RM2000

Static and Dynamic Analysis

of Spaceframes



Getting Started

TDV Ges.m.b.H

Januar 2003

Ι

Contents

С	ONTENTS	I
1	GENERAL	1-1
	1.1 Starting the program	
	1.2 DATA CONVERSION FROM RM7	1-1
2	THE INTRODUCTORY EXAMPLE	2-1
	2.1 SYSTEM:	2-1
	2.2 DESIGN CRITERIA	2-3
	2.3 MATERIALS:	2-4
	2.4 DESIGN LOADINGS:	
	2.4.1 Dead Load:	
	2.4.2 Live Load:	
	2.4.5 Inermal Forces:	
	2.4.4 Creep and Shrinkage 2.4.5 Pier settlement:	
_		
3	STARTING THE PROGRAM	
4	DESCRIPTION OF THE PROGRAM INTERFACE	4-1
	4.1 DESCRIPTION OF THE MAIN USER INTERFACE PARTS	4-1
	4.1.1 Tool bar	
	4.2 MAIN FUNCTIONS	
	4.2.1 Sub-functions	
5	THE DEFAULT – DATABASE	5-1
6	MODIFY A MATERIAL	6-1
7	CHECK THE CROSS SECTION	7-1
8	DEFINITION OF THE STRUCTURAL SYSTEM	8-1
	8.1 Nodes	8-1
	8.2 Elements	
	8.3 CROSS SECTIONS ASSIGNMENT	
	8.4 CALCULATION	
9	DEFINITION OF TENDONS	9-1
	9.1 DEFINITION OF TENDON GROUPS	
	9.2 DEFINITION OF THE TENDON GEOMETRY	
	9.3 DEFINITION OF THE TENDON STRESSING SCHEDULE	
1(0 DEFINITION OF LOADS	10-1
	10.1 DEFINING LOADS	
	10.1.1 Definition of a load set	
	10.1.2 Define a loading case	

RM2000

Contents

etting Started	II
10.1.3Assignment of Load set to Load case10.1.4Prestressing loading case10.1.5Creep and shrinkage loading case10.2DEFINITION OF A TRAFFIC LOAD	
11 DEFINITION OF A CONSTRUCTION SCHEDULE	11-1
11.1 DEFINITION OF CALCULATION ACTIONS	
12 CALCULATION OF THE STRUCTURAL SYSTEM	12-1
13 RESULTS	13-1
13.1 DIAGRAM PLOT 13.2 PLSYS	
14 STRESS CHECK	14-9
14.1 DEFINITION OF THE STRESS-LIMITS:	14-9
15 ULTIMATE LOAD CHECK	
16 SHEAR CAPACITY CHECK	
17 DATA BACKUP	
18 PLOT MACROS	
18.1 Plot-Macros	
18.1.1 Forces	
18.1.2 Fiber stress Plots	
18.1.5 Ottimate toda pior	
19 RESULT PLOTS	
19.1 SYSTEM (PLSYS)	
19.2 FORCES AND MOMENTS (DIAGRAM)	
19.5 FIBRE STRESS (DIAGRAM) 19.4 TENDON DRE-STRESSING AND CREED/SHRINK AGE	
19.5 INFLUENCE LINE	

Getting Started

1-1

1 General

The following items are briefly described in this introduction to the RM2000 space-frame analyser:

- Starting the program
- The user interface
- Importing material definitions
- Definition of materials
- Defining a cross section
- Defining the structural model
- Defining a tendon geometry
- Defining loads
- Defining a traffic loading case
- Defining a construction schedule
- Running the calculation
- Viewing the results
- Fibre stress check
- Ultimate load check
- Shear capacity check

This introduction is based on a simple example that the user should work through using the program RM2000 at the same time as following this text.

1.1 Starting the program

The program installation must be completed before any work can be started. The installation procedure automatically creates the following TDV icons for GP2000 and RM2000 on the desktop:



The program can be started by double-clicking the appropriate icon (shown above) or by selecting the icons via the Windows - "Start" – menu, (usually located in the bottom left hand corner of the screen). The GP2000 and RM2000. Icons are located in the file structure under the program group "TDV2000".

1.2 Data conversion from RM7

Refer to section 13. Data conversion from RM7 to RM2000 for further details

2 The introductory example

The three span hollow concrete box girder shown in Figure 1 below will be defined. This section contains several variable dimensions.



Figure 1. Structural system

The 140m long three-span bridge (40m + 60m + 40m) is located on a compound axis comprising a straight line, a circular curve and then another straight line.

2.1 SYSTEM:





STRUCTURAL MODEL: (program)



System axis: Horizontal plan

1 st .Part:Straight Line :	Station: 0-20 m
2 nd .Part: Spiral: A=100, R _{END} =200m	Station: 20-70 m
3 rd .Part: Circle: R=200	Station: 70-140 m

System axis: Vertical plan

1.Part: Line: 30m dZ=0,5m 2.Part: Circle : R=-2000m 3.Part: Line: 40m Station: 0-30 m Station: 30-100 m Station: 100-140 m

Numbering system:

Node numbers (span) : 101-111-126-136 Element numbers (span) : 101-110,111-125,126-135.

Supports: (defined by additional elements)



Getting Started



2.2 DESIGN CRITERIA

Clearance 20cm from bottom

The following criteria will be used for this design example:

Specifications, Codes, and Standards: AASHTO Bridge Design Specifications

Clearance 20cm from bottom

Clearance 20cm from bottom

Getting Started

2-4

2.3 MATERIALS:

Reinforcement: GRADE 460

Yield Strength: Modulus of Elasticity: 400000 kN/m² 200000000 kN/m²

Strain/Stress values



Strain/Stress values



Prestressing Steel:

Strand tendons shall consist of low-relaxation steel. Material Properties:

Ultimate Tensile Strength	1860000 kN/m ²
Yield Strength	1674000 kN/m ²

Apparent Modulus of Elasticity:

19700000 kN/m²



 $A_{\rm H}=0,0050{\rm m}^2/{\rm Duct}$

Getting Started

Allowable Tendon Stresses:

Jacking Force:	0,80 f _{pu}
At anchorages after anchoring	0,70 f _{pu}
At other location after anchoring	0,74 f _{pu}
At Service limit state after losses	0,80 f _{py}

_	Factor	0,8	0,7	0,74	0,8
fpu	1860000	1488000	1302000	1376400	
fpy	1674000				1339200

2.4 Design Loadings:

2.4.1 Dead Load:

Unit Weight of Reinforced Concrete (DC):	23,5 kN/m ³
Additional dead load:	30,0 kN/m

2.4.2 Live Load:

Apply one load train on one central lane in this example:



2.4.3 Thermal Forces:

Coefficient of expansion: Temperature Range Linear temperature gradient 10.8 x 10e-6 per °C 15°C +10°C at the top

2.4.4 Creep and Shrinkage:

Strains calculated in accordance with CEB-FIP 1990 Model Code for superstructures.

2.4.5 Pier settlement:

1 cm at each support.

3 Starting the program

The program installation must be completed before any work can be started. The installation procedure automatically creates the following TDV icons for *GP2000* and *RM2000* on the desktop:



The RM2000 program can be started either by double-clicking the RM2000 icon or by selecting the icons via the Windows - "Start" – menu, usually located in the bottom left hand corner of the screen.

 Double-click one of the these icons to start the program

After the installation the Default-Database is empty. Therefore the program try to create a Default-Database in the program directory (e.g. c:\Program Files\Tdv2000\rm8)

The appearing screen shown all available materials and formulas, which you can store now into the Default-Database.

Select <CS-AS90.RMD> and use blank to mark the first database. Then select <MAT-USA.RMD> and use blank to mark the second database.

Cs-hs54.rmd	Hong Kong Standard - Creep & Shrinkage Model	
Cs-hung.rmd		
Cs-nor.rmd	Norwegian Standard - Creep & Shrinkage Model	
Cs-oe47.rmd	ÖNORM 4750 - Kriech & Schwind Modell	
Cs-rsm90.rmd	RSM90 - Creep & Shrinkage Model	
Mat-bs.rmd	British Standard - Material Database	
MAT-DIN1.RMD	DIN - Material Datenbank (alt)	
Mat-din2.rmd	DIN - Material Datenbank (neu)	
Mat-Hung.rmd	Hungarian Standard - Material Database	
Mat-jap.rmd	Honkong Standard - Material Database	
Mat-nor.rmd	Norwegian Standard - Material Database	
MAT-OE1.RMD	ÖNORM - Material Datenbank (alt)	
Mat-oe2.rmd	ÖNORM - Material Datenbank (neu)	
Mat-por.rmd	Portogies Standard - Material Database	
Mat-usa.rmd	AASHTO - Material Database	•

CS-CEB90.RMD contains all necessary formulas and tables for the creep & shrinkage calculation according to CEB90.

MAT-USA.RMD contains all materials according to AASHTO.

This selection of databases appears only if the database is empty (e.g. after the installation or you delete this database in the program and you start the program again!).

> Select <**Ok**> to close this window.

The input screen shown below for starting a project appears following the program start. Any of the alternatives can be selected by choosing the appropriate radio button:

A new project must be stored in a new directory or sub-directory.

The structural data for this RM2000 example was prepared in GP2000 and exported to RM2000. The "new" directory therefore already exists!

Project Ne w	×
•	Subdirectory List
You can choose:	^
 Start a new project Continue project Import the data Read the demo example Finish RM2000 with backup Finish RM2000 	
Ok	Cancel

Select the arrow in the top left hand window of the input screen to open Windows Explorer. The Explorer directory selection screen will appear:

- > Select the appropriate directory path.
- Select "First Project"
- N.B. The database files shown in this directory were exported from GP2000.
- > Choose <Open> to accept the displayed directory as the desired project directory.

The full directory path will be shown in the top left hand window of the re-displayed project input screen.

The working directory is now defined.

> Select <**o**k> to start RM2000.

4 Description of the program interface

The main RM2000 screen is similar in design to most Windows programs.



4.1 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.

RMRM2000 8.27.01 [C:\work\FirstProject]				
File	Properties	Structure		

4.1.1 Tool bar





Opens a window listing the recorded actions.



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



Shows errors from the most recent calculations.



Opens the Windows Calculator program.



Opens the default editor program (Textpad or Notepad)



Opens a program for plotting graphic results.



Lists all freehand symbols for zooming functions.



Opens a dialog window for program parameters.



Prints plotfiles and other results.



Opens the RM2000 help files.



Opens the RM2000 online books.

Getting Started

4.2 Main functions

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

File	Properties	Structure	Loads and Constr.Schedule	Recalc	Results	Scripts

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 ge- ometry)
Loads and Con.Sch.	Definition of loads and constructions stages
Recalc	Start a calculation
Results	Viewing of result and creating of plots
Scripts	Using of Run- and Open TCL

N.B The 'up-arrow' symbol (' \hat{U} ') will be used in this document to identify a main function, e.g.: \hat{U} STRUCTURE.

4.2.1 Sub-functions

On selection of \hat{U} FILE, the following sub-functions list will be displayed on the right hand side of the screen.

N.B The 'right-arrow' symbol (' \Rightarrow ') is used to identify a sub-function in this document. e.g.: \Rightarrow IMPORT.

New	Start a new project in the current directory.
Defaults	All defaults, needed for the project
Open	Open an existing project or start a new one.
Import	Import a saved project (or part of it).
Export	Export (save) the current project (or part of it).
Demo	Select one of the RM2000 demo examples to be loaded for viewing.
Exchange	Change project information for viewing and editing into the desired format.
RM7	Import the RM7 steel cross section table for RM2000.
Optimise	Input of optimisation to accelerate the calculation.

RM2000

4-4

Getting Started

On selection of \hat{U} PROPERTIES, the following sub-functions list will be displayed on the right hand side of the screen.

Material
AddGrp
CS
Variable
Aero Cl

Modification of materials and material properties. Modification of reinforcement/stress groups. Modification of cross-sections and cross section properties. Modification of variables.

Definition of additional wind properties.

On selection of \hat{U} STRUCTURE, the following sub-functions list will be displayed on the right hand side of the screen.

Node
Element
Tendon
Special

Definition of nodes and their attributes.

Definition of elements and their attributes.

Definition of tendons and their attributes.

Definition of special commands.

On selection of \hat{U} LOADS AND CONSTR.SCHEDULE , the following sub-functions list will be displayed on the right hand side of the screen.

Loads
AddCon
Stage

Definition of load cases.

Additional constraints for optional DOF's.

Definition of constructions stages.

A dialogue window is opened on selection of \hat{U} RECALC. There is no sub function for 'Recalc'.

On selection of \hat{U} RESULT, the following sub-functions list will be displayed on the right hand side of the screen.

LCase	Loading case results in list form for nodes and elements.
Envelope	Envelope results in list form for nodes and elements.
PISys	File editor for the creation of plot-files.
PICrSh	Screen Plot - element by element - of creep and shrinkage.
Plinfi	Screen Plot of influence lines for all degrees of freedom.
Report	Result report for selected elements/nodes and load cases/envelopes.
Lists	Result report for selected elements/nodes and load cases/envelopes.

RM2000

4-5

Getting Started

On selection of \hat{U} SCRIPTS, the following sub-functions list will be displayed on the right hand side of the screen.

RunTcl	Run a tcl-script.					
OpenTcl	Open an existing tcl-s	cript.				
Action	Execute calculation Schedule".	action	independently	from	the	"Construction

C Variable

C:\work\FirstProject

5 The Default – Database

Define the properties of the materials to be used in the project.

- Import the materials necessary for the project
- Select ¹↓FILE ⇒DEFAULT to activate the Default-Database dialogue box shown below.

Material

Default database

The dialogue box contain two tables.

The left table show all information in the Default-Database. The right table show all information in the current project.

Now we copy all Materials from the Database into the current project.

- Select <Mark all> (the colour of all Material names will change into red)
- Select <->copy->> (All materials appears in the right table)
- concrete concrete C_35 C_40 C_45 C_50 C_55 C_60 DYWIDAG_EX concrete concrete concrete concrete External Tendons (1570/1770) bounded tendons (1570/1770) unbounded tendons (1570/1770) DYWIDAG NV DYWIDAG DV • Mark Unmark Close Mark all Material C Cross s C Variable <-- Copy <--Default database C:\work\FirstProjec --> Copy --> concret concret C_40 C_45 C_50 concrete concrete concrete concrete External Tendons (1570/1770) C 55 n en DYWIDAG_EX External Tendons (1570/1770) bounded tendons (1570/1770 DYWIDAG NV bounded tendons (1570/1770) unbounded tendons (1570/1770) ne (1570/1770 DYWIDAG OV • Mark Unmark Mark all Close

C Cross s.

<-- Copy <--

--> Copy -->

- Select 'Variable'
- Select <Mark all> (the colour of all Variable names will change into red)
- Select <->copy->> (All materials appears in the right table)
- Select <Close> to close the window.

C Material		C Cross s.		 Variable
Def	ault database	< Copy < > Copy>		C:\work\FirstProje
C78ba	Initial plastic flow for creep	according 🔺 🚺	78ba	Initial plastic flow for creep according
C78bf	Plastic flow according aplic	ation of Ic	78bf	Plastic flow according aplication of Ic
C78bs	Shrinkage flow - time 't' (CE	B-FIP78) 0	78bs	Shrinkage flow - time 't' (CEB-FIP78)
C78bs0	Shrinkage flow - time 't0' (0	EB-FIP78 0	78bs0	Shrinkage flow - time 't0' (CEB-FIP78
C78cf	Concrete factor depends of	n concret 0	78cf	Concrete factor depends on concret
C78cr	Creep coefficient after 28 (Jays (CEB 🛛 🗘	78cr	Creep coefficient after 28 days (CEB
C78es0	Basic shrinkage coefficien	(CEB-FIF C	78es0	Basic shrinkage coefficient (CEB-FIF
C78es1	Shrinkage coefficient depe	nds on er 🛛 🖸	78es1	Shrinkage coefficient depends on er
C78es2	Shrinkage coefficient depe	nds on no 🛛 🛛 🗘	78es2	Shrinkage coefficient depends on no
070-10	Modulus of elasticu at the	age tO (CE 🚽 👘 🕐	78et0	Modulus of elasticy at the age t0 (CE)

> Select ⇒PROPERTIES MATERIALS to activate the view/edit dialogue box shown on the next page:.

This window is described in detail as similar windows are frequently used in RM2000. The upper of the two displayed tables lists all the materials imported into the project.

lame	Group	Description		
_30	Concrete			
_35	Concrete			
2_40	Concrete			
2_45	Concrete			
C_50	Concrete			
C_55	Concrete			
c_60		Material-Characteristics for C_30	11eð	<u>•</u>
C_60	Concrete	Group Value	Unit	<u>×</u>
C_60	Concrete Concrete Description E-Modulus longitue F-Modulus longitue	Index State State 40000-000-00	Unit kN/m2 kN/m2	-
C_60	Concrete Concrete Description E-Modulus transve Poissons's table	Group Value inally Static +2.620+07* static +0.000+00 5 tatic +0.000+00 Static +0.000+00 0.20183 0.20183	Unit kN/m2 kN/m2	
E_60	Concrete Concrete Description E-Modulus longitur E-Modulus Innerve Proison's ratio Shear Modulus	Group Value inally Static +2.620+07 Static +0.000+00 Static 0.20183 Static 0.20183 Static +1.000+00	Unit kN/m2 kN/m2 kN/m2	•
Name E-Modi Poiss. G-Mod ALFA-T	Concrete Concrete Description E-Modulus transve Poissons's ratio Shear-Modulus Coeff. of themal et	Group Value Static +2.620e407 static +2.620e407 Static +2.620e407 Static +0.000e400 Static -0.20183 Static +1.000e407 Static +1.000e05 Static +1.000e05	Unit kN/m2 kN/m2 kN/m2 1/C	•
C_60	Concrete Con	Group Value inaly Static +2.630+47 traily Static +0.000+30 Static -0.2018-30 Static static +1.000+47 Static	Unit kN/m2 kN/m2 kN/m2 1/C kN/m3	×
Vame Mode Mode Mode ALFA-T Gamma Fc28	Concrete Concrete Description E-Modulus Ionghue E-Modulus Ionghue E-Modulus Ionghue Codus Ionghue Codus Coeff. of themal er Specific weight 28 day concrete og	Group Value inally Static +2.620e407 static +2.620e407 -2.620e407 Static -0.000e400 -2.620e407 Static -0.000e400 -2.610e Static -1.000e400 -2.610e Static +1.000e407 -2.610e Static +1.000e407 -2.430e401 finder strength Concrete +0.000e400	Unit kN/m2 kN/m2 1/C kN/m3 kN/m3	*

The lower table lists the properties of the material that is currently selected (blue line) in the upper table.

Material 'C_30' is selected from the upper list, in the above window and the material properties for 'C_30' are listed in the lower table.

The content of the tables can be changed using the various interactive buttons (described below):.

Insert a new line <u>before</u> the selected line



Edit the selected line.



Insert a new line <u>after</u> the selected line.



Make a copy of the selected line. The copy is inserted at the end of the list.



Show information about the selected line.



Delete one or more lines.

6-1

6 Modify a material

Select the material C_45 in the material list (upper table) Select the information button

An input/edit window will be displayed with the material properties. Most of the properties are 0 by default for a new material.

The creep calculation is based on the Ceb90 model in this example.

Assign this creep model to the material

The 28 day concrete cylinder strength and the type of cement is needed for creep calculation in accordance with Ceb90 or Ceb90.

> Select the PHI(t) arrow for creep.

- Select the correct model (AS 96cr) and confirm with <ok>. (see picture below).
- > Do the same for EPS(t) and EMOD(t).
- Input the other material properties by hand – use the same values shown in the screen shot on the last page.
- Confirm the material property inputs with <ok>.



The program will ask whether the properties of this material should permanently change.

Confirm with <YES>.

Close the material info window by clicking on the cross $\langle x \rangle$ at the top right hand corner of the window.

 $\hfill \ensuremath{\mathbb{C}}\hfill TDV-Technische Datenverarbeitung Ges.m.b.H.$

7 Check the cross section

> Select ûPROPERTIES ⇒CS to open the input window for cross section viewing and/or definition.



The table on the left side displays a list of all the cross sections that were defined in GP2000.

The selected cross section is displayed graphically on the right hand side of the screen.

The buttons at the bottom left hand side of the window have the following meanings:

CS	Cross section view.
Nodes	Cross section nodes.
Elem	Cross section elements.
Values	Cross section result values.
Comb	Composite cross section for hinge springs.
AddPnt	Definition of additional points (reinforcement, stress points)

With the button at the bottom right hand side of the window it's allowed to start the calculation in each position of the input procedure.

Definition of the structural system 8

8.1 Nodes

> Select ☆STRUCTURE ⇒NODE to open the input window for viewing and/or defining the nodes.

The	wind	ow	R ^M Structu	re : Node	e - Properties - Node p	osition							
displays		the		\checkmark	ter Er	i) i	(m),(m),(KN),(KNm),((KN/m2),(ø	C),(Deg) 🔽		Node Ele	ment Tendon	End
nodes a	and	co-	Node	Supp	x(m) v(m)	z (m)	Status	Node	Supp :	×(m) v(m	n) z(m)	Status	
ordinates	s for	the	101	No	0.000	0.000	0.000 Deact	120	No	75.863	0.738	2.918 Deact	-
			102	No	4.000	0.067	0.000 Deact	121	No	79.809	0.708	3.575 Deact	-1
project a	as it s	vas	103	No	8.000	0.133	0.000 Deact	122	No	83.740	0.671	4.311 Deact	
project t		vus	104	No	12.000	0.200	0.000 Deact	123	No	87.656	0.626	5.125 Deact	
alroady	dofi	her	105	No	16.000	0.267	0.000 Deact	124	No	91.556	0.572	6.018 Deact	
ancauy	uem	icu	106	No	20.000	0.333	0.000 Deact	125	No	95.436	0.511	6.988 Deact	
using CD	2000		107	No	24.000	0.400	0.001 Deact	126	No	99.296	0.442	8.036 Deact	
using OP	2000.		108	No	28.000	0.467	0.008 Deact	127	No	103.135	0.368	9.160 Deact	
-			109	No	32.000	0.532	0.029 Deact	128	No	106.950	0.295	10.361 Deact	
			110	No	36.000	0.591	0.068 Deact	129	No	110.741	0.221	11.638 Deact	
			111	No	39.999	0.642	0.133 Deact	130	No	114.505	0.148	12.991 Deact	
			112	No	43.998	0.684	0.230 Deact	131	No	118.242	0.075	14.419 Deact	
			113	No	47.996	0.719	0.366 Deact	132	No	121.949	0.001	15.921 Deact	
			114	No	51.992	0.746	0.546 Deact	133	No	125.625	-0.072	17.497 Deact	
			115	No	55.985	0.764	0.777 Deact	134	No	129.269	-0.145	19.146 Deact	
			116	No	59.974	0.775	1.066 Deact	135	No	132.879	-0.219	20.868 Deact	
			117	No	63.959	0.778	1.419 Deact	136	No	136.455	-0.292	22.662 Deact	
			118	No	67.936	0.772	1.842 Deact						
			119	No	71.905	0.759	2.340 Deact						-

The buttons at the bottom left hand side of the window have the following meanings:

Node Coordinates.
Node supports (spring constants).
Node support orientation and length.
Node support eccentricity.
Node masses (dynamic).

8.2 Elements

All the nodes and elements assignments were defined in the GP2000 getting started example so it is not necessary to define them here.

> Select ☆STRUCTURE ⇒ELEMENT to open the input window for to view the element definition.

The meaning of the button names at the bottom left hand side of the window are given below:

Elem	Mat	CS	Comp	Beta	Ecc	Hinge	Time	Shape	Checks	
------	-----	----	------	------	-----	-------	------	-------	--------	--

RM2000

8-2

Getting Started

Elem:	Element input (type definition, node assignment, sub-division).
Mat:	Element/material assignment. Material values are assigned, or Material is chosen from a material list that assigns values.
CS:	Element/cross-section assignment or definition of spring constants for all types of element springs.
Comp:	Composite elements and their sub-component parts (max 4).
Beta:	Element orientation and length.
Ecc:	Element eccentric connections.
Hinge:	Element begin and -end hinge releases.
Time:	Time dependent properties used for dynamic as well as creep and shrinkage calculations.
Shape:	Pre-deformation and pre-loading of elements.
Checks:	Element/Reinforcement assignment and checks definition.

The data input window changes on selection of another button.

- > Assign material properties to the elements.
- > Select the <Mat> button at the bottom left hand side of the screen to open the material assignment input window.

Structu	re : Element Proj	perties - Material	inicaicaisian caicaicais	anie anie iniezianie iniezianie	istic alicializiatic alic	an elarite ante antesan istanite	्यांत आदियांत ज्यात आदियां हरेत	n ains an siàr ill
) ,	<u> </u>	b 🖻 i	. (m),(m),(kN),(kNm),(kN	1/m2),(C),(Deg) 🛛 👻		lode Element	Tendon Special	End
Elem	Туре	Mat-Nam	E-Mod (kN/m2)	Poiss.	G-Mod (kN/m2)	Alpha-T (1/C)	Gamma (kN/m3)	
101	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
102	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
103	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
104	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
105	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
106	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
107	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
108	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
109	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
110	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
111	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
112	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
113	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
114	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
115	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
116	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
117	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
118	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	
119	Beam	C_45	+3.210e+07	0.197761	+1.340e+07	+1.080e-05	24.300	-
								_
Elem	n Mat	CS Comp	Beta Ecc	Hinge Time	Shape Check	:s		Recalc

Materials have already been imported and/or defined in GP2000 for this example. The Material properties can be modified by selecting the appropriate element and then the edit button.

8.3 Cross sections assignment

The cross sections have already been assigned in GP2000 for this example The cross section assignment can be modified by selecting the appropriate element and then the edit button. The TypBeg or TypEnd input window arrow can be chosen to assign a different cross section to the element. With EccTyp it's allowed to change the typ of eccentricity form the cross section.

Element types can be changed. The beam elements eccentricity type can be changed

Close the input screen by clicking the <x> in the upper right hand corner of the window or with <END> .

The graphic screen shot can be updated by using the redraw/re-zoom facility.

- > Use the freehand symbol 'V' to zoom all and redraw.
- > The freehand 'V' symbol must be drawn directly on the screen using the left mouse button whilst simultaneously holding the <ctrl> key on the keyboard down.

8.4 Calculation

The structural system definition is now complete except for the tendon geometry. The system can be calculated for the first time.

	월 Calculation				salic alic alic a			ic a le a le alle alle alle alle a		
> Select	Project text 1	First Projec	t							
 	Project text 2	3 span gird	ler with interr	al prestressing						
open the input	Units									
· · · · ·	Angle(structure)	(Deg)	-	Forces		(kN)	-	Time(Schedule)	Day	
window for	Angle(results)	Radians		Moment		(kNm)	•	Time(Load)	Second	
global project	Length(structure)	(m)	-	Stresses	/Reinforceme	nt (kN/m2)	-	Deflect. Factor	1000	
calculation	Length(CS)	(m)	•	Tempera	ature	(C)	-	Force Factor	1	
property defini	Structure type	Space fram	ne				-	Coord. system	Left	
property defini-	Active DOF	▼ Vx	I ▼ Vy	▼ Vz	🔽 Phi-x	🔽 Phi-y	🔽 Phi-z	Standard	AASHTO 👻	
tions.	Max/Min Displ.	∏ Vx	∏ Vy		🥅 Phi-x	🗖 Phi-y	🥅 Phi-z	Constr.start	13 12 2001	
	Max/Min Forces	▼ N	🔽 Qy	🔽 Qz	Mx 🔽	🔽 My	I ✓ Mz			
The pre-defined	Calculation			Calculation	уре			Special settings		
The pre-defined	Cross-section calculation			Ignore shear deformation				✓ Save tendon results (LC)		
parameters in the	Structure check			P-Delta effect				I▼ Save tendon results (Env)		
"Recalc" window	Stage activation			🔲 Stay Cable nonlinear				Update CS(+tendon sterministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerministerminis	el area)	
haven oon he	Stage actions	1.0		Large dis	placements			Update CS(-duct area)		
boxes can be	✓ Influence-lines cal	culation		Nonlinea	r Material prop	erties		I Update US(+fill area)	w mathead	
accepted or modi-	Plot to plot file				ate nermanent	load		Create primitate due to	TempVar	
fied as required.	Create image (Bitm	an)		Anniv Co	nstruction star	te constraints		Print creen and shrinka	ne factors	
neu us requireu.				C Accumul	- ate stiffness (S	umLC)		Store part.forces due to	creep	
				□ .				Calculate shear area fo	rCS	
				<u>[</u>						
	GrpFile	default.grp					•	SumLC 1000	Uk	
	Recalc	Co	nvergence	Dynamic	C	+S	Printer	CS Int	Cancel	

Heinz Pircher und Partner

•

8-4

Most of the default values are acceptable for this example.

The units for the input as well as the output can be specified by the user. The units can be changed <u>before</u> and <u>after</u> the calculation.

i.e. the calculation can be made using one set of units and results and can be viewed and printed out in a completely different set of units.

Each type of input you can have a separate unit. (Length of structure, Length of cross section, Force, Moment, stress, ...)

A brief description is given below for defining the input data units. Refer to chapter 12 for a description on how to modify the output units.

🕅 Length

Milimeter

C Centimeter

• Meter

C Inch

C Foot

C Yard

C Other

Πk

How to change the units (e.g. Moment)?

- Select the arrow on the right hand side of the Moment unit input.
 - Select the arrow on the right hand side of the Length unit input.
- Woment
 ▼

 Unit
 Factor to (KNm)

 Force
 (KN)

 Length
 (m)

 Moment
 (KNm)

 1
 1

 Ok
 Cancel

Factor to (m)

1000

100

39.3701

3.2809

1.0936

Cancel

Unit

(mm)

(cm)

(m)

(in)

(ft)

(yd)

- Define the unit for Force (kN for default)
- > Confirm with $< o\kappa >$.
- > Define the unit for Length (m for default)
- > Confirm with <ok>.
- > Confirm with <ok>.

Modify the following to suit this example (refer to the screen shot on the previous page):

- > Input a project text.
- > Switch to AASHTO.

Only a cross section calculation and a structure check can be done at this stage.

- ☑ Check 'Cross section calculation'.
- \square Check 'Structure check'.

© TDV - Technische Datenverarbeitung Ges.m.b.H.

 \square Uncheck all other Calculation options.

Confirm with <**Recalc>** to start the calculation.

9 Definition of tendons

9.1 Definition of tendon groups

> Select \hat{U} STRUCTURE \Rightarrow TENDON to open the tendon list.

All the tendons for the current project are listed in the upper table and the properties of the selected tendon are displayed in the lower table.



Select the append button to open the input window for tendon groups definition.

• Select 'Typ Internal' to define an internal tendon. Input the data shown in the adjacent screen shot.



Geometry

Confirm with <ok>

9.2 Definition of the tendon geometry

The three function buttons at the bottom of the screen have the following meaning:

Assignment	Tendon/Element assignment.
Geometry	Tendon geometry, type (intern/extern), material and cross-section
	properties.
3D-Values	The Calculated tendon geometry will be displayed graphically.

These functions are used to define the tendons.

Assign the tendon group to the elements.Select <Assignment> to open the appropriate input window.

The tendon groups are listed in the upper table and the elements assigned to the selected tendon are displayed in the lower table.

R ^M Elem-Te	Elem-Tendon 🛛 🔀										
El-from El-to	101										
El-step	1										
Ok		Cancel									

3D-Values

Se > Se

Select the tendon group.

Click the (lower) append button to open the assignment input window.

- > Input the data shown in the adjacent screen shot.
- > Confirm with <ok>.

Tendon 1 is assigned to Elements #101 to #113 here. A graphical display of the tendon geometry can be viewed as follows:

 Select the info button between the upper and

> The details of the tendon profile for the selected tendon and element are displayed in the left portion of the screen and the whole profile for the selected tendon together with a cross section plot

lower tables.

¥ Tendon	0101 Act	ive mode: C	Delete															E
Type	Normal			•		wEast 1.00	11 Set	1.00		itep dx (m)	Step dy I	(m) V	iew sometric		3D-Va	lues	<u></u>	
Ref.Elen	101	CS pnt		•		No dec 11.04	58	0.000	ssisection 1		10.100	(C)	oomouto	<u> </u>				++
	C	Giobal 🤇	C Local														1	+
×/I	e	y (m)	ez (m)														
0.000	0	0.000	0.00	0														
Relative	to	* Mada	00															
te Elem	,	Node	00															
Alpha1 (I	Deg)	Alpha	2 (Deg)															
99999.00)0	9999.	.000				1//	11		/ / /	/ /	'	/ /	/	/ /			
C Value	e 💽 Free	e OVa	lue 📀	Free			200 102 1	19 104 -	205 206	pol pol	+ 104	110 ×11	+12 7	13	t			
Relative	to	* Mada	00			ß	and ret	100 104	200 /10	e 107 /1	ਕੇ ਮੁੱਡ	hið	ul ju	R	3			
ter taltalli			00	- prin		- W	VI VI VI	Wr W	V V	a Par P	the free	V	12 12	· V				
Exten	n		r ju Ior															
Hadius (i	mj		Juit	,														
App	dy		Can	cel														
199		C	B.	~													_	•
<u></u>	1	1997	<u> </u>		I													
Туре	R.Elem	GI-Lo	Rel	CS pnt	x-x/l	y-ey	z-ez	Rel	dx-alph1	dy-alphi	2	dz-0	Stra	aight	NElem	Radius (r	1)	
Normal	101	Loc	Elem		0.000	0.000	0.000	Elem	Free	Free								1
																	- 1	
																		-1
																	_	10

of each element that the tendon profile passes through are displayed in the right portion of the screen. The parameters shown in the left portion of the screen correspond with the tendon profile at the position marked by the vertical line.

The tendon geometry is defined in 3-D space relative to an element (in the 'y' and 'z' directions) at any position along the element length (defined by x/l).



Click the append button to activate the input fields on the left.

- Select 'local' to define the tendon geometry locally relative to the selected element.
- > Input '101' as the reference element to define the cable geometry at the first element.
- Set X/L=0 to define the geometry as starting at the beginning of element 1 (X/L=1 defines the end of the element).
- > Set the eccentricity to 0 for both direction ('e_y', 'e_z'). The tendon location will then be at the centre of gravity of element 1.- on the centroidal axis.
- Select 'free' for 'Alph1' and 'Alph2' to let RM2000 calculate the angles. (The edit boxes for the angles will be deactivated following this selection).

> Select <**Apply**> to save the changes.

Make the next definition in 'Cross-section' view instead of 'Perspective-view'.

• Select 'Cross-section' at the top of the graphic screen.

**The order Step dz (m) 0.100 Step dy (m) View 0.100 C Iso ▼ 3D-Values > >> • tendon of CS pnt Fi-F definition is critical. CS pnl Always select the last line before Value
 Value
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V
 V se-**Tendon-AXIS** lecting the C CS pnt C Elem append button 0.0 Radius (m to ensure that Cancel the new data is appended to **(**F) 65 ۵, -/ X the previous R.Eler GI-Lo CS pnt 0.000 0.000 Elem Nod definitions. Fi-B 0.000 104 10000 •

The position of the tendon group centroid $(e_y \text{ and } e_z)$ will be constructed graphically for this element.

The centroid is defined by the intersection of the two dashed black lines (the 'tendon axis').

Eight tools are provided for moving the tendon axis. These tools are located above and to the left of the graphic screen.

The following tools are provided to move the vertical axis:



These buttons perform the following actions:

- << The vertical axis is moved to the extreme left edge of the cross section.</p>
- The vertical axis is moved to the left by one. 'dz-cursor' step.
- >> The vertical axis is moved to the extreme right edge of the cross section.
- > The vertical axis is moved to the right by one. 'dz-cursor' step.

The following tools are provided to move the horizontal axis:



These buttons perform the following actions:

- The horizontal axis is moved to the top of the cross section. ++
- +The horizontal axis is moved towards the cross section top by one 'dycursor' step.
- The horizontal axis is moved to the bottom of the cross section. ___
- The horizontal axis is moved towards the cross section bottom by one 'dycursor' step.

The dy (and dz) cursor step can be user defined – see below.

The eccentricity values e y and e z are refreshed automatically after each move of the axis.

The tendon group, in this example, at element 104 is located on the centre line of the cross section and at 0.20m above the bottom edge. (The vertical axis for the cable group stays on the centre line.)

Select the last line in the list.

Click the append button to start a new geometry definition.

- Select 'local' to define the tendon geometry locally relative to the selected element.
- > Open the CS-Point list and select 'bottom fibre'.
- > Input '104' as the reference element.
- \square Select relativ to QS pnt.
- > Input '0.10' in the window for 'Step-dy-cursor'.
- > Select < + > twice to move the horizontal axis up by 0.20m.
- Select 'Value' for 'Alph1' and 'Alph2'.
- > Keep the value '0' for Alfa1 and Alfa2
- > Select <**Apply**> to save the changes.
- > Check the cable geometry defined so far by changing the view to 'Perspective view'.

Define the next point using a different cable geometry tool

- × > Select <x> to close the info view.
 - > Select <Geometry> (bottom left) to open the geometry definition list.
 - Select the last defined point.



- Click the append button to activate the tendon point input window.
- > Input '111' as the reference element.



Getting Started

9-5

- > Open the CS-Point list and select 'top fibre'.
- Select 'Local' to define a local reference for the cable group centroid.
- > Input '0' for the element begin in X/L.
- > Input '-0.2' for e_y eccentricity.
- ☑ Select Relative to CS pnt.
- ⊙ Select 'Value' for 'Alph1' and 'Alph2'.
- > Keep the value '0' for Alfa1 and Alfa2
- > Select <ox> to save the changes.

> Input the next point similarly:

> Use the following table to complete the geometry for the tendon 1:

have at the state but s	STRUCTURE	TdNum		1(01	
Input the cable		Ref. Elem.	101	104	111	113
geometry	TENDON	CS pnt	-	bottom fibre	top fibre	-
		X/L	0	0	0	1
	GEOMETRY	e _Y [m]	0	0.2	-0.2	0
		e _z [m]	0	0	0	0
	Bottom table	Rel. to	☑ Elem	☑ CS pnt	☑ CS pnt	☑ Elem
	•	Alfa1	☑ Free	☑ Value	☑ Value	☑ Free
	!	Value	-	0	0	-
		Alfa2	☑ Free	☑ Value	☑ Value	☑ Free
		Value	-	0	0	-
		Rel. to	☑ Elem	☑ Node	☑ Node	☑ Elem
		Straight part				
		Extern				

The definition of the cable geometry for construction stage 1 is now complete.

Copy functions can be used to define the cable geometry.

- > Select the first cable group definition in the upper table.
- Click on the copy button to open the copy input window.
- > Input '102' in the 'New tendon' field.
- > Modify the Element begin to (108).
- Confirm with <ok>

Now, the Program had made a copy of the tendon 101 but translated to the start element 108. All other parameters are the same. The geometry definitions and the assignment must therefore be changed.

The element assignment must also be changed from #108-#120 to #108-#128.

- > Apply the changes.
- > Use the following table to complete the geometry for the tendon 2.

M Tendon Copy		×
Original tendon	101	-
New tendon	102	
Elem. begin	108	
Ok	Ok+Series	Cancel

TdNum			102		
Ref. Elem.	108	111	118	126	128
CS pnt	-	top fibre	bottom fibre	top fibre	-
Global/Local	• Local	• Local	OLocal	OLocal	• Local
X/L	0	0	0.5	0	1
e _Y [m]	0	-0.2	0.2	-0.2	0
e _z [m]	0	0	0	0	0
Rel. to	☑ Elem	CS pnt	CS pnt	CS pnt	☑ Elem
Alfa1	☑ Free	☑ Value	☑ Value	☑ Value	☑ Free
Value	-	0	0	0	-
Alfa2	☑ Free	☑ Value	☑ Value	☑ Value	☑ Free
Value	-	0	0	0	-
Rel. to	✓ Elem	☑ Node	☑ Node	☑ Node	☑ Elem
Straight part					
Extern					

- > Change the numbers of cable from 6 to 14.
- \blacktriangleright Close the geometry window by selecting $\langle x \rangle$.

Use the copy functions to define the third cable geometry.

- > Select the cable group definition 101 in the upper table.
- > Click on the copy button to open the copy input window.
- > Input '103' in the 'New tendon' field.
 - > Modify the Element begin to (123).
 - Confirm with <ok>

All other parameters are the same and can be copied directly.

The geometry definitions must be changed or created from scratch.

The element assignment must not be changed from #123-#135.

> Use the following table to complete the geometry for the tendon 3

Getting Started

TdNum		1(03	
Ref. Elem.	123	126	132	135
CS pnt	-	top fibre	bottom fibre	-
Global/Local	•Local	•Local	•Local	●Local
X/L	0	0	0	1
e _Y [m]	0	-0.2	0.2	0
e _z [m]	0	0	0	0
Rel. to	☑ Elem	CS pnt	CS pnt	☑ Elem
Alfa1	☑ Free	☑ Value	☑ Value	☑ Free
Value	-	0	0	-
Alfa2	✓ Free	☑ Value	✓ Value	✓ Free
Value	-	0	0	-
☑ Elem	☑ Elem	☑ Node	☑ Node	☑ Node
Straight part				
Extern				

 \blacktriangleright Close the geometry window by selecting $\langle x \rangle$.

The tendon definitions are now complete and will be displayed in the main graphic screen after using the redraw function. (The freehand 'V')

The screen, showing the cable profile, should look like this:

PRM200	0 8.49.01@d [C:\worl	k\FirstProject]					
	🔪 🔝 📰 🖌	Crt 🔍	😒 🍏 🍡 🔟 Licenced for TDV int	emal Main			
File	Properties	Structure	Loads and Constr.Schedule	Recalc	Results	Scripts	Main
Commanc	<u>د د د د</u>		<u></u>			en e	

The tendon profile is drawn in a turquoise colour.

[©] TDV – Technische Datenverarbeitung Ges.m.b.H.

9.3 Definition of the tendon stressing schedule

All the tendon stressing actions are defined in the construction schedule.

- Select û LOADS AND CONSTR.SCHEDULE ⇒STAGE to start the stage definitions.
- > Select <Tendon> (lower left side) to input the tendon actions.

All the actions that are applied to the tendons are defined in the two tables in this window.

The upper table lists all the actions applied to the tendons.

The lower table displays details of the action that is selected in the upper table. Define the following actions:

- 1. Stress the left end of tendon group 1 to a stress of 1.08 times the 'allowable stress'.
- 2. Losses due wedge slip on the left side (10mm)
- 3. Stress the right end of tendon group 1 to a stress of 1.08 times the 'allowable stress'.
- 4. Losses due wedge slip on the right side (10mm)

1 Stress the tendon on the left

()	۶	Select the (upper)	R ^M Tendon-List	₽^MTendon actions	×
	~	append button to open the tendon action input window.	TdNum Description 101 Tendon 1 102 Tendon 2 103 Tendon 3	Stress/Relax Type © PREL C Force C PRER © Factor C RELL C PELP	Wedge C WEDL C WEDR
•	~	window arrow. Choose 'Tendon 1' in the list		Tendon Number of cables	101 v
	۶	Confirm with <ok></ok>	Ok Cancel	Fact from SIGmax Stress-label	1.08 CS1
	0	Select 'PREL' as action	on type. The 'L' means on the	Description	

- left side (begin).
 Select 'Factor' to define a stress factor instead of a stress force.
- Input 1.08 as the factor.
- > Input 'CS1' as assignment to a construction stage in the edit box of the 'stress-label'.
- > Confirm with <ok>.

Cancel

Ok

RM2000

Getting Started

2

9-9

Wedge slip on the left side.

- Select the (upper) append button to open the tendon action input window.
- Select a tendon by clicking on the tendon window arrow.
 - > Choose '101' in the list.
 - > Confirm with <ox>.
 - Select 'WEDL' as action type for wedge slip on the left.
 - ⊙ Select 'Factor'.
 - > Input 0.01 as wedge slip (N.B. Units in metres)
 - Input 'CS1' as assignment to a construction stage in the edit box of the 'stress-label'.
 - ➤ Confirm with <ok>.

PM Tendon actions X Stress/Relax Туре Wedge O PREL C Force • WEDL C PRER Factor O WEDR **O RELL O BELB** 101 Tendon • 6 0.01 Wedge (m) CS1 Stress-label Description Ok Cancel

The next two actions are similar except that they are for the right hand side.

Create these next actions using the following parameters: PRER for stressing, factor of 1.08 and WEDR for a wedge slip on the right end.

The tendon schedule should now be the same as in the screen shot below:

	Tendon	Number	Туре	Data	Stress	label		De	scription				
PREL	101	6	Fact	1.0800	CS1			_					
VEDL	101	6	Fact	0.0100	CS1								
RER	101	6	Fact	1.0800	CS1								
VEDR	101	6	Fact	0.0100	CS1								
													•
		A	(Cha										
24	\sim	`)							X	Action list for ten	don 101		
Action	Number	Туре	Data	Stress-la	oel	Description	Action	Number	Туре	Data	Stress-label	Description	
REL	6	Fact	1.0800	CS1			1						
VEDL	6	Fact	0.0100	CS1									
nee	6	Fact	1.0800	CS1									
"NEN	6	Fact	0.0100	CS1									
VEDR													
VEDR													
VEDR													
WEDR													

The tendon force variation diagram as a result of friction, wobble and these actions can be seen graphically.

- i
- > Mark the last line in the top table.
- Press the 'info' button.

Getting Started

A screen plot of **all** the tendon schedule actions will be made when the 'last action' is selected in the upper table before pressing the 'Info' button.

To view the screen plot of the first 'n' actions, select the 'n'action before pressing the 'Info' button.

e.g.: To view a screen shot of the first two tendon actions, select the second action in the



upper table and then press 'Info' - only actions one and two will be displayed.

- Create the actions schedule for tendon 2
- > N.B. Change the stress-label field to 'CS2'
- Create the actions schedule for tendon 3
- > N.B. Change the stress-label field to 'CS3'

The tendon geometry definition and the tendon schedule is now complete.

10 Definition of 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
- 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 loading case can be added to another loading case or added/combined into an envelope.

10.1 Defining loads

Several loads can be combined into one LOAD SET

- Select ☆ LOADS AND CONSTR.SCHEDULE ⇒LOADS to start the load definition.
- > Select <LSET> to open the load definition input window.

The upper table contains a list of the load sets. The lower table contains the actual loading making up the Load Set.

10.1.1 Definition of a load set



Click the append button in the upper table to open the load set input window.

- > Input '101' as the load set number.
- Input 'self weight CS1' as the description for this load set. (Self weight with loading case for construction stage 1).
- > Confirm with <ok>.

Define the loading that makes up the load set.



M Loading 🛛 🔀						
C Concentrated Load	C Stressing					
Uniform Load	C Initial Stress/Strain					
C Partial Uniform Load	C Actions on Element End					
C Trapezoid and Triangular Load	C Wind Load (velocity)					
C Masses	Normal forces (Stiffness change)					
Load type	C Special					
Uniform concentric element load						
Uniform eccentric element load						
Self weight - load and mass						
Gelf weight - just as load						
Self weight - just as mass						
1	•					
Ok	Cancel					

© TDV – Technische Datenverarbeitung Ges.m.b.H.

Heinz Pircher und Partner
- Select 'Uniform Load' as the loading type.
- Select 'Self weight mass with load' from the list of uniform loading types.
- > Confirm with <ox>.

An input window for the self-weight parameters will be displayed.

The self weight load for construction stage 1 consists of elements #101 to #135.

- > Input the element parameters (101/135/1).
- > Input a specific weight of 24.3 $[kN/m^3]$.
- Input a load direction of '-1' in Ry. (i.e the load acts vertically downwards)
- > Confirm with <ox>.

	- 404
GO	From 101
	To 136
	Step 1
Direction vector	
Rx 0.0	Gam (kN/m3) 24.3
Ry -1	Real length
Rz 0.0	C Nodal load
01/	Ok+Series Cancel

The loading for the load set will now be displayed in the lower table.

10.1.2 Define a loading case

This loading case is to be made up from the above load set for later calculations. > Select <Lcase> to open the Construction schedule loading case input window :

۳,

Select the append button in the upper table.Input '101' as the loading case number.

- Choose the 'Type' window arrow to display the load type selection window.
- Select 'Load' for Load Type to indicate a static load (Load types definition is required for the creep calculation)

The different Load Types available are:

Load': Load remains on the structural system. *Load+Unload*' Load will be applied and removed after some time.

RM Load case	×
Number	101
Туре	Li
Load Info	
Location	lc0101.rm
Output File	lc0101.lst
Description	Self weight CS1
Ok	Cancel



Note: Selection of *<OK+Series>* confirms the input as well as opens the input window again. – speeds up data input preparation.

- > Mark the Loadset 101 in the upper Table.
- > Select the append button in the lower list.
- Choose the 'Load Set' window arrow to open the load set selection list
- > Select Load Set 101 from the list .
- > Input '1' for the 'Const-Fac' (static factor.)
- > Leave the dynamic factor blank.

^{®M} Load set	×
Load set	101 💌
Const-Fac	1
Var-Fac	
🔽 Load Set can be incre	ased
Ok	Cancel

Define the other loading cases in a similar way using the following values:

_				
		LOADS and CONSTR.SCHED.	Number	201
	Define Load Set's for		Loading	 Uniform load
		LOADS	Туре	201 © Uniform load Uniform concentric element load 101 135 1 0 -30 0 © Global - © Real length © Load/Unit © Load/Unit
		•	From	101
		LSET	to	135
			Step	1
		Bottom table	Qx [kN/m]	0
			Qy [kN/m]	-30
			Qz [kN/m]	0
			Direction	Global
			Load applica-	Real
			tion	length
			Definition	 Load/Unit length

Top table

	LOADS and CONSTR.SCHED.	Loading	✓ Add to load case			
Insert Load Set		Number	21	22	23	24
	LOADS	LCnr.	21	22	23	24
	LSET					

10-4

Getting Started

					-				
	LOADS and CONSTR.SCHED.	Number		21	22	2	23		24
Define Load Set's for the settlements		Loading	⊛ E End	Element- deforma- tions	 Eler Enddef tior 	nent- orma- is	 Element- Enddeformations 	- a-	 Element- Enddeforma- tions
	LOADS	Туре	Eler	nent-end ormation	Elemer deform	it-end ation	Element-en deformation	d 1	Element-end deformation
ı	·	From	1	1100	120	00	1300		1400
	LSET	to	1	1100	120	00	1300		1400
	•	Step		1	1		1		1
	Bottom table	Vx [m]		0	0		0		0
		Vy [m]	(0.01	0.0)1	0.01		0.01
		Vz [m]		0	0		0		0
		Direction	۲	Global	● Gl	obal	Ioba	I	 Global
		Rx [Rad]		0	0		0		0
		Ry [m]		0	0		0		0
		Rz [m]		0	0		0		0
		Where	۲	Begin	● Be	egin	Begin	1	Begin
Insert Load Set	LOADS and CONSTR.SCHED.	Loadi Numb LCn	ng per r.	 ☑ Add t load cas 31 31 	se loa	Add to d case 32 32]		
Define Load Set's for	LOADS and CONSTR.SCHED.	Num	ber	3	1		32		
ture load		Load	ing	 Initial st 	ress/strain	Initial	stress/strain		
	LOADS	Тур	е	lo	ad	Unitorni	load		
	_	Fror	n	10	01		101		
	LSET	to		13	35		135		
	<u>,</u>	Ste	р	-	1		1		
	Bottom table	Alfa	3	1.08	Be-5	1.	08e-5		
		DT-G	[°C]	1	5		0		
		DT- Y	[°C]	()		10		
		H-Y [<u>m]</u>	()		3.5		
		DT- Z	[°C]	()		0		

0

H-Z [m]

0

10.1.4 Prestressing loading case



Create a new loading set.

> Input the values shown in the adjacent screen shot

R ^M Load set	×
Number	501
Location	ls0501.rm
Description	Tendons
Ok	Cancel

Getting Started

Definition of loads

10-5

- Select the (lower) append button to add a loading case.
- Select 'Tensioning' as the load type.
- Select 'Tendon jacking' from the list.
- > Confirm with <ok>.



The input window for the tendon values will be displayed.

- > Input the tendon selection (101 to 103 in steps of 1).
- Select <LCASE> to define and assign the prestressing load set to a loading case.

Load case	×
Number	501
Туре	L-1 💌
Load Info	_
Location	lc0501.rm
Output File	lc0501.lst
Description	
Ok	Cancel



10.1.5 Creep and shrinkage loading case

Select <LCASE> to define and loading case for creep and shrinkage. It is not necessary do create load sets (<LSET>).

Getting Started

												_
Loads-L	oad set											
3,	<u> </u>) E _b	13	i	(m),(m),(kN),(kNm),(kN	1/m2),(C),(Deg)	- 🚬	Loads	AddCon	Stage	End	
Number	Location		Description			Number	Location	Description				
21	ls0021.rm		Settlement av	kis 1		201	ls0201.rm	Additional Load			_	
22	ls0022.rm		Settlement av	kis 2		501	ls0501.rm	Tendon				
23	ls0023.rm		Settlement a	kis 3								
24	ls0024.rm		Settlement av	kis 4								
31	Is0031.rm		Temperature	1								
32	IsUU32.rm		Femperature Solf woight G	2								
lin	150101.111		Jell Weight u	1								
و2	<u> </u>						>	Load set 0501				
Kw	From To	Step	Proj	Data1	Data2	Data3		Data4 Data5	Data	6		
TENDO	101 10	3 1	d	0.000	0.000	0.0	00	0.000 0.000)	.000		
											_	
1											-	
								1				
Comb	D LManage	LSet	LCas	e La	ane LTrain	Seismic	Wind				Recalc	1

The load set window table in **<LSET>** should be as shown in the adjacent screen shot.

The loading case window tables in **<LCASE>** should be as shown in the adjacent screen shot.

Loads-Lo	ad case													×
D ,		1		173	i	(m).(m)	.(kN).(kNm).(k	N/m2),(C),(Deg)	- >	<	Loads A	AddCon	Stage	End
Number	Туре		Load In	ifo	Location	n	Output-	File	Description					
24	Ŀ				lc002	4.rm	lc002	4.lst						
31	L-I				lc003	1.rm	lc003	1.lst						
32	LI				lc003	2.rm	lc003	2.lst						
101	닌				lc010	1.rm	lc010	1.lst						
201	LI				lc020	1.m	lc020	1.lst						
501	ĿI				lc050	1.rm	lc050	1.lst						
601	Ŀł				lc060	1.rm	lc060	1.lst						
B ,	\checkmark	٣		i					×	Load case	e 0601			ရ ပို့ယ
Load set	Const-F	ac V	ar-Fac	Increa	se I	Description	n	Load set	Const-Fac	Var-Fac	Increase	Descripti	on	
														_
														_
Comb	LMar	nage	LSet	LCase		Lane	LTrain	Seismic	Wind					Recalc

10.2 Definition of a traffic load

The definition of a traffic load for this simple example does not correspond to any known 'norm'. It is purely provided to demonstrate how a traffic load is defined.

Select <Lane> (lower left function list) to open the lane input window.



Getting Started

The upper table in this window lists all the defined traffic lanes, the lower table lists the properties of the selected lane (point on the lane for evaluation).



> Select the (upper) append button to open the input window for lane definition.

- > Input "1" for the Lane number.
- Confirm with <ok>



> Select the (lower) append button to open the input window for the lane properties definition.

On each point of the structure it's possible to calculate a influence line. For a serie of elements you can choose between several macros, who will generate this definitions.

.ction	Description	
/ACRO1X	Macro 1 - Main Girder (no eccentricity)-Longitudinal load	_
ACR01	Macro 1 - Main Girder (no eccentricity)-Vertical load	
ACR01Z	Macro 1 - Main Girder (no eccentricity)-Centrifugal load	
MACR02X	Macro 2 - Main Girder (with eccentricity)-Longitudinal load	
MACR02	Macro 2 - Main Girder (with eccentricity)-Vertical load	
ACR02Z	Macro 2 - Main Girder (with eccentricity)-Centrifugal load	
MACR03X	Macro 3 - Secondary Girder - Longitudinal load (distrib. to secondary girder)	
ACR03	Macro 3 - Secondary Girder - Vertical load (distrib. to secondary girder)	
ACR03Z	Macro 3 - Secondary Girder - Centrifugal load (distrib. to secondary girder)	
ACR04×	Macro 4 - Secondary Girder - Longitudinal load (distrib. to primary girder)	
ACR04	Macro 4 - Secondary Girder - Vertical load (distrib. to primary girder)	
ACR04Z	Macro 4 - Secondary Girder - Centrifugal load (distrib. to primary girder)	
POS3D	Position of Lane (3D)	
POSEL	Position of Lane (element) - LOCAL	
POSEG	Position of Lane (element) - GLOBAL	
POSERL	Position of Lane Relative to Element	
POSFG	Single element load with force	
POSFRG	Single load relative to element	-

- > Select 'Macro2' for lanes with eccentricity and vertical load.
- > Confirm with <ok>

The parameter list for Macro2 will appear. The list is empty as no lane parameters have been defined yet



Select the append button to open the macro parameter definition input window.

- > Input the data displayed in the adjacent screen shot.
- > Confirm with <ok>.

The data is the same for all elements.

The element series will be displayed in the list of parameters for macro2.



> Confirm with <ok>.

Further lanes could be defined at this stage, however, no further lane definition data is required for this example.

> Close the input window with <CANCEL>

Loads-La	ane								
		to Pa	i ³ i	(m),(m),(kN),(kNm),(kN	1/m2),(C),(Deg)	- 🔀	Loads Add	Con Stage	End
Number	Npos	Ninfl	Length (m) L	ocation	Output-File	Info-file	Des	cription	
1	70	0	139.998 la	ane0001.rm	lane0001.lst	lane0001.inf	Lan	e 1	
									-
∰,	1					X Laws 0001			
~									
Kw	Elem	Data1	Data2	Data3	Data4	Data5	Data6	Data7	
POSELY	101	0.0000	0.000	0.000				1.000	
POSFGY	101	0.0000	0.0000	0.0000	0.000	1.000	0.000		
POSELY	101	1.0000	0.000	0.000				1.000	
POSFGY	101	1.0000	0.0000	0.0000	0.000	1.000	0.000		
POSELY	102	0.0000	0.000	0.000				1.000	
POSFGY	102	0.0000	0.0000	0.0000	0.000	1.000	0.000		
POSELY	102	1.0000	0.000	0.000				1.000	
POSFGY	102	1.0000	0.0000	0.0000	0.000	1.000	0.000		•
Comb	LMan	iage LSet	LCase	Lane LTrain	Seismic	Wind			Recalc

The defined lanes will be displayed in the lane window tables. The lanes with lane numbers are displayed in the upper table and the points of the lane are displayed in the lower table.

The lane definition is complete.

Define the traffic loads on the lane.

> Select <LTRAIN> to open the input window for traffic loads.

All traffic load definitions are listed in the upper table. The load properties of the selected traffic load are shown in the lower table.

Getting Started

- > Leave the traffic load number as '1'
- > Leave the factors for maximum and minimum force as '1'.
- ► Confirm with <ok>.

Note: Some 'Design Code Norms' require different factors for maximum and minimum forces

The traffic loading in this example consists of a continuous uniform load of 60 kN/m that ends 3.0 metres before a 'Point Load' and starts again 3.0 metres after the same 'Point Load'. The single Point Load is 1000 kN.

Define the uniform load first:

- Select the (lower) append button to open the input window for traffic load parameter definitions.
- > Select 'Load Train Uniform Load'
- Confirm with <ok>

An input window for uniform load parameters will be displayed.

- Input -60 [kN/m] for 'Q'. The negative sign is necessary to indicate a downward force (in global coordinates.)
- Select 'Before+After'. this uniform load acts before and after a single Point Load.
- Select 'Function No' as the load is not defined by a function.
- > Confirm with <ok>.

The input window will remain open for further input. No further definitions are necessary.

- > Close the window with <Cancel>.
- Define the Point Load
- > Choose 'LIF' for live command.



Action	Description
LITEM	Load Train Uniform and Concentrated Load
LIQ	Load Train Uniform Load
LIF	Load Train Concentrated Load
LIA	Load Train Concentrated Load (AASHTO)
Ok	Cancel

₿ Load Train command

LIQV	Q (KN/m	ı) -60
	Qfunct	
Position of load	Load function	Lb=Min(L,2*A/Ymax)
C Continuos	Function No	🖸 No



10-9

X

The single point load has to be defined as a part of the live loading train.

- > Input -1000 for the load force.
- > Input 3 in 'Dmin' and 'Dmax' as the distance between the uniform load and the point load does not change.
- > 'Dstep' is 0 for the same reason.
- > Confirm with <ok>.
- > Close the window with <Cancel>.

There is a gap of 3 m after the point load before the uniform load starts again.

- > Input 0 as the next load force.
- > Input 3 in 'Dmin' and 'Dmax' as the distance between the new start of the uniform load and the point load does not change.
- > 'Dstep' is 0 for a fixed distance.
- > Confirm with <ok>.
- > Close the window with <Cancel>.
- > Also close the next window with <Cancel>.

The live window should contain the data displayed in the screen shot below:

Loads-Lo	oad Train								
i,	<u> </u>		i ³ i	(m),(m),(kN),	(kNm),(kN/m2),(C),(Deg		Loads	AddCon S	tage End
Number	Fact-min	Fact-max	Location	Fund	tion qlen	Triangle f.	Description		
	1.000	1.000	live0001.rm						^
									•
.		R _{iv}				×	Live 0001		
	q	F	I-from	l-to	I-step	q-Flag F	-Flag AASHTU	Function glen	
JUV JF	-60.000	-1000.000	3.000	3.000	0.000				·····
IF		0.000	3.000	3.000	0.000				
									_
									•
Comb	Manage	I Set	L Case	Lane	Train Seismic	Wind			Becalc
Comb	Livialiage	2360	20030	Lano I.	Jeisinic				Tiecaic

The live load definition is now complete.

🍽 Load Train Cor	centrated L	.oad 🛛 🗙
LIF	F (KN)	-1000
	Dmin (m)	3
	Dmax (m)	3
	Dstep (m)	0
Ok		Cancel

Ħ

11 Definition of a construction schedule

- > Open the construction schedule input window with ^①LOAD and CONSTR. SCHED-ULE ⇒ STAGE
- Select <Activation>

The entire 'activation plan' for the bridge construction is summarised in this window. The upper table displays a list of all the construction/activation stages

The lower table lists all the elements that are activated in the selected construction stage. Already activated or not activated elements are not shown in the lower list. Just new active elements are written into this table. Further the age of the new elements are defined in this table.

۶	Select	the	(upper)	append	button	to	open	the	input
	window	w for	the cons	struction	stage de	fin	ition.		

- > Input 'Load calculation' for the description
- > Confirm with <ok>.

90001.rm
j0001.lst
ad calculation
0

R^MElement Activation/Deactivation

ODeact

101

135

T

28

0

Cancel

Active

From

Τo

Step

Age (Day)

ts (Day)

Ωk

- Select the (lower) append button to open the input window for element activation/deactivation.
- > Activate elements #101 to #135 and the spring elements #1100 to #1400 for CS1 (all existing elements are aktiv now!).
- Input 28 [days] for the age. (This means that loadsincluding self weight- defined in this construction stage are applied to these elements 28 days after they were cast).
- > Input 0 days for shrinkage ('ts'). (This means that shrinkage starts immediately).
- > Confirm with <ok>.

The changes to the construction stage 'Load calculation' are shown in the activation window (Shown in the screen shot below).

It is useful, for calculation purposes, to define additional construction stages that have no element activation but are a separate part of the calculation. Define view more construction stages in this example - one for traffic load calculation, and the other one for the final results.

> Define a construction stage named 'traffic'.

RM2000

Getting Started

- > Define a construction stage named 'Superposition'.
- > Define a construction stage named 'Fibre stress check.
- > Define a construction stage named 'Ultimate load check.

۵,		2	1		13	i	(m),(m),(kN),(kl	Nm),(kN/m2),(C)	,(Deg)	-	×	[Loads	AddCon	Stage	End	
Status	Date			Number	Location	L	ist	Time (Day)	Duration	n (Day)) Descrip	tion					
	13	12	2001	1	stg0001.rm	s	tg0001.lst	0.0	10000	11011	Load ca	alculation				•	
	30	4	2029	2	stg0002.rm	s	tg0002.lst	10000	0.0		Traffic						
	30	4	2029	3	stg0003.rm	s	tg0003.lst	10000	0.0		Superp	osition					
	30	4	2029	4	stg0004.rm	s	tg0004.lst	10000	0.0		Fibre str	ress check					
	30	4	2029	5	stg0005.rm	s	tg0005.lst	10000	0.0		Ultimate	e load check					
	30	4	2029	6	stg0006.rm	s	tg0006.lst	10000	0.0		Shear c	apacity check	<				
Elfrom	FL		Elisten	á ce (Daul te	(Dau)	Action	Elérom	Ekto	F		Construction	i stage 0001 to (D	aul	Áction		
501	1110		1	1.90 (· · · · · · · · · · · · · · · · · · ·	0	Act		2110			1.90 (0.07)	() ()		1103011		
1100	14	00	100	20	0	0											
	1.1			0.0		•											
	-																
															_		
																-	
Ac	tivatio	m		Action	Te	ndon										Recalc	

11.1 Definition of calculation actions

> Open the construction schedule 'Actions' input window by selecting <action>

The upper table contains a list of the defined construction stages.

The lower table contains a list of the actions assigned to the selected construction stage.



- > Select construction stage 1.
- Select the (lower) append button to add an action.
- Select the 'Calculation actions' radio button.
- Select 'Calc' from the displayed list to initialise a loading case.
- > Confirm with <ok>.

• Laiculation	(static)	C Lheck actions	C List/piot actions	
C Calculation	(Dynamic)	C LC/Envelope actions	C System commands	
Action	Description			
Calc	Load case calcu	lation		
Stress	Tendon calculati	on		
Grout	End Prestressing	of a tendon series		
TStop	Time stop for eler	ment series		
Сгеер	Creep and shrink	age calculation		
UpdEmod	Update E-Modul	us and correct the results		
CabSag	Cable sagging (c	orrection of E-Modulus for element series	by factor)	
Infl	Influence line cal	culation		
LiveL	LTrain load calcu	lation		
TempVar	Variable tempera	ture (CS) calculation		
Buckle	Buckling analysis			
Failure	Buckling analysis	: till failure		
Reload	Considering of "N	o tension elements'		
OpenTcl	Open TCL file			
RunTcl	Run TCL file			
				•

© TDV - Technische Datenverarbeitung Ges.m.b.H.

Getting Started

11-3

- Input '101' for the 'Load case number'. '101' can be entered directly or can be chosen from the list displayed following selection of the Load case number 'window arrow'. No further input is necessary.
 - > Confirm with <ok>.

Loading case 101 will be added to the action list.

- Select the (lower) append button to add an action.
- Select the 'Calculation actions' radio button.
- > Select 'Calc' from the displayed list to initialise a loading case.
- > Confirm with <ok>.
- > Input '201' for the 'Load case number'
 - > Confirm with <ok>.

Loading case 201 will be added to the action list.

- Select the (lower) append button to add a third action.
- Select the 'Calculation actions' radio button.
- Select 'STRESS' from the displayed list to add the pre-stressing loading cases.
- > Confirm with <ok>.
- > Input 'CS1' for the stress
- > Confirm with $< o\kappa >$.

Action	×
CACtion will be skipped	
Command	Stress
Inp1: -	_
Inp2: Stress-label	CS1 💌
Out1: -	
Out2: List-file	×
Delta-T (Day)	0
Description	
Ok	Cancel

Repeat these stress actions for 'CS2' and 'CS3'

- > Confirm with <ok> in both cases.
- > Select the (lower) append button to add the next action.
- Select the 'Calculation actions' radio button.
- > Select 'CALC' from the displayed list to calculate a loading case..
- > Confirm with <ok>.
- > Input '501' for the 'Load case number'.
- > Confirm with <ok>.

Select the (lower) append button to add the next action.

- Select the 'Calculation actions' radio button.
- Select 'GROUT from the displayed list to add duct grouting for tendon 101-103.
- > Confirm with <ok>.

<mark>®</mark> MAction	×
Command	GROUT
Inp1: Tendon:From,To,Step	101,103,1
Inp2: -	
Out1: -	
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel



 \mathbf{T}

11-4

	Select the append button to add the next action.	R ^M Action	×
	Select the 'Calculation actions' radio button.		
Þ	Select 'CREEP' from the displayed list	Action will be skipped	
	Confirm with <ox>.</ox>	Command	
۶	Input '601' for the 'Loading case number'.	Inp1: -	· ·
	Input '3' for the 'Number of time steps' –	Inp2: Number of time-steps	3
	enough steps for one construction stage.	Out1: Load case number	601
	Input '10000' for 'Delta-T' to specify the creep	Out2: List-file	×
	duration.	Delta-T (Day)	10000
۶	Confirm with <ok>.</ok>	Description	
Λ	<i>lote: Creep loading cases need no definition in <lset>.</lset></i>	Uk	Cancel
_			
· 📰	Select the (lower) append button to add the next ad	ction.	
<u></u>	Select the 'Calculation actions' radio button.		
6	Select 'CALC' from the displayed list to calculate	a loading case	

- > Confirm with <ok>.
- > Input '31' for the 'Load case number'.

Confirm with <ok>.

Repeat these 'CALC' actions for loading case numbers 32; 21; 22; 23; & 24 > Confirm with <ox>. in all cases

All the above Stage 1 actions are displayed in the lower table refer to screen shot below.

R ^M Action										Þ
۵,	\checkmark		<u>i</u> m	l,(m),(KN),(KNm),(KN/m2)	.(øC),(Deg) _▼		Loads /	AddCon	Stage	End
Status	Number	Location	List	Time (Day)	Duration (Day)	Description				
	1	stg0001.rm	stg0001.lst	0	0	Load calculation				
	2	stg0002.rm	stg0002.lst	0	0	Traffic				
	3	stg0003.rm	stg0003.lst	0	0	Superposition				
	4	stg0004.rm	stg0004.lst	0	0	Fibre stress check				
	5	stg0005.rm	stg0005.lst	0	0	Ultimate load check				
										T
Status	Command	Inp1	Inp2	Out1	Out2	Construction Delta-T (Day)	stage 0001 Time (Day)	Descriptio	on	
	GROUT	101,103,1				0	0			A
	Creep		3	lc0601	×	0	0			
	Calc	lc0031			×	0	0			
	Calc	lc0032			×	0	0			
	Calc	lc0021			×	0	0			
	Calc	lc0022			×	0	0			
	Calc	lc0023			×	0	0			
	Calc	lc0024			×	0	0			_
Ac	tivation	Action	Tendon							Recalc

C Check actions

n file (only if unfan file (only if unfan file (only if unfavalues of the exi

for all val

, with +/-

Ok

Create the load case Add the load case to the LC or *.sup

C LC/Envelope act

C List/plot actions

C System command

Definition of a construction schedule

If the sequence of actions becomes scrambled, the 'copy' button can be used to copy the actions into the correct order onto the end of the list. The scrambled, actions can then be deleted.

C Calculation (Static)

LcInit LcAdd LcDel

> upOr upOr upOrX upSqr upSqrt upCom

C Calculation (Dynamic)

The traffic loading effects are calculated in construction stage 2. The results of the live load traffic calculation will be stored in a superposition file.

The superposition file must be initialised (set to zero) before starting the calculation!

- Select the (lower) append button to add the next action.
 - Select the 'LC/Envelope action' radio button.
 - Select 'Supinit' from the displayed list to initialise the envelope.
 - > Confirm with <ok>.
 - Input the superposition file name as 'live.sup' in the displayed input window.
 - > Confirm with <ok>.



Calculate the Influence lines – The traffic load evaluation is made via influence lines:

- > Append a new action for stage 2.
- Select 'Calculation actions' radio button.
- Select 'INFL' from the displayed list to calculate the influence lines.
- > Confirm with <ok>.
- > Input '1' for the 'Lane-number'.



Heinz Pircher und Partner

•

RM2000

Getting Started

Note:

Getting Started

11-6

- > Confirm with <ok>.
- Calculate the traffic loading results:
- > Append a new action for stage 2.
- Select the 'Calculation actions' radio button.
- Select 'LIVEL' from the displayed list to calculate the live load.
- > Input '1' for the 'Lane-number'
- > Input '1' for the 'Live load number'.
- > Set the output file name to 'live.sup'.



The actions for stage 2 will be shown in the lower table – refer to the screen shot below

tatus	Number	Location	List	Time (Day)	Duration (Day) Description				
	1	stg0001.rm	stg0001.lst	0	0	Load calculation				4
	2	stg0002.rm	stg0002.lst	0	0	Traffic				
	3	stg0003.rm	stg0003.lst	0	0	Superposition				
	4	stg0004.rm	stg0004.lst	0	0	Fibre stress check				
	5	stg0005.rm	stg0005.lst	0	0	Ultimate load check				
3,	×					Construction	stage 0002			
tatus	Command	Inp1	Inp2	Out1	Out2	Construction Delta-T (Day)	stage 0002 Time (Day)	Descrip	tion	2
tatus	Command	Inp1	Inp2	Out1	Out2	Construction Delta-T (Day) 0	stage 0002 Time (Day)	Descrip	tion	
tatus	Command Infl SupInit	Inp1	Inp2	Out1 Iane0001.inf live.sup	Out2	Construction Delta-T (Day) 0 0	stage 0002 Time (Day) 0 0	Descrip	tion	
tatus	Command Infl SupInit LiveL	Inp1 Iane0001	Inp2	Out1 lane0001.inf live.sup live.sup	Out2	Construction Delta-T (Day) 0 0	stage 0002 Time (Day) 0 0	Descrip	tion	
tatus	Command Infl SupInit LiveL	Inp1 Iane0001	Inp2	Out1 lane0001.inf live.sup	Out2 *	Construction Detta-T (Day) 0 0 0 0 0	stage 0002 Time (Day) 0 0	Descrip	tion	
tatus	Command Infl SupInit LiveL	Inp1 lane0001	Inp2	Out1 kane0001.inf live.sup	Out2	Construction Delta-T (Day) 0 0 0 0 0	stage 0002 Time (Day) 0 0	Descrip	tion	

 \mathbf{X} > Close the action window by clicking the $\langle \mathbf{x} \rangle$.

All loading results are stored in superposition files. This will be done in stage 3. Select stage 3 to add several actions in this stage.

Start with the Temperature envelope calculation.

The superposition file must be initialised (set to zero) before starting the calculation!

Getting Started

11-7

1 (100 () ()
1

Select the (lower) append button to add the next action.

- Select the 'LC/Envelope action' radio button.
- Select 'Supinit' from the displayed list to initialise the envelope.
- ► Confirm with <ok>.
- Input the superposition file name as 'temp.sup' in the displayed input window.
- > Confirm with <ok>.

Action	×
Action will be skipped	
Command	SupInit
Inp1: Input-file(*.sup)	-
Inp2: Factor (-)	
Out1: Output-file(*.sup)	temp.sup
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel



- Select the (lower) append button to add the next action.
- > Select the 'LC/Envelope action' radio button.
- Select 'SupAndX' from the displayed list to initialise the envelope.
- > Input 'temp.sup' for the Input-file
- > Input '31' for the 'Loadcase'.
- > Confirm with <ox>.

R ^M Action	×
Action will be skipped	
Command	SupAndX
Inp1: Input-file(*.sup)	temp.sup 🗨
Inp2: Add file(-,LC,*.sup)	31
Out1: Output-file(*.sup)	
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel



Select the (lower) append button to add the next action.

- > Select the 'LC/Envelope action' radio button.
- Select 'SupAnd' from the displayed list to initialise the envelope.
- > Input 'temp.sup' for the Input-file
- > Input '32' for the 'Loadcase'.
- > Confirm with <ox>.

Action will be skipped	
Command	SupAnd
Inp1: Input-file(*.sup)	temp.sup 💌
Inp2: Add file(-,LC,*.sup)	32 💌
Out1: Output-file(*.sup)	
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel

All these Inputs create an envelope temp.sup that combines the loading cases 31 and 32

Getting Started

11-8

Create the settlements envelope.

	Select	the	(lower)	append	button	to	add	the n	ext
9	action.								

- Select the 'LC/Envelope action' radio button.
- > Select 'Supinit' from the displayed list to initialise the envelope.
- > Confirm with $< 0\kappa >$.
- > Input the superposition file name as 'settle.sup' in the displayed input window.
- > Confirm with $< 0\kappa >$.

R ^M Action	×
C Action will be skipped	
Command	SupInit
Inp1: Input-file(*.sup)	
Inp2: Factor (-)	
Out1: Output-file(*.sup)	settle.sup
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel



- > Select the (lower) append button to add the next action.
 - > Select the 'LC/Envelope action' radio button.
 - > Select 'SupAnd' from the displayed list to initialise the envelope.
 - > Input 'settle.sup' for the Input-file
 - > Input '21' for the 'Loadcase'.
 - > Confirm with <**ox**>.

R ^M Action	×
🗖 Action will be skipped	
Command	SupAnd
Inp1: Input-file(*.sup)	settle.sup
Inp2: Add file(-,LC,*.sup)	21
Out1: Output-file(*.sup)	
Out2: -	
Delta-T (Day)	0
Description	
Ok	Cancel

Repeat this input for the loading cases 22,23 and 24.

All these loading cases (21; 22; 23 and 24) create the settle.sup envelope.

Finally create a superposition file total sup that combines everything into one envelope. Add the loadcases 101,201,501 and 601 (with SupAdd) and the superposition files temp.sup, live.sup and settle.sup (with SupAnd) into total.sup.

RM2000 Getting Started

12-1

12 Calculation of the structural system

- > Select **î** RECALC to open the input window for calculation.
- ☑ Check (select) all the calculation element selections ('Cross-section calculation' 'Create image').
- > Start the calculation with <ox>.

The main graphic screen must be redrawn after a recalc,.

> Use the short-cut 'zoom all' function (Drawing a 'V' with both the <Ctrl>-Key and the left mouse button pressed).

The following structural plot will be displayed:



Elements #101 to #135 and Elements #1100 to #1400 are now drawn with continuous lines indicating that they have been activated.

Note: A recalc can be started from most windows by choosing the RECALC> button in the lower right hand corner of the window.

[©] TDV – Technische Datenverarbeitung Ges.m.b.H.

13 Results

All actions that are made to both the structural system and to any calculation result manipulations must be specified under 'Construction Schedule'. Result plots must therefore be inserted into the actions of a construction stage. N.B some results can be shown without a plot-file definition in the action list!.

> Select \hat{U} RESULTS \Rightarrow LCASE to open the loading cases result viewer.

The adjacent	월 Resul	ts - Load Cas	e [Factor=	1000.0000]						×
screen shot	LC	101 -	Element	▼ Tendon	0 💌	LCase Envelope	PISys PICrSH	Plinfi Repo	rt Lists	End
shows the	Elem	101	Local	▼ □ Ndiv Pnt	(m),(rad) 🔹	Results: Normal	•			
local	Elem	Nod	х/1	Vx (m)*Fact	Vy (m)*Fact	Vz (m)*Fact	Rx (Rad)	Ry (Rad)	Rz (Rad)	
local	101	101	0.000	-1.83930	0.00198	0.00133	-0.00058	0.00011	0.31823	
deformations		102	1.000	-1.83982	-1.21733	0.00172	-0.00090	0.00008	0.27907	
deformations	102	102	0.000	-1.83982	-1.21733	0.00172	-0.00090	0.00008	0.27907	
for loading		103	1.000	-1.84014	-2.15162	0.00185	-0.00122	-0.00002	0.18046	
ioi ioauing	103	103	0.000	-1.84014	-2.15162	0.00185	-0.00122	-0.00002	0.18046	
case 101		104	1.000	-1.84027	-2.61966	0.00149	-0.00154	-0.00018	0.05069	
case 101.	104	104	0.000	-1.84027	-2.61966	0.00149	-0.00154	-0.00018	0.05069	
Other regults		105	1.000	-1.84018	-2.55341	0.00037	-0.00187	-0.00039	-0.08193	
Other results	105	105	0.000	-1.84018	-2.55341	0.00037	-0.00187	-0.00039	-0.08193	
aan ha		106	1.000	-1.83990	-1.99802	-0.00176	-0.00219	-0.00068	-0.18912	
can be	106	106	0.000	-1.83990	-1.99802	-0.00127	-0.00224	-0.00068	-0.18912	
· 1 1		107	1.000	-1.83942	-1.11187	-0.00465	-0.00259	-0.00102	-0.24258	
viewed by	107	107	0.000	-1.82023	-1.14303	-0.00174	-0.00296	-0.00107	-0.24257	
1		108	1.000	-1.82024	-0.19676	-0.00679	-0.00329	-0.00147	0.21524	
selecting	108	108	0.000	-1.81613	-0.23175	-0.00097	-0.00395	-0.00155	0.21523	
	100	109	1.000	-1.81710	0.43513	-0.00803	-0.00391	-0.00200	-0.10095	
other pa-	109	109	0.000	-1.82610	0.39577	0.00065	-0.00435	-0.00210	-0.10093	
P P P	110	110	1.000	-1.82846	0.47630	-0.00869	-0.00338	-0.00258	0.07741	
rameters	110	110	0.000	-1.83866	0.43531	0.00287	-0.00283	-0.00262	0.07743	<u> </u>
	Disp	lacement	Primary	Secondary	Total		Diagram	Print Min	Max	Recalc

Search functions are included in the result display and can be used to show minimum and maximum forces.

Loading cases having a prefix 'LC' are created internally – defined by the system. They were not defined by the user as a loading case!

Loading cases having numbers greater than or equal to 9000 are temporary loading cases that are created, by the program, when a creep loading case is divided into more than one time step.

The Results from pre-stressing and creep loading cases can be split into two forces:

- I State: V*e (for the pre-stressing loading case and the re-arranged forces for the • creep loading case).
- II State: The constraint forces for the pre-stressing and the creep loading case.
- I+II State: Sum of I State and II State.

The sum of these forces will be displayed on selection of <Total Forces>.

The type of unit for the results can be easily changed.

- > Select î RESULTS ⇒ENVELOPE to open the result viewer of envelops.
- > Choose 'live.sup' from the displayed list following the selection of the File 'window arrow'.

The presentation of the results for superposition а file is similar to those for loading case results. The must, user however, choose 'Max/Min' the line from the superposition file that defines the desired results to be viewed.

ile	live.sup	- Elemen	it 💌 MaxMz	▼ Tnd 0 ▼	LCase Envelope	PISys PICrS	h Plinfi F	Report Lists	End
lem	101	Local	💌 🗖 Ndiv F	Pnt (kN).(kNm)	▼ Results:	Normal 👻			
lem	Nod	×/I	N (kN)"Fact	Qy (kN)"Fact	Qz (kN)"Fact	Mx (kNm)"Fact	My (kNm)"Fact	Mz (kNm)*Fact	
01	101	0.000							2
	102	1.000	-26.613	·1596.787	-6.273	5.287	-25.006	6418.043	
02	102	0.000	-26.613	·1596.787	-6.273	5.287	-25.006	6418.043	
	103	1.000	-21.272	-1276.321	-6.327	5.478	-50.529	10962.015	
03	103	0.000	-21.272	-1276.321	-6.327	5.478	-50.529	10962.015	
	104	1.000	-15.969	-958.146	-6.375	5.863	-76.416	13929.400	
34	104	0.000	-15.969	-958.146	-6.375	5.863	-76.416	13929.400	
	105	1.000	-10.724	-643.408	-6.416	6.540	-102.555	15366.033	
05	105	0.000	-10.724	-643.408	-6.416	6.540	-102.555	15366.033	
	106	1.000	11.110	666.593	-6.445	7.636	-128.793	15336.091	
06	106	0.000	11.108	666.593	-6.445	3.546	-128.861	15336.092	
	107	1.000	16.181	971.001	-6.461	5.336	-154.994	13922.509	
07	107	0.000	-0.362	971.136	-6.461	-19.576	-155.054	13922.496	
	108	1.000	-0.470	1268.818	-6.458	-16.041	-180.806	11222.799	
80	108	0.000	-24.911	1268.573	-6.457	-55.427	-179.760	11222.690	
	109	1.000	-27.893	1420.650	-6.332	-44.850	-201.793	7375.547	
09	109	0.000	-58.612	1419.698	-6.332	-84.571	-199.318	7375.265	
	110	1.000	-2.122	50.052	-5.897	-47.539	-210.505	4282.812	
10	110	0.000	-3.276	49.953	-5.896	-79.585	-207.648	4282.476	
Disn	acement	Primaru	Secondaru	Total		Diagram	Print Min	May	Recalc

Choose the MaxN 'window arrow' and select 'MaxMz' from the displayed list of available choices. The results from the loading that caused the maximum Mz in this envelope will be displayed.

MinVv	MavVv	
MinVu	MaxVu	
MinVz	MaxVz	
MinBx	MaxBx	
MinBy	MaxBy	
MinBz	MaxBz	
MinN	MaxN	
MinQy	MaxQy	
MinQz	MaxQz	
MinMx	MaxMx	
MinMy	MaxMy	
MinMz	MaxMz	

Getting Started

13-3

13.1 Diagram plot

This function is used for the graphic representation of section forces, displacements, stresses and reinforcement requirements.

> Select ûLOADS AND CONSTR. SCHEDULE, ⇒STAGE ⇒ACTION and open the List/Plot actions.

C Calculatio	in (Static)	C Check actions	 List/plot actions 	
C Calculatio	n (Dynamic)	C LC/Envelope actions	C System commands	
Action	Description			
DoList	Create the resul	t list for LC,superpos.file,		
DoRep	Report creation			
PISys	Run ASCII plot	ïle		
PICross	Plot Cross-secti	on		
PICrSh	Plot Creep+Shri	nkage and E(t) curve		
Plinfi	Plot Influence li	ne la		
PITens	Plot Tendon Ac	tions and Tendon force (several LC)		
PIEITnd	Plot element sta	rt and end CS with all tendons		
PWind	Plot wind prope	ties		
PIUIt	Plot Ultimate loa	d check diagram		
PIShear	Plot Cross-section	on Shear stresses		
DgmVx	Diagram plot of	Vx deformation for LC/SUP		
DgmVy	Diagram plot of	Vy deformation for LC/SUP		
DgmVz	Diagram plot of	Vz deformation for LC/SUP		
DgmRx	Diagram plot of	Rx deformation for LC/SUP		
DgmRy	Diagram plot of	Ry deformation for LC/SUP		
DgmRz	Diagram plot of	Rz deformation for LC/SUP		
DgmNx	Diagram plot of	N force for LC/SUP		
DgmQy	Diagram plot of	Qy force for LC/SUP		
DgmQz	Diagram plot of	Qz force for LC/SUP		
DgmMx	Diagram plot of	Mx moment for LC/SUP		
DgmMy	Diagram plot of	My moment for LC/SUP		
DgmMz	Diagram plot of	Mz moment for LC/SUP		
DgmStr	Diagram plot of	stresses for LC/SUP and certain stress po	int	
DgmRei	Diagram plot of	reinforcement for certain reinforcement po	int	-

13.2 PISys

The required composition of any plot is defined using the plot file editor. The List and Plot actions required for preparing the plot can be selected from a list

> Select îrRESULTS ⇒PLSYS to open the plot file definition window.

'plstruct.r programgenerated This file to draw th screen plo A new f be easily ated by this file a ing it ir other file

ct.rm' is a	P ^M File editor								×
m-	🗾	🗇 i	>	<	LCase Envelope	PiSys PICrSh	Plinfi	Report Script	End
ted file.	plstruct.rm	▼ (m).(m)	.(KN),(KNm),(KN/m2),(a	sC),(Deg) 💌					
1	Key	Value-1	Value-2	Value-3	Value-4	Value-5	Value-6	Value-7	
lie is used	PETRAN	0.000000	90.000	45.000	1.000000	1.000000	1.000000		
.1 .	PLSIZE	25.000	0.000000	0.000000	0.000000	0.000000	0.000000		
v the main	PLINAC								
1 /	PLPEN	1	1	0.050000					
plot.	PLELEM	0	0	0	575				
r	PLPEN DLTVCZ	b 0.400000	U	0.050000					
v file can	PLELEM	0.400000	0	0	NUM				
••	PLPEN	2	1	0.050000	no.				
alv gener-	PLTXSZ	0.500000							
	PLNODE	0	0	0	SYS				
w editing	PLPEN	1	0	0.050000					
y cannig	PLNODE	0	0	0	NUM				
e and sav-	PLSTAG								
e and sav	PLPEN	1	0	0.050000					
into an-	PLELEM	0	U	0	575				
into an-	PLPEN DLTVCZ	6 0.400000	U	0.050000					
ila nama	PLELEM	0.400000	0	0	NUM				
ne name.	I cecent		Ū	0	HOM				
	Macro		Save	She	w Plot to file	Save as	1		Becalc
plat file			5446	5110	- Thorto hie	Jave as			Troodic
piot me									

definition input window will be displayed on selection of 'Edit'



The

> Select the last entry.

> Click on the append button.

The List and Plot action input window will be displayed.

- Select the 'Scale' radio button to define the plot scales.
- > Select 'PLSCAL' from the displayed list to open the input window for scaling parameters
- > Confirm with $< o\kappa >$.
- > Input 100 for 'Scalf' for the axial force and shear force scale.
- > Input 2000 for 'Scalm' for the moments scale.
- > Input 500 for 'Scald' for the deformations scale.
- Input 500 for 'Scals for the normal stress scale. ⊳
- > Confirm with <ox>.

Set the global direction:



- > Select the last entry.
- > Click on the append button. • Select the 'Value defaults' radio button



Getting Started

- Select 'PLGLOB' from the displayed list
- > Confirm with <ok>.

Insert a loading case plot:



Select the last entry.
 Click on the annead but

> Click on the append button.

- Select the 'Load case and Envelope' radio button to open the input screen for result plots.
- Select 'PLLC' from the displayed list to open the input window for a loading case number.
- > Confirm with <ok>.
- > Select loading case 101.
- > Confirm with <ok>.

Insert an internal force plot:

PLLC Values	×
LC No. 101	•
Ok	Cancel



Select the last entry.
 Click on the annual butt

- > Click on the append button.
- Select the 'Structure plot' radio button for plotting items of the structure.
- Select 'PLELEM' from the displayed list to open the input window for plotting elements.
- ► Confirm with <ok>.
- > An input window for the element to be plotted will be opened.
- > Leave the element selection as '0' (This is a shortcut for requesting all the elements in the structure).
- Select the Inp1 'window arrow' and choose Mz from the displayed list. Or enter Mz directly in the'Inp1' window.

The bending moments about the zaxis will be plotted. > Confirm with

<0K>.

The new plot input data is shown at the end of the list.

Save the changed file with a new name:



13-5

Getting Started

13-6

> Press the **<save as>** button.

Name the new file 'pl-L0101.rm'.

- ► Confirm with <ok>.
- > Select <Plot to file> to send the plot to a plot file
- > Accept the message that the plot file has been written with <ox>.
- > Select <**show**> to plot it on the screen.
- > Zoom all using the short-cut zoom facility- the free hand symbol 'V' to view the whole plot.



> Create a plot input file for loading cases 201, 501 and 601! Name the files as follows:

PLLC 201	Plot-Input-Filename	pl-L0201.rm
PLLC 501	Plot-Input-Filename	pl-L0501.rm
PLLC 601	Plot-Input-Filename	pl-L0601.rm

[©] TDV – Technische Datenverarbeitung Ges.m.b.H.

R ^M File				x
File	pl-L0101.rm	•		
0	. 1		Cancel	1

Add the plots to the construction schedule actions:

- > Select ûLOADS and CONSTR.SCHED.⇒ACTION
- Select construction stage 1 in the upper table.
- > Append a new action entry.
- Select the 'List and Plot actions' radio button
- Select 'PLSYS' from the displayed list to plot from an (ASCII-based) plot input file.
- > Confirm with <ok>.
- Choose 'pl-L0101.rm' from the displayed list on selection of the Input file (rm) 'window arrow' or enter it directly in the window.
- > Confirm with <ok>.

R ^M Action	×
Command	PISys
Input file(*.rm)	pl-L0101.rm
Plot number (-)	
Plot file(*.pl)	ж
Delta-T	0
Description	
Ok	Cancel

- > Add plots for the files 'pl-L0201.rm' 'pl-L0501.rm' and 'pl-L0601.rm' in the same way.
- > Add the following actions into the different construction stages:

Construction stage 1:



CONSTR.SCHED.
↓
STAGE
•
ACTION
•
Bottom table

LOAD AND

Action	List and Plot	List and Plot	List and Plot	List and Plot
ACION	actions	actions	actions	actions
Тур	Plsys	Plsys	Plsys	Plsys
Inp1	PI-L0101.rm	PI-L0201.rm	PI-L0501.rm	PI-L0601.rm
Inp2	-	-	-	-
Out1	*	*	*	*
Out2	-	-	-	-
Delta-T	-	-	-	-

Construction stage 2:

Input the calculation	LOAD AND CONSTR.SCHED.	Action	 List and Plot actions 	List and Plot actions
actions for the		Тур	Plinfl	Plsys
Slage 2	STAGE	Inp1	lane0001.inf,1	PI-live.rm
		Inp2	117	-
	ACTION	Out1	*	*
	•	Out2	-	-
	Bottom table	Delta-T	-	-

Construction stage 3:

Input the calculation actions for the stage 1

LOAD AND
CONSTR.SCHED.
STAGE
•
ACTION
•

AGE	Inp1	PI-te
,	Inp2	
ION	Out1	
,	Out2	
	L L	

Bottom table

Action	List and Plot	List and Plot	List and Plot
Action	actions	actions	actions
Тур	Plsys	Plsys	Plsys
Inp1	PI-temp1.rm	PI-settle1.rm	PI-total1.rm
Inp2	-	-	-
Out1	*	*	*
Out2	-	-	-
Delta-T	-	-	-

Getting Started

14 Stress check

> Select [↑]PROPERTIES ⇒CS <AddPnt> to edit the cross section and to define stresscheck points.

The upper table in this window lists all the cross sections defined for the project. The lower table lists all the reinforcement, stress and temperature points.

> Select the (lower) info button to open the interactive input screen.

nMccooo101

The 'stress check points' and 'reinforcement points' are created by using the intersection of two straight lines. All these points were defined in GP2000. It is, however, possible to define all these additional points in RM2000.

Point name Point type	bottom fibre Stress-check point [m].(m).(KN).(KNm).(K)	Group FIB-BI		TxtFact 1.000	Elem [▼Nod IZ EHN	umb 🔽 Nod-Ni	umb 🔽 Reinf	🔽 StrP 🕅 Tend
Definition by Node 1 Node 2 Dist/z (m) Angle (Deg) TMP (#C)	2 edges 45 • 0 • 0.000 0.0000 0.0000	Node 1 0 Node 2 0 Dist/y (m) 0.0 Angle (Deg) 0.0	• • 00 00 Carneel	- #			op fibre	risht-toprisht Toppen File Kes-Bot Kes-Bot File Kes-Bot File Kes-Bot	₩#
	_								
No mode act	ive	Reint name		Distance (m)	Augle (Dec)	Ned Ned	Distance (m)	Avala (Daa)	
No mode act	ive	Point name bottom fibre	Nod1 Nod	2 Distance (m) 0.0000	Angle (Deg)	Nod1 Nod2	Distance (m)	Angle (Deg)	TMP (#C)
No mode act	ive Group FIB-60TTOM FIB-TOP	Point name bottom fibre top fibre	Nod1 Nod	2 Distance (m) 0.0000 0.0000	Angle (Deg)	Nod1 Nod2 0 0 0 0	Distance (m) 0.0000	Angle (Deg) 0.0000 0.0000	TMP (#C) 0.0000
No mode act Type FIBPOI REIPOB	ive Group FIB-BOTTOM FIB-TOP REINF-BOTTOM	Point name bottom fibre top fibre left-bottom	Nod1 Nod 45 0 70 0	2 Distance (m) 0.0000 0.0000 0.0500	Angle (Deg) 0.0000 0.0000 0.0000	Nod1 Nod2 0 0 0 0 0 0	Distance (m) 0.0000 0.0000 0.0500	Angle (Deg) 0.0000 0.0000 0.0000	TMP (#C) 0.0000 / 0.0000 0.0000

14.1 Definition of the stress-limits:

- > Select î PROPERTIES ⇒ MATERIAL to modify the concrete.
 > Select the material C 45
- **i**

> Select the information button

- Select the arrow on the right side of <Fibre stress check>
- Insert the tension stress limit
- Insert the compressive stress limit (N.B. compression is -ve)
- > Confirm with $< o\kappa >$.
- Confirm the questions "Save the changes" with <yes>.
- Note: If the limits are exceeded, the program will give a message. (N.B. These stress limits can also be shown in the plot file.)

R ^M Material	char Fibre stress	chec	k	2	×
Mat-Nam	C_45				
Group	Concrete				
Descr.					
Unit	(KN/m2)				
Tension st	ress limits		Compressive	e stress limits	
General	3584		General	-20720	
Grp2	0		Grp2	0	
Grp3	0		Grp3	0	
Grp4	0		Grp4	0	
Grp5	0		Grp5	0	
Grp6	0		Grp6	0	
					1
CalculSt	atic			-	
Check-ste	el dim.			-	
Check-din	nensioning			•	
Principal s	stress check			•	
Ultimate lo	oad check			-	
<<	<		k	> >>	3

Getting Started

Definition of the Material of the stress points:

≻ Select ¹CPROPERTIES ⇒ADDGRP



Mark the first line in the table (FIB-BOTTOM) and click the append button

An input pad will open for the definition of the Material of the stress point.

- > Input 'C_45 as the material.
- Choose stress group 1
- Confirm with <ok>
- > Mark the second line in the table (FIB-TOP) and click the append button
- > Input 'C_45 as the material.
- Choose stress group 1
- Confirm with <ok>

Reinforcement	groups X
Group	FIB-BOTTOM
Material	C_45
StrGrp	1
Description	
Ok	Cancel

Insert the actions into the construction schedule:

- > Open the construction schedule input window with ¹LOAD and CONSTR. SCHED-ULE ⇒STAGE
- > Select <Activation>

۶	Append a new	R ^M Action
	action.	Loads Stopp End
\odot	Select the	Number Location List Time (Day) Duration (Day) Description
	'Calculation	1 support init support init support init support init 2 stg0002:m stg0002.bit 10000.000 0.000 Trafic 3 stg0003:m stg0003.bit 10000.000 0.000 Superposition
	action' group.	4 stg0004.tx 10000.000 D.000 Fibre stress check 5 stg0005.m stg0005.lst 10000.000 0.000 Ulimate load check
≻	Select the	
	'FibChk' action.	Des Construction stage 0004
≻	Set the input file	Command Inp1 Inp2 Out1 Out2 Deita-T (Day) Time (Day) Description
	name to	PibChk Total sup 1 * 0.000 10000.000 * PiSys pibfort m * 0.000 10000.000 * Pisco pibfort m * 0.000 10000.000 *
	'Total.sup'.	
۶	Confirm with	
	<0K>.	
		Activation Action Tendon Recalc

> Add a List and Plot action - use diagram input 'pl-MG1-total-FibT.pl' and 'pl-MG1-total-FibB.pl' to created a new plot in the next step!

Create two plot input file for plotting stress checks. Use the Diagram to define the necessary plot files for the fibre stress check.



Select the 'CRT' button to view, print or export the plot files generated.

14-11

Getting Started





Stress – Top and Bottom

15 Ultimate load check

Reinforcement material must be defined first.

Select ☆PROPERTIES ⇒ADDGRP to modify the Material of the reinforcement groups.



> Click the append button

An input pad will open for the definition of the Material of the reinforcement group. In our case the material assignment of both reinforcement groups is already made by using *GP2000*.

Confirm with <ok>

Define the additional material properties for the reinforcement. Select the materials used in the elements in this Ultimate load check and specify their stress/strain diagrams.

- Select ☆PROPERTIES ⇒MATERIAL to modify the reinforcement material.
- Select the material GRADE_460
- Select the information button
- Select the arrow on the right side of <Ultimate load check>

Define the stress-strain curve for the material.

RM Reinforcement	groups 🗙
Group	REINF-TOP
Material	GRADE_460
StrGrp	1
Description	
Ok	Cancel

¹ Materialch	ar Ultimate load ch	eck		×
Mat-Nam	RADE_460			
Group F	Reinforc.			-
Descr.				
Unit 🚺	kN/m2)			
EPS-PL	4	SIGMA*	4.6e+005	
EPS-*	20	SIG-0.2/E	2	
SIG-0.2	4e+005	SIG-ZUS	0	
Security facto	r	Cracking cheo	:k	
GAMA	1	XI	0	
EPS1-8	-20,-2,0,2,20,0,0,0		•	
SIG1-8	-400001,-400000,0,40	0000,400001,0,0	.0	
Static			-	
Check-steel	dim.		-	
Check-dimer	nsioning		-	
Fibre stress of	check		•	
Principal stre	ess check		-	
<<	<)k	> >>	1

- Select the EPS1-8 arrow
 - > Insert values (shown in the adjacent table).
 - > Confirm the material property inputs with <ox>.
 - > Insert the E-modulus ($2e8 \text{ kN/m}^2$)



RM2000	Ultimate load check
Getting Started	15-13

- > Confirm the material property inputs with <ok>.
- > Confirm the questions "Save the changes" with <yes>.

Define the additional material properties for the tendon material.

- ≻ Select \hat{v} PROPERTIES \Rightarrow MATERIAL to modify the prestressing steel.
- > Select the material PT1
- Select the information button
- Select the arrow on the right side of <ultimate load check>

Define the stress-strain curve for the material next.

- > Select the EPS1-8 arrow
 - > Insert the values (shown in the adjunct table).
 - > Confirm the material property inputs with <ox>.
 - > Insert the additional properties (see window below)
 - > Confirm the material property inputs with <ox>.
 - > Confirm the questions "Save the changes" with <YES>.

Define additional material properties for the concrete.

- Select ^①PROPERTIES ⇒MATERIAL to modify the \triangleright concrete.
- Select the material C 45
- Select the information button
- Select the arrow on the right side of <ultimate load check>

Define the stress-strain curve for the material.

- \mathbf{T} > Select the EPS1-8 arrow
 - > Insert the values (shown in the adjacent screen shot).
 - > Confirm the material property inputs with <ox>.
 - > Insert the additional properties (see window below)
 - > Confirm the material property inputs with <ox>.
 - > Confirm the questions "Save the changes" with <YES>. Define the location of the reinforcement in the section.
 - > Select ^①PROPERTIES ⇒CS <AddPnt>

Eps-1	-20	Sig-1	-1860000
Eps-2	-7.85	Sig-2	-1674000
Eps-3	0	Sig-3	0
Eps4	7.85	Sig-4	1674000
Eps5	20	Sig-5	1860000
Eps-6	0	Sig-6	0
Eps-7	0	Sig-7	0
Eps8	0	Sig-8	0
	_		_
OK]		CANCEL

Eps-1	-2	Sig-1	-51800
Eps-2	-1.429	Sig-2	-47570
Eps-3	-1.143	Sig-3	-42290
Eps4	-0.857	Sig-4	-34890
Eps5	-0.571	Sig-5	-25370
Eps-6	-0.286	Sig-6	-13740
Eps-7	0	Sig-7	0
Eps8	20	Sig-8	1e-5
OK			CANCEL

RM2000	Ultimate load check
Getting Started	15-14

All these points were defined in the GP2000 getting started example. It is also possible to define all these additional points in RM2000.

Select îrSTRUCTURE ⇒ELEMENTS <Checks> to modify the reinforcement in the element.

 R^MStructure : Element Properties - Reinforcement

The assignment of the reinforcement to the structural elements for the cross sections is displayed in the bottom table.

lem	Туре	Fib.C	hk. I	Jlt.Chk.	Shear cap.	RC Design	Elem	Туре	Fib.Chk.	Ult.Chk.	Shear cap.	RC Design	
01	Beam	Yes		Yes	Yes	Yes	108	Beam	Yes	Yes	Yes	Yes	
02	Beam	Yes		Yes	Yes	Yes	109	Beam	Yes	Yes	Yes	Yes	
03	Beam	Yes		Yes	Yes	Yes	110	Beam	Yes	Yes	Yes	Yes	
D4	Beam	Yes		Yes	Yes	Yes	111	Beam	Yes	Yes	Yes	Yes	
05	Beam	Yes		Yes	Yes	Yes	112	Beam	Yes	Yes	Yes	Yes	
D6	Beam	Yes		Yes	Yes	Yes	113	Beam	Yes	Yes	Yes	Yes	
J7	Beam	Yes		Yes	Yes	Yes	114	Beam	Yes	Yes	Yes	Yes	-
D o	\checkmark	Type	×14	×2/1	A1 (m2)	A2 (m2)	Addi Group	ional group list	forelement 10	I x2/I	A1 (m2)	A2 (m2)	
iroup	\checkmark	Type	×14	×2/1	A1 (m2)	A2 (m2)	Addi Group	ional group list	forelement 10	I x2/I	A1 (m2)	A2 (m2)	
iroup		Type Var	×1/1	×2/I	A1 (m2)	A2 (m2)	Group	ional group list	forelement10	і x2/I	A1 (m2)	A2 (m2)	-
roup EINF-BI		Type Var Var	×17	×2/I 1.000	A1 (m2)	A2 (m2)	Group	ional group list	: for element 10 Type x1/	1 ×2/1	A1 (m2)	A2 (m2)	-
roup EINF-BI EINF-TI HEAB		Type Var Var Fix	×1/1 0.000 0.000	×2/I 1.000 1.000	A1 (m2)	A2 (m2) 0.00000 0.00000 0.00000	Group	ional group list	for element 10	I x2/I	A1 (m2)	A2 (m2)	A
roup EINF-BI EINF-TI HEAR	OTTOM OP	Type Var Var Fix	×17 0.000 0.000 0.000	×2/I 1.000 1.000 1.000	A1 (m2) 0.00200 0.00200 0.00000	A2 (m2) 0.00000 0.00000 0.00000	Group	ional group list	forelement10	I ×2/I	A1 (m2)	A2 (m2)	
oup EINF-BI EINF-TI HEAR	OTTOM OP	Type Var Var Fix	×17 0.000 0.000 0.000	×2/I 1.000 1.000 1.000	A1 (m2) 0.00200 0.00200 0.00000	A2 (m2)	Group	ional group list	: for element 10 Type x1/	і ×2Л	A1 (m2)	A2 (m2)	
oup EINF-BI EINF-TI HEAR	OTTOM OP	Type Var Var Fix	×1/1 0.000 0.000	×2/I 1.000 1.000 1.000	A1 (m2) 0.00200 0.00000	A2 (m2)	Group	ional group list	for element 10	1 x2/1	A1 (m2)	A2 (m2)	
roup EINF-BI EINF-TI HEAR	OTTOM OP	Type Var Var Fix	×1/1 0.000 0.000	×2/I 1.000 1.000 1.000	A1 (m2) 0.00200 0.00000	A2 (m2)	Group	ional group list	: for element 10 Type x1/	x2/	A1 (m2)	A2 (m2)	



> Highlight the first line in the top table.> Select the 'edit' button.

For a certain element series it possible to choose which check should be done later.

In this example we can keep the default definition (all checks for all elements).

Element cl	necks			
El-from	101			
El-to	101			
El-step	1			
Fibre stress o	heck	€ Yes	C No	C No changes
Ultimate load	check	€ Yes	C No	C No changes
Shear capac	ity check	€ Yes	C No	C No changes
Reinf.Concre	te Design	• Yes	C No	C No changes
	"			Canad

Modify the reinforcement at the bottom of the cross section.

- Highlight the first line in the bottom table. (REINF-BOTTOM).
- Select the 'edit' button to activate the material assignment input window.
- Input the element series and the reinforcement area (A1) for the whole reinforcement group.
- Change the type into VAR to calculate the necessary rein-

Reinforce	ment 🗙]	Reinforce	ment
El-from	101	H	El-from	101
El-to	135	Ш	El-to	135
El-step	1	H	El-step	1
C Fix	© Var	Ш	C Fix	€ Var
Group	REINF-BOTTOM		Group	REINF-TOP
x1/I	0.0000		x1/I	0.0000
x2/I	1.0000		x2/I	1.0000
A1 (m2)	0.002000		A1 (m2)	0.002000
A2 (m2)	0.000000		A2 (m2)	0.000000
Ok	Cancel		Ok	Cancel

forcement for the group A2.

- > Modify the values to those shown in the adjacent screen shot.
- > Confirm with <ok>.

Modify the reinforcement in the top of the cross section.

- > Highlight the second line in the bottom table. (REINF-TOP)
- > Select the 'edit' button to activate the material assignment input window.
- > Change the type into VAR to calculate the necessary reinforcement for the group A2.
- > Modify the values to those shown in the adjacent screen.
- ► Confirm with <ok>.

Insert the ultimate load actions into the construction schedule:

- > Select ¹COADS and CONSTR. SCHEDULE ⇒STAGE <action>
- > Open the construction stage No.5.
- > Add the calculation action:
- Select the 'LC/Envelope action' radio button.
- > Select 'LcInit' from the displayed list to initialise a new load case.
- > Set the output name to '1000' [Out1].
- ► Confirm with <ok>.
- Select the 'LC/Envelope action' radio button.
- > Select 'LcAdd' from the displayed list to add a load case into 1000.
- > Set the input file name to '101' [Inp1].
- > Set the output name to '1000' [Out1].
- > Confirm with <ok>.
- Select the 'LC/Envelope action' radio button.
- > Select 'LcAdd' from the displayed list to add a load case into 1000.
- > Set the input file name to '201' [Inp1].
- > Set the output name to '1000' [Out1].
- > Confirm with <ok>.
- Select the 'LC/Envelope action' radio button.
- > Select 'LcAdd' from the displayed list to add a load case into 1000.
- > Set the input file name to '501' [Inp1].
- > Set the output name to '1000' [Out1].
- > Confirm with <ox>.



- Select the 'LC/Envelope action' radio button.
- > Select 'LcAdd' from the displayed list to add a load case into 1000.
- > Set the input file name to '601' [Inp1].
- > Set the output name to '1000' [Out1].
- > Confirm with <ok>.
- Select the 'LC/Envelope actions' group.
- > Append another action to initialise the ultimate load check:
- > Select the 'SupInit' function.
- > Set the output file name to 'UltMz.sup'[Out1].
- ► Confirm with <ok>.
- Select the 'Ceck action' group.
- > Select the 'ReinInit' action.
- > Confirm with <ok>.

The program set the reinforcement group A2 into zero.

- Select the 'Ceck action' group.
- > Select the 'UltRein' action.
- > Set the input file name to 'Total.sup' [Inp1].
- > Confirm with <ok>.

The program add necessary reinforcement into group A2. For further checks the program will use the total reinforcement A=A1+A2.

- > Append a new action.
- Select the 'Ceck action' group.
- > Select the 'UltChk' action.
- > Set the input file name to 'Total.sup' [Inp1].
- > Set the Character combination to 'UltMz' [Inp2].
- > Set the output file name to 'UltMz.sup' [Out1].
- > Confirm with <ok>.



Add a new Diagram with the name of "pl-MG1-total-Mz-Sec.pl" and "pl-MG1-UltMz-Mz-Sec.pl" into the action list. Show in this plot the results from UltMz.sup and Total.sup (Max/Min Mz) – but only the secondary forces.

15-17

Getting Started

Action													
۵,		2	1		r ³	(m).(m).(kN).(kNm),(kN/m2),(C),	(Deg) 💌		Loads	AddCon	Stage	End
Status	Date	,		Number	Location	List	Time (Day)	Duration (Da	y) Description				
Ok	13	12	2001	1	stg0001.rm	stg0001.lst	0	10000	Load calculation				
Ok	30	4	2029	2	stg0002.rm	stg0002.lst	10000	0	Traffic				
Ok	30	4	2029	3	stg0003.rm	stg0003.lst	10000	0	Superposition				
Ok	30	4	2029	4	stg0004.rm	stg0004.lst	10000	0	Fibre stress check				
Ok	30	4	2029	5	stg0005.rm	stg0005.lst	10000	0	Ultimate load check				
													-
D ,	1		1	₽ _₽					Constructio	in stage 0005			
Status	Date			Command	Inp1	Inp2	Out1		Out2	Delta-T (Day)	Time (Day)	Description	
Ok	30	4	2029	LoInit			lc1000			0	10000		
Ok	30	4	2029	LcAdd	lc0101		lc1000			0	10000		
Ok	30	4	2029	LcAdd	lc0201		lc1000			0	10000		
Ok	30	4	2029	LcAdd	lc0501		lc1000			0	10000		
Ok	30	4	2029	LcAdd	lc0601		lc1000			0	10000		
Ok	30	4	2029	SupInit			UltMz.su	1p		0	10000		
	30	4	2029	ReinIni						0	10000		
Ok	00								8	10	10000		
Ok Ök	30	4	2029	UltRein	total.sup					°	10000		
Ok Ok	30	4	2029	UltRein	total.sup						10000		

- > Select **î**RECALC to open the input window for global project calculation property definitions.
- ☑ Check 'Save tendon results (LC)'.
- \square Check 'Save tendon results (Env)'.
- > Insert the load case '1000' into SumLC.
- Recalculate the structure
- **Crt** > Select the 'CRT' button to view, print or export the plot files generated.

15-18

Getting Started



total.sup (Mz secondary)



UltMz.sup (Mz secondary)
Ultimate load check

15-19

Getting Started



Reinforcement Bottom

16 Shear capacity check

What is already defined?

Shear-area definition by using GP2000 Material assignment for the shear-reinforcement (☆PROPERTIES ⇒ADDGRP)

> Select ^①PROPERTIES ⇒ADDGRP to modify the Material of the group SHEAR.



Click the append button

An input pad will open for the definition of the Material of the reinforcement group. In our case the material assignment of both reinforcement groups is already made by using *GP2000*.

Reinforcement o	groups 🗙
Group	SHEAR
Material	GRADE_460
StrGrp	1 🗸
Description	
Ok	Cancel

Confirm with <ok>

Inserting the actions into the construction schedule:

- > Open the construction schedule input window with ℃CONSTR.SCHED.⇒STAGE
- > Select <Activation>



Select the (upper) append button to open the input window for the construction stage definition.

- > Input '6' for the number.
- > Input 'Shear capacity check' for the description.
- > All elements are already activated!
- > Confirm with <ok>.

Insert the shear capacity check actions into the construction schedule:

> Select ^①LOADS and CONSTR.SCHED.⇒STAGE <action>

- > Open the construction stage No.6.
- Select the 'Check actions' group.
- > Select the 'ShChk' function.
- > Set the input file name to 'Total.sup,501' [Inp1].
- > Set the group name to 'SHEAR' [Inp2].
- > Confirm with <ox>.

The superposition file Total.sup consists all necessary results. This file was created in the stage 3. Additional it's necessary to insert the prestressing loading case (501) and the group name of the shear definition. The results are stored in a list-file named "sheat total.lst".

Action									×
	\checkmark		<u>i</u> (ı),(m),(KN),(KNm),(KN/m2)	.(øC),(Deg)		Loads A	AddCon Stage	End
Status	Number	Location	List	Time (Day)	Duration (Day)	Description			
+	1	stg0001.rm	stg0001.lst	0	10000	Load calculation			
+	2	stg0002.rm	stg0002.lst	10000	0	Traffic			
+	3	stg0003.rm	stg0003.lst	10000	0	Superposition			
+	4	stg0004.rm	stg0004.lst	10000	0	Fibre stress check			
+	5	stg0005.rm	stg0005.lst	10000	0	Ultimate load check			
+	6	stg0006.rm	stg0006.lst	10000	0	Shear capacity chec	k		
									•
Status	Command	Inp1	Inp2	Out1	Out2	Construction s	stage 0006 Time (Day)	Description	
+	ShChk	total.sup.501	SHEAR	×	×	0	10000		
P									······
									-
Ac	tivation	Action	Tendon						Recalc

17 Data backup

Select \hat{U} FILE \Rightarrow EXPORT to activate the import dialogue box shown below. All data files are stored in the file rmexport.txi (Index-File) and rmexport.txd (Data-File). It is necessary to use both these files if these back-up files are copied to another directory.

\triangleright Confirm with <ok>.

Data base	rmexport		Files global.rm	C Tcl-Script	
Complete Pro Partial projec	ject (new project by import) t (add to the actual project)	Proper	ties 🗖 Stru	cture 🔲 Loads and Constr	.Schedule
└ Material └ AddGrp	material.rm	Cross s. C Variable n	ross.rm	Aero class aero.rm	•
Structure	struct.rm	•			
Tendon	tendon.rm	•	TDnr 1	ten0001.rm	·
Load set	loadset.rm	-	LSnr 1	Is0001.rm	
Load case	loadcase.rm		LCnr 1	lc0001.rm	
Lane	lane.rm	•	🗖 LNnr 🛛 🚺	lane0001.rm	
LTrain	live.rm	-	🗖 LVnr 📘	live0001.rm	
Seismic	seismic.rm	-	🗖 SMnr 🛛 1	seis0001.rm	
AddCon	addcon.rm	•	🗖 AChr 🛛 1	acon0001.rm	·
☐ Wind	wind.rm	•	WNnr 1	wind0001.rm	•
Stage	stage.rm	•	🗖 STnr 🛛 1	stg0001.rm	ŀ
Eut files	ascii.rm	•			

Getting Started

18-4

18 Plot Macros

18.1 Plot-Macros

It's possible to use plot macros to generate plot files by using the program.

- Select ☆RESULTS ⇒PLSYS to open the plot file definition window.
- > Select <Macro> to start the macros.
- > In the appeared window there are 7 different Plot-Macros.
- Mark the third line (Load case plot, forces)
- > Confirm with <ok>.



18.1.1 Forces

- Keep all default settings in the input window excepted Qy in the Results and change the Plot input file name into pl-L0101M.rm
- All these settings will create a Plot input file with the name of pl-L0101M.rm and will show Mz (Bending moment) and Qy (Shear force) from the load case 101.
- > Confirm with <ok>.

N Load case plot,	shape and structure	e				×
Presentation						
View	General isometric	set 0.0 90.0 45.0)			•
Scale	Given Scale 1 : 1	00.0				•
Structure						
Elements	All elements					•
Tendons	No tendons					•
Lanes	No lanes					•
Results						
Load cases	Set of loadcases	(101,101,1;)				•
	EN PQ	y 🗖 Qz	∏ Mx	🗖 Му	🔽 Mz	
	C Primary	C Secon	dary	Total		
	Local	🔿 Global				
Plot input file	pl_lcf.rm					•
Ok					Cance	

- In the appeared window are all generated input lines.
- Select <show> to plot it on the screen.
- Zoom all using the short-cut zoom facility- the free hand symbol 'V' to view the whole plot. At last we change the Plot-scale.

pl-L0101M.rm -	[m].(m).(m).	(KN),(KNm),(KN/m2),(ø(
 Key	Value-1	Value-2	Value-3	Value-4	Value-5	Value-6	Value-7	
PLTRAN	0.000000	90.000	45.000	1.000000	1.000000	0.500000		-
PLSIZE	100.000	0.000000	0.000000	0.000000	0.000000	0.000000		
PLSTAG								
PLPEN	1	0	0.025000					
PLELEM	0	0	0	SYS				
PLPEN	6	0	0.025000					
PLTXSZ	0.300000							
PLELEM	0	0	0	NUM				
PLTXSZ	0.200000							
PLNODE	0	0	0	SYS				
PLPEN	1	0	0.025000					
PLNODE	0	0	0	NUM				
PLSCAL	2000.000	15000.000	500.000					
PLLC	101							
PLVALUE	MAXMIN							
PLSTAT	ALLVAL							
PLSTAG								
PLPEN	3	0	0.025000					
PLELEM	0	0	0	Qy				-

- > Select \hat{T} RESULTS \Rightarrow PLSYS to open the plot file definition window.
- Mark the line 'PLSCAL' from the displayed list in the table and click the append button
- > Change 2000 into 500 for 'Scalf' for the axial force and shear force scale.
- > Change 15000 into 4000 for 'Scalm' for the moments scale.
- > Confirm with <ox>.
- > Select <**show**> to plot it on the screen.



18.1.2 Fiber stress Plots

> Add a List and Plot action - use plot input file 'pl-fibr1.rm' and 'pl-fibr2.rm' to created a new plot in the next step!

Create two plot input file for plotting stress checks. Use the Plot-Macro's to define the necessary plot files for the fibre stress check.

Additionally complete the plot-file with text information:

PLTXSZ	2.000					
PLPEN	1 0 0.020000					
PLFONT	RB GROTES					
PLFTXT	RB 16.000	-10.000	0.000000	"Getting	started	Example"
PLTXSZ	1.500					
PLPEN	6 0 0.020000					
PLFONT	LB GROTES					
PLFTXT	LB 10.000	-10.000	0.000000	"FIB-CHK	- END"	
PLTXSZ	1.500					
PLPEN	2 0 0.020000					
PLFONT	LB GROTES					
PLFTXT	LB 10.000	-13.000	0.000000	"FIB-CHK	TOTAL -	TOP"

These instructions have the following meaning:

PLTXSZ	height of the text in cm.
PLPEN	Colour, style and line thickness
PLFONT	Font-Type
PLFTXT	the coordinated start point of the text, orientation and text himself

Aditional text-info can be added in all the other plot-files!

Note: Use the editor and copy these lines into all other plot-files – speeds up data input preparation or use the plot macros!

Start a 'recalc' of the system

≻

Crt > Select the 'CRT' button to view, print or export the plot files generated.

Plot Macros

18-7

Getting Started



Plot generated by PlSys

18.1.3 Ultimate load plot

Add a new plot with the name of "pl-Ult1.rm" into the action list. Show in this plot the results from UltMz.sup and Total.sup (Max/Min Mz) – but only the secondary forces. (Insert the plot function: PLSTAT – SECOND from Value defaults)



≻

Select the append button to add the next action.

- Select the 'List and Plot actions' radio button
- Select 'PLSYS' from the displayed list to plot from an (ASCII-based) plot input file.
- > Confirm with <ox>.
- > Insert the name 'pl-Ult1'.
- > Confirm with <**ok**>.

Getting Started

18-2



Plot generated by PlSys

19 Result plots

19.1 System (PlSys)





Getting Started

Result plots





19.2 Forces and Moments (Diagram)



Getting Started





Result plots

Getting Started

19-4













19.4 Tendon pre-stressing and creep/shrinkage



19.5 Influence line

