Cervenka Consulting Ltd.
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 4-3

Tutorial for
Construction Process Modelling
in ATENA 2D

Written by
Zdeněk Janda, Jan Červenka

Prague, March 9, 2009

TABLE OF CONTENTS

1. INTRODUCTION

The objective of this tutorial is to show how the graphical environment of ATENA 2D can be used to model the construction process. The finite element solution core of ATENA supports the possibility to add or remove groups of finite elements. This feature can be used to model the construction process in ATENA. The graphical environment of ATENA 2D does not include a direct support for this feature, but the program environment is flexible enough that the user can exploit this functionality of the finite element core directly.

This feature can be exploited by direct editing of the ATENA input file and it will be demonstrated in this manual on an example of a small slab (see Figure 1-1), which is composed of two parts, which are not constructed at once. First, one part is made and a load is applied on top of it, which simulates the weight of the second part before the concrete is hardened. Then, the second part is added onto the deformed structure. Both structural parts are made from class 25/30 concrete. After 8 load steps, the top half is added and it will interact with the already present bottom part.

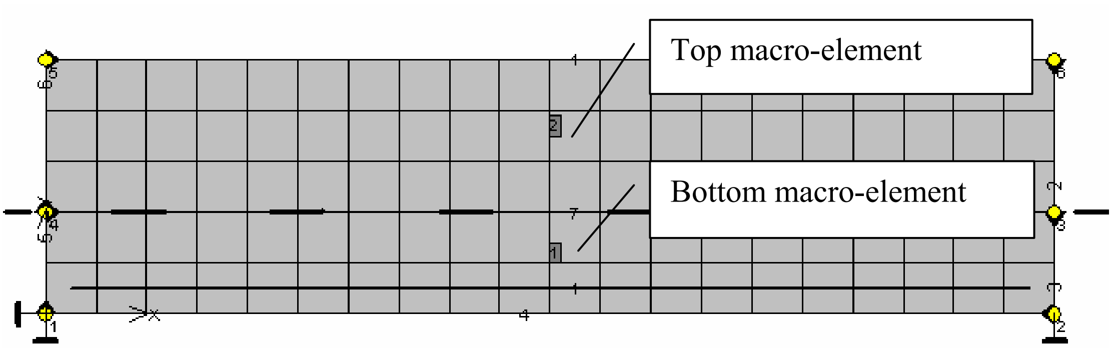


Figure 1-1: Model with two macro-elements. (LC1)

The basic idea of the construction process modeling in ATENA is the following: it is possible at any time to add or remove finite element groups. One possible approach would be to completely remove the finite elements of the top macro-element from the ATENA input file before executing the first step, and then add them later to the input file for the step 9. However, in many cases it is preferable to keep all the elements in the model throughout the whole analysis, and only to replace their material properties. In the steps, where these elements should not contribute to the structural stiffness, it is better to keep them in the model, but assume very soft material properties for them. If this approach is adopted the displacements of the finite element nodes forming the top part of the slab will follow the deflection of the bottom part, which is in a better agreement with the expected behavior. Then in the step 9, when the top part of the slab will start to interact with the bottom one, the element group of the top macro-element is removed and added again with new material properties.

2. GEOMETRY AND BOUNDARY CONDITIONS

First, it is necessary to construct the model of whole structure. Two separate macro-elements will be created for the first and second construction steps (see Figure 1-1).

Then, the load cases will be created for supports and loads. All together 4 load cases will be needed (Figure 2-1, Figure 2-2).

LC1 supports: this load case is used to define the basic boundary conditions to support the slab from the bottom.

LC2 force 1 construction step: this load case contains a distributed load along the top edge of the bottom macro-element. It represents the weight, which will be acting in the first construction step, i.e. on the bottom part of the slab.

LC3 force 2 construction step: this load case contains a distributed load along the top edge of the top macro-element. It represents the structural load acting on the final structure, after the top part of the slab is constructed.

LC4 support for top face of added part: this load case is basically small modeling trick to force the top nodes to remain horizontal. Without this boundary condition, the top nodes will deform following the deflection of the bottom slab. In some cases, this might be a required behavior, but in this case, it is assumed that during the concreting of the top slab, its top surface is leveled to a form a perfect horizontal surface (Figure 2-3).

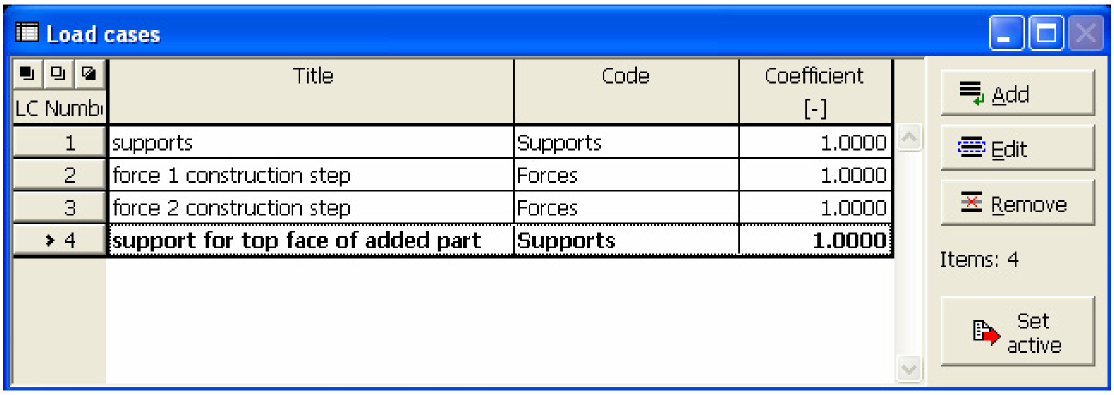


Figure 2-1: Load cases

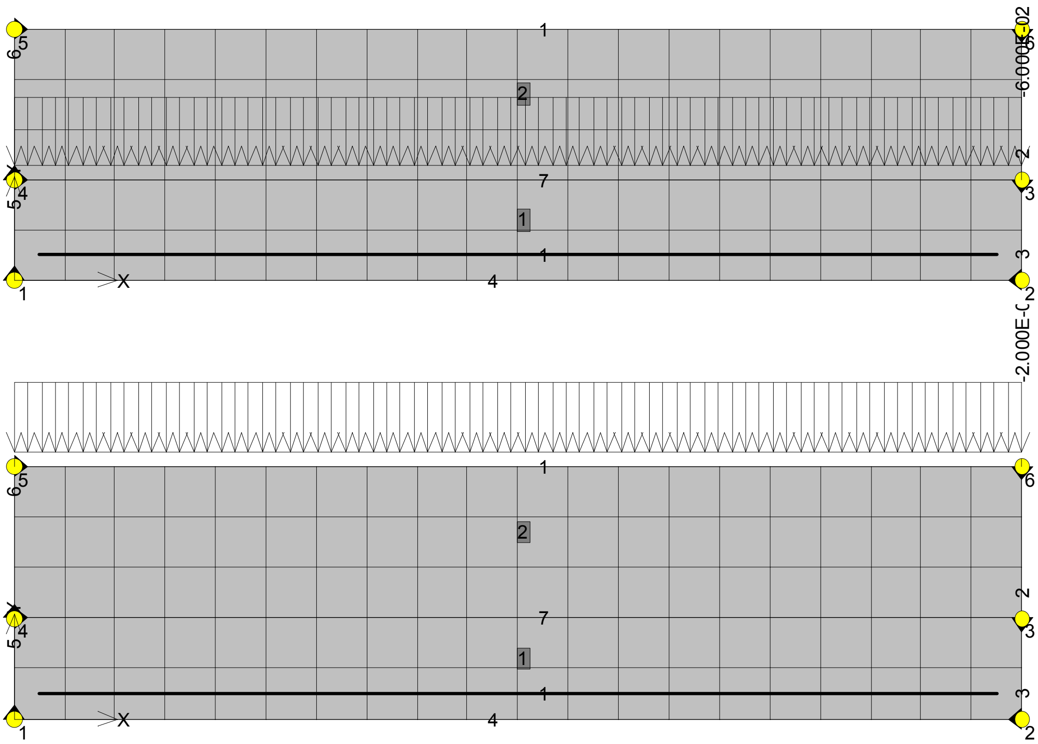


Figure 2-2: Load cases a) First construction step (LC2), b) second construction step (LC3)

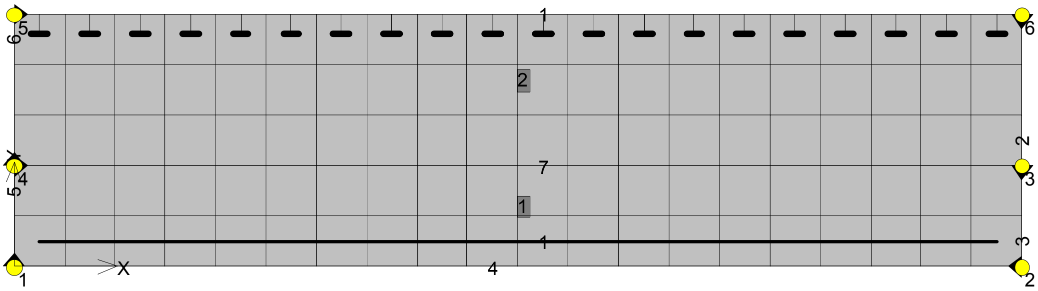


Figure 2-3: Boundary condition for added new part of construction (LC 4)

3. Material and Analysis Steps

In steps 1 – 8, only the bottom half of the structure should be present. For this steps we need to define a different material for the top macroelement (No. 2), a very soft one to behave almost like no material. We define a new Plane Stress Elastic Isotropic material (Figure 3-1) with Elastic modulus E = 1.000E-01 [MPa], i.e. about 1/10 000 of the modulus of the real material.

Steps 9-12 correspond to the second construction step, where both parts are present and subject to load (Figure 3-5).

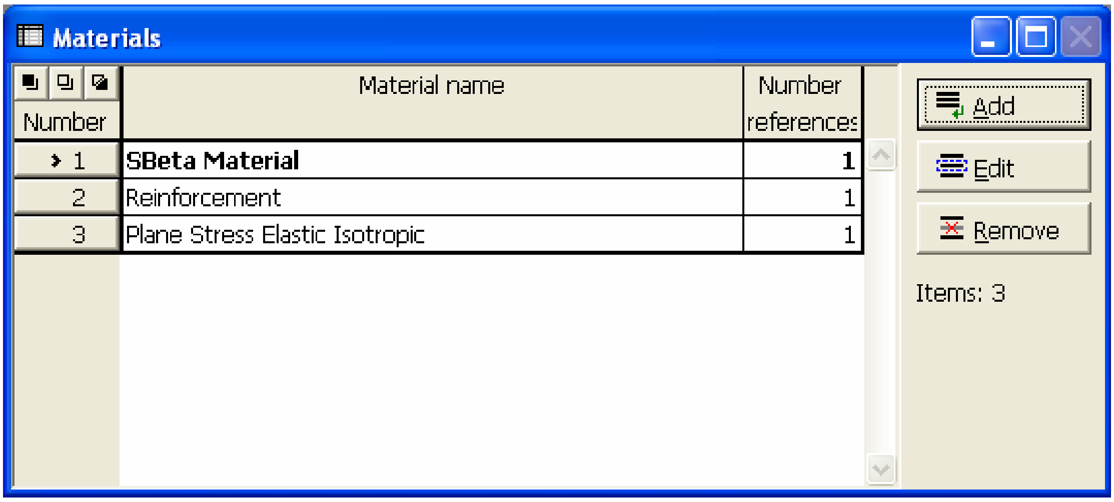


Figure 3-1: Materials

```
Material n. 1
 Name : SBeta Material
 Type: CCSBETAMaterial
 Elastic modulus E = 3.032E+04 [MPa]
 Poisson''s ratio sm = 0.200 [-]
 Tensile strength F_t = 2.317E+00 [MPa]
 Compressive strength F_c = -2.550E+01 [MPa]
 Type of tension softening : Exponential
 Specific fracture energy G_f = 5.793E-05 [MN/m]
 Crack model: Fixed
 Compressive strain at compressive strength in the uniaxial compressive test Eps_C = -1.682E-03 [-]
 Reduction of compressive strength due to cracks CompRed = 0.800 [-]
 Type of compression softening : Crush Band
 Critical compressive displacement Wd = -5.0000E-04 [m]
 Shear Retention Factor Variable
 Tension-compression interaction : Linear
 Specific material weight Rho = 2.300E-02 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]
```

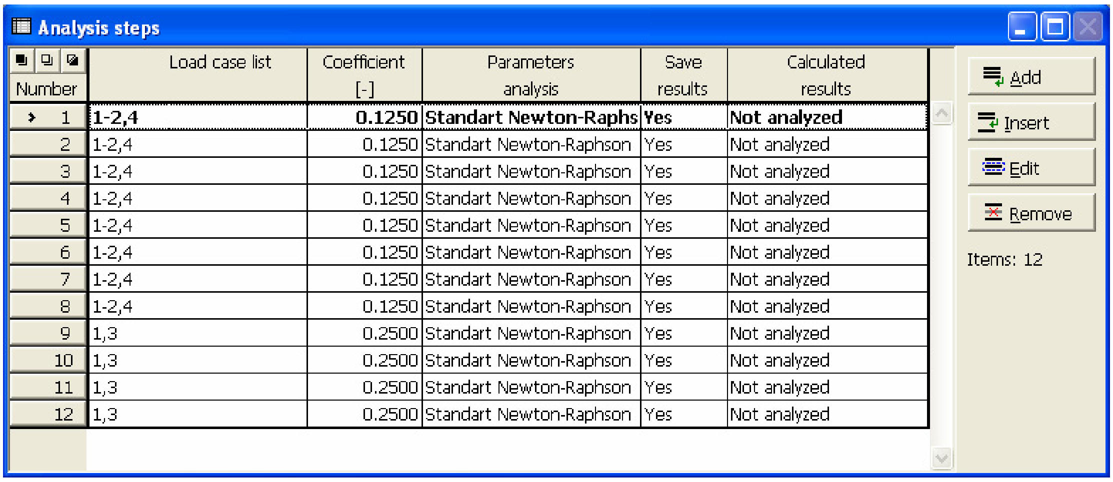
Figure 3-2: Material Sbeta for concrete

```
Material n. 2
 Name : Reinforcement
 Type: CCReinforcement
 Typ: BiLinear
 Elastic modulus E = 2.100E+05 [MPa]
 Sigma Y = 210.000 [MPa]
 Specific material weight RHO = 7.850E-02 [MN/m3]
 Coefficient of thermal expansion ALPHA = 1.200E-05 [1/K]
```

Figure 3-3: Material Reinforcement

```
Material n. 3
 Name : Plane Stress Elastic Isotropic
 Type: CCPlaneStressElastIsotropic
 Elastic modulus E = 1.000E-01 [MPa]
 Poisson''s ratio sm = 0.300 [-]
 Specific material weight Rho = 2.300E-02 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]
```

Figure 3-4: Soft material for the top macro-element in steps 1-8



Number	Load case list	Coefficient [-]	Parameters analysis	Save results	Calculated results
1	1-2,4	0.1250	Standart Newton-Raphs	Yes	Not analyzed
2	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
3	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
4	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
5	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
6	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
7	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
8	1-2,4	0.1250	Standart Newton-Raphson	Yes	Not analyzed
9	1,3	0.2500	Standart Newton-Raphson	Yes	Not analyzed
10	1,3	0.2500	Standart Newton-Raphson	Yes	Not analyzed
11	1,3	0.2500	Standart Newton-Raphson	Yes	Not analyzed
12	1,3	0.2500	Standart Newton-Raphson	Yes	Not analyzed

Figure 3-5: Analysis steps

4. RUNNING ANALYSIS

It will be necessary to edit the ATENA input file before executing the nonlinear analysis. The input file editing must be first activated by selecting the menu "**Option**" , "**Settings**". In this dialog the checkbox "**Enable input file editing before starting FE analysis**" (see Figure 4-1) must be selected prior the executing the analysis. Then it is possible to run the analysis by clicking the button "**Analysis**".

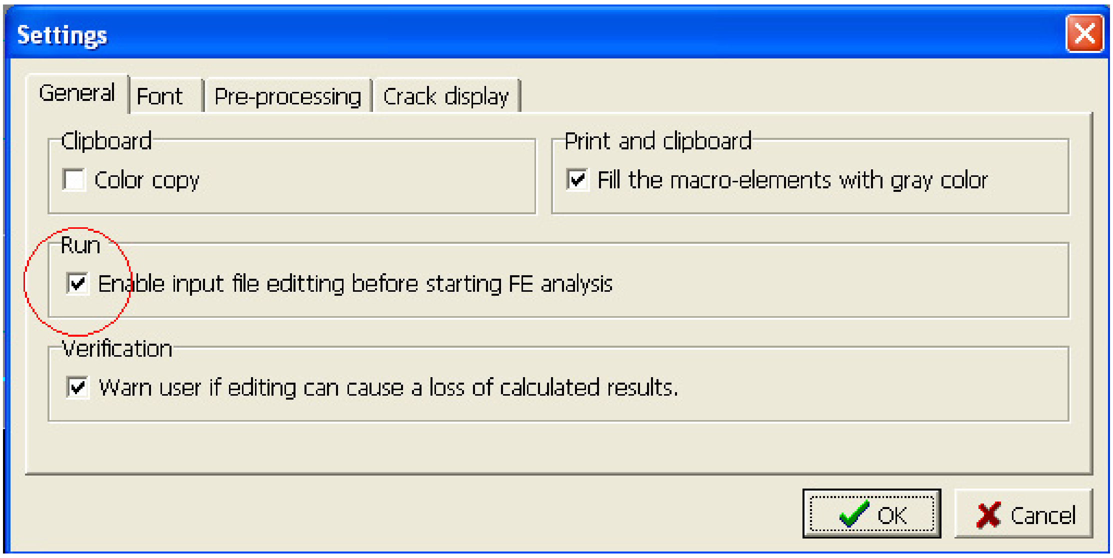


Figure 4-1: Option, Settings

When the run button is selected the following dialog appears (see Figure 4-2). In this dialog a user must select the checkbox "**Edit ATENA analysis input file**" and then click the button "**Analyse**".

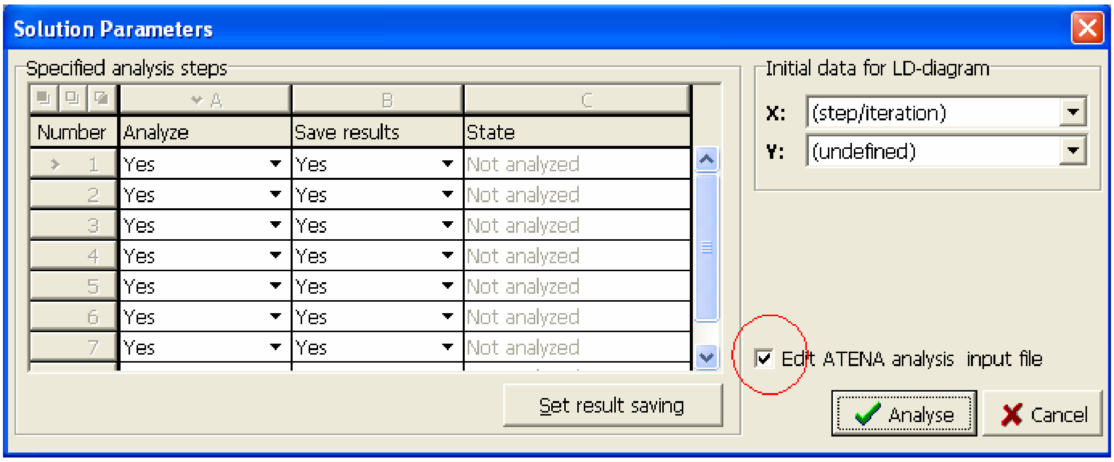


Figure 4-2: Solution Parameters

If the above procedure is used, the nonlinear analysis is not immediately executed, but rather a new window appears as shown in Figure 4-3. This window shows the content of the input file for ATENA finite element solution kernel. It is possible to directly modify its content. It should be noted that the file should be modified very carefully since certain mistakes may result in a program crash or loss of the input data. It is also recommended to save or backup the original input data file before pressing the run button.

The model will use the soft material for the element group 2 for the first 8 steps, and later it will be replaced while using the stronger concrete material Sbeta. It is therefore necessary to save the definition of the element group 2 for later use. This means that the content of the input file as shown in Figure 4-4 should be search to locate the definition of **Element group id 2** (top half of the structure). The whole definition of this macro-element should be selected and copied into the clipboard using **ctrl** + **c** (we will need it later for step 9). It is recommended to save it or paste it to some text editor (for instance notepad) for later use.

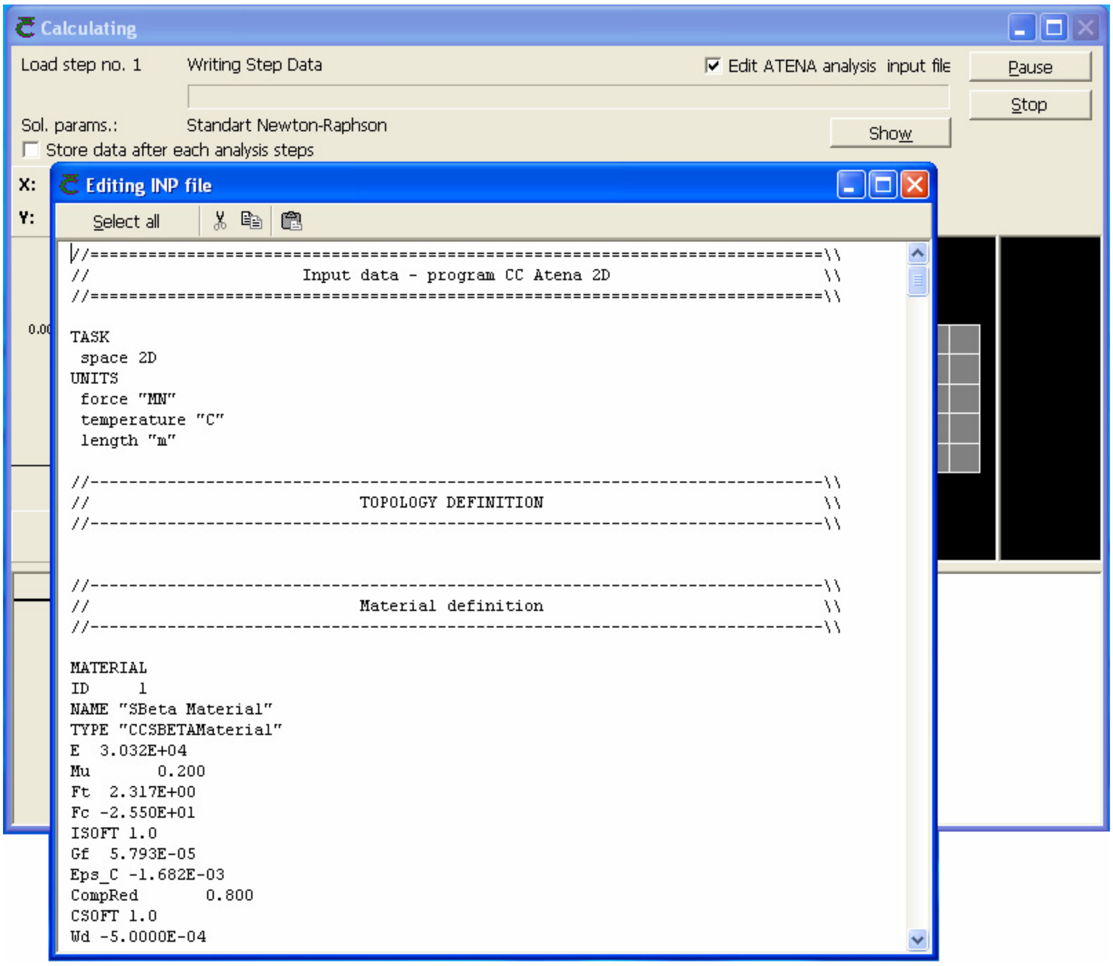


Figure 4-3: Window with the content of the ATENA input file before step 1.

The whole listing of the element group definition 2 is shown below:

ELEMENT GROUP

id 2

name "Quads - Macroelement No. 2"

type 1

material 3

geometry 2

ELEMENT INCIDENCES

1 64 65 66 67

2 67 66 68 69

3 69 68 70 71

.

.

60 142 146 147 143

After saving the definition of element group 2, it is possible to close the input file window by pressing the button and analysis will be started automatically. After the first step will be solved the window with input file for second step will popup. We need change the code for 9th step, so we need close all windows with input file for step 2-8 (Figure 4-4).

*Another option is to run only the first analysis step with the "**Edit ATENA analysis input file**" checkbox enabled. Then, run steps 2-8 with the input editing disabled, and only enable it again for step 9. This approach can save some clicking.*

In the input file for step 9 it is necessary redefine the material for all finite elements of the top macroelement No. 2. This means that the content of the input file for step 9 must be edited. It is necessary to first delete the element group 2 and then redefine it with the new material. The new commands should be included below the line with the command "*DELETE LOAD CASE 4 ;*".

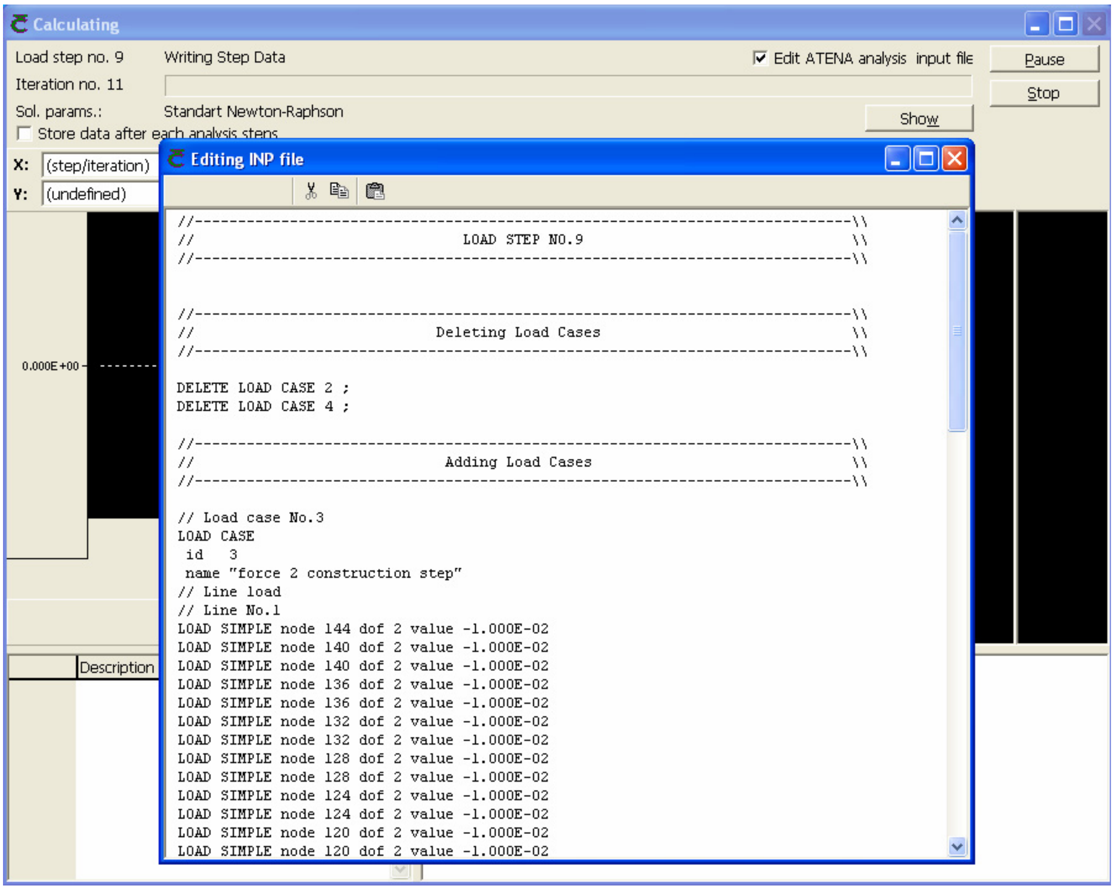


Figure 4-4: Window with input file for load step no. 9

The following command will be added to delete the old (soft) definition of macro-element No. 2

"*DELETE ENFORCED ELEMENT GROUP 2 ;*"

Next, the previously stored definition of the element group 2 should be pasted from the clipboard as it is shown in Figure 4-5. In order to use the normal concrete material, it is necessary to change the material number from 3 to 1 (i.e., back from the soft elastic isotropic to concrete), see Figure 4-5. The content of the whole input file code for load step no. 9 is added at the end of this document (6).

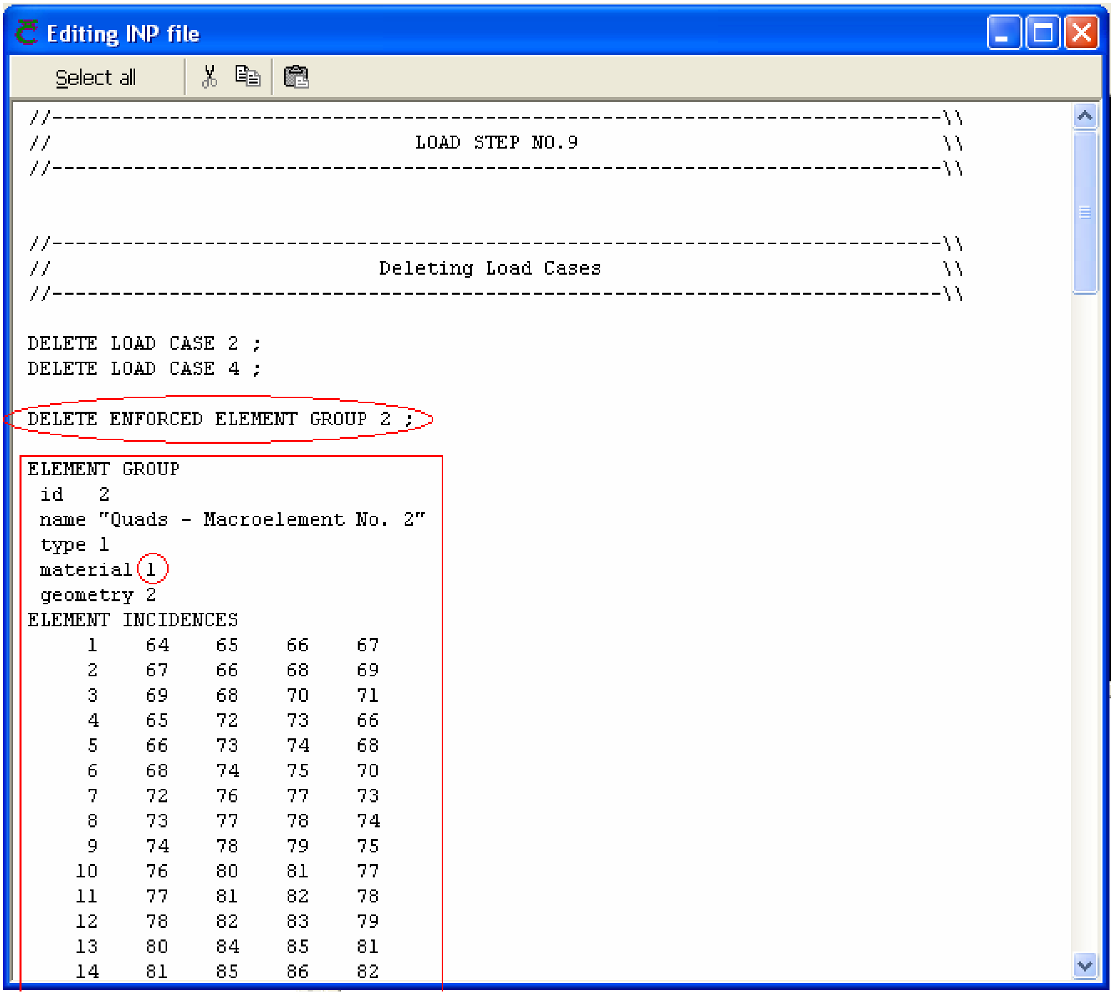


Figure 4-5: Input file for step no. 9 with added commands.

After this it is possible to close the input file and the analysis of step 9 will start automatically. Subsequently, it is recommended to uncheck the button "**Edit ATENA analysis input file**" on top of the analysis window to run the rest of the analysis without the popup windows for input file editing.

5. Analysis Results

The subsequent figures show the stress profiles at various steps during the analysis. The step 8 shows the results before the top macro-element is activated. While in step 12 it is clear that the top macro-element contributes to the stiffness of the slab with tensile stresses at the bottom and negative ones on the top. The results from step 12 also clearly show the stress jump between the old concrete and the new one.

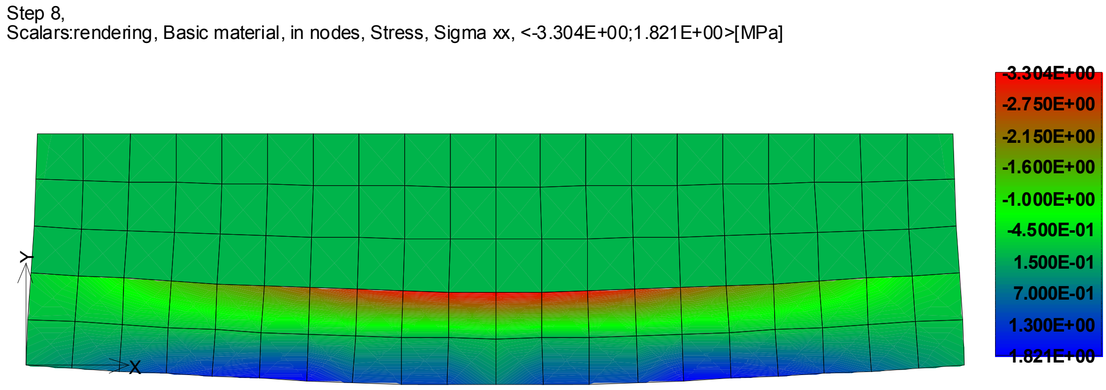


Figure 5-1: Stress xx after load step 8.

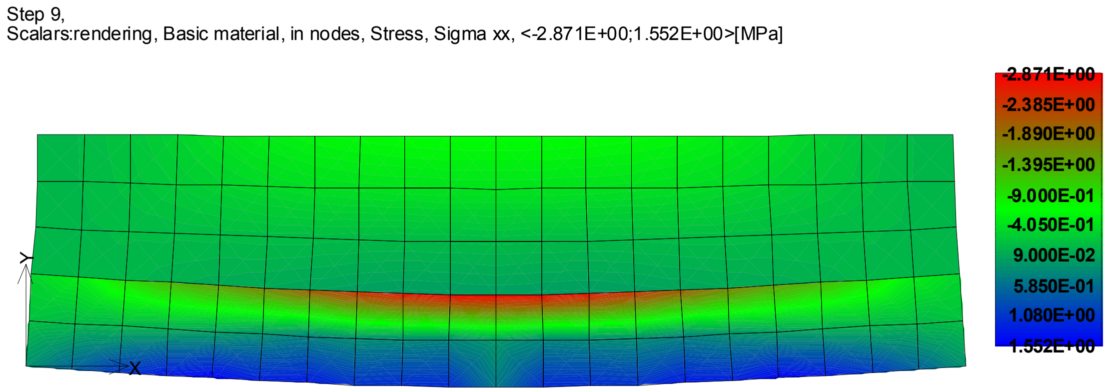


Figure 5-2: Stress xx after load step 9.

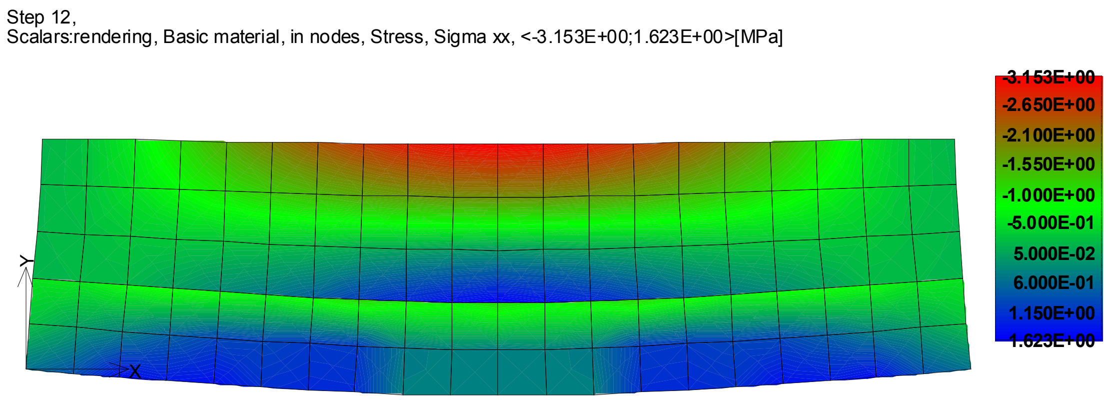


Figure 5-3: Stress xx after load step 12.

6. INPUT FILE FOR LOAD STEP NO. 9:

This section shows the whole content of the input file for step 9. This is the input file which was manually edited. The element group 2 with the original soft material is removed and then it is redefined using the normal concrete material number 1, i.e. the Sbeta material.

```
//-----------------------------------------------------------------------\\
//                          LOAD STEP NO.9                               \\
//-----------------------------------------------------------------------\\



//-----------------------------------------------------------------------\\
//                       Deleting Load Cases                             \\
//-----------------------------------------------------------------------\\

DELETE LOAD CASE 2 ;
DELETE LOAD CASE 4 ;

DELETE ENFORCED ELEMENT GROUP 2 ;

ELEMENT GROUP
 id   2
 name "Quads - Macroelement No. 2"
 type 1
 material 1
 geometry 2
ELEMENT INCIDENCES
     1    64    65    66    67
     2    67    66    68    69
     3    69    68    70    71
     4    65    72    73    66
     5    66    73    74    68
     6    68    74    75    70
     7    72    76    77    73
     8    73    77    78    74
     9    74    78    79    75
    10    76    80    81    77
    11    77    81    82    78
    12    78    82    83    79
    13    80    84    85    81
```

14	81	85	86	82
15	82	86	87	83
16	84	88	89	85
17	85	89	90	86
18	86	90	91	87
19	88	92	93	89
20	89	93	94	90
21	90	94	95	91
22	92	96	97	93
23	93	97	98	94
24	94	98	99	95
25	96	100	101	97
26	97	101	102	98
27	98	102	103	99
28	100	104	105	101
29	101	105	106	102
30	102	106	107	103
31	104	108	109	105
32	105	109	110	106
33	106	110	111	107
34	108	112	113	109
35	109	113	114	110
36	110	114	115	111
37	112	116	117	113
38	113	117	118	114
39	114	118	119	115
40	116	120	121	117
41	117	121	122	118
42	118	122	123	119
43	120	124	125	121
44	121	125	126	122
45	122	126	127	123
46	124	128	129	125
47	125	129	130	126
48	126	130	131	127
49	128	132	133	129
50	129	133	134	130
51	130	134	135	131
52	132	136	137	133
53	133	137	138	134
54	134	138	139	135
55	136	140	141	137
56	137	141	142	138

```
    57   138   142   143   139
    58   140   144   145   141
    59   141   145   146   142
    60   142   146   147   143

//------------------------------------------------------------------------\\
//                        Adding Load Cases                               \\
//------------------------------------------------------------------------\\

// Load case No.3
LOAD CASE
 id   3
 name "force 2 construction step"
// Line load
// Line No.1
LOAD SIMPLE node 144 dof 2 value -1.000E-02
LOAD SIMPLE node 140 dof 2 value -1.000E-02
LOAD SIMPLE node 140 dof 2 value -1.000E-02
LOAD SIMPLE node 136 dof 2 value -1.000E-02
LOAD SIMPLE node 136 dof 2 value -1.000E-02
LOAD SIMPLE node 132 dof 2 value -1.000E-02
LOAD SIMPLE node 132 dof 2 value -1.000E-02
LOAD SIMPLE node 128 dof 2 value -1.000E-02
LOAD SIMPLE node 128 dof 2 value -1.000E-02
LOAD SIMPLE node 124 dof 2 value -1.000E-02
LOAD SIMPLE node 124 dof 2 value -1.000E-02
LOAD SIMPLE node 120 dof 2 value -1.000E-02
LOAD SIMPLE node 120 dof 2 value -1.000E-02
LOAD SIMPLE node 116 dof 2 value -1.000E-02
LOAD SIMPLE node 116 dof 2 value -1.000E-02
LOAD SIMPLE node 112 dof 2 value -1.000E-02
LOAD SIMPLE node 112 dof 2 value -1.000E-02
LOAD SIMPLE node 108 dof 2 value -1.000E-02
LOAD SIMPLE node 108 dof 2 value -1.000E-02
LOAD SIMPLE node 104 dof 2 value -1.000E-02
LOAD SIMPLE node 104 dof 2 value -1.000E-02
LOAD SIMPLE node 100 dof 2 value -1.000E-02
LOAD SIMPLE node 100 dof 2 value -1.000E-02
LOAD SIMPLE node 96 dof 2 value -1.000E-02
LOAD SIMPLE node 96 dof 2 value -1.000E-02
LOAD SIMPLE node 92 dof 2 value -1.000E-02
LOAD SIMPLE node 92 dof 2 value -1.000E-02
LOAD SIMPLE node 88 dof 2 value -1.000E-02
```

```
LOAD SIMPLE node 88 dof 2 value -1.000E-02
LOAD SIMPLE node 84 dof 2 value -1.000E-02
LOAD SIMPLE node 84 dof 2 value -1.000E-02
LOAD SIMPLE node 80 dof 2 value -1.000E-02
LOAD SIMPLE node 80 dof 2 value -1.000E-02
LOAD SIMPLE node 76 dof 2 value -1.000E-02
LOAD SIMPLE node 76 dof 2 value -1.000E-02
LOAD SIMPLE node 72 dof 2 value -1.000E-02
LOAD SIMPLE node 72 dof 2 value -1.000E-02
LOAD SIMPLE node 65 dof 2 value -1.000E-02
LOAD SIMPLE node 65 dof 2 value -1.000E-02
LOAD SIMPLE node 64 dof 2 value -1.000E-02


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

// Parameters Standart Newton-Raphson
SET Static
SET SOLVE_LHS_BCS_OFF
SET Newton-Raphson
SET Iteration Limit 40
SET Displacement Error 0.010000
SET Residual Error 0.010000
SET Absolute Residual Error 0.010000
SET Energy Error 0.000100
SET iter_stop_displacement error factor 10000.0
SET step_stop_displacement error factor 1000.0
SET iter_stop_residual error factor 10000.0
SET step_stop_residual error factor 1000.0
SET iter_stop_energy error factor 1000000.0
SET step_stop_energy error factor 10000.0
SET Optimize Band Width Sloan
SET Line-Search
 LINE_SEARCH_WITHOUT_ITERATIONS
 MINIMUM_ETA       0.010
 MAXIMUM_ETA       1.000
SET Full_NR
SET TANGENT_PREDICTOR

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

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

STEP id 9 STATIC name "Load step No.9"
 LOAD CASE  1 * 0.25000000  3 * 0.25000000  65535 * 1.00000000
 EXECUTE

//------------------------------------------------------------------------\\
//                      END OF LOAD STEP NO.9                             \\
//------------------------------------------------------------------------\\
```

LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2009

[2] ATENA Program Documentation, Part 2-1, ATENA 2D User's Manual, CERVENKA CONSULTING, 2006

[3] ATENA Program Documentation, Part 3, ATENA Examples of Application, CERVENKA CONSULTING, 2000

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2009