



Cervenka Consulting Ltd.
Na Hřebenkách 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 5

ATENA-FEMAP User's Guide

written by

Vladimír Červenka, Jiří Niewald

Prague, March 03, 2005



Contents

1	INTRODUCTION	5
2	COMPONENTS	5
2.1	Script	5
2.2	Data files	5
2.3	Configuration and other files	5
3	USING ATENA-FEMAP	6
4	LIMITATIONS	8
5	USING FEMAP	8
6	SCRIPTS FOR COMMUNICATION FEMAP-ATENA	10
6.1	Generate Rod Elements	10
6.2	Create Atena Input	13
6.3	Write Atena Output	18
6.4	Import Atena Results	19
7	FINAL RECOMMENDATIONS	20
8	REFERENCES	20

1 Introduction

The purpose of this manual is to provide a guide for using program FEMAP in connection with ATENA. Program FEMAP is a product of Structural Dynamics Research Center (www.femap.com). It is a general program for pre- and post-processing of finite element analysis, and is used by many commercial finite element programs. ATENA is a product of Cervenka Consulting, and it is a special purpose package for nonlinear finite element analysis of concrete and reinforced concrete structures (www.cervenka.cz). In the case of using ATENA with FEMAP, only a core part of the ATENA system is used, namely, the finite element module AtenaConsole

2 Components

2.1 Script

FEMAP offers an option of the script language Easy Basic, which enables to write user routines for applications and features, which are not possible in the basic FEMAP environment. They can increase the capabilities of FEMAP application. This method is used for exporting the input data from FEMAP to ATENA as well as for the import of the output data from ATENA to FEMAP for post-processing. The scripts are also used to generate embedded reinforcing bars within the finite element mesh. The scripts can be executed in FEMAP by choosing an appropriate menu item from the main program menu.

2.2 Data files

TaskName.inp – The text input file for the analysis by AtenaConsole, where *TaskName* is chosen by the user. The file is created by FEMAP using the script CreateAtenaInput and is used by AtenaConsole.

TaskNameXXX.res – The text input file for AtenaConsole for the definition of ATENA output quantities. This file is created by FEMAP, and is used by AtenaConsole. It controls which output data will be extracted from AtenaConsole binary files to AtenaConsole text output files.

TaskNameXXX.out – The text output file from AtenaConsole. This file is created by AtenaConsole and is used by FEMAP by means of the script ImportAtenaResults. It serves for visualization of results.

2.3 Configuration and other files

Config.fem – This file is created in the working directory and contains global parameters of the finite element model and solution strategy (such as: *TaskName*, dimensions, numbers of elements and nodes, load steps, and other).

Atena_Cur.dir – This file is created in the directory containing FEMAP script files and it holds the current directory name.

TaskName.cfg – This file contains data about curve numbers of reinforcing bars. It is created by scripts for the embedded reinforcement and it is placed in the current working directory.

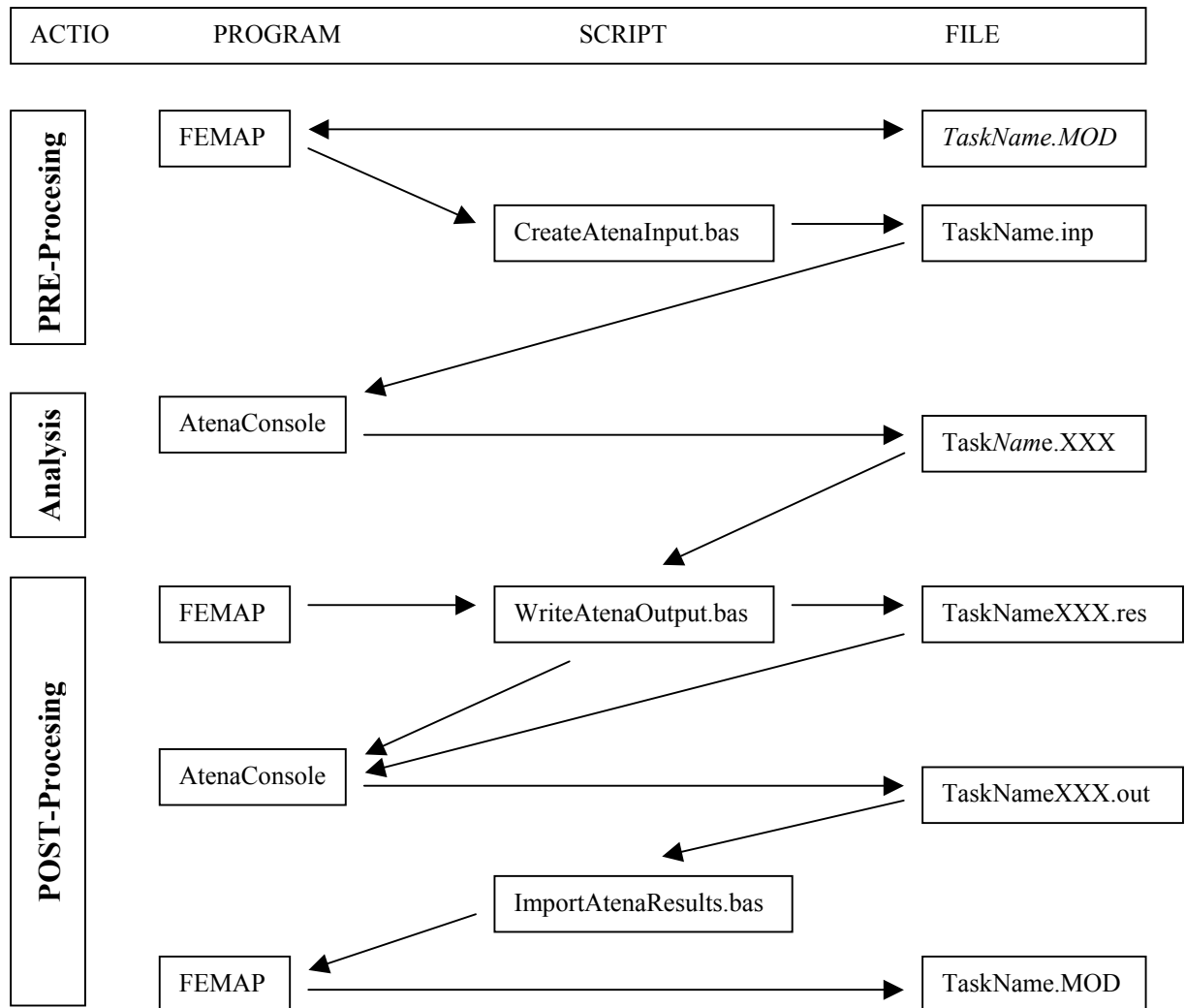
TaskName.cX - Data about reinforcing bars. It contains the weight coefficients of embedded reinforcement nodes or information about the connectivity of rebar nodes with the intersected solid elements. *X* is the curve number. The file is created by scripts for the embedded reinforcement and it is placed into the current working directory.

TaskName.pX - Data about reinforcing bars. It contains the weight coefficients of node bars elements or data connectivity with solid elements. *X* is the property number. The file is created by scripts for embedded reinforcement.

SpecVal.atf, *SpecVal.atn* – Lists of names of available output variables for FEMAP. File is created by the script WriteAtenaOutput.

3 Using ATENA-FEMAP

The use of ATENA-FEMAP is shown in the flowchart in Fig. 1.



Legend:

PROGRAM	AKTION	RESULT
- FEMAP	create finite element model	file <i>TaskName.MOD</i>
- FEMAP - script – CreateAtenaInput.bas	export of input file for AtenaConsole	file <i>TaskName.inp</i>
- AtenaConsole	FE analysis	binary file <i>TaskName.XXX</i>
- FEMAP - script – WriteAtenaOutput.bas	defining output data	file <i>TaskName.XXX.res</i>
- AtenaConsole	writing output data	file <i>TaskName.XXX.out</i>
- FEMAP - script – ImportAtenaResults.bas	importing output data to FEMAP	file <i>TaskName.MOD</i>

Fig. 1 Flow chart of ATENA-FEMAP system

As first the finite element model must be generated using the FEMAP environment. The details about FEMAP and its use can be found in the FEMAP manual. The special features concerning reinforcing bars are described in the Chapter 5. After completing model generation with FEMAP the system ATENA-FEMAP can be used for analysis and postprocessing. Operation of the system is controlled from the FEMAP environment. The sequence of operation steps is summarized below:

1. Build your finite element model in FEMAP. Scripts for generating reinforcement elements *Calculate2DRod*, *Calculate3DRod* can be used if required. For two-dimensional problems it is however, recommended to use the program ATENA 2D, which contains a powerful graphical user interface that simplifies many of the tasks, which are described in this manual.
2. By executing the script *CreateAtenaInput* the file *TaskName.inp* (example: *Test.inp*) is created. The file *Config.fem* with a list of load steps (to be executed) is generated.
3. Execute ATENA analysis by choosing the appropriate Run Option. File *TaskName.XXX* (example: *Test.001*) is created, where *XXX* is the load step number.
4. Execute the script *WriteAtenaOutput*. It reads the list of load steps from the file *Config.fem* and creates the files *TaskNameXXX.res* (example: *Test001.res*). This file contains the information for extracting output data from the Atena binary files.
5. Execute AtenaConsole (by Run Option). It reads the file *TaskNameXXX.res* and exports data from the binary file *TaskName.XXX* to the output text file *TaskNameXXX.out*.
6. Import the output data to FEMAP by executing the script *ImportAtenaResults*.
7. Perform desired post-processing in FEMAP.

The access to ATENA-FEMAP functions is made possible through the Menu.

The tools are activated by choosing "Atena", "SBeta" or "CCTools" menu (see Fig. 2)

Fig. 2. The ATENA – FEMAP tools are in menu "Atena". The SBETA – FEMAP tools is in menu "SBeta". In the menu "CCTools" is the other scripts for Export and Import Mesh, Monitoring - evolution value, Prescribe displacements from file.

Note: All files for the ATENA-FEMAP interface are text files. These files are editable and thus some of the operations described above can be replaced by a user file writing

or editing.



Fig. 2 Main menu in FEMAP (top), ATENA-FEMAP menu (right bottom).

4 Limitations

Following limitations with respect to FEMAP must be observed.

- Loading forces, or imposed displacements must be given in FE nodes.
- Supports can be only in FE nodes and in directions of global coordinate axes.
- ID number of Load Set and Constrain Set cannot exceed 999.

5 Using FEMAP

This chapter offers only a brief guide for FEMAP. For detail description please refer to the FEMAP User's Guide. In the following are mentioned only some hints for a case of 2D problem.

Surface. Simplest region is a rectangle. Choose menu **Geometry** → **Curve-Line** → **Rectangle**, enter corner points (bottom-left and top-right points). Choose **Menu Geometry** → **Boundary Surface** and **Select All** → **O.K.** By this a region for generation is created.

Material. In this menu material properties are generated: **Model** – **Material**. From all standard types only isotropic type is supported: **Type** → **Isotropic**. Only basic elastic properties can be entered at this place. Special ATENA materials can be entered by selecting **Other** types. Following ATENA material are available:

- CC1DElastIsotropic in template Elast Isotropic with Model ID 1
- CCPlaneStressElastIsotropic in template Elast Isotropic with Model ID 2
- CCPlaneStrainElastIsotropic in template Elast Isotropic with Model ID 3
- CC3DElastIsotropic in template Elast Isotropic with Model ID 4
- CCASymElastIsotropic in template Elast Isotropic with Model ID 5
- CC3DBiLinearSteelVonMises in template 3D BiLinear Steel VonMises with Model ID 6
- CCPlaneStressSteel in template 2D BiLinear Steel VonMises with Model ID 18
- CC3DCementitious in template 3D Cementitious with Model ID 7
- CC3DNonLinCementitious in template 3D Non-Lin Cementitious with Model ID 8
- CC3DNonLinCementitious2 in template 3D incr.Non-Lin Cementitious with Model ID 20
- CSBETAMaterial in template SBETA Material with Model ID 9
- CC2DInterface in template Interface with Model ID 10
- CC3DInterface in template Interface with Model ID 11

- CCRinforcement in template Reinforcement with Model ID 12
- CCCyclingReinforcement in template Cycling Reinforcement with Model ID 13
- CCSmearedReinf in template Smeared Reinforcement with Model ID 14
- CCSpringMaterial in template Spring Material with Model ID 15
- CC3DDruckerPragerPlasticity in template 3D Drucker Prager Plasticity with Model ID 16
- CCMicroplane4 in template Microplane M4 with Model ID 17
- CCCombinedMaterial in template Composite material with Model ID 19
- CCShellMaterial in template Shell Geom.material with Model ID 21

Description for these materials can be found in other Atena documents (Atena Input File Format, Atena Theoretical background).

The FEMAP templates make possible to enter parameters of the mentioned models. However, there is always a second possibility to edit these parameters in later stages, for example in the dialog box **Material properties** in the script for Atena Input (see chapter on scripts, Fig. 8).

Element Properties. Menu **Model** → **Property**. Following items must be selected:

Finite Element Types. Menu **Elem/Property Type** – In this menu only following types are supported:

- 1D element for reinforcing bars in the box **Line Elements** → **ROD**
- 2D element in the box **Plane Elements** → **MEMBRANE**
- 3D element in the box **Volume Elements** → **SOLID**

Material can be selected form a dialog box.

Dimensions can be entered in case of 2D element – thickness, and 1D element – cross section area.

Mesh generation. Variety of procedures is offered. Following items should be properly selected:

Mesh → **Mesh Control** – **Size on Surface** - Defines the element size on the surface.

Mesh → **Geometry** → **Surface (resp. Hex Mesh Solids)**- Selects the surfaces to be generated. If you want to use model the region using shell elements with the “ShellMaterial”, then check Midside Nodes. You will also need to generate quadratic Brick elements of topology Brick20 for it.

Generate the FE mesh using an appropriate dialog box.

Boundary conditions are defined in the menu item **Model** → **Constraint** → **Set**. Only nodal supports are accepted by ATENA. For this we use **Model** → **Constraint** → **Nodal** and select concerning nodes and constraints.

Loading is defined by the menu item **Model** → **Load** → **Set**. Only nodal forces are accepted by ATENA. For this we use **Model** → **Load** → **Nodal** and select concerning nodes. Supported loads are: **Force, Displacement**.

It is possible to generate arbitrary number of “Constraint–Sets” and “Load–Sets” and combine then into load steps for ATENA. In this way (almost) arbitrary load history can be defined.

For ATENA analysis there are recognized three sets constrains. These constrains are given ID numbers with following rules:

1. Loding forces and given displacement. ID numbering starts with 1001.
2. Supports. ID numbering starts with 1.
3. Embedded bars with kinematic constrains. ID numbering starts "MASTER-SLAVE definition in LOAD CASES" for "For each selected Curve separately" as curve ID+10.000, for "For ACTUAL Property ID" as property ID + 1.000.000.

These ID numbers are automatically assigned to the appropriate constrain types by scripts.

When the finite element model is completed the script *CreateAtenaInput* is used to export data to the program *AtenaConsole*.

6 Scripts for communication FEMAP-ATENA

The purpose of the scripts is to introduce special features of ATENA into FEMAP and provide a data interface between the two programs. The scripts are programs in Easy Basic and are stored in the directory CCScripts. The scripts used for ATENA are listed in the following Table1.

Table1. List of scripts for ATENA.

Name	Scope of Action	Function description
Calculate2DRod.bas	Preprocessing	Generates embedded bar elements for reinforcement in case of 2D mesh with quadrilateral elements.
Calculate3DRod.bas	Preprocessing	Generates embedded bar elements for reinforcement in case of 3D mesh with solid elements.
CreateAtenaInput.bas	Preprocessing	Generates input file for ATENA.
WriteAtenaOutput.bas	Postprocessing	Generates definition of output data in a load step.
ImportAtenaResults.bas	Postprocessing	Imports data from output file of ATENA to FEMAP.

6.1 Generate Rod Elements

These scripts are written for making possible the use of rod elements as embedded bars in ATENA for 3D element meshes. They calculate points of intersection of reinforcing bars with the finite element mesh and divide the reinforcing bars into series of rods embedded within the elements. These rod elements are considered as 1D elements in ATENA. In order to start the dialog choose the appropriate button from the Atena-Femap tool box (see Fig. 2). Then the sequence of dialog boxes follows as described further.

First the working directory must be chosen using the dialog window **Current Directory** as shown in the example below, Fig. 3.

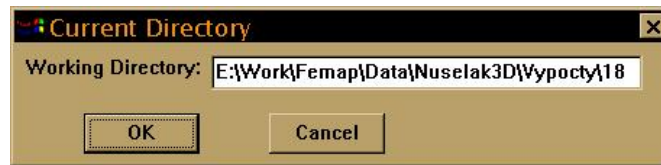


Fig. 3 Dialog box for choosing directory for generating reinforcement.

A task name is entered using the dialog window Task Name as shown in Fig. 4. This Task Name will be used through the work as a basis of various file names. The check box "Old output format for MASTER/SLAVE" enables to define kinematic constraints as weight coefficients of node bars elements or data connectivity with solid elements. In ATENA versions 1.1.3 and younger it is recommended to leave this box unchecked. For older versions it is necessary to select this option. The next field in this dialog is the "Default tolerance". It is the default value for the calculation of intersection points between the reinforcement rods and the solid finite elements. Next field is the possibility to read reinforcement curve information from a file: "Read Curve from file". The check box "Save continuously" is recommended when it is expected that large amount of reinforcement segments will be created.

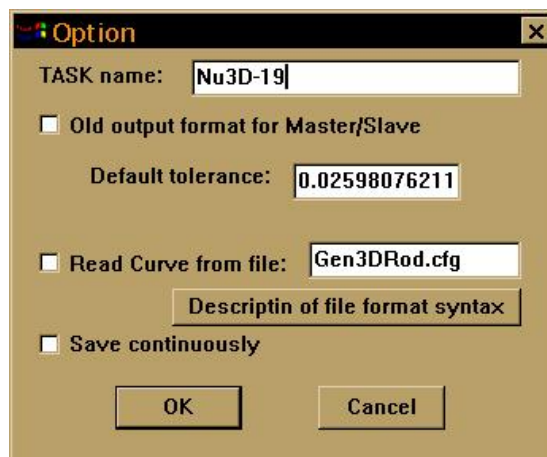


Fig. 4 Dialog box for entering task name and various reinforcement generation options.

Before calling scripts it is necessary to define material property and geometry ("curve") for concerned reinforcement. Supported geometry type is "Line". Properties are assigned to the reinforcement using the dialog "Reinforcement Segmentation" as shown in Fig. 5 below.

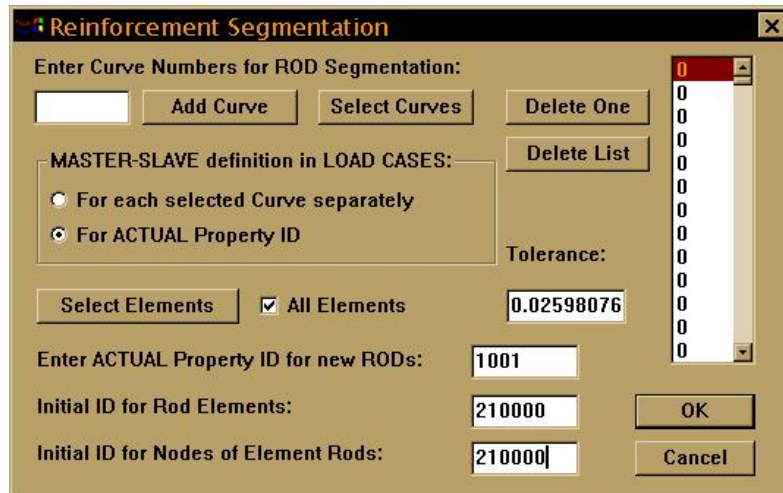


Fig. 5 Dialog box for entering reinforcement properties.

By pressing the button **Select Elements** it is possible to select a group of elements for checking their inter-section with the reinforcement bar. If the **All Elements** button is checked, then all elements will be checked for their intersection with the reinforcing bar. Initial ids for the generated rod elements and their nodes can be specified in this dialog as well.

In the input block "MASTER-SLAVE definition in LOAD CASES" it is possible to select a rule for the definition of load cases, which are used for connecting the reinforcement nodes with those of the solid elements. After pressing the button **OK**, new nodes and truss elements will be generated. These newly created reinforcement nodes must be connected to the rest of the mesh. In ATENA this is accomplished through a special load case, which contains commands for tying the reinforcement degrees of freedom with those of the solid elements. A separate load case can be created either for each curve (the check box "For each selected Curve separately") or for each property (the check box "For ACTUAL Property ID"). The ID of the created load case is calculated using the following rules:

- for each curve separately - $ID = \text{curve ID} + 10.000$
- for actual property - $ID = \text{property ID} + 1.000.000$

The tolerance is used for the calculation of the intersection points of reinforcement curves with the finite element model. The recommended value is about 1-5% of the size of the smallest finite element.

By pressing **OK** the generation of rod elements for given lines in the list is preformed. For next reinforcing bar this dialog can be repeated with another properties (List Curve number, Rod property ID, Selected Elements, Initial ID for Nodes and Elements). By pressing "Cancel" the rod generation is terminated.

The connectivity between the generated rod elements and the underlying FE mesh is stored for each bar into a file with the following naming convention: *TaskName.cX*, or *TaskName.pX*, where X denotes the id of the reinforcement curve, or property. The list of curves, or properties for which rod elements were created is stored into a file called *TaskName.cfg*. This file is totally regenerated each time this script is executed. When

ATENA input file is created, only the curves listed in this file will be included into the solution. This means that every time this script is executed, it is necessary to create the segmentation for all reinforcement curves that should be included in the model. In other words, this script should be executed only once for a given model. Otherwise the user should manually remove the generated reinforcement nodes and elements from the previous executions of the script.

This program can change contents of the files *Config.fem*, *Atena_Cur.dir* and creates the files *TaskName.cfg*, *TaskName.cX*, *TaskName.pX*.

6.2 Create Atena Input

The purpose of this script is to create a text input file for ATENA based on the FEMAP data and additional data special for ATENA. To start the dialog choose the button **Create Input** from the Atena-Femap tool box (see Fig. 2). Then the sequence of dialog boxes follows as described further.

The working directory can be chosen in the same way as described in the previous script, Fig. 3. Usually the same directory for all files is used. The basic parameters for the data interface are set through the dialog window **Initial Option**, Fig. 6.

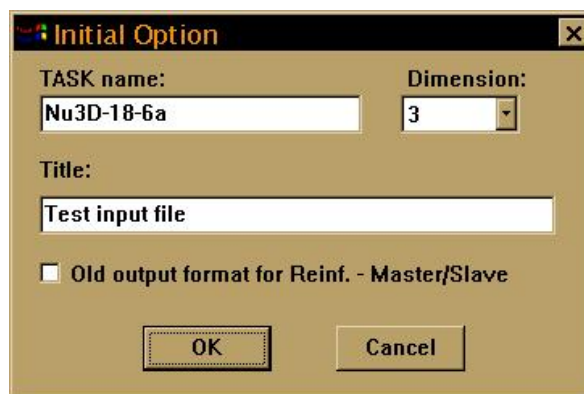


Fig. 6 Dialog box for data basic parameters.

Task name can be maintained unchanged. The dimension is 2 for 2D problem and 3 for 3D problem. Title can be any description.

The check box "Old output format for MASTER/SLAVE" must be selected if you have selected this check box for the generation of reinforcement with script *Calculate3DRod.bas* or *Calculate2DRod.bas* (Fig. 4), because script reads just previously generated kinematic constraints. Old method defines weight coefficients for each reinforcement node and degree of freedom, while the newer method requires only the nodal numbers, to which the displacements of the reinforcement nodes should be tied.

Next dialog, Fig. 7, offers an option to check for an existence of unused nodes. A similar option exists also in FEMAP and it is recommended to use it for the removal of free nodes.

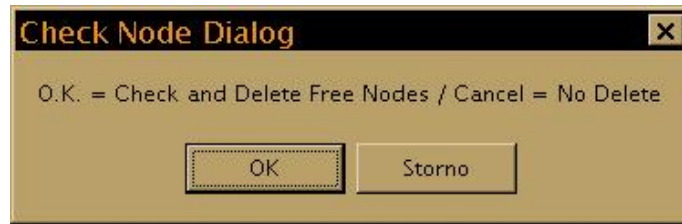


Fig. 7 Dialog box for counter clockwise element numbering.

The material properties entered in FEMAP can be viewed and changed in the dialog box **Material Properties**, Fig. 8.

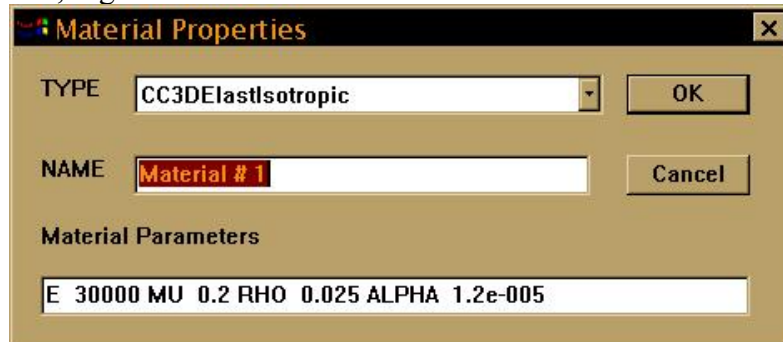


Fig. 8 Dialog box for viewing and editing material parameters.

Supported material types (CCSBETAMaterial, ...) as well as parameter identification names (E, MU, ...etc.) can be found in the document *Atena Input File Format*.

In the following dialog box an option of strain definition can be chosen, Fig. 9. The option NONLINEAR means large strains (Green-Lagrange strains), the option LINEAR means small strains (engineering strains).



Fig. 9 Large and small strain option.

The next dialog offers an option to check the numbering order of element nodes in 2D elements, Fig. 10. ATENA requires a counterclockwise numbering of element nodes. If clockwise numbering is detected FEMAP performs a renumbering. (This happens only seldom.)

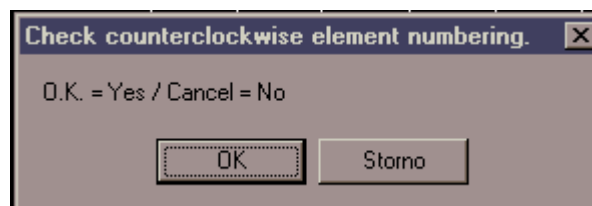


Fig. 10 Check element node numbering order.

Writing ELEMENT GROUPS:

The following tables describe the element type and group numbering that is created by FEMAP-ATENA interface for various element groups.

Element Name	Element Type
L ROD	1
L BAR	2
L TUBE	3
L LINK	4
L BEAM	5
L SPRING	6
L DOF SPRING	7
L CURVED BEAM	8
L GAP	9
L PLOT	10
L SHEAR	11
P SHEAR	12
L MEMBRANE	13
P MEMBRANE	14
L BENDING	15
P BENDING	16
L PLATE	17
P PLATE	18
L PLANE STRAIN	19
P PLANE STRAIN	20
L LAMINATE PLATE	21
P LAMINATE PLATE	22
L AXISYM	23
P AXISYM	24
L SOLID	25
P SOLID	26
L MASS	27
L MASS MATRIX	28
L RIGID	29
L STIFF MATRIX	30
L CURVED TUBE	31
L PLOT PLATE	32
L SLIDE LINE	33

Element Topology	2D	3D
Line2	0	
Line3		1
Tria3	2	
Tria6	3	
Quad4	4	
Quad8	5	
Tetra4		6
Wedge6		7
Brick8		8
Point1		9
Tetra10		10
Wedge15		11
Brick20		12
RigidList		13
Pyramid5		14
MultiList2		15
Contact2		16

(Solid) **Property** is identification of Material and Geometrical specification (for example: 2D element type has geometrical property-thickness, for Truss element is geometrical property-Area).

Calculation Constant	Value
K_PROP	10000
K_MATE	1000
K_ETYP	100
K_ETOP	10

The element type id is generated using the following formula:

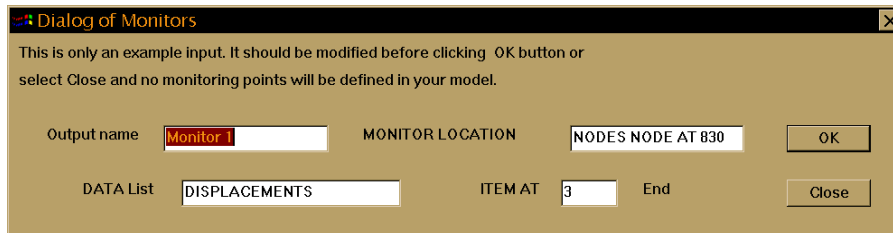
$$\text{ELEMENT_TYPE_ID} = \text{Elem_Type} * \text{K_ETYP} + \text{Elem_Topology} * \text{K_ETOP} + \text{Elem_Property} * \text{K_PROP}$$

In ATENA each element group consists of elements of the same type using the same material. The element group id is calculated using the following formula:

$$\text{ELEMENT_GROUP_ID} = \text{Incremental Index (for 3D Element Group)}$$

MONITORING

Dialog of Monitors has a few edit options. First is **Name of Output** monitor, next is **Location of monitor**, next is **Data list** and last is **Item** of Data List.



Parameters of analysis are set in the dialog box **Loading History**, Fig. 11. The meaning of parameters is clear from the screen. In this version only **Static** analysis is supported. Iteration method **Newton-Raphson** is suggested for the problems with given loads or prescribed deformations. In cases of ultimate load analysis and post-peak response the **Arc-Length** method is recommended. The parameters of nonlinear solution control the iterative process. **Residual Error** and **Absolute Residual Error** are related to force equilibrium. **Displacement Error** is related to displacement changes and **Energy Error** to energy dissipated in the current iteration. **Iteration limit** is the maximum number of iterations in load step. If reached, the solution has not converged under given tolerances.

Load case ID is increased by 1000 for the purpose of transfer to ATENA (example: load case 1 in the Input file has ID=1001). The load case ID is only a name (string of characters) and this method does not have any numerical consequences. The reinforcement is included in this dialog, because it makes possible to activate reinforcement in later stages of load history, for example is cases of structural strengthening or prestressing. Note that the scaling factor is ment for a value of force or displacement and thus in case of reinforcement is irrelevant and is considered always 1.0 regardless of user's choice.

The item **Store data for every STEP** makes sure that the binary files in ATENA contain data form all load steps. These data are needed for text output or post-processing. If text output or post-processing are not required this box should empty. By pressing OK the load steps in the dialog box are added to the steps previously generated.

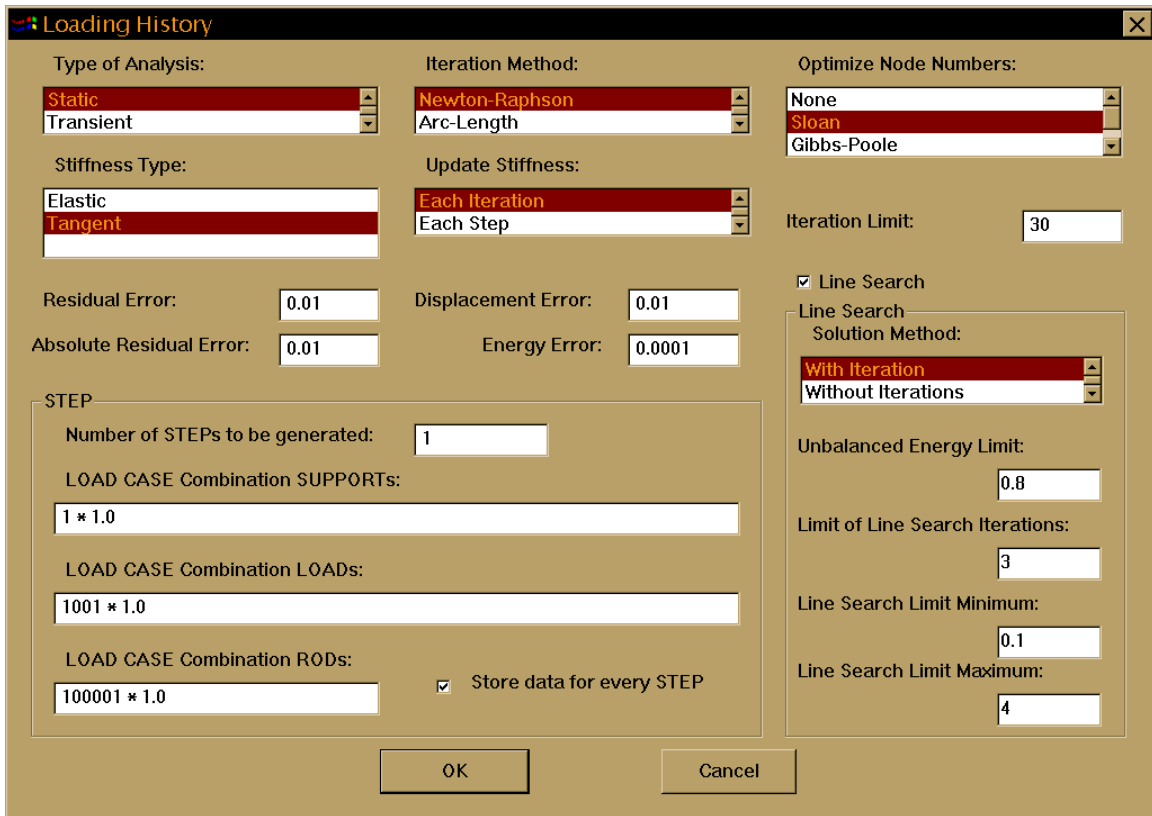


Fig. 11 Dialog box for load history and solution parameters.

In the dialog box **Analysis**, Fig. 12, it is optionally possible to modify the input file for ATENA using the button **Edit *.inp file**. The analysis by program ATENA is started by pressing the button „Atena Console Analysis *.inp file“, or „Atena Win Analysis *.inp file“.

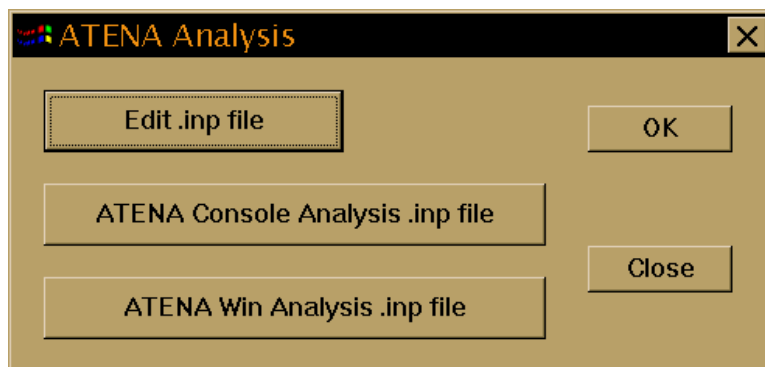


Fig. 12 ATENA Analysis option.

This script modifies files **Config.fem**, **Atena_Cur.dir**, and creates file **TaskName.inp**.

6.3 Write Atena Output

The purpose of this script is to define data which should appear in the text output file from Atena. Prerequisites are binary files **TaskName.XXX**, which contain data from an analysis. To start the dialog choose the button **Write Output** from the Atena-Femap tool box (see Fig. 2). Then the sequence of dialog boxes follows as described further.

First two dialog boxes for directory and task name are opened in the same way as in the previous scripts. Usually directory and task name should not be changed for the output. The next dialog box **Output Data**, Fig. 13, is used for definition of output data. The available load steps appear in the box **Load Step Binary Files**. If output data for all load steps should be generated the box **Generate all *.out files** should be selected. Alternatively, only one load step can be selected from the list. New **.out** files are created and existing **.out** files are replaced by new ones. In **Standard Data** the items required for the output should be checked.

In the option **Special Data** additional data can be entered. By pressing the button **Fill from Atena** it is possible to read all data items from Atena. Selected data, or all data, can be deleted from the box and only data types needed for output should be left. The buttons **Add User Data** and **Edit** are not active.

After pressing OK the script **WriteAtenaOutput** is executed. It generates the file **TaskNameXXX.res** (for definition of output data). It automatically generates also output files by executing ATENA only in case that box **Generate all *.out files is checked**. In other case the file **.res** is generated only for one load step and output file generating is preformed using the next dialog box **Output**, Fig. 14. This makes also possible to edit the **.res** file if desired.

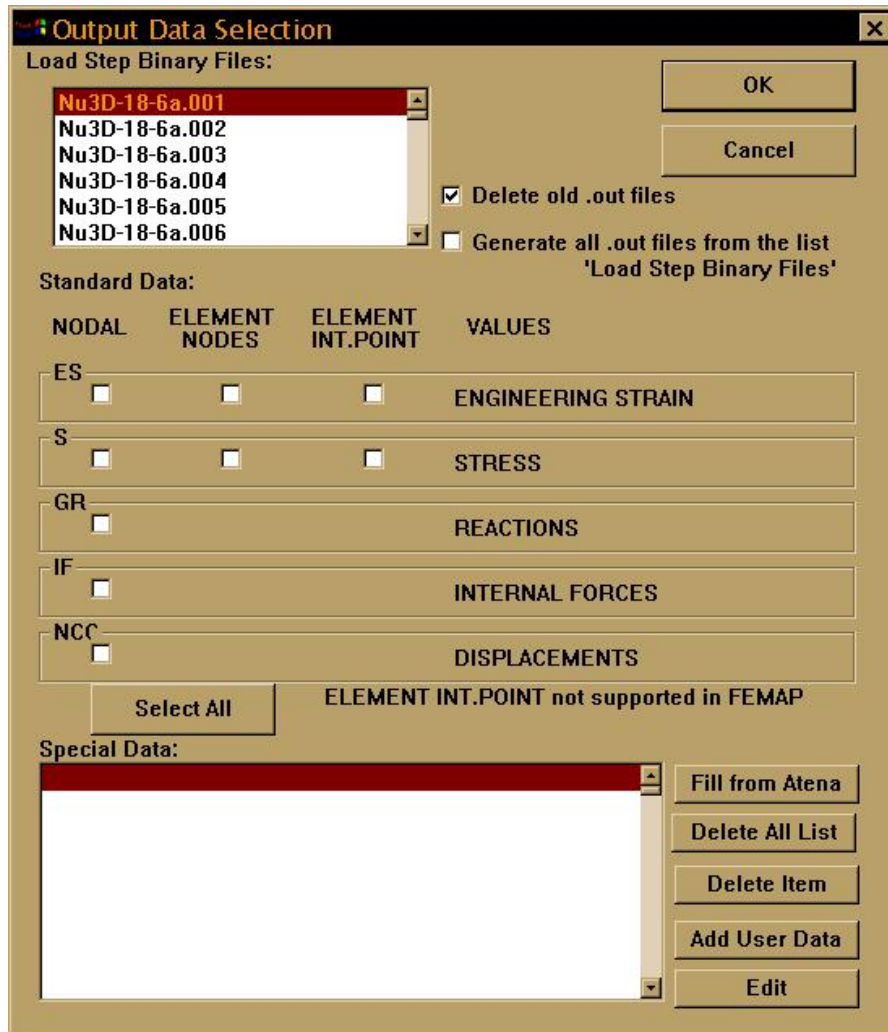


Fig. 13 Dialog box for output data.



Fig. 14 Dialog box for creating output files.

This script can modify the contents of the files *Config.fem*, *Atena_Cur.dir*. It creates the files *TaskNameXXX.res* and *TaskNameXXX.out*.

6.4 Import Atena Results

The purpose of this program is to import data from ATENA output files to FEMAP. Prerequisites are text output files **TaskNameXXX.out**, which contain output data form

analysis. To start dialog choose the button **Import Results** from the Atena-Femap menu item (see Fig. 2). Then the sequence of dialog boxes follows as described further.

First two dialog boxes for directory and task name are opened in the same way as in the previous scripts. Usually these parameters should not be changed for the output. The next dialog box **Input file name** is used for selecting needed output files, Fig. 15.

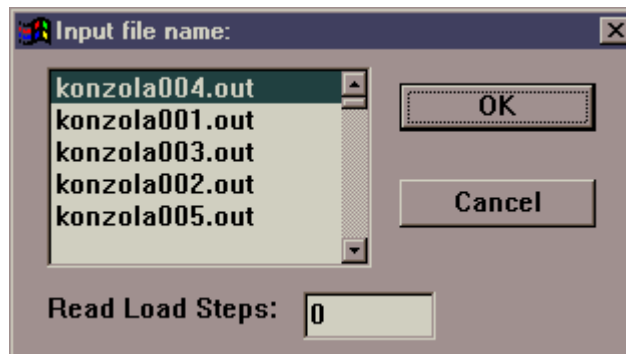


Fig. 15 Dialog box for output file names.

This script can modify the files *Config.fem*, *Atena_Cur.dir*. It reads data from the files *TaskNameXXX.out* and include them in the file *TaskName.mod*.

If it is required to read several output files, their number can be specified in the dialog box in Fig. 15. If for instance a number 3 would be specified Fig. 15, then the files “konzola004.out” to “konzola003.out” will imported into FEMAP. It is important to note the results from the first load step that is imported into FEMAP should contain all types of possible input data, otherwise an error message may occur during the data import for subsequent files.

7 Final recommendations

Viewing of results depend on the type of ,vector of result‘. When nodal or elemental results are to be post-processed, it is important to select an appropriate output type. For instance, in the case of vector type ELEMENT, it is necessary to be selected in the menu "View" → "Select" → "Deformed and Contour Data" → "Contour Options" → "Contour type" → Elemental. Without an appropriate selection of this option the display of data could be incorrect in FEMAP.

8 References

- [1]ATENA Program Documentation, Part 6, ATENA Input File Format, Cervenka Consulting, 2000.
- [2]FEMAP User Guide, Enterprise Software Products, 1986-1998
- [3]Language Reference, Cypress Enable Scripting, March 1998
- [4]FEMAP Basic Script Language - API Reference, Enterprise Software Products, March 1998
- [5]FEMAP Neutral File Format, Enterprise Software Products, March 1998