ČERVENKA CONSULTING
Červenka Consulting s.r.o.
Na Hrebenkach 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

ATENA Program Documentation
Part 6

ATENA Input File Format

Written by
Jan Červenka and **Libor Jendele**

Prague, September 1, 2020

Ackowledgements:

The software was developed with partial support of Eurostars funding program.

The following models were developed with the financial support of TA ČR: CC3DNonLinCementitious2HPRFC, CC3DNonLinCementitious2FRC, CC3DNonLinCementitious2SHCC, CCModelGeneral, CCTransportMaterial

Durability Analysis module has been developed during the project TA04031458, „Software for prediction and modelling of durability and safety of transportation structures" funded by TA ČR

ATENA3DPrint module has been developed during the project TF04000051, "digiCON2 – Simulation Service for Digital Concrete Production" funded by TA ČR.

ATENA modul CeSTaR – for modelling of prefabricated reinforced concrete segments with multi-spiral reinforcement was developed under the project TF05000040 „CeSTaR - Computer simulation and experimental validation - complex service for flexible and efficient design of pre-cast concrete" funded by TA ČR.

CONTENTS

1 INTRODUCTION AND SCOPE OF THE DOCUMENT

The program ATENA is a general-purpose finite element code with many special features for non-linear analysis of plain and reinforced concrete structures.

This document serves as a manual describing the syntax and format of ATENA input commands in its input file. This command file is often called also input file, and it is used to define finite element model, to specify the loading history and to activate the finite element non-linear analysis.

2 PROGRAM EXECUTION

There are several methods how to execute the finite element module ATENA. The heart of the analysis module is contained in a dynamically linked library ATENADLL.DLL. This module can be utilized either via its COM object interface CCCoAtena or from the command console by executing either AtenaConsole.exe or AtenaWin.exe or ATENAStudio.exe program. The CCCoAtena is used by AtenaGUI graphical pre and postprocessor and its use is described in a separate part of ATENA documentation. Here, the starting the analysis using AtenaConsole, AtenaWin, and ATENAStudio executables is described. The programs are executed as follows:

AtenaConsole [/D *path*] [/P] [/M *module_name*]
[/O] [*input_file*] [*output_file*] [*message_file*] [*error_file*]
[/reset_desktop] [/translate_ids]
[/extend_int_output_width] [/extend_real_output_width]
[/catch_fp_instructs] [/demo_mode]
[/silent] [/num_threads=*n*] [/min_chunk_size_per_thread=*m*]

AtenaWin [/D *path*] [/M *module_name*]
[/O] [*input_file*] [*output_file*] [*message_file*] [*error_file*]
[/translate_ids] [/extend_int_output_width] [/extend_real_output_width]
[/catch_fp_instructs] [/demo_mode]
[/silent] [/batch_execute] [/execute] [/rtf] [/inbuf_size=*i*] [/outbuf_size=*j*]
[/num_threads=*n*] [/min_chunk_size_per_thread=*m*]
[/num_unused_threads=*m*]

ATENAStudio [/D *path*] [/M *module_name*]
[/O] [/inp *input_file*]
[/extend_int_output_width] [/extend_real_output_width]
[/catch_fp_instructs] [/demo_mode]

[/execute] [/threads *n*]

[*input_file* | *project_file*]

AtenaConsole front-end is aimed for batch analyses. Hence, it works only with input and output files, produces no graphics and does not need any user interaction. On the other hand AtenaWin is a windows based application. On start it creates an editable window for each of ATENA's window. The user can use these windows to edit content of the files, inspect ATENA's output during the analysis etc. Of course, similar windows can be used for editing any other text file. It also provides graphical windows post processing and windows for 2D plots, which are useful for example for assessing load-displacement diagram of analyzed structure. Note that all windows in AtenaWin are updated already during the analysis.

In the above the following notation was used:

/D path = specifies path to the working directory where input and output files will be stored.

/P = this option forces the program to request manual specification of input and output files.

/M *module_name* = name of main DLL library used for execution. By default, it is assumed *CCStructures*. The *CCStructuresCreep* DLL is needed for creep analysis.

/O = specifies overwrite flag for *output_file, message_file* and *error_file* files. This means that during execution, (or re-execution within AtenaWin) the files are created or overwritten. By default the files are appended, i.e. output of the new analysis is added at the end of the files.

input_file = name of a file with Atena input commands. If not specified, standard input from keyboard is assumed.

output_file = name of a file for Atena output. If *output_file* doesn't exist, it is created. Otherwise it is appended. If *output_file* is not specified in the command line, then standard output to the screen is assumed.

message_file = name of a file for Atena message output. The message file contains compact information on Atena execution as for instance: a log of the execution start and end, convergence performances, severe warning and error messages during execution etc. If *message_file* doesn't exits, it is created. Otherwise it is appended. If *message_file* is not specified in the command line, then standard output to the screen is assumed.

error_file = name of a file for Atena error output. The error file contains full information on Atena execution as for instance: a log of the execution start and end, convergence performances, all warning and error messages during execution (incl. their place of invocation) etc. If *error_file* doesn't exits, it is created. Otherwise it is appended. If *error_file* is not specified in the command line, then standard output to the screen is assumed.

[/translate_ids] = this option is only for internal use for debugging. Don't use it.

[/extend_int_output_width] [/extend_real_output_width] = double number of digits used to output integer or real numbers, respectively.

[/catch_fp_instructs] = flag to catch, (i.e. unmask) floating point exceptions during the execution. Upon occurrence of such exception it will get caught, reported and the execution will be terminated. By default, floating point exceptions are ignored.

[/demo-mode] = flag for trial execution. All features are available in trial mode, however, there apply some restrictions towards size of the analyzed problem.

[/batch_execute] = option which forces AtenaWin automatically execute the given problem without any user intervention. After the execution all output data are saved and AtenaWin gets terminated. Use this option for batch execution.

[/execute] = option which forces AtenaWin automatically execute the given problem without any user intervention. After the execution the AtenaWin session remains running, thereby enabling a subsequent interactive postprocessing

[/silent] = flag that forces AtenaWin to output eventual error messages into message_file and error_file. By default, they are output to a message box on the screen. Use this option for batch execution.

[/num_threads=*n*] [/threads *n*] = use *n* threads during the execution. By default only a single processor core is used, i.e., sequential processing.

[/num_unused_threads=*m*]= same as the above but Atena will use number of processor's available threads minus *m*. The parameter [/num_threads=*n*] has higher priority.

[/min_chunk_size_per_thread=*m*][1] = chunk size for dynamic schedule, =0 for static load distribution; default=0

[/inp] = precedes the Input File name. ATENA Studio derives the .out, .msg, and .err filenames from the .inp filename by replacing the extension.

[input_file | project_file] = opens the Input file (.inp) or the Project file (.ccs).

If the Project file is given, it opens the existing project.

If the Input file is given there are two possibilities. If the directory, where Input file is located, contains one or more project files, which link on this Input file, ATENA Studio allows to choose, whether to open one of corresponding projects or create new one. Otherwise it creates a new project.

Table 1: Environmental variables for AtenaConsole, AtenaWin[2], and ATENA Studio execution

Command	
AtenaConsole 32-bit execution	
%AtenaConsole%	Basic AtenaConsole command, by default executes statics module
%AtenaConsoleD%	AtenaConsole execution for dynamics analysis
%AtenaConsoleC%	AtenaConsole execution for creep analysis
%AtenaConsoleT%	AtenaConsole execution for transport analysis
AtenaConsole 64-bit execution	
%AtenaConsole64%	Basic AtenaConsole 64-bit execution, by default

[1] Supported since version 5.7.0; replaces the option `NUM_ITERS_PER_THREAD` older versions

[2] AtenaWin program can be used for runtime visualization of the analysis progress and postprocessing. Starting from ATENA version 5, AtenaWin program is replaced by ATENA Studio. Please, check the program documentation of these programs for more details.

	executes statics module
%AtenaConsoleD64%	AtenaConsole 64-bit execution for dynamics analysis
%AtenaConsoleC64%	AtenaConsole 64-bit execution for creep analysis
%AtenaConsoleT64%	AtenaConsole 64-bit execution for transport analysis
AtenaWin 32-bit execution	
%AtenaWin%	Basic AtenaWin command, by default executes statics module
%AtenaWinD%	AtenaWin execution for dynamics analysis
%AtenaWinC%	AtenaWin execution for Creep analysis
%AtenaWinT%	AtenaWin execution for Transport analysis
AtenaWin 64-bit execution	
%AtenaWin64%	Basic AtenaWin command for 64-bit execution, by default executes statics analysis
%AtenaWinD64%	AtenaWin 64-bit execution for dynamics analysis
%AtenaWinC64%	AtenaWin 64-bit execution for creep analysis
%AtenaWinT64%	AtenaWin 64-bit execution for transport analysis
ATENA Studio 32-bit execution	
%AtenaStudio%	Start 32-bit ATENA Studio, the analysis type can be selected in a dialog
ATENA Studio 64-bit execution	
%AtenaStudio64%	Start 64-bit ATENA Studio, the analysis type can be selected in a dialog

3 INPUT COMMANDS

3.1 Changes of Input Commands Syntax in the New Version

With few exceptions, the current version of ATENA uses the same syntax of input commands the previous version did. The modified input command relates to

- &OUTPUT commands,
  The keywords for locations changed as follows

The old keyword	The new keyword
ATTRIBUTE	OUTPUT_DATA
LOAD	LOAD_CASES
ELEMENT	ELEMENTS
ELEMENT IP	ELEMENT_IPS
NODE	NODES
ELEMENT NODE	ELEMENT_NODES
LOAD	LOAD_CASES
MATERIAL	MATERIALS
GEOMETRY	GEOMETRIES
ELEMENT TYPE	ELEMENT_TYPES

  There are available several new or renamed output data, see the Table 129.

- &CREEP_ANALYSIS_PARAMS commands
  Creep and shrinkage analysis is a new analysis type not supported in the previous versions. Therefore, all related commands are new. Please refer to the appropriate section of this manual for more details. Note that some more creep commands are available in &CREEP_MATERIAL, &RETARDATION_TIMES, &HISTORY_IMPORT and analysis step definition &CREEP_STEP_DEFINITION

- &PREPROCESS commands.
  The preprocess commands can be used to easy FE model mesh generation by use of the T3D generator and for generation of embedded reinforcement bars.

- Boundary conditions, i.e. specification of concentrated loads and supports can now be defined via &SELECTION and modified &LOAD_PLACE and &LOAD_VALUE commands. List of loaded/supported nodes also can be automatically generated by T3D generator using ELEMPROP "*list_name*" and NODEPROP "*listname*" subcommands of T3D commands REGION, VERTEX, SURFACE etc.…

- &CCStructuresTransport commands, i.e. commands for analysis of moisture and humidity transport within structures. Although most input commands for temperature and humidity transport are the same as those for the other engineering modules, there are some exceptions. This section is devoted to the commands that are available only for the transport analysis.

- &CCStructuresDynamic module related commands, i.e. commands for dynamic analysis of structures including eigenvalues and eigenvectors analysis. It inherits also a few commands from creep and transport analysis.

3.2 General Rules

The following lines introduce general rules for composing Atena and Atena Pollute Transport input commands and syntax that is used to describe them.

- Each command has form of a sentence (not terminated by dot). The command consists of several tokens (or words) separated by one or more spaces or CR/LF characters.
- Tokens written in upper case letters with the 1st character being alphabetic denote keywords, e.g. DELETE.
- Tokens starting with & character refer to a more complicated input structures described elsewhere in the manual. They are not ATENA commands; rather they are to be replaced with an input structure they refer to. This syntax is used to simplify description of complicated commands. Cross-references to these input structures are indicated by & character.
- Tokens written in lower case *italic* letters denote value parameters, i.e. nodal coordinate. If name of such a token is enclosed in quotes, a string value (in quotes) is expected, i.e. "*file name*", otherwise numerical value is expected. Numerical tokens starting with *n* or *i* indicate integer values, whilst parameters starting with *x* denote real value.
- Interpretation of Atena keywords is case insensitive.
- Optional parameter (either a keyword or value) is enclosed in square brackets [].
- If an input token has to be one of several keywords and/or values, then all its admissible values are enlisted in curled brackets {} separated by vertical bar |, i.e. { NODE | ELEMENT | LOAD } . Default choice is underlined, (if it exists).
- Right curled bracket with "plus" subscript indicates that Atena input processor accepts one or more tokens from the above list of choice, $\{ \{ \text{NODE} \mid \text{ELEMENT} \mid \text{LOAD} \}_{+}$.
- Right curled bracket with integer subscript *n* indicates that Atena input processor requires just *n* times a token from the above list of choice, $\{ x \}_3$ means input of 3 real numbers .
- Features, which are currently not supported are denoted with ♥.
- The commands between two EXECUTE keywords can appear in any order. In case of multiple definition, the program accepts always the last definition before the EXECUTE command.
- The comment syntax corresponds to the C++ style. There are two comment types:
  - C-style comment, where the comment is started by "/*" (i.e. slash and star) characters and ended by "*/" (i.e. star and slash).
  - C++ style where it is assumed that everything following "//" (i.e. two slash) characters up to the end of line is considered to be a comment.

3.3 Main Input Commands

&MAIN_COMMANDS:
{ &TASK | &JOINT | &MATERIAL | &GEOMETRY | &ELEMENT | &DELETE |
&MACRO_DELETE | &FUNCTION | &INPUT | &LOAD | &LOCAL |
&MESSAGE | &ERROR | &OUTPUT | &RESTORE | &SET | &STEP |
&STORE | &UNITS | &T3D_SPEC | &DLL_NAME | &EMPTY |
&RETARDATION_TIMES | &ALLOCATE_NODAL_DOFS |
&HISTORY_IMPORT | &PREPROCESS | &TERMINATE | &BREAK |
&NODAL_IMPERFECTIONS | &SELECTION | &MACRO_JOINT |
&MACRO_ELEMENT | *module_name* | &EIGENVECTORS |
&PUSHOVER_ANALYSIS | &STATIC_INITIAL_CONDITIONS | &JUMP |
&LABEL | &DEBUG | &EVALUATE | &PYTHON | &MODULE |
&THREADING | &UPDATE_ELEMENT_CONSTRUCT_TIME |
&TRANSFORM_COORDS }

The above &MAIN_COMMANDS input structure represent general ATENA input command. Each &ENTRY represents a group of input command that is described later. Most of the present commands are used to define some entity for description of your finite element model. The exception to that is &STEP command that contains a keyword EXECUTE. Processing of this keyword forces ATENA to carry on the analysis.

The ATENA input commands can appear in any order in the input file, only the &TASK command has to be the 1st one, as it specifies dimension for many other entities, such as joint coordinates. It is possible to reference an entity prior it was even defined. Although it is not recommended, ATENA does accept that, but don't forget to define them later! If you do, ATENA will not issue any error or warning messages, as the program assumes default values for most of the undefined entities. Such an error remains usually untapped until issuing the STEP … EXECUTE command.

Note that it is possible at any time to modify the finite element model by adding, modifying or removing various modeling entities. The STEP … EXECUTE command uses always current settings of the finite element model.

Table 2: Main input commands

Keyword/Command	Keyword/Command description
&TASK	Define analysis identification.
&JOINT	Input joint parameters, such as coordinates etc.
&MATERIAL	Definition of material types.
&GEOMETRY	Definition of used geometry.
&ELEMENT	Element properties definition.
&DELETE	Delete various entities.
&FUNCTION	X-Y relationship definition.
&INPUT	Input redirection.
&LOAD	Loads and load cases definitions.
&LOCAL	Set joints using local coordinate system.

&MESSAGE	Message output redirection.
&OUTPUT	Output input data and results.
&RESTORE	Restore a previously saved analysis.
&SET	Miscellaneous settings.
&STEP	Step definitions and analyses.
&STORE	Store current analysis.
&UNITS	Sets program units.
&DLL_NAME	Name of dynamic link library, by which processor the following commands should be processed. Currently DLL_NAME is { CCFEMODEL | CCSTRUCTURES | CCSTRUCTURES_CREEP}.
&EMPTY	Forces the current DLL command processor to return to its "root" position, i.e. its main commands level.
&RETARDATION_TIMES	Generate retardation times.
&HISTORY_IMPORT	Import humidity and temperature history for creep analysis.
&TERMINATE	Immediately terminates the input commands stream
&SELECTION	Define list of entities, e.g. nodes, that are later used in another command, e.g. definition of boundary conditions.
&STATIC_INITIAL_CONDITIONS	Set structural initial conditions at nodes, such as reference tepmeratures.
module_name	Sets a top level for command parsing. *module_name* must be name of ATENA's FEM module. Default: nil E.g. CCStructures
;	This is to indicate end of the current input command. Control is returned to the top level (specified by *module_name*) for parsing a next command. Must be preceded by at least one SPACE character.
&JUMP, &LABEL	Jump to a particular label while parsing the input document, i.e., skip the commands between &JUMP and LABEL keywords.
&DEBUG	Set on/off debug mode during Atena execution.
&EVALUATE	Invoke Atena calculator.
&MODULE	Load, unload, select... additional engineering modules
&THREADING	Set parameters for multithreading execution.

3.4 Analysis Identification and Execution Settings

3.4.1 The Command &TASK

Syntax:
&TASK:
TASK [{ NAME "*task name*" | TITLE "*title*" | DIMENSION *n* | SPACE { 2D | 3D | AXISYMMETRIC } }+]

Table 3: &TASK command parameters.

Parameter	Description
NAME "*task name*"	Task name. E.g.: NAME "*task name*"
TITLE "*title*"	Title of the analysis. TITLE "*title*"
DIMENSION *n*	Problem dimension. *n* equals 2 or 3 for two or three-dimensional analysis. Note that setting of DIMENSION sets also SPACE type. If DIMENSION is 2, then 2D SPACE type is expected. Once DIMENSION type is set, it cannot be changed elsewhere.
SPACE	Set type of space approximation. It can be 2D, 3D or AXISYMMETRIC, i.e. 2D in axis *x* and *y* symmetric with respect to axis *y*. (Radius of rotation corresponds to axis *x*). Note that setting of SPACE type sets also problem DIMENSION. Once SPACE type is set, it cannot be changed elsewhere.
Note: This command should be the first input, as it specifies dimension several entities read later, i.e. nodal coordinates.	

3.4.2 The Command &ALLOCATE_NODAL_DOFS

Syntax:
& ALLOCATE_NODAL_DOFS:

ALLOCATE_DOFS {[NODE] ID *node_id* | [NODES] [AT] SELECTION *"selection_name"* DOFS_MASK *mask*

Explicit allocation of nodal dofs. Typically, the dofs are allocated automatically based on element that incidence at the node. However, there are cases, when an explicit allocation is needed and it can be done by this command

3.4.3 The Command &TERMINATE / &BREAK

Syntax:
& TERMINATE:

TERMINATE {[AT] [MODULE *module_name*] ID *break_id* [IGNORE_HITS *n_hits*]}
| {[" *user's string* "]}

&BREAK:
BREAK {[AT] [MODULE *module_name*] ID *break_id* [IGNORE_HITS *n_hits*]} |
{[" *user's string* "]}

Break Atena execution at a particular break point *break_id* at module *module_name* after *number_of_hits_to_ignor*e hits. The parameter *module_name* can be CCFEModel, CCStructures, CCFEModelGenerate.... If no MODULE is specified, the execution terminates at the given break point *break_id* at any module. If the parameter IGNORE_HITS *number_of_hits_to_ignore* is not specified, the execution is terminated at the first approach of the specified break point. Several break point ids are recognized, but break point ids 1 and 2 are probably the most important. The former one is placed at entry of a main execution routine of each Atena's modul. Similarly, the latter one is located at the exit of that routine.

Alternatevily this command terminates the input commands stream, (i.e. no ID *break_id* was input), thereby terminating the execution and optionally displays user's string.

If the execution is run from a GUI window, (e.g. AtenaWin), a dialog is displaied before the actual termination/break action that gives the user choice to either accept the break or ignore it and continue the analysis. Batch analyses are broken unconditionally, see the /batch_exec command line switch.

The commands BREAK and TERMINATE behave identically, the latter one supported only for input compatibility reasons.

Examples:

BREAK "Joints' coordinates read"

BREAK ID 1

BREAK AT MODULE CCFEModel ID 2 IGNORE_HITS 3

3.4.4 The Command &JUMP / &LABEL

Syntax:

&JUMP:

JUMP [TO] [LABEL] "*string with label name*"

&LABEL:

LABEL "*string with label name*"

The first command instructs Atena to ignore all subsequent input data until the second command is found. Thereafter, the input commands are processed in the usual way. Several &JUMP/&LABEL commands can be used in the same file providing they have unique "*string with label name*". Note that &LABEL commands are ignored, unless a &JUMP command is being processed.

3.4.5 The Command &DEBUG

Syntax:

&DEBUG:

DEBUG {ON | OFF }

Set debug mode on/off. If it is on, the execution stops after processing of each main command from input stream. The next command is executed by pressing "Execute after break" button or alternatively press "Execute from the cursor position" button to execute a command at the current cursor position.

3.4.6 The Command &MODULE

Syntax:

&MODULE:
MODULE [NAME "*module_name*"] [TYPE "*type_name*" { LOAD | UNLOAD | SELECT | CHECK | RENAME "*module_new_name*" | REPLACE }+

The command MODULE is used to manipulate available FE engineering modules during Atena execution. These are a default module, which is a module specified by the /M command line parameter and any number of loaded modules using MODULE command ... LOAD. For each loaded/default module you can specify its type "*type_name*", e.g. CCStructuresCreep, and its name "*module_name*", which is used for referencing to it. If "*module_name*" is not input, "*type_name"* i used instead. Each such module can be loaded, unloaded, selected, renamed, replace the current module and checked.

Examples:

Transport analysis followed by static structure analysis:

MODULE TYPE "CCStructures" CHECK // check that the default module is of type CCStructure

MODULE NAME "Transport" TYPE "CCStructuresTransport" LOAD SELECT // load (new) CCStructuresTransport module and select it to be the current module, (i.e. 2 modules are now loaded)

MODULE NAME "Transport" TYPE "CCStructuresTransport" CHECK // check that the current module is of type CCStructure

... // input data for transport analysis

MODULE NAME "CCStructures" SELECT // select CCStructures as the current module

MODULE NAME "Transport" UNLOAD // unload CCStructuresTransport and free all related memory

MODULE TYPE "CCStructures" CHECK

... // input data for static structures analysis

Replace a default module by CCStructuresTransport:

MODULE TYPE "CCStructuresTransport" REPLACE

// MODULE TYPE "CCStructuresTransport" CHECK

3.4.7 The Command &THREADING

Syntax:

&THREADING:
NUM_THREADS *num_threads* NUM_UNUSED_THREADS *num_unused_threads*
MIN_CHUNK_SIZE_PER_THREAD *min_chunk_size_per_thread*

Set parameters for multihreading. *num_threads* sets number of threads to be used for Atena execution. By default all available processor's cores are used. *num_unused_threads* is the same same as the above but Atena will use number of processor's available threads minus *num_unused_threads*. The parameter [/num_threads=n] has higher priority.
min_chunk_size_per_thread[3] sets chunk size for guided OMP schedule. By default 0 value is assumed, i.e. static load distribution.

3.4.8 The Command &EVALUATE

Syntax:

&EVALUATE:

{EVALUATE|EVAL} =expression_token_string

or

{EVALUATE|EVAL} "expression_token_string"

This command evaluates an expression from the expression_token_string and prints the result into Atena output file. In addition, it can store the result in a parameter for later use within another Atena command(s). The token must start with '=' and must not include a separator, such as a space character.

Alternatively, skip the '=' character and enclose the the expression_token_string. In this case the string can contain space chars, (which are removed prior execution of the string).

Note that in Atena input most integer and float numbers can be replaced by an expression_token_ring. This allows for parametric analyses, analyses with perturbated geometry etc.

The EVALUATE command supports the following features:

Operators:

& | << >>

= <> < > <= >=

+ -

* / % ||

^

[3] Supported since version 5.7.0; replaces the option `NUM_ITERS_PER_THREAD` older versions

!

Functions:

Abs, Exp, Sign, Sqrt, Log, Log10

Sin, Cos, Tan, ASin, ACos, ATan

Factorial. Erf, ErfInv, Atan2, Pow,

SOLVE_QUADRATIC_EQN, SOLVE_CUBIC_EQN,

Rand, SRand

Variables:

Pi, e

You can also define your own variable/parameter. e.g.

EVALUATE =cc=10" ->10

EVALUATE =cc+5 -> 15

Other:

Scientific notation supported, e.g. -1.24e-1

Error handling supported

Examples of parameters' definition:

EVALUATE =Poisson=0.3

EVALUATE =Dens=0.023

EVALUATE =FCompr=42.5

EVALUATE "FTens=0.3*pow(FCompr,0.6666)"

EVALUATE "Fract=0.000001*73.*pow(FCompr,0.18)"

EVALUATE =Young=21500.*0.9*pow(FCompr/10.,0.3333)

Examples of their use within other input commands:

MATERIAL ID 1 NAME "Concrete" TYPE "CC3DNonLinCementitious2"

E =Young MU =Poisson RHO =Dens ALPHA 0.000012 FT "FTens" FC "-FCompr GF" =Fract

WD -0.0005 EXC 0.52 BETA 0.0 FC0 -28.0 EPS_CP -0.00115 FIXED 0.7

VETRTEX 1 xyz =x_right+RAND(-dx,dx) =y_back+RAND(-dy,dy) =z_top+RAND(-dz,dz) nodeprop 'vertex1'

3.4.9 The Command &PYTHON

This command makes possible to use Python interpreter in preparing data for Atena execution. The data can be (for example) stored into EVAL variables and later used in place of any numeric values in an Atena command. They can be also used for TRANSFORM_COORDS etc.

Note that this command is available only in 64bit versions of Atena and it requires an instalation of a Python environment.

Syntax:

&PYTHON:
PYTHON { STRING “cmd_string” | FILE “cmd_file” | BLOCK cmd_block| { GET_REAL | | GET_INT | GET_STRING } “py_variable” | { SET_REAL | SET_INT | SET_STRING } “py_variable” py_value | FLUSH_IO_STREAMS | REDIRECT_IO_STREAMS | RESTORE_IO_STREAMS }+

Table 4: Table with the Python data

Parameter	Description
STRING “cmd_string”	Execute Python string.
FILE “cmd_file”	Execute Python commands in file “cmd_file”.
BLOCK cmd_block	Execute block of Python commands. The block is enclosed in lines containing "$->" and "<-$" tags. Note that content of the the border lines is otherwise ignored.
GET_REAL | | GET_INT | GET_STRING } “py_variable”	Get Python value of variable py_variable in the module __main__.
{ SET_REAL | SET_INT | SET_STRING } “py_variable” py_value	Set Python variable py_variable to value py_value in the module __main__.
FLUSH_IO_STREAMS	Flush Python out and err streams
REDIRECT_IO_STREAMS	Redirect Python out and err stream to Atena CCout and ccerr streams. This is needed for GUI Atena environments.
RESTORE_IO_STREAMS	Redirect Python out and err stream to stdout and stderr

Example:

```
PYTHON
  STRING "my_print()"
  FILE "d:\temp\test_py.py"
  GET_STRING var_jmeno
  SET_REAL thick_wall =thick_wall  // =thick_wall can be used in a later Atena command
  BLOCK


$-> ---------------------------------------------------------------------------
def interpolate(x,x1,y1,x2,y2):
  dx=x2-x1
  dy=y2-y1
  if dx!=0:
    x=(y1*(x2-x)+y2*(x-x1))/dx
  else:
    x=(y1+y2)/2
  return x
--------------------------------------------------------------------------- <-$
```

3.4.10 The Command &BREAK_DEBUG

Syntax:

&BREAK_DEBUG:

BREAK_DEBUG *break_id*

Break execution at specific points. This command is typically used to debug an input data file. The following data points are recognized:

Table 5: Table with the recognized execution breakpoints

Desired action	Value of *break_id*
Do not break.	0
Break on entry to the main model execution routine.	1
Break on exit to the main model execution routine.	2
Break on entry to the generator model execution routine.	4
Break exit entry to the generator model execution routine.	8
Break on entry to the global dofs mapping execution routine.	16
Break on entry to the global dofs mapping execution routine.	32
Break at any of the above points.	-1

More break points can be set. To do that set *break_id* as sum of the required individual break points.

3.4.11 The Command &SELECTION

Syntax:
&SELECTION :
SELECTION "*destination_name*" { SOFT | FORCE } { CLEAR | REMOVE_DUPLICATES | KEEP_DUPLICATES | SKIP_DUPLICATES | {COMBINE | SEPARATE} *"list1" "list2"* [*"list3"*] | RENAME "*source_name*" | {FROM | AT} *from_id* [TO *to_id* [BY *by_id*]] | LIST { *id* }+ | {INSERT | INCLUDE} *"selection_name"* | EXCLUDE "*selection_name*" | REVERSE "*selection_name*" | CONNECT "*selection_name*" | REMOVE | ELEMENT_CONSTRUCT_TIME *elem_constr_time* GROUP *group_id* | { ACTIVE | INACTIVE | { CONSTRUCT_TIME_DEPENDENT_ACTIVE | CONSTRUCT_TIME_DEPENDENT_REDUCED } GROUP *group_id* | [ENFORCED] DELETE {GROUP *group_id* | JOINT} | GENERATE { [NODES] | [ELEMENT] [OF] {GROUP|GROUP_FROM} *group_id*} [GROUP_TO *group_to*] [WITHIN] { BOX [MACRO] [NODES] *i1 i2 i3 i4* [*i5 i6 i7 i8*] | DISTANCE *x* FROM { POINT [MACRO] [NODES] *i1* | LINE [MACRO] [NODES] *i1 i2* | PLANE [MACRO] [NODES] *i1 i2 i3* } | NEAREST [MACRO] [NODES] *i1* | [{{IP | IPS}|{ENODE|ENODES}|{GNODE|GNODES}}] | {SOURCE_NODE_SELECTION *sel_nodes* | SOURCE_GROUP_SELECTION *sel_groups* SOURCE_GROUP} | [EXECUTE] | SORT
[{ +X | -X}] [{ +Y | -Y}] [{ +Z | -Z}] }+

Table 6: &SELECTION command parameters

Parameter	Description
„destination_name"	Name of the created or modified selection list.
CLEAR	Clear current content of the list but doesn't remove the selection itself
REMOVE_DUPLICATES	Remove duplicate entries in the selection
KEEP_DUPLICATES	Keep duplicate ids during INSERT and INCLUDE operations.
SKIP_DUPLICATES	Skip duplicate ids during INSERT and INCLUDE operations.
{ SOFT | FORCE }	If SOFT is defined, then ATENA tolerates non-existent source selection(s). Otherwise a input error exception is generated.
{COMBINE | SEPARATE} *"list1" "list2"* [*"list3"*]	Combines two or three selection lists into one list or split one list into two or three selection lists. Used to convert multi_selection lists into ordinary selection list and vice versa.
RENAME "*source_name*"	Rename selection "*source_name*" to

	„destination_name“
{ FROM | AT} *from_id* [TO *to_id* [BY *by_id*]] }	Set interval for entity *ids* to be generated. They are generated for recursive formula $id_1 = from\_id$ $id_n = id_{n-1} + by\_id$ up to $id_n <= to\_id$ By default $to\_id = from\_id, by\_id = 1$ Example: LIST AT 1 AT 10 FROM 100 TO 150 BY 10
LIST *id*	Entity to be added into the selection , e.g. LIST 23 26 100
INSERT *„selection_name“* INCLUDE *„selection_name“*	Insert entities from the *selection_name* selection into the selection *destination_name*. Source entities, which are already present in the selection *destination_name,* are not inserted, thus avoiding entities‘ duplication.
EXCLUDE *„selection_name“*	Remove entities defined in the *selection_name* selection from the selection *destination_name*. Source entities, which are already not present in the selection *destination_name,* are skipped.
REVERSE[4] “*selection_name”* |	Insert in reversed order the selection *selection_name.*
CONNECT “*selection_name”*	Connect the source selection "*selection_name*" with destination selection "*destination_name*". This is done in the following way: Loop from the first to the last entry of the source selection. For each such entry loop from the last to the first entry of the destination selection. If the current source and destination entries match, it is the place, where "*destination_name*" and "*selection_name*" should be connected. To do that, keep the current entry in the destination selection and remove all sbsequent entries. Append the source selection starting from the 1st entry behind the matching entry up to the end to the destination selection. If no match is found, the selection are appended with all the entries they originally include. Eg. Destination selection: {2,7,8,3,1,4}, source selection {9,3,5} -> yields destination selection : {2,7,8,3,5} The source selection remains unchanged.

[4] Supported since version 5.8.0

SORT [{ +X | -X}] [{ +Y | -Y}] [{ +Z | -Z}	This command has sense only for selection containing FE nodes!! Sort entries in the selection according to their reference coordinates. Note that sorting is executed immediatelly and thus it makes sense only for selection with all their entries (either previously inputed or with executed thier generation). For example: SORT +X - sort nodes referenced in the selection according with respect to their x coordinate, (from minimum t maximum), SORT -X - the same but in reverse order SORT +X +Y -Z - sort nodes N_i with reference coordinates (x_i,y_i,z_i) with respect to the value $x_i+y_i-z_i$. By default no sorting is applied.
REMOVE	Remove the modified selection list.
GENERATE { [NODES] | [ELEMENT] [OF] [{GROUP|GROUP_FROM} *group_id*} [GROUP_TO *group_id_to*]] } [WITHIN] BOX [MACRO] [NODES] *i1 i2 i3 i4* [*i5 i6 i7 i8*] [EXECUTE]	Data for the selection list generation. The list will include either all nodes or all elements of the group <*group_id.... group_id_to*> from within a box defined by the macro nodes *i1* thru *i8* (for 3D case) or a quadrilateral defined by *i1* thru *i4* (2D case and 3D case within plane defined by *i1 i2 i3 i4*). If *group_id* is specified, elements are generated, otherwise nodes are generated. The EXECUTE keyword forces to carry out the generation immediately. Otherwise it is done prior a first step execution.
SOURCE_NODE_SELECTION *sel_nodes*	Only nodes from selection *sel_nodes* become candidates for the generation. If not specified, all nodes from the model are considered.
SOURCE_GROUP_SELECTION *sel_groups* SOURCE_GROUP_SELECTION *sel_elements*	Only elements from selections *sel_groups; sel_elements* become candidates for the generation. If not specified, all elements from the model are considered.
GENERATE { [NODES] | [ELEMENT] [OF] [{GROUP | GROUP_FROM} *group_id*} [GROUP_TO *group_id_to*]] } [WITHIN] DISTANCE *x* FROM POINT [MACRO] [NODES] *i1* [EXECUTE]	Data for the selection list generation. The list will include either all nodes or all elements of the group <*group_id.... group_id_to*> from within distance *x* with respect to the point defined by the macro nodes *i1*. If <*group_id_....group_id_to*> is specified, elements are generated, otherwise nodes are generated. The EXECUTE keyword forces to carry out the generation immediately. Otherwise it is done prior a first step execution.
GENERATE { [NODES] | [ELEMENT] [OF] [{GROUP|GROUP_FROM}	Data for the selection list generation. The list will include either all nodes or all elements of the group <*group_id.... group_id_to*> from within distance *x*

group_id} [GROUP_TO *group_id_to*]] } [WITHIN] DISTANCE *x* FROM LINE [MACRO] [NODES] *i1 i2* [EXECUTE] [INSIDE]	with respect to the line defined by the macro nodes *i1* and *i2*. If *group_id* is specified, elements are generated, otherwise nodes are generated. The EXECUTE keyword forces to carry out the generation immediately. Otherwise it is done prior a first step execution. If the keyword INSIDE is used, the generation is reestricted only to entities with a node located between the macro node *i1 i2*.
GENERATE { [NODES] | [ELEMENT] [OF] [{GROUP|GROUP_FROM} *group_id*} [GROUP_TO *group_id_to*]] } [WITHIN] DISTANCE *x* FROM PLANE [MACRO] [NODES] *i1 i2 i3* [EXECUTE] [INSIDE]	Data for the selection list generation. The list will include either all nodes or all elements of the group <*group_id.... group_id_to*> from within distance *x* with respect to the plane defined by the macro nodes *I, i2* and *i3*. If *group_id* is specified, elements are generated, otherwise nodes are generated. The EXECUTE keyword forces to carry out the generation immediately. Otherwise it is done prior a first step execution. If the keyword INSIDE is used, the generation is reestricted only to entities with a node located between the macro node *i1 i2, i3*.
GENERATE { [NODES] | [ELEMENT] [OF] [{GROUP|GROUP_FROM} *group_id*} [GROUP_TO *group_id_to*]] } NEAREST [MACRO] [NODES] *i1* [EXECUTE]	Data for the selection list generation. The list will include the nearest node or element of the group <*group_id.... group_id_to*> with respect to the i1. If *group_id* is specified, an element is included, otherwise a node is added. The EXECUTE keyword forces to carry out the generation immediately. Otherwise it is done prior a first step execution.
[{{IP | IPS}|{ENODE|ENODES}|{GNODE|GNODES}}]	Generated a multiselection that includes integrated points (or element nodes) instead of global nodes. Use {GNODE|GNODES} to generate selection with global nodes, where each entry must be incidented by a element with *group_id*>=*grouip_id_from* and *group_id*<=*group_id_to*.
{ ACTIVE | INACTIVE | CONSTRUCT_TIME_DEPENDENT_ACTIVE } GROUP *group_id*	Make active, inactive or active on condition *constr_time* =< *current_time* all elements contained in the selection list that belongs to the group *group_id*
ELEMENT_CONSTRUCT_TIME *elem_constr_time* GROUP *group_id*	Set time of construction of all elements contained in the selection list that belongs to the group *group_id*. By default the elements not yet "constructed" are computed, but their matrices and vectors are multiplied by NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF, see Table 15). The *elem_constr_time* parameter is also accounted for by material models with variable material model parameters. This parameter is added to GROUP_CONSTRUCT_TIME *group_constr_time*, seeTable 53 . By default it is 0.

[ENFORCED] DELETE {GROUP *group_id* | JOINT}	Delete elements contained in the selection list that belongs to the group *group_id* or delete nodes *contained* in the selection list. If ENFORCED is not specified, all references to a deleted entity remain valid even after the deletion, thereby it is possible later to re-input the entity with new data. Otherwise, the entity and all references to it get unconditionally removed.
CONSTRUCT_TIME_DEPENDENT_ACTIVE | GROUP *group_id*	Set ACTIVE/INACTIVE status of all elements in the group group_id deneding on their time of construction. All elements not yet constructed are skipped.
CONSTRUCT_TIM[5]E_DEPENDENT_REDUCED GROUP *group_id*	Similar to the above, however all not yet constructed elements are computed and then multiplied by factor NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF, see &CONVERGENCE_CRITERIA, parameter NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF *x*.

Example:

```
SELECTION "all_nodes" FROM 1 TO 22
SELECTION "source" LIST 1 2 3 4 5 6
SELECTION "dest" LIST 3 5 12
SELECTION "source" INSERT "dest"
SELECTION "source" REMOVE "dest"
SELECTION “source” REMOVE
SELECTION "source" GENERATE ELEMENTS GROUP 1 WITHIN BOX 101 102
      103 104 106 107 108  // 3D case
SELECTION "source" GENERATE NODES WITHIN BOX MACRO NODES 101
      102 103 104 // 2D case
SELECTION "source" GENERATE NODES WITHIN DISTANCE 2.4 FROM POINT
      MACRO NODES 101
SELECTION "source" GENERATE NODES WITHIN DISTANCE 2.4 FROM LINE
      MACRO NODES 101 102
SELECTION "source" GENERATE NODES WITHIN DISTANCE 2.4 FROM PLANE
      MACRO NODES 101 102 103 GENERATE
SELECTION "source" GENERATE NODE NEAREST  MACRO NODE 101
      GENERATE
 SELECTION "nodes"  GENERATE SORT -Y  +X
SELECTION "border_nodes"  CONNECT "next_border_nodes"
```

Generate selection and monitor at IP:

```
SELECTION "IP_NEAREST_985001" GENERATE IPS NEAREST MACRO NODES
      985001 group_from 105 group_id_to 302 EXECUTE
```

[5] Supported since version 5.8.0

```
OUTPUT LOCATION OUTPUT_DATA DATA LIST
      "SELECTION_IDS_IP_NEAREST_985001" END ;
OUTPUT NAME "Monitor1_DISPLACEMENTS #100000" MONITOR_2
      LOCATION ELEMENT_IPS MULTI_SELECTION AT
      "IP_NEAREST_985001" DATA LIST
 "DISPLACEMENTS_AT_IPS" ITEM AT 1  End ;

Generate selection and monitor at NODE:
SELECTION "NODE_NEAREST_985001" GENERATE NODE NEAREST MACRO
      NODES 985001 EXECUTE
OUTPUT LOCATION OUTPUT_DATA DATA LIST
      "SELECTION_IDS_NODE_NEAREST_985001" END ;
OUTPUT  NAME "Monitor1_DISPLACEMENTS #100000" MONITOR_2
      LOCATION NODES NODE AT SELECTION "NODE_NEAREST_985001"
      DATA LIST  "DISPLACEMENTS" ITEM AT 1  End ;


SELECTION "ENODE_NEAREST_214" GENERATE ENODE NEAREST MACRO
      NODES 214 group_from 108 group_to 302 EXECUTE
OUTPUT LOCATION OUTPUT_DATA DATA LIST
      "SELECTION_IDS_ENODE_NEAREST_214" END ;

SELECTION "GNODE_NEAREST_214" GENERATE GNODE NEAREST MACRO
      NODES 214 group_from 108 group_to 302 EXECUTE
OUTPUT LOCATION OUTPUT_DATA DATA LIST
      "SELECTION_IDS_GNODE_NEAREST_214" END ;

SELECTION "InactiveElementsFromGroup218"    LIST 63 64 65 INACTIVE GROUP
      208 ELEMENT_CONSTRUCT_TIME  3. GROUP 208  ;
```

3.4.12 The Command &SELECTION_REAL

Syntax:
&SELECTION_REAL :
SELECTION_REAL *"destination_name"* { {SOFT | FORCE } { CLEAR | RENAME *"source_name"*] | {FROM | AT} *from_id* [TO *to_id* [BY *by_id*]] | LIST { *id* }+ | { INSERT | INCLUDE} *"selection_name"* | EXCLUDE *"selection_name"* | REMOVE }+

Table 7: &SELECTION_REAL command parameters

Parameter	Description
„destination_name"	Name of the created or modified selection list.
CLEAR	Clear current content of the list but doesn't remove the selection itself
{ SOFT | FORCE }	If SOFT is defined, then ATENA tolerates non-existent source selection(s). Otherwise a input error exception is generated.
RENAME *"source_name"*	Rename selection *"source_name"* to *„destination_name"*
{ FROM | AT} *from_id* [TO *to_id* [BY *by_id*]] }	Set interval for entity *ids* to be generated. They are generated for recursive formula $id_1 = from\_id$ $id_n = id_{n-1} + by\_id$ up to $id_n <= to\_id$ By default $to\_id = from\_id, by\_id = 1$ Example: LIST AT 1 AT 10 FROM 100 TO 150 BY 10
LIST *id*	Entity to be added into the selection , e.g. LIST 23 26 100
INSERT *„selection_name"* INCLUDE *„selection_name"*	Insert entities from the *selection_name* selection into the selection *destination_name*. Source entities, which are already present in the selection *destination_name,* are not inserted, thus avoiding entities' duplication.
EXCLUDE *„selection_name"*	Remove entities defined in the *selection_name* selection from the selection *destination_name*. Source entities, which are already not present in the selection *destination_name,* are skipped.
REMOVE	Remove the modified selection list.

3.4.13 The Command &SET

Syntax:
&SET:
SET { &ANALYSIS_TYPE | &LINEAR_SOLVER_TYPE |
&REFERENCE_CONFIGURATION | &CONVERGENCE_CRITERIA |
&SOLUTION_METHOD | &PREDICTOR_TYPE | &UPDATE_DISPLS_STRATEGY |
&ARC_LENGTH_PARAMS | &LINE_SEARCH_PARAMS | &OPTIMIZE_PARAMS |
&SERIALIZE_PARAMS | SOLVER_KEYS *n* | &FATIGUE_PARAMS |
&CREEP_ANALYSIS_PARAMS | &DYNAMIC_ANALYSIS_PARAMS | {
SOLVE_LHS_BCS_ON [OPTIMIZE] {VARIABLE_SLAVE_DOFS | SLAVE_DOFS |
DOFS } [AND] { VARIABLE_EQUATIONS | EQUATIONS}
| SOLVE_LHS_BCS_OFF}| { OPTIMIZE_SM_MAPPING_SIZE |
OPTIMIZE_SM_MAPPING_SPEED | SPARSE_MAP_INDEXING *n* } |
&MAX_REF_IDS | { EXTERNAL_IDENTIFIERS | INTERNAL_IDENTIFIERS }[6] |{
DISABLE_REPORT_TASK | ENABLE_REPORT_TASK | REPORT_LOCATION_STEP
n} | { DISABLE_REPORT_LOCATION | ENABLE_REPORT_LOCATION } | {
USE_BEST_ITERATION_FOR_CRITERION |
USE_BEST_ITERATION_FOR_CRITERIA } n_1 n_2 ... | {
UNUSE_BEST_ITERATION_FOR_CRITERION |
UNUSE_BEST_ITERATION_FOR_CRITERIA } n_1 n_2 ... | BEST_ITERATION_MIN_ID *n*
| STEP_LOAD_REDUCTION_ALLOWANCE *n* | REDUCE_STEP_LOAD_COEFF *v* |
MIN_LHS_BCS_MASTER_NODE_COEFF *n* | MIN_LHS_BCS_SLAVE_NODE_COEFF
n | MIN_LHS_BCS_SOLUTION_COEFF *n*}+

Table 8: &SET command parameters

Parameter	Description
&ANALYSIS_TYPE	Set what type of analysis is executed, i.e. static, transient etc.
&LINEAR_SOLVER_TYPE	Use direct or iterative solver (and set some vital parameters for the iterative solver).
&CONVERGENCE_CRITERIA	Convergence criteria during iteration process within each load step.
&SOLUTION_METHOD	Choose solution method for the analysis.
&ARC_LENGTH_PARAMS	Set parameters for Arc Length method.
&LINE_SEARCH_PARAMS	Set parameters for Line Search method.
&PREDICTOR_TYPE	Set which type of predictor should be used for building stiffness matrix, (i.e. elastic, tangential or secant).
&UPDATE_DISPLS_STRATEGY	Strategy for updating displacements during iterations, either each iteration or each load step.

[6] Not available in ATENA version 4.3.1 and older.

&OPTIMIZE_PARAMS	Sets whether bandwidth optimization is required and which type.
&SERIALIZE_PARAMS	Set depth of serialization. Change of this parameter is needed only under very special conditions and the user would normally use its default setting.
SOLVE_LHS_BCS_ON [OPTIMIZE] {VARIABLE_SLAVE_DOFS | SLAVE_DOFS | DOFS } [AND] { VARIABLE_EQUATIONS | EQUATIONS | SOLVE_LHS_BCS_OFF }	Turns on and off an advance LHS BCs management. For better stability it is possible to to reorded slave and master dofs and reorder specified LHS boundary conditions, see keywords VARIABLE_SLAVE_DOFS | SLAVE_DOFS | (all) DOFS for dofs and VARIABLE_EQUATIONS | (all) EQUATIONS for BCs. By default, it is ON and the above reordenig is allowed thru all dofs and BCs. Do not change this parameter to SOLVE_LHS_BCS_OFF unless unavoidable and all consequences being well understood.
OPTIMIZE_SM_MAPPING_SIZE | OPTIMIZE_SM_MAPPING_SPEED | SPARSE_MAP_INDEXING *n* }	Set how sparse matrix elementys are mapped. Optimize for size, speed or set manually, i.e. *n* = 1,2,4,8,16. By default, optimize for size is ussed, i.e. RAM amount is minimized..
SET SOLVER_KEYS *n*	This command specifies directly in binary form flags for the non-linear solver. It is not aimed for direct use by users. Every setting can be achieved in a more readable form using other parameters of the &SET command.
&FATIGUE_PARAMS	Parameters for fatigue analysis
&CREEP_ANALYSIS_PARAMS	Parameters for creep analysis.
&DYNAMIC_ANALYSIS_PARAMS	Parameters for dynamic analysis
&MAX_REF_IDS	Set maximum reference ids that are used by the automatic ATENA reference ids generator
DISABLE_REPORT_TASK | ENABLE_REPORT_TASK DISABLE_REPORT_LOCATION | ENABLE_REPORT_LOCATION | REPORT_LOCATION_STEP *n*	Disable or enable visualisation of task and location within the current execution. It is also possible to report location each *n* % of the total job. For example REPORT_LOCATION_STEP *10* ensures that for a system of say 200000 equations location is reported for each 20000th equation, e.g. 1, 20001, 40001... By default these information are enabled and location progress is reported always, so that the user has gets the best info about the analysis. This

	settings, however, involves some CPU overhead. To maximize the execution speed, disable these reports.
{ EXTERNAL_IDENTIFIERS | INTERNAL_IDENTIFIERS }	Set the way how, Atena entities are are identified. If external identifiers are required, Atena uses ids specified in the iput file. If intenal identifiers are required, Atena uses internal ids starting from 1 to number of a particular entities. Under normal conditions internal ids should not be used.
USE_BEST_ITERATION_FOR_CRITERION | USE_BEST_ITERATION_FOR_CRITERIA } n_1 n_2 ...	For n>0 and the iterating process within the current step does not yield a converged solution, then the solution is reverted to the best converged iteration based on the convergence criteria n_1, n_2, ... For n=0 the use of best iteration is reset to not using best_iteration feature. If divergence step's (or iteration's) stop criteria are met, the current step is marked as non-converged. When this option is combined with STEP_LOAD_REDUCTION_ALLOWANCE n, then the iteration is reverted only when (n-number of attempts to revert the current step)=0. By default n=0, i.e. this feature is N/A and v=1. , i.e. the step is marked as not converged step.
UNUSE_BEST_ITERATION_FOR_CRITERION | UNUSE_BEST_ITERATION_FOR_CRITERIA } n_1 n_2 ...	Same as the above but it removes the specified convregence criteria for best_iteration engine. If all criteria are removed, no best_iteration strategy is used .
BEST_ITERATION_MIN_ID n	Minimum iteration id, for which the iteration is always stored, i.e. regardless its convergence performance. Any subsequent iteration is stored only, if its convergence is better than convergence of any previous iteration.
STEP_LOAD_REDUCTION_ALLOWANCE n REDUCE_STEP_LOAD_COEFF v	If n>0 and the iterating process within the current step does not yield a converged solution, then the current step is re-executed for a reduced load increment. This step's re-execution is allowed n times and the load increment in the current re-exection is reduced by factor v^i, where i=1..n, i.e. number of the step re-execution. By default v=0.5 and n =0.
REFERENCE_CONFIGURATION	Set the current configuration, (i.e. structural

	shape) as the reference configuration. Subsequent displacements etc. will be computed with respect to it.

&ANALYSIS_TYPE:
{ STATIC | &TRANSIENT | &EIGENVALUES }

Table 9: &ANALYSIS_TYPE sub-command parameters

Parameter	Description
STATIC	Specify static analysis. There are no additional parameters
&TRANSIENT	Set transient analysis and set some parameters for it.
&EIGENVALUES	Set some parametyers for eigenvalues analysis.

&TRANSIENT:
TRANSIENT { [TIME] CURRENT *x* | [TIME] INCREMENT *x* |
TIME_INTEGRATION { {CRANK_NICHOLSON | THETA *x* }$_+$ |
ADAMS_BASHFORTH } | NEWMARK BETA *x* | NEWMARK GAMMA *x* |
HUGHES ALPHA *x* | WILSON THETA *x* | DAMPING { STIFFNESS
[COEFFICIENT] *x* | MASS [COEFFICIENT] *x* | &REGRESSION_DATA }$_+$

&REGRESSION_DATA:
REGRESSION { MODE *mode_id* | OMEGA *omega_val* | KSI *ksi_val* | WEIGHT
weight_val }$_+$ CALCULATE

Table 10: &TRANSIENT sub-command parameters♥

Parameter	Description
[TIME] CURRENT *x*	Sets current time.
[TIME] INCREMENT *x*	Sets time increment in steps.
TIME_INTEGRATION	Set type of temporal integration scheme. If this parameter is not input, then Newmark integration will be used.
CRANK_NICHOLSON	Use linear trapezoidal integration.
THETA *x*	θ parameter for trapezoidal integration. By default θ = 0.5. Several other linear temporal integration may be utilized depending on the θ, e.g. implicit Newton integration for θ = 1., explicit integration for θ = 0 etc. For good compromise between convergence and possibility of oscillations values about θ = 0.85 is recommended.
ADAMS_BASHFORTH	Adams – Bashforth quadratic temporal integration.
NEWMARK BETA *x*	Defines the Newmark's β parameter.

NEWMARK GAMA *x*	Defines the Newmark's γ parameter.
HUGHES_ALPHA *x*	Defines the Hughes α damping parameter
WILSON THETA *x*	Defines the Wilson θ damping parameter.
DAMPING STIFFNESS [COEFFICIENT] *x*	Defines stiffness matrix coefficient for proportional damping. E.g.: DAMPING STIFFNESS COEFFICIENT *0.8*
DAMPING MASS [COEFFICIENT] *x*	Defines mass matrix coefficient for proportional damping. E.g.: DAMPING MASS COEFFICIENT *0.8*
DAMPING REGRESSION MODE *mode_id* | OMEGA *omega_val* | KSI *ksi_val* | WEIGHT *weight_val*	Generate proportional damping coefficient based on input of modal damping parameters *ksi_val*. *mode_id* is id of an eigenmode, for which damping parameter *ksi_val* and associated weight factor *weight_val* is input. Values for at least 2 eigenmodes must be given. By default, *weight_val*=1. The keyword CALCUATE marks the end of the input and execute the regression procedure to transform the current input data for structural damping to the above DAMPING MASS and STIFFNESS coefficients. Example: SET TRANSIENT DAMPING REGRESSION MODE 1 OMEGA 2 KSI 0.002 WEIGHT 0.6 MODE 2 OMEGA 3 KSI 0.03 WEIGHT 0.8 MODE 3 OMEGA 7 KSI 0.04 WEIGHT 1.1 MODE 4 OMEGA 15 KSI 0.1 WEIGHT 0.9 MODE 5 OMEGA 19 KSI 0.14 WEIGHT 0.8 CALCULATE

&LINEAR_SOLVER_TYPE:
{ SOLVER { LU | DSS_LLT | DSS_LDLT | JAC | GS | ILUR | DCG | ICCG | DCGN | LUCN | DBCG | LUBC | DCGS | LUCS | DOMN | LUOM | DGMR | LUGM | PARDISO } | SLAP_ITERATION [LIMIT] *n* | SLAP_SAVED_VECTOR [LIMIT] *n* | SOLVER_BLOCK_SIZE *n* | EXTEND_ACCURACY_FACTOR *x* | PARDISO_REQUIRED_ACCURACY *y* | MIN_LHS_BCS_MASTER_NODE_COEFF *n* }+

Table 11: & LINEAR_SOLVER_TYPE sub-command parameters

Parameter	Description
{ SOLVER { LU | DSS_LLT | DSS_LDLT | JAC | GS | ILUR | DCG | ICCG | DCGN | LUCN | DBCG | LUBC | DCGS | LUCS | DOMN | LUOM | DGMR | LUGM }	Type of solver for computing linear problem Ax=y. It can be either a direct skyline storage solver, (i.e. LU), or direct sparse storage solver, (i.e. DSS_LLT, DSS_LDLT), or iterative sparse storage solver (i.e. the remaining types). Alternatively, it can be parallel direct sparse solver PARDISO from the MKL provided by Intel Visual Fortran. The skyline and sparse (SLAP) storage schemes are described in the Theoretical Manual for Atena software. The direct sparse solvers DSS_LLT and DSS_LDLT

	differ in type of factorization, they use. It is LL^T and LDL^T, respectively. In case of unsymmetric structural matrix both solvers use LU factorisation. The table below lists all the available solvers with their brief characteristic and recommendation for use. Default: LU
SOLVER_BLOCK_SIZE *n*	This value set granularity size for the solvers DSS_LLT and DSS_LDLT. It defines a block size during pre-factorisation process. The higher value, the lower number of structural blocks and smaller RAM overhead for mapping the structural matrix. On the other hand, a higher value results in higher waste of RAM to store the actual data of the matrix. It is recommended to set this value to something in range <2…6>. Default: 2
SLAP_ITERATION [LIMIT] *n*	Maximum number of iterations allowed within an iterative linear problem solver. Default: number of structural degree of freedom.
SLAP_SAVED_VECTOR [LIMIT] *nsave*	Number of direction vectors to save and orthogonalize against. This parameter is only used by the following iterative solvers: DOMN, LUOM (*nsave >=0*) and DGMR, LUGM (*nsave >0*). In all cases *nsave <=ndofs*, where *ndofs* is number of degree of freedom. Typically, the higher *nsave* , the better convergence but also the bigger memory required by the solver. Default value is *ndofs* /6 for DOMN, LUOM and *ndofs* /3 for DGMR, LUGM solver.
EXTEND_ACCURACY_FACTOR *x*	Factor, by which an iterative sparse matrix solver should increase its requirement upon accuracy. If *x*>0, the solver will employ residual forces convergence criterion with requested max. error "RELATIVE RESIDUAL ERROR" / *x* . If x <0, residual displacements convergence criterion will be used with max. error "RELATIVE DISPLACEMENTS ERROR" / *x*. Recommended values <1..10>. Default: 2
PARDISO_REQUIRED_ACCURACY [LIMIT] *y*	Accuracy required by PARDISO solver. For *y*=0, do not perform preconditioned Krylow-Subspace iterations and use LU factorisation instead. Otherwise the value of *y* controls accuracy of the built-in iterative solver further strenghten by the above EXTEND_ACCURACY_FACTOR factor *x*. The final required accuracy (expressed in number of non-negligible digits behing the decimal point) is $l=\log10(y/x)$. If the problem matrix is unsymmetric, (e.q. transport analysis), CGS iteration replaces the computation of LU. The preconditioner is LU that is computed at the previous step (the

	first step or last step with a failure) in a sequence of solutions needed for identical sparsity patterns. *l* controls the stopping criterion of the Krylow-Subspace iteration. $\varepsilon_{CGS} = 10^{(-l)}$ is used in the stopping criterion $\frac{\|dx_i\|}{\|dx_0\|} < \varepsilon_{CGS}$, with $\|dx_i\| = \|inv(LU)\, r_i\|$ and r_i is the residuum at iteration *i* of the preconditioned Krylow-Subspace iteration. If the problem matrix is symmetric (positive definite), (e.g. for static analysis), the same applies, but CG iteration replaces the computation of LU. Example: SET PARDISO_REQUIRED_ACCURACY limit 0.00000001 Default: 0
MIN_LHS_BCS_MASTER_NODE_COEFF *n* | MIN_LHS_BCS_SLAVE_NODE_COEFF *n* |MIN_LHS_BCS_SOLUTION_COEFF *n*	Set accuracy, (in its abs value) used to assemble and process lhs boundary conditions, particularly master nodes, slave nodes and solution coefficients. The latter value is the value used during lhs bcs solution, whilst the former two values are used to assemble/reassemble the boundary conditions. If the specified value is too high, although the solution is faster and needs less RAM, it can filter out some important relations within the boundary conditions. On the other hand, if the value is too small, the solution is slower and needs more RAM. In addition, it need not detect and eliminate all redundancies within the boundary conditions and can fail. Note that the effect of this solution parameter can be inspected in "Global matrix LHS BCs statistics" printed in ATENA output file. Example: SET MIN_LHS_BCS_MASTER_NODE_COEFF 1.e-5 Default: 1.e-6

Table 12: SOLVER TYPES

Type	D/I	Prep. phase	Exec. phase	Sym/ Non-sym	Temporary memory required	Description
LU	D	---	---	S,NS	-----	For smaller or ill-posed problems
JAC	I	ssds	sir	S,NS	*4*(11)+8*(1+4*n)*	Simple, not recommended
GS	I	---	sir	S,NS	*4*(11+nel+n+1)+8*(1+3*n+nel)*	
ILUR	I	ssilus	sir	S,NS	*4*(13+4*n+nu+nl)+8*(1+4*n+nu+nl)*	

DCG	I	ssds	scg	S	*4*(11)+8*(1+5*n)*	For large symmetric well-posed problems
ICCG	I	ssics	scg	S	*4*(12+nel+n)+8*(1+5*n+nel)*	For large symmetric problems, recommended
DCGN	I	ssd2s	scgn	S,NS	*4*(11)+8*(1+8*n)*	For large non-symmetric well-posed problems
LUCN	I	ssilus	scgn	S,NS	*4*(13+4*n+nl+nl)+8*(1+8*n+nl+nu)*	For large non-symmetric problems, recommended
DBCG	I	ssds	sbcg	S,NS	*4*(11)+8*(1+8*n)*	
LUBC	I	ssilus	sbcg	S,NS	*4*(13+4*n+nl+nu)+8*(1+8*n+nu+nl)*	
DCGS	I	ssds	scgs	S,NS	*4*(11)+8*(1+8*n)*	
LUCS	I	ssilus	scgs	S,NS	*4*(13+4*n+nl+nu)+8*(1+8*n+nu+nl)*	
DOMN	I	ssds	somn	S,NS	*4*(11)+8*(1+4*n+nsave+3*n*(nsave+1))*	
LUOM	I	ssilus	somn	S,NS	*4*(13+4*n+nu+nl)+8*(1+nl+nu+4*n+nsave+3*n*(nsave+1))*	
DGMR	I	ssds	sgmres	S,NS	*4*(31)+8*(2+n+n*(nsave+6)+nsave*(nsave+3))*	
LUGM	I	ssilus	sgmres	S,NS	*4*(33+4*n+nl+nu)+8*(2+n+nu+nl+n*(nsave+6)+nsave*(nsave+3))*	

In the above:

n is number of degree of freedom of the problem. *nel* is the number of nonzeroes in the lower triangle of the problem matrix (including the diagonal). *nl* and *nu* is the number of nonzeroes in the lower resp. upper triangle of the matrix (excluding the diagonal).

Table 13: EXECUTION PHASES

Phase name	Description
sir	Preconditioned Iterative Refinement sparse Ax = b solver. Routine to solve a general linear system Ax = b using iterative refinement with a matrix splitting.
scg	Preconditioned Conjugate Gradient iterative Ax=b solver. Routine to solve a symmetric positive definite linear system Ax = b using the Preconditioned Conjugate Gradient method.

scgn	Preconditioned CG Sparse Ax=b Solver for Normal Equations. Routine to solve a general linear system Ax = b using the Preconditioned Conjugate Gradient method applied to the normal equations AA'y = b, x=A'y.
sbcg	Solve a Non-Symmetric system using Preconditioned BiConjugate Gradient.
scgs	Preconditioned BiConjugate Gradient Sparse Ax=b solver. Routine to solve a Non-Symmetric linear system Ax = b using the Preconditioned BiConjugate Gradient method.
somn	Preconditioned Orthomin Sparse Iterative Ax=b Solver. Routine to solve a general linear system Ax = b using the Preconditioned Orthomin method.
sgmres	Preconditioned GMRES iterative sparse Ax=b solver. This routine uses the generalized minimum residual (GMRES) method with preconditioning to solve non-symmetric linear systems of the form: A*x = b.

Table 14: PREPARATION PHASES

Phase name	Description
ssds	Diagonal Scaling Preconditioner SLAP Set Up. Routine to compute the inverse of the diagonal of a matrix stored in the SLAP Column format.
ssilus	Incomplete LU Decomposition Preconditioner SLAP Set Up. Routine to generate the incomplete LDU decomposition of a matrix. The unit lower triangular factor L is stored by rows and the unit upper triangular factor U is stored by columns. The inverse of the diagonal matrix D is stored. No fill in is allowed.
ssics	Incomplete Cholesky Decomposition Preconditioner SLAP Set Up. Routine to generate the Incomplete Cholesky decomposition, L*D*L-trans, of a symmetric positive definite matrix, A, which is stored in SLAP Column format. The unit lower triangular matrix L is stored by rows, and the inverse of the diagonal matrix D is stored.
ssd2s	Diagonal Scaling Preconditioner SLAP Normal Eqns Set Up. Routine to compute the inverse of the diagonal of the matrix A*A'. Where A is stored in SLAP-Column format.

&CONVERGENCE_CRITERIA:
{ ABSOLUTE [ERROR] | RELATIVE [ERROR] } | RESIDUAL ERROR *x* |
DISPLACEMENT ERROR *x* | ENERGY ERROR *x* | STEP_STOP_RESIDUAL ERROR FACTOR *x* | STEP_STOP_DISPLACEMENT ERROR FACTOR *x* | STEP_STOP_ENERGY ERROR FACTOR *x* | ITER_STOP_RESIDUAL ERROR FACTOR *x* | ITER_STOP_DISPLACEMENT ERROR FACTOR *x* | ITER_STOP_ENERGY ERROR FACTOR *x* | NEGLIGIBLE_RESIDUAL *x* | NEGLIGIBLE _DISPLACEMENT *x* | NEGLIGIBLE_SIZE *x* | NEGLIGIBLE_TIME_FRACTION *x* | NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF *x* | ITERATION [LIMIT] *n* }+

Table 15: &CONVERGENCE_CRITERIA sub-command parameters

Parameter	Description
ABSOLUTE [ERROR]	The convergence criteria values are computed using the absolute norm that is using the maximal element of an array in its absolute value. The error is then computed by dividing an iterative value with the value cumulated within the whole step. Note that this keyword can be used also in conjugation with the input NEGLIGIBLE _SIZE *n*, in which case it has slightly different meaning, see below.
RELATIVE [ERROR]	The convergence criteria values are computed using the Euclidean norm. The error is then computed by dividing an iterative value with the value cumulated within the whole step. Note that this keyword can be used also in conjugation with the input NEGLIGIBLE _SIZE *n*, in which case it has slightly different meaning, see below.
RESIDUAL ERROR *x*	Convergence limit for absolute value of residual forces. Default value is 0.01. E.g. RESIDUAL ERROR *x*
DISPLACEMENT ERROR *x*	Convergence limit for absolute value of displacement increments. Default value is 0.01. E.g. DISPLACEMENT ERROR *x*
ENERGY ERROR *x*	Convergence limit for value of residual energy, i.e. norm of displacement increment multiplied by norm of residual forces. Not used in transport analysis. Default value is 0.01. E.g. RESIDUAL ERROR *x*
STEP_STOP_RESIDUAL ERROR FACTOR *x* | STEP_STOP_DISPLACEMENT ERROR FACTOR *x* | STEP_STOP_ENERGY ERROR FACTOR *x* | ITER_STOP_RESIDUAL ERROR FACTOR *x* | ITER_STOP_DISPLACEMENT ERROR FACTOR *x* | ITER_STOP_ENERGY ERROR FACTOR *x*	Factors for appropriate convergence criterion value. If a convergence criterion value multiplied by the appropriate factor exceeds the related calculated analysis error, then the execution is immediately killed. They are two sets of factors: the first one for checking each iteration and the other one to be exercised at the end of each step. The default value for iteration related factors is 1000, whilst the default value for step related factors is 10. E.g. SET Absolute stop_displacement error factor 15. Step_stop_displacement error factor 10. Step_stop_residual error factor 53 Iter_stop_displacement error factor 201 Iter_stop_residual error factor 203 SET Relative Step_stop_displacement error factor 54 Step_stop_energy error factor 55

	Step_stop_residual error factor 56 Iter_stop_displacement error factor 204 Iter_stop_energy error factor 205 Iter_stop_residual error factor 206
NEGLIGIBLE_SIZE *x*	Size that is already negligible. It affects accuracy of the analysis, particularly calculations of master/slave BCs, fixing of discrete reinforcement and the surrounding solids etc. For example points are assumed identical, if the distance between them is less than the absolute negligible size. Each element must have at each direction size greater than the absolute negligable size. Most iterative procedures compute with accuracy equal to the absolute negligible size. For all the comparisons only the ABSOLUTE negligible size is used. The relative negligable size is employed only to calculate the absolute negligible size, (if not input directly). If absolute negligible size is not specified, it is calculated as the product of relative negligible size and the minimum size (in x,y,z direction) of the analyzed problem. By default, relative negligible size is set to 1E-5.
NEGLIGIBLE_RESIDUAL *x* | NEGLIGIBLE _DISPLACEMENT *x*	Negligable values for norm of residual forces/displacements that can be ignored. By default they are set to 1.E-11. E.g. SET Absolute error Negligible_residual 0.1 Relative error Negligible_residual 0.2
NEGLIGIBLE_TIME_FRACTION *x*	Two time steps will be treated as different steps, if they apply at times differing more than *dt/x*, where *dt* is the current (minimal) time incerement. It is set by "SET" command for dynamic analysis, whilst default value 0.1 days is used for creep analysis.
NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF *x*	This coffecient is used to multiply element matrices and vector, if its DTIME_ELEMENT_AGE + DTIME_GROUP_AGE > time at the current age, see Table 6 and Table 53. By default it is equal o zero.
ITERATION [LIMIT} *n*	Limit on number of iterations within each step. E.g. ITERATION [LIMIT] *n*

```
&SOLUTION_METHOD
{ LINEAR |  NEWTON-RAPHSON | NEWTON-RAPHSON_AND_ LINE-SEARCH |
      ARC-LENGTH_AND_ LINE-SEARCH  |  { MODIFIED_NR | FULL_NR }  }+
      }
```

Table 16: &SOLUTION_METHOD sub-command parameters

Parameter	Description

NEWTON-RAPHSON	Use Newton Raphson nonlinear solver.
ARC-LENGTH	Use Arc Length nonlinear solver. Recommended for force loading up to peak load or behind, can scale (reduce) the load. Only for static analysis, i.e., not for probems involving time (transport, creep, nor dynamic analyses).
NEWTON-RAPHSON_AND_ LINE-SEARCH	Use Line Search nonlinear solver in combination with Newton-Raphson method.
ARC-LENGTH_AND_ LINE-SEARCH	Use Arc Length nonlinear solver in combination with Use Line Search nonlinear solver.
LINEAR	Use linear solver. (Note that geometrical non-linearity is disregarded and only linear material can be used).
MODIFIED_NR	Build stiffness matrix only in the 1st iteration and use it also for subsequent iteration of the step.
FULL_NR	Build new stiffness matrix in each iteration.

&PREDICTOR_TYPE:
{ ELASTIC_PREDICTOR | TANGENTIAL_PREDICTOR | SECANT_PREDICTOR }

Table 17: &PREDICTOR_TYPE sub-command parameters

Parameter	Description
ELASTIC_PREDICTOR	Elastic stiffness matrix shall be used to predict displacement increments from structural unbalanced forces. There are no additional parameters for this command. This is option is set by default
TANGENTIAL_PREDICTOR	Tangential stiffness matrix shall be used to predict displacement increments from structural unbalanced forces. There are no additional parameters for this command. By default elastic stiffness matrix is used.
SECANT_PREDICTOR	Secant stiffness matrix shall be used to predict displacement increments from structural unbalanced forces. There are no additional parameters for this command. By default elastic stiffness matrix is used

&UPDATE_DISPLS_STRATEGY:
{ UPDATE_IP_EACH_STEP | UPDATE_IP_EACH_ITERATION }

Table 18: &UPDATE_DISPLS_STRATEGY sub-command parameters

Parameter	Description
UPDATE_IP_EACH_STEP	Specify that material points, (i.e. integration points) should be updated at the end of each (converged) step, (i.e. load increment). It means that stress increments are calculated with respect to the beginning of step rather then previous iteration. It ensures stress increments to be calculated always from "converged" conditions, however as stress increments do not converged to zero (within current step), this approach is more demanding on evaluation of constitutive equations
UPDATE_IP_EACH_ITERATION	Specify that material points, (i.e. integration points) should be updated at the end of each iteration within a load increment). It means that stress increments are calculated with respect to the beginning of previous iteration. By default material points are updated with respect to loading increments, i.e. steps. See also SET UPDATE_IP_EACH_STEP

&ARC_LENGTH_PARAMS:
{ &ARC_LENGTH_TYPE | &CONSTRAINT_LENGTH_CONTROL |
&LOAD_DISPLACEMENT_RATIO | &LOCATION_PARAMS

Table 19: &ARC_LENGTH_PARAMS sub-command parameters

Parameter	Description
&ARC_LENGTH_TYPE	Set type of Arc Length method and associated constrain.
&CONSTRAINT_LENGTH_CONTROL	Set several parameters that control Arc Length method
&LOAD_DISPLACEMENT_RATIO	Control load – displacement scale for calculating Arc Length constrain.
&LOCATION_PARAMS	Set location where the Arc Length *step_length* and/or Line Search energy criterion should be calculated.

&ARC_LENGTH_TYPE:
{ CRISFIELD | NORMAL_UPDATE | CONSISTENTLY_LINEARISED |
EXPLICIT_ORTHOGONAL}

Table 20: &ARC_LENGTH_TYPE sub-command parameters

Parameter	Description
CRISFIELD	Crisfield variant of constant step length (including loading space) is to be used.
NORMAL_UPDATE	Updates of displacements within iteration kept normal to displacements within the step.
CONSISTENTLY_LINEARISED	Keeps constant projection of step length in the current iteration to direction of the previous iteration. It is linearized form of EXPLICIT_ORTHOGONAL method.
EXPLICIT_ORTHOGONAL	Keeps constant step length. Unlike CRISFIELD method, it is based on goniometric relationships, thus avoiding solving quadratic equation and difficulty with picking the correct root. From the mechanical point of view it poses identical constraint as CRISFIELD method.

&CONSTRAINT_LENGTH_CONTROL:
{ &ARC_LENGTH_BASE_STEP_LENGTH | &ARC_LENGTH_OPTIMISATION }

Table 21: &CONSTRAINT_LENGTH_CONTROL sub-command parameters

Parameter	Description
&ARC_LENGTH_BASE_STEP_LENGTH	Set base *step_length* .
&ARC_LENGTH_OPTIMISATION	Set the way how to optimize *step_length* in the current step based on base *step_length* and convergence performance in the previous step. The base *step_length* is defined by &ARC_LENGTH_BASE_STEP_LENGTH and by default it corresponds to *step_length* in the previous step.

&ARC_LENGTH_BASE_STEP_LENGTH
{ARC_LENGTH_PREVIOUS_STEP_LENGTH |
ARC_LENGTH_RESET_STEP_LENGTH | STEP_LENGTH *x* |
STEP_LENGTH_ONCE *x* | REL_STEP_LENGTH *x* |
REL_STEP_LENGTH_ONCE *x* | REL_REF_STEP_LENGTH *x* |
REL_REF_STEP_LENGTH_ONCE *x* | DLAMBDA_MIN *x* | DLAMBDA_MAX *x* | REF_DLAMBDA_MIN *x* | REF_DLAMBDA_MAX *x* |
MIN_STEP_LENGTH *x* | MAX_STEP_LENGTH *x* |
MIN_REL_STEP_LENGTH *x* | MAX_REL_STEP_LENGTH *x* |
MIN_REL_REF_STEP_LENGTH *x* | MAX_REL_REF_STEP_LENGTH *x* }

Table 22: &ARC_LENGTH_BASE_STEP_LENGTH &command parameters

Parameter	Description
ARC_LENGTH_ PREVIOUS_STEP_LENGTH	For the current step use base *step_length* (for possible optimization by &ARC_LENGTH_OPTIMISATION) from the previous step. In case of the 1st step, it acts according to ARC_LENGTH_RESET_STEP_LENGTH.
ARC_LENGTH_RESET_STEP_LENGTH	For the current step reset base *step_length*. The actual *step_length* is *step_length* resulting from applied load in the 1st iteration of the current step (for $\Delta\lambda = 1$). It is always calculated for the 1st step, 1st iteration.
STEP_LENGTH *x*	Set directly required step length to *x*. By default, it is initiated based on load increment, see ARC_LENGTH_RESET_STEP_LENGTH.
STEP_LENGTH_ONCE *x*	Same as the above but it is appkued only once.
REL_STEP_LENGTH *x* | REL_STEP_LENGTH_ONCE *x* | REL_REF_STEP_LENGTH *x* | REL_REF_STEP_LENGTH_ONCE *x*	Allows direct setting of $\Delta\lambda$ in the next step relative to previous or reference step length. It can be set only "ONCE", i.e. only in the next subsequent step or in all subsequent steps until a new relevant input. If *x*==-1, this input is ignored. By default, all these input valus are set to -1, i.e. they are ignored.
MIN_STEP_LENGTH *x* | MAX_STEP_LENGTH *x*	Set minimum and/or maximum value step length. If the *x* value is negative, this check is ignored. By default, *x*=-1. This input can overwrite DLAMBDA_MIN, DLAMBDA_MAX
MIN_REL_STEP_LENGTH *x* | MAX_REL_STEP_LENGTH *x*	Set minimum and/or maximum value of current step length related to the step length in the previous step. If the *x* value is negative, this check is ignored. By default, *x*=-1
MIN_REL_REF_STEP_LENGTH *x* | MAX_REL_REF_STEP_LENGTH *x*	Set minimum and/or maximum value of current step length related to the step length in firrst previous Arc-Length/ Line Srearch step. If the *x* value is negative, this check is ignored. By default, *x*=-1
DLAMBDA_MIN *x* | DLAMBDA_MAX *x*	Set minimum and/or maximum value of delta λ step load increment factor. If the *x* value is negative, this check is ignored. By default,

	x=-1. This input can be overwritten by MIN_STEP_LENGTH and MAX_STEP_LENGTH
REF_DLAMBDA_MIN *x* | REF_DLAMBDA_MAX *x*	Set minimum and/or maximum value of delta λ step load increment factor with respepect to reference load. If the *x* value is negative, this check is ignored. By default, *x*=-1. This input can be overwritten by MIN_STEP_LENGTH and MAX_STEP_LENGTH

&ARC_LENGTH_OPTIMISATION:
{ { ARC_LENGTH_CONSTANT |
ARC_LENGTH_VARIABLE_CONSERVATIVE_1/2 |
ARC_LENGTH_VARIABLE_CONSERVATIVE_1/4 |
ARC_LENGTH_VARIABLE_PROGRESSIVE } |
REFERENCE_NUMBER_OF_ITERATIONS }+

Table 23: &ARC_LENGTH_OPTIMISATION sub-command parameters

Parameter	Description
ARC_LENGTH_CONSTANT	For the current step use *step_length* unchanged from the previous step.
ARC_LENGTH_VARIABLE_CONSERVATIVE_1/2	Adjusts *step_length* for each load step based on the previous structural behavior: *step_length_new=* *pow(reference_number_of_iteration /last_number_of_iteration,1/2)*
ARC_LENGTH_VARIABLE_CONSERVATIVE_1/4	Adjusts *step_length* for each load step based on the previous structural behavior: *step_length_new=* *pow(reference_number_of_iteration last_number_of_iteration,1/4)*
ARC_LENGTH_VARIABLE_PROGRESSIVE	Adjusts *step_length* for each load step based on the previous structural behavior: *step_length_new=pow(last_number*

	_of_iteration/ reference_number_of_iteration,1/2)
REFERENCE_NUMBER_OF_ITERATIONS *n*	Set optimum number of iterations per load step to *n.* This value is used in Arc Length optimization of *step_length.* By default it is set to *n*=5.

&LOAD_DISPLACEMENT_RATIO:
{ LOAD_DISPLACEMENT_RATIO *x* |
LOADING_DISPLACEMENT_RATIO_CONSTANT |
LOADING_DISPLACEMENT_SCALE_CONSTANT |
LOADING_DISPLACEMENT_BERGAN_CONSTANT }

Table 24: &LOAD_DISPLACEMENT_RATIO sub-command parameters

Parameter	Description
LOAD_DISPLACEMENT_ RATIO *x*	Sets the parameter β_{ratio} to *x*. By default, it is 0.2.
LOADING_DISPLACEMENT_ RATIO_CONSTANT	The SW first (i.e. in the 1st load increment) calculates scaling factor $\beta=\beta_{ratio}\Delta\lambda/\Delta\|\|displacements\|\|$, where $\Delta\lambda=1$ and $\Delta displacements$ is derived from the loading increment. The calculated β is afterwards kept constant. The ratio $\Delta\|\|displacements\|\|/\Delta\lambda$ is called *bergan* coefficient.
LOADING_DISPLACEMENT_ SCALE_CONSTANT	Adjusts β(see the previous option) for each new load step as follows $\beta=\beta_{ratio} bergan_{last}$ This strategy tries to keep the same impact of changes happening in loading and geometric space.
LOADING_DISPLACEMENT_ BERGAN_CONSTANT	Adjusts β(see the previous option) for each new load step as follows $\beta=\beta_{last}\, bergan_{old} / bergan_{last}$ Subscript $_{old}$ stands for one before the $_{last}$ results. This strategy tries to keep the same ratio of influence of loading and geometric space.

&LOCATION_PARAMS:
LOCATION { NODE { AT *n* | FROM *n1* [TO *n2* [BY *n3*]] }+ DOF { AT *n* | FROM *n1*
[TO *n2* [BY *n3*]]}+ COEFF *x* | REMOVE}

Table 25: &LOCATION_PARAMS sub- command parameters

Parameter	Description
LOCATION	Specifies list of domains. Each from these domains contains list of structural DOFs and their coefficients used for calculation Arc-length step length.
REMOVE	It destroys list of domains and in the subsequent steps all structural DOFs will be accounted for.
NODE	It follows list of nodal intervals. Any number of intervals can be specified.
DOF	It follows list DOFs intervals. Any number of intervals can be specified.
AT n	Set location at node (or degree of freedom) n.
FROM n_1 [TO n_2 [BY n_3]]	Sets locations at nodes (or degrees of freedom) by interval. BY default $n_2 = n_1$ and $n_3 = 1$.
COEFF x	Weight factor for the specified DOF.

&LINE_SEARCH_PARAMS:
{ &LINE_SEARCH_ITERATION_CONTROL | &LIMIT_ETA_CONTROL |
REFERENCE_ETA x | UNBALANCED_ENERGY_LIMIT x |
&LOCATION_PARAMS }+

Table 26: &LINE_SEARCH_PARAMS sub-command parameters

Parameter	Description
&LINE_SEARCH_ ITERATION_CONTROL	Control several parameters for Line Search iteration process.
&LIMIT_ETA_ CONTROL	Set minimum and maximum value for η parameters etc.
REFERENCE_ETA x	Resets η to x.
UNBALANCED_ ENERGY_LIMIT x	Limit for relative work of out-of balanced forces within the "main" iteration. When satisfied, it stops Line search internal iteration loops. By default it is set to x=0.8 It says that Line search has by default reduce work of out-of balanced forces by 20%.

&LINE_SEARCH_ITERATION_CONTROL:
{ LINE_SEARCH_WITHOUT_ITERATIONS | {
LINE_SEARCH_WITH_ITERATIONS | LINE_SEARCH_ITERATION_LIMIT
n }+ }

Table 27: &LINE_SEARCH_ITERATION_CONTROL sub- command parameters

Parameter	Description
LINE_SEARCH_WITHOUT_ ITERATIONS	Do not carry internal Line search iteration loop within each "main" iteration.
LINE_SEARCH_WITH_ ITERATIONS	Carry on internal Line search iteration loop within each "main" iteration.
LINE_SEARCH_ ITERATION_LIMIT *n*	Set line-search iteration limit. Default value is 3 iterations.

&LIMIT_ETA_CONTROL:
{ LIMIT_ETA | MINIMUM_ETA *x* | MAXIMUM_ETA *x* }+

Table 28: &LIMIT_ETA_CONTROL sub-command parameters

Parameter	Description
LIMIT_ETA	Apply limit value for $\eta = \eta_{min} \ldots \eta_{max}$. Only η multiple of coordinate changes are applied to the next iteration. It is set automatically when issuing either of the commands MINIMUM_ETA *x* and/or MAXIMUM_ETA *x*.
MINIMUM_ETA *x*	Sets $\eta_{min} = x$. By default it is set to x=0.1
MAXIMUM_ETA *x*	Sets $\eta_{max} = x$ By default it is set to x=10.

&OPTIMIZE_PARAMS:
OPTIMIZE [BAND] WIDTH {SLOAN | GIBBS-POOLE | NONE}

Table 29: &OPTIMIZE_PARAMS sub-command parameters

Parameter	Description
BAND	Dummy keyword.
WIDTH	Activates bandwidth minimisation and set default method to SLOAN.
SLOAN	Use Sloan's algorithm for optimization process
GIBBS-POOLE	Use Gibbs-Poole's algorithm for optimization process
NONE	Don't optimize band-width. This is default setting.

&SERIALIZE_PARAMS:
SERIALIZE [MODEL] [STATE] { { BASICS | [AND] NODAL |
[AND] ELEMENT | ALL }+ | { DEEP | STANDARD } }+

Table 30: &SERIALIZE_PARAMS sub-command parameters

Parameter	Description
MODEL	Dummy keyword
BASICS	Stores just basic information about the model like number of nodes, materials etc.
AND	Dummy keyword
NODAL	Stores data related to nodes of the model, (e.g. displacements)
ELEMENT	Stores data related to elements of the model, (e.g. strains)
ALL	Same as coding BASICS NODAL ELEMENT; stores all data
STATE	Dummy keyword
STANDARD	Standard serialization depth, i.e. only essential object data is serialized.
DEEP	All data within objects are serialized.

&FATIGUE_PARAMS:
{ FATIGUE_TASK *f_task* | FATIGUE_CYCLES *f_cycles* |
FATIGUE_MAX_FRACT_STRAIN_MULT *f_mult* |
FATIGUE_COD_LOAD_COEFF *f_codcoeff* }+
These parameters only have influence on materials that support fatigue, see the description of the CC3DNonLinCementitious2Fatigue material.

Table 31: & FATIGUE_PARAMS sub-command parameters

Parameter	Description
FATIGUE_TASK *f_task*	The FATIGUE_TASK parameter determines the operation (fatigue calculation phase) for the analysis step. 0 - nothing to do with fatigue 1 - store base stress 2 - reset FATIGUE_MAX_FRACT_STRAIN 4 - calculate fatigue damage induced by FATIGUE_CYCLES load cycles. The calculated damage is added to FATIGUE_MAX_FRACT_STRAIN. 8 - apply the fatigue damage stored in FATIGUE_MAX_FRACT_STRAIN, multiplied by FATIGUE_MAX_FRACT_STRAIN_MULT To combine operations in one analysis step, the values are added together (combined by binary or), e.g. storing base stress and resetting fatigue max.fract.strain are requested by the value 3.

	Typically, FATIGUE_TASK is set to 3 (store base stress + reset fatigue max.fract.strain) before the first step of the load to be cycled and to 0 for the rest steps of the fatigue load, then to 12 (calculate + apply fatigue damage) before the first step applying the damage and to 8 for the rest damage application steps, then to 0 for any following static analysis
FATIGUE_CYCLES *f_cycles*	The number of cycles is determined by the FATIGUE_CYCLES parameter in the solutions parameters, set before the load step when the fatigue damage is calculated. The value of 0 means a non-cycling load.
FATIGUE_MAX_FRACT_STRAIN_MULT *f_mult*	Multiplier for max.fract.strain induced by fatigue, e.g. 0.2 if the damage is applied in 5 analysis steps
FATIGUE_COD_LOAD_COEFF *f_codcoeff*	Multiplier for the influence of the cycling crack opening displacements when calculating fatigue damage. Equivalent to changing the KSI_FATIGUE material parameter, but can be set separatly for each fatigue load

&CREEP_ANALYSIS_PARAMS:
{ SAMPLE_TIMES_PER_DECADE *ndecl* | RETARD_TIMES_PER_DECADE *ndecl_retard* | STOP_TIME *execution_stop_time* | FIRST_DTIME *first_time_increment* | {MP_METHOD | CS_METHOD} }+

Table 32: & CREEP_ANALYSIS_PARAMS sub-command parameters

Parameter	Description
SAMPLE_TIMES_PER_DECADE *ndecl*	Number of integration times per $\log_{10}$ of time span. Note that this command affects generation of integration (sample) times by the &CREEP_STEP_DEFINITION sub-command. Hence, the *ndecl* parameter must be set before the &CREEP_STEP_DEFINITION sub-command. This parameter defines the number of time steps, the program will use to integrate the structural behavior. Creep or other nonlinear effects will cause a redistribution of stresses inside the structure. In order to properly capture such processes a sufficiently small time steps are needed. This time spacing is defined by the number of sample times. Its definition depends on the type of the analyzed structure as well as on the choice of time units. For typical reinforced concrete structures and for the time unit being a day, it is recommended to set this parameter to 2. This will mean that for each load interval longer then 1 day, two sub-steps will be added. For a load that is interval longer then 10 days, 4 sub-steps will be added. For an interval

	longer than 100 days, it will be 6 sub-steps. Default value: 2.
RETARD_TIMES_PER_ DECADE *ndecl_retard*	Number of retardation times per $\log_{10}$ of time span. Note that this command affects generation of retardation times by the &RETARDATION command and hence it must be set beforehand. Alternatively, this value can be set directly in &RETARDATION. Example: If number of retardation times is set to 2, the creep law will be approximated by two points for each time unit in the logarithmic scale. This means two approximation points will be used for the time interval between 0 - 1 day, two points for the interval 1 - 10 days, then two points for 10 - 100 days, etc. So the proper values will depend on the choice of time units. If the time unit is a day, the recommended value is 1 - 2. Default value: 1.
FIRST_DTIME *first_time_increment*	Time increment for the first step of steps' serie that is automatically generated after issuing a STEP definition command, (without the option "RESUME_AT" option). Default value: 0.1 day Exanple SET FIRST_DTIME 0.08 ;
STOP_TIME *execution_stop_time*	Time at which the execution should stop [days]. This value must be input at leatest (or anywhere earlier) just before executing a step that should by stopped by this command. If it has not been specified, ATENA assumes STOP_TIME equal to *time_end* from the &retardation times command. The inputted value of STOP_TIME gets inserted in (automaticly generated) series of sample times but the higher sample times are not modified. Default value: 0 [days]
MP_METHOD♥ | CS_METHOD	Creep analysis method. CS_METHOD uses simplified approach, in which temperature and humidity in a material point depend only on cross sectional shape and average exterior temperature and humidity. The MP_METHOD uses accurate temperature and humidity at each structural material point and therefore it need additional analysis of moisture and heat transfer. Currently only CS_METHOD is supported. Default value: CS_METHOD.

&DYNAMIC_ANALYSIS_PARAMS:
{ STOP_TIME *execution_stop_time* | LAST_TIME *last_time* |
{NEWMARK_METHOD | HUGHES_ALPHA_METHOD |
WILSON_THETA_METHOD | MODIFIED_ WILSON_THETA_METHOD } }+

Table 33: & DYNAMIC_ANALYSIS_PARAMS sub-command parameters

Parameter	Description
STOP_TIME *execution_stop_time*	Time at which the execution should stop. If it is not defined, (i.e. *execution_stop_time=0*), then it is assumed *execution_stop_time=last_time.* Default value: 0
LAST_TIME *last_time*	Last time of the whole analysis. Default value: 0
NEWMARK_METHOD | HUGHES_ALPHA_METHOD | WILSON_THETA_METHOD | MODIFIED_WILSON_THETA_METHOD	Dynamic analysis method to be used. Default value: HUGHES_ALPHA_METHOD

&MAX_REF_IDS:
MAX_REF_ID { { MACRO_NODES_SMART_IDS_MAP |
MACRO_ELEMENTS_SMART_IDS_MAP |
MATERIALS_SMART_IDS_MAP | LOAD_CASES_SMART_IDS_MAP |
STEPS_SMART_IDS_MAP | FUNCTIONS_SMART_IDS_MAP |
GEOMETRIES_SMART_IDS_MAP | ELEMENT_TYPES_SMART_IDS_MAP
| NODES_SMART_IDS_MAP | ELEMENT_GROUPS_SMART_IDS_MAP |
ELEMENTS_SMART_IDS_MAP [FOR] [GROUP] *group_id* } *max_ref_id* }+

Table 34: & MAX_REF_IDS sub-command parameters

Parameter	Description
{MACRO_NODES_SMART_IDS_MAP | ….. | ELEMENTS_SMART_IDS_MAP [FOR] [GROUP] *group_id* } *max_ref_id* }+	Set maximum reference id for a specified data entity. The given value is typically used by the internal ATENA generator, when a request for next reference id is processed. Note that if it is specified max. ref_id for elements, i.e. the command ELEMENTS_SMART_IDS_MAP [FOR] [GROUP] *group_id* } *max_ref_id* , then the group id must be id of an already input element group. Any "forwards" specification is not allowed here. Default value: 50000 (for all queues).

3.4.14 The Command &UNITS

Syntax:
&UNITS:
UNITS { { &FORCE_UNITS | &TEMPERATURE_UNITS | &LENGTH_UNITS |
&MASS_UNITS | &TIME_UNITS } *"units"* }+

&FORCE_UNITS:

FORCE { N | kN | MN }

&TEMPERATURE_UNITS:
TEMPERATURE { ^{o}C | ^{o}F | ^{o}K | C | F | K }

&LENGTH_UNITS:
LENGTH { MM | M | IN }

&MASS_UNITS:
MASS { KG | TON | LB}

&TIME_UNITS:
TIME { sec | day }

Table 35: Description of available program units

Unit type	Unit type description	Supported Units
Force units	F	N, kN, MN, kips, lbf
Length units	L	mm, m, in
Temperature	T	^{o}C, ^{o}F, ^{o}K, C = ^{o}C, F=^{o}F, K=^{o}K,
Mass	M	kg, ton, lb
TIME	T	sec, day

Table 36: Description of derived units

Unit type	Unit type description	Supported units	Formula based on basic units (see { sec | day } Table 35)
Stress, pressure	S	Pa, kPa, MPa, psi, ksi	F/l^2

In some parts of the manual, the default values of certain material parameters are specified. If the parameter is not specified in the input manual, the default value is used. The used default value depends of coarse on the selected unit set. This means that the program converts the default value to the selected unit set. The conversion is done with the help of the following factors, whose value depends on the selected units.

Table 37: Value of factor f_F for the conversion of force default values

Jednotka	Faktor f_F
N	1 000 000
KN	1 000

MN	1
lbf	224809.024733489

Table 38: Value of factor f_l for the conversion of length default values

Jednotka	Faktor f_l
mm	1 000
cm	100
m	1
in	39.3700787401575

Table 39: Value of factor f_s for the conversion of stress units

Jednotka	Faktor f_s
Pa	1 000 000
kPa	1000
MPa	1
psi	145.037680789469
ksi	0.145037680789469

3.5 Topology Definition

3.5.1 The Command &JOINT

This command adds new finite element joints to the model.

Syntax:

&JOINT:
JOINT { &COORDINATES_SPEC }+

&COORDINATES_SPEC:
COORDINATES { [ID] *n* [NCOORDS] *ncoords* [X] { *x* }ncoords }+

Table 40: &JOINT command parameters.

This command is used to set model joint coordinates. Each joint coordinate should be on a separate line, e.g. [ID] *n* [X] x_1 x_2 x_3

If *ncoords* is not specified, it is by default equal to problem dimension, see &TASK.

3.5.2 The Command &LOCAL

This command specifies list of finite element joints, whose degree of freedom should be treated in element local coordinate system.

Syntax:
&LOCAL:
LOCAL DOFS JOINTS { *n* }+

Table 41: &LOCAL command parameters

Parameter	Description
LOCAL DOFS JOINTS { *n* }+	List of nodes with local degree of freedom. E.g. LOCAL DOFS JOINTS n_1, n_2, n_3, ... n_k

3.5.3 The Command &GEOMETRY

Syntax:
&GEOMETRY:
GEOMETRY ID *n* [NAME "*geometry name*"] TYPE &GEOMETRY_SPEC

Table 42: &GEOMETRY command parameters

Parameter	Description
ID	Geometry identification, e.g. ID *n*
NAME	User defined geometry name in quotes, also for identification. E.g.: NAME „*geometry name*“
TYPE	Geometry type in quotes and other geometry type dependent parameters, see &GEOMETRY_SPECIFICATION.

&GEOMETRY_SPEC:
{ &2D_GEOMETRY_SPEC | &3D_GEOMETRY_SPEC |
&TRUSS_GEOMETRY_SPEC | &SPRING_GEOMETRY_SPEC |
&EXTERNAL_CABLE_GEOMETRY_SPEC | &BEAM_GEOMETRY_SPEC |
&LAYERED_SHELL_GEOMETRY_SPEC | &BEAM_3D_SPEC |
&BEAM_1D_SPEC }+

&2D_GEOMETRY_SPEC:

{ "2D" THICKNESS *x* | { REF_V1_IDS *node1 node2* | REF_V1_VECTOR *x y* [*z*] | [DIAMETER *dia*]}+

Table 43: &2D_GEOMETRY_SPEC sub-command parameters

Parameter	Description
THICKNESS	Thickness of the two-dimensional object. E.g.: THICKNESS *x*
REF_V1_IDS *node1 node2*	Define position of an arbitrary vector $\bar{\tilde{v}}1$ used throughout definition of local coordinate system for plane 3D and 2.5D elements. The vector is set by coordinates of finite element nodes *node1* (tail) and *node2* (head). If it is input, it's projection into the element plane will yield X local coordinate axis. Otherwise, the procedure of establishing X local is written in the Atena theoretical manual.
REF_V1_VECTOR *x y z*	Same as the above, but the arbitrary vector is input directly.
DIAMETER *dia*	The diameter *dia* is used to specify diameter of rebars used in CCDiscretePlaneReinforcementME macro element. It is used solely for reinforcement corrosion analysis and is defined in Atena length units. By default, dia=28mm.

&3D_GEOMETRY_SPEC:
"3D" [{ REF_V1_IDS *node1 node2* | {REF_V1_VECTOR *x y* [*z*]}]

Table 44: &3D_GEOMETRY_SPEC sub-command parameters

Parameter	Description
REF_V1_IDS *node1 node2*	Define position of an arbitrary vector $\bar{\tilde{v}}1$ used throughout definition of local coordinate system for 3d gap elements. The vector is set by coordinates of finite element nodes *node1* (tail) and *node2* (head). If it is input, it's projection into the element plane will yield X local coordinate axis. Otherwise, the procedure of establishing X local is written in the Atena theoretical manual.
REF_V1_VECTOR *x y z*	Same as the above, but the arbitrary vector is input directly.

&TRUSS_GEOMETRY_SPEC:
"*Truss"* AREA *x*

Table 45: &TRUSS_GEOMETRY_SPEC sub-command parameters

Parameter	Description
AREA	Cross sectional area of a truss object. E.g.: AREA *x*

&SPRING_GEOMETRY_SPEC:
"*Spring*" { { AREA | THICKNESS} *x* | { LOCAL | GLOBAL } [SPRING]
DIRECTION { *x* }$_{\text{ncoords}}$ }$_2$

Table 46: &SPRING_GEOMETRY_SPEC sub-command parameters

Parameter	Description
AREA | THICKNESS	Cross-sectional area or spring "thickness" of a point spring or line spring object respectively. Default = 1.0. E.g.: AREA *x*
[{ LOCAL | GLOBAL }] [SPRING] DIRECTION	Spring direction in local or global coordinate system. Local coordinate system is applicable only for line or plane springs. By default, global coordinate system is assumed. *ncoords* coordinates defines direction vector, (*ncoords* equals to problem dimension from &TASK). The direction vector represents not only spring direction, but also its length that is significant in case of geometrically nonlinear analyses. E.g.: [LOCAL] DIRECTION x_1 x_2

&EXTERNAL_CABLE_GEOMETRY_SPEC:
"Cable" { AREA *x* | [FRICTION] COEFFICIENT *x* | [FRICTION] CONSTANT *x* |
RADIUS *x* | BOND_COHESION *bond_cohesion* | WOBBLE_COEFFICIENT
wobble_coeff | SLIP_UNLOAD_COEFF *slip_coeff_unload* | FUNCTION [SLIP]
slip_function_id | FUNCTION LOCATION *location_function_id* |
{FIXED|PRESTRESSED} [START | END | BOTH] | PERIMETER *x* |
PRECISION [FACTOR] *x* | DAMPING [FACTOR] *x* | RESET_SLIPS}$_{11}$

Table 47: &EXTERNAL_CABLE_GEOMETRY_SPEC sub-command parameters

Parameter	Description
AREA	Cross-sectional area or spring "thickness" of a point spring or line spring object respectively. Default = 1.0. E.g.: AREA *x*
COEFFICIENT $friction_{lin}$ CONSTANT $friction_{const}$ RADIUS *radius*	Parameters defining friction force at a deviator. $F_{frict} = ((1.- a) \max(F_{right}, F_{left}) + b)$, where For $friction_{lin} > 0$ $a = \exp(-\text{abs}(\varphi_{left} - \varphi_{right}) * friction_{lin} * perimeter)$ else $a = -friction_{lin}$ For $friction_{cons} > 0$

	b = abs(φ_{left} - φ_{right}) * $friction_{const}$ * *radius* * *perimeter* else b= -$friction_{const}$ φ = angle of cable (in radians), F = force in cable, *radius* = radius of deviator defined by parameter RADIUS, *perimeter* = perimeter of the bar $friction_{lin}$ = friction coefficient defined by parameter [FRICTION] COEFFICIENT $friction_{lin}$, $friction_{const}$ = friction coefficient defined by parameter. It has units of stress. Note that this value is further multiplied function *location_function_id.* CCExternalCable elements multiply it also by function *slip_function_id.* CCBarWith...Bond elements do NOT multiply it by function *slip_function_id.*
BOND_COHESION *bond_cohesion* SLIP_UNLOAD_COEFF *slip_coeff_unload*	Parameters defining cohesion for CCBarWith...Bond elements. *bond_cohesion* defines cohesion stress between the bar and a material, into which the bar is embedded [stress units]. It corresponds to σ_{c0} in the ATENA Theory manual. Note that this value is further multiplied by two coefficients. The first one is used for both load and unload regimes and it is defined by function *location_function_id.* The second one is used for load and unload regimes for all elements except elements with memory bond. It is defined by function *slip_function_id.* Memory bond elements uses for unloading *slip_coeff_unload*. Example: BOND_COHESION *bond_cohesion*
WOBBLE_COEFFICIENT *wobble_coeff*	*wobble_coeff* defines cohesion stress between the bar and a material, into which the bar is embedded [stress units]. It corresponds to $-f_w$ in the ATENA Theory manual.

FIXED [START | END | BOTH]	If specified, the starting node and/or the end node of the reinforcement bar is fixed with respect to the concrete, i.e. it cannot slip. By default, if FIXED command is not used, it can slip everywhere.
PRESTRESSED [START | END | BOTH]	Similar info as that above. PRRESTRESSED START means the same as FIXED LEFT etc.
FUNCTION [SLIP] *slip_function_id*	Id of a function, by which all the coefficients are multiplied, i.e. $friction_{lin}$, $friction_{const}$. If not specified, no multiplication occurs. The functional argument is current (total) deviator slip.
FUNCTION LOCATION *location_function_id*	Id of a function, by which all the coefficients are multiplied, i.e. $friction_{lin}$, $friction_{const}$. If not specified, no multiplication occurs. The functional argument is distance between the 1st node and the current node, for which the slip parameters are calculated. For cables, the two current friction values are calculated $friction_{const\_current} = friction_{const}\ fs(s)\ fd(dist)$, and $friction_{lin\_current} = friction_{lin}\ fs(s)\ fd(dist)$, where *fs(s)* stands for FUNCTION SLIP, and *fd(dist)* for FUNCTION LOCATION. If a function is not defined, a constant value of 1.0 is considered at its place. For bar with bond, only the first formula is used, defining the actual cohesion (i.e., the maximum possible bond stress): $C_{current} = friction_{const}\ fs(s)\ fd(dist)$ is used.
PERIMETER *x*	Perimeter of the reinforcement. This value is used only for CCBarWithBond / CCBarWithMemoryBond elements. Default: *x*=1 [m]
FRICTION UNLOAD COEFFICIENT x	This parameter is applicable only for the CCBarWithMemoryBond elements. It determines the maximum bond stress for the unloading branch, i.e., to which value the max. bond stress drops after the bond stress sign changes (by default, the bond strength – bond slip envelope is followed during unloading as defined for the loading). Admissible values: $\tau_{res} \le x \le \tau_{max}$ [stress units], where τ_{res} is the residual bond stress (last value from the bond strength - bond slip function) and τ_{max} the maximum bond stress (max. value from the bond strength – bond slip

	function).
PRECISION [FACTOR] *x*	Process of internal iterations will stop, if $\frac{x\sqrt{\sum(\Delta us_i)^2}}{l} \le error_{rel.displ}$, where Δus_i is change of slip at cable node *i* within the last iteration and $error_{rel.displ}$ is allowed relative displacement error of the problem, see &CONVERGENCE_CRITERIA. Default value: *x*=100000.
DAMPING [FACTOR] *x*	Factor for damping during the process of iterative calculation of nodal slips. The slips are updated as follow $us_i^{(j)} = us_i^{(j-1)} + x\ \Delta us_i^{(j)}$, where *j* indicates iteration id and *i* is cable node id. Default value: *x*=1
RESET_SLIPS	Reset total slips to zero. This may be needed e.g. after filling channels of the cables and redefining the slip function.

&BEAM_GEOMETRY_SPEC:
"Beam" { AREA *x* | [MOMENT] INERTIA_Y *x* | [MOMENT] INERTIA_Z *x* | [MOMENT] POLAR *x* | [MOMENT] TORGUE *x* | [MOMENT] SHEAR_Y *x* | [MOMENT] SHEAR_Z *x* | [WINKLER] [COEFFICIENT] C_1_X *x* | [WINKLER] [COEFFICIENT] C_1_Y *x* | [WINKLER] [COEFFICIENT] C_1_Z *x* | [PASTERNAK] [COEFFICIENT] C_2_X *x* | [PASTERNAK] [COEFFICIENT] C_2_Y *x* | [PASTERNAK] [COEFFICIENT] C_2_Z *x* | [LOCAL] [Z] [AXIS] DIR_X *x* | DIR_Y *x* | DIR_Z *x* | [{SIZE_LOCAL_Y | WIDTH} *x*] | [{SIZE_LOCAL_Z}|{HEIGHT} *x*] | [{KIRCHHOFF}|{MINDLIN}|{TIMOSHENKO}|{TIMOSHENKO_CSF}] | [REDUCE_TM_STIFF] | [REDUCE_MT_STIFF] | [RO_N *x*] | [EFF_WIDTH_FACTOR *x*] | [EFF_HEIGHT_FACTOR *x*] | [UPDATE_BEAM_DIR] | [MAX_NUMBER_OF_ITERATIONS_FOR_REDUCE_FORCES *n*] | [MAX_ERROR_FOR_REDUCE_FORCES *x*] | S_MIN *s_min* S_MAX *s_max* T_MIN *t_min* T_MAX *t_max*| [BARS NUMBER *n* { MATERIAL *n* BAR_AREA *x* BAR_LOCAL_Y *x* BAR_LOCAL_Z *x*}*n*] }

Table 48: &BEAM_GEOMETRY_SPEC sub-command parameters

Parameter	Description
AREA	Cross-sectional area of a beam object. Default = 1.0. E.g.: AREA *x*
INERTIA_Y	Cross-sectional inertia moment of a beam object with respect to local Y-axis. E.g.: INERTIA_Y *x*
INERTIA_Z	Cross-sectional inertia moment of a beam object with respect to local Z-axis.

	E.g.: INERTIA_Z *x*
POLAR	Cross-sectional polar moment of a beam object with respect to local X-axis. E.g.: POLAR *x*
TORGUE	Cross-sectional moment of a beam object in torque. E.g.: TORGUE *x*
SHEAR_Y	Cross-sectional shear moment of a beam object with respect to local Y-axis. E.g.: SHEAR_Y *x*
SHEAR_Z	Cross-sectional shear moment of a beam object with respect to local Y-axis. E.g.: SHEAR_Z *x*
C_1_X	Winkler (or C_1 Pasternak) coefficient with respect to local X-axis. E.g.: C_1_X *x*
C_1_Y	Winkler (or C_1 Pasternak) coefficient with respect to local Y-axis. E.g.: C_1_Y *x*
C_1_Z	Winkler (or C_1 Pasternak) coefficient with respect to local Z-axis. E.g.: C_1_Z *x*
C_2_X	C_2 Pasternak coefficient with respect to local X-axis. E.g.: C_2_X *x*
C_2_Y	C_2 Pasternak coefficient with respect to local Y-axis. E.g.: C_2_Y *x*
C_2_Z	C_2 Pasternak coefficient with respect to local X-axis. E.g.: C_2_Z *x*
DIR_X	X coordinate of a vector defining Z-axis of beam truss local coordinate system. Note that X local coordinate axis corresponds to beam direction and Y local axis is perpendicular to X and Z. E.g. DIR_X *x*
DIR_Y	Y coordinate of a vector defining Z-axis of beam truss local coordinate system. E.g. DIR_Y *x*
DIR_Z	Z coordinate of a vector defining Z-axis of beam truss local coordinate system.

	E.g. DIR_Z *x*
{SIZE_LOCAL_Y | WIDTH} *x*	Cross sectional width in direction of the local Y axis. Either of the two keywords can be used. E.g. WIDTH 0.25
{SIZE_LOCAL_Z}|{HEIGHT} *x*]	Cross sectional height in direction of the local Z axis. Either of the two keywords can be used. E.g. HEIGHT 0.25
KIRCHHOFF}|{MINDLIN}|{TIMOSHENKO}|{TIMOSHENKO_CSF}	Definition of which modification of the beam FE model should be used. By default, TIMISHERNKO element is selected. It is the only one element that supports nonlinearity. The others ignore it.
{REDUCE_TM_STIFF} | { REDUCE_MT_STIFF} {REDUCE_TM_COEFF | REDUCE_TM_COEFF *x*}	Flag for simulating process of material cracking. If it is set on, flexural and bending stiffness of the beam element is reduced by *x*. By default, it is off, i.e. full stiffness is applied. Default value of the reduction coefficient is 0.5, i.e. 50% reduction is used. Either of the two keywords can be used.
RO_N *x*	Coefficient for buckling length of comperessed columns. By default it is 1. E.g. RO_N 0.5
EFF_WIDTH_FACTOR *x*	Coefficient for buckling widtf of comperessed columns' cross section. By default it is 1. E.g. EFF_WIDTH_FACTOR 0.5
EFF_HEIGHT_FACTOR *x*	Coefficient for buckling height of comperessed columns' cross section. By default it is 1. E.g. EFF_HEIGHT_FACTOR 0.5
UPDATE_BEAM_DIR	Flag for updating beam's position already during iterations with a load step. By default it is updated only at e ach step.
MAX_NUMBER_OF_ITERATIONS_FOR_REDUCE_FORCES *n*	Maximum number of iterations for establishing force/moment equilibrium. Such procedure is needed typically after any of beam's nodal forces/moments have been reduced due to material nonlinearity. By default 30 iterations are allowed.
MAX_ERROR_FOR_REDUCE_FORCES *x*	Acceptable relative error for the iteration process described above. By default the value 0.01 is used.
S_MIN *s_min* S_MAX *s_max* T_MIN *t_min* T_MAX *t_max* BARS NUMBER *n* { MATERIAL *n* BAR_AREA *x* BAR_LOCAL_Y *x* BAR_LOCAL_Z *x*}*n*]	Definition of reinforcement bars in the cross section. First number of bars is read and then for each bar its material, area and coordinates are inputed. Note that all the values are specified in isoparametric coordinate system, i.e. in coordinates <*s_min..s_max*>, (for direction of the cross sectional width) and <*t_min…t_max*>,m(for height). By default, these intervals are set to <-1..1>, which corresponds to isoparametric coordtinates. If the intervals <*0..width*>,

	<0..height> are use. the the bar areas and coordinates are input in real coordina system with origin in the left bottom corner.

&LAYERED_SHELL_GEOMETRY_SPEC:

"*LayeredShell*" { DETECT_DEPTH {DETECT_VECTOR *x1 x2 x3* } | { REF_V1_IDS *node1 node2* | REF_V1_VECTOR *x y z* } | INTERFACE *interface_nodes_list* | [SOLID | REINFORCEMENT] LAYER *n* [{ [MATERIAL *mat_id*] [THICKNESS *thick*] [POSITION *pos*] DIAMETER *dia*]} | {SAME_AS *layer_id* }| REF_THICK *x* | IGNORE_REF_THICKNESS | { REDUCE_TAU_XZ_YZ | REDUCE_TAU_XY | FULL_TAU} }+ THICKNESS_EQN " *eqn_string"*

[REDUCE_TAU_XY] [REDUCE_TAU_XZ] [FULL_TAU]	Reduce shears by the factor 0.85.

Table 49: &LAYERED_SHELL_GEOMETRY_SPEC sub-command parameters

Parameter	Description
SOLID | REINFORCEMENT	The data that follow specify a solid, (i.e. concrete) or reinforcement, (i.e. steel) layer.
LAYER *n*	Id of an input layer.
[MATERIAL *mat_id*] [THICKNESS *thick*] [POSITION *pos*] [DIAMETER *dia*]	Parameters specification for the layer *n* . Material specification: Material type at an integration point can be defined as follows, (ordered in terms of priority): 1/ For each integration point separately; refer to &ELEMENT_MATERIALS, 2/ By layers, i.e. all IPs within the layer *n* share the same material *mat_id* . This achieved this subcommand using MATERIAL *mat_id* , 3/ Use a default material defined by element group definition command, refer to &ELEMENT_GROUP. Layer thickness *thick*: Layer thickness (for both solid and reinforcement layers) is defined in term of normalized layer coordinates η. Top and bottom shell surfaces have coordinates η=1 and η=-1, respectively. Total shell thickness is thus 1- (-1)=2, with respect to which all individual layer thickness is scaled. If some solid layers have zero thickness, it is automatically generated as (2.- sum (all solid layers non-zero thickness)) / number of solid layers with zero thickness.

	If total sum of solid layers thickness does not equal to 2., all input *thick* and *pos* parameters (for both solid and reinforcement layers) are scaled appropriately. Layer position *pos*: It specifies position of the reinforcement layer *n*. Again, the normalized layer coordinate η is used, see above. Note that the parameter applies only to reinforcement layers. Solid layers do not use the *pos* parameter, as it is assumed that they are located from bottom (layer 1) to top (the last solid layer) of the shell. The position is thus defined by their thickness. The diameter *dia* is used to specify diameter of rebars used in a particular reinforcement layer. It is used solely for reinforcement corrosion analysis and is defined in Atena length units, (not in s,t coordinates like e.g. THICKNESS, POSITION...!. By default, dia=28mm.
SAME_AS *layer_id*	Specifies that the layer *n* has the same properties as a previously defined layer *layer_id*
DETECT_DEPTH {DETECT_VECTOR *x1 x2 x3* }	Detect depth of shell elements and reorder element's incidences. If DETECT_VECTOR is not specified, the depth is chosen to comply with the smallest dimension of the element. Otherwise it is chosen to have the smallest angle with the given vector {*x1, x2, x3*}.
REF_V1_IDS *node1 node2*	Define position of an arbitrary vector $\bar{\bar{v}}1$ used throughout definition of a shell local coordinated system, see the Atena Theory Manual. The vector is set by coordinates of finite element nodes *node1* (tail) and *node2* (head). By default, this input need not be specified. In such a case, Atena kernel will construct $\bar{\bar{v}}3$ using the default definition from the Atena Theoretical Manual .
REF_V1_VECTOR *x y z*	Same as tha above, but the arbitrary vector is input directly.
REF_THICK *x*	Reference thickness used to transform normalized layer coordinates to real coordinates. By default, this value is not specified and in this case actual shell thicknesses at integration points are used instead. This input is particularly useful, if a reinforcement layer is placed at constant distance from the shell bottom or top surface, whereby the shell real thickness is variable.
IGNORE_REF_THICKNESS	This flag can be input for each reinforcement layer. If it is specified, the consequences implied by inputting above

	REF_THICK *x* are for this particular reinforcement layer ignored.
INTERFACE *interface_nodes_list*	Name of list that includes nodal ids, where all 6 shell DOFs should be retained. Use this feature to connect shell elements with other solid elements, e.g. bricks.
{ REDUCE_TAU_XZ_YZ | REDUCE_TAU_XY | FULL_TAU}+	Reduce the specified shear(s) by 1/6 of its original value to compensate for constant shear strain thru cross section. By default, no reduction is carried out, (recommended). . (Ahmad elements use always full shear strains without any reduction).
THICKNESS_EQN " *eqn_string*	String containing equation to caculate shell's thickness. It can conation placeholders "x", "y", "z" that are replaces by actual shell coordinates. Example: THICKNESS_EQN "0.2+x*0.001+y*0.002"

For explanation of transformation between isoparametric and global coordinate system see BEAM_1D_GEOMETRY_SPEC.

&BEAM_3D_GEOMETRY_SPEC:
"*Beam3D*" [DETECT_AXIS [DETECT_AXIS_VECTOR *x1 x2 x3*]]
[DETECT_HEIGHT [DETECT_HEIGHT_VECTOR *x1 x2 x3*]]
[NUMBER_OF_IPS_IN_R *n*] [SOLID] HEIGHTS NUMBER n VALUES val1, val2 .. val_n WIDTHS NUMBER n VALUES val1, val2 .. val_n DOMAINS NUMBER n MATERIAL {n|0} QUAD_IDS {FROM n [TO n [BY [n]]] | AT n | LIST *i1,i2…*} [[REINFORCEMENT] BARS NUMBER *n* {MATERIAL *mat_id* ST_AREA *a* S_COORD *s* T_COORD *t* [IGNORE_REF_HEIGHT] [IGNORE_REF_WIDTH]]n [REDUCE_TAU_XY] [REDUCE_TAU_XZ] [FULL_TAU] [REF_HEIGHT *x*] [REF_WIDTH *x*] [CS_ISO_WIDTH *x*][7] [CS_ISO_HEIGHT *x*][8]

Table 50: &BEAM_3D_GEOMETRY_SPEC sub-command parameters

Parameter	Description
SOLID | REINFORCEMENT	The data that follow specify a solid, (i.e. concrete) or reinforcement, (i.e. steel) layer.
HEIGHTS NUMBER *n* VALUES *val1, val2 .. val_n*	Total number of solid heights, i.e. number of rows of the *s,* t raster. It is followed of actual height values. Isoparametric coordinates are used. Otherwise, the input heights are scaled so that their sum will equal to 2.

[7] Available in ATENA version 5.7.0 and later

[8] Available in ATENA version 5.7.0 and later

WIDTHS NUMBER n VALUES *val1, val2 .. val_n*	Ditto for widths.
DOMAINS NUMBER n MATERIAL {*n*|*0*} QUAD_IDS {FROM *n* [TO *n* [BY [*n*]]] | AT *n* | LIST *i1,i2…*}	Definition of material domains. The quad_ids are counted rowvise starting from the bottom left corner. If material_id is zero, a hole is assumed.
[REINFORCEMENT] BARS NUMBER *n*	Number of reinforcement "bars", i.e. quads, where reinforcement is assumed
MATERIAL *mat_id* ST_AREA *a* S_COORD *s* T_COORD *t* [IGNORE_REF_HEIGHT] [IGNORE_REF_WIDTH]	For *n* bars specify its material id, area and position via s, t coordinates. Isoparametric coordinates are used, otherwise the scaling factors are applied. The factors are those used for scaling solid heights and widths.
DETECT_AXIS {DETECT_AXIS_VECTOR *x1 x2 x3* }	Detect axis of beam elements and reorder element's incidences. If DETECT_AXIS_VECTOR is not specified, the axial direction is chosen to comply with the biggest dimension of the element. Otherwise it is chosen to have the smallest angle with the given vector {*x1, x2, x3*}.
DETECT_HEIGHT {DETECT_HEIGHT_VECTOR *x1 x2 x3* }	Detect height of beam elements and reorder element's incidences. If DETECT_HEIGHT_VECTOR is not specified, direction of the beam's height is chosen to comply with the bigger dimension of the element's cross section. Otherwise it is chosen to have the smallest angle with the given vector {*x1, x2, x3*}.
[NUMBER_OF_IPS_IN_R *n*	Number of integration points in beam's longitudinal axis. By default 2 IPs are used, however especially in case of heavy material nonlinearity, more IPs may yield more accurate results, as the beam can better locate a material failure. Max. value is 6.
[REDUCE_TAU_XY] [REDUCE_TAU_XZ] [FULL_TAU]	Reduce shears by the factor 0.85.
[REF_HEIGHT *x*] [REF_WIDTH *x*]	Reference height and width of the beam's cross section used to transform normalized cross sectional local *s,t* coordinates to real coordinates, (unless they are ignored). By default, theses values are not specified and in this case the actual beam's dimensions at integration points are used. This input is particularly useful, if a reinforcement cell is placed at constant distance from surface of the beam, whereby the beam's cross section dimensions are variable.
[CS_ISO_WIDTH *x*]	Set isoparametric width and height of the cross section. By

[CS_ISO_HEIGHT *x*]	default it is computed based on dimension of the solid CS and its reinforcement. [9]

For explanation of transformation between isoparametric and global coordinate system see BEAM_1D_GEOMETRY_SPEC.

&BEAM_1D_GEOMETRY_SPEC:
"*Beam1D*" CS_WIDTH_EQN "*eqn_expression*" CS_HEIGHT_EQN "*eqn_expression*" VT_X_EQN "*eqn_expression*" VT_Y_EQN "*eqn_expression*" VT_ZEQN "*eqn_expression*" [NUMBER_OF_IPS_IN_R *n*] [SOLID] HEIGHTS NUMBER n_h VALUES $h_1, h_2 .. h_{nh}$ WIDTHS NUMBER n_w VALUES $w_1, w_2 .. w_{nw}$ DOMAINS NUMBER n MATERIAL {n|0} QUAD_IDS {FROM n [TO n [BY [n]]] | AT n | LIST *i1,i2…*} [[REINFORCEMENT] BARS NUMBER n_{reinf} {MATERIAL *mat_id* ST_AREA a_i S_COORD s_i T_COORD t_i [REF_HEIGHT *x*] [REF_WIDTH *x*]]$_n$] [REDUCE_TAU_XY] [REDUCE_TAU_XZ] [FULL_TAU] [CS_ISO_WIDTH *x*] [CS_ISO_HEIGHT *x*]

Table 51: &BEAM_1D_GEOMETRY_SPEC sub-command parameters

Parameter	Description
SOLID | REINFORCEMENT	The data that follow specify a solid, (i.e. concrete) or reinforcement, (i.e. steel) layer.
HEIGHTS NUMBER *n* VALUES $h_1, h_2 .. h_{nh}$	Total number of solid heights, i.e. number of rows of the *s,* t raster. It is followed of actual height values. Isoparametric coordinates are used. Otherwise, the input heights are scaled so that their sum will equal to 2.
WIDTHS NUMBER n VALUES $w_1, w_2 .. w_{nw}$	Ditto for widths.
DOMAINS NUMBER n MATERIAL {*n*|*0*} QUAD_IDS {FROM *n* [TO *n* [BY [*n*]]] | AT *n* | LIST *i1,i2…*}	Definition of material domains. The quad_ids are counted rowvise starting from the bottom left corner. If material_id is zero, a hole is assumed.
[REINFORCEMENT] BARS NUMBER n_{reinf}	Number of reinforcement "bars", i.e. quads, where reinforcement is assumed
MATERIAL *mat_id* ST_AREA a_i S_COORD s_i T_COORD t_i [REF_HEIGHT *x*] [REF_WIDTH *x*]	For *n* bars specify its material id, area and position via s, t coordinates. Isoparametric coordinates are used, otherwise the scaling factors are applied. The factors are those used for scaling solid heights and widths.

[9] Available in ATENA version 5.7.0 and later

CS_WIDTH_EQN "*eqn_expression*" CS_HEIGHT_EQN "*eqn_expression*"	Width and height of beam's cross section. Both are given in terms of algebraic expression f(x,y,z), in which the parameters x,y,z, (i.e. coordinates) are substituted automatically based on location a beam using this geometry. Example: CS_WIDTH_EQN "0.5+0.1*x" CS_HEIGHT_EQN "0.1"
VT_X_EQN "*eqn_expression*" VT_Y_EQN "*eqn_expression*" VT_ZEQN "*eqn_expression*"	Algebraic expressions for x,y,z coordinates of the vector vt. Theey are input in similar way to the above cross section's dimensions. Example: VT_X_EQN "0" VT_Y_EQN "0" VT_Z_EQN "0.3"
[NUMBER_OF_IPS_IN_R *n*	Number of integration points in beam's longitudinal axis. By default 2 IPs are used, however especially in case of heavy material nonlinearity, more IPs may yield more accurate results, as the beam can better locate a material failure. Max. value is 6.
[REDUCE_TAU_XY] [REDUCE_TAU_XZ] [FULL_TAU]	Reduce shears by the factor 0.85.
[REF_HEIGHT *x*] [REF_WIDTH *x*]	Reference height and width of the beam's cross section used to transform normalized cross sectional local *s,t* coordinates to real coordinates, (unless they are ignored). By default, theses values are not specified and in this case the actual beam's dimensions at integration points are used. This input is particularly useful, if a reinforcement cell is placed at constant distance from surface of the beam, whereby the beam's cross section dimensions are variable.
[CS_ISO_WIDTH *x*] [CS_ISO_HEIGHT *x*]	Set isoparametric width and height of the cross section. By default it is computed based on dimension of the solid CS and its reinforcement.

The geometry is specified in isoparametric coordinates. The following equations displays relation between isoparametric, (marked by *iso* index) and real dimensions, marked by *global* index.

Isoparametric coordinate system:

$h_{solid}^{iso} = \sum_{i}^{n_h} h_i^{iso}$ ... height and width of the solid part

$w_{solid}^{iso} = \sum_{i}^{n_h} w_i^{iso}$

$h_{reinf}^{iso} = \max\left(s_i^{iso} + \frac{1}{2}\sqrt{a_i^{iso}}\right) - \min\left(s_i^{iso} - \frac{1}{2}\sqrt{a_i^{iso}}\right),\ i = \langle 1 \ldots n_h \rangle$ ... height of the reinforcement part

$w_{reinf}^{iso} = \max\left(t_i^{iso} + \frac{1}{2}\sqrt{a_i^{iso}}\right) - \min\left(t_i^{iso} - \frac{1}{2}\sqrt{a_i^{iso}}\right),\ i = \langle 1 \ldots n_w \rangle$ ... width of the reinforcement part

$h_{tot}^{iso} = \max\left(h_{solid}, h_{reinf}\right), \quad w_{tot}^{iso} = \max\left(w_{solid}, w_{reinf}\right)$

Global coordinate system:

$w_i^{global} = w_i^{iso} \frac{w_{tot}^{global}}{w_{tot}^{iso}},\ h_i^{global} = h_i^{iso} \frac{h_{tot}^{global}}{h_{tot}^{iso}},\ i = \langle 1 \ldots n_h \rangle$

$a_i^{global} = a_i^{iso} \frac{w^{global}}{w_{tot}^{iso}} \frac{h_{tot}^{global}}{h_{tot}^{iso}}, \quad s_i^{global} = s_i^{iso} \frac{w_{tot}^{global}}{w_{tot}^{iso}},\ t_i^{global} = t_i^{iso} \frac{h_{tot}^{global}}{h_{tot}^{iso}},\ i = \langle 1 \ldots n_{reinf} \rangle$

Note that Atena internally computes with ccordinate system $h \in \langle -1 \ldots 1 \rangle,\ w \in \langle -1 \ldots 1 \rangle$ so that the geometruy output command print dimensions assuming $h_{tot}^{global} = w_{tot}^{global} = 1 - (-1) = 2$

3.5.4 The command &ELEMENT

Syntax:

&ELEMENT:
ELEMENT { &ELEMENT_GROUP | &ELEMENT_TYPE |
&ELEMENT_INCIDENCES | &ELEMENT_MATERIALS }+

Table 52: &ELEMENT command parameters

Parameter	Description
&ELEMENT_GROUP	This sub-command begins the definition of a new element group. This command should be followed by the definition of element connectivity by using the sub-command ELEMENT INCIDENCES
&ELEMENT_ TYPE	Define a new element type. This element type is later referred to by the sub-command &ELEMENT_GROUP to specify an element type/formulation for an element group.
&ELEMENT_INCIDENCES	This sub-command should follow the command &ELEMENT_GROUP. It is used to define element

	connectivities.
&ELEMENT_MATERIALS	This sub-command should follow the command &ELEMENT_GROUP. It sets material types individually for each material point of the element. If not specified, default material type from &ELEMENT_GROUP is used.

&ELEMENT_GROUP:
GROUP { ID *n* [NAME = "*element group name*"] TYPE *n* MATERIAL *n*
GEOMETRY *n* | DELETE | ACTIVE | INACTIVE |
GROUP_CONSTRUCT_TIME *group_constr_time* | ASSOC_LC_ID *lc_id*}+

Table 53: &ELEMENT_GROUP sub-command parameters

Parameter	Description
ID *n*	Element group identification E.g.: ID *n*
NAME "*element group name*"	Element group name in quotes, also for identification E.g. NAME "*element group name*"
TYPE *n*	Element type identification. E.g.: TYPE *n*
MATERIAL *n*	Identification number of material to be used for this element group. E.g.: MATERIAL *n*
GEOMETRY *n*	Identification number of geometry to be used for this element group. E.g.: GEOMETRY *n*
DELETE	Resets content of the element group to default, i.e. removes its all-previous input data.
ACTIVE | INACTIVE | CONSTRUCT_TIME_DEPEND_ACTIVE	Marks all elements within the group as active, inactive or active on *condition constr_time*=<*current time*. Active elements are included in the analysis, whereas inactive elements are ignored.
GROUP_CONSTRUCT_TIME *group_constr_time*	Set element group construction time of all elements within group *group_id*. By default the elements not yet "constructed" are computed, but their matrices and vectors multiplied are by NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF, see Table 15. The *group_constr_time* parameter is also accounted for by material models with variable material model parameters. This parameter is added to ELEMENT_CONSTRUCT_TIME *elem_constr_time,* see Table 6. By default it is 0.
ASSOC_LC_ID *lc_id*	Associated load case id. This input is generated automatically, however in some cases it allows to manually specify load case id associated with this group. For example, if discrete

	reinforcement bars are input manually, i.e. not generated, the *lc_id* says, which load case is used to bind the bar with the surrounding solids.

&ELEMENT_TYPE:
TYPE { ID *n* | NAME "*element type name*" | { LINEAR | NONLINEAR |
IGNORE_NEGATIVE_JACOBIAN |
IGNORE_ELEMENT_TYPE_EXCEPTION | SEMINONLINEAR } TYPE
"*element_type*" | GAMMA_REF *x* | GAMMA_COEFF *x* |
PREPARE_CALCULATION |
["DEFAULT_PROCESSING" | "INITIAL_STRAIN_ONLY_INTO_SOLID" |
"INITIAL_STRESS_ONLY_INTO_SOLID" |
"INITIAL_STRAIN_ONLY_INTO_REINF" |
"INITIAL_STRESS_ONLY_INTO_REINF"] }+

Table 54: &ELEMENT_TYPE sub-command parameters

Parameter	Description
ID *n*	Element type identification E.g.: ID *n*
NAME "*element type name*"	Element group name in quotes, also for identification E.g.: NAME "*My_CCIsoBrick*"
LINEAR	Forces to ignore all terms due to geometrical non-linearity. Material linearity still may exist.
NONLINEAR	Forces to account for all terms due to geometrical non-linearity. This is the default setting.
SEMINONLINEAR	Linear in the 1st iteration, nonlinear in the next iterations. This option is sometimes advantageous, if the structure is loaded by deformations.
IGNORE_NEGATIVE_JACOBIAN	By default, if an element at an IP has negative jacobian, (i.e. there is something wrong with its geometry), then the appropriate exception is thrown and the execution is terminated. If this keyword is specified, the negative value of the jacobian is ignored and the execution continues (, although with disputable results).
IGNORE_ELEMENT_TYPE_EXCEPTION	By default, if an exception occures within element calculation, then the appropriate exception is thrown and the execution is terminated. If this keyword is specified, execution of the offending element is skipped and remaining elements get proceeded.
TYPE "*element_type*"	Element type in quotes. E.g.: TYPE "*element_type"*, where "*element_type"* adopts form *name*<*xx_x…*>, where x and _ characters in the <> brackets indicate number and location of nodes for hierarchical finite element type *name*. For instance CCIsoTriangle<xxx_x> indicates a four nodes triangular element CCIsoTriangle with the fourth node located

	between node 2 and 3. Names of other element types are input directly without the <*xx*...> decoration, e.g. Spring. The system automatically distinguishes between 2D, 3D or axisymmetric variant of the element used. E.g.: TYPE "*CCIsoQuad*<*xxxx_xx*>.
GAMMA_REF *x*	Factor for accounting angle between mesh and crack direction. See theoretical manual for more description.
GAMMA_COEFF *x*	Factor for accounting angle between mesh and crack direction. See theoretical manual for more description.
PREPARE_CALCULATION	Force immediate preprocessing of the input element type for calculation. It is the user's responsibility to ensure that all needed data are already available, i.e. input. By default this flag is not specified and preprocessing of element types is delayed up to the very last moment prior the execution.
["DEFAULT_PROCESSING" "INITIAL_STRAIN_ONLY_INTO_SOLID "INITIAL_STRESS_ONLY_INTO_SOLID" "INITIAL_STRAIN_ONLY_INTO_REINF" "INITIAL_STRESS_ONLY_INTO_REINF"]	Special flag for processing initial strain/stress load for elements with embedded smeared reinforcement. By default, the load is applied to both solid and reinforcement parts of the element.

Table 55: Available element types

Element type name	Description
CCIsoBrick	Isoparametric brick element (hexahedron) E.g.: CCIsoBrick<xxxxxxxx>
CCIsoWedge	Isoparametric wedge element E.g.: CCIsoWedge<xxxxxx>
CCIsoTetra	Isoparametric tetrahedral element E.g. : CCIsoTetra<xxxx>
CCIsoTriangle	Isoparametric triangular element E.g.: CCIsoTriangle<xxx>
CCIsoQuad	Isoparametric quadrilateral E.g.: CCIsoQuad<xxxx>
CCQ10	4 nodes quadrilateral element composed of two triangle isoparametric elements. This element must be defined by at least four corner nodes. E.g.: CCQ10<xxxx>
CCQ10Sbeta	4 nodes quadrilateral element composed of two triangles. Four corner nodes must define this element. The material model at this element is evaluated at the element center. The constitutive secant matrix evaluated at the element center is

	used throughout the whole element to calculate element internal forces. E.g.: CCQ10Sbeta<xxxx>
CCSpring	Spring element defined by a single node. This element type should be used to define a spring support at given node.
CCLineSpring	Line spring element defined by two nodes. This element type should be used for spring supports along solid element edges.
CCPlaneSpring	Planar spring element defined by three nodes. This element type should be used for spring supports along faces of solid elements.
CCIsoTruss	Isoparametric truss element. E.g.: CCIsoTruss<xx>
CCIsoASymTruss	Isoparametric truss element for axisymmetric problems. The element contributes stiffness in direction of its axis. For adding also radial stiffness, combine this element with the CCCircumferentialTruss or CCCircumferentialTruss2 element. E.g. CCIsoASymTruss<xx>
CCIsoGap	Gap/Interface element. E.g.: CCIsoGap<xxxx>
CCCircumferentialTruss	Circumferential truss element. This element is defined by only one node and is used in axi-symmetric analysis to model circumferential reinforcement. It contributes also radial stiffness. E.g.: CCCircumferentialTruss
CCCircumferentialTruss2	Circumferential truss element. This element is defined by two nodes and is used in axi-symmetric analysis to model circumferential reinforcement. It is similar to the CCCircumferentialTruss element, however its “cross sectional area” is equal to its length multiplied by its thickness. For adding stiffness also in the element’s axial direction combine this element with the CCIsoASymTruss element. E.g.: CCCircumferentialTruss2
CCExternalCable	2D or 3D truss element for modeling external prestress cables. The bar is anchored at one end and prestressed at the other. The intermediate nodes are deviators, where frictional force is defined, see external geometry definition. The whole bar must consist of one or more elements. All the elements must compose the same element group.
CCBarWithBond	2D or 3D truss element for modeling reinforcement bars with specified cohesion with concrete. If exceeded, the bar will slip. The element type uses external cable geometry definitions to

	specify the appropriate solution parameters. The whole bar must consist of one or more elements. All the elements must compose the same element group.
CCAhmadElement33L CCAhmadElement32L CCAhmadElement33H CCAhmadElement32H CCAhmadElement22S	3D shell elements. The first and the second digits in the element name specify number of integration points for element bending and shear energy. E.g. the digit three says that the element is integrated in 3 IPs in X dir * 3 IPs in Y dir * number of layers. The last letter L,H and S stands for 9-nodes Lagrangian element, for 9 nodes Heterosis element and 8 nodes Serendipity element. See theoretical manual for more details. All the element must use a 3D material and LayeredShell geometry! They specified by 16 nodes, 8 for top and 8 for bottom surface similar to brick elements. The top and bottom middle points for Lagrangian and Heterosis elements (for the bubble functions) are generated automatically. At each node the elements have 3 degree of freedom. As top and bottom node have altogether 6 DOFs and shell theory uses only 5 DOFs per shell node, the z displacement of the bottom node is automatically constrained during the execution.
CCBeamNL	3D nonlinear beam element. The element uses quadratic interpolation along its axis, so that it can have curvilinear shape. Similar to the implemented CCAhmad elements it is also input as a 3D hexahedral box. Nevertheless, the usual axial nodal points are available (e.g. for checking resulting deformations and rotations. They are generated automatically.
CCBeam	3D linear beam element. The element is assumed for a simplified analysis with CCBeamMasonry and CCBeamRCMaterial materials.
CCIsoBeamBar<xx> CCIsoBeamBar<xxx>	Isoparametric 1D beam element with 2 or three nodes. The elements are similar to CCBeamNL but they are modelled as a bar 1D element. It resembles CCBeamNL element type without its nodes 1-12 to model element's 3D shape.
CCIsoShellBrick<xxxxxxx>... CCIsoShellBrick<xxxxxxxxxxxxxx>.	Isoparametric full 3D shell element (hexahedral curvilinear shape). They are compatible with the same materials as are CCIsoBrick elements. Unlike CCAhmadElement... elements it uses everywhere native 3dofs per node, i.e. no additional constraint of the element's bottom is needed. E.g.: CCIsoShellBrick <xxxxxxxxxxxxxxxxxx>
CCIsoShellWedge<xxxxxx>... CCIsoShellWedge<xxxxxxxxxxxx>.	Isoparametric full 3D shell element (wedge curvilinear shape). They are compatible with the same materials as are CCIsoBrick elements. Unlike CCAhmadElement... elements it uses everywhere native 3dofs per node, i.e. no additional constraint of the element's bottom is needed. E.g.: CCIsoShellBrick <xxxxxxxxxxxx>
CCIsoShellQuad<xxxx> ...	Nonlinear shell elements similar to Ahmad elements, however

CCIsoShellQuad<xxxxxxxxx>	they are specified by 2D curvilinear surface. In each node, they have 3 displacements and 2 rotations. As for material and geometry they use the same data as Ahmad elements defined above.
CCIsoShellTriangle<xxx> ... CCIsoShellTriangle<xxxxxx>	Nonlinear shell elements similar to CCIsoShellQuad elements, however they have triangular curvilinear shape. In each node, they have 3 displacements and 2 rotations. As for material and geometry they use the same data as Ahmad elements defined above.
CCIsoBeamBrick12_3D CCIsoBeamBrick8_3D	Isoparametric full 3D beam NL elements. The element uses quadratic interpolation along its axis, so that it can have curvilinear shape. The elements are compatible with materials suitable for full 3D analysis, i.e. material good for CCIsoBrick elements. As for geometry it uses (similar to CCBeamNL) CCBeam3DGeometry data.

Table 56: Element Type and Material Compatibility

	CCIso-Brick	CCIso-Quad	CCIso-Triangle	CCQ10	CCQ10Sbeta	CCSpring	CCLine-Spring	CCPlane-Spring	CCIso-Truss	CCIso-Gap	CCCircumferentialTruss
CC1DElastIsotropic (*)						X	X	X	X		X
CCPlaneStressElastIsotropic (*)		X	X	X	X	X	X	X	X		X
CCPlaneStrainElastIsotropic (*)		X	X	X	X	X	X	X	X		X
CC3DElastIsotropic (*)	X	X	X	X	X	X	X	X	X		X
CCASymElastIsotropic (*)		X	X	X		X	X	X	X		X
CC3DBiLinearSteelVonMises (*)	X	X	X	X		X	X	X	X		X
CC3DCementitious	X	X	X	X		X	X	X	X		X
CC3DNonLinCementitious	X	X	X	X		X	X	X	X		X
CC3DNonLinCementitious2 (*)	X	X	X	X		X	X	X	X		X
CC3DNonLinCementitious2User (*)	X	X	X	X		X	X	X	X		X
CC3DNonLinCementitious2Variable	X	X	X	X		X	X	X	X		X
CCSBETAMaterial		X	X	X	X	X	X	X	X		X
CC2DInterface										X	
CC3DInterface										X	
CCReinforcement						X	X	X	X		X

CCCyclingReinforcement						X	X	X	X		X
CCSmearedReinf	X	X	X	X	X						
CCCircumferentialSmearedReinf		X	X	X	X						
CCSpringMaterial						X	X	X			
CC3DDruckerPragerPlasticity	X	X	X	X		X	X	X	X		X
CCMaterialWithVariableProperties	X	X	X	X		X	X	X	X		X
CCMaterialWithTempDepProperties	X	X	X	X		X	X	X	X		X
CCMaterialWithRandomFields	X	X	X	X		X	X	X	X		X
CCCombinedMaterial	X	X	X	X		X	X	X	X		X

Table 57 : Element Type and Material Compatibility, (beam and shell elements)

	CCBeam	CCIsoBeamBar	CCIsoBeamBrick	CCBeamNL	Ahmad	CCIsoShellBrick	CCIsoShellWedge	CCIsoShellQuad	CCIsoShellTriangle
CC1DElastIsotropic (*)		X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]
CCPlaneStressElastIsotropic (*)									
CCPlaneStrainElastIsotropic (*)									
CC3DElastIsotropic (*)		X	X	X	X	X	X	X	X
CCASymElastIsotropic (*)									
CC3DBiLinearSteelVonMises (*)									
CC3DCementitious		X	X	X	X	X	X	X	X
CC3DNonLinCementitious		X	X	X	X	X	X	X	X
CC3DNonLinCementitious2 (*)		X	X	X	X	X	X	X	X
CC3DNonLinCementitious2User (*)		X	X	X	X	X	X	X	X
CC3DNonLinCementitious2Variable		X	X	X	X	X	X	X	X
CCSBETAMaterial									
CC2DInterface									
CC3DInterface									
CCReinforcement		X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]
CCCyclingReinforcement		X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]
CCSmearedReinf		X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]
CCCircumferentialSmearedReinf		X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]	X[10]
CCSpringMaterial									
CC3DDruckerPragerPlasticity		X	X	X	X	X	X	X	X

[10] For reinforcement.

CCMaterialWithVariableProperties		X	X	X	X	X	X	X	X
CCMaterialWithTempDepProperties		X	X	X	X	X	X	X	X
CCMaterialWithRandomFields		X	X	X	X	X	X	X	X
CCCombinedMaterial		X	X	X	X	X	X	X	X
CCBeamMasonryMaterial	X								
CCBeamRCMaterial	X								

Table 58 : Beam and shell elements and their element idealisation, material idealisation and geometry type

	CCBeam	CCIsoBeamBar	CCIsoBeamBrick	CCBeamNL	Ahmad	CCIsoShellBrick	CCIsoShellWedge	CCIsoShellQuad	CCIsoShellTriangle
Element geometry type	BEAM	BEAM_3D	BEAM_3D	BEAM_3D	LAYEREDSHELL	LAYEREDSHELL	LAYEREDSHELL	LAYEREDSHELL	LAYEREDSHELL
Element idealisation[11]	BEAM_3D	BEAM_NL_1D	BRICK	BEAM_NL_3D	LAYERED_SHELL	BRICK	BRICK	LAYERED_SHELL_2D	LAYERED_SHELL_2D
Material idealisation	ONE_D	BEAM_3D	THREE_D	BEAM_3D	SHELL	THREE_D	THREE_D	SHELL	SHELL
Element shape[11]	SHAPE_BAR	SHAPE_BAR	SHAPE_BRIC	SHAPE_BRICK	SHAPE_BRICK	SHAPE_SHELL_BRICK_3D	SHAPE_SHELL_WEDGE_3D	SHAPE_QUADRILATERAL	SHAPE_TRIANGLE

[11] Defined by a finite element that is used.

The above tables apply in full for static and dynamic analysis. As far as creep analysis is concerned, it uses time independent and time dependent materials:

Time independent material (as indicated by the name) does not change its behaviour with age. Such a material is e.g. used for reinforcement. Any material from the above table can be used as time independent material for creep analysis.

On the other hand, concrete is known to change its properties with time and therefore (within a creep analysis) it must be modelled by a time dependent material &CREEP_MATERIAL. Only materials marked with "*" (from the above table) can be used as the parameter "*short_term_material_type"*, (refering to the definition of &CREEP_MATERIAL).

Transport analysis uses completely different element types and element material models. They are described in Section 0. Any transport element type can be used in conjugation with any transport material model.

&ELEMENT_INCIDENCES:

[NNODES *num_nodes*]

$id_1 \ \{ n \}_{number\_nodes\_1}$

$id_2 \ \{ n \}_{number\_nodes\_2}$

...

$id_m \ \{ n \}_{number\_nodes\_m}$

Table 59: &ELEMENT_INCIDENCES sub-command parameters

Parameter	Description
[NNODES *num_nodes*]	Optional number of element incidences. If not defined, *num_nodes* is derived from the element's element type.
id	Element id. E.g.: *n*
$\{ n \}_{number\_nodes}$	Element incidences, i.e. ids of nodes incidenting with the element. *number_nodes* integer numbers is expected, where number *number_nodes* is number of element nodes for the particular element type E.g.: $n_1 \ \ n_2 \ \ [n_3] \ \ .... \ \ [n_{number\_nodes}]$
Note: This command has to follow the command ELEMENT GROUP. Each element incidences data must be input on a separate line.	

&ELEMENT_MATERIALS:

$id_1 \ \{ n \}_{number\_of\_material\_points}$

$id_2 \ \{ n \}_{number\_of\_material\_points}$

...

id_m { n }$_{number\_of\_material\_points}$

Table 60: &ELEMENT_MATERIALS sub-command parameters

Parameter	Description
id	Element id. E.g.: *n*
{ n }$_{number\_of\_material\_points}$	Material type at element's material point. By default, a positive integer value is expected for each material point of the element. If the input value *n* is zero, it indicates that this and all remaining material points use the default material type. If the input value *n* is negative, it indicates that this and all remaining material points are of type *(-n)*. If the element uses the same material types in all its material points, the &ELEMENT_MATERIALS command can be omitted and a default material type specified in &ELEMENT_GROUP is adopted. E.g.: 10 20 30 40 E.g. 10 -20
Note: This command has to follow the command ELEMENT GROUP. Each element material type's data must be input on a separate line.	

3.5.5 Geometrical imperfections &NODAL_IMPERFECTIONS

The following command can be used to specify initial imperfections of structural geometry. By default, zero nodal imperfections are assumed.

The nodal imperfections can be set by the input command &NODAL_IMPERFECTIONS:

Syntax:

```
&NODAL_IMPERFECTIONS :
NODAL_IMPERFECTIONS [SETTINGS] {
      &MANUAL_IMPEREFECTIONS_ENTRY |
      &GENERATED_IMPEREFECTIONS_ENTRY  }

&MANUAL_IMPEREFECTIONS_ENTRY:
{ NODE n { TOTAL | INCREMENT | INCREMENTAL} {VALUE | VALUES } val_x
      val_y [val_z]  }
```

Table 61: Nodal Initial Imperfections Definition (manual entries)

Sub-Command	Description
NODE *n*	Set initial conditions for node *n.*
{VALUE | VALUES} *val_x val_y* [*val_z*]	Specify initial nodal imperfections in direction of global coordinates. 3D problems need 3 values, 2D problems only two values..
{TOTAL | INCREMENT | INCREMENTAL }	Set input for total or incremental (with respect to the reference coordinates) values of the imperfect structural geometry.

&GENERATED_IMPEREFECTIONS_ENTRY:

NODAL IMPERFECTIONS [SETTING] SELECTION "*selection_name*" | { TOTAL | INCREMENT | INCREMENTAL} GENERATE CONST *const_vector* | COEFF_X *coeff_x_vector* | COEFF_Y *coeff_y_vector* | COEFF_Z *coeff_z_vector*}+

Table 62: Nodal Initial Imperfections Definition (generated entries)

Sub-Command	Description
SELECTION "*selection_name*"	Name of selection, for which the generation is requested.
{GENERATE GENERATE_VEL } CONST *const_vector* COEFF_X *coeff_x_vector* | COEFF_Y *coeff_y_vector* | COEFF_Z *coeff_z_vecor*	Keyword for entities to be generated. The values in global structural directions are generated as linear combination: $value_x = const(1) + x\ coeff_x(1) + y\ coeff_y(1) + z\ coeff_z(1)$ $value_y = const(2) + x\ coeff_x(2) + y\ coeff_y(2) + z\ coeff_z(2)$ $value_z = const(3) + x\ coeff_x(3) + y\ coeff_y(3) + z\ coeff_z(3)$ *x,y,z* are coordinates of nodes, where the generation is processed. The vector of values, e.g. *const_vector* must include 3 or 2 values for 2D or 3D problems, respectively.
{TOTAL | INCREMENT | INCREMENTAL }	Set input for total or incremental (with respect to the reference coordinates) values of the imperfect structural geometry.

Example:

```
NODAL_IMPEFECTIONS SETTINGS // 3D
  NODE  2 TOTAL VALUES  0. 0. 0.001
  NODE  3 INCREMENT VALUES  0. 0. 0.0015
  …….
```

NODAL_IMPEFECTIONS SETTINGS // 2D
NODE 2 TOTAL VALUES 0. 0.001
NODE 3 INCREMENTAL VALUES 0. 0.0015
…….

NODAL SETTING SELECTION "all_nodes" TOTAL
CONST 25. 12. 24. COEFF_X 0. 0. 0. COEFF_Y 0. 0. 0. COEFF_Z 0. 0. 0.01 GENERATE // 3D

3.6 Material Definition - The Command &MATERIAL

Syntax:
&MATERIAL:
MATERIAL ID *n* [NAME "*material_name*"] &MATERIAL_TYPE_PARAMS

Table 63: &MATERIAL command parameters

Parameter	Description
ID *n*	Material identification E.g.: ID *1*
NAME "*material_name*"	Material name in quotes, also for identification E.g.: NAME „*my_material*"
&MATERIAL_TYPE_PARAMS	Material type and type specific parameters

&MATERIAL_TYPE_PARAMS:
{ &LINEAR_ELASTIC_ISOTROPIC | &3DCEMENTITIOUS |
&3DNONLINCEMENTITIOUS | &3DNONLINCEMENTITIOUS2 |
&3DNONLINCEMENTITIOUS2VARIABLE |
&3DNONLINCEMENTITIOUS2USER |
&3DNONLINCEMENTITIOUS2SHCC |
&3DNONLINCEMENTITIOUS2SFATIGUE |
&3DNONLINCEMENTITIOUS3 | &SBETAMATERIAL |
&VON_MISES_PLASTICITY | &USER_MATERIAL |
&INTERFACE_MATERIAL | &REINFORCEMENT |
&REINFORCEMENT_WITH_CYCLING_BEHAVIOR |
&SMEARED_REINFORCEMENT | &SPRING |
&DRUCKER_PRAGER_PLASTICITY | &MICROPLANE |
&CREEP_MATERIAL | &COMBINED_MATERIAL |
&VARIABLE_MATERIAL |
&MATERIAL_WITH_TEMP_DEP_PROPERTIES |
&MATERIAL_WITH_RANDOM_FIELDS
&BEAM_MASONRY_MATERIAL | &BEAM_RC_MATERIAL |
&BEAM_REINF_BAR_MATERIAL}

Table 64: &MATERIALTYPE_PARAMS sub-command parameters

Parameter	Description
&LINEAR_ELASTIC_ISOTROPIC	Linear elastic isotropic materials for 1D, Plane Stress, Plane Strain, Axisymmetric and 3D analyses
&3DCEMENTITIOUS	Material suitable for rock or concrete like materials.
&3DNONLINCEMENTITIOUS	Materials suitable for rock or concrete like materials. Enhanced &3DCEMENTITIOUS material.
&3DNONLINCEMENTITIOUS2	Materials suitable for rock or concrete like materials. This material is identical to 3DNONLINCEMENTITIOUS except that this model is fully incremental.
&3DNONLINCEMENTITIOUS2VARIABLE	Materials suitable for rock or concrete like materials. This material is identical to 3DNONLINCEMENTITIOUS2 except that selected material parameters can be defined using a time or load step function.
&3DNONLINCEMENTITIOUS2USER	Materials suitable for rock or concrete like materials. This material is identical to 3DNONLINCEMENTITIOUS2 except that selected material laws can be defined by user curves.
&3DNONLINCEMENTITIOUS2SHCC	Strain Hardening Cementitious Composite material. Material suitable for fibre reinforced concrete, such as SHCC and HPFRCC materials.
&3DNONLINCEMENTITIOUS2FATIGUE	Based on the 3DNONLINCEMENTITIOUS2 material, suitable for fatigue analysis of rock or concrete like materials.
&3DNONLINCEMENTITIOUS3	Materials suitable for rock or concrete like materials. This material is an advanced version of 3DNONLINCEMENTITIOUS2 material that can handle the increased deformation capacity of concrete under triaxial compression. Suitable for problems including confinement effects.
&VON_MISES_PLASTICITY	Plastic materials with Von-Mises yield condition, e.g. suitable for steel.
&DRUCKER_PRAGER_PLASTICITY	Plastic materials with Drucker-Prager yield condition.
&USER_MATERIAL	User defined material (derived from elastic isotropic). The user provides a dynamic link library.

&INTERFACE_MATERIAL	Interface material for 2D and 3D analysis.
&REINFORCEMENT	Material for discrete reinforcement.
&REINFORCEMENT_WITH_ CYCLING_BEHAVIOR	Material for discrete reinforcement subject to cycling loading.
&SMEARED_REINFORCEMENT	Material for smeared reinforcement.
&SPRING	Material for spring type boundary condition elements, i.e. for truss element modeling a spring.
&MICROPLANE	Bazant Microplane material models for concrete
&CREEP_MATERIAL	Material for creep analysis. These are: CCModelB3 = Bazant-Baweja B3 model CCB3Improved = model same as the above with support for specified time and humidity history CCModelBP_KX = creep model developed by Bazant-Kim, 1991. CCModelCEB-FIP = creep model advocated by CEB-FIP 1978 CCModelACI_78 = creep model by ACI Committee in 1978. CCModelCSN731202 = model recommended by CSN731202 CCModelBP1 = full version of the creep model developed by Bazant-Panulla CCModelBP2 = simplified version of the above model CCModelGeneral = creep model for direct input of material compliance, strength and shrinkage at times typically measured in a laboratory. CCModelFIB_MC2010 = model by CEB-fib bulletin 65 from the year 2010. CCModelEN1992=creep model by Eurocode EN1992.1.1_2006.
&COMBINED_MATERIAL	This material can be used to create a composite material consisting of various components, such as for instance concrete with smeared reinforcement in various directions. Unlimited number of components can be specified. Output data for each component are then indicated by the label *#i*. Where *i* indicates a value of the *i*-th component.
&VARIABLE_MATERIAL	This material can be used as an envelope for

	other materials, whose parameters are not constant during the analysis. A function depending on time or load step can be specified for any material parameter. This can be used only in the connection with fully incremental materials.
&MATERIAL_WITH_TEMP_DEP_PROPERTIES	This material can be used as an envelope for other materials, whose parameters depend on temperature. This can be used only in the connection with fully incremental materials.
&MATERIAL_WITH_RANDOM_FIELDS	This material can be used to simulate the random spatial distribution of selected material parameters.
&BEAM_MASONRY_MATERIAL	Material for (reinforced) masonry structures modeled by CCBeal material.
&BEAM_RC_MATERIAL	Material for (reinforced) structures modeled by CCBeal material
&try_reduce_MyMz_keep_NBEAM_REINF_BAR_MATERIAL	Material for reinforcement bar used in solids modeled by either BEAM_RC_MATERIAL or BEAM_MASONRY_MATERIAL material.

3.6.1 Linear Elastic Isotropic Materials

3.6.1.1 Sub-command &LINEAR_ELASTIC_ISOTROPIC

Syntax:
&LINEAR_ELASTIC_ISOTROPIC:

TYPE { “CC1DElastIsotropic”“CCPlaneStressElastIsotropic” | “CCPlaneStrainElastIsotropic” | “CCASymElastIsotropic” | “CC3DElastIsotropic” } { E x | { MU | NY | POISSON } x | RHO x | ALPHA x | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D, SHELL, BEAM_3D, MEMBRANE_AXI} | DAMPING_MASS x_M | DAMPING_STIFF x_K }+

Table 65: &LINEAR_ELASTIC_ISOTROPIC sub-command parameters

Parameter	Description
Basic properties	
E x	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 210 x 10^3 f_F / f_l^2
{MU | POISSON | NY }	Poisson’s ratio.

x	Units: none Acceptable range: <-1; 0.5) Default value: 0.3
Miscellaneous properties	
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D "SHELL", "BEAM_3D", "MEMBRANE_AXI"} Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element, where it is used. So in most cases it is not needed to use this command. In certain cases, however, the program cannot determine correctly the idealisation to use. Such a case is for instance, if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.

3.6.2 Cementitious Materials

3.6.2.1 Sub-command &3DCEMENTITIOUS

Syntax:

&3DCEMENTITIOUS:
TYPE "*CC3DCementitious*" {E *x* | { MU | POISSON | NY } *x* | { FT | RT | F_T | R_T} *x* | { FC | RC | F_C | R_C} *x* | GF *x* | WD *x* | EXC *x* | BETA *x* | RHO *x* | ALPHA *x* | FT_MULTIP *x* | SHEAR_FACTOR *x* | UNLOADING *x* |

IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN,
AXISYMMETRIC, 3D} | DAMPING_MASS x_M | DAMPING_STIFF x_K}+

The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 66). This value should be specified in MPa and then transformed to the current units.

Table 66: &3DCEMENTITIOUS sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <-1; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; -2 FT) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number>

	Default value: 0.0001 f_F / f_l Generation formula: GF = 0.000025 FT
Compressive properties	
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
Miscellaneous properties	
EXC *x*	Eccentricity, defining the shape of the failure surface Units: Acceptable range: <0.5; 1.0> Default value: 0.52
BETA *x*	Multiplier for the direction of the plastic flow. Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used. Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none

	Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; +> Default value: 20
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .

3.6.2.2 Sub-command &3DNONLINCEMENTITIOUS

&3DNONLINCEMENTITIOUS:
TYPE "*CC3DNonLinCementitious*" { E *x* { MU | POISSON | NY } *x* | { FT | RT | F_T | R_T} *x* | { FC | RC | F_C | R_C} *x* | { FC0 | RC0 | F_C0 | R_C0} *x* | GF *x* | CRACK_SPACING *x* | TENSION_STIFF *x* | WD *x* | EPS_CP *x* | EXC *x* | BETA *x* | RHO *x* | ALPHA *x* | FT_MULTIP *x* | SHEAR_FACTOR *x* | UNLOADING *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K }+

The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 67). This value should be specified in MPa and then transformed to the current units.

Table 67: &3DNONLINCEMENTITIOUS sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <-1; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_C} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Default value: 0.0001 f_F / f_l

	Generation formula: GF = 0.000025 FT
CRACK_SPACING *x*	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
CRACK_SPACING_MIN *x*	Crack spacing minimal value. For extremely small elements during the crack initiation process more microcracks may develop than it is physically possible. For instance in case of concrete, if the finite element size is smaller than aggregate size, this parameter should be set to the size of aggregates. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF *x*	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0
Compressive properties	
EPS_CP *x*	Plastic strain at compressive strength. Units: none Acceptable range: <minimal real number; 0> Default value: -0.001 Generation formula: FC/E
{ FC0 | F_C0 | RC0 | R_C0 } *x*	Onset of non-linear behavior in compression. Units: $F/(l^2)$ Acceptable range: (FC,-2 FT) Default value: -20 f_F / f_l^2 Generation formula: FT*2.1
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
CRUSH_BAND_MIN *x*	Minimal value of crush band size for localization in

	compression. Typically this is calculated based on the finite element size as its size projected into the direction of minimal principal stress. If element sizes smaller than the minimal possible crushing zone size are used, this paremeter should be set to a nonzero value. In practical situations it can be set equal to the minimal dimension (thickness) of the structure at the location of the crushing. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
Miscellaneous properties	
EXC *x*	Eccentricity, defining the shape of the failure surface Units: Acceptable range: <0.5; 1.0> Default value: 0.52
BETA *x*	Multiplier for the direction of the plastic flow. Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used. Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none

	Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; +> Default value: 20
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .

3.6.2.3 Sub-command &3DNONLINCEMENTITIOUS2

&3DNONLINCEMENTITIOUS2:
TYPE "*CC3DNonLinCementitious2*" { E *x* { MU | POISSON | NY } *x* | { FT | RT | F_T | R_T} *x* | { FC | RC | F_C | R_C} *x* | { FC0 | RC0 | F_C0 | R_C0} *x* | GF *x* | CRACK_SPACING *x* | TENSION_STIFF *x* | WD *x* | EPS_CP *x* | FC_REDUCTION *x* | EXC *x* | BETA *x* | RHO *x* | ALPHA *x* | FT_MULTIP *x* | SHEAR_FACTOR *x* | AGG_INTERLOCK *x* | AGG_SIZE *x* | LIMIT_TAU_CRACK *x* | UNLOADING *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D}| DAMPING_MASS x_M DAMPING_STIFF x_K, | SUBSTEPS_PER_FT *x* | MAX_SUBSTEPS *x* }+

This material is identical to the previous material 3DNONLINCEMENTITIOUS but it is internally formulated purely incrementally, while in the previous material only the plastic part of the model is fully incremental, while the fracturing part is based on total formulation. The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 67). This value should be specified in MPa and then transformed to the current units.

Table 68: &3DNONLINCEMENTITIOUS2 sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <-1; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_C} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
GF *x*	Specific fracture energy Units: F/l

	Acceptable range: (0; maximal real number> Default value: 0.0001 f_F / f_l Generation formula: GF = 0.000025 FT
CRACK_SPACING *x*	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF *x*	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0
Compressive properties	
EPS_CP *x*	Plastic strain at compressive strength. Units: none Acceptable range: <minimal real number; 0> Default value: -0.001 Generation formula: FC/E
{ FC0 | F_C0 | RC0 | R_C0 } *x*	Onset of non-linear behavior in compression. Units: F/(l^2) Acceptable range: (FC,-2 FT) Default value: -20 f_F / f_l^2 Generation formula: FC*2/3
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
FC_REDUCTION *x*	Reduction of compressive strength due to cracking. When cracking occurs, depending on the tensile fracturing strain the compressive strength of the material is reduced using the formula from the modified compression field theory by Collins. The parameter of this command is the limiting relative value of the compressive strength reduction. Units: none Acceptable range: <0; 1>

	Default value: 0.2
Miscellaneous properties	
EXC *x*	Eccentricity, defining the shape of the failure surface Units: Acceptable range: <0.5; 1.0> Default value: 0.52
BETA *x*	Multiplier for the direction of the plastic flow. Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used. Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; +> Default value: 20

AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked if not higher than tensile strength. If yes, the shear stress on the crack is set to current value of the tensile strength. If set to 1, the aggregate interlock calculation based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D }

	Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
SUBSTEPS_PER_FT x	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

3.6.2.4 Sub-command &3DNONLINCEMENTITIOUS2VARIABLE

&3DNONLINCEMENTITIOUS2VARIABLE:
TYPE "*CC3DNonLinCementitious2Variable*" { E x { MU | POISSON | NY } x | { FT | RT | F_T | R_T} x | { FC | RC | F_C | R_C} x | { FC0 | RC0 | F_C0 | R_C0} x | GF x | CRACK_SPACING x | TENSION_STIFF x | WD x | EPS_CP x | FC_REDUCTION x | EXC x | BETA x | RHO x | ALPHA x | | FT_MULTIP x | SHEAR_FACTOR x | AGG_INTERLOCK x | AGG_SIZE x | LIMIT_TAU_CRACK x | UNLOADING x | PARAM "*parameter name*" F i | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K | SUBSTEPS_PER_FT x | MAX_SUBSTEPS x }$_+$

This material is identical to the previous material 3DNONLINCEMENTITIOUS2 but its selected material parameters can be changed during the analysis to simulate for instance material degradation.

Table 69: &3DNONLINCEMENTITIOUS2VARIABLE sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <-1; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_C} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Default value: 0.0001 f_F / f_l Generation formula: GF = 0.000025 FT
CRACK_SPACING *x*	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to

	finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF *x*	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0
Compressive properties	
EPS_CP *x*	Plastic strain at compressive strength. Units: none Acceptable range: <minimal real number; 0> Default value: -0.001 Generation formula: FC/E
{ FC0 | F_C0 | RC0 | R_C0 } *x*	Onset of non-linear behavior in compression. Units: $F/(l^2)$ Acceptable range: (FC,-2 FT) Default value: -20 f_F / f_l^2 Generation formula: FT*2.1
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
FC_REDUCTION *x*	Reduction of compressive strength due to cracking. When cracking occurs, depending on the tensile fracturing strain the compressive strength of the material is reduced using the formula from the modified compression field theory by Collins. The parameter of this command is the limiting relative value of the compressive strength reduction. Units: none Acceptable range: <0; 1> Default value: 0.2
Miscellaneous properties	
EXC *x*	Eccentricity, defining the shape of the failure surface Units: Acceptable range: <0.5; 1.0>

	Default value: 0.52
BETA *x*	Multiplier for the direction of the plastic flow. Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used. Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; +> Default value: 20
AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked if not higher than tensile strength. If yes, the shear stress on the crack is set to current value of the tensile strength. If set to 1, the aggregate interlock

	calculation based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { "1D", "PLANE_STRESS", "PLANE_STRAIN", "AXISYMMETRIC", "3D" } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element.

	Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
SUBSTEPS_PER_FT x[12]	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x[13]	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

3.6.2.5 Sub-command &3DNONLINCEMENTITIOUS2USER

&3DNONLINCEMENTITIOUS2USER:
TYPE "*CC3DNonLinCementitious2User*" { E x { MU | POISSON | NY } x | { FT | RT | F_T | R_T} x | { FC | RC | F_C | R_C} x | TENSION_SOFT_HARD_FUNCTION n | CHAR_SIZE_TENSION x | X_LOC_TENSION x | CRACK_SPACING x | TENSION_STIFF x | COMP_SOFT_HARD_FUNCTION x | CHAR_SIZE_COMP x | X_LOC_COMP x | FC_REDUCTION_FUNCTION n | SHEAR_STIFF_FUNCTION n | X_LOC_SHEAR x | SHEAR_STRENGTH_FUNCTION n | TENSILE_STRENGTH_RED_FUNCTION n | EXC x | BETA x | RHO x | ALPHA x | FT_MULTIP x | SHEAR_FACTOR x | AGG_INTERLOCK x | AGG_SIZE x | LIMIT_TAU_CRACK x | UNLOADING x | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K | SUBSTEPS_PER_FT x | MAX_SUBSTEPS x }+

This material is identical to the previous material 3DNONLINCEMENTITIOUS2 but it allows the user definition of the basic material curves such as tensile softening, compression softening, shear behavior of cracked concrete and tensile strength reduction based on the

[12] Supported since version 5.7.0

[13] Supported since version 5.7.0

applied compressive strength. The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 67). This value should be specified in MPa and then transformed to the current units. See ATENA theory manual for more detailed explanation of this material.

Table 70: Parameters for MATERIAL TYPE „CC3DNonLinCementitious2User". Suitable for rock or concrete like materials

Parameter	Description
Basic properties	
E	Elastic modulus. Format: E x Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
MU, POISSON, NY	Poisson's ratio. Format: MU x Units: none Acceptable range: <-1; 0.5) Default value: 0.2
FT, RT, F_T, R_T	Tensile strength Format: FT x Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$

FC , RC, F_C, R_C	Compressive strength Format: FC x Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30 f_F / f_l^2 Generation formula: FC = $-0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
TENSION_SOFT_ HARD_FUNCTION	Index of the function defining the tensile hardening/softening law. The horizontal axis represents strains and vertical axis tensile strength, which should be normalized with respect to $f_t^{'}$. Format: TENSION_SOFT_HARD_FUNCTION n Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.000 ; 1.00) ($\frac{0.75}{0.03}\frac{G_F}{f_t^{'}}$; 0.25) ($\frac{5}{0.03}\frac{G_F}{f_t^{'}}$; 0.00) where: GF = 0.000025 FT
CHAR_SIZE_TENSION	Characteristic size for which the various tensile functions are valid. Format: CHAR_SIZE_TENSION x Units: l Acceptable range: (0;maximal real number> Default value: 0.03 f_l Generation formula: none
X_LOC_TENSION	Strain value after, which the softening/hardening becomes localized, and therefore adjustment based on element size is needed.

	Format: X_LOC_TENSION *x* Units: none Acceptable range: <0;maximal real number> Default value: 0.0 Generation formula: none
CRACK_SPACING *x*	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF *x*	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0
Compressive properties	
COMP_SOFT_ HARD_FUNCTION	Index of the function defining the tensile hardening/softening law. The horizontal axis represents strains and vertical axis compressive strength, which should be normalized with respect to $f_c^{'}$. Format: COMP_SOFT_HARD_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.000 ; 0.25) (0.5*FC/E ; 0.80) (FC/E ; 1.00) (FC/E - 0.005 ; 0.00)

CHAR_SIZE_COMP	Characteristic size for which the various compressive functions are valid. Format: CHAR_SIZE_COMP x Units: l Acceptable range: (0;maximal real number> Default value: 0.10 f_l Generation formula: none
X_LOC_COMP	Strain value after, which the softening/hardening becomes localized, and therefore adjustment based on element size is needed. Format: X_LOC_COMP x Units: none Acceptable range: <0;-maximal real number> Default value: -0.001 Generation formula: FC/E
FC_REDUCTION_ FUNCTION n	Index of the function defining the compressive strength reduction due to cracking. The horizontal axis represents fracturing strains normal to a crack and vertical axis compressive strength, which should be normalized with respect to $f_c^{'}$. Format: FC_REDUCTION_FUNCTION n Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. It is recommended that this function does not decrease the compressive strength by more than 60%, i.e. 0.4 value. (0.000 ; 1.0) (0.001 ; 1.0) (0.005 ; 0.6) (0.01 ; 0.4)

Shear properties	
SHEAR_STIFF_ FUNCTION	Index of the function defining the shear retention factor evolution based on tensile strain in the crack direction. The horizontal axis represents strains and the vertical axis the relative reduction of the shear modulus. Format: SHEAR_STIFF_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.00000 ; 1.00) (1.e-7 ; 1.00) (1.e-6 ; 0.79) (1.e-5 ; 0.58) (0.00010 ; 0.36) (0.001 ; 0.15) (0.005 ; 0.01)
X_LOC_SHEAR	Strain value after, which the softening/hardening becomes localized, and therefore adjustment based on element size is needed. Format: X_LOC_SHEAR *x* Units: none Acceptable range: <0;maximal real number> Default value: 0.0 Generation formula: none

SHEAR_STRENGTH_ FUNCTION *n*	Index of the function defining the shear strength of a cracked concrete based on crack width in the crack direction. The horizontal axis represents strains and the vertical axis the relative value of shear strength with respect to the tensile strength FT. Format: SHEAR_STRENGTH_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.00000 ; 1.10) (0.0001 ; 0.87) (0.0005 ; 0.51) (0.0010 ; 0.34) (0.002 ; 0.20) (0.003 ; 0.15) (0.005 ; 0.09) (0.010 ; 0.05)
Tension-compression interaction	
TENSILE_STRENGTH_ RED_FUNCTION	Index of the function defining the tensile strength reduction based on the compressive stress in other material directions. The horizontal axis represents relative compressive stress normalized with respect to $f_c^{'}$ and the vertical axis the relative reduction of the tensile strength with respect to $f_t^{'}$. Format: TENSILE_STRENGTH_RED_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.000 ; 1.00) (1.000 ; 0.20)

Miscellaneous properties	
EXC	Excentricity, defining the shape of the failure surface Format: EXC x Units: Acceptable range: <0.5; 1.0> Default value: 0.52
BETA	Multiplier for the direction of the plastic flow. Format: BETA x Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO	Specific material density. Format: RHO x Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA	Coefficient of thermal expansion Format ALPHA x Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED	Fixed smeared crack model will be used. Format: FIXED x Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP x	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR x	This parameter is not used in this material as the shear stiffness

	of cracked concrete is defined according to the SHEAR_STIFF_FUNCTION defined for this material.
AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked according to the SHEAR_STRENGTH_FUNCTION provided for this material. If set to 1, the aggregate interlock calculation is based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS,

	PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
SUBSTEPS_PER_FT x[14]	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x[15]	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

3.6.2.6 Sub-command &3DNONLINCEMENTITIOUS2SHCC

&3DNONLINCEMENTITIOUS2SHCC:
TYPE "*CC3DNonLinCementitious2SHCC*" { E x { MU | POISSON | NY } x | { FT | RT | F_T | R_T} x | { FC | RC | F_C | R_C} x | FIBER_VOLUME_FRACTION x | FIBER_E_MODULUS x | FIBER_SHEAR_MODULUS x | FIBER_CROSS_SECTION_FACTOR x | FIBER_DIAMETER x | TENSION_SOFT_HARD_FUNCTION n | CHAR_SIZE_TENSION x | X_LOC_TENSION x | CRACK_SPACING x | TENSION_STIFF x | COMP_SOFT_HARD_FUNCTION x | CHAR_SIZE_COMP x | X_LOC_COMP x | FC_REDUCTION_FUNCTION n |

[14] Supported since version 5.7.0

[15] Supported since version 5.7.0

TENSILE_STRENGTH_RED_FUNCTION *n* | EXC *x* | BETA *x* | RHO *x* | ALPHA *x* | FT_MULTIP *x* | SHEAR_FACTOR *x* | AGG_INTERLOCK *x* | AGG_SIZE *x* | LIMIT_TAU_CRACK *x* | | UNLOADING *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} |DAMPING_MASS x_M DAMPING_STIFF x_K | SUBSTEPS_PER_FT *x* | MAX_SUBSTEPS *x* }+

This material is similar to the previous material 3DNONLINCEMENTITIOUS2USER but it includes features specific for modeling strain hardening cementitious composites or ultra-high performance fiber reinforced cementitious composite materials (SHCC, UHPFRCC. The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 67). This value should be specified in MPa and then transformed to the current units. See ATENA theory manual for more detailed explanation of this material.

Table 71: Parameters for MATERIAL TYPE „CC3DNonLinCementitious2SHCC". Suitable for strain hardening cementitious composites or fiber reinforced cementitious composites

Parameter	Description
Basic properties	
E	Elastic modulus. Format: E *x* Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 27 x 10^3 f_F / f_l^2
MU, POISSON, NY	Poisson's ratio. Format: MU *x* Units: none Acceptable range: <-1; 0.5) Default value: 0.2
FT, RT, F_T, R_T	Tensile strength Format: FT *x* Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 4 f_F / f_l^2
FC , RC, F_C, R_C	Compressive strength Format: FC *x* Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -16 f_F / f_l^2

Fiber reinforcement	
FIBER_VOLUME_FRACTION	Volume fraction of the fibers. Format: FIBER_VOLUME_FRACTION x Units: none Acceptable range: <0;1> Default value: 0.02
FIBER_E_MODULUS	Young's modulus of an individual fiber Format: FIBER_E_MODULUS x Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2
FIBER_SHEAR_MODULUS	Shear modulus of an individual fiber Format: FIBER_SHEAR_MODULUS x Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 0.15 x 10^3 f_F / f_l^2
FIBER_CROSS_SECTION_FACTOR	Fiber cross-section shape correction factor Format: FIBER_CROSS_SECTION_FACTOR x Units: none Acceptable range: <0; maximal real number> Default value: 0.9
FIBER_DIAMETER	Fiber diameter Format: FIBER_DIAMETER x Units: none Acceptable range: <0; maximal real number> Default value: 0.00004 f_l
Tensile properties	
TENSION_SOFT_ HARD_FUNCTION	Index of the function defining the tensile hardening/softening law. The horizontal axis represents strains and vertical axis tensile strength, which should be normalized with respect to f_t' . Format: TENSION_SOFT_HARD_FUNCTION n Units: none Acceptable range: <1;maximal int number>

	Default value: none Generation formula: default function should have the following points. (0.000 ; 1.00) (0.040 ; 1.25) (0.115 ; 0.00)
CHAR_SIZE_TENSION	Characteristic size for which the various tensile functions are valid. Format: CHAR_SIZE_TENSION x Units: l Acceptable range: (0;maximal real number> Default value: 0.08 f_l Generation formula: none
X_LOC_TENSION	Strain value after, which the softening/hardening becomes localized, and therefore adjustment based on element size is needed. Format: X_LOC_TENSION x Units: none Acceptable range: <0;maximal real number> Default value: 0.04 Generation formula: none
CRACK_SPACING x	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF x	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0

Compressive properties	
COMP_SOFT_ HARD_FUNCTION	Index of the function defining the tensile hardening/softening law. The horizontal axis represents strains and vertical axis compressive strength, which should be normalized with respect to $f_c^{'}$. Format: COMP_SOFT_HARD_FUNCTION n Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.000 ; 0.25) (0.5*FC/E ; 0.80) (FC/E ; 1.00) (FC/E - 0.005 ; 0.00) Note: the x-values should be negative.
CHAR_SIZE_COMP	Characteristic size for which the various compressive functions are valid. Format: CHAR_SIZE_COMP x Units: l Acceptable range: (0;maximal real number> Default value: 0.15 f_l
X_LOC_COMP	Strain value after, which the softening/hardening becomes localized, and therefore adjustment based on element size is needed. Format: X_LOC_COMP x Units: none Acceptable range: <0;-maximal real number> Default value: -0.0006, i.e. FC/E

Tension-compression interaction	
FC_REDUCTION_ FUNCTION *n*	Index of the function defining the compressive strength reduction due to cracking. The horizontal axis represents fracturing strains normal to a crack and vertical axis compressive strength, which should be normalized with respect to $f_c^{'}$. Format: FC_REDUCTION_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. It is recommended that this function does not decrease the compressive strength by more than 60%, i.e. 0.4 value. (0.000 ; 1.0) (0.001 ; 1.0) (0.005 ; 0.6) (0.01 ; 0.4)
TENSILE_STRENGTH_ RED_FUNCTION	Index of the function defining the tensile strength reduction based on the compressive stress in other material directions. The horizontal axis represents relative compressive stress normalized with respect to $f_c^{'}$ and the vertical axis the relative reduction of the tensile strength with respect to $f_t^{'}$. Format: TENSILE_STRENGTH_RED_FUNCTION *n* Units: none Acceptable range: <1;maximal int number> Default value: none Generation formula: default function should have the following points. (0.000 ; 1.00) (1.000 ; 0.20)
Miscellaneous properties	
EXC	Excentricity, defining the shape of the failure surface Format: EXC *x* Units: Acceptable range: <0.5; 1.0>

	Default value: 0.52
BETA	Multiplier for the direction of the plastic flow. Format: BETA *x* Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO	Specific material density. Format: RHO *x* Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA	Coefficient of thermal expansion Format ALPHA *x* Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED	Fixed smeared crack model will be used. Format: FIXED *x* Units: none Acceptable range: <0; 1> Default value: 1.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	This parameter is not used in this material since the shear stiffness of cracked concrete is defined according to the fiber content as described in ATENA Theory manual for this material.
AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked if not higher than tensile

	strength. If yes, the shear stress on the crack is set to current value of the tensile strength. If set to 1, the aggregate interlock calculation based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the

	program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
SUBSTEPS_PER_FT x[16]	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x[17]	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

3.6.2.7 Sub-command &3DNONLINCEMENTITIOUS2FATIGUE

&3DNONLINCEMENTITIOUS2FATIGUE:
TYPE "CC3DNonLinCementitious2Fatigue" { E x { MU | POISSON | NY } x | { FT | RT | F_T | R_T} x | { FC | RC | F_C | R_C} x | { FC0 | RC0 | F_C0 | R_C0} x | GF x | CRACK_SPACING x | TENSION_STIFF x | WD x | EPS_CP x | EXC x | BETA x | RHO x | ALPHA x | FT_MULTIP x | SHEAR_FACTOR x | AGG_INTERLOCK x | AGG_SIZE x | LIMIT_TAU_CRACK x | | UNLOADING x | BETA_FATIGUE x | KSI_FATIGUE x | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K | SUBSTEPS_PER_FT x | MAX_SUBSTEPS x }+

This material is based on the CC3DNONLINCEMENTITIOUS2 material, extended for fatigue calculation. It has an additional parameter, BETA_FATIGUE. It also has additional data attributes DAMAGE_FACTORS, FATIGUE_BASE_STRESS, FATIGUE_CYCLES_TO_FAILURE, FATIGUE_MAX_FRACT_STRAIN. See ATENA theory manual for more detailed description of this material. See the description of FATIGUE_PARAMS for details on fatigue analysis parameters.

[16] Supported since version 5.7.0

[17] Supported since version 5.7.0

Table 72: &3DNONLINCEMENTITIOUS2FATIGUE sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <-1; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; -FC/2) Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_C} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tensile properties	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Default value: 0.0001 f_F / f_l Generation formula: GF = 0.000025 FT
CRACK_SPACING *x*	Crack spacing – average distance between cracks after

	localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_STIFF *x*	Tension stiffening Units: none Acceptable range: <0; 1> Default value: 0.0
Compressive properties	
EPS_CP *x*	Plastic strain at compressive strength. Units: none Acceptable range: <minimal real number; 0> Default value: -0.001 Generation formula: FC/E
{ FC0 | F_C0 | RC0 | R_C0 } *x*	Onset of non-linear behavior in compression. Units: $F/(l^2)$ Acceptable range: (FC,-2 FT) Default value: -20 f_F / f_l^2 Generation formula: FC*2/3
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
Miscellaneous properties	
EXC *x*	Eccentricity, defining the shape of the failure surface Units: Acceptable range: <0.5; 1.0> Default value: 0.52
BETA *x*	Multiplier for the direction of the plastic flow. Units: Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0

RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used. Units: none Acceptable range: <0; 1> Default value: 0.25
FT_MULTIP *x*	Multiplier for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Default value: 2.1
SHEAR_FACTOR *x*	Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; +> Default value: 20
AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked if not higher than tensile strength. If yes, the shear stress on the crack is set to current value of the tensile strength. If set to 1, the aggregate interlock calculation based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock

	based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness
BETA_FATIGUE *x*	Exponent for fatigue calculation. Units: none Acceptable range: (0; +> Default value: 0.06
KSI_FATIGUE *x*	Factor for fatigue damage calculation based on crack opening and closing (ΔCOD). When set to -1, it activates the trilinear stress based damage evolution (and no COD based damage is calculated). Units: none Acceptable range: (0; 1> or -1 Default value: 0.0001
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration.

	Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
SUBSTEPS_PER_FT x[18]	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x[19]	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

3.6.2.8 Sub-command &3DNONLINCEMENTITIOUS3

&3DNONLINCEMENTITIOUS3:
TYPE "*CC3DNonLinCementitious3*" { E *x* |{ MU | POISSON | NY } *x* | { FT | RT | F_T | R_T} *x* |
{ FC | RC | F_C | R_C} *x* | GF *x* | CRACK_SPACING *x* |
TENSION_STIFFENING *x* |

[18] Supported since version 5.7.0

[19] Supported since version 5.7.0

EPS_VP *x* | { FC0 | RC0 | F_C0 | R_C0} *x* | SOFT_T *x* | EXC *x* | A *x* | B *x* | C *x* | ORDER *x* | RHO *x* | ALPHA *x* | FT_MULT *x* | SHEAR_FACTOR *x* | AGG_INTERLOCK *x* | AGG_SIZE *x* | LIMIT_TAU_CRACK *x* | | UNLOADING *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K | SUBSTEPS_PER_FT *x* | MAX_SUBSTEPS *x* }+

This material is an advanced version of 3DNONLINCEMENTITIOUS2 material that can handle the increased deformation capacity of concrete under triaxial compression. It is suitable for problems including confinement effects. The parameters for this material model can be calibrated based on compressive cylinder strength of concrete. Recommended values for various concrete compressive strengths are listed in the table after the parameter descriptions.

Table 73: &3DNONLINCEMENTITIOUS3 sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus Units: MPa Acceptable range: (0; maximal real number> Recommended value : From table below
{ MU | POISSON | NY } *x*	Poisson's ratio (ν) Units: none Acceptable range: <-1; 0.5) Recommended value : From table below
{ FT | RT | F_T | R_T} *x*	Tensile strength (f_t) Units: MPa Acceptable range: (0; -FC/2) Recommended value : From table below
{ FC | RC | F_C | R_C} *x*	Compressive strength (f_c) Units: MPa Acceptable range: <minimal real number; min(FC0, -2 FT)) Default value: -30
Tensile properties	
GF *x*	Specific fracture energy (G_f) Units: MN/m Acceptable range: (0; maximal real number> Recommended value : From table below

CRACK_SPACING *x*	Crack spacing – average distance between cracks after localization. If zero crack spacing is assumed to be equal to finite element size. Units: l Acceptable range: <0; maximal real number> Default value: 0.2
TENSION_STIFFENING *x*	Tension stiffening parameter Units: none Acceptable range: <0; 1> Default value: 0.4
Compressive properties	
EPS_VP *x*	Plastic volumetric strain at maximum compressive strength ($\varepsilon^{p}_{v,t}$). Units: none Acceptable range: <minimal real number; 0> Recommended value : From table below Generation formula: (FC/E) * (1-2*MU)
{ FC0 | F_C0 | RC0 | R_C0 } *x*	Onset of non-linear behavior in compression (f_{co}) Units: MPa Acceptable range: (FC,-2 FT) Recommended value : From table below
SOFT_T *x*	Slope of softening curve t Units: none Acceptable range: <0; maximal real number> Recommended value : From table below
X_LOC_COMP	Critical compressive displacement. Strain localization is not used in this model and this value is fixed to 1.0. Units: none Acceptable range: <0; maximal real number> Recommended value : 1.0
Miscellaneous properties	
EXC *x*	Eccentricity (e), defining the shape of the failure surface Units: none Acceptable range: <0.5; 1.0> Recommended value : From table below

A *x* B *x* C *x*	Plastic potential function parameters Units: none Acceptable range: any real number Recommended value : From table below
ORDER *x*	Polynomial order (n) of the plastic potential function Units: none Recommended value : 3
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M/f_t^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
FIXED *x*	Fixed smeared crack model will be used Units: none Acceptable range: <0; 1> Default value: 0
FT_MULT *x*	Multiplier (λ_t) for tensile strength in the plastic part of the fracture-plastic model in order to ensure that plastic surface and fracture surface intersect each other. Units: none Acceptable range: <0; +> Recommended value : From table below
SHEAR_FACTOR *x*	Shear factor (r_g) that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law. Units: none Acceptable range: <0; maximal real number> Default value: 20

AGG_INTERLOCK	This parameter activates or deactivates the aggregate interlock calculation (see AGG_SIZE parameter). If set to 0, the aggregate interlock calculation is deactivated. In this case the shear stresses on the crack are checked if not higher than tensile strength. If yes, the shear stress on the crack is set to current value of the tensile strength. If set to 1, the aggregate interlock calculation based on modified compression field theory by Colling is used to determine a shear strength of cracked concrete based on the current crack opening and aggregate size. Units: none Acceptable range: <0; 1> Default value: 0
AGG_SIZE *x*	Aggregate size for the calculation of aggregate interlock based on the modified compression field theory by Collins. When this parameter is set. The shear strength of the cracked concrete is calculated using the MDF theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material. Units: l Acceptable range: (0; +> Default value: 0.02 f_l
LIMIT_TAU_CRACK	If this parameter is set to 1, the shear stress on the crack is limited to the current value of tensile strength in case the AGG_INTERLOCK is activated. This means the shear strength provided by the modified compression field theory cannot be higher than the current values of tensile strength. This is the default behaviour if the AGG_INTERLOCK is set to 0. Units: none Acceptable range: <0; 1> Default value: 0
UNLOADING *x*	Unloading factor, which controls crack closure stiffness. Acceptable range: <0; 1) 0 - unloading to origin (default) $0.\overline{9}$ - unloading direction parallel to the initial elastic stiffness

IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command.
SUBSTEPS_PER_FT x[20]	This command activates substepping in the material model. This is used to eliminate huge strain increments, which may result in large inaccuracies in the evaluation of the new stress state. The value after this command defines how many steps should be made to reach the tensile strength. This is used also after the cracking, but the maximum allowable strain increment is kept constant to satisfy this criterion. Units: none Acceptable range: <1; maximal real number > Default value: 3
MAX_SUBSTEPS x[21]	This parameter is used in connection with the parameter SUBSTEPS_PER_FT. It defines the maximal number of steps allowed in the substepping algorithm. Units: none Acceptable range: <1; maximal real number > Default value: 10

[20] Supported since version 5.7.0

[21] Supported since version 5.7.0

Recommended values table :

FC	20	30	40	50	60	70
E	24377	27530	30011	32089	33893	35497
MU	0.2	0.2	0.2	0.2	0.2	0.2
FC	-20	-30	-40	-50	-60	-70
FT	1.917	2.446	2.906	3.323	3.707	4.066
FT_MULT	1.043	1.227	1.376	1.505	1.619	1.722
EXC	0.5281	0.5232	0.5198	0.5172	0.5151	0.5133
FC0	-4.32	-9.16	-15.62	-23.63	-33.14	-44.11
EPS_VP	$4.92 \cdot 10^{-4}$	$6.54 \cdot 10^{-4}$	$8.00 \cdot 10^{-4}$	$9.35 \cdot 10^{-4}$	$1.06 \cdot 10^{-3}$	$1.18 \cdot 10^{-3}$
SOFT_T	$1.33 \cdot 10^{-3}$	$2.00 \cdot 10^{-3}$	$2.67 \cdot 10^{-3}$	$3.33 \cdot 10^{-3}$	$4.00 \cdot 10^{-3}$	$4.67 \cdot 10^{-3}$
A	7.342177	5.436344	4.371435	3.971437	3.674375	3.43856
B	-8.032485	-6.563421	-5.73549	-5.430334	-5.202794	-5.021407
C	-3.726514	-3.25626	-3.055953	-2.903173	-2.797059	-2.719067
ORDER	3	3	3	3	3	3
GF	$4.87 \cdot 10^{-5}$	$6.47 \cdot 10^{-5}$	$7.92 \cdot 10^{-5}$	$9.26 \cdot 10^{-5}$	$1.05 \cdot 10^{-4}$	$1.17 \cdot 10^{-4}$

FC	80	90	100	110	120
E	36948	38277	39506	40652	41727
MU	0.2	0.2	0.2	0.2	0.2
FC	-80	-90	-100	-110	-120
FT	4.405	4.728	5.036	5.333	5.618
FT_MULT	1.816	1.904	1.986	2.063	2.136
EXC	0.5117	0.5104	0.5092	0.5081	0.5071
FC0	-56.50	-70.30	-85.48	-102.01	-114.00
EPS_VP	$1.30 \cdot 10^{-3}$	$1.41 \cdot 10^{-3}$	$1.52 \cdot 10^{-3}$	$1.62 \cdot 10^{-3}$	$1.73 \cdot 10^{-3}$
SOFT_T	$5.33 \cdot 10^{-3}$	$6.00 \cdot 10^{-3}$	$6.67 \cdot 10^{-3}$	$7.33 \cdot 10^{-3}$	$8.00 \cdot 10^{-3}$
A	3.245006	3.082129	2.942391	2.820644	2.713227
B	-4.871993	-4.745867	-4.637358	-4.542587	-4.458782
C	-2.659098	-2.611426	-2.572571	-2.540158	-2.512681
ORDER	3	3	3	3	3
GF	$1.29 \cdot 10^{-4}$	$1.40 \cdot 10^{-4}$	$1.50 \cdot 10^{-4}$	$1.61 \cdot 10^{-4}$	$1.71 \cdot 10^{-4}$

3.6.2.9 Sub-command &SBETAMATERIAL

&SBETAMATERIAL:
TYPE “CCSBETAMaterial” { E *x* | { MU | POISSON | NY } *x* |{ FT | RT | F_T | R_T} *x* |{ FC | RC | F_C | R_C} *x* | GF *x* | WD *x* | EPS_C *x* | SHEAR *x* | ISOFT *x* | C1 *x* | C2 *x* | C3 *x* | CSOFT *x* | COMPRED *x* | CD *x* | CS *x* | ROTATED CRACKS | RHO *x* | ALPHA *x* | DAMPING_MASS x_M DAMPING_STIFF x_K }+

The parameters for this material model can be generated based on compressive cube strength of concrete R_{cu} (see Table 74).). This value should be positive specified in MPa and then transformed to the current units.

Table 74: &CCSBETAMATERIAL sub-command parameters

Parameter	Description
Basic	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.2
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 3 f_F / f_l^2 Generation formula: $FT = 0.24 R_{cu}^{\frac{2}{3}}\ f_F / f_l^2$
{ FC | RC | F_C | R_C} *x*	Compressive strength Units: F/(l^2) Acceptable range: <minimal real number; 0) Default value: -30 f_F / f_l^2 Generation formula: $FC = -0.85 R_{cu}\ f_F / f_l^2$
Tension	
ISOFT *x*	Type of tension softening. Units: none Acceptable range: <1.0;5.0> 1.0=Exponential

	2.0=Linear 3.0=Local strain 4.0=SFRC 5.0=SFRC local strain Default value: 1.0
Case ISOFT = 1.0 (Exponential)	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Generation formula: GF = 0.000025 FT
C1 *x*	Softening parameter 1 Hidden
C2 *x*	Softening parameter 2 Hidden
C3 *x*	Softening parameter 3 Hidden
Case ISOFT = 2.0 (Linear)	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Generation formula: GF = 0.000025 FT
C1 *x*	Softening parameter 1 Hidden
C2 *x*	Softening parameter 2 Hidden
C3 *x*	Softening parameter 3 Hidden
Case ISOFT = 3.0 (Local strain)	
GF *x*	Specific fracture energy Hidden
C1 *x*	Softening parameter 1 Hidden
C2 *x*	Softening parameter 2 Hidden

C3 *x*	Softening parameter 3 Units: none Generation formula for minimum value: C30 = FT/E Acceptable range: <C30; maximal real number> Default value: C30
Case ISOFT = 4.0 (SFRC)	
GF *x*	Specific fracture energy Units: F/l Acceptable range: (0; maximal real number> Generation formula: GF = 0.00125 FT
C1 *x*	Softening parameter 1 Units: none Acceptable range: <0; 2> Default value: 1.
C2 *x*	Softening parameter 2 Units: none Acceptable range: <0; 1> Default value: 0.
C3 *x*	Softening parameter 3 Hidden
Case ISOFT = 5.0 (SFRC local strain)	
GF *x*	Specific fracture energy Hidden
C1 *x*	Softening parameter 1 Units: none Acceptable range: <0; 2> Default value: 1.
C2 *x*	Softening parameter 2 Units: none Acceptable range: <0; 1> Default value: 0.
C3 *x*	Softening parameter 3 Units: none Generation formula for minimum value: C30 = FT/E

	Acceptable range: <C30; maximal real number> Default value: C30
Compression	
EPS_C *x*	Compressive strain at compressive strength in the uniaxial compressive test. Normally should be equal to 2*FC/E. Units: none Acceptable range: <minimal real number; 0) Default value: 2*FC/E
COMPRED *x*	Reduction of compressive strength due to cracks. Units: none Acceptable range: <0; 1> Default value: 0.8
CSOFT *x*	Type of compression softening. Units: none Acceptable range: <1.0;2.0> 1.0=Crush band 2.0=Softening modulus Default value: 1.0
Case CSOFT = 1.0 (Crush band)	
WD *x*	Critical compressive displacement Units: l Acceptable range: <minimal real number; 0) Default value: -0.0005 f_l
CD *x*	Compression softening parameter Hidden
Case CSOFT = 2.0 (Softening modulus)	
WD *x*	Critical compressive displacement Hidden
CD *x*	Compression softening parameter Units: none Acceptable range: <0; maximal real number> Default value: 0.2
Shear	
SHEAR *x*	Shear retention factor. Could be fixed or variable.

	Format for fixed shear retention factor: (Picture, ”MISC_Shear_Retention_Fixed.bmp”) SHEAR FIXED x Format for variable shear retention factor: (picture, ”MISC_Shear_Retention_Variable.bmp”) SHEAR VARIABLE Units: none Acceptable range for fixed value: <0; 1.0> Default value: VARIABLE
CS x	Tension-compression interaction. Units: none Acceptable values: 0.2, 0.4, 0.6 0.6=Linear 0.4=Hyperbola A 0.2=Hyperbola B Default value: 0.6 (Linear)
ROTATED CRACKS	Activates rotated crack model. If not used fixed crack model is considered. Units: none Acceptable range: none Default value: not used
Miscellaneous	
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
RHO x	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA x	Coefficient of thermal expansion. Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012

3.6.3 Elastic – Plastic materials

3.6.3.1 Sub-command &VON_MISES_PLASTICITY and &DRUCKER_PRAGER_PLASTICITY

Syntax:

&VON_MISES_PLASTICITY:
TYPE "CC3DBiLinearSteelVonMises" { E *x* | { MU | POISSON | NY } *x* | YIELD [STRENGTH] *x* | HARDENING [MODULUS] *x* | {R *x*} | {K1 *x*} {K2 *x*} RHO *x* | ALPHA *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} | DAMPING_MASS x_M DAMPING_STIFF x_K }+

Table 75: &VON_MISES_PLASTICITY sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 210 x 10^3 f_F / f_l^2
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.3
YIELD *x*	Yield strength. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 200 f_F / f_l^2
HARDENING *x*	Hardening/softening modulus. HARDENING MODULUS *x* Units: F/(l^2) Acceptable range: <minimal real number; maximal real number> Default value: 0.0 f_F / f_l^2
Cycling behavior parameters	
R	Scaling of the initial yield surface. If equal to 0, no cycling behavior is considered. For values greater than 0 Bauschinger effect is included. If equal to 1.

	Format: R x Units: none Acceptable range: <0; 1> Default value: 0.7 (0 – no Bauschinger effect considered)
K1	Bauschinger hardening slope Format: K1 x Units: F/(l^2) Acceptable range: (0; maximal real number) Default value: 74 000 f_F / f_l^2
K2	Bauschinger memory Format: K2 x Units: none Acceptable range: (0; maximal real number) Default value: 1000
Miscellaneous properties	
RHO x	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA x	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the

	program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.

Syntax:

&DRUCKER_PRAGER_PLASTICITY:
TYPE "*CC3DDruckerPragerPlasticity*" { E *x* | { MU | POISSON | NY } *x* K *x* |
ALPHA_DP *x* | WD *x* | BETA *x* | RHO *x* | ALPHA *x* | IDEALISATION
{ 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D} |
DAMPING_MASS x_M DAMPING_STIFF x_K}+

The parameters for this material model can be generated based on compressive and tensile strength of the material R_c and R_t (see Table 76). These values should be specified in MPa and then transformed to the current units.

Table 76: &DRUCKER_PRAGER_PLASTICITY sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{ MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.2
ALPHA_DP *x*	Drucker-Prager criterion parameter Units: none Acceptable range: (0; maximal real number> Default value: 0.12 Generation formula:
K *x*	Drucker-Prager parameter k Units: F/(l^2) Acceptable range: <0; maximal real number>

	Default value: 0.0 f_F / f_l^2 Generation formula: $K = \lvert R_c \rvert \left(\sqrt{\frac{1}{3}} - \text{ALPHA\_DP} \right) f_F / f_l^2$
Compressive properties	
WD *x*	Critical compressive displacement Units: l Acceptable range: <0; maximal real number> Default value: -0.0005 f_l
Miscellaneous properties	
BETA *x*	Multiplier for the direction of the plastic flow. Units: none Acceptable range: <minimal real number; maximal real number> Recommended range: (-2; 2) Default value: 0.0
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.0023 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the

	program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.

3.6.4 User Material

3.6.4.1 Sub-command &USER_MATERIAL

Syntax:

&USER_MATERIAL :
TYPE { "CC3DUserMaterial" } { E x | {MU | POISSON | NY } *x* |
{*UserParameterN*} *x* | DAMPING_MASS x_M DAMPING_STIFF x_K}+

Table 77: &USER_MATERIAL sub-command parameters

Parameter	Description
Basic properties (inherited from elastic material)	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 210 x 10^3 f_F / f_l^2
{MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.3
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D

	configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D, SHELL, BEAM_3D, MEMBRANE_AXI} Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element, where it is used. So in most cases it is not needed to use this command. In certain cases, however, the program cannot determine correctly the idealisation to use. Such a case is for instance, if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
Advanced properties	
UserMaterialDLL "user_lib_name.dll"	The name of the user-provided dynamic link library (DLL) implementing the material model. Please note this parameter has to be the first one because the others, except for those inherited from the elastic material, are not be known to the kernel until the user DLL is loaded.
User defined properties	
{*UserParameterN*} *x*	The acual parameter names are defined in the DLL provided by the user. Only floating point parameters are supported.

3.6.5 Interface Material

3.6.5.1 Sub-command &INTERFACE_MATERIAL

Syntax:

&INTERFACE_MATERIAL :
TYPE { "CC2DInterface" | "CC3DInterface" } { {K_NN | KNN} *x* | {K_TT | KTT} *x* | COHESION *x* | FRICTION *x* | { FT | RT | F_T | R_T} *x* {TENSION_SOFT_HARD_FUNCTION *n* } | {COHESION_SOFT_HARD_FUNCTION *n* } | K_NN_MIN *x* | K_TT_MIN *x* | RESET_DISPLS[22] *n* TENSION_ELIPS *n* }+

Table 78: &INTERFACE_MATERIAL sub-command parameters

Parameter	Description
Basic properties	
{K_NN | KNN} *x*	Normal stiffness. Units: F/l^3 Acceptable range: (0; maximal real number>

[22] Available starting from ATENA version 4.3.1.

	Default value: 200 x 10^6 f_F / f_l^3
{K_TT | KTT}	Tangential stiffness. Units: F/l Acceptable range: (0; maximal real number> Default value: 200 x 10^6 f_F / f_l^3
{ FT | RT | F_T | R_T} *x*	Tensile strength Units: F/l^3 Acceptable range: <0; maximal real number> Default value: 0 f_F / f_l^2
COHESION *x*	Cohesion. Units: $F/(l^2)$ Acceptable range: <0; maximal real number> Default value: 0.0 f_F / f_l^2
FRICTION *x*	Friction coefficient. If zero, interface behaves like a no-tension element and full contact in compression is assumed. Units: none Acceptable range: <0; maximal real number> Default value: 0.0
TENSION_SOFT_HARD_FUNCTION	Function which defines uniaxial relative stress-displacement relationship. Relationship should be defined as a set of points starting from (0; 0) and only positive values should be specified. X-coordinates of this function mean normal displacement (units l, range <0,maximal real number), Y-coordinates represent the relative tensile strength with respect to FT (units NONE, range <0;maximal real number)) Default function values: X: 0.0; 0.0001 Y: 1.0; 0.0 Format: TENSION_SOFT_HARD_FUNCTION *n* Units: none Acceptable range: (1; maximal integer> Default value: none, see command FUNCTION
COHESION_SOFT_HARD_FUNCTION	Function which defines uniaxial relative stress-displacement relationship. Relationship should be defined as a set of points starting from (0; 0) and only positive values should be specified. X-coordinates of this function mean shear displacement (units l,

	range <0,maximal real number), Y-coordinates represent the relative tensile strength with respect to COHESION (units NONE, range <0;maximal real number)) Default function values: X: 0.0; 0.0001 Y: 1.0; 0.0 Format: COHESION_SOFT_HARD_FUNCTION *n* Units: none Acceptable range: (1; maximal integer> Default value: none, see command FUNCTION
Miscellaneous properties	
K_NN_MIN *x*	Minimal normal stiffness for numerical purposes. Units: F/l^3 Acceptable range: (0; maximal real number> Default value: K_NN / 1000
K_TT_MIN *x*	Minimal tangential stiffness for numerical purposes. Units: F/l^3 Acceptable range: (0; maximal real number> Default value: K_TT / 1000
RESET_DISPLS *n*	For $n>0$ this flag forces realignment of the bottom (slave) interface surface/lines of the gap element with respect to its top (master) surface/line, (i.e. the top surface/line is glued to the surrounding structure whilst the bottom surface/line is slipping). This happens at the end of each step. For $n<0$ the above applies in opposite way. For $n=0$ no realignment is carried out. The top surface/line of the gap element is the surface/line, whose nodal ids are entered firstly in the gap's incidences. If $n=\pm1$, each slave node is given coordinates of its master node. Consequently, this projection is suitable only for gap elements with zero thickness. If $n=\pm2$, slave nodal locations are calculated as the normal projection of the corresponding master nodes into surface/line defined by the deformed slave nodes. If $n=\pm3$, slave nodal locations are set to coincide with the corresponding master nodes and thereafter they are shifted in the direction to the original position of the slave nodes surface/line. The shift equals to the original gap thickness.
TENSION_ELIPS	Flag for activating/deactivating the ellipsoidal shape of the criterion in tension, i.e. for tensile normal stress. Units: none

	Acceptable range: -1 or 1 Default value: 1 If equal to 1, the elipse is used for the interface criterion in tension connecting the pure cohesion point and pure tension point. If <= 0 is specified, sharp corner is assumed in the interface criterion at the stress value corresponding to tensile strength.

3.6.6 Material Type for Reinforcement

3.6.6.1 Sub-commands &REINFORCEMENT, &REINFORCEMENT_WITH_CYCLING_BEHAVIOR , &SMEARED_REINFORCEMENT and &CIRCUMFERENTIAL_SMEARED_REINFORCEMENT

Syntax:
&REINFORCEMENT
TYPE "CCReinforcement" { E *x* | FUNCTION *n* | F_MULTIP *x*}+

Table 79: &REINFORCEMENT command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 210 x 10^3 MPa
FUNCTION *a*	Function which defines uniaxial stress-strain relationship. Relationship should be defined as a set of points starting from (0, 0) and only positive values should be specified. Same relationship will be used in compression. Units: none Acceptable range: (1; maximal integer> Default value: none, see command &FUNCTION.
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3

ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
F_MULTIP *x*	Function multiplier. Can be used to scale the function defining the stress-strain relationship. Units: none Acceptable range: (1; maximal real number> Default value: 1.0
COMPRESSION *x*	Compression flag. Can be used to deactivate the compressive response of the reinforcement. 0 – reinforcement cannot carry any compressive forces, but only tensile. 1 – reinforcement works both in tension and compression. Units: none Acceptable range: 0 or 1 Default value: 1

Syntax:
&REINFORCEMENT_WITH_CYCLING_BEHAVIOR:
TYPE "CCCyclingReinforcement" { E *x* | FUNCTION *n* }+

Table 80: &REINFORCEMENT_WITH_CYCLING_BEHAVIOR sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: $F/(l^2)$ Acceptable range: (0; maximal real number> Default value: 210 x 10^3 MPa
FUNCTION *n*	Function which defines uniaxial stress-strain relationship. Relationship should be defined as a set of points starting from (0, 0) and only positive values should be specified. Same relationship will be used in compression. Units: none Acceptable range: (1; maximal integer> Default value: none, see command &FUNCTION
R *x*	Bauschinger effect exponent of Menegotto-Pinto model. Units: none Acceptable range: (0; maximal real number>

	Default value: 20
C1 *x*	Menegotto-Pinto model parameter Units: none Acceptable range: (0; maximal real number> Default value: 0.925
C2 *x*	Menegotto-Pinto model parameter Units: none Acceptable range: (0; maximal real number> Default value: 0.15
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012

&SMEARED_REINFORCEMENT

TYPE "CCSmearedReinf" { E *x* | FUNCTION *n* | RATIO *x* | DIRECTION x_1 x_2 [x_3] | RHO *x* | ALPHA *x* | F_MULTIP *x* }+

Table 81: &SMEARED_REINFORCEMENT command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: $F/(l^2)$ Acceptable range: (0; maximal real number> Default value: 210 x 10^3 MPa
FUNCTION *a*	Function which defines uniaxial stress-strain relationship. Relationship should be defined as a set of points starting from (0, 0) and only positive values should be specified. Same relationship will be used in compression. Units: none Acceptable range: (1; maximal integer>

	Default value: none, see command &FUNCTION.
RATIO *x*	Cross-sectional area ratio of the smeared reinforcement with respect to the base material. Units: none Acceptable range: <0;1> Default value: 0.01
DIRECTION x_1 x_2 [x_3]	Unit vector defining the smeared reinforcement direction. The third component x_3 is required in case of 3D analysis. Units: l Acceptable range: <minimal real; maximal real number> Default value: 1 0 [0]
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
F_MULTIP *x*	Function multiplier. Can be used to scale the function defining the stress-strain relationship. Units: none Acceptable range: (1; maximal real number> Default value: 1.0
COMPRESSION *x*	Compression flag. Can be used to deactivate the compressive response of the reinforcement. 0 – reinforcement cannot carry any compressive forces, but only tensile. 1 – reinforcement works both in tension and compression. Units: none Acceptable range: 0 or 1 Default value: 1

&CIRCUMFERENTIAL_SMEARED_REINFORCEMENT

TYPE "CCCircumferentialSmearedReinforcement" { E *x* | FUNCTION *n* | RATIO *x* | RHO *x* | ALPHA *x* | F_MULTIP *x* }+

Table 82: & CIRCUMFERENTIAL_SMEARED_REINFORCEMENT command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 210 x 10^3 MPa
FUNCTION *a*	Function which defines uniaxial stress-strain relationship. Relationship should be defined as a set of points starting from (0, 0) and only positive values should be specified. Same relationship will be used in compression. Units: none Acceptable range: (1; maximal integer> Default value: none, see command &FUNCTION.
RATIO *x*	Cross-sectional area ratio of the smeared reinforcement with respect to the base material. Units: none Acceptable range: <0;1> Default value: 0.01
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
F_MULTIP *x*	Function multiplier. Can be used to scale the function defining the stress-strain relationship. Units: none Acceptable range: (1; maximal real number> Default value: 1.0

3.6.7 Material Type for Spring

3.6.7.1 Sub-command &SPRING

Syntax:
&SPRING:
TYPE "CCSpringMaterial" { K x | FUNCTION n | DAMPING_MASS x_M
DAMPING_STIFF x_K}$_+$

Table 83: &SPRING sub-command parameters

Parameter	Description
Basic properties	
K x	Initial stiffness. Units: F/l Acceptable range: (0; maximal real number> Default value: 1000.0
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
FUNCTION n	Function which defines uniaxial spring relationship. Relationship should be defined as a set of points starting in compression passing through (0, 0) and extending into tension. a Units: none Acceptable range: (1; maximal integer> Default value: none, see command &FUNCTION

3.6.8 Microplane Material Type for Concrete

3.6.8.1 Sub-command &MICROPLANE

Syntax:
&MICROPLANE:
{ &MICROPLANE4 | &CCM4 | &CCM4R | &CCM4RC }

&MICROPLANE4

The following microplane based models are supported in ATENA material library:

Material models	Description
&CCMICROPLANE4	Original version of the M4 microplane model for concrete developed by Prof. Bazant and Dr. Cannera, (Northwestern University, IL)
&CCM4	Enhanced version of the M5 developed by Prof. Bazant and Mr.

	Zi, (Northwestern University, IL). This version is prepared for being size independent (resulting in M5 model). A proper calibration is currently in progress and will be added in ATENA as soon as available.
&CCM4R	Extension of the CCM4 material for analysis taking into the effect of loading rate.
&CCM4RC	Extension of the CCM4R material model that also accounts for the effect of material creep and shrinkage.

&MICROPLANE4
TYPE "CCMicroplane4" { E *x* | NP *n* | K1 *x* | K2 *x* | K3 *x* | K4 *x* | BAND *x* |
IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN,
AXISYMMETRIC, 3D} } C1 *x* C2 *x* … C21 *x* }+

Table 84: &MICROPLANE sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.3
Special microplane parameters	
NP *i*	Number of microplanes Units: None Acceptable values: 21,28,37,61 Default value: 21
K1 *x*	Microplane parameter k_1. Units: None Acceptable range: <0; maximal real number> Default value: 1.5×10^{-4}

	Generation formula: $k_1 = 0.1156 \quad R_{cu} / E$
K2 *x*	Microplane parameter k_2. Units: None Acceptable range: <0; maximal real number> Default value: 500
K3 *x*	Microplane parameter k_3. Units: None Acceptable range: <0; maximal real number> Default value: 15
K4 *x*	Microplane parameter k_4. Units: None Acceptable range: <0; maximal real number> Default value: 150
BAND *x*	Crack band size. Units: l Acceptable range: <0; maximal real number> Default value: 0.003 f_l
Miscellaneous properties	
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not

	needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.
C1, C2, C3, …. C21	Microplane internal parameters. Program contains default values for these parameters, but the expert users or users familiar with the original work can modify them directly in order to obtain a better fit with experimental data. Syntax: Cx *x* Default values: (see theory manual for details) c1 =6.20e-1 Normal bound. param. c2 =2.76 Normal bound. param. c3 =4.00 Normal plasticity (EB_N) c4 =70.00 Strain ratio: normal/vol c5 =2.50 Tens. dev. bound. param. c6 =1.30 Comp. dev. bound. yield c7 =50.00 Deviatoric plasticity (EB_D) c8 =8.00 Compressive strength (FCP) c9 =1.30 Dev. bound. param. c10 =7.30e-1 Fric.b. initial slope c11 =2.00e-1 Fric.b.\sig_N inter.@\sig_V=0 c12 =7.00e+3 Fric.b.Speed\sig_N goes zero c13 =2.30e-1 Tensile vol.b. vert. scalar c14 =8.00e-1 Tensile vol.b. slope c15 =1.00 Tensile vol.b. horiz. yield c16 =2.00e-2 Unl. volumetric coeff. c17 =1.00e-2 Unl. volumetric coeff. c18 =1.000 Tensile vol.b. unload.coeff c19 =0.40 Unloading slope interpolator c20 =14.00e-2 Residual strength c21 =0.990 Unloading slope Int. in tens

&CCM4:
TYPE “CCM4” {&CCM4Params}+

&CCM4Params:

{ E *x* | Nplane *n* | K1 *x* | K2 *x* | K3 *x* | K4 *x* | ES0 *x* | VA *x* | FC *x* | TSH *x* | PSI *x* | ETA_V *x* | ETA_D *x* | ETA_N *x* | MY_U1 *x* | IDEALISATION { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D}

Table 85: &CCM4Params sub-command parameters

Parameter	Description
Basic properties	
E *x*	Elastic modulus. Units: F/(l^2) Acceptable range: (0; maximal real number> Default value: 30 x 10^3 f_F / f_l^2 Generation formula: $E = (6000 - 15.5R_{cu})\sqrt{R_{cu}}\ f_F / f_l^2$ (this formula is valid only if R_{cu} is compressive cube strength given as positive number in MPa.)
{MU | POISSON | NY } *x*	Poisson's ratio. Units: none Acceptable range: <0; 0.5) Default value: 0.3
Special microplane parameters	
Nplane *i*	Number of microplanes Units: None Acceptable values: 21,28,37,61 Default value: 28
K1 *x*	Microplane parameter k_1. Units: None Acceptable range: <0; maximal real number> Default value: 1.65×10^{-4} Generation formula: $k_1 = 0.1156\ R_{cu} / E$
K2 *x*	Microplane parameter k_2. Units: None Acceptable range: <0; maximal real number> Default value: 160
K3 *x*	Microplane parameter k_3. Units: None Acceptable range: <0; maximal real number>

	Default value: 6.4
K4 *x*	Microplane parameter k_4. Units: None Acceptable range: <0; maximal real number> Default value: 450
Shrinkage related parameters	
ES0 *x*	Ultimate shrinkage of thin cement paste on humidity=0.4. Units: None Default value: 0.00377
VA *x*	Volume fraction of aggregate. Units: None Default value: 0.8
FC *x*	Reference compressive strength in [MPa]. Units: MPa Default value: 39.42 MPa
TSH *x*	The time when shrinkage started in [days] Units: days Default value: 28
M5 related extra parameters (related to the material point size)	
PSI *x*	Ratio of the characteristic size of the material to the size of the current element. Units: None Default value: 1
ETA_V *x*	the ratio of the vertical line which approximates fracture affinity to epsilon plastic Units: None Default value: 1
ETA_D *x*	affinity scaling factor for the deviatoric stress boundary Units: None Default value: 1
ETA_N *x*	affinity scaling factor for the normal stress boundary Units: None Default value: 1
MY_U1 *x*	the ratio between ET and ED

	Units: None Default value: 1
Miscellaneous properties	
RHO *x*	Material density. Units: M/l^3 Acceptable range: <0; maximal real number> Default value: 0.00785 f_M / f_l^3
ALPHA *x*	Coefficient of thermal expansion Units: 1/T Acceptable range: <0; maximal real number> Default value: 0.000012
IDEALISATION	Defines the idealisation if material model with higher dimension is to be used in a finite element with lower dimension. For instance in case a 3D model is to be used in 2D configuration. Units: none Acceptable range: { 1D, PLANE_STRESS, PLANE_STRAIN, AXISYMMETRIC, 3D } Default value: program tries to determine a suitable idealisation based on the dimension of the material model and the dimension of the finite element where it is used. So in most cases it is not needed to use this command. In certain cases however the program cannot determine correctly the idealisation to use, such a case is for instance if a 3D model is to be used in 2D element. Then it is necessary to directly specify if plane stress or strain idealisation is to be used.

&CCM4R:
TYPE "CCM4R" { &CCM4RParams | &CCM4Params }+

&CCM4RParams:
{ REF_TEMPER *x* | QR *x* | CR0 *x* | CR2 *x* }

Table 86: &CCM4RParams sub-command parameters

Parameter	Description
REF_TEMPER *x*	Reference temperature. Units: ^0C Default value: 25 ^0C
QR *x*	Activation energy constant.

	Units: ${}^0K$ Default value: 1000 ${}^0K$
CR0 *x*	Boundary rate shape CR0 constant. Units: $\frac{1}{\text{sec}}$ Default value: 10^{-6} sec^{-1} 6.4
K4 *x*	Boundary rate shape CR2 constant. Units: $\frac{1}{\text{sec}}$ Default value: $8.5E^{-3}$

&CCM4RC:
TYPE "CCM4R" { &CCM4RCParams | &CCM4RParams | &CCM4Params }+

&CCM4RCParams:
{ TIME0 *x* | HUMIDITY0 *x* | TEMPERATURE0 | TAU1 *x* |
NUMBER_MAXWELL *n* | Q1 *x* | Q2 *x* | Q3 *x* | Q4
x | WC *x* | CC *x* | AC *x* | C *x* | C1 *x* | CREEP_DEGREE *x* | VOLUME_POW *x* |
LAMBDA0 *x* }

Table 87: &CCM4RCParams sub-command parameters

Parameter	Description
TIME0 *x*	Initial time. Units: Days Default value: 28 days
TEMPERATURE	Material initial temperature Units: ${}^0C$ Default value: 25 ${}^0C$
HUMIDITY	Material initial humidity. Units: None Default value: 0.94
TAU1 *x*	Te smallest relaxation time. Units: days Default value: 1.E-6 days
NUMBER_MAXWELL *n*	Number of Maxwell or Kelvin units Units: None

	Default value: 14
Q1 *x*	Creep parameter Q1, (refer to Bazant & Baweja Model B3). If negative, the parameter is estimated according to the above mentioned creep model. Units: $\frac{1}{MPa}$ Default value: -1
Q2 *x*	Creep parameter Q2, (refer to Bazant & Baweja Model B3). If negative, the parameter is estimated according to the above mentioned creep model. Units: $\frac{1}{MPa}$ Default value: -1
Q3 *x*	Creep parameter Q3, (refer to Bazant & Baweja Model B3). If negative, the parameter is estimated according to the above mentioned creep model. Units: $\frac{1}{MPa}$ Default value: -1
Q4 *x*	Creep parameter Q4, (refer to Bazant & Baweja Model B3). If negative, the parameter is estimated according to the above mentioned creep model. Units: $\frac{1}{MPa}$ Default value: -1
WC *x*	Water cement ratio. Units: None Default value: 0.4
CC *x*	Cement content. Units: $\frac{kg}{m^3}$ Default value: 100 $\frac{kg}{m^3}$
AC *x*	Aggregate cement ratio. Units: None Default value: 7.
C *x*	Proportionality constant between viscosity and microprestress

	Units: $\frac{1}{\text{MPa}^2\ \text{day}}$ Default value: $1.E^{-8}\ \frac{1}{\text{MPa}^2\ \text{day}}$
C1 *x*	Proportionality constant in computing the change of capillary tension Units: $\frac{MPa}{^0K}$ Default value: 4. $\frac{MPa}{^0K}$
CREEP_DEGREE *x*	Degree of creep function. Units: None Default value: 0.04
VOLUME_POW *x*	The power of volume fraction. Units: [None] Default value: 0.5
LAMBDA0 *x*	Slope of creep function. Units: None Default value: 1

3.6.9 &Creep Materials

The creep material definition includes a model for short-term material properties and a model for their variation in time. The former model is called BASE material model, while the latter one is CREEP model. The base model can be any material model that is written in incremental form. Models written in total formulation are not compatible with creep analysis. SHORT_TERM_MATERIAL_DATA entry comprises all short-term material parameters listed in a section describing the short-term material (starting with short tem material type name in quotes).

Syntax:

```
&CREEP_MATERIAL:
TYPE {&CCModelB3_DATA | &CCModelB3Improved_DATA |
      &CCModelBP_KX_Data | &CCModelCEB_FIP78_DATA |
      &CCModelACI78_DATA | &CCModelCSN731202_DATA  |
      &CCModelBP1_DATA | &CCModelBP2_DATA |  &CCModelGeneral_DATA |
      &CCModelFIB_MC2010_DATA | &CCModelEN1992_DATA } [
      MAT_CONSTR_TIME age ]  BASE [ {TYPE | MATERIAL} ]
      "short_term_material_type_name"
SHORT_TERM_MATERIAL_DATA
```

The parameter MAT_CONSTR_TIME *age* sets age of elements using this material. During the creep material execution it is subtracted from its loading and current times *t'*, *t*. The parameter BASE contains material type to be used for the short term material model. See Table 56 for more information about the available material models for this parameter. After that the parameters of the short term material will follow.

3.6.9.1 Sub-command &CCModelB3_DATA

&CCModelB3_DATA
CCModelB3 { CONCRETE *concrete_type* | THICKNESS *thick* | FCYL28 *fcyl28* | E28 *e28* | HUMIDITY *humidity* | DENSITY *density* | AC *ac* | WC *wc* | [SHAPE] FACTOR *sfactor* | {WATER | AIR | STEAM} [CURING] | [END] [OF] [CURING] TIME *endcuring* | TOTAL_LOSS *total_water_loss* | {LOAD | CURRENT} [TIME] *time* | {LOSS | SHRINKAGE | COMPLIANCE} *measured_val* }+

Table 88: & CCModelB3 sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported for static and types 1-4 for transport analysis. More information available in the Atena Theory Manual. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
E28 *e28*	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. Default value: calculated from *fcyl28*.
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
AC *ac*	Total aggregate/cement ratio. Default value: 7.04
WC *wc*	Water/cement ratio. Default value: 0.63

[SHAPE] FACTOR *sfactor*	Cross section shape factor. It should be 1, 1.15, 1.25, 1.3, 1.55 for slab, cylinder, square prism, sphere, cube, respectively. Default value 1.25
{WATER | AIR | STEAM} [CURING]	Curing conditions, either under in water or air under normal temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
TOTAL_LOSS *total_water_loss*	Total water loss (at zero humidity and infinite time). Default: 0 [kg]
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
{LOSS | SHRINKAGE | COMPLIANCE} *measured_val*	Measured water loss (at current humidity) | shrinkage | material compliance *measured_val* for previously specified load and current time. Units of water loss must correspond to units of *total_water_loss* , shrinkage is dimension-less and compliance is input in kPa^{-1}.

3.6.9.2 Sub-command &CCModelB3Improved_Data

&CCModelB3Improved_Data
CCModelB3Improved { CONCRETE concrete_type | THICKNESS thick | FCYL28 $f_{cyl,28}$ | E28 E_{28} | FCYL0_28 $f_{cyl0,28}$ | FT28 $f_{t,28}$ | GF28 $G_{f,28}$ | ALPHA α | HUMIDITY humidity | DENSITY density | AC ac | WC wc | [SHAPE] FACTOR sfactor | {WATER | AIR | STEAM} [CURING] | [END] [OF] [CURING] TIME endcuring | EPS_A_INF $\varepsilon_{a,\infty}$ | TAU_A τ_a | TIME_S t_s | H_A_INF $h_{a,\infty}$ | TOTAL_LOSS total_water_loss | {LOAD | CURRENT} [TIME] time | {LOSS | SHRINKAGE | COMPLIANCE} measured_val { HISTORY [TIME] time [HUMIDITY] humid [TEMPERATURE] temper }+ }+

Table 89: & CCModelB3Improved sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m].

	Default value: 0.0767 [m].
FCYL28 $f_{cyl,28}$	Cylindrical material strength in compression $f_{cyl}(28\,days)$. This value is crucial for the creep model's prediction, i.e. prediction of material compliance $\Phi(t,t')$ and cylindrical compression strength $f_{cyl}(t)$, shrinkage etc. The ratio of $f_{cyl}(t)/f_{cyl}(28\,days)$ may be used for overiding short f_{cyl}, f_t, G_f. Note that material compliance/rigidity is overwritten always. Default value: 35100 [kPa].
FCYL0_28 $f_{cyl0,28}$	The parameter $f_{cyl0}(28\,days)$. If specified, it is used to calculate $f_{cyl0}(t)$ and overide the corresponding value in the base material. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
GF28 $G_{f,28}$	The parameter fracture energy $G_f(28\,days)$. If specified, it is used to calculate $G_f(t)$ and overide the corresponding value in the base material. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
FT28 $f_{t,28}$	The parameter tensile strength $f_t(28\,days)$. If specified, it is used to calculate $f_t(t)$ and overide the corresponding value in the base material. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
E28 E_{28}	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. It is used by the creep model to predict material compliance $\Phi(t,t')$. If unspecified, the model calculates its value based on *fcyl28.* Default value: calculated from *fcyl28*.
ALPHA α	Coefficient of thermal expansion to be used for calculation $\Delta\varepsilon_t(\Delta T)$ within the creep material. Default value: 0
HUMIDITY *humidity*	Ambient relative humidity (0.3…1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
AC *ac*	Total aggregate/cement ratio. Default value: 7.04

WC *wc*	Water/cement ratio. Default value: 0.63
[SHAPE] FACTOR *sfactor*	Cross section shape factor. It should be 1, 1.15, 1.25, 1.3, 1.55 for slab, cylinder, square prism, sphere, cube, respectively. Default value 1.25
{WATER | AIR | STEAM} [CURING]	Curing conditions, either under in water or air under normal temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
TOTAL_LOSS *total_water_loss*	Total water loss (at zero humidity and infinite time). Default: 0 [kg]
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
{LOSS | SHRINKAGE | COMPLIANCE} *measured_val*	Measured water loss (at current humidity) | shrinkage | material compliance *measured_val* for previously specified load and current time. Units of water loss must correspond to units of *total_water_loss* , shrinkage is dimension-less and compliance is input in kPa^{-1}.
{ HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }+	For each entry of material history the data *time* , *temper* and *humid* must be input. If the data keywords are used, then it doesn't matter in which order the 3 data are input. Otherwise the indicated order is assumed. The units are days, degrees Celsius and dimension less humidity (in interval 0.3..1).
EPS_A_INF $\varepsilon_{a,\infty}$	Autogenous shrinkage at infinity time, (typically negative!). Default value= -0.
TAU_A τ_a	Half-time of autogenous shrinkage. Default value =30 days
TIME_S t_s	Time of final set of cement. Default value=5 days.
H_A_INF $h_{a,\infty}$	Final self-desiccation relatibe humidity. Default value=0.8

3.6.9.3 Sub- command &CCModelFIB_MC2010_DATA

&CCModelFIB_MC2010_DATA
CCModelFIB_MC2010 { CEMENT_CLASS { 32.5N | 32.5R | 42.5N | 42.5R | 52.5N | 52.5R } | AGGREAGETE { BASALTDENSELIMESTONE | QUARTZITE | LIMESTONE | SANDSTONE | LIGHTWEIGHTSANDSTONE } | THICKNESS *thick* | FCYL28 $f_{cyl,28}$ | E28 E_{28} | FCYL0_28 $f_{cyl0,28}$ | FT28 $f_{t,28}$ |

GF28 $G_{f,28}$ | ALPHA α | HUMIDITY *humidity* | DENSITY *density* | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* | { SHRINKAGE | COMPLIANCE} *measured_val* { HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }$_+$ }$_+$

Table 90: &CCModelFIB_MC2010 sub-command parameters

Parameter	Description
CEMENT_CLASS { 32.5N | 32.5R | 42.5N | 42.5R | 52.5N | 52.5R }	Type of cement, see e.g. http://www.cis.org.rs/en/cms/about-cement/standardization-of-cement : Strength classes of cement Cements are according to standard strength grouped into three classes, they being: • Class 32,5 • Class 42,5 • Class 52,5 Three classes of early strength are defined for each class of standard strength: • Class with ordinary early strength – N • Class with high early strength – R • Class with low early strength – L Class L can be applied only on CEM III cements. Default value: class_42_5N
AGGREAGETE { BASALTDENSELIMESTONE | QUARTZITE | LIMESTONE | SANDSTONE | LIGHTWEIGHTSANDSTONE }	Type of aggregate. Note that light weight concrete is detected for concrete with density below 2000kg/m^3 and some aditional meassures are taken for LIGHTWEIGHTSANDSTONE aggregate. Default value: QUARTZITE
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 $f_{cyl,28}$	Cylindrical material strength in compression $f_{cyl}(28\,days)$. This value is crucial for the creep model's prediction, i.e. prediction of material compliance $\Phi(t,t')$ and cylindrical compression strength $f_{cyl}(t)$, shrinkage etc. The ratio of $f_{cyl}(t)/f_{cyl}(28\,days)$ may be used for overiding short f_{cyl}, f_t, G_f. Note that material compliance/rigidity is overwritten always. Default value: 35100 [kPa].

FCYL0_28 $f_{cyl0,28}$	The parameter $f_{cyl0}(28\,days)$. If specified by a positive value, this value is used to calculate $f_{cyl0}(t)$ and overide the corresponding value in the base material. If it is specified as any negative value, $f_{cyl0}(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
GF28 $G_{f,28}$	The parameter fracture energy $G_f(28\,days)$. If specified by a positive value, this value is to calculate $G_f(t)$ and overide the corresponding value in the base material. . If it is specified as any negative value, $G_f(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
FT28 $f_{t,28}$	The parameter tensile strength $f_t(28\,days)$. If specified by a positive value, this value is used to calculate $f_t(t)$ and overide the corresponding value in the base material. If it is specified as any negative value, $f_t(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
E28 E_{28}	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. It is used by the creep model to predict material compliance $\Phi(t,t')$. If unspecified, the model calculates its value based on *fcyl28*. Default value: calculated from *fcyl28*.
ALPHA α	Coefficient of thermal expansion to be used for calculation $\Delta\varepsilon_t(\Delta T)$ within the creep material. Default value: 0
HUMIDITY *humidity*	Ambient relative humidity (0.3…1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]

{ SHRINKAGE | COMPLIANCE} *measured_val*	Measured shrinkage | material compliance *measured_val* for previously specified load and current time. Units of water loss must correspond to units of *total_water_loss* , shrinkage is dimension-less and compliance is input in kPa^{-1}.
{ HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }+	For each entry of material history the data *time* , *temper* and *humid* must be input. If the data keywords are used, then it doesn't matter in which order the 3 data are input. Otherwise the indicated order is assumed. The units are days, degrees Celsius and dimension less humidity (in interval 0.3..1).

3.6.9.4 Sub-command &CCModelEN1992_DATA

&CCModelEN1992_DATA
CCModel EN1992 { CEMENT_CLASS { 32.5N | 32.5R | 42.5N | 42.5R | 52.5N | 52.5R } | AGGREAGETE { BASALTDENSELIMESTONE | QUARTZITE | LIMESTONE | SANDSTONE | LIGHTWEIGHTSANDSTONE } | THICKNESS *thick* | FCYL28 $f_{cyl,28}$ | E28 E_{28} | FCYL0_28 $f_{cyl0,28}$ | FT28 $f_{t,28}$ | GF28 $G_{f,28}$ | ALPHA α | HUMIDITY *humidity* | DENSITY *density* | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* | { SHRINKAGE | COMPLIANCE} *measured_val* { HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }+ }+

Table 91: &CCModelEN1992 sub-command parameters

Parameter	Description
CEMENT_CLASS { 32.5N | 32.5R | 42.5N | 42.5R | 52.5N | 52.5R }	Type of cement, see e.g. http://www.cis.org.rs/en/cms/about-cement/standardization-of-cement : Strength classes of cement Cements are according to standard strength grouped into three classes, they being: • Class 32,5 • Class 42,5 • Class 52,5 Three classes of early strength are defined for each class of standard strength: • Class with ordinary early strength – N • Class with high early strength – R • Class with low early strength – L Class L can be applied only on CEM III cements. Default value: class_42_5N

AGGREAGETE { BASALTDENSELIMESTONE | QUARTZITE | LIMESTONE | SANDSTONE | LIGHTWEIGHTSANDSTONE }	Type of aggregate. Note that light weight concrete is detected for concrete with density below 2000kg/m^3 and some aditional meassures are taken for LIGHTWEIGHTSANDSTONE aggregate. Default value: QUARTZITE
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 $f_{cyl,28}$	Cylindrical material strength in compression $f_{cyl}(28\,days)$. This value is crucial for the creep model's prediction, i.e. prediction of material compliance $\Phi(t,t')$ and cylindrical compression strength $f_{cyl}(t)$, shrinkage etc. The ratio of $f_{cyl}(t)/f_{cyl}(28\,days)$ may be used for overiding short f_{cyl}, f_t, G_f. Note that material compliance/rigidity is overwritten always. Default value: 35100 [kPa].
FCYL0_28 $f_{cyl0,28}$	The parameter $f_{cyl0}(28\,days)$. If specified by a positive value, this value is used to calculate $f_{cyl0}(t)$ and overide the corresponding value in the base material. If it is specified as any negative value, $f_{cyl0}(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
GF28 $G_{f,28}$	The parameter fracture energy $G_f(28\,days)$. If specified by a positive value, this value is to calculate $G_f(t)$ and overide the corresponding value in the base material. . If it is specified as any negative value, $G_f(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]
FT28 $f_{t,28}$	The parameter tensile strength $f_t(28\,days)$. If specified by a positive value, this value is used to calculate $f_t(t)$ and overide the corresponding value in the base material. If it is specified as any negative value, $f_t(28\,days)$ is calculated by FIB_MC2010 based on $f_{cyl}(28\,days)$. Othewise, the value in the base material remains unchanged. Default value: 0 [MPa]

E28 E_{28}	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. It is used by the creep model to predict material compliance $\Phi(t,t')$. If unspecified, the model calculates its value based on *fcyl28*. Default value: calculated from *fcyl28*.
ALPHA α	Coefficient of thermal expansion to be used for calculation $\Delta\varepsilon_t(\Delta T)$ within the creep material. Default value: 0
HUMIDITY *humidity*	Ambient relative humidity (0.3…1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
{ SHRINKAGE | COMPLIANCE} *measured_val*	Measured shrinkage | material compliance *measured_val* for previously specified load and current time. Units of water loss must correspond to units of *total_water_loss* , shrinkage is dimension-less and compliance is input in kPa^{-1}.
{ HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }$_+$	For each entry of material history the data *time* , *temper* and *humid* must be input. If the data keywords are used, then it doesn't matter in which order the 3 data are input. Otherwise the indicated order is assumed. The units are days, degrees Celsius and dimension less humidity (in interval 0.3..1).

3.6.9.5 Sub-command &CCModelBP_KX_DATA

&CCModelBP_KX_DATA
CCModelBP_KX { CONCRETE *concrete_type* | THICKNESS *thick* | FCYL28 *fcyl28* | E28 *e28* | HUMIDITY *humidity* | DENSITY *density* | AC *ac* | WC *wc* | [SHAPE] FACTOR *sfactor* | {WATER | AIR | STEAM} [CURING] | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* | {SHRINKAGE | COMPLIANCE} *measured_val* { HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }$_+$ }$_+$

Table 92: & CCModelBP_KX sub-command parameters

Parameter	Description

CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
E28 *e28*	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. Default value: calculated from *fcyl28*.
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
AC *ac*	Total aggregate/cement ratio. Default value: 7.04
WC *wc*	Water/cement ratio. Default value: 0.63
[SHAPE] FACTOR *sfactor*	Cross section shape factor. It should be 1, 1.15, 1.25, 1.3, 1.55 for slab, cylinder, square prism, sphere, cube, respectively. Default value 1.25
{WATER | AIR | STEAM} [CURING]	Curing conditions, either under in water or air under normal temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
AS *as*	Total aggregate/find sand ratio. Default value 2.8
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
{ SHRINKAGE | COMPLIANCE} *measured_val*	Measured shrinkage | material compliance *measured_val* for previously specified load and current time. Units of water loss must correspond to units of *total_water_loss* , shrinkage is dimension-less and compliance is input in kPa^{-1}.
{ HISTORY [TIME]	For each entry of material history the data *time* , *temper* and

time [HUMIDITY] *humid* [TEMPERATURE] *temper* }+	*humid* must be input. If the data keywords are used, then it doesn't matter in which order the 3 data are input. Otherwise the indicated order is assumed. The units are days, degrees Celsius and dimension less humidity (in interval 0.3..1).

3.6.9.6 Sub-command &CCModelACI78_DATA

&CCModelACI78_DATA
CCModelACI78 { CONCRETE *concrete_type* | THICKNESS *thick* | FCYL28 *fcyl28* | HUMIDITY *humidity* | DENSITY *density* | AC *ac* | WC *wc* | AS *as* | SLUMP *slump* | AIR_CONTENT *air* | {WATER | AIR | STEAM} [CURING] | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* SHRINKAGE } *measured_val* }+

Table 93: & CCModelACI78 sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
DENSITY *density*	Concrete density [kg/m^3]. Default value: 2125. [kg/m^3].
AC *ac*	Total aggregate/cement ratio. Default value: 7.04
WC *wc*	Water/cement ratio. Default value: 0.63
AS *as*	Total aggregate/find sand ratio. Default value 2.8
SLUMP *slump*	Slump value [m]. Default value: 0.012m
AIR_CONTENT *air*	Air content [%]: Default value: 5%.

{WATER | AIR | STEAM} [CURING]	Curing conditions, either under in water or air under normal temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
SHRINKAGE *measured_val*	Measured shrinkage *measured_val* for previously specified load and current time. Unit of shrinkage is dimension-less.

3.6.9.7 Sub-command &CCModelCEB_FIP78_DATA

&CCModelCEB_FIP78_DATA
CCModelCEB_FIP78 { THICKNESS *thick* | FCYL28 *fcyl28* | E28 *e28* | HUMIDITY *humidity* | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* | SHRINKAGE *measured_val* }+

Table 94: & CCModelCEB_FIP78 sub-command parameters

Parameter	Description
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
E28 *e28*	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. Default value: calculated from *fcyl28*.
HUMIDITY *humidity*	Ambient relative humidity (0.3..1). Default value: 0.780
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
SHRINKAGE *measured_val*	Measured (at current humidity) shrinkage *measured_val* for previously specified load and current time. Unit of shrinkage is dimension-less.

3.6.9.8 Sub-command &CCModelCSN731202_DATA

&CCModelCSN731202_DATA
CCModelCSN731202 { CONCRETE *concrete_type*, THICKNESS *thick* | FCYL28 *fcyl28* | E28 *e28* | HUMIDITY *humidity* | [END] [OF] [CURING] TIME *endcuring* | {LOAD | CURRENT} [TIME] *time* | SHRINKAGE *measured_val* { HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }$_+$ }$_+$

Table 95: & CCModelCSN731202 sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
E28 *e28*	Short-term material Young modulus at 28 days, i.e. inverse compliance at 28.01 days loaded at 28 days [kPa]. Default value: calculated from *fcyl28*.
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{ HISTORY [TIME] *time* [HUMIDITY] *humid* [TEMPERATURE] *temper* }$_+$	For each entry of material history the data *time* , *temper* and *humid* must be input. If the data keywords are used, then it doesn't matter in which order the 3 data are input. Otherwise the indicated order is assumed. The units are days, degrees Celsius and dimension less humidity (in interval 0.3..1).
{LOAD	CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
SHRINKAGE *measured_val*	Measured shrinkage *measured_val* for previously specified load and current time. Unit of shrinkage is dimension-less.

3.6.9.9 Sub-command &CCModelBP1_DATA

&CCModelBP1_DATA
CCModelBP1 { CONCRETE *concrete_type* | THICKNESS *thick* | FCYL28 *fcyl28* |
HUMIDITY *humidity* | AC *ac* | WC *wc* | GS *gs* | SC *sc* | SA *sa* | CEMENT
[MASS] *cement_mass* | [SHAPE] FACTOR *sf* | {STEAM | WATER |AIR}
[CURING] | [END] [OF] [CURING] TIME *endcuring* | { LOAD | CURRENT }
[TIME] *time* SHRINKAGE *measured_val* }+

Table 96: & CCModelBP1 sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
AC *ac*	Total aggregate/cement ratio. Default value: 7.04
WC *wc*	Water/cement ratio. Default value: 0.63
GS *gs*	Coarse/fine aggregate ratio. Default value: 1.3
SC *sc*	Fine aggregate/cement ratio. Default value: 1.8
SA *sa*	Fine/total aggregate ratio. Default value: 0.4
CEMENT [MASS] *cement_mass*	Cement content. Default value: 320. kg/m^3
[SHAPE] FACTOR *sf*	Cross section shape factor. It should be 1, 1.15, 1.25, 1.3, 1.55 for slab, cylinder, square prism, sphere, cube, respectively. Default value 1.25
{STEAM | WATER	Curing conditions, either under in water or air under normal

|AIR} [CURING]	temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
SHRINKAGE *measured_val*	Measured (at current humidity) shrinkage *measured_val* for previously specified load and current time. Unit of shrinkage is dimension-less.

3.6.9.10 Sub-command &CCModelBP2_DATA

&CCModelBP2_DATA
CCModelBP2 { CONCRETE *concrete_type* | THICKNESS *thick* | FCYL28 *fcyl28* |
HUMIDITY *humidity* | AC *ac* | WC *wc* | GS *gs* | SC *sc* | SA *sa* | [SHAPE]
FACTOR *sf* | {STEAM | WATER |AIR} [CURING] | [END] [OF] [CURING]
TIME *time* | { LOAD | CURRENT } [TIME] xx SHRINKAGE *measured_val* }+

Table 97: & CCModelBP2 sub-command parameters

Parameter	Description
CONCRETE *concrete_type*	Type of concrete. Only type 1 and 3 are supported. Default value: 1
THICKNESS *thick*	Ratio volume [m^3] / surface area [m^2] of cross section. For long elements it is approximately cross sectional area [m^2] / perimeter [m]. Default value: 0.0767 [m].
FCYL28 *fcyl28*	Cylindrical material strength in compression [kPa]. Default value: 35100 [kPa].
HUMIDITY *humidity*	Ambient relative humidity (0.3...1). Default value: 0.780
AC *ac*	Total aggregate/cement ratio. Default value: 7.04
WC *wc*	Water/cement ratio. Default value: 0.63
GS *gs*	Coarse/fine aggregate ratio.

	Default value: 1.3
SC *sc*	Fine aggregate/cement ratio. Default value: 1.8
SA *sa*	Fine/total aggregate ratio. Default value: 0.4
[SHAPE] FACTOR *sf*	Cross section shape factor. It should be 1, 1.15, 1.25, 1.3, 1.55 for slab, cylinder, square prism, sphere, cube, respectively. Default value 1.25
{STEAM | WATER |AIR} [CURING]	Curing conditions, either under in water or air under normal temperature conditions (WATER | AIR) or steam condition (=STEAM). Default value: AIR
[END] [OF] [CURING] TIME *endcuring*	Time at beginning of drying, i.e. end of curing. [days]. Default value: 7 [days].
{LOAD | CURRENT} [TIME] *time*	Current or load time for the subsequent measured value. Default: 0 [days]
SHRINKAGE *measured_val*	Measured (at current humidity) shrinkage *measured_val* for previously specified load and current time. Unit of shrinkage is dimension-less.

3.6.9.11 Sub-command &CCModelGeneral_Data

&CCModelGeneral_Data
CCModelGeneral { T' *t'* | T *t* | FI *fi* | EPS *eps* | FCYL *fcyl* }+

Table 98: & CCModelGeneral sub-command parameters

Parameter	Description
T' *t'*	Set effective loading time *t'* for following data. Default value: none Units: t.
T *t*	Set effective observation time *t* for following data, i.e. a time, when an input value is measured. Default value: none Units: t.
FI *fi*	Value of material compliance *fi(t.t')* for times *t,t'* . Default value: none. Units: 1/S

EPS *eps*	Material shrinkage *eps(t)* at time of observation *t*. Default value: none Units: none
FCYL *fcyl*	Current cylindrical strength in compression *fcyl(t')* pertinent for loading time *t'* . Note that the value is input as positive value! Default value: none Units: S

3.6.10 Material Type for Combined Material

3.6.10.1 Sub-command &COMBINED_MATERIAL

Syntax:
&COMBINED_MATERIAL:
TYPE "CCCombinedMaterial"
COMPONENT *id1* [RATIO *x1*]
COMPONENT *id2* [RATIO *x2*]
….
COMPONENT *id3* [RATIO *x3*]

Table 99: &COMBINED_MATERIAL sub-command parameters

Parameter	Description
Basic properties	
COMPONENT *id*	Id of the previously defined material, which is to be used a one component of the combined/composite material. Units: none Acceptable range: (1; maximal integer> Default value: none
RATIO *x*	Relative contribution of this material to the overall behavior of the combined composite material. Units: none Acceptable range: <0; maximal real number> Default value: 1.0

3.6.11 Material Type for Material with Variable Properties

3.6.11.1 Sub-command &VARIABLE_MATERIAL

Syntax:
&VARIABLE_MATERIAL:
TYPE "CCMaterialWithVariableProperties" BASE *id*

PARAM "*name1*" [VALUE *x*] [F_T *id1*] [F_TEMP *id1*]
PARAM "*name2*" F *id2*
….
PARAM "*name3*" F *id3*

Table 100: &VARIABLE_MATERIAL sub-command parameters

Parameter	Description
Basic properties	
BASE *id*	Id of the previously defined base material, whose parameters will be modified based on the provided functions. Only the following base materials should be used as a base one: CC3DnonLinCementitious2, CC1DElastIstotropic, CCPlaneStressElastIsotropic, CCPlaneStrainElastIsotropic, CC3DelastIsotropic, CCASymElastIsotropic, CC3DDruckerPragerPlasticity, CC3DBiLinearSteelVonMises, CCReinforcement, CCSmearedReinf Units: none Acceptable range: (1; maximal integer> Default value: none
PARAM | PARAMETER "*name*"	Parameter name from the base material whose values will change based on the provided function. The original value of this parameter in the base material is overwritten by the values in the function. **The base material should not be used in any other combined material as well as a stand alone material. Otherwise results are unpredictable**. Units: none Acceptable range: any string Default value: none

VALUE *x*	Constant value of the parameter. By default *x*=1. The parameter is calculated by multiplying *x* by the function below.
F_T | FUNCTION_T *id*	Id of the previously defined function to account for effect of time. If *id*==0, the function is ignored . The parameter is calculated by multiplying the above *x* by the functions f_t(*time*) and f_temp(*temperature*). Units: none Acceptable range: (0; maximal integer> Default value: 0
F_TEMP | FUNCTION_TEMP *id*	Id of the previously defined function to account for effect of temperature. If *id*==0, the function is ignored . The parameter is calculated by multiplying the above *x* by this function f_t(*time*) and f_temp(*temperature*).. Units: none Acceptable range: (0; maximal integer> Default value: 0

3.6.12 Material Type for Material with Temperature Dependent Properties

3.6.12.1 Sub-command &MATERIAL_WITH_TEMP_DEP_PROPERTIES

This model is to be used to simulate change of material properties due to current temperature. The temperature fields can be imported from a previously performed thermal analysis.

Syntax:

```
&MATERIAL_WITH_TEMP_DEP_PROPERTIES:
TYPE "CCMaterialWithTempDepProperties"  BASE id
     PARAM  "name1"    F id1
     PARAM  "name2"    F id2
     ….
     PARAM  "name3"    F id3
     { EPS_T_F id4 | TOTAL n }
```

Table 101: &MATERIAL_WITH_TEMP_DEP_PROPERTIES sub-command parameters

Parameter	Description
Basic properties	
BASE *id*	Id of the previously defined base material, whose parameters will be modified based on the thermal loading and the provided function. Only the following materials should be used as a base material: CC3DNonLinCementitious2,

	CC1DElastIstotropic, CCPlaneStressElastIsotropic, CCPlaneStrainElastIsotropic, CC3DelastIsotropic, CCASymElastIsotropic, CC3DDruckerPragerPlasticity, CC3DBiLinearSteelVonMises, CCReinforcement, CCSmearedReinf Units: none Acceptable range: (1; maximal integer> Default value: none
PARAM | PARAMETER *"name"*	Parameter name from the base material whose values will change based on the thermal loading and provided function. The original value of this parameter in the base material is overwritten by the values in the function. **The base material should not be used in any other combined material as well as a stand alone material. Otherwise results are unpredictable**. Units: none Acceptable range: any string Default value: none

F | FUNCTION *id*	Id of the previously defined function that defines the dependence of the given material parameter on thermal loading. At each material point this function will define the value of the given material parameter based on the current thermal loading at this material point, i.e. integration point. Units: none Acceptable range: (1; maximal integer> Default value: none
EPS_T_F *id*	Id of the previously defined function that defines the evolution of thermal strains. It should be a function of initial strains based on the total temperature at a given point. When this function is defined the alpha parameter for the thermal expansion coefficient in the base material should be set to zero otherwise the thermal expansion is considered two times. Units: none Acceptable range: (1; maximal integer> Default value: none
TOTAL *n*	Activates the total formulation, i.e. the stress at each step will be calculated from zero by incremental application of the existing strain tensor. The parameter *n* defines the number of steps to reach the current strain valus. When this parameter is activated the material model does not consider the loading history, but it is necessary to accurately consider the changes of the elastic modulus in the incremental material formulation. Units: none Acceptable range: (1; maximal integer> Default value: 0

3.6.13 Material Type for Material with Properties Varying in Space

3.6.13.1 Sub-command &MATERIAL_WITH_RANDOM_FIELDS

This model is to be used to simulate a spatial distribution of material properties. For instance this model can be used to simulate a random distribution of material parameters over the structure.

Syntax:

```
&MATERIAL_WITH_RANDOM_FIELDS:
TYPE “CCMaterialWithRandomFields” BASE id
    FILENAME “name1”
```

Table 102: &MATERIAL_WITH_RANDOM_FIELDS sub-command parameters

Parameter	Description
Basic properties	
BASE *id*	Id of the previously defined base material, whose parameters will be modified based on the thermal loading and the provided function. Only the following materials should be used as a base material: CC3DNonLinCementitious2, CC1DElastIstotropic, CCPlaneStressElastIsotropic, CCPlaneStrainElastIsotropic, CC3DelastIsotropic, CCASymElastIsotropic, CC3DDruckerPragerPlasticity, CC3DBiLinearSteelVonMises, CCReinforcement, CCSmearedReinf Units: none Acceptable range: (1; maximal integer> Default value: none
FILENAME *"name"*	File name containing the spatial distribution of material parameters. Units: none Acceptable range: any string Default value: none

3.6.14 Material Types for Simplified Nonlinear Analysis Using CCBeam Element

3.6.14.1 Sub-command &BEAM_MASONRY_MATERIAL

This model can be used for nonlinear analysis of (reinforced) masonry structures modeled by CCBeam elements. It is used for solid part, i.e. masonry. An eventual reinforcements should be modeled by CCBeamReinfBarMaterial. The material conforms with recommendations given by Eurocode and similar codes for practice. The input "design" strengths overwrite values based on input of "characteristic" strengths.

Syntax:

& BEAM_MASONRY_MATERIAL :
TYPE "CCBeamMasonryMaterial" {[E *x*] | [MU *x*] | [RHO *x*] | [ALPHA *x*] | [F_K *x*] | [F_VK0 *x*] |[COEFF_F_VK *x*] | [F_VLT *x*] | [F_VLT_CONST *x*] | [F_VLT_COEFF *x*] | [F_XK_INPLANE *x*] | [{F_XK_OUTPLANE}|{F_XK} *x*] | [R_RATIO *x*] | [GAMMA_M] | [F_D *x*] | [F_VD *x*] | [F_XD_INPLANE *x*] | [{F_XD_OUTPLANE}|{F_XD} *x*] |[EPS_MU *x*] | [EPS_M *x*] | [LAMBDA *x*]

| [ETA *x*] | [REL_TOL *x*] | [ITER_MAX *n*] | [EPS_SMALL *x*] | [ALPHA_STEP *x*] | [ALPHA_TOL *x*] | [FLEX_DRIFT_COEFF *x*] | [SHEAR_DRIFT_COEFF *x*] | [STIRRUPS_SPACING *x*] | [STIRRUPS_AREA *x*] | [STIRRUPS_MATERIAL *n*] | DAMPING_MASS x_M DAMPING_STIFF x_K}

Table 103: & BEAM_MASONRY_MATERIAL sub-command parameters

Parameter	Description
E *x*	Young modulus. Units: stresses Default value: 0
MU *x*	Poisson ratio Units: none Default value: 0
RHO *x*	Material density Units: mass/volume Default value: 0
ALPHA *x*	Coefficient of thermal expansion Units: 1/T cceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
F_K *x*	Characteristic material compressive strength, (negative). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
F_VK0 *x*	Characteristic material initial shear strength, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
COEFF_F_VK *x*	Coefficient for normall stress to calculate F_VK. Units: none Default value: 0.4
F_VLT *x* F_VLT_CONST *x*	Characteristic material limit shear strength – constant part, (positive). Final value is calculated as $f_{vlt} = f_{vlt,const} + f_{vlt,coeff}\sigma_d$,

	where σ_d is element compression stress. This input is not used, if the corresponding design value is given. Units: none Default value: 0
F_VLT_COEFF *x*	
F_XK_INPLANE *x*	Characteristic material in-plane tensile strength in bending, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
{F_XK_OUTPLANE} | {F_XK} *x*	Characteristic material out-of-plane tensile strength in bending, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
R_RATIO *x*	Ratio of mortar thickness to the wall thickness Units: none Default value: 1
GAMMA_M *x*	Partial factor of safety Units: none Default value: 1
F_D *x*	Design material compressive strength, (negative) Units: stresses Default value: 0
F_VD *x*	Design material shear strength, (positive) Units: stresses Default value: 0
F_XD_INPLANE *x*	Design material in-plane tensile strength in bending, (positive) Units: stresses Default value: 0
{F_XD_OUTPLANE}|{ F_XD} *x*	Design material out-of-plane tensile strength in bending, (positive) Units: stresses Default value: 0
EPS_MU *x*	Maximum compressive strain at the corners of cross section, (negative)

	Units: none Default value: -0.0035
EPS_M *x*	Maximum compressive strain at the centre of cross section, (negative) Units: none Default value: -0.002
LAMBDA *x*	Coefficient to reduce compressed masonry area Units: none Default value: 1.
ETA *x*	Coefficient to apply for F_D Units: none Default value: 0.8
REL_TOL *x*	Relative acceptable error in moments/forces Units: none Default value: 0.001
ITER_MAX *n*	Maximum number of iterations for zeroizing of lateral bending moment. Note that the moments are calculated in a coordinate system, whose Y‘ axis is parallel to the resultant moment from M_y and M_z load. Therefore, moment along Z‘ must be equal zero. Units: none Default value:30
EPS_SMALL *x*	Strain value already assumed neglibable Units: none Default value: 0.001
ALPHA_STEP *x*	Angle step (for resultant moment load) at which the M-N diagram of cross section is cached. For zero or negative value nthing is cached and the appropriate M-N diragram is calculated on run-time basis. Units: none Default value: $\frac{\pi}{60}$
ALPHA_TOL *x*	Angle difference (for resultatnt moment load) thas is assumed negligible. Units: none Default value: $\frac{\pi}{360}$

FLEX_DRIFT_COEFF *x*	Coefficinet to check maximum flexural drift. By default *x*=0.008. If the criterion violated, corresponding beam's moments are reduced to zero.
SHEAR_DRIFT_COEFF *x*	Coefficinet to check maximum shear drift. By default *x*=0.004. If the criterion violated, corresponding beam's shear forces are reduced to zero.
STIRRUPS_SPACING *x*	Stirrups spacing. Units: length Default value: 0.0
STIRRUPS_AREA *x*	Area of reinforcement stirrups, (typically 2 x stirrup area). Units: length^2 Default value: 0.0
STIRRUPS_MATERIAL *n*	Id of material, froim which the tirrups are made. Units: none Default value: NONE

3.6.14.2 Sub-command &BEAM_RC_MATERIAL

This model can be used for nonlinear analysis of (reinforced) concrete structures modeled by CCBeam elements. It is used for solid part, i.e. concrete. An eventual reinforcements should be modeled by CCBeamReinfBarMaterial. The material conforms with recommendation given by Eurocode and similar codes for practice. The input "design" strengths overwrite values based on input of "characteristic" strengths.

Syntax:

& BEAM_RC_MATERIAL :
TYPE "CCBeamRCMaterial" {[E *x*] | [MU *x*] | [RHO *x*] | [ALPHA *x*] | [F_CK *x*] | [F_CVK *x*] | [F_CTK_INPLANE *x*] | [{F_CTK_OUTPLANE}|{F_CTK} *x*] | [GAMMA_M *x*] | [F_CD *x*] | [F_CVD *x*] | [F_CTD_INPLANE *x*] | [{F_CTD_OUTPLANE}|{F_CTD} *x*] |[EPS_CU *x*]|{EPS_C *x*}|{ LAMBDA *x*] | [ETA *x*] | [REL_TOL *x*] | [ITER_MAX *n*] | [EPS_SMALL *x*] | [ALPHA_STEP *x*] | [ALPHA_TOL *x*] | [FLEX_DRIFT_COEFF *x*] | [SHEAR_DRIFT_COEFF *x*] | [STIRRUPS_SPACING *x*] | [STIRRUPS_AREA *x*] | [STIRRUPS_MATERIAL *n*] | [STIRRUPS_K_I *x*] | [STIRRUPS_NI_1 *x*] | [STIRRUPS_EFFECTIVE_DEPTH *x*] | [STIRRUPS_C_RD_C *x*] | [STIRRUPS_NI_MIN *x*] | DAMPING_MASS x_M DAMPING_STIFF x_K}

Table 104: &BEAM_RC_MATERIAL sub-command parameters

Parameter	Description
E *x*	Young modulus. Units: stresses Default value: 0
MU *x*	Poisson ratio Units: none Default value: 0
RHO *x*	Material density Units: mass/volume Default value: 0
ALPHA *x*	Coefficient of thermal expansion Units: 1/T cceptable range: <0; maximal real number> Default value: 0.000012
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
F_CK *x*	Characteristic material compressive strength, (negative). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
F_CVK *x*	Characteristic material shear strength, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
F_CTK_INPLANE *x*	Characteristic material in-plane tensile strength in bending, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
[{F_CTK_OUTPLANE}|{F_CTK}*x*	Characteristic material out-of-plane tensile strength in bending, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0

GAMMA_M *x*	Partial factor of safety Units: none Default value: 1
F_CD *x*	Design material compressive strength, (negative) Units: stresses Default value: 0
F_CVD *x*	Design material shear strength, (positive) Units: stresses Default value: 0
F_CTD_INPLANE *x*	Design material in-plane tensile strength in bending, (positive) Units: stresses Default value: 0
[{F_CTD_OUTPLANE}|{F_CTD} *x*	Design material out-off-plane tensile strength in bending, (positive) Units: stresses Default value: 0
EPS_CU *x*	Maximum compressive strain at the corners of cross section, (negative) Units: none Default value: -0.0035
EPS_C *x*	Maximum compressive strain at the centre of cross section, (negative) Units: none Default value: -0.002
LAMBDA *x*	Coefficient to reduce compressed masonry area Units: none Default value: 1.
ETA *x*	Coefficient to apply for F_D Units: none Default value: 0.8
REL_TOL *x*	Relative acceptable error in moments/forces Units: none Default value: 0.001
ITER_MAX *n*	Maximum number of iterations for zeroizing of lateral bending moment. Note that the moments are calculated in a coordinate system, whose Y‘ axis is parallel to the resultant moment from

	M_y and M_z load. Therefore, moment along Z‘ must be equal zero. Units: none Default value: 20
EPS_SMALL *x*	Strain value already assumed neglibable Units: none Default value: 0.001
ALPHA_STEP *x*	Angle step (for resultant moment load) at which the M-N diagram of cross section is cached. For zero or negative value nthing is cached and the appropriate M-N diragram is calculated on run-time basis. Units: none Default value: $\frac{\pi}{60}$
ALPHA_TOL *x*	Angle difference (for resultatnt moment load) thas is assumed negligible. Units: none Default value: $\frac{\pi}{360}$
FLEX_DRIFT_COEFF *x*	Coefficinet to check maximum flexural drift. If the criterion violated, corresponding beam’s moments are reduced to zero. Units: none Default value: 0.008.
SHEAR_DRIFT_COEFF *x*	Coefficinet to check maximum shear drift. If the criterion violated, corresponding beam’s shear forces are reduced to zero. Units: none Default value: 0.004
STIRRUPS_SPACING *x*	Stirrups spacing. Units: length Default value: 0.0
STIRRUPS_AREA *x*	Area of reinforcement stirrups, (typically 2 x stirrup area). Units: length^2 Default value: 0.0
STIRRUPS_MATERIAL *n*	Id of material, froim which the tirrups are made. Units: none Default value: NONE
STIRRUPS_K_I *x*	Coefficient k_1. Typically no change is needed.

	Units: none Default value: 0.15
STIRRUPS_NI_1 *x*	Coefficient of compressive strut strength. Typically no change is needed. Units: none Default value: based on f_{ck}.
STIRRUPS_EFFECTIVE_DEPTH *x*	Effective depth of the section, typically distance between the centre of the longitudinal reinforcement and the top edge. Typically no change is needed. Units: length Default value: calculated automatically.
STIRRUPS_C_RD_C *x*	Coefficient based on National annex. Typically no change is needed. Units: none Default value: $0.18/\gamma_c$.
STIRRUPS_NI_MIN *x*	Minimal shear strength. Typically no change is needed. Default value $v_{\min} = 0.035\, k^{3/2} f_{ck}^{1/2}$

3.6.14.3 Sub-command &BEAM_REINF_BAR_MATERIAL

This model can be used for nonlinear analysis of (reinforced) concrete structures modeled by CCBeam elements. It is used for reinforcement part, i.e. steel. The solid part shoud be modeled by either CCBeamMasonryMaterial or CCBeamRCMaterial. The material conforms with recommendation given by Eurocode and similar codes for practice.

Syntax:

& BEAM_REINF_BAR_MATERIAL :
TYPE “CCReinfBarMaterial” {[E *x*] | [MU *x*] | [RHO *x*] | [ALPHA *x*] | [F_YK *x*] | [F_YVK *x*] | [GAMMA_M *x*] | [F_YD *x*] | [F_YVD *x*] | [E_YD_HARD *x*] | [EPS_YD_MAX *x*] DAMPING_MASS x_M DAMPING_STIFF x_K}

Table 105: & BEAM_REINF_BAR_MATERIAL sub-command parameters

Parameter	Description
E *x*	Young modulus. Units: stresses Default value: 0
MU *x*	Poisson ratio Units: none Default value: 0
DAMPING_MASS x_M DAMPING_STIFF x_K	Mass and stiffness damping factors specified for indiviual element group. They overwrite the same factor set for the whole structure by SET command .
RHO *x*	Material density Units: mass/volume Default value: 0
ALPHA_TOL *x*	Angle difference (for resultatnt moment load) thas is assumed negligible. Units: none Default value: $\frac{\pi}{360}$
F_YK *x*	Characteristic material compressive strength, (negative). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
F_YVK *x*	Characteristic material shear strength, (positive). This input is not used, if the corresponding design value is given. Units: stresses Default value: 0
GAMMA_M *x*	Partial factor of safety Units: none Default value: 1
F_YD *x*	Material strength, (positive) Units: stresses Default value: 0
F_YVD *x*	Material shear strength, (positive) Units: stresses Default value: 0

E_YD_HARD *x*	Hardening young modul Units: stresses Default value: 0
EPS_YD_MAX *x*	Max reinforcement tensile strain Units: none Default value: 0.01

3.7 Load and Boundary Conditions Definition

This command defines loads applied in a load case. The following main load types are supported:

Table 106: Load and boundary conditions definition types

Sub-Command	Description
&LOAD_DISPLACEMENT	Prescribed nodal displacement (i.e. Dirichlet boundary condition), either &SIMPLE_LOAD_DISPLACEMENT, or &COMPLEX_LOAD_DISPLACEMENT
&LOAD_FORCES	Prescribed nodal forces (i.e. Neumann boundary condition), either &SIMPLE_LOAD_FORCE or &COMPLEX_LOAD_FORCE
&LOAD_MASTER_SLAVE_NODES	Master slave node pairs – prescribed displacement as a linear combination of other displacements and constant value, (i.e. Cauchy boundary condition).
&ELEMENT_LOAD	Element loads, either &BODY_ELEMENT_LOAD or &ELEMENT_BOUNDARY_LOAD or &TEMPERATURE_ELEMENT_LOAD or &ELEMENT_INITIAL_STRAIN_LOAD or &ELEMENT_INITIAL_STRESS_LOAD or &LOAD_FUNCTION or &MASS_ACCELERATIONS or &ELEMENT_INITIAL_GAP_LOAD or &CHLORIDES or &CARBONATION or &ASR
&LOAD_FUNCTION	Time function id, i.e. id of time (or step id) function defining coefficient for the applied load. See &FUNCTION for the function definition.
&SPRING_DEFINITION	Spring support boundary condition.
&RIGID_BODY, &INVERSE_RIGID_BODY	Definition of rigid body and/or inverse rigid body constrains
&LOAD_SHELL_TO_SOLID	A special boundary condition similar to &LOAD_MASTER_SLAVE_NODES that is dsigned to connect nodes of shell 2D elements to solid elements.

3.7.1 The Command &LOAD

Syntax:

```
&LOAD:
LOAD CASE  { ID n | [NAME "load case name"] | &LOAD_DISPLACEMENT |
        &LOAD_FORCES | &LOAD_MASTER_SLAVE_NODES | &RIGID_BODY
        | &INVERSE_RIGID_BODY | &BEAM_NL_CONNECTION |
        &LOAD_SHELL_2D_TO_SOLID | &LOAD_BEAM_1D_TO_SOLID |
        &ELEMENT_LOAD }+
```

Table 107: General notes on LOAD command

The following are general notes on input of boundary conditions: • Load case ids > 900000 are reserved for internal use; thus input id <=900000. • Specified boundary condition of any type has cumulative character, i.e. if a loading force in a specified degree of freedom is input three times, the actual loading force is tripled. • The specified boundary conditions are incremental, i.e. they set change in a particular loading step, (execution time) with respect to the previous step, (previous time).

3.7.1.1 The Sub-command &LOAD_DISPLACEMENT

```
&LOAD_DISPLACEMENT:
SUPPORT [&DISPLACEMENT_TYPE] &LOAD_FUNCTION ]
        {&COMPLEX_LOAD_DISPLACEMENT |
        &SIMPLE_LOAD_DISPLACEMENT | &SPRING_DEFINITION}+

&DISPLACEMENT_TYPE:
TYPE {DISPLACEMENT | VELOCITY | ACCELERATION}
```

Note that displacements boundary conditions, (i.e type = "DISPLACEMENT"), are treated as incremental displacements load, whilst in case of velocities and/or accelerations, (i.e. type = "VELOCITY" or "ACCELERATION"), the input values are considered to be total load, not incremental load. Hence, "VELOCITY" and/or "ACCELERATION" BCs (because of its "total" character) must be specified in the group of "fixed" load within the dynamic load step definition. On the other hand, "DISPLACEMENT" type BCs are typically input within "increment" loads of the stepd definition.

3.7.1.2 The Sub-command &COMPLEX_LOAD_DISPLACEMENT

```
&COMPLEX_LOAD_DISPLACEMENT:
{COMPLEX  {&MASTER_NODES | &SLAVE_NODES | &LOAD_VALUE |
        RELAX}+ }+ [PROCESS_FLAG  {REFERENCE_COORDS |
        USE_CURRENT_COORDS | COPY_DEFORMATION |
        COPY_DEFORMATION_ONCE  | COPY_NO_DEFORMATION} ]
```

Table 108: COMPLEX_LOAD_DISPLACEMENT description

This type of Dirichlet boundary condition sets the following general boundary condition:

$$u_i = x + \sum_{j=1}^{N} u_j f_j \quad \text{, where} \quad i \neq j$$

In the above equation u_i represents all slave degrees of freedom (defined in &SLAVE_NODES), x is the prescribed value (defined in &LOAD_VALUE), u_j are the master degrees of freedom and f_j are multipliers for the master degrees of freedom (defined in &MASTER_NODES). The index i at the slave degree of freedom u denotes the possibility to enforce the above boundary condition for several slave nodes and their degrees of freedom.

The boundary condition has two forms: basic and relaxed. The relaxed form differs from the basic one in the way that during iteration process it transfers out-of-balance forces directly to reactions. This strategy is needed, if the specified boundary condition needs to be applied in form of extra Lagrangian multiplier, which in turn means that it may need an external force to realize the prescribed constrain.

In other words, use the relaxed form of the boundary condition for cases, when the structure is already stable before applying a new boundary condition and the new condition is used only to deviate the structure from those stable conditions to slightly different conditions. Use the basic form for cases, when you want connect some macroelements, when no master nodes are specified etc.

The PROCESS_FLAG input specifies a special generation of master-slave boundary conditions. These constraints can be generated using either current or reference coordinate system. The first or second method is invoked by inputing the keyword USE_CURRENT_COORDS or REFERENCE_COORDS, respectively.

Modeling construction processes typically generates the following problem: we need to connect previously erected (and loaded) parts of a structure with a part of the structure that is new in the construction step. The trouble is that the older part is already deformed and the deformed geometry on the border between the two parts is difficult to figure in the new part. Hence, ATENA offers to model the new part with undeformed shape and then to copy the border displacements (from the old part to the new part). It is achived by use of the option COPY_DEFORMATION, or alternatively COPY_DEFORMATION_ONCE . While the former option ensures copying of border displacements in every step, in which this load is employed, the latter keyword causes the displacements to be copied only once, i.e. in the next step and thereafter the option of COPY_NO_DEFORMATION is used.

3.7.1.3 The Sub-command &SIMPLE_LOAD_DISPLACEMENT

```
&SIMPLE_LOAD_DISPLACEMENT:
{SIMPLE { SAVE_SUPPORT |REMOVE_SAVED_SUPPORT } { &LOAD_PLACE |
      &LOAD_VALUE }+ }+
```

Table 109: SIMPLE_LOAD_DISPLACEMENT description

This type of Dirichlet boundary condition sets the following general boundary condition:

$$u = value$$

It is the simplest way to define prescribed deformation at a specified node and degree of freedom (defined in &LOAD_PLACE).

Location of the boundary condition is specified by id of supported node and its supported degree of freedom. Alternatively, the boundary condition can be set for all nodes (and the specified supported degree of freedom), whose ids are stored in a list of ids, see command &SELECTION . In this case, the BC's value is calculated as follow:

$u = const + x\,coeff\_x + y\,coeff\_y + z\,coeff\_z$, see &LOAD_VALUE command fragment.

In the above *x,y,z* are coordinates of node *id* from the list. This way it is possible to prescribe variable load that depends of coordinates of a node, to which it is applied. Typical example of such a load may by lateral (hydrostatic) pressure applied to a vertical wall of a water tank.

If *idof* >0, then the specified displacement applied at DOF *idof* . Otherwise current displacement at -*idof* is applied.

The optional flags SAVE_SUPPORT |REMOVE_SAVED_SUPPORT are used to save or remove the current load for later use. It is particularly useful for the case of *idof* >0, (e.g. for modelling of a tunnel excavation.

3.7.1.4 The Sub-commands &LOAD_FORCE, &COMPLEX_LOAD_FORCE and &SIMPLE_LOAD_FORCE

```
&LOAD_FORCE:
LOAD  TYPE {CONCENTRATED_LOAD | LUMPED_MASS }
      [&LOAD_FUNCTION]  { &COMPLEX_LOAD_FORCE  |
      &SIMPLE_LOAD_FORCE  }+

&COMPLEX_LOAD_FORCE:
{COMPLEX  { &SLAVE_NODES | &LOAD_VALUE }+ }+

&SIMPLE_LOAD_FORCE:
{SIMPLE { &LOAD_PLACE | &LOAD_VALUE }+ }+
```

Table 110: SIMPLE_LOAD_FORCE and COMPLEX_LOAD_FORCE description

Both these commands are similar to the above SIMPLE_LOAD_DISPLACEMENT and COMPLEX_LOAD_DISPLACEMENT. They specify an applied force (or mass) at a node, (instead of displacement at a node).

If *idof* >0, then the specified force is applied at DOF *idof* . Otherwise current partial internal force at -*idof* is applied.

The optional flags SAVE_SUPPORT |REMOVE_SAVED_SUPPORT are used to save or remove the current load for later use. It is particularly useful for the case of *idof* >0, (e.g. for modelling of a tunnel excavation.

3.7.1.5 The Sub-command for &MASTER_SLAVE_NODES

&LOAD_MASTER_SLAVE_NODES:
{ {MASTER | SOLID_TO_SOLID} { &MS_PAIRS | &MS_GROUPS | &MS_SELECTION } [&MS_PROCESS_FLAGS] }+

&MS_PAIRS:
[SLAVE] [NODAL] PAIRS [ACCEPT_OUTSIDE_ELEMENT] [DISTANCE *dx*] { n_i [{REPLACE | REPLACES}] i_i }+

&MS_GROUPS:
[SLAVE] [NODAL] GROUPS [ACCEPT_OUTSIDE_ELEMENT] [DISTANCE *dx*] [SHAPE *shape*] { { n_i }+ { REPLACE | REPLACES } i_i }+

&MS_SELECTION:
{ { SELECTIONS | LISTS } *list_of_masters list_of_slaves* [DISTANCE *dx*] } SEARCH_RADIUS *r* | { LIST_MASTER_NODES *list_of_masters* LIST_SLAVE_NODES *list_of_slaves* [DISTANCE *dx*] } | { LIST_SLAVE_NODES *list_of_slaves* LIST_MASTER_NODES *list_of_masters* [DISTANCE *dx*] }

&MS_PROCESS_FLAGS:
[PROCESS_FLAG {REFERENCE_COORDS | USE_CURRENT_COORDS} | {COPY_DEFORMATION | COPY_DEFORMATION_ONCE | COPY_NO_DEFORMATION}] | [SKIP_DOFS_MASK *skip_mask*] | [ALLOW_LDOFS_MASK *allow_ldofs_mask*] }+

Table 111: LOAD_MASTER_SLAVE_NODES description

The LOAD_MASTER_SLAVE_NODES command structure is a special case of &COMPLEX_LOAD_DISPLACEMENT, when all nodal degrees of freedom of the slave node have to equal to its corresponding master degrees of freedom. This is the case of the above command with “PAIRS” keyword, i.e. the 1st line of the command.

The command also can set that all slave degrees of freedom are to be replaced by linear combination of the appropriate degrees of freedom of several master nodes. In this case the “GROUPS” keyword used. For 2D case, master nodes must form line (i.e. 2 master nodes), triangle (i.e. 3 master nodes) or quadrilateral element (i.e. 4 master nodes). For 3D case, the master nodes must form line (i.e. 2 master nodes), tetrahedron (4 master nodes), triangle wedge (i.e. 6 master nodes) or cube element (i.e. 8 master nodes). The master nodes must be input in exactly the same order as used to describe element incidences for an element of the equal type.

If nonlinear elements are used, then SHAPE] *shape* input must specified. It describes shape of the embedded/adjacent elements. It is 1/2/3/4/5/6 for element of shape 3-nodes truss/ 6-nodes triangle/ 6, 8 or 9 nodes quadrilateral/ 16 or 18 or 20 nodes brick/ 10 nodes tetrahedron / 15 nodes wedge, respectively.

By default, the &MS_GROUPS and &MS_PAIRS boundary conditions are only accepted, if the slave nodes are located inside an element defined by the master nodes or closed to the master_node, respectively. The required accuracy is defined by the parameter DISTANCE. This behavior can be changed by using the flag ACCEPT_OUTSIDE_ELEMENT. If it is defined, the boundary conditions are always accepted. Note that specifying ACCEPT_OUTSIDE_ELEMENT causes skipping some topological checks of the input data that are aimed to trap an errorness user input. Hence, it should be used with the highest care. The ACCEPT_OUTSIDE_ELEMENT flag does not affect the &MS_SELECTION boundary conditions.

By default the "PAIRS" command alternative is assumed. The command allows definition one or more of such coupled pairs or groups.

Alternatively, master slaves pairs can be picked up from list of masters and list of slaves automatically. Such a pair is created, if master versus slave node coordinates from the respective lists are closer than absolute distance abs(*dx*).

If the above fails and value *dx* is negative, Atena tries to fix the slave node by the closest (master) element. It succeeds, if $abs(x_i - x_i^{approx}) \le \max(abs(dx), dx_{negligigle})$ for $i = 1,2,(3)$, where $\overline{x}$ and $\overline{x}^{approx}$ is respectively vector of coordinates of the slave node and a node resulting from its master element's approximation. If it does not succeed, the 2nd closest element is used etc. up to success or total failure. The isoparametric coordinates $\overline{r}^{approx}$ (that correspond to $\overline{x}^{approx}$) are typically required to be in range $\langle -1..1 \rangle$. This clampes x_i^{approx} and thus the previous inequation also tests, whether x_i^{approx} is in or outside of the master element. The parameter $dx_{negligigle}$ is set as the model's (absolute) NEGLIGIBLE_SIZE.

The PROCESS_FLAG input can be used to specify a special way of master-slave boundary conditions generation. These constrains can be generated using either current or reference coordinate system. Another option is to copy during the generation displacements from master points to slave points. It is useful in modeling of construction process. For a complete description of the PROCESS_FLAG options, see Table 108. *skip_mask* allows for definition of DOFs that are skipped, i.e. not connected. If *skip_mask* is not defined, all nodal DOFs are linked.

The SKIP_DOFS_MASK *skip_mask* is used to code, which nodal dofs should be skipped, i.e. which dofs should not be affected by the current master-slave condition. Displacement *x,y,* .. rotation *z* corresponds to 0b1, 0b10 ...0b100000. For example, let us want to constrain only displacements *x,y* and rotation *y* of nodes with 6 dofs, (3 displacements and three rotations). Using binary biwise notation, we need to constrain dofs 0b010011. The *skip_mask* is complement of 0b010011, i.e. 0b101100. Hence you must input *skip_mask* as integer number 44. (0b101100=0x2C=44).

The ALLOW_LDOFS_MASK is coded similar to the SKIP_DOFS_MASK. It defines, what master-slave degrees of freedom can be fixed, in spite they are local. By default only global dofs can be constrained.

SEARCH_RADIUS *r* set radius, where to look for surrounding elements to fix the master

slave node.

&LOAD_SHELL_2D_TO_SOLID:
{ SHELL_2D_TO_SOLID LIST_SHELL_NODES " *list_of_slaves* " |
LIST_SOLID_NODES " *list_of_masters* " | MAX_DISTANCE *x* |
SEARCH_RADIUS *r* | { FIX_ROTATION_BM | FIX_ROTATION_MT |
FIX_ROTATION_BT } | [SHELL_GROUP *shell_group_id*] | [EMBEDDED
SOLIDS_GROUPS *solid_group_ids_interval*] ROTATION_ARMS *n* }+

Table 112: &LOAD_SHELL_2D_TO_SOLID description

The LOAD_SHELL_2D_TO_SOLID command structure is similar to the command LOAD_MASTER_SLAVE_NODES described above and most its parameters are the same. The main difference is, however, that it connects 5 dofs of each slave node, (i.e. shell's 3 global displacements, 2 local rotations used by CCIsoShellTriangle<...> and CCIsoShellQuad<xxxxxxxxx> elements). Most 3D elements can serve as master elements, i.e. solid, shell 3D, beam 3D, beam 1D... Sign of MAX_DISTANCE *x* does not matter and correspondes to the processing of negative DISTANCE parameter by LOAD_MASTER_SLAVE_NODES. SEARCH_RADIUS *r* set radius, where to look for surrounding elements to fix the shell slave node. FIX_ROTATION_xx specifies, which shell's edge note should be used to fix rotations. The "x" can be B,M and T for bootom, middle and top, respectively.The optional parameter *shell_group_id* helps to indicate, what shell should be used to connect the slave nodes. If it is not specified, a first shell with the appropriate incidence is used. The same applies for *solid_group_ids_interval.* If not specified, any solid element{s) can be used. The ROTATION_ARMS *n* parameter specified, how many points are used to fix its CS's rotation. By default, n=1, i.e. 1 node and 1 node below the mid-plane. For n=2 we use nodes at -h, -h/2, h/2, h, where h is CS's height.

Example:

```
LOAD CASE ID 2 NAME "Supports shell to solid"  SHELL_2D_TO_SOLID
   LIST_SHELL_NODES "SHELL_TO_SOLID_CONTACT_NODES"
   LIST_SOLID_NODES "LIST_SOLID_NODES"
   MAX_DISTANCE 0.00001
   SHELL GROUP 1 EMBEDDED SOLIDS GROUPS FROM 100 TO 200
```

&LOAD_BEAM_1D_TO_SOLID:
{ LOAD_BEAM_1D_TO_SOLID LIST_BEAM_NODES " *list_of_slaves* " |
LIST_SOLID_NODES " *list_of_masters* " | MAX_DISTANCE *x* |
SEARCH_RADIUS *r* | { FIX_ROTATION_BM | FIX_ROTATION_MT |
FIX_ROTATION_BT } | { FIX_ROTATION_LM | FIX_ROTATION_MR |
FIX_ROTATION_LR } | BEAM_GROUP *beam_group_id* | EMBEDDED
SOLIDS_GROUPS *solid_group_ids_interval* | ROTATION_ARMS *n* }+

Table 113: &LOAD_BEAM_1D_TO_SOLID description

The LOAD_BEAM_1D_TO_SOLID command structure is similar to the command

LOAD_MASTER_SLAVE_NODES described above and most its parameters are the same. The main difference is, however, that it connects 6 dofs of each slave node, (beam's 3 global displacements, 3 global rotations used by CCIsoBeamBar<..> elements). Most 3D elements can serve as master elements, i.e. solid, shell 3D, beam 3D, beam 1D... Sign of MAX_DISTANCE *x* does not matter and correspondes to the processing of negative DISTANCE parameter by LOAD_MASTER_SLAVE_NODES. SEARCH_RADIUS *r* set radius, where to look for surrounding elements to fix the beam slave node. FIX_ROTATION_xx specifies, which beam's edge note should be used to fix rotations. The "x" can be B,M,T,L and R for bootom, middle, top, left, right, respectively. The optional parameter *beaml_group_id* helps to indicate, what beam should be used to connect the slave nodes. If it is not specified, a first beam with the appropriate incidence is used. The same applies for *solid_group_ids_interval*. If not specified, any solid element{s) can be used. The ROTATION_ARMS *n* parameter has similar meaning as for the above LOAD_SHELL_2D_TO_SOLID boundary condition. It affects fixing rotation about s,t local coordinate axes. By default, n=1, i.e. it uses nodes (0,h/2), (0,h/2), (-b/2,0), (b/2,0). For n=2 we use nodes (0,h/2), (0,h/2), (-b/2,0), (b/2,0) and additionally (0,h/4), (0,h/4), (-b/4,0), (b/4,0).

Example:

```
LOAD CASE ID 5 NAME "Supports_beam to solid"  BEAM_1D_TO_SOLID
   LIST_BEAM_NODES "B2"
   LIST_SOLID_NODES "N1N2N3N4"
   MAX_DISTANCE 0.00001
   BEAM_GROUP 2 EMBEDDED SOLIDS_GROUPS FROM 1 TO 1
```

&LOAD_VALUE:
{ [VALUE *value*] | { [CONST *const*] | [COEFF_X *coeff_x*] | [COEFF_Y *coeff_y*] | [COEFF_Z *coeff_z*] } }

Table 114: LOAD_VALUE description

This command can be used to define a general spatial distribution of loads in the form:

$$f(x,y,z) = (const + x\,coeff\_x + y\,coeff\_y + z\,coeff\_z)\,value$$

&SLAVE_NODES
SLAVE { [NODE] n_i [DOF] i_i }+

&MASTER_NODES
MASTER { [NODE] n_i [DOF] i_i [*] x_i }+

&LOAD_PLACE
{ NODE *node* | SELECTION *"list_name"* DOF *idof*

&LOAD_FUNCTION:
{ [INCREMENT | TOTAL] FUNCTION *i* }2

Table 115: LOAD_FUNCTION description

Most boundary conditions (specified by command structure &LOAD) can be adjusted according to the current time. The "adjustment" is defined by a time dependent functions

specified by &LOAD_FUNCTION, which in fact, specifies a coefficient for the given boundary condition.

The actual coefficint for mutiplying the load is calculated as follows:

$c_{t_i} = f_{incr}(t_i)\ (f_{tot}(t_i) - f_{tot}(t_{i-1}))$,

where c_{t_i} is load multiplier, $f_{tot}(t), f_{incr}(t)$ are values of the total and increment load functions at time t, t_i and t_{i-1} is time at current and previous step, respectively. The above formula is applicable for loads that have incremental character. For loads with total character the load multiplier is calculated by:

$c_{t_i} = f_{incr}(t_i)\ f_{tot}(t_i)$.

Examples of such (total) loads are &MASS_ACCELERATIONS, &CHLORIDES, &CARBONATION, &FIRE_BOUNDARY, &MOIST_TEMP_BOUNDARY_LOAD, boundary conditions with &DISPLACEMENT_TYPE == VELOCITY or ACCELERATION etc.

Of course, in practice you use either $f_{tot}(t)$ or $f_{incr}(t)$. Nevertheless, theoreticaly both functions can be used in the same time. If $f_{incr}(t)$ is not specified, its value is assumed equal one for any t. If $f_{tot}(t)$ is not specified, then it is assumed that $(f_{tot}(t_i) - f_{tot}(t_{i-1})) = 1$ for incremental and $f_{tot}(t_i) = 1$ for total load. If neither INCREMENT nor TOTAL keyword is given, then INCREMENT is assumed.

Note that the function applies only to “fixed” boundary constraints from &LOAD_VALUE and/or from &ELEMENT_LOAD and not to master-slave DOFs constrains, if the master is not fixed. Even if it is fixed, it applies only to its &LOAD_VALUE part.

It cannot be specified for the &LOAD_MASTER_SLAVE_NODES, because the slave degree of freedoms inherit this function from their master degrees of freedom.

3.7.1.6 The Sub-command &ELEMENT_LOAD

```
&ELEMENT_LOAD
LOAD [&LOAD_FUNCTION ] { &LOAD_FUNCTION | [INITIAL]
      &BODY_ELEMENT_LOAD |  &BOUNDARY_ELEMENT_LOAD |
      &TEMPERATURE_ELEMENT_LOAD |  &HUMIDITY_ELEMENT_LOAD
      &ELEMENT_INITIAL_STRAIN_LOAD |
      &ELEMENT_INITIAL_STRESS_LOAD | &PRESTRESSING_LOAD
      &FIXED_PRESTRESSING_LOAD | &FIXED_PRESTRAINING_LOAD |
      &MASS_ACCELERATIONS_LOAD | &ELEMENT_INITIAL_GAP_LOAD |
      &DURABILITY_ELEMENT_LOAD }
```

3.7.1.6.1 The Sub-command &LOADED_ELEMS and &LOAD_COEFF

```
&LOADED_ELEMS:
[ GROUP { group_id [ TO group_id_to [ BY group_id_by]] } | { SELECTION
      list_name }  [ ELEMENT  {  element_id [ TO element_id_to [ BY
```

element_id_by]] } | { SELECTION *list_name* }] [INSIDE_T_TDT_ONLY] [CONSIDER_CONSTR_TIME]]

The flag INSIDE_T_TDT_ONLY forces ATENA to apply the load only to elements, whose time of construction falls in range $\langle t \ldots t + \Delta t \rangle$. Consequently, they are loaded only once (within an execution history). The flag CONSIDER_CONSTR_TIME ensures that a time dependent load function receive argument $(t - t_{constr})$ instead of (t), where t is current time in the step. This is useful (for example) for shrinkage during digital printing of a concrete structure.

&LOAD_COEFF :
[COEFF *const*] [COEFF_X *coeff_x*] [COEFF_Y *coeff_y*] [COEFF_Z *coeff_z*]

3.7.1.6.2 The Sub-command &BODY_ELEMENT_LOAD

&BODY_ELEMENT_LOAD:
BODY [&LOADED_ELEMS] [&LOAD_COEF] [{LOCAL | GLOBAL}] { { X | Y | Z | ROT_X | ROT_Y | ROT_Z | DOF *idof* } [VALUE] *x* }+ | [MNODE_IDS *"mnode_ids"* MNODE_LOADS *"mnode_loads"* MNODE_DOFS_MASK *mask* [MNODE_MAX_DISTANCE *abs_max_dist*] [MNODE_DIM1 *n1*] [MNODE_DIM2 *n2*] [MNODE_DIM3 *n3*]]

3.7.1.6.3 The Sub-command &BOUNDARY_ELEMENT_LOAD

&BOUNDARY_ELEMENT_LOAD:
BOUNDARY [&LOADED_ELEMS] [&LOAD_COEF] [{LOCAL | GLOBAL}] [{ANY[23] | SURFACE| EDGE | EDGE_NO_DUPLICATES}] | [MULTIPLE {YES|NO}] | [NODES "*loaded_nodes*"] { { X | Y | Z | ROT_X | ROT_Y | ROT_Z | DOF *idof*} [VALUE] *x* }+ [MERGE [MERGE_STRING *str*] [NO_ELEM_OUTPUT] | [MNODE_IDS *"mnode_ids"* MNODE_LOADS *"mnode_loads"* MNODE_DOFS_MASK *mask* [MNODE_MAX_DISTANCE *abs_max_dist*] [MNODE_DIM1 *n1*] [MNODE_DIM2 *n2*]]

3.7.1.6.4 The Sub-command &TEMPERATURE_ELEMENT_LOAD

&TEMPERATURE_ELEMENT_LOAD
TEMPERATURE [&LOADED_ELEMS] [&LOAD_COEF] { REFERENCE [TIME] *t_ref* TARGET [TIME] *t_target* [IMPORT GEOMETRY *geometry_filename*] IMPORT [HISTORY] RESULTS *results_filename* } | { [VALUE *x*] | [REF_VALUE *ref_x*] | [NODE_ID *node_id* NODE_VALUE *node_value* | REF_NODE_VALUE *ref_node_value* |

[23] The option ANY is only available in 4.3.1 and older; starting 4.3.2, the default is "SURFACE" for 3D problems and "BOUNDARY" for 2D and axisymmetric problems.

{AUTOMATIC|MANUAL} | TIME_UNITS "*time_units*" | TIME_FNC_ID *fnc_id* }

3.7.1.6.5 The Sub-command &HUMIDITY_ELEMENT_LOAD

&HUMIDITY_ELEMENT_LOAD
HUMIDITY [&LOADED_ELEMS] [&LOAD_COEF] { REFERENCE [TIME] *t_ref* TARGET [TIME] *t_target* [IMPORT GEOMETRY *geometry_filename*] IMPORT [HISTORY] RESULTS *results_filename* } | { [VALUE *x*] | [REF_VALUE *ref_x*] | [NODE_ID *node_id* NODE_VALUE *node_value* | REF_NODE_VALUE *ref_node_value* | {AUTOMATIC|MANUAL} | TIME_UNITS "*time_units*" | TIME_FNC_ID *fnc_id* }

3.7.1.6.6 The Sub-command &ELEMENT_INITIAL_STRAIN_LOAD and &ELEMENT_INITIAL_STRESS_LOAD

&ELEMENT_INITIAL_STRAIN_LOAD:
[INITIAL] STRAIN [&LOADED_ELEMS] [&LOAD_COEF] [IP *ip_id*] { X | Y | Z | XY | YX | YZ | ZY | XZ | ZX } [VALUE] *x_element_initial_strain* }+

&ELEMENT_INITIAL_STRESS_LOAD:
[INITIAL] STRESS [&LOADED_ELEMS] [&LOAD_COEF] [IP *ip_id*] { X | Y | Z | XY | YX | YZ | ZY | XZ | ZX } [VALUE] *x_element_initial_stress* }+

3.7.1.6.7 The Sub-command &PRESTRESSING_LOAD, &FIXED_PRESTRESSING_LOAD and &FIXED_PRESTRAINING_LOAD

&PRESTRESSING_LOAD:
PRESTRESSING [&LOADED_ELEMS] [&LOAD_COEF] [VALUE] {START_NODE | END_NODE | START_AND_END_NODE } *prestres_val*

&FIXED_PRESTRESSING_LOAD :
FIXED_PRESTRESSING [&LOADED_ELEMS] [&LOAD_COEF] [DIRECTION] { START_TO_END | END_TO_START }] { [VALUE | VALUES] { *s_coord value_at_s* }+ | VALUE_FNC *i* }

&FIXED_PRESTRAINING_LOAD [&LOADED_ELEMS] [&LOAD_COEF] [DIRECTION] { START_TO_END | END_TO_START }] { [VALUE | VALUES] { *s_coord value_at_s* }+ | | VALUE_FNC *i* }

3.7.1.6.8 The sub-command &MASS_ACCELERATIONS_ELEMENT_LOAD

&MASS_ACCELERATIONS_ELEMENT_LOAD:
MASS_ACCELERATIONS [&LOADED_ELEMS] [&LOAD_COEF] {LOCAL | GLOBAL} { { X | Y | Z | DOF *idof* } [VALUE] *x* }+

3.7.1.6.9 The sub-command &ELEMENT_INITIAL_GAP_LOAD

&ELEMENT_INITIAL_GAP_LOAD:
[INITIAL] GAP [&LOADED_ELEMS] INIT_STEP_ID *n*

Example:

LOAD PRESTRESSING group 1 VALUE 10000

Table 116: ELEMENT_LOAD description

Use the above command structure to define loads applied to finite element(s).

If element a group and element ids range/selection is specified, then all (group_id,element_id) combinations are processed except for the case, when loaded elements are specified by two selections, one with group ids and another with element ids (of the loaded elements). If n_{group} and n_{elem} is number of entries in the appropriate selections, then in this case $\min(n_{group}, n_{elem})$ elements get loaded.

If no group and element ids range/selection is specified, all elements in the structure are loaded.

Currently the supported types are:

- Volumetric (mass or body) load in a general direction (defined as a vector in reference coordinate system), &BODY_ELEMENT_LOAD, (e.g. in units KN/m^3). It can be specified in global or local coordinate system. Note that some elements do not define a local coordinate system, in which case the option GLOBAL is the same as the LOCAL. In addition, body element load can also include concentrated load (applied to volume of the element). For more info refer to &BOUNDARY_ELEMENT_LOAD, where this load option is described.
- Surface/edge load in a general direction (defined as a vector in reference coordinate system), &BOUNDARY_ELEMENT_LOAD, (e.g. in units KN/m^2), the load is applied to finite nodes enlisted in the selection "*loaded_nodes*" . It can be specified in global or local coordinate system. Note that some elements do not define a local coordinate system, in which case the option GLOBAL is the same as the LOCAL. The {ANY[24]|SURFACE|{EDGE|EDGE_NO_DUPLICATES}} switch defines toward which type of element boundary is the load applicable. Important: one definition of a boundary load can load each element only at its one edge (or surface); otherwise an error is produced. If you need to load more element's edges/surfaces simultaneously, split the load into several boundary loads. EDGE_NO_DUPLICATES ensures that only one element can contribute the load along any part of the loaded edge. The EDGE and EDGE_NO_DUPLICATES keywords may be replaced with their synonyms LINE and LINE_NO_DUPLICATES with the same effect. The flag [MULTIPLE {YES|NO}] specifies, whether the boundary load is aplicable for multiple surfaces/edges or

[24] The option ANY is only available in 4.3.1 and older

only for a single surface/edge per one finite element.

In addition, boundary and body element load can also include a load at arbitrary location(s). It can take a form of a concentrated load at specific points, or a load applied along one or more lines, areas and volumes, (the last option is available only for element body load). Units of the load change respectively, i.e. force, force/length etc. The control location of the load, i.e. points, lines... need not coinside with the finite element mesh and can overlap several elements, (and/or an element can be loaded by more such loads). The load must be applied to the above defined element edge or surface, (the case of boudary load) or to a body, (the case of body load). It is input as MNODE_IDS *"mnode_ids"* MNODE_LOADS *"mnode_loads"* MNODE_DOFS_MASK *mask* [MNODE_MAX_DISTANCE *abs_max_dist*] [MNODE_DIM1 *n1*] [MNODE_DIM2 *n2*] [MNODE_DIM3 *n3*]], where *"mnode_ids"* is selection with macro node ids, where the concentrated load is applied, *"mnode_loads"* is a selection containing corresponding loads, (i.e. real numbers), defined by SELECTION_REAL command. The loads are sorted by loaded dofs and mnode_ids. *mask* is bitwise map indicating, which dofs are to be loaded.. For example mask=6 means loading of dir_y(=bit 2=2) + dir_z(=bit3=4), i.e. *"mnode_loads"* must comprise FY, FZ at mnode_1, followed by FY, FZ at mnode_2 etc. The parameter *abs_max_dist* sets an accuracy, with which ATENA checks, if a macro node is or is not located at the processed element surface. In the latter case the surface is not loaded. Finaly, the parameters *n1, n2, n3* specify type of loaded loacation. For *n1=n2=n3*=1 the load is applied to points.This is default behaviour. If just one of *nx*>1, then the load is applied to a line. If just two of *nx*>1, then the load is applied to an quadrilateral and similarly if all three of *nx*>1, then the load is applied to a hexahedral, (applicable for element body load only). Layout of the controle load objects resembles the layout used for finite element of the same shape and the actual value of *nx* specify number of points, into which the load should be distributed, (*n1,n2,n1* for directions in isoparametric *r,s,t* coordinates).

- The MERGE flag is used, if the current boundary load should be merged with a previous boundary load within the same load case. MERGE_STRING *str* allows merging only boundary loads with the same MERGE_STRING *str*. The merging is successful, if the current and the other boundary load are of the same type, (edge/surface) and have the same values. Other parameters, (e.g. *function_id, coeff_x* etc.) are not tested and values from the other boundary load are adopted. If the merging is not successful, then the current boundary load is processed in the same way as it would without the MERGE flag. The NO_ELEM_OUTPUT flag suppress element boundary related output at element level. Note that only single element surface or edge can be loaded within single boundary load. Hence, use MERGE option with caution.
- TYPE_STRING *str* is used only for output data aggregation.
- Element temperature load, &TEMPERATURE_ELEMENT_LOAD that corresponds to element initial strain load, where initial strains are calculated based on material expansion coefficient and specified temperature. The temperature history can also be imported from the associated CCStructuresTransport analysis. In this case one has to input IMPORT subcommand. If *results_file_name* is specified without *geometry_filename_name*, it means that imported and current models are identical. If *geometry_filename_name* is specified, an interpolation between the two

models is executed. For a proper import target and reference times are needed, (see REFERENCE [TIME] *t_ref* | TARGET [TIME] *t_target*. This is because any loading in ATENA is assumed to be of incremental character. Hence, the TEMPERATURE_LOAD is imported as temperature increments between the structural conditions at target and reference time. If omitted, *t_ref* and *t_target* are automatically derived from the current step time and its increment. Alternatively, temperature load increments at element nodes can be input directly using syntax { NODE_ID *node_id* NODE_VALUE *node_value* }. Note that element node related input is always added to average element temperature load, see [VALUE] *x*. Some material laws are temperature denpent and thus they need info about absolute temperatures, rather then temperature increments (used e.g. for element load due the material thermal expansion). These are input thru REF_VALUE *ref_x* and REF_NODE_VALUE *ref_node_value* in the similar way as temperature increments via VALUE *x* and NODE_VALUE *node_value*. Note that from the transport analysis, i.e. using the IMPORT command, they are imported automatically. The reference temperatures ignores any load coefficient coming from function definition, load case multiplier etc. The AUTOMATIC option causes Atena to automatically update TARGET and REFERENCE TIME according to time at the current and previous step. It is usefull particularly for element tremperature load during creep analysis. If AUTOMATIC, the load is imported from history files and no additional load is acceptable, (such as via VALUE and NODE_VALUE). By default, MANUAL regime is assumed. }. The TIME_UNITS "*time_units*" allows to specify, which time units were used to calculate and write the transpored analysis results in the file *results_file_name*. It is specified in the same way as in the Unit command. By default no time unit conversion is made. TIME_FNC_ID *fnc_id* specifies a function that map TARGET and REFERENCE TIME towards actual load time. This time is then used for interpolation within the imported load data. Note that time of imported data is converted according to TIME_UNITS "*time_units*".

- Element humidity load, &HUMIDITY_ELEMENT_LOAD. Its input and behaviour resembles that for TEMPERATURE_ELEMENT_LOAD.
- Initial element strains, &ELEMENT_INITIAL_STRAIN_LOAD, (usable e.g. for pre-stressed conditions)
- Initial element stresses, &ELEMENT_INITIAL_STRESS_LOAD
- Prestressing of external cables (i.e. elements CCExternalCable_2D a CCExternalCable_3D), &PRESTRESSING. The prestressing can be applied near the start node, (i.e. the 1st principal node, set by PRESTRESSING .. START_NODE), end node, (i.e. the last principal node set by PRESTRESSING .. END_NODE) or near both ends of the cable set by PRESTRESSING .. START_AND_END_NODE. It is specified as prestress increment. If it is specified in some steps and not specified in the higher steps, then in the higher steps the cable prestressing and nodal slips may change (as a consequence of an additional cable deformation). However the nodal slips at the cable ends will remain the same, i.e. they are fixed.
  Presstresing orientation can be also input via &EXTERNAL_CABLE_GEOMETRY_SPEC, however such info is overwritten by orientation info within the &PRESTRESSING command.

- Fixed prestressing, &FIXED_PRESTRESSING, is another type of loading that can be used to set cable prestressing. This is useful, if the cable prestress losses are calculated by a third party software. In fact this type of loading is equivalent to ELEMENT_INITIAL_STRESS_LOAD load, whereby the prestress value is input as a function of the longitudinal bar coordinate s . If this coordinate has the same orientation as the reinforcement bar incidences, than use DIRECTION START_TO_END. Otherwise use DIRECTION END_TO_START. This type of loading allow to prescribe only local sig_xx stress. It is specified as prestress increment. Fixed prestressing as a fuction of the longitudinal coordinaye can be specified directly whithin thi scommand or a seperate funtion can be used.
- Prestraining of external cable by per element specified initial strain, &FIXED_PRESTRAINING. It is specified as prestrain increment
- Special type of element "load" is introduced by &ELEMENT_INITIAL_GAP_LOAD. This load is used for gaps that are initially open. Size of the openning is derived from the gap element's thickness at step INIT_STEP_ID *n.* This load must be included only in a load case being used for the definition of step *n.* Other steps will ignore it.
- CHLORIDES, CARBONATION and ASR element load does not represent a real load and are described in Section 0. It merely forces Atena to calculate degradation of reinforced concrete elements due to progression of carbonation and/or chlorides from their outside surfaces.
- Volumetric (mass or body) load due to accelerations (increments) in a general direction (defined as a vector in reference coordinate system), &MASS_ACCELERATIONS (e.g. in units m/s^2). It can be specified only in global coordinate system. During the load assembling it is replaced by a concentrated force with value (-m*a), where "a" is the specified acceleration and "m" is nodal mass (from calculation of mass matrix, optionally increased by nodal lumped masses). If a load time function is specified, (i.e. being understood as the load accelerogram function), it is assumed that this function defines total accelerations in a time (and not load increments, as it is usual in most other load types). The corresponding load increment at time $t+\Delta t$ is then calculated as $a(f(t+\Delta t)-f(t))$, where $f(t)$ is the acceleration function and a is constant acceleration in a particular direction having been input within this load specification. This load is meanigful in dynamic analysis only and because of its "total" character, it must be specified in the group of "fixed" load within the dynamic load step definition, (i.e. not among "increment" loads!

The element load is aplied to element groups specified by GROUP *group_id* [TO *group_id_to* [BY *group_id_by*]] command tokens. Otherwise all element groups are loaded. For each element group it is possible to load only some elements. Their list is input in ELEMENT SELECTION *list_name* command tokens. If the list contains a non-existing element, the corresponding entry is ignored. Alternatively, the loaded elements can be input in form of interval ELEMENT *element_id* [TO *element_id_to* [BY *element_id_by*]]. In this case, however, one have to be cautious. *element_id* [TO *element_id_to* must exist in the group *group_id*. For the remaining element groups, i.e. up to *group_id_to*, internal element numbering is used. E.g. let group *group_id* has elements 100, 105, 108, 110, 120, 130 and *element_id*=105, *element_id_to*=110. Then the remaining loaded element groups

(i.e. groups up to *group_id_to*) receive the load into their second, third and forth element. (The elements within each group are sorted according to their element_id). As usuallly, by default all elements of the group are loaded.

In addition, it is possible to use linear spatial interpolation based on the element's centre-point coordinates and COEFF_X *coeff_x*] [COEFF_Y *coeff_y*] [COEFF_Z *coeff_z*] see Table 114. By default, *coeff_x*=0, *coeff_y*=0, *coeff_z*=0 and *const*=1. If only GROUP *group_id* is given (and [ELEMENT *element_id*] is omitted), then the load applies to all element of the specified element group. An exception to that is prestressing of external cable. This load is always applied in *element_id*=1 (and only once, if *element_id* is not specified).

Different values of element initial stress and strain can be applied at each material (i.e. integration) point, see IP *ip_id* input. If *ip_id*=0, the element load is applied into all material points. Hence, with *ip_id*=0 the user can specify "uniform" portion of a load (across the element) and then he can define the load deviation at a particular material point *ip_id.*

By default *ip_id*=0.

3.7.1.6.10 The Command ELEMENT_LOAD Options for &Durability Analysis

&DURABILITY_ELEMENT_LOAD:
[&CARBONATION] | [&CHLORIDES] | [&ASR]

&CARBONATION[25]:
CARBONATION { TIME_FNC_ID *id* | WATER_MASS *x* | CEMENT_MASS *x* | SCM_MASS *x* | CONCRETE_COVER *x* | K_CO2 *x* | CO2 *x* | RH *x* | NODES "*loaded_nodes*" " | A1 | A2 | A3 | F_T_CH | W_D | B | R_CORR | DX_CORR_DT_SPALLING | CEMENT_MASS | MAX_REINF_DEPTH | MAX_CORR_DT | RH | PSI }$_+$ [TYPE_STRING *str*] [MERGE [MERGE_STRING *str*]] [NO_ELEM_OUTPUT] [{IMMEDIATE_UPDATE_CORROSION_ON|IMMEDIATE_UPDATE_CORROSION_OFF}]}$_+$

&CHLORIDES[26]:
CHLORIDES { TIME_FNC_ID *id* | D_REF *x* | TIME_D_REF *x* | M_COEFF *x* | TIME_M_COEFF *x* | CONCRETE_COVER *x* | CS *x* | CL_CRIT *x* | NODES "*loaded_nodes*" | A1 | A2 | A3 | F_T_CH | W_D | B | R_CORR | DX_CORR_DT_SPALLING | CEMENT_MASS | MAX_REINF_DEPTH | T_AVER_OFFSET | MAX_CORR_DT | PSI }$_+$ [TYPE_STRING *str*] [MERGE [MERGE_STRING *str*]] [NO_ELEM_OUTPUT] [{IMMEDIATE_UPDATE_CORROSION_ON|IMMEDIATE_UPDATE_CORROSION_OFF}]}$_+$

&ASR[27]:

---

[25] development/testing implementation of CARBONATION, CHLORIDES, and ASR in version 5.3.x and older

[26] development/testing implementation of CARBONATION, CHLORIDES, and ASR in version 5.3.x and older

ASR { TIME_FNC_ID *id* | U_C *x* | U_L *x* | TAU_C_0 *x* | TAU_L_0 *x* | THETA_0 *x* | SAND_MASS *x* | REQUIRED_ALKALI_PER_REACTIVE_SILICA *x* | PROPORTION_REACTIVE_SILICA *x* | PROPORTION_REACTIVE_PARTICLES_IN_SAND *x* | MEASSURED_ASR_STRAIN *x* | THRESHOLD_ALKALI_IN_CONCRETE *x* | TOTAL_ALKALI_IN_MORTAR *x* | STEP_TIME_INCR *x* | MAX_NUMBER_OF_ITERS *n* | T_AVER_OFFSET *x* | S_L *x* | S_U *x* | EPS_CR *x* | EPS_U *x* | BETA_E *x* | BETA_FT *x* | BETA_GF *x* | H_MIN | H_AVER_OFFSET | BETA_SHAPE_FACTOR | ASR_MODE { APPLY_NOTHING | ASR_MODE APPLY_INIT_STRAINS | APPLY_REDUCTION_ALL | APPLY_INIT_STRESSES}+ }+

Table 117: Description of durability options for ELEMENT_LOAD command

This command extends the ELEMENT_LOAD command specifically for various durability analysis options. Use the above command structure to define loads applied to finite element(s) for durability load types.

CHLORIDES, CARBONATION and ASR element load does not represent a real load. It merely forces Atena to calculate degradation of reinforced concrete elements due to progression of carbonation and/or chlorides from their outside surfaces. The input data resembles &BODY_ELEMENT_LOAD. It applies to the parameters NODES "*loaded_nodes*" "*loaded_nodes*", MERGE, MERGE_STRING *str* and NO_ELEM_OUTPUT. The remaining parameters are:
- WATER_MASS, CEMENT_MASS and SCM_MASS - mass of water, cement and non-active suplementary cementitious material, SCM per 1m^3, [weight/volume], default 200kg/m^3, 400 kg/m^3, 50kg/m^3, respectively,
- CONCRETE_COVER : thickness of concrete cover layer, [length], default value 0.02m,
- K_CO2 : efficiency factor, [-], with typical values 0.3 for silica fume, 0.5 for low- calcium fly ash, 0.7 for high-calcium fly ash, effective only for concrete with SCM_MASS >0, i.e. not for Portland cement, default value 0.5,
- CO2 : content CO2 in the ambient air, [-], default 0.00036,
- RH : relative humidity of ambient air RH, [-], default 0.6,
- CL_CRIT : critical mass of chlorides per mass of SCM+cement for initialisation of reinforcement corrosion, [-] default 0.014.
- CS: mass of chlorides per mass of SCM+cement at surface, [-] default 0.103
-D_REF: reference chloride difussivity at TIME_D_REF , [length^2/time], default 1.e-12m^2/sec= 31.53mm^2/year
-TIME_D_REF: time at which D_REF is specified, [time], default 10 years,
-M_COEFF: exponent to calculate time evolution of chloride diffusion D, typically equal to 0.69/0.93/0.66 for structures submerged in salt water/suibject to high-low tide/air exposure regularly sprinkled by salt water
-TIME_M_COEFF: time, at which M_COEFF is valid, [time], default 30 years.
-A1: parameter with characteristic value, [length], default 7.44e-5 m,
-A2: parameter with characteristic value, [length], default 7.30e-6 m,
-A3: parameter with characteristic value, [length/stress], default -1.74e-5 m/MPa,
-F_T_CH: characteristic splitting tensile strength of concrete,[stress],default 3.5MPa,

[27] development/testing implementation of CARBONATION, CHLORIDES, and ASR in version 5.3.x and older

CEB_FIP(1991) f_t_ch_=0.3*f_c$^{2/3}$, ACI363R-92(1992) f_t_ch_=0.59*f_c$^{1/2}$, ACI318-99(1999) f_t_ch_=0.56*f_c$^{2/3}$
-W_D: critical crack width for spalling,[length], default 1 mm,
-B: parameter depending on the position of the bar, [-], default 9.5µm/µm,
-R_CORR: parameter, depends on the type of corrosion,[-], default 3, =1 for uniform corrosion(carbonation), =<2; 4> for pitting corrosion(typically chlorides),=<4; 5.5> for pitting corrosion(typically chlorides),
-DX_CORR_DT_SPALLING: corrosion rate after spalling, [length/time], default 30 µm/year,
-CEMENT_MASS: binder mass in m3 of concrete, [mass/length^3 , default 250kg/m^3,
-MAX_REINF_DEPTH: max. distance between loaded surface and reinforcement node, [m], default 0; if MAX_REINF_DEPTH>0, then the reinforcement corrosion analysis is carried out yielding a reduction of reinforcement cross sectional area,
-T_AVER_OFFSET: average offset for temperature of the loaded surface at,[temperature], default 0 Celsia; if available, the value is fetched from creep analysis and/or element temperature load,
-MAX_CORR_DT: max. time step for internal time integration, [time], default 1 month,
-RH: average concrete relative humidity,[-], default 0.6.; if available, the value is fetched from creep analysis,
-PSI: uncertainity factor, [-], default 1.
-IMMEDIATE_UPDATE_CORROSION_ON | IMMEDIATE_UPDATE_CORROSION_OFF The corrosion data can be updated immediately after the load's processing or at the end of the step. The former approach is preferable for corners and cases, when the next load should be added to the previous ones. The latter option is better for cases, when the same load is distributed into several loads (typically splitted by elements). Default is OFF.

For ByASR load:
- U_C - activation energy constant of the characteristic time tau_l, [temperature], default 5400 K,
- U_L 9126.85- activation energy constant of the latency time tau_l, [temperature], default 9400 K,
- TAU_C_0 - characteristic time for THETA_0, [time], default 80 days,
- TAU_L_0 - latency time for THETA_0, [time], default 145 days,
- THETA_0 - reference temperature for tau_l_0_, [time], default 38 days,
- SAND_MASS - total sand content in unit volume, [density], default 1613.4 kgm^{-3}
- REQUIRED_ALKALI_PER_REACTIVE_SILICA - amount of required alkali per kg of reactive silica, [-], default 0.154,
- PROPORTION_REACTIVE_SILICA - proportion of reactive silica, [-], default 0.218,
- PROPORTION_REACTIVE_PARTICLES_IN_SAND - proportion of reactive particles in the sand composition, [-], default 0.3,
- MEASSURED_ASR_STRAIN - ASR free expansion strain per kg/m^3 of the mixture, [-/(kg/m^3)], default 0.000001 m^3/kg, typically 8.93e-7 .. 1.34e-5 [m3/kg],
- THRESHOLD_ALKALI_IN_CONCRETE - threshold of alkali per m3 of concrete, [[density], default 3.7 kgm^{-3}
- TOTAL_ALKALI_IN_MORTAR- total available alkali content in a mortar, [density], default 6.2 kgm^{-3}
- STEP_TIME_INCR - maximum step time increments used for internal integration,

[time], default 30 days,
MAX_NUMBER_OF_ITERS - max number of iteration to calculate ksi, [-], default 30,
T_AVER_OFFSET - average offset for temperature of the loaded surface at,[temperature], default 0 Celsia; if available, the value is fetched from creep analysis and/or element temperature load,
S_L - limit compression stress_L = -0.3MPa for calculating Weight factor for ASR expansion
S_U - maximum compression stress_u = -10MPa for calculating Weight factor for ASR expansion
EPS_CR - ASR cracking strain Ɛcr= f0/Ec0
EPS_U - ASR ultimate strain Ɛu=Ɛcr+2Gf0/hf0,
BETA_E - min. coefficient for Young modulus E_residual/E_0, [-], default 0.1
BETA_FT - min. coefficient for tensile strength ft_residual/ft_0,[-], default 0.6
BETA_GF - min. coefficient for fracture energy Gf_residual/Gf_0, [-], default 0.6
H_MIN -min. relative humidity to start ASR,[-], default 0.75
H_AVER_OFFSET - average relative humidity minimum in the structure,[-], default 0.
BETA_SHAPE_FACTOR - shape beta factor, [-], default -2.
ASR_MODE { APPLY_NOTHING | ASR_MODE APPLY_INIT_STRAINS | APPLY_REDUCTION_ALL | APPLY_INIT_STRESSES}+ : specify, which actions related to ASR load should be considered.

The CHLORIDES, CARBONATION and d ASR element load use TIME_FNC_ID *id* function to project the "solution" time *t* to "degradation" time $t_d = f(t)$. (It is not a load's multiplier).

3.7.1.7 The Sub-command &SPRING_DEFINITION

&SPRING_DEFINITION:

SPRING DIRECTION $\{x\}_{\text{ncoords}}$ NODE *n* MATERIAL *n*

Table 118: &SPRING_DEFINITION sub-command parameters♥

Parameter	Description
DIRECTION $\{x\}_{\text{ncoords}}$	Spring direction. E.g. DIRECTION x_1 x_2 [x_3] Component x_3 is valid only in 3D problems. Positive internal spring force acts in the direction given by this vector.
NODE *n*	Node number, in which the spring is applied.
MATERIAL *n*	Spring stiffness material id.

Table 119: Other parameters for command &LOAD

Parameter	Description
ID *n*	Load case identification.

NAME "*load case name*"	Load case name in quotes, also for identification. E.g.: NAME "*load case name*"
MASTER { [NODE] n_i [DOF] i_i [*] x_i }+	List of master nodes, their degrees of freedom and multipliers. E.g.: MASTER NODE n_1 DOF i_1 * f_1 .... NODE n_k DOF i_k * f_k
SLAVE { [NODE] n_i [DOF] i_i }+	List of slave nodes and their degrees of freedom. They are ordered according to MASTER E.g.: SLAVE NODE n_1 DOF d_1 .... NODE n_k DOF d_k
VALUE x	Prescribed nodal value, either displacement or force depending on context. E.g.: VALUE x
MASTER [SLAVE] [NODAL] [PAIRS] { n_i i_i }+	Ids of master-slave nodal pairs. E.g.: MASTER [SLAVE] [NODAL] [PAIRS] n_1 i_1 , n_2 i_2 , n_3 i_3 ... n_i i_i
NODE n DOF n	Node and its DOF specifying a place, where the simple boundary condition is applied.
FUNCTION n	Id of time function applied atop of a specified boundary condition. E.g.: FUNCTION n
{ X | Y | Z | DOF *idof* } [VALUE]	Element body load components in reference coordinate system, (in force per volume unit). If DOF *idof* is used, the specified value applies to a DOF *idof* . E.g. X [VALUE] x Y [VALUE] x Z [VALUE] x
TEMPERATURE	Element temperature, (in deg).
STRAIN { X | Y | Z | XY | YX | YZ | ZY | XZ | ZX } [VALUE]	Component of element initial strain components in reference coordinates system.
STRESS { X | Y | Z | XY | YX | YZ | ZY | XZ | ZX } [VALUE]	Component of element initial stress components in reference coordinates system.
GROUP, ELEMENT	Group and element ids, where the ELEMENT_LOAD is applied.

3.7.1.8 The Sub-command &RIGID_BODY, &INVERSE_RIGID_BODY

&RIGID_BODY
RIGID_BODY MASTER_ID *n* SLAVE_SELECTION *list_of_slaves* FIX_DOFS *dofs_mask*

Table 120: RIGID_BODY description

The RIGID_BODY command structure is a special case of &COMPLEX_LOAD_DISPLACEMENT, when each slave node defined in the selection *list_of_slaves* should be fixed with respect to the master node *n,* so that the couple nodes behaves like a rigid frame in the structure. Only dofs specified in *dofs_mask* are affected. The mask is coded as a bitwise number with 1 for fixed dofs and 0 for skipped dofs. A dof 1 is the most right bit, a dof 2 is the next bit to the left etc. As an example, if you want to fix dislacement x, displacement y and rotation x, you need to set the mask as decimal number 11. (Decimal 11 is binary 1011).

&INVERSE_RIGID_BODY
INVERSE_RIGID_BODY SLAVE_ID *n* MASTER_SELECTION *list_of_masters*
FIX_DOFS *dofs_mask* MASTER_WEIGHTS (*w1,w2…*)

Table 121: INVERSE_RIGID_BODY description

The INVERSE_RIGID_BODY command structure is opposite to RIGID_BODY command. While RIGID_BODY specifies that each DOF (in the mask) of each slave from *list_of_slaves* is to be fixed by master node *master_id*, here each DOF of slave node should be fixed by DOFs of master nodes defined in *list_of_masters*, i.e. only number of DOFS constraint equations are generated (irrespective of number of masters!). Weighted average of master nodes DOFs is used, as specified in *master_weights.* Number of masters weight factors is ecpected to be entered.

3.7.1.9 The Sub-command &BEAM_NL_CONNECTION

&BEAM_NL_CONNECTION
BEAM_NL_CONNECTION LIST_OF_NODES *list_of_nodes* SKIP_DOFS_MASK
skip_mask MAX_COORDS_TOL *max_tol*

Table 122: BEAM_NL_CONNECTION description

The BEAM_NL_CONNECTION command forces ATENA to browse thru all CCBeamNL_3 element groups and elements in it. If position of one element (axial) end node is closed to the same of another element, the two end nodes are connected. If *list_of_nodes* is not defined, this operation is carried out for all detected nodes. Otherwise, only nodes from the list can be connected. In the same way: this boundary condition connects all detected nodal deggre of freedom, (i.e. typically 6), unless *skip_mask.*is defined. If it is defined, the DOFs with the corresponding bit set ON are skipped. The last parameter, i.e. *max_tol* defines proximity region, from where two points are assumed to be candidate for the connection. It is given in absolute length unit, i.g. 0.001.

3.8 Step and Execution Commands

3.8.1 The Command &STEP

Syntax:

&STEP:
STEP { ID n_1 [TO n_2 [BY n_3]] | &STEP_TYPE_AND_DATA | EXECUTE }+

Currently the following step types are available:

&STEP_TYPE_AND_DATA:
{&STATIC_STEP_DEFINITION | &TRANSIENT_STEP_DEFINITION |
&CREEP_STEP_DEFINITION | &DYNAMIC_STEP_DEFINITION}

Table 123: &STEP command parameters

Parameter	Description
ID n_1 [TO n_2 [BY n_3]]	Steps interval that would be executed by EXECUTE subcommand. By default n_3=*1*, n_2= n_1.
&STEP_TYPE_AND_DATA EXECUTE	Type and data for a particular load step. Currently STATIC, TRANSIENT, CREEP and DYNAMIC type are available.
EXECUTE	Forces the immediate execution of the steps in interval ID *n1* [TO *n2* [BY *n3*]].

&STATIC_STEP_DEFINITION:
[TYPE] STATIC { NAME "*step name*" | ID *n* }+ {[LOAD] [CASE] $n_i * x_i$ }+

Table 124: & STATIC_STEP_DEFINITION command parameters

Parameter	Description
STATIC	Static load step.
NAME "*step name*"	Step name in quotes that is going to be defined.
ID *a*	Integral identification of the step "*step name*".
[LOAD] [CASE] { $n_i * x_i$ }+	Linear combination of load cases for step *"step name"*, which are to be used in this step. E.g.: LOAD CASE *1 * 1.5 2 * 0.8*

&TRANSIENT_STEP_DEFINITION:
[TYPE] TRANSIENT { NAME "*step name*" | ID *n* }+ {[LOAD] [CASE] $n_i * x_i$ }+

Table 125: & TRANSIENT_STEP_DEFINITION command parameters

Parameter	Description
TRANSIENT	Transport analysis load step.
NAME "*step name*"	Step name in quotes that is going to be defined.
ID *a*	Integral identification of the step "*step name*".
[LOAD] [CASE] { $n_i * x_i$ }+	Linear combination of load cases for step *"step name"*, which are to be used in this step. E.g.: LOAD CASE *1 * 1.5 2 * 0.8*

&CREEP_STEP_DEFINITION
TYPE CREEP { NAME "*step name*" | ID *n* | {AT|RESUME_AT} *time* | [{FIXED | INCREMENT }] [LOAD] [CASE] $n_i * x_i$ }+

Table 126: & CREEP_STEP_DEFINITION command parameters

Parameter	Description
TYPE CREEP	Creep load step. As creep analysis involve numerical time integration, the creep step consists typically of several "static like" integration steps, one for each sample time. It starts at creep step *time* of the current creep step and stops at min(*time* of the next creep step, *execution_stop_time*) (see &CREEP_ANALYSIS_PARAMS.) The analysis cannot exceed *time_end*, see &RETARDATION.
NAME "*step name*"	Step name in quotes that is going to be defined.
ID *a*	Integral identification of the step "*step name*".
{AT|RESUME_AT} *time*	Time at the beginning of the current creep step, in [days]. If "AT" label is used, ATENA assumes that an additional loading is applied in this step and therefore it automatically refines time integration, (i.e. it resets step time incerements *dt* to 0.1 days). If "RESUME_AT" label is used, no additional loading is assumed and thus, no special time refinement is carried out. This option can be used for getting user control and produce some print outs, figures etc. during execution of creep analyses.
[LOAD] [CASE] | [{FIXED | INCREMENT }] { $n_i * x_i$ }+	Linear combination of load cases for step *"step name",* which are to be used in this step. The FIXED type of load is evenly distributed into all applied integration time sub-steps of the current creep step, whilst the INCREMENT type is used only in the 1st integration sub-step. In the remaining sub-steps they are applied, but load values are a priori zeroised. Typically loads are specified as of INCREMENT type and LHS boundary conditions as of FIXED type. By default the FIXED type is assumed. E.g.: LOAD CASE FIXED *1 * 1.5 2 * 0.8* INCRENENT 3 * *1.3 4 * 10.8*

&DYNAMIC_STEP_DEFINITION
TYPE DYNAMIC { NAME "*step name*" | ID *n* | AT *time* | [{FIXED | INCREMENT }] [LOAD] [CASE] $n_i * x_i$ }+

Table 127: & DYNAMIC_STEP_DEFINITION command parameters

Parameter	Description
TYPE DYNAMIC	Dynamic analysis related load step. As dynamic analysis

	involve numerical time integration, the dynamic step consists typically of several "static like" integration steps, one for each sample time. It starts at *time* of the current step and stops at min(step *time* of the next dynamic step, *execution_stop_time*). It behaves similarly to creep analysis, however, dynamic analysis uses equal size sub-step time lenghts.
NAME "*step name*"	Step name in quotes that is going to be defined.
ID *a*	Integral identification of the step "*step name*".
AT *time*	Time at the beginning of the current dynamic step, in [days]. If the step's id is defined in form of an interval, the value of *time´* is incremente based on current time increment *dt.*
[LOAD] [CASE] | [{FIXED | INCREMENT }] { n_i * x_i }+	Linear combination of load cases for step *"step name",* which are to be used in this step. The FIXED type of load is evenly distributed into all applied integration time sub-steps of the current dynamic step, whilst the INCREMENT type is used only in the 1st integration sub-step. In the remaining sub-steps they are applied, but load values are a priori zeroised. Typically loads are specified as of INCREMENT type and LHS boundary conditions as of FIXED type. By default the FIXED type is assumed. E.g.: LOAD CASE FIXED *1 * 1.5 2 * 0.8* INCRENENT *3 * 1.3 4 * 10.8*

3.9 Output Command

Apart from the following tables, please see also the ATENA 3D User's Manual, section 5.5 Output Data Attributes or the ATENA Studio User's Manual, section 4.4 Output Data Attributes for additional information about most of the available output quantities.

3.9.1 The Command &OUTPUT

Syntax:

```
   &OUTPUT :
{ OUTPUT | PRINT } [ { OPTIMIZE_IM_SPEED | OPTIMIZE_IM_SIZE |
INDEX_MAPPING n } ] { &OUTPUT_TYPE | &FORMAT_CSV | &REGEX | {
        SPLIT_MONITOR_DATA_BY_LOCATION |
        UNSPLIT_MONITOR_DATA_BY_LOCATION } | NAME "set_name" |
        &EXPORT_IMPORT | &SUPLEMENT_MONITOR
        [PRESERVE_OUTPUT_OPTIONS] | | REMOVE | { FILE_OPEN | FILE_CLOSE }
        "file_name" " [ { overwite | append } ] |     [MAXIMUM |
        MINIMUM|SUMMATION|AVERAGE]
          [RECORD] LENGTH x | &LOCATION | | {TRACK | RECORD} &DATA |
          TRACE { OFF | ON } | RECOVERY {LUMPED | VARIATIONAL|
                NEAREST_IP }   }+
```

```
&OUTPUT_TYPE:
{   STANDARD | {  MONITOR | MONITOR_1 | MONITOR_2 | MONITORS| PLOT |
      PLOT_1 | PLOT_2 }  [ EACH  {ITERATION | STEP}  ]   }

&EXPORT_IMPORT:
{ INTERPOLATE { FULL | NONE | STEP } EXPORT {DATA|CMDS} TO
      "filename" |  IMPORT {DATA|CMDS} FROM "filename_1", "filename_2"...
      "filename_n" }

& SUPLEMENT_MONITOR :
SUPLEMENT FROM n ARCHIVES "filename_1", "filename_2"... "filename_n""

&FORMAT_CSV:
 {FORMAT__CSV_ON | FORMAT_CSV_OFF}

&REGEX:
  {REGEX_ON | REGEX_OFF}

&LOCATION:
LOCATION  { ELEMENT_IPS |  ELEMENT_ NODES  | NODES |  GLOBAL |
      LOAD_CASES |  ELEMENT_TYPES | MATERIALS | GEOMETRIES |
      OUTPUT_DATA } &LOCATION_LIST


&LOCATION_LIST:
{{ GROUP[S]  &INTERVAL   [ ELEMENT[S] &INTERVAL [ IP[S] &INTERVAL ] ]
      | GROUP[S]  &INTERVAL   [ ELEMENT[S]  &INTERVAL [ ENODE[S]
      &INTERVAL ] ]  |  NODE[S] &INTERVAL |  ID[S] &INTERVAL  | LOC_1
      &INTERVAL   [ LOC_2 &INTERVAL [ LOC_3 &INTERVAL ] ] }+  }|{
      MULTI_SELECTION AT [SELECTION] multi_selection_list }


&INTERVAL:
 { AT { n | FROM n [TO n [BY n]] | SELECTION selection_list} }+

&DATA:
DATA { ALL | { ITEM n [TO n [BY n]] LIST  {"output_keyword" [RECALCULATE]
      { AT n1 FROM ITEM n1 [TO n2 [BY n3]] }+ END }+ }
```

Table 128: &OUTPUT command parameters

Parameter	Description
OUTPUT | PRINT	Two output channels are supported: the primary and secondary. The primary channel is activated using the keyword OUTPUT and it is managed in the same way as all the other standard I/O streams in Atena. The secondary channel is activated by use of the keyword PRINT. It is aimed for exporting Atena output to a third party

	application via a disc file. Both the output channels can operate simultaneously as required.
{ OPTIMIZE_IM_SPEED | OPTIMIZE_IM_SIZE | INDEX_MAPPING *n* }]	Type of output items mapping used within the OUTPUT command. *n* =<0..6>, where the higher value of *n*, the faster processing at the cost of a slightly higher demand for RAM. By default *n*=6, i.e. OPTIMIZE_IM_SPEED. OPTIMIZE_IM_SIZE is equivalent to *n*=3.
MONITOR | MONITOR_1 | MONITOR_2 | MONITORS EACH {ITERATION | STEP }	Adds output set *"set_name" into* monitor output requests. Output format is set to produce output data records versus time, in which all output data (for a particular step or iteration, i.e. for a particular time) are written into one line. The first word of such line contains *"set_name",* followed by current *step id* , *iteration id* and *time*, and then all output items are sequentially printed one after another. Use "grep *set_name*" or similar to extract output lines corresponding to *"set_name"* output data for their import into a thirty-party post-processing package like spreadsheets etc. The specified output command is processed after completing of every iteration or step. If the keyword MONITOR is specified, the MONITOR_1 set is used. Two output sets are available, one called MONITOR_1 and the other MONITOR_2. Both of them can be used for monitoring output data per iteration or per step, however, it is not recommended to mix ouput monitors per iteration with monitors per step into the same monitor set. (It would result in a table with data delivered by iterations with empty slot for data monitored per step, when convergence was not reached yet.). Hence, one of the monitors is typically used for monitoring output at each iteration and the other for output at each step. Two output sets are particularly useful, if AtenaWin/ Atena Studio is used for execution of the ATENA analysis. This is because AtenaWin/AtenaStudio can directly plot all the data from the monitors into 2D plots without need of any thirty-party SW. However, in this case it is recommended to use the set MONITOR_1 for output monitors per iteration and the set MONITOR_2 for monitors per step, because AtenaWin / AtenaStudio automatically allocates a monitor with information about analysis convergence called "ConvergenceMonitor" into the set MONITOR_1 and it produces convergence information per iteration. The monitor MONITOR_1 is thus pre-selected" to output monitors per iteration and MONITOR_2 remains free for step monitors.

<table>
<tr><td></td><td>The option “MONITORS” is used for export/import data or command from/to the both monitors, i.e. it operates on both sets MONITOR_1 and MONITOR_2. It has nothing to do with definition of a particular output data monitoring.</td></tr>
<tr><td>PLOT | PLOT_1 | PLOT_2 EACH {ITERATION | STEP }</td><td>The way of using the keywords PLOT | PLOT_1 | PLOT_2 is nearly the same as the use of the keyword MONITOR | MONITOR_1 | MONITOR_2. When specified, it (also) creates a set of data that can be printed or drawn in 2D plots. The following table points out the differences:
<table>
<tr><td>Keyword</td><td>PLOT | PLOT_1 | PLOT_2</td><td>MONITOR | MONITOR_1 | MONITOR_2</td></tr>
<tr><td>Output definition produces actual output:</td><td>Yes</td><td>No</td></tr>
<tr><td>Output is produced automatically at each step / iteration during execution:</td><td>No</td><td>Yes</td></tr>
<tr><td>Output data are arranged by lines where each line corresponds to</td><td>the current time, (single line marked t=0)</td><td>a time at automatic execution of the output command, (many lines marked with current t).</td></tr>
<tr><td>RAM requirements for storing output:</td><td>Small. Only current data are stored.</td><td>Large. Full history is maintained.</td></tr>
<tr><td>The data are typically drawn as 2D plots at:
(It need not always be the case).</td><td>a fixed time and many locations</td><td>at a single location at many times</td></tr>
</table></td></tr>
<tr><td>{FORMAT_CSV_ON | FORMAT_CSV_OFF}</td><td>Use standard or CSV output formatting</td></tr>
<tr><td>{REGEX_ON | REGEX_OFF}]</td><td>If REGEX_OFF , then output data request must be written exactly. In this case it can be set before the actuall output data is available. If REGEX_ON, the output data request is searched for using regular expression. Only substring must match. It cannot be defined beforehand.</td></tr>
<tr><td>SPLIT_MONITOR_DATA</td><td>Split the monitor by location or leave it untouched. By</td></tr>
</table>

_BY_LOCATION | UNSPLIT_MONITOR_DATA_BY_LOCATION	default the monitor is not splitted. For example, if we have monitor "NODAL_DISPLACEMENT", it can be split to separate monitors "NODAL_DISPLACEMENT_AT_NODE_1", "NODAL_DISPLACEMENT_AT_NODE_2" ... "NODAL_DISPLACEMENT_AT_NODE_n". ELEMENT_NODE, ELEMENT_IPS AND ELEMENT Location's data are splitted by elements, e.g. "FORCES_AT_GROUP_20_ELEMENT_4". (The level 3 is not accounted for).
STANDARD	Output format is set to "table" oriented form, i.e. items are printed in separate tables. Each line of such a table presents results for one location. Output command request is processed immediately after its issuing.
NAME *"set_name"*	Name of monitor output set.
INTERPOLATE { FULL | NONE | STEP } EXPORT {DATA|CMDS} TO "*filename*" | IMPORT DATA|CMDS} FROM *"filename_1", "filename_2"... "filename_n"*	Export/Import data from/to specified monitors. The "export" is always for the current step, i.e. time. The import is for time saved in import archives. When importing, linear interpolation of monitored output data can be requested. If "INTERPOLATE STEP" is specified, the imported output data are smoothly connected to the data from the recent step. If "INTERPOLATE FULL" is input, the imported data get connected to the lastly entered value, e.g. typically value for a last previous step, where the data were monitored for the last time. "INTERPOLATE NONE" suppresses any interpolation. "*filename*" is binary file into which the data are exported. *"filename_1", "filename_2"... "filename_n"* are filenames of previously exported data that should be now imported. The "DATA" and "CMDS" options are used to export/import actual output data/monitor output command requests.
SUPLEMENT FROM *n* ARCHIVES *"filename_1", "filename_2"... "filename_n"* [PRESERVE_OUTPUT_OPTIONS]	Force Atena to automatically add the output data history into the both monitors, (regardless of MONITOR_1/MONITOR_2 option). For each of the specified archive files it restores that file, (i.e. state), executes current output monitor requests and exports all results. After that, it restores back the current state and imports all the exported data, thereby adding output data history, (i.e. monitors) from the specified archives. This command is useful, if at a later time it is needed to add some monitored data from previous times, (i.e. from previous archives).

	PRESERVE_OUTPUT_OPTIONS causes to use for the supplemented monitor data current settings of the output data conditions, (such as recovery type etc.) rather then the settings, which were in use during the original execution.
REMOVE	Removes output set *"set_name"* from monitor output requests.
{ FILE_OPEN | FILE_CLOSE } *"file_name"* [{ overwite | append }]	This command specifies the name of the output file to be opened or closed. All output following this command will be redirected to this file or the default stream. The open file is overwrited or appended.
[RECORD] LENGTH *x*	Maximum length of output record. Default value = 120.
&LOCATION	Specification of location type, where the data should be output. If no location is specified, the whole model is assumed. Some data are available only on one location type, e.g. displacement are of type LOCATION NODES, the other have more, e.g. stress has LOCATION NODES, LOCATION ELEMENT NODE and ELEMENT INTERNAL POINT. The location is also used for TRACE ON/OFF specification (see below).
&LOCATION_LIST	Output location, i.e. list of nodes, elements etc., where the data should be output. By default output is done at all available locations. Hence for example, in case of LOCATION_IPS the location list GROUP 1 ELEMENTS 2 TO 5 prints data at all internal points of elements 2,3,4, and 5 of group no. 1., list GROUP 2 TO 5 produces output at all IPs of all elements for groups 2 through 5 etc.
&INTERVAL	Location interval for output. Alternatively location interval can be specified by *selection_list.*
MULTI_SELECTION *multi_selection_list.*	Location ids for output are set by the selection list *multi_selection_list.* E.g. Ids of integration points are input sequentially in the selection list as follows: {$group_i$, $element_i$, ip_i}, i=1, number of input IPs
&DATA	List of data to be output. Each data is characterized by associated *"output_keyword"*. Actual list of available *"output_keyword"* is in ATENA created dynamically based on current status of the analysis. This list can be printed out in self-explanatory format by the command OUTPUT LOCATION ATTRIBUTE DATA ALL. Some of these *"output_keyword" are* also explained in the following table. For more information about the available output data attributes, see also the GUE User Manuals - ATENA Engineering 2D, 3D, ATENA Studio.

	If only some items of *"output_keyword"* are desired, define them by ITEM *n* [TO *n* [BY *n*]]. For example, if only stress σ_x and σ_y are needed, type ITEM 1 TO 2. The list of *"output_keyword"* is terminated by keyword END. If all output data for a particular location type are requested, use keyword ALL (instead of LIST *"output_keyword_1" "output_keyword_2"* ...END structure). If "RECALCULATE" keyword forces to recalculate the requested output data even if they were previously computed and cached.
TRACE {OFF | ON}	Flag for tracing results during iterations. By default, data (e.g. at element IPs) can be traced even during iterations; (either by OUTPUT MONITOR EACH ITERATION ... or from ATENA GUI). As this extra output service costs not-negligible resources (mainly RAM), the user may find reasonable to switch off this service in case of extensive analyses (e.g. at areas being not critical for structural over-all behavior). This output is available only for the location ELEMENTS.
RECOVERY {LUMPED | VARIATIONAL | NEAREST_IP }	Method for recovering output data akin stress, strain etc. from IPs to element nodes. It can be either VARIATIONAL, in which case an energy based is used to do the recovery, or a simplified LUMPED method. The former one is more accurate and theoretically thorough, however it is also more costly in terms of CPU requirements. By default, the LUMPED approach is used. Note that LUMPED is usually preferred for linear elements, whilst VARIATIONAL is the best choice for nonlinear elements. The third option, i.e. NEAREST_IP set values in element nodes to be equal to those at the nearest integration point. This output is available only for the location ELEMENTS.
MAXIMUM | MINIMUM|SUMMATION| AVERAGE	Output only maximum, minimum, sum or average of all values over the printed domain, incl. loop over specified data items(=components).
{TRACK | RECORD}	This flag is significant only for MONITOR output. If TRACK is used, the monitored output data are stored for later output and they are also printed immediately. The keyword RECORD inhibits the immediate output and the data are only stored for later use. Default value: TRACK

Table 129: Output-type keywords understood by the command &OUTPUT for the location type OUTPUT_DATA

Output keyword	Description
CURRENT_OUTPUT_DATA_ATTRIBUTES	List of output data, (i.e. list of *"output_keyword"* currently available for output.
RETARDATION_TIMES	Retardation times used for approximation of creep material compliance function.
LOAD_TIMES	Times of creep load steps.
SAMPLE_TIMES	Integration times for creep analysis.
GENERATED_CREEP_DATA	Exact and approximated values of creep material compliance function generated by a creep material model.
STEP_LOAD	Load cases applied at the current step.
MEASURED_WATER_LOSS	Measured laboratory water loss in concrete for improving creep model accuracy.
MEASURED_SHRINKAGE	Measured laboratory shrinkage in concrete for improving creep model accuracy.
MEASURED_COMPLIANCE	Measured laboratory compliance of concrete for improving creep model accuracy.
MONITOR_SET_1_*set_name*	Output of previously monitored (and stored) output data set *set_name* in MONITOR 1 or PLOT 1
MONITOR_SET_2_*set_name*	Output of previously monitored (and stored) output data set *set_name* in MONITOR 2 or PLOT 2.
SELECTION_IDS_*selection_name*	List of entities in the selection list *selection_name*.
SELECTION_GEN	Data for selection lists generation.
DISCRETE_REINFORCEMENT	Data for discrete reinforcement generation. Superseded by data attribute DISCRETE_REINFORCEMENT within location type MACRO_ELEMENTS
ELAPSED_CPU_TIME	Info about execution CPU time within steps.
SMART_IDS_MAP_INFO	Info about maximum reference ids for the mapped ATENA entities, such as nodes, element groups etc.
EIGEN_VALUES	Print calculated structural eigenvalues.
BEAM_CHECK_M_N_DATA	M-N diagrams for CCBeam3D elements with CCBeamMasonryMaterial and/or CCBeamRCMaterial
CURRENT_RHS_BC	Current values of RHS forces at nodes.
CURRENT_LHS_BC	Current values of LHS boundary conditions at nodes.
CURRENT_SORTED_LHS_BC	Same as the above but sorted in different way.
FNC_*xxx_yyy*	Output values for function *xxx* generated by command *yyy,* see &FUNCTION command.

HUMIDITY_TEMPERATURE_HISTORY	History of humidity and temperature at creep material history.

Table 130: Output-type keywords understood by the command &OUTPUT for the location type GLOBAL

Output keyword	Description
FEMODEL_ CHARACTERISTICS	Characteristics of the finite element model.
TASK_NAME	Problem task name. The name specified using the TASK command will be printed to the output stream.
TASK_TITLE	Title as it was specified using the TASK command.
STEP_ID	Step identifications being currently executed.
SOLUTION_ CHARACTERISTICS	Several parameters characterising solution process.
EIGENVALUES_CHARACTERISTICS	A few parameters used by eignevalues and eigenvectors analysis
CONVERGENCE_CRITERIA	Parameters for assessing convergence performance.
ARC_LENGTH_PARAMS	Parameters relevant for Arc Length method.
LINE_SEARCH_PARAMS	Parameters relevant for Line Search method.
STEP_CONVERGENCE	Values of convergence characteristics as printed in "message" file
LOAD_CASES_CONTRIBUTION	Load cases contribution, i.e. sums of load cases coefficient from the previous steps multiplied by step lambda factor. Note that this values can only be monitored after step, not in iterations.
USER_LOAD_CASES_CONTRIBUTION	Same as the above, but it prints out only user defined load case. Internally generated load cases are skipped, (e.g. connection between reinforcement and surrounding solids).
PUSHOVER_ANALYSIS_PARAMS	Input parameters and results for/of Pushover analysis. Note that the analysis is available only for static analysis without creep.

Table 131: Output-type keywords understood by the command &OUTPUT for the location type LOAD_CASES

Output keyword	Description
SUPPORT_SLAVE_NODES	List of slave nodes in specification of LHS boundary conditions.
SUPPORT_MASTER_NODES	List of master nodes in specification of LHS boundary conditions.
LOAD_SLAVE_NODES	List of slave nodes in specification of RHS boundary conditions, i.e. nodal loads.

MASTER_SLAVE_NODES	For each Master-Slave BC lists id of slave and master nodes, together with their recommended values.
ELEMENT_LOAD	Data for element load, such as element initial stress/strain load, body/boundary load, prestressing … applied to elements

Table 132: Output-type keywords understood by the command &OUTPUT for the location type ELEMENTS

Output keyword	Description
ELEMENT_INCIDENCES	Element incidences, i.e. element nodal connectivity.
CRACK_ATTRIBUTES	Crack attributes at IP. See ATENA 2D User's Manual, section *2.8.5.29 Results - Load step i - Elements - Crack attributes* for details.
ELEMENT_MATERIAL_TYPES	Material types at element integration points
BEAM_NL_MIDPOINT PARAMS	Several parameters describing element state/conditions for CCBeam3D element at its middle point, (only for beam with a material derived from CCBeamBaseMaterial).
ASR_KSI	ASR load data, such as ksi etc.
ELEMENT_AGE[28]	Element age in digital printing of the structure

Table 133: Output-type keywords understood by the command &OUTPUT for the location type ELEMENT_IPS

Output keyword	Description
IP_COORDINATES	Coordinates of element internal points (i.e. material integration points).
DISPLACEMENTS_AT_IPS	Element displacements at its integration points.
STRAIN	Green-Lagrange strains, i.e. total strains minus initial trains due to temperature load and initial strains load.
TOTAL_STRAIN	Total strains corresponding to the deformations.
PRINCIPAL_STRAIN	Principal engineering strains.
STRESS	Element stresses.
PRINCIPAL_STRESS	Principal element stresses.
PERFORMANCE_INDEX	Index for material performance characteristics.
SBETA_STATE_VARIABLES	State variables for SBETA material model at element internal points. Similar output is available also for other materials. See ATENA 2D User's Manual, section *2.8.5.9 Results - Load step i - Nodes - Sbeta State Variables* for details.
EPS_MI	Value of internal creep variables.

[28] Not available in ATENA version 5.7.0 and older

ELEM_INIT_STRAIN_INCR	Current element initial strain increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRAIN	Current element initial total strain (total from all loads and all time steps).
ELEM_INIT_STRESS_INCR	Current element initial stress increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRESS	Current element initial total stress (total from all loads and all time steps).
ELEM_TEMPERATURE_INCR	Current element incrementally applied temperatures (total from all loads for the current time step).
ELEM_TOTAL_TEMPERATURE	Total temperatures
EPS_MI	Internal material variables for creep analysis using Dirichlet series.
BOND_STRESS	Bond stress between reinforcement and concrete.
CABLE_FORCE	Forces in external cables.
FRACTURE_STRAIN	Fracture strains
PLASTIC_STRAIN	Plastic strains
CRACK_ATTRIBUTES	Crack attributes containing the number of cracks, their direction, openings and surface stresses. See ATENA 2D User's Manual, section *2.8.5.29 Results - Load step i - Elements - Crack attributes* for details.
TENSILE_STRENGTH	Current values of tensile strength
MAXIMAL_FRACT_STRAIN	Maximal value of fracture strain reached during the analysis for each material direction.
MATERIAL_TRANSFORMATION_MATRIX	Coordinate transformation matrix from global to local material coordinate system.
CRACKING_MODULI	Crack opening stiffnesses for each material direction including shear components.
DIRECTION_STATUS	Cracking status information for each material direction.
PERFORMANCE_INDEX	Relative stress error in the evaluation of the material model.
YIELD/CRUSH_INFO	Yielding/crushing status information
SOFT/HARD_PARAMETER	Softening/hardening parameter
EQ_PLASTIC_STRAIN	Equivalent plastic strain. The calculation method depends on the used material model.
ELEM_MASS_ACCEL_LOAD_INCR	Element load increments due to the element's acceleration, (for a particular step), transformed into nodal concentrated forces.
TOTAL_MASS_ACCEL_LOAD	Total element load due to the element's acceleration transformed into nodal concentrated forces.
BEAM_ELEM_NL_PARAMS	A few parameters describing nonlinear behaviour of CCBeam3D elements.
ASR_MATER_PROPS_COEFFS	Coefficients for material parameters E, ft and Gf due to ASR degradation

ASR_TOTAL_ELEM_STRAIN	Total expansion strains due to ASR degradation
ELEM_MP_HUMID_TEMPER[29]	Humidity and temperature imported from the corresponding transport analysis into creep material history.

Table 134: Output-type keywords understood by the command &OUTPUT for the location type ELEMENT_NODES

Output keyword	Description
STRAIN	Green-Lagrange strains, see the same output in the above table.
TOTAL_STRAIN	Total strain in the structure.
PRINCIPAL_STRAIN	Principal engineering strains.
STRESS	Element stresses.
PRINCIPAL_STRESS	Principal element stresses.
SBETA_STATE_VARIABLES	State variables for SBETA material model at element nodes. Similar output is available also for other materials. See ATENA 2D User's Manual, section *2.8.5.9 Results - Load step i - Nodes - Sbeta State Variables* for details.
PERFORMANCE_INDEX	Index for material performance characteristics.
BOND_SLIP	Slips along the bar reinforcement with the reinforcement bond model.
BOND_STRESS	Bond stress between reinforcement and concrete.
CABLE_FORCE	Forces in external cables.
FRACTURE_STRAIN	Fracturing strains
PLASTIC_STRAIN	Plastic strains
TENSILE_STRENGTH	Current values of tensile strength
MAXIMAL_FRACT_STRAIN	Maximal value of fracture strain reached during the analysis for each material direction.
PERFORMANCE_INDEX	Relative stress error in the evaluation of the material model.
YIELD/CRUSH_INFO	Yielding/crushing status information
SOFT/HARD_PARAMETER	Softening/hardening parameter
EQ_PLASTIC_STRAIN	Equivalent plastic strain. The calculation method depends on the used material model.
ELEMENT_CRACK_VOLUME	Coordinates of shell's volume with cracks
ELEM_INIT_STRAIN_INCR	Current element initial strain increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRAIN	Current element initial total strain (total from all loads and all time steps).
ELEMENT_ORIENTATION	Element orientation for bricks, Ahmad and beam elements. Useful especially for checking reference depth vectors of shells and beams.

[29] Supported since version 5.8.0

ELEM_INIT_STRESS_INCR	Current element initial stress increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRESS	Current element initial total stress (total from all loads and all time steps).
ELEM_TEMPERATURE_INCR	Current element incrementally applied temperatures (total from all loads for the current time step).
ELEM_TOTAL_TEMPERATURE	Total temperatures
ELEM_HUMIDITY_INCR	Current element incrementally applied humidities (total from all loads for the current time step).
ELEM_TOTAL_HUMIDITY	Total humidities
INTEG_STRESS	Cross sectional forces and moments for bended elements
M_N_Q_BEAM	Same as the above for beam elements.
M_N_Q_SHELL	Same as the above for shell elements.
ELEM_MASS_ACCEL_LOAD_INCR	Element load increments due to the element's acceleration, (for a particular step), transformed into nodal concentrated forces.
TOTAL_MASS_ACCEL_LOAD	Total element load due to the element's acceleration transformed into nodal concentrated forces.
BEAM_FORCES	Nx,Vy,Vz,Kx,My,Mz beam forces for CCBeam3D element.
ULTIMATE_BEAM_FORCES	Ultimate Nx,Vy,Vz,Kx,My,Mz beam forces for CCBeam3D element, (only for beam with a material derived from CCBeamBaseMaterial).
BEAM_NL_PARAMS	Several parameters describing element state/conditions for CCBeam3D element, (only for beam with a material derived from CCBeamBaseMaterial).
CARBONATION_DATA_AT_*surface_name*	Data about concrete degradation due to carbonation progressing from surface *surface_name*
CHLORIDES_DATA_AT_*surface_name*	Data about concrete degradation due to chlorides progressing from surface *surface_name*
REINF_CORROSION	Data about reinforcement degradation due to chlorides and carbonation
ASR_MATER_PROPS_COEFFS	Coefficients for material parameters E, ft and Gf due to ASR degradation
ASR_TOTAL_ELEM_STRAIN	Total expansion strains due to ASR degradation
ELEM_MP_HUMID_TEMPER[30]	Humidity and temperature imported from the corresponding transport analysis into creep material history.

[30] Supported since version 5.7.0

Table 135: Output-type keywords understood by the command &OUTPUT for the location type NODES

Output keyword	Description
NODAL_DEGREES_OF_FREEDOM	Output number of all degrees of freedom or associated DOFs boundary conditions
REFERENCE_NODAL_COORDINATES	Reference nodal coordinates
CURRENT_NODAL_COORDINATES	Current nodal coordinates.
STRAIN	Green-Lagrange strains.
TOTAL_STRAIN	Total strain including initial strains due to element load.
PRINCIPAL_STRAIN	Principal engineering strains.
STRESS	Element stresses.
PRINCIPAL_STRESS	Principal element stresses.
SBETA_STATE_VARIABLES	State variables for SBETA material model at nodes. Similar output is available also for other materials. See ATENA 2D User's Manual, section *2.8.5.9 Results - Load step i - Nodes - Sbeta State Variables* for details.
PERFORMANCE_INDEX	Index for material performance characteristics.
DISPLACEMENTS	Current minus reference nodal coordinates, (i.e. nodal displacements).
PARTIAL_INTERNAL_FORCES	Internal forces at nodes
PARTIAL_EXTERNAL_FORCES	Applied nodal forces (i.e. loading).
PARTIAL_REACTIONS	Global reactions.
PARTIAL_RESIDUAL_FORCES	Applied nodal forces minus internal forces.
INTERNAL_FORCES	Internal forces at nodes (compacted).
EXTERNAL_FORCES	Applied nodal forces (i.e. loading). (compacted)
REACTIONS	Global reactions (compacted)
RESIDUAL_FORCES	Applied nodal forces minus internal forces (compacted).
EPS_MI	Value of internal creep variables.
BOND_STRESS	Bond stress between reinforcement and concrete.
CABLE_FORCE	Forces in external cables.
FRACTURE_STRAIN	Fracturing strains
PLASTIC_STRAIN	Plastic strains
TENSILE_STRENGTH	Current values of tensile strength
MAXIMAL_FRACT_STRAIN	Maximal value of fracture strain reached during the analysis for each material direction.
PERFORMANCE_INDEX	Relative stress error in the evaluation of the material model.
YIELD/CRUSH_INFO	Yielding/crushing status information

SOFT/HARD_PARAMETER	Softening/hardening parameter
EQ_PLASTIC_STRAIN	Equivalent plastic strain. The calculation method depends on the used material model.
ELEM_INIT_STRAIN_INCR	Current element initial strain increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRAIN	Current element initial total strain (total from all loads and all time steps).
ELEM_INIT_STRESS_INCR	Current element initial stress increment (total from all loads for the current time step).
TOTAL_ELEM_INIT_STRESS	Current element initial total stress (total from all loads and all time steps).
ELEM_TEMPERATURE_INCR	Current element incrementally applied temperatures (total from all loads for the current time step).
ELEM_TOTAL_TEMPERATURE	Total temperatures
ELEM_HUMIDITY_INCR	Current element incrementally applied humidities (total from all loads for the current time step).
ELEM_TOTAL_HUMIDITY	Total humidities
EIGENVECTORS_x	Structure eigenvectors of the mode x, e.g. EIGENVECTORS_1 to print the 1st eigenvector.
IMPERFECTIONS	Incremental values of imperfect structural geometry (with regards to its reference coordinates).
ACCELERATION	Total nodal accelerations within dynamic analysis. Note the difference: other BCs are typically input as an increment per step.
VELOCITIES	Total nodal accelerations within dynamic analysis. . Note the difference: other BCs are typically input as an increment per step.
ELEM_MASS_ACCEL_LOAD_INCR	Element load increments due to the element's acceleration, (for a particular step), transformed into nodal concentrated forces.
TOTAL_MASS_ACCEL_LOAD	Total element load due to the element's acceleration transformed into nodal concentrated forces.
BEAM_FORCES	Nx,Vy,Vz,Kx,My,Mz beam forces for CCBeam3D element.
CARBONATION_DATA_AT_*surface_name*	Data about concrete degradation due to carbonation progressing from surface *surface_name*
CHLORIDES_DATA_AT_*surface_name*	Data about concrete degradation due to chlorides progressing from surface *surface_name*
REFERENCE_BORDER_COORDINATE	Cummulated geometrical distance of output nodes with respect to the previous node. This output data is used as the horizontal coordinate for plots of value along some border, cutting lines etc.
ASR_MATER_PROPS_COEFFS	Coefficients for material parameters E, ft and Gf due to ASR degradation

ASR_TOTAL_ELEM_STRAIN	Total expansion strains due to ASR degradation
ELEM_MP_HUMID_TEMPER [31]	Humidity and temperature imported from the corresponding transport analysis into creep material history.

Table 136: Output-type keywords understood by the command &OUTPUT for the location type GEOMETRIES

Output keyword	Description
2DGEOMETRY	Parameters for 2D geometry.
3DGEOMETRY	Parameters for 3D geometry.
BEAM_GEOMETRY	Parameters for beam geometry.
CABLE_GEOMETRY	Parameters for type "external cable" geometry.
SPRING_GEOMETRY	Parameters for geometry of springs.
TRUSS_GEOMETRY	Parameters for truss geometry.
LAYRED_SHELL_GEOMETRY	Parameters for layered shell geometry, (e.g. used by Ahmad degenerated shell element.
BEAM_3D_GEOMETRY	Parameters for 3D curved beam element.

Table 137: Output-type keywords understood by the command &OUTPUT for the location type ELEMENT_TYPES

Output keyword	Description
ELEMENT_TYPE	List of defined element types.

Table 138: Output-type keywords understood by the command &OUTPUT for the location type MATERIALS

Output keyword	Description
MATERIALS	List of defined materials with their parameters.
CURRENT_MATERIAL_PARAMETERS	Values of current material parameters for creep analysis like Dirichlet series coefficients, material strength in compression etc.

Table 139: Output-type keywords understood by the command &OUTPUT for the location type MACRO_ELEMENTS

Output keyword	Description

[31] Supported since version 5.7.0

MACRO_ELEMENT_DATA	Input data characterizing macro elements. See also data MACRO_ELEMENT_INCIDENCES and MACRO_ELEMENT_PROPERTIES
DISCRETE_REINFORCEMENT	Data for discrete reinforcement generation. Supersedes data attribute DISCRETE_REINFORCEMENT within location type OUTPUT_DATA
MACRO_ELEMENT_INCIDENCES	List of principal macro nodes that define each macro element.
MACRO_ELEMENT_PROPERTIES	Properties of macroelements and their principal nodes
MACRO_ELEMENT_GENERATED_ELEMENTS	List of finite elements that were created during generation of each macro element.
MACRO_ELEMENT_GENERATED_NODES	List of FE nodes that were created during generation of each macro element.

Table 140: Output-type keywords understood by the command &OUTPUT for the location type MACRO_NODES

Output keyword	Description
MACRO_NODAL_COORDINATES	Coordinates of macro nodes.

Examples:

OUTPUT LOCATION OUTPUT_DATA DATA LIST "CURRENT_SORTED_LHS_BC" END

OUTPUT NAME "displ" MONITOR_1 EACH ITERATION LOCATION NODES NODE AT 132 DATA LIST "DISPLACEMENTS" ITEM AT 3 END

OUTPUT NAME "s_coord" PLOT_2 LOCATION NODES NODE AT SELECTION "border_nodes" DATA LIST "REFERENCE_BORDER_COORDINATE" END ITEM FROM 1 TO 1 ;

3.10 Creep Analysis Related Commands

The following section describes commands used for creep analysis. See also &CREEP_MATERIAL, &CREEP_ANALYSIS_PARAMS and &CREEP_STEP_DEFINITION sub-commands.

3.10.1 The Command &RETARDATION

The command is used to define retardation times for approximation of material creep compliance function by Dirichlet series. Coefficients of the approximation are set either by the Least Square Method, the case of using DISCRETE [SPECTRUM] keywords, or by Inverse Laplace Transformation, i.e. the case of CONTINUOUS [SPECTRUM]. By continuous is meant ATENA will use continuous rather then discrete retardation spectrum. By default, discrete approach is preferred. The 3rd derivation of the compliance function is employed to compute the Inverse Laplace Transformation. The retardation times will be generated from *time_start* to *time_end* (both inclusive) so that there will be *ndecl_retard* points evenly distributed at $\log_{10}$ time span. The exact meaning of these parameters slightly differs for the case of discrete and continuous approach. It is explained in more details in the ATENA theoretical manual. By default, it is generated one retardation time per $\log_{10}$ days. Note that it is not possible to carry on the analysis beyond *time_end* and it is not possible re-generate the retardation times later in the analysis, because it would result in serious inaccuracy of compliance function approximation.

Syntax:

&RETARDATION_TIMES:
RETARDATION [TIMES] [FOR] [EXECUTION] [{DISCRETE | CONTINUOUS}]
[SPECTRUM] [TIME[S]] FROM *time_start* TO *time_end*
RETARD_TIMES_PER_DECADE *ndecl_retard*

3.10.2 The command &HISTORY_IMPORT

The command forces ATENA to import data about humidity and temperature history at structural nodes that were before hand computed by CCStructuresTransport ATENA's execution module.

Syntax:

&HISTORY_IMPORT:
HISTORY { [IMPORT [GEOMETRY *geometry_filename*] | [RESULTS] *results_filename*]$_2$
| [NUMBER] | [OF] | [INTERVALS] | [FOR] | HUMIDITY *num_int_hum* |
TEMPERATURE *num_int_temp* | HUMIDITY_ABS_MAX_ERROR *err1* |
HUMIDITY_REL_MAX_ERROR *err2* | TEMPERATURE_ABS_MAX_ERROR *err3* |
TEMPERATURE_REL_MAX_ERROR *err4* | TIME_UNITS "*time_units*"}$_+$

Table 141: &HISTORY_IMPORT command parameters

Parameter	Description
results_filename	Name of binary file with the history. It must be the same as that specified for HISTORY EXPORT command in the CCStructuresTransport module. It should be enclosed in double quote character (“).
geometry_filename	Name of binary file with geometry of the imported model. It must be the same as that specified for HISTORY EXPORT command in the CCStructuresTransport module. It should be enclosed in double quote character (“). If omitted, identical imported and current models are

	assumed.
num_int_hum	Number of intervals into which nodal humidities at each time step should be sorted. By default *num_int_hum*=1.
num_int_temp	Number of intervals into which nodal temperatures at each time step should be sorted. By default *num_int_temp*=1.
HUMIDITY_ABS_MAX_ERROR *err1* | HUMIDITY_REL_MAX_ERROR *err2* | TEMPERATURE_ABS_MAX_ERROR *err3* | TEMPERATURE_REL_MAX_ERROR *err4*	Relative and absolute humidity and temperature "errors" that are considered as negligible. The values are used during mapping of moisture and humidity histories at structural material points. If the tested and master values differ less than as it is required by these maximum "errors", than no new history is created and the tested material point is mapped towards the master material point. By default, these "errors" are set to 0.1.
TIME_UNITS "*time_units*"	The TIME_UNITS "*time_units*" allows to specify, which time units were used to calculate and write the transpored analysis results in the file *results_file_name*. It is specified in the same way as in the Unit command. By default no time unit conversion is made.

3.11 Dynamic Analysis Related Commands

Dynamic analysis of structures has been developed in an engineering module CCStructuresDynamic. Hence, /M CCStructuresDynamic switch must be specified on the ATENA command line, in order to invoke the correct execution module.

The included eigenvalues and eigenvectors analysis is available in any engineering module derived for CCStructures, i.e. CCStructures, CCStructuresCreep and CCStructuresDynamic.

In general, the module CCStructuresDynamic is (similarly to CCStructuresCreep) an extension of the module CCStructures, from which it inherits many common services and input commands. Other services and input commands are borrowed from CCStructuresCreep and CCStructuresTransport modules.

The aim of this section is to describe additional input command that are specific for dynamic analysis and to point out small modification of the commands existing in other engineering modules to serve purposes of dynamic analyses.

3.11.1 Finite element and material model related data

Most structural finite element and any structural material available for static analysis can be used also for dynamic analysis. Of course, unlike in statics, dynamic analysis needs proper value of material density, i.e. the RHO parameter.

3.11.2 Dynamic initial values of state variables

The initial structural accelerations and velocities at finite nodes are set in a similar way to their specification within CCStructuresTransport module. By default, zero initial accelerations and velocities at nodes are assumed.

The nodal initial conditions can be set by the input command &DYNAMIC_INITIAL_CONDITIONS:

Syntax:

```
&DYNAMIC_INITIAL_CONDITIONS:
NODAL {ACCEL_VEL | VEL_ACCEL | ACCELERATION | VELOCITY }
      [SETTINGS] { &MANUAL_INITIAL_VALUES_ENTRY |
      &GENERATED_INITIAL_VALUES }

&MANUAL_INITIAL_VALUES_ENTRY:
{ NODE n VEL vel_x vel_y [vel_z] | ACCEL accel_x accel_y [accel_z] }
```

Table 142: Nodal Initial Conditions Definition (manual entries)

Sub-Command	Description
NODE *n*	Set initial conditions for node *n*.
VEL *vel_x vel_y* [*vel_z*]	Specify initial nodal velocities in direction of global coordinates. 3D problems need 3 values, 2D problems only two values..
ACCEL *accel_x accel_y* [*accel_z*]	Input initial nodal acceleration in similar way as the above initial velocities input.

&GENERATED_INITIAL_VALUES:

```
NODAL [SETTING] SELECTION "selection_name"  |  { { CONST const_vector |
      COEFF_X  coeff_x_vector | COEFF_Y coeff_y_vector | COEFF_Z coeff_z_vector
      }+ | { EQN "eqn"  } } {GENERATE_ACCEL  | GENERATE_VEL} }+
```

Table 143: Nodal Initial Conditions Definition (generated entries)

Sub-Command	Description
SELECTION "*selection_name*"	Name of selection, for which the generation is requested.
{GENERATE_ACCEL | GENERATE_VEL } CONST *const_vector* COEFF_X *coeff_x_vector* | COEFF_Y *coeff_y_vector* |	Keyword for entities to be generated. The values in global structural directions are generated as linear combination:

COEFF_Z *coeff_z_vecor* EQN "*eqn*"	$value_x = const(1) + x\ coeff_x(1) + y\ coeff_y(1) + z\ coeff_z(1)$ $value_y = const(2) + x\ coeff_x(2) + y\ coeff_y(2) + z\ coeff_z(2)$ $value_z = const(3) + x\ coeff_x(3) + y\ coeff_y(3) + z\ coeff_z(3)$ *x,y,z* are coordinates of nodes, where the generation is processed. The vecor of values, e.g. *const_vector* must include 3 or 2 values for 2D or 3D problems, respectively. Alternatively, the initial values can be input directly by "*eqn*", where it is also possible to use placeholders for the above *x,y,z* are coordinates of nodes.

Example:

NODAL VEL_ACCEL SETTING NODE 1 VEL 0.0030 0. 0. ACCEL -0.005370861556 0. 0.

NODAL VEL_ACCEL SELECTION "*my_selection*"

CONST 0.0030 0. 0. COEFF_X 0. 0. 0. COEFF_Y -0.6523648649 0. 0. COEFF_Z 0. 0. 0.1 GENERATE_VEL
CONST -0.005370861556 0. 0. COEFF_X 0. 0. 0. COEFF_Y 0. 0.1 0. COEFF_Z 0. 0. 0. GENERATE_ ACCEL.

NODAL VEL_ACCEL SETTING SELECTION "all_nodes"

EQNS "0." "-3.461644974+1.730822487*cos(0.3683608667*x)+1.493980135*exp(-0.3683608667*x)+0.2368423513*exp(0.3683608667*x)+2.277285924*sin(0.3683608667*x)" GENERATE_VEL // Set initial velocities

3.11.3 CCStructuresDynamic Set parameters

The standard SET parameters specified via the &ANALYSIS_TYPE, subcommand &TRANSIENT, are dynamic analysis extended. For more details see the enhanced version of the subcommand, i.e. &TRANSIENT.

Table 144: &ANALYSIS_TYPE sub-command parameters

Parameter	Description
&TRANSIENT	Set transient analysis and set some parameters for it.

Syntax:

&TRANSIENT:
TRANSIENT { [TIME] CURRENT *x* | [TIME] INCREMENT *x* | STOP_TIME *execution_stop_time* | LAST_TIME *last_time* | NEWMARK_METHOD | HUGHES_ALPHA_METHOD | } | NEWMARK BETA *x* | NEWMARK_GAMMA *x* | HUGHES_ALPHA *x* | DAMPING STIFFNESS [COEFFICIENT] *x* | DAMPING MASS [COEFFICIENT] *x* }+

Table 145: ANALYSIS_TYPE subcommands for the transport analysis

Parameter	Description
[TIME] CURRENT *x*	Sets current time.
[TIME] INCREMENT *x*	Sets time increment in steps.
STOP_TIME *execution_stop_time*	Time at which the execution should stop.
LAST_TIME *last_time*	Set the final time of the analysis.
NEWMARK_METHOD | HUGHES_ALPHA_METHOD	Dynamic analysis method to be used. Default value: HUGHES_ALPHA_METHOD
NEWMARK BETA *x* | NEWMARK_GAMMA *x* | HUGHES_ALPHA *x*	Defines the Newmark's β parameter, the Newmark's γ parameter and the Hughes α damping parameter. By default these parameters are 0.35, 0.6 and -0.05 respectively.
DAMPING STIFFNESS [COEFFICIENT] *x*	Defines stiffness matrix coefficient for proportional damping. E.g.: DAMPING STIFFNESS COEFFICIENT *0.8* Default value: 0
DAMPING MASS [COEFFICIENT] *x*	Defines mass matrix coefficient for proportional damping. E.g.: DAMPING MASS COEFFICIENT *0.8* Default value: 0

3.11.4 Step definition

Definition of the STEP within dynamic analysis is analogous to the definition for creep step, see &CREEP_STEP_DEFINITION. The only difference is that instead of "TYPE CREEP" you will know use "TYPE DYNAMIC".

3.11.5 Lumped masses

Structural lumped masses are input as a specification of loading case. They are input in the same way as concentrated loads; only LUMPED_MASSES keyword must be used, see simple support, see &LOAD_FORCES.

3.11.6 Eigenvalue and eigenvectors analysis

The analysis of structural eigenvalues and eigenvectors is available in any engineering module derived from CCStructures, Currently it comprises modules CCStructure, CCStructureCreep and, of course, CCStructuresDynamic. It uses Inverse subspace iteration methods to find a specified number of the lowest eigenvalues and eigenvectors of the structure.

There are few new SET &EIGENVALUES parameters as described below, see &SET, subparameter &ANALYSIS_TYPE

Table 146: &Eigenvalue Set sub-command parameters

Parameter	Description
&EIGENVALUES	Set some parametyers for eigenvalues analysis.

Syntax:

& EIGENVALUES:
{ NUMBER_OF_EIGENVALS *n* | MAX_EIGENVAL_ERROR *r* |
MAX_NUMBER_OF_SSPACE_ITERATIONS *n* |
REQUEST_STURM_SEQUENCE_CHECK {YES | NO} |
MAX_NUMBER_OF_JACOBI_ITERATIONS *n* | NUMBER_OF_PROJ_VECS *n* | SHIFT_EIGENVALUES *shift*, }+

Table 147: The eigenvalue analysis SET parameters

Parameter	Description
NUMBER_OF_EIGENVALS *n*	Sets number of the lowest eigenmodes that should be calculated. Default value: 10
MAX_EIGENVAL_ERROR *r*	Maximum eigenvalues error that is tolerated. Default value: 1.E-6
MAX_NUMBER_OF_SSPACE_ITERATIONS *n*	Max. number of subspace iterations. Default value: 16
STURM_SEQUENCE_CHECK {YES | NO}	Flag for requesting Sturm check that no eigenvalue got missed during the solution. This check is supported only by the direct skyline solver. Using of a sparse matrix solver will turn down eventual request for the Sturm check.
MAX_NUMBER_OF_JACOBI_ITERATIONS *n*	Max. number of iteration within Jacobi. The Jacobi procedure computes eigenmodes of the projected global eigenvalues problem via minimization of Rayleigh quotient. Hence, within each (“main”) iteration of inverse subspace iteration method another iterating process is executed in Jacobi. The value of *n* sets maximum number of these iterations that are allowed. Default value: 12
NUMBER_OF_PROJ_VECS *n*	Defines number of projection vector used by Rayleigh quotient method. It must be equal or bigger than the number of required eigenvalues. Default value: min(2**n_eigenvals, eigenvals*+8), where *n_eigenvals* is the number of required eigenvalues.

DAMPING STIFFNESS [COEFFICIENT] *x*	Defines stiffness matrix coefficient for proportional damping. E.g.: DAMPING STIFFNESS COEFFICIENT *0.8*
DAMPING MASS [COEFFICIENT] *x*	Defines mass matrix coefficient for proportional damping. E.g.: DAMPING MASS COEFFICIENT *0.8*
SHIFT_EIGENVALUES *shift*	Value by which the structural eigenvalues should be shifted. (Eigenvalue is 2nd power of structural circular eigenfrequency).
NORMALIZE_EIGENVECTORS {YES | NO}	Flag for request to normalize eigenvectors during iterations. Although this normalizing is source of a small CPU time overhead, it is recommended, because it improves numerical stability of the eigenmode analysis.

3.11.7 Eigenvalues and eigenvectors analysis execution command

Eigenvectors and eigenmodes analysis is executed by the following commands:

Syntax:

&EIGENVECTORS &STATIC_STEP_DEFINITION

Static step definition defines structural boundary Dirichlet conditons and is the same as for the case of static analysis.

3.11.8 Sample input data for transient dynamic analysis

The following lines are an example of input data to analyze a cantilever subject to harmonic concentrated load at its free end. The structure is modeled by a few shell elements. It has a proportional damping.

```
// Forced Vibration Analysis of a Spring Mass System (see vynucene_kmitani.mws)
// with proportional dumping
//
// 3 nonlinear shells + 4th shell as lumped mass at the end
//
// -for a finer analysis, change e.g. SET TRANSIENT TIME INCREMENT 0.02
// -for Nemark method, change eg. SET TRANSIENT HUGHES ALPHA -0.00 (or uncomment/comment the relevant lines)

TASK name "Test Ahmad elems"
dimension 3

//------------------------------------------------------------------------\\
//                    Material definition                                 \\
```

```
//------------------------------------------------------------------------\\

MATERIAL
 id   1
 name "Spring"
 type "CC3DElastIsotropic"
 E  30
 Mu       0.00
 Rho  0.000000000001
 Alpha  1.200E-05

 MATERIAL
 id   2
 name "Spring"
 type "CC3DElastIsotropic"
 E  30000000
 Mu       0.00
 Rho  156.
 Alpha  1.200E-05

//------------------------------------------------------------------------\\
//                      Element type definition                           \\
//------------------------------------------------------------------------\\

ELEMENT TYPE
 id   1
 name "1D Truss"
 type "CCAhmadElement33L9"

//------------------------------------------------------------------------\\
//                       Geometry definition                              \\
//------------------------------------------------------------------------\\

GEOMETRY ID 1 Name "Spring" TYPE "LayeredShell"
   SOLID
```

```
    LAYER 1  MATERIAL 1 THICKNESS 0.2
    LAYER 2  MATERIAL 1 THICKNESS 0.2
    LAYER 3  MATERIAL 1 THICKNESS 0.2
    LAYER 4  MATERIAL 1 THICKNESS 0.2
    LAYER 5  MATERIAL 1 THICKNESS 0.2
    LAYER 6  MATERIAL 1 THICKNESS 0.2
    LAYER 7  MATERIAL 1 THICKNESS 0.2
    LAYER 8  MATERIAL 1 THICKNESS 0.2
    LAYER 9  MATERIAL 1 THICKNESS 0.2
    LAYER 10 MATERIAL 1 THICKNESS 0.2

//-------------------------------------------------------------------------\\
//                    Joint coordinates definition                         \\
//-------------------------------------------------------------------------\\

JOINT COORDINATES


    1  0.00e+000  0.00e+000  1.0000000
    2  0.00e+000  0.5000000  1.0000000
    3  0.00e+000  1.0000000  1.0000000
    4  0.00e+000  0.00e+000  0.5000000
    5  0.00e+000  1.0000000  0.5000000
    6  0.00e+000  0.00e+000  0.00e+000
    7  0.00e+000  0.5000000  0.00e+000
    8  0.00e+000  1.0000000  0.00e+000
    9  0.5000000  0.00e+000  1.0000000
   10  0.5000000  1.0000000  1.0000000
   11  0.5000000  0.00e+000  0.00e+000
   12  0.5000000  1.0000000  0.00e+000
   13  1.0000000  0.00e+000  1.0000000
   14  1.0000000  0.5000000  1.0000000
   15  1.0000000  1.0000000  1.0000000
   16  1.0000000  0.00e+000  0.5000000
   17  1.0000000  1.0000000  0.5000000
```

18	1.0000000	0.00e+000	0.00e+000
19	1.0000000	0.5000000	0.00e+000
20	1.0000000	1.0000000	0.00e+000
21	1.5000000	0.00e+000	1.0000000
22	1.5000000	1.0000000	1.0000000
23	1.5000000	0.00e+000	0.00e+000
24	1.5000000	1.0000000	0.00e+000
25	2.0000000	0.00e+000	1.0000000
26	2.0000000	0.5000000	1.0000000
27	2.0000000	1.0000000	1.0000000
28	2.0000000	0.00e+000	0.5000000
29	2.0000000	1.0000000	0.5000000
30	2.0000000	0.00e+000	0.00e+000
31	2.0000000	0.5000000	0.00e+000
32	2.0000000	1.0000000	0.00e+000
33	2.5000000	0.00e+000	1.0000000
34	2.5000000	1.0000000	1.0000000
35	2.5000000	0.00e+000	0.00e+000
36	2.5000000	1.0000000	0.00e+000
37	2.9500000	0.00e+000	1.0000000
38	2.9500000	0.5000000	1.0000000
39	2.9500000	1.0000000	1.0000000
40	2.9500000	0.00e+000	0.5000000
41	2.9500000	1.0000000	0.5000000
42	2.9500000	0.00e+000	0.00e+000
43	2.9500000	0.5000000	0.00e+000
44	2.9500000	1.0000000	0.00e+000
45	2.9750000	0.00e+000	1.0000000
46	2.9750000	1.0000000	1.0000000
47	2.9750000	0.00e+000	0.00e+000
48	2.9750000	1.0000000	0.00e+000
49	3.0000000	0.00e+000	1.0000000
50	3.0000000	0.5000000	1.0000000
51	3.0000000	1.0000000	1.0000000
52	3.0000000	0.00e+000	0.5000000

```
  53  3.0000000  1.0000000  0.5000000
  54  3.0000000  0.00e+000  0.00e+000
  55  3.0000000  0.5000000  0.00e+000
  56  3.0000000  1.0000000  0.00e+000

//-----------------------------------------------------------------------\\
//                    Element group definition                           \\
//-----------------------------------------------------------------------\\

ELEMENT GROUP
 id   1
 name "Spring"
 type 1
 material 1
 geometry 1
ELEMENT INCIDENCES
   1     1    13    15     3     6    18    20     8     9    14    10     2    11    19    12     7     4    16    17
5
   2    13    25    27    15    18    30    32    20    21    26    22    14    23    31    24    19    16    28
29    17
   3    25    37    39    27    30    42    44    32    33    38    34    26    35    43    36    31    28    40
41    29
ELEMENT GROUP
 id   2
 name "Mass"
 type 1
 material 2

 geometry 1
ELEMENT INCIDENCES
   1    37    49    51    39    42    54    56    44    45    50    46    38    47    55    48    43    40    52
53    41



ELEMENT TYPE ID 1 PREPARE_CALCULATION
```

```
// Load case No.1
LOAD CASE
 id   1
 name "Permanent supports"
// Joint support

SUPPORT SIMPLE node   6  dof  1  value  0.0
SUPPORT SIMPLE node   6  dof  2  value  0.0
SUPPORT SIMPLE node   6  dof  3  value  0.0

SUPPORT SIMPLE node   4  dof  1  value  0.0
SUPPORT SIMPLE node   4  dof  2  value  0.0
SUPPORT SIMPLE node   1  dof  1  value  0.0
SUPPORT SIMPLE node   1  dof  2  value  0.0

SUPPORT SIMPLE node   7  dof  1  value  0.0
SUPPORT SIMPLE node   7  dof  3  value  0.0
SUPPORT SIMPLE node   8  dof  1  value  0.0
SUPPORT SIMPLE node   8  dof  3  value  0.0

SUPPORT SIMPLE node   5  dof  1  value  0.0
SUPPORT SIMPLE node   3  dof  1  value  0.0
SUPPORT SIMPLE node   2  dof  1  value  0.0

SUPPORT COMPLEX  master 49 1 * 1.0 slave 50 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 51 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 52 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 53 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 54 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 55 1
SUPPORT COMPLEX  master 49 1 * 1.0 slave 56 1


// Load case No.2
```

```
LOAD CASE
 id   2
 name "Concetrated force"
LOAD SIMPLE node 49 dof  1  value   0.25
LOAD SIMPLE node 51 dof  1  value   0.25
LOAD SIMPLE node 54 dof  1  value   0.25
LOAD SIMPLE node 56 dof  1  value   0.25

NODAL SETTING
node 49  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 50  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 51  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 52  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 55  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 54  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 55  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 56  vel 0.0030 0. 0. accel -0.005370861556 0. 0.

node 45  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 46  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 47  vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 48  vel 0.0030 0. 0. accel -0.005370861556 0. 0.

node 37 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 38 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 39 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 40 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 41 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 42 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 43 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 44 vel 0.0030 0. 0. accel -0.005370861556 0. 0.

node 1000007 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
node 1000008 vel 0.0030 0. 0. accel -0.005370861556 0. 0.
```

```
//-------------------------------------------------------------------------\\
//                      Options and switches                               \\
//-------------------------------------------------------------------------\\

// Parameters Solution Parameters
SET Static
SET Newton-Raphson
SET Iteration Limit 20
SET Displacement Error 0.010
SET Residual Error 0.010
SET Absolute Residual Error 0.010
SET Energy Error 0.010
SET STOP_TIME 3.5 LAST_TIME 3.5
SET TRANSIENT TIME CURRENT 0. INCREMENT 0.1

SET TRANSIENT HUGHES BETA 0.2505 GAMMA 0.5 ALPHA -0.05 DAMPING MASS
COEFFICIENT 1.789 STIFFNESS COEFFICIENT 0.
//SET TRANSIENT HUGHES BETA 0.2505 GAMMA 0.5 ALPHA -0.05 DAMPING MASS
COEFFICIENT 0. STIFFNESS COEFFICIENT 0.1396
SET HUGHES_ALPHA_METHOD

//SET TRANSIENT NEWMARK BETA 0.2505 GAMMA 0.5      DAMPING MASS
COEFFICIENT 1.789 STIFFNESS COEFFICIENT 0.
////SET TRANSIENT NEWMARK BETA 0.2505 GAMMA 0.5  DAMPING MASS
COEFFICIENT 0. STIFFNESS COEFFICIENT 0.1396
//SET NEMARK_METHOD

OUTPUT MONITOR_2 NAME "displ_node_1_X" EACH STEP LOCATION NODES Node
FROM 49 TO 56 BY 1
  DATA LIST "DISPLACEMENTS" END ITEM FROM 1 TO 1 ;

OUTPUT MONITOR_2 NAME "force_node_1_X" EACH STEP LOCATION NODES Node
FROM 49 TO 56 BY 1
  DATA LIST "PARTIAL_INTERNAL_FORCES" END ITEM FROM 1 TO 1 ;



//-------------------------------------------------------------------------\\
```

```
//                    Executing                    \\
//-------------------------------------------------------------------------\\

STEP id  1 TYPE DYNAMIC name "Load No.  1" AT  0.0    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.001094800003
STEP id  2 TYPE DYNAMIC name "Load No.  2" AT  0.1    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.001077716015
STEP id  3 TYPE DYNAMIC name "Load No.  3" AT  0.2    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.001043814628
STEP id  4 TYPE DYNAMIC name "Load No.  4" AT  0.3    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.000993624865
STEP id  5 TYPE DYNAMIC name "Load No.  5" AT  0.4    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.000927929917
STEP id  6 TYPE DYNAMIC name "Load No.  6" AT  0.5    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.847754933E-3
STEP id  7 TYPE DYNAMIC name "Load No.  7" AT  0.6    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.754351018E-3
STEP id  8 TYPE DYNAMIC name "Load No.  8" AT  0.7    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.649175706E-3
STEP id  9 TYPE DYNAMIC name "Load No.  9" AT  0.8    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.533870226E-3
STEP id 10 TYPE DYNAMIC name "Load No. 10" AT  0.9    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.410233878E-3
STEP id 11 TYPE DYNAMIC name "Load No. 11" AT  1.0    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.280195968E-3
STEP id 12 TYPE DYNAMIC name "Load No. 12" AT  1.1    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.145785694E-3
STEP id 13 TYPE DYNAMIC name "Load No. 13" AT  1.2    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * 0.9100483E-5
STEP id 14 TYPE DYNAMIC name "Load No. 14" AT  1.3    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.127726738E-3
STEP id 15 TYPE DYNAMIC name "Load No. 15" AT  1.4    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.262560826E-3
STEP id 16 TYPE DYNAMIC name "Load No. 16" AT  1.5    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.393297741E-3
STEP id 17 TYPE DYNAMIC name "Load No. 17" AT  1.6    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.517897375E-3
STEP id 18 TYPE DYNAMIC name "Load No. 18" AT  1.7    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.634415394E-3
STEP id 19 TYPE DYNAMIC name "Load No. 19" AT  1.8    LOAD CASE  FIXED  1 * 1.0
INCREMENT 2 * -0.741033573E-3
```

STEP id 20 TYPE DYNAMIC name "Load No. 20" AT 1.9 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.836088172E-3

STEP id 21 TYPE DYNAMIC name "Load No. 21" AT 2.0 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.918095893E-3

STEP id 22 TYPE DYNAMIC name "Load No. 22" AT 2.1 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.985777035E-3

STEP id 23 TYPE DYNAMIC name "Load No. 23" AT 2.2 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1038075457E-2

STEP id 24 TYPE DYNAMIC name "Load No. 24" AT 2.3 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1074175059E-2

STEP id 25 TYPE DYNAMIC name "Load No. 25" AT 2.4 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1093512517E-2

STEP id 26 TYPE DYNAMIC name "Load No. 26" AT 2.5 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1095786078E-2

STEP id 27 TYPE DYNAMIC name "Load No. 27" AT 2.6 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1080960265E-2

STEP id 28 TYPE DYNAMIC name "Load No. 28" AT 2.7 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1049266428E-2

STEP id 29 TYPE DYNAMIC name "Load No. 29" AT 2.8 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.1001199139E-2

STEP id 30 TYPE DYNAMIC name "Load No. 30" AT 2.9 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.937508473E-3

STEP id 31 TYPE DYNAMIC name "Load No. 31" AT 3.0 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.859188300E-3

STEP id 32 TYPE DYNAMIC name "Load No. 32" AT 3.1 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.767460782E-3

STEP id 33 TYPE DYNAMIC name "Load No. 33" AT 3.2 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.663757294E-3

STEP id 34 TYPE DYNAMIC name "Load No. 34" AT 3.3 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.549696096E-3

STEP id 35 TYPE DYNAMIC name "Load No. 35" AT 3.4 LOAD CASE FIXED 1 * 1.0 INCREMENT 2 * -0.427057074E-3

step id 1 execute

step id 2 execute

step id 3 execute

step id 4 execute

step id 5 execute

step id 6 execute

step id 7 execute

```
step id  8 execute
step id  9 execute
step id 10 execute
step id 11 execute
step id 12 execute
step id 13 execute
step id 14 execute
step id 15 execute
step id 16 execute
step id 17 execute
step id 18 execute
step id 19 execute
step id 20 execute
step id 21 execute
step id 22 execute
step id 23 execute
step id 24 execute
step id 25 execute
step id 26 execute
step id 27 execute
step id 28 execute
step id 29 execute
step id 30 execute
step id 31 execute
step id 32 execute
step id 33 execute
step id 34 execute
step id 35 execute

/* end of file */
```

3.11.9 Sample input data for eigenvalues and eigenvectors analysis

The following as an example of input data for eigenvalue analysis of the structure from the previous section.

```
// Eigenvalue analysis
//
// A cantilever modelled by 4 nonlinear shells
// Cross sectional dimension width=height=1; length=40
//
// Exact solution: (see
c:\AtenaExamples\Examples\Dynamics\SpringWithLumpedMass\Eigenvalues\cantilever.mw
s )
//
//    f1=0.0443Hz
//    f2=0.278Hz
//    f3=0.775Hz
//
// Calculated:
//
//    f1=0.0445Hz
//    f2=0.299Hz
//    f3=0.945Hz



TASK name "Test Ahmad elems"
dimension 3

//------------------------------------------------------------------------\\
//                        Material definition                             \\
//------------------------------------------------------------------------\\

MATERIAL
 id   1
 name "Spring"
 type "CC3DElastIsotropic"
 E  30000000
 Mu       0.00
 Rho  156.
 Alpha  1.200E-05
```

```
//-----------------------------------------------------------------------\\
//                    Element type definition                            \\
//-----------------------------------------------------------------------\\

ELEMENT TYPE
 id   1
 name "1D Truss"
 type "CCAhmadElement33L9"

//-----------------------------------------------------------------------\\
//                      Geometry definition                              \\
//-----------------------------------------------------------------------\\

GEOMETRY ID 1 Name "Spring" TYPE "LayeredShell"
  SOLID
    LAYER 1  MATERIAL 1 THICKNESS 0.2
    LAYER 2  MATERIAL 1 THICKNESS 0.2
    LAYER 3  MATERIAL 1 THICKNESS 0.2
    LAYER 4  MATERIAL 1 THICKNESS 0.2
    LAYER 5  MATERIAL 1 THICKNESS 0.2
    LAYER 6  MATERIAL 1 THICKNESS 0.2
    LAYER 7  MATERIAL 1 THICKNESS 0.2
    LAYER 8  MATERIAL 1 THICKNESS 0.2
    LAYER 9  MATERIAL 1 THICKNESS 0.2
    LAYER 10 MATERIAL 1 THICKNESS 0.2

//-----------------------------------------------------------------------\\
//                  Joint coordinates definition                         \\
//-----------------------------------------------------------------------\\

JOINT COORDINATES


   1  00.0e+000  0.00e+000  1.0000000
```

2 00.0e+000 0.5000000 1.0000000
3 00.0e+000 1.0000000 1.0000000
4 00.0e+000 0.00e+000 0.5000000
5 00.0e+000 1.0000000 0.5000000
6 00.0e+000 0.00e+000 0.00e+000
7 00.0e+000 0.5000000 0.00e+000
8 00.0e+000 1.0000000 0.00e+000
9 05.000000 0.00e+000 1.0000000
10 05.000000 1.0000000 1.0000000
11 05.000000 0.00e+000 0.00e+000
12 05.000000 1.0000000 0.00e+000
13 10.000000 0.00e+000 1.0000000
14 10.000000 0.5000000 1.0000000
15 10.000000 1.0000000 1.0000000
16 10.000000 0.00e+000 0.5000000
17 10.000000 1.0000000 0.5000000
18 10.000000 0.00e+000 0.00e+000
19 10.000000 0.5000000 0.00e+000
20 10.000000 1.0000000 0.00e+000
21 15.000000 0.00e+000 1.0000000
22 15.000000 1.0000000 1.0000000
23 15.000000 0.00e+000 0.00e+000
24 15.000000 1.0000000 0.00e+000
25 20.000000 0.00e+000 1.0000000
26 20.000000 0.5000000 1.0000000
27 20.000000 1.0000000 1.0000000
28 20.000000 0.00e+000 0.5000000
29 20.000000 1.0000000 0.5000000
30 20.000000 0.00e+000 0.00e+000
31 20.000000 0.5000000 0.00e+000
32 20.000000 1.0000000 0.00e+000
33 25.000000 0.00e+000 1.0000000
34 25.000000 1.0000000 1.0000000
35 25.000000 0.00e+000 0.00e+000
36 25.000000 1.0000000 0.00e+000

```
  37  30.000000  0.00e+000  1.0000000
  38  30.000000  0.5000000  1.0000000
  39  30.000000  1.0000000  1.0000000
  40  30.000000  0.00e+000  0.5000000
  41  30.000000  1.0000000  0.5000000
  42  30.000000  0.00e+000  0.00e+000
  43  30.000000  0.5000000  0.00e+000
  44  30.000000  1.0000000  0.00e+000
  45  35.000000  0.00e+000  1.0000000
  46  35.000000  1.0000000  1.0000000
  47  35.000000  0.00e+000  0.00e+000
  48  35.000000  1.0000000  0.00e+000
  49  40.000000  0.00e+000  1.0000000
  50  40.000000  0.5000000  1.0000000
  51  40.000000  1.0000000  1.0000000
  52  40.000000  0.00e+000  0.5000000
  53  40.000000  1.0000000  0.5000000
  54  40.000000  0.00e+000  0.00e+000
  55  40.000000  0.5000000  0.00e+000
  56  40.000000  1.0000000  0.00e+000

//------------------------------------------------------------------------\\
//                      Element group definition                          \\
//------------------------------------------------------------------------\\

ELEMENT GROUP
 id   1
 name "Spring"
 type 1
 material 1
 geometry 1
ELEMENT INCIDENCES
   1     1    13    15     3     6    18    20     8     9    14    10     2    11    19    12     7     4    16    17
5
```

```
   2    13   25   27   15   18   30   32   20   21   26   22   14   23   31   24   19   16   28
29   17
   3    25   37   39   27   30   42   44   32   33   38   34   26   35   43   36   31   28   40
41   29
   4    37   49   51   39   42   54   56   44   45   50   46   38   47   55   48   43   40   52
53   41

ELEMENT TYPE ID 1 PREPARE_CALCULATION



// Load case No.1
LOAD CASE
 id   1
 name "Permanent supports"
// Joint support

SUPPORT SIMPLE node   6  dof  1  value  0.0
SUPPORT SIMPLE node   6  dof  2  value  0.0
SUPPORT SIMPLE node   6  dof  3  value  0.0

SUPPORT SIMPLE node   4  dof  1  value  0.0
SUPPORT SIMPLE node   4  dof  2  value  0.0
SUPPORT SIMPLE node   1  dof  1  value  0.0
SUPPORT SIMPLE node   1  dof  2  value  0.0

SUPPORT SIMPLE node   7  dof  1  value  0.0
SUPPORT SIMPLE node   7  dof  3  value  0.0
SUPPORT SIMPLE node   8  dof  1  value  0.0
SUPPORT SIMPLE node   8  dof  3  value  0.0

SUPPORT SIMPLE node   5  dof  1  value  0.0
SUPPORT SIMPLE node   3  dof  1  value  0.0
SUPPORT SIMPLE node   2  dof  1  value  0.0

//-------------------------------------------------------------------------\\
//                        Options and switches                             \\
```

```
//-----------------------------------------------------------------------\\

// Parameters for dynamic analysis

 SET NUMBER_OF_EIGENVALS 5
 SET MAX_EIGENVAL_ERROR 0.0001
 SET MAX_NUMBER_OF_SSPACE_ITERATIONS 14
 SET REQUEST_STURM_SEQUENCE_CHECK YES
 SET MAX_NUMBER_OF_JACOBI_ITERATIONS 10
 SET NUMBER_OF_PROJ_VECS 15

 //SET solver ICCG
 SET Optimize width Sloan

//-----------------------------------------------------------------------\\
//                          Executing                                    \\
//-----------------------------------------------------------------------\\

EIGENVECTORS  LOAD CASE  1 * 1.0

// STEP ID 1 STATIC NAME "BCs and load" LOAD CASE 1 * 1.0  EXECUTE

/* end of file */
```

3.12 Miscellaneous Commands

3.12.1 The Command &FUNCTION

This command defines an x-y relationship that can be referred to by other commands, when a law or function needs to be specified.

Syntax:

```
   &FUNCTION:
   FUNCTION  ID n   NAME "name"   TYPE  { &MULTILINEAR_FUNCTION_DATA
          |   &ANALYTIC_ FUNCTION_DATA | &PYTHON_FUNCTION_DATS  }
           [ { OUTPUT_X | OUTPUT_Y | OUTPUT_INTEGRATE_Y |
          OUTPUT_DERIVATE_Y  | OUTPUT_NONE  |
           OUTPUT_MIN_X  output_min_x  | OUTPUT_MAX_X  output_max_x |
          OUTPUT_INCR_X  output_incr_x  |
           OUTPUT_SUFFIX "suffix_string"  |
```

OUTPUT_FFT_Y | OUTPUT_FFT_ERR_Y |
OUTPUT_FFT_SPECT_A | OUTPUT_FFT_SPECT_T |
OUTPUT_FFT_SPECT_O |
OUTPUT_FILTERED_FFT_Y | OUTPUT_FILTERED _FFT_ERR_Y |
OUTPUT_FILTERED _FFT_SPECT_A | OUTPUT_FILTERED_FFT_SPECT_T
| OUTPUT_FILTERED _FFT_SPECT_O |
OUTPUT_SIEVED_X | OUTPUT_SIEVED_Y] $_+$

Currently the multilinear and analytic types are supported:

&MULTILINEAR_FUNCTION_DATA:
"CCMultiLinearFunction" [ALLOC_POINTS *dim*]
{ XVALUES { { x_i }$_+$ | RETRIEVE *"output_data_X"* }
YVALUES { { y_i }$_+$ | RETRIEVE *"output_data_Y"* } }|
GENERATE &ANALYTIC_FUNCTION_DATA }
[{ REMOVE_ALL | REMOVE_ALL_FILTERS | REMOVE_ALL_GENERATED |
{ ADD_OMEGA_FILTER *omega_min omega_max*] }$_n$ |
SIEVE_Y | SIEVE_FFT_Y | SIEVE_FILTERED_FFT_Y |
COPY_FFT_Y | COPY_FILTERED_FFT_Y | COPY_SIEVED |
FFT | X_DOWN x_l | X_UP x_h | N_MAX n_{max} | DX_APPROX *dx* |
PRINT_Y PRINT_FFT_Y | PRINT_FILTERED_FFT_Y |
PRINT_COEFFS |PRINT_FILTERED_COEFFS |PRINT_SIEVED_POINTS }] $_+$

[ALLOC_POINTS *dim*] allocates space for *dim* additional points. It does not effect the actual number of input points. It merely speeds up input of the subsequent *dim* points by preparing beforehand a cumulative storage they will need.

&ANALYTIC_FUNCTION_DATA:
"CCAnalyticFunction" Y_EQN "*y_string*" [X_MIN *x_min*] [X_MAX *x_max*]
[DX *dx*]

where "*y_string*" contains string with agebraic expression of argument *x*, *x_min, x_max* is min/max value of *x*, *dx* is used to calulate numerical integral or derivative of the function. By default, *dx*=1.E-5, *x_min*=- *x_max*=-1.E20

Example: TYPE "CCAnalyticFunction" Y_EQN "-1.*12.56^2*sin(12.56*x)"

&PYTHON_FUNCTION_DATA:
"CCPythoncFunction" MODULE "module_name" FUNCTION "function_name"
[X_MIN *x_min*] [X_MAX *x_max*] [DX *dx*]
where "module_name" and "function_name" is module and function name to be used for the calculation. The actual module and function must be input by the PYTHON command. The remaining parameters are the same as before.

The optional intput OUTPUT_X | OUTPUT_Y | OUTPUT_INTEGRATE_Y | OUTPUT_DERIVATE_Y | OUTPUT_NONE | OUTPUT_MIN_X *output_min_x* | OUTPUT_MAX_X *output_max_x* | OUTPUT_INCR_X *output_incr_x* | OUTPUT_SUFFIX *"suffix_string"* is for printing and plotting of X, Y and other values of the specified function. Upon issuing this sub-command, Atena creates a new output in OUTPUT_DATA category. The name of the output is assembled as "FUNC_*n_type_suffix_string*". *n*, *type*, *suffix_string* are respectively function *id*, one of X, Y, INTEGRATE_Y, DERIVATE_Y depending on OUTPUT_... request and user defined output name suffix. The function is derivated with respect to X and integrated with respect to X within *min_val_x* and *x* . If *incr_val_x* is specified, the requested function values are printed for *min_val_x, min_val_x+ incr_val_x*, *min_val_x+ 2*incr_val_x*,.... *max_val_x*. Otherwise the values are printed only at definition points that falls into interval *min_val_x... max_val_x*. More output requests can be issued within one FUNCTION command. In case of redefining, i.e. recreating FUNC_*n_type_suffix_string* output it is sometimes necessary to set on recalculate flag within the OUTPUT command to print the actual data, i.e. use command OUTPUT .... LOCATION OUTPUT_DATA DATA LIST " FUNC_*n_type_suffix_string*" END ... RECALCULATE.
The input FFT | X_DOWN x_l | X_UP x_h | N_MAX n_{max} | DX_APPROX *dx* is for executing Fourier spectral analysis. It approximates the input function by Fourier series, (i.e. FFT) from x_l to x_h , whereby it applies *dx* shift in the FFT approximation functions. The results can by printed by tne commands PRINT_Y | PRINT_FFT_Y |

PRINT_COEFFS and output by OUTPUT_Y | OUTPUT_FFT_Y | OUTPUT_FFT_ERR_Y | OUTPUT_FFT_SPECT_A | OUTPUT_FFT_SPECT_T | OUTPUT_FFT_SPECT_O. Such analysis calculates approximation of the "Y" values by Fourier serie, (i.e. OUTPUT_FFT_Y), relative errors of the approximation, (i.e. OUTPUT_FFT_ERR_Y) and a graph with results of actual spectral analysis of the function.. Its horizontal axis shows circular frequencies or periods of the FFT approximation frequencies, (i.e. OUTPUT_FFT_SPECT_T or OUTPUT_FFT_SPECT_O) and their magnitudes of excitation are depicted on vertical axis of the graph, (i.e. OUTPUT_FFT_SPECT_A).
The FFT approximation uses by default full modal spectrum, however, it can be filtered by the commands $\{$ ADD_OMEGA_FILTER *omega_min omega_max*] $\}_n$. The results are output by OUTPUT_FILTERED_FFT_Y | OUTPUT_FILTERED _FFT_ERR_Y | OUTPUT_FILTERED _FFT_SPECT_A | OUTPUT_FILTERED_FFT_SPECT_T | OUTPUT_FILTERED _FFT_SPECT_O and printed by PRINT_FILTERED_FFT_Y |PRINT_ FILTERED_COEFFS.

All specified or generated {X,Y} data pairs can be deleted by REMOVE_ALL subcommand. All generated data are deleted by REMOVE_ALL_GENERATED. All filteres are released by REMOVE_ALL_FILTERS.

The optional SIEVE_Y | SIEVE_FFT_Y | SIEVE_FILTERED_FFT_Y | MAX_ERROR *x* performs sieving of respective points. It attempts removing as much points as possible but still preserving accuracy given by maximum error *x*. The error is defined as relative error and is by default 0.001.

Finaly, the results from FFT and/or SIEVE can overwrite the original X,Y data. It is achived by the commands COPY_FFT_Y | COPY_FILTERED_FFT_Y | COPY_SIEVED.

Use command OUTPUT PLOT..... to define horizontal and vertical series that can be later plotted in Atena 2D graph window.

Example:

Create output series x and $\int_0^{200} y\,dx$ for a multilinear function id 500, (note that the function must be defined beforehand). The new output data FNC_500_X_REDEFINED and FNC_500_INTEGRATE_Y_REDEFINED are created by command:

```
FUNCTION id 500
MIN_VAL_X 0 MAX_VAL_X 200 INCR_VAL_X 10 OUTPUT_SUFFIX "_REDEFINED"
OUTPUT_X OUTPUT_INTEGRATE_Y
```

The series can be plotted using commands:

```
OUTPUT PLOT_2 NAME "new_plot1_fnc_500_X" EACH STEP LOCATION
OUTPUT_DATA DATA LIST "FNC_500_X_REDEFINED" END ;
OUTPUT PLOT_2 NAME "new_plot1_fnc_500_INTEGRATE_Y" EACH STEP
LOCATION OUTPUT_DATA DATA LIST "FNC_500_INTEGRATE_Y_REDEFINED"
END ;
```

Example of spectral analysis of a function:

```
# Generate a function
FUNCTION id   1  name "Test multilinear function"
 type "CCMultiLinearFunction"
 generate X_MIN 0. X_MAX 3.14 DX 0.157 Y_EQN "0.5+sin(5*x+1.)"  ;
 # Generate output data for the FFT analysis
 FUNCTION id   1
 OUTPUT_X OUTPUT_Y  OUTPUT_SUFFIX "_SUFF"
OUTPUT_FFT_Y OUTPUT_FFT_ERR_Y OUTPUT_FFT_SPECT_A
OUTPUT_FFT_SPECT_T OUTPUT_FFT_SPECT_O
 X_DOWN 0.2 X_UP 3. DX_APPROX -0.1 N_MAX 100 PRINT_COEFFS PRINT_POINTS
FFT     ;
# Define monitors to allow for 2D graphical presentation of the results
OUTPUT  PLOT_2 NAME "fnc_1_X" EACH STEP LOCATION OUTPUT_DATA DATA
LIST "FNC_1_X_SUFF" END ;
OUTPUT  PLOT_2 NAME "fnc_1_Y" EACH STEP LOCATION OUTPUT_DATA DATA
LIST "FNC_1_Y_SUFF" END ;
OUTPUT  PLOT_2 NAME "fnc_1_FFT_Y" EACH STEP LOCATION OUTPUT_DATA
DATA LIST "FNC_1_FFT_Y_SUFF" END ;
OUTPUT  PLOT_2 NAME "fnc_1_FFT_ERR_Y" EACH STEP LOCATION
OUTPUT_DATA DATA LIST "FNC_1_FFT_ERR_Y_SUFF" END ;
OUTPUT  PLOT_2 NAME "fnc_1_SPECT_A" EACH STEP LOCATION OUTPUT_DATA
DATA LIST "FNC_1_SPECT_A_SUFF" END ;
OUTPUT  PLOT_2 NAME "fnc_1_SPECT_T" EACH STEP LOCATION OUTPUT_DATA
DATA LIST "FNC_1_SPECT_T_SUFF" END ;
```

OUTPUT PLOT_2 NAME "fnc_1_SPECT_O" EACH STEP LOCATION OUTPUT_DATA DATA LIST "FNC_1_SPECT_O_SUFF" END ;

FUNCTION id 1 PRINT_POINTS sieve MAX_ERROR 0.0005 ;

Note that in order to visualize these plots, (using Atena's Graph Series dialog) don't forget to check the "Values'profile for fixed time" checkbox and set horizontal and vertical fixed time to zero, see description of the PLOT output option.

3.12.2 The Command &PRE-CRACK♥

Syntax:

PRE-CRACK ELEMENT GROUP *n* ELEMENT *n* INTEGRATION [POINT] *n* NORMAL x_1 x_2 [x_3]

Table 148: &PRE-CRACK command parameters ♥

Parameter	Description
ELEMENT GROUP *n*	Element group id in which the pre-defined crack is to be inserted.
ELEMENT *n*	Element id in which the pre defined crack is to be inserted.
INTEGRATION [JOINT] *n*	Integration point id in which the pre defined crack is to be inserted. This is an optional parameter, if it is not specified, crack is inserted into all integration points.
NORMAL x_1 x_2 [x_3]	Crack normal direction.

3.12.3 The Command &DELETE

Syntax:

&DELETE:
DELETE [ENFORCED] [{ [{ ELEMENT { GROUP [ID] *n* [ELEMENT [ID] *n*] | LIST *"multi_list"* | TYPE [ID] *n* } } | GEOMETRY [ID] *n* | JOINT { [ID] *n* | LIST "*list*" } | LOAD [CASE] [ID] *n* | MATERIAL [ID] *n* | STEP [ID] *n* | FUNCTION [ID] *n* }+]

Table 149: &DELETE command parameters

Parameter	Description
ELEMENT GROUP [ID] *n* [[ELEMENT [ID] *n*]	Delete element group from the model or a single element from the specified element group. E.g. ELEMENT GROUP *3* [ELEMENT *4*]
ELEMENT LIST	Delete elements from the list. The list contains pairs {

"multi_list"	group_id; element_id }+ and can be created e.g. by Selection combine/separate commands.
ELEMENT TYPE [ID] *n*	Delete element type from the list of element type definitions. E.g. ELEMENT TYPE *2*
GEOMETRY [ID] *n*	Delete geometry from the model. E.g. GEOMETRY *6*
JOINT [ID] *n* | LIST *"list"*	Delete joint from the model. Alternatively delete nodes from the list. E.g. JOINT *3* ; JOINT LIST "INTERNAL_NODES" …
LOAD CASE [ID] *n*	Delete load case from the model. E.g. LOAD CASE *4*
MATERIAL [ID] *n*	Delete material from the list of material types. E.g. MATERIAL *20*
STEP [ID] *n*	Delete step *n* from the model. E.g. STEP *4*
FUNCTION [ID] *n*	Delete function from the model E.g. FUNCTION *5*
ENFORCED	If not specified, all references to a deleted entity remain valid even after the deletion, thereby it is possible later to re-input the entity with new data. Otherwise, the entity and all references to it get unconditionally removed.

3.12.4 The Command &MACRO_DELETE

Syntax:

&MACRO_DELETE:
MACRO_DELETE [ENFORCED] [{ [{ MACRO_ELEMENT [ID] *n* |
MACRO_NODE [ID] *n* }+]

Table 150: &DELETE command parameters

Parameter	Description
MACRO_ELEMENT	Delete macro element from the model. E.g. MACRO_ELEMENT *3*
MACRO_NODE	Delete macro node from the model E.g. MACRO_NODE *3*
ENFORCED	If not specified, all references to a deleted entity remain valid even after the deletion, thereby it is possible later to re-input the entity with new data. Otherwise, the entity and all references to

	it get unconditionally removed.

3.12.5 The Command &INPUT

Syntax:
&INPUT:
INPUT { FILE_OPEN | FILE_CLOSE } "*file name*"

Table 151: &INPUT FILE sub-command parameters

The command specifies the name of the input file to be opened or closed. Following this command the ATENA input stream will be redirected into this file or the default stream. E.g. INPUT FILE "*file name*"

3.12.6 The Command &MESSAGE

Syntax:
&MESSAGE:
MESSAGE { FILE_OPEN | FILE_CLOSE } "*file name*" [{ overwite | append }]

Table 152: &MESSAGE FILE command parameters

This command specifies the name of the message file to be opened or closed. All messages following this command will be redirected to this file or the default stream. The open file is overwrited or appended. E.g. MESSAGE FILE "*file name*" [{ overwite	append }]

3.12.7 The Command &ERROR

Syntax:
&ERROR:
ERROR { FILE_OPEN | FILE_CLOSE } "*file name*"" [{ overwite | append }]

Table 153: &ERROR FILE command parameters

This command specifies the name of the error file to be opened or closed. All errors following this command will be redirected to this file or the default stream. The open file is overwrited or appended. E.g. ERROR FILE "*file name*"

3.12.8 The Command &RESTORE

Syntax:
&RESTORE:
RESTORE FROM "*file name*"

Table 154: &RESTORE command parameters

This command reads the finite element model state from the given binary file name. The content of the finite element model is overwritten by the file contents. E.g. RESTORE FROM "*file name*"

3.12.9 The Command &STORE

Syntax:
&STORE:
STORE TO "*file name*" [EACH *n* [{STEP|STEPS}|{SUBSTEP|SUBSTEPS}]

Table 155: &STORE command parameters

This command writes the finite element model state to a binary file. It can write immediately, e.g. STORE TO "*file name*", or it can autimatically serialize each *n*-th, e.g. STORE TO "*file name*" EACH *n* STEPS, or it can carry out the serialization each step and *m*-th substeps, e.g. STORE TO "*file name*" EACH *m* SUBSTEPS, (for dynamic and creep analyses only). In the case of automatic serialization by steps the filename is appended by "*.step_id*". The serialization by substeps appends the file name by "*_substep_id.step_id*". If *n*==0, then it the automatic serialization is stopped.

3.12.10 The Command &PUSHOVER_ANALYSIS

An usual static analysis can be accompanied by the Pushover analysis as advocated in Eurocode. In this case the structure is loaded incrementally and its load-displacement diagram is recorded. After each step the pushover analysis is carried out (using the recorded LD

diagram) and if the criteria of the pushover analysis are met, any additional loading, (i.e. subsequent load steps) are ignored.

Syntax:
&PUSHOVER_ANALYSIS:
PUSHOVER_ANALYSIS { IS_ACTIVE *n* | MONITOR_ID *n* | FORCE_MONITOR_NAME *"name"* | FORCE_ITEM_ID *n* | DISPLS_MONITOR_NAME *"name"* | DISPLS_ITEM_ID *n* | GAMMA_FACTOR_D *x* | GAMMA_FACTOR_F *x* | GAMMA_FACTOR *x* | MASS_NORM *x* | MASS *x* | PERIOD_T_B *x* | PERIOD_T_C *x* | PERIOD_T_D *x* | ETA_FACTOR *x* | BETA0 x | SOIL_FACTOR *x* | ACCEL_GROUND *x* | ACCEL_GROUND_D *x* | P_D *x* | P_F *x* | EXT_P_F *x* | PO_STOP_IF_ULS_AND_DLS_FLAG *n* | PO_STOP_ONLY_IF_UNSTABLE_FLAG *n* | STOREY_NODES_IDS { *n* }+ | VERTICAL_AXIS_ID *n* | HORIZONTAL_AXIS_ID *n* | STOREY_DLS_COEFF *x* | EXECUTE}n

Table 156: &PUSHOVER_ANALYSIS command parameters

IS_ACTIVE *n*	If *n*=1, carry out pushover analysis at the end of execution of each CCStructures's step. If the Eurodoce requirements are met, the STOP_FLAG (see below) is set to 1 and any subsequent STEP ..EXECUTE command is ignored. The analysis can resume, only if STOP_FLAG is manually set to 0. Units: none Default: 0
MONITOR_ID *n*	Id of a monitor, where LD diagram from the analysis is stored. It can be 1 or 2 to utilize output monitor 1 or 2. Units: none Default: 1
FORCE_MONITOR_NAME *"name"*	Name of the monitor to record forces (used in the LD diagram). Units: none Default: "LD_DIAGRAM_VALUE_Y"
FORCE_ITEM_ID *n*	Item number used by the above. Units: none Default: 1
DISPLS_MONITOR_NAME *"name"*	Name of the monitor to record displacementss (used in the LD diagram). Units: none Default: "LD_DIAGRAM_VALUE_X"
DISPLS_ITEM_ID *n*	Item number used by the above.

	Units: none Default: 1
GAMMA_FACTOR_D *x*	Tansformation factor for deformations between MDOF and SDOF, (called Gamma in Eurocode) Units: none Default: 1.
GAMMA_FACTOR_F *x*	Tansformation factor for forces between MDOF and SDOF, (called Gamma in Eurocode) Units: none Default: 1.
GAMMA_FACTOR *x*	Tansformation factor for forces and deformations between MDOF and SDOF, (called Gamma in Eurocode). Supported for compatibility reasons. Now replaced by GAMMA_FACTOR_D and GAMMA_FACTOR_F Units: none Default: 1.
MASS_NORM *x*	Equivalent mass of SDOF, (called m_star in Eurocode) Units: weight, (e.g. kg) Default: 1
MASS *x*	Equivalent mass of MDOF, (used e.g. by Romanian Building Code) Units: weight, (e.g. kg) Default: 1
PERIOD_T_B *x*	Time period T_b from Eurocode, (called T_b in Eurocode) Units: time Default: 0
PERIOD_T_C *x*	Time period T_c from Eurocode, (called T_b in Eurocode) Units: time Default: 0
PERIOD_T_D *x*	Time period T_b from Eurocode, (called T_d in Eurocode)

	Units: time Default: 0
ETA_FACTOR *x*	Damping correction factor from Eurocode, , (called eta in Eurocode) Units: time Default: 1, (i.e 5. % of viscous damping)
BETA0 x	Dynamic amplification factor to calculate elastic response spectrum Se(T). Units: none Default: 2.5
SOIL_FACTOR *x*	Soil factor from Eurocode, (called S in Eurocode) Units: time Default: 0
ACCEL_GROUND *x*	Ground acceleration, (ULS), (called a_g in Eurocode) Units: length/time^2 Default: 0
ACCEL_GROUND_D *x*	Ground acceleration, (DLS), (called a_Dg in Eurocode) Units: length/time^2 Default: 0
P_D *x*	Relative displacement stopping value, (called p_d in Eurocode) Units: none Default: 1.5
P_F *x*	Relative force drop down coefficient to violate PO ULS criterion, (called p_f in Eurocode). Units: none Default: 0.8
EXT_P_F *x*	Relative force drop down coefficient to declare the analysis unstable and stop the execution. Units: none. Default: 0.2
PO_STOP_ONLY_IF_UNSTABLE_FLA	If *n*=1 the analysis continues until the stability

G *n*	criterion is failed (irrespective of the pushover analysis status). If *n*=0, the pushover analysis is completed based on the pushoover analysis status and the flag PO_STOP_IF_ULS_AND_DLS_FLAG. Default: *n*=0
PO_STOP_IF_ULS_AND_DLS_FLAG *n*	If *n*=1, the pushover analysis is completed after both ULS and DLS criteria are met. If *n*=0, to complete the analysis it suffices to fulfill only the ULS critera. Default: *n*=0
STOREY_NODES_IDS { *n* }+	List of node ids for all floors fo the structure. The nodes must be input sorted from the ground to the heigest floor. If an id *n=0*, then the associated displacement are assumed zero. (It is typically used for gound floor). If the structure has *m* stories, *m+1* node ids are expected. If node node ids are input, DLS check in the Pushover analysis is skipped. Note: For expert users only. Others are discouraged to input this parameter. Atena maintains this parameter automatically and no intervention from the user is needed. Units: none Default: none Example: 0 249 693
VERTICAL_AXIS_ID *n*	Id of model axis to be considered vertical, i.e. axis, where gravity load applies. Units: none Default: 3, (i.e. Z axis)
HORIZONTAL_AXIS_ID *n*	Id of model axis, where the ground acceleration is applied. Units: none Default: 1, (i.e. X axis)
STOREY_DLS_COEFF *x*	Coefficient $coeff_{DLS}$ to calculate maximum interstory drift: $d_r \le coeff_{DLS}\, h$. h is height of store and d_r is relative storey drift. Units: none

	Default: 0.005
EXECUTE	Carry out pushover analysis immediately. (By default, this command is not needed, as the analysis is calculated automatically at the end of execution of each load step).

3.12.11 Static initial values of state variables

The initial structural state variables at finite nodes are set in a similar way to their specification within CCStructuresTransport module. At the moment, this approach can be used to set only nodal reference temperature and humidity in the structure but it is expected to extend in the future. The nodal initial conditions can be set by the input command &STATIC_INITIAL_CONDITIONS:

Syntax:

```
&STATIC_INITIAL_CONDITIONS:
NODAL {HUMIDITY | TEMPERATURE} [SETTINGS]
      { &STATIC_MANUAL_INITIAL_VALUES_ENTRY |
      &STATIC_GENERATED_INITIAL_VALUES }+

& STATIC_MANUAL_INITIAL_VALUES_ENTRY:
{ [{BASE_HUMIDITY | BASE_TEMPERATURE} base_value ] [NODE n
      {HUMIDITY |TEMPERATURE} nodal_value] }
```

Table 157: Static Nodal Initial Conditions Definition (manual entries)

Sub-Command	Description
NODE *n*	Set initial conditions for node *n*.
{HUMIDITY|TEMPERATURE} *base_value*	Specify initial nodal humidity / temperature for node *n*. This value is added to the base humidity / temperature below. Units: [-]/[/T] Default: 0.
{BASE_HUMIDITY | BASE_TEMPERATURE} *nodal_value*	Initial base temperature. This value is used for all nodes of the structure. Units: [-]/[T] Default: 0.

& STATIC_GENERATED_INITIAL_VALUES:

NODAL [SETTING] SELECTION "*selection_name*" | CONST *const* | COEFF_X *coeff_x* | COEFF_Y *coeff_y* | COEFF_Z *coeff* | {GENERATE_HUMIDITY|GENERATE_TEMP }

Table 158: Static Nodal Initial Conditions Definition (generated entries)

Sub-Command	Description
SELECTION "*selection_name*"	Name of selection, for which the generation is requested.
CONST *const* COEFF_X *coeff_x* | COEFF_Y *coeff_y* | COEFF_Z *coeff_z* G {GENERATE_HUMIDITY| GENERATE_TEMP}	Generate reference humidity/temperature for nodes in the selection "*selection_name*" . The values are generated as linear combination: $temperature = base\_temp + const + x\,coeff_x + y\,coeff_y + z\,coeff_z$ where *x,y,z* are coordinates of nodes of nodes in the selection. Units: COEFF_F, COEFF_M, COEFF_Z: [-/L]/[T/L] CONST: [-]/[T] Default: all constants are set to zero.

Note that initial reference humidity and or temperatures can be set also by applying element humidity / temperature load that import humidity / temperature history from a previous transport analysis of the structure. In this case the reference nodal humidity / tepleratures corresponds to structural conditions at reference time of the first applied element humidity / temperature load. As such values typically represent actual real humidities / temperatures in the structure, the input described in this paragraph is not needed, (actually humidity / temperatures from element temperature load would be added to humidities / temperatures from the command &STATIC_INITIAL_CONDITIONS).

Example:

// initials for temperatures

NODAL SETTING
NODE i TEMPERATURE temp

NODAL SETTING SELECTION "all_nodes"
CONST 25. COEFF_X 0.1 COEFF_Y -0.6523 COEFF_Z 0.8
GENERATE_TEMPERATURE

NODAL SETTING
BASE_TEMPERATURE base_temp // this value is added to specific node temperature,

3.13 Preprocessor commands

The following section describes ATENA commands for the ATENA native preprocessor to generate FE models. These include mainly commands for running T3D preprocessor and commands for generating reinforcement bars through the analysed structure.

Syntax:

&PREPROCESS:
{&T3D_SPEC | &T3D_EXPAND | &T3D_FIT_NURBS | &MACRO_JOINT |
&MACRO_ELEMENT_SPEC | &UPDATE_ELEMENT_AGE }

3.13.1 The Command &T3D_SPEC

T3D FEM mesh generator has been incorporated into ATENA. It is a powerful 3D generator for generating nodes and elements of a FE model. All the T3D related commands must be enclosed between **T3D_GENERATE** and **T3D_END** or **T3D_GENERATE** and **RETURN** ATENA input commands. The main idea of the generation is to define macro nodes, macro lines, patches etc. that are subsequently used to generate 3D regions. Patch and surface type domains are supported as well. The current implementation of the generator can also be used to generate lists of nodes, see command &SELECTION . Such list is then simply used for definition of Dirichlet and Von Neumann boundary conditions, see subcommands &LOAD_PLACE and &LOAD_VALUE (commands &LOAD_DISPLACEMENT, &LOAD_FORCE).

All T3D related commands are described in a separate PDF document.

The T3D command line options, see Chapter 7 of T3D documentation, should follow

T3D_GENERATE command. They must not change in all subsequent call T3D_GENERATE command.

The following are new features of T3D that have not been yet documented in it:

3.13.1.1 The NODEPROP / ELEMPROP parameter

Commands **CURVE, SURFACE, PATCH, SHELL** and **REGION** can now include additional parameters:

NODEPROP ‘nodeprop’
ELEMPROP ‘elemprop’

In similar way, the command **VERTEX** can additionally include:

NODEPROP ‘nodeprop’

The parameter NODEPROP and/or ELEMPROP is used to generate the above mentioned selection lists. Such a list is given name *‘nodeprop’* resp. *‘elemprop’* (notice use of single quote ‘ instead of usual double quote “ !) and it will contain identification ids of all internal FE nodes, resp. elements that were used to generate the T3D entity with the additional parameters. Specify the parameters NODEPROP and ELEMPROP also for boundary entities, (such as for surfaces of T3D region), if the generated list should include also boundary nodes and elements of the T3D entity.

3.13.1.2 The subcommand RETURN

There is a new T3D command **RETURN**. It is similar to **T3D_END** in that it forces command parser to return from T3D back to ATENA. However, **T3D_END** generates FE mesh before it returns, whilst **RETURN** does not. Use the command **RETURN** to specify T3D commands that (for some reason) are mixed with ATENA commands.

3.13.1.3 The parameter ELEMGROUP

The commands **CURVE, SURFACE, PATCH, SHELL** and **REGION** can include additional parameter **ELEMGROUP.** The syntax is as follows

CURVE *curve_id* …… ELEMGROUP *truss_group_id* ….
SURFACE *surface_id* …..ELEMGROUP *triangle_group_id quad_group_id*…
PATCH *patch_id* ….ELEMGROUP .*triangle_group_id quad_group_id*…
SHELL *shell_id* ….ELEMGROUP *triangle_group_id quad_group_id*…
REGION *region_id* … ELEMGROUP *tetra_group_id pyram_group_id*
wedge_group_id hexa_group_id

The parameter has to be used in order to say to ATENA, what element group should be used for the generated elements. As T3D generator is capable of generating mixed type FE mesh, i.e. a mesh of several element types, and as (in ATENA) one element group can contain only one element type, it is necessary to input for 2D T3D entities two element groups, one for triangle and the other for quadrilateral elements and similarly four element groups for 3D T3D regions, (tetrahedra, pyramids, wedges and hexahedra (i.e. bricks)).

Note that model id, i.e. id from a T3D command will probably differ from generated FEM entity id. For example vertex id will probably differ from generated FEM node id at the same location. This is particularly the case, if T3D is used also for optimisation of solution matrix band.

3.13.1.4 The subcommand REMOVE

T3D command REMOVE removes entity and all dependent entities dependent on it from the model. The command syntax is:

REMOVE {VERTEX *vertex_id* | CURVE *curve_id* | SURFACE *surface_id* | PATCH *patch_id* | SHELL *shell_id* | REGION *region_id* || ALL }

Use of the above new T3D commands and subcommands is demonstrated in the enclosed sample AtenaWin analyses.

3.13.1.5 The parameter EQUIDISTANT

The keyword equidistant ensures equidistant distribution of finite elements within an entity. It can be used for any entity with exception of vertices, e.g. curve, surface, region etc. Except for curves, the equidistant property is only applicable for an entity, which is created via a procedure of mapping. For curves, it is applicable subject to no vertices are fixed to that curve. To alleviate this restriction, create a copy of the curve, split it to more curves (already without a fixed vertex) and fixed them to the original curve. Note that the EQUIDISTANT property is automatically propagated to all neighboring entities.

Example:

surface 11 curve 102 100 103 12 equidistant

The subcommand EQUIDISTANT can also be used for unstructured meshes. In this case, however, no curve with the EQUIDISTANT property is allowed to have fixed vertices and splitting of a copied curve (as described above) will help.

Note also, that the EQUIDISTANT is not always 100% accurate, especially in case of a higher order meshes.

3.13.1.6 The subcommand OUTPUT

The subcommand OUTPUT is used to explicitly control, whether a generated entity should be output (to ATENA), or not. It works in the same way as the OUTPUT parameter from entity definitions.

Its main use is to allow editing of FE data from the T3D generator. Suppose you have a T3D model that has been already used to generate a FE model into ATENA and you need to edit that model. The model has been serialized. The procedure of editing the model would be as follows:

1/ Restore the original model.

2/ Go back to T3D.

3/ Using OUTPUT commands suppress output (from T3D to ATENA) of all entities that didn't change.

4/ Re-define the edited entities.

3/ Re-generate the whole model (and output all the changes into ATENA).

Syntax:

OUTPUT { YES | NO} { Vertex | CURVE | ….. | REGION } *entity_id*

3.13.1.7 The subcommand SLAVE

The subcommand SLAVE allows connecting of two overlapping surfaces (or neighboring curves and nodes). Its use is rather simple: define the first entity of the pair in a usual way. Define the second entity of the pair and include the keyword SLAVE in its definition.

Note that SLAVE is applied only for internal joints, therefore SLAVE must be specified also for all boundary entities and their subentities up to level of boundary vertices. It behaves in exactly the same way as ELEMPRO and NODEPROP keywords.

Example:

curve 100 vertex 101 104 slave

Only vertices with nearly the same coordinates get connected. The "same" property is judged based on 1% octree mesh size. Octree is a special technique by which the 3D space around

the model is subdivided into brick shaped regions in order to facilitate faster searching methods. It works for both structured and unstructured meshes. An error message is produced and the generation is terminated, if for a SLAVE node no master node is found.

3.13.2 The command T3D_FIT_NURBS

This command is used to optimize parameters for T3D curves and surfaces to obtain the best fit of their points. The procedure of modelling is as follows: Start T3D generator as usually and define all T3D curves and surfaces. Estimate all unknown T3D NURBS parameters. Exit T3D by RETURN command, so that no finite elements are generated. For each optimized NURBS execute T3F_FIT_NURBS to optimize values of their parameters. Start T3D again and complete the T3D model definition and exit by T3D_END. It will generate finite nodes and elements of the FE model using the updated T3D NURBS' parameters.

T3D_FIT_NURBS command optimizes primarily coordinates of the NURBS's control points. If requested, it can also optimize weights of the control points and knot vector values. At the moment it cannot change number of control points and knot vector dimension.

&T3D_FIT_NURBS :

T3D_FIT_NURBS {CURVE | SURFACE} *nurbs_id* [OPTIMIZE] [AGRESSIVE] [CTRL_POINTS] [WEIGHTS] [KNOTS] [RESET] { POINTS *num_points* { {u | v} *coord_uv* XYZ { *coord_x coord_y coord_z* }$_{num\_points}$ }$_n$

Table 159: T3D_FIT_NURBS command parameters

{CURVE | SURFACE} *nurbs_id*	Optimize T3D curve or surface with identification *nurbs_id.*
POINTS *num_points*	Use the following *num_points* curve/surface points for the optimization.
{u | v} *coord_uv*	For surfaces only ! Set {u | v} *coord_uv* of a line of surface points. Note that complementing coordinates {v | u} are calculaed automatically. Several lines can be input, some in u, other in v direction.
XYZ { *coord_x coord_y coord_z* }$_{num\_points}$ }	Cartesian coordinates of the curve/surface points.
[OPTIMIZE] [CTRL_POINTS] [WEIGHTS] [KNOTS] [RESET]	By default only the control points' coordinates are optimized. The CTRL_POINTS, WEIGHTS and KNOTS keywords allows to set explicitely what to optimize (i.e. fit). The RESET keywords clears all optimize requests. The OPTIMIZE keyward has no special function. It is included only to improve readeness of the command.

[AGRESSIVE]	By default, only the NURB's internal points are optimized. By using AGRESSIVE keyword ATENA will optimize all the points.

Example:

```
T3D_fit_NURBS
surface 3
points 7
v 0.3
xyz
0.897058815224913 3.98823524913495 0.
0.411764693923875 3.94705875875432 0.
u 0.6
xyz
0.6536585362962522 3.32926826337002 0.
2.57692308970414  4.38461531005917 0.
3.53235294217301   3.70588229896194 0.
v 0.8
xyz
2.76206897862664 3.90517235604934 0.
3.24137932922235 4.40344819271106 0.

T3D_fit_NURBS
curve 10
points 5
xyz
 3.94567705313272e-002    2.96741080965606    1.43577379680776e-002
 0.821474616564413        5.58497313672873    3.90681992068942e-003
 1.5                 6.8             4.e-003
 3.1681394677438          5.58619089453943    5.06473440571477e-003
 3.9435700552385          2.96658986175115    1.9541001617481e-002
```

3.13.3 The command T3D_EXPAND_SELECTIONS

The command is used to compile regular and expanded selection lists with finite elements and nodes for a particular geometrical entity by T3D generator. These lists are used to connect a geometrical T3D model with an associated (T3D generated) finite element model.

The regular selection lists includes only nodes or elements within the entity and outside its boundary. They are created automatically during the mesh generation by T3D and they are using an actual setting of &T3D_EXPAND_SETTINGS during the generation. The expanded selection lists are regular selection lists expanded by adding nodes and elements on

boundaries of the appropriate entity. They are created by commands &T3D_EXPAND_SETTINGS after the T3D mesh generation, i.e. in time, when the regular lists are available. The command T3D_EXPAND_SELECTIONS ALL expands all currently define curves, surfaces, shells, patches and regions.

Syntax:
&T3D_EXPAND:

T3D_EXPAND_SELECTIONS { [&T3D_EXPAND_SETTINGS] }+ {
[&T3D_EXPAND_ENTITY] }+

&T3D_EXPAND_SETTINGS :

[PROP_GENERATION {NONE | SEMIATOMATIC | AUTOMATIC }] |
[EXPAND_SUFFIX "*expand_str*"] | [GROUP_SUFFIX *"group_str*"] |
[DEF_VERTEX_FMT_FOR_NODES "*vertex_fmt*"] |
[DEF_MNODE_FMT_FOR_NODES "*mnode_fmt*"] |
[DEF_CURVE_FMT_FOR_NODES "*curve_fmt*"] |
[DEF_PATCH_FMT_FOR_NODES "*patch_fmt*"] |
[DEF_SURFACE_FMT_FOR_NODES "*surface_fmt*"] |
[DEF_SHELL_FMT_FOR_NODES "*shell_fmt*"] |
[DEF_REGION_FMT_FOR_NODES "*region_fmt*"] |
[DEF_MELEMENT_FMT_FOR_NODES "*melement_fmt*"] |
[DEF_BAR_REINFORCEMENT_FMT_FOR_NODES "*rc_fmt*"] |
[DEF_BAR_REINFORCEMENT_FMT_FOR_PRINCIPAL_NODES "*prc_fmt*"]
| [DEF_CURVE_FMT_FOR_ELEMENTS "*curve_fmt*"] |
[DEF_PATCH_FMT_FOR_ELEMENTS "*patch_fmt*"] |
[DEF_SURFACE_FMT_FOR_ELEMENTS "*surface_fmt*"] |
[DEF_SHELL_FMT_FOR_ELEMENTS "*shell_fmt*"] |
[DEF_REGION_FMT_FOR_ELEMENTS "*region_fmt*"] |
[DEF_MELEMENT_FMT_FOR_ELEMENTS "*melement_fmt*"] |
[DEF_BAR_REINFORCEMENT_FMT_FOR_ELEMENTS "*rc_fmt*"]]

&T3D_EXPAND_ENTITY:

{ALL| {{CURVE | SURFACE | SHELL | PATCH | REGION } *entity_id*1} }+

Table 160: &T3D_EXPAND_SELECTIONS command parameters

PROP_GENERATION {NONE | SEMIATOMATIC | AUTOMATIC }	Specify mode for creation selection lists of finite nodes and finite elements that are associated with geometrical entities like vertex, curve etc. NONE means that no expanded lists are created, (i.e. a commands akin &T3D_EXPAND_SETTINGS are ignored) and regular selection lists are created only, if NODEPROP or ELEMPROP param is explicitly defined.

	SEMIAUTOMATIC means that regular and expanded selection lists are created only, if NODEPROP or ELEMPROP param is explicitly. In case of vertices, the NODEPROP param need not be explicitly set. In that case the automated name generation is invoked using DEF_VERTEX_FMT_FOR_NODES. AUTOMATIC mode forces to do the same as the SEMIATOMATIC mode does, but it also creates additional set of lists using the automated name generation. This mode is used to automatically create selection lists of finite nodes and elements for all geometrical entities used in the T3D model, (e.g. vertices, curves etc.)
EXPAND_SUFFIX "*expand_str*"	Defines suffix string. All subsequently compiled names of expanded selection lists will be given names that equal the original (T3D) selection lists' names appended by "*expand*_str". Default: "_&T" Example: "_Expanded". In this case, e.g. an original selection list name "Curve_1" will expand to "Curve_1_Expanded.
GROUP_SUFFIX *"group_str*"	Defines suffix string. All subsequently compiled names of selection lists with elements ids will be accompanied also by selection lists with group ids and they will be given names that equal the original (T3D) element ids selection list appended by "*group_str*" . Default: "_&G" Example: "_AssocGroups". In this case, e.g. an original selection list name "Curve_1" will expand to "Curve_1_AssocGroups.
DEF_VERTEX_FMT_FOR_NODES "*vertex_fmt*"	Defines formatting string akin the "C" language *printf*(…) function. All subsequently T3D generated names of selection lists that includes list of nodes associated with vertices will be assigned a name that equal to str.Format("*vertex_fmt*", vertex_id). If a vertex has got explicitly specified the nodeprop parameter, the associated selection list will be given that name. The above applies for PROP_GENERATION=NONE and PROP_GENERATION=SEMIAUTOMATIC. If

	PROP_GENERATION equals to AUTOMATIC, then the nodeprop is ignored, (or reserved) and DEF_VERTEX_FMT_FOR_NODES "*vertex_fmt*" definition is used instead. Default: "NV%i" Example: “NVertex%i”. In this case, e.g. all finite nodes associated with a vertex 13 will be listed in a selection list that calls NVertex13.
DEF_MNODE_FMT_FOR_NODES "*mnode_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for macro nodes. Default: "NMN%i" Example: “NMacroNode%i”. In this case, e.g. all finite nodes associated with a macro node 13 will be listed in a selection list that calls NMacroNode13.
DEF_CURVE_FMT_FOR_NODES "*curve_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for curves. Default: "NC%i" Example: “NCurve%i”. In this case, e.g. all finite nodes associated with a curve 13 will be listed in a selection list that calls NCurve13.
DEF_PATCH_FMT_FOR_NODES "*patch_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for patches. Default: "NP%i" Example: “NPatch%i”. In this case, e.g. all finite nodes associated with a patch 13 will be listed in a selection list that calls NPatch13.
DEF_SURFACE_FMT_FOR_NODES "*surface_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for surfaces. Default: "NS%i" Example: “NSurface%i”. In this case, e.g. all finite nodes associated with a surface 13 will be listed in a selection list that calls NSurface13.

DEF_SHELL_FMT_FOR_NODES "*shell_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for shells. Default: "NH%i" Example: “NShell%i”. In this case, e.g. all finite nodes associated with a shell 13 will be listed in a selection list that calls NShell13.
DEF_REGION_FMT_FOR_NODES "*region_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for regions. Default: "NR%i" Example: “NRegion%i”. In this case, e.g. all finite nodes associated with a region 13 will be listed in a selection list that calls NRegion13.
DEF_MELEMENT_FMT_FOR_NODES "*melement_fmt*"]	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for macro elements. The list will also include boundary nodes, i.e. it is “expanded: list. Default: " NME%i" Example: “NMacroElement%i”. In this case, e.g. all finite nodes associated with a macro element 13 will be listed in a selection list that calls NMacroElement13.
DEF_BAR_REINFORCEMENT_FMT_FOR_NODES "*rc_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for reinforcement bar nodes. The list will also include boundary nodes, i.e. it is “expanded: list. Default: "NBR%i" Example: “NBar%i”. In this case, e.g. all finite nodes associated with a reinforcement bar 13 will be listed in a selection list that calls NBar13.
DEF_BAR_REINFORCEMENT_FMT_FOR_PRINCIPAL_NODES "*prc_fmt*"	The same definition as the above for DEF_VERTEX_FMT_FOR_NODES, however, it applies for principal nodes of reinforcement bars. The list will also include boundary nodes, i.e. it is “expanded: list. Default: " NPBR%i"

	Example: “NPrincBar%i”. In this case, e.g. all finite nodes associated with a principal nodes of a reinforcement bar 13 will be listed in a selection list that calls NPrincBar13.
DEF_CURVE_FMT_FOR_ELEMENTS "*curve*NODES "*melement_fmt*" DEF_PATCH_FMT_FOR_ELEMENTS "*patch_fmt*" DEF_SURFACE_FMT_FOR_ELEMENTS "*surface_fmt*" DEF_SHELL_FMT_FOR_ELEMENTS "*shell_fmt*" DEF_REGION_FMT_FOR_ELEMENTS "*region_fmt*" DEF_MELEMENT_FMT_FOR_ELEMENTS "*melement_fmt*" DEF_BAR_REINFORCEMENT_FMT_FOR_ELEMENTS "*rc_fmt*"	The same formatting strings as the above, but they are used to assign names to generated list of finite elements. Default: "EC$%i", "$E$P%i", "$E$S%i", "$E$H%i", "$E$R%i", "$EME%i", "EBR$%i" Example: “$N$MacroNode%i”. In this case, e.g. all finite nodes associated with a region 13 will be listed in a selection list that calls NRegion13.

3.13.4 The Command &MACRO_JOINT

Syntax:

&MACRO_JOINT:
MACRO_JOINT { &COORDINATES_SPEC | [ENFORCED] [ID] *n* DELETE }+

&COORDINATES_SPEC:
COORDINATES { [ID] *n* [NCOORDS] *ncoords* [X] { *x* }ncoords }+

Table 161: &MACRO_JOINT command parameters

This command adds new macro joints to the model. The joints are used for example for reinforcement bar generation. Each macro joint coordinate should be on a separate line, e.g. [ID] *n* [X] x_1 x_2 x_3 If *ncoords* is not specified, it is by default equal to problem dimension, see &TASK. This command adds new macro joints to the model or deletes the existing one. The joints are used for example for reinforcement bar generation. Each macro joint coordinate should be on a separate line, e.g. [ID] *n* [X] x_1 x_2 x_3 If *ncoords* is not specified, it is by default equal to problem dimension, see &TASK. The “ENFORCED” keyword has the same meaning as in “DELETE” command.

3.13.5 The Command &MACRO_ELEMENT

These commands are used to define or remove a macroelement definition, which is employed to generate finite element nodes and elements of a FE model to be analysed. Several types of macroelements exist and one can think of macroelement the same was as about finite element types. Each type of a macroelement set exactly a method for how some finite elements and their nodes should be generated. Input data for a macroelement consists of two parts: macroelement-specific part and macroelement-common part. Each macroelement has its unique name (that conforms with object class name, into which the macroelement is coded). This name must be input exactly and is case-sensitive. Again, the same applies for finite element types.

Table 162: &MACRO_ELEMENT supported types

CCIsoMacroElement	Macroelement to generate a block of elements of a general hexahedral shape (3D case) or a quadrilateral shape (2D case).
CCCopyElementSelection	Macroelement to create one or more copies of already generated elements. The copied elements can be rotated, shifted and translated.
CCCopyNodeSelection	Macroelement to create one or more copies of already generated nodes. The copied nodes can be rotated, shifted and translated.
CCOverlayElementSelection	Generate overlaied elements over source elements. Useful for example in the case when the original, (e.g. gap) elements have spurious energy modes.
CCExtrudeElementSelection	Macroelement to generate elements as an extrusion from a specified surface. Used advantageously to generate interphase elements between surfaces of two solid blocks.
CCDiscreteReinforcementME	Macroelement definition of discrete reinforcement bars. This macroelement definition supersedes the legacy REINFORCEMENT BAR *id* GENERATTE … command.
CCDiscretePlaneReinforcementME	Macroelement definition of discrete reinforcement smeared planes.
more macroelement types to come soon….	

3.13.5.1 Macroelement common data

These are input for all macroelement types, irrespective of their type. Macroelement specific input MACRO_ELEM_DATA_SPEC is described later for each type separately.

Syntax:

&MACRO_ELEMENT
MACRO_ELEMENT *melem_id* { &GENERATE_SPEC | &UPDATE_SPEC |
&DELETE_SPEC)

&GENERATE_SPEC:
GENERATE TYPE *"type_str"* { [THROUGH] NODES { *mnode_id* }$_+$ |
GROUP *group_id* | COUNTER [{BASE | ELEMENT_BASE | NODAL_BASE}]
base_id | NAME *"melem_name"* |
ELEMPROP *"elem_prop*" | NODEPROR *"node_prop"* { ID *id*}$_+$ |
MACRO_ELEM_DATA_SPEC | EXECUTE }$_+$

&DELETE_SPEC:
{ ENFORCED DELETE } | { DELETE }

Table 163: &MACRO_ELEMENT command parameters

melem_id	Unique integer number for the macroelement's identification. Note that macroelements ids need not be continuous.
&GENERATE_SPEC | &UPDATE | &DELETE_SPEC	Request to generate, update or remove the macroelement *melem_id* and input of the corresponding data (for generation only). Meaning of the keyword "ENFORCED " is the same in "DELETE" command.
"type_str"	Type of macroelement to be used for finite element generation, see the table &MACRO_ELEMENT supported types above.
{ [THROUGH] NODES { *mnode_id* }$_+$	List of ids of macro nodes, which defines geometry of the macroelement. Typically these are ids of some important macroelement boundary nodes are defined but it need not be always the case. For more information refer to description of a particular macroelement.
GROUP *group_id* | COUNTER [{BASE | ELEMENT_BASE | NODAL_BASE}] *base_id*	Id of a group that comprises the generated finite elements. Each macroelement is composed of one or more elements, all of them being from the GROUP *group_id* . COUNTER [{BASE | ELEMENT_BASE | NODAL_BASE}] *base_id* allows to set base ids for numbering of generated finite elements and nodes. By default *base_id* is 50000, so that the first generated element and node will be assigned id 50001. *base_id* can be set separately for nodes and elements .
ELEMPROP *"elem_prop*"	Defines a property that is assigned to each generated finite element. During generation of finite elements a selection list called *"elem_prop*" is automatically

	generated (see command &SELECTION) that contains ids of the generated elements. This selection can be later used for e.g. element load definition etc.
NODEPROP *"node_prop"* { ID *id*}+	Defines a property that is assigned to generated finite element node. Its use is similar to *"elem_prop*" and exact meaning of *"node_prop" ids* depends on a type of macroelement.
MACRO_ELEM_DATA_SPEC	Macroelement type specific data.
EXECUTE	Forces to generate finite elements immediately. By default, the generation is postponed up to the time when elements are needed, i.e. typically analysis step execution.

3.13.5.2 CCIsoMacroElement MACRO_ELEM_DATA_SPEC data

CCIsoMacroElement can be used to generate a quadrilateral or hexahedral block of elements. Geometry of the block is defined by its corner macronodes, see input data { [THROUGH] NODES { *mnode_id* }+ of input data common to all macroelements. The corner nodes are input in exactly the same way as element incidences of quadrilateral or hexahedral finite isoparametric elements, e.g. the same order of input corner ids is assumed.

Both linear and hierarchical quadratic macroelements are supported, i.e. a quadrilateral/hexahedral meshed domain can be specified by 4 to 9 / 8 to 20 macronodes. The macroelement is defined the same way as corresponding isoparametric elements.

As for NODEPROP *"node_prop"* { ID *id*}+ , (see input data common to all macroelements), the following system for finite nodes identification is used:

- Finite element nodes that coincide with macronodes are given *node_prop* from the corresponding macronodes, (if available).
- Finite element nodes located on an edge of the macroelement are given *node_prop* being a concatenation of nodal properties of macronodes defining the edge. Both edge's macronodes must have been assigned nodal property string in order to generate nodal property for intermediate finite element nodes.
- The same concept is applied for nodal properties for elements on the macroelement surface.

Syntax:

SHAPE {BAR | QUAD | HEXA } { DIR *dir_id* | DIVISION *nr* | DR {*dr*}+ }+ {
LINEAR | QUADRATIC }

Table 164: MACRO_ELEM_DATA_SPEC for CCIsoMacroElement macro element parameters

SHAPE {BAR | QUAD|HEXA}<xx…x>	Specifies shape of the macroelement. 1D can specify bar shape, 2D problems quadrilateral shape and 3D problems can use hexahedral shape (akin an isoparametric brick). The xx…x> string is so called macroelement type decoration, (akin isoparametric element types) and it specifies what macroelement macronodes are input. For example

	QUAD<xxxx> defines linear quadrilateral macroelement, QUAD<xxxxxxxx> is quadratic quadrilateral macroelement with Serendipity approximation etc.
DIR *dir_id* | DIVISION *nr* | DR $\{dr\}_+$	*nr* is number of finite elements generated in each principal direction *dir_id*. By default, elements' size *dr* in principal direction *dir_id* is 1/*nr* . However, it is possible to assign *dr* explicitly. *nr* values are expected for each *dir_id*. If less values are input, the list is toped up with the last input value. If sum of all input *dr* (for a particular *dir_id*) doesn't match 1., it is adjusted appropriately. For example: DIR 2 DIVISION 5 DR 1 2 will generate 5 elements in direction *s* , the first of them having half size of the others.
LINEAR | QUADRATIC	Linear or quadratic finite elements will be generated. Note that this input should not be mixed with linear or quadratic shape of macroelement in use.

Example:

```
MACRO_ELEMENT 1000 GENERATE TYPE
      "CCIsoMacroElement<xxxxxxxx_x_x>" THROUGH NODES 201 202 204 203
      101 102 104 103 205 206
   GROUP 1 COUNTER ELEMENT_BASE 1 NODAL_BASE 1 NAME "Macro
      block 1"
   ELEMPROP "Block_1"
   NODEPROP "N1" ID 1
   NODEPROP "N2" ID 2
   NODEPROP "N3" ID 3
   NODEPROP "N4" ID 4
   NODEPROP "N5" ID 5
   NODEPROP "N6" ID 6
   NODEPROP "N7" ID 7
   NODEPROP "N8" ID 8
   QUADRATIC
   SHAPE HEXA      DIR 1 DIVISION 4      DIR 2 DIVISION 3      DIR 3
      DIVISION 2     DR 0.2 0.2
   EXECUTE
```

3.13.5.3 CCCopyElementSelection MACRO_ELEM_DATA_SPEC data

This type of macroelement is used, when a group of elements are repeated in the FE model. In this case it is necessary to input (or generate) only the first occurrence of the elements. These elements are then assigned an element property, so that they can be referred to during creating their copies. The CCCoppyElementSelection macroelement takes responsibility for the process copying of the "master" finite elements.

CCCopyElementSelection macroelement can be used for element extrusion, mirroring, rotating etc. The transformation of copied elements is defined by principal SOURCE_NODES $\{id\}_3$ | $\{id\}_4$ (i.e. the macroelement's specific input data) and destination { [THROUGH] NODES { *mnode_id* $\}_+$, i.e. the macroelement's common input data.

Syntax:

SOURCE_NODES $\{id\}_3$ | $\{id\}_4$ | SOURCE_ELEMPROP "*elemprop*" | SOURCE_GROUP *id* | SOURCE_NODEPROP "*nodeprop*" | ACCOMPLISH *count* | [TIMES] $\}_+$

Table 165: MACRO_ELEM_DATA_SPEC for CCCopyElementSelection macro element parameters

Parameter	Description
SOURCE_NODES $\{id\}_3$ | $\{id\}_4$	Defines ids of source macronodes, whose coordinates should be transformed into destination coordinates of nodes { [THROUGH] NODES { *mnode_id* $\}_+$. Note that this input data only defines transformation of the model and no actual macronodes will be copied. 2D resp. 3D problem needs 3 resp. 4 of such nodal source-destination nodal pairs.
SOURCE_ELEMPROP "*elemprop*"	All elements defined in the selection "*elemprop*" will be copied. If this parameter is not specified, then all (current) elements from SOURCE_GROUP are copied.
SOURCE_NODEPROP "*nodeprop*"	Selection list "*nodeprop*" of source nodes, whose copy should be included in a new node selection. Name of the selection will be concatenation of destination "*elemprop*" and "*nodeprop*" . If more copies are generated, (see ACCOMPLISH *count* TIMES data), the name is appended by "*$n*", where *n* is number of additional copy. The same applies for destination "*elemprop*" .
SOURCE_GROUP *id*	Id of element group that contains the elements SOURCE_ELEMPROP "*elemprop*". By default, GROUP *group_id* is used.
ACCOMPLISH *count* | [TIMES]	Specifies number of copies to be generated. By default one copy is created, i.e. *count*=1.
KEEP_ELEM_IDS	Copy source element id to target element id. Otherwise a new target id is generated, which is behaviour by default. This function is only available for the first copy of the elements, (i.e. count=1) and at the same time target and source group ids must be different.

Example:

MACRO_ELEMENT 1001 GENERATE TYPE "CCCopyElementSelection" THROUGH NODES 102 107 104 202

GROUP 1 NAME "Macro block 2"

ELEMPROP "Block_2"

SOURCE_NODES 101 102 103 201 SOURCE_ELEMPROP "Block_1" SOURCE_NODEPROP "N1N4N5N8" "N5N6N7N8" "N5N8" "N5N6"

EXECUTE

3.13.5.4 CCCopyNodeSelection MACRO_ELEM_DATA_SPEC data

This type of macroelement is used, when a group of nodes are repeated in the FE model. In this case it is necessary to input (or generate) only the first occurrence of the nodes. These are then assigned a node property, so that they can be referred to during creating their copies. The CCCoppyNodeSelection macroelement takes responsibility for the copying of the "master" finite nodes. Note that the generated nodes have no associated degree of freedom unless thay are later connected to a finite element or the command "ALLOCATE_DOFS NODES...." is executed.

CCCopyNodeSelection macroelement can be e.g. used for connection of 2D shell elements, which use local degrees of freedom and which are connected via an arbitrary nodes having all dofs global. The transformation of copied nodes is defined by principal SOURCE_NODES $\{id\}_3$ | $\{id\}_4$ (i.e. the macroelement's specific input data) and destination { [THROUGH] NODES { *mnode_id* }$_+$, i.e. the macroelement's common input data.

Syntax:

SOURCE_NODES $\{id\}_3$ | $\{id\}_4$ | INPUT_NODEPROP "*prop_in*" |
OUTPUT_NODEPROP "*prop_out*" | SOURCE_NODEPROP "*nodeprop*" |
ACCOMPLISH *count* | [TIMES] }$_+$

Table 166: MACRO_ELEM_DATA_SPEC for CCCopyNodeSelection macro element parameters

Parameter	Description
SOURCE_NODES $\{id\}_3$ | $\{id\}_4$	Defines ids of source macronodes, whose coordinates should be transformed into destination coordinates of nodes { [THROUGH] NODES { *mnode_id* }$_+$. Note that this input data only defines transformation of the model and no actual macronodes will be copied. 2D resp. 3D problem needs 3 resp. 4 of such nodal source-destination nodal pairs.
INPUT_NODEPROP "*prop_in*"	All nodes defined in the selection "*prop_in*" will be copied.
SOURCE_NODEPROP "*nodeprop*"	Selection list "*nodeprop*" of source nodes, whose copy should be included in a new node selection. Name of the selection will be concatenation of destination "*elemprop*" and "*nodeprop*" . If more copies are generated, (see ACCOMPLISH *count* TIMES data), the name is appended by "*$n*", where *n* is number of additional copy. The same applies for destination "*elemprop*" .
OUTPUT_NODEPROP "*prop_out*"	Identification of output nodes. It defines a property that is assigned to each generated finite node. During generation of finite nodes a selection list called *"prop_out*" is automatically generated (see command &SELECTION) that contains ids of the generated nodes. This selection can be later used for e.g. node load definition etc.
ACCOMPLISH *count* | [TIMES]	Specifies number of copies to be generated. By default one copy is created, i.e. *count*=1.

Example:

MACRO_ELEMENT 1003 GENERATE TYPE "CCCopyNodeSelection" THROUGH NODES 200 201 202 203

OUTPUT_NODEPROP "xxx" NAME "Macro block - nodes copied"

INPUT_NODEPROP "LIST_SOLID_NODES"

SOURCE_NODEPROP "OX" "OY" "OZ"

SOURCE_NODES 100 101 102 103 ACCOMPLISH 3 TIMES EXECUTE

3.13.5.5 CCOverlayElementSelection MACRO_ELEM_DATA_SPEC data

This type of macroelement is used, when we need to generate elements over a particular volume of the structure, which has already been meshed. For example gap elements suffer from spurios energy modes. They only exhibits stiffness for deformation modes, when the upper surface moves relative to the bottom surface. Other displacements modes produce/consume zero energy, so that stiffness matrix features (for 3D gaps) 7 and more zero eigenvalues. (A regular 3D finite elements has 6 zero eigenvalues: three displacements and three rotations). This problem can be solved by supporting the gap elements by ambient elements or to overlay them with other elements of the same shape. Such elements have typically only some small rigidity to remove singularity of the structure. CCOverlayElementSelection helps to generate the auxillary elements. It can prove useful also for modelling fibreconcrete, concrete with smeared reinforcement etc.

CCOverlayElementSelection macro element behaves similar to CCCopyElementSelection macro element, however spource and target elements share the same volume. Hence, no coordinates transformation is involved, no new finite nodes are generated, target and source elements share incidences. The input SOURCE_NODES and SOURCE_NODEPROP (for CCCopyElementSelection macro element) is superfluous.

Syntax:

SOURCE_ELEMPROP "*elemprop*" | SOURCE_GROUP *id* | | ACCOMPLISH *count* | [TIMES] }+

Table 167: MACRO_ELEM_DATA_SPEC for CCOverlayElementSelection macro element parameters

Parameter	Description
SOURCE_ELEMPROP "*elemprop*"	All elements defined in the selection "*elemprop*" will be copied. If this parameter is not specified, then all (current) elements from SOURCE_GROUP are copied.
SOURCE_GROUP *id*	Id of element group that contains the elements SOURCE_ELEMPROP "*elemprop*". By default, GROUP *group_id* is used.
ACCOMPLISH *count* | [TIMES]	Specifies number of copies to be generated. By default one copy is created, i.e. *count*=1.
KEEP_ELEM_IDS	Copy source element id to target element id. Otherwise a new target id is generated, which is behaviour by default. This function is only available for the first copy of the elements, (i.e. count=1) and at the same time target and source group ids must

	be different.

Example:

MACRO_ELEMENT 1001 GENERATE TYPE "CCCopyElementSelection" THROUGH NODES 102 107 104 202

GROUP 1 NAME "Macro block 2"

ELEMPROP "Block_2"

SOURCE_NODES 101 102 103 201 SOURCE_ELEMPROP "Block_1" SOURCE_NODEPROP "N1N4N5N8" "N5N6N7N8" "N5N8" "N5N6"

EXECUTE

3.13.5.6 CCExtrudeElementSelection MACRO_ELEM_DATA_SPEC data

This type of macroelement is used, when some elements should be generated as an extrusion of elements on a surface. Such an extrusion can be accomplished several times, thereby generating e.g. a set of layers for modeling a complex interphase between two solid blocks. The macroelement reads element group and ids of nodes of the source surface, (from which the extrusion takes place) and it also reads a vector of the extrusion, (defined by NODE and SOURCE_NODE macro nodes). The vector can be of zero length.

At the end, the macroelement generates selection lists, (for the two surfaces of extruded elements). They are named as ELEMPROP+"$n"+SOURCE_NODEPROP+"<-" (bottom surface) and ELEMPROP+"$n"+SOURCE_NODEPROP+"->" (top surface), where n is number of copies. If n==0, i.e. the 1st layer, the whole string "$0" is omitted. For example, the sample below would generate the following selections:

"Block_3_Block_2_N2N3N6N7<-"

"Block_3_Block_2_N2N3N6N7->"

"Block_3$1_Block_2_N2N3N6N7<-"

"Block_3$1_Block_2_N2N3N6N7->"

"Block_3$2_Block_2_N2N3N6N7<-"

"Block_3$3_Block_2_N2N3N6N7->"

Syntax:

SOURCE_GROUP *id* | SOURCE_NODE *id*| SOURCE_GROUPPROP "*groupprop*" | SOURCE_ELEMPROP "*elemprop*" | SOURCE_NODEPROP "*nodeprop*" | ACCOMPLISH *count* | [TIMES] }+

Table 168: MACRO_ELEM_DATA_SPEC for CCCopyElementSelection macro element parameters

Parameter	Description
SOURCE_NODE *id*	Defines id of a bottom macronode for the extrusion vector. The top node is defined by NODE *id.*
SOURCE_GROUPPROP	This selection contains information, from which groups the

"*groupprop*"	elements from "*elemprop*" comes, i.e. each entry in "*elemprop*" has associated entry in "*groupprop"*.
SOURCE_ELEMPROP "*elemprop*"	All elements defined in the selection "*elemprop*" with nodes defined in SOURCE_NODEPROP "*nodeprop*" will be used as a base for the extrusion.
SOURCE_NODEPROP "*nodeprop*"	See above.
SOURCE_GROUP *id*	If SOURCE_GROUPPROP is omitted, SOURCE_GROUP *id* sets element group that contains the elements SOURCE_ELEMPROP "*elemprop*".
ACCOMPLISH *count* | [TIMES]	Specifies number of copies to be generated. By default one copy is created, i.e. *count*=1.

Example:

MACRO_ELEMENT 1002 GENERATE TYPE "CCExtrudeElementSelection" THROUGH NODE 110

GROUP 2 NAME "MB_3"

ELEMPROP "Block_3"

SOURCE_NODE 107 SOURCE_ELEMPROP "Block_2" SOURCE_NODEPROP "Block_2_N2N3N6N7" SOURCE_GROUP 1 ACCOMPLISH 3 TIMES

EXECUTE

3.13.5.7 CCDiscreteReinforcementME MACRO_ELEM_DATA_SPEC data

This macroelement is used to generate discrete reinforcement bars. The element supersedes the legacy command REINFORCEMENT BAR …. The "{ [THROUGH] NODES { *mnode_id* }$_+$" data from the MACRO_ELEMENT command defines macro nodes, thru which the reinforcement bar should pas; the mnode_1 and mnode_n being the first and the last macro node of the bar.

Syntax:
[SIZE] MINIMUM *x* | [EMBEDDED] [IN] [SOLID] [SOLIDS] { AT | FROM} *solid_group_id_*1 [TO *solid_group_id_2*] | {NORMAL | TINY [SIZE]} | PROCESS_FLAG { USE_REFERENCE_COORDS | USE_CURRENT_COORDS | COPY_DEFORMATION | COPY_DEFORMATION_ONCE | COPY_NO_DEFORMATION } | REPEAT *n* | DX *dx1 dx2 dx3...*| DY *dx1 dy2 dy3...* | DZ *dz1 dz2 dz3...*| RESET_EMBEDDED | RECONNECT_NODES }$_+$

Table 169: MACRO_ELEM_DATA_SPEC for CCReinforcementME MACRO_ELEM_DATA_SPEC element parameters

Parameter	Description
[EMBEDDED] [IN]	Interval of element groups defining the "master" material, i.e.

[SOLID] [SOLIDS] { AT | FROM} *solid_group_id_*1 [TO *solid_group_id_2*]	solids ids, where the bar should be generated. In other words, the bar will be embedded in the specified material groups.
{ NORMAL | TINY [SIZE] }	If TINY size is defined, then the algorithm used to generate elements of the bar works correctly even in the case, that more neighboring NODES are located with the same elements. If it is not the case, use of NORMAL size is preferable, as it results in much faster element generation. Default value: NORMAL
[SIZE] MINIMUM *x*	Minimum length of generated element. If not satisfied, newly generated node is ignored. Default value: 0 [length units]
REPEAT *n*	How many additional macro elements should be generated or reconnected. By default *n*=0, i.e. only one macro element is produced. This option make possible to generate a serie of macro elements using just one input definition. [32]
DX *dx1 dx2 dx3...* DY *dx1 dy2 dy3...* DZ *dz1 dz2 dz3...*	Distance in X direction between generated macro elements due to REPEAT *n*>0. If less then *n* values are input, the missing entries are derived from the most recent DX input. By default *dx*=0. The same for DY and DZ input.
RESET_EMBEDDED	Clear all input in EMBEDDED] [IN] [SOLID] [SOLIDS] { AT | FROM} *solid_group_id_*1.....
RECONNECT_NODES	Reconnect generated nodes into the surronding solids. Useful for the case of macro elements' update needed in simulating a construction process.
PROCESS_FLAG {...}	Process flags have the same meaning as for master-slave boundary conditions used to connect reinforcement bars to tye surrounding solids.

Example:

```
MACRO_ELEMENT 1001 GENERATE TYPE "CCDiscreteReinforcementME"
  THROUGH NODES 100 101 NAME "Bottom reinforcement" MINIMUM 0.
  GROUP 2 EMBEDDED AT 1
  ELEMPROP "Bar_1"
  NODEPROP "N1" ID 1
  NODEPROP "N2" ID 2
  REPEAT 2 DX 0 DY 0.02 0.02 DZ 0 // can be only  REPEAT 2 DY 0.02 as it remembers the last value
```

[32] Not available in ATENA version 4.3.1 and older.

EXECUTE

MACRO_ELEMENT 1000011 UPDATE REPEAT 9 RESET_EMBEDDED RECONNECT_NODES

3.13.5.8 CCDiscretePlaneReinforcementME MACRO_ELEM_DATA_SPEC data

This macroelement is used to generate discrete smeared reinforcement planes. Each reinforcing plane can be of triangular or quadrilateral shape. Its corner boundary nodes are defined by 3 or 4 macro nodes.

Syntax:

{PLANE *n* THROUGH NODES { *n1 n2 n3 n4* | *n1 n2 n3* }}$_+$ MINIMUM [SIZE] *x* | [EMBEDDED] [IN] [SOLID] [SOLIDS] { AT | FROM} *solid_group_id_*1 [TO *solid_group_id_2*] | NORMAL | TINY [SIZE] }$_+$

Table 170: MACRO_ELEM_DATA_SPEC for CCDiscretePlaneReinforcementME MACRO_ELEM_DATA_SPEC element parameters

Parameter	Description
{PLANE *n* THROUGH NODES { *n1 n2 n3 n4* | *n1 n2 n3* }}$_+$	Specify 3 or 4 macronodes ids defining triangular or quadrilateral reinforcement plane.
[EMBEDDED] [IN] [SOLID] [SOLIDS] { AT | FROM} *solid_group_id_*1 [TO *solid_group_id_2*]	Interval of element groups defining the "master" material, i.e. solids ids, where the bar should be generated. In other words, the bar will be embedded in the specified material groups.
NORMAL | TINY [SIZE]	If TINE size is defined, then the algorithm used to generate elements of the smeared reinforcement planes works correctly even in the case, that more neighboring NODES are located with the same elements. If it is not the case, use of NORMAL size is preferable, as it results in much faster element generation. Default value: NORMAL
[SIZE] MINIMUM *x*	Minimum distance between nodes of generated element. If not satisfied, newly generated node is ignored. Default value: 0 [length units]

Example:

MACRO_ELEMENT 1001 GENERATE TYPE "CCDiscretePlaneReinforcementME"

PLANE 1 THROUGH NODES 1001 1005 1006 1004

PLANE 2 THROUGH NODES 1005 1002 1003

PLANE 3 THROUGH NODES 1005 1003 1006

NAME "Bottom reinforcement"

```
  MINIMUM 0.
  GROUP 10 EMBEDDED AT 1
  ELEMPROP "Plame_1"
  NODEPROP "N1" ID 1
  NODEPROP "N2" ID 2
  NODEPROP "N3" ID 3
  NODEPROP "N4" ID 4
  EXECUTE

MACRO_ELEMENT 1001 GENERATE TYPE "CCDiscreteReinforcementME"
  THROUGH NODES 100 101 NAME "Bottom reinforcement" MINIMUM 0.
  GROUP 2 EMBEDDED AT 1
  ELEMPROP "Bar_1"
  NODEPROP "N1" ID 1
  NODEPROP "N2" ID 2
```

3.13.6 The command &TRANSFORM_COORDS

This section describes the command used to change/transform coordinates of earlierly inputed finite nodes. Their ids must be specified in the selection “node_list”. Using this command a flat surface can be simply modified to a spherical surface. New coordinates can be calculated either by EVAL Atena evaluator or the PYTHON interpreter can be employed. In the former case input “eqn_str”, which contains an mathematical expression to do the calculation. In the latter case specify “pymodule_name” and “py_eqn_name” of a python function to be used. The module and function have to be defined beforehand, see PYTHON command. If “py_module_name” is ””, i.e. empty string, then “__main__” is assumed.

Syntax:

```
&TRANSFORM_COORDS:
   TRANSFORM_COORDS NODE_IDS “node_list” { { EQUATION_X |
         EQUATION_Y | EQUATION_Z}  “eqn_str” | { PYTHON_EQUATION_X |
```

PYTHON_EQUATION_Y | PYTHON_EQUATION_Z} “py_module_name” “py_eqn_name” }+

Example

```
TRANSFORM_COORDS NODE_IDS "Nmiddle_3Nmiddle_4Nmiddle_7Nmiddle_8"
  EQUATION_X
"thick_front/2.+thick_middle+thick_honeycomb/2.*(1.+sin(1.57+2*3.14/length_period*x))"
  PYTHON_EQUATION_Y "__main__" "transform_eqn_x_front"
  EQUATION_Z "z"
  EXECUTE
  ;
```

3.13.7 The command &UPDATE_ELEMENT_CONSTRUCT_TIME used for digital printing[33]

This section describes a command used to update delta element time *dtime* during digital printing of the structure. The *dtime* value is subtracted from actual structural step time *t* and the result is used for time functions for materials with time variable parameters. Also, elements with *t*< *dtime* are assembled in “reduced” form, i.e. prior assembly their matrices and vectors are multiplied by NEGLIGIBLE_ELEMENT_CONTRIBUTION_COEFF *x* coefficient. By default, they are not drawn, (unless overwritten by a particular draw condition).

[33] Not available in ATENA version 5.7.0 and older

The actual process of digital printing is specified by a polygon of printing head's motion. The *track_nodes_ids_name* selection lists sequentially all nodes thru which the head moves within the first layer. The second layer is printed similarly to the first one but it is elevated by THICKNESS *thick* at DIRECTION *x y z.* The same for the third and next layers.

If the polygon is not continuous, insert *node_id=0* in the selection at appropriate place.

Note that for the sake of simplicity ATENA works with coordinates of elements' centres, whilst the printing head polygon indicates position, (i.e. vectors) of right bottom edge of the printed layer. Hence, in order to check position of an element with respect to the printing polygon, its element centre coordinates $[x_1\ x_2\ x_3]$ must be shifted by

$$x_i^{'} = x_i - \left(thick\, n_i - width\, v_i \right) / 2$$

where vector $\overline{v}$ is cross product of vector $\overline{n}$ and a vector of the current head's motion, (i.e. along its appropriate motion segment). If we need opposite direction of $\overline{n}$ or $\overline{v}$, use negative *thick* and/or *width.*

Syntax:

```
&UPDATE_CONSTRUC_TIME:
   UPDATE_ELEMENT_CONSTRUCT_TIME
      TRACK "track_nodes_ids_name"
      [ GROUPS "printed_groups_ids_name" ]
      VELOCITY vel
        DEVIATION_DELAY_FNC n
      [LAYER] THICKNESS thick  WIDTH width DIRECTION n1 n2 n3
        [ START_TIME start_time ]
        [ TIME_BETWEEN_LAYERS dtime ]
        [ ALLOW_REVERSE_PRINTING ] [{IS_POINT | IS_EDGE} ]  [ {
        USE_2_OPT | USE_3_OPT | USE_4_OPT | USE_5_OPT}
        OPTIMIZE_TRACK ]  [ MIN_REL_HEIGHT ] [ MAX_REL_HEIGHT ]  [
        ADD_MAX_CONSTR_TIME ] [ VERBOSE ] [TRACK_OFFSET offset ]
      [EXECUTE]  [&ESTIMATE]

   &ESTIMATE :
   ESTIMATE [G x]  [Q x]  [V_N x]  [T_L x]  [H x]  [ { B | SPAN } x]  [E0 x]  [XI_E x]
         [NU x]  [RHO x]  [PHI0 x]  [XI_PHI x]  [C0 x]  [XI_C x] [K x]  [SIGMA_P0 x]
         [XI_SIGMA_P x]  [ [VERTICAL]  [EDGES]  [EDGE] ] {
        FREE_UNSUPPORTED |  SIMPLE_SUPPORTED |  CLAMP_SUPPORTED }
        [FETCH_PARAMS_FROM_MATERIAL] [CALCULATE]
```

Table 171: & UPDATE_ELEMENT_CONSTRUCT_TIME parameters

TRACK "*track_nodes_ids_name*"	Name of a selection with track nodes' ids. If a *node_id*=0, e.g. the TRACK selection includes [….*i, j,0,k,l* ...], then we add construction times for motion between nodes *i,j* and *k,l* and ignore segment (and

	time for motion) between nodes *j,k*. If *node_id*<0, e.g. the TRACK selection includes [….*i, j,-k,l,m* ...], then we add construction times for motion between all nodes *i,j,abs(k),l,m*, however, at segments *j,abs(k)* and *abs(k),l* no structural element can be printed.
GROUPS "*printed_groups_ids_ name*"	Name of a selection with groups' ids of the printed element. If not specified, all present groups are considered.
VELOCITY *vel*	Velocity of the printing head.
DEVIATION_DELAY_FNC *n*	Id of function fnc(*delta_angle*) that defines printing delay due to direction change of neighbouring polygon printing segments. The angle is in rad. If unspecified, no delay is introduced.
THICKNESS *thick*	Velocity of the printing head.
WIDTH *width*	Width of the printing head. Not used for 2D and axisymmetric analyses.
DIRECTION n_1 n_2 n_3	Direction of corresponding nodes in subsequent printed layers. Typically it is [0,0,1] for 3D analyses and [0,1] for 2D and axisymmetric analyses
START_TIME *start_time*	Time to be added to all points in the track polygon. By default *start_time*=0
TIME_BETWEEN_LAYERS *dtime*	Extra time to be added while moving between adjacent layers. By default *dtime*=0
ALLOW_REVERSE_PRINTING	Odd and even layers are printed in forward and backward direction, respectively. By default all is printed in forward direction, (referred in respect to track printing polygon).
{USE_2_OPT | USE_3_OPT | USE_4_OPT | USE_5_OPT} OPTIMIZE_TRACK	Try to optimize the printing polygon, i.e. optimize order of nodes in TRACK "*track_nodes_ids_name*" to make the printing path shorter. Use Nearest neighbor and one of K-OPT method.
VERBOSE	Verbose partial steps of the above optimization
MIN_REL_HEIGHT *min_z* MAX_REL_HEIGHT *max_z*	Change construction time only for elements with height above *min_z* and below *max_z*. The height is considered relative to the points in TRACK.
ADD_MAX_CONSTR_TIME	Increment *start_time* by the time needed to print previous elements.
TRACK_OFFSET *offset*	Set track offset used to calculate layer id. It is usually used in cooperation with ADD_MAX_CONSTR_TIME. By default, *offset* =0.
IS_POINT | IS_EDGE	Allow oprimization of printing edges or points. Note

	that in far most cases printing of edges is required.
EXECUTE	Execute the element update now. By default, it is executed at the beginning of a subsequent STEP EXECUTE command.
ESTIMATE	Estimate maximum height of the wall based on plasticity strength and linear buckling failure criteria.

Table 172: &ESTIMATE command parameters

G *x*	Gravity acceleration Default: 9.806 m/s^2
Q *x*	Material volume discharged from the printing nozzle per unit time Default: 54427.E-9 m^3/s
V_N *x*	Horizontal speed of the printing nozzle Default: 0.104 m/s
T_L *x*	Period required for printing an individual layer Default: B/V_N s
H *x*	Wall thickness Default: 0.055 m
{ B | SPAN } *x*	Wall width Default: 0.25 m;
E0 *x*	Initial elastic stiffness printing material E0 Default: 48564 Pa
XI_E *x*	Curing rate elastic stiffness xi_E[s^(-1)]; E = E0*(1+xi_E) Default 0.895 Pa/s
NU *x*	Poisson's ratio printing material Default: 0.5 [-]
RHO *x*	Density printing material Default: 2100 kg/m3
PHI0 *x*	Friction angle. If nonzero, shear strength criterion is employed. Default: 0 rad, i.e. use SIGMA_P0 and XI_SIGMA_P

XI_PHI *x*	Linear curing rate of the yield strength with respect t o phi. If nonzero, shear strength criterion is employed. Default: 0 1/s , i.e. use SIGMA_P0 and XI_SIGMA_P
C0 *x*	Cohesion of fresh printing material. If nonzero, shear strength criterion is employed. Default: 0 Pa, , i.e. use SIGMA_P0 and XI_SIGMA_P
XI_C *x*	Linear curing rate of the yield strength with respect t o c. If nonzero, shear strength criterion is employed. Default: 0 1/s, i.e. use SIGMA_P0 and XI_SIGMA_P
SIGMA_P0 *x*	Material compression uniaxial yield strength of the fresh printing material. Default: 0 Pa
XI_SIGMA_P *x*	Linear curing rate of the yield strength, used as sigm a_p = sigma_p0 *(1+xi_sigma*t) Default: 0 1/s
K *x*	Min(sigma_y/sigma_x, sigma_z/sigma_x), is the coe fficient of lateral to vertical stress at the wall bottom , used for Mohr Coulomb yield criterion , Default: 0 [-]
[VERTICAL] [EDGES] [EDGE]] { FREE_UNSUPPORTED | SIMPLE_SUPPORTED | CLAMP_SUPPORTED }	Specify boundary conditions along the wall's vertical edges. Default: FREE_UNSUPPORTED
FETCH_PARAMS_FROM_MATERI AL *material_id*	Fetch a copy of parameters required by &ESTIMATE command from corresponding parameters already input in the command &UPDATE_ELEMENT_CONSTRUCT_TIME. Material related parameters are computed using input data for material *material_id.* Note that any of the &ESTIMATE parameters can be subsequently changed and its change does not edit value in the &UPDATE_ELEMENT_CONSTRUCT_TIME.
CALCULATE	Execute guess of maximum wall heigth

Example for 3D analysis:

```
SELECTION "printed_groups_ids_name" LIST 208 ;
SELECTION "track_nodes_ids_name" LIST 109   91 0 49   31 44 ;
UPDATE_ELEMENT_CONSTRUCT_TIME
TRACK "track_nodes_ids_name"
GROUPS "printed_groups_ids_name"
VELOCITY 0.05
LAYER THICKNESS 0.15 WIDTH -0.2 DIRECTION 0. 0. 1.
START_TIME 0.1 TIME_BETWEEN_LAYERS 0.
EXECUTE   ;
```

Example for 2D/axisymmetric analysis

```
SELECTION "PRINTED_GROUPS_IDS_NAME" LIST 1 ;
SELECTION "TRACK_NODES_IDS_NAME" KEEP_DUPLICATES  INSERT "N1"
INSERT "N2"
SET negligible_element_contribution_coeff 0.0001
UPDATE_ELEMENT_CONSTRUCT_TIME
  TRACK "TRACK_NODES_IDS_NAME"
  GROUPS "PRINTED_GROUPS_IDS_NAME"
  VELOCITY =VELOC TIME_BETWEEN_LAYERS 16.956
  LAYER THICKNESS =DY*10.  DIRECTION 0. 1. START_TIME 0.
  ALLOW_REVERSE_PRINTING
  EXECUTE
```

3.14 Transport Analysis Related Commands

The moisture and humidity transport analysis in ATENA has been developed in a CCStructuresTransport engineering module. Hence, the " /M *module_name* " parameter from the ATENA command line must read:

```
/M CCStructuresTransport:
```

The CCStructuresTransport module is an extension of CCFEModel, (being the base for all engineering modules in ATENA) and hence most input command for the transport analysis are the same as those e.g. for static analysis of structures. This section describes additional commands that are relevant only for the transport analysis.

Generally, it is important to recognize similarity between static and transport analyses. Primary unknowns (i.e. LHS) and loading (i.e. RHS) variables for static analysis are deformations and load forces, respectively. The equivalent entities for the transport analysis are vector of psis (i.e. LHS variables) and vector of fluxes (i.e. RHS variables). The psis encompass nodal relative humidity and temperature. Similarly the vector of fluxes includes moisture ant heat fluxes at structural nodes. If Dirichlet boundary conditions are given that means we are going to fix somewhere humidity and/or temperature value. The same applies for Von Neumann boundary conditions. Similar to static analysis, both LHS and RHS boundary conditions have incremental character, however, sign of Von Neumann boundary conditions now depends on flux's orientation with respect direction of normal of the surface, where the boundary condition is applied, (and thus unlike in CCStructures the direction of global coordinate axes is irrelevant). Plus sign means an inflow, i.e. flow going in the surface, i.e. in the body and minus sign means an outflow, flow in the surface, i.e. losses. At beginning of the analysis, i.e. at time t=0 a degree of freedom without any LHS and/or RHS boundary condition means a degree of freedom belonging to impermeable surface.

There are a few input commands that are meaningful only for transport analysis. These are commands:

- related to temporal time integration, &Transport Set parameters (and problem's time step marching execution as it is),
- needed for definition of transport finite element, &Transport finite elements,
- specifying transport constitutive material model, &Transport constitutive material,
- inputting structural initial state conditions, &Transport initial value of state variables,
- &History export related commands
- &Transport analysis additional output data.

Note also that only Modified Newton-Raphson or Full Newton-Raphson execution method can be used.

3.14.1 Transport constitutive material model

The &MATERIAL_TYPE_PARAMS from &MATERIAL command for the case of transport analysis reads:

```
&MATERIAL_TYPE_PARAMS TYPE  {&CCModelBaXi94_PARAMS |
      &CCTransportMaterial_PARAMS |  &CCTransportMaterialLevel7_PARAMS }

&CCModelBaXi94_PARAMS   "CCModelBaXi94" [CONCRETE  CONCRETE TYPE
      n_type  RATIO_WC ratio  [CEMENT_WEIGHT  cem_weight ]] [
      TEMPERATURE  { [K_TEMP_H x] | [K_TEMP_TEMP x] | [K_TEMP_W x] |
      [K_TEMP_GRAV x] |  [C_TEMP_H x] | [C_TEMP_TEMP x] |  [C_TEMP_W x] |
      [K_TEMP_H_FNC_ID x] | [K_TEMP_TEMP_FNC_ID x] |
      [K_TEMP_W_FNC_ID x] | [K_TEMP_GRAV_FNC_ID x] |
```

[C_TEMP_H_FNC_ID x] | [C_TEMP_TEMP_FNC_ID x] |
[C_TEMP_W_FNC_ID x] }+

&CCTransportMaterial_PARAMS TYPE "CCTransportMaterial"
[TEMPERATURE
{ [K_TEMP_H K_{Th}^0] | [K_TEMP_TEMP K_{TT}^0] | [K_TEMP_W K_{Tw}^0] |
[K_TEMP_GRAV K_{Tgrav}^0] | [C_TEMP_H C_{Th}^0] | [C_TEMP_TEMP C_{TT}^0] |
[C_TEMP_W C_{Tw}^0] | [C_H_T C_{Tt}^0] | [K_TEMP_H_FNC_TEMP_ID $f_{K_{Th}}^T$] |
[K_TEMP_TEMP_FNC_TEMP_ID $f_{K_{TT}}^T$] | [K_TEMP_W_FNC_TEMP_ID
$f_{K_{Tw}}^T$] | [K_TEMP_GRAV_FNC_TEMP_ID $f_{K_{Tgrav}}^T$] |
[C_TEMP_H_FNC_TEMP_ID $f_{C_{Th}}^T$] | [C_TEMP_TEMP_FNC_TEMP_ID $f_{C_{TT}}^T$] |
[C_TEMP_W_FNC_TEMP_ID $f_{C_{Tw}}^T$] | [C_TEMP_T_FNC_TEMP_ID $f_{C_{Tt}}^T$] |
[K_TEMP_H_FNC_H_ID $f_{K_{Th}}^h$] | [K_TEMP_TEMP_FNC_H_ID $f_{K_{TT}}^h$] |
[K_TEMP_W_FNC_H_ID $f_{K_{Tw}}^h$] | [K_TEMP_GRAV_FNC_H_ID $f_{K_{Tgrav}}^h$] |
[C_TEMP_H_FNC_H_ID $f_{C_{Th}}^h$] | [C_TEMP_TEMP_FNC_H_ID $f_{C_{TT}}^h$] |
[C_TEMP_W_FNC_H_ID $f_{C_{Tw}}^h$] | [C_TEMP_T_FNC_H_ID $f_{C_{Tt}}^h$] |
[K_TEMP_H_FNC_T_ID $f_{K_{Th}}^t$] | [K_TEMP_TEMP_FNC_T_ID $f_{K_{TT}}^t$] |
[K_TEMP_W_FNC_T_ID $f_{K_{Tw}}^t$] | [K_TEMP_GRAV_FNC_T_ID $f_{K_{Tgrav}}^h$] |
[C_TEMP_H_FNC_T_ID $f_{C_{Th}}^t$] | [C_TEMP_TEMP_FNC_T_ID $f_{C_{TT}}^t$] |
[C_TEMP_W_FNC_T_ID $f_{C_{Tw}}^t$] | [C_TEMP_T_FNC_T_ID $f_{C_{Tt}}^t$] }+]
[WATER
{ [D_H_H D_{wh}^0] | [D_H_TEMP D_{wT}^0] | [D_H_W D_{ww}^0] | [D_H_GRAV D_{wgrav}^0] |
[C_H_H C_{wh}^0] | [C_H_TEMP C_{wT}^0] | [C_H_W C_{ww}^0] [C_H_T C_{wt}^0] |
[D_H_H_FNC_H_ID $f_{D_{wh}}^h$] | [D_H_TEMP_FNC_H_ID $f_{D_{wT}}^h$] |
[D_H_W_FNC_H_ID $f_{D_{ww}}^h$] | [D_H_GRAV_FNC_H_ID $f_{D_{wgrav}}^h$] |
[C_H_H_FNC_H_ID $f_{C_{wh}}^h$] | [C_H_TEMP_FNC_H_ID $f_{C_{wT}}^h$] |
[C_H_W_FNC_H_ID $f_{C_{ww}}^h$] | [C_H_T_FNC_H_ID $f_{C_{wt}}^h$] |
[D_H_H_FNC_TEMP_ID $f_{D_{wh}}^T$] | [D_H_TEMP_FNC_TEMP_ID $f_{D_{wT}}^T$] |
[D_H_W_FNC_TEMP_ID $f_{D_{ww}}^T$] | [D_H_GRAV_FNC_TEMP_ID $f_{D_{wgrav}}^T$] |
[C_H_H_FNC_TEMP_ID $f_{C_{wh}}^T$] | [C_H_TEMP_FNC_TEMP_ID $f_{C_{wT}}^T$] |
[C_H_W_FNC_TEMP_ID $f_{C_{ww}}^T$] | [C_H_T_FNC_TEMP_ID $f_{C_{wt}}^T$] |
[D_H_H_FNC_T_ID $f_{D_{wh}}^t$] | [D_H_TEMP_FNC_T_ID $f_{D_{wT}}^t$] |
[D_H_W_FNC_T_ID $f_{D_{ww}}^t$] | [D_H_GRAV_FNC_T_ID $f_{D_{wgrav}}^t$] |
[C_H_H_FNC_T_ID $f_{C_{wh}}^t$] | [C_H_TEMP_FNC_T_ID $f_{C_{wT}}^t$] |
[C_H_W_FNC_T_ID $f_{C_{ww}}^t$] | [C_H_T_FNC_T_ID $f_{C_{wt}}^t$] }+]

&CCTransportMaterialLevel7_PARAMS TYPE "CCTransportMaterialLevel7"
[SPECIFIC
{ [DOH_FNC_ID] | [DOH25_FNC_ID] | [B1 *val*] | [B2 *val*] | [ALPHAINF *val*] | [ETA *val*] | [A *val*] | [QH_POT *val*] | [QW_POT *val*] | [TH_INIT *val*] | [ALPHA_INIT *val*] | [TH_INCR_MIN *val*] | [TH_INCR_MAX *val*] | [TEMPERATURE_INCR_MAX *val*] | [CEMENT_MASS *val*] | [AGGREGATE_MASS *val*] | [FILLER_MASS *val*] | [CEMENT_DENSITY *val*] | [WATER_DENSITY *val*] | [AGGREGATE_DENSITY *val*] | [FILLER_DENSITY ival] | [C_AGGREGATE_TEMP_TEMP *val*] | [C_FILLER_TEMP_TEMP *val*] | [C_CEMENT_TEMP_TEMP *val*] | [C_WATER_TEMP_TEMP *val*] | [K_AGGREGATE_TEMP_TEMP *val*] | [K_FILLER_TEMP_TEMP *val*] | [K_CEMENT_TEMP_TEMP *val*] | [K_WATER_TEMP_TEMP *val*] | [K_AIR_TEMP_TEMP *val*] | [W_F *val*] | [H80 *val*] | [W80 *val*] | [TEMP0 *val*] | [A_WV *val*] | [A_W *val*] | [MI_WV *val*] | [TEMP0_ICE *val*] | [A_WV_ICE *val*] | [EA *val*] }$_+$]
[TEMPERATURE
{ [K_TEMP_H K^0_{Th}] | [K_TEMP_TEMP K^0_{TT}] | [K_TEMP_W K^0_{Tw}] | [K_TEMP_GRAV K^0_{Tgrav}] | [C_TEMP_H C^0_{Th}] | [C_TEMP_TEMP C^0_{TT}] | [C_TEMP_W C^0_{Tw}] | [C_H_T C^0_{Tt}] | [K_TEMP_H_FNC_TEMP_ID $f^T_{K_{Th}}$] | [K_TEMP_TEMP_FNC_TEMP_ID $f^T_{K_{TT}}$] | [K_TEMP_W_FNC_TEMP_ID $f^T_{K_{Tw}}$] | [K_TEMP_GRAV_FNC_TEMP_ID $f^T_{K_{Tgrav}}$] | [C_TEMP_H_FNC_TEMP_ID $f^T_{C_{Th}}$] | [C_TEMP_TEMP_FNC_TEMP_ID $f^T_{C_{TT}}$] | [C_TEMP_W_FNC_TEMP_ID $f^T_{C_{Tw}}$] | [C_TEMP_T_FNC_TEMP_ID $f^T_{C_{Tt}}$] | [K_TEMP_H_FNC_H_ID $f^h_{K_{Th}}$] | [K_TEMP_TEMP_FNC_H_ID $f^h_{K_{TT}}$] | [K_TEMP_W_FNC_H_ID $f^h_{K_{Tw}}$] | [K_TEMP_GRAV_FNC_H_ID $f^h_{K_{Tgrav}}$] | [C_TEMP_H_FNC_H_ID $f^h_{C_{Th}}$] | [C_TEMP_TEMP_FNC_H_ID $f^h_{C_{TT}}$] | [C_TEMP_W_FNC_H_ID $f^h_{C_{Tw}}$] | [C_TEMP_T_FNC_H_ID $f^h_{C_{Tt}}$] | [K_TEMP_H_FNC_T_ID $f^t_{K_{Th}}$] | [K_TEMP_TEMP_FNC_T_ID $f^t_{K_{TT}}$] | [K_TEMP_W_FNC_T_ID $f^t_{K_{Tw}}$] | [K_TEMP_GRAV_FNC_T_ID $f^h_{K_{Tgrav}}$] | [C_TEMP_H_FNC_T_ID $f^t_{C_{Th}}$] | [C_TEMP_TEMP_FNC_T_ID $f^t_{C_{TT}}$] | [C_TEMP_W_FNC_T_ID $f^t_{C_{Tw}}$] | [C_TEMP_T_FNC_T_ID $f^t_{C_{Tt}}$] }$_+$]
[WATER
{ [D_H_H D^0_{wh}] | [D_H_TEMP D^0_{wT}] | [D_H_W D^0_{ww}] | [D_H_GRAV D^0_{wgrav}] | [C_H_H C^0_{wh}] | [C_H_TEMP C^0_{wT}] | [C_H_W C^0_{ww}] [C_H_T C^0_{wt}] | [D_H_H_FNC_H_ID $f^h_{D_{wh}}$] | [D_H_TEMP_FNC_H_ID $f^h_{D_{wT}}$] | [D_H_W_FNC_H_ID $f^h_{D_{ww}}$] | [D_H_GRAV_FNC_H_ID $f^h_{D_{wgrav}}$] | [C_H_H_FNC_H_ID $f^h_{C_{wh}}$] | [C_H_TEMP_FNC_H_ID $f^h_{C_{wT}}$] | [C_H_W_FNC_H_ID $f^h_{C_{ww}}$] | [C_H_T_FNC_H_ID $f^h_{C_{wt}}$] | [D_H_H_FNC_TEMP_ID $f^T_{D_{wh}}$] | [D_H_TEMP_FNC_TEMP_ID $f^T_{D_{wT}}$] |

[D_H_W_FNC_TEMP_ID $f^T_{D_{ww}}$] | [D_H_GRAV_FNC_TEMP_ID $f^T_{D_{wgrav}}$] |
[C_H_H_FNC_TEMP_ID $f^T_{C_{wh}}$] | [C_H_TEMP_FNC_TEMP_ID $f^T_{C_{wT}}$] |
[C_H_W_FNC_TEMP_ID $f^T_{C_{ww}}$] | [C_H_T_FNC_TEMP_ID $f^T_{C_{wt}}$] |
[D_H_H_FNC_T_ID $f^t_{D_{wh}}$] | [D_H_TEMP_FNC_T_ID $f^t_{D_{wT}}$] |
[D_H_W_FNC_T_ID $f^t_{D_{ww}}$] | [D_H_GRAV_FNC_T_ID $f^t_{D_{wgrav}}$] |
[C_H_H_FNC_T_ID $f^t_{C_{wh}}$] | [C_H_TEMP_FNC_T_ID $f^t_{C_{wT}}$] |
[C_H_W_FNC_T_ID $f^t_{C_{ww}}$] | [C_H_T_FNC_T_ID $f^t_{C_{wt}}$] }+]

Table 173: &Parameters of the & CCModelBaXi94 within the transport analysis

Parameter	Description
CONCRETE TYPE *n_type*	Type of concrete, resp. type of cement. *n_type* = <1..4>, *n_type* =1 for Portland cement etc. Default value: 1
RATIO_WC *ratio*	Water cement ratio. The allowed range is <0.3..0.7. Default value : 0.56
CEMENT_WEIGHT *cem_weight*	This parameter is used to account for moisture loss due to hydration. When the CCModelBaXi94 material model is used, *cem_weight* should be set 0, because the model takes hydration into account automatically. This option is prepared for some less elaborated material models that cannot deal with hydration moisture loss directly and the (Bazant and Thonguthai 1978; Bazant 1986) model should be used instead. For more information refer to the ATENA Theoretical Manual, section Transport analysis. Default value: 0
[K_TEMP_H *x*] [K_TEMP_TEMP *x*] [K_TEMP_W *x*] [K_TEMP_GRAV *x*]	Coefficients defining heat flux. The heat flux is computed by $-\overline{J_T} = [k_{Th}]\overline{\nabla h} + [k_{Tw}]\overline{\nabla w} + [k_{TT}]\overline{\nabla T} + \overline{k}_{T0}$, see the ATENA Theoretical manual. Usually, all these coefficients are zero, except $[k_{TT}]$=K_TEMP_TEMP=x. Default value: K_TEMP_TEMP = 2.1 W/C/m
[C_TEMP_H *x*] [C_TEMP_TEMP *x*] [C_TEMP_W *x*]	Coefficients defining heat material capacity. The $LHS_T = \frac{\partial}{\partial t}(C_T) = c_{Th}\frac{\partial h}{\partial t} + c_{Tw}\frac{\partial w}{\partial t} + c_{TT}\frac{\partial T}{\partial t} + c_{T0}$, see the ATENA Theoretical manual. Usually, all these coefficients are zero, except $[c_{TT}]$=C_TEMP_TEMP=x. Default value: C_TEMP_TEMP =2.55E6 [J/m^3/C].
[K_TEMP_H_FNC_ID *x*] [K_TEMP_TEMP_FNC_ID *x*] [K_TEMP_W_FNC_ID *x*]	All the above heat flux and capacity coefficients are constant with respect to state variables, i.e. humidity and temperature,

[K_TEMP_GRAV_FNC_ID *x*] [C_TEMP_H_FNC_ID *x*] [C_TEMP_TEMP_FNC_ID *x*] [C_TEMP_W_FNC_ID *x*]	but can vary in time. This is achieved by multiplying each of the above parameters by a time function. Ids of such a function are specified here. The whole concept is similar to time varying boundary conditions, parameters for material models in static etc. The time functions themselves are given by &FUNCTION.

Table 174: &Parameters of the & CCTransportMaterial within the transport analysis

Input parameters for user-defined constitutive law for flow governing equations

Heat :

$$\frac{\partial W}{\partial t} = -div(\underline{q}_w)$$

$$C_{Th}\frac{\partial h}{\partial t} + C_{TT}\frac{\partial T}{\partial t} + C_{Tw}\frac{\partial w}{\partial t} + C_{Tt} = div\left(K_{Th}\,grad(h) + K_{TT}\,grad(T) + K_{Tw}\,grad(w) + K_{Tgrav}\right)$$

Moisture :

$$\frac{\partial Q}{\partial t} = -div(\underline{q}_T)$$

$$C_{wh}\frac{\partial h}{\partial t} + C_{wT}\frac{\partial T}{\partial t} + C_{ww}\frac{\partial w}{\partial t} + C_{wt} = div\left(D_{wh}\,grad(h) + D_{wT}\,grad(T) + D_{ww}\,grad(w) + D_{wgrav}\right)$$

W, Q states for total amount of moisture per unit volume, [kg/m^3] and total amount of energy per unit volume, [J/m^3]. Note that positive value of C_{Tt}, C_{ht} causes consumption, so that e.g. hydration heat must be input as negative number. Input always a label followed by an associated real value, (for constant parameter) or integer id of a previously defined function, (for a function definition). If a parameter is skipped, it is assumed either zero or the associated function is assumed to have value 1, i.e. neglected. The T subscript for temperature related parameters is replaced by TEMP string. The subscripts for humidity, water content and time, i.e. sink related terms remain unchanged, i.e. H, W, t respectively. For example C_{TT} is entered as C_TEMP_TEMP etc. All functions are defined separately. Each such a definition is refered by its id, i.e. a integer number. This integer is then specified as a value following the appropriate label. For example the function $f^t_{C_{TT}}(t)$ is defined with id *k*. Then, the material data input would read C_TEMP_TEMP_FNC_ID *k.*

Significance of the parameters is as follows:

$$C_{Th} = C_{Th}^0 \, f_{C_{Th}}^h(h) \, f_{C_{Th}}^T(T) \; f_{C_{Th}}^t(t)$$
$$C_{TT} = C_{TT}^0 \, f_{C_{TT}}^h(h) \, f_{C_{TT}}^T(T) \; f_{C_{TT}}^t(t)$$
$$C_{Tw} = C_{Tw}^0 \, f_{C_{Tw}}^h(h) \, f_{C_{Tw}}^T(T) \, f_{C_{Tw}}^t(t)$$
$$C_{Tt} = C_{Tt}^0 \, f_{C_{Tt}}^h(h) \, f_{C_{Tt}}^T(T) \; f_{C_{Tt}}^t(t)$$
$$C_{wh} = C_{wh}^0 \, f_{C_{wh}}^h(h) \, f_{C_{wh}}^T(T) \; f_{C_{wh}}^t(t)$$
$$C_{wT} = C_{wT}^0 \, f_{C_{wT}}^h(h) \, f_{C_{wT}}^T(T) \; f_{C_{wT}}^t(t)$$
$$C_{ww} = C_{ww}^0 \, f_{C_{ww}}^h(h) \, f_{C_{ww}}^T(T) \; f_{C_{ww}}^t(t)$$
$$C_{wt} = C_{wt}^0 \, f_{C_{wt}}^h(h) \, f_{C_{wt}}^T(T) \; f_{C_{wt}}^t(T)$$

$$K_{Th} = K_{Th}^0 \, f_{K_{Th}}^h(h) \, f_{K_{Th}}^T(T) f_{K_{Th}}^t(t)$$
$$K_{TT} = K_{TT}^0 \, f_{K_{TT}}^h(h) \, f_{K_{TT}}^T(T) \, f_{K_{TT}}^t(t)$$
$$K_{Tw} = K_{Tw}^0 \, f_{K_{Tw}}^h(h) \, f_{K_{Tw}}^T(T) \, f_{K_{Tw}}^t(t)$$
$$K_{Tgrav} = K_{Tgrav}^0 \, f_{K_{Tgrav}}^h(h) \, f_{K_{Tgrav}}^T(T) \, f_{K_{Tgrav}}^t(t)$$
$$D_{wh} = D_{wh}^0 \, f_{D_{wh}}^h(h) \, f_{D_{wh}}^T(T) \, f_{D_{wh}}^t(t)$$
$$D_{wT} = D_{wT}^0 \, f_{D_{wT}}^h(h) \, f_{D_{wT}}^T(T) f_{D_{wT}}^t(t)$$
$$D_{ww} = D_{ww}^0 \, f_{D_{ww}}^h(h) \, f_{D_{ww}}^T(T) f_{D_{ww}}^t(t)$$
$$D_{wgrav} = D_{wgrav}^0 \, f_{D_{wgrav}}^h(h) \, f_{D_{wgrav}}^T(T) \, f_{D_{wgrav}}^t(t)$$

Default values: All functions are constant and equal to one, i.e. they are disregarded. All other parameters are by default zero with the following exceptions:

$$C_{hh} = 225 \frac{kg}{m^3}, \; D_{hh} = 1.5E-6 \frac{kg}{s\,m}$$
$$C_{TT} = 2.55E6 \; \frac{J}{m^3 C}, K_{TT} = 2.1 \frac{J}{s\,m\,C}$$

Table 175: &Parameters of the &CCTransportMaterialLevel7 within the transport analysis

Parameter	Description
DOH_FNC_ID *id*	Id of degree of hydration DoH(time) function. It prevails input of DOH25_FNC_ID and analytical calculation of DoH(time) using B1, B2, ALPHAINF and ETA.
DOH25_FNC_ID *id*	Id of degree of hydration DoH25(time) function, i.e. DoH function for reference temperature 25°C and relative humidity 1. It is

	overwriten by DOH_FNC_ID and prevails analytical calculation of DoH(time) using B1, B2, ALPHAINF and ETA
B1 *val*	B_1 hydration parameter, (see Atena Theory manual). Units: [time^{-1}] Default value: 0.5 hour^{-1}=0.0001389sec^{-1}
B2 *val*	B_2 hydration parameter, (see Atena Theory manual). Units: [-] Default value: 0.001
ALPHAINF *val*	Ultimate hydration degree α_∞ Units:[-] Default value: 0.85
ETA *val*	Microdiffusion of free water through formed hydrates $\overline{\eta}$ Units: [-] Default value: 7.
A *val*	Material parameter a in Eqn. to compute β_h reduction of capillary moisture. Units: [-] Default value: 7.5
QH_POT *val*	$Q_{h,pot}$ is potential hydration heat Units: [energy/kg of cement] Default value: 500000 J/kg of cement
QW_POT *val*	$Q_{w,pot}$ is potential hydration moisture consmption Units: [mass of water/mass of cement, i.e. unitless] Default value: 0.24 kg of water / 1kg of cement
TH_INIT *val*	Initial time t_{ini} for which α_{ini} has been calculated. Typically it is zero. Units: [time] Default value: 0 hour
ALPHA_INIT *val*	Initial value of α maturity factor. For fresh

	and hydrated concrete $\alpha = 0$, $\alpha = 1$, respectively. Typically it is zero. Units: [-] Default value: 0
TH_INCR_MIN *val*	Units: $\Delta t_{\min}$ minimum time increment for integration of α maturity factor Units: [time] Default value: 1 second
TH_INCR_MAX *val*	$\Delta t_{\max}$ maximum time increment for integration of α maturity factor Units: [time] Default value: 1 hour
TEMPERATURE_INCR_MAX *val*	Time increment for for integration of α maturity factor is calculated as follows: $\Delta t = \exp(0.03674066933\Delta T_{\max} + \log(t)$ $\Delta t_{\min} \le \Delta t \le \Delta t_{\max}$ TEMPERATURE_INCR_MAX *val* states for $\Delta T_{\max}$ parameter in the above equation. Units: [temperature] Default value: 0.1 ^{0}C
CEMENT_MASS *val*	Cement mass in concrete m_{cement}. Units: [mass] Default value: 161 kg
AGGREGATE_MASS *val*	Fine and coarse aggregeate mass in concrete $m_{aggregate}$. Units: [mass] Default value: 2086 kg
FILLER_MASS *val*	Filler mass in concrete m_{filler}. Units: [mass] Default value: 69 kg
CEMENT_DENSITY *val*	Cement density. Units: [mass/length^3] Default value: 3220 kg/m^3
WATER_DENSITY *val*	Water density. Units: [mass/length^3]

	Default value: 1000 kg/m^3
AGGREGATE_DENSITY *val*	Density of coarse and fine aggregate. Units: [mass/length^3] Default value: 2800 kg/m^3
FILLER_DENSITY *val*	Density of filler. Units: [mass/length$^{3]}$ Default value: 2400 kg/m^3
C_AGGREGATE_TEMP_TEMP *val*	Heat capacity of aggregate per unit volume $C_{aggregate}$. Units: [energy/(lenght^3 ^{0}C)] Default value: 2.352E6 J/(m^3 ^{0}C)
C_FILLER_TEMP_TEMP *val*	Heat capacity of filler per unit volume C_{filler}. Units: [energy/(lenght^3 ^{0}C)] Default value: 2.268E6 J/(m^3 ^{0}C)
C_CEMENT_TEMP_TEMP *val*	Heat capacity of cement per unit volume C_{cement}. Units: [energy/(lenght^3 ^{0}C)] Default value: 2.415E6 J/(m^3 ^{0}C)
C_WATER_TEMP_TEMP *val*	Heat capacity of water per unit volume C_{water}. Units: [energy/(lenght^3 ^{0}C)] Default value: 4.18E6 J/(m^3 ^{0}C)
K_AGGREGATE_TEMP_TEMP *val*	Heat conductivity of aggregate $\lambda_{aggregate}$. Units: [energy/(length time temperature)] Default value: 1.9 J/(m second ^{0}C)
K_FILLER_TEMP_TEMP *val*	Heat conductivity of filler λ_{filler} Units: [energy/(length time temperature)] Default value: 0.6 J/(m second ^{0}C)
K_CEMENT_TEMP_TEMP *val*	Heat conductivity of cement λ_{cement} Units: [energy/(length time temperature)] Default value: 1.55 J/(m second ^{0}C)

K_WATER_TEMP_TEMP *val*	Heat conductivity of water λ_{water} Units: [energy/(length time temperature)] Default value: 0.604 J/(m second ^{0}C)
K_AIR_TEMP_TEMP *val*	Heat conductivity of air λ_{air} Units: [energy/(length time temperature)] Default value: 0.035 J/(m second ^{0}C)
W_F *val*	Free water saturation w_f Units: [mass/length^3] Default value: 127 kg/m^3
H80 *val*	Relative humidity h_{80} for w_{80}. Units: [-] Default value: 0.8
W80 *val*	Water saturation w_{80} for h_{80}. Units: [mass/length^3] Default value: 40 kg/m^3
TEMP0 *val*	Parameter T_0 to calculate saturaturated water vapour pressure p_{sat} for temperatures $T \geq 0\,^0C$. Units: [temperature] Default value: 234.18 ^{0}C.
A_WV *val*	Parameter a to calculate saturated water vapour pressure p_{sat} for temperatures $T \geq 0\,^0C$. Units: [-] Default value: 17.08
A_W *val*	Water absorption coefficient *A*. Units: [mass/(length^{2}time$^{0.5}$)] Default value: 0.01 kg/(m^2 second$^{0.5}$)
MI_WV *val*	Water vapour diffusion resistance factor μ Units: [-] Default value: 210.
TEMP0_ICE *val*	Parameter T_0 to calculate saturatated water

	vapour pressure p_{sat} for temperatures $T < 0\ ^0C$ Units: [temperature] Default value: 272.44 ^{0}C.
A_WV_ICE *val*	Parameter a to calculate saturated water vapour pressure p_{sat} for temperatures $T < 0\ ^0C$. Units: [-] Default value: 22.44
EA *val*	Acxtivation energy E_a Units: [energy/mol] Default value: 38300 J/mol
All remaining input data in the sections TEMPERATURE and WATER:	They are the same as those for &CCTransportMaterial_PARAMS, except by default $C_{hh} = 0.0,\ D_{hh} = 0.$ $C_{TT} = 0.,\ K_{TT} = 0.$

3.14.2 Transport finite elements

The transport analysis uses different types of finite elements. They are input in exactly the same way as for static analysis. The following tables lists all transport analysis element. For each of the supported element the table below also presents name of corresponding a finite element for static analysis, which has the same geometry and nodal ids marking.

Table 176: Finite elements to transport analysis with Newton-Cotes integration.

Element	Description	Equivalent element for static analysis with the same geometry
IsoQuad4_2D ... IsoQuad9_2D	2D quadrilateral isoparametric elements	CCIsoQuad4_2D ... CCIsoQuad9_2D
IsoQuad4_Asym ... IsoQuad9_2ASym	Axisymmetric quadrilateral isoparametric elements	CCIsoQuad4_Asym ... CCIsoQuad9_ASym
IsoTriangle3_2D ...	2D triangular isoparametric elements	CCIsoTriangle3_2D ...

IsoTriangle6_2D		CCIsoTriangle6_2D
IsoTriangle3_ASym ... IsoTriangle6_ASym	Axisymmetric triangular isoparametric elements	CCIsoTriangle3_ASym ... CCIsoTriangle6_ASym
IsoBrick8_3D ... IsoBrick20_3D	Hexahedral isoparametric elements	CCIsoBrick8_3D ... CCIsoBrick8_3D
IsoWedge6_3D .... IsoWedge15_3D	Wedge isoparametric elements	CCIsoWedge6_3D .... CCIsoWedge15_3D
IsoTetra4_3D .... IsoTetra10_3D	Tetrahedral isoparametric elements	CCIsoTetra4_3D .... CCIsoTetra10_3D
IsoTruss2_2D, IsoTruss3_2D, IsoTruss2_3D, IsoTruss3_3D, IsoTruss2_ASym, IsoTruss3_ASym,	Truss isoparametric elements, 2D, 3D and axisymmetric.	CCIsoTruss2_2D, CCIsoTruss3_2D, CCIsoTruss2_3D, CCIsoTruss3_3D, CCIsoTruss2_ASym, CCIsoTruss3_ASym,

Table 177: Finite elements to transport analysis with Gaussian integration.

Element	Description	Equivalent element for static analysis with the same geometry
IsoQuadGauss4_2D ... IsoQuad Gauss 9_2D	2D quadrilateral isoparametric elements	CCIsoQuad4_2D ... CCIsoQuad9_2D
IsoQuad Gauss 4_Asym ... IsoQuad Gauss 9_2ASym	Axisymmetric quadrilateral isoparametric elements	CCIsoQuad4_Asym ... CCIsoQuad9_ASym
IsoTriangle Gauss	2D triangular isoparametric elements	CCIsoTriangle3_2D

3_2D ... IsoTriangle Gauss 6_2D		... CCIsoTriangle6_2D
IsoTriangle Gauss 3_ASym ... IsoTriangle Gauss 6_ASym	Axisymmetric triangular isoparametric elements	CCIsoTriangle3_ASym ... CCIsoTriangle6_ASym
IsoBrick Gauss 8_3D ... IsoBrick Gauss 20_3D	Hexahedral isoparametric elements	CCIsoBrick8_3D ... CCIsoBrick8_3D
IsoWedge Gauss 6_3D .... IsoWedge Gauss 15_3D	Wedge isoparametric elements	CCIsoWedge6_3D .... CCIsoWedge15_3D
IsoTetra Gauss 4_3D .... IsoTetra Gauss 10_3D	Tetrahedral isoparametric elements	CCIsoTetra4_3D .... CCIsoTetra10_3D

3.14.3 Transport initial values of state variables

Each transient analysis, the transport analysis included, needs to know initial values of the structural state variables prior any execution. This is achieved by the following commands:

Syntax:

```
&INITIAL_CONDITIONS:
NODAL { MAT_H_TEMP | MAT_TEMP_H| H_TEMP_MAT | TEMP_H_MAT |
        TEMPERATURE | HUMIDITY | MATERIAL} [SETTINGS] {
        &MANUAL_INITIAL_VALUES_ENTRY |
        &GENERATED_INITIAL_VALUES }

&MANUAL_INITIAL_VALUES_ENTRY:
{ NODE n |  TYPE type  | H h | W w | TEMP temp }+
```

Table 178: Nodal Initial Conditions Definition (manual entries)

Sub-Command	Description
NODE *n*	Set initial conditions for node *n*.
TYPE *type*	Specify type of material used in node *n*. Note that transport analysis is integrated in finite nodes rather than integration nodes in finite elements and hence material model is related to finite nodes (and not finite elements).
H *h* | W *w*	Set initial condition for relative humidity *h*. Moisture conditions can be equivalently also set by setting the amount of water content *w*, see the ATENA Theoretical manual for definition of *w*(.
TEMP *temperature*	Set initial temperature in the node [Kelvin]

&GENERATED_INITIAL_VALUES:

NODAL [SETTING] SELECTION "*selection_name*" { TYPE *type* | GENERATE_ H | GENERATE_W | GENERATE_TEMP | CONST *const* | COEFF_X *coeff_x* | COEFF_Y *coeff_y* | COEFF_Z *coeff_z*}+

Table 179: Nodal Initial Conditions Definition (generated entries)

Sub-Command	Description
SELECTION "*selection_name*"	Name of selection, for which the generation is requested.
TYPE *type*	Specify type of material used in nodes in the selection.
{GENERATE_H | GENERATE_W | GENERATE_TEMP }1 CONST *const* | COEFF_X *coeff_x* | COEFF_Y *coeff_y* | COEFF_Z *coeff_z*	Keyword for entities to be generated. The value is generated as linear combination: $value = const + x\ coeff_x + y\ coeff_y + z\ coeff_z$ *x,y,z* are coordinates of nodes, where the generation is processed

Example:

NODAL MAT_H_TEMP SETTING NODE 1 MATERIAL TYPE 1 H 1. TEMP 20

NODAL SELECTION "*my_selection"* GENERATE_TYPE 1

CONST 0.5 COEFF_X 0. COEFF_Y -0.6523648649 COEFF_Z 0. GENERATE_H

CONST -10. COEFF_X 0. COEFF_Y 0. COEFF_Z 0 GENERATE_T

3.14.4 Transport Set parameters

The transport analysis SET related input is specified via the ANALYSIS_TYPE subcommand.

Table 180: &ANALYSIS_TYPE sub-command parameters

Parameter	Description
&TRANSIENT	Set transient analysis and set some parameters for it.
&CONVERGENCE_CRITERIA	Convergence criteria for the transport analysis

&TRANSIENT:
TRANSIENT { [TIME] CURRENT *x* | [TIME] INCREMENT *x* |
TIME_INTEGRATION { {CRANK_NICHOLSON | THETA *x* }+ |
ADAMS_BASHFORTH } | REFERENCE_ETA *eta*}+

Table 181: ANALYSIS_TYPE subcommands for the transport analysis

Parameter	Description
[TIME] CURRENT *x*	Sets current time.
[TIME] INCREMENT *x*	Sets time increment in steps.
TIME_INTEGRATION	Set type of temporal integration scheme. If this parameter is not input, then CRANK_NICHOLSON integration will be used.
CRANK_NICHOLSON	Use linear trapezoidal integration.
THETA *x*	θ parameter for trapezoidal integration. By default θ = 0.5. Several other linear temporal integration may be utilized depending on theθ, e.g. implicit Newton integration for θ = 1, explicit integration for θ = 0 etc. For good compromise between convergence and possibility of oscillations values about θ = 0.85 is recommended.
ADAMS_BASHFORTH	Adams – Bashforth quadratic temporal integration.
REFERENCE_ETA *eta*	Damping factor. $\psi_{t+dt} = \psi_t + \eta \Delta \psi_{t+dt}$. $\eta =< 0.3..1 >$; $\eta = 1$ set totally un-damped analysis. Default: 1

&CONVERGENCE_CRITERIA:
{ ABSOLUTE [ERROR] | RELATIVE [ERROR] } | TEMPERATURE ERROR *x* |
HUMIDITY ERROR *x* | STEP_STOP_TEMPERATURE ERROR FACTOR *x* |
STEP_STOP_HUMIDITY ERROR FACTOR *x* |
ITER_STOP_TEMPERATURE ERROR FACTOR *x* | ITER_STOP_HUMIDITY

ERROR FACTOR *x* | NEGLIGIBLE_TEMPERATURE *x* | NEGLIGIBLE _HUMIDITY *x* }+

Table 182: &CONVERGENCE_CRITERIA sub-command parameters

Parameter	Description
ABSOLUTE [ERROR]	The convergence criteria values are computed using the absolute norm that is using the maximal element of an array in its absolute value. The error is then computed by dividing an iterative value with the value cumulated within the whole step.
RELATIVE [ERROR]	The convergence criteria values are computed using the Euclidean norm. The error is then computed by dividing an iterative value with the value cumulated within the whole step.
TEMPERATURE ERROR *x*	Convergence limit for absolute value of temperature increments. Default value is 0.01. E.g. TEMPERATURE ERROR *x*
HUMIDITY ERROR *x*	Convergence limit for absolute value of humidity increments. Default value is 0.01. E.g. HUMIDITY ERROR *x*
STEP_STOP_TEMPERATURE ERROR FACTOR *x* | STEP_STOP_HUMIDITY ERROR FACTOR *x* | ITER_STOP_TEMPERATURE ERROR FACTOR *x* | ITER_STOP_HUMIDITY ERROR FACTOR *x*	Factors for appropriate convergence criterion value. If a convergence criterion value multiplied by the appropriate factor exceeds the related calculated analysis error, then the execution is immediately killed. They are two sets of factors: the first one for checking each iteration and the other one to be exercised at the end of each step. The default value for iteration related factors is 1000, whilst the default value for step related factors is 10. E.g. SET Absolute Step_stop_humidity error factor 15. Step_stop_ temperature error factor 53 Iter_stop_ humidity error factor 201 Iter_stop_ temperature error factor 203 SET Relative Step_stop_ humidity error factor 54 Step_stop_ temperature error factor 56 Iter_stop_ humidity error factor 204 Iter_stop_ temperature error factor 206
NEGLIGIBLE_TEMPERATURE *x* | NEGLIGIBLE _HUMIDITY *x*	Values that are negligible, i.e. that can be ignored. By default they are set to 1.E-11. E.g. SET Absolute error Negligible_temperature 0.1 Relative error Negligible_temperature 0.2

3.14.5 The &HISTORY EXPORT command

The command forces ATENA to export data about humidity and temperature history at structural nodes. These data can be later imported into static analysis by the command &HISTORY_IMPORT.

Syntax:
&HISTORY_EXPORT:
HISTORY [{APPEND | OVERWRITE}] [EXPORT] [TO] [GEOMETRY
geometry_filename] | [RESULTS] results_filename] 2

Table 183: Transport analysis HISTORY_EXPORT command parameters

Parameter	Description
results_filename	Name of binary file with the history. It must be the same as that specified for HISTORY IMPORT command in the CCStructuresCreep module. It should be enclosed in double quote character (“).
geometry_filename	Name of binary file with geometry of the exported model. It must be the same as that specified for HISTORY IMPORT command in the CCStructuresCreep module. It should be enclosed in double quote character (“). If omitted, identical imported and current models are assumed.
[{APPEND | OVERWRITE }]	Open option for the file. By default, the file gets during execution overwritten.
[EXPORT] [TO]	Ignored keywords.

3.14.6 &Transport element load

The transport analysis supports the following types of element load:

- &BOUNDARY_ELEMENT_LOAD
- &BODY_ELEMENT_LOAD
- &FIRE_BOUNDARY LOAD
- &MOIST_TEMP_BOUNDARY_LOAD

3.14.6.1 The Sub-command &FIRE_BOUNDARY LOAD

&FIRE_BOUNDARY _LOAD:
FIRE_BOUNDARY [GROUP *group_id* [TO *group_id_to* [BY *group_id_by*]]
[ELEMENT { *element_id* [TO *element_id_to* [BY *element_id_by*]] |

SELECTION *list_name* }]] [COEFF *const*] [COEFF_X *coeff_x*] [COEFF_Y *coeff_y*] [COEFF_Z *coeff_z*] [[FIRE] [TYPE] { GENERIC | NOMINAL_HC, MODIFIED_HC }] [CONVECTION h_c] [EMISSIVITY ε_r]
[TEMPERATURE_MAX $T_{g,ref}$] [TEMPERATURE_MIN $T_{g,\min}$]
[TIME_FUNCTION *time_id*] [NODES "*boundary_nodes_list*"] [{EDGE | EDGE_NO_DUPLICATES} | SURFACE }] [MERGE [MERGE_STRING *str*]] [MULTIPLE {YES|NO}] [NO_ELEM_OUTPUT] [{ CONSTANT_IN_STEP | VARIABLE_IN_STEP | SEMIVARIABLE_IN_STEP }]

Important: Note that unlike other types of static loads (that are input in incremenental manner), the fire boundary load has character of a load potential and thus it must be input in total form. Therefore the load describes (total) fire load conditions !

Table 184: FIRE_BOUNDARY_LOAD parameters for element load

Parameter	Description
[FIRE] [TYPE] { GENERIC | NOMINAL_HC, MODIFIED_HC USER_CURVE	Type of fire load to be applied.
[CONVECTION h_c	Convection heat transfer coefficient [W/m2/K]. Default value 50 W/m^{2}K.
EMISSIVITY ε_r	Emissivity parameter. Default value 0.56.
TEMPERATURE_MAX $T_{g,ref}$	Max. temperature parameter.
[TIME_FUNCTION *time_id*	Id of an user-defined time dependent function. It acts as an extra multiplier of the generated or directly inputed fire boundary load.
TEMPERATURE_MIN $T_{g,\min}$	Ambient temperature prior the fire broke up. (Any generated temperature cannot fall below this value).
NODES "*boundary_nodes_list*"	List of boundary load that are load.
({EDGE | EDGE_NO_DUPLICATES } | SURFACE }	Type of boundary load, that is applicable for the given fire load. For more explanation see &BOUNDARY_ELEMENT_LOAD.
[MERGE [MERGE_STRING *str*]] [NO_ELEM_OUTPUT]	These parameteres are described in &BOUNDARY_ELEMENT_LOAD , where they are used in the same way.

{ CONSTANT_IN_STEP | VARIABLE_IN_STEP | SEMIVARIABLE_IN_STEP }	Set how the load should be treated: CONSTANT_IN_STEP = load values are calculated based on the model state at the beginning of step, VARIABLE_IN_STEP = load values are calculated based on the current model state (within each iteration), SEMIVARIABLE_IN_STEP = same as the above, but the load stiffness predictor neglects the load's variability. It (to some degree) degrades convergency but it may improve solution stability. It can turn out to be useful particularly for nonlinear elements.
[MULTIPLE {YES|NO}]	Allow the load to be applied to more surface/edges of one element

3.14.6.2 The Sub-command &MOIST_TEMP_BOUNDARY_LOAD

&MOIST_TEMP_BOUNDARY_LOAD:
MOIST_TEMP_BOUNDARY &ELEM_LOAD_DATA
&MOISTURE_FLUX_DUE_TO_RELATIVE_HUMIDITY_GRADIENT
&MOISTURE_ FLUX_DUE_TO_HUMIDITY_RATIO_GRADIENT
&MOISTURE_ FLUX_DUE_TO_CEMSTONE_CALC
&HEAT_ FLUX_DUE_TO _TEMPERATURE_GRADIENT
&HEAT_ FLUX_DUE_TO_EVAPORATED_MOISTURE
&COMMON_ MOIST_TEMP _BC_DATA [MERGE [MERGE_STRING *str*]]
[NO_ELEM_OUTPUT] { CONSTANT_IN_STEP | VARIABLE_IN_STEP |
SEMIVARIABLE_IN_STEP | [MULTIPLE {YES|NO}] }

&ELEM_LOAD_DATA: [GROUP *group_id* [TO *group_id_to* [BY *group_id_by*]]
[ELEMENT *element_id* [TO *element_id_to* [BY *element_id_by*]]]] |
SELECTION *list_name* }]] [COEFF *const*] [COEFF_X *coeff_x*] [COEFF_Y
coeff_y] [COEFF_Z *coeff_z*]

&MOISTURE_FLUX_DUE_TO_RELATIVE_HUMIDITY_GRADIENT:
[{ACCOUNT|NEGLECT} [GRADIENT] [OF] RELATIVE_HUMIDITY]
[CONVECTION_W h_{cw}]

&MOISTURE_ FLUX_DUE_TO_HUMIDITY_RATIO_GRADIENT:
[{ACCOUNT|NEGLECT} [GRADIENT] [OF] HUMIDITY_RATIO]
[EVAPORATION_MOISTURE Θ] [AIR_PRESSURE *p*] [AIR_VELOCITY
v] [AIR_VELOCITY_FUNCTION *air_velocity_fnc_id*]

&MOISTURE_ FLUX_DUE_TO_CEMSTONE_CALC:
[{ACCOUNT|NEGLECT} [GRADIENT] [OF]
HUMIDITY_CEMSTONE_CALC]

&HEAT_ FLUX_DUE_TO _TEMPERATURE_GRADIENT:
[{ACCOUNT|NEGLECT} [GRADIENT] [OF] TEMPERATURE]
[CONVECTION_T h_{cT}] [EMISSIVITY_T ε_{rT}]

&HEAT_ FLUX_DUE_TO_EVAPORATED_MOISTURE:
[{ACCOUNT|NEGLECT} [GRADIENT] [OF] EVAPORATED_MOISTURE]
[EVAPORATION_HEAT h_{we}]

&COMMON_ MOIST_TEMP _BC_DATA:
[AMBIENT_HUMIDITY h_g] [MOIST_FUNCTION *moist_fnc_id*]
[AMBIENT_TEMPERATURE T_g] [TEMP_FUNCTION *tempt_fnc_id*]
[NODES "*boundary_nodes_list*"] [{ {EDGE | EDGE_NO_DUPLICATES} | SURFACE}]

Important: Note that unlike other types of static loads (that are input in incremenental manner), the moisture-temperaturee boundary load has character of a load potential and thus it must be input in total form. Therefore the load describes (total) moisture-temperature load conditions !

Table 185: MOIST_TEMP_BOUNDARY_LOAD parameters for element load

Parameter	Description
AMBIENT_HUMIDITY h_g	Ambient air relative humidity, [-]. Default value: 0.6
AMBIENT_TEMPERATURE T_g	Ambient temperature, [^{o}C]. Default: 20 ^{o}C
CONVECTION_W h_{cw}	Convection moisture transfer coefficient [kg/s/m^2]. Default value 0. kg/s/m^2
EVAPORATION_MOISTURE Θ	Evaporation moisture transfer coefficient [kg/(m^{2}s)]. Default value `(25+19*v_)/(3600.)` kg/s/m^2, where `v_` is air velocity in ms^{-2}.
AIR_PRESSURE p	Total (absolute) ambient air pressure, [Pa], (=sum of partial dry air pressure and partial water vapour pressure). Default: normal pressure 101325Pa
AIR_VELOCITY v	Average ambient air velocity, [m/s], Default =0.m/s
CONVECTION_T h_{cT}	Convection heat transfer coefficient [W/m2/K]. Default value 20 W/m^{2}K.
EMISSIVITY_T ε_{rT}	Heat emissivity parameter, [-] Default value 0.85.

EVAPORATION_HEAT h_{we}	Evaporation heat transfer coefficient [J/kg]. Default: this coefficient is automatically set to consume 2270000 J per 1kg of evaporated water.
[MOIST_FUNCTION *moist_fnc_id*] [TEMP_FUNCTION *tempt_fnc_id*] [AIR_VELOCITY_FUNCTION *air_velocity_fnc_id*]	Id of an user-defined time dependent function for ambient moisture, ambient temperature and air velocity, respectively. It acts as an extra multiplier of the generated or directly inputed moisture-temperature boundary load.
[{ACCOUNT|NEGLECT} [GRADIENT] [OF] [RELATIVE_HUMIDITY] [TEMPERATURE] [HUMIDITY_RATIO] [EVAPORATED_MOISTURE] [HUMIDITY_CEMSTONE_CALC]	Acount for or neglect various kinds of moisture/heat flux contribution: RELATIVE_HUMIDITY - usual Darcy mositure flux due to gradient of relative humidities, TEMPERATURE - usual heat flux due to temperature gradient HUMIDITY_RATIO - moisture flux due to evaporation, i.e. due to gradient of air humidity ratio gradient, EVAPORATED_MOISTURE - heat flux due to flux of evaporated moisture CEMSTONE_CALC-moisture flux due to evaporation calculated according to http://www.cemstone.com/concrete-evaporation-forecast-engineers.cfm
NODES "*boundary_nodes_list*"	List of boundary load that are load.
({EDGE | EDGE_NO_DUPLICATES } | SURFACE }	Type of boundary load, that is applicable for the given fire load. For more explanation see &BOUNDARY_ELEMENT_LOAD.
[MERGE [MERGE_STRING *str*]] [NO_ELEM_OUTPUT]	These parameteres are described in &BOUNDARY_ELEMENT_LOAD , where they are used in the same way.
[MULTIPLE {YES|NO}]	Allow the load to be applied to more surface/edges of one element
{ CONSTANT_IN_STEP | VARIABLE_IN_STEP | SEMIVARIABLE_IN_STEP }	Set how the load should be treated: CONSTANT_IN_STEP = load values are calculated based on the model state at the beginning of step, VARIABLE_IN_STEP = load values are calculated based on the current model state (within each iteration), SEMIVARIABLE_IN_STEP = same as the above, but the load stiffness predictor neglects the load's variability. It (to some degree) degrades convergency but it may improve solution stability. It can turn out to be useful particularly for nonlinear elements.

3.14.7 &Transport analysis additional output data

In addition to standard output the transport analysis offers also the following output data

Table 186: Transport analysis related Output-type keywords understood by the command &OUTPUT for the location type NODES

Output keyword	Description
Q_W	Moisture nodal fluxes.
Q_T	Heat nodal fluxes.
CURRENT_PSI_VALUE	Current values of nodal state variables in nodes at time $t + \Delta t$. i.e. at the end of the current time step.
START_PSI_VALUE	Values of nodal state variables in nodes at time *t,* i.e. at the start of the current time step.

Table 187: Transport analysis related Output-type keywords understood by the command &OUTPUT for the location type NODES

Output keyword	Description
TRANSPORT_CONVERGENCE_CRITERIA	Parameters for assessing convergence performance of the transport analysis.

4 SAMPLE INPUT FILE

4.1 Input file for a sample static analysis.

```
/*

Sample analysis:
-----------------
Analysis of a simple 2D wall comprising quadrilateral and
triangle
elements, subject to displacement load at nodes 600 and 700.

Nodal pairs 300-800 and 200-500 are constrained to have the
same displacements.
The analysis has several "dummy" entities in order to test
deletion
process in ATENA input file.

        |y
        |
        |
        |               300
        |400            800                    700
   -x-> ---------------------------------------| -> 3.33e-6 *
        |                  |                 /|     function 20
        |                  |               /  |
        |                  |     20     /     |
        |                  |          /       |
        |     10           |        /         |
       ^|                  |      /           |
       ||                  |    /       15    |
       x|                  |  /               |    3.33e-6 *
       ||                  |/                 | ->  function 20
   -x-> ---------------------------------------------------------
-
        100               200                600
        (50)              500


                                                                x
*/


// Testing input data format
TASK name "Test"
TITLE "Test MASTER-SLAVE"
DIMENSION 2

// Coordinate definition
JOINT COORDINATES
 50  0. 0.  // dummy object for deletion checking
```

```
100  0.  0.
200  0.1 0.
700  0.2 0.1
300  0.1 0.1
500  0.1 0.
400  0.  0.1
600  0.2 0.
800  0.1 0.1

// Material definition
MATERIAL ID 71 NAME "Steel" TYPE "CCPlaneStressElastIsotropic"
E 210000 mu 0.2 rho 0.0023 alpha 1.2e-5
MATERIAL ID 70 NAME "Steel" TYPE "CCPlaneStressElastIsotropic"
E 210000 mu 0.2 rho 0.0023 alpha 1.2e-5     // dummy object
for deletion checking

// Geometry definition
GEOMETRY ID 81 Name "Steel thickness" TYPE "2D" thickness 0.1
GEOMETRY ID 80 Name "Steel thickness" TYPE "2D" thickness 0.1
// dummy object for deletion checking

// Element type definition, Should be referred from ELEMENT
GROUP
// definition
ELEMENT TYPE ID 92 NAME "Stupid 2D Triangle #1" TYPE
"CCIsoTriangle<xxx>"
ELEMENT TYPE ID 91 NAME "Stupid 2D Quad #1" TYPE
"CCIsoQuad<xxxx>"
ELEMENT TYPE ID 90 NAME "Stupid 2D Quad #1" TYPE
"CCIsoQuad<xxxx>" // dummy object for deletion checking

// Element group definition
ELEMENT GROUP ID 500 TYPE 90 NODES 4 MATERIAL 70 GEOMETRY 80
ELEMENT INCIDENCES      // dummy object for deletion checking
10  100  200  300  400

ELEMENT GROUP ID 2000 TYPE 92 NODES 3 MATERIAL 71 GEOMETRY 81
ELEMENT INCIDENCES
20  500 700 800
10  500 600 700
15  100 200 300       // dummy object for deletion checking

ELEMENT GROUP ID 1000 TYPE 91 NODES 4 MATERIAL 71 GEOMETRY 81
ELEMENT INCIDENCES
10  100  200  300  400

// Load function definition
FUNCTION ID 20 NAME "Load function" TYPE
"CCMultiLinearFunction" XVALUES 0. 2. YVALUES 0. 1.
FUNCTION ID 10 NAME "Load function" TYPE
"CCMultiLinearFunction" XVALUES 0. 1. YVALUES 1. 1.

// Load case 60 definition
```

```
LOAD CASE ID 60 NAME "Supports"     // dummy object for
deletion checking
SUPPORT SIMPLE
node 100 dof 1 value 0.0
node 100 dof 2 value 0.0
400 dof 1 value 0.0

// Load case 61 definition
LOAD CASE ID 61 NAME "Supports"
SUPPORT SIMPLE
node 100 dof 1 value 0.0
node 100 dof 2 value 0.0
node 400 dof 1 value 0.0

// Load case 63 definition
LOAD CASE ID 63 NAME "Loads"
SUPPORT SIMPLE node 600 dof 1 VALUE 3.33e-6     FUNCTION 20
SUPPORT SIMPLE node 700 dof 1 value 3.33e-6     FUNCTION 20

// Load case 62 constraints
LOAD CASE ID 62 NAME "Constraints"
SUPPORT COMPLEX
MASTER node 200 dof 1 * 1.0 SLAVE node 500 dof 1 value 0.0
MASTER node 200 dof 2 * 1.0 SLAVE node 500 dof 2 value 0.0
MASTER node 300 dof 1 * 1.0 SLAVE node 800 dof 1 value 0.0
MASTER node 300 dof 2 * 1.0 SLAVE node 800 dof 2 value 0.0

// SUPPORT MASTER SLAVE NODAL PAIRS  5  2   8  3

// Set analysis options/switches
SET Static
SET Newton-Raphson
SET Displacement error 0.01
SET Residual error 0.01
SET Absolute residual error 0.1
SET Iteration limit 20

// Testing of deletion
DELETE ELEMENT GROUP 500
DELETE JOINT 50
DELETE ELEMENT GROUP 2000 ELEMENT 15
DELETE GEOMETRY 80
DELETE ELEMENT TYPE 90
DELETE MATERIAL 70
DELETE LOAD CASE ID 60
DELETE FUNCTION 10

// Apply 1 load steps
STEP ID 31 STATIC NAME "Step 1" LOAD CASE  61 * 1.0  62 * 1.0
63 * 1.0 EXECUTE
OUTPUT LOCATION GLOBAL DATA ALL
OUTPUT LOCATION ELEMENT INTERNAL POINTS
  group from 1000 to 1000 element from 10 to 20 ip from 1 to 4
```

```
      group from 2000 to 2000 element from 10 to 20 ip from 1
    to 3
      DATA ALL
OUTPUT LOCATION ELEMENT NODES DATA ALL
OUTPUT LOCATION ELEMENT DATA ALL
OUTPUT LOCATION NODAL DATA ALL
OUTPUT LOCATION LOAD CASE DATA ALL
/* end of file */
```

4.2 Input file for a sample transport analysis

/*

Testing input data format - LHS and RHS boundary conditions; their values and sign.

(for 3D version see transp2_bricks_test.inp)

Structure:

2D structure of vertical quadrilaterals

Total dimension width*thickness*height=0.15*10.*1.

Discretisation: 4 elements per height, one ter width

Location: left bottom node (x,y)=(0,0), top right node (x,y)=(0.15,1.)

Loading (per step): vertical flux of heat (to the bottom)

Initial condition: dT/dy= -20/1.= -20; dT/dx=0; dh/... irrelevant, h=fixed everywhere

Flux: qy = K_TEMP_TEMP * dT/dy = 103680 * -20 = -2073600

External forces: sum(Q) = qy*width*thick= -2073600 * 0.15 * 10. = 3110400

Individual force: Q = sum(Q)/2 = 3110400/2 = 1555200

Sign of internal and external forces:

Internal forces: positive value corresponds to the flow in direction of outwards normal to the boundary surface

External load: positive value corresponds to the flow in direction of inwards normal to the boundary surface

In the example below:

dT/dy = negative ...-> flow to the bottom; i.e. in direction -y.

top surface (nodes 9,10), i.e. y=1 .... internal forces negative, i.e. -1555200; external load positive, i.e. 1555200

bottom surface (nodes 1,2), i.e. y=0 .... internal forces positive, i.e. 1555200; external load negative, i.e. -1555200

ALL EXTERNAL LOADS as well as NON_ZERO LHS BCs (i.e. fixing psi, h) HAVE INCREMENTAL CHARACTER.

This means that e.g. LOAD SIMPLE SELECTION "all9-10" dof 2 const 1555200. applied to all steps

will produce external forces 1555200. in the 1st step, 3110400. in the 2nd step.... The same applies to nonzero SUPPORT SIMPLE ..... specification.

To steps are applied:

step 1 .... see the load level defined above, (load_case 1)

step 2 .... doubles the above load, (load_case 2 (using "deformation" load increment)

or load_case 3 (using "nodal force" load increment)

or load_case 4 (using boundary load increment)

Use any one of load_case 2-4 to achieve the same loading

Initial conditions for the example:

NODAL SETTING

NODE 1 MATERIAL TYPE 1 H 1. TEMP 20

NODE 2 MATERIAL TYPE 1 H 1. TEMP 20

NODE 3 MATERIAL TYPE 1 H 1. TEMP 25

NODE 4 MATERIAL TYPE 1 H 1. TEMP 25

NODE 5 MATERIAL TYPE 1 H 1. TEMP 30

NODE 6 MATERIAL TYPE 1 H 1. TEMP 30

NODE 7 MATERIAL TYPE 1 H 1. TEMP 35

NODE 8 MATERIAL TYPE 1 H 1. TEMP 35

NODE 9 MATERIAL TYPE 1 H 1. TEMP 40

NODE 10 MATERIAL TYPE 1 H 1. TEMP 40

Boundary conditions:

SELECTION "all" list 1 2 3 4 5 6 7 8 9 10;

```
SELECTION "all3-8" list 3 4 5 6 7 8 ;
SELECTION "all9-10" list 9 10 ;
SELECTION "all1-2" list 1 2 ;

SUPPORT SIMPLE SELECTION "all" dof 1 const 0.     // fix h
SUPPORT SIMPLE SELECTION "all3-8" dof 2 const 0.     // fix T
LOAD SIMPLE SELECTION "all9-10" dof 2 const 1555200.     // fix T
LOAD SIMPLE SELECTION "all1-2" dof 2 const -1555200.     // fix T

Equivalent BC (compared only for ONE step of analysis!!!)
SUPPORT SIMPLE SELECTION "all" dof 1 const 0.     // fix h
SUPPORT SIMPLE SELECTION "all" dof 2 const 0.     // fix T

*/

TASK name "Test analysis for RHS and LHS BCs"
TITLE "2D quadrilateral in Y direction with vertical flux of heat to the bottom"
DIMENSION 2

// Set analysis options/switches
SET Static
SET Newton-Raphson
//SET Full_NR
SET Absolute Displacement error 0.00000001
SET Absolute Residual error 0.00000001
SET Displacement error 0.00000001
SET Residual error 0.00000001

// SET Optimize band width
SET TRANSIENT TIME CURRENT 0. INCREMENT 0.00069
SET TRANSIENT TIME_INTEGRATION CRANK_NICHOLSON THETA 1.0
//SET REFERENCE_ETA 0.8

// Coordinate definition
JOINT COORDINATES   // 4 elements 0.15*0.25 placed vertically
```

```
    1  0.   0.
    2  0.15 0.
    3  0.   0.25
    4  0.15 0.25
    5  0.   0.5
    6  0.15 0.5
    7  0.   0.75
    8  0.15 0.75
    9  0.   1.
   10  0.15 1.

// Material definition
MATERIAL ID 1 NAME "Baxant-Xi"
  TYPE "CCModelBaXi94"
  CONCRETE
    CONCRETE TYPE 1
   RATIO_WC 0.5
    CEMENT_WEIGHT 0.27
  TEMPERATURE
    K_TEMP_TEMP 103680
   C_TEMP_TEMP 0.000008

// initial values for psi
NODAL SETTING  // temperature gradient dT/dy=-20.
NODE    1  MATERIAL TYPE 1  H 1.  TEMP 20
NODE    2  MATERIAL TYPE 1  H 1.  TEMP 20
NODE    3  MATERIAL TYPE 1  H 1.  TEMP 25
NODE    4  MATERIAL TYPE 1  H 1.  TEMP 25
NODE    5  MATERIAL TYPE 1  H 1.  TEMP 30
NODE    6  MATERIAL TYPE 1  H 1.  TEMP 30
NODE    7  MATERIAL TYPE 1  H 1.  TEMP 35
NODE    8  MATERIAL TYPE 1  H 1.  TEMP 35
NODE    9  MATERIAL TYPE 1  H 1.  TEMP 40
NODE   10  MATERIAL TYPE 1  H 1.  TEMP 40
```

```
// Geometry definition
GEOMETRY ID 1 Name "Concrete column" TYPE "2D" thickness 10.

// Element type definition, Should be referred from ELEMENT GROUP
// definition
ELEMENT TYPE ID 1 NAME "2D Iso quadratic" TYPE "IsoQuad<xxxx>"

// Element group definition
ELEMENT GROUP ID 1 TYPE 1 MATERIAL 1 GEOMETRY 1

ELEMENT INCIDENCES
   1    1    2    4    3
   2    3    4    6    5
   3    5    6    8    7
   4    7    8   10    9

SELECTION "all" list 1 2 3 4 5 6 7 8 9 10;
SELECTION "all3-8" list 3 4 5 6 7 8 ; // intermediate nodes
SELECTION "all9-10" list 9 10 ;  // top surface
SELECTION "all1-2" list 1 2 ;    // bottom surface



// Steady state boundary conditions
LOAD CASE ID 1 NAME "LC-1"  // for fixed nodes dT/dx=-20 from initial conditions and
equivalent external load
SUPPORT SIMPLE SELECTION "all" dof 1 const 0.    // fix h
SUPPORT SIMPLE SELECTION "all3-8" dof 2 const 0.    // fix T
LOAD SIMPLE SELECTION "all9-10" dof 2 const 1555200.    // fix T
LOAD SIMPLE SELECTION "all1-2"  dof 2 const -1555200.    // fix T

// initialisation
STEP ID 1  STATIC  NAME "BCs and load" LOAD CASE  1 * 1.0  EXECUTE
OUTPUT LOCATION NODES  DATA LIST "Q_T" "CURRENT_PSI_VALUES"
"EXTERNAL_FORCES" "INTERNAL_FORCES" "REACTIONS" END
```

```
// break "Execute 2nd step to obtain dT/dx=2* (-20)" ;


// load alternative 1 - additional temperature increment induced solely by dT/dy
LOAD CASE ID 2 NAME "LC-2 -additional temperature increment" // total dT/dx=-40, i.e. increment at fixed nodes -20, (incr. of Q already in LC-1)
SUPPORT SIMPLE SELECTION "all3-8" dof 1 const 0.  // fix h; not all DOFs fixed to avoid case of no structural DOFs
SUPPORT SIMPLE NODE 1 DOF 2 VALUE  0
SUPPORT SIMPLE NODE 2 DOF 2 VALUE  0
SUPPORT SIMPLE NODE 3 DOF 2 VALUE  5
SUPPORT SIMPLE NODE 4 DOF 2 VALUE  5
SUPPORT SIMPLE NODE 5 DOF 2 VALUE 10
SUPPORT SIMPLE NODE 6 DOF 2 VALUE 10
SUPPORT SIMPLE NODE 7 DOF 2 VALUE 15
SUPPORT SIMPLE NODE 8 DOF 2 VALUE 15
SUPPORT SIMPLE NODE 9 DOF 2 VALUE 20
SUPPORT SIMPLE NODE 10 DOF 2 VALUE 20 ;


// load alternative 2 - additional temperature increment induced by dT/dy and dQ at the top and bottom
LOAD CASE ID 3 NAME "LC-2 -additional temperature increment" // total dT/dx=-40, i.e. increment at fixed nodes -20, (incr. of Q already in LC-1)
SUPPORT SIMPLE SELECTION "all" dof 1 const 0.     // fix h
SUPPORT SIMPLE NODE 3 DOF 2 VALUE  5
SUPPORT SIMPLE NODE 4 DOF 2 VALUE  5
SUPPORT SIMPLE NODE 5 DOF 2 VALUE 10
SUPPORT SIMPLE NODE 6 DOF 2 VALUE 10
SUPPORT SIMPLE NODE 7 DOF 2 VALUE 15
SUPPORT SIMPLE NODE 8 DOF 2 VALUE 15
LOAD SIMPLE SELECTION "all9-10" dof 2 const 1555200.    // fix T
LOAD SIMPLE SELECTION "all1-2"  dof 2 const -1555200. ; // fix T


// load alternative 3 - additional temperature increment induced by dT/dy and dQ at the top and bottom
LOAD CASE ID 4 NAME "LC-2 -additional temperature increment" // total dT/dx=-40, i.e. increment at fixed nodes -20, (incr. of Q already in LC-1)
SUPPORT SIMPLE SELECTION "all" dof 1 const 0.     // fix h
```

SUPPORT SIMPLE NODE 3 DOF 2 VALUE 5

SUPPORT SIMPLE NODE 4 DOF 2 VALUE 5

SUPPORT SIMPLE NODE 5 DOF 2 VALUE 10

SUPPORT SIMPLE NODE 6 DOF 2 VALUE 10

SUPPORT SIMPLE NODE 7 DOF 2 VALUE 15

SUPPORT SIMPLE NODE 8 DOF 2 VALUE 15

LOAD BOUNDARY group 1 TO 1 BY 1 VALUE DOF 2 2073600 NODES "all9-10"

LOAD BOUNDARY group 1 TO 1 BY 1 VALUE DOF 2 -2073600 NODES "all1-2" ;

//STEP ID 2 STATIC NAME "BCs and load" LOAD CASE 2 * 1.0 EXECUTE // step execute command for the load alternative 1

//STEP ID 2 STATIC NAME "BCs and load" LOAD CASE 3 * 1.0 EXECUTE // step execute command for the load alternative 2

STEP ID 2 STATIC NAME "BCs and load" LOAD CASE 4 * 1.0 EXECUTE // step execute command for the load alternative 3

OUTPUT LOCATION NODES DATA LIST "Q_T" "CURRENT_PSI_VALUES" "EXTERNAL_FORCES" "INTERNAL_FORCES" "REACTIONS" END

/* End of File */

5 ATENA INPUT FILE KEYWORDS

1

2

3

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z