Appendix 1

Ascii Command Input

Table of Contents

1 Introduction.

The entire NovaFrame and NovaDesign model can be generated from an ascii text input. The ascii text input consists of a series of command lines. A command line (also called statement) is a single text line starting with a command key word, followed by command values, For example;

```
REFLINE, 1, 1, “Veilinje”
```

All valid command key words are presented in section 5 of this appendix. The commands are listed in the order we recommend for your input file. For example defining a reference line (REFLINE) should come before defining horisontal geometry (RFLHORI) for the same reference line.

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Project name  :  Eksempel
% Database name :  D:\Eksempel.GDD
% Date          :  28/4-2000 - 12:12
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%


%%% REFERENCE LINES  %%%
REFLINE, 0, 0, "Sections"

%%% CONCRETE MATERIAL INPUT %%%
MCONCR, 1, 45.00, 2200, 27836, 28.00, 27174, 2.00, 2.01, 3.48, 0.000000, 0.20, 27174, "C45"

%%% REINFORCEMENT STEEL MATERIAL INPUT %%%
MREINF, 1, 500.0, 200000, 10.00, "B500C"

%%% CONCRETE DESIGN PARAMETER INPUT %%%
DPCONCR, 1, 1.40, 1.20, 1.00, 3.484, 3.484, 3.484, "Default values"

%%% REINFORCEMENT DESIGN PARAMETER INPUT  %%%
DPREINF, 1, 0.4, 1.5, 0.30, 40.0, 1, 1.25, 1.10, 1.00, 5.00, 5.00, 5.00, "Default values"

%%% SECTION DESIGN PARAMETER INPUT %%%
DPSECT, 1, 1, 0.000, 0.000, 10.00, 45.00, 0.666, 2.50, 0.4, 0.30, 1, "Default values"

%%% DESIGN SETUP INPUT %%%
DCSETUP, 2, 0, 1, "All elements (frame model)"

%%% DESIGN SETUP OPTION INPUT %%%
DCOPT, 2, CONCR, 1, 0, 0, 1, 1, 0, 0, 0, 1, 0, 0

%%% DESIGN SETUP SECTION INPUT %%%
DCSECT, 2, -1, -1, -1.000, -1.000

%%% DESIGN SETUP COMBINATION INPUT %%%
DCCOMB, 2, ULS, -1, -1
DCCOMB, 2, PLS, -1, -1
DCCOMB, 2, SLS, -1, -1

%%% END OF INPUT
```

Fig. A1.1 Example acii input

2 Import and export of ascii commands

2.1 Import and export from file

A complete import and export of ascii commands can be done from the **File** menu.

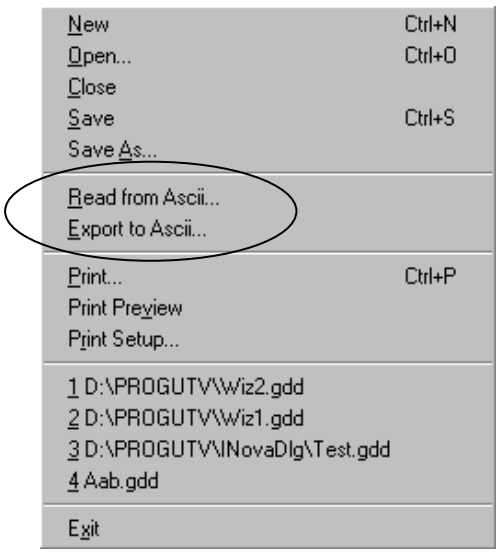

Fig. A1.1 'File' menu

In order to import command input from an existing ascii input file, select ***Read from Ascii…*** from the ***file*** menu. This menu option will display a dialog for specifying file name for ascii input file, see figure below. The 'File name' droplist holds your last input file names (up to 12 files).

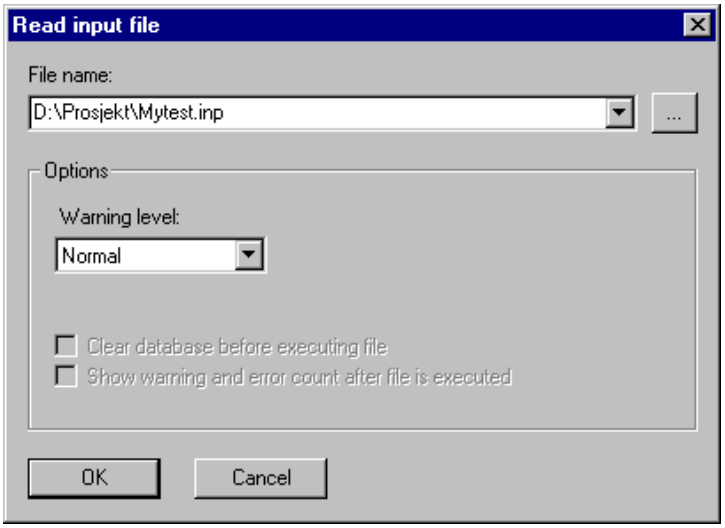

Fig. A1.2 'Read input file' dialog

In order to write all model data to an ascii file, select ***Export to Ascii…*** from the ***file*** menu. This menu option will display a standard 'Save As' dialog which allows you to select file name for the ascii file.

2.2 Automatic ascii backup file (*.bak)

A complete backup of all ascii input is made automatically whenever you save your database. The filename for this ascii backup is always <*database name*>*.bak*. For example, if your database name is *Mytest.gdd*, the file name for the ascii backup will be *Mytest.bak*.

2.3 Running batch

In some situations it is preferred to work using only input files which both trigger analyses, design and creating output to file directly. Working like this is called running batch.

If-statements and Do-loops are also only available when reading from files and not through the dialog input.

Commands in chapter 5.17 and 5.18 are commands which are only available in batch.

3 Error and warning messages

For every ascii input line the syntax and values of the statement are checked before the command is processed. If any errors are found in the input line, a warning- or error message will be displayed. If any errors are found, the statement will be ignored. Statements generating only warnings will however be processed.

Sample input sequence:

```
      …
      NODEIN, 1, , , 0, 0, 0     ! This command will generate an error message
                                 ! because NODEIN is not a valid command
NODEINS, 1, , , 0, 0, 0    ! Creates node 1 at location (0, 0, 0)
NODEINS, 1, , , 2, 2, 2    ! This command will generate a warning message
                                 ! because node 1 is already defined, however
                                 ! the statement is accepted and node 1 is now
                                 ! located at (2, 2, 2)
…
```

The warning and error messages are displayed in message boxes, see figure below. You must press the ‘OK’ button in these message boxes to continue the input sequence. The messages are also sent to an error file (*.err). This file can be opened after the input sequense is finished to check the generated errors and warnings

The program will prompt you a maximum of 5 times with message boxes for each input sequence. Any following warning or error messages will be sent to the error file only, and not displayed.

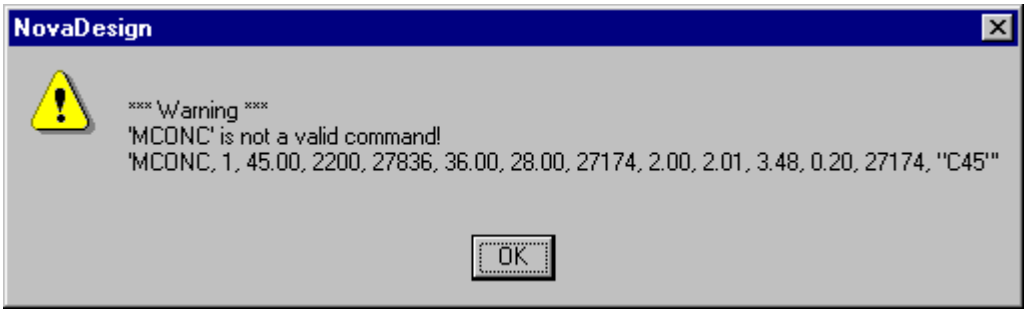

Fig. A1.3 Warning message

4 Rules for ascii input syntax

4.1 Input values

An input statement contains a command key word followed by a number of input values, and in some cases a “name” for indentification. All input data may be entered as decimal (float) values. However if an integer value is expected at an input data field, the decimals are removed (1.92 becomes 1)

Usually the input values are precalculated, but in some cases it may be convenient to enter whole mathmatic expressions in the text input. The program offers a number of valid operators for such expressions. Valid operators are:

+
-
*
/
sin
cos
tan,
^ (raised to the power of)
sqr (square)
sqrt (square root).

Parentheses ( ) can be used in combination with all operators.

Sample input sequence:

```
      …, 2+3,…                ! (expression equals 5)
…, 2*(1+4), …           ! (expression equals 10)
…, sin(30), …           ! (expression equals 0.5)
…, sqr(4.4), …          ! (expression equals 19.36)
…, sqrt(10.24), …       ! (expression equals 3.2)
…, (0.4/0.7)^0.3, …     ! (expression equals 0.8455)
```

The program is not case sensitive (both ‘sin’ or ‘SIN’ are valid operators).

Note!
The mathmatic expressions may also include user-defined alias values (see later section).

4.2 Value separators

Both comma, blank, tabulator or a mix of these are allowed as separators between input values in all commands. However, using comma as separator allows you to take advantage of the default values for each command. If no value is entered between to commas, the default value will be used.

4.3 Comments in text input

Any text following '!' or '%' in an ascii input line is taken as a comment. This text will be ignored when the program executes the ascii input. Comments can be used to explain the input in order to make the input sequence more readable. Any number of comment text lines can be added to the ascii input. It is also possible to add a comment to the end of a statement.

Sample input sequence :

```
      …
! This line is is a comment
NODEINS, 1, 1, 1        ! This text is a command
!NODEINS, 2, 2, 1       ! This NODEINS statement will be ignored
      …
```

4.4 Program defined alphanumeric values (aliases)

Several predefined alphanumeric values, or aliases, are available. An alias is an alphanumeric text that represents a numeric value. For example, the text 'ALL' equals the numeric value -1. Using these aliases can make the ascii input more readable. Available predefined aliases are presented together with the different ascii commands in this appendix. In addition you may define your own aliases, as explained below. The predefined aliases are reserved and can not be overridden.

4.5 User defined alphanumeric values (aliases)

You can associate an alphanumeric value with a numeric value by using the '=' operator. The alphanumeric value must be a single word (no space allowed). The alphanumeric value can be used as an alias for the numeric value later in the input sequence.

Sample input sequence:

```
…
FCK = 45.0
…
MCONCR, 1, FCK          ! FCK will be replaced with the value 45.0
…
```

The alphanumeric values are not case sensitive (upper/lower case). Predefined alias names can not be used (such as 'ALL' or NA). There is no limitation in the number of characters allowed for each alias. If an alphanumeric value is redefined within the ascii input, the value associated with this alias will be replaced without any warning message.

4.6 Multi line commands

A command should normally be written in a single text line, however a command line with many characters may be split into several lines using the and-operator '&'. This could be useful if a command line is too long to fit within normal page width.

Sample input sequence:

```
…
REAS,  1. 12, 201, 1, 1, ,176,,177, 178, 179, 180, 190,191,  &
                          277, 278, 279, 280, 290, 291
…
```

4.7 IF statements

The text input may include any number of IF statements. The IF statement will include or skip input lines based on a test expression.

Syntax: IF, value, test-operator, limit-value
ELSEIF, value, test-operator, limit-value
ELSE
ENDIF

Valid test operators are

EQ	(equal to)
LT	(less than)
GT	(greater than)
LE	(less or equal)
GE	(greater or equal)

The test acceptance limit is 1E-10, meaning; testing if 1.00000000009 equals to 1, would return 'true'. An IF statement must always be closed with an ENDIF command as shown in the sample below. The ELSE and ELSEIF commands are optional.

Sample input sequence:

```
…
CODE = 1      ! switch
…
IF,CODE,LE,0       ! test if CODE <= 0
   PTOT = 100.0
ELSEIF,CODE,GT,1    ! test if CODE > 1
   PTOT = 200.0
ELSE                ! default, if no tests was true
   PTOT = 100+CODE*100
ENDIF               ! remember to close expression
…
```

The IF statements may be encapsulated by other IF-statements (maximum 50 layers of encapsulation). IF-statements may also be used in combination with DO-loops.

Note!
Please note that when IF statements have been used in the ascii-input file these statements will not be imported into the ascii text input windows for each input command. This implies that the IF statements information will be lost if the ‘Apply’, ‘Apply All’ or ‘OK’ buttons are pressed in the corresponding ascii input dialogs. Consequently the solution (‘Solve’ or ‘Solve Analysis’) should be run without editing or re-applying the ascii text input windows for the commands inside the IF block.

4.8 DO-loops

The text input may include any number of DO-loops. A DO-loop is a sequence of input lines between a DO- and a corresponding ENDDO-statement. The DO-loop makes it possible to run the sequence of input lines more than one time.

Syntax: DO, loop-variable, from, to, step
ENDDO

The loop variable must be given a start value, a limit value and a step value. The loop variable is incremented with the step value after each loop. The program continues to run the input sequence as long as the loop variable is less or equal to the limit value. The loop variable may be used not only to count the number of loops. It can also be used directly in the input command lines within the DO-loop, or in mathematical expressions.

A DO-loop may encapsulate another DO-loop. The maximum number of encapsulation layers is 50. Note that the loop-variable (N in the sample below) must be different in all layers.

The loop-variable does not have to be an integer value, which makes it more useful in combination with arithmetic expressions

Sample input sequence:

```
…
DO,N,0,45,1
   ANG = 2*I
   NODE, N+1, N+1,1,, 5.0*SIN(ANG), 5.0*COS(ANG)
ENDDO
…
```

The DO loop may be used in combination with other mathematical expressions, such as a simple counter variable. The DO-loop may also be used in combination with IF-statements.

Sample input sequence:

```
…
DO,N,0,45,1
   ANG = 2*I
   NODE, N+1, N+1,1,, 5.0*SIN(ANG), 5.0*COS(ANG)

   NLOOP = NLOOP + 1  ! counter variable
```

```
   IF,NLOOP,GE,12
      NODE, N+100, N+100,1,2.500, 5.0*SIN(ANG), 5.0*COS(ANG)
   ENDIF
ENDDO
…
```

Note!
Please note that when DO loops have been used in the ascii-input file these statements will not be imported into the ascii text input windows for each input command. This implies that the DO loops information will be lost if the ‘Apply’, ‘Apply All’ or ‘OK’ buttons are pressed in the corresponding ascii input dialogs. Consequently the solution (‘Solve’ or ‘Solve Analysis’) should be run without editing or re-applying the ascii text input windows for the type of commands inside the DO loops.

5 Input commands

5.1 Project data input

PROJECT, PROJID, CRULES, SRULES, ”NAME”

	Default	*Unit*	*Description*	
PROJID	0	-	Project number	
CRULES	**NS3473**	-	Concrete designs rules. Available design rules are:	
		1	= Norwegian code NS 3473 ed 5	(**NS3473**)
		2	= Sweedish code BBK94	(**BBK94**)
SRULES	**NS3472**	-	Steel design rules. Available design rules are:	
		1	= Norwegian code NS 3472	(**NS3472**)
”NAME”		-	Optional project name. All characters allowed including space	

Use the PROJECT command to assign project identification and design rules

5.2 Material data input

MCONCR, ID, FCK, RHO1, ECK, FCCK, FCN, ECN, FTN, EPCO, EPCU, VFRAME, EFRAME, ALFA, RH,"NAME"

NS 3473

	Default	*Unit*	*Description*
ID	1	-	Concrete Material number
FCK	45.	MPa	NS 3473 tab 5.
RHO1	2200.	kg/m^3	NS 3473 cl 9.2.1 and 11.1. (ρ_1)
ECK	(*)	MPa	NS 3473 cl. 9.2.1
FCCK	(*)	MPa	NS 3473 tab 5.
FCN	(*)	MPa	NS 3473 tab 5.
ECN	(*)	MPa	NS 3473 cl. 11.3
FTN	(*)	MPa	NS 3473 tab 5.
EPCO	(*)	o/oo	NS 3473 cl. 11.3
EPCU	(*)	o/oo	NS 3473 cl. 11.3
VFRAME	0.2	-	Poisson's ratio for frame analysis
EFRAME	(*)	MPa	Young's modulus for frame analysis, by default equal to ECK
ALFA	1.0e-5	1/^{o}C	Thermal expansion coefficient
RH	70	%	Relative humidity
"NAME"	(*)	-	Optional name. All characters allowed including space

Only values for ID, FCK and RHO1 are required input. Default values according to NS 3473 rev. 5 will be calculated for the other (*) concrete material parameters.

NS-EN 1992

MCONCR, ID, FCK, LB, RHO, VFRAME, EFRAME, CEM, "NAME"

	Default	*Unit*	*Description*
ID	1	-	Concrete Material number
FCK	45.	MPa	NS-EN 1992 table 3.1.
LB	0	-	0 = ordinary concrete 1 = light weight aggregate concrete (LB)
RHO	2200.	kg/m^3	NS-EN 1992 table 11.1
VFRAME	0.2	-	Poisson's ratio for frame analysis
EFRAME	E_{cm}	MPa	Young's modulus for frame analysis, -1 or blank = Use default value; E_{cm} corresponding to specified FCK
CEM	1	-	Cement class. NS-EN 1992 appendix B 0=S, 1=N, 2=R
"NAME"	(*)	-	Optional name. All characters allowed including space

Only values for ID, FCK and are required input. Default values according to NS-EN 1992-1-1 will be calculated for the other (*) concrete material parameters.

BBK94

MCONCR, ID, CLASS, RHO, VFRAME, EFRAME, "NAME"

	Default	*Unit*	*Description*
ID	1	-	Concrete Material number
CLASS	45.	MPa	Concrete strength ("Hållfasthetsklass")
RHO	2400.	kg/m^3	Density of aggregate (LWA if RHO<2400)
VFRAME	0.2	-	Poisson's ratio for frame analysis
EFRAME	(*)	MPa	Young's modulus for frame analysis, by default equal to ECK
"NAME"	(*)	-	Optional name. All characters allowed including space

(*) Default values are acc. to BBK94 for entered value for FCCK and RHO

BS5400

MCONCR, ID, FCK, RHO, VFRAME, EFRAME, "NAME"

	Default	*Unit*	*Description*
ID	1	-	Concrete material number
FCK	45.	MPa	Concrete compressive strength
RHO	2300	kg/m^3	Density of aggregate (LWA if RHO<2300)
VFRAME	0.2	-	Poisson's ratio for frame analysis
EFRAME	(*)	MPa	Young's modulus for frame analysis, by default equal to EC
"NAME"	(*)	-	Optional name. All characters allowed including space

(*) Default values are acc. to BBK94 for entered value for FCCK and RHO

MCONCR creates a new concrete grade. Only ID and compressive cube strength (FCK) are required entries in order to create a concrete grade according to NS 3473 (for example: *MCONCR, 1, 65*.).

MREINF, ID, FSY, ESY, EPSU, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Reinforcement Material number
FSY	500.	MPa	Characteristic steel strength
ESY	200000.	MPa	Young’s modulus
EPSU	10.	o/oo	Ultimate strain limit
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992

MREINF, ID, FYK, ES, EPSUD, CLASS, CURVE, FT, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Reinforcement Material number
FYK	500.	MPa	Characteristic steel strength
ES	200000.	MPa	Young’s modulus
EPSUD	30.	o/oo	Ultimate strain limit
CLASS	3.	-	Ductility class. 1=A, 2=B, 3=C, 4=User defined
CURVE	1.	-	Stress-strain curve 1=Capacity limited by ultimate strength f_t 2= Capacity limited by yield stress f_{yk}
FT	520.	MPa	Ultimate strength at EPSUD
”NAME”		-	Optional name. All characters allowed including space

MREINF creates a new reinforcement steel grade.

MTEND, ID, FSY, ESY, EPSU, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Tendon Material number
FSY	1670.	MPa	Characteristic tendon steel strength
ESY	195000.	MPa	Young’s modulus
EPSU	10.	o/oo	Ultimate strain limit
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992 (Eurocode-2)

MTEND, ID, FP01K, EP, EPSUD, CLASS, CURVE, FT, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Reinforcement Material number
FP01K	1670.	MPa	Characteristic steel strength
EP	195000.	MPa	Young’s modulus
EPSUD	10.	o/oo	Ultimate strain limit
CLASS	1.	-	Ductility class. 1=default, 4=User defined
CURVE	1.	-	Stress-strain curve 1=Capacity limited by ultimate strength f_t 2= Capacity limited by yield stress f_{p01k}
FT	1736.8.	MPa	Stress at ultimate strain limit
”NAME”		-	Optional name. All characters allowed including space

MTEND creates a new tendon steel grade.

MSTEEL, ID, FY, ES, RHO,FU, EPSU, ALFA, VFRAME, EFRAME, EY, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Steel Material number
FY	355.	MPa	Characteristic yield strength
ES	210000.	MPa	Characteristic Young’s modulus
RHO	7850.	kg/m3	Density
FU	490.	MPa	Ultimate strength
EPSU	12.	o/oo	Ultimate strain limit

ALFA	1.2e-5	1/°C	Thermal expansion coefficient
VFRAME	0.3	.	Poisson's ratio for frame analysis
EFRAME	210000.	MPa	Young's modulus for frame analysis
EY	0.	MPa	Young's modulus in yield
"NAME"		-	Optional name. All characters allowed including space

MSTEEL creates a new user defined construction steel grade.

MSTEEL, ID, TYPE, "NAME"

	Default	*Unit*	*Description*
ID	1	-	Steel Material number
TYPE	-.	-	Predefined steel grade alias: ST37, ST44, ST52, S235, S275, S355, S275N , S355N, S420N, S460N, S275M, S355M, S420M, S460M
"NAME"		-	Optional name. All characters allowed including space

MSTEEL creates a new predefined construction steel grade.

e.g:
MSTEEL, 11, S355N, "S355N"

5.3 Parameter set input

DPSECT ID, TYPE, LEX, LEY, LNMIN, LNMAX, KA, CREEP, BETA, KIT, TOL, - ,TOLX, TOLY, ”NAME”

NS 3473

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
TYPE	1	-	Material type
LEX	0.0	m	Theoretical buckling length about section x-axis (L-axis in NovaFrame)
LEY	0.0	m	Theoretical buckling length about section y-axis (N-axis in NovaFrame)
LNMIN	10	-	Lower limit for load dependent slenderness (NS 3473, 12.2.5)
LNMAX	45	-	Upper limit for load dependent slenderness (NS 3473, 12.2.4)
KA	0.66	-	Reinforcement scaling factor (NS 3473, 12.2.4)
CREEP	2.5	-	Total creep factor. (NS 3473, A.12.2)
BETA	0.4	-	Load frequency factor (NS 3473, A.15.6.2)
KIT	1.0	-	Scaling factor for torsion stiffness applied for cross in analysis, May be used in general for scaling of the torsion stiffness calculated by the program, or in particular to accommodate recommendations in NS 3473 cl. A.12.4.1. The values specified in cl. A.12.4.1 are based on E-modulus and not G- modulus. In order to obtain the stiffness reduction according to cl. A.12.4.1 the following values are suggested for KIT: 0.72 un-cracked concrete (corresponds to $0.3 \cdot EI_t$) 0.24 if concrete is cracked due to bending (corresponds to $0.1 \cdot EI_t$) 0.12 if concrete is cracked due to shear or torsion (corresponds to $0.05 \cdot EI_t$)
TOL	0	-	Include eccentricity in axial force due to construction tolerances. Applies only to ULS and PLS and if axial force is compressive (less than zero) 0 = include only for slender structural members (NS 3473, 12.2.3) 1 = include always (NS 3473, 12.1.2)
TOLX	-1	m	Override of tolerances in section x-dir. Default: -1 means calculated according to regulations. Given value lager than zero, gives the tolerance directly.
TOLY	-1	m	Override of tolerances in section y-dir. Default: -1 means calculated according to regulations. Given value lager than zero, gives the tolerance directly.
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992

DPSECT ID, TYPE, LEX, LEY, RMX, RMY, CURV, CREEP, BETA, KIT, TOL, DIMP,”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
TYPE	1	-	Material type
LEX	0.0	m	Theoretical buckling length about section x-axis (L-axis in NovaFrame)
LEY	0.0	m	Theoretical buckling length about section y-axis (N-axis in NovaFrame)
RMX	1.0	-	Member end moment factor for moments about section x-axis (NA.5.8.3.1)
RMY	1.0	-	Member end moment factor for moments about section x-axis (NA.5.8.3.1)
CURV	0	-	Code for curvature used in calculation of additional moment due to curvature for slender members 0 = Use default curvature (cl. 5.8.8.3 (1)) 1 = Use actual curvature calculated from applied moments including additional moments from construction tolerances.
CREEP	2.5	-	Total creep factor. (cl . 5.8.4)
BETA	0.4	-	-Not in use-
KIT	1.0	-	Scaling factor for torsion stiffness applied for cross in analysis.
TOL	2	-	Include eccentricity in axial force due to construction tolerances and geometric imperfections. Applies only to ULS and PLS and if axial force is compressive (less than zero). 0 = No not include tolerances 1 = Include tolerances in both x- and y-direction 2 = Include tolerances in x-direction only (My moment) 3 = Include tolerances in y-direction only (Mx moment)
DIMP	400	-	Denominator in ratio LEX/DIMP and LEY/DIMP for calculation of eccentricity due to geometric imperfections. Recommended value according to Eurocode-2 is 400 (le/400), while special rules may apply according to Statens vegvesens håndbok 185 or other specifications. See also description for parameter TOL.
”NAME”		-	Optional name. All characters allowed including space

BBK 94

DPSECT ID, TYPE, LEX, LEY, LNMIN, LNMAX, KA, CREEP, BETA, KIT, TOL, CLASS, TOLX, TOLY, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
TYPE	1	-	Material type. Valid types are: 1 = concrete (**CONCR**) 2 = steel (**STEEL**)
LEX	0.0	m	Theoretical buckling length about section x-axis (L-axis in NovaFrame)
LEY	0.0	m	Theoretical buckling length about section y-axis (N-axis in NovaFrame)
LNMIN	10	-	Lower limit for load dependent slenderness (NS 3473, 12.2.5)
LNMAX	45	-	Upper limit for load dependent slenderness (NS 3473, 12.2.4)
KA	0.66	-	Reinforcement scaling factor
CREEP	2.5	-	Total creep factor.
BETA	0.5	-	Load frequency factor
KIT	0.3	-	Scaling factor for torsion stiffness when used in frame analysis.
TOL	0	-	Include eccentricity in axial force due to construction tolerances. Applies only to ULS and if axial force is compressive (less than zero) 0 = include only for slender structural members () 1 = include always ()
CLASS	3	-	Safety class (BBK94, 1.1.1.4)
TOLX	-1	m	Override of tolerances in section x-dir. Default: -1 means calculated according to regulations. Given value lager than zero, gives the tolerance directly.
TOLY	-1	m	Override of tolerances in section y-dir. Default: -1 means calculated according to regulations. Given value lager than zero, gives the tolerance directly.
”NAME”		-	Optional name. All characters allowed including space

DPSECT ID, TYPE, CLASS, MET, LEX, LEY, CEX, CEY, LVX, LVY, f*MVIOX, f*MVIOY, CV , ”NAME” (STEEL)

STEEL

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
TYPE	1	-	Material type. Valid types are: 1 = concrete (**CONCR**) 2 = steel (**STEEL**)
CLASS	1	-	Section Class (NS3472, 5.2.2) NOT IN USE!
MET	0	-	Production method (**CALC** = 0 = Default**, ROLLED** = 1, **WELDED** = 2)
LEX	0.0	m	Theoretical buckling length about section x-axis (L-axis in NovaFrame)
LEY	0.0	m	Theoretical buckling length about section y-axis (N-axis in NovaFrame)
CEX	0	-	Buckling Curve about Section y-axis (NS 3472, 5.4.1) 0 = **CALC** = Default, **CURVEA**=1, **CURVEB**=2, **CURVEC**=3.
CEY	0	-	Buckling Curve about Section y-axis (NS 3472, 5.4.1) 0 = **CALC** = Default, **CURVEA**=1, **CURVEB**=2, **CURVEC**=3.
LVX	0.0	m	Theoretical lateral buckling length about section x-axis (L-axis in NovaFrame)
LVY	0.0	m	Theoretical lateral buckling length about section y-axis (N-axis in NovaFrame)
f*MVIOX	0.0	-	Factor on lat. Buckl. Mom. about section x-axis (L-axis in NovaFrame)
f*MVIOY	0.0	-	Factor on lat. Buckl. Mom. about section y-axis (N-axis in NovaFrame)
CV	0	-	Lateral buckling Curve about Section x and y-axis (NS 3472, 5.4.1) 0 = **CALC** = Default, **V15** = 1, **V20** = 2.
”NAME”		-	Optional name. All characters allowed including space

DPSECT defines a set of general section parameters.

DPCONCR ID, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS

NS 3473

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
GAMULS	1.40	-	Material safety factor, ULS, acc. to NS3473
GAMPLS	1.20	-	Material safety factor, PLS, acc. to NS3473
GAMSLS	1.00	-	Material safety factor, SLS, acc. to NS3473
EPSULS	3.50	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	3.50	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	3.50	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
GAMULS	1.50	-	Material safety factor γ_c, ULS (cl. NA.2.4.2.4(1))
GAMPLS	1.20	-	Material safety factor γ_c, PLS (cl. NA.2.4.2.4(1))
GAMSLS	1.00	-	Material safety factor γ_c, SLS (cl. NA.2.4.2.4(2))
EPSULS	3.50	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	3.50	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	3.50	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

BBK 94

ID	1	-	Parameter set number
GAMULS	1.50	-	Total material safety factor ($\eta*\gamma_m*\gamma_n$), ULS
GAMPLS	1.20	-	Total material safety factor ($\eta*\gamma_m*\gamma_n$), PLS
GAMSLS	1.00	-	Total material safety factor, SLS
EPSULS	3.50	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	3.50	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	3.50	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

DPCONCR defines a set of concrete specific design parameters for a cross section.

DPSTEEL ID, GAMULS, GAMPLS, EPSULS, EPSPLS, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
GAMULS	1.15	-	Material safety factor, ULS, acc. to NS3473
GAMPLS	1.00	-	Material safety factor, PLS, acc. to NS3473
EPSULS	12.0	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	12.0	o/oo	Strain limit for capacity charts and design calculations, PLS
”NAME”		-	Optional name. All characters allowed including space

DPSTEEL defines a set of steel specific design parameters for a cross section.

DPREINF ID,BOND, KT, WD, C1, SCALEWK, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

NS 3473

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
BOND	0.75	-	Bond coefficient f_{tk}/τ_{bk}, acc. to NS3473, A.15.6.2
KT	1.5	-	NS 3473, Table 10
WD	0.3	mm	Allowable crack width. NS 3473, Table 9
C1	55.	mm	Minimum concrete cover, acc. to NS3473 table 12 and cl 17.1.1
SCALEWK	**OFF**	-	Scale calculated crack widths with ratio c_1/c_2 acc NS3473 (**ON**, **OFF**)
GAMULS	1.25	-	Material safety factor, ULS, acc. to NS3473
GAMPLS	1.10	-	Material safety factor, PLS, acc. to NS3473
GAMSLS	1.00	-	Material safety factor, SLS, acc. to NS3473
EPSULS	5.00	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	5.00	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	5.00	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992

DPREINF ID, K1, KC, WMAX, -, -, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
K1	0.8	-	Bond factor (cl. 7.3.4).
KC	1.0	-	Concrete cover factor (cl. NA.7.3.1)
WMAX	0.3	mm	Crack width limit. (table NA.7.1N) exclusive of k_c-factor

-			-not in use-
-			-not in use-
GAMULS	1.15	-	Material safety factor γ_s, ULS (cl. NA.2.4.2.4(1))
GAMPLS	1.00	-	Material safety factor γ_s, PLS (cl. NA.2.4.2.4(1))
GAMSLS	1.00	-	Material safety factor γ_s, SLS (cl. NA.2.4.2.4(2))
EPSULS	5.00	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	5.00	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	5.00	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

BBK94

DPREINF ID, KAPPA1, KSI, WD, C1 , - ,GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
KAPPA1	1.2	-	BBK94, 4.5.5
KSI	1.5	-	Crack coefficient. BBK94, table 4.5.4
WD	0.2	mm	Allowable crack width. BBK94, table 4.5.4
C1	45.	mm	Minimum concrete cover, acc. to table 3.9.5b
-			-not in use-
GAMULS	1.15	-	Total material safety factor ($\eta*\gamma_m*\gamma_n$), ULS
GAMPLS	1.00	-	Total material safety factor ($\eta*\gamma_m*\gamma_n$), PLS
GAMSLS	1.00	-	Total material safety factor, SLS
EPSULS	5.00	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	5.00	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	5.00	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

DPREINF defines a set of reinforcement specific design parameters.

DPTEND ID, BOND, WD, C1, SCALEWK, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

NS 3473

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
BOND	1.15	-	Bond coefficient f_{tk}/τ_{bk}, acc. to NS3473
KT	2.0	-	
WD	0.2	mm	Allowable crack width
C1	65.	mm	Minimum concrete cover, acc. to NS3473 table 12 and cl 17.1.1
SCALEWK	**OFF**	-	Scale calculated crack widths with ratio c_1/c_2 acc NS3473 (**ON**, **OFF**)
GAMULS	1.15	-	Material safety factor, ULS, acc. to NS3473
GAMPLS	1.00	-	Material safety factor, PLS, acc. to NS3473
GAMSLS	1.00	-	Material safety factor, SLS, acc. to NS3473
EPSULS	10.0	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	10.0	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	10.0	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

NS-EN 1992

DPTEND ID, K1, KC, WMAX, -, -, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
K1	1.6	-	Bond factor (cl. 7.3.4).
KC	1.0	-	Concrete cover factor (cl. NA.7.3.1)
WMAX	0.2	mm	Crack width limit. (table NA.7.1N)
-			-not in use-
-			-not in use-
GAMULS	1.15	-	Material safety factor γ_s, ULS (cl. NA.2.4.2.4(1))
GAMPLS	1.00	-	Material safety factor γ_s, PLS (cl. NA.2.4.2.4(1))
GAMSLS	1.00	-	Material safety factor γ_s, SLS (cl. NA.2.4.2.4(2))
EPSULS	5.00	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	5.00	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	5.00	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

BBK94

DPTEND ID, KAPPA1, KSI, WD, C1 , -, GAMULS, GAMPLS, GAMSLS, EPSULS, EPSPLS, EPSSLS, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	Parameter set number
KAPPA1	1.2	-	BBK94, 4.5.5
KSI	2.0	-	Crack coefficient. BBK94, table 4.5.4
WD	0.1	mm	Allowable crack width. BBK94, table 4.5.4
C1	45.	mm	Minimum concrete cover, acc. to table 3.5.9b
-			-not in use-
GAMULS	1.38	-	Total material safety factor ($\eta^*\gamma_m^*\gamma_n$), ULS
GAMPLS	1.20	-	Total material safety factor ($\eta^*\gamma_m^*\gamma_n$), PLS
GAMSLS	1.00	-	Total material safety factor, SLS
EPSULS	10.0	o/oo	Strain limit for capacity charts and design calculations, ULS
EPSPLS	10.0	o/oo	Strain limit for capacity charts and design calculations, PLS
EPSSLS	10.0	o/oo	Strain limit for capacity charts and design calculations, SLS
”NAME”		-	Optional name. All characters allowed including space

DPTEND defines a set of tendon specific design parameters.

5.4 Spectra input

EQSPEC, NO, TYPE, PAR1, PAR2, PAR3, ”NAME”

	Default	*Unit*	*Description*
NO	1	-	Earthquake specter number
TYPE	2	-	0 = Predefined by design code 1 = User defined by pseudo-acceleration specter points 2 = User defined by pseudo-velocity specter points 3 = User defined by pseudo-displacement specter points
TYPE = 0:			
PAR1	1.0	-	Seismic design code 1 = NS-EN 1998-1:2004+NA:2008
PAR2	3.0	-	Soil type factor S (Table NA.3.3) 1 = Soil type A 2 = Soil type B 3 = Soil type C 4 = Soil type D 5 = Soil type E
PAR3	1.5	-	Construction factor q
TYPE = 1, 2 or 3			
PAR1	1	*)	Normalization factor
PAR2	5	%	Actual damping ratio for specter
PAR3	0	-	Method for scaling specter values if damping ratio in an EARTHQUA command is different from 5%. 0 = No correction 1 = ELOCS 85 2 = Eurucode-8
”NAME”	-	-	Optional name of the spectra

*) Unit depends on actual input for the spectra points, can be m/s^2 when the normalization is g=9.81 m/s2.

EQSPEC defines an earthquake specter.

EQSPECPT, NO, DIR, SPECPT, FREQ, VAL

	Default	*Unit*	*Description*
NO	0	-	Eartquake spectra number.
DIR	1	-	Spectra value direction: 1 = Horizontal (**HOR**), 2 = Vertical (**VER**),
SPECPT	0	-	Spectra point number; 1, 2, 3, 4,… maximum=20
FREQ	-1	Hz	Frequency
VAL	-1	*)	Spectra value

*) Unit can be acceleration [m/s^2], velocity [m/s] or displacement [m]

EQSPECPT defines an earthquake specter point. Maximum is 20 points in each direction. Valid only for specters defined with TYPE = 1, 2 or 3 (ref. command EQSPEC). If no vertical spectra points are given the horizontal spectra points will be used.

NOTE: The specter points must be entered with increasing frequency.

WINDSPEC, NO, TYPE, REFHEIGHT, CAERO ”NAME”

	Default	*Unit*	*Description*
NO	0	-	Wind spectra number
TYPE	1	-	1: NS3491-4 , 2: Von Karman
REFHEIGHT	0	m	Reference height. If 0 then maximum calc. Z by nodes is used
CAERO	1	-	1: Include aerodynamic dampening, 0:Not include…
”NAME”	-	-	Optional name of the spectra

WINDSPEC, Defines a windspectra.

WINDLEN, NO, xLU, xLV, xLW

	Default	*Unit*	*Description*
NO	0	-	Wind spectra number

xLU	0	m	Length scale
xLV	0	m	Length scale
xLW	0	m	Length scale

WINDLEN, Used for overriding spectra values derived from actual spectra definition at reference height. 0 : will not override.

WINDTURB, NO, IU, IV, IW

	Default	*Unit*	*Description*
NO	0	-	Wind spectra number
IU	0	-	Turbulence intensity
IV	0	-	Turbulence intensity
IW	0	-	Turbulence intensity

WINDTURB, Used for overriding spectra values derived from actual spectra definition at refernce height. 0 : will not override.

WINDCOH, NO, COHU, COHW

	Default	*Unit*	*Description*
NO	0	-	Wind spectra number
COHU	9	-	Coherence decay factor, wind in u-direction (local L-axis). Corresponds to C_{uy} in SVV Håndbok 185.
COHW	6	-	Coherence decay factor, wind in w-direction (local N-axis) Corresponds to C_{wy} in SVV Håndbok 185.

WINDCOH, Used for defining the coherence for the actual spectra.

5.5 Reference line data input

REFLINE No, Type, Name

	Default	*Unit*	*Description*
No:	1	-	Reference Line number.
Type:	1	-	Type of reference line ; 1 = Horizontal projection (Global X-Yplane), 2 = Vertical projection (Global Z-Axis), 3 = Stations by length of line.
Name:		-	Optional name. All characters allowed including space.

REFLINE defines the reference line properties and its name.

RFLAXIS No RefLineNo StationNo Name

	Default	*Unit*	*Description*
No:	1	-	Axis number.
RefLineNo:	1	-	Reference Line No.
StationNo:	0.0	m	Station on reference line [m].
Name:		-	Optional name. All characters allowed including space.

RFLAXIS defines the name of a specific station on a reference line.

RFLHORI No RefLineNo Type Stat1 X1 Y1 Stat2 X2 Y2 R1 R2 A

	Default	*Unit*	*Description*
No:	1	-	Segment number.
RefLineNo:	1	-	Reference Line No.
Type:	0	-	Discrete segments: Type 1 = straight line, type 2 circle, type 3 Clothoide. Chained segments: Type 11 = straight line, type 12 circle, type 13 Clothoide.
Stat1:	1	m	Start station on reference line. (Not in use for type 11 - 13).
X1:	1	m	Start x -ordinate. (Not in use for type 11 - 13).
Y1:	0	m	Start y -ordinate. (Not in use for type 11 - 13).
Stat2:	1	m	End station on reference line.
X2:	1	m	End x -ordinate. (Not in use for type 11 - 13).
Y2:	0	m	End y -ordinate. (Not in use for type 11 - 13).
R1:	1	m	Start Radius.
R2:	0	m	End Radius.
A:	0		Clothoide parameter.

RFLHORI defines segments of geometry in the global X-Y-plane. This is usually the projection of the road in the X-Y-plane.

NOTE: All lines must start with minimum one discrete segment.

RFLVERT No RefLineNo Type Stat1 Z1 Stat2 Z2 R

	Default	*Unit*	*Description*
No:	1	-	Segment number.
RefLineNo:	1	-	Reference Line No.
Type:	0	-	Type 1 = straight line, type 2 circle.
Stat1:	1.	m	Start station on reference line.
Z1:	0.	m	Start z -value.
Stat2:	1.	m	End station on reference line.
Z2:	1.	m	End z -value.
R1:	1.	m	Start Radius.

RFLVERT defines segments of vertical geometry for the horizontal segments.
NOTE: Horizontal segments must be defined for the reference line before the vertical segments can be defined.

RFLCOORD No RefLineNo Type X Y Z StationNo.

	Default	*Unit*	*Description*
No:	1	-	Coord number.
RefLineNo:	1	-	Reference Line No.
Type:	0	-	Type 1 = first coord (station number is given).

		-	Type 2 = coordinates with no station numbering (station is calculated).
X:	1.	m	X - ordinate.
Y:	1.	m	Y - ordinate.
Z:	0.	m	Z - ordinate.
StationNo:	0.	m	StationNo on reference line. (Only for type = 1)

RFLCOORD defines coordinates to be chained on order to define the geometry of a reference line.
NOTE: All lines defined are defined by a sequence and must start with one and only one COORD of type 1.

RFLCOLUM ColRfl RoadRflNo Type StationNo/AxisNo Offset Angle TopElev. BotElev.

	Default	*Unit*	*Description*
ColRfl:	1	-	Reference line no for this column. Reference line must be of type 2 (=Station by vertical projection).
RoadRflNo:	1	-	Reference Line to which the column is attached.
Type:	0	-	Type 1 = Position by station, top of column to be automatically calculated. Type 2 = Position by Axis no., top of column to be automatically calculated. Type 3 = Position by station, top of column is given directly. Type 4 = Position by Axis no., top of column is given directly.
StationNo/AxisNo:	0.	m/-	For type 1 and 3: station no. For type 2 and 4 axis no.
Offset:	0.	m	Offset normal to reference line at given station/Axis.
Angle:	0.	°	Angle in [deg.] from RoadRflNo line to the column.
TopElev:	0.	m	Top elevation of column
BotElev:	0.	m	Bottom elevation of column

RFLCOLUM defines a column attached to a reference line.

5.6 Cross section geometry input

XSECT, RFL, PRO, TYPE, MAT, MATNO, DESNO, SDESNO, ”NAME”

	Default	*Unit*	*Description*	
RFL	0	-	Reference line number	
PRO	0.	m	Station number (profile number)	
TYPE	13	-	Code for section geometry type. Valid types are:	
			1 = Stiffness values, No concrete/steel design!	(**MSTIFF**)
			2 = Circular section, No concrete/steel design!	(**MCIRC**)
			3 = Rectangular section, No concrete/steel design!	(**MRECT**)
			4 = Box section, No concrete/steel design!	(**MBOX**)
			5 = T-section, No concrete/steel design!	(**MT**)
			11 = Massive rectangular. Parametric shape	(**RECT**)
			12 = Massive circular. Parametric shape	(**CIRC**)
			13 = Massive general. Based on section point input	(**GENERAL**)
			14 = Panel general Panel section based on section point input	(**PANEL**)
			20 = Deformation Spring.	(**MDEFSPR**)
			22 = 6x6 Spring matrix.	(**MSPRMTX**)
			21= Rotation spring.	(**MROTSPR**)
MAT	1	-	Material type	
			1 = concrete	(**CONCR**)
			2 = steel	(**STEEL**)
MATNO	1	-	Material number	
DESNO	1	-	Material design parameter set	
SDESNO	1	-	Section design parameter set	
”NAME”		-	Optional cross section name. All characters allowed including space	

XSECT defines a new cross section. This command replaces the command SECT.

DIM, RFL, PRO, DIM1, DIM2, …DIMn

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0.	m	Station number (profile number). Section number for RFL=0.
DIM1	0.	var.	Cross section measure number 1
DIMn	0	var.	Cross section measure number n

DIM cross section measures for non design sections. Valid shapes are:

Stiffness values (1):	DIM1 = Area, [m2]
	DIM2 = I_L, [m4]
	DIM3 = I_M, [m4]
	DIM4 = I_N, [m4]
Circ. section (2):	DIM1 = Outer diameter, [mm]
	DIM2 = Inner diameter, [mm]
Rect. section (3):	DIM1 = Width, [mm]
	DIM2 = Height, [mm]
Box section (4):	DIM1 = Outer Width, [mm]
	DIM2 = Outer Height, [mm]
	DIM1 = Web thickness, [mm]
	DIM2 = Flange thickness, [mm]
T-section (5):	DIM1 = Total Width, [mm]
	DIM2 = Total Height, [mm]
	DIM1 = Web thickness, [mm]
	DIM2 = Flange thickness, [mm]

DIM cross section measures for parametric section shapes. Valid shapes are:

Rectangular (11):	DIM1 = width	
	DIM2 = height	
	DIM3 = thickness of left web	(0.0 if massive cross section)
	DIM4 = thickness of right web	(0.0 if massive cross section)
	DIM5 = thickness of upper slab	(0.0 if massive cross section)
	DIM6 = thickness of lower slab	(0.0 if massive cross section)
Circular (12):	DIM1 = Outer radius	
	DIM2 = tubular wall thickness	(0.0 if massive cross section)

DIM definitions for spring sections:

Def. Spring (20):	DIM1 = K_L [kN/m]

DIM2 = K_M [kN/m]
DIM3 = K_N [kN/m]

Rot. Spring (21): DIM1 = $K_{\varphi L}$ [kNm/rad]
DIM2 = $K_{\varphi M}$ [kNm/rad]
DIM3 = $K_{\varphi N}$ [kNm/rad]

Spring Matrix (22): Require 6 lines of dim commands.

DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7
1	K_{LL}	K_{LM}	K_{LN}	$K_{L\varphi L}$	$K_{L\varphi M}$	$K_{L\varphi N}$
2	K_{ML}	K_{MM}	K_{MN}	$K_{M\varphi L}$	$K_{M\varphi M}$	$K_{M\varphi N}$
3	K_{NL}	K_{NM}	K_{NN}	$K_{N\varphi L}$	$K_{N\varphi M}$	$K_{N\varphi N}$
4	$K_{\varphi LL}$	$K_{\varphi LM}$	$K_{\varphi LN}$	$K_{\varphi L\varphi L}$	$K_{\varphi L\varphi M}$	$K_{\varphi L\varphi N}$
5	$K_{\varphi ML}$	$K_{\varphi MM}$	$K_{\varphi MN}$	$K_{\varphi M\varphi L}$	$K_{\varphi M\varphi M}$	$K_{\varphi M\varphi N}$
6	$K_{\varphi NL}$	$K_{\varphi NM}$	$K_{\varphi NN}$	$K_{\varphi N\varphi L}$	$K_{\varphi N\varphi M}$	$K_{\varphi N\varphi N}$

Note: Matrix is assumed symmetric, only upper right half is used!
Units: [kN/m], [kNm/rad].

PT, RFL, PRO, ID, TYPE, X, Y

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	1	m	Station number (profile number)
ID	1	-	ID for section point (unique within actual cross section)
TYPE	**COOR**	-	Section point type. Valid types are: 0 = coordinate point (local X- and Y- coordinates) (**COOR**) -1 = curve point (Radius for curved line between two coordinate points) (**RAD**)
X	0.	mm	X-coordinate for section point, or radius if TYPE = -1
Y	0.	mm	Y-coordinate for section point, unused if TYPE = -1

PT defines a new section point for a defined cross section. Cross section must have type = 13 (free shape)

PANEL, RFL, PRO, ID, PT1, PT2, THICK

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0.	m	Station number (profile number)
ID	1	-	Panel number
PT1	0	-	Start point for panel
PT2	0.	-	End point for panel
THICK	0.	mm	Panel thickness

Defines a new panel within the actual cross section. The cross section must be of type 14 (PANEL). The section points PT1 and PT2 must have been previously created with the PT command.

CONN, RFL, PRO, CONNTYPE, CONPT, OFFL, OFFN

Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0.	m	Station number (profile number)
CONNTYPE	1	-	Code for method used for specifying location of reference line intersection point 1 = Reference line intersects at CoG, or at an offset OFFX and OFFY from GoG 2 = Reference line intersects at specified section point PT, or at an offset OFFX and OFFY from section point PT.
CONPT	0	-	Section point used for defining reference line intersection point
OFFL	0.	m	Offset in section L(X)-direction *from* reference line intersection point *to* GoG or PT
OFFN	0.	m	Offset in section N(Y)-direction *from* reference line intersection point *to* GoG or PT

Defines location of reference line intersection point in the cross section plane. This command is *not* valid for sections connected to reference line 0.

SECT, RFL, PRO, TYPE, CMAT, CDES, SDES, CONN, PT, OFFX, OFFY, ”NAME”

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0.	m	Station number (profile number)
TYPE	13	-	Code for section geometry type. Valid types are: 11 = rectangular. Parametric shape (**RECT**) 12 = circular. Parametric shape (**CIRC**) 13 = general. Free shape based on section point input (**GENERAL**)
CMAT	1	-	Concrete material type
CDES	1	-	Concrete design parameter set
SDES	1	-	Section design parameter set

CONN	1	-	Code for method used for specifying location of reference line intersection point 1 = Reference line intersects at CoG 2 = Reference line intersects at an offset OFFX and OFFY from GoG 3 = Reference line intersects at specified section point PT 4 = Reference line intersects at an offset OFFX and OFFY from section point PT
PT	0	-	Section point used for defining reference line intersection point
OFFX	0.	m	Offset in section X-direction *from* reference line intersection point *to* GoG or PT
OFFY	0.	m	Offset in section Y-direction *from* reference line intersection point *to* GoG or PT
”NAME”		-	Optional cross section name. All characters allowed including space

Use XSECT command in stead.

SECTFACE , RFL, PRO, FROM, TO, STEP, FACE

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0	m	Station number (profile number)
FROM	0	-	From section point no
TO	0	-	To section point no
STEP	1	-	Step
FACE	-1	-	Actual Face association: -1: No face association 0: If section point y-ordinate is larger than cog y- then section point is upper face, else it will be lower face. 1: Section point is upper face (**UPPER**) 2: Section point is lower face (**LOWER**)

SECTFACE Labels section points to a spescific face. Only points included in a sectface command will be assosiated with a face.

5.7 Cross section reinforcement and tendon input

RETYP, ID, TYPE, V1, V2, V3, V4

	Default	*Unit*	*Description*		
ID	0	-	reinforcement type number		
TYPE	2	-	Code for amount specification. Valid types are:		
			1 = Number of budles given (N).		**NUM**
			2 = Bundle spacing given (cc)		**CC**
			3 = Total As given (As)		**AS**
			TYPE = 1	TYPE = 2	TYPE = 3
V1	0	-/-/mm^2	N	n	As
V2	0	-/mm/-	n	ø	n
V3	0	mm	ø	cc	ø
V4	0	-/-/mm	-	-	cc

RETYP defines a new reinforcement amount definition. Number of bars pr. bundle (n) can be from 0.5 to 4.0 in step of 0.5.

RE, RFL, PRO, ID, PT1, PT2, C1, C2, C3, C4, HCEF, LOC, ROT, CALCWK, ”NAME”

	Default	*Unit*	*Description*	
RFL	0	-	Reference line number	
PRO	1	m	Station number (profile number)	
ID	1	-	ID for reinforcement group (unique within actual cross section)	
Rectangular cross section (cross section type = 11)				
PT1	1	-	Face code :	1 = upper, 2 = lower, 3 = left, 4 = right
PT2	1	-	Face code:	1 = inner, 2 = outer
C1	50	mm	Concrete cover at start point	
C2	50	mm	Concrete cover at end point	
C3	50	mm	Lateral concrete cover at start point	
C4	50	mm	Lateral concrete cover at end point	
Circular cross section (cross section type = 12)				
PT1	1	°	Start of sector. Angle given relative to section X-axis	
PT2	1	°	End of sector. Angle given relative to section X-axis	
C1	50	-	Face code:	1 = inner, 2 = outer
C2	50	-	Edge code:	1 = bundle on edge, 2 = not bundles on edge
C3	50	mm	Concrete cover	
C4	50	-	-	
General cross section (cross section type = 13)				
PT1	1	-	Reference section point 1	
PT2	1	-	Reference section point 2	
C1	50	mm	Concrete cover at section point 1	
C2	50	mm	Concrete cover at section point 2	
C3	50	mm	Lateral concrete cover at section point 1	
C4	50	mm	Lateral concrete cover at section point 2	
HCEF	500	mm	Maximum effective cross section height (NS 3473 cl. A.15.6.2.1)	
LOC	1	-	Code for adjustment of lateral location of group	
			1 = centric	
			2 = move group towards start of distribution line	
			3 = move group towards end of distribution line	
ROT	0.	°	Rotation of bundles	
CALCWK	1	-	Select if crack widths are to be calculated for this reinforcement group, or not	
			0 = do not calculate crack widths	**NO**
			1 = calculate crack widths	**YES**
”NAME”		-	Optional reinforcement group name. All characters allowed including space	

RE defines a new reinforcement group

NS-EN 1992

CALCWK	1	-	Option for crack width calculation for this reinforcement group,.

0 = Omit crack width calculation for this reinforcement group.
1 = Calculate crack widths based on distribution length.
2 = calculate crack widths based on bar spacing.

REAS, RFL, PRO, ID, NMAT, NPAR, TABLE, CUR, LOCK, RETYP, ….

	Default	*Unit*	*Description*
RFL	0	-	Reference line number for actual cross section
PRO	0	m	Station number (profile number) for actual cross section
ID	1	-	ID for reinforcement group this amount specification is assigned to
NMAT	1	-	Reinforcement material
NPAR	1	-	Reinforcement design parameter set
TABLE	-1	-	-not in use-
CUR	0	-	ID for selected reinforcement amount (RETYP)
LOCK	0	-	Lock code: 0 = allow amount to be stepped 1 = lock at current amount
RETYP	0	-	Reinforcement amount in step table (up to 60 entries in table)

REAS specifies selected reinforcement amount and amount step table for a reinforcement group.

SHRETYP, ID, TYP, V1, V2, V3

	Default	*Unit*	*Description*	
ID	0	-	Reinforcement type number	
V1	0	-	n	Bars pr. bundle
V2	0	mm	ø	Bar diameter
V3	0	mm	cc	Bundle spacing

SHRETYP defines a new shear reinforcement amount definition. Number of bars pr. bundle (n) can be from 0.5 to 4.0 in step of 0.5.

SHRE, RFL, PRO, ID, SHAPE, ”NAME”

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	1	m	Station number (profile number)
ID	1	-	ID for shear reinforcement unit (unique within actual cross section)
SHAPE	1	-	Shape code. Valid shapes are: 1 = General
”NAME”		-	Optional name. All characters allowed including space

SHRE defines a new shear reinforcement unit

SHREPT, RFL, PRO, ID, TYPE, PT, C1, C2, …

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0	m	Station number (profile number)
ID	1	-	ID for reinforcement group (unique within actual cross section)
TYPE	0	-	Methode for calculating location relative to section point PT. Valid types are: 0, 1,2 and 3 (see users manual)
PT	0	-	Reference cross section point
C1	0	mm	Concrete cover C1 relative to section point PT
C2	0	mm	Concrete cover C2 relative to section point PT

SHREPT defines geometry for a defined shear reinforcement unit. Each SHREPT command can consist of up to 20 points. For each new point add a , TYPE, PT, C1, C2 sequence.

SHREAS, RFL, PRO, ID, NMAT, NDES, TABLE, CUR, LOCK, SHRETYP, …

	Default	*Unit*	*Description*
RFL	0	-	Reference line number for actual cross section
PRO	0	m	Station number (profile number) for actual cross section
ID	1	-	ID for reinforcement unit this amount specification is assigned to
NMAT	1	-	Reinforcement material
NPAR	1	-	Reinforcement design parameter set
TABLE	-1	-	-not in use-
CUR	0	-	ID for selected reinforcement amount (SHRETYP)
LOCK	0	-	Lock code: 0 = allow amount to be stepped 1 = lock at current amount
SHRETYP	0	-	Reinforcement amount in step table (up to 60 entries in table)

SHREAS specifies selected shear reinforcement amount and amount step table for a shear reinforcement unit.

TE, RFL, PRO, ID, TYPE, PT1, PT2, C1, C2, C3, C4, HCEF, LOC, "NAME"

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	1	m	Station number (profile number)
ID	1	-	ID for reinforcement group (unique within actual cross section)
TYPE	1	-	1 = location defined by 2 reference section points or sector angles 2 = location defined in coordinates for start and end of group

TYPE = 1
Rectangular cross section (cross section type = 11)

PT1	1	-	Face code : 1 = upper, 2 = lower, 3 = left, 4 = right
PT2	1	-	Face code: 1 = inner, 2 = outer
C1	100	mm	Concrete cover at start point
C2	100	mm	Concrete cover at end point
C3	100	mm	Lateral concrete cover at start point
C4	100	mm	Lateral concrete cover at end point

Circular cross section (cross section type = 12)

PT1	1	°	Start of sector. Angle given relative to section X-axis
PT2	1	°	End of sector. Angle given relative to section X-axis
C1	100	-	Face code: 1 = inner, 2 = outer
C2	100	-	Edge code: 1 = bundle on edge, 2 = not bundles on edge
C3	100	mm	Concrete cover
C4	100	-	-

General cross section (cross section type = 13)

PT1	1	-	Reference section point 1
PT2	1	-	Reference section point 2
C1	100	mm	Concrete cover at section point 1
C2	100	mm	Concrete cover at section point 2
C3	100	mm	Lateral concrete cover at section point 1
C4	100	mm	Lateral concrete cover at section point 2

TYPE = 2

PT1	1	mm	X1
PT2	1	mm	Y1
C1	100	mm	X2
C2	100	mm	Y2
C3	100	-	-not in use-
C4	100	-	--not in use-

HCEF	500	mm	Maximum effective cross section height (NS 3473 cl. A.15.6.2.1)
LOC	1	-	Code for adjustment of lateral location of group 1 = centric 2 = move group towards start of distribution line 3 = move group towards end of distribution line
CALCWK	0	°	Option for crack width calculation for this tendon group, 0 = do not calculate crack widths **NO** 1 = calculate crack widths **YES**
"NAME"		-	Optional reinforcement group name. All characters allowed including space

TE defines a new tendon group

<u>NS-EN 1992</u>

CALCWK	0	°	Option for crack width calculation for this reinforcement group, -1 = calculate crack widths at tendon location by scaling crack widths for a governing reinforcement group selected automatically by the program. 0 = do not calculate crack widths **NO** >0 calculate crack widths at tendon location by scaling crack widths for the specified governing reinforcement group CALCWK.

TEAS, RFL, PRO, ID, NMAT, NPAR, EPSI, DUCT, ASCODE, VAL1, VAL2

	Default	*Unit*	*Description*
RFL	0	-	Reference line number for actual cross section
PRO	1	m	Station number (profile number) for actual cross section

ID	1	-	ID for reinforcement group this amount specification is assigned to	
NMAT	1	-	Tendon material	
NPAR	1	-	Tendon design parameter set	
EPSI	0.0	o/oo	Initial strain	
DUCT	50	mm	Duct diameter	
ASCODE	1	-	Code for type of As input (VAL1, VAL2). Valid codes are: 1 = n x As entered 2 = total As and tendon spacing entered	
			ASCODE = 1	ASCODE = 2
VAL1	2	-/mm^2	n	Astot
VAL2	100	mm^2/mm	As	cc

TEAS specifies tendon section area and properties.

5.8 Cross section sub area input

SUBAR, RFL, PRO, ID, TYPE, CRNA, CRNB, CRNC, CRND,PT, PT, …

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0	m	Station number (profile number)
ID	1	-	ID for sub area (unique within actual cross section)
TYPE	0	-	Sub area type. Valid types are: 0 = beam 1 = column 2 = slab 3 = web 4 = wall
CRNA	0	-	Section point defining corner A (upper left corner) of sub area. If set to 0, the program will automatically choose all corner points A to D. The values 'CRNB', 'CRNC' and 'CRND' are then ignored.
CRNB	0	-	Section point defining corner B (lower left corner) of sub area
CRNC	0	-	Section point defining corner C (lower right corner) of sub area
CRND	0	-	Section point defining corner D (upper right corner) of sub area
PT	-	-	ID of first section point on circumference of sub area
PT…	-	-	ID if following section points defining circumference of sub area.

NS-EN 1992

SUBAR, RFL, PRO, ID, TYPE, STRUTANG, CRNA, CRNB, CRNC, CRND,PT, PT, …

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0	m	Station number (profile number)
ID	1	-	ID for sub area (unique within actual cross section)
TYPE	0	-	Sub area type. Valid types are: 0 = beam 1 = column 2 = slab 3 = web 4 = wall
STRUTANG	40	deg	Shear strut angle (cl. 6.2.3 (2)): > 0 = user specified strut angle -1 = use minimum allowable strut angle -2 = use maximum allowable strut angle
CRNA	0	-	Section point defining corner A (upper left corner) of sub area. If set to 0, the program will automatically choose all corner points A to D. The values 'CRNB', 'CRNC' and 'CRND' are then ignored.
CRNB	0	-	Section point defining corner B (lower left corner) of sub area
CRNC	0	-	Section point defining corner C (lower right corner) of sub area
CRND	0	-	Section point defining corner D (upper right corner) of sub area
PT	-	-	ID of first section point on circumference of sub area
PT…	-	-	ID if following section points defining circumference of sub area.

SUBAR defines a new sub area for a defined cross section.

SHEAR, RFL, PRO, ID, DIR, SUB

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	1	m	Station number (profile number)
ID	1	-	ID for shear area (unique within actual cross section)
DIR	1	-	Shear force direction 0 = default (based on sub area type) 1 = section X-direction (based on sub area type) 2 = section Y-direction (based on sub area type)
SUB	1	-	ID of sub area defining area extent

SHEAR defines a new shear area for a defined cross section.

TORAR, RFL, PRO, ID, TYPE, SUB1, SUB2, SUB3, SUB4

	Default	*Unit*	*Description*
RFL	0	-	Reference line number
PRO	0	m	Station number (profile number)
ID	1	-	ID for section point (unique within actual cross section)
TYPE	0.	-	Torsion area type. Valid types are: 0 = massive 1 = box
SUB1	0.	-	ID of sub area defining left web of box, or whole torsion area if TYPE = 1
SUB2	0.	-	ID of sub area defining right web of box. Unused if TYPE = 1.
SUB3	0.	-	ID of sub area defining lower slab of box. Unused if TYPE = 1
SUB4	0.	-	ID of sub area defining upper slab of box. Unused if TYPE = 1

TORAR defines a new torsion area for a defined cross section.

5.9 Cross section external force input

Applies only to cross sections connected to reference oine 0

SECTFORCE, ID, LSTATE, SECT, PERM, N, MX, MY, VX, VY, T

	Default	*Unit*	*Description*
ID	1	-	Load combination number
LSTATE:	ULS	-	Limit State; 1=ULS, 2=PLS, 3=SLS. (**ULS, PLS, SLS**)
SECT	1	-	Cross section number (section on reference line 0)
PERM	-1	-	ID for loadcombination containing section forces from permanent loads -1 = not specified
N	0	kN	Axial force
MX	0	kNm	Bending moment about section x-axis
MY	0	kNm	Bending moment about section y-axis
VX	0	kN	Shear force in section x-axis
VY	0	kN	Shear force in section y-axis
T	0	kNm	Torsion moment.

SECTFORCE defines a new set of cross sectional forces for design calculations

5.10 Frame geometry ascii input

NODEINS FromNode ToNode NodeIncr StartX StartY StartZ DeltaX DeltaY DeltaZ
NODEDEL From Node ToNode NodeIncr

FromNode, ToNode and NodeIncr:	Actual node numbers.
StartX, StartY and StartZ:	Global coordinates for first node in meter.
DeltaX, DeltaY and DeltaZ:	Increment in meter in global coord. between two following nodes.

NODEINS inserts nodes and NODEDEL deletes nodes.

RFLNODE FromNode ToNode NodeIncr RefLineNo StartStation EndStation

FromNode, ToNode and NodeIncr:	Actual node numbers.
RefLineNo;	Actual Reference Line.
StartStation, EndStation	Stations on Reference line [m].

AXISNODE FromNode ToNode NodeIncr RefLineNo FromAxisNo ToAxisNo RelFromAxis RelToAxis

FromNode, ToNode and NodeIncr:	Actual node numbers.
RefLineNo;	Actual Reference Line.
FromAxisNo, ToAxisNo:	Actual Axis No. (Step is always = +1). Step in negative axis order is not supported.
RelFromAxis, RelToAxis:	Dist from axis along RefLine [m].

AXIS2NOD FromNode ToNode NodeIncr RefLineNo FromAxisNo ToAxisNo NoOfElem

FromNode, ToNode and NodeIncr:	Actual node numbers. (Note: ToNode is automatically calculated based on the other values.)
RefLineNo;	Actual Reference Line.
FromAxisNo, ToAxisNo:	Actual Axis No. (Step is always = +1). Step in negative axis order is not supported.
NoOfElem:	Number of elements between each axis.

NODECOORD FromNode ToNode NodeIncr RefLineNo FromCoord StepCoord

FromNode, ToNode and NodeIncr:	Actual node numbers.
RefLineNo:	Actual Reference Line.
FromCoord:	First coord no.
StepCoord:	Step in coord no.

ELEMINS FromElem ToElem ElemIncr LeftNode RightNode NodeIncr
ELEMDEL FromElem ToElem ElemIncr

FromElem, ToElem and ElemIncr:	Actual element numbers.
LeftNode, RightNode:	Node number on left and right side of first element.
NodeIncr:	Increment of node number between two following elements.

ELEMINS inserts elements and ELEMDEL deletes elements.

SECTINS No Type E-mod Poisson A1 A2 A3 A4 Name
SECTDEL No

No:	Geometric section number.
E-mod, Poisson:	Young's modulus [Mpa] and Poisson's number for actual section.
Type:	1: General section: A1: Area, A2: I about L, A3: I about M, A4: I about N 2: Circular section: A1: Outer diameter, A2: Inner diameter 3: Rectangular section: A1: Width, A2: Height 4: Box section: A1: Width, A2:: Height, A3: Web thickness, A4: Flange thickness 5: T-section: A1: Width, A2: Height, A3: Web thickness, A4: Flange thickness 20: Deformation spring:

A1: K in L-dir., A2:. K in M-dir., A3: K in N-dir.
21: Rotation spring:
A1: R about L, A2: R about M, A3: R about N
Name: Optional name. All characters allowed including space.

Values given in [m], [kN/m] and [kNm/Rad].
A-values not used can be omitted.

SECTINS inserts geometric sections and SECTDEL deletes geometric sections.

ELSPINS FromElem ToElem ElemIncr SectNo SectIncr Type A1 A2
ELSPDEL FromElem ToElem ElemIncr

FromElem, ToElem and ElemIncr: Actual element numbers.
SectNo, SectIncr: Sect. no. for first elem. and sect. no. incr. between elem..
Type: Type of direction description for local axis:
Type 1: Node 3 method:
A1: Node in MN-plane for first elem., A2: Node incr.
Type 2: Alpha method:
A1: Alpha value for first element, A2: Alpha increment
Type 3: N-direction method:
A1: N-axis direction (1=X, 2=Y, 3=Z, 4=-X, 5=-Y, 6=-Z)
Type 4: Default method.
Type 5: Column method. N-direction is associated with the direction of a reference line at a specific station or axis no.

ELSPINS inserts element specification and SECTDEL deletes element specifications.

BOUNDINS FromNode ToNode NodeIncr XTran YTran ZTran XRot YRot Zrot
BOUNDDEL FromNode ToNode NodeIncr

FromNode, ToNode and NodeIncr: Actual node numbers.
XTran, YTran, ZTran: 1: Fixed in X-, Y- and Z-direction 0: Not fixed *)
XRot, YRot, ZRot: 1: Fixed for rotation about X-, Y- and Z-axis 0: Not fixed

*) If master-slave connection (Indicated by a XTran number higher than 1:
XTran is first master node number (1 is not allowed).
YTran is master node increment (0 if all actual nodes have the same master node).

BOUNDINS inserts boundary conditions and BOUNDDEL deletes boundary conditions.

JOINTINS FromElem ToElem ElemIncr FirstCode SecondCode
JOINTDEL FromElem ToElem ElemIncr

FromElem, ToElem and ElemIncr: Actual element numbers.
FirstCode, SecondCode: Code for respectively first and second node on element.
Each code has 6 digits.
The first 3 digits represent the DOF in L-, M-, and N-direction.
The last 3 digits represent the DOF about L-, M-, and N-axis.
1 is released (A joint), 0 is fixed.

JOINTINS inserts joints and JOINTDEL deletes joints.

DESGINS FromElem ToElem ElemIncr FirstSedt LastSect SectIncr
DESGDEL FromElem ToElem ElemIncr

DESGINS inserts design sections and DESGDEL deletes design sections.

FromElem, ToElem and ElemIncr: Actual element numbers.
FirstSect, LastSect, SectIncr: Relative distance from first node. Value between 0 and 1.

MASS FromElem ToElem ElemIncr MassType Direction A1 A2

FromElem, ToElem and ElemIncr: Actual element numbers.
Type: Type of mass.
Type 1: Mass by density
Type 3: Distributed mass.
Type 4: Point mass.
Type 5: Rotational mass.
Direction: Direction of mass.
0 - All directions (Available for type 1, 3 and 4)
1 – Only X-direction (Available for type 4 and 5)
2 – Only Y-direction (Available for type 4 and 5)
3 – Only Z-direction (Available for type 4 and 5)

A1, A2:	Depending on type: Type 1: Mass by density: A1: density [ton/m^3], A2 not in use. Type 3: Distributed mass. A1: distributed [ton/m], A2 not in use. Type 4: Point mass. A1: point mass [ton], A2 distance from node 1. Type 5: Rotational mass. A1: rot. mass [ton*m^2], A2 distance from node 1.

MASS inserts element mass properties.

NOTE: Multiple mass definitions on the same element will be added.

WINDFACT, FROM, TO, STEP, L-$(CA)_D$, L-$(CA)_{L0}$,L-$(CA)_{L1}$, L-$(CA)_{M0}$, L-$(CA)_{M1}$, N-$(CA)_D$, N-$(CA)_{L0}$, N-$(CA)_{L1}$, N-$(CA)_{M0}$, N-$(CA)_{M1}$

	Default	*Unit*	*Description*
FROM	-	-	From element number
TO	-	-	To element number
STEP	1	-	Step in element number
L-$(CA)_D$	0	NOTE3	For wind in L-direction, Drag component
L-$(CA)_{LO}$	0	NOTE3	For wind in L-direction, Lift component,
L-$(CA)_{L1}$	0	NOTE4	For wind in L-direction, derivative Lift component
L-$(CA)_{M0}$	0	NOTE5	For wind in L-direction, torsion component
L-$(CA)_{M1}$	0	NOTE6	For wind in L-direction, derivative torsion component
N-$(CA)_D$	0	NOTE3	For wind in N-direction, Drag component
N-$(CA)_{L0}$	0	NOTE3	For wind in N-direction, Lift component
N-$(CA)_{L1}$	0	NOTE4	For wind in N-direction, derivative Lift component
N-$(CA)_{M0}$	0	NOTE5	For wind in N-direction, torsion component
N-$(CA)_{M1}$	0	NOTE6	For wind in N-direction, derivative torsion component

WINDFACT inserts element wind area properties.

NOTE1: Multiple wind area definitions on the same element will be added.
NOTE2: (CA) in the factors is the shape factor C multiplied with the unit wind area of the element. This is the heigt/width of the element perpendicular to the wind direction.
NOTE3: The unit of the non-derivative drag and lift factors is [m^2/m]
NOTE4: The unit of the derivative lift factors is [m^2/(m*rad)]
NOTE5: The unit of the non-derivative torsion factors is [m*m^2/m]
NOTE6: The unit of the derivative torsion factors is [m*m^2/(m*rad)]

NOTE7: Positive angle (alfa) for derivative components: Relative wind angle rotates counterclockwise from local L-axis to local N-axis.

TORSIONAD, FROM, TO, STEP, BTETHA

	Default	*Unit*	*Description*
FROM	-	-	From element number
TO	-	-	To element number
STEP	1	-	Step in element number
BTETHA	0	-	Constant b_θ in expression for aerodynamic torsional damping

TORSIONAD adds aerodynamic torsional damping to the specified elements in calculation of dynamic wind response. For elements which b_θ is not set the torsional damping will by default be taken as 0.0.

5.11 Tendon ascii input

TENDON TenNo TenType TenMatParNo TenDesParNo TenArea DuctDia GroutFck Name

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
TenType	0	-	Tendon type (**POST, PRE**) 0 = Post-tensioned 1 = Pre-tensioned
TenMatParNo	1	-	Material parameters
TenDesParNo	1	-	Tendon design parameters
TenArea	0	mm^2	Stressing Area of tendon
DuctDia	0	mm	Diameter of duct
GroutFck	45	MPa	Grout strength (not currently in use

TENDON describes a tendon with parameters and size.

TENGROUP TenNo NoOfTendons Space-L Space-N RelToTenNo Offset-L Offset-N

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
NoOfTendons	1	-	Total number of tendons
Space-L	0	mm	If NoOfTen >1, then this is spacing between tendons in L-direction.
Space-N	0	mm	If NoOfTen >1, then this is spacing between tendons in N-direction.
RelToTenNo	0	-	The geometry of this tendon is relative to RelToTenNo.
Offset-L	0	mm	Offset from RelTenNo to this tendon in L-direction
Offset-N	0	mm	Offset from RelTenNo to this tendon in N-direction

TENGEO TenNo GeoType RefLineNo RelativeCode SectPointNo

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
GeoType	0	-	Type of geometry: (**BYELEM, BYRFL, BYAXIS**) 0: Geometry is given by sequence of elements 1: Geometry is based on stations along a reference line. 2: Geometry is based on axes along a reference line.
RefLineNo	0	-	Current reference line, only for Geotype = 2 or 3.
RelativeCode	0	-	Used to specifying how the tendon is position is to be derived: (**RELCOG, RELUPPER, RELLOWER, RELRFL, RELPOINT**) 1: Geometry is relative to th Element axis i.e COG of the current cross section. 2: The tendon geometry in N-direction is relative to Upper Face of the cross section. 3: The tendon geometry in N-direction is relative to Lower Face of the cross section. 4: The tendon geometry is relative to the reference line. 5: The tendon geometry is relative to SectPointNo.
SectPointNo	0	-	Section point number, only in use when RelativeCode = 5.

TENGEON TenNo CurveType From To Step OffsetA OffsetB DistC
TENGEOL TenNo CurveType From To Step OffsetA OffsetB DistC

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
GeoType :	0	-	Type of geometry segment. 1)- Start Left; (y''(Left = 0) and y'(right = 0) 2)- Inflection curve: (y'(Left = 0) and y'(right = 0), inflection in middle. 3)- End Right; (y''(Right = 0) and y'(left = 0) 4)- Straight line 5)- Inflection curve: (y'(Left = 0) and y'(right = 0), inflection at DistC 6)- Start Left; (y''(Left = 0), y'(Left = FI) and y'(right = 0) 7)- End Right; (y''(Right = 0), y'(Right = FI) and y'(left = 0) 8)- Start Left; (y''(Left = 0), y'(Left = FI) and y'(right = 0), starts with 1m straight line.

9)- End Right; (y''(Right = 0),), y'(Right = FI) and y'(left = 0), ends with 1m straight line.

From	-	-	From element no or, station no, or axis no.
To	-	-	To element no or, station no, or axis no.
Step	-	-	Step elements (increment).
OffsetA:	0	m	Start offset at end 1 of FromEl.
OffsetB:	0	m	End offset at end 2 of ToEl.
DistC:	0	1	For type 5, relative distance to inflection point. For type 6,7,8,9, this is angle FI in radians.

TENGEON & TENGEOL defines tendon geometry respectively in N and L-direction.

Default Geometry: Dist to COG = 0.0 for entire tendon.

The element numbers must be given in the sequence they appear along the tendon. The element Local M-axis must be in the same direction as the sequence of elements.

For geometry based on stations on a reference line, elements associated with nodes derived by the reference line must appear in a positive order with increasing element number.

TENLOSS TenNo Friction Wobble Wedgedraw Creep Shrincage S1 S2 T2

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
Friction	0.2	1/rad	Friction coefficient.
Wobble	0.001	1/m	Wobble coefficient
WedgeDraw	0	mm	If tendon type Post-tensioned: Wedge draw in at cable active ends. If tendon type is Pre-Tensioned: this value is interpreted as the bond length, i.e. length where force increases linearly.
Creep	0	o/oo	Effect of creep. Strain, with sign!.
Shrincage	0	o/oo	Effect of shrincage. Strain, with sign.
S1	0	%	Relaxation, stress with no losses. [% of S_{02}]
S2	0	%	Relaxation, stress with losses T2. [% of S_{02}]
T2	0	%	Relaxation, loss at stress S2.[% - loss]

TENSTRESS TenNo End1 End2 First

This command defines the stressing of a tendon.

	Default	*Unit*	*Description*
TenNo	0	-	Tendon no
End1	0	%	Stressing in % of S_{02}.
End2	0	%	Stressing in % of S_{02}.
First:	1	-	Defines, which of the cable ends who is stressed first: 1 – End 1 is first, 2 – End 2 is first.

5.12 Load ascii input

LOADINS No FromElem ToElem ElemIncr Type Dir A1 A2 A3 A4 Name
LOADDEL No FromElem ToElem ElemIncr

No: Loadcase number.
FromElem, ToElem and ElemIncr: Actual element numbers.
Type:
1: Dead weight:
- A1: Weight [kN/m3]

2: Temperature:
- A1: Temperature increase [Deg.]
- A2: Temperature gradient L, [dT/L-widtht].
- A3: Temperature gradient N, [dT/N-width].
- A4: Temperature coefficient on alfa *10E-5.

3: Distributed load:
- A1: Load [kN/m]
- A2: Eccentricity in L-dir. [m]
- A3: Eccentricity in N-dir. [m]

4: Concentrated load:
- A1: Load [kN]
- A2: Distance from left node [m]

5: Moment:
- A1: Moment [kNm]
- A2: Distance from left node [m]

7: Prestressing Load: (Parasite type)
- A1: Axial Load at left end [kN] (compression = positive)
- A2: Eccentricity in N-dir. at end 1 [m]
- A3: Increment in eccentricity in N-dir. at end 2 [m]
- A4: No of cables, defaults to 1.0.
- Load direction is not in use. Defaults to M-direction

8: Trapezoidal load:
- A1: Load at left end [kN/m]
- A2: Dist. from left node [m]
- A3: Load increase at right end [kN/m]
- A4: Dist. from right node [m]

10: Static wind
- A1: Wind speed at 10 m height [m/s]
- A2: Wind direction relative to X-axis [deg.]
- A3: Not in use
- A4: Exponent for wind profile. (>0 to override default)

11: Prestressing Load: (Non parasite type)
- A1: Axial Load at left end [kN] (compression = positive)
- A2: Eccentricity in N-dir. at end 1 [m]
- A3: Increment in eccentricity in N-dir. at end 2 [m]
- A4: : No of cables, defaults to 1.0.
- Load direction is not in use. Defaults to M-direction

12: Shrinkage Load:
- A1: Shrinkage strain in 0/00.
- A2-A4: Not In use
- Load direction is not in use. Defaults to M-direction

15: Tendons (Activate tendons defined by TENDON commands):
- A1: Tendon no. To activate.
- A2: 1= paraisite forces, 0: full forces.
- A3: Stressing phase; 0 default = all, 1 = first end only.
- A4: Time dependant losses. CSR = Creep, Shrinkage, Relaxation.
  - 000 = no losses, 111 = all losses, 101= C and R ….
- Load direction is not in use. Defaults to M-direction

20: Gradient X
- A1: Load value at ordinate X1, [kN/m]
- A2: Global ordinate X1
- A3: Load value at ordinate X2, [kN/m]
- A4: Global ordinate X2, note: (X2 > X1)

21: Gradient Y
- A1: Load value at ordinate Y1, [kN/m]
- A2: Global ordinate Y1
- A3: Load value at ordinate Y2, [kN/m]
- A4: Global ordinate Y2, note: (Y2 > Y1)

22: Gradient Z
- A1: Load value at ordinate Z1, [kN/m]
- A2: Global ordinate Z1
- A3: Load value at ordinate Z2, [kN/m]
- A4: Global ordinate Z2, note: (Z2 > Z1)

30: Hydrostatic pressure
A1: Global ordinate Z, zero water level
A2: Density of water [kg/m^3]
Load direction: only L- and M-direction is in use.
40: Soil pressure
A1: Global ordinate Z, Reference soil level
A2: Density of soil [kg/m^3]
A3: Soil pressure at reference soil level [kN/m]
A4: soil coefficient k
Load direction: only X- and Y-direction is in use.

Dir.:	Load direction: 1=X, 2=Y, 3=Z, 4=L, 5=M, 6=N
Name:	Optional name. All characters allowed including space.

A-values not used can be omitted.

LOADINS inserts loads and LOADDEL deletes loads.

Traffic line syntax

TRAFLINE No FromElem ToElem ElemIncr pos MinPosDist Name

TRAFLINE inserts a line of elements called traffic line.

No:	Traffic line number.
FromElem, ToElem and ElemIncr:	Actual element numbers.
pos:	Number of position for traffic load on each element.:
MinPosDist	Minimum distance between positions (optional)
Name:	Optional name. All characters allowed including space.

NOTE: when using minimum distance between positions the following rules apply:
- Minimum 1 position on each element.
- The maximum number for each element is given by the value of pos.

Traffic load syntax

TRAFLOAD No TrackNo TrafLineNo Type EccMin EccMax Q P1 P2 P3 A-dist B-dist Name

TRAFLOAD inserts a traffic load with a track on a traffic line.

No:	Traffic load number
Track No:	Traffic track number
TrafLine No:	Traffic line number
Type:	Type of traffic load:
	1: Road
	2: Railway UIC
	3: Railway SW/0 and SW/2
	4: Subway
	6: AASHTO traffic loads
For Type of Traffic Load = 1:	
EccMin, EccMax:	Min and max eccentricity
Q:	Distributed load (Positive in neg. Z-direction)
P1, P2, P3:	Point loads (Positive in neg. Z-direction)
A-dist:	Minimum distance [m] between P1 and P2.
B-dist:	Minimum distance [m] between P2 and P3.
For Type of Traffic Load = 2:	
EccMin:	Not in use.
EccMax:	Eccentricity.
Q:	Distributed load (Positive in neg. Z-direction)
P1:	Point load used 4 times (Positive in neg. Z-direction)
P2, P3:	Not in use.
A-dist:	Distance [m] between the four point loads.
B-dist:	Not in use.
For Type of Traffic Load = 3:	
EccMin:	Not in use.
EccMax:	Eccentricity.
Q:	Distributed load (Positive in neg. Z-direction)
P1, P2, P3:	Not in use.
A-dist:	Length [m] of applied load.
B-dist:	Length [m] between applied load.
For Type of Traffic Load = 4:	
EccMin:	Not in use.

EccMax:	Eccentricity.
Q:	Distributed load (Positive in neg. Z-direction)
P1:	Point load.
A-dist:	Dist. between two P1 in a pair [m].
B-dist:	Dist between the two pairs of P1 [m].
P3:	Distance without Q, [m]
P2:	Not in use.

For Type of Traffic Load = 6:

EccMin, EccMax:	Min and max eccentricity
Q:	Distributed load (Positive in neg. Z-direction)
P1, P2, P3:	Point loads (Positive in neg. Z-direction)
A-dist:	Minimum distance [m] between P1 and P2 and the distance between P1 and P3 or P2 and P3 (alters the truck direction).
B-dist:	Maximum distance [m] between P1 and P2.

Name: Optional name. All characters allowed including space.

Dynamic load syntax

EARTHQUA No FromMode ToMode SpectraNo Ga kx ky kz Dampening

EARTHQUA defines a loadcase where results from an earthquake analysis are to be saved.

No:	Loadcase no.
FromMode ToMode:	Actual eigen modes, step = 1.
SpectraNo:	Spectra number.

For EARTHQUA:

Ga:	Ground acceleration, [m/s^2].
Kx:	Scaling of ground acceleration in global X-dir.
Ky:	Scaling of ground acceleration in global X-dir.
Kz:	Scaling of ground acceleration in global Z-dir. (vertical).
Dampening:	Mechanical damping [o/o]. E.g. use value 5 for 5 % dampening

Name: Optional name. All characters allowed including space.

Dynamic load syntax: Stochastic Wind

DYNWIND No FromMode ToMode SpectraNo Speed Dir Roughness Dampening

DYNWIND defines a loadcase where results from a stochastic wind analysis are to be saved.

No:	Loadcase no.
FromMode ToMode:	Actual eigen modes, step = 1.
SpectraNo:	Spectra number.
Speed:	Wind speed at 10m height [m/s].
Dir:	Wind direction [deg] relative to global X-Axis (in the horizontal plane XY).
Roughness/category:	Terrain roughness or terrain roughness category depending on used spectra. Category if spectra is NS3491-4 type. Category acc. to table 5.1.
Dampening:	Mechanical damping [-]. E.g. use value 0.008 for 8% dampening
Ground level:	Correction to the ground elevation [m].
Name:	Optional name. All characters allowed including space.

NovaFrame includes a stochastic wind analysis for determining the dynamic response of the unsteady flow of the wind.

External displacements syntax

DISPINS Type LoadNo NodeNo Wx Wy Wz Rx Ry Rz Name

Type: 1: Loadcase
2: Loadcombination
LoadNo: Loadcase or loacombination number
NodeNo: Node number
Wx, Wy, Wz: External given displacements in global X-, Y- and Z-direction [m].

Rx, Ry, Rz: External given rotation about global X-, Y- and Z-axis [Radians].
Name: Optional name. All characters allowed including space.

External forces syntax

FORCINS Type LoadNo ElemNo DesSectNo PL PM PN ML MM MN Name

Type: 1: Loadcase
2: Loadcombination
LoadNo: Loadcase or loacombination number
ElemNo: Element number
PL, PM, PN: External given section forces in local L-, M- and N-direction [kN].
ML, MM, MN: External given moments about local L-, M- and N-axis [kNm].
Name: Optional name. All characters allowed including space.

Creep combination syntax

CREEPCMB No LC1 Fac1 LC2 Fac2 LC3 Fac3 Name

CREEPCMB defines a combination of loadcases giving a force level to be used for creep loads.

No: Creep combination no.
LC<n> Fac<n> Loadcase and corresponding factor (Max. 3 pairs on each line).
Name: Optional name. All characters allowed including space.

Creep load syntax

CREEPLC No FromElem ToElem ElemIncr CreepCmb Type A1 A2 A3 Name

CREEPLC defines a loadcase where results from a creep analysis is to be saved.

No: Creep Loadcase no.
FromElem, ToElem and ElemIncr: Actual element numbers.
CreepCmb: Creep combination no to be used.
Type: 1: Calculate automatic:
A1: Age of concrete when first loaded [days]
A2: Start time [days].
A3: End time [days].
2: Value:
A1: Partial creep factor.
Name: Optional name. All characters allowed including space.

5.13 Buckling data

Axial syntax

AXIAL FromElem ToElem ElemIncr Type A1 A2 A3 A4 A5 A6

AXIAL defines an axial force level for the specified elements.

FromElem, ToElem and ElemIncr:	Actual element numbers.
Type:	Type of axial force: 1: Value A1 = axial force [kN]. A2-A6 not in use 2: By loadcases A1 = LC1. A2 = fac LC1 A3 = LC2. A4 = fac LC2 A5 = LC3. A6 = fac LC3

Buckling syntax

BUCKLING FromElem ToElem ElemIncr Euler_L Lateral_L Mvio_L Euler_N Lateral_N Mvio_N

BUCKLING defines euler buckling lengths and lateral buckling lengths to be associated with each element.

FromElem, ToElem and ElemIncr:	Actual element numbers.
Euler_L:	Euler buckling length in L-direction.
Lateral_L	Lateral buckling length in L-direction
Mvio_L:	Lateral buckling, moment factor L-direction
Euler_N:	Euler buckling length in N-direction
Lateral_N:	Lateral buckling length in N-direction
Mvio_N:	Lateral buckling, moment factor N-direction

Member syntax

MEMBER Member no. Type FromElem ToElem ElemIncr “Name”
MEMBER defines a sequence of elements which is one memeber.

Member No.	Actaul member No.
Type	Member type: 1: **COLUMN**, 2: **BEAM**
FromElem, ToElem and ElemIncr:	Actual element numbers.
“Name”	Member Name.

5.14 Models and analysis ascii input

Model syntax

MODEL No UseMod Code E1 E2 dE Name

MODEL inserts a new model definition.

No:	Model number.
UseMod:	Use this existing model to add or delete from Use: –1 if no existing model is to be used Use: 0 if you want to start with the “default all” model.
Code:	Code for what to be done: 1: Add models -1: Delete models 3: Add elements -3: Delete elements
E1, E2 and dE:	Actual model numbers if code is 1 or –1. Actual element numbers if code is 3 or –3;.
Name:	Optional name. All characters allowed including space.

Model modification syntax

MODMODIF No Code From To Step data1 data2 data3 data4 data5 data6

MODMODIF inserts a new modification to a specific model.

No:	Model number.
Code:	Code for what to be done: 1: Node, change Boundary conditions for specified nodes. 2: Node, change master slave connections for specified nodes 10: Elements, change joints for specified elements. 11: Elements, change cross sections associated with specified elements. 20: Elements, change mass by adding mass to specified elements. 21: Elements, change mass by replacing existing mass on specified elements. 30: Elements, change axial force by adding to existing axial definition for specified elements. 31: Elements, change axial force by replacing existing axial definition on specified elements. 32: Elements, change axial force by giving new axial value on specified elements. 33: Elements, change axial force by Loading From Database. 40: Elements, change wind areas by adding wind areas to specified elements. 41: Elements, change wind areas by replacing existing wind areas on specified elements.
data1 -> data6:	See next page:

Code		From	To	Step	Data1	Data2	Data3	Data4	Data5	Data6	Comment
1	Nodes	X	X	X	FixCode						FixCode (1 = fixed, 0 free). 111111; i.e. all are fixed
2	Nodes	X	X	X	Master Node	Incr Master Node					See general decription on master-slave connections
10	Elements	X	X	X	Joint Code End 1	Joint Code End 2					JointCode (1 = free, 0 fixed). 111111 i.e. all are free.
11	Elements	X	X	X	Start section No	Step Section					
20	Elements	X	X	X	Mass Type	Mass Dir.	Value	Value			See description concerning mass input
21	Elements	X	X	X	Mass Type	Mass Dir.	Value	Value			See description concerning mass input
30	Elements	X	X	X	LcNo	Fact	LcNo	Fact	LcNo	Fact	
31	Elements	X	X	X	LcNo	Fact	LcNo	Fact	LcNo	Fact	
32	Elements	X	X	X	Value						
33	Elements										Aplies to all elements with axial definition
40	Elements	X	X	X	Value	Value	Value	Value	Value	Value	See description conc-erning wind-area input
41	Elements	X	X	X	Value	Value	Value	Value	Value	Value	See description conc-erning wind area input

Table defining model modify input.

Calculation group syntax

CALCGRP No AnType LC1 LC2 dLC Name

CALCGRP inserts a new calculation group.

No:	Calculation group number.
AnType:	Type of analysis: 1- Ordinary static analysis 2- Traffic load analysis 2- Creep analysis 4- Eigenmode analysis 6- Earthquake analysis
LC1, LC2, dLC:	For Analysis type = 1; (Ordinary static analysis) LC1 = From loadcase no LC2 = To loadcase no dLC = Step loadcase For Analysis type = 2; (Traffic analysis) LC1 = From traffic load no LC2 = To traffic load no dLC = Step traffic load For Analysis type = 3; (Creep analysis) LC1 = Creep loadcase no LC2 = Not in use dLC = Not in use For Analysis type = 4; (Eigenvalue analysis) LC1 = No of modes to be calculated LC2 = Not in use dLC = Not in use For Analysis type = 6; (Earthquake analysis) LC1 = From Dyn. loadcase no LC2 = To Dyn. loadcase no dLC = Step loadcase
Name:	Optional name. All characters allowed including space.

Analysis syntax

ANALYSIS No ModNo CGNo SetupNo Time Name

ANALYSIS inserts a new analysis.

No:	Analysis number.
ModNo:	Model number to be used in analysis.
CGNo:	Calculation group to be used in analysis.
Setup No:	Currently not in use
Time:	Currently not in use.
Name:	Optional name. All characters allowed including space.

ANSETUP, NO, GEOSTIFF, 0, 0, 0, ”NAME”

	Default	*Unit*	*Description*
NO	0	-	Analysis setup number
GEOSTIFF	0	-	Include geometric stiffness matrix. 0=NO, 1=YES.
NAME			Optional name

5.15 Combination ascii input

ORDCOMB, No, Lstate, Type, LC1, Fac1, LC2, Fac2, LC3, Fac3, Name

	Default	*Unit*	*Description*
No:	-	-	Loadcombination number.
LState:	NA	-	Limit State; 1=ULS, 2=PLS, 3=SLS. (**NA = 0, ULS, PLS, SLS**)
Type:	LC	-	Type of results to be combined; 1 = Loadcases (**LC**) 2 = Loadcombinations (**OC**)
LC<n> Fac<n>	-	-	Loadcase or combination and corresponding factor (Max. 3 pairs on each line).
Name:	-	-	Optional name. All characters allowed including space.

A loadcombination can be defined on as many lines you want by using the same comb. number.
Each line must consist of only previously defined combinations or loadcases, but a combination can consist of both (defined on several lines).
You only have to include as many loadcase/comb.-factor pairs on each line as you need.

SORTCOMB, No, Lstate, Type, Method, LC1, Fac1, LC2, Fac2, LC3, Fac3, Name

	Default	*Unit*	*Description*
No:	-	-	Sort combination number.
LState:	NA	-	Limit State; 1=ULS, 2=PLS, 3=SLS, (**NA = 0, ULS, PLS, SLS**)
Type:	LC	-	Type of results to be combined; 1 = Loadcases, (**LC**) 2 = Ordinary loadcombinations, (**OC**) 3 = Traffic Loads, (**TL**) 4 = Sorted combinations, (**SC**)
Method:	0	-	Method for combining: 0 = Add all. (Default) (**ADD**) 1 = Only the most unfavourable for each component. (**WORST**) 2 = As for 1 but the sign is positive and negative. (**SWORST**) 3 = Adds if unfavorable. (**ADDIF**) 4 = Adds if unfavorable, but the sign is positive or negative. (**SADDIF**) 10 = Root mean square summation. (**RMS**) 13 = Adds dynamic wind results with unfavorable sign for both governing force component and accompanying force components. (**SDYNW**). NOTE: The dynamic wind load case must be the first load case (LC1) specified in the input command line for method=13 for this method to work properly.
LC<n> Fac<n>	-	-	Loadcase/ Ord. combination/Traffic load or Sorted combination and corresponding factor. (Max. 3 pairs on each line).
Name:	-	-	Optional name. All characters allowed including space.

A Sorted load combination can be defined on as many lines you want by using the same Sort. comb. number.
Each line must consist of only previously defined Loadcase, Ord. comb.Traffic load or Sorted comb. , but a combination can consist of both (defined on several lines).
You only have to include as many Load./Comb.-factor pairs on each line as you need.
The method can not change within a sorted combination.

SORTLINE, NO, LIM, FromEl, LastEl, StepEl, FromSC, LastSC, StepSC, CalcCode, ”Name”

	Default	*Unit*	*Description*
NO	0	-	Sorted combination line number
LIM	ULS	-	Limit State; 1=ULS, 2=PLS, 3=SLS, (**ULS, PLS, SLS**)
FromEL	0	-	First Element
ToEL	0	-	Last Element.
StepEL	0	-	Step Element
FromSC:	0	-	First sorted combination.
LastSC:	0	-	Last sorted combination.
StepSC:	0	-	Step sorted comb.
CalcCode:	111111	-	Which force componets to calculate: Example1: (110101 = will calc: max/min of PL, PM, ML and MN.) Example2: (111111 = will calc: all max/min components.)

Name:	-	-	Optional name. All characters allowed including space.

SORTLINE Calculates the worst sorted combination for each section force component, for each design section for the given elements. Only the force components specified by CalcCode will be available for listing.

BUCKLDES, TYPE, CMB NO, BYSECT, BYELEM, BYMODEL, MODEL NO

	Default	*Unit*	*Description*
TYPE	1	-	2: Ord Comb, 4:Sort Comb line
CMB NO	0	-	Actual Combination
BYSECT	1.	-	By Section DESPAR
BYELEM	1.	-	By element
BYMODEL	1.	-	By Model
MODEL NO	-1	-	Only in use when BYMODEL = 1. Default **ALL** = -1

BUCKLDES Defines the method for calculating bucling lengths for a loadcombination or a sorted combination line.

5.16 Design calculation input

DCSETUP, ID, SECTTYPE, COMBTYPE, ”NAME”

	Default	*Unit*	*Description*
ID	1	-	ID for design calculation setup
SECTTYPE	0	-	Section code: 0 = by element, 1 = by reference line
COMBTYPE	1	-	Combination code: 1 = ordinary load combinations, 2 = sorted combination lines
”NAME”		-	Optional setup name. All characters allowed including space

DCSETUP defines a new design calculation setup.

DCSECT, ID, FROM, TO, FROM, TO

	Default	*Unit*	*Description*
ID	1	-	ID for actual design calculation setup
Section specification code = 0 (by element)			
FROM	**ALL**	-	From element number (**ALL**)
TO	-	-	To element number
FROM	**ALL**	-	From X/L (**ALL**)
TO	-	-	To X/L
Section specification code = 1 (by reference line)			
FROM	**ALL**	-	From reference line number (**ALL**)
TO	-	-	To reference line number
FROM	**ALL**	-	From station number (**ALL**)
TO	-	-	To station number

DCCOMB, ID, LIM, FROM, TO

	Default	*Unit*	*Description*
ID	1	-	ID for actual design calculation setup
LIM	**ALL**	-	Limit state (**ALL, ULS, PLS, SLS**)
Combination code = 1 (ordinary combination)			
FROM	**ALL**	-	From combination number (**ALL**)
TO	-	-	To combination number
Combination code = 2 (sorted combination lines)			
FROM	**ALL**	-	From sorted combination line number (**ALL**)
TO	-	-	To sorted combination line number

DCOPT, ID, MAT, OPT1, OPT2, …, OPT14

	Default	*Unit*	*Description*
ID	1	-	ID for design calculation setup
MAT	**CONCR**	-	Material type. Concrete is the only valid option: 0 = concrete (**CONCR**)
OPT1	1	-	Include shear forces
OPT2	1	-	Include torsion moments
OPT3	0	-	Sort max/min forces (not applicable for sorted combination lines)
OPT4	1	-	Include slenderness effects
OPT5	1	-	Lock all reinforcement amounts at current level
OPT6	0	-	Use crack widths as a design criteria
OPT7	0	-	Allow stepping of reinforcement in compression zone
OPT8	0	-	Reset all reinforcement amounts to first entry in table
OPT9	1	-	Lock all shear reinforcement amounts at current level
OPT10	0	-	Use shear reinforcement spacing as a design criteria (torsion)
OPT11	0	-	Reset all shear reinforcement amounts to first entry in table
OPT12	1	-	Include calculated tendons
OPT13	1	-	Include given tendons
OPT14	1	-	Include long term losses

Use DCOPT to switch on/off calculation options for a specified design calculation setup. 0 = off, 1 = on

DCSOLVE, ID

	Default	*Unit*	*Description*
ID	1	-	ID for design calculation setup to run. (**ALL**)

DCSOLVE runs the specified design calculation(s)

DCCONV, ITER, CRIN, CRIMX, CRIMY, INCR

	Default	*Unit*	*Description*
ITER	150	-	Maximum number of equilibrium iterations.
CRIN	0.001	-	Convergence acceptance parameter for axial force N. Smaller value gives a stricter convergence criterion. Recommended value is between 0.1 and 1.0.
CRIMX	0.001	-	Convergence acceptance parameter for moment Mx. Smaller value gives a stricter convergence criterion. Recommended value is between 0.1 and 1.0.
CRIMY	0.001	-	Convergence acceptance parameter for moment My. Smaller value gives a stricter convergence criterion. Recommended value is between 0.1 and 1.0.
INCR	**OFF**	-	Use incremental loading (**ON**, **OFF**).

DCCONV sets the convergence options for concrete design calculation

5.17 Capacity chart input

CHART, ID, TYPE, RFL, PRO, LIMIT, EPSOPT; EPSC; EPSS; EPST ”NAME”

	Default	*Unit*	*Description*
ID	1	-	ID for section point (unique within actual cross section)
TYPE	1	-	Chart type. Available types are:
			1 = Moment-axial force capacity chart (MN)
			2 = Moment-curvature chart (MC)
			3 = Biaxial moment capacity chart (MM)
			4 = Moment-concrete strain chart (ME)
RFL	0	-	Reference line number for cross section
PRO	1	m	Station number (profile number) for cross section
LIMIT	1	-	Limit State; 1=ULS, 2=PLS, 3=SLS
EPSOPT	1	-	Option for selecting strain limits for chart
			0 = use EPSC, EPSS and EPST values
			1 = use strain limits as defined in parameter sets for actual cross section
EPSC	-3.50	o/oo	Strain limit for concrete
EPSS	10.0	o/oo	Strain limit for reinforcement
EPST	10.0	o/oo	Strain limit for tendons
”NAME”		-	Optional chart name. All characters allowed including space

CHART creates a new capacity chart, including a default chart line.

CHARTLN, ID, VAL1, VAL2, VAL3, SMOOTH

	Default	*Unit*	*Description*			
ID	1	-	ID for catual capacity chart			
		MN-chart	MC-chart	MM-chart	ME-chart	
VAL1	0	°	Ø	Ø	Øfrom	Ø
VAL2	360	°	-	-	Øto	-
VAL3	0	kN	-	N	N	N
SMOOTH	3	-	Code for number of points calculated for each chart line			
			(1 = 15, 2 = 25, 3 = 45, 4 = 95)			
”NAME”		-	Optional setup name. All characters allowed including space			

CHARTLN defines a new chart line for an existing capacity chart

5.18 Miscellaneous ascii input

CLEAR, OPT

	Default	*Unit*	*Description*	
OPT	**ALL**	-	Specify what data to delete. -1 = all, 1 = calculated results	(**ALL, RES**)

CLEAR clears current data in memory (not on file)

STOP

Stops execution of the rest of the input file

SAVE, FILENAME

	Default	*Unit*	*Description*
FILENAME	**-**	-	Saves model to databs

SAVE saves model data to specified database file. The database file may be new or existing.
Complete file name including path (e.g.“C:\Arkiv\Model_1”) or just generic file name (“Model_1”) may be given. If just generic file name is given the database will be saved to current working folder.

CALCCOMB, TYPE

	Default	*Unit*	*Description*		
TYPE	1	-	Type of load combinations to calculate.	-1 = all	**(ALL)**
				1 = all ordinary load combinations	
				2 = all sorted load combinations	
				3 = all sorted combination lines	

CALCCOMB recalculates all load combinations of the specified type

SOLVE, TYPE, NMODE

	Default	*Unit*	*Description*		
TYPE	**STATIC**	-	Specify type of solution .	-1 = all	**(ALL)**
				1 = Static analysis	(**STATIC**)
				2 = Trafic analysis	(**TRAFFIC**)
				3 Creep analysis	(**CREEP**)
				4 = Eigen mode calculation	(**EIGEN**)
				5 = Stochastic wind analysis	(**DYNWIND**)
				6 = Earth quake analysis	(**EQUAKE**)
				7 = Buckling analysis	(**BUCKL**)
NMODE	1	-	Number of modes to calculate if solve type = 4 or 7 (calculate eigen values)		

SOLVE starts solution of frame analysis. If several analysis are defined the ANSOLVE command must be used in stead

ANSOLVE, FROM, TO, STEP

	Default	*Unit*	*Description*		
FROM	**-**	-	From analysis number	-1 = all	**(ALL)**
TO	**-**	-	To analysis number		
STEP	**-**	-	Step		

ANSOLVE starts solution of a specified frame analysis in a multippel analysis.

INPUT, ”FILENAME”

Default	*Unit*	*Description*

”FILENAME” **-** - Input file name (whole path including extension, f.ex: ”D:\Prosjekt\Test.inp”)

INPUT reads ascii input from the specified file name

OUTPUT, CAT, TYPE, VAL1, VAL2, … , ”FILENAME”

	Default	*Unit*	*Description*		
CAT	**-**	-	Output listing category	1 = model data 2 = analysis results 3 = concrete design results	**(MODEL)** **(ANRES)** **(CONRES)**
Model data listing(category = 1)					
TYPE	**-**	-	Type of model data to list	201 = nodes 203 = boundary conditions 204 = master-slaves 205 = elements, etc…	
VAL1	0	-	From model item number	If = -1 **(ALL**), VAL2 is skipped	
VAL2	0	-	To model item number		
Analysis result listing (category = 2)					
TYPE	0	-	Type of analysis results:	0 = displacements 1 = section forces 2 = support reactions 3 = equilibriium check 4 = eigen modes 5 = buckling modes 6 = accelerations 7 = steel UR 8 = elastic stress 9 = steel stress	
VAL1	0	-	Load combination type:	1 = Loadcases, 2 = Ordinary loadcombinations, 3 = Traffic Loads, 4 = Sorted combination Line,	**(LC)** **(OC)** **(TL)** **(SCL)**
VAL2	0	-	From element/node number	If = -1 **(ALL**), VAL3 is skipped	
VAL3	0	-	To element/node number		
VAL4	0	-	From combination number	If = -1 **(ALL**), VAL5 is skipped	
VAL5	0	-	To combination number		
VAL6	0	-	Element number	If Displacements are selected then the displacement will be transformed to the local axes of this element.	
VAL7	0	-	Not in use!		
VAL8	0	-	Not in use!		
VAL9	0	-	0 – New file, 1 - Append to existing file.		
Concrete design result listing (category = 3)					
TYPE	0	-	Type of analysis results:	0 = N/M sumary 1 = V/T summary 2 = additional moments due to slenderness 3 = slenderness results in detail 4 = concrete strain, stress etc. 5 = reinforcement strain, stress etc. 6 = tendon strain, stress etc. 7 = shear reinforcement results 8 = sub area results 9 = shear area results 10 = torsion area results 11 = section forces	
VAL1	0	-	Load combination type:	2 = Ordinary loadcombinations, 4 = Sorted combination lines,	**(OC)** **(SCL)**
VAL2	0	-	Present as UR or value	0 = UR 1 = Value	
VAL3	0	-	Compress option	0 = no compression 1 = Compress combinations (max each design section) 2 = Compress sections (max each combination)	
VAL4	0	-	Reference line	Specity reference line to list, (use VAL5 and VAL6 to enter station number interval) -1 = list results pr element, (use VAL5 and VAL6 to enter	

				element number interval)
VAL5	0	-	From element/station no	NB! Interval must be specified!
VAL6	0	-	To element/station no.	
VAL7	0	-	From combination number	NB! Interval must be specified!
VAL8	0	-	To combination number	
VAL9	0	-	0 – New file, 1 - Append to existing file.	
”FILENAME”	-	-	Output file name	

OUTPUT writes listings of model data, analysis results or concrete design results to a text file.

DELETE, ITEM, FROM, TO, STEP

	Default	*Unit*	*Description*		
ITEM	-	-	Type of items to delete. Valid items are:	1 = Nodes	**(NODE)**
				2 = Nodes	**(ELEM)**
FROM	-	-	From item number	-1 = all	**(ALL)**
TO	-	-	To item number		
STEP	-	-	Step		

DELETE commands deletes the specified items

IF, VALUE, TEST, LIMIT

	Default	*Unit*	*Description*		
VALUE	0	-	Value to compare with LIMIT		
TEST	1	-	Boolean test (true or false)	Equal to	**(EQ)**
				Less than	**(LT)**
				Greater than	**(GT)**
				Less or equal	**(LE)**
				Greater or equal	**(GE)**
			Test acceptance limit is 1E-10		
LIMIT	0	-	Limit for boolean test		

IF is used in combination with the commands ELSE, ELSEIF and ENDIF to select which input lines that are to be executed. If the IF expression is TRUE all following command lines will be executed until next ENDIF command. Otherwise the execution will jump to the next ELSEIF, ELSE or ENNDIF command. IF blocks can be placed within other IF-blocks or DO-loops.

Example:

```
IF,R,GT,5.7
   ANG = 23.2
ELSEIF,R,LE,4.8
ANG = 19.7
   ELSE
ANG = 21.0
   ENDIF
```

ELSEIF, VALUE, TEST, LIMIT

	Default	*Unit*	*Description*		
VALUE	0	-	Value to compare with LIMIT		
TEST	1	-	Boolean test	Equal to	**(EQ)**
				Less than	**(LT)**
				Greater than	**(GT)**
				Less or equal	**(LE)**
				Greater or equal	**(GE)**
			Acceptance accuracy is 1E-10		
LIMIT	0	-	Limit for		

ELSEIF is used in combination with the commands IF, ELSE and ENDIF to select which input lines that are to be executed. If one of the previous IF or ELSEIF expression are true, the ELSEIF test is skipped and all following command lines will be skipped until next ENDIF command. An IF block may include any number of ELSEIF tests.

ELSE

ELSE is used in combination with the commands IF, ELSEIF and ENDIF to select which input lines that are to be executed. If both corresponding IF and ELSEIF command expressions are false, the input lines between ELSE and ENDIF will always be executed.

ENDIF

ENDIF marks the end of an IF-block. The command is used in combination with IF, ELSE and ELSEIF.

DO, ALIAS, FROM, TO, STEP

	Default	*Unit*	*Description*
ALIAS	-	-	Any uppercase alfanumeric name (alias). This alias will have the value FROM in the first loop and then increment with the value STEP with each loop
FROM	0	-	Start value (may be a real value, f.ex 1.2)
TO	1	-	End value (may be a real value, f.ex 7.0)
STEP	1	-	Step value (may be a real value, f.ex 0.2)

DO is used in combination with the commands ENDO to create a loop of input commands. The loop will run until the value of the DO variable (alias) is greater than the value of TO. IF blocks can be placed within other IF-blocks or DO-loops.

Example:

```
DO,N,0,45,1
   ANG = 2*I
   NODE, N+1, N+1, 1,, 5.0*SIN(ANG), 5.0*COS(ANG)
ENDDO
```

ENDDO

ENDDO marks the end of a DO-loop. The command is used in combination with DO.

5.19 Commands with special use

This section describes commands or use of commands which is only available when running batch.

Special use of the XSECT command which allow the user to copy a cross section from an existing section. The command will copy everything including reinforcement and tendons:

XSECT, RFL, PRO, COPY, FROMRFLNO, FROMPRONO, Not in use, Not in Use, ”NAME”

FROMRFLNO -The ref. line from which to copy.
FROMPRONO -The profile id of the section to copy.

In general NovaFrame does not calculate displacements for traffic loads and sorted combinations. The user can however pin point which nodes that displacement are to be calculated for. Please note that the displacements are calculated based on the sorting of the forces and not of the displacements**.**

TRAFDISP, ELNO, X/L, NODENO

	Default	*Unit*	*Description*
ELNO	**-**	-	Element number
X/L	**-**	-	Relative position of design section
NODENO	**-*)**	-	Optional NodeNo where displacement are calculated

TRAFDISP, node deformations will be calculated for forces at given design section in node no. Required to get sorted displacements corresponding to sorted section forces.

*) NODENO is optional. Default behaviour is:
X/L <= 0.500, Selects the Node in element end 1.
X/L > 0.500, Selects the Node in element end 2.

5.20 Obsolete input commands

SELEINS
SELEDE

```
// Codes for defining listing type
LIST_CONCR_MATPAR          101
LIST_REINF_MATPAR          102
LIST_PREST_MATPAR          103
LIST_STEEL_MATPAR          104
LIST_CONCR_DESPAR          111
LIST_REINF_DESPAR          112
LIST_PREST_DESPAR          113
LIST_STEEL_DESPAR          114
LIST_SECTION_DESPAR        121

LIST_NODE                  201
LIST_BOUNDARY              203
LIST_MASTER                204
LIST_ELEMENT               210
LIST_JOINTS                213
LIST_ELEMENT_STIFF         214
LIST_MASS                  230
LIST_WINDAREA              231
LIST_BUCKLING_DATA         241

LIST_LOAD                  301
LIST_LOADCASES             302
LIST_TRAFFIC_LINES         310
LIST_TRAFFIC_LINE_POS      311
LIST_TRAFFIC_LOADS         312
LIST_DYN_LOAD              313
LIST_CREEP_LOAD            320
LIST_CREEP_LOAD_CALCDATA   321
LIST_CREEP_COMB            322
LIST_AXIAL_RES             330

LIST_REFLINE               401
LIST_REFLINE_GEOM          402
LIST_SECTION_GEOM          403
LIST_SECTION_GEOM_DETAIL   404
LIST_REFLINE_CALC_GEOM     405
LIST_REFLINE_COLUMN        406
LIST_NYLP_GEOM             410
LIST_REFLINE_SECT_DATA     420
LIST_REFLINE_SECT_GEOM     421
LIST_REFLINE_SECT_PARAM    422
LIST_REFLINE_SECT_CONNECT  423
LIST_REFLINE_SECT_XYZ      424

LIST_MODEL                 501
LIST_MODEL_ALL_PROPERTIES  502
LIST_MODEL_ELEM_RES        503
LIST_MODEL_MODIFICATIONS   504
LIST_MODEL_EIGENMODES      505
LIST_CALC_GROUPS           510
LIST_ANALYSES              511
LIST_CONSEQ                512

LIST_ORD_COMB              601
LIST_SORT_COMB             602
LIST_SORT_COMB_LINE        603
LIST_COMB_TRACER           604

LIST_COORDSYS              701
LIST_FORCE                 702
LIST_RUN_STATISTICS        710
LIST_DEFINED_SPECTRAS      711
LIST_EQUILIBRIUM_CHECK     712
LIST_MEMBERS               720
LIST_MODAL_MASS            721
LIST_PARTISIPATE_MASS      722

LIST_CHANGES               801
LIST_DATABASE_STATUS       802
LIST_TEST                  803
LIST_SAVE_REPORT           804
LIST_DOC_STATUS            805
LIST_START_COMB_TRACER     806

LIST_TENDONS               1101
LIST_TENDON_LOSSES         1102
LIST_TENDON_GEOM           1103
LIST_TENDON_AMOUNT         1104
LIST_TENDON_GEOM_DETAIL    1105
LIST_PCSPENN_INPUT         1106
LIST_TENDON_DESIGN         1107
LIST_ELAST_STRESS_CALC     1108
LIST_TENDON_LOSSES_DETAIL  1109
LIST_TENDON_ELONGATION     1110
LIST_TENDON_POSITION       1111   //Distance from bot. scaffolding
```