analysis of the bearing behaviour in main (longitudinal) direction with one singe girder using the total superstructure cross-section. (e.g. to determine the influence of a single wheel load to the moments in transverse direction in the roadway plate).

In order to do this it is possible to cut a strip out of the cross-section and to model it as plane frame. The different parts of the cross-section (upper plate, webs, bottom plate) are modelled as beams with rectangular cross-section. For analysing a strip along a support line (above the bridge bearings) the support elements may be used in the same way, than for the longitudinal bridge model. For investigating a mid-span cross-section the elastic indirect support must be simulated in a suitable way by fictitious support elements (e.g. in the centre of the webs).

Example: Modelling a cross-section as plane frame for investigating the bearing behaviour in transverse direction.

<u>Transverse section:</u>

Note: *this section is used only separately for the transverse analysis – This section is **NOT** used for the longitudinal analysis (where the overall section properties are required!)*



4.3.3 Connection of the Superstructure with the Sub-structure

The main area of difficulty in modelling bridge structures is at the connections of different members – particularly connecting the sub-structure to the superstructure.
The following types of connections are generally used between the sub-structure and the superstructure of the more common bridges:

- Built-in connection
- Fixed bearing connection
- Uni-directional bearing connection
- Multi-directional bearing connection

The bearing configuration at a typical connection is normally either a 'Single' or 'Double' bearing connection depending on the overall structural stability and on the torsion capacity of the bridge deck. The 'Double bearing configuration' provides torsion fixity and is typically provided at the ends of the bridge deck and even at certain internal piers on long viaduct type bridges. The double bearings are usually placed under a stiff diaphragm (in box girder bridges) so their position is not constrained by the positions of the webs.
Given below are a series of sketches that depict the recommended modelling of these 'Typical Connections' for a Bridge deck diaphragm (typically in a concrete box girder) to single Column connection.

All the bearing connections between the deck and the substructure are modelled with spring elements.

- The spring elements do not actually have any dimension.
- The spring elements are located at the actual position of the bearings in space.
- The spring elements are connected to the sub-structure and the superstructure with 'Rigid elements' that do not deform in any way.
- The 'Rigid elements' are not user defined and have no identification name or number.
- The 'Rigid elements' are defined as 'Eccentric Connections' between the ends of the elements and a node. The sign convention for 'Eccentric connections' is in the 'Global coordinate directions' and is considered as being a **vector from the end of the element to the node**.

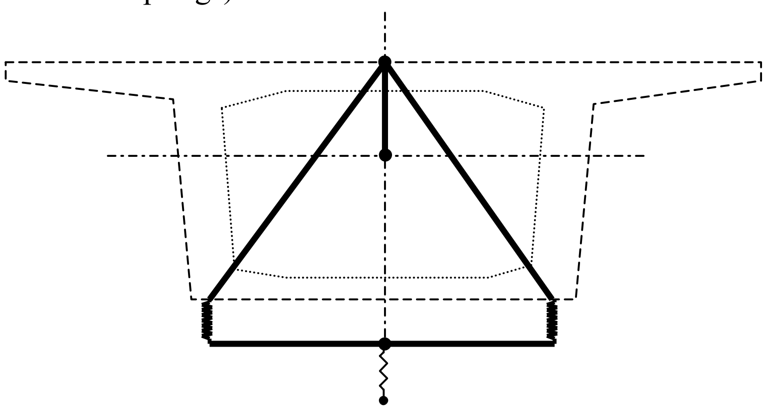
Referring to the sketch below (Bearing arrangement taken to be 0.5m from top of pier to soffit of deck.):
The column – (or abutment) element no. 103 is connected to the ground (Node '0')
Spring element no. 101 (Left hand bearing) is connected to the node at the top of the column: Z-Direction eccentricity (Spring element 101 to node 100) + 2.500m
Y-Direction eccentricity (Spring element 101 to node 100) – 0.250m

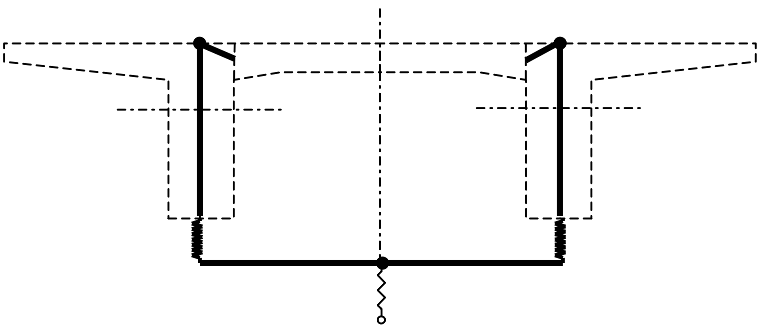
Spring element no. 101 (Left hand bearing) is connected to the node at the top of the deck: Z-Direction eccentricity (Spring element 101 to node 1) + 2.500m
Y-Direction eccentricity (Spring element 101 to node 1) + 3.000m

Spring element no. 102 (Right hand bearing) is connected to the node at the top of the column: Z-Direction eccentricity (Spring element 102 to node 100) - 2.500m
Y-Direction eccentricity (Spring element 102 to node 100) – 0.250m

Spring element no. 102 (Right hand bearing) is connected to the node at the top of the deck: Z-Direction eccentricity (Spring element 102 to node 1) - 2.500m
Y-Direction eccentricity (Spring element 102 to node 1) + 3.000m

Details at the Sub-structure/Super-structure connections.
(The bearings are shown as springs)





The node at the top of the deck is automatically connected to the centroid of the section with an 'Eccentric connection' when allocating the cross section to the element (see chap. 3.3).

The type of bearing –'Rigid'; 'Multi-directional'; or 'Uni-directional' is modelled by defining the 'Releases' for the individual spring elements (Vx; Vy; Vz; φx; φy; φz).

Typical values for a 'Rigid' connection for a spring element is 1E10. Therefore a 'Multidirectional Bearing' allowing free rotation in all directions and movement in the Global 'X' and 'Z' directions would be given the following values in the Global Coordinate System: 0, **1E10**, 0, 0, 0, 0.

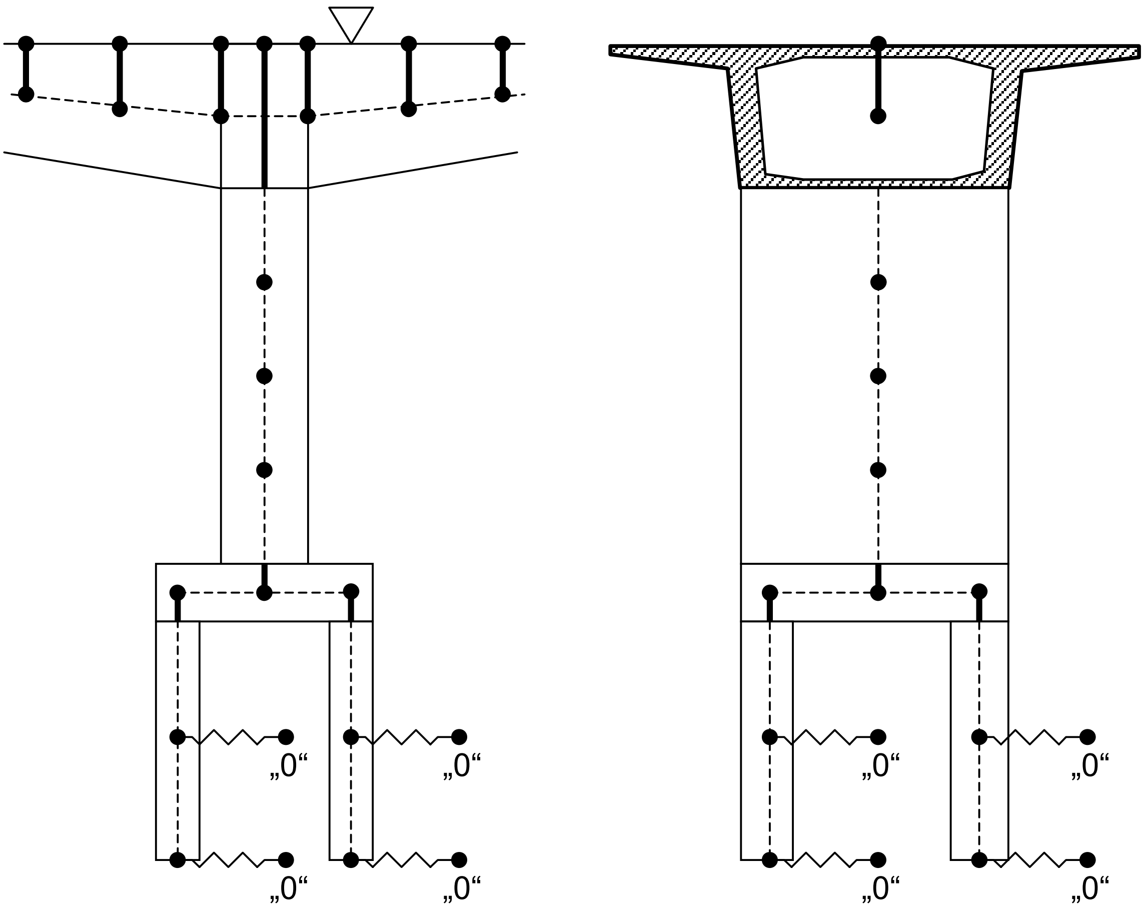
A 'Uni-directional bearing' restricting transverse movement but allowing full rotation in all directions would be given the following values in the Global Coordinate System: 0E10, **1E10**, **1E10**, 0, 0, 0.

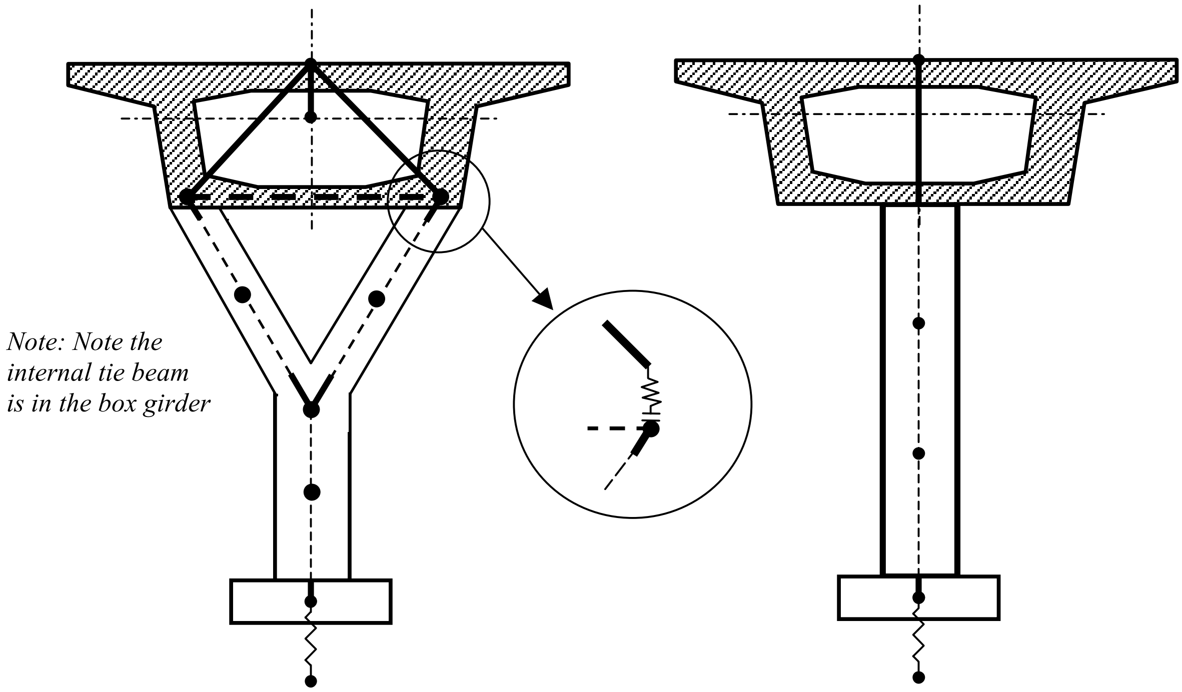
4.3.4 Substructure Modelling

Tablature

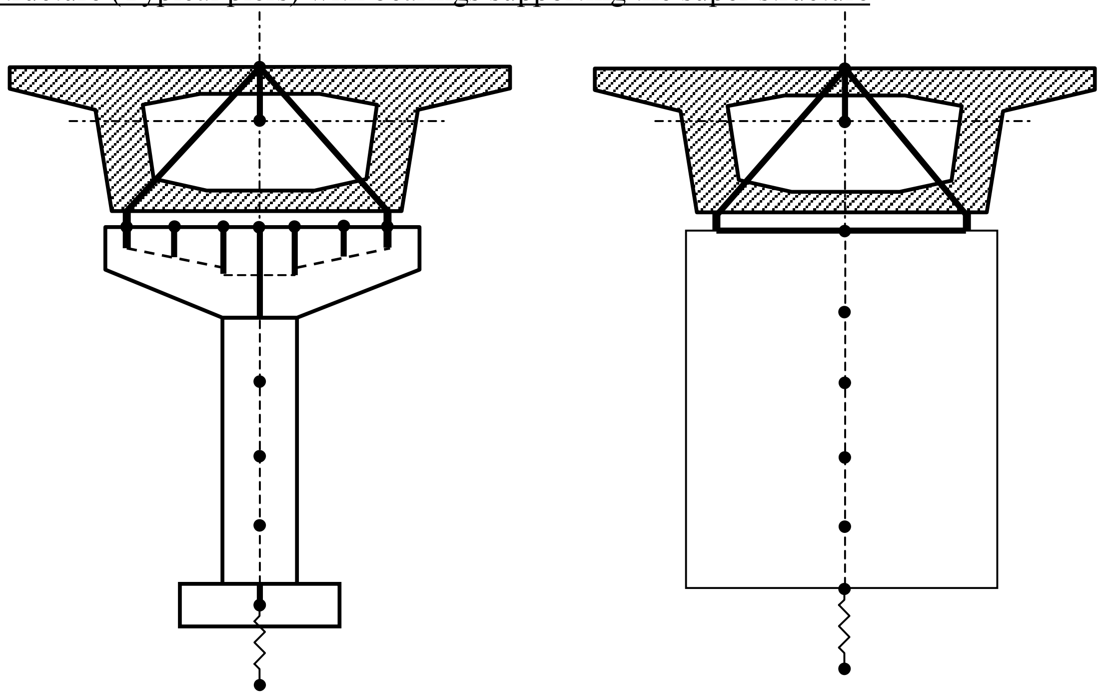
Thick Solid Line	Stiff (rigid) connection
Dashed line	Element Centroid
Dot	Nodal point

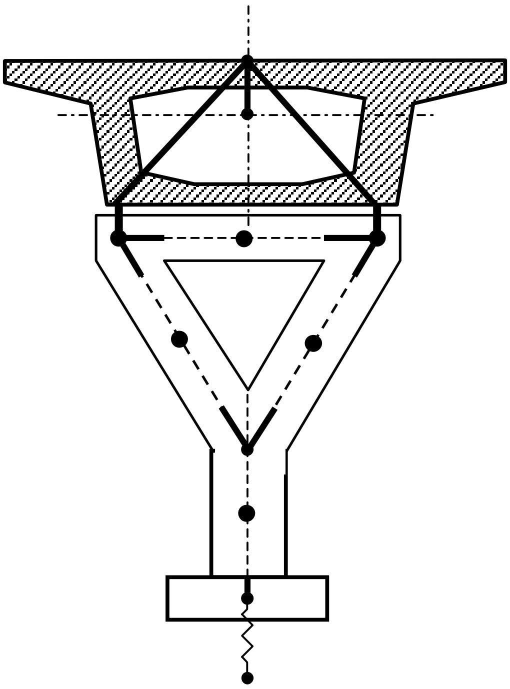
Substructure – built in to the superstructure (no bearings)
(Showing some typical piers)



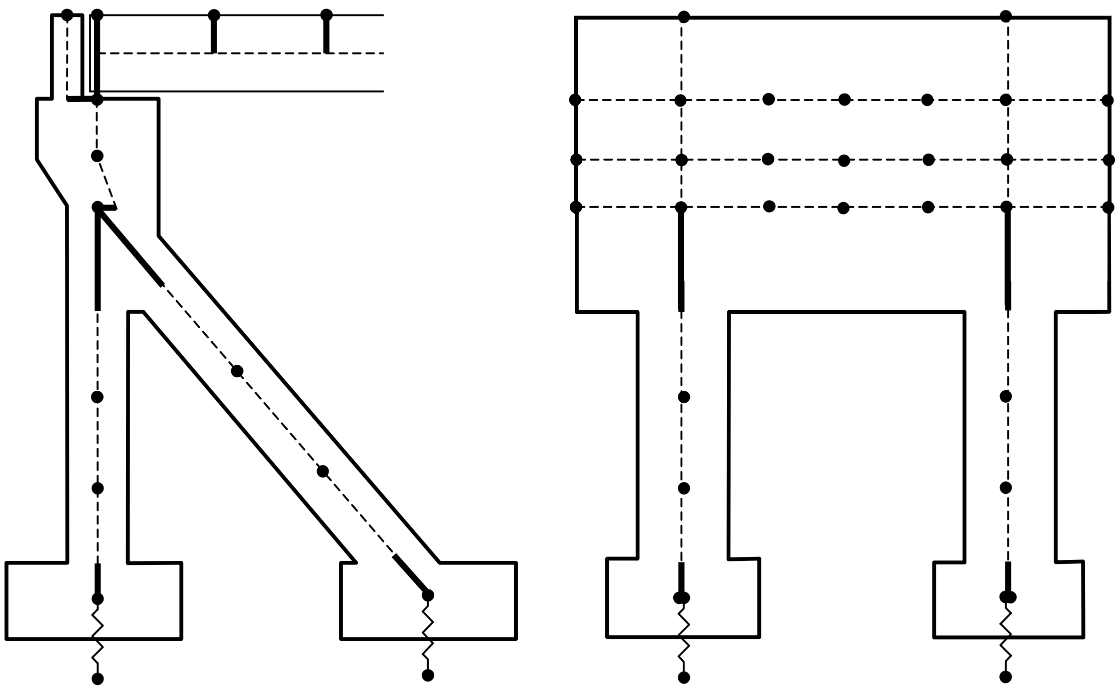


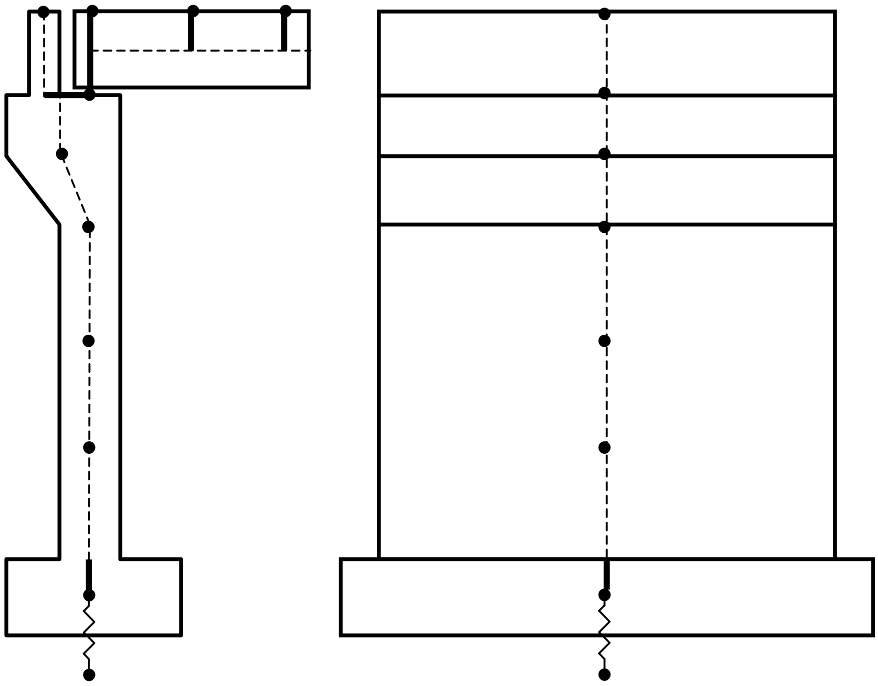
Substructure (Typical piers) with bearings supporting the super structure





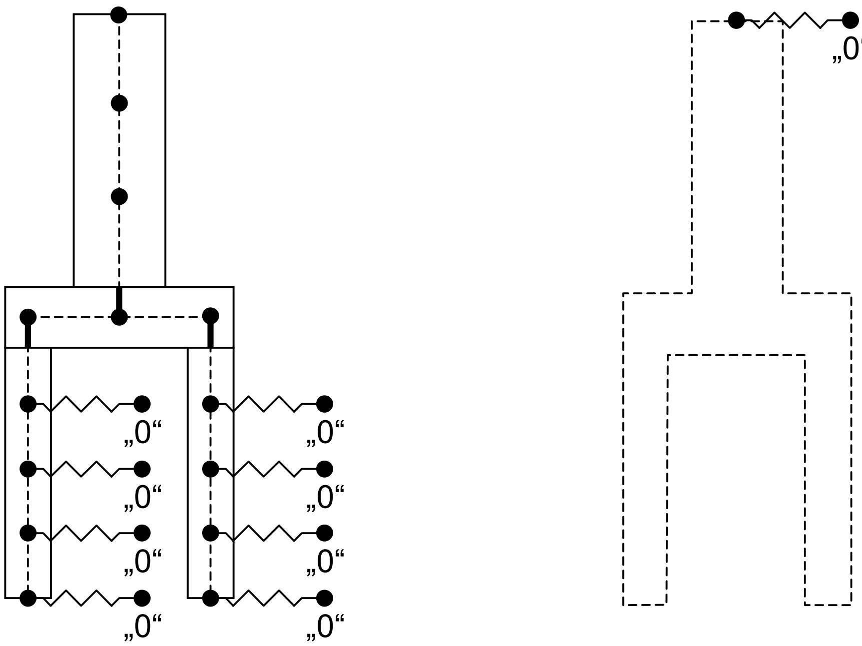
Substructure (typical abutments) with bearings supporting the super structure





Stiffness matrix / Flexibility matrix

The entire substructure can be replaced with a single matrix representing the action of the whole support system below the deck girder at this point.



4.4 Composite Structures

4.4.1 Composite Cross-sections

Cross-sections of composite elements are basically specified in the same manner than those of ordinary beam elements (see chap. 3.4). A direct element related definition of the cross-section values is not allowed for composite elements, because the geometric arrangement of the different cross-section parts is in this case essential for calculating the stiffness matrices. Therefore, all start and end cross-sections of composite elements have to be defined as database objects in the function ⇧PROPERTIES ⇨CS in the RM2000 GUI, or in the Geometric Preprocessor GP2000.

The geometry definition (FE-mesh) has to be done separately for the different partial cross-sections forming the composite cross-section. The Finite Element mesh of the composite cross-section is automatically built by the program using the different parts. To enable this composition process the partial meshes must coincide along the connection line (see chap. 3.4.7). All cross-section values of the composite cross-section as well as for the partial cross-sections are then individually calculated by the program.

There is one difference between „normal" cross-section values and „composite" cross-section values: in the case of composite structures it is necessary to consider the elastic modulus of the materials of the different cross-section parts. The integration domain is not the true area, but a weighted area, where the weights for the different parts are the ratio's between the elastic modulus of that part and a „reference modulus" assigned to the composite cross-section. All results are then related to this reference modulus, and all later calculations can be performed in the standard manner, using this reference modulus and the related cross-section values.

4.4.2 Nodes and Elements of the Structural System

The basic concept for the analysis of composite structures in RM2000 is, that partial elements are combined to composite elements. Each part of the composite element (steel girder, concrete plate,...) has to be defined as a separate partial element, and the related partial cross-sections have to be assigned to the start and endpoints of these elements. A separate composite element has to be defined for every combination of partial elements being at least once active in the construction schedule. The composite cross-section of these elements is automatically built by combining the cross-sections of the partial elements.

The nodes assigned to partial elements and the composite elements may also be different, if this is required for correctly analysing the different construction stages.

4.4.3 Construction Stages and System Activation

Composite cross-sections and elements respectively may be supplemented step by step during the construction sequence. **The composite element** representing the actual combination of partial elements **and all** actual **partial elements** have to be activated. No composite element has to activated if - in an early stage - only one single partial element is active.

4.4.4 Calculation of Internal Forces

Primary results of an analysis of a composite structure are the internal forces of the composite element.

This primary forces may be divided up in the function „SPLIT“ to the partial elements by using the result presentation type **“SPLITTED“** in ⇧RESULTS ⇨LCASE or ⇧RESULTS ⇨ENVELOPE**.** For the normal forces and bending moments Nx, My and Mz this may be done using the well known formulas, and the user may feel free checking the split results by manually doing this transformation as an exercise. For the shear forces and torsion moment Qy, Qz and Mt however, the FE model must be used to calculate the shear stress distribution in the composite cross-section. These stresses are integrated over the partial cross-sections yielding the contributions of the different parts to the total shear forces and torsion moment. This procedure might require a considerable amount of computation time in big systems.

<u>**Important:**</u> A back-calculation to get the stress distribution by using the partial values of Qy, Qz, and Mt is not possible. The correct stress distribution can only be calculated using the total composite cross-section and the total internal forces. The contributions to the different parts may therefore only be used for a general evaluation and must not be used for further stress- or ultimate load calculations!

A switch for invoking the function „SPLIT“ and presenting split or total results (or both) is also available in the following calculation actions:

- PLSYS (Graphic presentation of results)
- DOLIST (Generating output lists of results)
- Computation of shear key forces (see below)
- ADDCON (Constraints for determining the stressing sequence of stay cables of a cable stayed bridge)

The inverse function **„JOIN“** (inverse with respect to „SPLIT“) allows to transform the results of a partial element to the system axis of a later in the construction sequence activated composite element and to superimpose them with the results of other partial elements or of the composite element itself. The transformation is made for Nx, My, Mz using the well known formulas. For Qy, Qz, Mt analogous formulas are used, derived from the equilibrium conditions.

Important: The transformed and accumulated internal forces may not be used for computing the normal and shear stresses in the cross-section. The sum of the internal forces does finally not contain the information about the origin of the different contributions.

ATTENTION: The „accumulated“ value must therefore only be used for a general evaluation of the bearing behaviour. They must not be used for further stress or ultimate load calculations.

The „JOIN“ function is available in the following calculation actions:

- PLSYS (Graphic presentation of results)
- DOLIST (Generating output lists of results)
- ADDCON (Constraints for determining the stressing sequence of stay cables of a cable stayed bridge)

4.4.5 Computation of Stresses

Normal stresses (from Nx, My, Mz) and shear stresses (from Qy, Qz, Mt) may be calculated for predefined „stress points“ (see chap. 3.3). These reference points are defined at the beginning of the data input process for the analysis (either in GP2000 or in RM2000). They are later available throughout the whole analysis progress. Related to the actual activation state, the stresses are calculated on the actually valid composite cross-section, or – if only one partial element is active – on this partial cross-section.

The normal stresses are computed using the well known basic formulas, the shear stresses are computed using the FE model of the cross-section. The stresses of the different stress points may by accumulated throughout all construction stages and different partial or composite states respectively.

Attention must be paid to the Finite Element subdivision of the cross-section. The mesh must be sufficiently fine, not only with respect to the calculation of the stiffness matrix, but also to guarantee accurate stresses in the „stress points“. *RM2000* offers a function for an automatic mesh refinement in the module for the calculation of the cross-section values.

4.4.6 Computation of Shear Key Forces

The shear forces in the connection face are calculated for the „Ultimate serviceability state“ in order to determine the necessary shear key dimensions.

The following procedure is generally applied in traditional „by hand“ calculations: A horizontal section with the width „b“ is placed in the cross-section at the level of the connection face. The statical moment „Sz“ of the cross-section part cut away is calculated. The shear stress may then be calculated using the well known formula

$$\tau_{xy} = \frac{Q_y * S_z}{J_z * b}.$$

But the validity of this formula is limited (connection face parallel to the element axis, constant cross-section, etc.).

A more general approach is therefore used in *RM2000.* The shear stresses in the connection face must correspond to the change of the normal force (dN / dx) transmitted in that part of the composite element, which is separated by the considered connection face. These normal forces are available for all partial elements if the above described function „SPLIT“ has been used. Only the normal force difference between the start and the end of a partial element has to be calculated to get the total shear force to be transmitted by the pins over the element length. This process avoids the restrictions of the above shear stress formula and the result is consistent to all other system modelling assumptions.

However, there is an other problem arising due to the fact, that the load case superposition of traffic loads and other life loads does not yield the maximum values for normal force differences, but only for the normal forces. This problem is solved by using so called „combination elements“.

RM2000 allows the user to define arbitrary linear combinations of basic element results as results of „combination elements“. These linear combinations may for instance be displacement differences or, as used in the above described context, normal force differences between start and end of the element. The required linear combinations are built in RM2000 when a load case is calculated, and the results are stored for the „combination element“ in the same way than the basic results for the structural elements.

An element number different to the numbers of the structural elements has to be defined for every „combination element“. But this number does in this case not identify a real structural element, but is related to another (structural) element, whose additional result values are stored under this element number.

The „combination elements“ must be activated in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTIVATION and are then considered in all functions of load case superposition and influence line calculation. Thus, these elements also allow to calculate and evaluate influence lines for the shear key forces of a composite element. and to determine the maximum forces in the shear keys also for complex traffic load combinations.

RM2000 supports the use of „combination elements“ for the calculation of shear key forces by automatically creating such elements for all partial elements of composite elements in the database. The numbers of these elements are generated by adding 10000 to the element number of the related partial element.

Example:

- The steel girder of a composite bridge is numbered from 101 to 150
- The automatically established related combination elements get the numbers 10101 to 10150.

No further input is required for specifying the „combination elements“ if the user agrees with this numbering scheme. The elements have however to be specified in the activation list of the construction stage where the shear keys become active and the transmission of shear forces starts.

Note: *Actually no input facilities for defining “combination elements” are provided. Therefore a general use of these elements is not yet possible. They are automatically established for composite structures with the above mentioned normal force difference as related result value.*

The design codes of several countries also claim an ultimate load check in addition to the described shear key calculation for the ultimate serviceability state.

The German code (DIN) requires for instance, that the „Plastic moment“ of the composite cross-sections has to be calculated and the related „total compression forces“ in the concrete part - and the related tension forces in the reinforcement of the cracked zone respectively - be determined. The maximum value of these compression and tension forces is determined for different sections of the girder (mid-span region, support region) and the chosen shear key amount must be able to transmit these forces. But these maximum forces may be reduced using the ratio between the actual „Ultimate load moment“ (moment times safety factor) and the „Plastic moment“. This reduction factor must however not be under 0.5.

RM2000 gives the values of the „Ultimate load moment“ in the result listing of the computation of the ultimate moment of the composite cross-section. The „Plastic moment“ of the composite cross-section is calculated by performing the ultimate moment calculation for a zero internal force combination. This is created by initialising a load case (LCINIT) and assigning it without superimposing any calculated load cases to the ultimate moment calculation action.

4.4.7 Pre-stressing of Composite Girders

All partial elements of a composite element may individually be pre-stressed. In accordance with the actual state at the tensioning time the pre-stressing has to be applied at the partial element or composite element representing the actual construction state.

“External” and “internal” pre-stressing require different procedures.

External pre-stressing: It is assumed that the friction along the idle roll is sufficient to guarantee that the connection between pre-stressing tendon and structural system is fully actuated by adherence. All relevant design code rules for the **ultimate serviceability state** are based on that assumption. Therefore, all hints given further below for the internal pre-stressing after establishing the grouting effect are also valid for the area along the idle roll of an external tendon.

It is up to the user, whether this area is modelled in detail – with system nodes on both ends of the idle roll and maybe further intermediate system nodes and detailed description of the tendon geometry – or whether the idle roll is modelled in a rough way with only one single system node and a kink in the tendon geometry.

In between the idle rolls the external tendon is modelled by a separate structural “linear cable element”. This cable element is a straight connection between the end of the one and the begin of the other idle roll. The cable element is at both ends connected to the system node by “eccentric connections”. The system parameters of the cable elements are automatically created when the input for pre-stressing is made. The user must only assign element numbers to the cable elements. This is also done in the input sequence for pre-stressing.

Unfortunately no practically applicable hints for for treating the adhesion along the idle roll in the **ultimate limit state** are actually available in the relevant design codes. This is also true for internal pre-stressing states before the adhesion due to grouting has been established.

Assuming fixed end anchorage and free or friction-governed sliding inbetween is the basis for calculating the friction losses during the stressing procedure. But this assumption would lead to a very complex non-linear computation process if used for any later applied Load Cases and especially for traffic loads. Such analyses are in principal possible, but the related huge computation effort can in most cases not be afforded. Some design codes (e.g. DIN) require treating the ultimate limit state principally in the same way, than an internal pre-stressing with adhesion. The only difference is, that no additional strains due to later applied loadings are transferred from the cross-section to the tendon. But it is allowed to add an additional tendon stress – mostly a percentage of the yield stress - to the stress state applied in the tendon by the stressing procedure.

Internal pre-stressing: For internal pre-stressing and external pre-stressing in the idle roll area all application rules of the general pre-stressing functions as described in chapter 5 are valid. The following special hints are to be considered in the context of composite structures:

- a
- b
- c

“Primary” and “secondary” results must be distinguished and separately calculated and stored for the load type “Pre-stressing”. The primary state (also called „$V * e$“-state) contains the direct effect of the tensioning process onto the pre-stressed structural elements (internal stress state without taking into account external constraints). The applied pre-stressing force is transformed into the components N_x, Q_y, Q_z, acting in the direction of the element axis and perpendicular to it. The eccentric position (e_y and e_z) of the tendon with respect to the centre of gravity results the moments $M_x = Q_y * e_z + Q_z * e_y$, $M_y = N_x * e_z$, $M_z = N_x * e_y$. These internal forces of the primary state is related to deformations of the pre-stressed elements, yielding deformations of the total system. External constraints (boundary conditions) will in the general case yield restraint stress resultants. The secondary state contains these internal forces due to restraint and the deformations of the total system within the boundary conditions.

4.5 Cable Stayed Bridges

4.5.1 General

The analysis of a structure of this type can often be subdivided into 2 general procedures:

- Design work on the "Final Structure" resulting in final cable stressing forces, Element sizes and an acceptable force diagram to be aimed at in the Construction Stage analysis.

- Construction Stage Analysis phases – Stage cable forces manipulated to achieve the acceptable force diagram derived from the "Final Stage analysis" above.

RM2000 is designed to fulfil both of the above parts of the calculation even whilst taking several special effects such as creep & shrinkage, cable sagging, P-Delta effects etc into account.

Each cable stayed structures is unique, it is therefore not possible to provide a universally applicable recipe on how to carry out the calculation. TDV, however, proposes that the bridges can be categorised into 4 general types. It is possible to consider all or none of the special effects (non-linearities, P-Delta,…) for each type. It is, of course, up to the engineer to decide which of the various functions and combinations are required to suit his needs.

The table below shows TDV's proposal which is based on our experience and the special features available in *RM2000*. The influence of each function on a large cable stayed structure was carefully studied and is shown as a percentage of the results derived without consideration of the function.

Note: *The percentage will, naturally, vary from project to project.*

4.5.2 Available Options

P-Delta……..	consideration of P-Delta effects in the analysis
Cable non-linear…	Cable sagging for cable elements is considered. Note: a transverse load (at least the self weight) and a program internal subdivision of the Cable element (NTEL = 4,5, 6,8, … under STRUCTURE / ELEMENT / ELEMENT) must be defined.
3rd Order Theory….	Consideration of the “Large displacements” theory.
Shear displacement…	Consideration of this effect. Where the cross section geometry is not accurately defined (e.g. stiffeners for steel cross sections are smeared and not modelled in detail) we suggest that an equivalent G-modulus is specified for the material.
Elastic compression compensation…	The pylon as well as the deck will be shortened due to the applied normal force from the cables etc. This shortening will influence the structural behaviour and must somehow be compensated. We propose that a “stress free element length” load is used.
Ernst E-Modulus	Consideration of reduced E-Modulus in accordance with the Ernst’s formulas.
Damper elements	Use of special damping elements for dynamic analysis.

Smaller structures can be understood to be structures with a relatively high girder stiffness where effects such as P-Delta, Cable sagging etc do not have a significant effect on the structural behaviour.

These effects will have a high influence for larger structures.

4.5.2.1 Classification and available functions:

Procedure	Classification	P-Delta	Cable (non-lin)	IIIrd Order	Shear dis-placement	Compression compensation	E-modulus (Ernst)	Damper Elements
1	Smaller Structures Final System	☺			☺	☺	☺	☺
2	Smaller Structures Constr. Stages	☺	☺		☺	☺	☺	☺
3	Larger Structures Final System % diff in results	☺ 10 – 20%	☺ 10 – 20%	☺ 10%	☺ 3 – 7%	☺ 10%		☺
4	Larger Structures Constr. Stages	☺	☺		☺	☺		☺

☺ .... Function can be used.

4.5.2.2 Explanation:

<u>Procedure 1 (Small Structures *Final System*):</u>

Consideration of P-Delta: All LSET must first be defined in one single LCASE, select “P-delta effect” in RECALC window.
Consideration of Shear: Can always be selected. No special settings required in RECALC window.
Compression Compensation: No special settings required in RECALC window.
Damper elements: No special settings required in RECALC window.

Procedure 2 (Small Structures *Construction Stage Analysis)*:

Consideration of P-Delta:	Must be selected in combination with “accumulate stiffness” in the RECALC window .
Cable non-linear:	In construction stages effects of cable non-linearity only can be considered by modification of the cable elements! A transverse load (at least the self weight) must be defined.
Consideration of Shear:	Can always be selected. No special settings required in RECALC window.
Compression Compensation:	No special settings required in RECALC window.
E-modulus Ernst:	Ernst modulus is defined by the user as a loading. as a variation of „cable non-linear“ (Either one or the other).
Damper elements:	No special settings required in RECALC window.

Note: *Taking account of IIIrd order in a construction stage analysis causes philosophic problems. It can, however, be considered with the GEOMETRY UNDER LOAD FROM THE FINAL SYSTEM CALCUALTION (result from procedure 1 or 3) as a basic geometry for the CONSTRUCTION STAGE CALCUALTION. The consideration of P-Delta effects alone is then sufficient.*

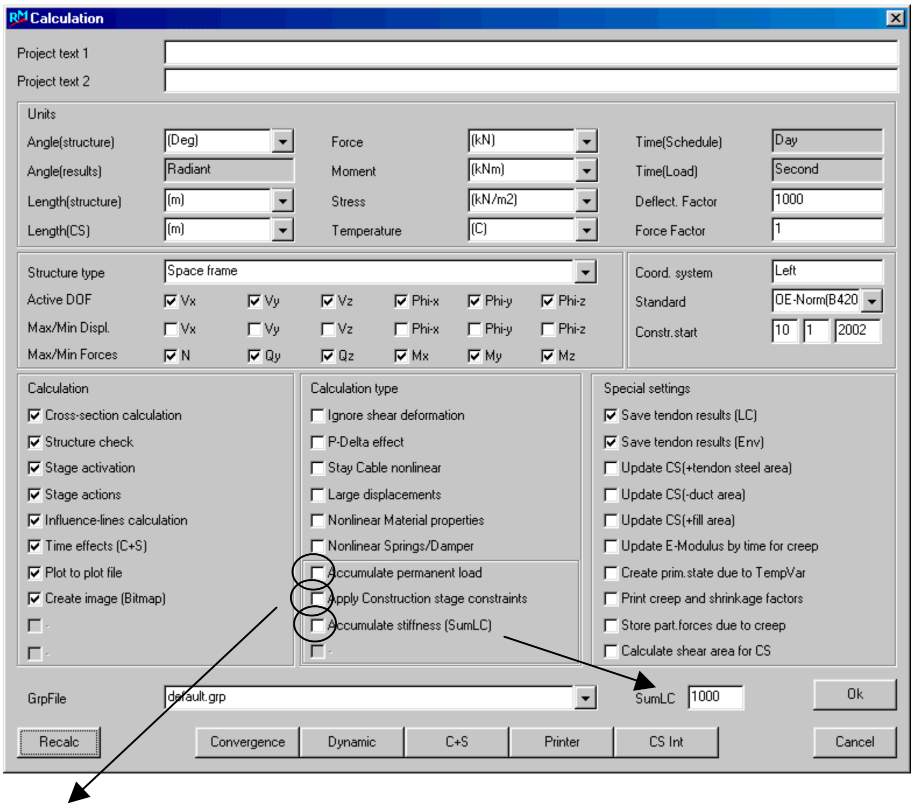
Procedure 3 (Large Structures *Final System)*:

Consideration of P-Delta:	All LSET must first be defined in one single LCASE, select “P-delta effect in RECALC window.
Cable non-linear:	A transverse load (at least the self weight) and a program internal subdivision of the Cable elements must be defined!
IIIrd Order Theory:	All LSET must first be defined in one single LCASE, select “IIIrd Order Theory” in RECALC window.
Consideration of Shear:	Can always be selected. No special settings required in RECALC window.
Compression Compensation:	No special settings required in RECALC window.
E-modulus Ernst:	Ernst modulus is defined by the user as a loading. as a variation of „cable non-linear“ (Either one or the other).
Damper elements:	No special settings required in RECALC window.

<u>Procedure 4 (Large Structures *Construction Stage Analysis)*:</u>

Consideration of P-Delta:	Must be selected in combination with "accumulate stiffness" in the RECALC window .
Cable non-linear:	Effects of cable non-linearity in construction stage analysis can only be considered by modification of the cable elements! A transverse load (at least the self weight) must be defined.
Consideration of Shear:	Can always be selected. No special settings required in RECALC window.
Compression Compensation:	No special settings required in RECALC window.
E-modulus Ernst:	Ernst modulus is defined by the user as a loading. as a variation of „cable non-linear" (Either one or the other).
Damper elements:	No special settings required in RECALC window.

Note: *Taking account of IIIrd order in a construction stage analysis causes philosophic problems. It can, however, be considered with the GEOMETRY UNDER LOAD FROM THE FINAL SYSTEM CALCUALTION (result from procedure 1 or 3) as a basic geometry for the CONSTRUCTION STAGE CALCUALTION. The consideration of P-Delta effects alone is then sufficient.*



Accumulate Permanent load: Many loading cases are calculated in the Construction Schedule. All the permanent loads (self weight, pre-stressing,…) are automatically identified as being permanent loads. The loads are automatically accumulated by the program so that the same construction stage preparation for a linear calculation can be used for a non-linear calculation as well.

Note: *Must be in combination with "SumLC" in order to consider all influences from previous loads correctly!*

Apply construction stage constraint : The structure deflects in each stage due to loads. If the loads are accumulated (see above) it is also necessary to apply these displacements in the opposite direction in order to have full consistency. The program does this automatically if "Construction stage constraints" is selected.

Accumulate stiffness SumLC : The sum of ["N" (normal force) from SumLC (TDV proposes that LC1000 should be used) + the "N" from the current LCs] is taken to adjust the stiffness of the currently active structure by the this resultant normal force.

Note: *Must be in combination with "SumLC" in order to consider correctly all influences of previous loads!*

4.5.3 Proposed Procedure

TDV's philosophy and understanding of the behaviour of this type of structure is based on the experience gathered from several projects. The software design process, RM2000's capabilities and the incorporated functions were influenced by this experience to such a degree that the program reflects the state of Technology Know How for this type of structure at this time. - RM2000 for this procedure must be applied considering this TDV philosophy.

The following 'TDV' procedure divides the analysis into two parts, as previously mentioned:

- the design process on the "Final State System"
- the construction stage design based on the results from the "Final State System"

4.5.3.1 Final State System:

The "Final state" is used for the design of cables, girder and pylon. The subsequent construction stage calculations will end up with the same forces used in the "Final state"

The girder without cables is a continuous beam & will have the same characteristic shape of, say, the bending moment diagram (peak moment at the pylon support & high sagging moment at mid span).

Installing the cables will slightly reduce this behaviour, the support and span moment will be decreased. The cables themselves are very soft and hardly influence the structure.

Stressing the cables will bring the girder moments to the desired order of magnitude. This stressing must be evaluated in the design process.

Note: *The time dependent effects will always have the tendency to creep in direction of the "continuous beam condition".*

The TDV tool for this design process (evaluation of cable forces) is the Unit Load method, function ADDCON (apply ADDitional CONstraint).
The user must:

- Define a unit (stressing) load for each cable (possibly together with another unit load such as support jacking)
- Apply all the permanent loads to the structural model.
- Define the required design components (e.g. the bending Moment at certain points in the girder, a deformation at the pylon tip, etc.) to be aimed for by the program.

ADDCON finds the appropriate multiplication factors for the unit loads to achieve the above design components specified by the user.

All the additional calculations (traffic, additional loads, dynamic…) required to complete the design of the structure can be carried out immediately as this exact result will be reached at the end of the construction stage analysis by using ADDCON!
The whole design is highly influenced by the time dependent effects (creep and shrinkage) it is therefore very important that the time (age) from start of construction for the Final State System is correctly specified.

Note: *It is recommended to use the Load Type "Zero-stress length" (LX0) or the equivalent Load Type „initial normal force" (Load Type „FX0") for specifying the pre-stressing of the stay cable elements, when one or more non-linear conditions are taken into account.*

4.5.3.2 Construction stage analysis:

Every detail of the structures erection procedure (in relation to the time axis!) must be considered in the construction stage calculation.

The final cable forces as well as the element forces (defined as ADDitional CONditions) at the construction stage calculation will, automatically, be the same as those found in the "Final State" System Calculation under the same loading condition.

The same unit loads as defined for the "Final state" must be defined (cable forces – use load type FX0 , and possibly support jacking).

The construction sequence should be modelled in detail including creep & shrinkage. effects.

Note: *Creep & shrinkage will make the calculation non-linear since the change of the unit loads (multiplied by the resulting factor) will change the creep & shrinkage effects as well. It might be more appropriate during the preliminary design phase to use a pure linear (no creep & shrinkage, no other non-linearities) approach.*

Once the desired loading condition for the structure has been achieved using this construction stage analysis, more accurate calculations can be made. All the various non linear conditions (see above table) could be applied – if appropriate – to achieve a more accurate solution.

Note:	**The calculation time will considerably increase with each additional non-linear effect considered.**

There are a very large number of possible combinations for considering non linear effects – individually or in partial combination - beware of randomly applying all the possible combinations of non-linear effects as this may achieve little more than a longer calculation time!

The cable sagging can be considered either by changing the E-modulus of the cables according to the Ernst formula or by calculating “Stay cable non-linear”.

The reduced E-modulus can be retrieved from the Final State Calculation (if cable sagging was considered). This reduced E-modulus (or a modified value of this) can then be separately defined as a constant factor for each Construction Stage by selecting ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION – e.g.:

Select “Insert” or “Modify” – then:

⊙	Calculation (Static)	
➢	CabSag	Cable sagging (correction of E-Modulus for Element Series by factor)
⊙	Element: From,To,Step	Elements to be considered
⊙	Factor	Multiplication factor for E-Modulus, defined by user.

If Cable sagging effects are to be considered in the Construction stage by using

⊙ Stay Cable nonlinear - in the ⇧ RECALC window,

then the cables must first be modified and transferred from the final system geometry (so that they can be assigned an accurate “stiffness matrix”).

Two intermediate steps (Steps 2 & 3 below) must be made between the calculation of the final state system (Step 1) and the calculation of the construction stage sequence (Step 4).

4.5.4 Four Step stay cable geometry adaptation

<u>Step 1:</u> **"DESIGN"** (Final State System)

This step is the same as the calculation of the "final state system" (described above):

- The final system must be active for the calculation (no construction stages!)
- Internal subdivision of the Cable elements must be specified as the definition of the new cables nodes are based on this subdivision.
- All loads must be defined in one single (final) loading case.
  (alternatively, as defined in detail in Step 4, different loading cases, complying with the sequence of load application, can be defined and calculated by selecting
- ⊙ Accumulate Permanent Load in the ⇧ RECALC box)

These conditions are needed for the design of the cables, the girder and the pylon.
Apart from the cross-section design, the most important result from this design process is the determination of the stressing forces of the cables.

The following options are provided in RM2000 to supply these initial forces:

⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LSET:

⊙ Stressing	➢ Cable / external tendon stressing	FCAB
⊙ Initial Stress	➢ Uniform temperature load	T
⊙ Initial Stress	➢ Initial normal force	FX0
⊙ Initial Stress	➢ Stress free element length	LX0
⊙ Actions on Element End	➢ Element end displacements	VGA

We recommend that the initial forces are applied either as "Initial normal force" FX0 or as "Stress free element length" LX0 which are equivalent.

The "Initial normal forces" FX0 should be defined as unit loads for each cable in the design calculation (using ADDCON)

N.B. *The initial nominal "unit loading" to the cable (First Loading Case) consists of two load sets, the first set being the normal "stressing force" (FXO) and the second set being a transverse load (the self weight).Since in a non-linear calculation it is not possible to vary the stressing force "FXO" and maintain equilibrium with the self weight part of the load, a Second Loading Case must be defined that contains the normal "stressing force" –FXO - on its own. This second loading case is then set up as the variable load and is operated on and modified by ADDCON.*

Every newly activated cable therefore must have 3 load sets acting on them in this calculation: an initial force FX0 plus a transverse load forming the first Loading Case and a second Loading Case with the "initial force" defined on its own and having the variable factor (factor updated by ADDCON).

Note: *Generally a non-linear ADDCON calculation can be accelerated by using better starting conditions. These "good" starting values can be found by first running ADDCON with linear cables!*

Summary of Step 1:

- Design of the structure using ADDCON on the "Final State System".
- A single "Final state LC" containing all dead loads and the initial cables forces (the first calculated Loading Case after activation of a non-linear cable must contain one initial force - LX0 or FX0 - and at least one transverse force - e.g. cable self-weight)
- To run ADDCON, more LSet's and LC's with initial cables forces have to be defined. – each of them corresponding to the cable stressing procedure.
- All LC's must be accumulated in LC1000.
- Use *"Accumulate Permanent Load"* for calculation.
- Generally it is helpful to use an increment of 10 for cable-numbering.

Note: *TDV has special macro tools for simplifying the input of large numbers of LSet's and LC's.*

<u>Step 2:</u> "Modification of the stay cable geometry" (first intermediate step)

The cable geometry is modified to the (un-deformed) basic system in this step.
All system nodes must be rigidly supported. - The user should select "step2-nodesupp" to define the "high" spring constants for simulating a rigid support. (see chapter 4.5.1)
The deformation of the subdivided cable is found, by the program, including the displacement at every subdivision point by calculating using this new system and using the loads, defined in Step 1.
These points define the node coordinates of the new cable elements, specified in Step 3.
N.B. The primary node support conditions must be restored before proceeding to the next step (The user should select "step2-undosupp" to restore the old constraints)

<u>Summary of Step 2:</u>

- Calculation at the undeformed system. (needed to find the "undeformed" position of the cable subdivision points.)
- All nodes of the primary structure must be rigidly supported for this calculation.
- The original node support conditions must be restored after the calculation.

Note: *TDV has special macro tools for steps 2 and 3.*

<u>Step 3:</u> "Input of the new cable elements" (second intermediate step)

The full cable lengths in step 1 are replaced by the shorter cable elements from the step 2 cable geometry calculation in this step.

Define the new cable nodes first.

The new node coordinates for these cables comes from the displaced subdivision point of the cables in step 2. - The user can recalculate the coordinates of the displaced subdivision point of the cables from the known coordinates of the undeformed cable subdivision points.

<u>Note:</u> *Nodes between cables need to have rigid conditions in rotation - these nodes, therefore, must be rigidly constrained against rotations (DOF 4,5,6) -. (see chapter 4.5.1)*

The new cables can now be defined and activated between these nodes with the same parameters as the old cables.

The initial forces, defined in Step 1 for the single cable have to be recalculated to one initial load - “stress free element length” LX0 - being subdivided according to the cable subdivision and be given as unit loads to the new cable elements.
The old cables must be deactivated or deleted before proceeding to the next step.
The cables are now ready for the non-linear calculation in the construction stage sequence (step 4).

<u>Summary of Step 3:</u>

- The position of the subdivision points of the cables are recalculated from the result of Step 2 *(LC 1000)* calculation.
- The nodes of the new cable elements are defined
- The new (shorter) cables are between these nodes
- The new “subcables” have the same cross section and material properties as the original full length cable.
- The cable from Step 1 is deactivated and the new cables are activated in the construction stages.

Note: *TDV has special macro tools for steps 2 and 3.*

<u>Step 4:</u> “Construction stage sequence”

The construction stages can be calculated now that the system geometry is defined.

The first loading case calculated on any cable after its activation in this construction schedule must contain the initial cable force. This loading case must also contain at least one vertical load on the cable (generally the cable self-weight).

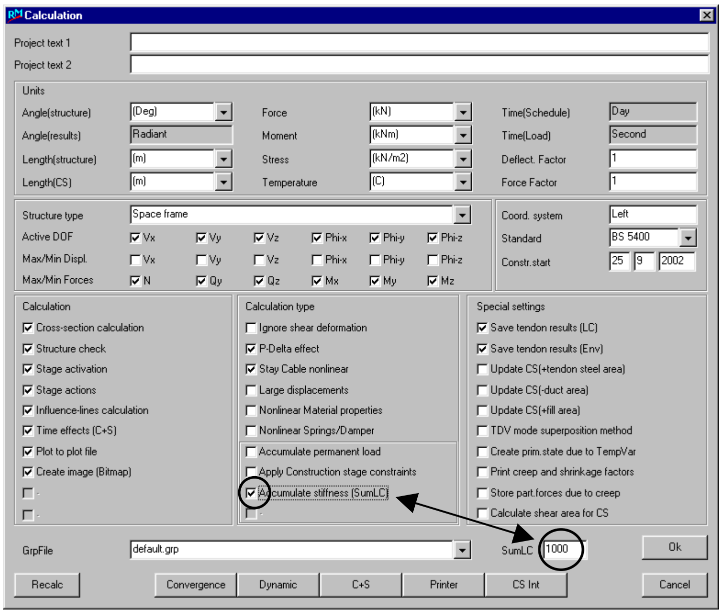
It must be repeated, because of its importance, that the “Stress free element length” LX0 of a cable is input as a load but is to be understood as a characteristic of the cable.

Note: *Initial internal forces do not give a contribution to the load vector in the system of equations. They are only superimposed on the analysis results as they might affect the element stiffness matrices in the case of a geometrically non-linear or in a stability analysis.*

Because the “cable stiffness” is load-dependent it is of particular importance that the load sequence must be in the correct order!

To take into account this stiffness change due to changing load select:

⊙ Accumulate stiffness (SumLC) in the ⇧ RECALC box



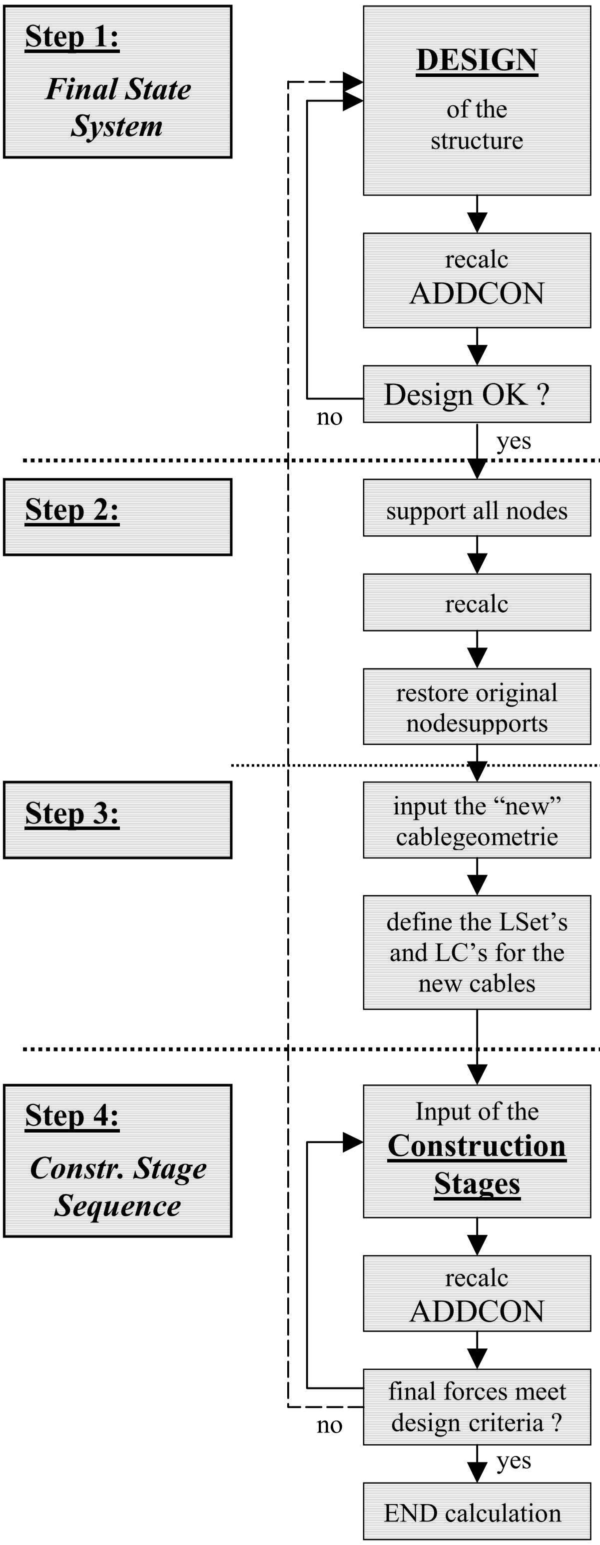
In this case all loads have to be accumulated into one defined load case. (default: SumLC=1000) The load management. (see chapter 6.6) can be used advantageously for this.

If the design conditions for the Construction Stage Sequence and the Final State System are as defined in step 1, then the same ADDCON input can be used for this step (step 4).

N.B a creep and shrinkage calculation it is not *compatible with a selection of:*

⊙ Non-linear Material properties

4.5.4.1 Proposed procedure to use non-linear cable elements in construction stage sequence:



Step 1:

- final state system
- cables with subdivision points
- all loads in one final state LC
- LSet's with initial force and selfweight of the cables
- LSet's with the additional initial force for all cables
- use *"Accumulate Permanent Load"*
- ADDCON converged faster, if the addcon first is calculated with linear cables and after that with nonlinear cables

Step 2:

- same LSet's and LC's as in Step 1
- results for a defined SumLC

Step 3:

- recalculate the position of the cable subdivision points from the result in SumLC of Step 2
- these are nodecoordinates of the new cables
- deactivate the old cable
- transfer all initial forces, defined in Step 1, to a "stress free elementlength" of the old cable
- this length divided through the number of the new cables gives the initial force of the new cables

Step 4:

- activate the cables in the construction stages
- the first calculation on a new activated cable has to contend the initial force and the self-weight of the cable
- use *"Accumulate stiffness (SumLC)"* in the recalc-box

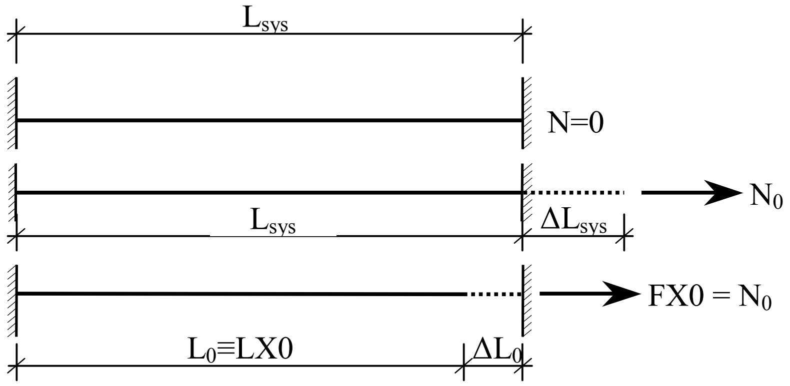
4.5.5 Use of the Load Types FX0, LX0 for Cable Stayed Bridges

The „Stressing Process“ for a cable corresponds physically to a „Shortening“ of the cable. This shortening is simulated by applying an initial force. Several different Load Types are provided in RM2000 to simulate this process. They are specified in the function ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LSET.

⊙ Initial Stress/Strain	➢ Initial normal force	FX0
⊙ Initial Stress/Strain	➢ Stress free element length	LX0
⊙ Initial Stress/Strain	➢ Uniform temperature load	T
⊙ Actions on Element End	➢ Element end deformation	VGA
⊙ Stressing	➢ Cable/external tendon stressing	FCAB

Only the the use of the recommended Load Types „FX0“ and „LX0“ is discussed in this chapter, the other types are typically used in other contexts.

4.5.5.1 Relation between FX0 and LX0



The system length L_{sys} of a cable element is generally defined as the straight distance between the start and end points (i.e. the start and end nodes if no eccentric connection is defined).

Stressing this cable with the force N_0 yields an elongation ΔL_{sys}. Applying the Load Types FX0 and LX0 means placing this stretched element with the original length into the structural system, i.e. the elongation is cut away and the original (stress-free) length is assumed to be smaller than L_{sys}.

This reduced length is called ***„stress-free length L_0“*** (≡ LX0).

Relation:

$$\mathbf{L_0 = \frac{EA \cdot L_{sys}}{EA + N_0}}$$

with: E ... E-Modulus of the cable
A ... Cross-section area of the cable
L_0 ... stress-free element length (=LX0)
L_{sys} ... un-deformed element length
N_0 ... initial normal force (=FX0)

The ***„initial normal force FX0“*** (=N_0) is defined as the force, which is required to stretch the cable with the original length L_0 by the amount of ΔL_0 in order to reach the system length L_{sys}, which is necessary to place the element in the un-deformed system.

Relation:

$$\mathbf{FX0 = \frac{EA}{L_0} \cdot (L_{sys} - L_0)}$$

with: E ... E-Modulus of the cabel
A ... Cross-section area of the cable
L_0 ... stress-free element length (=LX0)
L_{sys} ... un-deformed element length
FX0 .. initial normal force

This shows that the 2 Load Types are totally equivalent.

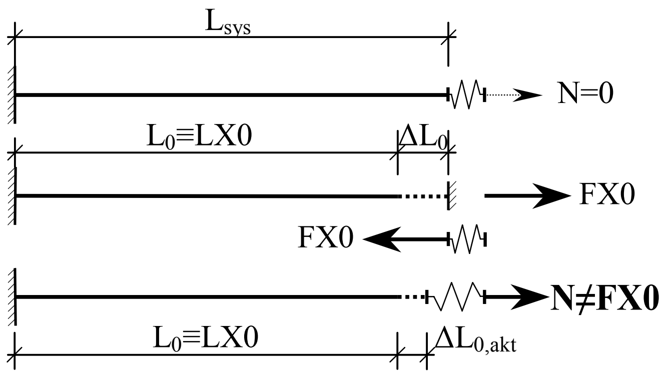
The load type „initial normal force FX0“ is a further possibility of defining a reduction of the original length with respect to the system length L_{sys}.

4.5.5.2 Relationship between FX0 and the final normal force N in the cable

One of most frequently asked questions is, why - after calculating a load case with FX0 - **the resulting normal force is not the same than the initial force FX0**.

The answer is, that this is due to the flexibility of the remaining structural system. In the special case, that the support on both ends of the cable is absolutely rigid, the applied normal force remains in the cable (i.e. FX0=N), otherwise the reaction of the structural system causes a relaxation in the cable and N will be smaller than FX0.

The following example simulates the flexibility of the structural system by one spring (with spring constant *k*) applied in one of the 2 nodes:



A length reduction is applied by using the Load Type FX0 to the element with the system length L_{sys}. This causes a force FX0 acting on the structural system represented by the spring. The system is deformed by this force (the spring is stressed), causing a reduction of the actual length of the cable ($L_{akt} < L_{sys}$).

This means, that the effective elongation ($\Delta L_{0,akt} = L_{akt} - L_0$) is smaller than the applied initial value ΔL_0. The remaining normal force in the cable is the force, which is necessary to strain the cable by the amount of $\Delta L_{0,akt}$ instead of ΔL_0.

Equilibrium exists when

$$N = EA \cdot \frac{\Delta L_{0,akt}}{\Delta L_0} = k \cdot \left(\Delta L_0 - \Delta L_{0,akt}\right)$$

We see, that due to the deformation of the primary structural system the initial normal forces FX0 does not anymore act in the cable, but a reduced force corresponding to the equilibrium condition.

If, as it is often the case, a predefined normal force should act in the cable after the stressing process (stressing process against the structural system), then it is necessary to use an iterative process to define the stressing load by LX0 or FX0. TDV provides in RM2000 the function ⇧LOADS AND CONSTR.SCHEDULE ⇨ADDCON (see chap. 6.11 Additional Constraints).

4.5.5.3 Load Type FCAB

Cable jacking simulation in RM2000 can also be performed by using the Load Type FCAB. This function does simulate the real physical procedure including the structural response on the jacking force application. In simple linear calculations you get in fact right results without iteration, and the applied normal force will fully be in the cable after the load case has been calculated. However, this load type is not applicable in non-linear analyses, and – due to the fact, that stressing a cable influences the force in all previously stressed elements - for determining the required jacking forces with ⇧LOADS AND CONSTR.SCHEDULE ⇨ADDCON to have a certain normal force in the cable after all cables have been jacked.

The following procedure is performed internally in the program: The stressed element is extracted from the system (deactivated) and replaced by the specified force acting at the begin and the end of the cable. This calculation does not give a result for the considered element itself, because it is not active. Then the cable element is – without a new redistribution analysis - with the prescribed normal force as result value again integrated in the non-deformed system.

A possibly known cable force can therefore be directly introduced in the structural system. Other than in function FX0, the specified force will be in the cable after the calculation of the Load Case.

This Load Type must not be used for complex, non-linear calculations (especially not for non-linear cable elements). Cable sagging, large displacements, 2nd order theory etc. are not compatible with FCAB. Due to the cable element being inactive, the consideration of the non-linear behaviour will be incomplete.

4.5.5.4 TDV Recommendation – conclusion

In the case the cable force is known (checking a certain situation), then FCAB is applicable if the non-linearity is not considered. In the case that the erection sequence respectively stressing strategy should be evaluated, then FX0 should be used together with ADDCON. FCAB is not applicable in this case, because every cable stressing Load Case is calculated on a different system and the interaction is not taken into account.

Also in the case of a non-linear behaviour FCAB should not be used, because the used cable end forces are related to the direction of the original cable and a rotation of the normal force is not considered.

4.6 Suspension Structures

The following chapter shows the required steps for calculating a structure using the 3rd order theory (large deflection theory) which is entirely implemented in the program.

4.6.1 General

Some basics need to be clarified before starting to calculate a suspension structure:

- ➢ The superposition rule is no longer applicable (results from different construction stages can not be accumulated)
- ➢ Each construction stage changes the global stiffness of the new structure dramatically and has therefore its own geometry.
- ➢ As a consequence it is also clear that loads applied in an earlier stage have a different result in following stages (in case of construction stages). All loads for the current stage have to be recalculated for each stage again. That means that the self weight of the elements representing stage 1 is considered in stage 2, 3, etc as well. For each stage we get an individual result. A total envelope for all components can be created to find out the worst situation of forces and displacements for each element.

All these necessary definitions are done at certain positions in the program. Each definition is done interactively in the program, an ASCII file of all definitions can be created for eventual Editor – modifications when selecting ⇧ FILE ⇨ EXPORT. The names of the files are mentioned in the following description as well.

The structure is prepared in the common way. The required calculation rule (3rd order – large deflection) is selected when starting the calculation with ⇧ RECALC. All results are finally available as usual.

More about loading:

Because of large displacement feature element can change its orientation in space (angles BETA, ALFA1, ALFA2).
Some extra definitions are needed:

1.) X/L loading position along element local X-axis remains constant.
2.) Locally defined components of loading and eccentricity *ey, ez* are automatically redefined corresponding to the new element orientation.
3.) Globally defined components of loading and eccentricity *ey* and *ez* remain constant.

More about results:

Element results of large displacement analysis are those corresponding to the new structure geometry under the load and **not to reference structure** before loading:

1.) Element displacement results do not consider the 'rigid body large displacement' part but only the 'small distortion result' part.
2.) Element forces are those corresponding to new element position.
3.) Node displacement results consider the total large displacement result (difference node position before loading – node position under load).

More about convergence:

The calculation uses the Newton-Raphson-method.

$$^{m}\psi = {}^{m}[k_{\delta}] \cdot {}^{m}\{\delta\} - {}^{m}f_{\delta} \qquad (1)$$

$$^{m}\Delta\delta = {}^{m}\left[k_{\delta}{}^{T}\right]^{(-1)} \cdot {}^{m}\psi \qquad (2)$$

$$^{m+1}\{\delta\} = {}^{m}\{\delta\} + R \cdot {}^{m}\Delta\delta \qquad (3)$$

The meanings of the variables are:

m	= current iteration number
ψ	= equilibrium of internal and external forces
$[k_{\delta}]$	= displacement dependent stiffness matrix
$\{\delta\}$	= displacement vector for current stage of approximation
f	= displacement dependent external forces
$\Delta\delta$	= improvement of displacement
$[k^{\delta}{}_{T}]$	= displacement dependent tangent stiffness matrix
R	= relaxation factor

The solution process is stopped under the following conditions:

m	>	n_{max}
TOL1	<	$\Sigma^{m}\psi^{2}{}_{i}$
TOL2	<	$\max\left\|{}^{m}\psi_{i}\right\|$
TOL3	<	$\Sigma^{m}\delta^{2}{}_{i}$
TOL4	<	$\max\left\|{}^{m}\delta_{i}\right\|$

4.6.2 Explanation

The following small example shows, how a structure can be calculated according to the theory of large displacements.

The goal is: Geometry and internal forces of structure under load

using ⇧SYSTEM ⇨NODE and ⇨ELEMENT a reference geometry is entered, the closer to final geometry under load the better.
The geometrical behaviour can be influenced using the initial forces as a load type for the cable elements (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET load type 'Initial Normal Force') in that way that the geometry under load is as wanted (the tolerance is up to the user). The result is 'true geometry of structure under load'. The more the reference geometry is inaccurate, the clearer the manipulation of the cable forces.
All loading case combinations have to be prepared as loading sets because of the invalidity of the superposition law. Any security factor should be considered as load multiplication factors. A loading set combination as [self weight + delta temperature + Wind] will form one single loading case.

A calculation can be for instance as follows:

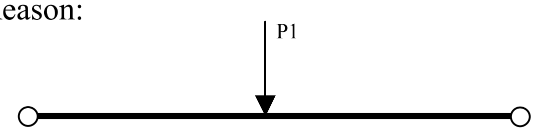
[reference system] + [self weight] + [wind] + [temperature] = [new geometry]

but not like this:

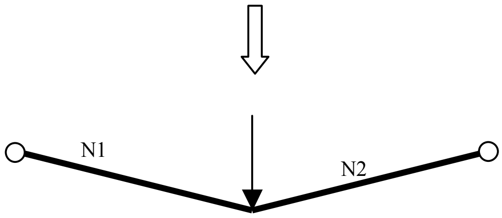
[reference system] + [self weight.] = [geometry 1]

[geometry 1] + [wind] + [temperature] = [geometry 2] ≠ [new geometry]

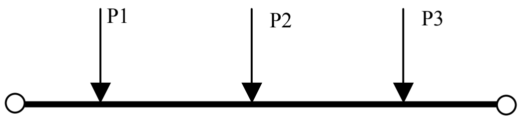
Reason:



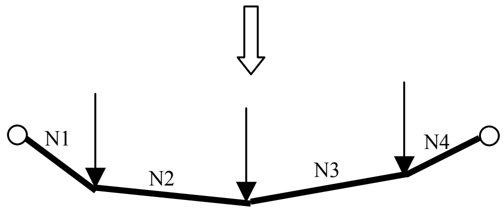
a) LC [P1] results; displacement and N in cable



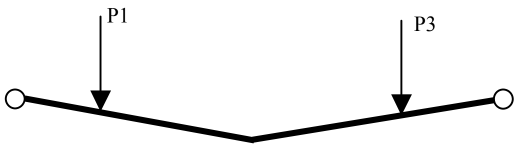
b) Result: GEOMETRY 1



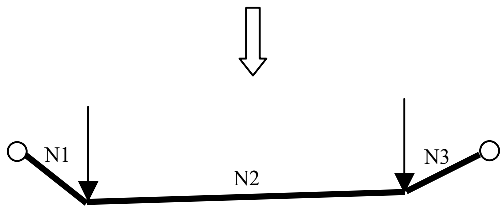
c) LC [P1/P2/P3] results: displacements and N in cable



d) Result: GEOMETRY 2



e) LC [P1//P3] applied on GEOMETRY 1 results again in displacement and N in cable



Result: GEOMETRY 3 and **not** GEOMETRY 2

That means, all load components have to act on the structure the same time!
When all load cases (containing certain load sets) are calculated, an unfavourable combination of the internal forces can be done.

4.6.3 System Definitions for Suspension Structures

To find out the correct geometry of the structure at the final situation (geometry under all dead load) is in most of the cases the first step. That means that it is necessary to find out the equilibrium between the stiffness of the structure and all permanent load. When applying all permanent load on the structure we need to get ‘0’ deformation resp. no difference between defined structure and structure under load.
This predefined geometry of the structure will be called ‘reference geometry’ in the following.

All cables in the structure will get a certain ‘initial force’ which is used to define a cable shortening. The cable shortening is used as a geometry correction value to identify the correct reference geometry.

A calculation of a suspension structure without these ‘initial forcers’ for all cable elements is not possible. These ‘initial forcers’ have to be considered in all stages for any loading case to be calculated.

The definition of the ‘reference geometry’ is not supported by the program. The user has to identify the cable geometry himself by using any other tool like Excel for instance.

4.6.4 Reference Geometry

A possible approach to define the final geometry under load (‘reference geometry’) can be as follows:

- Create a model for the beam elements (girder) only and define a spring element for each point where a hanger cable is connected (vertical stiffness only).
- Define all other support conditions as in reality.
- Define all loadings representing the final situation where all permanent loads are acting.
- Calculate this loading case (self weight + all permanent load)
- The resulting support reaction in the spring elements representing the cable attachments is the normal force that will be applied to the cable element for this loading.
- This normal force (plus the hanger self weight) can than be used as the hanger component when defining the geometry of the main cable.

4.6.5 System Parameters

Elements representing the cables in the structure need to be identified as being cables (⇧STRUCTURE ⇨ELEMENT ⇩ELEM - Select 'Cable' for 'Element type'). All necessary hinges at the end of the cables are automatically considered by the program, all rotations (φ_x, φ_y, $\varphi_{z)}$ of the main cable elements need to be fixed (1E10).

The user defines an area (A_x) only for all cable elements (no inertias) as well as any wanted material (⇧STRUCTURE ⇨ELEMENT ⇩CS and ⇩MAT).

All nodes of the cable representing the connection of the main cable with an hanger has to be defined by coordinates. No additional nodes between these main cable – hanger – nodes are requested (element subdivision is possible).

All other elements can be defined as usual (beam elements with any cross section and material properties or spring elements).

4.6.6 Load Input for Suspension Structures

All loads acting on a certain structure need to applied as one loading case as already mentioned. All loads are prepared separately in Load Sets, a certain combination of these load sets will then form a loading case (maybe containing multiplication factors for the load sets). The load set containing the 'Initial forces' for the cable has to be considered in all loading cases, otherwise the geometry would be wrong.

4.6.6.1 Load Sets

Load sets are defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨ LOADS ⇩LSETS. The upper table shows all existing load sets. A new load set is defined by selecting the appropriate line in the upper table and by selecting either the 'Insert before' or 'Insert after' button.
Any number for the load sets as well as a general description are requested.

The lower table shows the actual loading for the load set selected in the upper table. To define the 'Initial forces' for the cable elements we can add a new load by selecting the appropriate line in the lower table in by selecting either the 'Insert before' or 'Insert after' button.
Select

- ⊙ Initial normal force as a load type
- ➢ And 'Initial normal force as a sub type again

A new window appears asking for the actual loads for certain elements

- El-from the load will be
- El-to applied to this
- El-step series of elements
- Fx Normal force

All other relevant loadings need to be specified in the usual way by using any other load type. There is no limitation of load types for a suspension structure.

All existing load sets can then be combined to form loading cases.

4.6.6.2 Load Cases

Select ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE to create new loading cases or to modify existing ones.
The upper table shows all existing loading cases. A new loading case is defined by selecting the appropriate line in the upper table and by selecting either the 'Insert before' or 'Insert after' button.
A new input window appears asking for the following input:

- Number Number of the new loading case (see TDV's load numbering scheme)
- Type Define type of load. A multiple choice is available by clicking the arrow symbol next to the input field
- ⊙ Load the (later) defined Load Set(s) are acting as a load
- ⊙ Load+Unload the (later) defined Load Set(s) are loaded and unloaded again (e.g. temporary traffic load)
- Load Info related Load Info previously defined in LMANGE
- Location Name of automatically assigned ASCII input file
- Output File Name of automatically assigned ASCII output file
- Description Descriptive text (max. 80 characters)

The lower table shows all assigned load sets to the selected loading cases in the upper table. A new load set is assigned to a loading case by selecting the appropriate line in the lower table and by selecting either the 'Insert before' or 'Insert after' button.
A new input window appears asking for the following input:

- Load Set Number of the new load set to be assigned to the currently selected loading case.
- Const-Fac Constant multiplication factor for this load set.
- Var-Fac Variable multiplication factor which will be added to Const-Fac (dynamic calculation, see chap. 9.7)

The difference between a loading case for 'stage n' and a loading case for 'stage n+1' is very often only the new self weight for the new elements (in case of construction stage calculation). The easiest way to generate the loading cases for all stages is therefore to copy one loading case again and again and to add the appropriate new loading sets for the new stage.

4.6.7 Calculation of Suspension Structures

Check whether the following necessary definitions are available before starting the calculation with ⇧ RECALC:

- Cable elements are specified in the structure
- The loading set 'Initial force' is specified and appears in all loading cases
- All loading cases for all stages contain always the complete loading

Select ⇧ RECALC if everything is available.
The already described general calculation pad appears where the following buttons are important for a suspension structure:

⊙ P-Delta effect is selected
⊙ Stay cable non-linear is selected
⊙ Large displacements is selected

Additional information for the convergence control of the calculation can be defined. Select ⇩Convergence
to open a new window where the constants for the Non-linear calculations are to be set:

- ➢ Relax Relaxation factor for the Newton Raphson Iteration (Default 1).
- ➢ Niter Maximum number of iterations to stop the iteration
- ➢ Tol-1 Tolerances
- ➢ Tol-2 for the
- ➢ Tol-3 control of
- ➢ Tol-4 the iteration

The calculation can be started now by selecting ⇩RECALC.

4.6.8 Traffic Load on Suspension Structures

The calculation of traffic load is a delicate due to the non-linear behaviour. Principally it would be requested to calculate all relevant load positions individually in order to consider the different stiffness for different load.

The amount of loading cases is quickly very high and the amount of work necessary for the preparation of all the loading cases as well.

The relationship between <u>permanent load</u> and <u>traffic load</u> needs to be estimated. If this relationship shows that the traffic load is having a relatively small compared to the dead load and the structure is not showing a very high non-linear behaviour it is possible to use a certain simplification.

The program allows the user to identify a certain reference load (which is usually all permanent load at the final stage) to establish a tangent matrix. The stiffness of the structure for this reference load is then used to run the traffic load calculation as usual be use of influence lines and also the superposition law.

A comparison between the resulting values of the traffic load envelope and a single loading case (all permanent load + one traffic position) has to be done in any case to verify the values and to find out if this simplification is applicable or not.

If this simplification can be applied it is sufficient to define the reference load case when adding the Influence line calculation into the ⇧LOADS AND CONSTR.SCHEDULE.

Supposing that all Lane definitions and all Load trains are available (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LANE and ⇩LTRAIN) the following actions can de added into the LOADS AND CONSTR.SCHEDULE. Select the appropriate construction stage in the upper table (⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION) and also the appropriate line in the lower table for the selected stage. Use either the 'Insert after' and 'Insert before' button to add the action for influence line calculation.

The following input is requested in the appearing window:
- ⊙ Calculation Actions and
- ➢ Infl

The parameters then required in the new input pad are:

➢ Command	The user selected command is displayed.
➢ Lane-number	Select the wanted lane for the calculation of the influence lines (interactive selection possible by clicking the arrow symbol next to the input field).
➢ Reference LC(-)	Define the loading case containing the load for the structure that is taken for the creation of the 'tangent matrix'.
➢ Influence-file(*.inf)	Name of the binary output file containing the influence lines. An '*' stands for 'default' – a file called 'lane0001.inf' will be created for lane 1.

- List-file — Name of the ASCII output file containing the influence lines. An '*' stands for 'default' – a file called 'lane0001.lst' will be created for lane 1.
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

The new action will be displayed in the lower table.

The action for the live load calculation using the now available influence lines needs to be added together with all the other eventually necessary superposition actions.

4.7 Incremental Launching Method (ILM)

4.7.1 General

The erection procedure for an incremental launching bridge requires specific support from the software for several reasons. The amount of construction stages is much higher than for any other erection procedure, the launching process itself must be simulated to get the typical envelopes of results and finally the structural model must be completed by additional elements such as launching nose, casting yards and temporary supports (spring elements).

All this is supported by the program. RM2000 takes care that all the construction stages from positioning the launching nose at the starting point to the final position of the completely launched bridge deck are generated based on a user defined launching concept. A couple of specific data is necessary to define in addition to the usual definitions, as already mentioned. The following description will illustrate these specific additional definitions.

4.7.2 System preparation (GP2000 and RM2000)

The preparation of the general structure (final situation) is done in the usual way using the features of GP2000 and (possibly) RM2000.

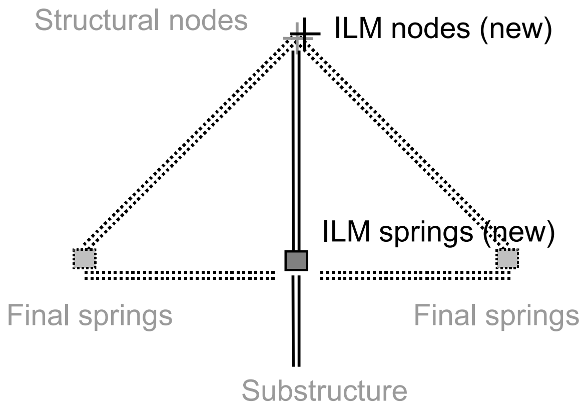
The user should be aware of the fact that the subdivision of the structure into structural nodes and structural elements needs <u>not</u> be laid out for the launching sequences. The segment lengths however need to fit with the user defined element subdivision. It is part of the ILM features that the structure will automatically be adjusted for all necessary intermediate situations.

4.7.3 Conditions to be considered

- The road alignment in plan view must at the moment be straight (no curves allowed). The extension for curved structure is under work.
- The road alignment in elevation is supposed to be straight (horizontal).
- The cross section height should be constant.
  Variable cross section heights are possible if the variation is stepped element per element.
- The bridge deck's structural nodes need to be on one level (same Y – coordinate, most frequently at top of cross section. This is also valid for the launching nose!
- The specific ILM – supports (spring elements in addition to the ‚ordinary' supports) should be located at one level (same Y – coordinate) as well.

4.7.4 Required Additional System Definitions

- Definition of the launching nose (Elements, Nodes, Cross section and material properties)
- Additional spring element at end of launching nodes to stabilise the structure during launching against torsion, transverse and longitudinal displacement.
- Additional spring elements for the launching procedure itself.
  The substructure needs to be coupled with the superstructure using spring elements. These specific spring elements are defined using the already existing nodes (existing since the substructure is supposed to be already defined) and nodes at the same location as the real structural nodes. Since the structural nodes remain at the same position and cannot be launched, it is necessary to specify additional nodes at the same location (at the final position of the piers) but without any connection to the real structural nodes. See sketch above.
  These springs (and nodes) can as far as possible be defined in GP2000. Elements located outside the structure (casting yard) can of course be defined in RM2000 directly.



- The spring constants of the specific ILM supports should be defined as follows:
  Cx = 1e10, Cy,z = very small to avoid constraints, Rx,y,z = very small to avoid constraints.

Pre-stressing (Tendon definition, stressing sequence, etc.) as well as Creep & Shrinkage are defined in RM2000 in the usual way.

4.7.5 Construction Schedule – Preparations (RM2000)

For the user there are actually only minor differences to be considered in case of an ILM simulation. All spring elements (real supports and ILM springs) are activated and deactivated in the usual way in the right order as well as the removal of the launching nose at the end of the launching procedure.

All ILM supports are being shifted at the right location according to the launching procedure by the program.

The user defines all the „real“ construction stages. These “real” stages are the ones where the newly launched segment has arrived at the right location. In these user defined stages all pre-stressing actions, creep & shrinkage as well as all possibly necessary design code checks (fibre stress check, etc.) are defined.
The launching itself is handled by the program.

4.7.6 Necessary additional Construction Schedule definitions:

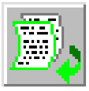
- After the user defined activation of a new segment an empty stage must be defined. Empty means: no activation, no action required from the user. These empty stages will be used for the ILM simulation later.
- Empty LoadSets need to be set up for the ILM loading situation as well. (maybe the 300++ series according to our recommended numbering scheme).
- Empty LoadCases need to be set up for the ILM loading situation as well. (maybe the 300++ series according to our recommended numbering scheme).

4.7.7 Launching – Definitions (RM2000)

All data is defined under SYSTEM / ILM.

4.7.7.1 SEGMENTS

The upper listing shows all individual segments that will be launched.



Use “Insert before“ or „Insert after“ to create a new segment. Note: order in the listing is important!

The structure is split into segments, the relation between the segments is defined as well.

Segments:

- Nose segment — Needs to be the first definition! All other segments belong to one nose.
- Girder — New segment that will be added to structure and that will be launched
- Fix Support — Piece of structure that will not be launched, but which will be used for the launching (piers, or support springs).
- Temp. Support — Piece of structure that will not be launched, but which will be used for the launching (piers, or support springs).

Possible definitions:

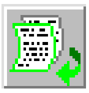
Segment name:	Each segment is from now one identified by its name (e.g. “A”, max 23 characters).
Nose segment:	A girder segment belongs to one already defined nose segment that must be defined here. Only necessary for girder segments! In case the launching is done from one side only there will only be one nose. This only nose will be defined for all girder segments.
Spring:	Element number of support spring between deck and substructure (ILM springs)
Nodes:	Node number (ILM – Nodes) belonging to the ILM springs, usually situated on top of cross section, identical to strucutural nodes respectively.
NewNdNo:	The element subdivision does mostly not fit into the launching steps, but since each step requires a node and an element the program will subdivide the user defined structure into a much finer subdivision wherein all necessary intermediate launching steps are considered. These new nodes and elements will be created and numbered by the program, the starting number of the nodes are defined here. Begin and end nodes of the original element are not changed.
NewElNo:	Same as for „NewNdNo“, but for starting element number.

In case the option “Continuous“ is set, all segments starting with the corresponding launching nose segment are renumbered. As a result the user will get a continuous numbering of new nodes and new elements (TDV recommends to use this option, it will make the definitions for the final situation much easier thinking about traffic lane definitions, plotting, etc.)

Tolerance: The intersections of piers (supports) and the launched segments need to be found during an ILM calculation. In case one of several ILM supports are used at the same longitudinal location (eccentric bearings, one left and one right) a tolerance for the Z – direction (transversal) needs to be defined so that all ILM springs are found.

Description: General text for identification (max. 127 characters).

The lower listing shows all elements that will represent the individual segments. (SEGMENT – OLD).



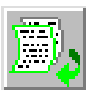
Use „Insert before“ or „Insert after“ to assign elements to the created segments.

SEGMENT - OLD Input of elements belonging to the segments. An element can only be member of one segment! After inserting new elements it is now possible to apply a Y – coordinate (precamber) to the structure. “Modify“ will allow this input..

SEGMENT - NEW Elements that will be generated by the program will be listed in this column after the calculation. This is a result, no input required.

4.7.7.2 ILM

The launching steps and their order are defined here.



Use “Insert before” or “Insert after” to define the launching of a new segment.

Input of launching:

- Nose position Before any segment can be launched the nose must be put into the correct starting position.
- Segment moving Once the nose position is defined, the segments belonging to this nose can be launched.

Possible definitions:

Segment name: User defined name of segment to be launched.

Number of steps: Number of steps to get the wanted step length. (e.g. 20 steps per segment). Input required for "Segment launching" only. According to this definition the subdivision of the new "ILM" structure (new elements, new nodes) will be generated. The step therefore defines the element length of the newly established structural system.

Moving length: Total length of launching (e.g. segment length). "Launching length" divided by "number of steps" gives to final stepping length.

Stage No.: Number of existing empty construction stage as defined in the RM2000 project (project in the original directory).
After calculating the ILM definitions the program will generate all ILM actions in the corresponding stages automatically.

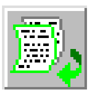
LCase No: Number of existing empty Loading Case as defined in the RM2000 project (project in the original directory).
After calculating the ILM definitions the program will generate all ILM loadings in the corresponding loading cases stages automatically.

LCase No: Number of existing empty Loading Sets as defined in the RM2000 project (project in the original directory).
After calculating the ILM definitions the program will generate all ILM load data in the corresponding loading sets stages automatically.

The lower listing allows the definition of additional RM2000 actions if necessary to complete the launching simulation..

ILM-ACTIVIATION All found intersections of girder/nose with substructure are listed here.

ILM - ACTION Each step requires a series of specific ILM actions that are listed here. Note that the special requirements for ILM bridges (e.g. lifting of nose to reach support) are generated by the program as well.


Use „insert before" or „ Insert after" to add any user wanted action into the action listing. (Plotting of results, producing listings, stress checks, etc.)

4.7.7.3 New project, TCL, Recalculate

NEW PROJECT	A new ILM session can be started.
TCL	Export and Import of ILM – data (tcl Format).
RECALC	This button will interpret the data in the described way. A new element subdivision will be found, actions and activations will be set up, etc. All this will be transformed into a new data base. The user can also choose between “System check“, “Calculate Element subdivision“, “new numbering“ or “Export in new directory“. A selection of the wanted new directory can also be done, default is subdirectory with name ILM.. The summation of all dead load into one loading case should also be done here (SumLC), Loading case number 1000 is recommended by TDV. The calculation can take a while since a huge amount of data will be read and generated.

4.7.7.4 END

Finish ILM input.

In case the calculation could be done successfully (no error messages) the user can close the current RM2000 session and restart RM2000 in the generated new directory.
The calculation can be performed immediately since all empty LSETS, LCASES and STAGES have been filled up with the necessary input.
Please note the all further analysis such as traffic load need to be performed on the new system in order to guarantee compatibility of the results.

5 Pre-stressing

5.1 General

Two different types of pre-stressing of structures are principally known:

- **„Internal“ pre-stressing,** where the pre-stressing tendons are installed in ducts poured into the concrete cross-section and
- **„External“ pre-stressing**, where the tendons are located outside the concrete cross-section.

The internal pre-stressing is in the following generally meant whenever the term „pre-stressing“ is used without a special indication.

The simulation of internally pre-stressed structures is done by assigning „pre-stressing tendons“ to the structural elements. The physical parameters (material, cross-section area, ...) of these tendons and their location within the cross-section of the structural elements have to be specified.

Tendons with the same geometric and physical parameters may be grouped together in „tendon groups“ also called “Tendon Profiles”. However, the expression „(pre-stressing) tendon“ is for simplicity often used in this manual for describing features equally related to Tendon Profiles than to single tendons.

Note: *Grouping together several cables essentially eases the input. It is obviously allowed if more than one tendon with same geometry are arranged side by side in the cross-section and only vertical loading occurs. It is however often possible to group together with sufficient accuracy also tendons arranged on top of each other. A tendon geometry related to the centroid of the tendon group is used in this case and the moment of inertia of the group with respect to the centroid is neglected.*

Performing the pre-stressing analysis of a pre-stressed structure requires the following definitions and actions:

Definition of the pre-stressing tendons:

- ➢ Material properties for the tendons
- ➢ Cross-sections of the tendons and numbers of tendons in the tendon groups
- ➢ Cable geometry for the tendons
- ➢ Structural elements assigned to the tendons

Stressing process:

- ➢ Definition of the type and sequence of stressing actions
- ➢ Calculating the related impacts on the structural system

Determination of the system reactions:

- ➢ Assigning the stressing process to the pre-stressing Load Cases
- ➢ Calculation of these Load Cases in the construction sequence
- ➢ Grouting of the pre-stressed tendons

These definitions are made in different positions in the program. The definitions is made interactively in the GUI. An ASCII file containing these definitions can be created by selecting ⇧FILE ⇨EXPORT. This file can later be used for performing the input process by importing the data from file.

5.2 Material of Pre-stressing Tendons

A material type must be assigned to each tendon. This material must be defined in the material table in the project database together with the required parameters. Some specific additional material parameters described below are required for pre-stressing materials besides the standard parameters for static analyses.

Entering new materials into the material table is described in detail in chapter 3. The group name „Prestr. steel“ must be assigned to materials used for pre-stressing tendons.

Example for material definition:
- ➢ Name of the material (e.g. VT 16-100)
- ➢ Descriptive Text (e.g. tendon 1)

Material group
- ⊙ Prestr.steel

The following material parameters are required for pre-stressing tendons:

- Static calculation: E-modl (longitudinal E–modulus, e.g. 1.95E8 kN/m²)
- Prestr. process: E-Modex (E-modulus for extension calculation, e.g. 1.95E8 kN/m²)
- SIG-allow-pr (max. tensile stress in the tendon after the stressing process (eventually including wedge slip); e.g. 1.315e+06 kN/m²
- SIG-allow-SA (max. tendon stress in the final state after creep and shrinkage under the live loading, e.g.1.315E6 kN/m²)

E-Modl The modulus of elasticity of the pre-stressing steel is used for computing the composite cross-section after grouting and determining the stresses respectively tension forces in the tendons due to loading cases applied after grouting. It is generally assumed that tendons consisting of winded strands have a smaller stiffness than the material itself, even after grouting. The modulus of elasticity of steel is therefore mostly slightly decreased in this context, typically to 2.05E8 kN/m^2.

E-Modex An additional tension flexibility of the tendon is to be considered for computing the cable extension in the stressing process. This additional flexibility counts for considering lateral deviations of the tendon within the duct and effects due to differential displacements between the strands etc. Usually a value of 1.95E8 kN/m^2 is used for tendons consisting of winded strands.

SIG-allow-pr This allowable stress value is a reference value used in the stressing actions for limiting the tension force at the end of the stressing process (see chap. 5.6.2). It is only used if the tension force is not directly specified, but as a factor of an "allowable tension force". This allowable tension force is then computed by multiplying SIG-allow-pr with the cross-section area of the tendon. The notation of SIG-allow-pr is in the Austrian code σ_{pm0}, and required to be approximately 0.7*f_{pk} (70% of the characteristic value of the tension strength). A typical value is 1.315E6 kN/m^2. The basic value f_{pk} is usually fixed in the approval documents of the used pre-stressing system.

SIG-allow-SA SIG-zul-GZ ist die Grenzspannung im Spannstahl für den Endzustand nach Kriechen und Schwinden unter maximaler Gebrauchslast (Bemessungswert = „seltene Einwirkungskombination“). Der Wert ist in der ÖNORM B4750 (11.4.5(4)) SIG-allow-SA is the limit stress in the maximum serviceability load state (reference value in accordance with the design code) in the final state after creep and shrinkage. The value is given in ON B4750 with 0.7*f_{pk}, i.e. the same value than SIG-allow-pr. (1.3150E6 kN/m^2).

XI The parameter XI is used in the crack propagation check in accordance with Austrian code ON B4750. This check is actually not performed for other design codes. XI (ON-notation k_b) describes the adhesion behaviour between concrete and steel. The value 1.0 indicates full adhesion and 0.0 is used for tendons in ducts without grouting. The Austrian code does never allow to assume full adhesion.

The required values in the design code are for post-tensioned tendons after grouting:

- 0.2 for smooth bars
- 0.4 for shaped bars or strands
- 0.6 for ribbed bars

For immediate adhesion (pre-stressing in a pre-stressing bed) the values are:

- 0.6 for shaped bars or strands
- 0.8 for ribbed bars

5.3 Definition of Tendons (Tendon Profiles)

5.3.1 Creating New Tendon Profiles

The tendon is part of the structure and the geometry for all tendons is defined in ⇧STRUCTURE ⇨TENDON. The tendon table is shown in the upper part of the screen after selecting this function. The parameters related to the tendons are:

- physical parameters (material, cross-section area, number of tendons, ...)
- assigned structural element series
- tendon geometry

The physical parameters are directly displayed in the corresponding line in the tendon table (upper table in the GUI window). The lower table, depending on user selection, either shows the related element assignment table or the tendon geometry table. Two different presentation types are available for the tendon geometry table (see chap. 5.4.3).

A new tendon is created by pressing the ‘Insert before’ or ‘Insert after’ button above the tendon table. The related input pad asks for the following parameters:

- No. of the new Tendon Profile
- Type of the new tendon (internal or external)

Note: *A tendon is defined to be “external” if it lies over the whole length outside of the cross-section of the structural elements and does not contain sections, where it is inside in a later grouted duct. Tendons with both, grouted sections inside the structural element cross-section and free external sections in between, must be defined as “internal” tendons.*

- Material name (interactive selection from the material table)
- Number of tendons in the new tendon group
- Steel cross-section area for one tendon of the Tendon Profile
- Duct cross-section area for one tendon of the Tendon Profile

Note: *Steel area and duct area are cross-section parameters and therefore measured in the unit* ***[Length(CS)]^2*** *and not in the unit [Length(Structure)]^2.*

- Friction coefficient μ (for external tendons only for the region of the turning blocks) (tangent of the friction angle)
- Accidental deviation value **β (not K!)** (in [angle unit per length unit]), describing the wobbling of the tendon. (For external tendons only for the region of the deviator blocks)
- Name of an ASCII-file containing the tendon data (is automatically created by the program, no user definition possible).
- Descriptive text (optional)

Attention:	**Details of the calculation of friction losses are given in chap. 5.4, Stressing Procedure. Note, that in literature and design codes in the German world the Accidental Deviation Angle β, measured in [degrees/meter], is commonly used to describe the wobbling of the tendons, whereas the Wobble Factor $K = \mu * \beta_{rad}$ is used in England and the USA. If the factor K is known, then the value β must be correctly determined before entering it into the program database. If – as it is often the case – degrees are used as angle unit, then the value of β to be entered will be** $\beta_{degrees} = (K / \mu) * (180. / \pi)$

The parameters may later on easily be changed by using the „Modify" button. Especially the number of tendons in the group will often be later increased or decreased in the tendon design process.

An effective way for creating new tendons is by using the copy function. The physical and geometry parameters of a tendon can be copied to a new tendon at any time by selecting the appropriate line in the upper table and clicking the 'Copy' button. All currently existing data will be copied to the new tendon. Only a new start element for the assigned element series has to be entered. This allows to translate very efficiently tendon groups e.g. from one span to the next span.

However, this function is also applicable when some of the geometry parameters are different. The values to be changed are simply adapted by using the „Modify" function after the „Copy" function. The „Copy" function may either be used before or after the definition of the geometry data, depending on whether only the physical parameters should be transferred or also the geometry parameters.

5.3.2 Assignment of Structural Elements

The information, where the tendon starts and ends, and which structural elements it passes through – i.e. which structural elements are assigned -, must be specified for all internal Tendon Profiles. This assignment is done in ⇧STRUCTURE ⇨TENDON ⇩ASSIGNMENT by adding or modifying the data in the assignment table. This table is shown below the tendon table after the assignment function has been selected.

Generally tendons start and end at start and end points of structural elements. Therefore only the element series (El-from, El-to, El-step) to be assigned to the Tendon Profile is asked for, when data are added using the „Insert before" or „Insert after" functions.

But RM2000 also allows tendons starting somewhere between the start and end point of an element. The modify function in the window must in this case **later** be used to add this information. The Info window contains a graphic presentation of the tendon on the right side, and on the left side the related parameters with the possibility to modify them.

The value x/l defines the position in the element, where the tendon starts or ends. The value is related to the Clear Length of the element (it is for instance 0.5, when the tendon starts in the centre point of the beam element).

5.4 Tendon Geometry

5.4.1 General

The geometry of the tendon is specified by defining the position of „Constraint Points“ (P_i) and possibly the tangent direction of the tendon (F_i) in these points (“fix value” or “free”) arbitrarily along the assigned element series. The program calculates a 3rd order curve matching all these Constraints. This curve represents the tendon geometry with friction losses becoming a minimum.

The geometry definition via Constraint Points may be applied directly on the actual tendons (**option Tendon geom. NORMAL**), or on a “master tendon” which itself is not considered in the analysis process – regardless of which cross section characteristics are defined (**option Tendon geom. MASTER PROFILE**). This profile – or the relevant part of it - may then be assigned to actual tendons (**option Tendon geom. SLAVE PROFILE**). No Constraint Points must be entered for a slave tendon, they are taken over from the master tendon – even for start end end points of the actual tendon not matching a Constraint Point of the master profile.

The Constraint Points are entered or modified in ⇧STRUCTURE ⇨TENDON ⇩GEOMETRY. The table of the defined constraint points (Constraint Point table, Tendon Geometry table) is displayed below the tendon table. The definition of the position of the Constraint Points is usually done in terms relative to the structural elements, and the values in the table are also presented in this form if ⇧STRUCTURE ⇨TENDON ⇩GEOMETRY is selected.

Another presentation type is available for the Tendon Geometry table, where the position of the Constraint Points is given in terms of global coordinates. This presentation type is active if ⇧STRUCTURE ⇨TENDON ⇩3D-VALUES is selected. Not dependent on the selected presentation function, the parameters describing the tendon geometry may also either be defined relative to the elements or in terms of global coordinates.

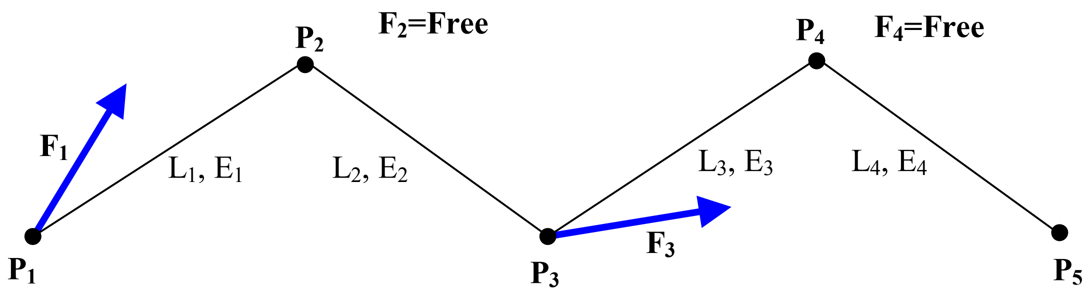
The input is either performed in a alpha-numeric input pad by using the „Insert before" or „Insert after" button, or in a combined numeric-graphic pad activated by clicking the Info button. Using interactive graphics makes the definition of the position of the constraint points easier.

5.4.2 Basics of the Geometry Calculation

The ensuing description is made for internal tendons, where the geometry is specified via constraint points of the type normal. The procedure used for external tendons or tendon segments is partially different. The deviations are described in detail in chapter 5.5 "External Tendons".

The drawing below shows an example for user defined constraint conditions (P_1, P_3, P_2, P_4, P_5, F_1, F_3). In this example the tangent vectors at points P_2, P_4 and P_5 are "free", i.e. not prescribed.

Note: *The following figures are drawn in 2D. The presented polygons and curves are however general curves and polygons in space.*



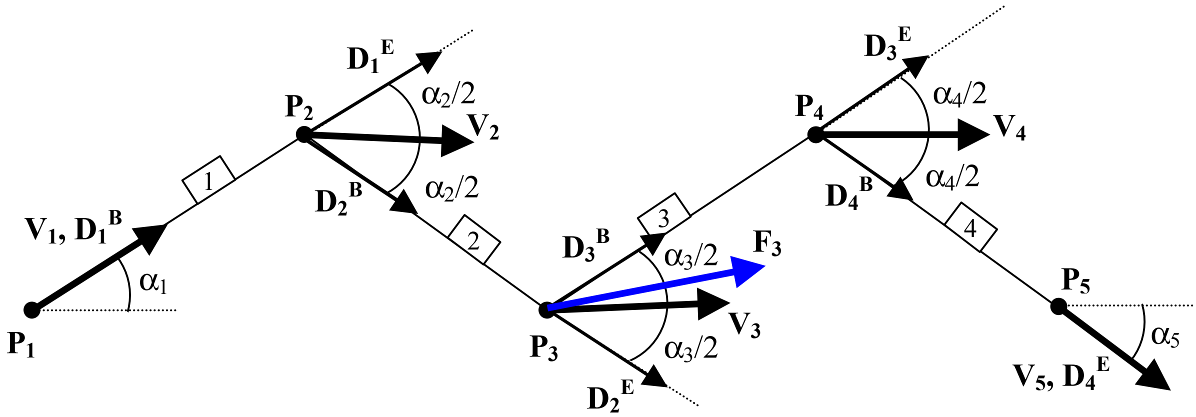
i	No. of the constraint point
L_i	Length between two constraint points
E_i	Fictitious E-Modulus of the "tendon members"
P_i	Constraint points
F_i	Prescribed tangent vectors with fixed angles at a specified constraint point

Step 1:
The basic reference geometry is the polygon formed by the straight connections between the constraint points. The tendon part between 2 constraint points is called "tendon segment". The straight connection forms a beam element ("spare beam").

The first approximation of the tendon geometry is now calculated assuming a prescribed tangent direction in all constraint points. The deviation of this direction from the direction of the connection line is applied separately on each segment as a prescribed element end rotation, and the resultant bending lines of the spare beams form the basic tendon geometry. The initially prescribed tangent directions are at the start and end points the directions of the connection lines to the next or former constraint point respectively. The initial tangent direction in intermediate points is the median line between the 2 directions from the former and to the next point.

This initial geometry fits the requirement of continuous direction changes and the curve has at the beginning and at the end tangents in the direction of the spare beams and in the intermediate points in the direction of the median. The compensating bending line of the total spare beam sequence is the superimposed in the 2nd step to this initial geometry.

The vectors and angles are presented in the figure below.



Step 2:
A compensation calculation is in the 2nd step made for the total sequence of the spare beams (bending line of a continuous beam). The tangent directions are in the calculation adapted to match the prescribed direction conditions and the minimum energy condition. The minimum energy condition automatically implies, that the friction losses become a minimum, because both, the deformation energy and the friction losses are determined by the curvature integral over the tendon length.

The bending lines are calculated separately for the X-Y-plane and the X-Z-plane. The resulting curves are superimposed to the initial curve. The resulting space curve of the tendon geometry is then a cubic spline curve.

Bending line in the X-Y-plane (y is normal to the spare beam in the X-Y-plane):

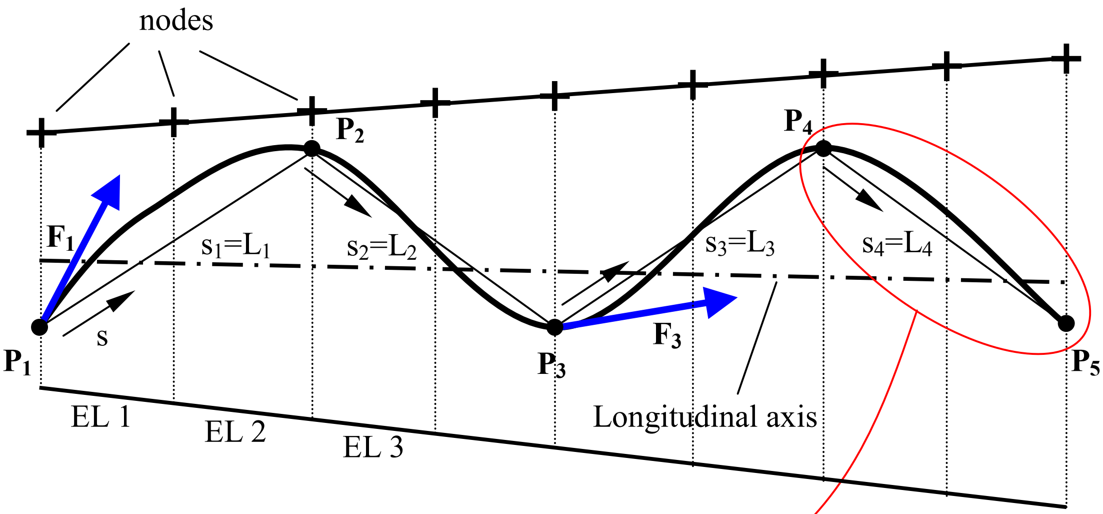
$$x \circ y: \qquad y(S - S_0) = y(x) = a_0 + a_1 \cdot x + a_2 \cdot x^2 + a_3 \cdot x^3$$

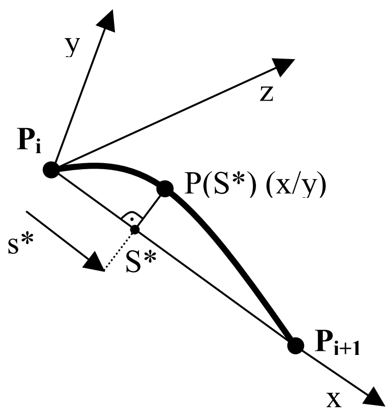
Bending line in the X-Z-plane (z is normal to the spare beam in the X-Z-plane):

$$x \circ z: \qquad z(S - S_0) = z(x) = b_0 + b_1 \cdot x + b_2 \cdot x^2 + b_3 \cdot x^3$$

In order to calculate the position of the tendon in intermediate cross-sections between the constraint points, the intersection between the analytically given space curve and the cross-section plane is calculated. The position s^* of the corresponding point (S^*) on the spare beam is the calculated by using the orthogonality condition.

The below presented figure shows schematically the geometry.





Element	X/L	s
1	0	0.00
1	1	5.52
2	0	5.52
2	0.5	6.43
2	1	7.34

S* Any point along the "tendon members".
P(S*) Calculated point of the tendon based on S* (in 3D) (see figure above).
s* Length from P_1 to S* along the "tendon elements"

$$s^* = s_1 + ... + s_{(i-1)} + \overrightarrow{P_i S^*}$$

Summary of the procedure:

- Determination of the stiffness matrices [k_{TMj}] of the different segments and calculation of the fixed end forces of the different spare beams {p_j}= [k_{TMj}] * {d_j}
- Specification of the additions stiffness terms [k_{Zi}] for the points with prescribed tangent direction and calculation of the equivalent forces: {$p_{z,i}$}= [k_{Zi}] * {Δv_i}
- Assembling the element stiffness matrices to the total matrix [K], considering the additional stiffness at the points, where the tangent direction is prescribed.
- Solving the equation system: $\{p\}+[K]\cdot(\Delta\alpha)=0$

Notations in the above formulae:

i	No. of the constraint point
j	No. of the segment between 2 constraint points
p_j, $p_{z,i}$	Equivalent load vectors (fixed end values)
$k_{TM.j}$	Stiffness matrix of the segment j (straight spare beam)
$k_{Z,i}$	Rotation stiffness of the constraint point (with prescribed tangent direction)
d_j	Deformation vector (element end rotations) of the considered spare beam
Δv_i	Prescribed deviation of the vector F_i from V_i.
$\Delta\alpha$	Resultant deviations of the final tangent directions from V_i.
F_i	Prescribed direction vector in the constraint point
Vi	Vector in the constraint point; orientation in the median line direction of 2 subsequent spare beam directions

Straight Parts:

The above procedure is slightly changed in order to consider straight parts of a cable:

1. The vectors V_i at the beginning and at the end of the straight part are not prescribed in the median direction, but both in the direction of the connection line (spare beam direction). Therefore the initial geometry has in this part no deviation from the straight connection line.
2. The fictitious stiffness of the spare beam in the straight part is considerably increased compared to the other segments. This guarantees that the segment remains straight in the compensatory bending line calculation.

Attention has to be drawn to the fact that 2 straight segments must not be arranged immediately one after the other, and that the tangent direction in the straight part cannot be prescribed because it is unconditionally determined by the direction of the connection line.

5.4.3 Definition of the Constraint Points

The following parameters are required for the definition of the position of constraint points:

⊙ Space point The definition of the position is done via global coordinates
⊙ Structure element The definition is done relative to a structural element

5.4.3.1 Parameters for global definition

The definition of the position of constraint point via global coordinates in space is nut very often used, because usually the geometry is given in relation to the structural system. This option is provided in RM2000 only for special cases.

Besides the below described Constraint Point Type the global coordinates in space of every constraint point describing it's position are entered in this case. Additionally, the direction of the tangent of the tendon geometry <u>must</u> be entered for all points. In this input mode it is not possible to set free the tangent direction of some constraint points, i.e. using this input mode is only allowed if the tangent direction in all constraint points is known.

- x, y, z Coordinates of the Constraint Point in Space
- dx, dy, dz Vector for describing the tangent direction

5.4.3.2 Parameters for element related definition

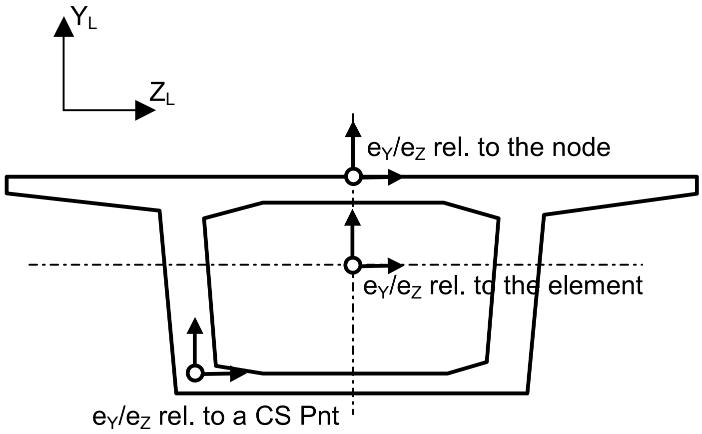
The input field 'Ref. Elem' contains the number of the reference element. This element number describes the position of the point in the structural system together with the value x/l describing the position between element start and element end.

The position of the tendon point within the cross-section defined by the element number and x/l may be defined in local or global directions and either

- in terms relative to the centroid,
- in terms relative to the nodal point (often on top of the cross-section) or
- in terms relative to a specified cross-section reference point.

(see figure below).

The specification of the position related to the longitudinal direction of the element is performed via the clear length related coordinate x/l. The eccentricities in y- and z-directions may be specified in the local or in the global coordinate system. (Selection switch „Elem“, „Node“ and „CS pnt“ below the input fields x/l, e_y and e_z).



The **direction of the tangent** in the constraint point can be prescribed as a compulsory condition additionally to the position. This direction is specified by the angles Alpha1 (“vertical angle”) and Apha2 (“horizontal angle”), being the angles in the vertical plane and in the horizontal plane respectively. These angles may be related – similar to the position – to 3 different axes: the element axis, the system line or the line connecting the specified reference points (selection switch „Elem“, „Node“ and „CS pnt“ below the input fields Alpha1 and Alpha2).

Alpha1 Vertical angle of the tendon at the current position. Switch ‘Free’ (no direction constraint) or ‘Value’ for a user defined constraint value.

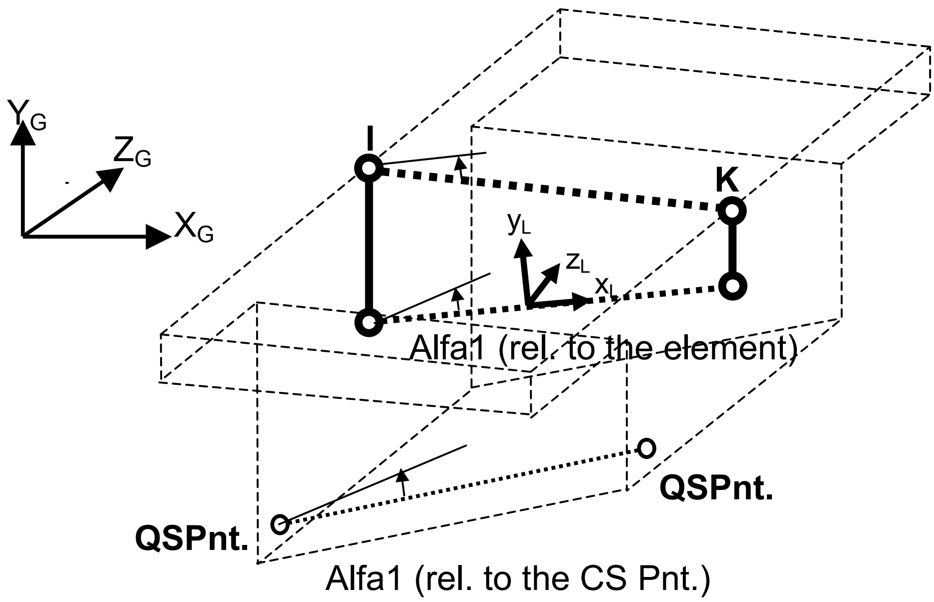
Alpha2 Horizontal angle of the tendon at the current position. Switch ‘Free’ (no direction constraint) or ‘Value’ for a user defined constraint value.

Attention: **The meaning of “vertical angle” (resp. elevation plane) and “horizontal angle” (resp. horizontal plane) is in this context not exactly the same than in the structural element definition part. See description below.**

The **vertical angle** is the angle between the **reference line** (element axis, system line, connection line of reference points) and the normal projection of the tangent to the “elevation plane“ (from the reference line to the projection of the tangent anticlockwise positive). The „elevation plane“ is in this case built by **using the reference line** for creating a new local coordinate system in accordance with the general rules, and **considering** a possibly for the structural element prescribed **angle Beta**. The y_L-axis of this new local coordinate system forms the elevation plane together with the reference line.

The **horizontal angle** is the angle between the **reference line** (element axis, system line, connection line of reference points) and the normal projection of the tangent to the “plan“ (from the reference line to the projection of the tangent anticlockwise positive).

The „plan“ is in this case built by **using the reference line** for creating a new local co-ordinate system in accordance with the general rules, and **considering** a possibly for the structural element prescribed **angle Beta**. The z_L-axis of this new local coordinate system forms the “plan” plane together with the reference line.



There is also the possibility to prescribe a **straight part** between 2 constraint points. This is done by setting the switch „Straight part“ at the constraint point, where the straight section begins. The straight section the extends to the next constraint point. Prescribed direction constraints at the begin and the end of the straight section are ignored, the tangent direction on both sides is equivalent with the direction of the straight part of the tendon. A fold in the tendon geometry curve is generally not allowed.

5.4.3.3 Summary of input parameters

- Type — Type of the considered tendon segment (see chap. 5.4.4)
- Ref.Elem — No. of the assigned structural element
- CS pnt — Name des reference point in the cross-section

Note: *Every defined cross-section reference point may be used for defining the reference line for the specification of the tendon constraint point. But the specified point is only used, if also the switch (⊚ CS Pnt) is set, defining that the geometry is related to the connection line of reference points rather than to the element axis or system line.*

⊙	Global	The entered eccentricities are defined in global axis directions
⊙	Local	The entered eccentricities are defined in local axis directions

Attention: The switch "local" always means the local coordinate system of the considered element - built by using the element axis (centroid line) and not by using the reference line if ⊙Node or ⊙CS Pnt has been selected.

➢	x/l	Position in longitudinal direction (related distance from the element start)
➢	ey	Eccentricity in (global or local) y-direction
➢	ez	Eccentricity in (global or local) z-direction
⊙	Elem	Reference line is the element axis (centroid)
⊙	Node	Reference line is the system line (straight connection between the start node and end node of the element)
⊙	CS pnt	Reference line is the connection line of a reference point in the start and end cross-section
➢	Alpha1	„Vertical angel" of the tendon tangent
➢	Alpha2	„Horizontal angle" of the tendon tangent

Attention: For the exact meaning of the terms „Vertical angle" and „Horizontal angle" see chap. 5.4.3.2.

⊙	Value	„Vertical angle" and „Horizontal angle" respectively are prescribed (the above defined value is prescribed as constraint condition)
⊙	Free	Alpha1 and Alpha2 respectively are free (not prescribed)
⊙	Elem	Reference axis for Alpha1 or Alpha2 resp. is the element axis
⊙	Node	Reference axis for Alpha1 or Alpha2 resp. is the system line
⊙	CS Pnt	Reference axis for Alpha1 or Alpha2 resp. is the connection line between reference points
☑	Extern	The straight section is outside the cross-section
➢	Number	Element number (new structural element) of the external tendon segement
➢	Radius	Radius of the curved segment

Note: *A new free structural element no. has to be assigned to external tendon segments.*

5.4.4 Choice of Tendon Constraint Point Types

The following Constraint Point Types are available to specify the tendon geometry.

Normal: Using the type "Normal" we specify fixed points in space where an internal or external tendon has to go through. The tangent at this point can be free or constraint (value). Elevation and plan are separately calculated, i.e. the direction may be prescribed in the elevation and free in the ground plane ore vice versa. This point type is set by the program as default, and generally used in the definition of internal tendons for all points except the start points of straight sections (For the correct use of this function for external tendons see chapter 5.5 External Pre-stressing).

Line: This type serves for defining a start point of a straight tendon section. This might be the begin of a straight part of an internal tendon or the begin of a straight part of an external tendon. The direction of the tangent is automatically defined by the position of the point and the position of the next point. If the type "Line" is used in the external tendon definition for defining the start of a straight section after a tangent intersection point, then the entered position in the cross-section is not used, but the intersection point of the straight connection line between the tangent intersection points and the cross-section plane.

The following types are (besides the type "Line") only provided for (mostly external) tendons, where alternating straight sections and more or less circular deviation sections are arranged:

Line (free Y): As with "Line", the begin of a straight tendon part is specified with this type. But with this type, this is done for start points after a straight section and a subsequent curved segment. Together with the next segment the 3 segments must be in a common plane. Therefore *RM2000* calculates the plane built by the 2 previous points and the subsequent point, and changes the Y coordinate of the actual point such that the point lies in the prescribed plane (see also chapter 5.5.2 and 5.5.3).

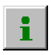
Line (free Z): Same as "Line (free Z), but the Z coordinate is adapted instead of the Y-coordinate.

Intersection point: With this function an intersection point of two tangents or a fixed constraint point can be defined (e.g. tangent intersection point of a deflection sheave or end point of an external tendon).

Free node at element: With “Free node at the element” a point of the tendon with variable position can be defined. The exact position will be calculated with the specified constraints (e.g. transition of a straight external part to the internal part in the region of the deviator or intersection point of the straight part to the curved deviator block, see also chap. 5.5.2).

Intersection point (free): An intersection point (free) can be used if the position of this point should be calculated by the program in the way that it becomes a point of the plane defined by three constraint points (see chapter 5.5.3 Geometry Definition by Specification of Straight Segments (Type 2).

5.4.4.1 Graphic input facilities

Selecting the  button above of the constraint point table a graphic presentation of the assigned element series together with the tendon geometry and the cross-sections opens. The user can select if he wants to see the tendon geometry in the cross section or a view of the definiton (Vertical, side, ground view).

If a super-elevated view is wanted this has to be defined in ‚plstruct.rm’. It is possible to change in the first line the parameters Scf-x, Scf-y and Scf-z (file editor or in ⇧RESULTS ⇨PLSYS).
The super-elevation is equivalent to the one defined in PLSYS.

The new window is split into three parts:
a) The interactive graphic screen (right, central)
b) The input part (left, only active for ‘new’ input or for editing)
c) The table at the bottom of the window displays the geometry points already input.

a) Graphic presentation window:

The radio buttons at the top of the window can be used to display either a
⊙ Cross section or a
⊙ Perspective view

The value
- TxtFact

is used to influence the size of the presented element labels.

A vertical line presents the current position of the tendon definition in both views.

There are four buttons at the top of the graphic screen:
These buttons use the dz step for the cursor if Cross Section View is selected and use the dx step for the cursor if the Perspective View is selected.

Cross-section view:

- << move the cursor to the extreme left side of the cross section or
- < move to the left using the dz step for the cursor or
- > move to the right using the dz step for the cursor or
- >> move the cursor to the extreme right side of the cross section or

Isometric view:

- << move the cursor to the end of the first assigned element.
- < move in the longitudinal direction (+X) using the dx step for the cursor
- > move in the longitudinal direction (-X) using the dx step for the cursor
- >> move the cursor to the end of the last assigned element.

Four further buttons are on the right hand side of the screen. They are equally used for the cross-section view and the isometric view:

- ++ move the cursor to the top of the cross section
- \+ move upwards using the dy step for the cursor (which is user defined)
- \- move downwards using the dy step for the cursor (which is user defined)
- -- move the cursor to the bottom of the cross section

b) Constraint point table

The constraint point table is displayed on the bottom edge of the window in order to keep the information about the actually active constraint point resident. The effect of changes is there immediately visible.

Changing the position in the graphics does not change the active constraint point. Only the parameters shown in the input part of the window are adapted. These parameters are transferred to the active constraint point if the function „Apply“ is selected in the „Modification“ case.

The active constraint point in the constraint point table also determines the position, where a new constraint point is inserted when the function „Apply“ is selected in the „Insert“ function.

c) Input part

The user can define the eccentricity directly or use to use the graphical facility of stepping in either direction inside the cross sections as described above.

Step dx Step dy Step dz	Increment for moving the cursor cross lines around the inside of the cross section or along the element axis. It is mainly used to specify a tendon constraint point relative to the top, bottom, left or right of the cross section (not referring to the centre of gravity)
Apply	Select this button to store the current definitions for the new or modified tendon constraint point. The new tendon constraint point definitions will be immediately displayed in the input table at the bottom of the screen.

The Main Input table (not the table below the graphical screen displayed with ‘Info’):

All tendon constraint points for the tendon geometry are defined/modified/deleted in this table. (N.B. the point can only be deleted in the main data table – not via the ‘Info’ button!) To insert a new tendon constraint point select the appropriate line and use the ‘insert after’ or ‘insert before’ buttons to activate the input. Choosing the ‘Edit’ button for the selected line also activates the input.

The tendon profile defined via this ‘main Input table’ can be checked graphically afterwards by selecting the ‘Info’ button.
The 3-dimensional coordinates of the profile constraint points can also be checked by choosing ‘3D-Values’.

5.5 External Pre-stressing

5.5.1 General

Tendons or tendon parts outside of the element cross-section may also be modelled with RM2000 (external pre-stressing). We distinguish in this context between **“external tendons”**, located over the whole length outside the cross-section, and **“external segments”** of internal tendons.

“External segments” are always straight. A straight part is automatically created by the program when the radio button “Extern” in the Constraint Point definition menu is set. These external segments **must get an individual structural element number**. They are only at the beginning and at the end connected to the structural system.

Attention:	**The activation of these additional elements is not later done in ⇨STAGE ⇩ACTIVATION by the user, but automatically by the program when used in a Stress Action in ⇨STAGE ⇩ACTION. They are however listed in the element table in ⇧STRUCTURE ⇨ELEMENT ⇩ELEM, but they can not be modified there.**

The geometry definition of external segments of internal tendons is done as described in the previous chapter 5.4. The only difference is, that a Constraint Point of the type “Line” is used for specifying the begin of the straight part, and a new structural element number is assigned.

“External Tendons” are also in the curved parts located outside of the cross-section. They are in these sections however assigned to structural concrete elements for transmitting deviation and frictions forces. But it is assumed that the contact line is outside of the cross-section (along a deviation block) and the tendon can not be grouted and never contribute to a net or composite cross-section.

An external tendon always requires after a straight segment at least one ensuing structural element simulating the deviator block. Folds in the tendon geometry are not allowed. Each deviator block requires at minimum 1 structural element between 2 straight parts. Two elements are required if also the summit of the curve should be defined. Note that the begin and the end of the deviator block should already be considered in the modelling process of the structural system (nodal points approximately at the begin and end of the deviator), in order to guarantee a proper transmission of the deviation forces to the structure.

The straight segments (with a start Constraint Point of the type “Line”) are usually defined as “External segments”, i.e. separate structural elements simulating the tendon are created. These elements are connected at the begin and end to the structural system. No friction calculation is performed in these segments.

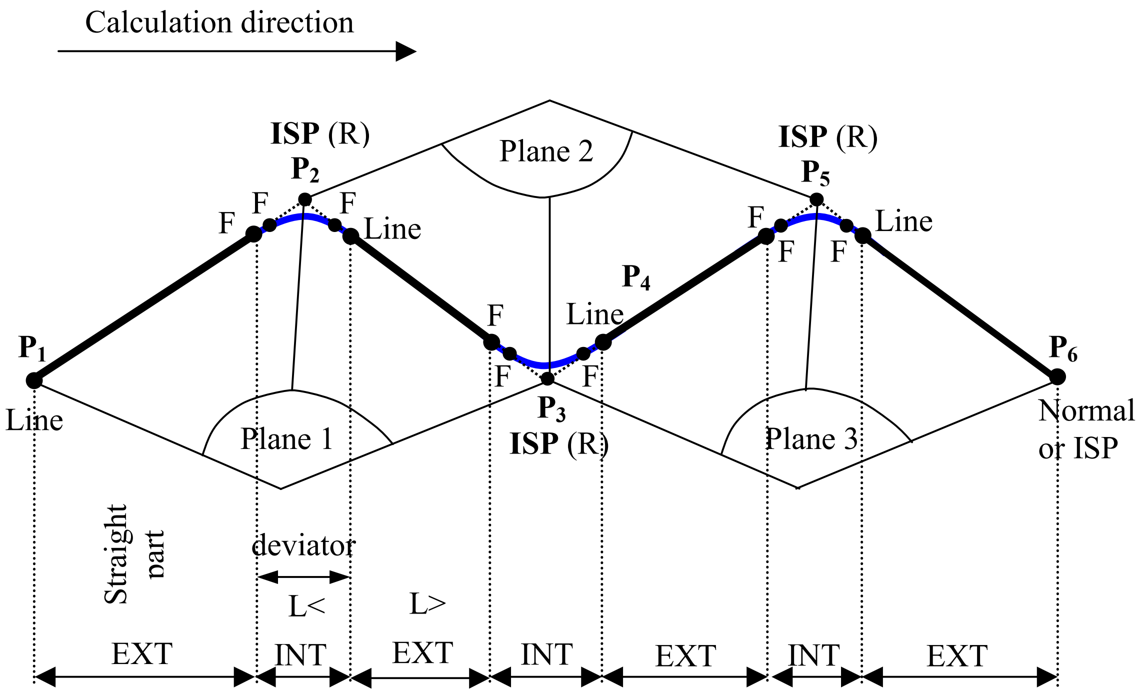
Note: *It is formally possible to treat straight parts of external tendons with respect to friction and tendon force calculation in the same manner than internal tendons. This is done by not setting the switch in the constraint point definition menu to "external" and not assigning a new element number. They are however not considered for the calculation of net and composite cross-section values.*

Due to the high friction and the high deviation forces it may be assumed, that along the deviator block external tendons may be treated like internal tendons (rigidly connected to the structural element). The friction calculation is therefore performed in these section in the same manner than for internal tendons. The accidental deviation angle should be zero if there are straight (or nearly straight) parts within the deviator region, in order to avoid wrong additional friction forces.

The geometry definition of external tendons may be performed in 2 different ways:
- Definition via tangent intersection points (Type 1)
- Definition of the straight parts, where the curved segments are fitted in between.

5.5.2 Geometry Definition via Tangent Intersection Points (Type 1)

The user defines in this case the start and end points of the tendon and the tangent intersection points as Constraint Points of the tendon geometry (points P_1, P_2, P_3, P_4, P_5 of the below figure).



Minimum required point definitions:

Besides the tangent intersection points (ISP) at least one free point F (in calculation direction) before and one point of the type “Line” after each intersection point have to be specified. The free point before the the tangent intersection point marks the end of the “external segment”, the “Line” point after the intersection point defines the begin of the next external segment. Only the position in longitudinal direction (x/l) has to be specified for these points. The specified position in the cross-section is irrelevant, the eccentricities are automatically calculated by the program (intersection between the ISP connection line with the cross-section plane).

2 subsequent sections between the tangent intersection points form a plane, defined by 3 Constraint Points (see planes 1, 2, 3 in the above figure). The curve representing the tendon geometry between the straight sections is then calculated as 3^{rd} order parabola, with the positions and tangent directions of the start and end points (F and L), and the specified Radius as curvature radius at the vertex are constraint conditions.

Note:	**The resulting 3rd order curve in the deflection area is more or less different to the mostly required form "Straight line – Circle – Straight line". The size of the deviation may be controlled by an suitable choice of the vertex curvature radius – too small radii yield opposite curvatures at the start and end points, too large radii may yield begin and end points of the curved part being outside of the predefined deflection area. A better approximation can be achieved by defining "additional free points" (see below).**

Additional „Free Points":

The deflection area, where the tendon is treated as internal tendon for calculating the friction losses, is often much larger than the effective deflection area, where the tendon is really in contact with the deflection block. I.e. the actual tendon geometry within the deflection area consists of a circular arc in the vertex region and additional straight pieces to the connection points of the (external) straight segments. This shape can only approximately be represented by the 3rd order curve calculated as described above. The main shortcoming is, that opposite curvatures usually arise at the start and end points, influencing the friction losses.

A much better approximation can be achieved by defining additional "Free points" between the tangent intersection point and the start and end points of the curved section. The position of these points is calculated by the program by fitting a circle with the specified radius into the angle between the 2 straight connection lines forming the tangents of this circle. The 3rd order curve is now calculated between these theoretical new points. Straight pieces are then arranged between these new points and the start and end points of the internal section. The resulting shape in the curved area is now much closer to the exact circular form (see chap. 5.5.4).

5.5.3 Geometry Definition by Specification of Straight Segments (Type 2)

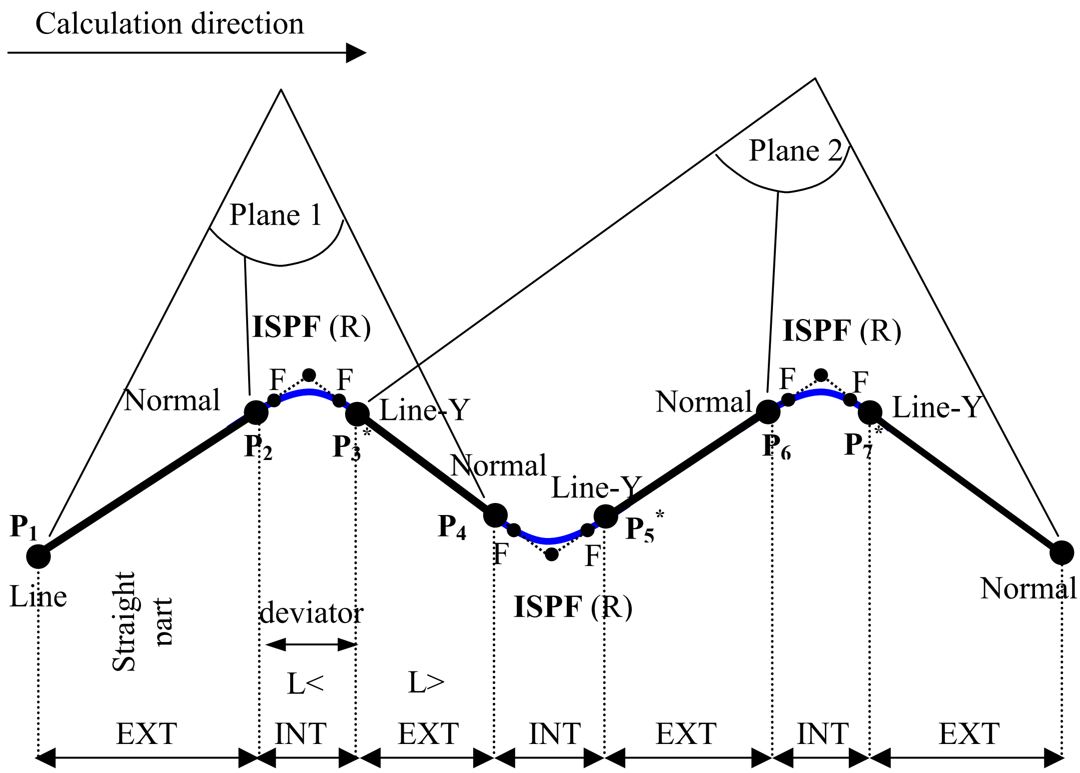
The calculation process is in this case in some details different to the procedure used in the 1st method with tangent intersection points:

- the planes for the calculation of the curved parts are characterized by other Constraint Points, and
- the position of the tangent intersection point is calculated by the program.

The geometry of the tendon is in this case calculated as described below:

The start point of the tendon is a point of the type "Line", the end points of each straight (external) segment are points of the type "Normal". The start points of all further straight segments (from the 2nd onwards) are either

- **„Line (free Y)" for the definition of the plan geometry,** or
- **„Line (free Z)" for the definition of the elevation geometry** respectively.



* Point position is corrected fit in plane

Three constraint points (P1, P2, P4), specified by the user, build a plane (e.g. plane 1, see figure above). The final position of point P_3, describing the transition point from the curved internal section to the ensuing straight segment, is calculated by the program. The condition, that this point is in the plane built by P1, P2 and P4, must be fulfilled. Dependent on whether the Y or the Z coordinate should be adapted to pull the point into the defined plane, P_3 is specified as Constraint Point of the type "Line (free Y)" or "Line (free Z)".

Attention:	**"Line (free Z)" has to be used for defining the elevation geometry, "Line (free Y)" for defining the plan geometry. The type "Line" must not be used, for program internal reasons even not, if the point is already located in the right plane.**

If no further free points are defined besides the start and end points of the “straight segments”, then the curve between the 2 points describing the begin and the end of the curved region automatically becomes a 2nd order parabola with prescribed tangents at the start and end points. The curvature radius at the vertex can in this case not be defined by the user, but will be an implicit result of the calculation.

If a certain curvature radius should be prescribed for the vertex point, then the user has the possibility to insert the tangent intersection point as “„Free point with radius“ (ISPF). The calculation is then performed in the same manner as described in 5.5.2 (Inserting a 3rd order curve with prescribed curvature radius at the vertex). Again opposite curvatures within the deflection area may arise as described in the previous chapter 5.5.2.

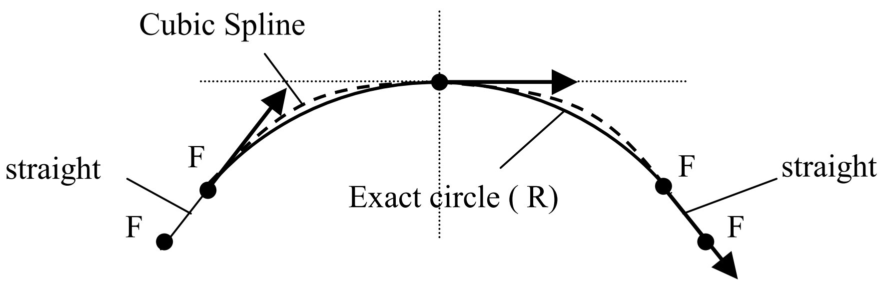
Similar to the procedure described in chap. 5.5.2, **additional Free Points** may be defined in order to get a better approximation of the geometry within the deflection area.

5.5.4 Approximate Geometry in the Region of the Deviator Block

If in the region of the deviator all “Free points” (see above) are defined, then the program calculates the exact transition from the straight part of the tendon to the circle (condition: the straight segment is the tangent to the circle with the radius R). The curve between the vertex and these transition points is approximated by a 3D cubic spline curve.

If the additional free points are omitted, then the cubic parabola is fitted between the start and end point of the deflection area and in consequence the approximation is worse (the resulting geometry is more inaccurate).

The parabola deviate in both cases more or less from the circular shape (curvature radius at the vertex is smaller than at the transition points). Therefore the radii listet in TENDON.LST are not equal to the prescribed radius.



5.6 Simulation of the Stressing Procedure

5.6.1 Computing the Friction Losses

The angle changes of the tendons are calculated and measured in 3-dimensions. The accumulated angle change ($\sum\alpha$) at any position in the tendon profile is used to calculate the friction loss up to that position in the tendon profile using the friction loss formula given below:

$$\mathbf{Z_i = Z_o * e^{-\mu(\Sigma\alpha + \beta * l)}}$$

μ	Coefficient of friction
α	Angular deviations (measured internally in Radians)
β	Accidental deviation angle (rad/m) (entered in °/m; internally transformed to rad/m)
l	Member length

> NB: The formula for losses used by RM2000 is different from that defined in the British/American code ($\mathbf{Z_i = Z_o * e^{-(\mu*\Sigma\alpha + K*l)}}$), the conversion from the British/American formula to be compatible with the formula used in RM2000 requires the WOBBLE FACTOR 'K' value to be multiplied by ((180/(π * μ)) in order to get the input value $\beta_{Degrees}$ required by the program.
>
> The values given in various codes for the wobble and friction coefficients are based on experimental data – they are empirical and are designed to give an expected result from angular changes in the tendon profiles and the unexpected deviations (wobble) for different types (stiffness, material and profile) of tendon ducts.

It is only possible to stress a tendon (profile) after the geometry for it as well as its material properties have been defined. All the affected tendons need to be stressed before the related pre-stressing Load Case can be calculated in the construction stage.

The actions that can be applied to the tendons include 'initial stressing', 'release', 're-stressing' and 'wedge slip' from the left end or the right end of the tendon. Each action must refer to a 'Stress-label' (an assigned name that can be used to represent a series of actions on various tendons – 'Construction Stage' commands can then refer to this 'name'). Thus: It is possible to stress several tendons for the first construction stage including applying wedge slip, apply a single stress label to these actions (such as 'stage1') and then simply refer to the stress label when calculating the pre-stressing load case in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOAD.

5.6.2 Stressing Actions – Tensioning, Releasing, Wedge Slip

The exact definition of the stressing sequence is performed in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩TENDON. The upper table (Stressing Actions Table) appearing in the GUI pad lists the Stressing Actions for all tendons. The lower table shows all the actions belonging to the same group than the active action of the upper table. The lower table is purely for viewing, all the definitions are made in the upper table.

Existing definitions can be modified/deleted by selecting the appropriate line and choosing the 'Edit' or 'Delete' button. A new stress action for a tendon can be added by selecting the appropriate position in the upper table and choosing 'Insert before' or 'Insert after'. The following actions on the tendon can be made via this input screen:

- ⊙ PREL Initial stress action on the tendon at the left end of the tendon
- ⊙ PRER Initial stress action on the tendon at the right end of the tendon
- ⊙ RELL Release the tendon at the left end of the tendon
- ⊙ RELR Release the tendon at the right end of the tendon
- ⊙ WEDL Wedge slip at the left end of the tendon
- ⊙ WEDR Wedge slip at the right end of the tendon

A further switch can be set for all force related actions (PREL, PRER. RELL, RELR), defining whether the stressing force is entered directly or via specifying a factor related to the „allowable pre-stressing force". This reference value is defined as the product of the cross-section area of the cable(s) and the allowable stress defined in the material definition function (⇧PROPERTIES ⇨MATERIAL – parameter SIG-allow-pr).

- ⊙ Force The stressing force is specified directly as a force
- ⊙ Factor The stressing force is defined via a factor related to the allowable force (e.g. 0.95 represents 95% of the allowable force).

A wedge slip occurs in most pre-stressing methods, and is usually the last action on the tendon before grouting.

Note:	**The input is in global units [length(structure)] – usually metres. Thus, if [m] is the unit and the wedge slip is 6mm, then the input must be 0.006).**

After selecting the type of the stressing action by the above switches, the related parameters have to be entered:

- The actually related tendon must be identified by either typing the number into the input file for 'Tendon' or by selecting the pull-down menu arrow and choosing the appropriate tendon from the displayed list.
- The number of tendons per tendon group is taken from the previous definition and is shown as an information.

- The next entry is either a ‘Factor’, ‘Force’, or ‘Wedge Slip’ value, depending on the selected action type.
- A label is assigned to every stressing action. The program identifies by this “Stress Label”, which actions should be considered when the Load Case pre-stressing, where this Label is referenced, will be calculated.
- The description can be a general text for the user.

Each stress action can immediately be viewed by clicking on . The displayed graphics screen shows the reaction (tendon force diagram) for the selected and previous actions on the specified tendon. Different colours are used for presenting the effects of the different previously applied actions. The user can also store this picture in a plot-file.

An ASCII file called STRESS.LST is created automatically when running the calculation (⇧RECALC). This file contains all important information such as tendon elongation and tendon forces for all tendons.

5.7 The Pre-stressing Load Case

The program requires a prepared Load case number to be able to calculate the pre-stressing loading case in the LOADS AND CONSTR.SCHEDULE. The procedure is similar to a general Loading case calculation:

- ➢ The user creates various Load Sets containing the loading data for the loading case.
- ➢ These Load Sets are then used to create a Load Case.

This procedure is also valid for a pre-stressing loading case.

5.7.1 Definition of the Load Sets for Pre-stressing

Select ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET to create a new Load Set. First define a new Load Set in the upper table by selecting a position in the upper table and choosing ‘Insert before’ or ‘Insert after’. The related tendons are then inserted in the load table (lower table in the GUI).

Note: *It is also possible to define several Load Sets and to combine them in one Load Case.*

The following input must be inserted in the upper table:

- Number Load Set number
  (TDV has a recommended numbering scheme which is useful for future identification – e.g. Pre-stressing Load sets/cases should be numbered between 501 and 599. Load Set 501 for the

Construction stage 1 pre-stressing loading set, Load Set 502 for Stage 2 pre-stressing loading, etc.)

- Location — The program creates an ASCII file automatically without any user input.
- Description — Descriptive text (max. 80 characters)

The loading data for this new Load Set must then be defined in the lower table:

- Select a position in the upper table
- Select 'Insert before' or 'Insert after' at the top of the lower table to open the Load Type selection window.
- Select the Load Type
  - ⊙ **Stressing**
  - ➢ **Select 'Tendon stressing' (TEND0)**

Define the Tendons to be assigned to this Load Set in the displayed input window:

- ➢ From/To/Step — Series of tendons to be assigned to the currently selected Load Set (501… 599)

Note: *It is possible to use in a Load Set respectively Load Case only some of the tendons grouped together in the same "Stress Label". However, before using the Load Set in the action CALC for calculating the pre-stressing Load Case, the actual stressing forces have to be determined for all related tendons in the stressing action STRESS.*

- ⊙ Increment-Force — The difference with respect to a previous stressing process is considered
- ⊙ Total-Force — The total force is considered in the calculation

Note:	**The setting of the above switch is only important, if a tendon is stressed more than once at different stages in the Construction Schedule. The previous tension force is zero if a tendon is stressed for the first time, therefore the difference will be equal to the total tension force.**

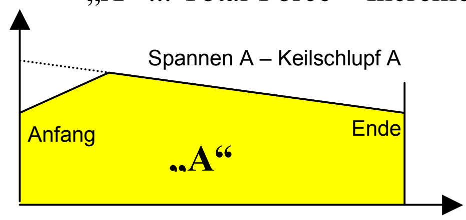
If the switch is set to „⊙ Increment-Force", then the previous tendon forces are subtracted from the new forces when the forces acting on the structural system are calculated, i.e. only the tendon force difference yields a secondary (constraint) stress state in the structural elements. Using this option is required when tendons are partially stressed in one construction stage and re-stressed to the full tendon force in a later stage (e.g. stressing to 60% in the first stage and to 100% not before the last stage.

Attention has to be paid to the fact, that the 2 stressing cycles have to be grouped in 2 different "Stress Labels". The stress label for stressing to 60 % has to be assigned to the first „STRESS" action. The "STRESS" action for fully stressing the tendons must be placed in the construction schedule not before the calculation of the related pre-stressing load case (storing the related tendon forces in the database) has been performed. In the ensuing „CALC" action the previous tendon forces are subtracted from the forces computed in the „STRESS" action.

If the switch is set to „◉ Total-Force", then the full force is applied if the same tendon is stressed in a later Load Case. This is mostly not meaningful. This option may be useful in special cases, e.g. for approximately considering a pre-stressing steel relaxation (as percentage of the initial force), or for simulating later removal of a tendon (see below).

Example:
Construction stage 1: First stressing to the tendon force state "A"
The selection of the switch has no influence on the results

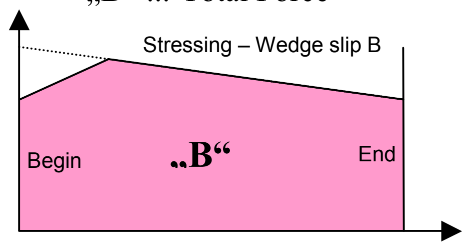
„**A**" ... Total-Force = Increment-Force



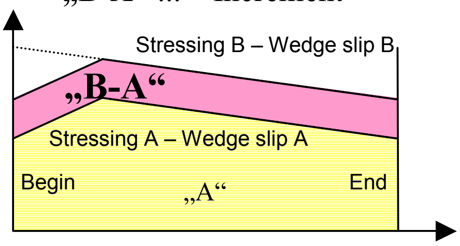
The stress state "A" is applied to a tendon in the first construction state (function STRESS in the Construction Schedule – see chapter 5.8 "Calculation actions for Pre-stressing in the Construction Schedule). Selecting "Total-Force" or "Increment-Force" is in this case of no importance, because no previous tendon forces exist.

Construction stage 2: Re-stressing of the tendon to the tendon force state "B"

„**B**" ... Total Force



„**B-A**" ... Increment



The stressing force state "B" is assigned to this tendon in the 2nd construction stage. The total tendon force "B" is used in the Load Set if „◉ Total-Force" is selected. If this Load Case is summarized in a superposition file, then the result is superimposed to that of the first construction stage, and the sum "A+B" will be as final result in superposition file, instead of the correct state "B".

If „⊙ Increment-Force“ is selected, then the force state “B-A” will be used as loading. The sum of the 1st and 2nd construction state will now be the state due to the tendon force state “B”. I.e. the choice of the switch „⊙ Increment-Force“ is important if a tendon is stressed in different construction stages, and a standard Load Case accumulation is performed.

A practical application of the option „⊙ Total-Force“ may be an approximate consideration of the pre-stressing steel relaxation. Considering the Load Set in the Load Case with a factor of –0.2 means a relaxation of 20%. An other application of „Total-Force“ is the removal of tendons (Load Set with Total Force is assigned to the Load Case with factor (-1.0)).

<u>Example for relaxation:</u>

Assumption:
A tendon is stressed up to 100% as usual. The stressing sequence is available, a Load Set (LSET 501 – Tendon number – „Increment“) and a Load Case 501 with factor 1.0 for the Load Set 501 are defined in calculated within the construction schedule.

The relaxation occurs over a certain amount of time (e.g. 8 days). It is ‚0’ at application time of the Load Case and e.g. 10% after 8 days.

A further Load Set (LSET 502 – Tendon number – „Total-force“) and a Load Case 502 with factor (-0.10) for the Load Set 502 are defined and calculated in the construction schedule, 4 days (as approximation) after the Load Case 501.

5.7.2 Definition of the “Load Case Pre-stressing”

This is done in the same manner than defining other Load Cases and described in detail in chap. 6.4. ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LCASE is selected to create a new Load Case. The upper table shows the existing Load Cases, the new Load Case is inserted in the table.

Input for a new Load Case:

- Number — Load Case number. (TDV numbering system recommended – see above under Load Set) Loading Case 501 should be the pre-stressing load case for Construction stage 1, etc.
- Type — Definition of the Permanence Code.
  - Load — Load acts permanently
  - Load+Unload — Load is applied and removed after a period of time. **Not appropriate for pre-stressing loading!**
- Load Info — Refers to the ‘Load Management’ table which supports an automatic combination of loads during the construction sequence. See ⇩LMANAGE for detailed description. The pull-down menu arrow opens a list of all the available Load Groups. The current loading case can be assigned to one of the Load Groups. (PT for instance)
- Location — Name of the ASCII input file - automatically generated by the program. (Can not be changed by the user.)
- Output File — Name of the ASCII output file automatically generated by the program. (Can not be changed by the user.)
- Description — Descriptive text (max. 80 characters)

The appropriate Load Sets must be assigned to the Load Case in the lower table in the GUI once the Load Case is generated.

- Load Set — Selection of the Load Set, interactive selection via the pull-down menu arrow.
- Const-Fac — Constant multiplication factor (for all static calculations).
- Var-Fac — Variable multiplication factor (for dynamic calculations).

Note:	**A Load Set is multiplied by the sum of both factors (LoadSet * (Const-Fac + Var-Fac). Var-Fac is often a function of time, using definitions entered in ⇧PROPERTIES ⇨VARIABLE**

5.7.3 Calculation of the Load Case „Pre-stressing“ and Results

The calculation of the Load Case “Pre-stressing” is in some details different to the calculation of “normal” other Load Cases. The applied - in fact external - stressing forces (deflection forces, anchor forces, friction forces) are transformed to an equivalent “internal” stress state or internal force state respectively. This equivalent internal force state is generally called **“primary state”** or **“V*e state”**.

This internal force state is on the one hand stored in the result file as the “primary internal force state”, and – on the other hand – used to calculate an equivalent strain state to be applied as a loading (similar to a temperature loading) on the structural system. Solving the equation system for this loading gives a 2nd part of the internal force state, the **“secondary state”** or **“constraint state”**.

I.e. the total internal force state is separated into 2 parts,
- the V*e-state (respectively **primary state**) and
- the constraint state (respectively **secondary state**)

The 2 parts are separately stored and controlled in the database. They are only added in the superposition processes if required. This separation into 2 parts cares for the design rules in most national codes, where often the primary and secondary states have to be treated differently with respect to safety conditions.

Some approximation are assumed in the process for calculating the V*e-state:
- Cross-sections remain plane (linear stress distribution in the cross-section)
- Friction forces are neglected (theoretically there were shear components due to friction)
- Reference cross-section is always the original cross-section (without subtracting the duct holes or adding the weighted tendon cross-sections)

These approximations are generally made in engineering and may be seen as allowable in most cases. However, to user must be conscious of these approximations when evaluating and judging the results, because they may cause in special contexts essential deviations from the exactly resulting state.

Primary state and secondary state are separately stored in the result files and may be separately printed or viewed in the function ⇧RESULTS ⇨LCASE. This holds for true for internal forces of the structural elements as well as for the tendon forces.

The primary state is always the state directly resulting from the stressing action. The results of all later applied loads, specially also tendon force losses due to later stressed other tendons, are stored as secondary forces. Only tendon force losses due to creep and shrinkage are themselves again separated into a primary and secondary part.

5.8 Tendon Calculation in the Construction Schedule

Given below is a check list of the items that must have been defined before the calculation of the pre-stressing load case can be made:

- Material properties for the tendons (⇧PROPERTIES ⇨MATERIAL)
- Tendon geometry for the tendons (⇧STRUCTURE ⇨TENDON)
- Number of tendons per tendon group (⇧STRUCTURE ⇨TENDON)
- Stressing procedure (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE button ⇩TENDON)
- Load Sets for pre-stressing (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS button ⇩LSET)
- Load Case for pre-stressing (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS button ⇩LCASE)

Complete the missing definitions if necessary before adding the calculation action into the Construction Schedule. The ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE button ⇩ACTION can be selected once the element activation for at least one construction stage has been done.

The construction state to be considered must be selected in the upper table (construction state table). The necessary actions are then inserted in the action table (lower table of the GUI) (refer chap. 7, Construction Schedule). First the stressing actions have to be inserted, they calculate the primary state but do not perform a Load Case analysis:

⊙ **Calculation Actions**

- **STRESS – tendon calculation**

The Action "Stress" performs the simulation of the stressing process. The resulting forces (fixed end forces equivalent to the effectively acting forces, primary state of the pre-stressing) are assigned to the related Load Set and may be used in the Load Case analysis.

The "Stress label", specified in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩TENDON in order to group together different "Stressing actions" is now entered to define the stressing actions which should be performed in the actual Action "STRESS". If different Stress labels have been assigned in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩TENDON to stressing actions being performed before the actual Load Case is calculated, then the Action "STRESS" has to be subsequently performed for each "Stress label" before proceeding to the Action CALC for calculating the secondary internal force state.

Note: _It is normally possible (if no tendons are later on re-stressed) to assign the same Stress Label to all Stressing Actions, and to perform the action "STRESS" in advance for all tendons. The primary internal forces actually being assigned and used for the Load Case calculation are then selected be the tendons to be stressed as specified in the Load Set definition_

Input data for the Calculation Action "STRESS":

- Inp2: Stress label — Label identifying the required tendon stressing procedure at this stage
- Out2: List file — '*' means, that the default name STRESS.LST is used. (actually it is not possible to assign a user defined file name)
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

The Load Case calculation action is then selected after the required STRESS action(s) have been defined:

⊙ **Calculation Actions**

- **CALC – Load Case calculation**

The input for this Action is performed in the same manner than described in chapter 7.3:

- Inp1: Load Case No. — Selection of the Load Case to be analysed – interactively by using the pull-down menu arrow.
- Out2: List file — '*' means that a list file with the default name is created (e.g. LC0509.LST for Load Case 509). The user can overwrite the '*' and define any name for the output listing containing the Load Case results (displacements, primary and secondary internal forces).
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

The final step is to grout the considered ducts. This is performed in the Action GROUT.

⊙ **Calculation Actions**

- **GROUT – Grouting the ducts (establishing adhesion between the tendon and the cross section (for internal tendons only).**

No system reaction are calculated in the Action GROUT. The only effect is, that the composite cross-section is established (cross-section values are adapted) for the concrete stress calculation (not for further Load Case calculations!).

This adaptation of the cross-section values is however only performed, if the related calculation option (+Tendon area) in the function ⇧RECALC is also set to yes (see chap. 5.9, Calculation Options).

- Inp1: Tendon: From, to, step — Series of tendons to be grouted
  The tendons to be grouted are not specified via the "Stress label", but directly by specifying here a series of tendons. This allows to grout only some of the previously stressed tendons and grouting others later on. It is also possible to let some of the tendons un-grouted till the end.
- Out1: List file — Provided for Action related output data, actually not active (no output data are written)
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

The numerical results of all Load Cases can be viewed in the output listing (lc????.lst or user defined) or interactively in ⇧RESULTS ⇨LCASE. The Load Case results can also be viewed graphically by producing a plot file in ⇧RESULTS ⇨PLSYS.

5.9 Calculation Options for Pre-stressing related Actions

As already mentioned, several simplifications and approximations have been introduced for the treatment of the pre-stressing in the analysis process. These assumptions and pre-conditions allow a more or less accurate analysis. The methods and simplifications used in the analysis can be controlled by some calculation options defined in the ⇧RECALC input pad.

5.9.1 Treatment of Tension Force Losses

Grouted tendons:

Tendons already stressed and grouted normally get stress changes due to any ensuing Load Cases. These stress changes are equivalent to changes of the separately stored tension forces assigned to the tendons rather than to the structural elements. These change are called "Tension Force Losses" in this context, although they also might increase the tension forces in the tendon. Changes of the tendon forces due to external Load Cases (weight loads, traffic, temperature, etc.) are automatically calculated for all grouted tendons, and stored in the database if the option for storing the tendon forces is set (see next section).

However, for later pre-stressing Load Cases only the secondary part (constraint part) of the internal force state is automatically considered without setting the appropriate option:

In order to take into account also the tension force changes due to the primary state of later applied pre-stressing Load Cases, it is necessary to set the option

- Calc. losses for el. compression V*e

to yes!

Attention:	**The option „Calc. losses for el. compression" <u>should always be set,</u> if different tendons of a certain section of the structural system are stressed and grouted at different times. Only the secondary (constraint) part is considered for the calculation of tendon force losses if this option is not set.**

Un-grouted tendons:

No automatic calculation of tendon force losses is actually provided in RM2000 for un-grouted tendons, even not for internal forces due to normal external Load Cases or the constraint part of pre-stressing Load Cases. This shortage can approximately be overcome by treating un-grouted tendons as grouted tendons with respect to the tendon force calculation (not for the concrete stress evaluation).

The option

- Losses ungrouted = grouted

allows to consider un-grouted tendons as grouted tendons in the tendon force loss calculation process. This is a useful assumption especially in the case where the friction is very high (e.g. deviator blocks of external tendons). However, also for normal tendons this assumption is mostly better than completely omit the losses due to later Load Cases.

5.9.2 Storing the Tendon Results

The tendon forces are separately allocated in the database and not automatically stored. They need not be stored for each Load Case if they are not needed for later design checks (e.g. only fibre stress check but no ultimate load check is performed). But generally the 2 options

- Save tendon results (LC) and
- Save tendon results (Env)

should be set to „yes" in order to have the correct tendon forces available in all eventually required later performed design checks (e.g. for checking the additional strains in the tendon).

5.9.3 Calculation of Concrete Stresses

The primary results of static analyses of beam structures are always deformations and internal forces, where the full original concrete cross-section (gross cross-section) is always used for calculating the beam element stiffness matrices. The real cross-section in the case of pre-stressed structures is however initially weakened by the duct holes (net cross-section). After grouting the tendons the cross-section is strengthened with respect to the original one due to the higher Young's modulus of the pre-stressing steel with respect to the concrete module (composite cross-section).

It is generally not necessary to adapt the cross-section for the standard static analysis determining the internal force state, because the internal forces do not vary very much with stiffness changes. It is however mostly required to introduce the correct cross-section values for calculating the concrete stresses, because the deviations of the stresses may be essential.

In order to perform the required cross-section adaptations for the concrete stress calculation RM2000 offers 3 options in the menu ⇧RECALC „Special settings":

- Update CS (+ tendon steel area)
- Update CS (- duct area)
- Update CS (+ fill area)

Update CS (+ tendon steel area):
If this option is set, then the tendon steel area factored by $n = E_s/E_c$ is taken into account for calculating the composite cross-section to be used for evaluating the concrete stresses. Usually selecting this option is only meaningful together with the option for subtracting the duct area described below. Otherwise the concrete stiffness in the tendon steel area would be taken into account additionally to the steel stiffness.

If no duct exists (pre-stressing in a pre-stressing bed), then it is recommended nevertheless to define a duct area (equal to the steel area) in order to get the correct composite cross-section.

The composite cross-section is used for the stress evaluation for all Load Cases applied after grouting (Action GROUT).

Update CS (- duct area):
If this option is set, then the net cross-section values (cross-section weakened by the duct holes) are used for evaluating the concrete stresses due to all Load Cases applied before grouting of the tendons. Subtracting the duct holes is performed for all Load Cases, after grouting the steel area is again added if the above described option is set.

Update CS (+ fill area):
This option governs the treatment of the grout material in the calculation process of the composite cross-section. If the option is set, then for the grout material the full area be-

tween the tendon and the duct is considered with the same Young's modulus than the structural concrete.

Some design codes do not allow to take into account the longitudinal stiffness of the grout material, in this case the option must not be set. A possibility for partially considering this stiffness is actually not provided in RM2000.

6 Loading

6.1 General

The term "Load" or "Loading" is used within this guide in a general sense and comprises not only external loads (forces) but also any other impacts on the structure, like temperature changes or prescribed deformations, and also other conditions influencing the stress and deformation state, like initial stresses or – in the dynamic analysis – masses and possibly initial velocity or acceleration fields and response spectra.

The loading is applied on the structure in "Load Cases", characterizing either a total state, or a differential state compared to a previous state. Note, that linear analyses allow the later superposition of load case results, and hence the separate treatment of different impacts in separate load cases, whereas non-linear analyses always require working with total states, where all loads acting at a certain moment are applied in one load case.

A Load Case is made up by "Load Sets" and is the primary unit to apply loading for any different type of analysis done by the program. The Load Sets form the basis for most of the loading (exception: traffic loads).

"Load Types" describing the different types of impacts and the different kinds of distribution on the system, are used to effectively apply almost any conceivable applied bridge load on the structure. These Load Types can be applied to the elements or nodes, and include e.g. point loads, uniformly or trapezoidally varying distributed loads, displacements, temperatures and other effects.

Both the construction loading and the final stage loading (except for the traffic loading) must be split up into Load Sets that are inseparable and always act together. The Load Set can consist of a single load or several loads.

- Every load is defined separately.
- Several loads can be combined into one LOAD SET
- Several LOAD SETS can be combined to form one LOAD CASE
- Every LOAD SET can be factored (constant and/or variable) within a LOAD CASE.
- The results from LOAD CASES can be combined in many ways to form envelopes.
- Result envelopes can be combined with other result envelopes to form an envelope of the envelope.
- All the loading cases can be individually factored before being combined into an envelope.

- All the envelopes can be individually factored before being combined into another envelope.
- The results from an individual load case can be added to another load case or added/combined into an envelope.

6.2 Load Set

A Load-Set is a selection of individual loads that are to be applied to the structure collected together into one group of loads.

The Load-Set is the basic loading group and cannot be sub-divided.
(The Load-Case, refer below, is made up from a combination of Load-Sets)

A Load-Set may consist of one single load (of any type – refer to Load-Types below) or a group of loads applied to a single element or a group of elements.

6.3 Load Types

1. Concentrated load
2. Uniform load
3. Partial uniform load
4. Trapezoid and Triangle load
5. Masses
6. Stressing
7. Initial stress/strain loads (temperature, …)
8. Action on element end
9. Wind load (velocity)
10. Normal forces (stiffness change)

6.3.1 Concentrated Loads

Concentrated Loads are used to apply concentrated forces and moments at arbitrary locations on nodes or on beam elements. The direction of loading may be specified in the global coordinate system or in the local element coordinate system.
The location of the load may be specified in one of the following ways:

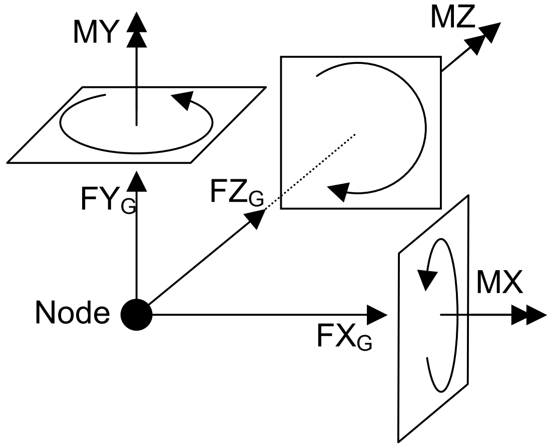
- Specifying a node
- Specifying a relative distance from the element begin to the point of application of the load. x/l must satisfy following condition: $0 \leq x/l \leq 1$.

Any number of concentrated loads may be applied to each node or element. Loads applied on elements may be specified either in global coordinates or in the local element coordinate system. Multiple loads that are applied at the same location are added together.

6.3.1.1 Concentrated nodal loads

F Point loads acting on nodes (Forces and Moments defined in the global coordinate system).

Note: *Be aware, that the moments are right-hand turning although the global system is a left-hand system.*



6.3.1.2 Concentrated element loads (eccentric forces)

This group of Load Types allows the specification of point loads by specifying the 3 force components and the location, where this force acts on the system. The force may be specified either in the global coordinate system, or in the local element coordinate system. The location in the element is specified by the relative distance x/l to the element begin and the eccentricities given in the local element coordinate system. The moments are internally calculated using the specified eccentricities.

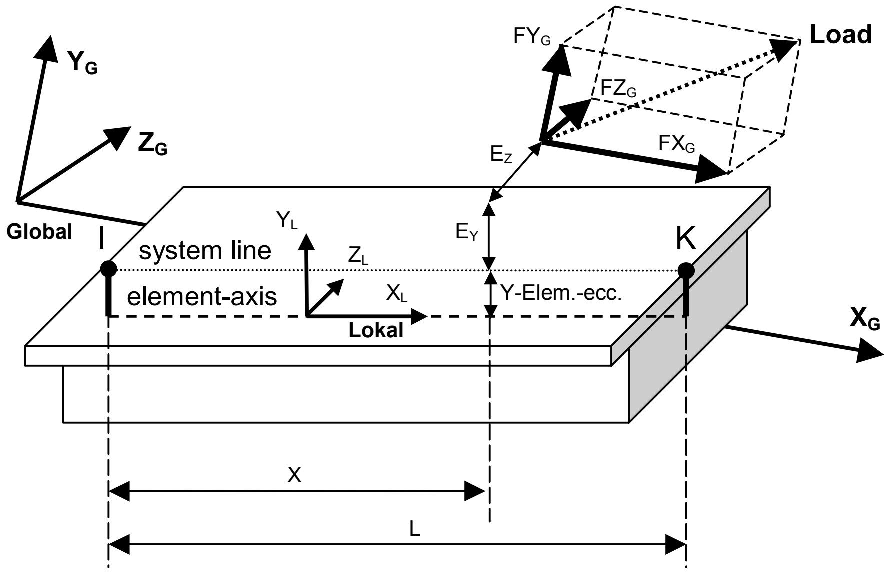
FSG Concentrated element load in the global coordinate system.
FSL Concentrated element load in the local element coordinate system.
The eccentricity is defined in the local element coordinate system (vector from the element axis to the load application point).

FSGY Concentrated element load in the global coordinate system.
FSLY Concentrated element load in the local element coordinate system.
Similar to FSG and FSL resp., but the specified load eccentricity in Y-direction (e_y) is not measured from the centroid (element axis) but from the connection line between the two structural nodes of the element (system line). The Y-component of the structural eccentricity between the connection line of the nodes and the element centroid is automatically added to the specified load eccentricity.

Note: *The structural eccentricity is affected by the following:*
- The cross-section properties
- The cross-section assignment (nodes at the top, nodes at the bottom)
- Additional element eccentricity assignment in global or local coordinate system

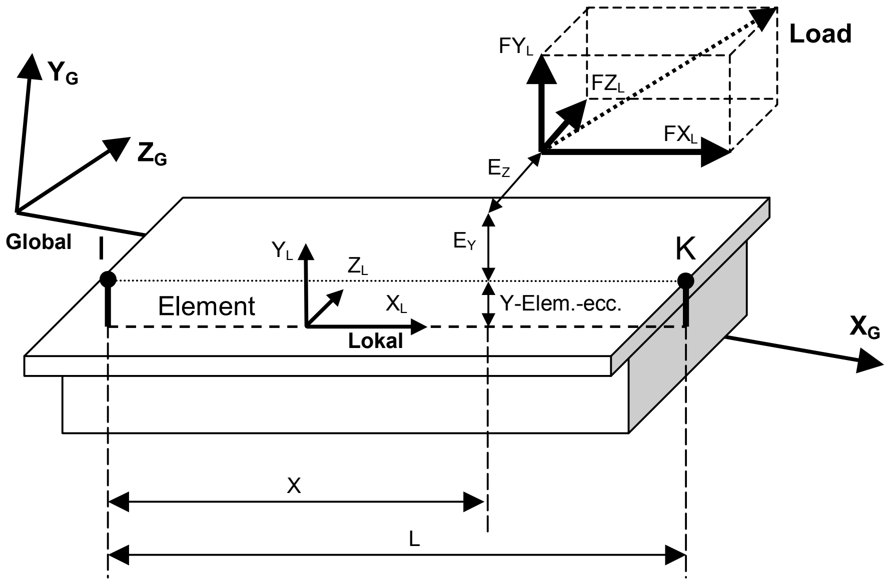
Concentrated element load FSGY

E_y = vector from the **system line** to the load application point,
E_z = vector from the **element axis** to the load application point.



Concentrated element load FSLY

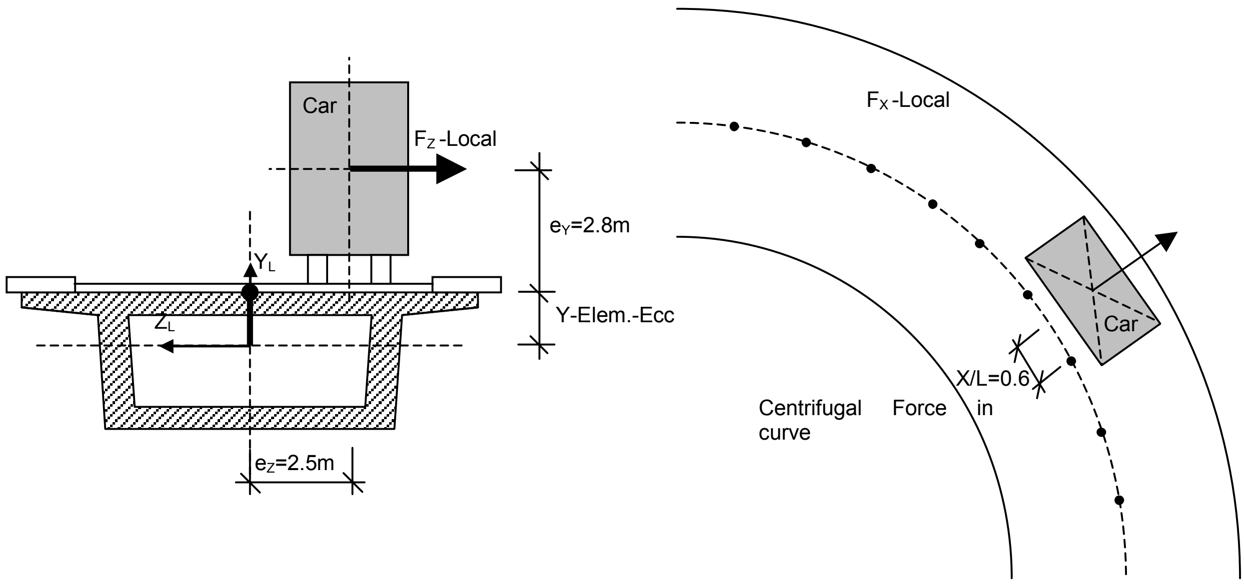
E_y = vector from the **system line** to the load application point,
E_z = vector from the **element axis** to the load application point.



Application examples:

This Load Type will be typically and advantageously used in the case, where the position of the centroid is not a priori known and a lateral load (e.g. wind load on or centrifugal or braking force of a truck) acts at a given height above the roadway surface. This load type allows in this case a much easier specification of the eccentricity, being the same for the whole superstructure even when the cross-section height is varying.

Example 1: centrifugal force on a curved bridge

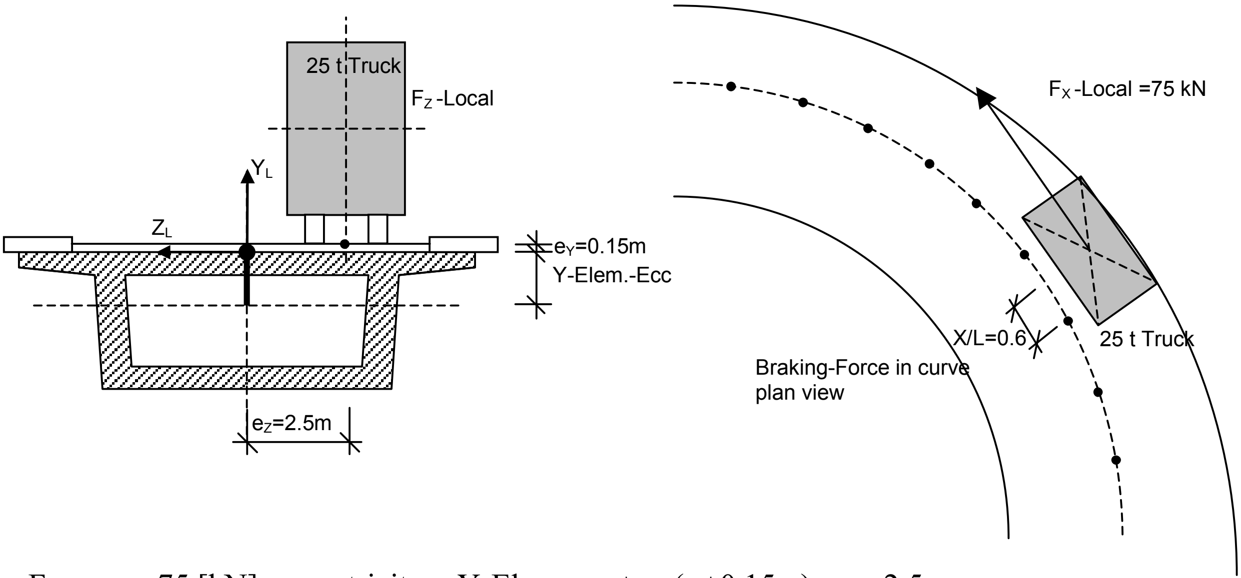


Eccentricity: Y-Elem.ecc.+ local e_Y (=+2.8m), local e_Z (=-2.5m)

Example 2: BRAKING FORCE in acc. with AUSTRIAN Standard:

- 30% of the heaviest regular vehicle
- acting at the top of the pavement

Example: 25 t Truck => Braking force: 0,3*250 = 75 kN



$F_{X\text{-LOCAL}}$=75 [kN], eccentricity = Y-Elem.ecc.+e_Y (=+0.15m), e_Z=-2.5m

FSGZ Concentrated element load in the global coordinate system.

FSLZ Concentrated element load in the local element coordinate system.
Similar to FSGY and FSLY resp., but the specified load eccentricity in Z-direction (e_z) is not measured from the centroid but from the connection line between the two structural nodes of the element. The Z-component of the structural eccentricity between the connection line of the nodes and the element centroid is automatically added to the specified load eccentricity.

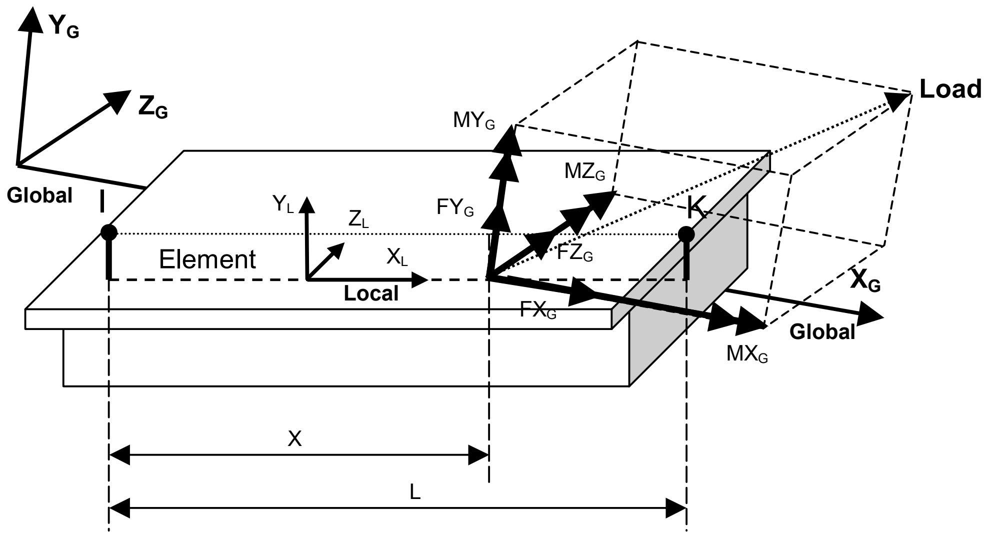
E_y = vector from the element axis to the load application point,
E_z = vector from the system line to the load application point.

Application example:
This load type will be typically and advantageously used in the case, where horizontal structural eccentricities exist, and the given horizontal load eccentricity is related to the connection line of start and end nodes rather, than to the centroid line. This will happen more seldom than in the vertical direction, but in the case of a changing roadway width varying horizontal structural eccentricities this load type might be helpful.

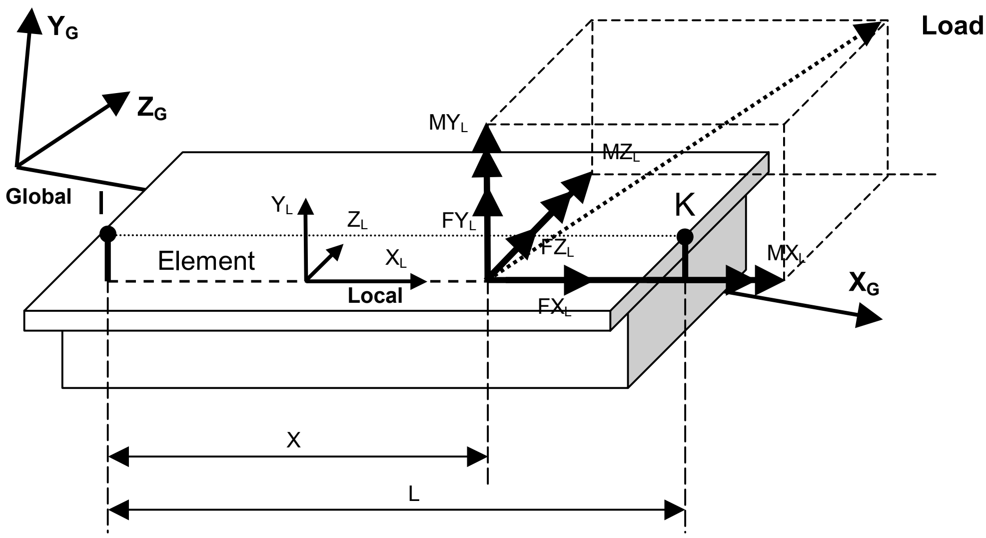
6.3.1.3 Concentrated element loads (forces and moments)

This group of Load Types allows the specification of point loads by specifying the full force vector consisting of the 3 force components and 3 moments acting on the system. The forces and moments may be specified either in the global coordinate system, or in the local element coordinate system. The location in the element is specified by the relative distance X/L to the element begin.

FSGM Concentrated element load (force and moment) in the global coordinate system.



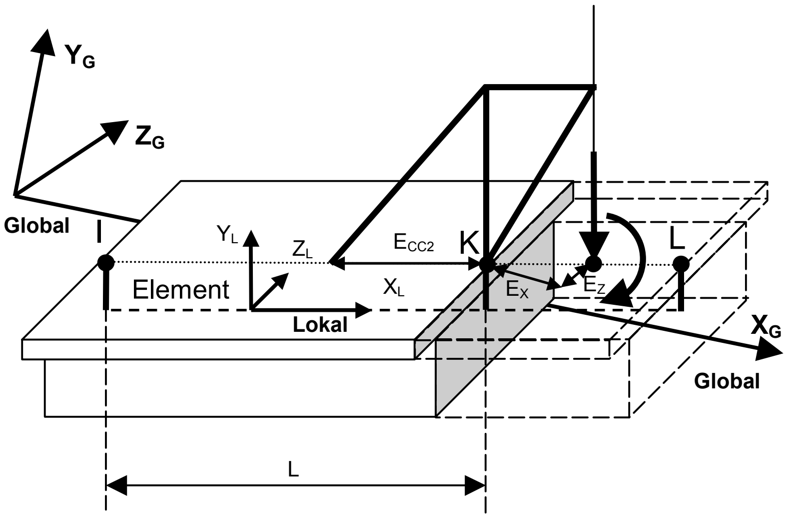
FSLM Concentrated element load (force and moment) in the local element coordinate system.



6.3.1.4 Concentrated element load as node load

This Load Type has been provided with in-situ segmental cantilever construction in mind. The inactive 'wet concrete' weight is specified by defining the specific weight of the material (GAMMA) for the relevant elements. The volume is taken from the cross-section and length definitions provided previously in the database. This makes in many cases the load definition much easier, especially in the case of varying cross-sections.

FSY Distributed Element loads integrated over a series of elements and applied to a node. Self weight of inactive elements applied as nodal forces.



The load vector acting on the specified node is defined by the definition of an element series and the appropriate specific weight GAMMA. A positive GAMMA will yield a force in the negative global Y-direction.
The position of the load vector is not automatically calculated by the program. The definition of an eccentricity vector E_X,E_Y,E_Z in the global coordinate system allows applying the load in an eccentric position.
An eccentricity ECC2 may be specified to split the moment Mz into an appropriate couple with lever arm ECC2.

Note: *The conversion into a couple with ECC2 is currently neither implemented nor available.*

6.3.2 Uniformly Distributed Loads (UDL)

The UDL is used to apply distributed forces and moments on beam elements over the whole element length. The direction of loading may be specified in the global coordinate system or in the local element coordinate system.

Basically, the UDL Load Type is described as a line load related to the length of the element (force per unit length). This distributed line load is transformed internally in the program into forces and moments acting on the nodes in accordance with the deformation method theory for beams.

Deviating from this basic definition, the UDL may be defined as

- Projected load
- Nodal load
- Surface load

as described below.

Property "Projected Load":
Projected loads are related to the projection of the element in load direction rather, than to the real element length. This is commonly used e.g. for the definition of distributed snow or wind loads, where the load intensity is measured per unit length of the element projection. The intensity would be the depth of snow or the wind speed; the projected element length is measured in a plane perpendicular to the direction of loading.

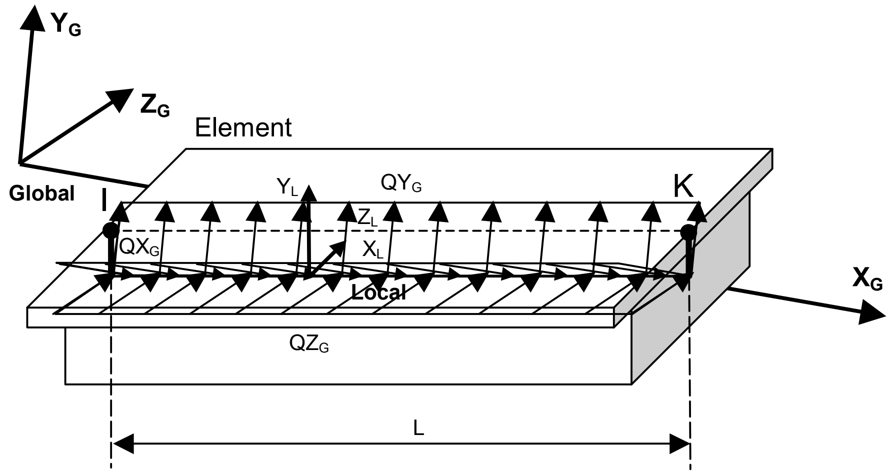
Property "Nodal Load":
An UDL specified as nodal load will be transformed into two equivalent point loads acting on the start and end nodes of the element. This means, that the end moments theoretically resulting from the distribution of the load over the element length will be neglected or not taken into account. An application example might be, that the load is indirectly applied on the nodes of the system, where the loading is only specified as distributed to take advantage of the element lengths and possibly widths already existent in the database.

Property "Surface Load":
A distributed load defined as a surface load [kN/m²] will be automatically multiplied by the height or the width of the cross section to get the appropriate line load value for the calculation of the nodal forces and moments used in the equation system.

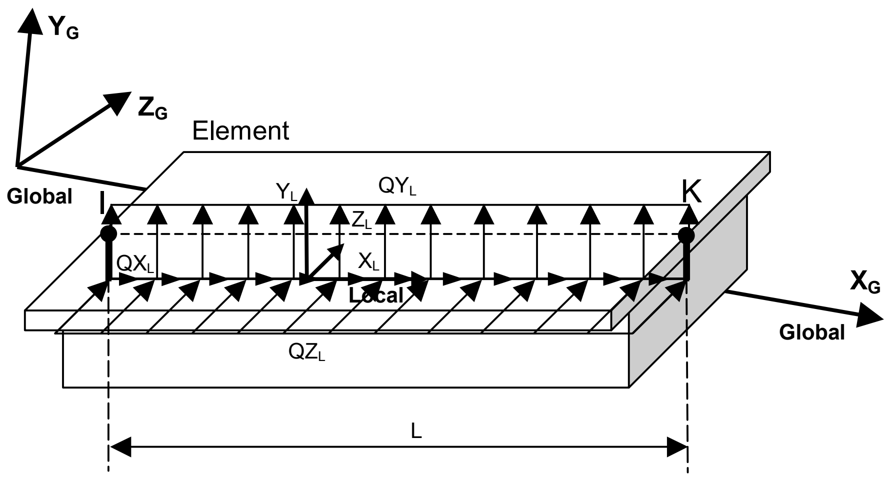
Note: *Surface loads will automatically yield a linearly varying distributed line load (LDL) in the case, that the cross-section widths or heights differ at the element ends.*

6.3.2.1 Uniform concentric element load

QG Concentric UDL (global) - Uniformly distributed concentric element load defined in the global coordinate system acting over the whole element length.



QL Concentric UDL (local) - Uniformly Distributed concentric element load defined in the local element coordinate system acting over the whole element length.

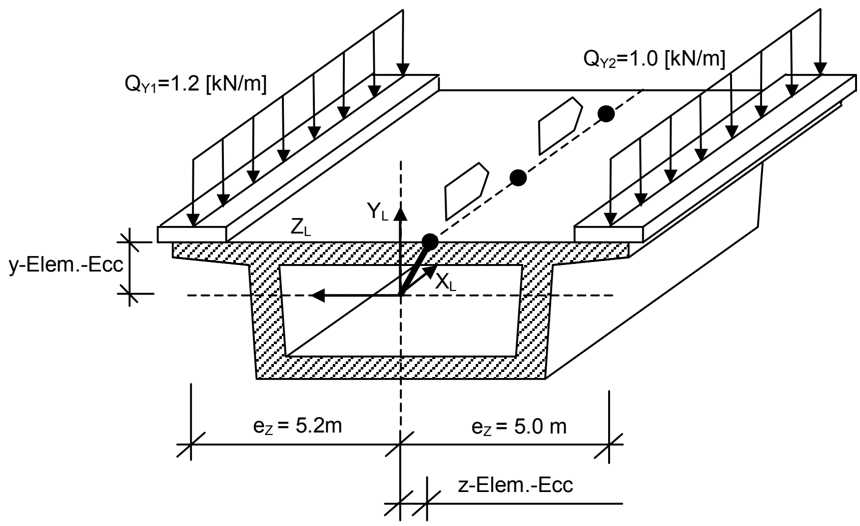


6.3.2.2 Uniform eccentric element load

QEXG Eccentric UDL (global) - Uniformly distributed eccentric element load in the global coordinate system acting over the whole element length. The eccentricity is defined in the local element coordinate system with reference to the **cross-section centroid**.

QEXL Eccentric UDL (local) - Uniformly distributed eccentric element load in the local element coordinate system acting over the whole element length. The eccentricity is defined in the local element coordinate system with reference to the **cross-section centroid.**

Example 1: superimposed dead load - walkway
This is only an application example and the load can be defined with other load types.



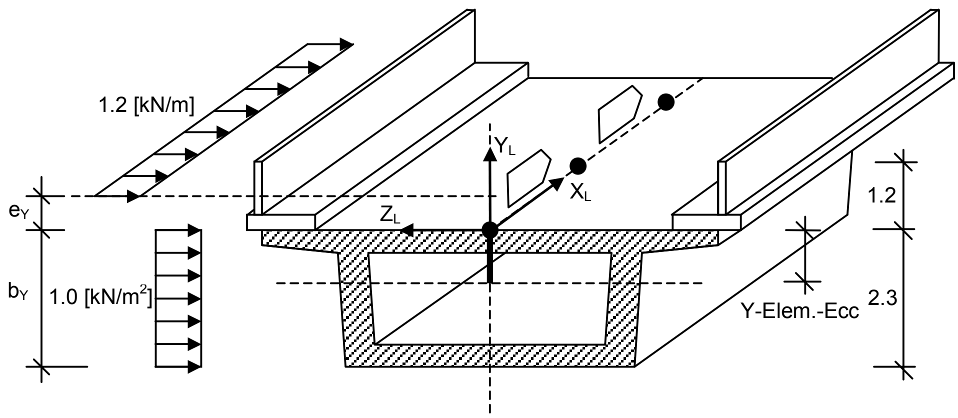
Qy=-1.2 [kN/m], Eccentricity: Y-Elem.ecc., local e_z (=+5.2m)
Qy=-1.0 [kN/m], Eccentricity: Y-Elem.ecc., local e_z (=-5.0m)

Note: *Basically in RM2000 load eccentricities are defined in barycentric local coordinates (QEXG, QEXL). Using the load types QEYG, QEYL the load eccentricity in local y-direction is based on the system line. All other eccentricities are based on the element axis. For the load types QEZG, QEZL this applies analogously in local z-direction.*

QEYG Eccentric UDL in global direction acting on the whole length of the element.
QEYL Eccentric UDL in local direction acting on the whole length of the element. Similar to QEXG and QEXL resp., but the specified load eccentricity in Y-direction (e_y) is not measured from the centroid but from the connection line between the two structural nodes of the element. The Y-component of the structural eccentricity between the connection line of the nodes and the element centroid is automatically added to the specified load eccentricity.

This load type will be typically and advantageously used in the case, where the load is acting on top or bottom of the cross-section and the position of the centroid is either not known or varying along the span.

Example 2: Transverse wind load on the structure
This is only an application example and might be different for different codes.



Wind load on parapet: Load Type: QEYG or QEYL
Qz=-1.2 [kN/m], Eccentricity: global Y-Elem.ecc. + local e_y (=+1.2m/2=+0.6m)

Wind load on cross-section: Load Type: QG or QL
Qz=-1.0 [kN/m^2], with "Q_z load mult. by CS depth"

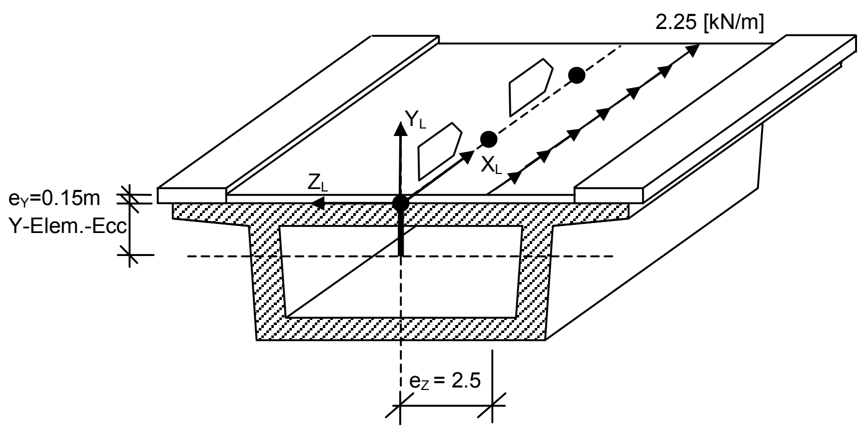
Note: *A distributed wind load defined as a surface load of 1.0 kN/m² will automatically be multiplied with the height of the cross section yielding a distributed concentric element load even when the centroid is not in the middle of the loaded projected surface. This theoretical load eccentricity is neglected!*

Example 3: BRAKING FORCE according to AUSTRIAN Standard
Design force according to the code is the worst of:

- 30% of the heaviest vehicle as point load
- 10 kN * Roadway width in m
- 5% of total uniform load as uniform load

All loads are acting at the top of the pavement

Example:	Length L=34+44+44+34=156 m		Width of the roadway = 9 m
	Heaviest vehicle	250 kN	Uniform traffic load = 5 kN/m^2
	=> 0.3 * 250 =	75 kN	
	=>10.0 * 9.0 =	90 kN	
	=>9.0*156*0.05*5 =	351 kN	=> decisive force
	=> q_x = 351/156 =	2.25 kN/m	or 0.25 kN/m^2



QEZG Eccentric UDL in global direction acting on the whole length of the element.
QEZL Eccentric UDL in local direction acting on the whole length of the element. Similar to QEYG and QEYL resp., but the specified load eccentricity in Z-direction (e_y) is not measured from the centroid but from the connection line between the two structural nodes of the element. The Z-component of the structural eccentricity between the connection line of the nodes and the element centroid is automatically added to the specified load eccentricity.

Note: _All uniformly distributed load-types listed above (except self weight) may by defined as line loads (force per unit length) or as surface loads being multiplied in the analysis by the cross-section height or width. They also may either be related to the real element length or to the projection of the element length to a plane normal to the force direction._

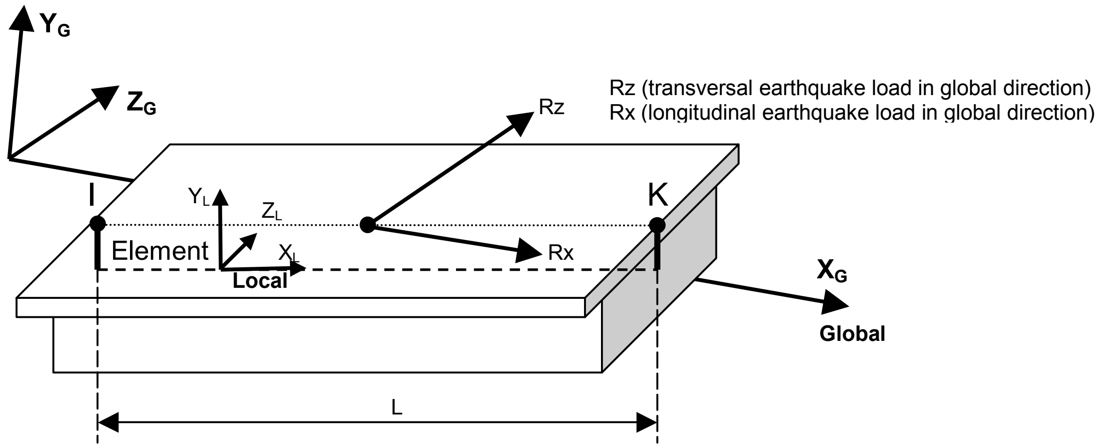
6.3.2.3 Self weight

G Self-Weight Load activates the self-weight of all specified elements. For a beam element, the self-weight is a force distributed along the element length. The input value „Gam“ specifies the specific weight to be considered. The material parameter „Gamma“ is used if no value Gam is specified (Gam=0.0). The magnitude of the self-weight is equal to the weight density, gamma, multiplied by the cross-sectional area. For the Self-Weight Load you also have to define the direction vector of the load (usually 0/-1/0 to define, that the load acts in the negative Y direction).

Note: *The average cross-section area is taken if the cross-sections at the element start and the element end differ. This average area is multiplied by the specific weight giving the appropriate UDL value.*
The defined direction vector is a ***normalized vector****. The magnitude of the vector is always 1.00, even if you define 0/0/0.20, the program generates a normalized vector 0/0/1.0!*

Example: Static earthquake loading

Assumption: 2 Load Sets have to be applied in 2 separate Load Cases for simulating either an earthquake in the longitudinal direction or an earthquake in lateral direction. This is only an application example and might be different in different codes or cases.



Load Set 1: Static earthquake in longitudinal direction:
Rx=1.00, Ry=0, Rz=0 density $\gamma = 25kN/m^3$ (100% of the self-weight in x-direction),
Load Case 1 with Load Set 1 and a constant vactor of 0.05 (5% of Load Set 2 in x-direction),

Load Set 2: Static earthquake in transversal direction:
Rx=0, Ry=0, Rz=1.00 density $\gamma = 1.25kN/m^3$ (5% of the self-weight in z-direction),
Load Case 2 with Load Set 2 and with a constant vactor of 1.00.

6.3.2.4 Self weight – just as load

G0 Same as “G”.
But in a dynamic analysis the self weight defined with G0 is considered as a static or dynamic load, but **without mass**.

Note: *Quod vide chapter 9.2.3 „Definition of the masses”*

6.3.2.5 Self weight – just as mass

GM Same as “G”.
The self weight defined with GM is only considered as a mass in a dynamic analysis, but not as a static or dynamic loading.

Note: *Quod vide chapter 9.2.3 „Definition of the masses”*

6.3.3 Partial Uniformly Distributed Loads

The partial UDL is used to apply distributed forces and moments on beam elements over a part of the element length. The direction of loading may be specified in a global coordinate system or in the local element coordinate system.

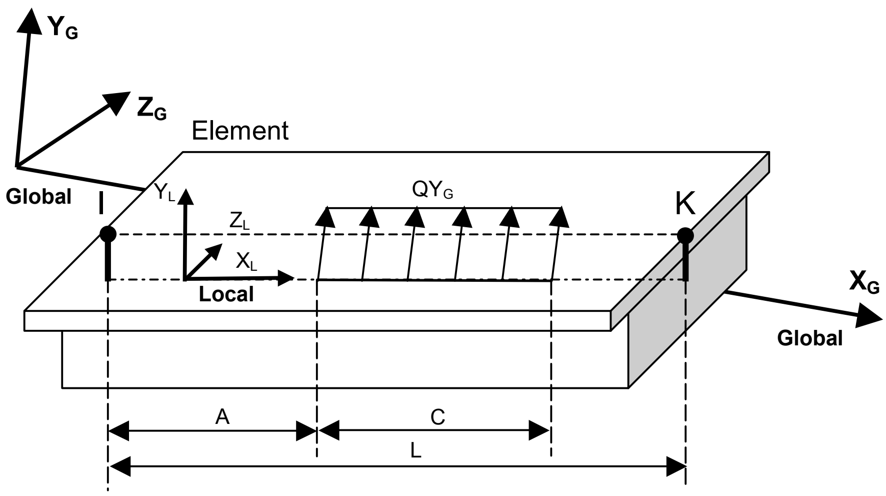
As for the above described Load Type “UDL over the whole element length”, the partial UDL is basically described as a line load related to length (force per unit length). This distributed line load is transformed internally in the program into forces and moments acting on the nodes in accordance with the deformation method theory for beams.

Deviating from this basic definition, the UDL may be defined as
- Projected load (related to the projection length or area)
- Nodal load (neglecting fixed end moments)
- Surface load (related to the surface area instead of the length)

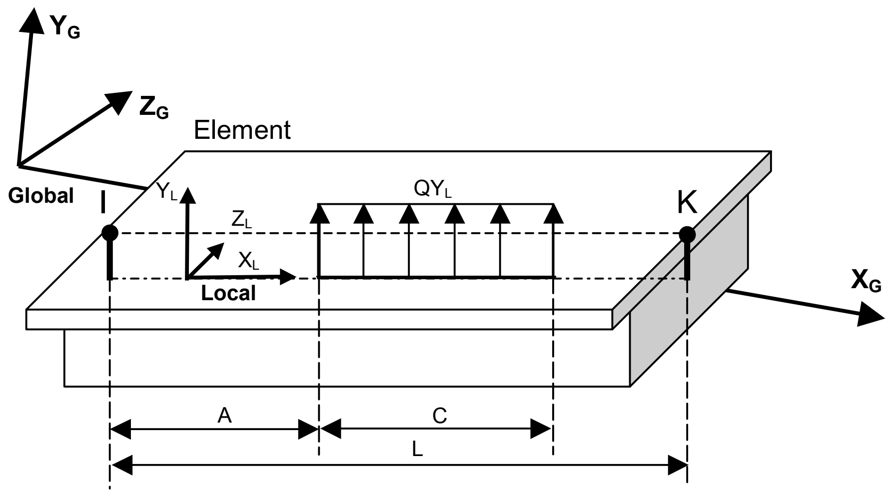
as described in detail in the previous section.

6.3.3.1 Concentric partial uniform element load

QTG Concentric partial UDL (**vector Q_X, Q_Y, Q_Z global**) - Uniformly distributed concentric element load defined in the global coordinate system acting over a part of the element length.



QTL Concentric partial UDL (**vector Q_x, Q_y, Q_z local**) - Uniformly distributed concentric element load defined in the local element coordinate system acting over a part of the element length.



6.3.3.2 Eccentric partial uniform element load

QTZG Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **global** with local **z-eccentricity**).

QTZL Eccentric partial UDL acting over a part of the element length (vector Q_x, Q_y, Q_z **local** with local **z-eccentricity**).

QTYG Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **global** with local **y-eccentricity**).

QTYL Eccentric partial UDL acting over a part of the element length (vector Q_x, Q_y, Q_z **local** with local **y-eccentricity**).

QZZG Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **global** with local **z-eccentricity related to the system line**). The system eccentricity in z-direction (z-distance between system line and element axis) is automatically added.

QZZL Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **local** with local **z-eccentricity related to the system line**). The system eccentricity in z-direction is automatically added.

QYYG Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **global** with local **y-eccentricity related to the system line**). The system eccentricity in y-direction (y-distance between system line and element axis) is automatically added.

QYYL Eccentric partial UDL acting over a part of the element length (vector Q_X, Q_Y, Q_Z **local** with local **y-eccentricity related to the system line)**. The system eccentricity in y-direction is automatically added.

Note: *Eccentric partial element loads can only have either a y-eccentricity or a z-eccentricity. Load vectors with both, a y-* ***and*** *a z-eccentricity, have to be separated into 2 parts (e.g. a vector in z-direction with y-eccentricity and a vector in y-direction with z-eccentricity).*

keyword	orientation of the load	eccentricity of the load	system eccentricity
QTZG	global X, Y, Z	z	no
QTZL	local x, y, z	z	no
QTYG	global X, Y, Z	y	no
QTYL	local x, y, z	y	no
QZZG	global X, Y, Z	z	yes (only z)
QZZL	local x, y, z	z	yes (only z)
QYYG	global X, Y, Z	y	yes (only y)
QYYL	local x, y, z	y	yes (only y)

Note: *All uniformly distributed Load Types listed above (except self weight) may by defined as line loads (force per unit length) or as surface loads being multiplied in the analysis by the cross-section height or width. They also may either be related to the real element length or to the projection of the element length into a plane normal to the force direction.*

6.3.4 Linearly Varying Distributed Loads (LDL) (Trapezoidal or Triangular shape)

The linearly varying distributed load is used to apply non-uniformly distributed forces and moments on beam elements over the whole or a part of the element length. The direction of loading may be specified in a global coordinate system or in the local element coordinate system.

As for uniformly distributed loads, the LDL is basically described as a line load related to length (force per unit length). This distributed line load is transformed internally in the program into forces and moments acting on the nodes in accordance with the deformation method theory for beams.

Deviating from this basic definition, the LDL may be defined as

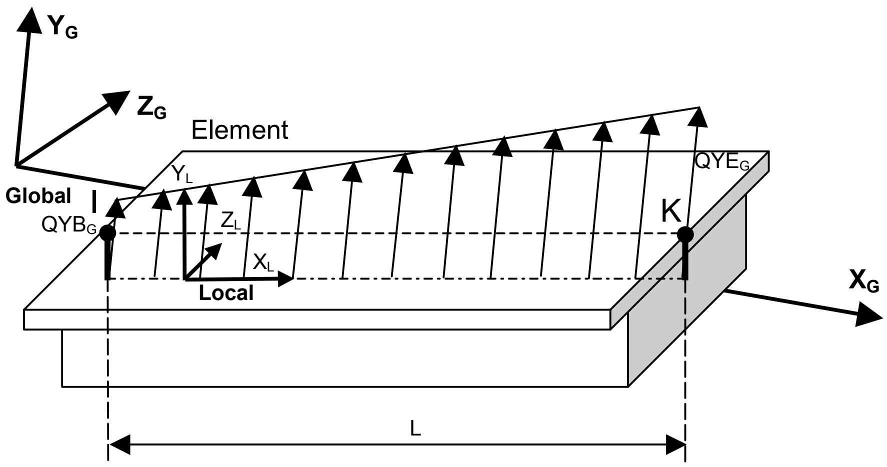
- Projected load (related to the projection length or area)
- Nodal load (neglecting element end moments)
- Surface load (related to the surface area instead of the length)

as described in detail in the previous sections.

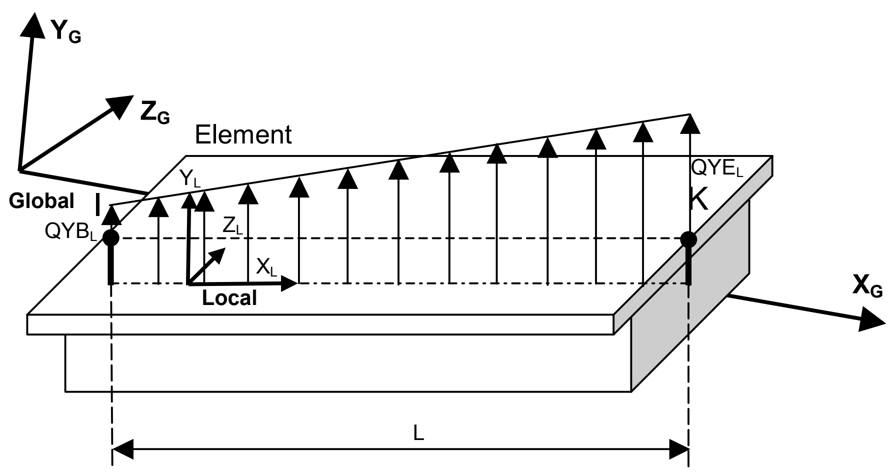
The variation of load intensity may be trapezoidal or triangular.

6.3.4.1 Trapezoidal element load

TG Trapezoidal LDL (**vector Q_X, Q_Y, Q_Z global**) - Linearly varying distributed concentric element load (trapezoidal shape) defined in the global coordinate system acting over the whole element length.



TL Trapezoidal LDL (**vector Q_x, Q_y, Q_z local**) - Linearly varying distributed concentric element load (trapezoidal shape) defined in the local coordinate system acting over the whole element length.

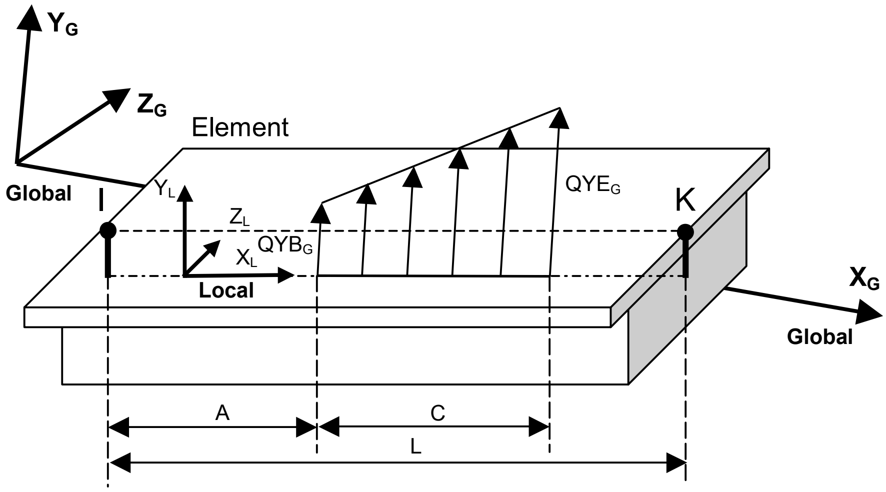


6.3.4.2 Partial trapezoidal element load

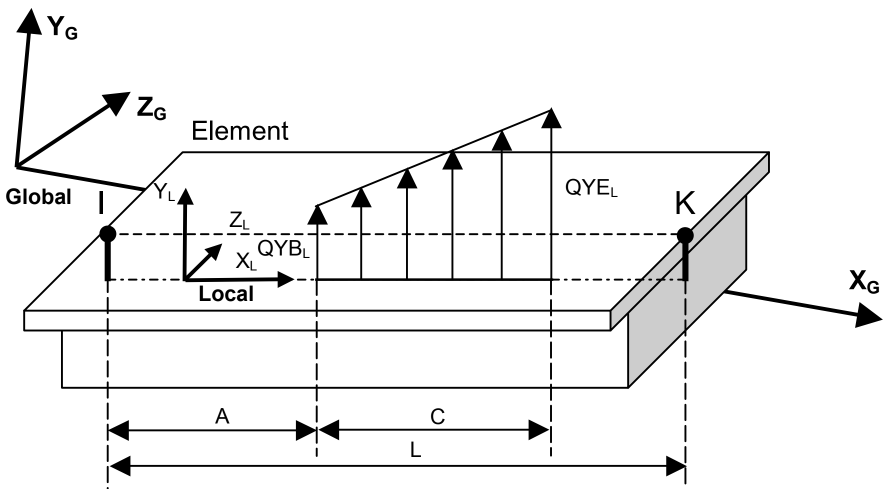
PTXG Partial Trapezoidal LDL (**global X-direction**) –Distributed concentric element load in global x-direction varying linearly over a part of the element.

PTXL Partial Trapezoidal LDL (**local x-direction**) –Distributed concentric element load in local x-direction varying linearly over a part of the element.

PTYG Partial Trapezoidal LDL (**global Y-direction**) –Distributed concentric element load in global y-direction varying linearly over a part of the element.



PTYL Partial Trapezoidal LDL (**local y-direction**) –Distributed concentric element load in local y-direction varying linearly over a part of the element.



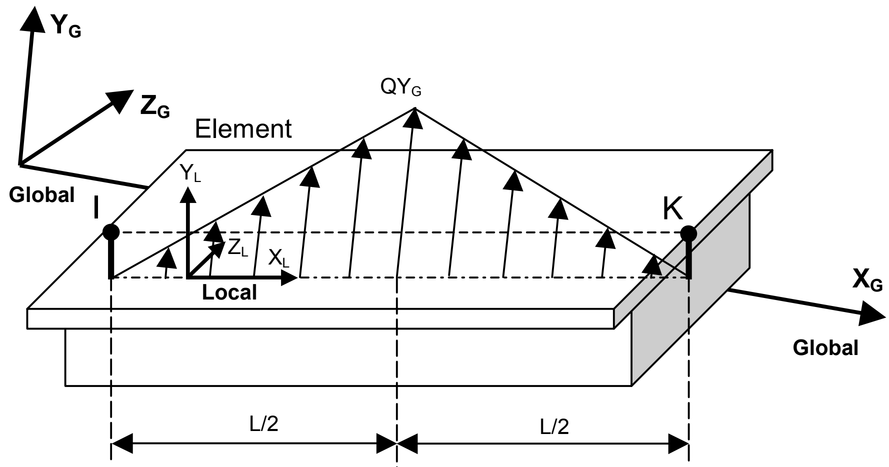
PTZG Partial Trapezoidal LDL (**global Z-direction**) –Distributed concentric element load in global z-direction varying linearly over a part of the element.

PTZL Partial Trapezoidal LDL (**local z-direction**) –Distributed concentric element load in local z-direction varying linearly over a part of the element.

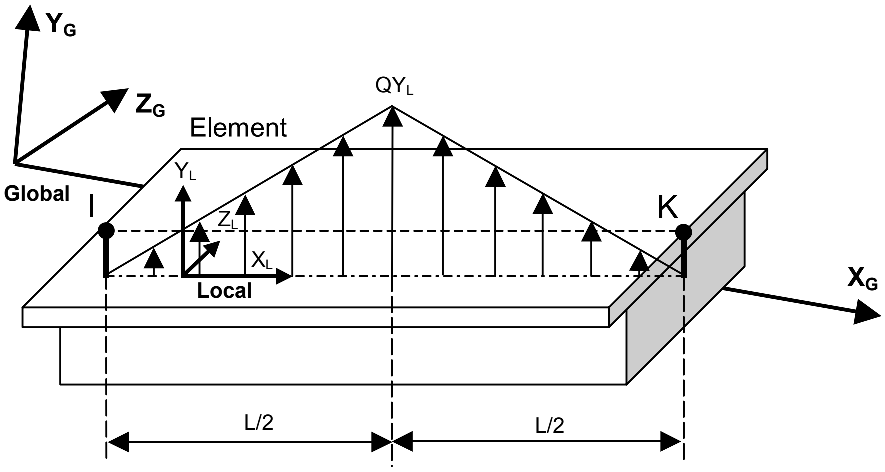
keyword	coordinate system	orientation of the load	eccentricity
PTXG	global	X	no
PTXL	local	x	no
PTYG	global	Y	no
PTYL	local	y	no
PTZG	global	Z	no
PTZL	local	z	no

6.3.4.3 Triangular element load

DREIG Triangular element load (global) – Distributed concentric element load defined in the **global** coordinate system, with the load intensity raising linearly from 0 to the specified value Q_{XG}, Q_{YG}, Q_{ZG} from both element ends to the element centre.



DREIL Triangular element load (local) – Distributed concentric element load defined in the local coordinate system, with the load intensity raising linearly from 0 to the specified value Q_{XG}, Q_{YG}, Q_{ZG} from both element ends to the element centre.



6.3.5 Masses

Three Load Type groups allow a mass definition:

- Nodal masses (Load Type 'MASSES – NODE MASS')
- Element masses (Load Type 'MASSES – ELEMENT UNIFORM MASS')
- Self weight (Load Type 'UNIFORM LOAD – SELF WEIGHT')

Note: *Masses are always entered* ***as forces*** *in the appropriate unit system. The program automatically transforms the value to real masses in the dynamic analysis, where they are required, using the defined gravity acceleration value. Masses may also be defined for static analyses where they are treated like nodal or element forces.*

The use of and the necessary specifications for the definition of masses are described in detail in chap. 9.

6.3.6 Pre/Post tensioning

6.3.6.1 Tensioning of Tendons

The Load Type "Tendon stressing" is used to input the loading due to pre-stressing a structural system.

TEND0 Applying the pre-stressing, defining the tendons to be stressed.

⊙ Increment-Force The difference with respect to a previous stressing process is considered
⊙ Total-Force The total force is considered in the calculation

Details see chapter 5.7. „Load Case Pre-stressing".

6.3.6.2 Cable / external tendon stressing

FCAB The pre-stressing force is directly assigned to an element of the type "cable" or to an external pre-stressing element. Exactly this normal force will be in the cable element after the load case calculation.

The function FCAB simulates the physical process „Cable tensioning" fully force related. The results will be exact for simple linear analyses.
The stressed element is separated from the system and the force FCAB is applied instead of it on the remaining structural system. The resulting internal forces and deformations are computed, then the element is again installed in the un-deformed structural system and the normal force in the element is set to FCAB.

If the cable force at the end of the stressing process is known, then this Load Type can be used to stress the force directly into the system. Other than in the case of using e.g. "FX0" the normal force in the stressed element will be exactly the applied force.

Attention: **This Load Type should not be used for complex, non-linear calculations! Analyses considering p-delta effects (2nd order theory), large displacements or non-linear cable element are not fully compatible with the loading function FCAB. The related cable elements are not active, therefore the non-linear effects cannot be fully considered. It is recommended to use in these cases an other Load Type instead (e.g. LX0, FX0).**

Note: _Since, by calculation of a Load case with this Load Type, the stressed element is separated from the system, no other Load should be defined on the stressed element in this Load case._

6.3.7 Initial Stress/Strain Loads - Temperature

6.3.7.1 Temperature loading

The temperature load creates a thermal strain in the beam element. This strain is given by the product of the material coefficient of thermal expansion and the temperature change. In accordance with the beam theory without warping, the temperature strain may vary linearly over the cross-section. This variation is described by 3 components of the temperature change:

- The temperature change part being constant over the cross-section (DT-G).
- The temperature gradient in the local y-direction (TG_y).
- The temperature gradient in the local z-direction (TG_z).

Some design codes require a non-linear variation of the temperature over the cross-section to be investigated (e.g. AASHTO). *RM2000* offers a possibility to take this demand into account by defining the temperature distribution in the cross-section very detailed and selecting the function TempVar to calculate the appropriate integrals characterizing the uniform temperature change and the gradients. This function is more in detail described in the next section.

The variation of the temperature load in longitudinal direction (over the element length) is actually always assumed to be constant over the whole element length. A suitable subdivision of the structure into small elements has to be made to simulate by piecewise constant sections a temperature load essentially varying in longitudinal direction.

Note: *The program does not have the possibility to enter an initial temperature (i.e. the temperature characterizing the initial stress-less state). All temperature values such as DT-G or the temperatures assigned to Cross Section Temperature Points are therefore not absolute values but differences with respect to the initial temperature of the structure. The average element temperatures assigned to the elements in ⇧STRUCTURE⇨ELEMENT ⇩TIME are not considered for the calculation of the temperature loading.*

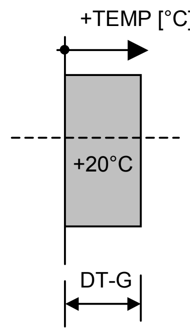
Input parameters:

Alpha Actual temperature expansion coefficient to be used. The value <u>has unconditionally to be entered here</u>. It is not taken from the material or system table if it is not entered (i.e. if Alpha=0.0). No loading is applied in this case! The value is approximately 1.0E-5 [1/°C] for all steel and concrete types, but some design codes require slightly different values to be used.

Note: *In the case of reinforced or pre-stressed concrete analyses it is always assumed, that steel and concrete have the same expansion coefficient and no internal primary stresses occur due to constant or linearly distributed temperature changes. The coefficient value assigned* *to the reinforcement or pre-stressing steel material is therefore never used, except for external sections of pre-stressing tendons treated like structural elements.*

DT-G Temperature change with respect to the stress-less or a previous state, which is constant over the cross section and produces only axial strains

DT-G=+20°C and DT-Y=0°C



The temperature gradients (TG_y, TG_z), and consequently the related strain gradients (κ_y) and (κ_z), are specified by value pairs of temperature difference and related length or width (DT-Y, H-Y and DT-Z, H-Z respectively).

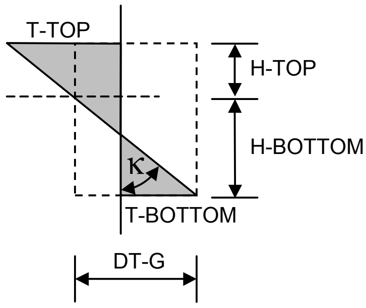
DT-Y, H-Y Temperature difference DT-Y and related height H-Y, describing the temperature gradient TG_y=DT-Y/H-Y in the local y-direction, producing only bending strains in the X-Y plane

DT-Z, H-Z Temperature difference DT-Z and related height H-Z, describing the temperature gradient TG_z=DT-Z/H-Z in the local z-direction, producing only bending strains in the X-Z plane

Note: *Temperature gradients are specified as the change in temperature per unit length. The temperature gradients are positive if the temperature increases (linearly) in the positive direction of the element local axis. The gradient temperatures are assumed to be zero at the neutral axes, hence no axial strain is induced.*

Example 1:

T_{TOP}=-35°C
T_{BOTTOM}=+20°C
H-Y=1.0
H-TOP=0.4
H-BOTTOM=0.6



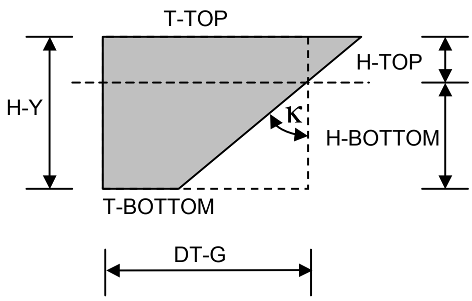
DT-Y=T_{TOP}-T_{BOTTOM}=-35-(+20)=-55°C

$$DT - G = T_{BOTTOM} + \frac{(T_{TOP} - T_{BOTTOM})}{H - Y} \cdot H_{BOTTOM}$$

=> DT-G=20 + (-35-20)*0.6/1.0 = -13°C

Example 2:

T_{TOP}=+30°C
T_{BOTTOM}=+10°C
H-Y=1.0
H-TOP=0.3
H-BOTTOM=0.7



DT-Y=T_{TOP}-T_{BOTTOM}=+30-(+10)=+20°C

$$DT - G = T_{BOTTOM} + \frac{(T_{TOP} - T_{BOTTOM})}{H - Y} \cdot H_{BOTTOM}$$

=> DT-G = +10+(+30-10)*0.7/1.0 = 24°C

6.3.7.2 Non-linear Temperature Distribution

Some design codes require a non-linear variation of the temperature over the cross-section to be investigated. This is for instance the case in the

- AASHTO Code
- Australian Standard
- British Standard BS 5400
- Korean Standard

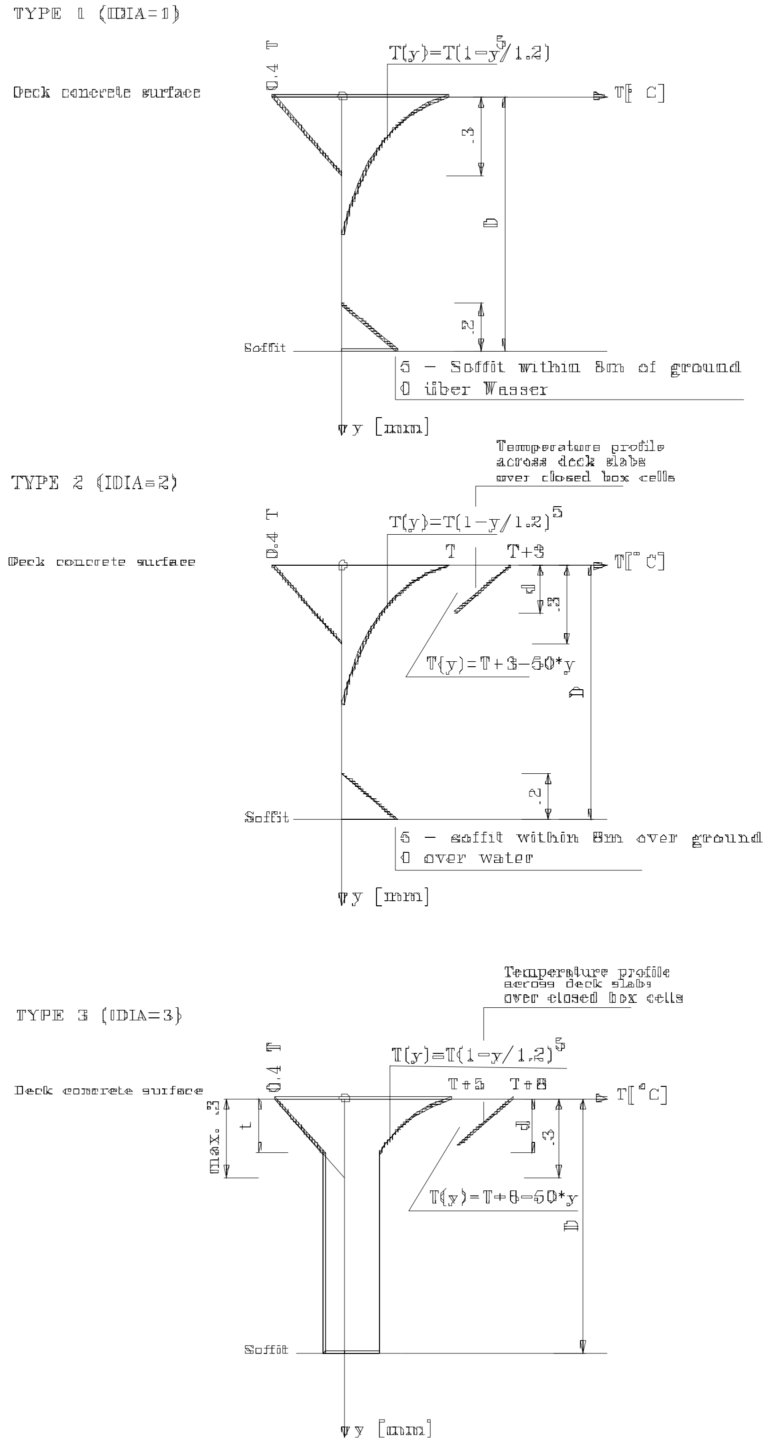
All these Standards only require to consider a non-linear variation in the vertical direction. A constant distribution may be assumed for the horizontal direction and may be combined with the non-linear case. This fact has been considered in the program *RM2000* and therefore only a distribution in the local y-direction may be defined.

Some codes prescribe the temperature state to be investigated as a function of the distance of the point from the upper surface and the bottom surface of the cross-section, other formulations refer to the related distances (with respect to the total cross-section height). Also mixed formulations can occur.

Two cases have usually to be investigated: a temperature increase and a temperature decrease. The required variations for these 2 cases generally differ and 2 separate Load Cases have therefore to be investigated.

A summary of the different demands of the above mentioned Design Codes is given below:

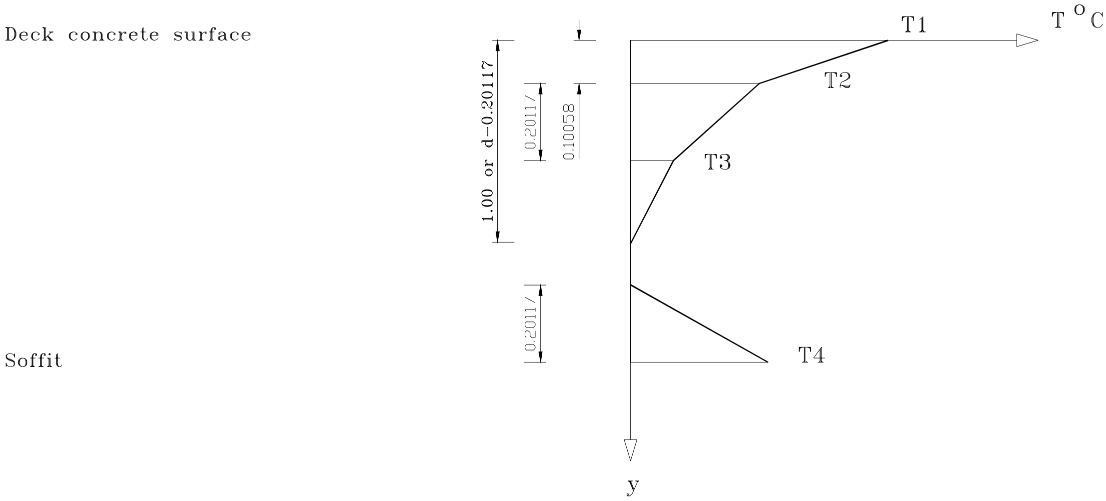
Australian Standard



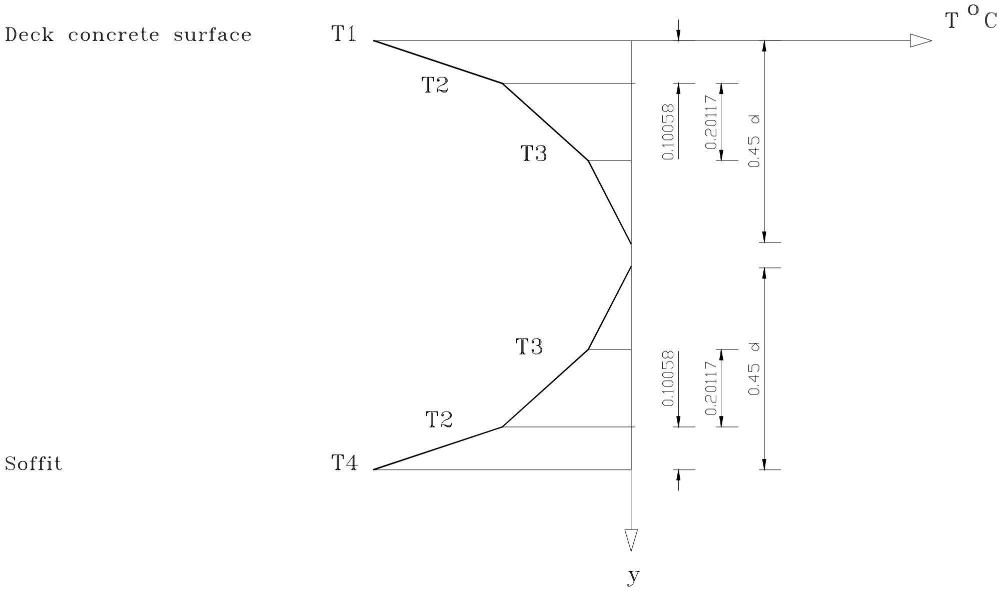
Temperature distribution through the cross sections in accordance with the Australian Bridge Standard.

AASHTO-Code

TYPE 1 (IDIA = 4, 14)

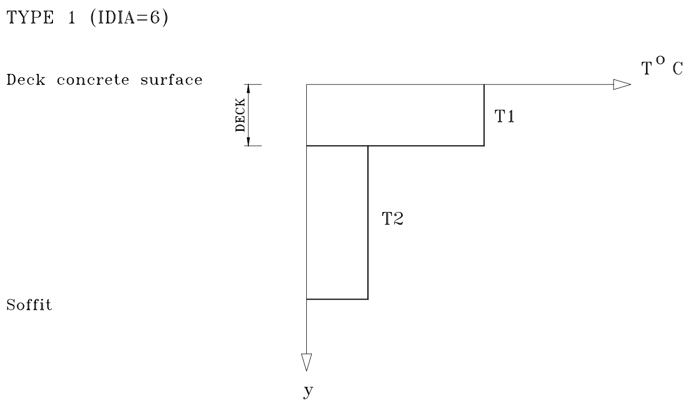


TYPE2 (IDIA = 5, 15)



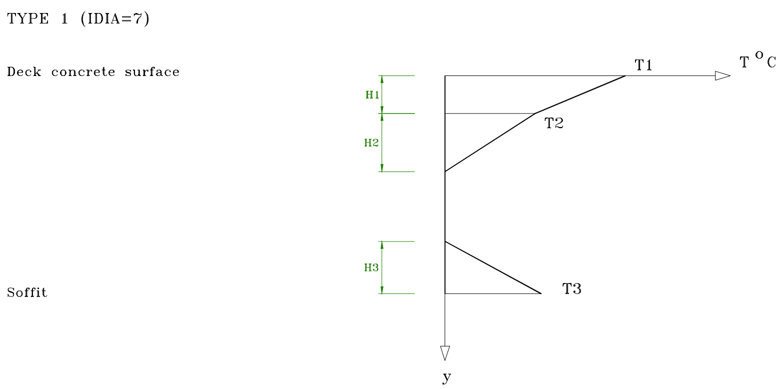
Temperature distribution through the cross-sections in accordance with the AASHTO-Code

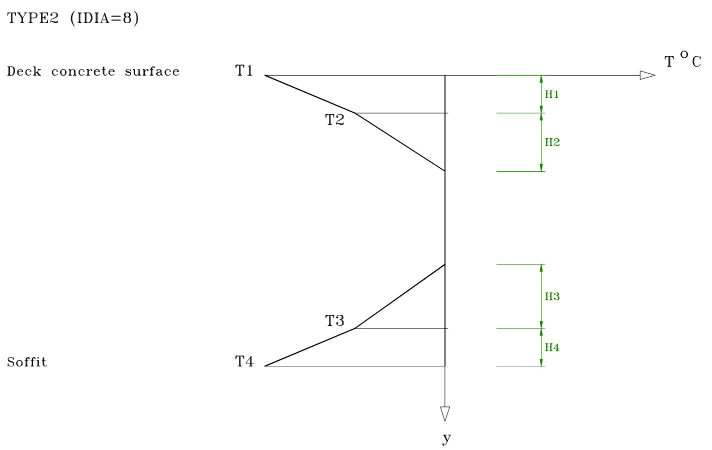
Korean Standard



Temperature distribution through the cros-sections in accordance with the Korean Standard

BS 5400-Code





Temperature distribution through the cross sections in accordance with the BS 5400-Code

To investigate a non-linear temperature distribution case it is necessary to

1. Define a group of cross section points with their temperature for all cross-sections.
2. Create an empty Load Set in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET.
3. Assign this Load Set to a Load Case
4. Select the Calculation Action "TempVar" for this Load Case
5. Calculate this Load Case

Defining the temperature distribution diagram (Cross Section Points with their temperatures):

The input of the temperature distribution diagrams is performed in the function ⇧PROPERTIES ⇨ADDGRP and ⇧PROPERTIES ⇨CS ⇩ADDPNT by specifying reference points in the cross-section along the y_L axis in those positions, where the temperature diagram has folds. The relevant temperature value and a common group name is assigned to all points of a specific diagram (Details see chap. 3.3.7).

The required reference points in the cross-sections may either be defined directly in *RM2000* in the function ⇧PROPERTIES ⇨CS ⇩ADDPNT as described in chap. 3.3., but also graphically in the geometric pre-processor *GP2000.* Especially in the case of main girders with variable cross section it is recommended to use the functions provided in *GP2000*, which are much more efficient than the direct definition in *RM2000*.

Definition of the Load Sets and assignment to the Load Case:

The Load Set characterizing the temperature loading state with non-linearly varying temperature over the cross-section has to be created as an empty Load Set in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET, and must be assigned to the required Load Case. The actual Load Set data are created, when the Action TempVar is performed. The Load Case Calculation Action for this Load Case can therefore only be performed after the appropriate TempVar Action.

Action „TempVar":

The function "TempVar" has been provided in *RM2000* to take into consideration temperature distributions like those described above. TempVar is a Calculation Action to be placed in the Action Table before the calculation of the Load Case. The action is generating a global temperature change value DT-G and a temperature gradient TG_y in the local y-direction by integrating a given distribution in y-direction over the cross-section area. These values are entered in the loading table of the Load Set to be used in the load case calculation for determining the system response.

Calculating the Load Case:

The calculation action for the temperature load case is placed in the Action Table after the action “TempVar”. The analysis is then performed when ⇧RECALC is selected (see also descriptions in chap. 6.2, 6.4 und 7.3).

6.3.7.3 Secondary component – bending part TB

This and the ensuing 3 Load Types are generated automatically by the program in the case of a non-linear temperature distribution. They describe the primary and secondary part of the temperature diagram in terms of element end forces. It is also possible to enter them directly, but this is not recommended because the correct usage requires a deep insight in the theoretical contexts.

The normal force and bending part described with TB is normally the sole part causing a system reaction in the beam model.

6.3.7.4 Secondary component – shear part TS

A secondary shear part does normally not arise due to in longitudinal direction constant strain gradients. A shear force arises due to equilibrium conditions if the end moments related to the temperature gradient are different on both ends of the element.

6.3.7.5 Primary component – bending part TB0

The primary part is also defined in terms of internal forces, although it characterises an internal equilibrium state. The fictitious internal force characterising this state is defined to be the force producing the correct stress in 2 points of the cross-section (the upper and lower edge).

6.3.7.6 Primary component – shear part TS0

Normally not existent.

6.3.7.7 Stress-free element length LX0

LX0 Stress-free „Fitting“-length
A stress free element length LX0 differing from the system length is assigned to the element.

The initial strain required to produce the elongation characterised by difference between the specified length and the system length is calculated. This strain is applied in the same manner than a temperature change DT-G. The stress-free length LX0 is used as well for determining the initial strain (ε_0 = (LX0-LX)/LX0) and the related end force FX0 = E*A* ε_0 , as for calculating the linear normal force stiffness (E*A / LX0).

Note: _See also chap. 4.5.5 „Use of Load Types FX0, LX0 for Cable Stayed Bridges“_

6.3.7.8 Initial Normal Force (Stress/Strain)

FX0 Initial normal force in the element
A stress free element length differing from the system length is assigned to the element by specifying the normal force required for producing this length difference. The effect of this Load Type is identical to LX0. The stress-free length LX0 is first internally computed by using the normal force FX0 (LX0 = LX + (FX0*LX) / (E*A –FX0)), then the program proceeds in the same manner than for the Load Type LX0.

Both above Load Types correspond physically to the installation process of a pre-stressed cable, stressed in a pre-stressing bed (not against the system). (Stressing against the system is simulated by the Load Type FCAB, see chap. 6.3.6.2). This pre-stressed element is installed in the actual system. The actual distance between the connection points characterises the length in the fully (with FX0) pre-stressed state. LX0 is the length arising in the case that the connection to the system is dropped. The calculation process simulates removing the pre-stressing bed. The pre-stressing forces are then acting on the system at both connection points. The system will give way and the resulting force in the cable will be (more or less, depending on the system stiffness) smaller than the specified fixed end value FX0.

Note: _See also chap. 4.5.5 „Use of Load Types FX0, LX0 for Cable Stayed Bridges“_

6.3.8 Actions on the Element Ends

6.3.8.1 Element End Displacement

The Load Type "Element end displacement" does not prescribe a global displacement value to the point, where it is assigned, but it induces a **displacement difference** between the specified element end and the nodal point to which it is connected, i.e. a gap or overlapping distance between the element end and the appropriate node is prescribed. The global deformation behaviour is calculated as a reaction of the structural system to this prescription.

This load type is for instance typically used for simulating support settlements. In this case, an element end displacement in the vertical direction will be applied on the support element (typically a spring element with one node fixed). The end of the support element will be moved by the specified amount in relation to the node. When the appropriate node is fixed, the specified movement will act like an absolute displacement of the element end point.

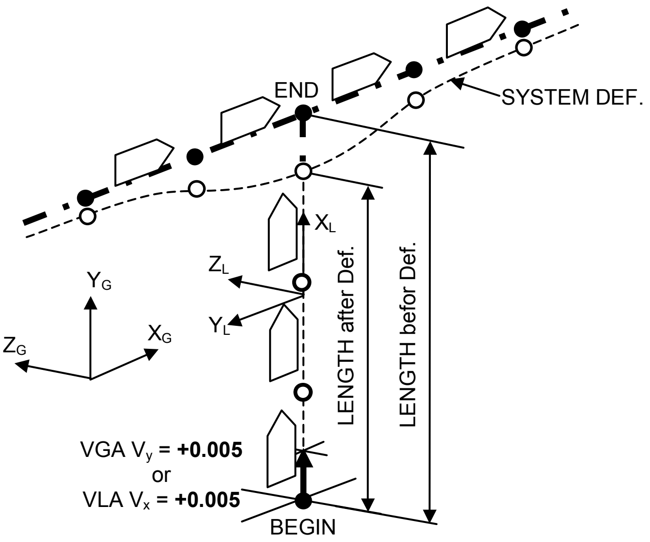
If this Load Type is applied to an element of the superstructure, the resulting deformations will represent an influence line for the appropriate internal force at the regarded point.

The Element end displacements may be specified at the start or at the end of an element, and they may be defined either in the global or in the local coordinate system.

Mind the sign conventions:
The element end displacements are defined as vectors **from the element end to the appropriate node** in the regarded coordinate system, i.e. the node is moved away from the element end by the specified amount. This convention applies also to rotations, where the node is rotated right hand turning in relation to the original position at the element end.

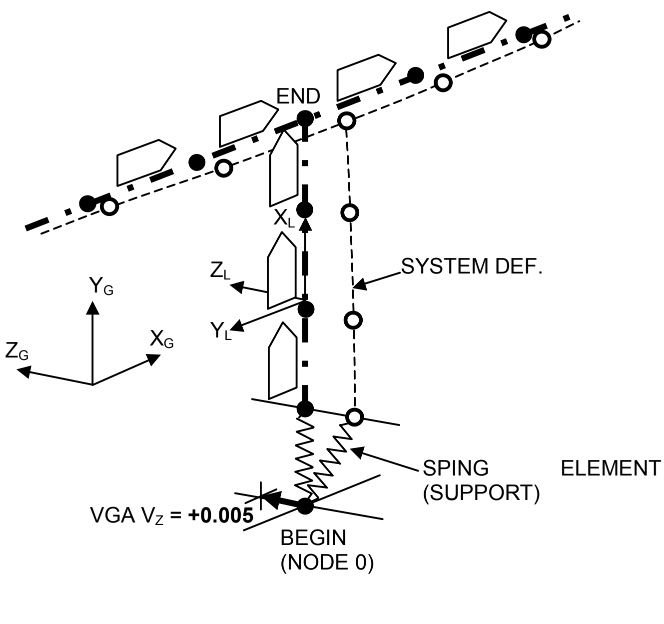
Note: *The global deformations and the internal forces which result from these prescribed deformations are dependent upon the various constraint conditions (typically from the supports). Whenever the DOF's of the node, to whom the element end displacement is applied, is restrained, the element end will move in the opposite direction than specified for the node.*

Example: This example typically describes a 5mm downward support settlement (displacement) of a bridge pier. This settlement is simulated by an element end displacement in the global Y direction applied at the element begin of the support element (e.g. spring element 501). By applying a global element end displacement (VGA) V_y or a local element end displacement (VLA) V_x of +0.005m the program will try to move the start node upwards by that amount. But as the start node is restrained the element begin will be moved downwards by 5mm instead.



VGA End-displacement (global – at the start node) – Prescribed displacements and rotations defined in the global coordinate system applied between the element start and the start node.

VLA End-displacement (local – at the start node) – Prescribed displacements and rotations defined in the local coordinate system applied between the element start and the start node.



VGE End-displacement (global – at the end node) – Prescribed displacements and rotations defined in the global coordinate system applied between the element end and the end node.
VLE End-displacement (local – at the end node) – Prescribed displacements and rotations defined in the local coordinate system applied between the element end and the end node.

6.3.8.2 Element End Displacement without statical effect

DSPLA The input for applying a displacement to the begin of the element is prepared with this load type.
DSPLE The input for applying a displacement to the end of the element is prepared with this load type.

These load sets cause no internal forces in the structure (no statical effects).
The load sets are used for incremental launching method and later for nonlinear calculation.

6.3.8.3 Load Type DEMO – Support Removal

DEMO Support removal – simulation of removing a previously active support element. This will typically but not necessarily be a spring element.

The basic requirements for a correct analysis of this loading are:
- The sum of all the internal forces in the structure resultant from all applicable load cases accumulated from all the previous construction stages must be stored in a special load case. (e.g.: LC 1000)
- The accumulated internal forces of the specified element(s) from this specified load case are now automatically applied to the redefined structure and redistributed on the structure in accordance with the normal rules of statics. This case must be considered as a normal load case and must be combined with all other previous load cases to get the total result.

6.3.8.4 Cable end displacement correction

DISCOR The input for applying a displacement to the ends of a cable is prepared with this load type.

Different from the load types VGA, VLA, ... “Element End Displacement“ with this option a displacement of cable nodes (e.g. for consideration of the fabrication shape) can be defined directly.

6.3.9 Wind Load

This Load Type calculates the effective forces acting on the structure due to wind impact. These forces are determined by using the wind direction, the wind speed and aerodynamic shape coefficients dependent on the cross-section shape and the “angle of flow”. These coefficients are usually determined in wind tunnel tests and given in form of diagrams describing the angle-of-flow dependency of the drag-, lift- and pitch coefficients for a special cross-section shape. The static part of the acting forces is herein calculated as well as the power spectrum characterizing the dynamic part.

Note:: *This Load Type is usually only used for highly sophisticated wind dynamics analyses, because usually no shape coefficients are available for standard cases due to missing wind tunnel tests. The wind loads are in these cases usually modelled by standard distributed loads as described in chap. 6.3.2 „Uniformly Distributed Loads“.*

Usage and correct application of this Load Type “Wind Load” is described in detail in chap. 9.8 "Wind Dynamics".

The following special Types are available for describing the wind loading:

6.3.9.1 Mean wind load (macro) WINDM

6.3.9.2 Mean drag for longitudinal wind component DRAGML

6.3.9.3 Mean drag DRAGM

6.3.9.4 Mean lift LIFTM

6.3.9.5 Mean pitch PITCHM

6.3.10 Normal Forces (Stiffness Change)

In the case of a non-linear solution (p-delta effects, large displacements) the superposition of load cases is theoretically not allowed. The normal forces used for the calculation of the stiffness matrices must therefore represent total states. It is therefore necessary to define an initial state of the normal force distribution in the system, if differential loading cases are investigated. This might be done for single Load Cases by specifying these initial normal forces as a Load Set assigned to the Load Case to be calculated.

For these Load Sets the following two load types are available:

6.3.10.1 Assign normal force direct (just stiffness)

PDFOR Direct assignment of a user defined normal force intensity to an element series.

With this function you can define which elements should be assigned with the defined normal force (Fx internal normal force in kN). First you must choose the elements which are needed for a P-Delta calculation (From, To, Step) and then you can define the normal force for these elements directly (Fx).

6.3.10.2 Assign normal force from LC (just stiffness)

PDFLC The normal force is read from a Load Case result and assigned to an element serie.

With this function you can define witch elements should be assigned with a normal force from an other load case (LC - loading case). First you must choose the elements which are needed for a P-Delta calculation (From, To, Step) and then you can define the load case for these elements directly (LC).

6.3.11 Special

6.3.11.1 Pier Dimensioning

DIMPR Additional information for pier dimensioning by Austrian standard ÖN B4700 are prepared with this load type.

The following values have to be defined by the user:

L0y [m] Buckling length of single beam in local xy-plane.
(relates to Mz)
L0z [m] Buckling length of single beam in local xz-plane.
(relates to My)
ni-y Imperfection factor for calculation of the additional load eccentricity $e_{a,y}$ out of the Buckling length L0y. (correspond to Mz)
ni-z Imperfection factor for calculation of the additional load eccentricity $e_{a,z}$ out of the Buckling length L0z. (correspond to Mz)

6.3.12 Load Type Creep & Shrinkage

There is no separate Load Type for the specification of creep and shrinkage parameters. The behaviour of the structure due to creep and shrinkage is defined by

- The creep and shrinkage potential depending on material parameters, cross-section parameters and environmental conditions (humidity, temperature)
- The stress state subjected to creep
- The time history (Duration and start points of creep inducing stresses)

The parameters describing the creep and shrinkage potential of the structural materials are specified in the function ⇧PROPERTIES ⇨MATERIAL, ⇧PROPERTIES ⇨VARIABLES, ⇧PROPERTIES ⇨CS and described in chap. 3, Structural Properties.

The parameters describing the time history are specified in the functions ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTIVATION and ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION. They are described in the chap. 7.4, Creep & Shrinkage.

The internal force redistributions due to creep an shrinkage will nevertheless be stored and referenced as Load Case results. A Load Case Number will be assigned in the LOADS AND CONSTR.SCHEDULE for every time section where creep and shrinkage is considered. The Load Case needs not to be created before assigning it to the Creep Action, but it has to be created previously in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE as an empty Load Case, if it is used in the Load Management function.

Creep and shrinkage cannot be mixed up with other load types in one Load Case. A detailed description of the treatment of creep and shrinkage can be found in chap. 7.4, Creep & Shrinkage.

Note: *Creep & Shrinkage may not be treated together with other Load Types in the same Load Case, but only separately for a certain period.*

6.4 Load Case

6.4.1 General

The Load Case is made up from a single Load Set or a group of Load Sets and is the primary unit for applying load to the structural model.

The different Load Sets must have been defined first. These Load Sets can then be combined and factored (if required) in different ways to form various Load Cases that can be applied as a loading to the structural model. The factors might be constant for static purpose or variable (time dependent) for dynamic analysis.

Load Cases are primarily identified by a **Load Case Number** and defined by the appropriate Load Sets. A descriptive text may be assigned as an additional information in order to allow an easier allocation in the checking or result interpretation process.

Several Load Case Attributes may be assigned to each Load Case, describing the required treatment of the Load Case in the analysis process. These attributes are:

- The Permanence Code
- A Load Case Info Table, describing the rules for accumulation and superposition of the load case (function ⇩LMANAGE, see chap.6.4.3 and chap. 6.6.

6.4.2 Permanence Code

This Load Case attribute is important for the proper treatment of the time dependent effects (creep behaviour and stress accumulation over the construction stages). It must only be specified if creep and shrinkage behaviour is taken into account in the analysis, indicating whether the Load Case is “permanent”, and thus creep inducing.

This code is additionally used in the case of considering P-delta effects in the construction schedule by accumulating all the previous permanent dead load cases (calculation option „Accumulate permanent loads, see chap. 7.5.4).

The Permanence Code (GUI notation Load Type) may have the values:

- Load (**L**)
- Load + Unload (**LU**)

Load Cases marked with “Load” are permanent and therefore considered in the creep analysis. Load Cases marked with “Load + Unload” are live loads and not considered for creep, or in the accumulation process for P-delta calculations.

- **Load** Permanent loads like self-weight, superimposed dead load and so on. This load is active during the whole period from being applied over the whole remaining construction procedure and operation time to infinity. It is assumed to be creep inducing throughout this time.

- **Load+Unload** Live loads, acting on the structure only for short periods. These loads do therefore not induce creep and are not considered in the creep and shrinkage analysis.

6.4.3 Load Case Info Table

Load Case Info Tables are created in the function ⇩LMANAGE and contain the information on how the different types of load cases (self weight cases, creep&shrinkage cases etc.) should be superimposed throughout the construction schedule. The Load Case Info Tables must have been created and defined before it can be assigned to a Load Case. The definition and application of the Load Case Info Tables is fully described in chap. 6.6, Load Management.

A Load Info Table may be assigned to the Load Case by specifying the name of the table created in the function ⇩LMANAGE as a Load Case attribute. The Info Table is used to accumulate the Load Case into various other load case numbers and into superposition files. The accumulation is automatically made immediately when the Load Case has been calculated in the construction schedule.

6.5 Combinations

6.5.1 General

Load Case results can be combined in any way provided that the deformation behaviour of the structure is linear.
This is the normal way of handling loads applied in different construction stages on different partial systems – to superimpose Load Case results from the different stages and to use these values for design code checks.

Load cases may be combined into new loading cases or into combination files.
Combination files may be combined into new combination files.
The combination files may be **"Envelopes"**, containing maximum and minimum values built from a set of Load Cases (or previously created Envelopes/combination files) using explicitly defined superposition rules.

The Load Cases and/or combination files may be multiplied by "Superposition Factors" before being combined into the new combination file. The factors are typically used for applying Load Factors to get "Ultimate Limit States" (U.L.S.).

6.5.2 Creating Superposition Load Cases

A Superposition Load Case created by combining other previously calculated Load Cases is initialised in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION, selecting

- ⊙ LC/Envelope Actions
- ➢ Action LcInit

This action prepares the required storage area for the results of the new Load Case.

The results of calculated Load Cases may then be added to the "initialised" Load Case by using ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION, and selecting

- ⊙ LC/Envelope Actions
- ➢ Action LcAdd

A multiplication factor can be specified for each added Load Case. This might e.g. be a safety coefficient in order to get a special design value of the internal force state.

A further possibility to create Superposition Load Cases is to use the load management function (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LMANAGE). The combination rules are there allocated in a table, and the combination is automatically performed for the affected Load Cases when they are calculated. This function is described in detail in chap. 6.6.

6.5.3 Envelopes

Envelopes generally consist of a set of Result Vectors (internal forces, deformations), each vector containing the maximum or the minimum of a certain component (**Characteristic Component**) and the related other result components. An envelope is identified by a user defined name, which is used as a filename for storing the related result values.

The characteristic components (maxima/minima) characterizing a certain Envelope Vector may be internal forces and/or deformations. The definition of the envelope composition - which maxima and minima are to be assembled - is set by the user in the ⇧RECALC window. This definition is valid for all the Envelopes in the project and can not be assigned selectively to some of the Envelopes. It is not possible to select only maxima or minima to be evaluated (e.g. only maximum tension force), but maxima **and** minima will be created for each selected result component.

Envelopes are created by first initialising the appropriate Superposition File (**Action SupInit**) and then superimposing Load Case results or other Envelopes, using defined superposition rules, into this file. The superposition rules are specified by the “Superposition Code” and the “Superposition Factor” assigned to each Loading Case or Envelope which should be superimposed into another envelope. The following Superposition Codes are available:

- SupAdd — Unconditional superposition (RM7 – Combination Code 1)
- SupAnd — Superposition if structurally unfavourable (RM7 – Combination Code 2)
- SupAndX — Superposition with the structurally unfavourable sign (+/-) (RM7 – Combination Code 3)
- SupOr — Exchange if structurally unfavourable (RM7 – Combination Code 12)
- SupOrX — Exchange if structurally unfavourable with sign change (+/-) (RM7 – Combination Code 13)

SupAdd:
Unconditionally **Add** the appropriately factored new results, into the combination file.

This code is applied for permanent loads such as self weight, pre-stressing, earth pressure etc.

Two superposition factors (F1 & F2) are associated with the SupAdd code and are input in Inp2. If no values are defined then both values are assumed to be 1.0

F1: The **"favourable"** factor to be applied to the results before the combination. This factor is adopted if the results after combination become structurally **less** critical**.**

F2: The **"unfavourable"** factor to be applied to the results before the combination. This factor is adopted if the results after combination become structurally **more** critical**.**

SupAnd:
Conditionally **Add** the appropriately factored results into the combination file.

Add the new results, multiplied by the defined **factor F1**, only **if**, after combination, the total becomes more unfavourable (structurally more critical) with respect to the Characteristic Component. I.e. the results will be added to the Maximum Vectors only if the value of the Characteristic Component is positive, and to the Minimum Vectors only if it is negative. The results will not at all be added if the value of the Characteristic Component is exactly zero.

Note: *In general, result values will seldom be exactly zero, therefore the results will in most cases be superimposed either in the maximum or in the minimum vector. In cases, where the characteristic values are theoretically zero but in fact slightly different, the vectors will be randomly superimposed (or not if exactly zero), and the related other components might show a strange distribution in the structural system.*

The superposition factor F1 entered in the Inp2 input field is used in the SupAnd function. F2 is not used in this case. The value 1.0 is used if no factor is entered.

This function is generally used for live loads, such as traffic loads or snow etc..

SupAndX:
Conditionally **Add** the appropriately factored results (with a change of sign if necessary) into the combination file.

Add the new results, multiplied by the defined factor, if necessary with applying an appropriate sign change to make the total more unfavourable (structurally more critical) with respect to the Characteristic Component. I.e. the results will be added to the Maximum Vectors with the original sign, if the sign of the Characteristic Component is positive, and with the opposite sign, if it is negative. Vice versa for the Minimum Vectors.

The only instance when the results will not be added is when the value of the Characteristic Component is exactly zero (see also the above note).

The superposition factor F1 entered in the Inp2 input field is used in the SupAndX function. F2 is not used in this case. The value 1.0 is used if no factor is entered.

This code is generally used for loads which can change the direction, e.g. temperature loads or wind loads.

Attention:	**Using the function SupAndX for superimposing already existent other envelopes is dangerous, because maximum values might be added with the opposite sign to the minimum values in the actual superposition file and vice versa.**

SupOr:
Conditionally **replace** the old values in the combination file with the appropriately factored new results.

Example:

in the memory:	maxMZ = 200	minMZ = -300
values to superimpose:	maxMZ = 225	minMZ = -125
new values of the envelope:	maxMZ = 225	minMZ = -300

Replace the old values **if** the new results, multiplied by the defined factor, are more unfavourable (structurally more critical) with respect to the Characteristic Component.

The superposition factor F1 entered in the Inp2 input field is used in the SupOr function. F2 is not used in this case. The value 1.0 is used if no factor is entered.

This code is typically used for finding the worst state from two or more states which cannot occur at the same time (e.g. heavy vehicle load trains (such as a crane gantry) applied at different positions on the structure).

Note: *Care should be taken when applying this code to Envelopes containing unconditionally superimposed Load Cases. Zero states containing none of the superimposed values may occur, if these Envelopes are superimposed with SupOr into a newly initialised Envelope. The code SupAdd should be used for superimposing the first Envelope, if one of the states excluding each other should unconditionally be considered.*

SupOrX:
Conditionally **replace** the old values in the combination file with the appropriately factored new results (with a change of sign if necessary).

<u>Replace</u> the old values **<u>if</u>** the new results , multiplied by the defined factor, – or the results with opposite sign multiplied by the defined factor - are more unfavourable (structurally more critical) with respect to the Characteristic Component.

The superposition factor F1 entered in the Inp2 input field is used in the SupOrX function. F2 is not used in this case. The value 1.0 is used if no factor is entered.

This code has a limited practical application range. It might be used to detect worst influence of different loading cases which exclude each other and may each have varying sign (e.g. wind or earthquake in longitudinal and lateral direction, some design codes allow to assume that design earthquake loads and design wind loads will not occur at the same time).

6.5.4 Creating Envelopes

Envelopes are created in the construction schedule - in a similar way than Superposition Load Cases as described in chap. 6.5.2. - by applying the appropriate Envelope Actions in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION (Initialization with SupInit, Adding with SupAdd, SupAnd etc. (see above)).

These Actions allow to initialise new envelopes and adding or combining Load Cases or other envelopes in accordance with the desired superposition (see chap. 6.5.3 “Envelopes” or chap. 7.3.1, Envelope-Actions).

Besides this explicit direct definition of all superposition actions in the Construction Schedule, there is a further possibility to create Envelopes in a more convenient way: The function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB may be used to create a Combination Table, where the superposition rules for the different Load Cases are summarized. It is then sufficient to select the Envelope Action SupComb in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION to create the required envelope (see chap. 6.5.5):

⊙ Envelope Actions
➢ Aktion SupComb

6.5.5 Creating a Combination Table

A table containing the combination rules for up to **24 Envelopes (called Combinations**) can be established with the function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB. The Combinations are initially identified by a Combination Number (from 1 to 24) and later assigned to Envelope or Combination names via **SupComb** in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION.

This function is advantageously used for establishing the result database to be used for design code checks in the Construction Stage as well as in the Final Stage for both the Serviceability and the Ultimate Limit States.

- An arbitrary number of Load Cases or Envelopes can be assigned to each available combination (1 to 24).
- The superposition process defined in the Combination Table starts at the moment that the Envelope Action **"SupComb"** is called for the required combination. **"SupComb"** may be called as often as desired, also in different Construction stages.
- The superposition rule for the Load Cases or Envelopes is defined by the Superposition Code and one or two factors.

Note: *SupAdd has two factors (**F1 for the favourable** case and F2 for the unfavourable case), SupAnd & SupOr have only one factor (**F1 for the unfavourable** case).*

Construction Stage Calculation
Since Load Cases and Envelopes are continuously updated during the construction sequence calculation, the following procedure is recommended:
The action **"SupComb"** may be used in every Construction Stage using the same combination number but assigning the results to different Combination File names – viz:
Create different Combination Files with names related to the Stage Number using the same Combination number:

E.g.:	Comb1_1.sup	Combination 1 for Stage 1
	Comb1_2.sup	Combination 1 for Stage 2
	Comb1.sup	Combination 1 for the final Stage

The result files "Comb1_1.sup" etc may be used at any subsequent time in the calculation for design checks etc. for the appropriate stage.

The Combination Table is displayed and modified on selection of ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB. Selection of the insert function opens the input data screen for inserting all the required input data. The displayed Combination Table only shows 12 of the 24 allowed Combinations. Select the arrow at the top of the screen to view Combinations 13 to 24.

The table contains the Load Cases and Envelopes to be considered (1st row) together with the superposition code to be used (SupAdd, SupAnd, etc.) in the 2nd row. The fields beneath the Combination Numbers display the superposition factors to be applied for that particular combination to the Load Case or Envelope.

6.6 Load Info Tables (Function ⇩LMANAGE)

Load Case results can be automatically added into new Load Cases or combined into Superposition Files using any superposition command, during the calculation process. This automatic accumulation is controlled by the Load Management function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LMANAGE. Any Load Info Table may be assigned to a Load Case in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE.

All the accumulation and superposition processes would otherwise have to be separately defined in every Construction Stage.

Note: *Working with Load Info Tables is not absolutely necessary, but it essentially eases the creation of the required combinations and envelopes and reduces the required amount of input data.*

Each Load-Info-Table is a named database object with a user defined name. The name usually refers to the type of the Load Case being relevant for the treatment in the superposition process.
Example:

- G1 Self weight load acting on the net cross-section
- G2 Additional permanent load acting on the composite section
- G3 same as G2 (such as formwork traveller, scaffolding, wet-concrete etc.)
- PT Pre-stressing - post or pre tensioning Load Case
- C&S Creep and Shrinkage Load Case

The allocation of the Load Cases to the Load Information Tables is done in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE (see chap. 6.4.4). The table, which is relevant for treating the actual Load Case with respect to automatic superposition, is assigned to each Load Case (e.g. table G1 to all self weight Load Cases, table PT to all pre-stressing Load Cases, etc.).

Up to 3 Load Cases and up to 3 Envelopes, where the assigned Load Case should be accumulated, may be defined in such a Load Info Table.

The accumulation into the defined Load Cases is unconditional, i.e. the Load Cases are superimposed in the same way than dead loads and independently of whether they diminish or increase the internal forces. No factors can be applied in this automatic accumulation procedure.

The superposition into the defined Envelopes may be

- Unconditionally additive (SupAdd)
- Additive when unfavourable (SupAnd) *
- Additive when unfavourable with optional sign change (SupAndX) *
- Replacing if unfavourable (SupOr) *
- Replacing if unfavourable with optional sign change (SupOrX) *

The combinations are defined by the Combination Code (see detailed description in chap. 6.5.3) shown in brackets.

** N.B.* *"Unfavourable" means that the addition/combination of this loading case will make the loading case/combination file more structurally critical.*

All the Load Cases and Envelopes used in the automatic accumulation procedures must be initialised in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION in the first construction stage (⇨LC/EnvelopeActions ⇨LcInit/SupInit).

e.g.: initialise Load Case 1000 and the combination file envelope.sup

➢	LcInit	-	-	1000
➢	SupInit	-	-	envelope.sup

6.7 Recommended Load Case Numbering Scheme

6.7.1 Basic Definition

The intention of this proposal is to have a clear and unified system for the numbering of the individual Load Cases calculated in the LOADS AND CONSTR.SCHEDULE process as well as for the numbering for the total sum and the subtotals of the permanent Loading Cases up to a any stage.

It is strongly recommended that this numbering system is used for all projects.
The benefits from using a unified numbering system is both the ease of interpretation of the results as well as the the ease of communication and understanding with the TDV staff during hot-line/e-mail support for project work.

This recommended numbering system uses a 3 digit Loading Case Number and will work for up to 99 Construction Stages. The user is able to use an equivalent numbering system for 999 Construction Stages by increasing the system to a 4 digit numbering system.

6.7.2 Numbering of Individual Load Cases

Constitutive relation: Load Case Number = [F] [CN]
where

- [F] A single digit number signifying the Load Case Group - as specified in "Load Management"
- [CN] The Construction Stage Number

The recommended Load Case Group Numbers are:

1	G1 Loading Case
2	G2 Loading Case
3	G3 Loading Case
4	reserve
5	Pre-stressing Loading Case
6	Creep & Shrinkage Loading Case
7	reserve
8	reserve
9	reserve

The last two digits identify the Construction Stage Number.

e.g.: [F]=1 and [CN]=05 defines Loading Case No 105 = 100+5, where the 100 series defines G1 Loads and 05 indicates Construction Stage 5.

The individual Loading Case Numbers (groups) must be defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCase.

- G1 100+Stage (group G1 loads: 105 means G1 Loading Case in Construction Stage 5)
- G2 200+Stage (group G2 loads – additional loads, such as formwork traveller, scaffolding, wet concrete)
- G3 300+Stage (group G3 loads – similar to G2)
- PT 500+Stage (group PT loads – Pre-stressing - post or pre tensioning Loading Cases)
- CS 600+Stage (group CS loads – Creep and shrinkage Loading Cases)

6.7.3 Numbers of Construction Stage (sub)totals

The sums of the different loading groups should be stored in the following Loading Cases:

- G1 LC 100 (i.e.: 100 = 101+102+103+104+105+....)
- G2 LC 200 (i.e.: 200 = 201+202+203+204+205+....)
- G3 LC 300
- PT LC 500
- C&S LC 600

The sum of the sums of the loading groups should be stored in Loading Case 1000:
(G1+G2+G3+PT+CS) of Stage 1 +
(G1+G2+G3+PT+CS) of Stage 2 +
(G1+G2+G3+PT+CS) of Stage 3 +
+… = 1000

I.e. (100+200+300+500+600) = 1000

<u>Recommended numbering of the subtotals during the Action Schedule:</u>

1000+100+..... Total load after a Loading Case Action

- after G1 1100+Stage total of all previous Loading Cases plus the current G1 Load Case

- after G2 1200+Stage total of all previous Loading Cases plus the current G2 Load Case
- after G3 1300+Stage total of all previous Loading Cases plus the current G3 Load Case
- after PT 1500+Stage total of all previous Loading Cases plus the current PT Load Case
- after C&S 1600+Stage total of all previous Loading Cases plus the current C&S Load Case

e.g.: ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION
- LcInit 1000 - 1601

This input in `<Action>` creates the Loading Case 1601 and copies the contents of loading case 1000 directly into it.
If this `<Action>` is entered in the LOADS AND CONSTR. SCHEDULE immediately after the calculation of loading case 601 and provided that the numbering system and calculation sequence recommended above is followed then Loading Case 1601 will contain the sum of the stage construction results up to and including Loading Case 601 – i.e up to and including the Creep and Shrinkage Loading Case in Construction Stage 1.

Similarly Loading Case 1508 contains the sum of the stage construction results up to and including the Pre-stressing Loading Case 508 in Construction Stage 8.

6.7.4 Camber

$$C'St_n' = (\sum\delta'St_{all}' - \sum\delta'St_{n-1}')*-1$$

Where:

$C'St_n'$ = The pre-camber for the stage about to be built.
$\sum\delta'St_{all}'$ = The sum of all permanent load deflections for all the stages (The whole construction)
$\sum\delta'St_{n-1}'$ = The sum of all permanent load deflections up to the end of the previous construction stage.

The main girders* of bridge structures are usually precambered during construction to eliminate the deflection caused by the permanent loads so that the “design profile” of the structure is achieved at the end of construction or at a certain time after that (considering creep effects).
A special storage scheme and result manipulation of the loading cases is required for this pre-camber calculation.

This procedure is also valid for structures that are built in several construction stages – where every single construction stage must be built with a certain pre-camber to arrive at the final pre-camber:

6.7.4.1 Loading Types

Any load type and condition can physically be used in the pre-camber calculation, however, the normal loading cases to be included are all the permanent loads on the structure – such as:

- Dead Load (self weight) – G1 Loading
- Additional dead Load
  (such as premix (surfacing), balustrades, sidewalks etc), - G2 Loading
- Pre-stressing. – PT Loading
- Creep& Shrinkage (to time infinity) – C&S Loading

Some national design codes also call for a certain percentage of the live loading effects to be included in the pre-camber calculations.

* Sub-structure elements are also precambered in certain large bridges – such as the Pylons to large cable stayed bridges.

6.7.4.2 Load Management

RM2000's “Load Management” facility (described above) can be usefully used for finding the required pre-camber at any stage of the construction of the bridge.

All the loading case results which form part of the “Permanent Loading” are algebraically added into Loading Case 1000 as described above under “Load Management” (The results in loading case 1000 are thus updated each time that a new permanent loading case is applied to the structure.

The deflections stored in the last loading case 1000 – at the end of construction – will then represent the final deflected shape of the structure if NO pre-camber is applied to it.

This last loading case 1000 multiplied by minus 1 is the “Camber Line”.

The required pre-camber for any stage is the Camber Line reduced by the total Deflection up the end of the previous stage.
This is true whether it is the formwork pre-camber for the stage or the pre-shaping of a pre-cast segment (steel or concrete).

Therefore: LC 1000*(-1)=LC 2000

e.g. using *RM2000*: ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩Action

- LcInit1000 -1 2000 (This input under **`<Action>`** creates the camber line)

Therefore:

- Remaining pre-camber after G1 2100+CS
- Remaining pre-camber after G2 2200+CS
- Remaining pre-camber after G3 2300+CS
- Remaining pre-camber after PT 2500+CS
- Remaining pre-camber after C&S 2600+CS

Where 'CS' = Construction Stage Number.

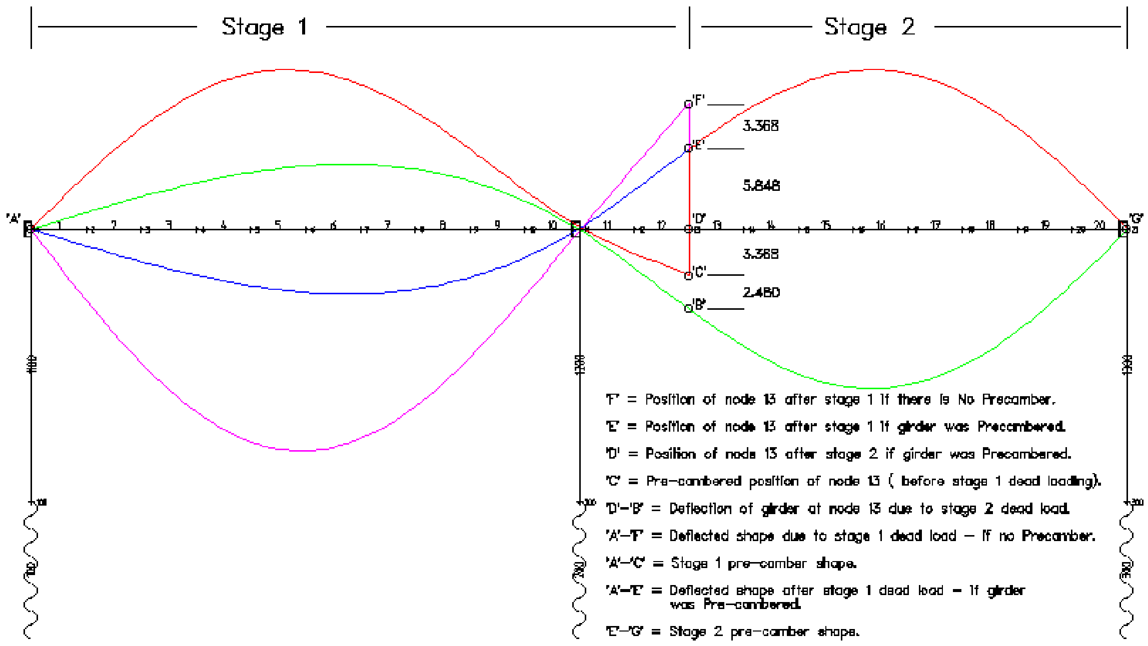
e.g.: the deflected shape stored in loading case 2508 (defined below) is the camber line after Pre-stressing (PT) in Construction Stage 8.

e.g. using *RM2000*: ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩Action

- LcInit 1508 1 2508 (1508 is copied into an initialised 2508)
- LcAdd 2000 1 2508 (LC2000 is added to LC2508)

First input in **`<Action>`** makes a copy of 1508 to 2508
Second input in **`<Action>`** adds the Camber line (LC 2000) to 2508

Example: Consider a 2 span beam built in 2 stages:

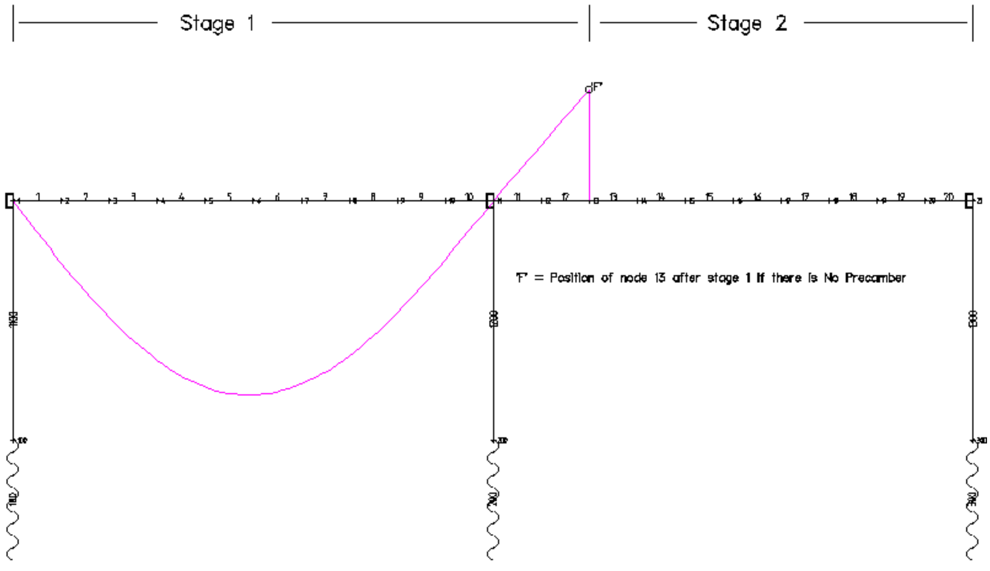


Pre-Camber and Deflected Shapes for 2 span beam built in 2 Stages

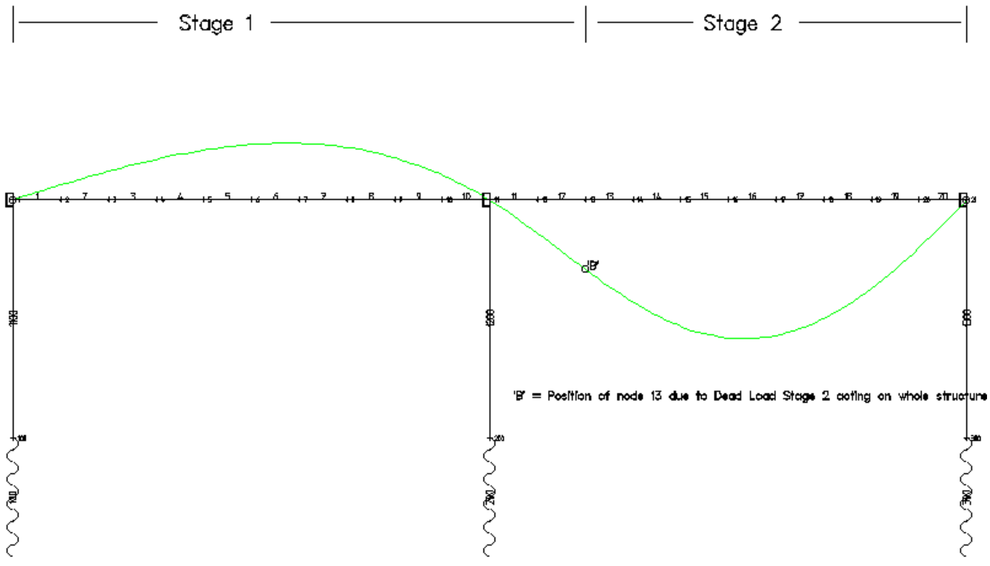
(Stage 1 & Stage 2 loading is Self Weight only - UDL)

By pre-cambering the girder, the position of node 13 at the end of stage 1 is 'E' (instead of 'F') – as can be seen E-F = D-C.
The deflection of Node 13 due to stage 2 loading is 'D-B' (=3.368 + 2.48) which is equal to but of opposite sign to the pre-camber position 'E'.
After applying stage 2 loading Node 13 will therefore be at position 'D'.

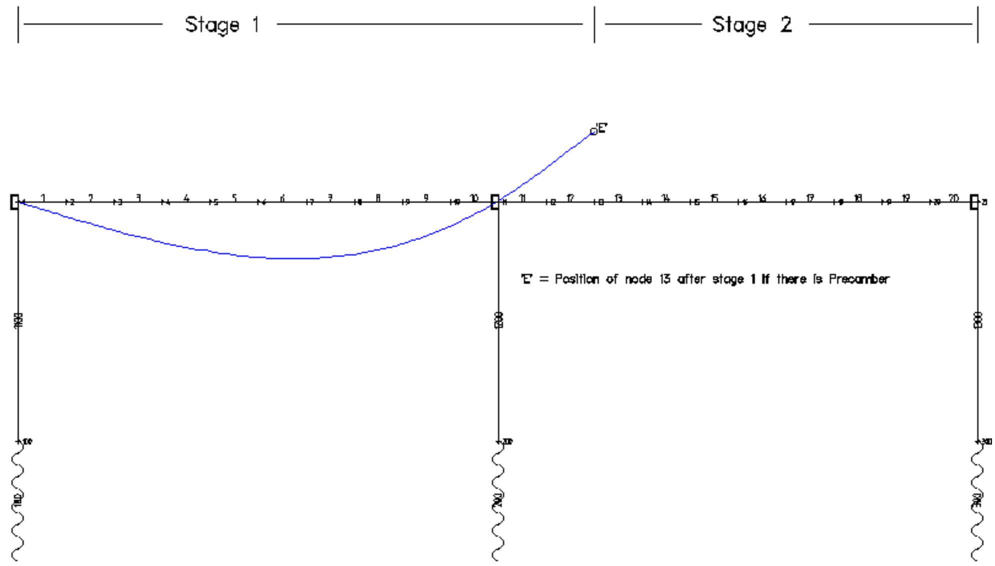
De-composed deflections and precamber for clarification:



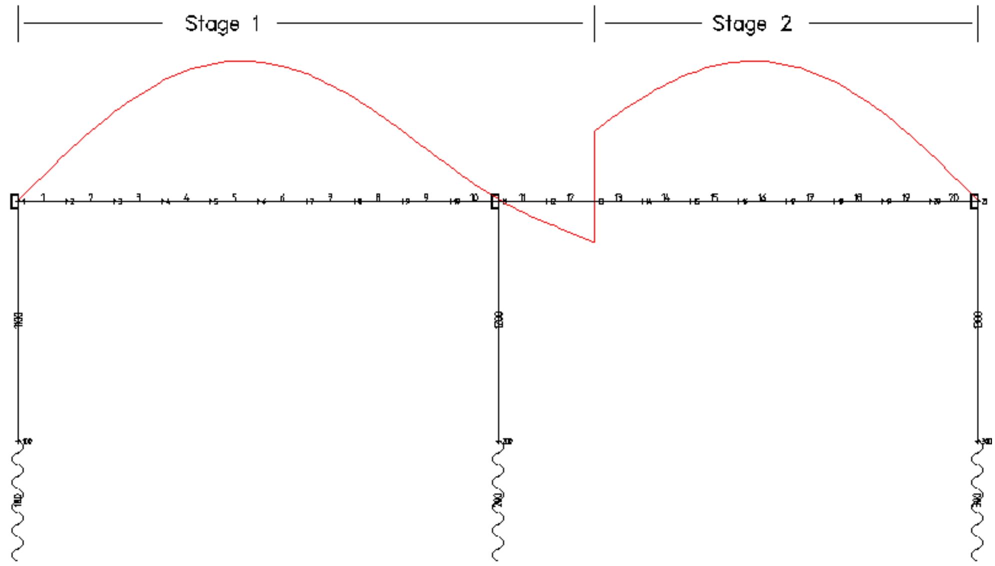
Deflected Shapes due to Dead Load Stage 1 only



Deflection due to Stage 2 Dead Load only



Deflected Shape after Dead Load Stage 1 and Pre–Camber



Camber Line for 2 span beam built in 2 Stages

6.8 Traffic Load Calculation

6.8.1 General

The calculation of traffic loads may be done in *RM2000* in the standard manner, defining a certain number of Load Cases representing the possible traffic load cases and superimposing them by the standard procedures described in chap. 6.5. This process will however often require a huge amount of Load Cases to be defined, analysed and superimposed to get sufficiently accurate answers for the maximum and minimum internal forces due to possible traffic loadings.

RM2000 therefore offers an alternative way to perform the **calculation of traffic loads by evaluation of influence lines**. This is in fact the standard procedure for treating traffic loads in *RM2000*, allowing a very accurate solution with acceptable computation and input preparation effort. It also allows to take into account arbitrarily sophisticated rules for traffic load application demands of the different design codes.

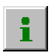
This chapter shows the requirements for the calculation of stresses due to traffic loads based on the influence line theory. This procedure requires the specification of

- The lanes identifying the positions of the vehicles relative to the deck elements
- The actual loads representing the considered vehicles (Load Trains)
- The calculation of the influence lines for all lanes
- The evaluation of the influence lines for all Load Trains

It is strongly recommended to prepare all the necessary basic information before starting the input. Such as:

- Number and eccentricities of lanes
- Loading scheme
- Superposition resp. combination of the individual results per lane

Each lane can be presented graphically by creating a plot file in ⇧RESULTS ⇨PLSYS.

The evaluation of influence lines may be a very great computational effort, especially if large systems (bridges with many spans) are analysed. Loads applied far away from the considered result point give however often a neglectable contribution. Therefore the program offers the possibility to define tolerance values for the influence lines. The evaluation will then not be done for the structural parts, where the influence line values do not exceed the predefined tolerance limit. These limit values are 1.E-7 per default and may be reset by the user by clicking the  button in the Load Train Table.

The same pad may be used to set multiplication factors for the influence line evaluation results. This functionality is required in AASHTO, where separate rules are valid for moments, normal forces and shear forces respectively.

6.8.2 Calculation and Evaluation of Influence Lines

The influence line definition and calculation procedure used in RM2000 is slightly different from the common approach found in literature. The influence lines for the different points and result components (displacements, internal forces) are not calculated directly by applying an appropriate local strain at the considered point, but by calculating a series of unit Load Cases and collecting the the results for to the considered point gained in the different Load Cases.

Point forces of 1 kN are applied in these unit Load Cases in a series of points along one or more lines over the whole superstructure. These lines are in the program called **Traffic Lanes** or simply **Lanes**. The definition of these traffic lanes is described in chap. 6.9. The influence lines created by collecting the results of the Load Cases of such a lane are often called **"related influence lines"**, because they are related to the lanes. But for simplicity they are mostly only called "influence lines" within this document.

Note: *The point load applied for calculating the influence lines is independent from the selected unit system and always 1 KN. The applied load is shown in the GUI and in the influence line result listings in terms of the selected units (e.g. 0.225 kips).*

This special influence line definition has been chosen in RM2000 because the traffic loading on bridges generally moves along lanes eccentric or even skew to the system line or element axis. This general load application condition cannot be taken into account in the classical influence line approach. The disadvantage of this procedure is, that it is not possible to calculate one single influence line for a special point. A simultaneous calculation of all influence lines for all points of the lane is required, and, in order to get sufficiently accurate results, a sufficient amount of lane points must be specified (generally all element start and end points).

The evaluation of the influence lines is done by placing a predefined traffic load set such that the influence on the result is a maximum. These traffic load sets are called **"Load Trains"**. The definition of these Load Trains is described in chap. 6.10. A special facility is available for influence line evaluation at points where there is an abrupt change of sign of the influence line. (This facility complies with the British Standard for traffic loading).

6.8.3 Performing the Traffic Load Analysis

6.8.3.1 Overview

All necessary Lanes and Load Trains (defined in LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LANE and ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LTRAIN) have to be available before the traffic load calculation can be started.

The traffic load calculation is done in several steps in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE by applying the appropriate Actions:

- First all temporary and final superposition files have to be created and initialised **(Actions SupInit)**
- Then the influence lines for all relevant Lanes are calculated **(Actions Infl)**
- The Envelopes for all Lanes are created by evaluating the existing influence lines using the prepared Load Trains **(Actions LiveL)**
- Finally the intermediate Envelopes related to the different lanes are combined to get the final Envelope describing the min./max. internal forces due to traffic **(Actions SupOr, SupAnd, etc.)**

6.8.3.2 Superposition file initialisation

The initialisation is made in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE by applying the appropriate Actions, selecting

- ⊙ Envelope Actions
- ➢ SupInit Superposition file initialisation

This Action requires the definition of or shows the following data:

- ➢ Command — The user selected command is displayed.
- ➢ Input file (*.sup) — It is possible to take an existing superposition file and to write it directly into the new superposition file when initialising the new one. The existing SP-file and the new SP-File will then be identical. Leave this input empty in order to get an empty new SP-file.
- ➢ Factor (-) — the multiplication factor is not applicable for this function.
- ➢ Output file (*.sup) — Name of the new superposition file to be initialised
- ➢ - — input field not used
- ➢ Delta-T — Duration of the Action (not needed for this Action)
- ➢ Description — Descriptive text (max. 80 characters)

6.8.3.3 Influence line Calculation Action

The calculation of influence lines is selected by inserting the appropriate Actions into the Action Table in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE. This is done by selecting

- ⊙ Calculation actions
- ➢ Infl Calculation of Influence Lines

This Action requires the definition of or shows the following data:

- ➢ Command The user selected command is displayed.
- ➢ Lane-number Select the wanted lane for the calculation of the influence lines (interactive selection possible by clicking the arrow symbol next to the input field).
- ➢ Reference LC(-) Only to be used for geometrically non-linear structures (the selected Load Case will be used to establish a 'tangent matrix' to allow the traffic calculation by using this stiffness)
- ➢ Influence-file(*.inf) Name of the binary output file containing the influence lines. A '*' stands for 'default' –a file called 'lane0001.inf' will be created for lane 1.
- ➢ List-file Name of the ASCII output file containing the influence lines. A '*' stands for 'default' –a file called 'lane0001.lst' will be created for lane 1.
- ➢ Delta-T Duration of the Action (not needed for this Action)
- ➢ Description Descriptive text (max. 80 characters)

6.8.3.4 Graphic presentation of influence lines

All influence lines for all elements can be displayed graphically on the screen (⇧RESULTS ⇨PLINFL) after the calculation (⇧RECALC) has been performed.

Plot files presenting the influence lines can be created after they have been calculated. This is done by inserting the appropriate Plot Action in the Action Table, selecting

- ⊙ Plot actions
- ➢ PlInfl Plot influence lines

This Action requires the definition of or shows the following data:

- ➢ Command The user selected command is displayed (PlInfl).
- ➢ Influence-file(*.inf) Select the wanted influence line file being created when using the action INFL.
- ➢ Element Select the element we want to see influence lines for.

- Plot file — Name of the graphic (plot) file that will be created in this Action. A '*' stands for default –a file called 'lane0001.pl' will be created for the influence lines for lane 1.
- - — input field not used
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

6.8.3.5 Evaluation of influence lines

When all the necessary superposition files and the necessary influence lines for the relevant lane(s) are available the Action for the calculation of the traffic load (evaluation of influence lines) can be inserted at the appropriate position in the Action Table. This is done by selecting

- ⊙ Calculation actions
- LiveL — Live load calculation by evaluation of influence lines

This Action requires the definition of or shows the following data:

- Command — The user selected command is displayed.
- Lane-number — Select the wanted lane for the calculation of the Load train (interactive selection possible by clicking the arrow symbol next to the input field).
- Load Train number — Select the wanted Load Train (interactive selection possible by clicking the arrow symbol next to the input field).
- Output file(*.sup) — Name of the (existing) superposition file (either empty but initialised or already containing results of previous actions).
- List-file — Name of the ASCII output file containing the influence lines. An '*' stands for 'default' –a file called 'lane0001.lst' will be created for lane 1.
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

6.8.4 Non-linear calculation of Traffic Load Cases (LiveSet)

The calculation of traffic load cases by evaluating influence lines is based on the superposition principle and therefore in general not allowed for non-linear calculations. However, it is in many cases allowed to use a process, where the dead loads are calculated with taking into account the non-linearity of the system, whereas the behaviour under traffic loading is considered to be linear. The stiffness matrix used for the traffic load analysis is in this case the tangent matrix based on the deformed shape under dead loading. This approximation might be suitable when the traffic loads are small compared to the dead loads.

Assuming linearity of the behaviour under traffic loading allows using the tangent matrix for calculating and evaluating the influence lines to get the worst stressing state due to traffic loading. This process is used in the program if one or more non-linearity options are set in the menu ⇧RECALC and an influence line evaluation is selected in the construction schedule.

But sometimes a full non-linear calculation of traffic loads is required (e.g. in the case of a highly non-linear behaviour or if the traffic loading is a considerable part of the total loading). This is performed by calculating traffic load cases directly like any other load case, creating Load Cases and Load Sets in the related database tables by using the standard Load Types provided in the program.

The effort for directly defining the Load Sets describing complex load trains moving on arbitrary lanes may however be considerable, especially if the worst position of the load train for a certain internal force component has to be considered. In order to support this task the function (Calculation Action)

- **LiveSet** has been provided in RM2000.

This Action generates - by using the influence lines - a Load Set describing the loading due to a Load Train acting at the worst position with respect to a (user defined) characteristic result value (internal force or deformation component, maximum or minimum). This Set is assigned to a Load Case and a full non-linear calculation may be performed for this case.

Note: *This process is based on the assumption that the relevant position of the Load Train is not affected by the non-linearity, i.e. in general, that the zero-crossing of the influence lines do not move due the non-linearity.*

Input parameters for the function LiveSet:

- Inp1: Lane, LTrain — Choice of the Lane and the Load Train (interactive selection by using the arrow symbol). One existing Lane **and** one existing Load Train **must** be selected (no default).
- Inp2: El., Ntel, DOF — Choice of the required characteristic value (interactive selection by using the arrow symbol). The position in the system is specified by El. and Ntel (radio-buttons for element start and end points, otherwise no. of subdivision point). DOF specifies the result component (internal force, deformation) becoming a maximum or minimum.
- Out1: Lset, Init(-) — No. of the Load-Set to be generated, and code, whether the set should be initialised prior to adding the actual loads to it. The Load Set must have been previously created (typically as an empty Load Set) in the corresponding database table.
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

Such a fully non-linear calculation will usually only be performed for a restricted number of characteristic max./min. values in order to avoid an exploding calculation effort. However, proceeding in this manner often allows a very good estimation of the influence of non-linearity by comparing for some values the fully non-linear results with the results of the influence line evaluation process.

6.9 Traffic Lanes

6.9.1 General

Lanes are fixed paths along which Load Trains (refer next chapter) can move. Only Load Trains used for the influence line evaluation and described in chap. 6.10 – Traffic Load Trains can be related to Lanes, all other Load Sets used in numbered Load Cases have to be related directly to the structural system (elements or nodes).

A lane consists of a series of points, where point loads in a specified direction are applied, giving unity result vectors called **“(related) influence lines”** in this document. These unity result vectors are later multiplied with the actual load intensity values in the influence line evaluation process (see chap. 6.8.1).

The Lane Points are defined relative to the structural elements and do not need to be parallel to the axis of the structural model (skew lanes are allowed, e.g. in transition areas). A series of Lanes can be interconnected in any random pattern if desired – thus if a traffic lane on a bridge traverses a bridge at a highly skew angle it can be easily defined using this flexible ‘Lane’ definition.

The automatic determination of the position of the lane points by the provided macros may be supplemented or refined by direct definitions in the GUI.

The loading can be defined as being eccentric to the ‘Lane’.
Any number of lanes may exist on a structure at any one time.

Any Lane is positioned relative to the members of the structural model, usually the beam elements forming the bridge deck. The beam elements forming the deck are dependent from the structural model, generally we find either

- one series of longitudinal elements (continuous beam model used e.g. for a box girder) or
- several series of longitudinal elements being connected with a series of transverse beam elements (grid of beams model used e.g. for multiple main girder structures)

Considering these two possibilities, the Lanes can either be related to

- longitudinal elements, or to
- transversal elements

Lanes related to longitudinal elements are defined by the eccentricities of their centre lines in the local y and z directions. Loads are applied on these lanes as distributed element loads, causing an additional distributed torsion moment due to the eccentricity of the Lane.

Lanes related to transversal elements are identified by their relative position of the centre line on the element. This position is characterised by the ratio x/l between the distance of the Lane from the element begin (x) and the total elastic element length (l).

6.9.2 Definition of Lanes

Lanes are defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LANE. The Lane Table (upper table of the input pad) lists all currently existing Lanes. Lanes can be deleted, modified and copied by using the appropriate buttons. A new Lane is added to the Lane Table by selecting the ‘Insert before’ or the ‘Insert after’ button.

The Lane definition pad requires the definition of or shows the following data:

- ➢ The Number of the new Lane
- ➢ The Location – this is the name of the created ASCII file containing the input (the name cannot be modified by the user)
- ➢ The Output file – this file will contain the Lane listing after having run the calculation (ASCII file, the name cannot be modified by the user)
- ➢ The Name of the ASCII file containing all the influence line data (cannot be modified by the user)
- ➢ An Arbitrary descriptive text for better identification of the Lane

After having confirmed the input the new lane will be displayed in the Lane Table:

- The lane number in the first column (Number).
- The second column (Npos) shows the number of Load Points along the Lane (this value will be ‘0’ after having created the lane. The number of positions will be displayed once the calculation has been run).
- The third column (Ninfl) shows the number of influence lines (this column will be ‘0’ after having created the lane. The number of influence lines will displayed once the calculation has been run).
- The other columns have similar meanings as in the input pad.

Lane Point Property Data

Lanes are basically defined by **specifying the position of all Lane Points** and the unit loads acting in these points in the Lane Point Property Table (the lower table in the Lane definition pad).

Using the below described standard functions for entering all data of the Lane Property Table is however a tedious work and it is recommended to use the provided macro commands to generate a series of Lane Points very easily. These macros are later described. The basic property definition functions are described below to illuminate the database interrelations.

Two sets of basic input functions are provided for the definition of Lanes:
- Functions for describing the position of the Lane Points
- Functions for describing the load direction in the Lane Point

One of the functions for describing the position and one of the functions for describing the unit load have to be selected for every lane point.

Functions for describing the position of the Lane Points:
- POS3D
- POSEL
- POSEG
- POSERL

Functions for describing the unit loads acting on the lane:
- POSFG
- POSFRG

POS3D: The position of the Lane Points is defined by the global coordinates. This function may only be used if the Lane Points coincide with nodal points of the structure. The related influence lines are calculated by applying unit nodal point forces on the structural nodes coinciding with the Lane Points.

Required input data for POS3D:
- X global coordinate components
- Y to identify the load point on
- Z the lane directly
- Factor multiplication factor for this lane

Note: *All unit loads applied at that point are multiplied by the multiplication factor. Per default this factor is set to 1, but it is often used for specifying the dynamic load coefficient. This allows to specify a basic Load Train without consideration of the dynamic coefficient in ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LTRAIN.*

POSEL or POSEG: The position of the Lane Points is related to the elements of a main girder (elements approximately parallel to the Lane). The unit loads are in this case acting on the system as distributed loads or point loads within the element. The specified eccentricity causes additional torsion moments to be applied on the elements. The specified unit load may act in the local element directions (POSEL) or in the global directions.

Required input data:

Element	Number of element to which the point is related
dl/L	Distance from the defined point to the next point expressed as the proportion of the distance between these two points.
Factor	Multiplication factor for this lane
⊙ Local/Global	Switch defining the eccentricity being given in the local or global coordinate system.
➢ x/l	Distance of the Lane Point from the begin of the defined element (if defined locally expressed as the ratio between the distance and the element length).
➢ ey, ez	Eccentricity of the Lane Point to the defined element (from the Cross Section Centroid to the Lane in the specified coordinate system).
⊙ No ecc.	No cross section internal eccentricity is considered
⊙ Ygl, Zgl	The internal Cross section eccentricity in y or z direction will additionally be considered.

POSERL: The position of the Lane Points is related to the transversal elements of a multiple girder system (elements approximately perpendicular to the Lane). The unit loads are in this case acting on the system as point loads acting at the specified position of the specified transverse element.

Required input data:

Element	Number of element to which the Lane Point is related.
dl/L	Distance from the element begin to the defined Lane Point expressed as the ratio between the distance (dl) and the element length (L).
Factor	Multiplication factor for this lane.

POSFL or POSFG: Definition of the unit load acting on the Lane Point specified with POSEL or POSEG.

Required input data:

Element	Number of element to which the input refers.
Fx/Fy/Fz	Components of the unit load distributed from the point on the lane to this element in the local (POSFL) or global (POSFG) coordinate system.

- ⊙ Local/Global — Switch selecting whether the load vector is given in the local or global coordinate system.
- ➢ x/l — Distance of the lane point from the begin of the defined element expressed as the ratio between the distance and the element length.
- ➢ Ey, Ez — Eccentricity of the load from the element (always local!)
- ⊙ Local — The local element coordinate system is used for the eccentricities Ey and Ez
- ⊙ Local + Y Ele.-ecc — The eccentricity Ey refers to the top or the bottom (SYSTEM definition) of the cross section (internal eccentricity is additionally considered)
- ⊙ Local + Z Ele.-ecc — The eccentricity Ez refers to the most left or right edge (SYSTEM definition) of the cross section (internal eccentricity is additionally considered)

POSFRG or POSFRL: Definition of the unit load acting on the Lane Point specified with POSERL.

Required input data:

- Element — Number of element to which the unit load is related.
- dl/L — Distance from the element begin to the defined Lane Point expressed as the ratio between the distance (dl) and the element length (L).
- Fx/Fy/Fz — Components of the unit load distributed from the point on the lane to this element.
- ⊙ Local/Global — Switch selecting whether the load vector is given in the local or global coordinate system.

Lane Point Property Table:

The defined Lane Point Properties are displayed in the Lane Point Property Table:

- The name of the function as described above in the first column (including the code “Y” or “Z” if internal eccentricities are additionally considered).
- The second column (Elem) shows the number of the related element. The code “d” is shown in the function POS3D where no element is related.
- The other columns (Data1 to Data7) contain the related data depending on the selected function.
    - Data1 to Data3 are either the coordinates of the Lane Point (POS3D) or the defined eccentricities (POSEL, …).
    - Data4 to Data6 are the components of the unit load vector (POSFL, POSFG).
    - Data4 is “dl/L” in the functions POSEL,…
    - Data7 is always the multiplication factor

6.9.3 Macros for the Definition of Lanes

As already mentioned, defining a Lane using the above described standard functions is a very tedious and time consuming work because two definitions in the Lane Point Property Table have to be made for every point on the lane, where related influence lines should be calculated. These are in general every element begin and element end of the main girder elements and maybe also intermediate points.

A set of 4x3 Macros has therefore been provided to generate a part of or the whole Lane Point Property Table with one command and the specification of few data. The Macro creates automatically all Lane Point Property data of the selected Lane for all elements of the deck. This data is stored in the Lane Point Property Table for the calculation. The result of a macro (the data stored in the Lane Point Property Table) can be modified using the above described input functions, but the Macro input data is not available any more. In order to re-create a Lane it is necessary to delete it first and then to run the Macro again.

The provided macros are offered in the Lane Point definition pad where also the basic Lane Point Definition Functions described above are selected. The following macros are available:

- MACRO1X concentric lane on main girder, longitudinal load (braking)
- MACRO1 concentric lane on main girder, vertical load
- MACRO1Z concentric lane on main girder, centrifugal load

- MACRO2X eccentric lane on main girder, longitudinal load (braking)
- MACRO2 eccentric lane on main girder, vertical load
- MACRO2Z eccentric lane on main girder, centrifugal load

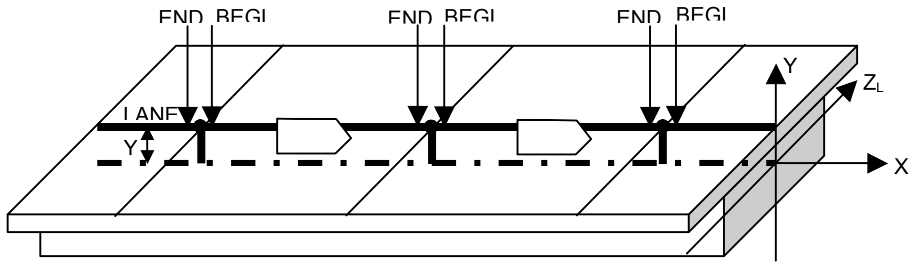
- MACRO3X lane on cross beams, longitudinal load (braking)
- MACRO3 lane on cross beams, vertical load
- MACRO3Z lane on cross beams, centrifugal load

- MACRO4X lane on cross beams, longitudinal load (braking)
- MACRO4 lane on cross beams, vertical load
- MACRO4Z lane on cross beams, centrifugal load

One of these Macros will be applicable in most practical cases.

MACRO1:

Elements forming an individual longitudinal deck are defined. The lane is concentric on this element series having only a vertical eccentricity in order to apply the load at the surface of the deck.

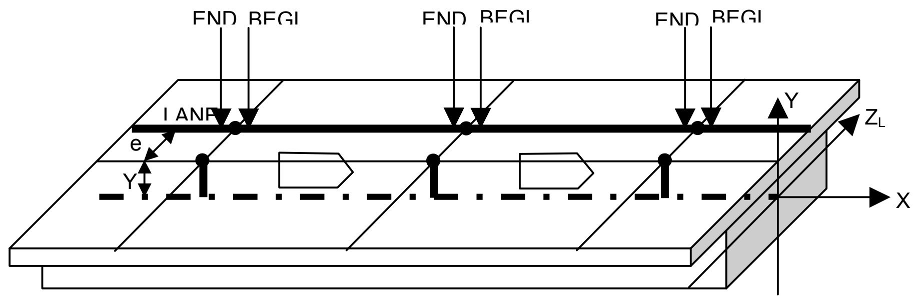


MACRO1X generates an equivalent lane for forces in the longitudinal direction

MACRO1Z generates an equivalent lane for forces in the transverse direction

MACRO2:

Elements forming an individual longitudinal deck are defined. The lane can be eccentric to this element series and can also have a vertical eccentricity in order to apply the load at the surface of the deck.

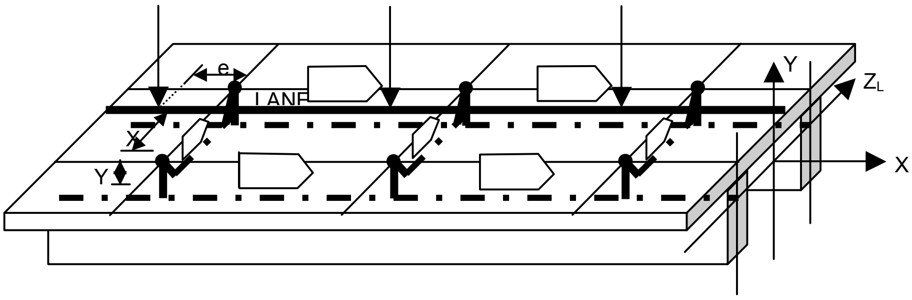


MACRO2X generates an equivalent lane for forces in the longitudinal direction

MACRO2Z generates an equivalent lane for forces in the transverse direction

MACRO3:

The deck is defined by two or more series of longitudinal elements being connected by one or more series of transverse elements. The lane can be relative to the transverse elements (cross members) as well as relative to the longitudinal elements. Lanes between two series of longitudinal elements will usually be related to the cross members, lanes outside the longitudinal elements will be relative to these longitudinal elements.



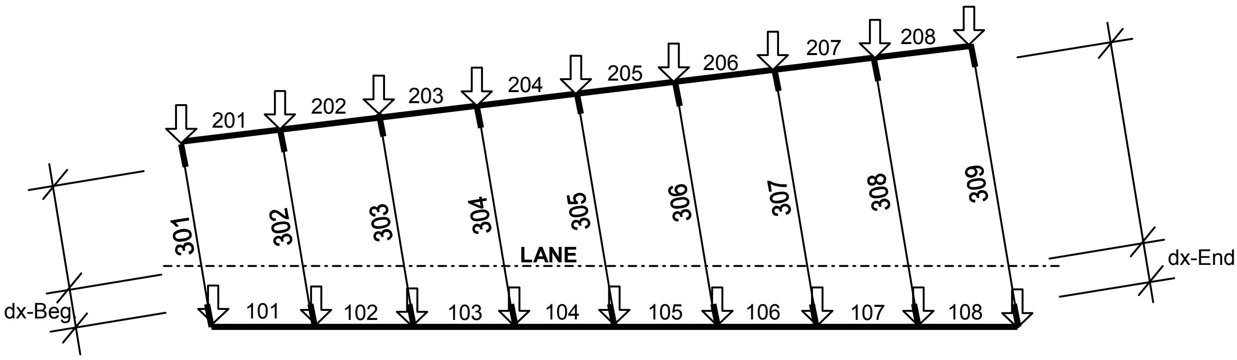
MACRO3X generates an equivalent lane for forces in the longitudinal direction

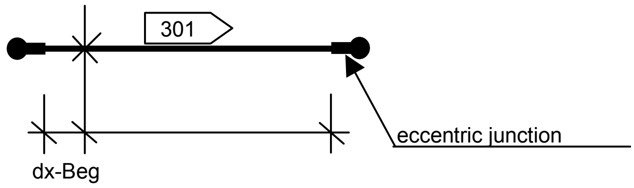
MACRO3Z generates an equivalent lane for forces in the transverse direction

MACRO4:

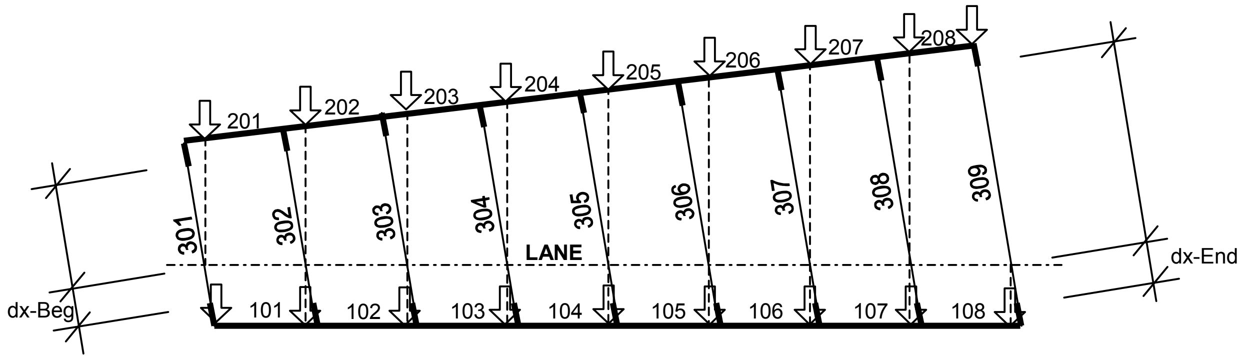
The loading is related to transverse girders as in Makro3, but the forces are not applied directly on the cross girders but transferred to the longitudinal elements without creating fixed end moments.

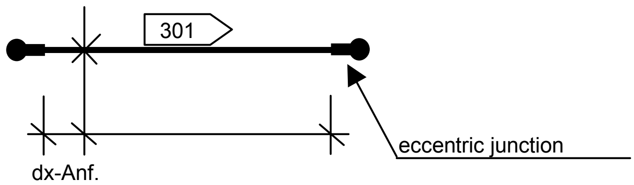
The lane points are described by their distance from the element begin of the transverse elements. This distance can be expressed as the percentage of the clear element length [%], or as the distance itself [Length(structure)]. By defining different distances for the first element (x/l-Beg or dx-Beg) and the last element (x/l-End or dx-End) it is possible to define lanes running in a skew direction over the cross girders.





With the additional information ORTHOGONAL TO LANE the primary elements (i.e. on the longitudinal girder) will be found orthogonal to the lane direction.





MAKRO4X same as above, for longitudinal load
MAKRO4Z same as above, for centrifugal load

Input data for the MACROS:

The macro pad contains a table of macro input data. This table is empty after selecting the Macro Definition Function. Data can be entered in this table by using the “Insert before” or “Insert after” button. Several lines in this tables may be created. As long as the user does not exit the Macro Definition Function he can modify the macro input data by selecting the line to be modified and clicking the “Modify” button.

After exiting the Macro Definition Function the appropriate data of the Lane Point Property Table will be generated and inserted, and the macro data will not be available anymore, if the Macro Definition function is selected again.

The Macro Input Data Table contains the following data:

- Kw — Shows the Position Definition Function to be used in the Lane Point Property Table (e.g. POSEG)
- Elem-from — series of elements
- El-to — to be considered
- El-step — for this lane
- x/l — **Only for MACRO3, MACRO3X, MACRO3Z:** distance of the lane from the begin of the loaded element (cross member) expressed as the ratio between the distance and the element length.
- ey — **Only for MACRO2, MACRO2X, MACRO2Z and MACRO3, MACRO3X, MACRO3Z:** vertical eccentricity of the lane relative to the centre of gravity of the element
- ez — **Only for MACRO2, MACRO2X, MACRO2Z and MACRO3, MACRO3X, MACRO3Z:** horizontal eccentricity of the lane relative to the centre of gravity of the element
- Phi — (dynamic) coefficient for this lane; this value is taken as multiplication factor (Data7) in the Lane Point Property Table.
- Ndiv — Number of subdivisions for each considered element
- Type — **Only for MACRO1, MACRO1X, MACRO1Z** and **MACRO2, MACRO2X, MACRO2Z:** program internal code for the type of load application

Inserting or modifying these data creates an input pad requiring the following definitions:

For all Macros:

- ⊙ No ecc. — No cross section internal eccentricity is considered
- ⊙ Ygl — The global cross section internal Y eccentricity is considered
- ⊙ Zgl — The global cross section internal Z eccentricity is considered

- ➢ El-from — elements to be
- ➢ El-to — considered for
- ➢ El-step — this lane definition

For MACRO3, MACRO3X, MACRO3Z:

- ➢ x/l — position of the lane inside the element

For MACRO2, MACRO2X, MACRO2Z and MACRO3, MACRO3X, MACRO3Z:

- ➢ ey — vertical eccentricity of the lane
- ➢ ez — horizontal eccentricity of the lane

For all macros:

- ➢ Phi — (dynamic) coefficient for this lane, stored as multiplication factor for the lane
- ➢ Ndiv — Number of subdivisions for each considered element

For MACRO1, MACRO1X, MACRO1Z and MACRO2, MACRO2X, MACRO2Z:

Switch describing where the unit loads should be applied:

- ⊙ Begin+End of Elements
- ⊙ Only begin of Element
- ⊙ Only end of Element
- ⊙ Begin/End/Begin/End of Element
- ⊙ End/Begin/End/Begin of Element

The Macros for generating lanes related to main girder elements will create load positions at the begin and/or end of elements (possibly among others defined by Ndiv > 1). The expression for “x/l” will be 0.0001 for the begin of an element and 0.9999 for the end of an element (column DATA1 in the Lane Point Property Table) in order to get a correct presentation of sudden changes of internal forces at the nodes.

The switch describing whether Lane Points should be created at the begin or the end of the elements is set per default to “Begin+End”, that means the unit loads are applied at the begin and the end of the specified elements. In this case the unit load will be applied at intermediate nodes shortly before and shortly behind the node, giving almost the same influence line twice. It is possible to save computing time by defining Lane points only at the begin or the end of the elements, but internal forces” jumping” at the nodes (e.g. shear forces) will in this case not be maximised or minimised correctly at all element ends.

The functions Begin/End/Begin/End and End/ Begin/End/Begin allow to set a Lane Point at both sides of every 2nd node, e.g. at the end of the 2nd element and the begin of the 3rd element, at the end of the 4th element and the begin of the 5th element and so on.

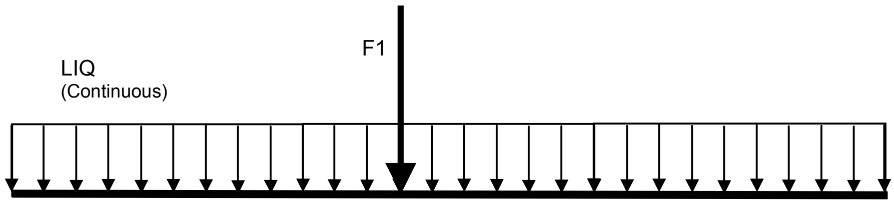
6.10 Traffic Load Trains

6.10.1 General

A set of loads moving over the structure along Lanes is in *RM2000* called 'Load Train'. There is no limit to the number of Load Trains that can be defined. The loads building a Load Train are

- a distributed (line) load acting on the whole length of a Lane (if unfavourable)
- point loads describing vehicles or vehicle axes

The simplest case of a Load Train is a **continuous** uniform line load and one point load characterizing the additional load due to a heavy vehicle.

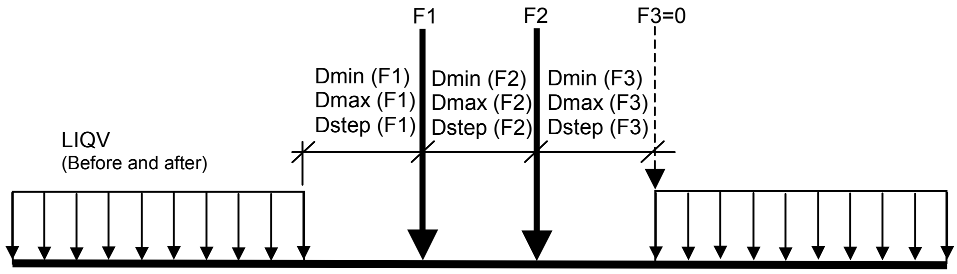


E.g. assuming the basic uniform traffic load being 5 kN/m^2, the dynamic factor being 1.10 and the total width of the 2 lane roadway (defined here as 1 Lane) being 7.5 m, the continuous uniform load LIQ would be -5.0 * 1.10 * 7.5 = -41.25 kN/m. Further assuming 2 parallel heavy vehicles of 250 kN each moving along the Lane and covering a length of 6.0 m, the point load to be superimposed to the uniform load will be

F1 = 2 * (-250) *1.1 – 6 * (-41.25) = -302.5 kN

Note: *The above example considers the dynamic factor as part of the Load Train definition. An alternative would be to define the dynamic factor as multiplication factor in the Lane definition (Data7 of the Lane Point Property Table), and to use the basic traffic loads for the specification of the Load Trains.*

Another example for a typical Load Train is:



The above example characterizes a traffic load consisting of a vehicle with 2 axes (F1, F2) and a uniformly distributed traffic load in front and behind of the vehicle. Note that a fictitious point load F3 (value 0) has to be defined in order to be able to specify the distance between the point load F2 and the start of the uniform load behind.

The 'Load Train' (better the point load group of the Load Train) is positioned on the structure in a way that the resulting internal forces are maximised or minimised. The distributed load of the Load Train is applied in all parts of the Lane, where it gives unfavourable contributions to the internal forces.

The positions of Load Trains are not related directly to the structural elements, but the Trains are necessarily related to 'Lanes', which themselves are defined relative to the elements as described above. I.e. all Load Trains are moved along Lanes and cannot be assigned directly to the structural system.

The intensity of the uniformly distributed loads may be defined as varying in accordance with the total length of the loaded portions of the bridge. This functionality is for instance required in the British Standard BS5400. The variation of the load intensity may be defined in the form of a table or a formula.

BS 5400 furthermore requires to consider 2 heavy vehicles behind each other, both with 2 axis loads. The distance between the vehicles can vary within certain limits and must be chosen to give the most unfavourable results. This variation is done automatically in the program, if the possibility to specify a maximum and minimum distance between the point loads has been used.

A special facility is available for influence line evaluation at points where there is an abrupt change of sign of the influence line. (This facility complies with the British Standard for traffic loading).

6.10.2 Definition of Load Trains

The user can define an unlimited number of Load Trains independent from the Lane definitions. Lanes and Loads are related to each other later in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION, when the calculation is performed.

All Load Trains are defined in the function ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LTRAIN. They consist of a distributed line load over the whole length of the lanes, and of concentrated loads applied at defined distances from each other.

The Load Train Table (upper table in the definition pad) lists all currently available Load Trains. Existing Load Trains can be deleted, modified and copied by using the appropriate buttons. A new Load Train is added by selecting the 'Insert before' or the 'Insert after' button, where a new input pad will be opened, requiring the following data to be input:

- Number of the new load train
- Fac-min (multiplication factor for the minimum evaluations)
- Fac-max (multiplication factor for the maximum evaluations)
- Location is the name of the created ASCII file containing the input (cannot be modified by the user)
- Arbitrary descriptive text for the defined Load Train

The Load Train Property Table (lower table in the input pad) contains the load data describing the Load Train actually selected in the Load Train Table. To enter or to modify the load data, the appropriate Load Train is selected in the upper table, and the detailed information can be entered or modified in the Property Table by either selecting the 'Insert before', the 'Insert after' or the 'Modify' button.

Inserting a new load will open a selection menu where the type of the load is specified:

- LIQ   a uniform load being part of the Load Train
- LIF   a concentrated load
- LIA   a concentrated load according to AASHTO

Direction of the applied loads:
The direction of the loads of the load train is not entered as a Load Train Property, but implicitly defined by the Lane, to which the Load Train is applied, i.e. separate Lanes have to be defined for traffic loads acting in vertical direction and loads acting in horizontal direction (wind, centrifugal, braking forces,…). Vertical traffic loads act in the global Y direction, horizontal forces in the local x or z directions respectively.

Attention:	**The loads of the Load Trains are always acting in the positive coordinate direction generated by the Lane Definition Macro, Load Trains for the definition of vertical traffic loads acting usually in the negative global Y-direction have therefore to be entered with negative sign.**

Input data for LIQ (distributed load):

Q   Intensity of the uniformly distributed load [load/unit length].

Note: *Take care that the intensity value Q has the unit [force/length], whereas the function value Qfunct is a scalar value not subjected to changes due to changes of the unit system.*

f(qlen)   Function describing the variation of the distributed load (Q). The intensity Q is multiplied by this function to get the actually valid intensity Q if the load intensity is variable (this is required in

BS5400). Qfunct must be a function of the intrinsic variable QLEN, describing the effective length of the applied distributed load.

Interruption Code:

- ⊙ Continuous — The uniform load is applied everywhere along the deck where the influence line is identifying either a favourable or an unfavourable result.
- ⊙ Before+After — The uniform load is applied everywhere along the deck where the influence line is identifying either a favourable or unfavourable result, but interrupted by some concentrated loads (LIF)

Variability code:

- ⊙ Function No — The load intensity (Q) is **not** variable but constant, the function Qfunct (if specified) is not used
- ⊙ Function Yes — The load intensity is variable. The program uses the formula defined for 'Qfunct' to multiply it with the intensity input value Q.

Length Reduction Code:

- ⊙ Lb = Min (L , 2*A / Ymax): No — No reduction of base lengths
- ⊙ Lb = Min (L , 2*A / Ymax): Yes — Reduction in acc. With BS5400

This code can only be set if the variability code is set to "Yes". It governs the assumptions for the lengths of the loading (QLEN) to be considered. The influence line is normally divided into several sections with varying sign. The program investigates the cases that the load is distributed over the full base length of

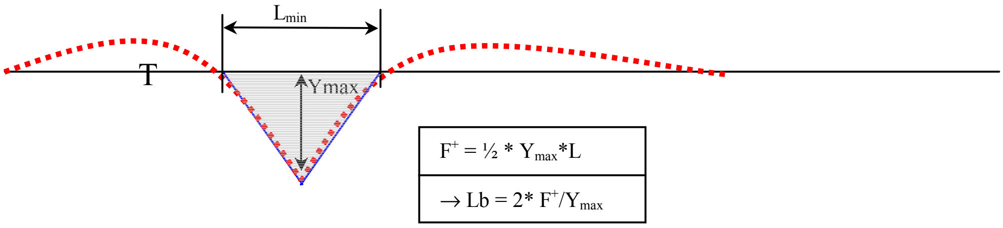
c) the section, where the result point is
d) two sections with the same sign
e) three sections with the same sign, etc.

BS 5400 has additionally the rule, that a reduced effective length has to be investigated in the case that the influence line has a cusped shape and lies wholly within a triangle joining the extremities of its base to its maximum ordinate (Ymax). The reduced length must be calculated using QLEN = 2*A / Ymax , where

Ymax — The maximum ordinate of the influence line
A — The Area under the influnce line

This reduced length has to be used if it is smaller than the base length L.

QLEN for the case a) (one section of the influence line) is therefore calculated using the formula QLEN = L_{min} = Min(L , 2*A / Ymax) if the reduction code is set to „Yes".



<u>Input if LIF is selected:</u>

F	Load intensity [load unit] in the specified direction
Dmin	Minimum distance of the concentrated load F from the end of the uniform load or from the begin of the load train or from a previous concentrated load.
Dmax	Maximum distance of the concentrated load F from the end of the uniform load or from the begin of the load train or from a previous concentrated load.
Dstep	Increment of distance between ‘Dmin’ and ‘Dmax’

Note: *Dmax and Dstep need not to be input if the distance is constant (as it is the case for most design code vehicles). All distances defined by Dmin, Dmax, Dstep are separately investigated by the program, the worst case being considered in the evaluation.*

<u>Input if LIA is selected:</u>

F	Load intensity [load unit] in global coordinate direction

6.10.3 Summary of Traffic Load Design Code Rules

6.10.3.1 OENORM (Austrian Code)

The Austrian Code requires either a group of trucks or a single tracked vehicle to be moved over the bridge. The group of trucks is applied in combination with a uniformly distributed surface load on the roadway including sidewalks. The group consists of up to 2 trucks with a weight of 250 kN, and 160 kN trucks on the remaining Lanes.

The tracked vehicle has a weight of 600 kN and is applied without distributed load (only a distributed load on the sidewalks has to be applied).

The truck group is normally relevant for the main design of major bridges, whereas the tracked vehicle may become relevant for small bridges or for the design in transverse direction. But a truck on the sidewalk has additionally to be investigated for the transverse design.

Additionally, a dynamic factor depending on the length of the spans is to be considered.

6.10.3.2 DIN

The German Norm distinguishes between a “Main Lane”, and “Secondary Lanes” which are differently treated. The width of the lanes is 3 m. A 600 kN Heavy Vehicle (SLW) with 3 axes with a spacing of 1.6m has to be moved along the “Main Lane” additionally to the uniform load of 5 kN/m^2 in front and behind of the vehicle. Both loads are to be multiplied with the dynamic factor.

The secondary lanes are only loaded with the uniformly distributed load without multiplying it with the dynamic factor. Any lane of the bridge can become the main lane, therefore a separate Load Train has to be defined for the basic uniform load and for the main lane load exceeding the basic load.

The Load Train describing the basic uniform load is applied to every lane and superimposed with “SupAnd” to get the worst case. The Load Train describing the exceeding load on the main lane is also applied to every lane, but separately superimposed with “SupOr”. Finally this Envelope is superimposed with “SupAnd” to the Envelope of the basic uniform load.

6.10.3.3 British Standard BS5400

The British Standard has the most complicated rules for traffic load application. The load intensity of the UDL part is here dependent on an effective load application length. This length is variable. The program has to investigate different cases and to select the worst case. The intrinsic variable QLEN is used for the effective length, and the load intensity has to be defined in the program as a function of QLEN.

Additionally, 2 heavy vehicles with 2 axes each have to be moved over the lanes. The distance between the two vehicles may also vary within given limits. The program again investigates different cases to get the worst case.

6.10.3.4 AASHTO Code

The speciality of the AASHTO Code is, that it requires a different procedure for standard cross sections and support cross sections. Whereas only one heavy vehicle has to be considered for the standard sections, 2 vehicles located on both sides of the support line have to be applied for evaluation of the forces in the support cross section.

Additionally different load intensities have to be used for evaluating shear forces and bending moments.

6.11 Additional Constraints

6.11.1 General

This function allows to determine superposition factors for previously calculated Load Cases to get a prescribed state in the structure by combining those Load Cases. The use of this function is restricted to linear analyses because the superposition rules must be valid.

The prescribed state is defined by a couple of internal force components and/or deformation components in specified structural points. These prescribed state variables are called „Constraint Conditions“. The database object “Additional Constraint” is specified by a set of such Constraint Conditions together with the associated Load Cases building the requred state.

The number of Load Cases to be superimposed with a variable factor to be determined must correspond to the number of Constraint Conditions, i.e. the number of constraints must be equal to the number of variables in order to get an equation system, which can be solved.

Load Cases or Combinations without a variable factor may be specified to be additionally considered for the calculation of the required state variables.

Typical applications for this function may be found in analyzing cable stayed bridges, where stay cable stressing forces should be determined, which guarantee, that the internal forces in the girder or in the cables remain within prescribed limits.

Example:

The following Load Cases are given for a cable stayed bridge:
- self weight
- a series of unity Load Cases, each of them corresponding to the stressing procedure of one cable

Problem to be solved:
Calculation of the final pre-stressing forces to guarantee admissible strength after combining self weight and all pre-stressing loading cases.

Prerequisites:
The admissible forces for each cable have to be defined as Additional Constraints. The Load Cases to be applied unconditionally and the stressing Load Cases for each cable have to calculated and stored in the database. The stressing Load Cases are unity cases, which are factorized by the calculation.

6.11.2 Input Sequence

Additional Constraints are numbered objects and specified in ⇧LOADS AND CONSTR.SCHEDULE ⇨ADDCON. The Additional Constraint Table (upper table in the input pad) contains the Number, the Location, and a descriptive text for every specified Additional Constraint. Inserting a new Additional Constraint opens an input pad, where the user defines the Number and the descriptive text. The file name for the 'Location' is generated automatically from the Constraint Number.

Two property tables (lower table in the input pad) are attached to the Additional Constraint: The Constraint Conditions Table listing all Constraint Conditions, and the Constraint Load Table, defining the Load Cases and Envelopes to be considered.

Note: *These property tables are separately defined and the program does not check at this stage, whether the condition, that the number of Constraint Conditions must be equal to the number of variable Load Cases is fulfilled. This check is done later when the calculation action is performed. The user will then get an error message, that the constraint calculation cannot be performed.*

The Constraint Load Table can be defined or modified by selecting the considered Additional Constraint and the function ⇧LOADS AND CONSTR. SCHEDULE ⇨ADDCON ⇩LOADS. The sum of all Load Cases and Envelopes defined in this table will be considered for the constraint calculation.

Selecting the "Insert" or "Modify" button shows an input pad requiring the data:

- ⊙ Load Case — Load Cases should be considered
- ⊙ Envelope — An Envelope should be considered
- ⊙ Fix — The factor is constant and predefined
- ⊙ Variable — The factor is variable

Envelopes can only be considered with predefined constant factors. The Fix/Variable-switch is in this case disabled. The following data is required, if "Load Case" has been selected:

- ➢ LC from — Load Cases
- ➢ LC to — to be
- ➢ LC step — considered
- ➢ Factor — "VAR" if variable or predefined value

The following data is required, if "Envelope " has been selected:

- ➢ DOF-from — Characteristic component of the Envelope (MaxVx … MinMz)
- ➢ DOF-to — should be left empty
- ➢ DOF-step — should be left empty
- ➢ Sup.File — Name of the Envelope to be considered
- ➢ Factor — predefined value

The Constraint Condition Table can be defined or modified by selecting the considered Additional Constraint and the function ⇧LOADS AND CONSTR. SCHEDULE ⇨ADDCON ⇩ELEMENTS. The Constraint Conditions may be related to Nodes or to Elements.

Selecting the “Insert” or “Modify” button shows an input pad requiring the data:

- ⊙ Node — Conditions related to nodes
- ⊙ Element (Begin) — Conditions related to the beginning of the elements
- ⊙ Element (End) — Conditions related to the end of the elements
- ⊙ Element — Conditions related to the Ndiv-point of the elements
- ➢ From — Elements or Nodes
- ➢ To — where
- ➢ Step — DOF’s or Internal Forces are constraint
- ➢ Ndiv Pnt — Subdivision Point (this data is only considered if “Element” is selected and results in subdivision points have been calculated
- ➢ Wmin — Minimal value of the Constraint Variable
- ➢ Wmax — Maximal value of the Constraint Variable
- ➢ Operator — the boundary condition might be limited with equal, smaller a variable or bigger a value
- ➢ DOF — degree of freedom of the constrain (deformation and/or internal forces)
- ➢ Selection of results in composite cross-sections
- ⊙ Normal — The results are made for the active (composite) cross-section.
- ⊙ Split — The results are made separately for the different parts of a composite cross-section (e.g. concrete or steel part).
- ⊙ Join — The results are made for both, composite and partial cross-sections.

6.11.3 Addition Function to Simplify the Input Procedure

How to use the additional check box FIX LC/SUP (VAR)

Without using the function FIX LC/SUP (VAR):

It is necessary to split up all used loading cases into fix and variable loading cases.

e.g.:
LC 1000 is the permanent loading case and consists = 101 + 102 + 103. All these loading cases are used. One of them should be variable. All three loading cases have to insert separately.

LC 101 is variable
LC 102 is fix
LC 103 is fix

It’s not allowed to use the loading case 1000 in the additional constraint definition.

With using the function FIX LC/SUP (VAR):

e.g.:
LC 1000 is the permanent loading case and consists = 101 + 102 + 103. All these loading cases are used. One of them should be variable. Insert only loading case 1000 and use the additional check box FIX LC/SUP (VAR).

LC 1000 is fix

It’s allowed to use the loading case 1000 in the additional constraint definition.

Therefore, it’s not necessary to insert following constraint definitions:

LC 101 is variable
LC 102 is fix
LC 103 is fix

7 Construction Schedule and Analysis Process

7.1 General

A sequence of calculations, file manipulations and combinations are carried out in the construction schedule in *RM2000* in a similar way to the batch file concept.
The principal of the construction schedule is that every single action that is defined takes place within a specific project related time frame.
The loading of the elements and the manipulation of the results is all done in combination with the currently active elements in the structure. (The activation stage represents the current state of construction of the project that also ties the construction state with the time frame for the project.)

Before any construction schedule action can be carried out, the following must have been completely defined.

- **The structural model**:
  Material, Cross-sections, Nodes, Elements, Support conditions which are defined in ⇧PROPERTIES ⇨MATERIAL / CS and ⇧STRUCTURE ⇨NODES / ELEMENT respectively
- **All Load Sets and Load Cases** for all the Construction Stages: Permanent loads, pre-stressing, traffic, additional loads, etc which are defined in: ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS)
- **All variables**:
  If Creep & Shrinkage are being considered in the construction schedule, the desired Creep Model must have been defined – this can be defined by the user in (⇧PROPERTIES ⇨VARIABLE) or can be imported from the data base (⇧FILE ⇨IMPORT) - ASCII file -Partial Project-Variable

7.2 System Activation

7.2.1 General remarks

The complete structural model must first be defined – as mentioned above.
The concept then is to define the elements that are active in the particular construction stage – thus representing the actual structure at that stage of construction.
Once an element has been activated, then it is assumed to be active throughout the Construction provided that it is not ‘de-activated’ – Thus only the newly active members for any particular construction stage need to be defined.
Elements can be ‘de-activated’ and subsequently ‘re-activated’ if desired.

The concept of the time related construction schedule is to accommodate elements that can Creep and/or Shrink. These elements must have their activation age defined.
This 'activation age' represents the age of the concrete elements at the moment that they become active .The age for Creep can be different from the age for Shrinkage.

7.2.2 The System Activation

The system activation is defined in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTIVATION. The input window consists of two tables, the upper table (Stage List) showing the defined Construction Stages with the age at the start of the activation, the duration of the activation stage and a general description, and the lower table (Activation List) showing the active elements for the selected stage.

The (Construction) Stage List:

The Construction Stages are identified by their number. The construction stage number is shown in the first column of the Stage List and is arbitrarily defined by the user. It is recommended that a sequentially increasing numbering scheme is adopted following the actual time axis for the project. (N.B. Even thought the program allows a higher number to be used before a lower one and also has no problems with gaps in the numbering sequence, the practice of using strange numbering systems is not recommended!)

The other items in the Stage List are the Location, the List file, the Time, the Duration and a Descriptive Text.

The 'Location' is the name of the program generated ASCII file for the system activation. This name is automatically built by the program and depends on the Stage Number.

The 'List' is the name of a generated ASCII file for the system activation listing. It is also automatically defined.

The 'Time' and the 'Duration' locate the Construction Stage on the time axis.
'Time' defines the start time (in days) of the construction stage.
Time 0 is always the start time of the 1st stage.

The start time of the subsequent stages cannot be specified by the user but is the same as the 'time' at the end of the previous stage, which is defined by the start time and the Duration.

'Duration' is only implicitly defined by the durations of the time dependent Actions (creep intervals) in the related Action List.

'Description' is an arbitrary alpha-numeric text describing the Construction Stage.

A new stage can be created by selecting the appropriate line in the Stage List and using the 'Insert before' or the 'Insert after' symbol at the top of the screen. An existing stage can be modified or deleted using the 'Edit' or 'Delete' symbol as appropriate. The stage can be 'renumbered' using the 'Edit' function.

The Activation List:

The list of activated/de-activated elements for the selected Construction Stage is shown below the Stage List. The list contains the elements to be activated or de-activated in the selected construction stage, the concrete age at the activation time, the start time for the shrinkage behaviour and the Activation Code.

The elements are defined in a series using a 'from, to, step' concept. The Activation Code will be 'ACT', if the series of elements is to be added to the currently active structure, or 'DACT' if the series of previously active elements are to be de-activated.

The Activation List does not contain all the currently active elements for the particular Stage, but only the changes with respect to the previous Stage – i.e. it only shows the newly activated/de-activated elements.

New entries in the Activation List are made by either using the 'Insert before' or 'Insert after' symbol of the Activation List.

Existing definitions can be modified/deleted using the 'Edit' or 'Delete' symbol. The following data is entered in the appropriate definition pad:

The Activation Code

- ⊙ Activate — elements are activated (added to the current structure)
- ⊙ Deact — elements are deactivated (removed from the current structure)

The element range:

- ➢ El-from; El-to; El-step — Numbers of elements that are to be activated or deactivated in the current construction stage

Parameters for Creep & Shrinkage (see also chap.7.4, Creep & Shrinkage):

- Age Concrete age of activated elements at Construction Stage start
- ts Age of the concrete, when shrinkage starts.

Note: *The start time of the Construction Stage is generally the moment, when the first load is applied, i.e. the time, when the formwork is removed or the pre-stressing is applied. 'Age' therefore is usually the time between pouring the concrete and removing the form-work/applying the pre-stressing. - Used for the Creep calculation*

At least one stage activation must have been defined before any calculation action on the structure can be made.

7.3 Call of Actions on the Structure

The actions that are applied to the structure for a selected construction stage are defined in the lower table (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION). Select the appropriate stage in the upper table and insert the desired action in the lower table using either the ‘Insert before’ or ‘Insert after’ symbol at the top of the lower table. A defined action can be modified/deleted using the ‘Edit’ or ‘Delete’ symbol.

The sequence and time history of the actions within the construction schedule is very important and must correspond to the actual physical construction sequence at the construction site!

The following action groups can be chosen by selection the radio button in the displayed input window:

- ⊙ Calculation actions (Static)
- ⊙ Calculation actions (Dynamic)
- ⊙ Check actions
- ⊙ Load case actions
- ⊙ Envelope actions
- ⊙ List actions
- ⊙ Plot actions
- ⊙ System command actions

The actions available under each action group are briefly described below.

Note: *The action must be chosen from the list and the selection confirmed with* `<OK>` *before the detail input pad for that action is opened.*

7.3.1 Available Actions for a Construction Stage

Calculation Actions (Static):

Calc: Calculation of a loading case containing any kind of external load (all loads except Creep&Shrinkage and Traffic).
The appropriate Load Case with its Load Sets must be defined before this action (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET and ⇩LCASE This action requires an existing)

Stress: Stress a set of pre-stressing cables (tendons). The pre-stressing loading case calculation can only be calculated after the relevant cables have been stressed. (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩TENDON).

Grout: Grout the tendons.
The action of grouting fills the duct with cement grout, making the tendon composite with the cross section and thus modifying the cross section properties. (Cross section properties updated).

TStop: Simplified (reduced) input for representing creep and shrinkage effects in a structure during stage construction where there is a repetitive sequence of construction. i.e. for a balanced cantilever bridge where several piers are built at different times but using the same sequence.
The structure is first defined as being built at the same time and then the groups of elements representing the different cantilevers can be 'frozen' for the appropriate period of time (Stop the Creep) to bring the creep time in line with the actual construction time. A considerable amount of input effort can be saved using this function as the number of prepared construction stages for the calculation can be greatly reduced.

Creep: Calculation of a Creep&Shrinkage loading case.
The Creep model for the concrete material as well as the Loading Case and Loading Sets must be defined before this action (⇧PROPERTIES ⇨MATERIAL or ⇨VARIABLE

UpdEmod: Youngs modulus for the materials is updated to accommodate its variation with time.
The original ('old') E-modulus is defined by the user but the updated ('new') E-modulus is used to calculate the loading case results.
The E-modulus variation with time must have previously been defined in ⇧PROPERTIES ⇨MATERIAL.

Infl: Calculation of influence lines.
Traffic lanes must have been defined before this action can be called. (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LANE).

LiveL: Calculation of the traffic loading effects using the existing influence lines for a lane.
Traffic loads and Influence Lines must first be defined (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LTRAIN

TempVar: A temperature variation diagram in accordance with any design code specifications can be prepared and applied to the structure. (⇧PROPERTIES ⇨ADDGRP)

Buckle: Evaluation of the buckling load (safety factor). Buckle determines linearly the stability divergence point of the structural system.

Failure: Evaluation of the failure load of the structural system (full geometrically non-linear analysis)

OpenTcl: Opens a *.tcl file
RunTcl: Executes a *.tcl file

Calculation Actions (Dynamic):

Tint: Time History calculation.
A table for the load variation over the time must have been previously specified (⇧PROPERTIES ⇨VARIABLE).

Eigen: Calculation of the eigen-frequencies (eigen-modes) for the structure.
A minimum requirement for this calculation is the specification of the self weight load using the program internal specific weight definition. Other masses can additionally be defined using the function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS.)

RespS.: Calculation of a response spectrum.
Eigen modes and the response spectrum table must first have been defined/calculated (⇧PROPERTIES ⇨VARIABLE)

Excit: Random analysis for excitation spectrum

Wind: Calculation of wind turbulence with aerodynamic effect

FFT: Power Spectrum Calculation using Fast Fourier Transformation

Check actions

FibChk: Fibre stress check.
The fibre stress check at specified points in the cross section for a specific Combination or Load Case. – Ref: ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB.
The Fibre Stress points in the cross section and the load cases and combinations must have been previously defined/calculated (⇧PROPERTIES ⇨CS ⇩REIN).

FibRpt: A detailed output of fibre stresses is created.

FibTend: Stresses of pre-stressing tendons are checked for selected loading cases

TndChk: Stress Check for pre-stressing tendons.

FibII: Special stress check for new Austrian Standard – no tensile stresses allowed.

UltChk: Ultimate moment; shear; or axial force check.
The Ultimate load check for any loading case or superposition file (result will be compared with a combination defined in ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩COMB).
The program creates a superposition file containing the max/min ultimate moment (shear, normal force) of resistance. This can then be compared to the factored envelope prepared by the user (see: ⇧RESULT ⇨ENVELOPE for numerical values or ⇧RESULT ⇨PLSYS for graphical output).
N.B. The superposition file must be initialised first (Envelope Action ‘SupInit’).

ShChk: Shear Capacity check – similar input as for UltChk

ReinIni: Initialisation of A2 reinforcement area.

Attention:	**All reinforcement areas A2 resulting from previous reinforcement design calculations will be lost in the database.**

UltRein: Dimensioning of carrying capacity

FibIIRe: Fibre stress dimensioning (State II)

Restart: The use of so called 'unit loading cases' requires a restart of the calculation once the multiplication factors for the unit loading cases are available (evaluated by ADDCON feature). The complete calculation is repeated again and again until a certain tolerance limit is reached. Than the next Construction Schedule will continue.

<u>Loading case actions:</u>

LcInit: Initialise a loading case (create the storage area) or create a new loading case out of a *.mod file (Calculation action 'Eigen') or make a new loading case by factorising an existing loading case.

<u>Initialise a loading case</u>
Inp 1: Blank
Inp 2: Blank
Out 1: Loading Case Number

<u>Create a new loading case out of a *.mod file</u>
Inp 1: Name of the Modal file to be made into a loading case.
Inp 2: Mode number from the modal file
Out 1: Loading Case Number for the modal file results

<u>Make a new loading case by factorising an existing loading case.</u>
Inp 1: Loading Case Number to be factorised
Inp 2: Multiplication factor
Out 1: Loading Case Number of New file.

LcAdd: A calculated loading case can be factorised and added into a superposition file or into another loading case.
Inp 1: Loading Case Number to be added
Inp 2: Multiplication factor
Out 1: The results of Inp1 * Inp2 will be added to this Loading Case Number/Superposition File

LcDel: Load case can be deleted (e.g. if this Load Case is needed several times during the calculation process).

<u>Envelope actions:</u>

SupInit: Every superposition file must be initialised (the storage area set to zero) before it can be used. SupInit initialises a superposition file (the storage area is set to zero) or creates a new superposition file by factorising an existing superposition file.

Initialise a superposition file
Inp 1: Blank
Inp 2: Blank
Out 1: The superposition file name

Create a new superposition file by factorising an existing superposition file.
Inp 1: The superposition file name to be factorised
Inp 2: Multiplication factor
Out 1: The new superposition file name.

SupAdd: Add the factored results from a loading case or superposition file to the results from another superposition file and put these results back into the original superposition file or into a completely new superposition file.
Inp 1: existing superposition file to which the results will be added (If the superposition file is new, it must be initialised before any action can be made on it – SupInit).
Inp 2: Number of the loading case or name of the superposition file that is to be factored by F1 & F2 (optional) and added to file defined in 'Inp1'.
F1: The multiplication factor for the loading case/superposition file if this force/moment/deflection reduces the total force/moment/deflection on the structure
F2: The multiplication factor for the loading case/superposition file if this force/moment/deflection increases the total force/moment/deflection on the structure
Out 1: superposition file containing result of 'Inp1 + Inp2'. If nothing is defined then 'Inp1' is replaced by the result i.e 'Inp1' = 'Out1'.

SupAnd: Factor the results from a superposition file and **Add** them to another superposition file **if** the result of the addition is structurally more critical (unfavourable).
Inp 1: The superposition file to which the results will be added. (If the superposition file is new, it must be initialised before any action can be made on it – SupInit).
Inp 2: Number of the loading case or name of the superposition file that is to be factored by F1 (optional) and added to the file defined in 'Inp1'.
F1: The multiplication factor for the loading case/superposition file (Will be taken as 1.0 if undefined).

Out 1: Superposition file containing result of 'Inp1 + Inp2'. If nothing is defined then 'Inp1' is replaced by the result i.e 'Inp1' = 'Out1'. (This new superposition file must be initialised before any action can be made on it – SupInit).

SupAndX: Factor the results from a superposition file and **Add** them to another superposition file, changing the sign where necessary, to make the result of the addition structurally more critical (unfavourable).

Inp 1: The superposition file to which the results will be added.
(If the superposition file is new, it must be initialised before any action can be made on it – SupInit).

Inp 2: Number of the loading case or name of the superposition file that is to be factored by F1 (optional) and added, with change of sign where appropriate, to the file defined in 'Inp1'.
F1: The multiplication factor for the loading case/superposition file (Will be taken as 1.0 if undefined).

Out 1: Superposition file containing result of 'Inp1 + Inp2'. If nothing is defined then 'Inp1' is replaced by the result i.e 'Inp1' = 'Out1'. (This new superposition file must be initialised before any action can be made on it – SupInit).

SupOr: Replace the old values in the combination file with the appropriately factored new results, conditionally.
Replace the old values in a superposition file (Inp1) if the new results, multiplied by the defined factor (Inp2), are more structurally critical.

Inp 1: Superposition file for comparison. The values in this file will be replaced if the values in the file being compared with it (Inp2) are more structurally critical.
(If the superposition file is new, it must be initialised before any action can be made on it – SupInit).

Inp 2: Number of the loading case or name of another superposition file that is to be factored and compared with the file defined in 'Inp1'. These factored values, where more structurally critical, will replace those in the file defined in Inp1.
F1: The multiplication factor for the loading case/superposition file (Will be taken as 1.0 if undefined).

Out 1: Superposition file containing the more structurally critical results from 'Inp1 or Inp2'. If no file is defined then the results are placed in Inp1: 'Inp1' = 'Out1'
(If the superposition file is new, it must be initialised before any action can be made on it – SupInit).

SupOrX: Replace the values in a superposition file with the factored results from another superposition file, changing the sign where necessary, if these values (with or without change of sign) are structurally more critical.

Inp 1: Superposition file for comparison. The values in this file will be replaced if the values in the file being compared with it (Inp2) - (with or without change of sign) are more structurally critical.
(If the superposition file is new, it must be initialised before any action can be made on it – SupInit).

Inp 2: Number of the loading case or name of another superposition file that is to be factored and compared with the file defined in 'Inp1'. These factored values, (with or without change of sign) where more structurally critical, will replace those in the file defined in Inp1.
F1: The multiplication factor for the loading case/superposition file (Will be taken as 1.0 if undefined).

Out 1: Superposition file containing the more structurally critical results from 'Inp1 or Inp2'. If no file is defined then the results are placed in Inp1: 'Inp1' = 'Out1'
(If the superposition file is new, it must be initialised before any action can be made on it – SupInit).

SupComb: Create a superposition file from the combination table.

Inp 1: The column number in the combination table (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB) that is to be made into a combination (superposition) file.

Out 1: Name of the Superposition file containing the result of the combination defined in Inp1 (File must be initialised first before any action can be made on it – SupInit).

SupImp: Evaluation of a load impact factor. The factor is derived from dividing the forces from the dynamic analysis by the forces from the static analysis of the same loading train.
The user must specify the two relevant superposition files.
(StaticSup * factor = DynamicSup).

List and plot actions:

DoList: Print loading case results into an ASCII list file
DoRep: Create a ‘report’ file containing specific user requested result output.

PlCross: Plot one selected cross section of the structure together with the cross section properties on an A4 paper.
PlCrSh: Plot the Creep and Shrinkage curves for a selected element of the structure on a A4 paper.
PlInfl: Plot the influence line for a selected element of the structure on a A4 paper.
PlSys: Use a user defined (existing) ASCII File to create the corresponding plot file.
PlTens: Plot the stress actions on a pre-stressing tendon on a A4 paper.

System command actions:

GoCopy: Copy any file to a new one
GoCrt: Show an existing plot file
GoDel: Delete any existing file
GoRen: Rename any existing file
GoSet: Change general options
GoWait: Interrupt the run of the calculation for a certain time interval (specified by the user in seconds). If no value is entered, the program waits until the user hits the <Return> key.

Note: *The **length of the file names** in all system command actions is limited to **24 characters**. This limit cannot be extended by the user. This restriction might especially be a problem when using **GoCopy** for copying a file to an other directory using a full path-name.*

7.3.2 Adding Actions into the Construction Schedule

The principle of the required Input data after having selected an action and confirmed with <OK> is described below::

- Command: The selected action is shown – can not be edited.
- Input 1: Existing files for manipulation are defined here.
- Input 2: Not always active – the input field has different meanings for different actions (such as multiplication factor, time steps for manipulating Input 1)
- Output 1: The manipulated files from ‘Inp 1’ is stored in ‘Output 1’
- Output 2: An ASCII List file (output printout) can be defined (’*’ tells the program to create the default name for the list file)
- Description: Descriptive text (max. 80 characters)

7.3.3 Start Single Actions Immediately

Under ⇧RESULT ⇨SCRIPT ⇩RUN STAGE ACTION following actions can start in this way:

CHECK-Actions:

FibChk Fibre stress check
TndChk Tendon stress list for a certain load case

UltChk Ultimate load check
ShChk Shear capacity check

LIST/PLOT-Actions:

All list and plot actions are available

SYSTEM COMMANDS-Actions:

All system commands actions are available

7.4 Creep & Shrinkage

7.4.1 General

Creep and shrinkage effects in concrete structures and in pre-stressed concrete structures considering all the construction stages (times/dates) and the final stage can be considered. Time (for creep and shrinkage) is measured in days in *RM2000*.

The time dependent inelastic (viscous) deformations consist of a stress-independent part (shrinkage), and a stress dependent part (creep).

The creep behaviour (stress dependency) is generally assumed to be linear, i.e. the creep strain ε_{cc} is a linear function of the elastic strain or the stress respectively ($\varepsilon_{cc} = \varphi * \varepsilon_e = \varphi * \sigma_c/E_c$). This assumption allows a beam model to be analysed, using internal forces instead of stresses (internal forces yield a linear strain distribution in the cross-section and a linear stress distribution, i.e. the cross-section remains plane in all states).

The creep coefficient φ is generally a function of time, of material, of the cross section properties and of the climatic conditions on site. Various creep models, describing the creep coefficient as a function of these parameters, have been developed. The most common of these have been prepared as a predefined set of variables to be imported into the database. These models are described below.

The shrinkage behaviour (the stress independent shrinkage (or swelling) strain as a function of time) is defined by the shrinkage coefficient $\varepsilon_{cs.}$ The shrinkage strain is assumed to be constant over the cross-section, thus the condition that cross-sections remain plane is fulfilled. Similar mathematical models exist for the calculation of the shrinkage coefficient as for the creep coefficient.

Creep and shrinkage are commonly treated together - *RM2000* combines these two influences into one Load Case. The creep and shrinkage process in a structure from the construction start time to time infinity is generally divided into Creep Periods. The creep periods are usually related to the construction stages or to major changes of loading. Calculating the C&S influence for one Creep Period is called performing a Creep Action. Every Creep Action has an associated Load Case for the results.

The C&S Load Case for one Creep Period is calculated by applying the appropriate creep strain $\varphi*\varepsilon_e$ and shrinkage strain ε_{sc} as initial strains on the structure (similar to a temperature loading). The values φ and ε_{sc} for a creep period are the differences between the diagram values of the start and the end of the period:

$$\varphi * \varepsilon_e = (\varphi(t_e,\ldots) - (\varphi(t_a,\ldots)) *\varepsilon_e$$
$$\varepsilon_{cs} = \varepsilon_{cs}(t_e\text{-}t_s) - \varepsilon_{cs}(t_a\text{-}t_s)$$

where

t_a = age of the concrete (in days) at the beginning of the creep period (if the element has been activated and the load has been applied before the beginning of the creep period)

or

= age of the concrete at the load application time (for creep, if the load is applied after the beginning of the creep period)

or

= age of the concrete at element activation time (for shrinkage, if the element is activated after the beginning of the creep period)

t_e = age of the concrete (in days) at the end of the creep period
t_s = age of the concrete (in days) at the beginning of the shrinkage process

Applying the strains on the structure yields redistribution forces within the cross section (called **internal redistribution forces** in this document) and forces due to external constraints (called **external redistribution forces**). The primary part (internal redistribution forces) and the total forces are stored in the database. The secondary part (external redistribution forces) is recalculated from these values when required.

<u>Preparation for the "Creep & Shrinkage" Load Case:</u>
The following items must have been prepared or calculated before a Creep & Shrinkage calculation is performed:

- Creep & Shrinkage model definition (either user defined or imported from the Standard Database)
- Material properties - relevant definitions for C&S.
- Element properties - relevant definitions for C&S.
- Calculation of the creep inducing loading (stresses).
- Definition of the Load Case for storing the creep & shrinkage effects
- Placing the Calculation Action for the Creep & Shrinkage Load Case in the construction schedule.

7.4.2 User Defined Creep & Shrinkage Models

The creep and the shrinkage parameters PHI(t, t_0, …) & EPS(t, t_s, …) of the material may be defined by the user in ⇧PROPERTIES ⇨VARIABLE and assigned to the material in ⇧PROPERTIES ⇨MATERIAL.

The function PHI is the ratio between the elastic strain and the creep strain. The creep coefficient PHI_∞, characterizing the creep strain reached after a very long period of time, is generally in the order of magnitude of 2 (for concrete in a European climatic environment), i.e. the creep strain is approximately twice the elastic strain of the creep inducing stress state.

The shrinkage coefficient is in terms of strain. I.e. it has no units! (not in per mille!). The final shrinkage coefficient is often in the order of magnitude of -15.E-5, which is equivalent to a temperature decrease of 15°C.

Attention:	**The shrinkage coefficient must be defined with a negative sign, the program does not automatically set it negative. Entering a positive value will yield a negative shrinkage, i.e. a swelling behaviour.**

Note: *The functions PHI(t, t_0,...) and EPS(t, t_s, ...) have to be defined for the whole calculation interval, i.e. in general for the time interval t_a to infinity. They are, as indicated, mostly not only functions of the actual concrete age t, but also functions of the age t_0, where the load is applied, and the age t_s, where shrinkage starts. The consideration of shrinkage starts with the element activation t_a, shrinkage between t_s and t_a is not taken into account.*

Examples:

1.)
Assuming that the functions PHI(t, t_0,…) and EPS(t, t_s, …) are independent from the load application time t_0 and other variables, and simply increase exponentially with the concrete age t from zero at the age zero to a final value PHI_∞ at time infinity.

E.g. $PHI(t) = PHI_\infty * (1.-e^{-t})$, $EPS(t) = EPS_\infty * (1.-e^{-t})$

Assuming PHI_∞ were 2 and EPS_∞ were 15.E-5, the appropriate variables with arbitrary names (e.g. PHICR and EPSSH) have to be defined as formulas:

$PHICR = 2.* (1.-e^{-t})$
$EPSSH = 15.E\text{-}5.* (1.-e^{-t})$

These variables have to be assigned to the material as creep and shrinkage coefficients respectively.

2.)
Assuming that PHI(t, t_0,…) is given in the form of a table describing the creep coefficient as a function of an effective time TEFF, the effective time being T-T0.

E.g.	TEFF = 0	PHICRTAB = 0
	TEFF = 1 day	PHICRTAB = 0.5
	TEFF = 7 days	PHICRTAB = 1.0
	TEFF = 30 days	PHICRTAB = 1.5
	TEFF = 365 days	PHICRTAB= 1.8
	TEFF = 10000 days	PHICRTAB = 2.0 (= infinity)

The above table for the variable PHICRTAB has to be entered using the variable definition function and selecting

- ⊙ Table
- ➢ Name PHICRTAB
- ➢ Description Descriptive text (max. 80 characters)

The creep coefficient PHICR may now be defined in terms of the effective time by defining the expressions relating TEFF to the intrinsic variables T and T0, and PHICR to the table values PHICRTAB:

TEFF = T-T0
PHICR = PHICRTAB(TEFF)

The shrinkage coefficient may be similarly defined. The variables defining the creep and shrinkage coefficients have to be assigned to the material in ⇧PROPERTIES ⇨MATERIAL.

7.4.3 Standard Creep & Shrinkage Models

7.4.3.1 Definition as Variables

The following creep & shrinkage models have been prepared by TDV as a set of variables to be imported into the program database. They are stored in the program directory and are part of *RM2000*. The theoretical background for these models is described in detail in the *RM2000* "Technical description".

- AASHTO Model (based on CEB/FIP 90) (cs-as96.rmd)
- British Standard Model (BS5400/1990 Part 4) (cs-bs54.rmd)
- Hong Kong Standard Model (based on BS5400/1990 Part 4) (cs-hs54.rmd)
- CEB/FIP Model Code 78 (cs-ceb78.rmd)
- CEB/FIP Model Code 90 (Standard = Product Model) (cs-ceb90.rmd)
- CEB/FIP Model Code 90 (Improved Summation Model) (cs-rsm90.rmd)
- DIN-Norm 1045/I (cs-di45.rmd)
- ON B4750 (cs-oe47.rmd)
- Norwegian Model (modified CEB90) (cs-nor.rmd)
- Hungarian Model (modified CEB90) (cs-hung.rmd)

The Variable set describing the creep behaviour in accordance with one of the above creep models can be imported using the standard file import function ⇧FILE ⇨IMPORT, selecting

⊙ ASCII Files
⊙ Partial project
☑ Variable

and the file name of the file to import located in the program directory.

The variables that are used in the formulas defining the creep and shrinkage coefficients have to be assigned to the creep parameter PHI(t, t_0,…) and the shrinkage parameter EPS(t, t_0, …) of the material used. The parameters are listed below:

Creep	Shrinkage	Code
AS96cr	AS96sh	AASHTO 1996 based on CEB-FIP 90 (cs-as96.rmd)
BS54cr	BS54sh	BS5400 Part 4 1990 (cs-ceb78.rmd)
HS54cr	HS54sh	Hong Kong Standard (cs-hs54.rmd)
C78cr	C78sh	CEB/FIP Model Code 78 (cs-ceb78.rmd)
C90cr	C90sh	CEB/FIP Model Code 90 (cs-ceb90.rmd)
SM90cr	--	Improved Summation Model (cs-rsm90.rmd)

DI45cr	DI45sh	German Code DIN 1045/1 (2002) (cs-DI45.rmd)
OE47cr	OE47sh	Austrian Code OENORM B4750 (cs-oe47.rmd)
NORcr	NORsh	Norwegian Standard NS 3473 (cs-nor.rm)
HUNGcr	HUNGsh	Hungarian Standard HUNG-UT 2/13 (cs-hung.rmd)

Note: _The above sets of variables contain many "intermediate variables" used for the calculation of the final variables representing the creep and shrinkage coefficients. All these variables have the prefix C78, C90, SM90,B54,DI45,H54, NOR or OE47 respectively. Variable names with this prefix should not be used for other purposes, to avoid the destruction of some of the imported variables._

The assignment of the creep variables and the appropriate material parameters is done in ⇧PROPERTIES ⇨MATERIAL by specifying the variable names for PHI(t) and EPS(t) respectively.

7.4.3.2 Program internal Standard Creep Models

The 10 creep models listed above are directly integrated in the program. They can therefore be used without importing the appropriate variable set. Using these integrated creep models has indeed great advantages, especially with respect to computing time consumption. The actual creep and shrinkage coefficients have to be calculated very often in the time stepping procedure for analysing creep and shrinkage in the different construction stages. Avoiding the overhead of evaluating large variable sets again and again leads to considerable computing time savings.

The option

☑ **C+S variable calculation by program**

has to be set in ⇧FILE ⇨OPTIMISE, if the calculation of the creep and shrinkage coefficients should be performed internally without evaluating the appropriate variable set.

Changing this optimisation option requires exiting the program and starting it anew in order to activate the change (see also chap. 2.7.1 – Optimising the Calculation Performance).

7.4.4 Parameters for Modelling Creep & Shrinkage

A set of material parameters and structural system variables must be input for the creep and shrinkage models. The material parameters are defined in ⇧PROPERTIES ⇨MATERIAL and assigned internally to the variables which are used in the formulas describing the model. The system parameters are directly assigned to the variables in ⇧PROPERTIES ⇨MATERIAL.
The various models defined in different design codes use different parameters to define the creep and shrinkage coefficients. The dependencies of the different models for the variables are listed below :

7.4.4.1 Material parameters

Emodl (E28)	Modulus of Elasticity
Fc28	Design value for the concrete compressive strength.
CF	Fresh concrete consistency parameter.
ZF	Cement hardening parameter.
CECO	Cement content (Cement weight per concrete volume)
WCR	Water/cement ratio

<u>Emodl - Modulus of Elasticity</u>

The modulus of elasticity might be used in some models for the description of the C&S behaviour, but it is not used in the models provided in *RM2000*.
The modulus of elasticity is only used for the definition of a time dependency of the elasticity modulus. It is assigned internally to the variable E28 to be used for further user defined variable definitions.

Usage and dependencies for EModl (E28) in the standard creep models:

Modell	Creep	Shrinkage	Emod(t)
AS96	-	-	-
BS5400, HS5400	-	-	-
CEB78, CEB90	-	-	+
DIN1045, OE4750	-	-	-
RSM90	-	-	-
Norway	-	-	+
Hungarian	-	-	-

(-) .... not used (+) used

Fc28 - Design value of concrete compressive strength

The design value Fc28 of the concrete compressive strength is defined in most design codes as the compressive strength of a concrete cylinder at an age of 28 days. This parameter is assigned internally to the variable Fc28 to be used for further user defined variable definitions.

Usage and dependencies for Fc28 in the standard creep models:

Modell	Creep	Shrinkage	Emod(t)
AS96, CEB78	-	-	-
BS5400, HS5400	-	-	-
CEB90	+	+	+
DIN1045, OEN4750	+	+	-
RSM90, Norway	+	+	-
Hungarian	-	-	-

(-) .... not used (+) used

CF –Consistency coefficient of the fresh concrete [-]

The parameter CF characterizes the consistency of the fresh concrete at the time of casting the concrete. Three fresh concrete consistency groups are distinguished in accordance with EN V 206:

CF	= 1	stiff	(small water-cement ratio)
	= 2	plastic	(medium water-cement ratio)
	= 3	semi-fluid	(high water-cement ratio)

The parameter CONC is assigned internally to the variable CF to be used for further user defined variable definitions. This variable is used to determine the necessary coefficients for the evaluation of the actual creep values φ in CEB78 (instead of Fc28 used in CEB90, RSM90 and Norway-Model).

Usage and dependencies for CF:

Modell	Creep	Shrinkage
AS96	-	-
BS5400, HS5400	-	-
CEB78	+	-
CEB90, RSM90	-	-
DIN1045, OE4750	+	-
Norway, Hungarian	-	-

(-) ... not used (+).....used

Note: *Fractional numbers between 1 and 3 may be entered, the related coefficients will then be determined by interpolation procedures using the table values for 1,2 and 3.*

ZF – Cement hardening parameter [-]

The hardening parameter ZF characterizes the type of cement used for the concrete. Three standard cement quality types are on the market:

ZF	= 1	slowly hardening cement (SL)
	= 2	normal and rapid hardening cement (N, R)
	= 3	rapid hardening high strength cement (RS)

The parameter ZF is assigned internally to the variable ZF to be used for further user defined variable definitions.

Usage and dependencies for ZF in the standard creep models:

Model	Creep	Shrinkage
AS96	-	-
BS5400, HS5400	-	-
CEB78	-	-
CEB90	+	+
RSM90	+	-
DIN1045	+	-
OE4750	+	+
Norway	+	+
Hungarian	-	-

(-).....not used (+).....used

Note: *Fractional numbers between 1 and 3 may be entered, the related coefficients will then be determined by interpolation procedures using the table values for 1, 2 and 3.*

Note: **The use of the relaxation factor is not yet implemented. The time dependent elasticity modulus is not used for the evaluation of φ and ε_{cs} even though the function EMOD(t) is set up in the standard models. This function is only used in the ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION function UPDEMOD, where a stress redistribution analysis due to a changing elasticity modulus is performed (see chap. 7.3).**

CECO – Cement content [Cement weight per concrete volume]

The parameter CECO characterizes the content of cement in the concrete (cement weight as a percentage of the concrete weight).

The parameter CECO is assigned internally to the variable CECO to be used for further user defined variable definitions.

Usage and dependencies for CECO in the standard creep models:

Model	Creep	Shrinkage
AS96	-	-
BS5400, HS5400	+	+
CEB78, CEB90, RSM90	-	-
DIN1045, OE4750	-	-
Norway, Hungarian	+	+

(-).....not used (+).....used

WCR – Water cement ratio [-]

The parameter WCR characterizes the ratio between the content of water (weight related) and the content of cement (CECO) in the concrete.

The parameter WCR is assigned internally to the variable WCR to be used for further user defined variable definitions.

Usage and dependencies for WCR in the standard creep models:

Model	Creep	Shrinkage
AS96	-	-
BS5400, HS5400	+	+
CEB78, CEB90, RSM90	-	-
DIN1045, OE4750	-	-
Norway, Hungarian	+	+

(-).....not used (+).....used

7.4.4.2 System parameters for creep and shrinkage

Other parameters influencing the creep and shrinkage behaviour are related to the following:

- The environment of the construction site, especially the climatic conditions.
- The cross-section geometry and reinforcement content
- The casting time.

All these parameters are **element parameters**, related to the structural elements and constant over the element length. This applies also to the cross-section parameters described below, which are average values related to a characteristic intermediate cross-section if the cross-section is not constant over the element.

Climatic parameters:

RH(%) Relative humidity at the construction site for the element being considered (time independent average value) in [%].
TMP Average global temperature for the element being considered.

The climatic parameters used in RM2000 are the average environmental temperature TMP and the relative humidity RH at the construction site. Although they are generally the same for all elements of a structure, they can be entered in ⇧STRUCTURE ⇨ELEMENT ⇩TIME as element attributes together with the parameters related to casting or activation time respectively. Different values may, for instance, be used for the parts of the structure exposed to the air, for parts beneath the ground surface and for parts under water.

The specified values are, however, constants and may not be specified as time dependent functions taking into account the different seasons during construction time. An approximate approach for considering the season influence could be in forming weighted average values related to the casting season, taking into account the higher creep rate in the first months.

The values RH and TMP are assigned internally to the variables TMP and RH. These variables may also be used for other user defined Variable definitions.

Element (Cross-Section) parameters:

A_x	Cross Section Area
U	Outside perimeter exposed to drying
UIN	Inside perimeter (of a hollow box cross-section)
RPR	Stiffness ratio steel/concrete $((A_s*E_s)/(A_c*E_c))$

These parameters are used for the calculation of the "notional member size" or "notional thickness", which influences the time development of the creep and shrinkage functions which themselves are dependent on the speed of drying.

Note: *They normally do not influence the final value of the C&S coefficients.*

These parameters (except RPR) need not and can not be entered by the user as they are automatically calculated for the cross sections of the elements that can creep. The average values will be taken if the cross sections at the element begin and end differ.

These parameters may however be reset by the user in ⇧PROPERTIES ⇨VARIABLE for the "what if?" checking procedures for studying their influence on the creep and shrinkage curve (see chap. 7.4.5).

<u>RPR – Stiffness ratio [-]</u>

The element parameter RPR (defined in ⇧STRUCTURE ⇨ELEMENT ⇩TIME) characterizes the ratio between the normal force stiffness of the reinforcement and that of the concrete cross-section $((A_s*E_s)/(A_c*E_c))$. It is used in some creep laws (e.g. Hongkong Standard and British Standard) for determining the creep coefficient and/or the shrinkage coefficient.

However, it does normally not affect very much the resultant values of the coefficients, therefore it is mostly sufficient to neglect the influence of the reinforcement (RPR=0), or to enter a rough approximate value.

The parameter RPR is assigned internally to the variable RPR to be used for further user defined variable definitions.

Usage and dependencies for WCR in the standard creep models:

Model	Creep	Shrinkage
AS96	-	-
BS5400, HS5400	+	+
CEB78, CEB90, RSM90	-	-
DIN1045, OE4750	-	-
Norway, Hungarian	+	+

(-).....not used (+).....used

<u>Time parameters:</u>

The below described time parameters are in all creep and shrinkage models used for calculating the values of the creep and shrinkage coefficients.

AGE (Day) Age of the concrete at the element activation time (difference between the time of casting and the time of activation) measured in days.

t_S (Day) Start time for shrinkage relative to the casting time (i.e it is not necessarily relative to the activation time) measured in days.

The time t=0 of the creep model is the time that the concrete for that element is cast (casting time). The creep curve diagram starts at time = 0 (days) even though the element only starts to creep when load is applied to it. – i.e. at activation time – this is when the formwork is removed or the pre-stressing is applied (self weight will act on the concrete plus pre-stressing and construction loading where applicable.)
The parameter AGE is used to locate the casting time on the global time axis. Therefore AGE is the time (in days) on the creep curve before the element activation time $T_{G,A}$.

Some C&S models allow the starting time of shrinkage to be different from the starting time of creep. The starting time for shrinkage is defined by the parameter t_S.
t_S is the age of the concrete when the shrinkage process starts, i.e. shrinkage will start at the time $T_{G,A}$-AGE+t_S =TS (in days).

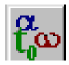
7.4.5 Checking the Time Dependency Coefficients

The mathematical formulas for the calculation of the creep and shrinkage coefficients and the time dependent elasticity modulus in accordance with the creep models described above are very sophisticated and the influence of the different parameters is very much interrelated and hard to estimate.

RM2000 provides a checking tool for viewing the effects on the time dependent values of modifying one or more of the variables. The results of the modifications can be viewed in the form of an output listing or graphically in a time diagram enabling a quick and easy plausibility check for the parameter settings.

7.4.5.1 Checking Variables in ⇧PROPERTIES ⇨VARIABLE

Individual values may be viewed in the function ⇧PROPERTIES ⇨VARIABLE, where resetting some of the variables will immediately show the influence on the other variables depending on them.

A list of all the values of the variables will be displayed on selection of the  - button located at the top right hand corner of the lower input list.
Any of these variables can be modified by selecting it and clicking the ‘Modify’ button. After confirmation of the modification with “Ok” the dependent values of the user defined variables will be immediately changed in the appropriate list.

Note: *The modifications to the variables are only valid locally within the function ⇧PROPERTIES ⇨VARIABLE and do not affect any analysis procedure. The database parameters assigned to these variables are reassigned in any calculation process.*

7.4.5.2 Viewing time diagrams of the coefficients

The time dependent functions (creep curve; shrinkage curve, time dependent elasticity modulus) for each element in the structure can be viewed in ⇧RESULTS ⇨PlCrSh immediately after they have been assigned to the material and after the material has been assigned to the elements. The evaluation of the variables is done automatically without having to select ⇧RECALC (see figure below).

The variables can be changed using the function ⇩SET in ⇧RESULTS ⇨PlCrSh. On confirmation of the modification, the time diagram is automatically redrawn taking the changes to the variables into account.
⇩REDRAW temporarily resets the variables to the new values for viewing.

Note: *The modifications of the variables are, as above, only valid locally and do not affect any analysis procedure.*

The plot of the time diagram is done in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION selecting

- ⊙ Plot Actions
- ➢ Action PlCrSh
- ➢ Description Descriptive text (max. 80 characters)

Note: *The time dependency diagrams should be thoroughly checked to ensure that no errors were made in the definitions.*

7.4.6 Creep Inducing Stress State and Load Case Definition

7.4.6.1 Creep inducing stress state

The creep inducing stress state cannot be explicitly specified by the user. It is calculated implicitly taking into account the results of all Loading Cases applied prior to the end date of the creep interval, and marked as “permanent” Load Cases. Load Cases are marked as “permanent” by setting the “Permanence Code” to “Load” (see chap. 6.4, Load Case).

Note: *It is not easy to calculate shrinkage without creep or creep without shrinkage. Separating the influences requires setting one of the 2 coefficients to zero.*

Most C&S Models, including the TDV models, relate the creep coefficient not only to the age of the concrete, but also to the load application time. This is based on observations and the model assumption, that the total creep strain can be divided in a “delayed elasticity” part, arising shortly after the load application, and a “flow” part that only depends on the age of the concrete.

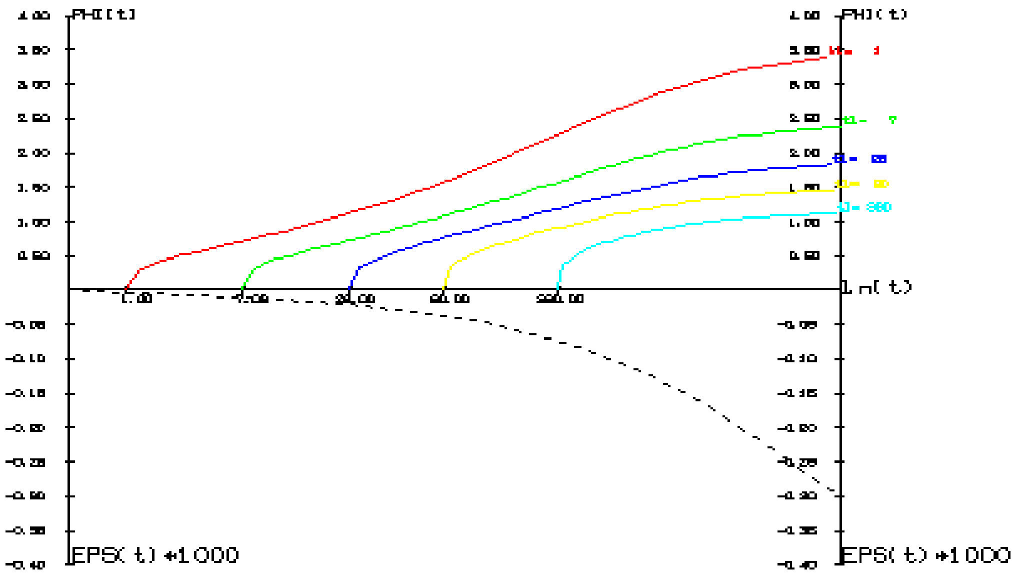


Fig.: Time diagram of creep and shrinkage coefficients

The above diagram shows the creep and shrinkage variation with time for a typical concrete in accordance with the CEB Model Code 90. The different curves for the creep coefficient φ are related to different load application times t_0.

The dependency from the load application time requires, that the creep inducing stress state may not be taken into account as a total state, but must be separated into different parts related to the different Load Cases. The creep coefficient must therefore be separately determined for each creep inducing Load Case, giving partial creep strains for the creep period, which must be superimposed to give the total initial creep strain applied to the structure.

Note: *This procedure is used in all permanent Load Cases applied prior the end time of the particular Creep Period, it is applied in all the previous as well as in the current Construction Stage.*

7.4.6.2 Load Case “Creep&Shrinkage” and Superposition

A Load Case for the results from the calculation of deformations and stress redistributions due to creep and shrinkage for a certain time interval must be defined. This Load Case is generally generated in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE prior to assigning it to the Creep Action in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION, but may also be directly defined when assigning it to the Creep Action. But it must necessarily be generated prior to assigning it to the Creep Action if it should be considered it in the Load Management function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LMANAGE. (Refer to Chap. 6.5, Load Management, for details).

Note: *The Load Case number for the creep & shrinkage loading case can also be assigned directly in the ‘Creep’ input pad (Out1) – but this is not recommended. If this direct assignment is done then this Load Case can **NOT** be considered in the automatic superposition procedure specified in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LMANAGE. The Load Case is however stored and available for all other Load Case manipulations.*

Insert a new Load Case:

- Number — Load Case Number for creep & shrinkage results
- Type — Permanence Code
- Load Info — Load Info Code, used for the load management process
- Description — Descriptive text (max. 80 characters)

Load Case Number
Any number can be chosen for the C&S Load Case (other than those already used for other Load Cases!) However, TDV recommends that the Load Case numbers for the C&S Load Cases should be 601 for the 1st stage, 602 for the 2nd, etc. (see chap. 6.6).

Permanence Code
This code must be set to “Load” in order to mark the Loading Case as permanent. All permanent loads are considered in further stress redistribution in later construction stages or time intervals.

Load Info Code
This code specifies the group defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LMANAGE, to which the Load Case belongs. All the superposition rules prescribed for the Load Cases of this group will then be applied.
TDV recommends that a group named C&S should be defined in the load management function, describing the superposition rules for all C&S Loading Cases.
The Load Info Code will then be C&S.

7.4.7 Creep & Shrinkage Calculation Action

Any calculation with *RM2000* is strictly related to a global time axis (starting at the "birthday" of the first active element and ending at time infinity). All Actions in the analysis process are related to an Action Application Time defining the moment that the Action occurs.
For a Creep Action this is defined as the time at the end of the Creep Period, i.e. the time when the Action is terminated. This end time of the Creep Period is defined in the input by 'Delta-T', characterizing the time interval between the Application Time of the previous Action, and the end time of the Creep Period.

All system activations and Loading Case calculations are in a defined order. The program memorizes the application time for each Loading Case and the activation time for each element. The required definitions are therefore:

- To assign the Creep Action to a Construction Stage
- To place the Creep Action at the right position in the Action Schedule
- The definition of the number of time steps to be used
- The definition of the end time of the creep period by specifying the time interval Delta-T between the previous Action and the Creep Action

The appropriate construction stage is selected in the upper list in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION. The existing Action Schedule of this stage is shown in the lower list of this input pad.

The Creep Action is placed in the Action Schedule by selecting the appropriate position and inserting the new Action using the 'Insert before' or 'Insert after' button, and selecting

- ⊙ Calculation actions
- ➢ Creep — Creep and shrinkage calculation

The number of time steps, the Load Case number, the time difference and some additional parameters have now to be defined:

- ➢ Inp2 — Number of time steps for this creep period.
- ➢ Out1 — Load Case number for the C&S Load Case.

- Out2 — Name of ASCII list file containing the results (if '*' is set, the default name LCxxxx.lst is used, xxxx referring to Out1).
- Delta-T (Days) — Time interval to the end of this creep & shrinkage period.
- Description — Descriptive text (max. 80 characters)

Number of time steps:
The number of time steps specified in the input governs the accuracy of the results as the development of stresses and strains due to C&S variation within a time interval is highly non-linear. A linear development of these stresses is assumed by the program within the time interval in the analysis process and therefore the more time steps used, the more accurate the result of the calculation.

Although the time development is generally non-linear, assuming a linear variation within a time interval for the secondary effects is usually sufficiently accurate. Therefore it is common practice to use only one time step for a time interval, and using more than one creep interval in construction stages where this assumption is not justified.

Note:	**Non-linearity of the creep strain development is not only caused by the non-linear time dependency of the C&S coefficients, but – more important – by major stress changes within the Creep Period. Therefore it is recommended to relate the Creep Periods to such changes, i.e. to apply separate Creep Actions between the Calculation Actions of essential Load Cases.**

Load Case Number:
For details see the previous section "Load Case Creep & Shrinkage".

Note: *The user defined Loading Case Number is used for storing the results for the creep and shrinkage analysis of the particular Creep Period. If the number of time steps is greater than one, automatically numbered Load Cases (starting with number 9001) will be generated, containing the results of the individual steps.*

Duration (Considered C&S Period)
The beginning of the considered C&S Period is principally defined by the end of the previous Creep Action, or – for the very first period – the global time 0.

The input parameter 'Delta-T' specifies the time interval between the previous Action and the end of the actual creep period. 'Delta-T' is added to the Action Application Time of the previous Action, giving the Action Application Time of the Creep Action. Thus, the **end of the Creep Period is** defined as the **Action Application Time** for the Creep Action.

It should be noted, that 'Delta-T' is not necessarily the duration of the Creep Period. This is only the case when all other Actions occur without delay at the end of the previous Creep Action or at time 0. The duration of the Creep Period is the sum of the 'Delta-T's' of all actions from the end of the previous creep action to the end of the current creep action.

Attention:	**Inserting a new Action with a time offset 'Delta-T' before the Creep Action will not automatically reset the 'Delta-T' value of the Creep Action to keep the Creep period unchanged, but 'Delta-T' of the Creep Action will remain unchanged and the end time of the Creep Period will be offset.**

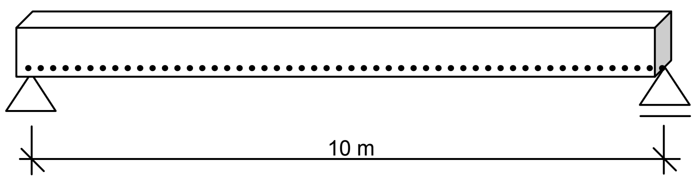
The values 'Delta-T' characterizing the time difference to the previous action, and the Application Time on the global time axis are shown in the Action Schedule Table in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION.

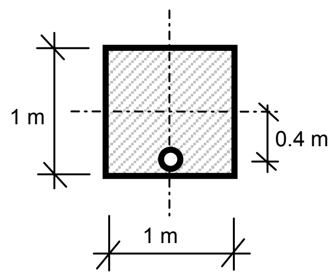
7.4.8 Output Description for LC Creep&Shrinkage

The description of the output listings for creep and shrinkage load cases is made on behalf of a small example.

7.4.8.1 Example – Structural System:

Single span girder consisting of 2 elements (101 and 102) with a total length of 10 m (statically determined support).





A=1m^2
J=1/12=0,0833m^4
E=3.03e7 kN/m^2

The structure is pre-stressed (1 straight tendon).

E=2e8 kN/m^2 A_K=0.0010 m2
Sig_{All}=1e6kN/m^2

This pre-stressing yields a normal force of 1000kN and a bending moment of 400 kNm.

All elements are simultaneously activated in construction stage 1. The age of the concrete at activation time (T=0) is 7 days for element 101 and 14 days for element 2. The creep and shrinkage coefficients are determined in accordance with CEB90. One time step is used for each C&S Load Case.

5 Load Cases are applied in the following sequence:

- LC 101 Self weight 25 kN/m^3 at T=0
- LC 501 Pre-stressing at T=0
- LC 601 Creep and shrinkage from T=0 to T=100 days
- LC 201 Permanent load 20 kN/m^2 applied at T=100 days
- LC 602 Creep and shrinkage from T=100 to T=1100 (over 1000 days)

The creep coefficients (or curves respectively) are - in accordance with CEB90 - dependent on the age of concrete at the load application time t_0. The load application time of the creep loading itself is assumed to be halfway in the time interval (i.e. T=50 days for LC 601, T=600 days for LC 602).

The determination of the creep coefficients is therefore based on the curves for the following concrete ages at load application:

- Element 1: t_0 = 7 days; 57 days; 107 days; 607 days; 1107 days
- Element 2: t_0 = 14 days; 64 days; 114 days; 614 days; 1114 days

The following tables show the creep curves for the above load application times.

CREEP CEB/FIP mod. 90

Loading case 601 - time interval from 0 to 100 (for w2=0.5 => t = 50)
Loading case 101 and 501 applied at time t = 0
Element 101 7 days in scaffolding
Element 102 14 days in scaffolding

A/u	0.25000	RH	75	ZT	2	T	20
lamda	1.5065	h0	500	HF	6750	Fcm	46
		Hunit	100	h0'	5	FCM'	4.6

	El 101	El 102	El 101	El 102
	LC 101 and 501		LC 602	LC 602
T/t0	7	14	57	64
1				
2				
7	0.0000			
14	0.4510	0.0000		
21	0.5542	0.3958		
57	0.8043	0.6759	0.0000	
64	0.8350	0.7059	0.3031	0.0000
107	0.9777	0.8412	0.5405	0.5062
114	0.9960	0.8582	0.5611	0.5286
607	1.5089	1.3214	0.9967	0.9723
614	1.5123	1.3244	0.9993	0.9749
1107	1.6759	1.4696	1.1183	1.0926
1114	1.6775	1.4710	1.1194	1.0937

RM Variables	
t0	14
t	114
c90t0	14
c90frh	1.3178
c90frt	1.0000
C90frht	1.3178
c90bf	2.4711
c90bt0	0.5570
c90bh	1112.57
c90bht	1111.96
c90bc	0.4731
c90f0	1.8140
PHI	0.8582

Loading case 602 - time interval from 100 to 1100 (for w2=0.5 => t = 600)
Loading case 101 and 501 applied at time t = 0
Loading case 601 applied at time 50
Loading case 201 applied at time t = 100
Element 101 7 days in scaffolding
Element 102 14 days in scaffolding

A/u	0.25000	RH	75	ZT	2	T	20
lamda	1.5065	h0	500	HF	6750	Fcm	46
		Hunit	100	h0'	5	FCM'	4.6

	El 101	El 102	El 101	El 102	El 101	El 102	El 101	El 102
	LC 101 and 501		LC 601	LC 601	LC 201	LC 201	LC 602	LC 602
T/t0	7	14	57	64	107	114	607	614
1								
2								
7	0.0000							
14	0.4510	0.0000						
21	0.5542	0.3958						
57	0.8043	0.6759	0.0000					
64	0.8350	0.7059	0.3031	0.0000				
107	0.9777	0.8412	0.5405	0.5062	0.0000			
114	0.9960	0.8582	0.5611	0.5286	0.2686	0.0000		
607	1.5089	1.3214	0.9967	0.9723	0.8662	0.8532	0.0000	
614	1.5123	1.3244	0.9993	0.9749	0.8687	0.8557	0.1919	0.0000
1107	1.6759	1.4696	1.1183	1.0926	0.9834	0.9704	0.6190	0.6158
1114	1.6775	1.4710	1.1194	1.0937	0.9845	0.9715	0.6208	0.6177
2000	1.8092	1.5875	1.2125	1.1855	1.0714	1.0580	0.7375	0.7354
5000	1.9457	1.7077	1.3068	1.2780	1.1574	1.1432	0.8219	0.8200
delta	0.6982	0.6128	0.5778	0.5651				

7.4.8.2 Description of the Output Listing of LC 602

A detailed protocol of the creep, shrinkage and relaxation behaviour is given directly in the output listing of LC 602 because only 1 time step has been used. If more time steps had been used then this information were given in the output listings of the partial load cases generated automatically for each time step (e.g. creep with 3 time steps produces 3 partial load cases numbered 9001, 9002 und 9003).

In order to get this detailed information in the output listing it is further necessary to activate the option „Output of Creep and Shrinkage Coefficients“ in ⇧RECALC – SPECIAL SETTINGS.

The total output listing of a C&S Load Case consists of 11 tables. The first 3 tables are specific for C&S cases (different from the output listing of other Load Cases). These 3 tables are therfore described in detail. The 4th table contains the load integrals. These are always zero for C&S cases.

The internal forces are output in 2 separate tables, containing
1. the primary part of the internal forces and
2. the total internal forces due to creep and shrinkage

The „primary part" means the internal forces due to internal redistribution at cross-section level (stress redistribution from concrete to steel - or vice versa in the relaxation case). The total forces contain additionally the redistribution forces due to external constraints. Primary and total internal forces are identical in this example, because the secondary part is zero (no external constraints exist).

Parameters for describing the system behaviour (modification of the left-hand side for considering the stress changes within the creep period)

```
                          LOAD CASE :  602

  TIME INTERVAL         100.000     600.000    1100.000

   ELEM       CREEP   SHRINKAGE   RELAXATION       E-MOD
  -----------------------------------------------------
    101   0.6190307 -0.27385E-04   0.0000000 0.33659E+08
    102   0.6176510 -0.27239E-04   0.0000000 0.33661E+08
```

Table 1: Parameters for the system behaviour

TIME INTERVAL

1st value	T_A Start time of the creep period
2nd value	T_M Integration point in the time interval (mostly centre point)
3rd value	T_E End time of the creep period

ELEM	Element number
CREEP	Creep coefficient for stresses arising in the creep period due to creep and shrinkage ($\varphi(t_E)$ - $\varphi(t_M)$)
SHRINKAGE	Shrinkage coefficient for the time interval T_M to T_E (only for information – not used in the solution algorithm)
RELAXATION	Relaxation coefficient for the time interval T_M to T_E (actually not used)
E-MOD	Modulus of elasticity at the end time of the creep period (not used for the static analysis, used in some creep laws for determining the creep coefficient φ.

Hints:
The described C&S Action starts at T_A=100, lasts for 1000 days and ends at T_E=1100. The line „TIME INTERVAL" shows the start time and the end time of the creep period as well as the fictitious application time of the stress redistributions due to creep and shrinkage.

This time is specified via the ratio value w2. This value is by default set to 0.5 (centre of the interval). This setting has not been changed in the example, therefor $T_M = (T_A+(T_E-T_A)*w2 = 100+(1100-100)*0,5 = 600$.

The table below this header line contains the parameters (creep coefficient, shrinkage coefficient, relaxation coefficient and modulus of elasticity) for each element for the time interval from T=600 to T=1100 based on the curve for the load application time T=600.

Attention must be paid - when checking the relationships - to the fact, that the concrete age at time T=0 has to be considered for the evaluation of the creep formulas. I.e. the creep curves for t_0 = 7 or 14, 107 or 114, 607 or 614 respectively are to be considered. The modulus of elasticity is given for the age t=1107 or 1114 respectively.

Parameters for building the load vector (right-hand side of the equation system)

```
                              LOAD CASE :  602

  TIME INTERVAL                              100.000    1100.000

   TIME      LC  ELEM       CREEP    SHRINKAGE  RELAXATION       E-MOD
  -------------------------------------------------------------------
     0.0    101   101   0.6981721 -0.76687E-04   0.0000000 0.14509E+07
     0.0    501   101   0.6981721 -0.76687E-04   0.0000000 0.14509E+07
    50.0    601   101   0.5778032 -0.76687E-04   0.0000000 0.14509E+07
   100.0    201   101   0.9834273 -0.76687E-04   0.0000000 0.14509E+07
     0.0    101   102   0.6128284 -0.75817E-04   0.0000000 0.13887E+07
     0.0    501   102   0.6128284 -0.75817E-04   0.0000000 0.13887E+07
    50.0    601   102   0.5651234 -0.75817E-04   0.0000000 0.13887E+07
   100.0    201   102   0.9715044 -0.75817E-04   0.0000000 0.13887E+07
```

Table 2: Parameters for building the load vector (right-hand side)

TIME INTERVAL
1st value — T_A Start time of the creep period
2nd value — T_E End time of the creep period

TIME	Application time of the Load Case. This time is used to calculate the age of the concrete at loading application. Must creep laws define the development of the creep strains to be dependent on that age at the load application time. The creep curve related to that Load Case may then be determined using this age.
LC	Load Case No. (of the creep inducing load)
ELEM	Element no.
CREEP	Creep coefficient for the considered time interval $\varphi(t_E) - \varphi(t_A)$
SHRINKAGE	Shrinkage strain for the time interval T_A to T_E. The shrinkage strain is not load case dependent and considered only once for each element, although ist is printed in every line of the table.
RELAXATION	Relaxation coefficient for the time interval T_A to T_E (actually not impemented)
E-MOD	**Change** of the elasticity modulus in the time interval T_A bis T_E

Hints:
The described C&S Load Case starts at T_A =100 days, lasts 1000 days and ends at T_E=1100. This is shown in the header line „TIME INTERVAL". The table below contains for every element the creep inducing Load Cases with their load application times (on the global time axis of the constrution schedule). The next rows contain the related creep coefficients and shrinkage strains. It can be seen that the first C&S Load Case is applied at time 50 (because w2=0.5 and the duration of the Load Case ist 100 days). The Load Case 201 is applied at time 100. The last column contains the change of the elastic modulus in this time interval (E(t=1107)-E(t=107) or E(t=1114)-E(t=114) respectively).

ITERATION PROTOCOL

```
                            LOAD CASE :  602

                                          RELAXATION FACTOR  1.000
 ITERATION PROTOCOL

 ITER      PROZ[%]        RSUM        RMAX        XMAX       XVMAX
 -----------------------------------------------------------------
    1     100.000%   4162.5388 2982.246820      0.0000    0.000000
    2       0.000%      0.0000    0.000000     -0.0010   -0.000966
    3       0.000%      0.0000    0.000000     -0.0010    0.000000
```

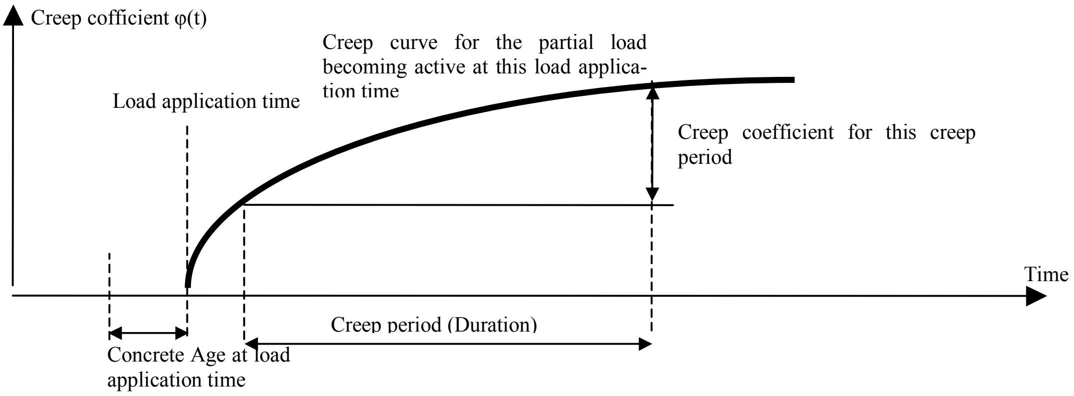
Table 3:
ITERATION PROTOCOL

The iteration protocol shows the development of basic iteration parameters during the iteration process. An iterative solution is required because the redistribution forces evolve again a priori unknown creep strains. Only internal redistribution occurs in the above example, only few iterations are therefore necessary. If there were considerable constraint internal forces, the number of required iterations would be much higher.

The following equation shows the implicit relationship requiring an iterative solution. The internal moment due to redistribution M_{602} is dependent on the creep strains induced by M_{602} itself.

$$\frac{\boxed{M_{602}}}{E \cdot J} = \frac{M_{101}}{E \cdot J} \cdot \varphi_{101} + \frac{M_{501}}{E \cdot J} \cdot \varphi_{501} + \frac{M_{601}}{E \cdot J} \cdot \varphi_{601} + \frac{M_{201}}{E \cdot J} \cdot \varphi_{201} + \frac{\boxed{M_{602}}}{E \cdot J} \cdot \varphi_{602}$$

The creep coefficients $\varphi_{101}, \varphi_{501}, \varphi_{601}, \varphi_{201}$ are used in the „Right-hand equation side", but φ_{602} on the „Left-hand side".



7.4.9 "TSTOP" - Interrupt Creep & Shrinkage

The C&S process for specified structural parts can be halted for specified periods using TSTOP.

This "Interrupt C&S function", can be advantageously used in a construction schedule where there is a repetitive nature in the construction –it greatly simplifies the input and analysis process resultant from complicated construction time schedules.

A typical example of the advantageous use of this function is in the analysis of a balanced cantilever bridges, where the different piers and cantilever arms are not erected at the same time. Each pier with its cantilever arms is an independent structure itself until the balanced cantilevers are connected to each other. The limited amount of construction equipment usually available dictates that a contractor builds the balanced cantilevers at different times or at least with staggered time intervals. The cycle of construction, however, is of a repetitive nature and can therefore be simplified (for analysis purposes) using TSTOP.

<u>Example</u>:
Pier 1 is erected during global time = 0 to global time = 100 [days]
Pier 2 is erected during global time = 20 to global time = 120 [days]
Pier 3 is erected during global time = 40 to global time = 140 [days]
The piers are linked at global time 140 [days]

The standard simulation method would require an exact modelling of the time schedule, activating only pier 1 at time 0, pier 1 and 2 at time 20, pier 1,2,and 3 at time 40.
Thus a standard Action Schedule would contain the following:

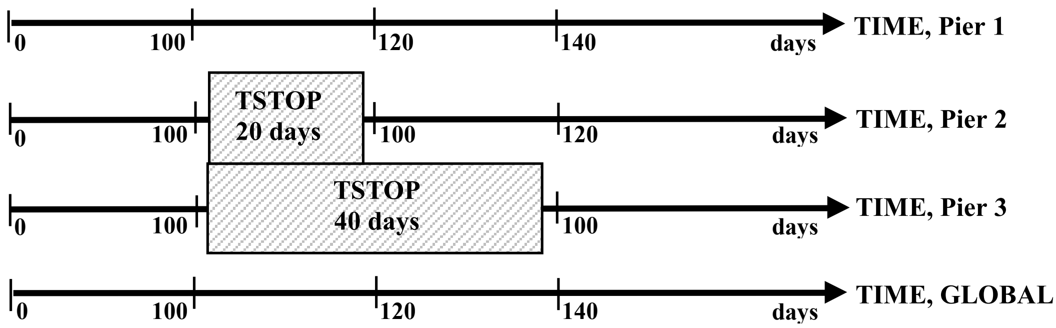
- All actions for pier 1 from time = 0 to time = 20
- All actions for pier 1 and pier 2 from time = 20 to time = 40
- All actions for pier 1, pier 2 and pier 3 from time = 40 to time 140
- All Actions for the combined system from 140 days to time infinity.

This process will yield a large number of Load Cases to be analysed as all the loads on the separate piers must be treated as separate Load Cases (self weight, creep & shrinkage etc). This will also require a big calculation time (being dependent on the number of Load Cases to be calculated).

The alternative process, using TSTOP, reduces the calculation of the erection phase of all 3 separate piers into one construction stage as follows:

- Perform all Actions for pier 1, 2 and 3 for 140 days.
- Interrupt C&S in pier 2 for 20 days and pier 3 for 40 days by applying 'TSTOP' to the corresponding elements at time 120 and 100 respectively.
- Perform all Actions for the combined system from 140 days to infinity.

The 'TSTOP' Action shifts the time axis by 20 days for pier 2 and by 40 days for pier 3.



Note:	**The TSTOP-Action must actually <u>not be applied more than once</u> to each element throughout the whole construction schedule. For different element groups it may however be placed on different positions in the Action Table.**

7.5 Structural Analysis Process (Options and Methods)

7.5.1 Starting the Analysis Process

Once the active structures and all the required actions have been defined for each construction stage, the execution of the complete calculation run may be started by selecting ⇧RECALC.

The ⇧RECALC input pad provides a variety of possible option settings, ranging from the selection of output units to options for partly excluding the performance of certain calculation functions or including special computation features such as taking non-linear behaviour into account. A summary of the available options is given in the next section.

Selecting ⇧RECALC ⇩RECALC after setting the required options will start the calculation process.

All results of the analysis are stored in the Database and may be viewed, printed or plotted using the function ⇧RESULTS after the analysis process has been finished.

7.5.2 Overview over Analysis Options

The options to be set in the ⇧RECALC pad are grouped in 3 sets:

- Calculation (selecting the calculation functions to be performed)
- Calculation type (calculation methods to be used)
- Special settings

7.5.2.1 Function Selection Group

- Cross-section calculation
- Structure check
- Stage activation
- Stage actions
- Influence lines calculation
- Time effects (C&S)
- Plot to plot-file

7.5.2.2 Calculation Options

- Ignore shear deformations
- P-Delta effect — Chapter 7.5.3
- Stay cable non-linear — Chapter 4.5.4
- Large displacements
- Non-linear material properties
- Non-linear springs/dampers
- Accumulate permanent load — Chapter 7.5.4
- Apply construction stage constraints — Chapter 7.5.4
- Accumulate stiffness(SumLC) — Chapter 7.5.4
- Calc. losses for el. compression — Chapter 5.9.1
- Losses ungrouted = grouted — Chapter 5.9.1

7.5.2.3 Special Settings

- Save tendon results (LC) — Chapter 5.9.2
- Save tendon results (Env) — Chapter 5.9.2
- Store slave tend. geom. as 3D points
- Calculate shear area for CS
- Update CS (+ tendon steel area) — Chapter 5.9.3
- Update CS (- duct area) — Chapter 5.9.3
- Update CS (+ fill area) — Chapter 5.9.3
- TDV mode superposition method
- Create primary state due to TempVar — Chapter 6.3.7.2. (ff)
- Print creep and shrinkage factors
- Store partial forces due to creep

7.5.3 P-Delta Effects (2nd Order Non-linear Calculation)

7.5.3.1 Load Case Calculation Considering P-Delta Effects

If P-Delta effects should be considered in the analysis, the option „P-Delta effect“ has to be set active in ⇧RECALC – CALCULATION TYPE. The program will in this case perform an iteration process, starting with a linear solution. The normal forces resulting from this calculation will be used to calculate the 2nd order theory stiffness matrix to be used for the solution of the next iteration step. The iteration process stops when the differences between the computed normal forces and those used for the stiffness matrix calculation are below the tolerance limit.

Due to the non-linearity of the solution the superposition of load cases is theoretically not allowed. The normal forces used for the calculation of the stiffness matrices must therefore represent total states. It is therefore necessary to define an initial state of the normal force distribution in the system, if differential loading cases are investigated. This might be done for single Load Cases by specifying these initial normal forces as a Load Set assigned to the Load Case to be calculated (see chap. 6.3).

Complex normal force distributions however mostly occur in construction stage analyses, which are difficult to describe and which are usually not a priori known to allow a direct specification in the Load Case. RM2000 therefore provides the possibility to take over the initial normal forces from a previously calculated Load Case. The number of this Load Case is specified in ⇧RECALC in the input field “SumLC”. No - or only directly specified - initial normal forces will be taken into account if SumLC=0.

Note that the Load Case no. “SumLC” is unique throughout the whole analysis process. This requires a special treatment of construction stage analyses with considering P-Delta effects, providing througout the construction schedule a summation load case containing the total actual normal forces at any time in the schedule. The next section shows how to proceed in this case.

7.5.3.2 Construction Stage Calculation with P-Delta (non-linear)

For the calculation of Load Cases with considering P-Delta effects it is necessary to use the 2nd order stiffness matrices based on the actual normal forces in the elements (total state – i.e. also the normal forces of all previous permanent Load Cases must be considered). This requires steadily accumulating the relevant Load Cases into one Load Case (for example LC 1000). This Load Case Number must be specified in ⇧RECALC in the input field “SumLC” (e.g. SumLC=1000).

The Calculation Option “Accumulate stiffness (LCSum)” must also be activated.

RM2000 offers 2 possibilities to accumulate Load Cases into one load case (LC 1000 recommended).

a) by directly specifying all required superposition actions
b) by using the Load Management facility

Example for a direct definition:

Initializing Load Case 1000	LCINIT	1000		
Calculating Load Case 101	CALC		101	
Adding LC 101 to LC 1000	LCADD	1000		101
Calculating Load Case 102	CALC		102	
Adding LC 102 to LC 1000	LCADD	1000		102
Calculating Load Case 103	CALC		103	
Adding LC 103 to LC 1000	LCADD	1000		103

etc.

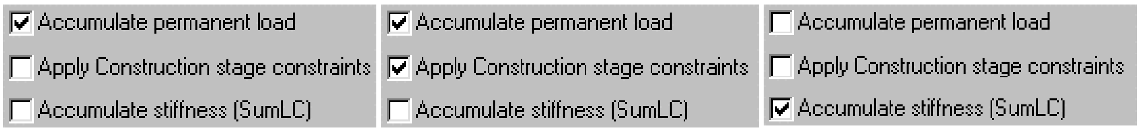
Using the Load Management facility:
This facility allows to define Load Case groups and to specify rules, how to handle the load cases of the different groups in the construction schedule (e.g. automatic accumulation). The Load Management facility is described in detail in chap. 6.6.

If the accumulation Load Case Number is assigned in ⇧RECALC and the "Accumulate stiffness" option is selected, then the program knows the initial normal forces for each load case to be calculated. To calculate in our example the load case 103 (CALC 103) with the P-Delta effects, the program will takes the normal forces from LC 1000 (LC101 + LC102) and the normal forces of the actual load case (LC 103) and starts the stiffness matrix calculation: $K_{II} = (N_{103} + N_{1000})$.

7.5.4 Considering Structural Non-linearity in Stage-wise Analyses

7.5.4.1 Related Options in ⇧RECALC

The well known fact, that the superposition principle is not valid anymore in the case of non-linear analyses, requires to apply special techniques to allow using the standard construction sequence procedure in the case of considering structurally non-linear effect in a RM2000 analysis. Some **options** in the ⇧RECALC window have therefore been provided to activate special program functions which allow an approximate considera-tion of non-linear effects when using the standard procedure with calculating incre-mental Load Cases in the Construction Schedule:



- ***Accumulate permanent load***

Whenever a new Load Case is calculated, not only the Load Sets belonging to this Load Case are considered, but additionally the Load Sets of all previously calculated „perma-nent“ Load Cases (a Load Case is defined to be permanent by selecting the appropriate Permanence Code (Load Type L-I; see chap. 6.4.2)). These Load Cases must also have been accumulated in the “Summation Load Case” specified in the input field “SumLC” of the ⇧RECALC window.
This option allows to use the incremental Load Case definition when a non-linear analy-sis is performed on the final structural system (no system changes in the different con-struction stages. The primary results will be deformations and stresses due to the total load. The accumulated results of the previous permanent Load Cases will be subtracted and the differential state stored as result of the incremental Load Case.

Note, that this option (without additionally selecting the next option) must not be used when the structural system changes throughout the construction sequence (activation and de-activation of elements), because the results will obviously be wrong (an example showing the reason in detail is found in the ensuing subsection).

Note: *This option must not be used by consideration of loads with initial stresses which are only available as results but not as input (e.g.: creep and shrinkage, tendon stressing, composite structures).*

- ***Apply construction stage constraints***

This function is a supplement to the above described option. It must not be used without also selecting „Accumulate permanent load“. Applying this option „Apply construction stage constraints“ together with the option „Accumulate permanent load“ makes the program store additional constraints (initial deformations and stresses) in the internal database, which allow to consider the previous permanent Load Cases as acting on the appropriate previously active system. This allows to apply the option „Accumulate permanent load“ also to construction schedule analyses, where the structural system changes from stage to stage.

Note that the combination of these 2 options must not be applied to these structural systems and loading types:

- Composite structures
- Creep and Shrinkage
- Element removing (DEMO - Element removing)
- Stressing tendons (TENDO – Tendon stressing
  FCAB – Cable /external tendon stressing)

- ***Accumulate stiffness (SumLC)***

This option activates the consideration of the Load Cases accumulated in the "Summation Load Case" SumLC for computing the stiffness matrix of the structural system. I.e. if the option "P-Delta" is set, then the normal forces of SumLC are used in addition to those of the actual Load Case for computing the initial stress matrix K_σ; if the option "Large displacements" is set, then the deformations of SumLC are used for evaluating the large displacement matrix K_L in addition to those of the actual Load Case.

Note: *The Summation Load Case SumLC is advantageously created by using the Load Management function (see chap. 6.6).*

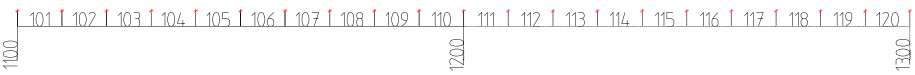
The restrictions with respect to applicability, listed above for the option „Apply construction stage constraints“, do not hold for this option. This makes it obvious to prefer using this option in most cases instead of the other options.

Summary of TDV's recommendations for using these options:
„Accumulate permanent loads“
Non-linear calculation on the final system (no element activation or de-activation while calculating the different Load Cases
„Accumulate permanent loads“ + „Apply constuction stage constraints"
Non-linear construction schedule calculation (only few element activations and de-activations while calculating the Load Cases, no creep and shrinkage, no prestressing tendons, no composite cross-section
„Accumulate Stiffness (SumLC)“
General non-linear construction stage calculation

Note: *The simultaneous application of all 3 functions will in a general case give wrong results and is therefore not allowed (the stiffness governing parameters will be considered twice, explicitly as accumulated initial values and implicitly values of the actual Load Case).*

7.5.4.2 Example – Linear Construction Stage Calculation

The following small example shows the problems arising when using these options in the context of changing structural systems in the construction schedule. The basic considerations are made on a system, where the non-lineariy does not influence the results. The example is a simple 2-span superstructure with equal spans and uniformly distributed vertical self-weight loading.



In a 1st construction stage the first span is built and activated (the self weight is applied). The result will be the standard bending moment distribution with zero values at both ends and the maximum value $q*l^2/8$ in the middle of the first span.

In the 2nd stage, the 2nd span is built and the self-weight activated, acting now on the total system and producing therefore a negative clamped end moment above the central support. The actual bending moment distribution of the different cases is shown in the ensuing figure table. The option settings have obviously no influence on the results of the 1st load case. All calculations give the same standard distribution in the left span.

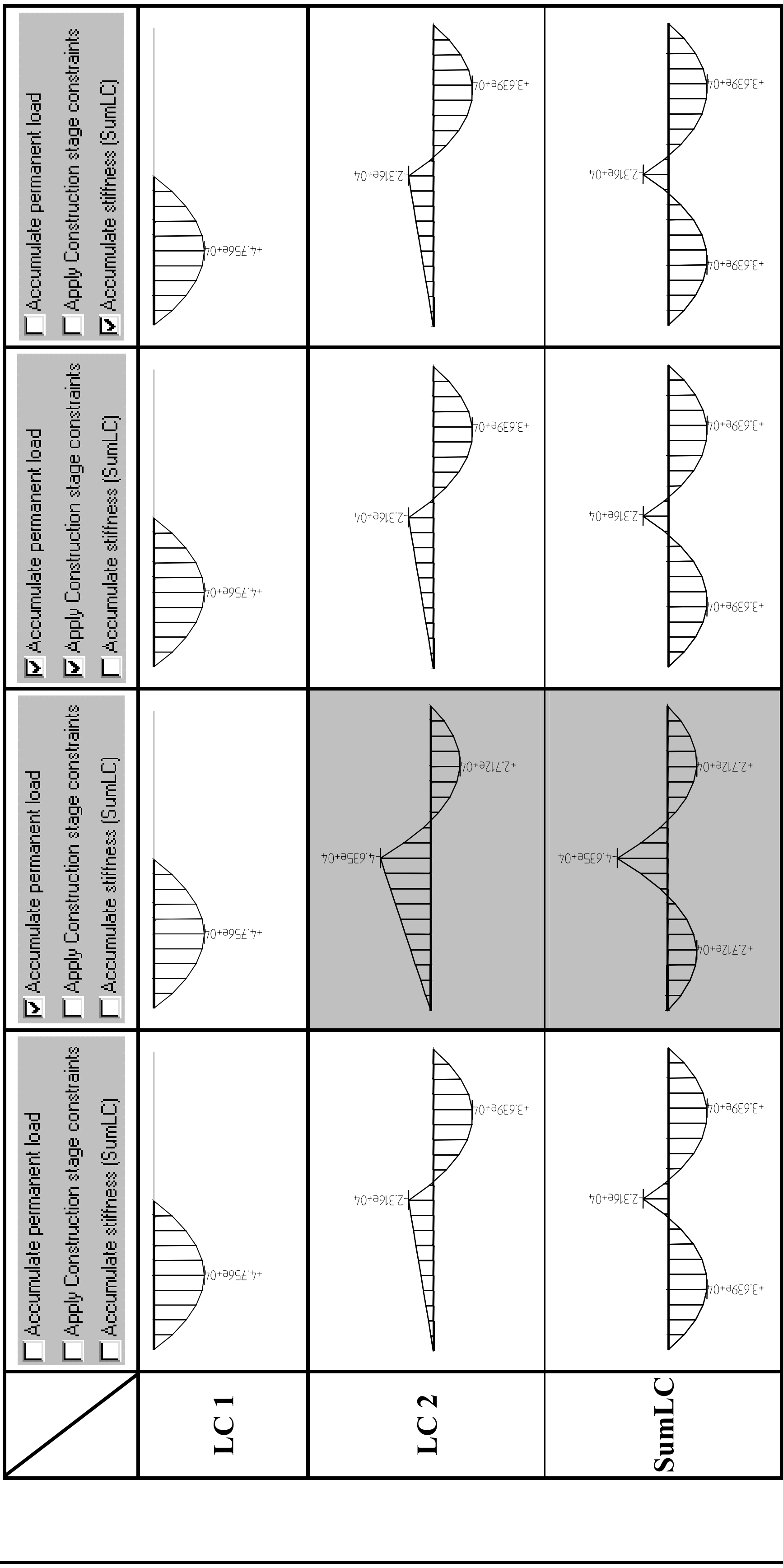
The self-weight of the 2nd span is applied as Load Case 2 on the total system. The result of this Load Case is then combined with LC 1 and stored in SumLC (e.g. LC 1000). Without selecting one of these options the results in SumLC will obviously be the sum of LC 1 (calculated on the partial system) and LC 2 (calculated on the total system).

If the option **„Accumulate permanent loads“** is used, then the total loads of both load cases are used for coputing LC 2 on the total system. The negative moment above the central support will be higher than in the previous analysis (considering the loading of the left span implies a symmetry condition equivalent to a rigid clamping condition for the 2nd span, whereas previously an elastic clamping condition was effective).

This primary result is assumed to be the total result and stored in SumLC (i.e. SumLC will contain the results for a uniformly over the whole length loaded 2-span girder). The results of the previous SumLC are then subtracted from those of the new SumLC and stored in LC2. It is obvious, that the results is different to those without the option.

If **„Apply construction stage constraints“** is additionally selected, then additional constraint conditions are stored, making the loads of Load Case 1 included in the total loading acting on the structural system of the first construction stage. This allows using „total loads“ also for construction stages with changing structural systems.

Note: *For several special structural model conditions (see above), the necessary constraints cannot be fully formulated, therefore this option is not generally applicable!*


Accumulate permanent load
Apply Construction stage constraints
Accumulate stiffness (SumLC)
LC 1
LC 2
SumLC
+4.756e+04
+3.639e+04
-2.316e+04
+2.712e+04
-4.635e+04

If selecting the option **„Accumulate stiffness (SumLF)“,** then the accumulated results of the previous load cases are not included in the total load vector, but directly considered (additionally to the iteration values of the actual load case) in the evaluation process of the non-linear stiffness matrix of the system (i.e. the stiffness matrix based on the initial normal force and displacement state of SumLC is used for calculating the new Load Case. Obviously, this option has no influence in the case of a linear analysis.

7.5.4.3 Example – Non-linear Construction Stage Calculation

In order to get an essential influence of the non-linear behaviour, the above example is modified. A (large) normal force is additionally applied in the 1st construction stage, and the system is calculated with considering P-Delta-effects (2nd order theory). The normal force is applied in a separate initial Load Case 11 to point out the differences.

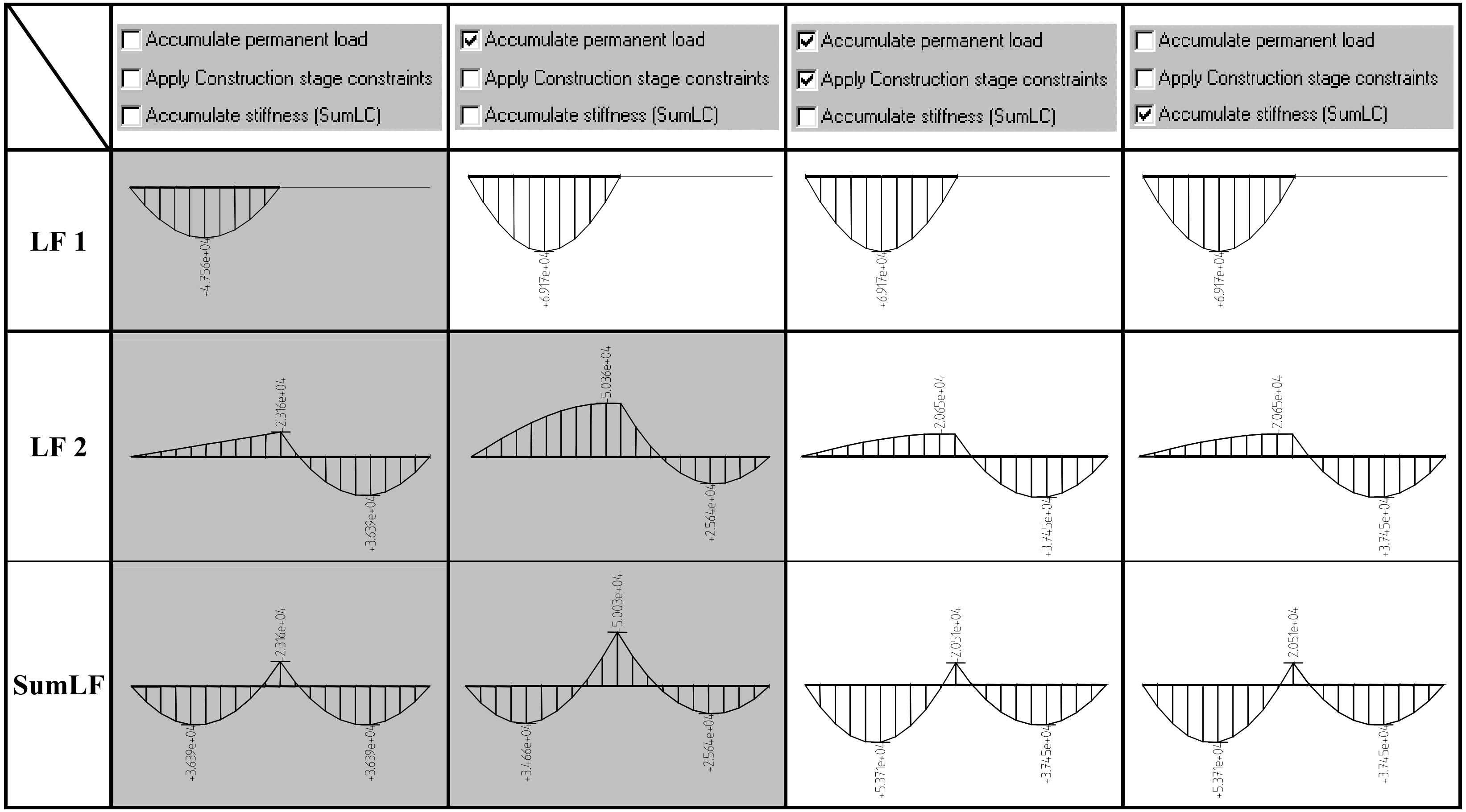
At first, the analysis is performed without selecting any of the described options. We get as results the same values, than in the linear analysis (which are obviously wrong because the P-Delta-effect of the normal force is not considered.

Selecting **„Accumulate permanent loads“** takes into account the normal forces of all previous Load Cases when calculating the subsequent ones. Because the normal force (LC 11) is considered in Load Case 1, the result will be correct. But because the structural system changes in construction stage 2 and the loads of LC11 and LC1 are assumed to act on the new system, the results of LC2 will be wrong (as in the linear analysis), unless the option “Apply construction stage constraints” is also selected.

Using the additional option **„Apply construction stage constraints“** allows performing also non-linear construction schedule analyses in accordance with this method. In our example again the LC 1 will give the correct results, but also LC 2 and SumLF will now get the correct results for the construction stage calculation with system changes and considering P-Delta effects (restrictions see above).

The option **„Accumulate stiffness (SumLF)“** has been developed in order to overcome the shortcomings (restrictions) of the constraint method. As described above, the stiffness matrix based on the initial normal force and displacement state of SumLC is used in this case for calculating the new Load Case. This option is generally applicable.

7.5.4.4 Construction schedule analysis for a non-linear system



8 Design Code Checks

8.1 Fibre Stress Check

8.1.1 General

This chapter shows all the necessary steps for performing a fibre stress check.

The following principal actions are required:
- Definition of the needed material properties for the used materials
- Definition of the cross-section geometry with stress points (points inside the cross-section, where the stress check should be performed)
- Calculation of Load Case(s) and/or creation of superposition file(s) and/or creation of several combination(s)
- The calculation of the fibre stresses in the construction sequence
- Plot of stresses

8.1.2 Material properties

A material has to be created either in ⇧STRUCTURE ⇨ELEM ⇩MAT or (recommended and described in the following) in ⇧PROPERTIES ⇨MATERIAL. The upper table lists all currently available materials.
View the existing material properties by opening the overall material properties window using the button (or by checking the corresponding values individually in the lower table). The appearing window shows all existing properties for the current material. The block titled 'Calcul.-Static' needs to be completed in order to calculate the fibre stresses.

Some additional data is necessary if a graphic for the stresses is required containing the stress limits for the elements. These stress limits can be defined when switching to the corresponding input pad by clicking the arrow symbol at the right of 'FIBRE STRESS CHECK'.
The appearing window allows the definition of 6 stress limits for tension and compression. The two limits for 'General' are the important ones, Grp2 to Grp6 are for special (not implemented) use only. Define the tensile and compressive stress limits (Units!) to have this information available for graphics.

8.1.3 Fibre stress points

It is necessary to specify the positions in the cross section for which the fibre stress check will be done. These points are called stress points.

It is strongly recommended to create the fibre stress points when preparing the structure in *GP2000*.
Existing stress points can be viewed resp. new stress points can be defined in ⇧PROPERTIES ⇨CS ⇩REINF. The upper table shows all existing cross sections, the lower table shows the existing stress points per cross section. Not only stress points but also any reinforcement in the cross section can be specified here. The information about which point we are looking at is to be seen in the first column (titled ‘kw’). A fibre stress points is identified as ‘FIBPOI’.
Existing Fibre stress points can be modified, new ones can be added using the ‘Insert before’ or the ‘Insert after’ buttons.
A new window asking for the definition of a stress point will appear when adding a new point into the lower table.

Select the appropriate type first (REIPSI is default – reinforcement point) by clicking the arrow next to the input filed and select FIBPOI – stress check point. A ‘group’ – definition is only necessary for reinforcement, not for stress points.

Each stress point is defined as an intersection of to lines inside the cross section. Each lines is defined by two nodes of the cross section. A certain distance (Dist/z and Dist/y) as well as a certain angle (Angle) to the defined lines can be specified (Units!)
Hit **`<OK>`** to confirm (or **`<CANCEL>`**).

It is obvious that it is necessary to know the node numbers of the cross section. We either need a plot file of the cross section to do the stress point input, or we define our points interactively. This interactive input is available when clicking on the i at the top of the lower table.
A new window appears showing the cross section and the stress point input. The table at the bottom is handling the input by offering the ‘Insert before’, ‘Insert after’, ‘Modify’ and ‘Delete’ buttons.
When hitting one of the Insert buttons, the stress point input becomes active. The same input rules as described above applies, parallel to the input we can view the cross section where we can also modify the presentation by using the buttons above the cross section graphic.

Example:
- ☑ Select Nod-Numb and modify maybe also the
- ☑ TxtFact in order to read the node numbers in the cross section (or use the Zoom-functions)

Each node input is confirmed by hitting **`<APPLY>.`**
The program is automatically numbering the stress points starting with 1 and incrementing with 1. Memorize the numbers for later use (Plot file)!
The successfully defined points are shown in blue colour in the graphic.
Quit the window when all stress points are defined.

Note: _The most appropriate way to define these reinforcement points is the graphic input in GP2000 where all reinforcement points for all cross sections can be done (here it is individually – reinforcement points per cross section)._

8.1.4 Load Combination to be Checked

Any existing and calculated loading case as well as any existing and calculated superposition file can be used for a fibre stress check.
The resulting listing of the fibre stress check will not contain any specific individual component if an individual result (loading case or superposition file) is being checked.
Example:

Final.sup = LC101 + LC201 + LC501 + traffic.sup

Fibre stresses only for 'Final.sup' will be available, no fibre stresses for LC101, LC201, LC501 or traffic.sup.

In order to get the stresses for all components we need to specify a certain 'combination'.
These combinations are done in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB.
24 possible combinations can be specified here using the appropriate buttons ('Modify', 'Insert after', 'Insert before', 'Copy').
Each loading case resp. superposition file can appear several times, a max. number of 24 combinations can be created. Each loading case or superposition file can have two multiplication factors **(F1 for 'favourable' and F2 for 'unfavourable').**

Attention:	**In case of a 'favourable' or 'unfavourable' combination of for instance' Self weight' + 'traffic' it is essential, that the self weight is added into the combination after the traffic. Otherwise the program will not know which factor to take for the self weight loading case if the current values in the combination are all 'zero'.**

If the original (not factorised) Loading case (or sup. File) <0
and the factorised result (Combination) <0
Then the 'unfavourable factor (F2)' is taken
If the original (not factorised) Loading case (or sup. File) >0
and the factorised result (Combination) >0
Then the 'unfavourable factor (F2)' is taken
If the original (not factorised) Loading case (or sup. File) >0
and the factorised result (Combination) <0

Then the 'favourable factor (F1)' is taken
If the original (not factorised) Loading case (or sup. File) <0
and the factorised result (Combination) >0
Then the 'favourable factor (F1)' is taken

The table we get is interpreted by the program in that way that each column is considered for the selected check (fibre stress check in this case). The table does not suppose that loading cases are existing, but the correct interpretation requires that all defined loading cases in the table as well as all defined superposition files are calculated and available when calculating the fibre stresses.

8.1.5 Fibre Stress Calculation

Go to ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION. All relevant loading cases and superposition files must be available before adding the fibre stress check action into the LOADS AND CONSTR. SCHEDULE.
Select the appropriate stage in the upper table and also the appropriate position in the lower table where all actions for the selected stage are shown. Use either the 'Insert before' or the 'Insert after' button to add a new action into the table.
Two possibilities are now available:

- Stress check for a user defined combination specified in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB.
- Stress check for a specific loading case or superposition file

8.1.5.1 Stress for a user defined combination

Select 'Insert before' or 'Insert after', select

⊙ Envelope actions
and click the line for

- SupComb – Combination for fibre check

The next window is asking for the combination number defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOAD ⇩COMB.

- Command program defined name of action
- Inp1:Combination number Input of the number of an existing combination (1,2,3, 4, …12).
- Inp2: Not active.
- Out1 Outputfile (*.sup)put-file Name of the superposition file, where the results of this combination are prepared for the following fibre stress check.
- Out2 Not active.

- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

The result of this action will be a superposition file used for the fibre stress check in the following.

Select the appropriate line in the lower table for the wanted stage and use 'Insert before' or 'Insert after'. Select

- ⊙ Calculation actions
  and click the line for
- FibChk – Fibre stress check

The appearing window is asking for:

- Command program defined name of action
- Inp1:Input file(*.sup), LC Input of the name of an existing superposition file or the number of a loading case which will be taken for the fibres tress check (The file specified in SUPCOMB before).
- Inp2:Fact1,Fact2(*SIGMA)Stress multiplication factors (Fac1 for max, Fac2 for min. stresses)
- Out1 Not active.
- Out2 Listfile Name of the ASCII list file containing the numeric results. Default name (if '*' is set) is fib[name of *.sup file].lst
- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

8.1.5.2 Stress for Load case or Superposition file

The 'SUBCOMB' action is not necessary, we can immediately switch to the FIBCHK action in the LOADS AND CONSTR. SCHEDULE.

Select the appropriate line in the lower table for the wanted stage and use 'Insert before' or 'Insert after'. Select

- ⊙ Calculation actions and select in the action selection table the action
- FibChk – Fibre stress check

The appearing window is asking for:

- Command program defined name of action
- Inp1:Input file(*.sup), LC Input of the name of an existing superposition file or the number of a loading case which will be taken for the fibres tress check (The file specified in SUPCOMB before).
- Inp2:Fact1,Fact2(*SIGMA) Stress multiplication factors for the **allowable** stresses (Fac1 for max, Fac2 for min. allowable stresses)
- Out1 Not active.

- Out2 Listfile Name of the ASCII list file containing the numeric results. Default name (if '*' is set) is fib[name of *.sup file].lst
- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

8.1.6 Fibre Stress Graphics

The fibre stresses are calculated, we can ask for a graphical presentation of the stresses for certain stress points in the cross sections in ⇧RESULTS ⇨PLSYS.

The important definitions in the ASCII input file are:

- PLSCAL set the scale for stresses
- PLSUP or PLLC select Loading case number or superposition file to be used for the graphic presentation of fibre stresses.
- PLFIBP select the stress point of the cross section
- PLELEM / FIBSUP / MIN or /MAX or
  PLELEM / FIBLC to plot stresses from a loading case or from a superposition file (min and max available).
- PLELEM / FIBMA or
  PLELEM / FIBMI to plot the defined stress limits for tensile and compressive stresses (specified in ⇧PROPERTIES ⇨MATERIAL and FIBRE STRESS CHECK).

8.2 Fibre Stress Check with Cracked Tension Zone (FibII)

8.2.1 General

The check of stresses in state II (cracked tension zone) as e.g. required in OENORM B4750 is done using the same rules as used in FibChk (see above). It is based on the assumption that the concrete behaves linearly elastically in the compression zone and does not bear any stresses in the tension zone.

Un-cracked concrete (state I) may be assumed as long as the average tension strength of the concrete is not exceeded at the tensioned surface of the cross-section. If it is exceeded, then the stresses in the concrete and in the reinforcement are determined using the "working-load method" for the state II (cracked tension zone).

In order to perform such an analysis it is necessary to do first a standard fibre stress check (FibChk). This check shows, whether the limit stresses of the material (tension and compression) are exceeded. No check with cracked tension zone (FibII) is required if the limit stress (stress limit group 1) is not exceeded.

If the limit stresses are exceeded the user has the possibility to change the cross-section or the material and to repeat the analysis. Or he can perform a new fibre stress check with FIBII, considering cracked tension zone. A check with FibII is made after a computation with FibChk in order to learn to know, whether the limits are exceeded or not. The program can not decide, whether a FibII check should be performed.

The stress limit values used in FibII are automatically taken from line 6 of the table defined in ⇧PROPERTIES ⇨MATERIAL Fibre stress check (limit group 6).

8.3 Ultimate Load Carrying Capacity Check

8.3.1 General

The principles and method of input preparation for the ultimate moment check calculations for pre-stressed concrete beams using *RM2000* is described below:

The following must be specified for the ultimate moment check:

- Material properties relevant to the ultimate moment check (σ-ε diagram)
- Reinforcement Groups
- The position of the reinforcement in the Cross-section
- Structure\Element\reinforcement – define reinforcement areas for the elements.
- The factored loading combination output files to be used in the check.
- The call for the ultimate moment capacity calculation in the LOADS AND CONSTR.SCHEDULE.

8.3.2 Ultimate Moment material characteristics

The properties for the materials relevant to the ultimate moment calculation need to be updated – Go to ⇧PROPERTIES ⇨MATERIAL and select the material in the upper table.

Define the necessary material properties by opening the overall material properties window using the button (or by editing the corresponding values individually in the lower table). This needs to be done for all materials used in the calculations!

Select ULTIMATE LOAD CHECK, then select EPS1 - 8 and SIG1 - 8. The stress/ strain values for 8 points can now be defined (be sure of consistent Units!).

Note: It is important that $\varepsilon_I < \varepsilon_{I+1}$ and $\sigma_I < \sigma_{I+1}$

> (The stress and strain values may not be the same as previous values – the maximum stress value for any material may not, for instance, be constant - there must be a slight variation (0.001% or even less is OK!) – numerical computational reasons.)

Example:

Eps-1:	-2.000	Sig-1:	-22500
Eps-2:	-1.667	Sig-2:	-21880
Eps-3:	-1.333	Sig-3:	-20000
Eps-4:	-1.000	Sig-4:	-16880
Eps-5:	-0.667	Sig-5:	-12500
Eps-6:	-0.333	Sig-6:	- 6875
Eps-7:	0.000	Sig-7:	0
Eps-8:	10.000	Sig-10:	1e-6

A reinforcement material must exist before defining the Reinforcement Groups.

8.3.3 Reinforcement Groups

Go to ⇧PROPERTIES ⇨REINGRP and add a new Reinforcement Group (INPUT: name, material, and stress group) by using the ‘Insert before’ and ‘Insert after’ buttons. The definition of a stress group is not essential. This refers to the six possible stress limits that the user can define for different checks for each material (⇧PROPERTIES ⇨MATERIAL, select material in the upper table and hit the [I] button at the top of the upper table. Select FIBRE STRESS CHECK to view and input the stress limits).

8.3.4 Cross-section reinforcement geometry

Go to ⇧PROPERTIES ⇨CS (Cross-Section) ⇩ADDPNT. and select a cross-section and then click on the  at the top of the lower table. The reinforcement geometry that is shown in the interactive window can now be added and/or edited. It is also possible to enter the reinforcement points directly in the lower table without the interactive input facility.

Select the required position in the lower table and choose either ‘Insert before’ or ‘Insert after’ to activate the window for the definition of a new reinforcement point. Choose the type (e.g. REIPSI for a single point) and the group assignment that was defined in ⇧PROPERTIES ⇨ADDGRP.

Every reinforcement point has to be defined relative to two edges. The cross section view can be manipulated at the top of the screen.

- ☑ Select Nod-Numb and modify
- ☑ TxtFact can be used to increase the scale of the displayed node numbers in the cross section (the Zoom-functions can also be used for this)

Define two edges using two points. Node 1 and 2 are two nodes in the cross section the line between them defines the edge. A distance from the edge defined by the two points can also be defined as well as an angle plus a distance (Dist/z and Dist/y resp. Angle – Units!). Hit **<APPLY>** to confirm (or **<CANCEL>**). The defined points will be displayed in the lower table and in the cross-section. Define further reinforcement points or close the window.
A reinforcement point can also be on a node of the cross section.

Note: *The simplest way to define these reinforcement points is using the graphic input in GP2000. All the reinforcement points for each cross section type can be defined at once using GP2000 whereas each reinforcement point must be individually defined in RM2000 for eachr cross section.*

8.3.5 Element– reinforcement

Go to ⇧STRUCTURE ⇨ELEMENT ⇩REINF and add the reinforcement area. New Reinforcement Groups cannot be defined in this window, only the ‘Modify’ button is active. Click the line of an existing Reinforcement Group and the ‘Modify’ button to enter an area for this group (Units!). The assignment of the reinforcement to the elements is done automatically when assigning the cross sections to the elements.

8.3.6 Relevant Combinations

Go to ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB and add the relevant combination of existing loading cases and/or superposition files to be considered for the ultimate moment check by using the appropriate button (‘Modify’, ‘Insert after’, ‘Insert before’, ‘Copy’).
Each loading case as well as each superposition file may appear several times, a max. number of 24 combinations can be created. Each loading case or superposition file can have two multiplication factors **(F1 for ‘favourable’ and F2 for ‘unfavourable’).**

Note:	**It is essential, that the self weight is added into the combination after the traffic when ‘favourable’ or ‘unfavourable’ are being considered. - for instance ‘Self weight’ + ‘traffic’. Otherwise the program will not know which factor to take for the self weight loading case if the current values in the combination are all ‘zero’.**

If the original (un-factored) Loading case (or sup. File) <0
and the factored result (Combination) <0
Then the ‘unfavourable factor (F2)’ is taken
If the original (un-factored) Loading case (or sup. File) >0
and the factored result (Combination) >0
Then the ‘unfavourable factor (F2)’ is taken
If the original (un-factored) Loading case (or sup. File) >0
and the factored result (Combination) <0
Then the ‘favourable factor (F1)’ is taken
If the original (un-factored) Loading case (or sup. File) <0
and the factored result (Combination) >0
Then the ‘favourable factor (F1)’ is taken

The resultant table is interpreted by the program in such a way that each column is considered for the selected check (ultimate moment in this case). The table function does not check that the defined loading cases exist, but the correct interpretation requires that all defined loading cases in the table as well as all defined superposition files are calculated and available when calculating the ultimate moment.

8.3.7 Ultimate Moment calculation

Go to ⇧LOADS AND CONSTR.SCHEDULE STAGE ⇩ACTION. All relevant loading cases and superposition files must be available before adding the ultimate moment action into the LOADS AND CONSTR. SCHEDULE.
Select the appropriate stage in the upper table and also the appropriate position in the lower table where all actions for the selected stage are shown. A new action can be added to the table by using either the 'Insert before' or the 'Insert after' button.

Select
- ⊙ Calculation Actions
  and click the line for
- ULTCHK – ultimate load carrying capacity

- ➢ Command — Program defined name of action
- ➢ Inp1 Input-file — The reference loading case/combination – The number of an existing factored Loading case or Superposition file (e.g. 100 or total.sup) -interactive selection is available via the pull-down menue arrow.

N.B. ***The reference loading case/superposition file is the applied ultimate loading that is to be compared with the Ultimate resistance (Typically the ultimate moment of resistance of the section compared with the applied ultimate moment). If the element being checked is a pre-stressed element then it is essential that the pre-stressing loading case is included in the reference loading case/superposition file. The "SumLC" must be set in the RECALC pad as well as the "save tendon result" box. If the pre-stressing loading case is not included then the "Pre-strain" in the tendons from the pre-stressing action will be ignored in the calculation of the ultimate resistance.***

- ➢ Inp2 UltNxMyMz *(See below for further clarification) — 'Ult' - 'equilibrium of all internal forces'
  'Nx' - 'ultimate load check for normal forces' only
  'My' - 'ultimate load check for bending moment My' only
  'Mz' - 'ultimate load check for bending moment Mz' only
  The program uses 'Ult' if nothing is chosen by the user.
- ➢ Out1 Output-file — Superposition file name for the results - this file can be combined further if desired
- ➢ Out2 List file — Output file name for the Ultimate load check results. (Listing) '*' tells the program to create a list file with the default name.

- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

The output result contains the Ultimate Resistance (typically the Ultimate Moment orf Resistance) of the cross section under the selected ‘reference’ loading case/combination – which should be factored by the ultimate load factors specified in the relevant design code.
The result presentation is in the form of an output listing file (can be edited and viewed) and also a superposition file (user defined name). This superposition file can be graphically compared with the applied ultimate loading (moment or axial force) - create a plot file in ⇧RESULTS ⇨PLSYS.

Possible input variations for UltChk

Ult: *RM2000* calculates the ultimate moment of resistance (using the factored external forces – reference loading - and the material properties) and stores the max/min results in a user defined superposition file (say: ultimate.sup)

Ult<u>Nx</u>: *RM2000* balances the ultimate applied loading (using the factored external forces – reference loading - and material properties) and finds the max/min for <u>Nx</u> (normal force). My and Mz are taken from the reference loading file – say ‘ult1.sup’ and remain ‘frozen’

Ult<u>Mz</u>: *RM2000* balances the ultimate applied loading (using the factored external forces – reference loading - and material properties) and finds the max/min <u>Mz</u> (bending moment). Nx and My are taken from the reference loading file – say ‘ult1.sup’ and remain ‘frozen’.

Ult<u>My</u>: *RM2000* balances the ultimate applied loading (using the factored external forces – reference loading - and material properties) and finds the max/min <u>My</u> (transverse moment). Nx and Mz are taken from the reference loading file – say ‘ult1.sup’ and remain ‘frozen’.

All max/min results (Nx, My, Mz) can be subsequently combined as required.

Examples:

Ult<u>NxMz</u>: *RM2000* balances the ultimate applied loading (using the factored external forces – reference loading - and the material properties) and finds max/min <u>Nx and Mz</u>.
My and Mz are taken from the reference loading file – say ‘ult1.sup’ and remains ‘frozen’during the calculation of the max/min Nx.
My and Nx are taken from the reference loading file – say ‘ult1.sup’ and remains ‘frozen’during the calculation of the max/min Mz.

UltNxMyMz: *RM2000* balances the ultimate applied loading (using the factored external forces – reference loading - and the material properties) and finds max/min Nx, My and Mz.
My and Mz are taken from the reference loading file – say 'ult1.sup' and remains 'frozen'during the calculation of the max/min Nx.
My and Nx are taken from the reference loading file – say 'ult1.sup' and remains 'frozen'during the calculation of the max/min Mz.
Mz and Nx are taken from the reference loading file – say 'ult1.sup' and remains 'frozen'during the calculation of the max/min My.

Typical Results listing for UltChk

```
       **** ELEMENT 104 PNT 1 ULTIMATE STATE (ITER 11) ****

PART  MAT   |         NX         MY         MZ |         AX         EY         EZ
------------+----------------------------------+-----------------------------
104   C_45  |   -5810.28   -5279.88    7799.34 |     9.5114     1.3423    -0.9087
R-B   GRADE_|     400.00       0.00     822.79 |     0.0010    -2.0570     0.0000
R-T   GRADE_|     -16.42      -9.01      22.05 |     0.0010     1.3430    -0.5490
101   PT 1  |    5426.70      -0.35   10354.01 |     0.0096    -1.9080     0.0001
TOTAL ULT   |       0.00   -5289.24   18998.19 |
.................................................................................
PART  MAT   |    EPS-MAX    SIG-MAX         EY |    EPS-MIN    SIG-MIN         EY
------------+----------------------------------+-----------------------------
104   C_45  |    2.82961        0.0    -2.1070 |   -0.15055    -7233.0     1.3930
R-B   GRADE_|    2.78737   400000.0    -2.0570 |    2.76477   400000.0    -2.0570
R-T   GRADE_|   -0.06584   -13167.7     1.3430 |   -0.09836   -19672.0     1.3430
101   PT 1  |    2.65081   565281.2    -1.9080 |    2.65081   565281.2    -1.9080
```

BLOCK 1

```
       **** ELEMENT 104 PNT 1 LOAD CAPACITY MIN-NX (ITER 31) ****

PART  MAT   |         NX         MY         MZ |         AX         EY         EZ
------------+----------------------------------+-----------------------------
104   C_45  |-470568.33    -5268.03   24369.17 |     9.5114     0.0518    -0.0112
R-B   GRADE_|    -268.23      -6.04    -551.74 |     0.0010    -2.0570    -0.0225
R-T   GRADE_|    -389.22     -15.35     522.73 |     0.0010     1.3430    -0.0394
101   PT 1  |   -2799.82       0.18   -5341.98 |     0.0096    -1.9080     0.0001
TOTAL MIN-NX|-474025.59    -5289.24   18998.19 |

PART  MAT   |    EPS-MAX    SIG-MAX         EY |    EPS-MIN    SIG-MIN         EY
------------+----------------------------------+-----------------------------
104   C_45  |   -1.31264   -45421.8    -2.1070 |   -2.00000   -51800.0     1.3930
R-B   GRADE_|   -1.32188  -264376.8    -2.0570 |   -1.36038  -272075.4    -2.0570
R-T   GRADE_|   -1.91840  -383680.3     1.3430 |   -1.97379  -394758.7     1.3430
101   PT 1  |   -1.36764  -291647.7    -1.9080 |   -1.36764  -291647.7    -1.9080
```

BLOCK 2

Each element normally needs at least 2 different materials for the Ultimate load check to function properly (Concrete and reinforcement for instance) because concrete is usually defined as having no tensile strength (only compressive).

The example above has 4 materials: Concrete, Reinforcement Bottom, Reinforcement top & Pre-stressing.

The program first finds the strain condition to exactly balance the applied calculated forces and Moments (factored by the appropriate load factors for the Ultimate State) as stored in the user defined reference file (Combination file (*.sup))

The output listing displays the balanced compressive and tensile forces for the 4 materials, the moment of resistance contribution from each of the materials acting about the Centre of Gravity of the Cross Section and the total Moment of Resistance of the Cross Section for the chosen loading combination (Nx, My, Mz).

Ax	Area of material
Ey	The vertical distance of the force (for the material) from the centre of gravity (CG) of the cross section.
Ez	Transverse distance of the force (for the material) from the centre of gravity (CG) of the cross section.

The lower part of block 1 shows the derived stresses and strains for each material for balancing the applied Ultimate Load.

EPS-MAX	max strain in [per mille]
SIG-MAX	max stress in [force unit / length unit ^{2}]
EY	Eccentricity of max stress/strain (EZ is not printed)
EPS-MIN	min strain in [per mille]
SIG-MIN	min stress in [force unit / length unit ^{2}]
EY	Eccentricity of min stress/strain (EZ is not printed)

Depending on the user defined code in the Ultimate check action in the LOADS AND CONSTR.SCHEDULE (UltMz, UltMy,Mz, UltNxMyMz) the program ‘freeze’s two of the three force components (My and Mz are frozen for min/max Nx) and increases the ‘unfrozen’ force component up to the defined limit (Failure).
If a balanced force can not be reached (the material arrangement and consequently the initial Ultimate State (BLOCK 1) cannot allow an increase of the force component) the program prints ???????? instead of values.

Block 2 in the example above shows the MIN NX evaluation where the forces for MY (-5289.24) and MZ (18998.19) are frozen and NX is changed up to the max. allowable value (-474025.59).
The printed stress/strains are the value for the evaluated forces.

Similar output is printed for all other force components.

8.4 Shear Capacity Check

8.4.1 EUROCODE Shear Capacity Check – OENORM B4750

The shear capacity check is performed *RM2000* in accordance with OENORM B4750 (with references to B4702/B4700), the Austrian version of Eurocode, if OENORM (B4700) is selected as active design code and the action ShChk has been invoked in the construction schedule.

The following section shows the formulas used in *RM2000* for the shear capacity check in accordance with the above mentioned design code and describes the required data preparation.

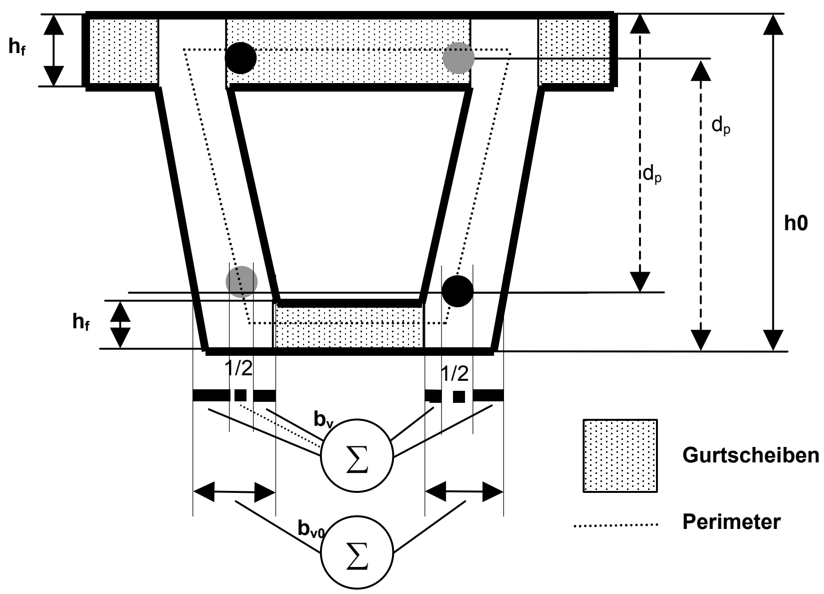
8.4.1.1 Notation

Design values:

Symbol	RM Equivalent	Meaning
V_{Sd}		Design value of the shear force.
V_{Rdc}		Design value of the resistance against shear force, if the strength of the compression diagonals is decisive.
V_{Rds}		Design value of the resistance against shear force, if the strength of the skew shear reinforcement is decisive.
v_{Sd}		Design value of shear flow in flange plates
v_{Rdc}		Design value of the resistance against shear flow between web and flange plates, if the strength of the compression diagonals is decisive.
v_{Rds}		Design value of the resistance against shear flow between web and flange plates, if the strength of the skew shear reinforcement is decisive.
V_{od}	Q_y	Design value of the shear force in the total cross-section due to the decisive loading combination (without direction, positive value).
V_{cd}		Shear force component of the inclined longitudinal compression force in the flange plates.
V_{sd}		Shear force component of an inclined tension flange (conventional reinforcement)
V_{pd}	Q_y(LC pre-stressing)	Shear force component of the inclined pre-stressing tendons (positive if acting in opposite direction to the shear from external loads)
T_{Sd}	M_x	Factorised torsion moment

$\sigma_{cp,eff}$		Compression stress in the centre of gravity
ε_p		Strain in the pre-stressing steel - total
$\varepsilon_p^{(0)}$		Strain in the pre-stressing steel – component due to tensioning
$\varepsilon_p^{(1)}$		Strain in the pre-stressing steel – additional component due to external loading

Geometric data:



Half of the diameter of the ducts is is subtracted from the width, if it is greater than $b_w/8$. Half of the diameter of empty ducts is unconditionally subtracted.

d_{eb} and d_{et} are the distances of the reinforcement and pre-stressing tendons respectively from the opposite cross-section surface.

Symbol	Meaning
d_p	Distance of the tendons from the opposite (compression) surface.
Z	Lever arm; >= $0.9 * d_p$
b_w	Minimum web width
h_f	Cut line width of flange plates
A_k	Perimeter area
u_{kl}	Perimeter length

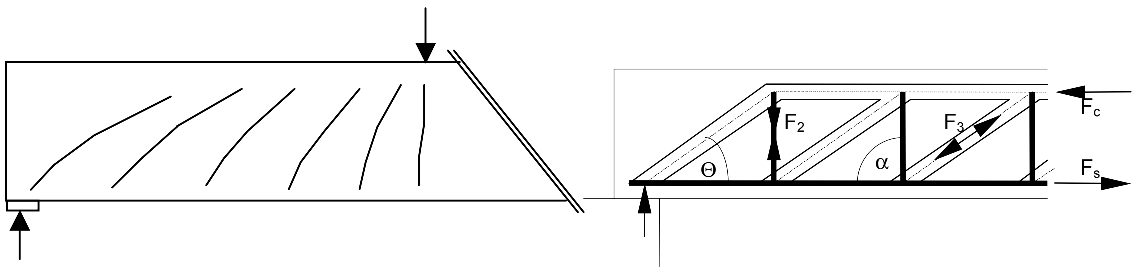
Material data:

Symbol	RM-Equivalent	Meaning

f_{cd}	fc28	Design value of compressive strength of the concrete
f_{yd}	Sigma-*	Limit stress of reinforcement
γ_c	GAMMA	Safety coefficient for concrete
γ_y	GAMMA	Safety coefficient for reinforcement steel

Additional data:

Symbol	Meaning
α	Inclination angle of the stirrups. A fixed value of 90° is assumed in *RM2000*.



Result data:

Symbol	Meaning
V_{Rdc}	Design value for resistance against compression
V_{Rds}	Design value for resistance against tension
β	Inclination of the compression diagonals with respect to the element axis
$A_{sw}(Q_y)$	Vertical shear reinforcement for shear force reception
$A_{sl}(Q_y)$	Longitudinal reinforcement for shear force reception
$A_{sw}(M_x)$	Vertical shear reinforcement for torsion reception
$A_{sl}(M_x)$	Longitudinal reinforcement for torsion reception
$A_{sw}(Q_y+M_t)$	Vertical shear reinforcement for bearing the shear flow due to sum of shear force and torsion

8.4.1.2 Theoretical Background

The validity of following formulas is required:

$$V_{Sd} \leq V_{Rdc}$$
$$V_{Sd} \leq V_{Rds} \qquad \text{(B4750: 48, 49)}$$

The design value of the shear force V_{Sd} is herein calculated with:

$$V_{Sd} = V_{od} - V_{cd} - V_{sd} - V_{pd} \qquad \text{(B4750: 50)}$$

8.4.1.3 Inclination of the compression diagonals

The inclination angle β of the compression diagonals in the concrete may vary within the following limits:

$0.6 <= \tan(\beta) <= 1.0$ bei $\varepsilon_p^{(1)} >= 0.025$ (B4750: 51a)
$0.4 <= \tan(\beta) <= 1.0$ bei $\varepsilon_p^{(1)} <= 0$ (B4750: 51b)
I.e. the angle β may vary between 21,8° or 31° and 45°.

The design value of the resistance against shear force is essentially determined the size of the angle β. The chosen angle β must be used for the design checks for shear force and for torsion.

An additional reduction due to lateral strain effects of the stirrups must be considered in the calculation of V_{Rdc}. This diminution coefficient is called ν and calculated using

$$\nu = 0.7 - \frac{1.5 * f_{cd}}{200} \geq 0.5 \quad \left(f_{cd} \ in \ N/mm^2 \right) \qquad \text{(B4750: 52)}$$

V_{Rdc} is finally calculated using

$$V_{Rdc} = \frac{b_w * z * \nu * f_{cd}}{\cot\beta + \tan\beta} = b_w * z * \nu * f_{cd} * \sin\beta * \cos\beta \qquad \text{(B4750: 53)}$$

The value b_w of the above formula is the minumum web width. This value must be reduced if ther exist ducts in the web whose diameter is greater than 1/8 b_w. Half of the sum of the duct diameters has to be subtacted in this case.

$$b_{w,norm} = b_w - ½ * \Sigma d_h \qquad \text{(B4750: 54)}$$

RM2000 automatically determines the most unfavourable tendon group and makes the subtraction if all tendon groups consist only of one tendon. Otherwise the task is geometrically undeterminate and the user has to enter the amount to be subtracted.

A further reduction of V_{Rdc} is required if the cross-section is under compression (e.g. due to pre-stressing):

$$V_{Rdc,red} = 1.67 * V_{Rdc} * \left(1 + \frac{\sigma_{cp,eff}}{f_{cd}} \right) \leq V_{Rdc} \qquad \text{(B4750: 55)}$$

RM2000 uses the stress in the centre of gravity for $\sigma_{cp,eff}$. Considering the formula 56 of B4750 is therefore obsolete.

The condition $V_{RDC,red} <= V_{SD}$ allows to determine the maximum allowed inclination of the compression diagonal. The interaction condition $V_{SD}/V_{RDC,red} + T_{SD}/T_{RDC} <= 1$ must additionally be fulfilled if torsion exists. All further computations have to be done using this inclination of the compression diagonal.

8.4.1.4 Skew shear reinforcement

The design value of the resistance of the skew shear reinforcement against shear force is:

$$V_{Rds} = \frac{A_{Sw}}{s} * z * f_{yd} * \cot\beta \qquad \text{(B4750: 57)}$$

The required skew shear reinforcement is therefore:

$$A_{Sw} = \frac{V_{Sd}}{z * f_{yd} * \cot\beta} \quad (cm^2/m)$$

8.4.1.5 Minimum reinforcement

The minimum shear reinforcement required by the design code is:

$$A_{Sw,\min} = \frac{15 * f_{ctm}}{f_{yd}} * b_w \qquad \text{(B4700: 62)}$$

This minimum value must be compared with the computed required reinforcement at every investigated cut line.

8.4.1.6 Skew shear reinforcement in the flange plates

Compression flanges (B 4700 pages 37 ff)

Using

$$v_{Sd} = \frac{\Delta F_{Sd}}{\Delta l} \text{ with } \Delta F_{Sd} = \frac{\Delta M}{h} \text{ and } \Delta M = Q * \Delta l$$

the total compression force change will be:

$$\Delta F_{Sd} = \frac{Q * \Delta l}{h}$$

The compression force change of the flange part may then be calculated using:

$$\Delta F_{Sd} = \frac{Q * \Delta l}{h} * \frac{D_{B,gurt}}{D_{B,gesmt}}$$

The design value of the shear force in the flange plate is then determined using:

$$V_{Sd} = \frac{Q}{h} * \frac{D_{B,gurt}}{D_{B,gesmt}}$$

The resistance is sufficient if the following relation is true:

$$V_{Sd} \le V_{Rdc} \qquad \text{(B4700:34)}$$

with

$$V_{Rdc} = \frac{h_f * \nu * f_{cd}}{\cot\beta + \tan\beta} \qquad \text{(B4700:35)}$$

where h_f is the cut width of the flange plate.

A sufficient resistance against compression force change must be established by an appropriate skew shear reinforcement:

$$V_{Sd} \le V_{Rds} \qquad \text{(B4700:36)}$$

with

$$V_{Rds} = A_{sf} * f_{cd} * \cot\beta \qquad \text{(B4700:37)}$$

Tension flanges (B 4700 page 38 paragraph 9)

The checks with the above formulas may be omitted in the design for tension flange plates, if the flange plates only contain structural reinforcement, which is not considered to be required for bearing bending moments and normal forces, and if the cross reinforcement has the same amount than the longitudinal reinforcement (quadratic grid).

8.4.1.7 Longitudinal tension reinforcement due to shear force

The shear force induces a longitudinal force in the tension zone due to the skew compression diagonal. This force is given by

$$F_{tw} = \frac{V_{Sd}}{2} * \cot\beta \qquad \text{(B4750:58 (3))}$$

The required longitudinal tension reinforcement due to shear force is therefore:

$$A_{Sl} = \frac{V_{Sd}}{2 * f_{yd}} * \cot\beta$$

8.4.1.8 Torsion

The basic check relation for torsion is:

$$t_{Sd} \leq t_{Rdc} \qquad \text{(B4700: 50)}$$

$$t_{Sd} \leq t_{Rds} \qquad \text{(B4700: 51)}$$

The shear flow due to torsion along the characteristic surface of the cross-section or the substitute hollow box is calculated using:

$$t_{Sd} = \frac{T_{Sd}}{2 * A_k} \qquad \text{(B4700: 49, B4750: 59)}$$

The design value of the resistance is calculated with:

$$t_{Rdc} = \nu * f_{cd} * d_{ef} * \frac{1}{\cot\beta + \tan\beta} \qquad \text{(B4700: 52)}$$

and

$$t_{Rds} = A_{sw} * f_{yd} * \cot\beta \qquad \text{(B4700: 53)}$$

where d_{ef} is the thickness value of the substitute hollow box cross-section. For solid cross-sections this is 1/6 of the diameter of the maximum circle fit into. For real hollow box cross-sections this is the minimum web or flange plate width.

Equation (B4700: 53) gives the required shear reinforcement for torsion:

$$A_{sw} = \frac{t_{sd}}{f_{yd} * \cot\beta}$$

The required longitudinal reinforcement for torsion is gained using:

$$A_{sl} = \frac{u_{kl} * t_{Sd} * \cot\beta}{f_{yd}} \qquad \text{(B4700: 54,55)}$$

8.4.1.9 Combined shear force and torsion

The following condition must be fulfilled if shear force and torsion act together on the cross-section at the same time:

$$\frac{V_{Sd}}{V_{Rdc}} + \frac{t_{Sd}}{t_{Rdc}} \leq 1 \qquad \text{(B4700: 56, 57)}$$

The shear force and the torsion are always assumed to act together at the same time if they exist. Therefore the above approach is always used for the calculation of β.

8.5 Shear Capacity Check for AASHTO/LRFD Bridge Design Specifications 1998

Shear capacity check in *RM2000* according to AASHTO/LRFD Bridge Design Specifications 1998. This check is done when the AASHTO Standard Code is chosen and the module ShChk is called within the LOADS AND CONSTR.SCHEDULE.

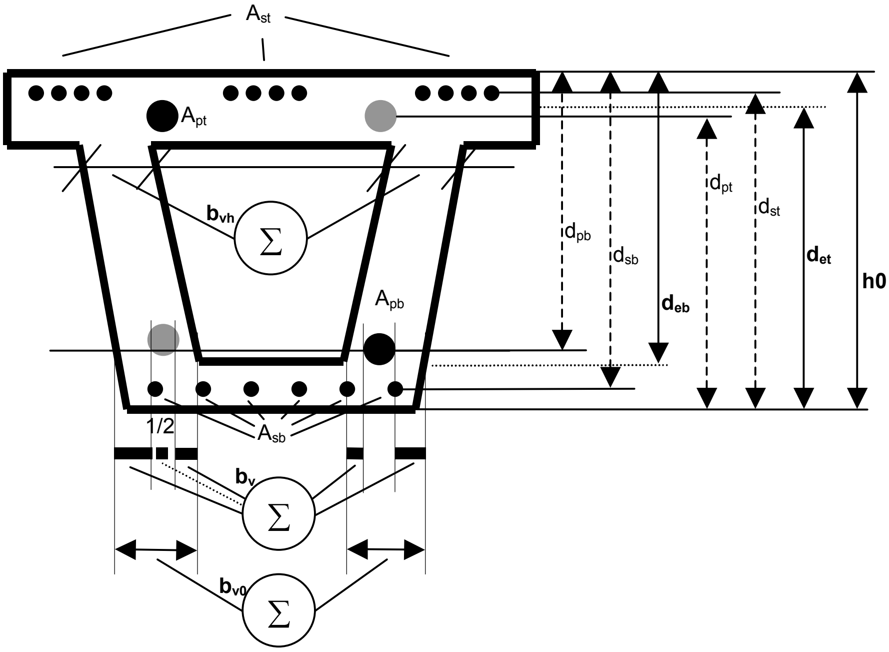
The formulas used in *RM2000* for shear capacity check and the way how to prepare the check are described in this chapter.

8.5.1.1 Notation

Loading data:

Symbol	Unit in AASHTO formula	RM equivalent	Meaning
N_u	N	N_x	Factored axial force (taken as positive if tensile!)
V_u	N	Q_y	Factored shear force (always taken positive)
T_u	Nmm	M_x	Factored torsion moment (always taken positive)
M_u	Nmm	M_z	Factored moment
$A_{ps}*f_{p0}$	N	N_x (prestr. lc)	Component in x–direction of the effective pre-stressing force
V_p	N	Q_y (prestr. lc)	Component in the direction of the applied shear of the effective pre-stressing force; positive if resisting the applied shear
f_{pc}	MPa		stress in concrete at the centroid of the cross-section; 0 if in tension

Geometric data:



If not grouted, the diameter of ducts is subtracted from the web width; half of the diameter is subtracted if grouted.
d_{eb} / d_{et} are the weighted (fy/fp) centroids of steel at bottom/top.

The formulas described below are not fully unit consistent. The used units are described in the tables. But RM2000 uses internally the standard units m, kN, rad.

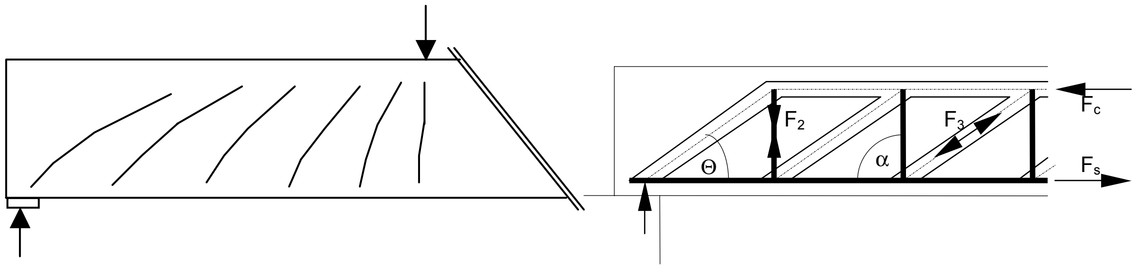
Symbol	Unit in formula	Meaning
A_{c0}	mm²	Total area of the cross section
H_0	mm	Height of the cross section
d_e	mm	Effective depth from the extreme compression fibre to the centroid of the tensile force in the tensile reinforcement (either d_{eb} or d_{et})
d_v	mm	Effective shear depth; the greater of 0.9*de and 0.72*h
b_v	mm	Effective web width taken as the minimum web width minus the diameter of not grouted ducts or one-half of the diameter of grouted ducts
b_{v0}	mm	b_v without correction due to ducts
b_{vh}	mm	Effective web width taken as the minimum web width
A_{cp}	mm²	Total area enclosed by the outside perimeter of concrete cross-section; the area of the perimeter A_{oh} is used in *RM2000*
P_{cp}	mm	Length of the outside perimeter of the concrete cross-section; the length of the perimeter ph is used in *RM2000*
A_p	mm²	Total area of grouted pre-stressed tendons (either A_{pb} or A_{pt})
A_s	mm²	Total area of reinforcement steel (either A_{sb} or A_{st})
isbox		1 if the cross-section is a box-section, 0 otherwise

<u>Material data:</u>
Steel data is summed for all steel parts (reinforcement and tendons) in the tension zone (bottom or top) of the member. The tension zone is defined by the moment Mz.

Symbol	Unit in formula	RM equivalent	Meaning
E_c	MPa	E-modl	Modulus of elasticity of concrete
f'_c	MPa	Sig-p	Compressive strength of concrete
E_p*A_p	N		Sum of E-modl * area (grouted pre-stressing steel)
E_s*A_s	N		Sum of E-modl * area (reinforcement steel)
A_p*f_p	N		Sum of area * yield strength (grouted pre-stressing steel)
A_s*f_y	N		Sum of area * yield strength (reinforcement steel)

Other data:

Symbol	Unit in formula	meaning
φ	1	Resistance factor (predefined 0.85 for shear, 0.95 for moment)
A	1	Angle of inclination of transverse reinforcement to longitudinal axis (90°, predefined)



Result data

Symbol	Unit in formula	meaning
V_n	N	Nominal shear resistance
V_c	N	Nominal shear strength provided by concrete
V_s	N	Nominal shear strength provided by shear reinforcement
T_{cr}	Nmm	Torsion cracking moment
B		Factor indicating ability of diagonally cracked concrete to transmit tension
Q		Angle of inclination of diagonal compressive stresses
$A_v(Q_y)$	mm²/m	Total vertical reinforcement area for shear only
$A_l(Q_y)$	mm²/m	Total longitudinal reinforcement area for shear only
$A_v(M_x)$	mm²/m	Total vertical reinforcement area for torsion only
$A_l(M_x)$	mm²/m	Total longitudinal reinforcement area for torsion only
$A_v(Q_y+M_t)$	mm²/m	Total vertical reinforcement area for shear and torsion
$A_l(Q_y+M_t)$	mm²/m	Total longitudinal reinforcement area for shear and torsion

8.5.1.2 General considerations

Torsion reinforcement is not required, if

$$T_u \le 0.25*\phi*T_{cr} \tag{5.8.2.1-3}$$

If the factored torsion moment is less than on quarter of the pure torsion cracking moment, the reduction in shear capacity can be neglected. The torsion cracking moment is defined by:

$$T_{cr} = 0.328*\sqrt{f_C^{'}}\frac{A_{cp}^2}{p_c}*\sqrt{1+\frac{f_{pc}}{0.328*\sqrt{f_C^{'}}}} \tag{5.8.2.1-4}$$

Shear reinforcement is only required, if

$$V_U \ge 0.5*\phi*(V_C+V_P) \tag{5.8.2.4-1}$$

or if torsion has to be considered. V_c is calculated from (5.8.3.3-3).

8.5.1.3 Sizing of shear reinforcement

Design of cross sections subject to shear shall be based on, $V_u \le \Phi*V_n$, where V_n shall be determined by the lesser of:

$$V_n = 0.25*f_C^{'}*b_v*d_v + V_P \tag{5.8.3.3-2}$$

or

$$V_n = V_C + V_S + V_P \tag{5.8.3.3-1}$$

for which:

$$V_c = 0.083*\beta*\sqrt{f_C^{'}}*b_v*d_v \tag{5.8.3.3-3}$$

and

$$V_S = A_V*f_V*d_V(\cot\Theta+\cot\alpha)*\sin\alpha \tag{5.8.3.3-4}$$

In *RM2000*, α is predefined to 90 degree.

Out of this formula, the necessary reinforcement area per unit length A_v can be determined. If any reinforcement is needed according to (5.8.2.4-1), a minimum reinforcement of

$$A_V = 0.083 * \sqrt{f_C^{'}} \frac{b_V}{f_y} * 1000 \qquad (5.8.2.5\text{-}1)$$

must be provided.

In order to determine the angle of inclination of diagonal compressive stresses Q and, hence, the appropriate factor b, table (5.8.3.4.2-1) is used. The row in this table is selected by the relation of shear stress to f´c. The column is selected by an iterative process computing e_x, which depends on cotQ. The formulas used:

Shear stress in the concrete:

$$v = \frac{V_U - \Phi * V_P}{\Phi * b_V * d_V} \qquad (5.8.3.4.2\text{-}1)$$

The strain in the reinforcement on the flexural tension side of the member:

$$\varepsilon_x = \frac{\frac{M_U}{d_V} + 0.5 * N_U + 0.5 * V_U * \cot\Theta - A_{PS} * f_{po}}{E_S * A_S + E_P * A_{ps}} \leq 0.002 \qquad (5.8.3.4.2\text{-}2)$$

The term A_{ps}*f_{po} in the *RM2000* shear capacity check is approximated by the total normal force due to pre-stressing in the member (N_{xp}).
If the value e_x is negative, it shall be multiplied by the factor F_e taken as

$$F_\varepsilon = \frac{E_S * A_S + E_p * A_{ps}}{E_C * A_C + E_S * A_S + E_p * A_{ps}} \qquad (5.8.3.4.2\text{-}3)$$

The term E_c*A_c is determined by the product of the minimal web width with the cross section height: b_{vh} * h_0.

8.5.1.4 Longitudinal reinforcement

The amount of longitudinal reinforcement A_s depends on the amount of pre-stressing steel and the results in the member:

$$A_S * f_y + A_{ps} * f_{ps} \geq \frac{M_U}{\phi * d_v} + \frac{0.5 * N_U}{\phi} + \cot\Theta * \left(\frac{V_U}{\phi} - 0.5 * V_s - V_p \right) \qquad (5.8.3.5\text{-}1)$$

8.5.1.5 Torsion reinforcement

If torsion has to be considered, torsion reinforcement must be provided to obtain a torsion resistance. T_r must satisfy:

$$T_r = \phi * T_n \geq T_u \qquad (5.8.2.1\text{-}1)$$

where

$$T_n = \frac{2 * A_0 * A_t * f_y * \cot\Theta}{s} \qquad (5.8.3.6.2\text{-}1)$$

s…spacing of stirrups

8.5.1.6 Combined Shear and Torsion

If shear and torsion have to be considered together, the formulas for shear have to be applied with V_u replaced by:

$$V_U = \sqrt{V_U^2 + \left(\frac{0.9 * p_h * T_U}{2 * A_o}\right)^2} \qquad (5.8.3.6.2\text{-}2)$$

The parameters band Q must be calculated using

$$v = \frac{V_U - \phi * V_p}{\phi * b_v * d_v} + \frac{T_U * p_h}{\phi * A_{oh}^2} \qquad (5.8.3.6.2\text{-}3)$$

for box sections, or

$$v = \sqrt{\left(\frac{V_U - \phi * V_p}{\phi * b_v * d_v}\right)^2 + \left(\frac{T_U * p_h}{\phi * A_{oh}^2}\right)^2} \qquad (5.8.3.6.2\text{-}4)$$

for other sections.

The longitudinal reinforcement shall be proportioned to satisfy:

$$A_S * f_y + A_{ps} * f_{ps} \geq \frac{M_U}{\phi * d_v} + \frac{0.5 * N_U}{\phi} + \cot\Theta * \sqrt{\left(\frac{V_U}{\phi} - 0.5 * V_s - V_p\right)^2 + \left(\frac{0.45 * p_h * T_U}{2 * A_o * \phi}\right)^2} \qquad (5.8.3.6.3\text{-}1)$$

As shear and torsion never occur separated from each other, *RM2000* **always** uses the changed formula for V_u and n in order to compute band Q. If there is no torsion moment, these formula automatically become the formulas for shear only.

8.5.2 Preparation of data for the shear capacity check

8.5.2.1 Geometric parameters for the shear capacity check

a) Perimeter (for determining the shear flow due to torsion):

The evaluation of shear stresses due to torsion requires the definition of a **characteristic perimeter.** This perimeter should define the area enclosed by the centreline of exterior closed transverse torsion reinforcement.

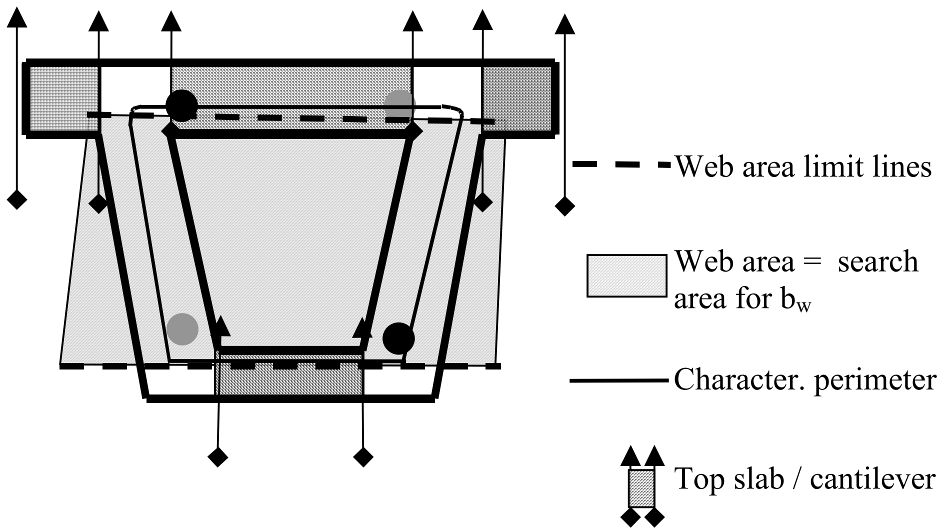
The definition of the perimeter is done in the cross-section definition process (mostly in GP2000). The perimeter is defined as a sequence of polygon points (cross-section reference points (see chap. 3.3)). All points must belong to the same reference point group. This group name is the input parameter for the ShChk module.

b) Border lines limiting the web area:

This group must additionally contain **2 border lines limiting the web area**, where ducts of tendons may be located. This definition is required to enable the program to look for the minimum web width taking into consideration the reduction due to the ducts (see chapters 3.3.8., 8.4.1.3). The angle of the cut through the web can be modified using this line definition.

c) Cut lines between web(s) and top slab:

The definition of the flange plates is also made by pairs of cut lines defined by reference points. 2 cut lines have to be defined for every flange plate. The name of the line (start



point / end point) is used for identifying the cut line in the result listing. All cut line pairs for specifying flange plates must be grouped in a common reference point group, but other than the perimeter point group. This group is also an input parameter for the ShChk module.

The material number of the shear reinforcement is defined by the material assigned to the specified reference point groups.

8.5.2.2 Loading

All load cases and envelopes have to be factorised and combined in accordance with the required design code into one load case or envelope.

All pre-stressing actions have to be separately combined in a pre-stressing load case (e.g. 500).

8.5.2.3 Performing the shear capacity check (ShChk-module)

The module for performing the shear capacity check is started within the construction stage. Input parameters for the action start are:

- Relevant external load envelope name and relevant pre-stressing load case number, separated by comma, in the input field “Inp1” and
- “Perimeter” reference point group name and “Flange plate” reference point group name in the input field “Inp2”.

Example:

External load envelope:	total.sup
Pre-stressing load case:	500
Reference point group for perimeter definition	Shear
Reference point group for top slab definition	Slab

Action selection:

ShChk	total.sup,500	Shear, Slab

8.5.3 Output

The shear capacity check module creates a report file called „sheaxxxx.lst“, where xxxx represents either the load case number of the relevant load case or the first 4 characters of the relevant superposition file. The results are printed element-wise. Elements may be excluded from the shear capacity check – and thus from being presented in the output file - by deactivating the shear capacity check in the the function ⇧STRUCTURE ⇨ELEMENT ⇩REINF (similar to other design checks).

File format:

```
ELEM POS              FC         FY        ACP        PCP          H
        RESULTS:       N         QY         MX         MZ         VP   APS*FP0
      PARAMETER:       B         -B       BETA      THETA         VC       TCR
     REINF. AREA: AV(QY)     AT(MX) AV(QY+MX)     AL(QY) AL(QY+MX)
---------------------------------------------------------------------------
  101 1         51800.00 460000.00      18.44      17.83       3.50
     1:  MINQy -13893.70  -6705.25     653.57       0.00    3729.92 -15456.51
     2:  MAXMy -13814.90  -1320.25   10618.67       0.00    3729.92 -15456.51
     1:   PARA      1.23      0.00       1.72      43.00    3180.87  57358.90
     2:   PARA      1.23      0.00       1.72      43.00    3180.87  57270.28
     2:   PARA WITHOUT TORSION           1.81      42.21    3349.02
     1:  REINF     37.68     (NONE)     37.73
     2:  REINF               (NONE)                          163.93   TOP
     2:  REINF               (NONE)                                   BOTTOM
```

For each element begin and end point, a result block is produced. If no shear reinforcement is needed, the result is (none). If any reinforcement is needed, the first line of output displays important parameters calculated during shear check:

B:	minimum web width where shear check has been done
-B:	with corrected by ducts.
ACP	area of perimeter
PCP	length of perimeter
H	height of cross section

Then, for every result, all corresponding forces (N, Qy, Mx, Mz) and important parameters (BETA, THETA, VC, TCR) used to get the results are displayed. Below the forces and parameters, the reinforcement areas (results) are displayed.

In case of a load-case – shear check, only one line is possible. In case of a shear check based on a superposition file, for every combination shear check is done. The combination resulting in the absolute maximum reinforcement for all reinforcement types (vertical, longitudinal, based on shear only, torsion only or shear and torsion) can be different for different reinforcement types (see example). In this case, all different lines are listed.

In the example above, the element 101 at begin has a minimum web width of 1.23m. The area of the circumference is 18.44m², the length is 17.83m. The height of the cross section is 3.5m.

The combination line MinQy has the following results:

BETA	1.72
THETA	43.00
VC	3180.87
TCR	57358.90

This results in a vertical shear reinforcement need due to shear (AV(QY)) of 37.68* 0.0001 m2/m.

The internal results (BETA, THETA, VC, TCR) can be different for all combination. Therefore, they are listed in the output file for each combination, even when they are the same.

8.6 British Standard BS 5400 1990

The Shear capacity check in accordance with BS5400 Steel, concrete and composite bridges 1990, Part 4 (Concrete bridges) as carried out using RM2000 is described below together with the formulae used and a description of the required data preparation.
This shear check is carried out when the BS 5400 Standard Code is chosen (in the "RECALC" Pad) and the ShChk module, with its appropriate input data, is called in the construction schedule.

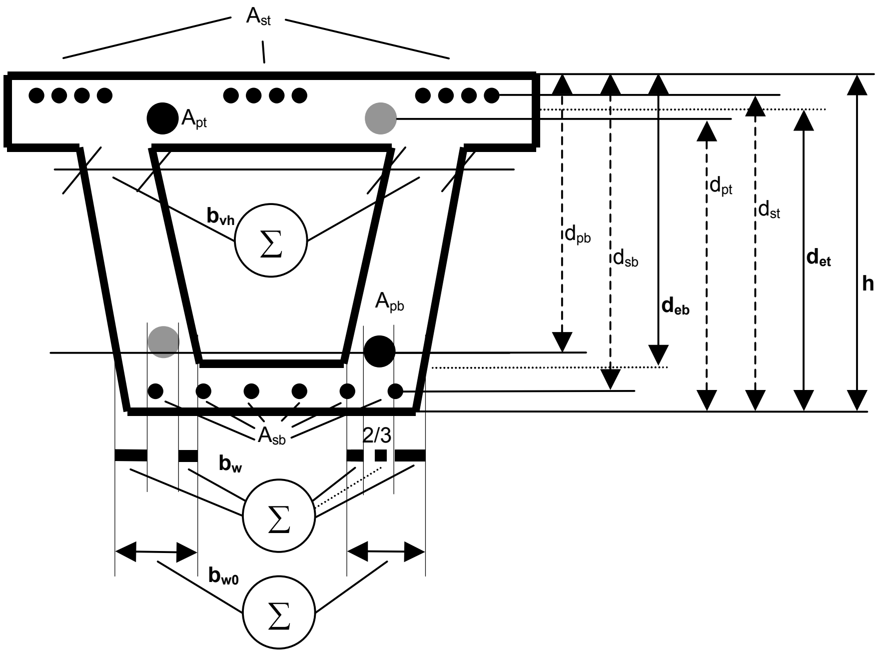
Loading data:

symbol	RM equivalent	meaning
V	Q_y, Q_z	Shear force due to ultimate loads (always taken positive)
T	M_x	Torsion moment due to ultimate loads (always taken positive)
M	M_z, M_y	Bending moment due to ultimate loads (always taken positive)
f_{cp}	(prestr. lc)	Stress at centroid due to total pre-stressing (Primary + Secondary); after all losses * γ_{fL} [taken as positive if in compression]
f_{pt}	(prestr. lc)	Stress at bottom/top fibre due to total pre-stressing (Primary + Secondary); after all losses * γ_{fL} [taken as positive if in compression] at the fibre that is in tension due to external loading.
f_{pe}	(prestr. lc)	Stress in the tendons due to pre-stressing, after all losses * γ_{fL}
f_{pu}		Characteristic strength of the tensioned steel
V_p	Q_y, Q_z (prestr. lc)	Component of the effective pre-stressing force * γ_{fL} in the direction of the applied shear; positive if resisting the applied shear
γ_{fL}		Product of partial load factors γ_{f1} & γ_{f2}

Geometric data:

The sum of the duct diameters in a single line must be subtracted from the total web width if ungrouted; (*2/3 if grouted).

d_{sb} / d_{st} are the distances to the unstressed steel centroids (bottom /top). [User defined input]



symbol	meaning
h	Overall depth of the cross section
d_s	effective depth from the extreme compression fibre to the centroid of the reinforcement in the tension zone (either d_{sb} or d_{st})
d_p	Effective depth from the extreme compression fibre to the centroid of the tendons in the tension zone (either d_{pb} or d_{pt})
d_e	Effective depth from the extreme compression fibre to the centroid of the steel area in the tension zone (either d_{eb} or d_{et})
d_t	Distance from the compression fibre to the longitudinal reinf in the stirrup corner or the distance to the centroid of the tendons (whichever is greater)
b, b_w	Effective web width = minimum web width minus ∑duct diameter (not grouted) or 2/3 duct diameter (grouted) – user defined for tendon groups (see below).
A_0	Area enclosed by the median wall line. The median wall line is user defined in GP2000.
P_0	Perimeter of the area enclosed by the median wall line. The median wall line is user defined in GP2000.
A_p	Total area of grouted pre-stressing tendons in (either A_{pb} or A_{pt})
A_s	Total area of reinforcing steel (either A_{sb} or A_{st})
I	Second moment of area of section (Moment of Inertia)
y_c	Distance from centroid to the fibre under tension due to external loading.(M_{cr})
y_o	Distance from centroid to the centroid of the steel in the tension zone (M_o).

Material data: The steel components (reinforcement and tendons) in the tension zone (bottom or top of member) are considered individually as well as combined. The "tension zone" in this check is defined by the section properties and the sign of moment Mz.

symbol	unit in formula	RM equivalent	meaning
E_c	MPa	E-modl	Short term modulus of elasticity of concrete
f_t	MPa		Maximum principle tensile stress (calculated)
f_{cu}	MPa	Sig-p	Characteristic (28 day) concrete cube strength
E_p*A_p	N		sum of E-mod * area (grouted pre-stressing steel)
E_s*A_s	N		sum of E-mod * area (reinforcing steel)
A_p*f_p	N		sum of area * yield strength (grouted pre-stressing steel)
A_s*f_y	N		sum of area * yield strength (reinforcing steel)
γ_m			Material factor

Result data:

symbol	unit in formula	meaning
V_{co}	N	Ultimate shear resistance of the section un-cracked in flexure
V_{cr}	N	Ultimate shear resistance of the section cracked in flexure
M_{cr}	kNm	Cracking moment at the section considered

M_0	kNm	Moment required to produce zero stress in the concrete at the centroid of the "Total steel area" in the tension zone.
A_{sv}	mm²/m	Total vertical reinforcement area for shear only
A_s	mm²/m	Longitudinal reinforcement area for shear only
A_{st}	mm²/m	Total (both sides) vertical reinforcement area for torsion only
A_{sL}	mm²/m	Longitudinal reinforcement area for torsion only
v	mm²/m	Shear stress due to shear force only
v_c	mm²/m	Allowable ultimate shear stress (Table 8 BS5400)
v_t	mm²/m	Shear stress due to torsion only

8.6.1 BS 5400 (British Standard)

The theory and the required input data for the Ultimate Shear Check in accordance with the British Standard BS5400 are described below.

8.6.1.1 Shear capacity

The ultimate shear resistance of the concrete alone for Class 1 or 2 structures is treated differently from Class 3 structures.

Theoretical background – formulae from BS5400.

SHEAR FORCE

- ***Stress v***

$$v = \frac{V}{b*d}$$

- ***max shear force V_{max}***

In no circumstances should the shear force V due to ultimate loads, exceed the appropriate value given in table 2 multiplied by b*d (table 28 in BS5400)

- ***Cracking moment Mcr***

$$M_{cr} = (0.37*\sqrt{f_{cu}} + f_{pt})\frac{I}{y}$$

- ***Shear force, uncracked section V_{co}***

$$V_{co} = 0.67*b*h*\sqrt{f_t^2 + f_{cp}*f_t}$$

$$f_t = 0.24*\sqrt{f_{cu}}$$ Where f_t is taken as positive

- ***Vertical component of pre-stressing for sections uncracked in flexure***

$$V_u = V_{co} + V_P * \gamma_{fl}$$

- ***Shear force, cracked section V_{cr}***

Shear force, cracked section class 1 and class 2

$$V_{cr} = 0.037 * b_{w*} d * \sqrt{f_{cu}} + \frac{M_{cr}}{M} * V$$

$$V_{cr} \geq 0.1 * b_w * d * \sqrt{f_{cu}}$$

Shear force, cracked section class 3

$$V_{cr} = \left(1 - 0.55 * \frac{f_{pe}}{f_{pu}}\right) * v_c * b_w * d_s + \frac{M_0 * V}{M}$$

$$M_0 = f_{pt} * \frac{I}{y}$$

For cases where both tensioned and untensioned steel are contained in the steel area A_s, f_{pe}/f_{pu} may be taken as:

$$\frac{f_{pe}}{f_{pu}} = \frac{P_t}{A_{s(t)} * f_{pu(t)} + A_{s(u)} * f_{yL(u)}}$$

Where

P_t	is the effective pre-stressing force after all losses
$A_{s(t)}$	is the area of tensioned steel
$A_{s(u)}$	is the area of untensioned steel
$f_{pu(t)}$	is the characteristic strength of the tensioned steel
$f_{yL(u)}$	is the characteristic strength of the untensioned steel

The component of pre-stressing force normal to the longitudinal axis of the member should be ignored for sections cracked in flexure with inclined tendons (i.e. the vertical component of the pre-stressing cables)

- ***Ultimate Shear Resistance of concrete V_c***

$V_c = V_{cr}$ or V_{co} whichever is the smaller

- ***Calculation of the required shear reinforcement (Stirrups).***

$$A_{SV}\ \min = \frac{(0.4 * b\ )}{0.87 * f_{YV}}$$

V > V_c

$$A_{SV}\ = \frac{(V + 0.4 * b * d_t - V_c)}{0.87 * f_{YV} * d_t}$$

- ***Calculation of the required longitudinal shear reinforcement A_{SL}***

Where links are used, the area of longitudinal steel in the tensile zone should be such that:

$$A_{SL}\ = \frac{V}{2(0.87 f_Y)}$$

A_{SL}	is the area of effectively anchored longitudinal tensile reinforcement and pre-stressing tendons (excluding debonded tendons) additional to that required at the ultimate limit state for other purposes;

8.6.1.2 TORSION

- ***Box section:***

$$v_t = \frac{T}{2 * h_{wo} * A_o} \qquad \frac{\Delta v_t}{v_t} = \frac{h_{wo}^2}{2 * A_o} * \left(\frac{b}{h_t} + \frac{b}{h_b} + \frac{2 * d}{h_{wo}}\right)$$

h_{wo}	is the wall thickness where the stress is determined
h_t	is the wall thickness in the top flange
h_b	is the wall thickness in the bottom flange

- ***Rectangular section:***

$$v_t = \frac{2 * T}{h_{\min}^2 * \left(h_{\max} - \frac{h_{\min}}{3}\right)}$$

h_{min}	is the smaller dimension of the section ;
h_{max}	is the larger dimension of the section;

8.6.1.3 Interaction of torsion and shear force:

In no case should the sum of the shear stresses resulting from shear force and torsion ($v + v_t$) exceed the value of the ultimate shear stress v_{tu} from table 10 BS5400 except that for concrete grades above 40 v_{tu} may be increased to

$$v_{tu} = 0.75 * \sqrt{f_{cu}} \le 5.8 N / mm^2$$

<u>Reinforcement design: A_{ST}, A_{SL}</u>

$v_t > v_{tmin}$ (from table 10 BS5400)

N. B.: The formulae have been modified to suit the computer output.

- ***Box section:***

$$A_{ST} \ge \frac{T}{A_o * 0.87 * f_{YV}} \qquad A_{SL} \ge A_{SV} * \left(\frac{f_{YV}}{f_{YL}}\right) * \frac{(perimeter.of.A_o)}{2}$$

A_{ST} is the area of the closed links per metre (both legs) required for torsion – steel in both webs for a box girder.
A_{SL} is the total area of the longitudinal reinforcement, around the section required for torsion
f_{yv} f_{yv} & f_{yl} are taken as = f_y in RM2000 (not greater than 460 N/mm^2)

- ***Rectangular section:***

$$A_{ST} \ge \frac{T}{0.8 * x_1 * y_1 * 0.87 f_{YV}} \qquad A_{SL} \ge A_{SV} * \left(\frac{f_{YV}}{f_{YL}}\right) * (x_1 + y_1)$$

A_{ST} is the area of the closed links per metre (both legs) required for torsion.
A_{SL} is the total area of the longitudinal reinforcement around the section required for torsion
x_1 is the smaller centre line dimension of the links;
y_1 is the larger centre line dimension of the links;

<u>Allowable Ultimate shear stress</u> due to shear force for concrete Table 8 in BS5400:
The following formula is used:

$$v_c = \frac{0.27}{\gamma_m} * \left(\frac{100 * A_s}{b_w * d}\right)^{1/3} * (f_{cu})^{1/3}$$

$$0.15 \le \frac{100 * A_s}{b_w * d}$$

γ_m is taken as 1.25
f_{cu} should not exceed 40

Allowable shear stress due to shear force Table 28 in BS5400:
The following formula is used:

$$v_{\max} = 0.75 * \sqrt{f_{eu}} \leq 5.8 N / mm^2$$

Allowable shear stress due to torsion as well as due to shear plus torsion for concrete Table 10 in BS5400:

The following formulae are used:

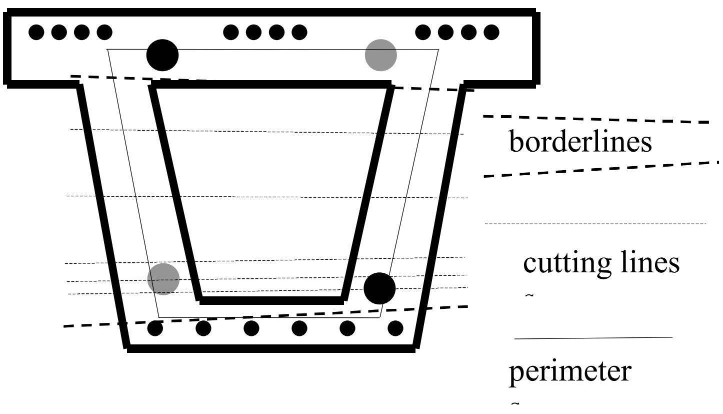
$$v_{tu} = 0.75 * \sqrt{f_{cu}} \leq 5.8 N / mm^2$$

$$v_{t\min} = 0.067 * \sqrt{f_{cu}}$$

8.6.2 Preparing data for the shear capacity check

8.6.2.1 The median wall line

The median wall line must be defined prior to the shear capacity check calculation. This line passes along the centreline of the individual elements forming the box section (top slab; bottom slab & webs) and encloses the area A_o which is used in the calculation for torsion.
The definition of the median wall line consists of a set of polygon points that are defined in GP2000. All these (additional) points must be assigned to a group name which is subsequently used as an input parameter in the ShChk module.
Two borderlines defining the web limits within the section must also be defined. The program looks for the minimum section width between these two borderlines.



N.B. The material input for the group name assigned to the median wall line definition is used as the material for the shear reinforcement design.

8.6.2.2 Automatic effective width calculation (of web)

When the tendon group or groups are defined as single tendons, the web width is automatically reduced by the full duct diameter (for un-grouted tendons) and 2/3^{rds} of the duct diameter (for grouted tendons)
Where a tendon or tendon group contains more than one tendon, the web width reduction must be user defined. (The program can not make the automatic web width reduction, in this case, as the tendons arrangement within the tendon group is not known).

8.6.2.3 User defined web width reduction

When the program is unable to calculate the effective web width (or web width reduction - refer clause above), the user may define the reduction manually:

The reduction to the web width can be defined for each element begin and end via the 'Element Reinf' input window:
Structure\Element\Reinf\b-beg(m) and b-end(m)
The values to be entered for b-beg(m) and b-end(m) are the actual reductions (not the reduced value!)
Example:
If there are 3 ducts in a horizontal line in each of 2 webs then the reduction will be:
3 * 2 * ϕ. If the duct diameter ϕ = 85 mm then b-beg (and b-end) = 0.51 metres. (or 2/3 of this value if the ducts are grouted)

8.6.3 Loading

The RM2000 shear check calculation requires a single, appropriately factored, superposition file (or loading case) name as input. This file must contain all the loads defined in the design code for the particular combination being checked.

This above superposition file (or loading case) must include the pre-stressing loading case (primary plus secondary pre-stressing)

In addition to this superposition file (or loading case), a separate loading case only containing the pre-stressing actions (primary plus secondary pre-stressing) must also be identified.

8.6.4 Partial safety factors γ_{fl} for Pre-stressing and γ_m for reinforcement.

The partial safety factors γ_{fl} for pre-stressing – Refer Cl:6.3.4.2 & 6.3.4.3 and γ_m for reinforcement used in the design formulae must be input for the two materials under Properties\Material\ (*material name*)\GAMMA. A value of 1.15 should be entered for both the pre-stressing material and the reinforcing material to be consistent with the design formulae used for shear design in BS5400.
.

8.6.5 Input Data for Module ShChk

The shear capacity check calculation is defined and started from the “Construction Schedule” - “Action” part of the program using the module “ShChk”.

Inp1:	Superposition file, Pre-stressing load case number
Inp2:	Group name, ‚Structure Class (ref BS5400 Cl 4.1.1.1(b))
Out1:	Output file name for a detailed result listing
Out2:	Output file name for a summarised result listing (Automatic if left blank)

Superposition file:	A single, factored, combination file containing all the loads (including pre-stressing) for the combination.
Pre-stressing load case number:	The number of the pre-stressing loading case (Primary + Secondary pre-stressing) that is included in the above combination file.
Group name:	Group name defined in GP2000 for identifying the median wall line and the associated material (also specified in GP2000) for the reinforcement used in this shear design.
Structure Class:	The design class of the structure being checked – Class 1; 2 or 3 (ref BS5400 Cl 4.1.1.1(b))
Out1:	Output file name for a detailed result listing. If this is left blank then a detailed listing will not be made. (N.B. The detailed listing is not required for design but is useful for cross checking)
Out2:	Output file name for a summarised result listing. (A default name will be automatically given if the pre-defined * is not deleted) N.B. All the result information necessary for the design is provided in this listing.

e.g.

Superposition file:	total.sup
Pre-stressing Loading case:	500
Group name:	shear
Structure Class:	2
Out1:	detshear.lst
Out2:	*

Module call	ShChk
Inp1	total.sup,500
Inp2	shear,,2
Out1	detshear.lst
Out2	*

N.B. the double ,, (comma) is important!!

N.B. The program automatically subtracts the primary pre-stressing forces *1.0 from the combination file (total.sup used in this example).
The combination file total.sup. must therefore include the primary pre-stressing forces – viz:

Typically the shear and associated forces from the following factored loading cases are included in a shear design check:
Dead Load + Additional Dead Load + Live Loading envelope + Temperature/wind/settlement etc + Secondary pre-stressing.

The program automatically subtracts the primary pre-stressing forces *1.0 from the combination file therefore the combination file for the RM2000 shear check calculation should then typically contain the following factored loading cases:
Dead Load + Additional Dead Load + Live Loading envelope + Temperature/wind/settlement etc + Secondary pre-stressing + Primary pre-stressing.

8.6.6 Defining the Median Wall Line in GP2000

[An abbreviated description on the method of defining the median wall line is given below - Refer to the GP2000 Getting Started for a more detailed explanation]

The median wall line, which is required for calculating the torsional stresses around a box girder is defined in GP2000 in the following way:

8.6.6.1 Define the “Group name”

The median wall line is identified by a name defined in the “Reference Point Group” window. Select the Reference Point Group arrow to open the input window.

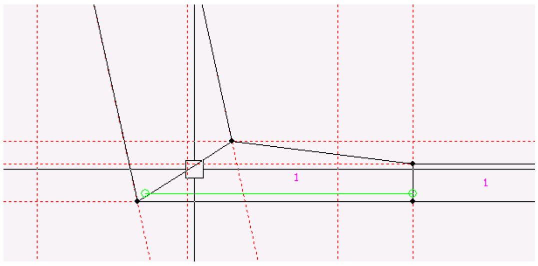
Select the ‘Append’ button to insert a new group and create the group “SHEAR” (say).

Insert “GRADE_460” for the material

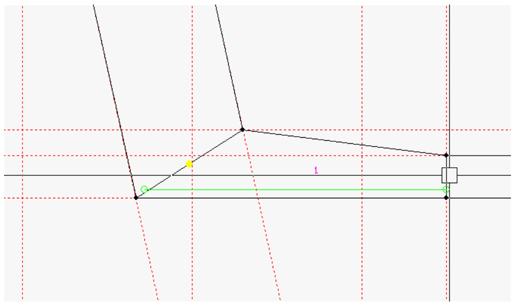
(N.B. This is the most convenient place for defining the reinforcement material that is to be used in the shear design calculation – it can also be defined in RM2000 under Properties\Material\AddGrp)

Select “SHEAR” as the active group.

- Select the icon for point construction relative to a node.

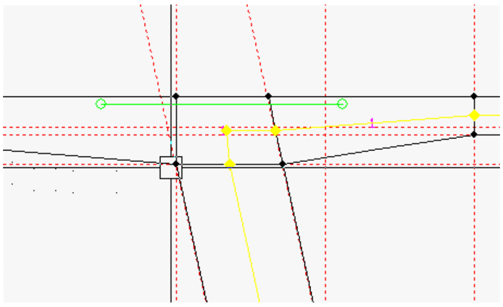


- Select an element point as shown in the adjacent screen. A screen for the definition of the point automatically opens.
- Select the type of point as "Perimeter Point".
- Confirm with `<OK>`. A single point will be displayed on the screen
- Continue around the box by selecting the next the element point – as shown in the adjacent screen and then the next and the next etc until the closed box is defined.

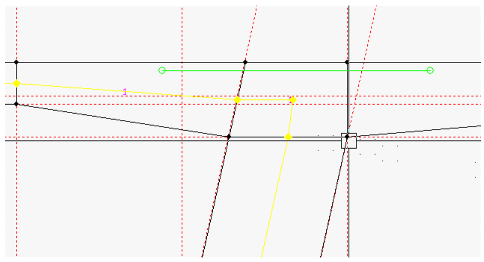


<u>Define the two border lines required to identify the "Webs"</u>

- Select 'INTERSECTION', to create a start point for the border line.



- Click on the intersection point below the left cantilever. A screen for the definition of the point automatically opens.
- Select the type of point as "Borderline Start" named: We-T.
- Confirm with `<OK>`. A triangle will be displayed on the screen.
- Select 'INTERSECTION', to create an end point for the border line.



- Select the intersection point on the out-side of the other web (below the right canti-lever slab)
- Select the type of point as "Borderline End" named: We-T from the new screen.
- Confirm with `<OK>`. A triangle will be displayed on the screen
- Repeat this input procedure to insert the second border line indicating the bottom of the web named: We-B.

8.7 Principal Tensile Stress check (DIN 4227 Part 1)

The principal tensile stress check in *RM2000* is carried out in accordance with DIN 4227 (part 1).The calculation is made when DIN is selected as the active design code, all used materials stress limits are defined and the actions PrDinU (for ultimate state) or PrDinS (for serviceability) are called in the construction schedule. The internal forces and stresses from the different loading cases (self weight, permanent loads, traffic loads and additional loads) must be available in the associated files (envelopes or loading cases) from a previous *RM2000* analysis.
The fibre stress check for serviceability has to be calculated in accordance with section 12.2 of DIN 4227 (part 1) and for the ultimate state in accordance with section 12.3 to be correctly made.
The section below gives the formulae used in *RM2000* for the principal tensile stress check in accordance with the above design code and describes the required input data preparation.

8.7.1 General Calculation of basic data

8.7.1.1 Shear stress in condition 1 due to shear force

Shear forces (T) for beams with continuous cross sections can be calculated in accordance with the following equation:

$$b \,.\, \tau_1 = \frac{Q_y \,.\, S_1}{I_z} = T$$

Iz Second moment of inertia about the z-axis
S_1 Static moment of the cut-off cross section area
B width of the webs

The additional influence of varying cross section properties (due to the cable geometry) must be considered the following way:

$$b.\tau_1 = \frac{Q_y \cdot S_1}{I_z} + \frac{d}{dx}\left(M_z \frac{S_1}{I_z} + N \frac{A_1}{A}\right)$$

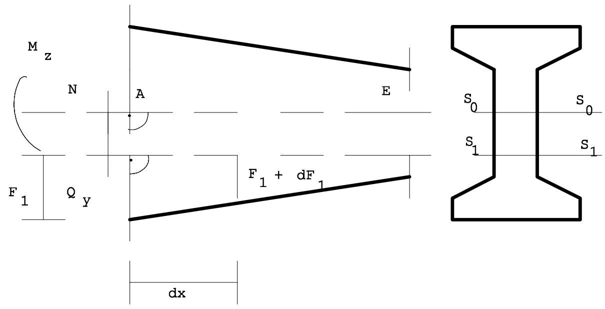


Fig. Varying cross sections

This equation is solved by defining the terms

$$\frac{d}{dx}\left(\frac{S_1}{I_z}\right) \text{ and } \frac{d}{dx}\left(\frac{A_1}{A}\right)$$

as differential quotients:

$$\frac{d}{dx}\left(\frac{S_1}{I_z}\right) = \frac{1}{\Delta X}\left[\left(\frac{S_1}{I_z}\right)_B - \left(\frac{S_1}{I_z}\right)_E\right]$$

$$\frac{d}{dx}\left(\frac{A_1}{A}\right) = \frac{1}{\Delta X}\left[\left(\frac{A_1}{A}\right)_B - \left(\frac{A_1}{A}\right)_E\right]$$

ΔX	Element length
S_1	Static moment for cut off cross sectional area (cut 1-1)
I_z	Moment of inertia about Z-axis due to the ideal (composite) cross section
A_1	Cut off cross sectional area
A	Cross sectional area due to the ideal (composite) cross section
B	Element begin
E	Element end

Cross section changes between one element end and the next element begin are not considered.

8.7.1.2 Shear stress in condition 1 due to torsion moments

Hollow box

$$T = \frac{M_T}{2 \cdot F_k} \rightarrow \tau_{MT} = \frac{T}{b}$$

F_K Perimeter area
b Concrete thickness

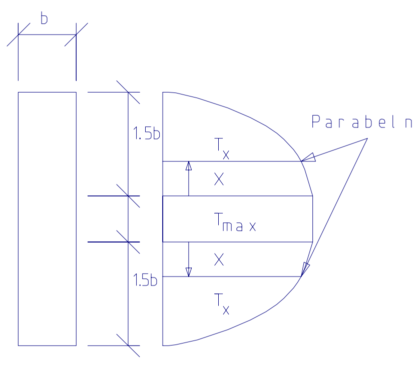
T- beam, rectangle:

$$\tau = \frac{M_T}{A_c t_{eff}}$$

A_c Area defined by the center line of the elements of the equivalent box cross section
t_{eff} Effective width of the equivalent box cross section

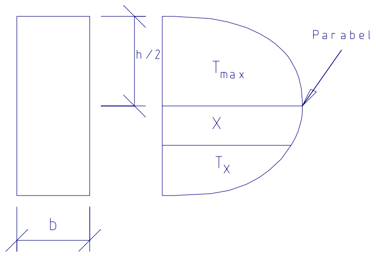
The max. shear stress due to torsion will be computed in accordance with the above formula valid for rectangles and plate-beams.

A parabolic distribution of the shear stresses is assumed.
According to figure 9.4 and 9.5 two cases must be distinguished



$$\tau_x = \frac{\tau_{\max}}{2.25\,b^2}\,x^2 + \tau_{\max}$$

Fig. Distribution of stresses for h/2 < 1.5b



$$\tau_x = -\frac{\tau_{\max}}{0.25\,h^2}\,x^2 + \tau_{\max}$$

Fig. Distribution of stresses for h/2 > 1.5b

The shear stresses for condition 1 will be computed for the service load stage and the calculated ultimate load stage.

The principal stresses and associated angles are evaluated considering the axial stresses s_x, stresses s_y due to transverse pre-stressing or shear connectors and shear stresses.

8.7.2 Evaluation of stresses due to service and ultimate load

8.7.2.1 Evaluation of stresses due to service load

The principal tensile stress due to service load will be computed by the following formula using the formulas for τ_Q and τ_{Mt} (condition 1).

$$\sigma_{I,II} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + (\tau_Q + \tau_{MT})^2}$$

8.7.2.2 Evaluation of stresses due to computed ultimate load.

Different zones depending on the computed ultimate load (section 12.3. of the associated German standard):

Upon exceeding the allowable edge stresses the cross section is considered to be in ZONE A.

• **Zone A: Principal compressive stress as a result of shear force**

The relevant principal compressive stress between the web and the compression flange will be determined for zone A as follows:

$$\sigma_{II\,ZU} = -\frac{\tau_u - \sigma_{yu}}{\sin\delta \cdot \cos\delta}$$

$$\tan\delta = \tan\delta_I \left(1 - \frac{\Delta\tau}{\max\tau_u} \right) \geq 0.4$$

$$\tan\delta_I = \frac{\sigma_{Ium} - \sigma_{yum}}{\tau_{um}}$$

τ_u	The shear stress from shear force at the position being considered.
σ_{yu}	Associated normal stress (σ_{yn}= 0) for condition 1 and the ultimate load state
δ	The calculated angle between the compression diagonal and the axis of the cross section due to the associated loading case.
σ_{1um}	The principal tensile stress at the cut through the flange cross section.
σ_{ym}	The associated normal stress
τ_{um}	The associated shear stress.
max τ_u	The shear stresses from the shear force at the cut.

All stresses will be computed according to the condition 1 rules and the computed ultimate load state.

$\Delta\tau$	60 % of the stress limit for the inclined principal tensile stress resp. shear stress without the calculation of shear reinforcement (zone A and zone B).

The shear stresses from condition 1 are taken into account instead of the principal compressive stresses for tensioned flanges. The limits for these stresses can be found in the instruction for "Basic values of the shear stresses in the computed ultimate state in zone B and in the tensile flanges of zone A".

• Zone A: Principal compressive stress due to torsion

The principal compressive stresses due to torsion will be computed considering an assumed angle of 45° of the compression member.

$$\sigma_{II\,zu} = -2\,\tau_u$$

τ_u The shear stressed due to torsion for condition 1 and the computed ultimate state

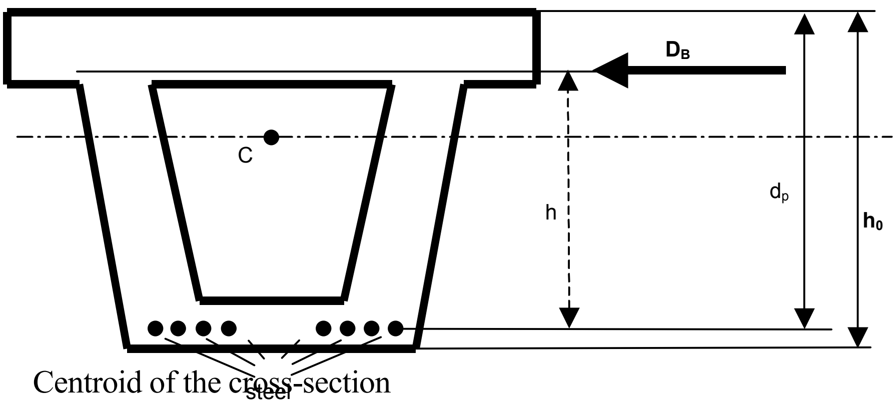
If there is torsion and shear force acting at the same time, the calculated stress components due to both forces are added algebraically. Stresses exceeding the limits will be marked on the left side of the output with a star (‘*’).

• Zone B: Shear and principal compressive stress.

Shear stresses from the shear force in condition 2 are the relevant evaluations for zone B.

$$\tau = \frac{Q_{Tr} + D_b \sin\alpha}{h \cdot b}$$

Q_{Tr} Shear force from the computed ultimate load.
D_b Resultant internal compressive concrete force for the corresponding load combinations.
sin α Angle between the centroidal axis and the line of the resulting compressive concrete force D_b.
h Inner lever arm of the relevant load combination for the corresponding bending moment. The lever arm is a result of the difference between the resultant D_B (sums of the internal forces in the compressive zone) and Z (sums of the internal forces in the tensile zone) but not less than $0.8 \cdot d_p$.



C Centroid of the cross-section

h_0 The total height of the cross-section
d_p Distance between the resultant of the sum of the internal forces in the tensile zone and the compressive edge

• Zone B: Principle compressive stress in the bending compression zone

$$\sigma_{II\,ZU} = \frac{-\tau_R}{\sin\delta \cdot \cos\delta}$$

$$\tan\delta = (1 - \Delta\tau / \tau_R) \geq 0.4$$

τ_R Value for shear stress in condition 2
$\Delta\tau$ 60 % of the stress limit for the inclined principal stress from shear stress without checking the shear reinforcement (zone A and B).

Compression zones will be treated as in zone A, taking into account τ_u and σ_u in condition 2.

σ_U Max. edge stress in the cracked state from the relevant load case or superposition file.
τ_U Shear stress in condition 2 evaluated for the cuts.

$$\tau_U = \tau_R \frac{D_b^{\,T}}{D_b}$$

τ_R Value of the shear stress in condition 2.
D_b Resulting concrete compressive force for the corresponding load combination.
D_b^T Resulting concrete compressive force in the dropped part of the cross section for the corresponding load combination.

• Zone B: Shear and principal compressive stress due to torsion

Ref. zone A.

8.7.3 Calculation of reinforcement to take tensile forces

The shear reinforcement calculation will be performed for those cuts, in which one of the specified limits are exceeded. In these cases, tensile force per unit length is calculated for the computed ultimate load state. The shear reinforcement must carry these tensile forces.

• Zone A: web and compression flange

$$Z_Q = b \cdot \frac{\tau \cdot \sin\delta + \sigma_{yu} \cdot \cos\delta}{\cos\delta} \qquad A_Q = b \cdot \frac{Z_Q}{\beta_s}$$

b	Concrete width of the cut under consideration
τ	Shear stress in condition 2 as a result of the computed ultimate load.
d	See principal compressive stress for zone A.
σ_{yu}	See principal compressive stress for zone A.
βs	Yield strength of the structural steel.
A_Q	Calculated area of reinforcement due to shear force Q

• Zone B: Web

$$Z_Q = b \cdot \frac{\tau_R \cdot \sin\delta}{\cos\delta} \qquad A_Q = b \cdot \frac{Z_Q}{\beta_s}$$

b	Concrete width of the examined cut.
τ_R	Evaluated shear stress in condition 2.
δ	See principal compressive stress in zone B.
βs	Yield strength of the structural steel.
A_Q	Calculated area of reinforcement due to shear force Q

• Zone B: Compression flange

The computation of the tensile force will be performed as for zone A, but the stresses τ_u and $\sigma_{x,u}$ refer to the values of the bending tensile zone in condition 2.

• Torsion

The reinforcement calculations will be performed for the equivalent lattice work box with a tensile flange and tensile diagonals in equilibrium with the longitudinal and bending reinforcement. The tensile force per unit length for vertical stirrups.

$$Z_{MT} = \frac{M_{TU}}{2\,F_K} \qquad\qquad A_{MT} = b \cdot \frac{Z_{MT}}{\beta_s}$$

M_{TU}	Torsion moment from the computed ultimate load.
F_K	Core area of the hollow cross section. An equivalent hollow cross section will be computed for T-or rectangular cross sections.
βs	Yield strength of the structural steel.
A_{MT}	Calculated area of reinforcement due to torsion moment MT

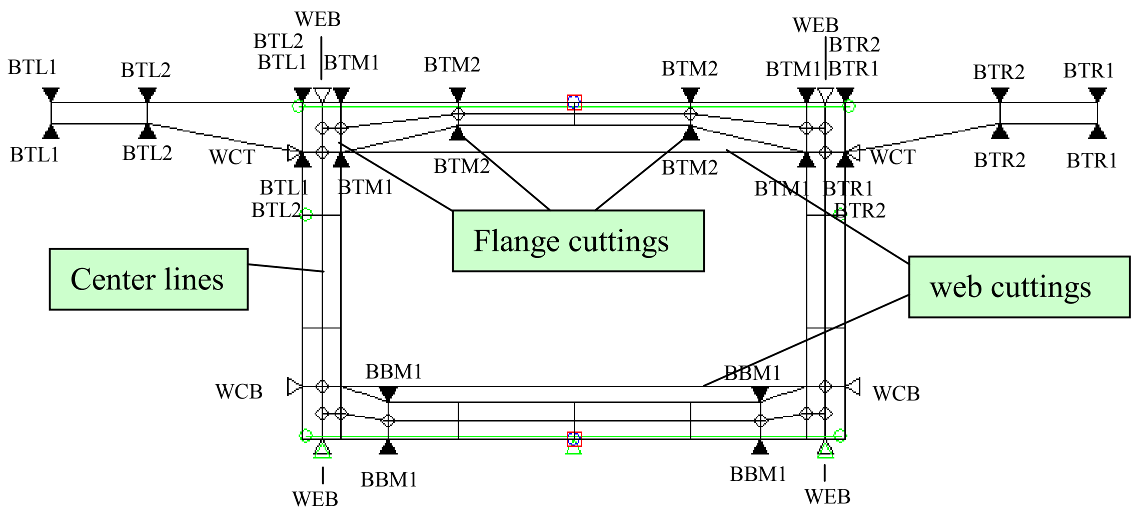
There is no reduction of the axial tensile stresses due to torsion in areas of axial compressive stresses.

8.7.4 Preparation of the Cross-section (*GP2000*)

For the principal tensile stress check it is necessary to make some preparations in GP2000. First a perimeter has to be defined (see picture). Also cutting lines which intersect the webs and flanges of the cross-section have to be created. The principal tensile stress calculation also necessitates the definition of cutting lines which have to be positioned exactly in the middle of the webs (vertically). The designation of these 'center lines' must be **'WEB'**.

The definition of points of correlating cuts must have the same designation (e.g. all four points for the cutting line BTM2 have to be named with BTM2).

The definition types of the flange cuttings as well as of the lines in the middle of the webs are 'Borderline Start' and 'Borderline End'. The perimeter has to be generated with 'Perimeter Points'.
The web cuttings and the perimeter points should be defined in the same reference group (e.g. SHEAR). The flange cuttings are defined in a different group (e.g. BORDERS).



BTL1/2	Border Top Left 1/2	BBM1	Border Bottom Middle 1
BTR1/2	Border Top Right 1/2	WCT	Web Cutting Top
BTM1	Border Top Middle 1	WCB	Web Cutting Bottom
BTM2	Border Top Middle 2	WEB	Center Line in the middle of webs

Note that the designations of these defined points should not have more then four characters.

8.7.5 Input for the principal tensile stress check (*RM2000*)

The calculation of principle tensile stresses due to service or ultimate loadings have to be started with the check actions **PrDinS** (for serviceability) and **PrDinU** (for ultimate state) at each construction stage. The associated superposition files and the result files for the considered loading cases have to be calculated beforehand.

8.7.5.1 Serviceability:

For starting the action **PrDinS** a superposition file or result file for a loading case has to be input. Also the borderlines which may have been defined in GP2000 or in RM2000 beforehand have to be activated.
The order of the input data is important. First the reference group in which the perimeter and the border lines for the horizontal cuts are defined have to be specified. As the second input select the reference group where all vertical cuts are defined and separate it with a '**,**' from the first input (e.g. SHEAR,BORDERS). Currently only the first input can be selected interactively by clicking on the arrow, the second input has to be written into the input field explicitly.
Also an output file name (*.lst) for a detailed list of the calculated results can be specified in the appropriate input line.

8.7.5.2 Ultimate State:

For this action an ultimate check (UltChk) has to be calculated with the corresponding superposition file or loading case. Only **Ult** has to be specified for this action. Then the action **PrDinU** can be started. The first input must be the superposition file or result file for a loading case for which the check should be performed and the second input is the created **UltChk** superposition file. In the same way as for the **PrDinS** action the reference groups have to be specified.
After a **RECALC** the calculated results are available.

8.7.5.3 Correct input for a principal tensile stress analysis (SLS and ULS)

```
MODULE   "PrDinS"  "SLS-H.sup"              "SHEAR,Borders" "" "PrDinS-SLS-H.lst"  0 ""
MODULE   "SupInit" ""             ""        "ULS-Hres.sup"  ""                     0 ""
MODULE   "UltChk"  "ULS-H.sup"    "Ult"     "ULS-Hres.sup"     "ULS-Hres.lst"      0 ""
MODULE   "PrDinU"  "ULS-H.sup,ULS-Hres.sup" "SHEAR,Borders" "" "PrDinU-ULS-H.lst"  0 ""
```

In this example the reference group for the perimeter and the web cutting lines is 'SHEAR' and the reference group for the flange cuttings is 'Borders'. The results of the principal tensile stress calculation are written into the list files PrDinS-SLS-H.lst (SLS) and PrDinS-ULS-H.lst (ULS).

8.7.6 Output and results

The results are written into the corresponding *.lst files. The following list shows which results of the calculated checks are listed.

```
                PRINCIPLE TENSILE STRESSES (SERVICEABILITY)
                       SUPERPOSITION FILE: SLS-HZ.sup
                       Units : Force(kN),Moment(kNm)

 ELEM POS         CS        MAT                           ACP              H
 NR:               N         QY        MX         MZ
      (MAX)      SIG-X     TAU-Q     TAU-MT     SIG1      SIG2  (NR1,NR2)  B
* NR:            SIG-X     TAU-Q     TAU-MT     SIG1      SIG2   ALPHA     B
---------------------------------------------------------------------------
  4: MAXMz  -55064.1   -1845.0    3568.0    -461.4
     CENT1   -5197.3    1896.2     316.3     669.6   -5850.0 (1,1)     0.500
     CENT2   -5197.3    1896.2     316.3     669.6   -5850.0 (1,1)     0.500
     BLT11   -4990.5    1826.1     316.3     875.8   -5192.0 (2,4)     0.500
     BLT12   -4990.5    1826.1     316.3     875.8   -5192.0 (2,4)     0.500
     BLU11   -9444.2    1630.0     316.3     333.5   -9722.3 (1,2)     0.500
     BLU12   -9444.2    1630.0     316.3     333.5   -9722.3 (1,2)     0.500
     BLU21   -9916.8     215.7      26.8       5.6   -9921.4 (1,2)     5.900
     BLU22   -9916.8     215.7     143.8      12.4   -9922.6 (3,2)     1.100
     BLT21   -4911.0     130.4      41.2       8.4   -4912.3 (2,4)     3.842
     BLT22   -4911.0     130.4      23.1       7.9   -4912.1 (2,4)     6.842
     STL1    -4873.5     442.1       0.0      97.9   -4882.1 (2,4)     0.270
     STL2    -4916.7     648.7       0.0     165.1   -4935.5 (2,4)     0.650
     STML1   -4916.7     179.4       0.0      13.4   -4918.1 (2,4)     0.650
     STML2   -4876.9     388.7       0.0      74.7   -4883.9 (2,4)     0.300
     STMR1   -4916.7     179.4       0.0      13.4   -4918.1 (2,4)     0.650
     STMR2   -4876.9     388.7       0.0      74.7   -4883.9 (2,4)     0.300
     STR1    -4873.5     442.1       0.0      97.9   -4882.1 (2,4)     0.270
     STR2    -4916.7     648.7       0.0     165.1   -4935.5 (2,4)     0.650
     SUML   -10340.9    2133.5       0.0     472.9  -10724.1 (1,2)     0.359

  134  2"hollowbox.011" "B_45"                            22.0          3.768
---------------------------------------------------------------------------
  1: MINNx  -50996.3   -2486.0    2275.1  -22431.9
  2: MINMz  -50993.1   -2322.7   -1454.2  -32943.0
  3: MAXRz  -50990.2   -2175.1    2756.5  -31082.0
  4: MAXMx  -50969.4   -1111.4    7732.9  -13805.7
  5: MAXMz  -50931.9     799.8    4749.1    9691.3
     CENT1   -4934.0    1163.0     351.7     306.8   -5227.9 (1,1)     0.500
     CENT2   -4934.0    1163.0     351.7     306.8   -5227.9 (1,1)     0.500
     BLT11   -5147.2    1127.8     351.7     348.4   -5158.7 (1,5)     0.500
     BLT12   -5147.2    1127.8     351.7     348.4   -5158.7 (1,5)     0.500
     BLU11   -7725.9    1044.2     351.7     185.3   -7875.6 (1,2)     0.500
     BLU12   -7725.9    1044.2     351.7     185.3   -7875.6 (1,2)     0.500
     BLU21   -8042.7     140.3      29.8       3.1   -8045.2 (1,2)     5.900
```

The header indicates if the results are calculated for the serviceability check or for the ultimate state check, the applied result file and the units.
The second table lists all internal forces and stresses for which the results are calculated and important additional values on which the calculation is based are displayed.
The first line displays the element number and the position 1 or 2 for the begin or the end of the element. In addition the name of the cross-section and the included area of the perimeter (ACP) as well as the hight (H) of the cross section are listed.

```
 ELEM POS         CS        MAT                           ACP              H
```

Explanation of displayed result lines:

First an internal number for the listed superposition line is assigned. For example **1** is been assigned for the MINNx superposition. All calculated results in this line are outputs of the MINNx analysis.

```
1: MINNx  -50996.3   -2486.0    2275.1  -22431.9
```

In the following lines the calculated shear stresses and principal stresses are listed. On the left side of these lines the defined name of the corresponding cross-section cut is written.

```
  (MAX)    SIG-X     TAU-Q     TAU-MT     SIG1      SIG2  (NR1,NR2)   B
 BLT11   -5147.2    1127.8     351.7     348.4   -5158.7    (1,5)  0.500
```

The results SIG-X, TAU-Q and TAU-MT are the maximum values which are calculated for the cut BLT11.
The expression (1,5) means that the result of SIG1 comes from the analysis which is marked with 1 (in this example MINNx) and the result of SIG2 comes from the superposition results which is marked with 5 (here: MAXMz).

Each line which is marked with a star (*) at the begin of a line shows that the defined stress limits are exceeded.

8.8 Reinforced concrete design

This chapter shows all the necessary steps for the design of unstressed reinforcement (for bending and axial force) in pre-stressed or non-pre-stressed cross-sections.

The following steps are required for performing the reinforcement design:

- Definition of material properties for the design (σ-ε diagrams of all used materials – concrete, reinforcement, ...) used materials
- Definition of Reinforcement groups
- Definition of reinforcement points (location in the cross-section)
- System \ Elements \ Reinforcement – Definition of the reinforcement for the elements
- Internal forces – Establishing the relevant combinations (envelopes) for the design
- Adding the design actions in the construction stage
    - REININI Initialising the reinforcement area A2 (zero)
    - ULTREIN Calculation of the reinforcement required for the Load Case or Envelope under consideration

Note: *An ultimate load carrying capacity check using the computed unstressed reinforcement content can afterwards be performed. The module ULTCHK does not increase the area of reinforcement!).*

8.8.1 Material properties for the reinforcement design

The material properties needed for the reinforcement design module are the same as those for the ultimate load carrying capacity check (see chap. 8.3.2)

8.8.2 Reinforcement point groups

The reference points describing the position of reinforcement in the cross-sections have to be assigned to groups in the same way as for the ultimate load carrying capacity check (see chap. 8.3.3). The reference points must have been defined before the group can be assigned to them.

Note: *Maximum 24 reference point groups may actually be defined for one cross-section.*

8.8.3 Position of the reinforcement in the cross-section

The position of the reinforcement in the cross-section is defined by the reference points. These must be specified in the same way as for the ultimate load carrying capacity check (see chap. 8.3.3)

8.8.4 Reinforcement content in the elements

The actual reinforcement content in the elements may be viewed in ⇧STRUCTURE ⇨ELEMENT ⇩CHECKS. Reinforcement groups may not be created in this function (the reinforcement groups are automatically assigned to the elements based on the assignment to the start and end cross-sections of the element). The area of reinforcement belonging to the group may however be specified using this function. 2 separate reinforcement areas may be assigned:

A1 Fixed reinforcement
A2 Fixed or variable reinforcement

The reinforcement area A1 must be specified by the user and will not be changed in the reinforcement calculation process. It is e.g. meant for defining the minimum reinforcement. This amount will also not be changed when initialising the reinforcement with the REININI action in the construction schedule. This part of the reinforcement is therefore a fixed reinforcement amount!

The amount A2 may be defined as being fixed (as A1) or variable. This part of the reinforcement may therefore be predefined by the user, or changed by the program in the design process. If A2 is defined as being variable, the required reinforcement (additionally to A1) will be stored under A2, and may be viewed in the element reinforcement table. The total reinforcement of the group will be A = A1 + A2.

At least one appropriate reinforcement group must have a variable A2 in order to enable the computation of the required reinforcement amount. Otherwise an ultimate load carrying capacity check will be made instead of ULTREIN, illustrating the information whether the predefined reinforcement content is sufficient or not.

8.8.5 Relevant Combinations

The relevant loading combinations must have been established prior to carrying out the reinforcement calculation (ULTREIN) in the same way as for the ultimate load carrying capacity check (see chap. 8.3.5).

8.8.6 Calculating the reinforcement

The calculation action is then placed in the construction schedule with ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION, selecting

⊙ Check actions
- ReinIni (if necessary) and
- UltRein

The function ReinIni for initialising A2 does not require additional parameters to be input. The function UltRein requires:

- Inp1 Number of the load case or name of the superposition case (*.sup). Interactive selection in the pull-down menu.
- Inp2 -
- Out1 -
- Out2 Name of the list file for storing the results
- Delta-T -
- Description Descriptive text (max. 80 characters)

The results are listed in an output file. A result diagram can be plotted for every reinforcement group (similar to the stress diagram for stress points). This is generally done by creating a plot file in ⇧RESULTS ⇨PLSYS.

General computation parameters for the stress integration over the cross-section in the function UltRein are defined in ⇧RECALC ⇨CSINT.

Iteration Maximum number of iterations
The program compares the difference between the internal forces due to loading and the resultant forces due to integrating the stresses related to an assumed strain state. The program finishes the iteration, if the difference remains under the defined tolerance value or if the maximum iteration number is reached. Each attempt is an iteration.

Recursion level should be between 1 and 3

2 inaccurate, but fast
3 intermediate
4 accurate, but slow

Improvement factor should be between 0,25 and 0,5

0.5 inaccurate but fast
0.25 accurate but slow

Relaxation factor

Tolerance Accuracy tolerance (see “Iteration” above)

Option: Attempt with optimum reinforcement
The program tries to set the area of reinforcement groups with very small A2 (from a previous design process) to zero. If this is not possible, these values are increased and the design for the other groups is repeated.

Option: Reduction of small values
The program tries to set the area of reinforcement groups with very small A2 (from a previous design process) to zero. The other areas are again determined afterwards.

8.9 Linear Buckling Analysis

A linear buckling analysis is performed with the calculation action BUCKLE in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION. All relevant Load Cases must be available before adding the buckle action into the construction schedule.

Attention:	**Note that the Option "Consideration of P-Delta effects" must have been activated in ⇧RECALC in order to allow performing a buckling analysis.**

Select

- ⊙ Calculation Actions
  and click the line for

- BUCKLE — Linear Buckling analysis

- ➢ Command — Program defined name of the action
- ➢ Inp1: — Number of natural modes The user can select how many natural modes the program should find (a warning will appear if less natural modies than selected are found while performing the calculation action) – interactive definition between type 1 and type 2 is available via the pull-down menu arrow. *(See below for further clarification)
- ➢ Inp2: — Reference Load Case – The number of an existing factored Load Case (e.g. 101) - interactive selection is available via the pull-down menu arrow.
- ➢ Out1: — Output-Load Case; Enter the number of the first free Load Case.
- ➢ Out2: — Output-file; Name of the list file containing internal forces and displacements for the reference Load Case and all found natural modes.
- ➢ Delta-T — Duration of the Action (not needed for this Action)
- ➢ Description — Descriptive text (max. 80 characters)

The result of each natural mode is stored by the program. The first natural mode can be found in the file specified for 'Out1: Start Output LC'. The ensuing natural modes are stored in files, where the specified Load Case Number part is incremented by '1'. All natural modes can also be presented graphically by using the function ⇧RESULTS ⇨PLSYS where the deformed shape for each natural mode (selected by the assigned filename) can be plotted.

Two types are available (choose between these two types by open the arrow button on INP1):

Type 1: All Load Sets
Type 2: Only variable Load Sets

Type 1:

Increase all used Load Sets in the Load Case

How does the program find the buckling load?

$[K_1]$ stiffness matrix (linear = P-Delta without loads)
$[K_2]$ stiffness matrix (P-Delta with all loads)

$$\left|[K_1]+\lambda\cdot([K_2]-[K_1])\right|=0 \quad \lambda=...$$

Type 2:

Increase only the variable Load Sets in the Load Case

How does the program find the buckling load?

$[K_1]$ stiffness matrix (P-Delta with constant loads only)
$[K_2]$ stiffness matrix (P-Delta with all loads)

$$\left|[K_1]+\lambda\cdot([K_2]-[K_1])\right|=0 \quad \lambda=...$$

8.10 Buckling Analysis till Failure (Non-linear buckling)

Failure due to non-linear buckling is calculated by using the calculation action FAILURE in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION. All relevant Load Cases must be available before adding the failure action into the construction schedule.

Attention:	**Note that the Option "Consideration of P-Delta effects" must have been activated in ⇧RECALC in order to allow performing a buckling analysis.**

Select
- ⊙ Calculation Actions
  and click the line for

- FAILURE Buckling analysis till failure

- ➢ Command Program defined name of the action

- ➢ Inp1: Reference Load Case – The number of an existing factored Load Case (e.g. 101) - interactive selection is available via the pull-down menu arrow.

- ➢ Inp2: Factor from, Factor to, Tolerance
  - ▪ Factor from: intensity factor, lower limit
  - ▪ Factor to: intensity factor, upper limit
  - ▪ Tolerance for interactive computation of the critical intensity factor

- ➢ Out1: Output-Load Case; Enter the number of the first free Load Case.

- ➢ Out2: Output-file; Name of the list file containing internal forces and displacements for the reference Load Case and all found natural modes.

- ➢ Delta-T Duration of the Action (not needed for this Action)
- ➢ Description Descriptive text (max. 80 characters)

The result of each natural mode is stored by the program. The first natural mode can be found in the file specified for 'Out1: Start Output LC'. The ensuing natural modes are stored in files, where the specified Load Case Number part is incremented by '1'. All natural modes can also be presented graphically by using the function ⇧RESULTS

⇨PLSYS where the deformed shape for each natural mode (selected by the assigned filename) can be plotted.

The program varies the intensity factor within the given limits and computes the corresponding stiffness matrix and the value of the determinant for each factor. The program determines just the first intensity factors for which the determinant becomes 0 within the given limits. This is carried out iteratively and is controlled by the given tolerance.

Two types are available (choose between these two types by open the arrow button on INP1):

Type 1: All Load Sets

Type 2: Only variable Load Sets

Type 1:

Increase all used Load Sets in the Load Case

Type 2:

Increase only the variable Load Sets in the Load Case

9 Dynamics

In addition to standard static analyses *RM2000* also supports the solution of different structural dynamics problems, such as:

- Calculation of Eigenfrequencies on the un-damped system
- Calculation of the related Eigenvectors (natural modes)
- Calculation of forced vibrations with or without consideration of modal damping
- Evaluation of a response-spectrum (earthquake response analysis)
- Time stepping procedure with direct time integration for arbitrary load histories (including 'moving masses' and 'moving load')
- Wind dynamics (stochastic excitation)

9.1 General

The basic discrete equation system for solving structural dynamics problems with viscous damping is written:

$$[M]\,\{\ddot{a}\} + [C]\,\{\dot{a}\} + [K]\{a\} + f = 0$$

where [M] = mass matrix
[C] = damping matrix
[K] = stiffness matrix
{f} = force vector
{a} = deformation vector
$\{\dot{a}\}$ = velocity vector
$\{\ddot{a}\}$ = acceleration vector

All these quantities may in general be time dependent, however, the stiffness, mass and damping matrices are mostly assumed to be constant in the time domain.

The above equation is generally valid and can be physically interpreted as the condition of equilibrium (the sum of all forces acting in each direction must be zero at any time). The following restrictions are however often introduced in order to allow an efficient solution of this equation:

1) diagonalization of the mass matrix (lumped mass)
2) small damping (≤ 15 %) (to allow the calculation of eigenfrequencies on the undamped system)
3) damping proportional to velocity (linear damping)
4) damping matrix can be represented as a linear combination of mass matrix and stiffness matrix

The requirements for performing structural dynamics analyses with RM2000 are:

- The structural model must be created (as for static analyses*)
- The active masses need to be specified (in any case)
- The damping parameters must be specified (for modal analysis and time history)
- The (time dependent) loading has to be specified (for standard modal analysis and time stepping procedure)
- A response spectrum diagram needs to be entered (for Response Spectrum analysis only)
- The required computation actions need to be specified in the Construction Schedule

** Note:* *There are some additional modelling requirements for dynamic analyses compared to static analyses. These requirements are described in chap. 9.2.*

The required calculations will be made when the above definitions have been entered and the function ⇧RECALC is invoked.

9.2 Structural requirements, Mass matrix and Damping matrix

9.2.1 Structural model requirements

The concept of *RM2000* is generally to perform dynamic analyses within the construction schedule together with static analyses on the same model according to the active construction stage. This demands that all requirements for either static or dynamic analyses must be taken into consideration in the structural modelling process.

In many cases the standard requirements for modelling the structure as described in chapter 4 are also sufficient for the dynamic analysis, except that additional parameters like masses, damping, time dependency of loads etc. have to be specified. There are however some restrictions that must be taken into consideration to allow the same structural model being used for static and dynamic analyses.

The most important of these restrictions is, that the consideration of the influence of masses distributed over the element length is not exact, but approximate. The mass matrix is usually lumped, i.e. the distributed element masses are integrated over the element length and applied as point masses on the structural nodes.

Therefore it is absolutely necessary for dynamic calculations, that the total deformation shape may be sufficiently well described by the nodal displacements alone. The part of the local deformation shape within an element, which is caused by the distributed masses, must be small enough to be neglected. This requires a sufficiently fine subdivision of the structural parts into elements.

As a general rule, each span of the superstructure should be subdivided into at least 10 elements to get a sufficiently good answer for the dynamic behaviour. High piers have also to be subdivided, if the mass of the pier has a considerable influence.

Other additional structural requirements may concern the boundary conditions or the admissibility of model simplifications, such as for instance calculating a 6 or 7 span bridge instead of a 30 span bridge with constant span lengths. The user has to check from case to case, whether the chosen the model assumptions allow a sufficiently accurate solution of the required dynamic analysis.

9.2.2 Mass matrix

As a prerequisite for the solution of any structural dynamics problem the mass matrices have to be established, as can easily be seen in the above basic equation.

The generation of mass matrices requires, that the masses of the system have been previously defined. *RM2000* can handle distributed element masses and point masses.

The masses and moments of inertia are **defined as forces and moments** in the menu ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS. These defined force values are in the dynamic analysis (for the calculation of mass matrices) divided by the gravity acceleration value to get the real masses and moments of inertia used in the solution process. The gravity acceleration value is in the program per default set to 9.81 m/s^2, and may be changed by the user in the menu ⇧RECALC ⇩DYNAMIC.

Attention:	Masses are generally scalar values, acting in the same manner in all possible acceleration directions. Thus, the definition of masses as force vectors requires to enter the same value for all 3 force components. ***RM2000*** **does not set zero force components automatically equal to the non-zero value, as it is done for the self weight masses**. This enables the user to take into account special effects (e.g. to exclude or reduce the vibration in a certain direction, considering hydrodynamic masses etc.), but it requires to be careful and not to forget to specify all 3 components in the standard case.

The masses must be grouped in Load Sets and assigned to Load Cases in the same manner than static loadings. Chap. 6, Loading, shows in detail, how to create Load Sets and Load Cases. This section only shows the special requirements for the definition of masses.

9.2.3 Definition of the Masses

Three Load Type groups allow a mass definition:

- Nodal masses (Load Type 'MASSES – CONC. NODAL MASS')
- Element masses (Load Type 'MASSES – ELEMENT UNIFORM MASS')
- Self weight (Load Type 'UNIFORM LOAD – SELF WEIGHT')

The masses defined as a self weight load are automatically applied as a loading **and** as a distributed mass when the Load Set is assigned to the Load Case used in the required dynamic analysis. Masses defined as Nodal masses or Element masses may either be used for both, mass matrix **and** load vector computation, or only for the mass matrix calculation.

Note: *The definition of masses comprises also the definition of the rotational mass inertia terms. These are usually given either in terms of moments of mass inertia or in terms of the radii of mass inertia. RM2000 offers only the possibility to specify these parameters in terms of moments of mass inertia. If the radii of inertia are given instead of the moments, the user has to transform them into moments to specify the rotational inertia behaviour.*

9.2.3.1 Load Type group 'MASSES – CONC. NODAL MASS'

All Load Types of this group require the definition of the nodes, where the mass should be applied, a switch, defining, whether the mass should only be applied as a mass or also as a load, and a load vector, describing the nodal mass:

➢ From, to, step	series of structural nodes for the mass definition.
⊙ Dynamic	the specified load vector will only be applied as a mass (no static or dynamic load is generated).
⊙ Static + Dynamic	the specified load vector will be applied as a mass **and** as a static or dynamic load*.
➢ Load vector	specification of the mass parameters according to the Load Type (as described below).

* *Note:* *The function for considering a mass also as a load (switch "Static+Dynamic" is not yet implemented. Point mass, which should also be considered as point loads, must be additionally specified as point loads.*

The NODAL MASS group comprises the following Load Types for the definition of node masses:

NDMAS **Concentrated nodal mass and moments of inertia**.

This Load Type allows to define concentrated masses assigned **concentrically** to the structural nodes. The masses are specified as forces with possibly different components in the 3 global directions. The moments of inertia are specified as moments around the global coordinate axes.

Load vector to be entered:

- g*mx [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global X-direction.
- g*my [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global Y-direction.
- g*mz [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global Z-direction.
- g*Imx [unit] moment of mass inertia for the rotation φ_X around the global X-axis, multiplied by the gravity constant [g].
- g*Imy [unit] moment of mass inertia for the rotation φ_Y around the global Y-axis, multiplied by the gravity constant [g].
- g*Imz [unit] moment of mass inertia for the rotation φ_Z around the global Z-axis, multiplied by the gravity constant [g].

Note: *The input of nodal masses refers to the structural nodes and **not** to the centre of gravity of the cross-section! A possibly eccentric location of the mass centroid with respect to the node must be explicitly specified by the user (see below).*

NDMASE **Eccentric concentrated nodal mass**

This Load Type allows to define concentrated masses assigned **eccentrically** to the structural nodes. The masses are specified as forces with possibly different components in the 3 global directions. The masses are assumed to be point masses without own rotational inertia terms. The rotational inertia terms related to the structural nodes are calculated by the program using the eccentricity vector.

Load vector to be entered:

- g*mx [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global X-direction.
- g*my [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global Y-direction.
- g*mz [unit] concentrated mass multiplied by the gravity acceleration [g] acting in the global Z-direction.
- ex [unit] eccentricity of the point mass in global X-direction.
- ey [unit] eccentricity of the point mass in global Y-direction.
- ez [unit] eccentricity of the point mass in global Z-direction.

Sign convention: The eccentricity values are given as vectors from the node to the mass centroid in the global coordinate system. This corresponds to the eccentricity definition of point loads (and is different from the definition of system eccentricities).

The mass matrix used in the program will in this case be established by applying the standard eccentricity transformation to the basic matrix containing the diagonal terms g*mx, g*my and g*mz.

Note: *Differing values of the mass vector components will also influence the rotational terms resulting from the eccentricity transformation.*

NDMASI Full tensor of moments of inertia

This Load Type allows to define the full tensor of mass moments of inertia to describe a propriety mass moment of inertia of a concentric or eccentric point mass.

The translation part of the mass matrix and the coupling terms due to eccentricity are not affected. This Load Type is used in addition to NDMAS or NDMASE, where the mass vector g*mx, g*my, g*mz and the eccentric location of the mass have been specified. The propriety inertia tensor is superimposed to the one calculated in NDMAS or NDMASE.

Load vector to be entered:

- g*Imx [unit] Moment of mass inertia for the rotation φ_X around the global X-axis, multiplied by the gravity constant [g].
- g*Imy [unit] Moment of mass inertia for the rotation φ_Y around the global Y-axis, multiplied by the gravity constant [g].
- g*Imz [unit] Moment of mass inertia for the rotation φ_Z around the global Z-axis, multiplied by the gravity constant [g].
- g*Imxy [unit] Coupling term of mass inertia for the rotations φ_X and φ_Y if the global axes are not the principal axes.
- g*Imyz [unit] Coupling term of mass inertia for the rotations φ_Y and φ_Z if the global axes are not the principal axes.
- g*Imzx [unit] Coupling term of mass inertia for the rotations φ_X and φ_Z if the global axes are not the principal axes.

NDMASA Principal moments of inertia and directions

This Load Type is in fact equivalent to NDMASI, except that the principal moments and the angles describing the principal axes of inertia are specified rather than the full tensor in terms of global coordinates. The full tensor of mass moments of inertia to describe the propriety mass inertia is calculated internally using the direction of the axes.

The rules for establishing the local axes are the same than those for the local element coordinate systems.

As in NDMASI, the translation part of the mass matrix and the coupling terms due to eccentricity are not affected. This Load Type is used in addition to NDMAS or NDMASE, where the mass vector g*mx, g*my, g*mz and the eccentric location of the mass have been specified.

Load vector to be entered:

- g*Imx [unit] moment of mass inertia for the rotation φ_x around the local x-axis, multiplied by the gravity constant [g].
- g*Imy [unit] moment of mass inertia for the rotation φ_y around the local y-axis, multiplied by the gravity constant [g].
- g*Imz [unit] moment of mass inertia for the rotation φ_z around the local z-axis, multiplied by the gravity constant [g].
- Alpha1 angle between the local x-axis and the global X-Z-plane
- Alpha2 angle between the local.x–global.Y-plane and the global X-Y-plane
- Beta angle between the local.x–global.Y-plane and the local x-y-plane

See also chap. 2.4.3 for the definition of local axes!

9.2.3.2 Load Type group 'MASSES – ELEMENT UNIFORM MASS'

All Load Types of this group require the definition of the elements, where the distributed mass should be applied, a switch, defining, whether the mass should only be applied as a mass or also as a load, and a load vector, describing the distributed mass:

- From, to, step series of elements where distributed masses are applied.
- ⊙ Dynamic the specified load vector will only be applied as a mass (no static or dynamic load is generated).
- ⊙ Static + Dynamic the specified load vector will be applied as a mass **and** as a static or dynamic load*.
- Load vector specification of the mass parameters according to the Load Type (as described below).

* _Note:_ _The function for considering a mass also as a load (switch "Static+Dynamic" is not yet implemented. Masses to be considered also as loads must be additionally specified as distributed element loads._

The element masses are related to the element length, i.e. they are given as loads per unit length. The nodal values used in the analysis are calculated by integrating the element mass over the element length and applying half of the total mass to the element start and end respectively, considering only the torsion inertia term and neglecting the rotational inertia terms around the local y and z axes. A standard eccentricity transformation is eventually applied to this local mass matrix, if the position of the element ends is eccentric to the nodes.

The ELEMENT MASS group comprises the following Load Types for the definition of element masses:

ELMAS **Element uniform mass and moments of inertia**.

- g*mx [unit] uniformly distributed mass in **local** x–direction multiplied by the gravity constant [g]
- g*my [unit] uniformly distributed mass in **local** y–direction multiplied by the gravity constant [g]
- g*mz [unit] uniformly distributed mass in **local** z–direction multiplied by the gravity constant [g]
- g*Imx [unit] uniformly distributed moment of mass inertia around the **local** x-axis multiplied by the gravity constant [g]
- g*Imy [unit] not used
- g*Imz [unit] not used

ELMASE **Eccentric uniform element mass**

- g*mx [unit] uniformly distributed mass acting in **local** x–direction multiplied by the gravity constant [g]
- g*my [unit] uniformly distributed mass acting in **local** y–direction multiplied by the gravity constant [g]
- g*mz [unit] uniformly distributed mass acting in **local** Z–direction multiplied by the gravity constant [g]
- g*Imx [unit] uniformly distributed moment of mass inertia around the **local** x-axis (torsion inertia) multiplied by the gravity constant [g]
- ey [unit] eccentricity of the point mass in **local** y-direction.
- ez [unit] eccentricity of the point mass in **local** z-direction.

Attention:	**The eccentricity e_y, e_z defined in ELMASE is not related to the element axis (centroid line), but to the system line (connection line between start and end node). Thus using ELMASE with a zero eccentricity yields a different mass matrix than ELMAS if the system line differs from the element axis.**

9.2.3.3 Load Types 'UNIFORM LOAD – SELF WEIGHT'

G	**Selfweight – load and mass**
G0	**Selfweight – just as load**
GM	**Selfweight – just as mass**

These Load Types are generally also used for static analyses, therefore the input values are described in detail in chap. 6, Loading. If these Load Types (except G0) are assigned to a dynamic calculation, appropriate masses will automatically be generated and used in the analysis.

The generated masses are equivalent to concentric element masses as defined with ELMAS, where the mass terms g*mx, g*my and g*mz are calculated by multiplying the specific weight GAM with the cross-section area A of the element (the average area, if start and end cross-sections differ). The direction vector is not used for the determination of the masses, all three components will have the same value g*mx = g*my = g*mz = GAM * A.

The torsion mass inertia g*Imx will be calculated by multiplying GAM with the polar moment of inertia of the cross-section Ip, being the sum of the bending moments of inertia Iy and Iz (Ip = Iy + Iz).

g*Imx = GAM * Ip

9.2.3.4 Load Case specification

The Load Sets containing the specified masses are assigned to Load Cases in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LCASE. Note, that all loads of these Load Sets, which are not specified by the above described 3 load types, will not be used for the calculation of the mass matrices in the dynamic analysis. This allows to include loads in the same Load Set, which should not be taken into account as masses, but only as loads in the static or in the time history analysis.

Whereas the definition of the masses is sufficient for the eigenvalue calculation, the time history analysis requires additionally the definition of time dependent loads (and possibly masses). These loads may be defined by any Load Type offered in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET. These Load Sets describing the time dependent loads must be included in the same Load Case with the definition of the mass distribution.

2 multiplication factors may be assigned to the specified Load Sets to factorise the related loading. Besides the factor 'Const-Fac' for the static analysis (multiplication factor

for loads) a variable factor 'Var-Fac' can be assigned to the Load Set. This "dynamic"-factor is defined as an arbitrary expression giving a constant value as well as a variable value – normally expressed as a function of time (see ⇧PROPERTIES ⇨VARIABLE).

Note: *The specified factors will be applied on the whole specified load set. That means, that masses* ***and*** *loads will be factorised in the same manner, if they are in the same Load Set. As the masses (or at least the greatest part of the masses) is constant in time, the rolling masses and loads must usually be placed in a different Load Set.*

The Load Set will be multiplied by the sum of 'Const-Fac' and 'Var-Fac' (LoadSet * [Const-Fac + Var-Fac]).

Once the Load Case for the dynamic analysis is specified, it will be assigned in the function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION to the required "Dynamics" Actions (E.g. 'EIGEN' or 'TIME HISTORY').

9.2.4 Damping matrix

The damping matrix is only required when a time stepping analysis is performed. The calculation of Eigenfrequencies and Eigenforms is performed on the undamped system, the modal method for the analysis of forced vibrations or of the earthquake response behaviour requires only the percentage of the critical damping (see further below).

The damping matrix is generally represented as a linear combination of [M] and [K]:

$$[C] = \alpha * [M] + \beta * [K]$$

α and β are the so-called Rayleigh-damping coefficients for which the following equation is valid:

$$\xi_i = \frac{\alpha + \beta\,\omega_i^2}{2\,\omega_i}$$

ξ_i is the valid damping constant for the eigenfrequency ω_i (some authors use the variable ζ for the damping constant). It expresses the actual damping related to the critical damping and is in literature often given in percent. The smallest value ξ_{min} results, when $\omega = \omega_{min}$, where

$$\xi_{\min} = \sqrt{\alpha \cdot \beta}$$

$$\omega_{\min} = \sqrt{\frac{\alpha}{\beta}}$$

Often the logarithmic decrement $\delta = 2\pi\xi(1-\xi^2)^{-1/2}$ or the linearised value $\Delta = 2\pi\xi$ is used. This value describes the decrease of the amplitude within one period and is also mostly given in percent. *RM2000* uses the damping constant ξ (input as factor, not in percent) in the modal analysis, whereas either the Rayleigh-coefficients α and β or the damping constant ξ_i for 2 frequencies ω_i or the minimal value ξ_{min} for ω_{min} must be specified in the time stepping procedure.

As the damping values are generally very hardly measurable, the Rayleigh-coefficients are mostly recomputed from 2 estimated damping percentages using the above equations.

E.g.: If we use ω_1=4 rad/sec with ξ_1=3% and ω_2=17 rad/sec with ξ_2=12%, we get the following equation system:

$$\alpha + \omega_1^2\beta = \alpha + 16\beta = 2*\omega_1*\xi_1 = 0.24$$
$$\alpha + \omega_2^2\beta = \alpha + 289\beta = 2*\omega_2*\xi_2 = 4.08$$

In this case the result will approximately be: $\alpha=\beta=0.015$.

When using the minimum values ξ_{min} and ω_{min}, α and β can be recomputed from the equations

$$\alpha = \xi_{min}\omega_{min}$$
$$\beta = \xi_{min}/\omega_{min}$$

It must be noted that theoretically different damping constants may be used for different elements in the direct time integration method. But in the modal analysis the damping parameters must be only one characteristic system value. Otherwise the orthogonality condition of the total damping matrix to the eigenvectors is not valid anymore.

The damping parameters as characteristic system values are defined in *RM2000* in the menu ⇧RECALC ⇩DYNAMIC. Damping values related to the different elements of the system are entered in ⇧STRUCTURE ⇨ELEMENT ⇩TIME. The damping behaviour is either independent from the Eigenfrequencies and described by the variable Xsi representing the logarithmic decrement, or dependent on the Eigenfrequencies, described by the value pair α and β, which may be directly entered by the user or automatically calculated from value pairs ξ_1, ω_1 and ξ_2, ω_2 using the above mentioned equations.

9.3 Eigenvalues and Eigenforms

9.3.1 Mathematical Background

In the special case of un-damped free oscillation the damping term [C]*{a} and the force vector {f} disappear.

In this case

$$a = a_0 \cos \omega t \ (\omega = \text{frequency})$$

is valid and thus

$$\ddot{a} = -\omega^2 a_0 \cos\omega t$$

By using this relation we obtain the homogeneous system of equations

$$([K] - \omega^2 [M]) \{a\} = 0$$

This equation describes a general Eigenvalue problem.

The square roots of the eigenvalues represent the natural angular frequencies of the structure. The eigenvectors corresponding to the natural frequencies represent the natural modes or mode shapes in which the structure can vibrate.

Mathematically, an *N degree of freedom equation system* has exactly *N eigenvalues* and *N eigenvectors* associated with it. However, in practise only few (low) eigenfrequencies give a substantial contribution to the deformation behaviour of the structural systems. Therefore only the lowest eigenvalues will normally be determined. I.e. only a limited frequency region has to be investigated in nearly all practical applications of dynamic analyses.

The user therefore specifies upper and lower limits for the range of frequencies interesting for the specific problem to be analysed. A simple iterative algorithm is used to find all eigenfrequencies in the given frequency range. This algorithm is based on calculating the determinant of the dynamic system for different frequencies. A changing sign of the determinant value indicates an eigenfrequency. The user has to specify a 'step' for the iterative variation of the frequency and the 'tolerance' for defining the convergence limit.

The related eigenvector will then be computed for each determined eigenfrequency. For an eigenfrequency the system of equations is theoretically singular. To compute the associated eigenvector, the value for one degree of freedom is set to '1' (node no. and degree of freedom are either chosen automatically by the program or defined by the user). This eliminates the singularity of the matrix and the solution gives a resulting eigenvector. This vector will afterwards be normalized with respect to the maximum value (the maximum value will be 1 instead of the one used in the solution process).

9.3.2 Calculation of Eigenfrequencies in RM2000

Eigenvalues and eigenfrequencies (also called natural frequencies) are calculated on the undamped structural system. Thus no damping parameters need to be specified prior to performing this analysis.

The active masses for the eigenvalue analysis must be defined under a single Load Set or a group of Sets that are combined into a single Load Case to be used in the analysis (see previous section). In the function ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION this Load Case will be assigned to the Action 'EIGEN' for the eigenvalue calculation.

- Eigen Natural modes of the system

The following input parameters must be defined for this Action:

- Inp1: Number of natural modes The user can select how many natural modes the program should find (a warning will appear if less natural modes than selected are found while performing the calculation action).
- Inp2: Reference LC Number of the Load Case containing the mass definitions. An interactive selection is possible when clicking the arrow button next to the input field.
- Out1: Output file (*.mod) Name of the binary file containing the results. The name **must** start with 'eig' followed by an unused Load Case number (e.g. the next free number) and end with the extension '.mod' (e.g. eig1001.mod). The detail results can be transformed into ASCII format using the 'DOLIST' action.
  When there is a subsequent response spectrum analysis this file name must also correspond to the input for the seismic combination rule defined in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC.
- Out2: List-File Name of the list file containing internal forces and displacements for the reference Load Case as well as the mass participation factors and frequencies of all the found natural modes.
- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

The result of each natural mode (vibration shape) is stored by the program in the 'Out1' file (eigxxx.mod) and can be viewed in ASCII format using the DOLIST function described above.

The deformation results from the eigenvalue analysis are automatically stored under loading case numbers for plotting purposes. The automatic numbering system assigns the 'number that is used for Out1' described above (e.g. 1001) for the first case (eigenmode 1 is stored in lc1001) and the results for the subsequent eigenmodes are stored under the ensuing loading case numbers (i.e. incremented by 1).

All natural modes can also be presented graphically by using the function ⇧RESULTS ⇨PLSYS where the deformed shape for each natural mode (selected by the assigned filename) can be plotted.

9.4 Modal Analysis – Damped Vibrations

9.4.1 Mathematical Background

The basic differential equation for the movement of an elastic structure with viscous damping has been shown in chap. 9.1. For small damping, it may be assumed that the eigenfrequencies are independent of the damping. They will therefore be computed for the un-damped system (see chap. 9.3.1). The computation and storing of the eigenvalues is described in chap. 9.3.2.

The eigenvalues of the system are collected in the modal matrix [φ], and the eigenfrequencies are stored on the diagonal of the otherwise empty matrix [ω]. Additionally the "generalized mass" is defined:

$$[\, M_G \,] = [\, \varphi \,]^T . [\, M \,] . [\, \varphi \,]$$

There exists an orthogonality relationship for the eigenvectors. Therefore the matrix $[M_G]$ will have non-zero terms only in the diagonal. Due to this orthogonality it is also possible to represent any deformed shape {a} of the structural system as a linear combination of the eigenvectors:

$$\{a\} = [\, \varphi \,] . \{\, q \,\}$$

{q} contains the associated proportionality factor for each eigenvector. These are called **'normal coordinates'**.

Using this definition, the partial differential equation for {a} may be rewritten into an equivalent ordinary differential equation for the normal coordinates (the derivation is not shown in this manual).

This differential equation is:

$$[M_G]*\{\ddot{q}\}+[\varphi]^T*[C]*[\varphi]*\{\dot{q}\}+[K]*\{q\}=\{F\}*[\varphi]$$

When the damping is neglected, this differential equation consists of matrices with non-zero terms on the diagonal only. The different rows of the system of equations, representing the individual eigenvectors, are consequently no more coupled. That means, that we can now consider independent equations, each of them describing the time dependent variation of one normal coordinate q, valid for one related eigenvector.

Instead of the original system, a set of single mass oscillators is considered, each of them defined by mass and stiffness adjusted such that their eigenfrequencies coincide with those of the original system. The solution for all the single masses may then be substituted in the equation

$$\{a\} = [\varphi] * \{q\}$$

and used to solve the original system.

When damping is specified for the single mass system the influence of damping can be considered in a simple way. In general damping it is defined by the logarithmic damping decrement (=difference between the amplitudes of two subsequent oscillations in percent).

Then it is no longer necessary to compute the damping matrix [D] of the complete system. This makes treating the damping much easier, considering also that in most practical cases it will hardly be possible to specify or compute the necessary coefficients of the full damping matrix. The damping matrix in the differential equation is now replaced by a matrix with non-zero terms on the diagonal only. These coefficients are conforming with the logarithmic damping decrement and can be specified globally or for each eigenvector (modal damping).

A further advantage of the modal method is, that only a limited number of eigenfrequencies are usually necessary to be taken into consideration. It can be shown, that eigenfrequencies located far enough away from the exciting load frequencies do not give a substantial contribution to the total result. Only eigenfrequencies near to the excitation frequencies have therefore to be determined and evaluated.

9.4.2 Forced Vibrations (by harmonic loading)

The calculation of forced vibrations is based on the modal method.
This function is not yet implemented in *RM2000*.

9.5 Earthquake Analysis using the Response Spectrum Method

9.5.1 General

The Response Spectrum Analysis is based on the modal method. This method is one of the methods for the solution of the general motion equation shown at the beginning of this chapter. The mathematical background is described in the previous chapter 9.4.

It is characterized by the procedure, that the coupled system of equations representing the general equation of motion, will be decoupled by modal transformation. We get the decoupled differential equation system:

$$m_i \ddot{q}_i + 2\, m_i \xi_i \dot{q}_i + \omega_i^2\, m_i q_i = -\, P_i(t) \qquad , i = 1,\ldots,n$$

We now assume that the solutions for the above equations are given in terms of a response spectrum. I.e. the amplitude values of accelerations, velocities or displacements respectively are known for the n eigenforms. If this is the case, then the response of the actual system may be determined by suitably combining the responses of the different eigenvectors.

Such “response spectra” for an irregular dynamic excitation (e.g. earthquake) are commonly specified in the relevant national design codes (general response spectra) or available from national authorities or research institutes (spectra for local earthquake events).

First of all a load vector must be specified. For earthquake analyses mostly value and direction of the ground acceleration are known. Sometimes alternatively ground displacement or ground velocity may be given. In those cases it is easy to compute the corresponding acceleration (D_=V, V_=A). The load vector can then be setup by multiplying the mass matrix of the structural system by this acceleration. For other applications the load vector itself may be known and can be specified directly.

Response spectra are diagrams, where for each frequency (abscissa value) a related amplitude (ordinate value = deflection, velocity or acceleration) is defined.

The response spectrum values are interpreted by the program as "single mass solution" for the eigenfrequency according to the modal superposition method. For each eigenvector a normal coordinate is computed. The different contributions are superimposed.

As the response spectrum gives no information about the phase shift of the different exciting frequencies, the individual contributions must be superimposed using stochastic principles.

9.5.2 Combination rules for seismic analysis

This combination is a straight forward process adding the contributions of the eigenvectors determined by the modal transformation. However, there exists one severe difficulty for the superposition of the factorised eigenvectors: the response spectra do not give any information on phase differences between the different eigenvectors. Therefore appropriate probabilistic techniques have to be used for the superposition. The response-spectrum method may therefore be described as a half-deterministic procedure.

Several approaches for this combination procedure are found in literature, e.g.:

a) **ABS-Rule** (sum of absolute values)

The total response is computed by adding the absolute values of all individual contributions.

$$R_{ges} = \sum_{i=1}^{n} | R_i |$$

R_t Total response
R_i Individual response for eigenvector no. i

This rule assumes full correlation between the different eigenfrequencies, all maxima are reached at the same time. This is an upper limit and a very unfavourable estimate. The ABS-rule is suitable for structures, where the relevant eigenvalues are situated close to each other.

b) **SRSS-rule** (Square Root of Sum of Squares)

This rule supposes, that the individual eigenfrequencies are completely uncorrelated.

$$R_{ges} = \sqrt{\sum_{i=1}^{n} R_i^2}$$

The contributions R_i of the different eigenfrequencies are added in 'Pythagorean' manner. This rule is used in many existing design codes and will give good results if the considered eigenfrequencies are distributed over a wide range of frequencies and they are not situated too close one to each other.

c) **DSC-rule** (Newmark/Rosenblueth)

This rule takes into account that a correlation exists between the contributions of the individual eigenfrequencies.

$$R_{ges} = \sqrt{\sum_{i=1}^{n}\sum_{j=1}^{n} \rho_{ij}\ R_i R_j}$$

with $$\rho_{ij} = (1 + (\frac{\omega_i{}' - \omega_j{}'}{\xi_i{}'\omega_i + \xi_j{}'\omega_j})^2)^{-1}$$

ω_i, ω_j two of the considered eigenfrequencies
ζ_i, ζ_j modal damping of those eigenfrequencies

The rule allows the user to take into account different damping for different eigenfrequencies. This makes however only sense if additional information specifying the frequency dependency of the damping is available. But in most cases the response spectrum is valid for one single global damping value.

The theoretical base of the rule also allows to take into account that the correlation between two eigenfrequencies depends on the duration of the earth quake:

$$\zeta_i = \zeta_i^* + \frac{2}{s\,\omega_i}$$

s duration of the earthquake
ζ_i^* original modal damping or global damping on which the response spectrum is based
ζ_i modified modal damping to be used in the above formula defining ρ_{ij}

It is obvious, that the influence of the duration of the earthquake is very small for higher eigenfrequencies.

A detailed description of the theoretical background is given in 'Fundamentals of Earthquake Engineering' by Newmark/Rosenblueth, Prentice-Hall, 1971.

d) **CQC-rule**

This rule is based on a more complex theory modelling the correlation between the different eigenfrequencies. It gives good results, if the duration of the earth quake is at least 5 times higher than the longest considered period of vibration.

$$R_{ges} = \sqrt{\sum_{i=1}^{n}\sum_{j=1}^{n} \rho_{ij} R_i R_j}$$

with $$\rho_{ij} = \frac{8\sqrt{\xi_i \xi_j}(\xi_i + r\xi_i) r^{3/2}}{(1-r^2)^2 + 4\xi_i\xi_j(1+r^2) + 4(\xi_i^2\xi_j^2) r^2}$$

and $r = \omega_j/\omega_i \quad (\omega_j < \omega_i)$

A detailed explanation of the theoretical background is given in 'A response spectrum method for random vibration analysis of MDF systems' by Armen der Kiureghian, Earthquake Engineering and structural dynamics, Vol. 9, 419-435 (1981).

Unless which rule is used in the analysis it is the user´s responsibility to make sure, that a sufficient number of eigenfrequencies is considered in the analysis. In order to help the user to check this requirement the program computes the '**effective mass percentages**' for the X, Y and Z directions of the global coordinate system.

$$\% EM_x = \frac{(\sum M_x \phi_x)^2}{\sum M \phi^2} * \frac{100}{\sum M_x}$$

$$\% EM_y = \frac{(\sum M_y \phi_y)^2}{\sum M \phi^2} * \frac{100}{\sum M_y}$$

$$\% EM_z = \frac{(\sum M_z \phi_z)^2}{\sum M \phi^2} * \frac{100}{\sum M_z}$$

where

$$\Sigma M\phi^2 = \Sigma M_x\phi_x^2 + \Sigma M_y\phi_y^2 + \Sigma M_z\phi_z^2 + \Sigma M_{xx}\phi^2 + \Sigma M_{yy}\phi^2_{yy} + \Sigma M_{zz}\phi_{zz}^2 = 1.0$$

and M_x represents the x-translational mass values and f_x represents the x-translational mode shape components. Similar for M_y, f_y, and M_z, f_z, and respectively for M_{xx}, f_{xx}, M_{yy}, f_{yy} and M_{zz}, f_{zz} as they relate to rotational mass values and rotational mode shape components.

These percentage values reach 100% for all directions if all eigenfrequencies are considered in the analysis.

The formulae c) and d) work with correlation factors between zero and one, where the factors come close to one, when the difference between both eigenvalues goes to zero. If there are big frequency differences, the values of the correlation factors get close to zero.

The formulae a) and b) represent special cases of the general relationship, where in case a) all correlation factors are one, and in case b) $\rho_{ij} = 1$ for i=j and $\rho_{ij} = 0$ for $i \neq j$.

9.5.3 Input of the necessary parameters

The combination rule as well as the unit being related to the horizontal and vertical axis of the defined response spectrum is selected in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC.

The input window shows 2 tables:

The upper table shows all existing definitions. Select the appropriate line in the upper table and use either the 'Insert before' or the 'Insert after' button to define a new calculation rule ('Delete' and 'Modify' buttons are also available).
The appearing input window requires the following input:

- Number — Load Case number of the seismic load case
- Modal File — Name of the output file containing the results of the modal analysis. The file name is user defined and must be identical to the file name that is selected in the calculation in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION. If such a calculation is already available the user can select the file with the arrow button next to the input field.
- Location — Name of the ASCII file containing the input data (no input possible, name is defined by the program).
- Rule — Selection of the combination rule that is used to combine the different frequencies. Pressing the pull-down arrow allows an interactive selection out of the following combination rules:
  - ABS — The total response is computed by adding the absolute values of all individual contributions
  - SRSS — Pythagorean addition. This rule supposes, that the individual Eigen frequencies are completely uncorrelated
  - DSC — This rule takes into account that a correlation exists between the contributions of the individual eigenfrequencies. The individual damping for each eigenfrequency can be considered.
  - CQC — This rule is based on a more complex theory modelling the correlation between the different eigenfrequencies. It gives good results, if the duration of the earth quake is at least 5 times higher than the longest considered period of vibration.
- Duration — Duration in [sec] – not used in this context
- Description — Descriptive text (max. 80 characters)

The lower table contains the definition of the mass vector, the damping coefficient and the assignment of the available response spectrum function defined previously under ⇧PROPERTIES ⇨VARIABLE.

Select the appropriate line in the upper table for the additional definition and use either the ‘Insert before’ or the ‘Insert after’ button to define the additional information for the calculation rule in the lower table (‘Delete’ and ‘Modify’ buttons are also available).
The appearing input window requires the following input:

- ⊙ d the ground displacement is used for the response spectrum
- ⊙ v the ground velocity is used for the response spectrum
- ⊙ a the ground acceleration is used for the response spectrum
- ➢ Vec-Vx global motion components
- ➢ Vec-Vy defining the direction of the
- ➢ Vec-Vz displacement/velocity/acceleration
- ➢ Damp-Fact damping constant ξ (ratio between actual damping and critical damping (see chap. 9.2.4))
- ➢ Var-Name(Graph) name of the formula describing the relationship between the frequency and the motion value (defined in ⇧PROPERTIES ⇨VARIABLE as described below).

9.5.4 Input of a response spectrum diagram

A response spectrum is a diagram describing the relationship between the angular velocity OMEGA (abscissa value) and the related ground motion amplitude (ordinate value).

Response spectra for different geographical regions are generally available from design codes or from the local earthquake investigation institutes or authorities. Mostly they are given in terms of Hertz (instead of the angular velocity OMEGA) and accelerations related to the gravity acceleration value g (factors of g instead of real accelerations). Sometimes the abscissa value might also be given in a logarithmic scale.

Generally the available response spectra may also be given in other units. The frequency may either be given in Hertz [Hz] = Rotations/sec, or as an angular velocity Omega (Radians/sec), or in terms of the period T [sec/rotation]. The response value describing the related ground motion may be given in terms of displacements, velocities or accelerations.

In *RM2000*, the response spectrum diagram must be specified as a named variable, defined in ⇧PROPERTIES ⇨VARIABLE, representing a table or a formula, or even a piecewise valid set of different formulas.

This variable describes the ordinate value as a function of the abscissa value, where the variable may generally be either a ground displacement amplitude, a velocity amplitude or an acceleration amplitude in a certain direction. A switch in the menu ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC described in the above section is provided, to specify whether the assigned response spectrum is given in terms of displacements (d), velocities (v) or accelerations (a). The direction of the described motion is also specified in the same menu.

Attention:	**The ordinate values of the response spectrum must be given in the units internally used in the calculation process [m, sec], (see chapter 2.3.1). A suitable previous transformation has to be applied for spectra given as motion amplitudes in other units, or as factors of the gravity constant.**

The abscissa value must be the **angular velocity OMEGA**, which is defined as an intrinsic variable in *RM2000*. A suitable transformation formula has to be related to the entered abscissa values describing the spectrum diagram, if they are defined in terms of Hz or Periods. Such a transformation rule must also be applied, if the abscissa values are given in terms of logarithms of frequency or period respectively.

Example:
We assume that the Variable describing the Response Spectrum were named **RESP.**
The variable will now have the form

RESP = f (ABSCISSA)

RESP may either be a displacement, velocity or acceleration amplitude. This property is assigned in ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC as described above, and not specified in the variable definition function.

However, in the case that the ordinate value is not specified as a displacement, velocity or acceleration term **(in the internal units!)**, an additional transformation has to be used to describe the response spectrum. This can for instance be done by introducing an additional variable ORDINAT. If for instance the spectrum is entered in terms of factors of g, then the appropriate formulation will be:

ORDINAT = f (ABSCISSA)
RESP = f (ORDINAT) = ORDINAT * g

RM2000 uses internally as abscissa value the angular velocity Omega in [Radians/sec] defined by an intrinsic variable OMEGA. This variable name is reserved and must not be used for other purposes. It is necessary to establish the relationship between this intrinsic variable and the actually used abscissa value, to have the correct spectrum to be evaluated. When the spectrum is given in terms of angular velocities, the appropriate variable definition for assigning OMEGA will simply be:

ABSCISSA = f (OMEGA) = OMEGA

For spectra given in terms of Hertz or the Period T, the appropriate necessary variable definition would be
ABSCISSA = f (OMEGA) = OMEGA/2π or
ABSCISSA = f (OMEGA) = 2π/OMEGA.
Similarly, for spectra given in terms of logarithms of Hertz or the Period T, the appropriate necessary variable definition would be
ABSCISSA = f (OMEGA) = log (OMEGA/2π) or
ABSCISSA = f (OMEGA) = log (2π/OMEGA).

Summary of the required steps to define a response spectrum:

- Select ⇧PROPERTIES ⇨VARIABLE (see chap. 3.6, Structural Properties)
- Define the response spectrum as table or formula as given in the design code (ORDINAT=f(ABSCISSA))
- Define the calculation value of the ordinate as a function of the given ordinate value (RESP = f(ORDINATE))
- Define the given abscissa value as a function of the internal variable OMEGA (ABSCISSA = f(OMEGA))

- Select ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC

- Indicate the type of the ground movement amplitude (displacement, velocity or acceleration)
- Define the direction vector of the ground movement

9.5.5 Performing the Response Spectrum Analysis

The Response Spectrum analysis is performed in the Action **RespS**. Prerequisites for performing this Action are:

- A response spectrum (⇧PROPERTIES ⇨VARIABLE) is defined as a Variable.
- The seismic load (name and type of the response spectrum, direction vector) is defined (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC).
- Natural modes have been calculated (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION ‘Eigen’).
- Initialisation of the superposition file that will contain the results (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION ‘SupInit’).
- Calculate the seismic forces by using the existing response spectrum
- Create List file and/or graphic to view results

9.5.5.1 Superposition file initialisation

Select either the ‘Insert before’ or the ‘Insert after’ button to assign the loading sets to the loading case (‘Delete’ and ‘Modify’ buttons are also available).
Select

- ⊙ Envelope actions and
- ➢ SupInit Superposition file initialisation

The appearing input window requires the following input:

- ➢ Command Program internal name of the action. Can not be modified by the user.
- ➢ Inp1: Input-file (*.sup) Any existing superposition file can be used to initialise and to write the results at the same time into the file defined under ‘Out1’.
- ➢ Inp2: Factor Multiplication factor for file ‘Inp1’.
- ➢ Out1: Output file (*.sup) Name of the binary superposition file containing the results. The results can be transformed into ASCCI format using the ‘DOLIST’ action.
- ➢ Out2: Not in use.
- ➢ Delta-T Duration of the Action (not needed for this Action)
- ➢ Description Descriptive text (max. 80 characters)

9.5.5.2 Response spectrum Action

Select either the 'Insert before' or the 'Insert after' button to assign the load sets to the load case ('Delete' and 'Modify' buttons are also available).
Select

- ⊙ Calculation actions and
- ➢ RespS Response spectrum

The appearing input window requires the following input:

- ➢ Command Program internal name of the action. Can not be modified by the user.
- ➢ Inp1: Number of SEISM Number of seismic load defined under ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩SEISMIC. An interactive selection is available when clicking the arrow symbol next to the input field.
- ➢ Inp2 Not in use.
- ➢ Out1: Output file (*.sup) Name of the binary superposition file containing the results. The file must be initialised before! The results can be transformed into ASCCI format using the 'DOLIST' action.
- ➢ Out2: Not in use.
- ➢ Delta-T Duration of the Action (not needed for this Action)
- ➢ Description Descriptive text (max. 80 characters)

The results are stored in the file defined in 'Out1: Output file (*.sup)' and can be presented graphically using ⇧RESULTS ⇨PLSYS.

9.6 Time Stepping Analysis

9.6.1 General

In many cases the transformation to the modal space is not possible (e.g. different damping behaviour of different structural parts, big damping not allowing the linearisation etc.) or economically not useful (many eigenvectors to be taken into consideration, etc.). In these cases the method of direct time integration can be applied.

The equations are thereby integrated in a numerical step-by-step procedure, without previous transformation. The basic idea is the decomposition into time intervals, where the basic equation is not exactly valid at any time, but only at the beginning and at the end of each time interval. A shape function approximately describes the changing behaviour of the state variables within the time step.

Different procedures are proposed in literature. Basicly, a distinction between explicit and implicit procedures is made. The advantage of the explicit procedures is the simplicity of the algorithms and the small number of arithmetical operations for one time step. The most important disadvantage is, that these procedures are only stable if the length of the time steps stays below some critical value. This value is normally very small, so that a large amount of time steps is usually needed for a sufficiently accurate explicit procedure.

In implicit processes the unknown velocity- and acceleration values at the end of a time step are not only expressed as a function of the known quantities at the beginning of the interval, but also as a function of the unknown deformations at the end of the time interval. Normally these methods are unconditionally stable, but you will get less accuracy or even wrong results if the time steps become too big. Big time steps yield numerical damping effects of the higher-frequency deformation parts.

The Newmark-method turned out to be the most reliable of the implicit methods suggested in literature. This method is therefore implemented in *RM2000*.

2 Parameters δ and α influencing the integration process are needed in the Newmark method. They describe the approximated variation of the state variables within one time step. The following set of these parameters has been proven to give the best results and is therefore provided as a default in *RM2000*:

$$\delta = c1 = 0{,}50$$
$$\alpha = c2 = 0{,}25 * (0{,}5 + \delta)^2 = 0{,}25$$

These values may be changed by the experienced user in the menu ⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩SEISMIC. Note that these constants are notated as c1 and c2 in *RM2000*, in order to avoid mixing them up with the Rayleigh coefficients α and β used for the damping matrix calculation.

The time stepping procedure additionally requires the damping matrices established as a sum of the stiffness and mass matrices multiplied by the Rayleigh coefficients α and β. These values are also defined in the menu ⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩SEISMIC, either directly or by specifying 2 value pairs ξ_1, ω_1 and ξ_2, ω_2 in order to allow the program calculating the appropriate Rayleigh coefficients α and β as described in 9.2.

9.6.2 Defining Loads and Masses as a function of time

9.6.3 Starting the Time History Analysis

9.7 Moving Loads and Moving Masses

The following part shows all the necessary steps for the dynamic calculation of moving loads resp. moving masses on a structure.

9.7.1 General

The corresponding functions can be used supposing that the structural system is completely defined (elements - nodes, material, cross sections, etc.) before starting the input of the necessary data.
The loads (masses) need to be converted into nodal loads (masses). A time dependent load (mass) definition is required for each node of the deck resp. each point where a load (mass) applies on the structure.
The following basic information is needed:

- Velocity of the load (mass)
- Length of the deck (used to calculate the duration of load (mass) application)
- Number of elements representing the deck (load – time diagram for each element)
- Load (mass) intensity

The necessary data is entered in three steps:

- ➢ The definition of all necessary variables
- ➢ Loads depending from the variables (LoadSet and LoadCase) for each node of the structure resp. for each position of the load (mass) on the deck
- ➢ Call of the special action in the LOADS AND CONSTR.SCHEDULE and start of calculation

Note: *The necessary data input for the calculation of Moving Masses and Moving Loads are principally identical, the only difference between the two analyses concerns the load input. A 'moving masses' – calculation needs a mass definition <u>additional</u> to the load definition.*

All these necessary definitions are done at certain positions in the program. Each definition is done interactively in the program, an ASCII file of all definitions can be created for eventual Editor – modifications when selecting ⇧FILE ⇨EXPORT. The names of the files are mentioned in the following description as well.

9.7.2 Variable definition

A typical example for the application of this feature is subsequently used to illustrate the required program input.

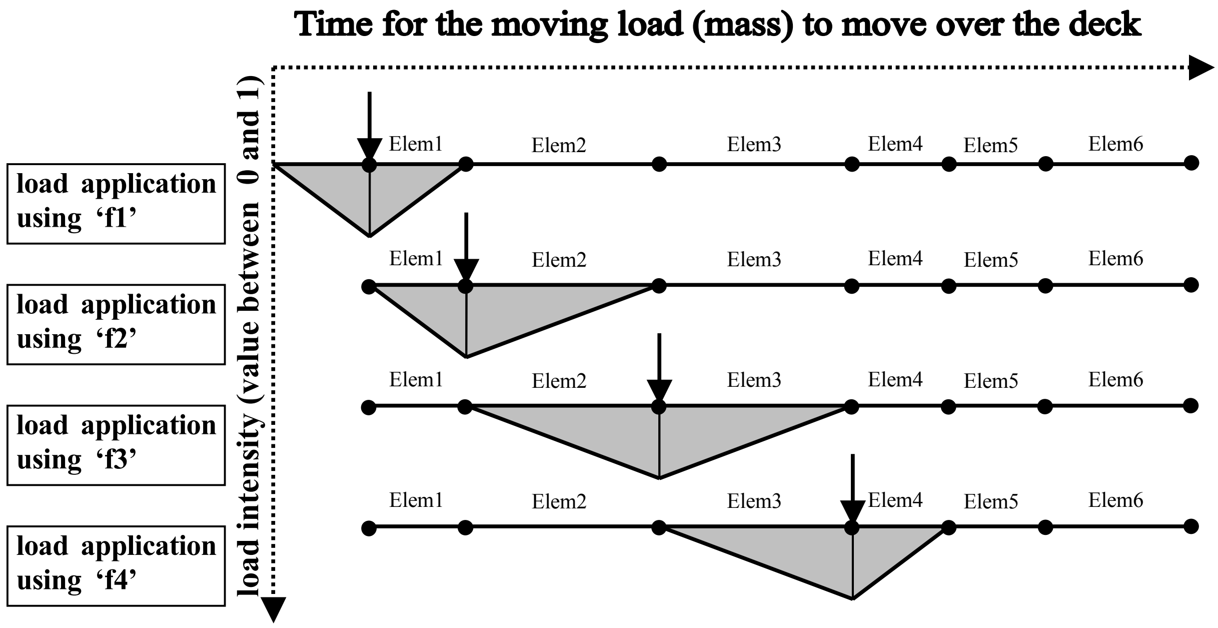
The input of the necessary variables is done under ⇧PROPERTIES ⇨VARIABLES.
The upper table shows all existing variables and formulas. New variables and formulas can be entered by using the 'Insert before' and 'Insert after' button (the 'Delete' and 'Modify' buttons are also available). See also chap. 3.6 for a more detailed explanation of the input procedure.
Each new input is either a formula or a table. A 'NAME' and a 'Description' are required for both, an additional 'Expression' is needed in case of a formula.

- Define the speed of the moving load (mass).
  Input as 'formula': v = 350km/h
- The program requires a [m/s] unit (resp. [feet/s] etc.), therefore we need to define:
  Input as 'formula': vm = v*1000/3600
- Define the length the deck
  Input as 'formula': ltot = 140[m]
- Define the duration for the load (mass) to move over the deck:
  Input as 'formula': tott = ltot/vm
- Define the time step for each element (more then one input if element lengths are varying!!) (35 elements in this example)
  Input as 'formula': dt = tott/35
- Define the time for the time history calculation:
  Input as 'formula': tint = t – tstart

Where 't' is the actual time on the global time axis of the structure (is defined by the duration time input in ⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION for the time dependent actions and where 'tstart' is the program internal expression for the absolute time (relative to day '0' of the LOADS AND CONSTR. SCHEDULE) when a certain action takes place (any position on the global time axis).

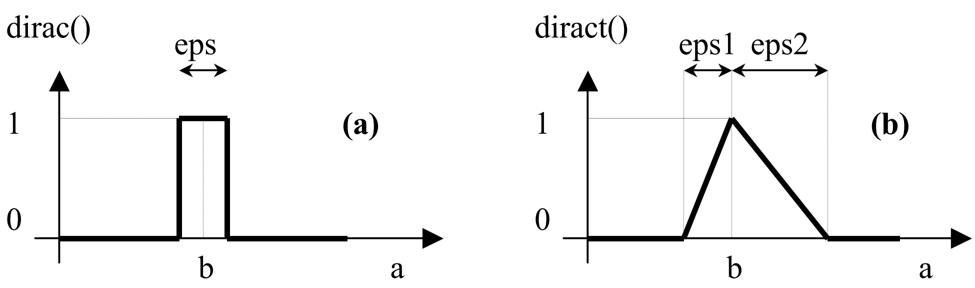
Note: The unit [second] for 't' is required for time history calculations!

Define the functions representing the time dependency of the load application:



f1 = diract(tint,0.0*dt,dt,dt)
f2 = diract(tint,1.0*dt,dt,dt)
f3 = diract(tint,2.0*dt,dt,dt)
etc.

The program internal function for this kind of analysis is 'diract' (diract(a,b,eps1, eps2) = triangular interpolation). A summary of all available program internal functions and variables is given in chap. 3.6, Variables.



9.7.3 LoadSet definition

A LoadSet is defined for each position of the load (mass) on the deck (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩LSET).
Define individual concentrated load sets (example : Fy -240kN for each node of the deck).
Masses are additionally necessary in case of moving masses.
See chap. 6.2, Load Set, for more details.

9.7.4 LoadCase definition

One load case containing all load sets is defined (⇧LOADS AND CONSTR. SCHEDULE ⇨LOADS ⇩LCASE). Each load set requires an additional information for the variable factor (Var-Fac). The constant factor (Const-Fac) for static loading is set to '0'. The actual load is multiplied by the product of [Const-Fac * Var-Fac] if both factors are set.
The single load case contains all loading sets, in our explanation example:

LoadSet for first position:	Const-Fac=0	Var-Fac=f1
LoadSet for second position:	Const-Fac=0	Var-Fac=f2
LoadSet for third position:	Const-Fac=0	Var-Fac=f3
etc.		

9.7.5 Construction schedule

All elements of the structure are active in order to call the action for the time integration. The appropriate position in the upper table showing all available stages is selected and the necessary actions are defined in the corresponding lower table by using either the 'Insert before' or the 'Insert after' button (the 'Delete' and 'Modify' buttons are also available.).

Select
- ⊙ Calculation actions
- ➢ TInt Time history

- ➢ Command program defined name of action – no input
- ➢ Inp1 loading case number of the loading case containing all load (and mass) positions
- ➢ Inp2 delta T = duration of the time history analysis (no relation to any other time definition in *RM2000*)

- Out1 Output file (*.sup) containing the binary results in form of internal forces and displacements. This file must be initialised before using it.
- Out2 List file containing the results in ASCII
- Delta-T Duration of the Action (not needed for this Action)
- Description Descriptive text (max. 80 characters)

9.7.6 Calculation Control

All data is available to run the calculation. Two more definitions are required before actually starting the calculation.

Select ⇧RECALC ⇩DYNAMIC and specify

- dt = time increment for time integration

How many objects allowed? What is presented (time diagram of diplacement, int. force)?

Select ⇧RECALC ⇩DEFAULT.GRP (hit the arrow symbol next to the input field to activate the interactive selection) and specify the wanted result points on the structure to be viewed on the screen.

Each component for each node or element of the structure can be viewed on the screen during the calculation. Use either the 'Insert before' or 'Insert after' button ('Delete' button is also available) to define

- Object Node or element
- From first wanted 'object'
- To increment for wanted objects
- Step last element of the series of wanted objects
- Position only for 'elements', Element Begin and/or End
- Print DOF select the wanted component to be printed in the list file defined in ⇧LOADS AND CONSTR. SCHEDULE ⇨STAGE ⇩ACTION for the 'Tint' action.
- Plot DOF select the wanted component to be plotted on the screen and into a plot file.

The calculation can be started by selecting ⇧RECALC ⇩RECALC

9.7.7 Automatic Load Definition by using TCL

The function ⇧STRUCTURE ⇨SPECIAL "Pre-processor for moving load" allows to define TCL Script Files for specifying moving loads or masses. Using this pre-processor eases considerably the input effort for describing moving loads or masses. An input menu is shown on the screen after selecting this function, where all required definitions may be made (all Load Sets, Load Case, Velocity, temporary loaded elements, type of the loading (nodal load or mass, element load or mass) etc.).

This pre-processor creates a TCL file (name: ‚MLOAD.TCL') and finally new TCL files (*.tcl), which may be imported subsequently. Only the following things have then to be done for performing the analysis:

- Adding the Load Set containing the mass to the new Load Case
- Adding the Aalculation Action TINT to the Action table of the construction schedule
- Definition of the file: default.grp
- Starting the analysis with ⇧RECALC ⇩RECALC

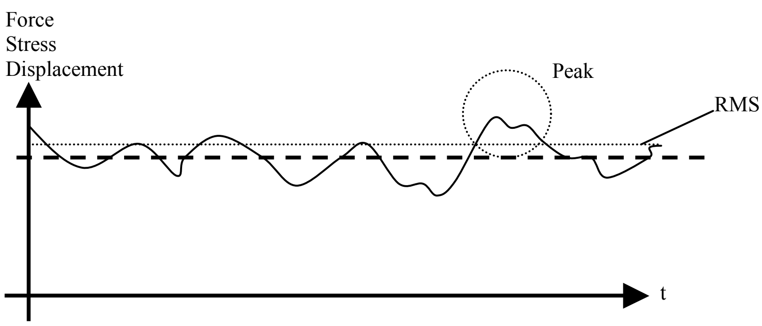
9.8 Wind Dynamics

9.8.1 General

This chapter shows how to proceed for considering dynamic wind loading in the structural analysis with RM2000.

Loading by wind effects is generally a stochastic event, where the intensity as well as the direction is varying with time. The long term variation is usually considered by investigating different stationary cases (mean wind with a constant speed blowing in a certain direction). These different cases are treated separately as different Load Cases, each of them consisting of a static part related to the mean wind load, and a dynamic part related to the short term variation of the wind.

Note: *The design wind speed is in most design codes given as a direction independent value, although the maximum speed is in reality often distinctly different for different directions.*



The static part is defined by the "mean wind", basically characterized by a reference pressure value q (called dynamic pressure or velocity pressure) dependent on the design wind speed. This value is given by the formula

$$q = \frac{1}{2}\rho * v^2$$

where ρ is the mass density of the air, and
v the design wind speed respectively.

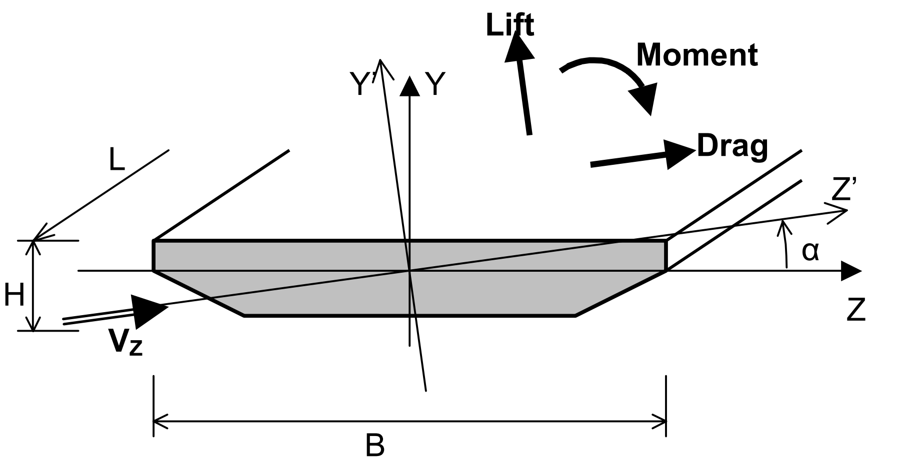
Note: *In many design codes the design value q is directly given instead of the design speed, as a function of the level above the ground surface.*

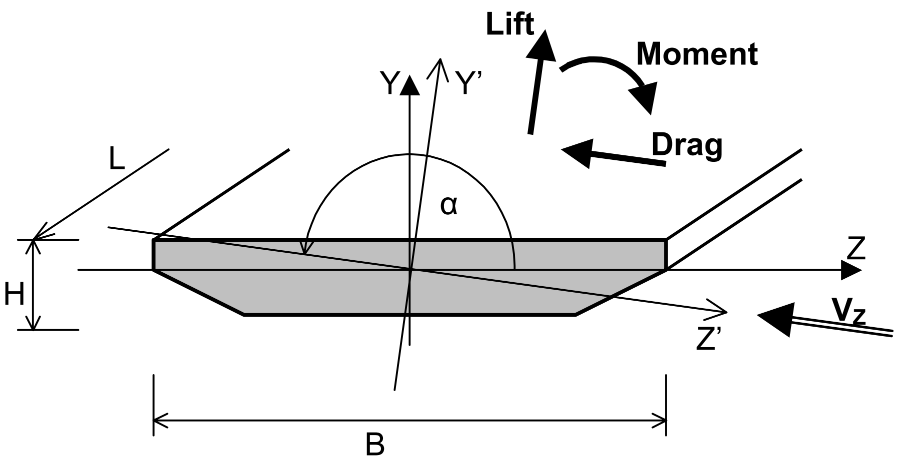
The dynamic part is defined by a characteristic deviation value of the actual wind speed with respect to the mean speed. This deviation is stochastic, different approaches exist for specifying the characteristic deviation values. Most commonly used is the root mean square value (RMS) of the deviation as the basic value for characterizing the time dependent part of the wind loading. Some design codes require to use other values, e.g. the absolute peak value or another limit value based on the probabilistic theory.

9.8.2 Specification of the Static (stationary) Wind Loading

The actual wind load is then calculated by multiplying the dynamic pressure *q* by appropriate factors, called *aerodynamic coefficients*. These coefficients are dependent on the distribution of the aerodynamic pressure on the surface of the structure, representing integral values of the pressure related to a characteristic length.

The aerodynamic pressures yield in general not only forces in the wind direction (drag component q*C_D), but also forces in perpendicular direction (lift component q*C_L) and overturning moments (pitch component q*C_M).

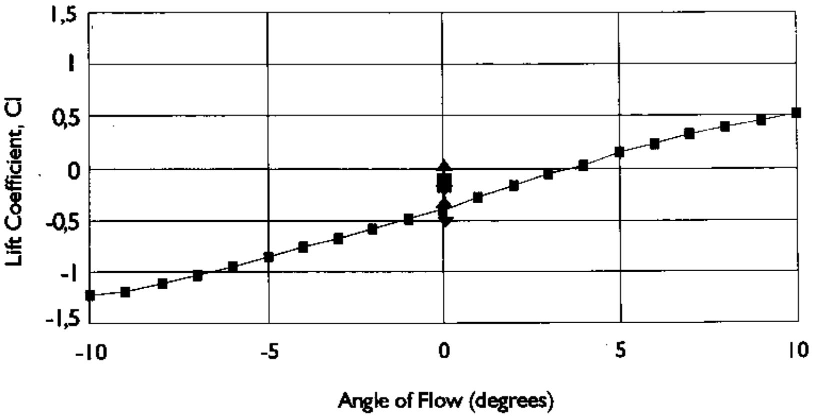


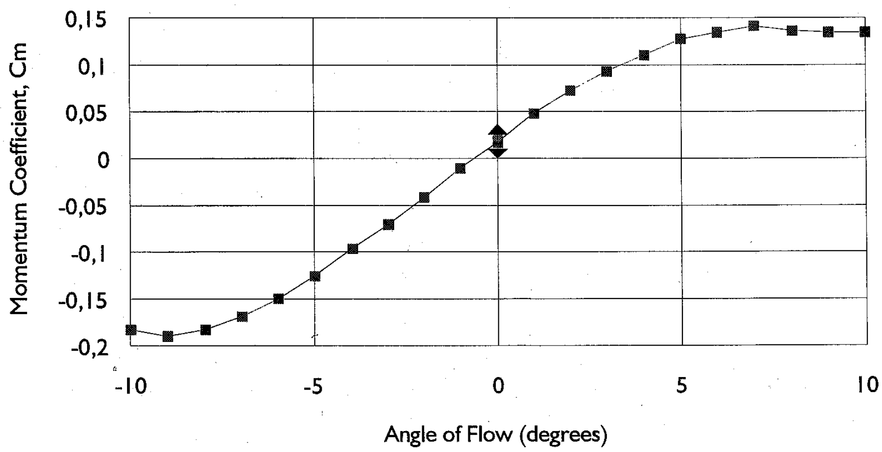


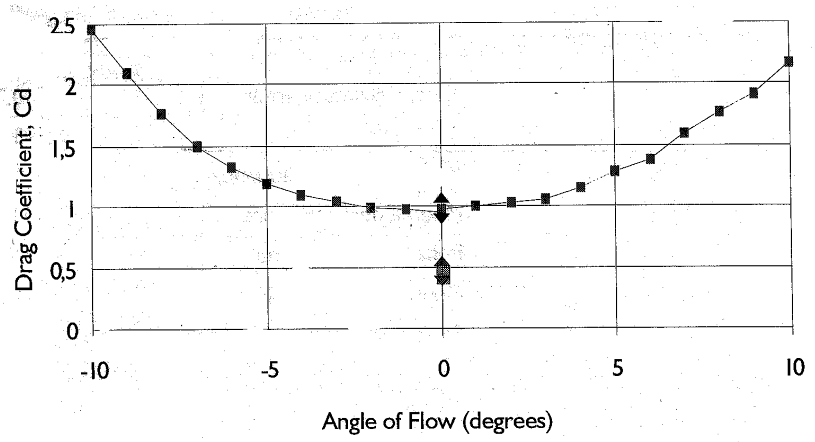
The values of these coefficients depend on the shape of the structure (i.e. of the shape of the cross-section in the case of bridge superstructures), and on the wind direction (the "attack angle" α defined as the angle between the wind direction and the bridge deck surface). Approximate values for these coefficients, found in literature, may be used for minor structures, but for major bridges subject to strong wind (especially for large suspended or cable stayed bridges) sophisticated wind tunnel tests should be performed in order to determine accurate values for these coefficients for the whole range of possible wind directions.

Note: *The dynamic stability behaviour is not only dependent on the coefficients themselves, but – more important – on the variation of these values with changing attack angle α! Considering the tangential deviation of the deck surface due to the overturning moment and the related changes of the loading values may cause vibrations or stability failure even in the case of constant wind conditions.*

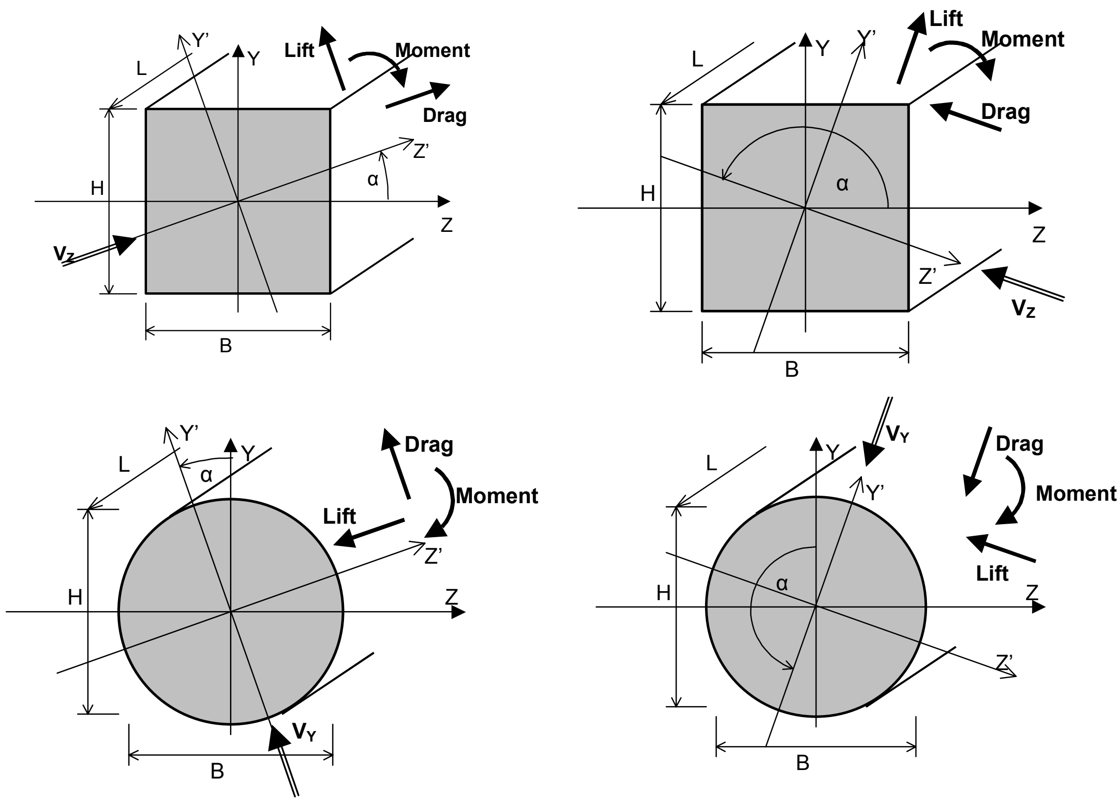
The figures below show the variation of the aerodynamic coefficients for lifting forces (C_L), overturning moments (C_M) and drag forces (C_D) with the attack angle α, for a typical bridge cross-section.







Note: _In case of a deck girder a range of this angle α between –10° and +10° is of interest. For pylons and piers this range might be from 0° to 360°, but symmetric conditions are often helpful._



The above sketches show the direction of lifting forces (lateral forces) as well as the sign of overturning moments due to wind acting in the local z and y directions.

9.8.3 Time Dependent (Dynamic) Wind Loading

9.8.4 Considering Wind Effects in RM2000

The total analysis can be split into 3 parts:
(1) Static analysis for the “mean wind”
  - the results are stored as a standard Load Case

(2) Dynamic spectral analysis of “wind turbulences”
  - the results are stored in a superposition file – with RMS (Root Mean Square) of displacements and forces

(3) Maximal expected values:
  - Result is a superposition file = (1) + [Peak factor *(2)]

Steps (1), (2) and (3) need to be executed for each different wind direction. The results should be combined following the stochastic rules.

The input for ***RM2000*** consists of the following steps:
- Input for aerodynamic classes of cross sections
- Assignment of aerodynamic classes to the elements
- Wind load input
- Calculation actions
- Correct application in the construction schedule

All this input is done at different locations in the program.

All (interactive) input can also be done in form of ASCII input together with file operations. These operations are available under ⇧FILE ⇨EXPORT.

9.8.5 Aerodynamic Cross-section Classes – Shape Coefficients

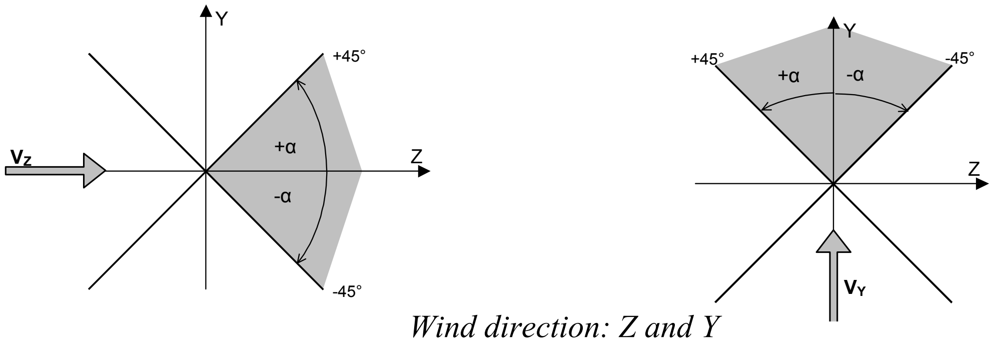
9.8.5.1 Wind Directions on cross-sections

This input is done under ⇧PROPERTIES ⇨AERO_CL. The upper listing shows the available Aerodynamic cross section classes. Existing definitions can be modified or deleted, new definitions can be inserted using the known buttons for Insert before and Insert after:
- Aero CS No.
- Description

Multiplication dimensions – choose between width (b) and height (h)
- ⊙ Drag
- ⊙ Lift
- ⊙ Moment

The lower listing allows the input and/or manipulation of the drag/lift/moment coefficients. A coefficient in the local (cross section related) Z and Y direction is required for Drag, Lift and Moment. Existing definitions can be modified or deleted, new definitions can be inserted using the known buttons for Insert before and Insert after:

- Alpha direction of wind
- C_x Coefficient for Drag, Lift, Moment



Wind direction: Z and Y

Note: *The only interesting direction for deck cross section is the (local) Z-direction, due to symmetry condition reason. For pylon and pier cross section the local Y- and Z-direction are considered.*

The user defines a table for Drag, Lift and Moment. The complete input can be checked graphically by using the button above the upper table. The defined input can be viewed as a curve. Existing definitions from previous projects can be imported as well.
The graphic shows, that the order of input is important!

9.8.6 Element – assignment of aerodynamic cross section classes

This input handles the assignment of available aerodynamic coefficients to elements. Theses coefficients might be different for several elements of the structure. The aerodynamic class can be specified to either a set of elements or an individual element.
The input is done under ⇧STRUCTURE ⇨ELEMENT ⇩CS.
The element must be existing before this input can be done! Hit the 'Edit' button to activate the cross section definition window.

- Select wanted set of elements (from – to – step)
- Select „CS class" and
- assign wanted (and available!) aerodynamic CS class (defined under ⇧PROPERTIES ⇨WINDCS)

9.8.7 Input of Wind Loading in Load Set

The load intensity for the actual wind load is defined under ⇧CONSTRUCTION SCHEDULE ⇨LOADS ⇩LSET
The upper listing shows all available Load Set definitions, new Load Sets can be entered and existing ones modified/deleted by using the known buttons above the listing.
Select 'Insert before' or 'after' at the appropriate position and define:

- Load set number
- Load Case to which the Load Set is assigned to (if wanted, select above!)
- Enter description for load set for better identification

The lower listing allows the definition of the actual loading. Select either the 'Insert before' or 'Insert after' button and choose

⊙ Wind loads

The following load types are available:

- Mean wind
- Mean drag
- Mean lift
- Mean pitch

9.8.7.1 Mean wind

Mean wind is like a Macro and covers all three others. This can only be selected, if all definitions under ⇧PROPERTIES ⇨WINDCS and ⇧STRUCTURE ⇨ELEMENT ⇩CS are available!
If not, Mean drag, lift and pitch can be entered individually.

Define

- Wanted element series (from – to – step)
- Wind direction vector
- Number of Wind

Remark:	**Dynamic effects caused by wind can also be calculated without Cross section and Element specific aerodynamic data:**

9.8.7.2 Mean drag

Define

- Wanted element series (from – to – step)
- Density and attack angle (Ro and Alpha)
- Wind velocity at element begin and end (v-beg, v-end)
- Define drag coefficient (cd)
- Multiply load intensity either by cross section width or height (Qy, Qz)

9.8.7.3 Mean lift

Define

- Wanted element series (from – to – step)
- Density and attack angle (Ro and Alpha)
- Wind velocity at element begin and end (v-beg, v-end)
- Define drag coefficient (cd)
- Multiply load intensity either by cross section width or height (Qy, Qz)

9.8.7.4 Mean pitch

Define

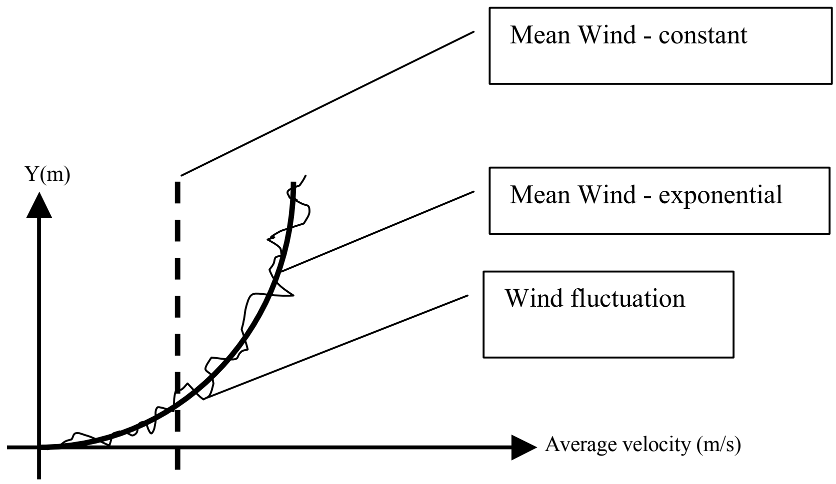
- Wanted element series (from – to – step)
- Density and attack angle (Ro and Alpha)
- Wind velocity at element begin and end (v-beg, v-end)
- Define pitch coefficient (cp)
- Multiply load intensity either by cross section width or height (Qy, Qz)

9.8.8 Wind Load Definition

RM2000 needs 4 general describing parameters for Wind Loads. The corresponding input is done under ⇧CONSTRUCTION SCHEDULE ⇨LOADS ⇩WIND.
The upper listing shows all available load interpretations (use available buttons for insert, modify and delete actions), the part where usually the lower listing is located handles the input of

- Mean wind velocity
- Wind Turbulence Intensity
- Wind Fluctuation Spectrum
- Coherence data

9.8.8.1 Mean velocity

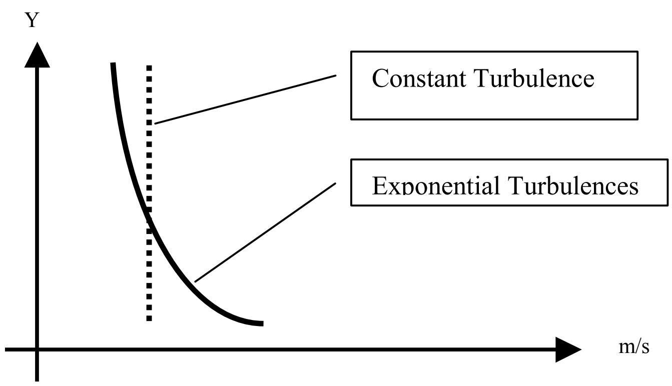


The user can switch between
- Exponential flow
- Constant flow by clicking the arrow key next to the input field.

The corresponding variables can be defined, we refer to the technical description for further explanations.
- Windd
- Cc
- Vref
- T(s)
- Alpha
- Y0
- Yref

These variables define the velocity curve either for an exponential or constant function

9.8.8.2 Wind Turbulence Intensity



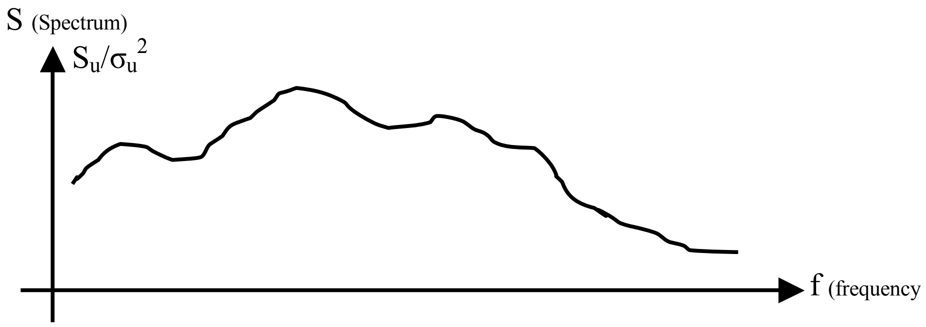
The ‘friction’ of the wind close to the ground is usually greater than the ‘friction’ further away from the ground.
The Wind Turbulence Intensity is relative to the Mean Wind

The user can switch between
- Constant
- Exponential low

By clicking the arrow key next to the input field.
The corresponding Factors can be defined, we refer to the technical description for further explanations.
- Fact Z0 resp.
- W-length
- W-vert
- W-lateral
-

9.8.8.3 Fluctuation Spectrum

RM2000 offers currently the Kaiman spectrum and a constant fluctuation spectrum



σ….. Turbulence intensity define above

The user can switch between
- Kaiman spectrum (one value and the spectrum is defined) and
- Constant (White noise) spectrum (constant spectrum value in 3 directions)

9.8.8.4 Coherence data

The user needs to identify the correlation between simultaneous load applications of the wind at several locations on the structure by defining the type of Coherence law.

The user can switch between
- ➢ Full coherence (everywhere, everything simultaneously) and
- ➢ Coherence type 1 (using Decay constants) – matrix 3*3

Once a ‘Wind Number’ is defined in the upper listing it is possible to change any definition in the lower block by clicking the ‘Modify’ button (only active button).

9.8.9 Construction Schedule actions

The necessary calculation actions need to be added into the construction schedule at the appropriate position once all cross section, element and load specific data is available.
The actions are defined under ⇧CONSTRUCTION SCHEDULE ⇨STAGE ⇩ACTIONS.
Select
⊙ Calculation actions
to view the wind dynamic actions, too.

9.8.9.1 Action FFT

➢ Command	name of action, cannot be changed
➢ Inp1 :	insert table
➢ Inp2 :	time step
➢ Out1:	table containing the spectrum
➢ Out2:	list file
➢ Delta-T	Duration of the Action (not needed for this Action)
➢ Description	Descriptive text (max. 80 characters)

This action is not needed if empirical spectrum are used!

All results (available after calculation started with ⇧RECALC) can be viewed via the created listing file (*.lst) or interactively under ⇧RESULTS ⇨ENVELOPE.
Any result from a superposition file can also be viewed graphically using ⇧RESULTS⇨PLSYS.

9.8.9.2 Action Excitation Spectrum

- Command — name of action, cannot be changed
- Inp1: — select loading case and number of spectrum to be used
- Inp2: — select modal file (eig3001.mod) and damping constant (e.g. 0.03 for 3%)
- Out1: — name of initialised (action SUPINIT) superposition file containing the results
- Out2: — name of listing file, * means default name
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

All results (available after calculation started with ⇧RECALC) can be viewed via the created listing file (*.lst) or interactively under ⇧RESULTS ⇨ENVELOPE.
Any result from a superposition file can also be viewed graphically using ⇧RESULTS⇨PLSYS.

9.8.10 Action Wind – calculation of wind turbulences with aerodynamic effects

- Command — name of action, cannot be changed
- Inp1: — Wind number, Modal File
- Inp2: — Damping value, dx, dy, dz
- Out1: — name of initialised (action SUPINIT) superposition file containing the results
- Out2: — name of listing file, * means default name
- Delta-T — Duration of the Action (not needed for this Action)
- Description — Descriptive text (max. 80 characters)

All results (available after calculation started with ⇧RECALC) can be viewed via the created listing file (*.lst) or interactively under ⇧RESULTS ⇨ENVELOPE.

Any result from a superposition file can also be viewed graphically using ⇧RESULTS⇨PLSYS.

10 Results

10.1 General

Results can be grouped in the categories
- intermediate results,
- structural analysis results and
- design check results.

Intermediate results are values (parameters) needed for the calculation and due to the complexity of their determination procedure not entered by the user, but calculated internally in RM2000 using other more basic parameters. Some of these intermediate results – which are interesting for the user for the evaluation of the structural behaviour – may also be output. This is often very helpful to judge the structural behaviour or the plausibility of the calculation results.

Intermediate results are for instance:
- cross-section values
- time dependent creep & shrinkage coefficients
- time dependent Young's modulus
- influence lines

Calculation results are (in a static analysis):
- deformations
- internal forces
- stresses

Dynamic analyses give additional results:
- eigenvalues and eigenforms,
- amplitudes of deformations, velocities, accelerations, internal forces and stresses in a modal analysis,
- time dependent deformations, velocities, accelerations, internal forces and stresses in a time step calculation.

Design results are:
- comparison of actual with allowed values (e.g. stresses)
- cracking moment
- needed reinforcement

During the calculation RM2000 produces a set of List Files including most of the input data, intermediate results, calculation results and design check results. In addition to this output the function RESULTS provides an application showing selected results on diagrams or in lists.

10.2 Automatically generated result lists

Following list files are generated automatically:

Intermediate results:

Struct.lst	geometry-data cross-section data from element to element material data from element to element geometry of the tendon, etc.
Cross.lst	detailed arrangement of the cross-section values
Material.lst	detailed arrangement of the used materials
Stress.lst	arrangement of the pre-stressing actions (pre-stressing force)
Lanexxx.lst	data of traffic loads

Calculation results:

LCxxx.lst	deformations and internal forces of the load case xxx

Design results:

FBxxxx.lst	Fibre Stress Checks
Ultxxxx.lst	Ultimate Load Checks

The following intermediate results are not listed in detail in automatically generated files:

- time dependent values of creep & shrinkage coefficients
- time dependent values of the Young's modulus
- influence lines

These values must be selected in the function ⇧RESULTS in order to be displayed.

10.3 Program function ⇧RESULTS

The calculation results can be viewed and presented in many forms.
The following 'types of result' from the calculation are available:

- Numerical plus Plotted results for individual loading case calculations.
- Numerical plus Plotted results for Envelopes (combinations of loading cases)
- Numerical plus Plotted results for Fibre Stresses
- Plot of the material property variation with time (Creep & Shrinkage + Elastic Material Modulus)
- Plot of the Influence lines used for the live load calculation
- Plot of results from a history file (time dependent calculation)
- Printed results - numerical list files plus plotted results (hard copy)

10.4 Individual Load Case Results

Results – Load Case

LC [ ] ∇ [Element] ∇ [KN, kNm] ∇

Elem [ ] [Global] ∇ [ ] Ndiv Pnt

Elem	Nod	x/l	N (Vx)	Qy (Vy)	Qz (Vz)	Mx (Rx)	My (Ry)	Mz (Rz)
:	:	:	:	:	:	:	:	:
:	:	:	:	:	:	:	:	:
:	:	:	:	:	:	:	:	:

[Displacement] [Primary] [Secondary] [Total] [Print] [Min] [Max] [Recalc]

The displayed results for the ‘Forces’ ‘Moments’ and ‘Deflections’ from any individual loading case calculation can be selected using the alternatives described below:

LC (Loading Case) The loading case number to be viewed must be entered in the box or chosen from the displayed list in the pull-down menu.

Element & Node & Cable The results to be viewed can be either Element, Node or Cable related – choose the toggle button to change between them.
‘Element’ results, comprising both ‘Force and Deflection’ output, are provided for the beginning and end of each element as well as for all the intermediate points provided ‘Ndiv Pnt’ is selected and the element was sub-divided in the element definition. (⇧STRUCTURE ⇨ELEMENT)

.

‘Node’ results are provided at the node positions, which may be different from the element begin and element end respectively.

kN, kNm The results output units can be modified by choosing from the displayed list in the pull-down menu.

.

Elem The element number in the window can be user defined by editing the displayed value. The defined element number is shown at the bottom of the screen display.

Global & Local The results can be viewed in the Local or the Global co-ordinate system - choose the toggle button to change between them.
‘Force results are usually viewed in the ‘Local co-ordinate directions’ **-** It is particularly useful to view the results in the ‘Local co-ordinate system’ when the element is not parallel or perpendicular to the ‘Global co-ordinate system’.
‘Deflection results are usually viewed in the ‘Global co-ordinate directions’ as deflections from loads dependent on gravity are in the vertical direction irrespective of the direction of the element!

Ndiv Pnt The values for the intermediate points on the element will be displayed when ‘Ndiv Pnt’ is selected and provided the element was sub-divided in the element definition. (⇧STRUCTURE ⇨ELEMENT)

Deflection The loading case values will change from ‘Forces’ to ‘Deformations’ following the selection of ‘Deflection’.

x/l The distance from the start of the member expressed as a proportion of the total length. (Used with Ndiv)

Vx, Vy, Vz Deflections in the x, y, & z directions respectively. The units for the deflections are the same as those used in all the input and output data except they may be multiplied by a factor defined in the input screen for 'Recalc' (Default = 1000)

Rx, Ry, Rz Rotations about the x, y, & z axes respectively. . The units for the deflections are the same as those used in all the input and output data except they may be multiplied by a factor defined in the input screen for 'Recalc' (Default = 1000).

Total, Primary, Secondary The loading case values will change from 'Deformations' to 'Forces' 'following the selection.

Total 'Primary + secondary State' displays the output for normal loading case and envelope results as well as the Primary PLUS the Secondary effects for the pre-stressing and creep and shrinkage loading cases only.

Primary 'Primary State' displays the output for the Primary Loading part of the loading case only (for pre-stressing and creep and shrinkage loading cases only).

Secondary 'Secondary State' displays the output for the Secondary effects part of the loading case only (for pre-stressing and creep and shrinkage loading cases only).

x/l The distance from the start of the member expressed as a proportion of the total length. (Used with Ndiv)

N Normal force in the x direction.

Qy, Qz Shear force in the y & z directions respectively.

Mx, My, Mz Moments about the x, y, & z axes respectively.

Print The currently displayed Listing will be written into an ASCII file. Wanted elements, results and file name need to be defined.

Min Select Min to view the minimum value for the loading case. (A pull-down window will automatically open for choice of the particular type of force or deflection defining the minimum – i.e Qy or Qz or Mx etc)

Max Select Max to view the maximum value for the loading case. (A pull-down window will automatically open for choice of the particular type of force or deflection defining the maximum – i.e Qy or Qz or Mx etc)

Recalc Start the calculation again – a complete choice of items to be calculated is offered. Any one or all the items may be selected to be recalculated.

10.5 Superposition results (Envelope)

Results – Envelope

File [ ] ∇ [Element] ∇ [MinN] ∇ [KN, kNm] ∇

Elem [ ] [Global] ∇ [ ] Ndiv Pnt

Elem	Nod	x/l	N (Vx)	Qy (Vy)	Qz (Vz)	Mx (Rx)	My (Ry)	Mz (Rz)
:	:	:	:	:	:	:	:	:
:	:	:	:	:	:	:	:	:
:	:	:	:	:	:	:	:	:

[Deflection] [I State] [II State] [I + II State] [Min] [Max] [Recalc]

The displayed results for the 'Forces' 'Moments' and 'Deflections' from any Result Envelope can be selected using the alternatives described below:

File (Envelope file) The required envelope to be viewed must be entered in the box or chosen from the displayed list in the pull-down menu.

Element & Node The results to be viewed can be either Element or Node related – choose the toggle button to change between them.
'Element' results, comprising both 'Force and Deflection' output, are provided for the beginning and end of each element as well as for all the intermediate points provided 'Ndiv Pnt' is selected and the element was sub-divided in the element definition. (⇧STRUCTURE ⇨ELEMENT)

'Node' results are provided at the node positions, which may be different from the element begin and element end respectively.

MinN The leading value from the 'Envelope' that the user wishes to view together with the other 'corresponding values' must be defined in this box - choose from the list in the pull-down menu.

kN, kNm The results output units can be modified by choosing from the displayed list in the pull-down menu.

.

Elem The element number in the window can be user defined by editing the displayed value. The defined element number is shown at the bottom of the screen display.

Global & Local The results can be viewed in the Local or the Global co-ordinate system - choose the toggle button to change between them.
'Force results are usually viewed in the 'Local co-ordinate directions' **-** It is particularly useful to view the results in the 'Local co-ordinate system' when the element is not parallel or perpendicular to the 'Global co-ordinate system'.
'Deflection results are usually viewed in the 'Global co-ordinate directions' as deflections from loads dependent on gravity are in the vertical direction irrespective of the direction of the element!

Ndiv Pnt The values for the intermediate points on the element will be displayed when 'Ndiv Pnt' is selected and provided the element was sub-divided in the element definition. (⇧STRUCTURE ⇨ELEMENT)

Deflection The loading case values will change from 'Forces' to 'Deformations' following the selection of 'Deflection'.

x/l The distance from the start of the member expressed as a proportion of the total length. (Used with Ndiv)

Vx, Vy, Vz Deflections in the x, y, & z directions respectively. The units for the deflections are the same as those used in all the input and output data except they may be multiplied by a factor defined in the input screen for 'Recalc' (Default = 1000)

Rx, Ry, Rz Rotations about the x, y, & z axes respectively. . The units for the deflections are the same as those used in all the input and output data except they may be multiplied by a factor defined in the input screen for 'Recalc' (Default = 1000).

I+II; I; II The loading case values will change from 'Deformations' to 'Forces' 'following the selection of 'I+II; I; or II.

I+II State 'I+II State' displays the output for normal loading case and envelope results as well as the Primary PLUS the Secondary effects for the prestressing and creep and shrinkage loading cases only.

I State ‘I State’ displays the output for the Primary Loading part of the loading case only (for pre-stressing and creep and shrinkage loading cases only).

II State ‘II State’ displays the output for the Secondary effects part of the loading case only (for pre-stressing and creep and shrinkage loading cases only).

x/l The distance from the start of the member expressed as a proportion of the total length. (Used with Ndiv)

N Normal force in the x direction.

Qy, Qz Shear force in the y & z directions respectively.

Mx, My, Mz Moments about the x, y, & z axes respectively.

Min Select Min to view the minimum value for the loading case. (A pull-down window will automatically open for choice of the particular type of force or deflection defining the minimum – i.e Qy or Qz or Mx etc)

Max Select Max to view the maximum value for the loading case. (A pull-down window will automatically open for choice of the particular type of force or deflection defining the maximum – i.e Qy or Qz or Mx etc)

Recalc Start the calculation again – a complete choice of items to be calculated is offered. Any one or all the items may be selected to be recalculated.

10.6 PlSys

All of the graphical outout facilities in *RM2000* are accessed via this button - (⇧RESULTS ⇨PlSys)

10.6.1 General

The data for the plot file can either be defined from scratch using the appropriate plot commands described below or can be basically defined using the ‘Macro’ button. A general plot input file, following the users requirements, is produced under ‘Macro’ – the user can view the plotted result and is free to modify the input by editing or adding additional plot commands to the input file.

File name extension.
All the plot input files must have the extension ***.rm** - if this extension is not present or if the file has a different extension, then the program will not recognise the file as a plot file input.
Files can be saved under their own name or under a new name for future editing using the 'Save as' feature.

Definitions
The 'Buttons' displayed at the bottom of the main plot screen (opened on selection of ⇧RESULTS ⇨PlSys) have the following function:

10.6.2 Macro

7 'basic' plot file inputs can be easily prepared using 'Macro' – these input files can be edited later for presentation purposes (see plotting facilities below for a detailed explanation):

Structure	The structure in the un-deformed state is plotted.
Load case plot, shape & structure	The deflected shape resulting from applying a loading case to the structure is plotted together with the structure in the un-deformed state is plotted.
Load case plot, forces	The resultant forces (moments, shears etc) on the structure from applying a loading case is plotted together with the structure in the un-deformed state.
Load case plot, stresses	The resultant fibre stresses (top and bottom fibre) on the structure from applying a loading case is plotted together with the structure in the un-deformed state.
Superposition File plot, shape & structure	The deflected shape stored in the superposition file is plotted together with the structure in the un-deformed state is plotted
Superposition File plot, forces	The resultant forces (moments, shears etc) stored in the superposition file is plotted together with the structure in the un-deformed state
Superposition File plot, stresses	The resultant fibre stresses (top and bottom fibre) stored in the superposition file is plotted together with the structure in the un-deformed state

10.6.2.1 Save
Changes made to the plot file input is saved to the original file name using this button.

10.6.2.2 Show
A screen plot is produced and immediately displayed on selection of this button. The content of the plot is defined by the commands in the displayed plot file input screen.

10.6.2.3 Plot to File
A plot file output specially formulated for hard copy printout is produced with this button. The content of the plot is defined by the commands in the displayed plot file input screen.

10.6.2.4 Save as
The plot file input shown on the screen can be saved under a new name by using the 'Save as' button. This new file can then be edited to suit the users needs.

10.6.3 Plot Actions

The plot commands that are required for preparing the plot file input are selected via the "input before or after" icons that are at the top of the Plot File Editor input screen which is opened on selection of the button (⇧RESULTS ⇨PlSys).

10.6.4 Presentation capabilities

There are several additional "Plot Functions" available for enhancing the general appearance of the plot files for presentation purposes.

The additional "Plot Functions" include:

Scaling facilities	The structural model as well as the output results can be plotted to any scale (independently)
Font	Any font for the text can be chosen
Pen	A selection of normal pen colours is available for plotting – any colour can be used for any different part of the plot.
Text size	Any text size can be used
Border	A border can be requested for the plot
Free Text	Text can be placed within any part of the plot – for convenient labelling
Elements	A selection of which elements forming the structural model should be plotted can be made:

- All the structural elements can be selected to be plotted
- All the active structural elements can be selected to be plotted
- All the structural elements that are in a particular construction stage can be selected to be plotted
- All the inactive structural elements can be selected to be plotted

10.6.5 Type of Plots

The following types of plots can be made:
- Structural Elements including numbering
- Structural Nodes including numbering
- Pre-stressing Tendon profiles including numbering
- Traffic lanes including numbering
- Individual loading Case results – any or all of the 6 forces (including moments) and the 6 deflections (including rotations)
- Superposition file results (envelopes) for any chosen Maximum or Minimum value in the matrix– any or all of the 6 forces (including moments) and the 6 deflections (including rotations)

10.6.6 Superposition of Plots

There is no limitation on the superposition of plots – for instance the deflections (to any scale) can be superimposed over the bending moment diagram and the structural model plot.

10.6.7 Plot Commands

The different plot commands for the plot file input can be selected from this list. The commands are sorted into the groups, which can be viewed by pressing the radio buttons – the groups are summarised below:

⊙ Paper size, scale, border
⊙ Scale
⊙ Pen, text size, …
⊙ Value defaults
⊙ Load case and Envelope
⊙ Structure plot
⊙ Free plot

10.6.7.1 Paper size, scale, border

The paper size, the scale for the structural model as well as whether or not a border line should be drawn around the plot can be input under this group.

PLTRAN: The projection angle of the view and the scale factors for each axis will be defined. The parameters can be varied to plot an isometric view -looking from any direction - or simply a 2 dimensional view.

Radio button General isometric set After confirming the radio button the values for projection angles (0.0,90.0,45.0) and scale factors (1.0,1.0,1.0) will automatically be set.

Radio button Ground plan view After confirming the radio button the values for projection angles (0.0,0.0,90.0) and scale factors (1.0,0.0,1.0) will automatically be set.

Radio button Vertical plan view After confirming the radio button the values for projection angles (0.0,90.0,0.0) and scale factors (1.0,1.0,0.0) will automatically be set.

Radio button Side plan view After confirming the radio button the values for projection angles (0.0,90.0,0.0) and scale factors (0.0,1.0,1.0) will automatically be set.

Phi-x;Phi-y;Phi-z The angles between the plotter x-axis and the x, y, z axes of the plotted view. *N.B The angle is measured in the anti-clockwise direction in degrees.*

Scf-x;Scf-y;Scz-z The scale factors for the plotted elements / forces / deflections in the x, y, z directions respectively.

PLSIZE: The size of the plot and the scale of the system is defined with this pad.

Scale: The scale for the plot - 100 means that the drawing (structural model) will be plotted at a scale of 1:100 of full size.

Delx/Dely: The dimensions (in cm) of the paper for the plot in the x and y plotter directions.

NOTE: If only 'SCALE' will be defined, the paper size will be evaluated automatically.
If only DELX/DELY will be defined, the scale will be evaluated automatically.

Spntx;Spnty;Spntz: The intersection point of the plotted axes can be moved from 0,0,0 to any other point in the coordinate system with this input.

PLBORD: The outside border lines of the plot are defined with this pad. The lines are defined relative to the edge of the paper which itself was defined in (PLSIZE).

Bord Left/Right/Above/Below (cm): The distance to the left/right/top/bottom border lines from the edge of the paper.

10.6.7.2 Scale

The scale for the deformations, forces, moments and stresses can be set under this group. It should be noted that this command must be entered in the Plot command list before calling for the actual plots of the deformations, forces, moments or stresses!

PLSCAL: The scale for plotting the Forces, Moments, Displacements and Stresses are defined. (N.B. The scale refers to the hard copy plot)

Scalf-1cm = ...Units Scale for plotting the Forces and Reinforcement
Scalm-1cm = ...Units Scale for plotting the Moments
Scald-Fact = The deflections will be multiplied by this factor.
Scals-1cm = ...Units Scale for plotting the stresses

PLSCFAC: The scale factor for plotting of Forces, Moments, Displacements and Stresses is defined. Already defined scales will be multiplied by the factor.

Factor Scale factor

IMPORTANT!	**The changed scale will be used for all further commands till the command with another scale factor will be input. The factor may be also negativ in order to show all results with the opposite sign (Factor = 1.0 means origin scale).**

10.6.7.3 Pen, text size,...

The pen (line colour), text size, pen thickness and font type can be set under this group. The command, similar to other commands, can be several times in one plot.. The attributes will be used immediately in the next plotting function and thereafter until changed.

PLPEN: The pen (colour, line thickness and line type) is defined with this pad.

Pen No Number of pen colour will be defined:
- 1 white
- 2 red
- 3 green
- 4 blue
- 5 yellow
- 6 cyan (bright blue)
- 7 magenta

Ltyp No The line style to be plotted:
- 0 continuos
- 1 short dashed

- 2 middle dashed
- 3 long dashed
- 4 short dashed-dotted
- 5 middle dashed-dotted
- 6 long dashed-dotted

Lin-thick The line thickness to be plotted.

PLTXSZ: The size of the text in the plot is defined with this pad.

Txtsiz The height of the text in cm (absolut value, not scale dependend).

PLFONT: The free text reference points as well as the font types can be viewed and modified via this pad.

Position The position of the free text in the plot area is defined by the coordinated start point (Refer: PLTFTXT) and the position within the line of text that the coordinated start point refers to.
Select the arrow to activate the text reference point pad and choose between
LA:LC:LB: left above/centre/below
CA:CC:CB: centre above/centre/below
RA:RC:RB: right above/centre/below

Font Type The font for the text can be chosen via this pad.
Select the arrow to activate the Font Type list and choose between the six different fonts displayed:
- norm
- pica
- greece
- kursiv
- grotesk
- clasic

10.6.7.4 Value defaults

Certain defaults can be changed using this group.
Type of result annotation - All values, max/min only, or no values
Type of element to be plot - All elements, only beam elements, only springs,...,only active elements, only inactive elements,…
Type of forces for pre-stressing - Only primary, only secondary state or total forces
The point on the cross section for which the fibre stresses will be plotted.

PLALL: Plott all (active and inactive elements)
PLSTAG: Plot only stage

PLINAC:	Plot only inactive elements
PLGLOB:	Set the global direction (forces)
PLLOC:	Set the local direction (forces)
PLMARK:	Select the element type to be ploted
PLVALUE:	Select the value-marking-type
PLSTAT:	Select the type of forces (prim., sec. state)
PLFIBP:	Select the cross-section point for fibre stress checck
PLNORM:	Set the result type (normal)
PLSPLT:	Set the result type (split-partial element of composite)
PLJOIN:	Set the result type (join-composite and partial element)

10.6.7.5 Load case and Envelope

The type and name/number of the results file is selected and defined in this group – whether it is a loading case or an envelope results file.

PLLC: The loading case number to be plotted is defined with this pad. Press the arrow to open the list of existing load cases.

PLSUP: The superposition file to be plotted is defined with this pad. Press the arrow to open the list of existing superposition files.

10.6.7.6 Structure plot

A fundamental requirement for plotting anything is that the 'Structure Plot' group is selected.
All the forces, moments, stresses and deflections are plotted relative to an element so ELEMENT **must be** selected and then the 'Pull Down Menu' for Input 1 selected to open the Command selection window for forces, moments, stresses and deflections.

The command to plot FORCES, MOMENTS, STRESSES AND DEFLECTIONS can **only** be accessed via Structure Plot\Element.

The user can also define on which elements the forces should be plotted – default is 0; 0; 0 which plots all the structural elements.

Nodes, lanes, and tendons can also be plotted with various extra features accessed via the 'Pull Down Menu' for Input 1

PLELEM: All the element related output can be defined with this pad:

Forces, Moments, Deflections, the Elements themselves...

Elem-From/to/Step	The element series for which some plot output is required.
Inp 1	Select the arrow to define the type of output that should be plotted for the selected elements.
Inp 2	Select this arrow if further definition is required.

All the element related output to be plotted can be selected from this pad:

SYS	Plot the defined elements - or, if the elements are not defined plot every element in the structure unless this is constrained by a previous command (i.e PLSTAG).
SYSQ	Plot the structural cross sections for the defined elements -or for all the elements as described for 'SYS'
SYSO	Plot the local structural axes for the defined elements -or for all the elements as described for 'SYS'
NUM	Plot the element numbers for all the the defined elements -or for all the the elements as described for 'SYS'
FIBLC	Plot the stresses of the certain cross-section point for all the defined elements -or for all the elements as described for 'SYS'. The values will be read from certain loadcase.
FIBSUP	Plot the stresses of the certain cross-section point for all the defined elements -or for all the elements as described for 'SYS'. The values will be read from certain superposition file, so the INP2 is necessary.
FIBMAX	Plot the limit max stress of the certain cross-section point for all the defined elements -or for all the elements as described for 'SYS'. The values will be read from material file.
FIBMIN	Plot the limit min stress of the certain cross-section point for all the defined elements -or for all the elements as described for 'SYS'. The values will be read from material file.
PLREIN	Plot reinforcement of certain reinforcement group for all the defined elements -or for all the elements as described for 'SYS'.

N	Plot the Normal (Axial) Force acting on the elements for the previously defined loading case or superposition file.
Qy	Plot the Shear Force Qy acting on the elements for the previously defined loading case or superposition file.
Qz	Plot the Shear Force Qz acting on the elements for the previously defined loading case or superposition file.
Mx	Plot the Bending Moment Mx (acting around the X axis) on the elements for the previously defined loading case or superposition file.
My	Plot the Bending Moment My (acting around the Y axis) on the elements for the previously defined loading case or superposition file.
Mz	Plot the Bending Moment Mz (acting around the Z axis) on the elements for the previously defined loading case or superposition file.
SHAPE	Plot the deformed shape of the structure resulting from a previously defined loading case
Vx;Vy;Vz	Plot the deformed shape of the structure resulting from a previously defined loading case or superposition file with Vx (or Vy or Vz as appropriate) as the primary value in the line of the superposition matrix.
Rx;Ry;Rz	Plot the deformed shape of the structure resulting from a previously defined loading case or superposition file with Rx (or Ry or Rz as appropriate) as the primary value in the line of the superposition matrix.
NSUPF	Plot the forces in the supports resulting from a previously defined loading case or superposition file.
NSUPM	Plot the moments in the supports resulting from a previously defined loading case or superposition file.

PLNODE: All the node related output to be plotted can be selected from this pad:

SYS	Plot the defined nodes - or, if the nodes are not defined plot every node in the structure unless this is constrained by a previous command (PLSTAG)

SHAPE The deflected shape of the structure is plotted with this command.

NUM Plot the node numbers for all the the defined nodes -or for all the the nodes as described for 'SYS'

NSUPF Plot the forces in the supports resulting from a previously defined loading case or superposition file.

NSUPM Plot the moments in the supports resulting from a previously defined loading case or superposition file.

10.6.7.7 Free plot

Various plot enhancement features for presentation purposes are accessed via this group such as lines, texts, circles as well as the coordinate system axis marks all in any position on the plot.
A plot file can also be imported from any directory via this group.

PLFTXT: The coordinated start point for the position of the free text, the orientation of the text and the text itself are defined with this pad.

Position Select the arrow to activate the text reference point and choose between
LA:LC:LB: left above/centre/below
CA:CC:CB: centre above/centre/below
RA:RC:RB: right above/centre/below
if the position of the text relative to the defined coordinated point needs to be changed.

X/Y-cm 'X' and 'Y' coordinates defining the start point for the text (in cm) with reference to the plot file axes.

Alpha The orientation of the text measured in degrees from the the plotter 'X' axis (+ve is anticlockwise)

Text Any Alpha-numeric text that needs to be inserted.

PLLINE: Any straight line can be plotted anywhere in the plot area using this pad.

X1/Y1-cm The coordinates of the start point of the line (in cm) in the plotfile coordinate system.

	X2/Y2-cm	The coordinates of the end point of the line (in cm) in the plotfile coordinate system.
PLCIRC:		Any size circle can be plotted anywhere in the plot area using this pad.
	XR/YR-cm	The coordinates of the centre point of the circle (in cm) in the plotfile coordinate system.
	R-cm	The radius of the circle (in cm)
PLCOSY:		The coordinate system axes symbol can be positioned anywhere in the plot area with this pad.
	Position X /Position Y (cm)	The coordinated position of the symbol, measured in cm, relative to the 0,0 position of the plot.
PLIMP:		A plotfile can be imported from any directory using this pad.

10.7 Fibre Stress results

The extreme fibre stresses can be printed out and plotted for each element for:

- Individual loading cases
- Several defined individual loading cases
- Superposition files like traffic loading envelopes

The fibre stresses are based on the forces and moments acting on the particular element together with the defined element section properties at the time of the loading calculation.

10.7.1 Fibre Stress Output list Files

The fibre stress results can be requested in both a detailed output listing format and a summary output listing format.
The summary output listing only contains the top and bottom fibre stresses for the particular combination file (or loading case)
The detailed output listing contains a full break down of the top and bottom fibre stresses resulting from each individual loading case and/or superposition file making up the combination file.
The results, in both cases, are bannered (#) where the values exceed the defined allowable values.

10.7.2 Requesting a Fibre Stress Output list File

The Fibre Stress Output list Files can only be requested via the (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION) - **FibChk**

10.7.2.1 Minimum data requirements

The following must have been pre-defined/calculated before the Fibre Stress Output list Files can be requested:

- Load Combinations. These are defined using (⇧LOADS AND CONSTR.SCHEDULE ⇨LOADS ⇩COMB) or directly in (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION) - **Envelope action**
- The superposition file name for the above loading combination must also be defined if the file was created using SupComb - (⇧LOADS AND CONSTR.SCHEDULE ⇨STAGE ⇩ACTION) - **Envelope actions\SupComb.** If the superposition file was created directly in 'Action' then the file already has a name.

- The points for fibre stress checks within the cross section (Normally defined in _GP2000_)

10.7.2.2 Typical input for a summary output listing

MODULE	FibChk	Command for fibre stress calculation
Inp1	comb6.sup	comb6.sup will have been created via SupComb (refer above)
Inp2	0.75,0.90	max, min factors to define the max and min allowable stresses – the allowable stress is defined by multiplying the general allowable stress by these factors respectively (The general allowable stress is defined under ⇧PROPERTIES ⇨MATERIAL Concrete Fibre stress check \General (e.g. 2600 for tension, -16000 for compression)
Out2	fibcomb6.lst	It is not necessary to define an output file name but it can be useful!

10.7.2.3 Typical input for a detail output listing

MODULE	FibChk	Command for fibre stress calculation
Inp1	comb6.sup,6	comb6.sup will have been created via SupComb (refer above) the additional 6 refers to comb 6 again and tells _RM2000_ to prepare a detailed list.
Inp2	0.75,0.90	max, min factors to define the max and min allowable stresses – the allowable stress is defined by multiplying the general allowable stress by these factors respectively (The general allowable stress is defined under ⇧PROPERTIES ⇨MATERIAL Concrete Fibre stress check \General (e.g. 2600 for tension, -16000 for compression)
Out2	fibcomb6x.lst	It is not necessary to define an output file name but it can be useful! – Clearly this file name should be different from the one above as otherwise the one will overwrite the other!

10.7.2.4 Fibre Stress Output Plot Files

The fibre stresses at the defined fibre stress points can also be plotted using all the normal colour and line style plotting facilities – refer to PlSys above.

10.7.2.5 Minimum data requirements

The minimum pre-defined data/calculation is the same as for Fibre Stress output listing files – see above.

10.8 Time integration result - PlCrSh

10.8.1 PlCrSh

The Creep and Shrinkage variation with time for each element can be viewed in graphical form and plotted out (hard copy).

'Set' - The special facility "Set" allows the user to be able to view the effects of modifying one or even several of the constants influencing the creep and shrinkage. The modified values will be automatically re-set to the pre-defined values when **'Redraw'** is selected.

10.8.2 E(t)

The Young's Modulus variation with time for the concrete for each element can be viewed in graphical form and plotted out (hard copy).

'Set'/'Redraw' The "Set" and "Redraw" facilities – refer description under 'PlCrSh' are also available with E(t) allowing the user to view the effects on the Youngs Modulus curve of modifying one or even several of the constants.

10.9 Influence Lines - PlInfl

The 12 Influence lines for each lane on each element will be displayed in graphical form on selection of this 'Button'. The plots can be magnified (zoom in or out) using the normal free hand symbols.