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 2-1

User's Manual for ATENA 2D

Written by:

Vladimír Červenka and **Jan Červenka**

Prague, May 2015

CONTENTS

1 INTRODUCTION

The purpose of this manual is to provide a full description of the 2D graphic user interface for the program **ATENA**. It is determined to support a usage of **ATENA** program system. This document is compatible with the **ATENA** version 2.0.1 released in 06.2002.

The **ATENA** program, which is determined for nonlinear finite element analysis of structures, has got tools specially designed for computer simulation of concrete and reinforced concrete structure behaviour.

ATENA program consists of the solution core and the user interface. The solution core has got capabilities for the 2D and 3D analysis of continuum structures. It has libraries of finite elements, material models and solution methods.

ATENA User Graphic Interface for 2D is a program, which enables access to the **ATENA** solution core. It is limited to 2D graphical modeling and covers the states of plane stress, plane strain, and rotational symmetry.

This manual is devoted only to the description of **ATENA** user interface and its functions. The questions of the **ATENA** solution core and the theoretical background are mentioned only marginally and are included in the other volumes of the **ATENA** documentation: ATENA Part 1 - Theory, Part 6 - ATENA Input file.

Although some examples are included in this manual to support the description of some functions, a systematic treatment of examples is not covered. It is also a subject of a separate document ATENA Part 5 – Examples.

ATENA version 2.0.1 includes some extensions and improvements, comparing to previous versions. The main changes, which can be noted by used are:

Import and export of data in other formats.

Bond properties of bar reinforcement.

Deleting results in chosen load steps after analysis.

Incompatibility of data in results with respect to lower versions. Input data are compatible.

ATENA installation.

The loss of data compatibility was an unavoidable result of the implementation of a new data structure. This greatly improved the numerical performance and potentials for future development. Working with old ATENA files and the installation are described in the separate manuals ( ).

2 FILE

This chapter explains the menu item **File** in the main menu toolbar. Creating a new file or opening an old file is possible by using submenu **File**.

There are four parts in the menu item **File** as shown in Fig. 2-1.

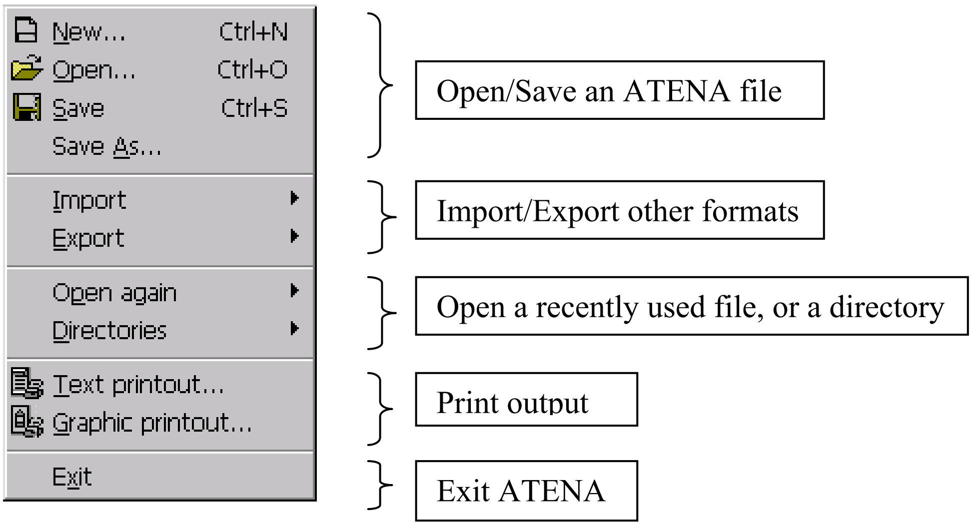


Fig. 2-1 Menu item 'File'.

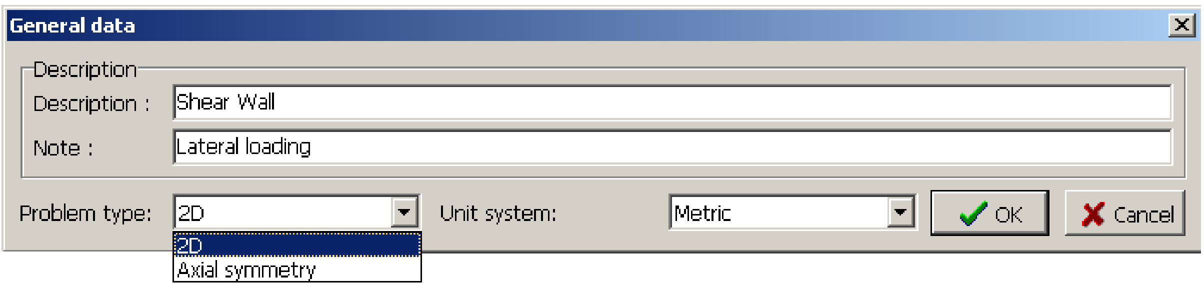


Fig. 2-2 New file dialog box.

2.1 Open New File (File | New)

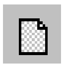
As first step a problem type must be defined using the menu item **File | New** as shown in Fig. 2-2 . The dialog box **General data** has three data fields:

- **Description**. Task description associated with this ATENA file. This text shall appear in the top blue field of the window. This can be used to identify the ATENA window, for example, in case of more simultaneously processes ATENA tasks.
- **Note**. Additional accompanying text for a detail explanation.

Problem type. Here, two types are possible. '2D' is for a plane stress/strain problem. This is a default option. In this problem definition two orthogonal axes, X, Y appear in the graphical window. 'Axial symmetry' is for an axisymmetrical (also called rotationally symmetrical) problem. In this case the graphical window shows the radial axis R (horizontal) and axis of rotation 'Z' (vertical). The axes R, Z are used only in the graphical window in order to clearly indentify the axially symmetrical problem. However, in the graphical and numerical post-processing these axes are denoted as X, Y axes.

Unit system. This option allows the user to choose unit system that will be used throughout the whole analysis. Two unit systems are available now: ‘metric’ system using SI units and ‘imperial’ system using American unit system. In the case of ‘metric’ system distances will be measured in meters ([m]), forces in mega-Newton ([MN]) and temperature in degrees of Celsius ([°C]). In the case of ‘imperial’ units distance are given in inches ([in]), forces in kilo-pound-force ([kip]) and temperature in degrees Fahrenheit ([°F])

If a current file in **ATENA** exists, whose data has not been saved, a small dialogue box appears after using this command, which will ask whether the current file should be saved or not. After saving/unsaving the current file, a new file is opened.

Clicking the mark  on the toolbar has the same effect as this command.

2.2 Open an ATENA File (File | Open)

An old ATENA file can be opened by this command. A window shown in Fig. 2-3 will appear after using this command. A file can be selected in this window. **ATENA** 2D files have extension `.cc2`.

If there is already a current file in **ATENA**, after using this command, a small dialogue box will appear, which will ask whether the edited file should be saved or not? After saving/unsaving the edited file, another ATENA file can be opened.

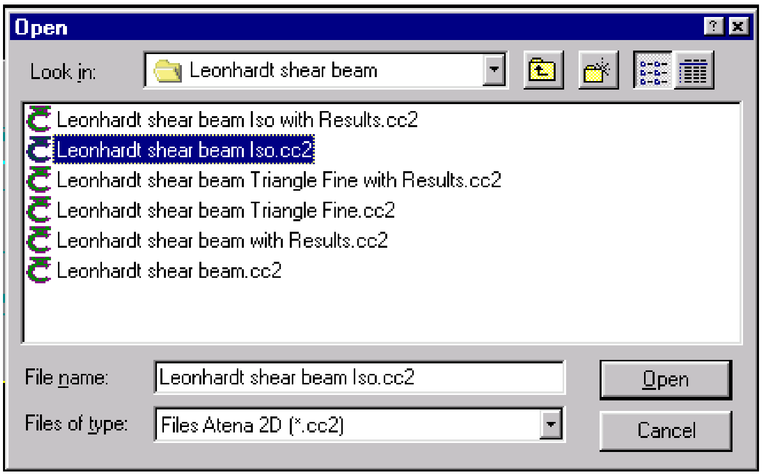


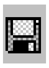
Fig. 2-3 Open existing file.

Clicking on the mark  in toolbar has the same effect as this command.

2.3 Saving a File (File | Save)

A current ATENA file can be saved using this command. A dialogue window shown in

Fig. 2-4 appears if the file has not yet been given a name. A file name should be written in this dialogue box and by clicking the **Save** button the file is saved.

Clicking on the mark  in the toolbar has the same effect as this command.

2.4 Saving a File Under a Specified Name (File | Save as)

It is possible to save a file under a chosen name. A dialogue window shown in

Fig. 2-4 will appear after clicking on the **File | Save as** command. A file name can be entered through the **File name** box and then the file can be saved.

One of the following file types can be chosen:

1. Files Atena 2D (*.cc2): Saves both the input data and result data.

Files Atena 2D – input data only (*.cc2): Saves only input data generated by pre-processing,

2. Fig. 2-5.

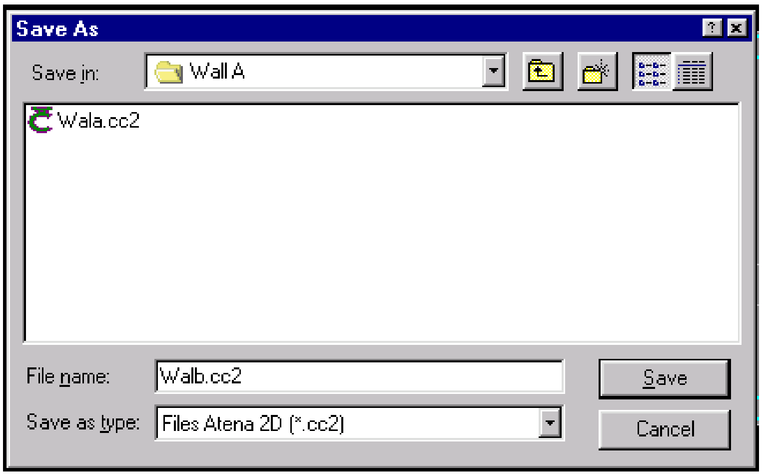


Fig. 2-4 Save ATENA file.

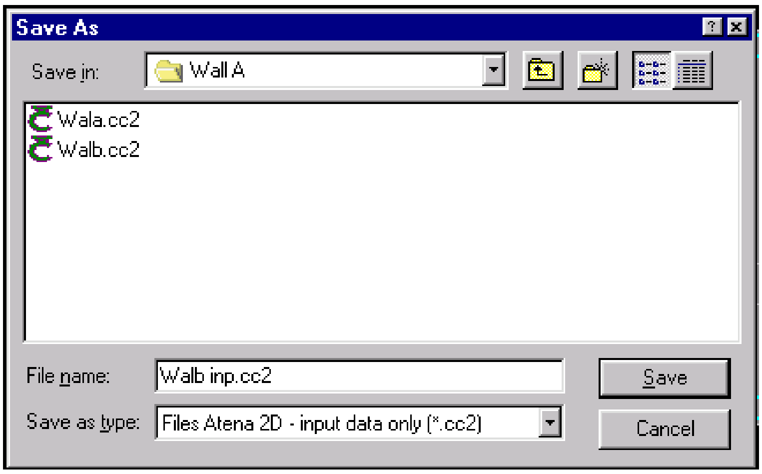


Fig. 2-5 Option Save as "Files Atena 2D - input data only (*.cc2)".

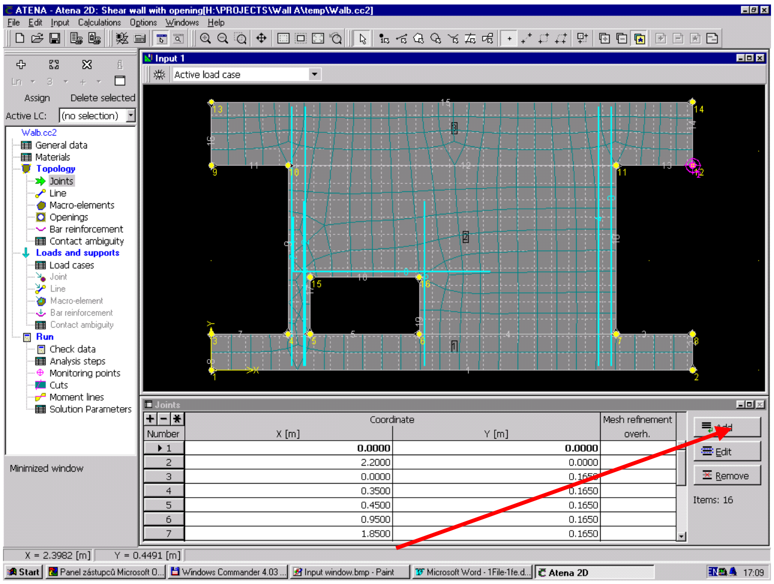


Fig. 2-6 Original file name remains in ATENA after saving input only

Note: In case of saving a file as “Files Atena 2D - input data only (*.cc2)” the original file stays in **ATENA** after saving input data only. Example: we are working on a file named Walb.cc2. Then we save it under the name “Walb inp.cc2” by using “Files Atena 2D - input data only (*.cc2)”. After the saving process, the file in **ATENA** is still Walb.cc2 and not the file Walb inp.cc2, see Fig. 2-6.

In the case of saving a file as “Files Atena 2D (*.cc2)”, the file displayed in the main window after the saving process will be the newly named file.

2.5 Import/Export Data of Other Formats

2.5.1 Input Data Formats

The input data can be imported form several formats. The formats offer different range of input data transfer according to data origin. Thus imports from DXF an IFC are limited to some geometrical data and those from CCT and SBD cover most of ATENA input data.

SBD – Data file from the **SBETA** program. **SBETA** was the program preceding **ATENA**. Most **SBETA** data is transferred into **ATENA**. It is strongly recommended to check the data after import, since the meaning of some data has changed.

CCT – ATENA text data format.

DXF – Autocad data format. Lines and points are imported.

IFC – According to the standard Industry Foundation Class version 2X. Currently only two objects can be imported: beams and columns.

After data import into **ATENA** it is recommended to check and consolidate the geometry by executing the command **Correct topology** form the menu **Edit**.

2.5.2 Structure of CCT Data

The following is a very brief description of the syntax of this file format. In order to understand this command in more detail it is recommended to export CCT file for a known problem and then study the created output. The following symbols are used in the paragraphs below:

<value_description> - indicates a description of a value or character set that should be inserted at this position in the CCT file.

\ - backslash character indicates that the commands on the following line should continue on the same line.

(*– indicates comment beginning

*) – denotes a comment end. Please note that no comments are allowed in the CCT file. Here the comments are used only to provide additional description.

```
FILETYPE ATENA_TEXT
VERSION <version, currently 0>

BEG_HEADER
    DESCRIPTION "<task name>"
    COMMENT "<comment to task>"
    NUMPLANES <no. of layers of smeared reinforcement>
    STRUCTYPE "<analysis type, 2D or AxiSymmetric>"
END_HEADER

BEG_MATERIALS
COUNT <no. of materials>

  BEG_MAT <material number id>
   ver_mat <version zápisu materiálu>
   <CCM file>
  END_MAT <material number id>

END_MATERIALS

BEG_TOPOLOGY

  BEG_JOINTS
  COUNT <number of joints>
    <joint no.> <X> <Y> for each joint
  END_JOINTS

  BEG_LINES
  COUNT <number of lines>
    <line no.> <type> continuation according to typu:
        line LI <joint no. beginning> <joint no. end>
        arc  AR <joint no. beginning> <joint no. end>
                   <center coordinate X> <center coordinate X>
                   <radius> <orientation (+) or (-)>
        circle CI <center coordinate X> <center coordinate  Y>
                  <radius> <orientation (+) or (-)>
                  <fictitious origin at 0>
  END_LINES

  BEG_MACROELEMENTS
  COUNT <number of macroelements>
    <no.> <material no.> <thickness > <list of lines on border>
```

```
          (* for each macroelement *)
          (* for undefined material no.=-1 *)
  END_MACROELEMENTS

  BEG_OPENINGS
  COUNT 0
    <opening no.> <list of border lines> (* for each opening *)
  END_OPENINGS

  BEG_REINFORCEMENTS
  COUNT <no. of reinforcements>
    <reinforcement no.> <material id> <section area> <type> \
                        <no. of segments>
         (* type: NORM for normal, EXTC for cable, *)
        (* TANG for circumferential *)
        (* for each segment
        <type>
          beginnin BEG <X> <Y>
          line LIN <X end> <Y end>
          arc ARC <X end> <Y end> <X center> <Y center> \
              <radius> <orientation (+) or (-)>
          circele CIR <X center> <Y center> <radius>
  END_REINFORCEMENTS

END_TOPOLOGY

BEG_LOADING
COUNT <number of load cases>

  for each load case
  BEG_LOADCASE
    NAME "<name of load case>"
    CODE <load case type>
    COEFFICIENT <koefficient>
    according to code (load case type)
      dead load: SW
        DIRECTION  <X unit vector component> <Y unit vector component>
      force: FO
        BEG_JOINTLOAD
        COUNT <number of joints>
           <joint number> <macroelement no., -1 for undefined > \
           <orientation GLBX, GLBY, GLRT> <amplitude> \
               [X rotation for GLRT] [Y rotation for GLRT]
        END_JOINTLOAD
        BEG_LINELOAD
        COUNT <number of lines>
           <line no.> <macroelement no., -1 for undefined>
           <type> (* depending on type *)
             (* continuous: COFL *)
               <dir 1 - see bellow> <value> [X rotation] [Y rotation]
             (* point force: PTLD *)
               <dir 1 - see bellow> <value> <distance from origin of line>
                   [X rotation] [Y rotation]
             (* trapezoid partial: QUPA *)
               <dir 1 - see bellow> <value origin> <value end> \
                                    <distance from origin of line>
               <length> [X rotation] [Y rotation]
             (* trapezoid on full: QUFL *)
               <dir 1 - see bellow> <value origin> <value end>
                   [X rotation] [Y rotation]
             (* dir1 : GLXP global on projection X,
                       GLYP global on projection Y,
                       GLXA global along length X,
                       GLYA global along length Y,
                       GLRA global alogn lengt rotated,
                       LOXA local along length X,
                       LOYA local along length Y,
                       LORA local along length rotated
```

```
            dir2 : GLBX globl X, GLBY globl Y, GLRT globl rotated,
                   LOCX local X, LOCY local Y, LORT local rotated *)
          [rotation] (* only for rotated *)
        END_LINELOAD

      (* supports: SU  *)
        BEG_JOINTLOAD
        COUNT <number of loaded joints>
          <joint number> <macro-element, -1 for undefined> \
                         <direction GLOB, GLRT> <support X FREE, FIXD> \
                         <support Y FREE, FIXD> [X rotation] [Y rotation] \
        END_JOINTLOAD
        BEG_LINELOAD
        COUNT <number of loaded lines>
          <line number> <macro-element, -1 for undefined> \
                        <direction GLOB, GLRT, LOCL, LCRT> \
                        <support X FREE, FIXD> <support Y FREE, FIXD> \
                        [X rotation] [Y rotation]
        END_LINELOAD
      (* prescribed def. DF *)
        BEG_JOINTLOAD
        COUNT <number of loaded joints>
          <joint number> <macro-element, -1 for undefined> \
                         <direction GLOB, GLRT> <support X FREE, FIXD> \
                         [X value for FIXD] <support Y FREE, FIXD> \
                         [Y value for FIXD] [X rotation] [Y rotation]
        END_JOINTLOAD
        BEG_LINELOAD
        COUNT <number of loaded lines>
          <line number> <macro-element, -1 for undefined> \
                        <direction GLOB, GLRT, LOCL, LCRT> \
                        <support X FREE, FIXD> [X value for FIXD] \
                        <support Y FREE, FIXD> [Y value for FIXD] \
                        [X rotation] [Y rotation]
        END_LINELOAD
      (* temperature TM *)
        BEG_MACROLOAD
        COUNT <number of loaded macro-elements>
          <macro-element number> <-1> <delta T>
        END_MACROLOAD
        BEG_REINFLOAD
        COUNT <number of loaded reinforcement bars>
          <bar number> <-1> <delta T>
        END_REINFLOAD
      (* shrinkage SH *)
        BEG_MACROLOAD
        COUNT <number of loaded macro-elements>
          <macro-element number> <-1> <epsilon>
        END_MACROLOAD
      (* pre-stressing PS *)
        BEG_REINFLOAD
        COUNT <number of loaded reinf. bars>
          <bar number> <-1> <force>
        END_REINFLOAD
    END_LOADCASE

END_LOADING
```

2.6 Open a Recently Used ATENA File (File | Open Again)

It is easy to open a recently used ATENA file by this command. Select the file from the list as shown in Fig. 2-7.

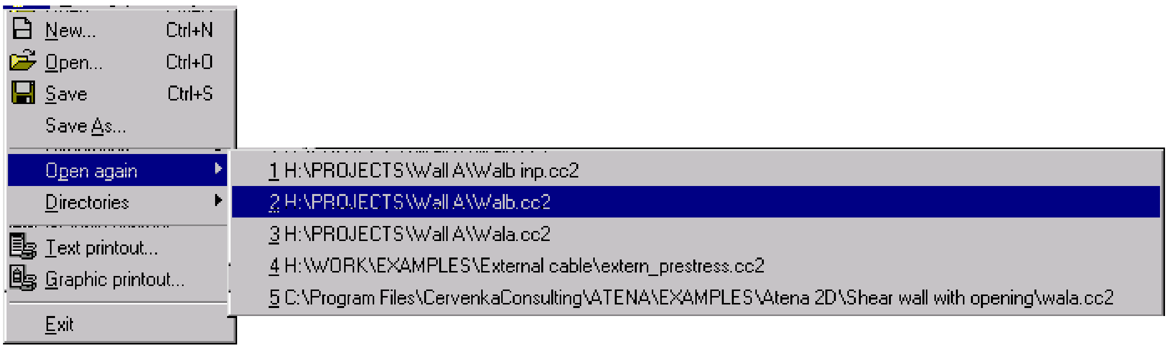


Fig. 2-7 Open file again.

2.7 Using a Recently Used Directory (File | Directories)

It is easy to open an ATENA file in a recently used directory by this command. After clicking on **File | Directories** shown as in Fig. 2-8, a directory file list will appear and a file can be selected and opened.

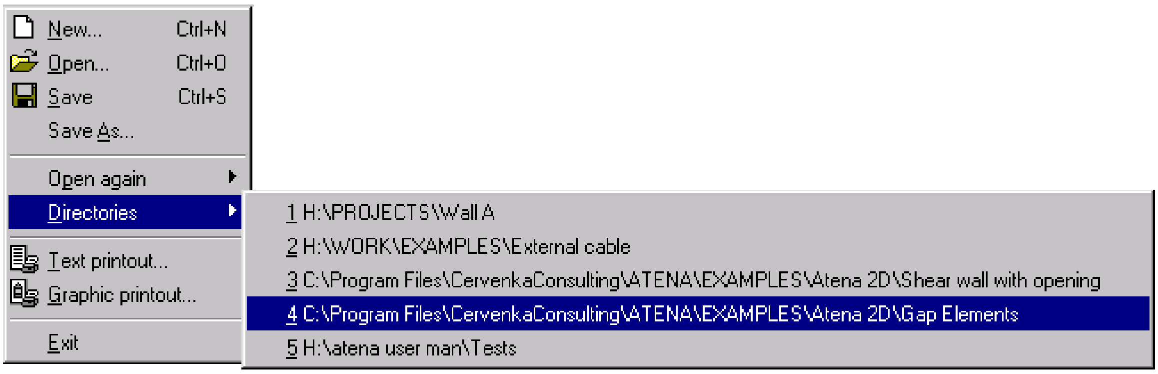


Fig. 2-8 Open directory again.

2.8 Display and Printing the Numerical Output Data (File | Text Printout....)

Display and/or printing of the analysis data and results can be done using this command. After clicking on **File | Text printout....**, a window shown in Fig. 2-9 will appear.

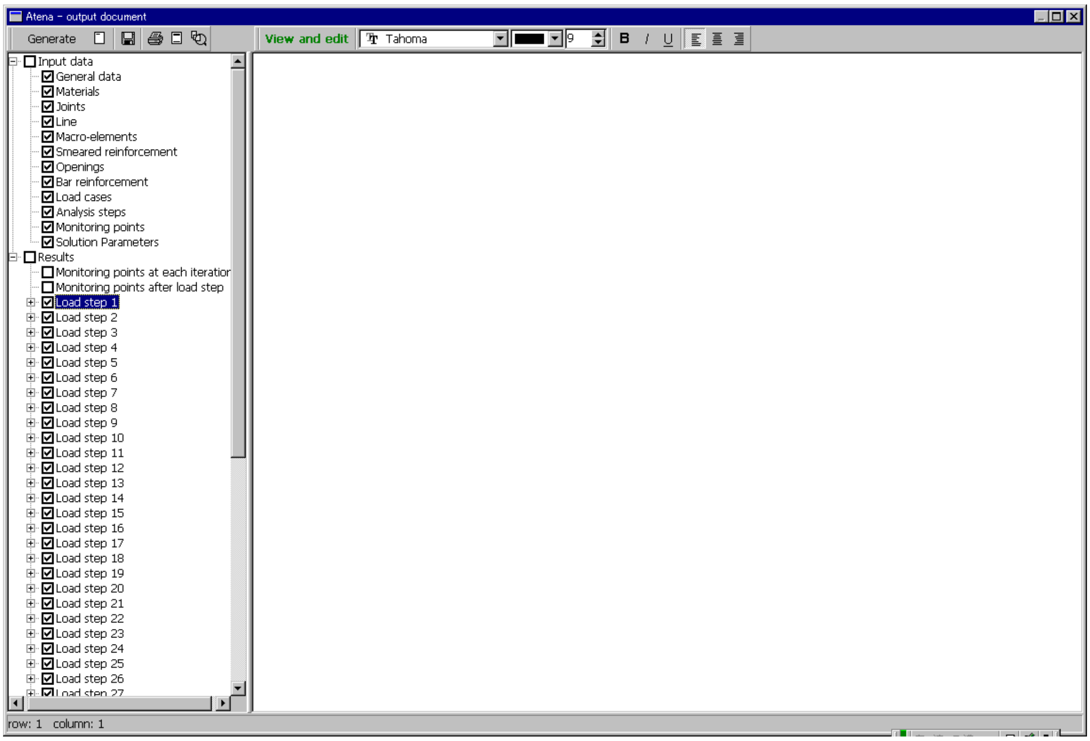


Fig. 2-9 Window for numerical output data.

2.8.1 Displaying Text Data

It is possible to display data in a text form by the following procedure.

(1) A data tree structure is displayed on the left side of the window as shown in Fig. 2-9. Desired output items must be marked in corresponding check boxes. However, the check boxes must be marked all the way from the top level to the desired item. Marking only the check box of a desired item will produce no data display. For details see the *Example* below.

(2) By clicking the 

button on the left side of the toolbar shown in Fig. 2-9, an alphanumerical text output will be created in the right field.

Example:

It is desired to generate a numerical output of principal stresses in the load step 1 in the finite element mesh nodes. This option is ensured by checking the boxes: **Results**, **Load step 1**, **Nodes**, **Principal Stress** as shown in Fig. 2-10. If some of the box checks in the tree are omitted, for example **Results** the data will NOT BE GENERATED.

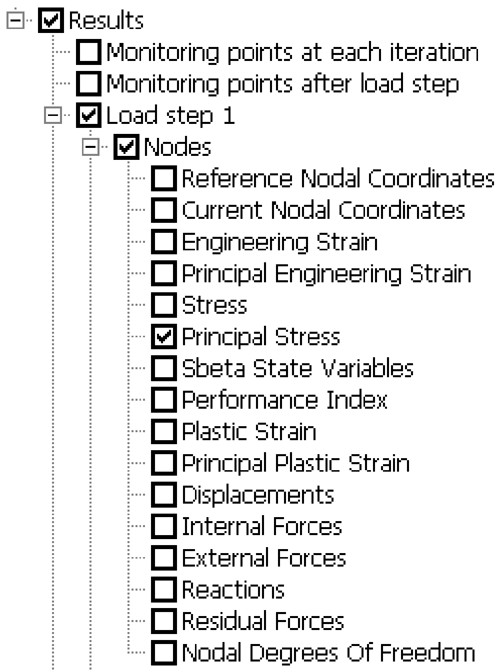


Fig. 2-10 Checking desired numerical data output.

2.8.2 Input Data Structure

This section deals with the input data, which define the numerical model, and are sent to the solution core for analysis. They are displayed in graphical form on the screen and can be also presented in a text form. A brief summary of data is listed in the table below and examples of data are shown in the following subsections.

Data item	*Contents*
Input data -	
General data	General data: analysis title and data specified in **Input | Basic data** section. Data displayed: General data
Materials	Material properties and data specified in the section **Input | Materials**. Data displayed: Materials
Joints	Displays joints coordinates and data specified in **Input | Topology | Geometrical joints** section. Data displayed: Joints
Lines	Line data. Joint numbers defining each line and data specified in **Input | Topology | Geometrical lines** section. Data displayed: Line
Macro-elements	Macro-element and data specified in **Input | Topology | Geom. Macro-elements** section. Data displayed: Macro-elements
Smeared reinforcement	List of Smeared reinforcement
Openings	Line numbers defining openings and data specified in **Input | Topology | Openings** section. Data displayed: Openings
Bar reinforcement	Steel reinforcement bar's cross-sectional area and data specified in **Input | Topology | Bar reinforcement** section. Data displayed: Bar reinforcement
Load cases	Loading and support conditions and data specified in **Input | Loads and supports | Load cases** section. Data displayed: Load cases Joint Line Macro-element Bar reinforcement
Analysis steps	Loading load cases, solution methods, coefficient and data specified in **Input | Analysis Data | Analysis steps** section. Data displayed: Analysis steps
Monitoring points	Monitoring points and data specified in **Input | Analysis Data | Monitoring** section. Data displayed: Monitoring points
Cuts	Definition of cut geometry. Data displayed: Cuts
Moment lines	Definition of moment line geometry. Data displayed: Moment lines
Solution parameters	Solution methods and parameters. Data specified in **Input | Analysis Data | Solution parameters** section. Data displayed: Solution Parameters

2.8.3 *Input Data Display – Examples*

2.8.3.1 Input Data - General Data

An example of general data display:

```
General data
Desc.   :Leonhardt shear beam
Note     : Using Q10Sbeta elements and SBETA material
Num. of smeared reinf. layers   :2
```

2.8.3.2 Input Data - Materials

Material properties of all defined materials are displayed.

An example of material data display:

```
Materials
Material n. 1
 Name : SBeta Material
 Type: CCSBETAMaterial
 Elastic modulus E = 3.172E+04 [MPa]
 Poisson''s ratio sm = 0.200 [-]
 Tensile strength F_t = 1.640E+00 [MPa]
 Compressive strength F_c = -2.848E+01 [MPa]
 Type of tension softening : Exponential
 Specific fracture energy G_f = 1.000E-04 [MN/m]
 Crack model: Fixed
 Compressive strain at comp. strength in the uniax. test Eps_C = -1.795E-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-03 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]

Material n. 2
 Name : Plane Stress Elastic Isotropic
 Type: CCPlaneStressElastIsotropic
 Elastic modulus E = 2.000E+05 [MPa]
 Poisson''s ratio sm = 0.300 [-]
 Specific material weight Rho = 7.850E-03 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]

Material n. 3
 Name : Reinforcement
 Type: CCReinforcement
 Typ: BiLinear
 Elastic modulus E = 2.080E+05 [MPa]
 Sigma Y = 560.000 [MPa]
 Specific material weight RHO = 7.850E-03 [MN/m3]
 Coefficient of thermal expansion ALPHA = 1.200E-05 [1/K]
```

2.8.3.3 Input Data - Joints

Example of geometrical joint data:

Joints

Joint topology

Number	Coordinates X [m]	Y [m]
1	0.0000	0.0000
2	0.2500	0.0000
3	0.3500	0.0000
4	1.2750	0.0000
5	1.2750	0.3200
6	1.1475	0.3200
7	1.0725	0.3200
8	0.0000	0.3200
9	0.2500	-0.0300
10	0.3000	-0.0300
11	0.3500	-0.0300
12	1.0725	0.3500
13	1.1100	0.3500
14	1.1475	0.3500
15	1.0725	0.0000

Mesh refinement at joints

No joint mesh refinement is specified

Joint springs

No joint springs are specified

2.8.3.4 Input Data - Line

Example of geometrical line data:

Line

Line topology

Number	Segment line	Joints Beg.	End	Center X [m]	Y [m]	Radius R [m]	Orient. [+/-]	Fictiv beg. [°]
1	Line	1	2					
2	Line	2	3					
3	Line	3	15					
4	Line	4	5					
5	Line	5	6					
6	Line	6	7					
7	Line	7	8					
8	Line	8	1					
9	Line	2	9					
10	Line	9	10					
11	Line	10	11					
12	Line	11	3					
13	Line	7	12					
14	Line	12	13					
15	Line	13	14					
16	Line	14	6					
17	Line	15	4					
18	Line	15	7					

Mesh refinement. at lines

No line mesh refinement is specified

Line contacts

Number line	Connection type	Material	Thickness [m]

```
----------------------------------------------
    2 fixed
    6 fixed
   18 fixed
```

Line springs

```
No line springs are defined
```

2.8.3.5 Input Data - Macro-elements

Example of macro-element data:

Macro-elements

Macro-element topology

```
------------------------------------------------------------------------------
Number Material           Thickness Line list
                               [m]
------------------------------------------------------------------------------
    1 SBeta Material        0.1900 1, 2, 3, 7, 8, 18
    2 Plane Stress El       0.1900 2, 9, 10, 11, 12
    3 Plane Stress El       0.1900 6, 13, 14, 15, 16
    4 SBeta Material        0.1900 4, 5, 6, 17, 18
```

Mesh generation parameters

```
-------------------------------------------------------------------------
Number Mesh type       Elem. size  Smoothing  Quad type   Method
                               [m] Mesh       elem.       analysis
-------------------------------------------------------------------------
    1 quadrilaterals        0.0250 yes        CCQ10SBeta  linear
    2 quadrilaterals        0.0250 yes        CCIsoQuad   linear
    3 quadrilaterals        0.0250 yes        CCIsoQuad   linear
    4 quadrilaterals        0.0250 yes        CCQ10SBeta  linear
```

Data on macro-element topology are displayed.

2.8.3.6 Input Data - Smeared Reinforcement

Example of smeared reinforcement data:

Smeared reinforcement

```
---------------------------------------
Number  Layer Material
 macr.
---------------------------------------
     1      1 [SBD] Horizontal smeared
     1      2 [SBD] Vertical smeared in
     2      1 [SBD] Horizontal smeared
     2      2 [SBD] Vertical smeared in
     3      1 [SBD] Horizontal smeared
     3      2 [SBD] Vertical smeared in
```

Note: The mark [SBD] means that this data was imported form SBETA file.

2.8.3.7 Input Data - Openings

Example of opening data:

Openings

Openings topology

```
------------------------------------------------------------------------------
Number Line list

------------------------------------------------------------------------------
  1 5, 6, 7, 8
```

2.8.3.8 Input Data - Bar Reinforcement

Example of bar reinforcement data:

```
Bar reinforcement
Reinforcement top.
-----------------------------------------------------------------------------
Number Topology - segments [m]
-----------------------------------------------------------------------------
    1 Beg. (0.3700, 0.0200), Lin.to(0.3700, 1.2050)
    2 Beg. (0.4300, 0.0200), Lin.to(0.4300, 1.2050)
    3 Beg. (1.8300, 0.0200), Lin.to(1.8300, 1.2050)
    4 Beg. (1.7700, 0.0200), Lin.to(1.7700, 1.2050)
    5 Beg. (0.4250, 0.0200), Lin.to(0.4250, 0.7700)
    6 Beg. (0.9750, 0.0200), Lin.to(0.9750, 0.7700)
    7 Beg. (0.3750, 0.7000), Lin.to(0.3750, 0.4500)
    8 Beg. (0.3750, 0.4500), Lin.to(1.2750, 0.4500)

Reinforcement properties
-----------------------------------------------------------------------------
Number Segment Material             Area              External cable
                                    [m2]  Act.anchor  Coeff.[-]  C[MPa]  R[m]
-----------------------------------------------------------------------------
    1 norm.  Bar d12           2.260E-04
    2 norm.  Bar d12           2.260E-04
    3 norm.  Bar d12           2.260E-04
    4 norm.  Bar d12           2.260E-04
    5 norm.  Bar d10           7.900E-05
    6 norm.  Bar d10           7.900E-05
    7 norm.  Bar d8            5.000E-05
    8 norm.  Bar d8            5.000E-05
```

2.8.3.9 Input Data - Load Cases

Example of load case data:

```
Load case 1
Properties
Name:           Support
Coefficient :         1.000 [-]
Code :          Supports

Joint support
No joint supports are prescribed

Line support
--------------------------------------------------------
 Line    Support    Direction        Axis X rotation
numbe X      Y                         X [m]       Y [m]
--------------------------------------------------------
    1 fixed  fixed  Global

Load case 2
Properties
Name:           Loading
Coefficient :         1.000 [-]
Code :          Prescribed deformation

Joint deformation
----------------------------------------------------------------------------
Join.        Support and deformation        Direction       Axis X rotation
numbe X              [m] Y             [m]                  X [m]       Y [m]
----------------------------------------------------------------------------
   12 fixed   -1.000E-04 free               Global

Line deformation
No line deformations are prescribed
```

2.8.3.10 Input Data - Analysis Steps

Example of analysis step data:

```
Analysis steps
-----------------------------------------------------------------------------
Number Parameters      Coefficient Load case list
                              [-]
------------------------------------------------------------------------------
    1 Arc Length           1.000 1, 2
    2 Arc Length           1.000 1, 2
    3 Arc Length           1.000 1, 2
    4 Arc Length           1.000 1, 2
    5 Arc Length           1.000 1, 2
    6 Arc Length           1.000 1, 2
    7 Arc Length           1.000 1, 2
    8 Arc Length           1.000 1, 2
    9 Arc Length           1.000 1, 2
   10 Arc Length           1.000 1, 2
```

2.8.3.11 Input Data - Monitoring Points

Example of monitoring data:

```
Monitoring points
----------------------------------------------------------------------------
Number Location         Coordinate                       Specification
                       X [m]       Y [m] Value                 Item
----------------------------------------------------------------------------
    1 Node            2.2000      0.9360 Displacements         Component 1
    2 Node            2.2000      0.9350 Reactions             Component 1
```

2.8.3.12 Input Data - Solution Parameters

Example of solution parameter data:

```
Solution Parameters
Solution parametrs n.1
 Name : Standard solution parameters
 Method: Newton-Raphson
 Iteration Limit: 20
 Displacement Error 0.010 [-]
 Residual Error 0.010 [-]
 Absolute Residual Error 0.010 [-]
 Energy Error 0.010 [-]
 Optimize Band-Width:
 Line Search: On
  Line Search Type: Without Iterations
  Minimum Eta: 0.100 [-]
  Maximum Eta: 10.000 [-]
 Update Stiffness: Each Step
 Stiffness Type: Tangent

Solution parametrs n.2
 Name : Arc Length
 Method: Arc-Length
  Arc-Length Method: Explicit Orthogonal
  Arc-Length Adjustment Method: Constant
  Load-Displacement Ratio       0.200 [-]
  Loading-Displacement Method: Constant
  Reference Number Of Iterations: 5
  Step Length: Based On First Load Step
```

```
 Arc-Length Location: All Nodes
Iteration Limit: 30
Displacement Error 0.010 [-]
Residual Error 0.010 [-]
Absolute Residual Error 0.010 [-]
Energy Error 0.010 [-]
Optimize Band-Width:
Line Search: On
 Line Search Type: With Iterations
 Unbalanced Energy Limit: 0.800 [-]
 Line Search Iteration Limit: 2
 Minimum Eta: 0.100 [-]
 Maximum Eta: 1.000 [-]
Update Stiffness: Each Iteration
Stiffness Type: Tangent
```

2.8.4 Result Data Structure

This section deals with the result data, which come out of the finite element analysis. They are displayed in a graphical form on the screen in the post-processing mode and can be also presented in a text form. A brief contents of data is listed in the table below and examples of data are shown in the following subsections.

2.8.4.1 Result Data of Monitored Points

		Contents
Results	Monitoring points at each iteration	Monitored data in each iteration.
	Monitoring points after load step	Monitored data in each load step.

2.8.4.2 Result Data Evaluated in Nodes

The meaning of nodal values depends on data type. The state variables, such as stress and strain, are weighted average values of element nodal values, see Sect.2.8.4.5. The weights are proportional to the element volumes. Thus, in case of regular mesh, with elements of equal sizes the nodal value is an average of element nodal values.

The averaging is made only for nodes belonging to the same group. This is made possible by considering two nodes in the same location on a line connecting two groups of elements (macro-elements). Thus, one node can be a part of only one group. The rigid (or other) connection of nodes between groups is facilitated by the 'Master-slave' method.

The vector variables, such as displacements, forces and coordinates, describe the vector components. Following table summarizes data contents in the tree structure.

				Contents
Results	Load step *i*	Nodes	Reference Nodal Coordinates	Initial nodal coordinates.
			Current Nodal Coordinates	Coordinates of each node in deformed configuration.

			Engineering Strain	Strains $\varepsilon_x, \varepsilon_y, \gamma_{xy}$ of basic group (see 2.8.4.8).
			Principal Engineering Strain	Principal strains and a direction vector in nodes.
			Stress	Stress components σ_x, σ_y, τ_{xy} of basic group (see 2.8.4.8).
			Principal Stress	Principal stresses in nodes of basic group (see 2.8.4.8).
			Sbeta State Variables	State variables of 'Sbeta Material' in each node.1). See section 2.8.5.9 for details.
			Performance Index	Relative error in the evaluation of the material model at each node.1)
			Engineering Strain Smeared *j*	Engineering strain of smeared reinforcement. 2)
			Principal Engineering Strain Smeared *j*	Principal engineering strain of smeared reinforcement. 2)
			Stress Smeared *j*	Stress of smeared reinforcement. 2)
			Principal Stress Smeared *j*	Principal stress of smeared reinforcement. 2)
			Plastic Strain Smeared *j*	Plastic strain of smeared reinforcement. 2)
			Plastic Strain	Plastic strain of concrete, steel, or bar reinforcement in nodes.
			Principal Plastic Strain	Principal plastic strain of concrete, steel, or bar reinforcement in nodes.
			Displacements	Displacements of each node.
			Nodal Degrees Of Freedom	A location in the global stiffness matrix.
			(Compact/Partial) Internal Forces	Nodal forces.
			(Compact/Partial) External Forces	External forces acting in nodes.
			(Compact/Partial) Reactions	Reactions in nodes.
			(Compact/Partial) Residual Forces	Residual forces in nodes. (Sum of external and internal forces acting in node.)
			Total Elem Init Strain	Initial strains (typically used to represent prestressing or shrinkage).
			Total Elem Init Stress	Initial stresses (typically used to represent prestressing).
			Elem Total Temperature	Thermal load.
			Total Elem Body Load	Typically used for dead weight.

			Von Mises Stress	Equivalent plastic stress for the von Mises yield criteria.
			Total Strain	Strains including initial strains.
			Principal Total Strain	Principal components of strains including initial strains.
			Tensile Strength	The current tensile strengths (in principal directions).
			Fracture Strain	Strain corresponding to crack opening.
			Principal Fracture Strain	Principal components of strain corresponding to crack opening.
			Maximal Fract Strain	The maximum fracture strain during the loading history so far.
			Rate factors	Note used, equal to 1.0
			Soft/Hard parameter	Hardening/softening parameter for the plastic model in Cementitious fracture-plastic materials. It corresponds to the ratio of current compressive strength to the one specified in the material definition: $f_c\left(\varepsilon_{eq}^{p}\right) / f_c$
			Eq Plastic Strain	Equivalent plastic strain for the Cementitious fracture-plastic material
			Rate factor	Note used, equal to 1.0
			Fc Reduction	Reduction of compressive strength due to concrete cracking in Cementitious fracture-plastic models
			Reference Border Coordinate	Coordinates for drawing 2D plots showing evolution of certain quantity along the path given by these values
			Interface Stress	Stress on contact elements.
			Interface Displacements	Relative displacements at contact elements.
			Interface Plast Displacements	Nonlinear part of the relative displacements at contact elements.
			Bond Slip	Reinforcement displacement relative to surrounding material.
			Bond Stress	Stress in the reinforcement contact.
			Cable Force	Force in reinforcement.

Notes:

1) Displayed only if 'Sbeta Material' is used.

2) The index 'j' stands for material number. Example: In the item 'Stress Smeared 5' the number 5 indicates the number of material in the material list as shown in Fig. 2-11.

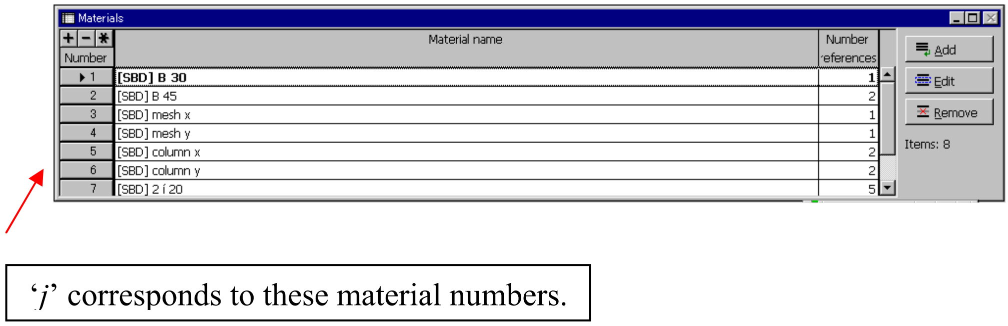


Fig. 2-11 Material numbers

In case of bar and smeared reinforcements the stress state is uniaxial and the values of ‘stress’ and ‘principal stress’ are identical. The same holds for strains in reinforcement.

2.8.4.3 Result Data Evaluated in Elements

Results	Load step *i*	Elements	Element Incidences	Nodal number of elements.
			Crack Attributes	Crack attributes of elements. See section 2.8.5.29 for details.
			Element Material Types	The material types used for each element.
			Element Output Options	Output-related settings for each element (activity, interpolation, etc.)

The crack attributes define the crack direction and length within the element, crack opening and stress components, normal and shear, on the crack plane.

2.8.4.4 Result Data Evaluated in Element Integration Points

These data have the same type of information as in the nodes, see sections 2.8.4.5 and 2.8.4.2. It should be realized, that the integration points are the ‘true’ locations, where the response of material model is evaluated (except for true nodal quantities like Displacements, Reference/Current Nodal Coordinates, Bond Slip, Compact/Partial Internal/External/Residual Forces, Reactions). The material state variables displayed in other locations, such as element nodes, can be somewhat changed due to extrapolation.

Results	Load step *i*	Elements	IP Coordinates	Coordinates of the integration points.
			Material Transformation Matrix	Transformation matrix used in Cementitious fracture-plastic materials for transformation from global to local coordinate system of the cracked material

			Cracking Moduli	Cracking moduli $\mathbf{E}^{'cr}$ of the Cementitious fracture-plastic material.
			Direction Status	Status variable in the Cementitious fracture-plastic materials indicating material fracture state: 1-virgin, 2-crack, 4-fully cracked.
			yield/crush info	Status variable in the Cementitious fracture-plastic material indicating crushing state: 1-virgin, 2-plastic hardening, 4-softening
			Reference Ip Coordinates	Initial coordinates of the integration points.
			Displacements At Ips	Displacements of the integration points

2.8.4.5 Result Data Evaluated in Element Nodes

These data have the same type of information as in integration points, see section 2.8.4.4. Their values are extrapolated from integration points to element nodal points. The values of material variables (stresses, strains) in the same node, but in different element, can be different. (However, in mesh nodes, sect. 2.8.4.2, only one average value is shown in nodes belonging to the same element group.)

2.8.4.6 Global Data

				Contents
Results	Load step *i*	Global	FE model Characteristics	Numbers of: nodes, groups, element types, materials, dimensions.
			Task Name	Task name. 1)
			Task Title	Title.
			Solution Characteristics	Parameters of used solution methods.
			Arc Length Params.	Arc Length parameters
			Line Search Params.	Line Search method parameters.
			Convergence Criteria	Convergence status.
			Step Id	Load step number.
			Step Convergence	Convergence status at step end.
			Eigenvalue Characteristics	Parameters used for the eigenvalue solution

Notes: 1) Default name and title is 'Test'.

2.8.4.7 Load Cases Data

				Contents
Results	Load step *i*	Load cases	Support Slave Nodes	List of support slave nodes.
			Support Master Nodes	List of support master nodes.
			Load Slave Nodes	List of load slave nodes (loaded nodes).
			Master Slave Nodes	List of Master-Slave constraints

2.8.4.8 MNQ forces in nodes

Results for moment lines (see 4.7.5 and 8.3), global nodes interpolation (see also 2.8.4.2).

2.8.4.9 MNQ forces in element nodes

Results for moment lines (see 4.7.5 and 8.3), element nodes interpolation (see also 2.8.4.5).

2.8.4.10 Element Groups in Results

The data are grouped according to their locations in a basic group and layers. In this classification the basic group includes elements with the basic material (for example Sbeta, Von Mises, etc.) and discrete reinforcement. For example, if one checks a data box for results in 'stress', this makes a request for data of all concrete elements and all truss elements of reinforcing bars. The stress data will include three stress components for concrete and only one stress component for bars.

The additional layers include smeared reinforcement. One smeared reinforcement in each layer. Description of a data box is composed of a data name and a material number. Example: **Stress smeared 5** means that request is made for stress in smeared reinforcement of the material type 5.

2.8.5 Result Data - Examples

2.8.5.1 Results - Load Step - Monitoring Points at Each Iteration

Example of monitoring two values at the same location: horizontal displacement and horizontal reaction. This output can be used to observe the convergence of the nonlinear solution.

Monitoring points at each iteration

```
Monitoring p. specif.
Data typ: Displacements - x(1)
Spec. location : X: 2.2000 [m], Y: 0.9360 [m], FE node
Calculated location: X: 2.2000 [m], Y: 0.9350 [m], Mesh node 429
Results at monitoring point
-----------------------
 Step Iter.       Value
                    [m]
-----------------------
    1     1  -8.000E-05
          2  -8.000E-05
    2     1  -1.600E-04
          2  -1.600E-04
    3     1  -2.400E-04
          2  -2.400E-04
          3  -2.400E-04
          4  -2.400E-04
          5  -2.400E-04
```

```
Monitoring p. specif.
Data typ: Reactions - Dof(1)
Spec. location : X: 2.2000 [m], Y: 0.9350 [m], FE node
Calculated location: X: 2.2000 [m], Y: 0.9350 [m], Mesh node 429
Results at monitoring point
-----------------------
 Step Iter.       Value
                   [MN]
-----------------------
    1     1  -4.308E-02
          2  -4.292E-02
    2     1  -8.588E-02
          2  -8.572E-02
    3     1  -1.286E-01
          2  -1.268E-01
          3  -1.282E-01
          4  -1.275E-01
          5  -1.276E-01
```

2.8.5.2 Results - Load Step - Monitoring Points After Load Step

Example of monitoring two values at the same location: horizontal displacement and horizontal reaction.

Monitoring points after load step

```
Monitoring p. specif.
Data typ: Displacements - x(1)
Spec. location : X: 2.2000 [m], Y: 0.9360 [m], FE node
Calculated location: X: 2.2000 [m], Y: 0.9350 [m], Mesh node 429
Results at monitoring point
-----------------
 Step       Value
              [m]
-----------------
    1  -8.000E-05
    2  -1.600E-04
    3  -2.400E-04
```

```
Monitoring p. specif.
Data typ: Reactions - Dof(1)
Spec. location : X: 2.2000 [m], Y: 0.9350 [m], FE node
Calculated location: X: 2.2000 [m], Y: 0.9350 [m], Mesh node 429
Results at monitoring point
-----------------
 Step       Value
             [MN]
-----------------
    1  -4.292E-02
    2  -8.572E-02
    3  -1.276E-01
```

2.8.5.3 Results - Load Step - Nodes - Reference Nodal Coordinates

Example of reference nodal coordinates. It contains a list of original nodal coordinates in the undeformed state.

```
Output data for request: REFERENCE_NODAL_COORDINATES
Description: Reference nodal coordinates
  Step: 3  Iteration: 5 at Time: 3
--------------------------------------------------------------------------------
 Node       X(1)       X(2)
Units          m          m
    1  0.0898000  0.0825000
    2  0.0898000  0.1650000
    3  0.00e+000  0.1650000
    4  0.00e+000  0.0825000
    5  0.00e+000  0.00e+000
    6  0.0899000  0.00e+000
    7  0.1802000  0.00e+000
    8  0.1796000  0.0828000
    9  0.1794000  0.1650000
   10  0.2709000  0.0846000
```

2.8.5.4 Results - Load step - Nodes - Current Nodal Coordinates

Example of current nodal coordinates. It contains a list of nodal coordinates in the current deformed state.

```
Output data for request: CURRENT_NODAL_COORDINATES
Description: Current nodal coordinates
  Step: 3  Iteration: 5 at Time: 3
----------------------------------------------------------------------------------
 Node       X(1)       X(2)
Units          m          m
    1  0.0897998  0.0825000
    2  0.0897998  0.1650000
    3 -2.21e-007  0.1649999
    4 -1.33e-007  0.0824999
    5  0.00e+000  0.00e+000
    6  0.0899000  0.00e+000
    7  0.1802000  0.00e+000
    8  0.1795996  0.0827999
    9  0.1793999  0.1649999
   10  0.2708991  0.0845991
```

2.8.5.5 Results - Load Step - Nodes - Engineering Strain

Example of engineering strain data in nodes.

```
Output data for request: ENGINEERING_STRAIN
Description: Engineering Strain
  Step: 10  Iteration: 3 at Time: 10
-------------------------------------------------------------------------------------
 Node     eps_xx     eps_yy   gamma_xy
Units       None       None       None
    1 -0.0002095  0.0000214 -0.0000792
    2 -0.0002214  0.0000115 -0.0000908
    3 -0.0003265  0.0000159 -0.0001343
    4 -0.0002977  0.0000359 -0.0000888
    5 -0.0002545  0.0000491 -0.0000426
    6 -0.0001878  0.0000314 -0.0000622
    7 -0.0000582  2.54e-006 -0.0000715
    8 -0.0000585 -1.64e-006 -0.0000609
    9 -0.0000588 -2.02e-006 -0.0000501
   10  0.0000768 -6.12e-006 -0.0000826
```

2.8.5.6 Results - Load step - Nodes - Principal Engineering Strain

Example of principal engineering strain data in nodes.

```
Output data for request: PRINCIPAL_ENGINEERING_STRAIN
Description: Principal Engineering Strain
  Step: 10  Iteration: 3 at Time: 10
------------------------------------------------------------------------------------
 Node       Max.       Min.       v1_x       v1_y       v2_x       v2_y
Units       None       None          m          m          m          m
    1  0.0000280 -0.0002161 -0.1644553  0.9863845 -0.9863845 -0.1644553
    2  0.0000200 -0.0002300 -0.1846883  0.9827971 -0.9827971 -0.1846883
    3  0.0000286 -0.0003392 -0.1857439  0.9825982 -0.9825982 -0.1857439
    4  0.0000417 -0.0003035 -0.1296837  0.9915554 -0.9915554 -0.1296837
    5  0.0000506 -0.0002560 -0.0695833  0.9975761 -0.9975761 -0.0695833
    6  0.0000357 -0.0001922 -0.1379067  0.9904452 -0.9904452 -0.1379067
    7  0.0000191 -0.0000747 -0.4197372  0.9076457 -0.9076457 -0.4197372
    8  0.0000116 -0.0000717 -0.3986226  0.9171151 -0.9171151 -0.3986226
    9  7.47e-006 -0.0000683 -0.3539666  0.9352581 -0.9352581 -0.3539666
   10  0.0000939 -0.0000232  0.9242318 -0.3818320  0.3818320  0.9242318
```

Legend:

Max – maximal principal strain,
Min – minimal principal strain,
v1_x, v1_y – components of direction unit vector of Max,
v2_x, v2_y – components of direction unit vector of Min.

Principal strains represent the extreme values of strain tensor components. For details see the documentation *ATENA Theory, section 2.1.1 Basic assumptions*.

2.8.5.7 Results - Load step - Nodes – Stress

Example of data on stress nodes.

```
Output data for request: STRESS
Description: Stress
  Step: 10  Iteration: 3 at Time: 10
-------------------------------------------------------------------------------------
 Node   sigma_xx   sigma_yy     tau_xy
```

```
Units        MPa        MPa        MPa
    1 -6.3276102 -0.6350027 -0.9821697
    2 -6.7385814 -1.0067875 -1.1163010
    3 -9.7677146 -1.4927037 -1.6222217
    4 -8.8145835 -0.7143098 -1.0757320
    5 -7.4999568 -0.0564204 -0.5214479
    6 -5.6342595 -0.1963931 -0.7817011
    7 -1.8324320 -0.2927624 -0.9194646
    8 -1.8689030 -0.4286103 -0.7843007
    9 -1.8850199 -0.4429914 -0.6473548
   10  0.6516723 -0.2410783 -0.5942988
```

2.8.5.8 Results - Load step - Nodes - Principal Stress

Example of principal stress data in nodes.

```
Output data for request: PRINCIPAL_STRESS
Description: Principal Stress
  Step: 10  Iteration: 3 at Time: 10
---------------------------------------------------------------------------------------
 Node       Max.       Min.       v1_x       v1_y       v2_x       v2_y
Units        MPa        MPa          m          m          m          m
    1 -0.4703095 -6.4923034 -0.1653742  0.9862309 -0.9862309 -0.1653742
    2 -0.7970555 -6.9483134 -0.1846504  0.9828043 -0.9828043 -0.1846504
    3 -1.1860496 -10.074369 -0.1857439  0.9825982 -0.9825982 -0.1857439
    4 -0.5738849 -8.9550084 -0.1294407  0.9915872 -0.9915872 -0.1294407
    5 -0.0200686 -7.5363086 -0.0695445  0.9975788 -0.9975788 -0.0695445
    6 -0.0862532 -5.7443993 -0.1395196  0.9902193 -0.9902193 -0.1395196
    7  0.1365945 -2.2617889 -0.4231067  0.9060799 -0.9060799 -0.4231067
    8 -0.0839851 -2.2135282 -0.4022817  0.9155159 -0.9155159 -0.4022817
    9 -0.1950218 -2.1329895 -0.3577058  0.9338343 -0.9338343 -0.3577058
   10  0.9485614 -0.5379674  0.8945839 -0.4469001  0.4469001  0.8945839
```

Legend:

Max – maximal principal stress,
Min – minimal principal stress,
v1_x, v1_y – components of direction unit vector of Max,
v2_x, v2_y – components of direction unit vector of Min.

Principal stresses represent the extreme values of stress tensor components. For details (of 2D) see the Theory manual, section 2.1.1 Basic assumptions.

2.8.5.9 Results - Load step - Nodes - Sbeta State Variables

Sbeta state variables contain data produced by the material model Sbeta. They contain a complete set of data available in the model and include some data available in other sections (principal stress and strain) as well as additional data (tension stiffening, shear retention factor, etc.). Example of Sbeta State Variables:

```
Output data for request: SBETA_STATE_VARIABLES
Description: SBETA Attributes
  Step: 3  Iteration: 5 at Time: 3
---------------------------------------------------------------------------------------
 Node         Y1         Y2        Rc1        Rc2        Rt1        Rt2         s1         s2
rec         ret       sc(1)      sc(2)      sc(3)     eps(1)     eps(2)     eps(3)      sd(1)
sd(2)      st(1)      st(2)      sx(1)      sx(2)      sx(3)      yd(1)      yd(2)      angle
tevol       Redg     rsm(1)     rsm(2)      ns(1)      ns(2)        ncr
Units       None       None       None       None       None       None       None       None
None       None       None       None       None       None       None       None       None
None       None       None       None       None       None       None       None       None
None       None       None       None       None       None       None
1   31689.222   31689.222   0.00e+000   0.00e+000   0.00e+000   0.00e+000   0.0073270  -0.0407077
1.0304559    0.9991744   -0.0263163   -0.0070643   -0.0163904   -7.86e-007   -5.69e-008   -1.24e-006
```

```
0.00e+000   0.00e+000   0.00e+000   0.00e+000  -0.0263163  -0.0070643  -0.0163904   0.00e+000
0.00e+000   1.4525297   0.00e+000   1.0000000   0.00e+000   0.00e+000   0.00e+000   0.00e+000
0.00e+000
```

Legend:

Node	Node number.
Y1	Secant modulus in direction 1 (max).
Y2	Secant modulus in direction 2 (min).
Rc1	Biaxial compressive strength in direction 1 at failure. See Theory 2.1.5.1.
Rc2	Biaxial compressive strength in direction 2 at failure. See Theory 2.1.5.1.
Rt1	Biaxial tensile strength in direction 1 at failure. See Theory 2.1.5.2.
Rt2	Biaxial tensile strength in direction 2 at failure. See Theory 2.1.5.2.
s1	Concrete stress in direction 1.
s2	Concrete stress in direction 2.
Rec	Compressive strength factor due to the biaxial stress. See Theory 2.1.5.1.
Ret	Tensile strength factor due to the biaxial stress. See Theory 2.1.5.2.
Sc(1)	Concrete stress normal component in dir. x, σ_{cx}.
Sc(2)	Concrete stress normal component in dir. y, σ_{cy}.
Sc(3)	Concrete stress shear component in dir. xy, τ_{cxy}.
Eps(1)	Strain component, normal, in dir. x, ε_x.
Eps(2)	Strain component, normal, in dir. x, ε_x.
Eps(2)	Strain component, shear, in dir. xy, γ_{xy}.
Sd(1)	Smeared reinf. stress. NOT USED in this version.
Sd(2)	Smeared reinf. stress. NOT USED in this version.
st(1)	Tension stiffening stress. NOT USED in this version.
st(2)	Tension stiffening stress. NOT USED in this version.
Sx(1)	Total stress component, normal, in dir. x, σ_x. (Here same as σ_{cx})
Sx(2)	Total stress component, normal, in dir. y, σ_y. (Here same as σ_{cy})
Sx(3)	Total stress component, normal, in dir. xy, τ_{xy}. (Here same as τ_{cxy})
Yd(1)	Smeared reinf. modulus. NOT USED here.
Yd(2)	Smeared reinf. modulus. NOT USED here.
Angle	Angle of principal strain or material axis 1.
Tevol	Volumetric strain due to temperature or shrinkage.
Redg	Shear retention factor.
Rsm(1)	Yield stress in smeared reinforcement. NOT USED here.
Rsm(2)	Yield stress in smeared reinforcement. NOT USED here.
Ns(1)	Material state number in dir. 1. (See Theory, 2.1.2.1)
Ns(2)	Material state number in dir. 2. (See Theory, 2.1.2.1)
Ncr	Crack state number.

Note: Units of Sbeta variables are not shown. They have units of stress and strain.

2.8.5.10 Results - Load Step - Nodes - Performance Index

The performance index is used for some materials only.

2.8.5.11 Results - Load step - Nodes - Engineering Strain Smeared

Engineering strains in smeared reinforcement are listed in nodes. Example of data:

```
Output data for request: ENGINEERING_STRAIN_SMEARED_5
Description: Engineering Strain
  Step: 3  Iteration: 5 at Time: 3
-----------------------------------------------------------------------------------------
 Node     eps_xx
Units       None
  509 -5.69e-008
  510  9.63e-008
  511  6.53e-008
  512 -1.83e-007
  513 -4.31e-007
  514 -2.09e-007
  515 -2.40e-006
  516 -1.29e-006
```

Strain is in a local direction of reinforcement. It should be realized, that the strains are averaged in the nodes. In the above example '..smeared_5' refers to the material type number.

The node numbers refer to the smeared reinforcement nodes. These nodes belongs to the smeared reinforcement layer and are different than those for the basic material – concrete.

2.8.5.12 Results - Load Step - Nodes - Principal Engineering Strain Smeared

In case of smeared reinforcement, which is in an uniaxial stress state, the principal strains are identical with strains.

2.8.5.13 Results - Load Step - Nodes - Stress Smeared

Stresses in smeared reinforcement are listed in nodes. Example of data:

```
Output data for request: STRESS_SMEARED_5
Description: Stress
  Step: 3  Iteration: 5 at Time: 3
-----------------------------------------------------------------------------------------
 Node   sigma_xx
Units        MPa
  509 -0.0113735
  510  0.0192659
  511  0.0130689
  512 -0.0365487
  513 -0.0861566
  514 -0.0418414
  515 -0.4803048
  516 -0.2570962
```

Stress is in a local direction of reinforcement. It should be realized, that the stresses are averaged in the nodes. In the above example '..smeared_5' refers to the material type number.

The node numbers refer to the smeared reinforcement nodes. These nodes are due to the smeared reinforcement layer and are different than those for the basic material – concrete.

2.8.5.14 Results - Load Step - Nodes – Principal Stress Smeared

In case of smeared reinforcement, which is in the uniaxial stress state, the principal stress is identical with stress.

2.8.5.15 Results - Load Step - Nodes - Plastic Strain Smeared

Plastic strains in smeared reinforcement are listed in the nodes. Average values in nodes are shown.

2.8.5.16 Results - Load Step - Nodes - Plastic Strain

Plastic strains in basic element group are listed in nodes. Example of data:

```
Output data for request: PLASTIC_STRAIN
Description: Plastic Strain
  Step: 3  Iteration: 5 at Time: 3
-------------------------------------------------------------------------------------
 Node   eps_p_xx
Units       None
 1288 -3.33e-021
 1289 -2.62e-021
 1290 -6.74e-021
 1291 -1.01e-020
 1292 -3.60e-021
 1293 -3.21e-021
 1294 -2.94e-021
```

Average values in nodes are shown. The above example shows the plastic strains in reinforcement in a local direction along the reinforcement. They are near zero in this load stage.

Plastic strains are calculated only in models based on the theory of plasticity. Examples of such materials are: 3D cementitious, Von Mises, Drucker-Prager, reinforcement. Therefore, in some materials small or zero plastic strains occurs or they are not presented at all. In 2D or 3D material models 2 or 3 plastic strain components can appear in the list.

2.8.5.17 Results - Load Step - Nodes - Principal Plastic Strain

Principal values of plastic strain tensor are listed in nodes.

```
Output data for request: PRINCIPAL_PLASTIC_STRAIN
Description: Principal Plastic Strain
  Step: 3  Iteration: 9 at Time: 3
------------------------------------------------------------------------------------
 Node       Max.       Mid.       Min.      vmax_x     vmax_y     vmax_z     vmid_x     vmid_y
vmid_z     vmin_x      vmin_y     vmin_z
Units       None       None       None           m          m          m          m          m
m          m          m          m
   80  0.00e+000  0.00e+000  0.00e+000  1.0000000  0.00e+000  0.00e+000  0.00e+000  1.0000000
0.00e+000  0.00e+000  0.00e+000  1.0000000
```

Number of principal components depends on the material model. In the above example, the material is '3D nonlinear cementitious' and thus has all three stress components. The principal directions are described by unit vector components vmax_x, vmax_y, vmax_z, for Max, vmid_x, vmid_y, vmid_z, for Mid, vmin_x, vmin_y, vmin_z, for Min.

In case of reinforcement in a uniaxial stress state, only one component is shown. This strain is in a local direction of the truss element. Furthermore, the principal and normal values of stresses and strains in reinforcement are identical.

2.8.5.18 Results - Load Step - Nodes - Displacements

Nodal displacement data are displayed. Example of displacement data:

```
Output data for request: DISPLACEMENTS
Description: Nodal coordinates change
  Step: 10  Iteration: 3 at Time: 10
----------------------------------------------------------------------------------
 Node       X(1)       X(2)
Units          m          m
    1  0.0000526 -0.0009465
    2  0.0000493 -0.0009801
    3  0.0000634 -0.0009813
    4  0.0000866 -0.0009438
    5  0.0001036 -0.0009015
    6  0.0000595 -0.0009037
    7  0.0000163 -0.0009047
    8  0.0000171 -0.0009464
    9  0.0000219 -0.0009810
   10 -0.0000225 -0.0009457
```

2.8.5.19 Results - Load Step - Nodes – Partial Internal Forces

Partial internal force data show the nodal force equivalents of element stresses in each element group. Summation of forces includes only elements of the same group. Example of partial internal force data:

```
Output data for request: PARTIAL_INTERNAL_FORCES
Description: Current internal forces, ie. LHS
  Step: 10  Iteration: 3 at Time: 10
-------------------------------------------------------------------------------------
 Node     DOF(1)     DOF(2)
Units         MN         MN
    1  9.34e-013 -6.46e-014
    2 -0.0478981 -0.0032716
    3 -0.0505934 -0.0145738
    4  1.90e-013  3.53e-013
    5 -5.79e-013  1.53e-014
    6 -9.46e-013  2.17e-013
    7 -1.32e-012 -6.41e-014
    8  5.06e-013 -6.96e-015
    9 -0.0252674 -0.0062000
   10 -0.0000301 -0.0000229
```

Note: The sums of nodal force in nodes of adjacent elements are shown. In nodes without external loading and kinematic constrains, such as supports and slaves, in an equilibrium state these values should be near to zero within the tolerance specified by convergence criteria. In the above example nodes 2,3 and 9 are on the border line with another element group and the nodal forces are not added across this border. However, they should be in equilibrium with the nodal forces of the other group. It is better to observe equilibrium in the data of residual forces.

2.8.5.20 Results - Load Step - Nodes – Internal Forces

Internal force data show the nodal force equivalents of element stresses regardless of element groups. Summation of forces includes all elements regardless of the group. In older ATENA versions this output data type was labeled Compact Internal Forces. Example of internal force data:

```
Output data for request: INTERNAL_FORCES
Description: Current internal forces, ie. LHS
  Step: 10  Iteration: 3 at Time: 10
-------------------------------------------------------------------------------------
```

```
 Node      DOF(1)      DOF(2)
Units          MN          MN
    1  9.34e-013 -6.46e-014
    2  1.62e-012  3.27e-013
    3 -6.99e-013  5.18e-014
    4  1.90e-013  3.53e-013
    5 -5.79e-013  1.53e-014
    6 -9.46e-013  2.17e-013
    7 -1.32e-012 -6.41e-014
    8  5.06e-013 -6.96e-015
    9  7.08e-013  1.37e-013
   10 -0.0000301 -0.0000229
```

Notes: The sums of nodal force in nodes of adjacent elements are shown. The summation includes all elements. In nodes without external loading and kinematic constrains, such as support s, in an equilibrium state these values should be near to zero within the tolerance specified by convergence criteria. In the above example the nodes 2,3 and 9 are on the border line with another element group and the nodal forces are averaged across this border. Compare with internal forces in Sect. 2.8.5.19 .

The difference between internal forces and residual forces is in the range of summation. The internal forces do not include the external force and reactions acting in nodes. These external actions are of course important for equilibrium. Therefore, it is better to observe equilibrium on compact residual forces.

The relative high unbalanced force in node 10 is due to a crack in one of the adjacent element. This violates equilibrium within required tolerance and the unbalanced force can be decreased by decreasing the equilibrium convergence limit if required.

2.8.5.21 Results - Load Step - Nodes – Partial External Forces

Nonzero partial external forces are listed in all nodes (master and slave). The external forces are due to given loading. Example of data:

```
Output data for request: PARTIAL_EXTERNAL_FORCES
Description: Current external forces, ie. loads (only nonzero values)
  Step: 6  Iteration: 7 at Time: 6
---------------------------------------------------------------------------------------
 Node      DOF(1)      DOF(2)
Units          MN          MN
    3         --- -0.0300000
  178         --- -0.0022500
  179         --- -0.0013218
  185         --- -0.0009282
```

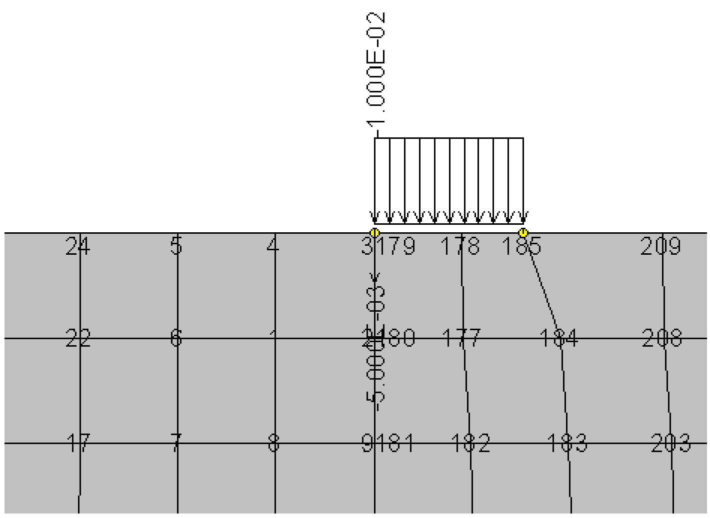


Fig. 2-12 External loading forces in double nodes located on the border of element groups.

In this example nodes 3 and 179 are in the same location on the border of two element groups. They are related by 'Master-slave method'. Node 3 is a master node, node 170 is the slave node. Each of the nodes on the border is loaded by an external force. Nodes 178, 185 are not on the border and are not related. In this data output all forces are listed separately for all master and slave nodes.

2.8.5.22 Results - Load Step - Nodes – External Forces

In this case External forces are listed in master nodes only and only nonzero loading values are shown. Example of data:

```
Output data for request: EXTERNAL_FORCES
Description: Current external forces, ie. loads (only nonzero values)
  Step: 6  Iteration: 7 at Time: 6
---------------------------------------------------------------------------------------
 Node     DOF(1)     DOF(2)
Units         MN         MN
    3        --- -0.0313218
  178        --- -0.0022500
  185        --- -0.0009282
```

In this case the forces in nodes 3 and 179 are added (compacted) and displayed only in the master node 3. Compare with Sect. 2.8.5.21. The compacted forces are normally chosen for output. In older **ATENA** versions this output type was called Compact External Forces.

2.8.5.23 Results - Load Step - Nodes – Partial Reactions

Partial reactions are listed in all nodes with kinematic constrains. The kinematic constrains can be of two types: (a) fixed supports or prescribed displacements in nodes, (b) slave nodes. Partial reactions are treated in a way similar to partial external forces, see Sect. 2.8.5.21.

Example of reaction data:

```
Output data for request: PARTIAL_REACTIONS
Description: Current reactions, ie. LHS at constrained nodes
  Step: 6  Iteration: 3 at Time: 6
--------------------------------------------------------------------------------------
 Node     DOF(1)     DOF(2)
Units         MN         MN
    3        --- -0.0087023
  166 -0.0016941 -0.0041058
  168        ---  0.0163789
  171 -0.0003921 -0.0073400
  172  0.0020862 -0.0049331
  179  0.0217322 -0.0076766
  180  0.0244204  0.0011976
  181  0.0116517  0.0021368
  187 -0.0008122  0.0020979
  188 -0.0120575  0.0015021
  193 -0.0157219  0.0006518
  194 -0.0084090  0.0000664
  200  0.0125404        ---
  201  0.0005051        ---
  202 -0.0136231        ---
  205  0.0162205        ---
  207  0.0083205        ---
  210 -0.0271216        ---
  211 -0.0183808        ---
  212  9.92e-006 -4.45e-009
  213 -7.61e-006  3.42e-009
  214  3.91e-006 -1.75e-009
```

```
  215  4.53e-006 -2.02e-009
```

In the above example the nodes 3, 168, 200, 200, etc. represent external supports. The nodes 166, 171, 172, etc. are slave nodes of macro-elements and nodes 212, 213, etc. are the slave nodes of bar reinforcement elements.

Since the slave nodes are kinematicaly related to the master nodes they are regarded as supports to master nodes and thus provide reaction forces to master nodes. This occurs, for example in nodes on the border of element groups and in reinforcing bars. In this model reactions represent the forces acting between element groups.

2.8.5.24Results - Load Step - Nodes – Reactions

Reactions are listed in all nodes with kinematic constrains. In reactions only external kinematic constrains such as supports and prescribed displacements are considered. Reactions are treated in a way similar to external forces, see Sect. 2.8.5.22. In older versions of Atena this data type used to be called Compact Reactions.

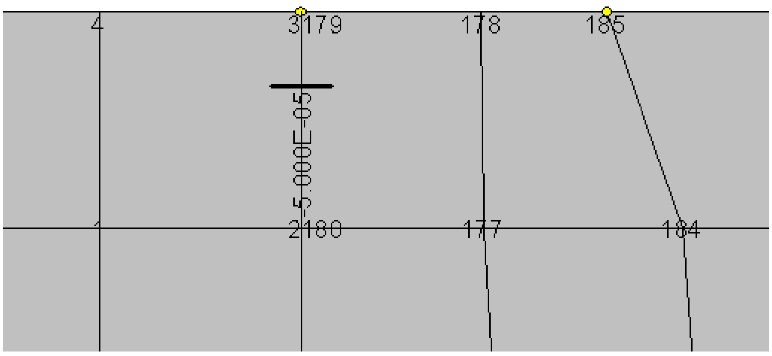


Fig. 2-13 Compact reactions.

Example of compact reaction data:

```
Output data for request: REACTIONS
Description: Current reactions, ie. LHS at constrained nodes
  Step: 6  Iteration: 3 at Time: 6
---------------------------------------------------------------------------------------
 Node      DOF(1)      DOF(2)
Units          MN          MN
    3         --- -0.0163789
  168         ---  0.0163789
  179         --- -0.0076766
  200  0.0125404         ---
  201  0.0005051         ---
  202 -0.0136231         ---
  205  0.0361629         ---
  207  0.0096367         ---
  210 -0.0271216         ---
  211 -0.0183808         ---
```

In this case the compact reactions in the nodes with supports are shown. If a reaction is in a double-node the value in a master node is shown. In the above example nodes 3 and 179 are the double nodes, each belonging to a different element group. Node 3 is master and node 179 is slave. (This can be found in the menu **Calculations | Analysis progress info.. | Input**.) In the compact form the reaction in node 3 shows the total reaction (i.e. the sum of reactions in nodes 3 and 179).

2.8.5.25 Results - Load Step - Nodes – Partial Residual Forces

The residual force shows the unbalanced force in nodes within element groups. Example of partial residual force data:

```
Output data for request: PARTIAL_RESIDUAL_FORCES
Description: Current residual forces, ie. RHS-LHS
  Step: 6  Iteration: 3 at Time: 6
------------------------------------------------------------------------------------
 Node      DOF(1)      DOF(2)
Units          MN          MN
    1 -8.68e-013  1.18e-013
    2  0.0244204  0.0011976
    3  0.0217322        ---
    4 -5.09e-013 -3.51e-014
    5  9.40e-014 -8.21e-014
```

Partial residual forces describe equilibrium state. They are calculated as a sum of internal forces, external forces and reactions in a node under consideration and within the element group only. In an equilibrium state the residual forces should be zero. Any nonzero residual forces indicate an error in the non-linear solution. In the case of (this) non-compact output the sum of nodal forces is made only within the element group. Thus unbalanced forces can occur on the group border as shown in the above example in nodes 2 and 3.

2.8.5.26 Results – Load Step - Nodes – Residual Forces

The residual force shows a global unbalanced force in the node. Example of residual force data:

```
Output data for request: RESIDUAL_FORCES
Description: Current residual forces, ie. RHS-LHS
  Step: 6  Iteration: 3 at Time: 6
-------------------------------------------------------------------------------------
 Node      DOF(1)      DOF(2)
Units          MN          MN
    1 -8.68e-013  1.18e-013
    2  4.92e-013 -3.47e-013
    3  1.23e-013        ---
    4 -5.09e-013 -3.51e-014
    5  9.40e-014 -8.21e-014
```

Residual forces describe equilibrium state. They are calculated as a sum of internal forces, external forces and reactions in a node under consideration. In an equilibrium state the residual forces should be zero. Any nonzero residual forces indicate an error in the non-linear solution. In the case of compact output the sum of nodal forces is made for all elements adjacent to the node. In the above example the residual forces in the master nodes 2, 3 are about the same order as in other nodes and are near zero.

2.8.5.27 Results - Load Step - Nodes - Nodal Degrees of Freedom

This list relates original nodal numbers 'Node Units' to the DOF numbers in the global stiffness matrix line numbering. DOF(1) and DOF(2) are degrees of freedom in X and Y directions, respectively. The DOF numbering system is changed due to optimizing procedures.

```
Output data for request: NODAL_DEGREES_OF_FREEDOM
Description: Nodal degrees of freedom (if negative, then -boundary condition id)
  Step: 6  Iteration: 7 at Time: 6
-------------------------------------------------------------------------------------
 Node  DOF/BC(1)  DOF/BC(2)
Units       None       None
    .
    .
  164        144        145
  165        146        147
  166         -9        -10
  167        140        141
  168        139         -1
  169        127        128
```

The minus signs in front of DOF numbers indicate DOF with kinematic constrains e.i. BC. They include supports and master-slave relations.

2.8.5.28 Results - Load Step - Elements - Element Incidences

This data describe element node incidences. Example of element incidences:

```
Output data for request: ELEMENT_INCIDENCES
Description: Finite elements incidences
  Step: 6  Iteration: 7 at Time: 6
---------------------------------------------------------------------------------------
Group Elem.       Type      Geom.      Mater      No. 1      No. 2      No. 3      No. 4
Units             None       None       None       None       None       None       None
    1     1         12          1          1          1          2          3          4
    1     2         12          1          1          1          4          5          6
    1     3         12          1          1          1          6          7          8
    1     4         12          1          1          1          8          9          2
    1     5         12          1          1         10         11          9          8
    1     6         12          1          1         10          8          7         12
```

Legend:

Group units	Element group – macroelement number
Elem.	Element number
Type	Element type (CCQ10Sbeta, CCIsoquad. etc.)
Geom.	Element group – macroelement number
Mater.	Material number
No.1,2,3,4	Node numbers

2.8.5.29 Results - Load Step - Elements - Crack Attributes

Crack attributes in elements are listed. The attribute values are average values of all integration points. Example of data on element crack attributes:

```
Output data for request: CRACK_ATTRIBUTES
Description: Element Crack Attributes
  Step: 6  Iteration: 7 at Time: 6
--------------------------------------------------------------------------------------
Group Elem. N.cracks  Dim  N.attr  N1(1)    N1(2)      Size        COD     Sigma_n    Sigma_t
Units           None  None None       m        m          m          m        MPa        MPa


  1   116     1     2     4  0.9994837 -0.0321294  0.0278658  8.25e-006  1.3654179  0.0350714
  1   117     1     2     4  0.9980102 -0.0630523  0.0290361  0.0000103  1.3051162  0.0709954
  1   118     1     2     4  0.9983929 -0.0566705  0.0287913  0.0000131  1.2275692  0.0495634
```

Legend:

Group units	Element group – macroelement number
Elem.	Element number
N.cracks	Number of cracks in element
Dim	Dimensionality of the model
N.attr	Number of crack attributes
N1(1),(2)	Crack normal direction unit vector components
Size	Crack length
COD	Crack opening displacement – crack width
Sigma_n	Normal stress on the crack plane
Sigma_t	Shear stress on the crack plane

2.8.5.30 Results - Load Step – Integration Points

The state variables in integration points are the 'true' values corresponding to a material response as obtained from material models. The important feature of the output in integration points is that all state variables, such as stresses and strains are not changed due to interpolations.

The results in integration points can be displayed and printed in the similar manner as was shown for nodes. They are displayed in all integration points of finite elements. For example in case of the element type Q10IsoQuad there are 4 integration points and in case of the truss element 2 integration points. The output contents is otherwise identical with the output in the nodes described above. Therefore, for brevity, only the examples of stress data is shown.

```
Output data for request: STRESS
Description: Stress
  Step: 6  Iteration: 3 at Time: 6
--------------------------------------------------------------------------------------
Group Elem.    IP   sigma_xx   sigma_yy     tau_xy
Units                    MPa        MPa        MPa
    1     1     1 -5.2649450 -0.2162333 -0.2720579
    1     1     2 -8.1206743 -0.8075407 -0.5800538
    1     1     3 -4.8479660 -1.1314540 -1.7830366
    1     1     4 -8.1344743 -1.7843078 -2.0953607
    1     2     1 -4.6828729  0.1615989 -0.7348336
    1     2     2 -4.8095775  0.0311175 -0.1400682
    1     2     3 -6.0911229 -0.1316863 -0.6829434
    1     2     4 -6.1556918 -0.2478999 -0.0965083
    1     3     1 -3.7405848 -0.6009045 -1.0980024
    1     3     2 -2.1863795 -0.2885931 -1.0123273
    1     3     3 -3.8301404 -0.3966465 -0.5014873
    1     3     4 -2.2878196 -0.0840749 -0.4106617
    1     4     1 -3.8748170 -0.6652509 -0.1981763
    1     4     2 -3.5738772 -0.4768670 -0.7165896
    1     4     3 -2.6563775 -0.4205244 -0.2529727
    1     4     4 -2.2994999 -0.2202494 -0.7766643
```

2.8.5.31 Results - Load Step – Element Nodes

The resulting state variables are presented in element nodes. Their values are interpolated from the integration points to the nodal locations using the element shape function. The output contents is otherwise identical with the output in the nodes described above. Therefore, for brevity, only the example of stresses in element type CCIsoQuad is shown:

```
Output data for request: STRESS
Description: Stress
  Step: 6  Iteration: 3 at Time: 6
--------------------------------------------------------------------------------------
Group Elem.  Node   sigma_xx   sigma_yy     tau_xy
Units                    MPa        MPa        MPa
    1     1     1 -5.8067579 -0.5166762 -0.6229856
    1     1     2 -5.6191713 -1.0508604 -1.4930651
    1     1     3 -7.4580803 -1.4194179 -1.6727949
    1     1     4 -7.5025365 -0.8647846 -0.8012776
    1     2     1 -5.0053828  0.0716517 -0.5937402
    1     2     4 -5.8105642 -0.0958690 -0.5648367
    1     2     5 -5.8554075 -0.1646794 -0.2253723
    1     2     6 -5.0709301 -0.0019128 -0.2515002
    1     3     1 -3.4317069 -0.4914576 -0.9528219
    1     3     6 -3.4848615 -0.3734987 -0.6077989
    1     3     7 -2.5929611 -0.1930670 -0.5559852
    1     3     8 -2.5358451 -0.3111126 -0.9027251
    1     4     1 -3.5532179 -0.5744056 -0.3162409
    1     4     8 -2.8431320 -0.4317055 -0.3485023
    1     4     9 -2.6439626 -0.3175558 -0.6501369
    1     4     2 -3.3726897 -0.4642186 -0.6161165
```

In the above example the stresses in four elements are listed. Note, that the stress values in the node number 1, which is a common node for all four elements, are different. These values are averaged if an output in nodes is requested, see Sect.2.8.5.7.

2.8.5.32 Results – Load Step – Global

Example of global data output.

FE-Model:

```
Output data for request: FEMODEL_CHARACTERISTICS
Description: Number of some main FE model entities
  Step: 6  Iteration: 3 at Time: 6
---------------------------------------------------------------------------------------
     Nodes El. groups  El. types Geometries  Materials  Dimension

       257          5         12          5          3          2
```

Task Name:

```
Output data for request: TASK_NAME  Step: 6  Iteration: 3  Time: 6 Name : Leonhardt shear beam
```

Task Title:

```
Output data for request: TASK_TITLE  Step: 6  Iteration: 3  Time: 6 Title :     Atena
```

Solution Characteristics:

```
Output data for request: SOLUTION_CHARACTERISTICS
```

```
Description: Solution characteristics
  Step: 6  Iteration: 3 at Time: 6
-------------------------------------------------------------------------------------
Analysis type                                  Solution method      Predictor type    Update
displs. strategy Optimize params                              Serialize params

      STATIC               NONLINEAR LINE_SEARCH TANGENT_MATRIX   TANGENT_PREDICTOR
UPDATE_IP_EACH_STEP            SLOAN     STANDARD BASICS NODAL STATE ELEMENT STATE
```

Arc-Length Prameters:

```
Output data for request: ARC_LENGTH_PARAMS
Description: Arc-Length Parameters
  Step: 6  Iteration: 3 at Time: 6
-------------------------------------------------------------------------------------
         ARC_LENGTH_TYPE      ARC_LENGTH_BASE_STEP_LENGTH STEP_LENGTH REFERENCE_DLAMBDA
ARC_LENGTH_OPTIMISATION REFERENCE_NUMBER_OF_ITERATIONS LOAD_DISPLACEMENT_RATIO
LOAD_DISPLACEMENT_RATIO_OPTIMISATION NODE(FROM,TO,BY) DOF(FROM,TO,BY) COEFFICIENT
                                                                    m
MN/m
              CRISFIELD  ARC_LENGTH_USE_PREVIOUS_STEP_LENGTH     0.001123                1
ARC_LENGTH_CONSTANT                               5                    0.2
LOADING_DISPLACEMENT_RATIO_CONSTANT                              All model dofs
```

Line Search Parameters:

```
Output data for request: LINE_SEARCH_PARAMS  Step: 6  Iteration: 3  Time: 6 Line-Search
Parameters :
```

Convergence Criteria:

```
Output data for request: CONVERGENCE_CRITERIA
Description: Convergence criteria parameters
  Step: 6  Iteration: 3 at Time: 6
-------------------------------------------------------------------------------------
RelDisplE. RelResidE. RelEnergyE AbsDisplE. AbsResidE. Iter.Limit
      None       None       None          m         MN       None
 0.0100000  0.0100000  0.0100000  0.0100000  0.0100000         40
```

Step ID:

```
Output data for request: STEP_ID
Description: Load step No.6
  Step: 6  Iteration: 3 at Time: 6
--------------------------------------------------------------------------------------
Steps from   Steps to   Steps by

         6          6          1
```

2.8.5.33 Results – Load Step – Load Cases – Support Slave Nodes

The information in this data section describes supports and loading, i.e. boundary conditions with details about master-slave relations. They serve as additional information for cases of more complex kinematic constrains. These cases are not accessible through the graphical user interface **ATENA 2D**, but can be used via in out commands and **ATENA Console**. In standard cases of supports, such as those generated by the graphical user interface **ATENA 2D**, the data obtained from data items **Partial Reactions**, **Reactions**, **Partial External Forces** and **External Forces** provide sufficient information.

2.9 Printing the Graphic Output Data (File | Graphic Printout….)

By choosing this item the graphical contents of an active window will be sent to a printer. An appropriate printer can be chosen in the standard Windows menu **Printer**.

2.10 Ending ATENA (File | Exit)

By choosing this menu item the **ATENA** program can be terminated. If data contents in **ATENA** was changed a dialog box will appear and makes possible a data saving before exiting the program.

3 EDIT

The menu item **Edit** has only one function. It allows to save a graphics of the active windows as a bitmap on a clipboard in Windows. This data is then available for further processing in Windows.

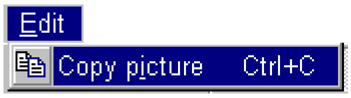


Fig. 3-1 Edit-Copy picture menu.

The process of transferring pictures from **ATENA** to other documents can be done in two ways as described below. The first prerequisite is, that the graphical window to be copied must be an *active window*. Such window is marked by the top menu bar in blue color.

Direct transfer in metafile format:

1. Select the item **Edit | Copy picture** and click on it. (This will make temporary copy of the contents of the active window in the clipboard memory of the computer).
2. Position the cursor in a document where you wish to insert the graphics.
3. Execute the command **paste**. This will place the contents of the active window (from the clipboard) to the document. The picture is an object in the metafile format with optimal features.
4. Change the properties of the inserted picture frame if necessary, such as size, position and text flow.

Bitmap:

1. Select the item **Edit | Copy picture** and click on it.
2. Open a graphical tool, for example Paint, and paste this picture in it.
3. Save this picture under some name in an appropriate directory as a bitmap.
4. Open a document, where you intend to insert the picture. This can be, for example, a Word document.
5. In this document place the cursor in a position, where you want have this picture and choose **Insert | picture | from file**. Select the saved bitmap file and insert it in the document.

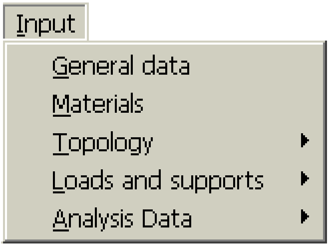
The first procedure gives a optimal picture quality and has advantage of an object manipulation. Disadvantage is that you do not maintain a separate file of the picture. Obviously, this can be overcome by creating a separate document file with the picture. The second method creates s bitmap file. A change of picture size may result in a poor quality.

The contents of the **Edit** menu changes when results are present after running the analysis. This will be described later.

4 INPUT

4.1 Access to Input Data Generation

The tools for development of a numerical model are included in the menu item **Input**. It serves for definition of all input data. The item is active only in the pre-processing mode. The proper selection of pre- or post-processing modes can be made from the menu item **Calculations**. The **Input** menu tree is shown in Fig. 4-1.



The functions for data input can be also reached from the graphical access tree structure in the left field of the pre-processing window shown in Fig. 4-2. Thus, there are always two ways how to generate input data: (1) Using the menu in Fig. 4-1, or (2) using the access tree in Fig. 4-2. The later way is usually chosen for its greater efficiency.

Fig. 4-1 Menu 'Input'.

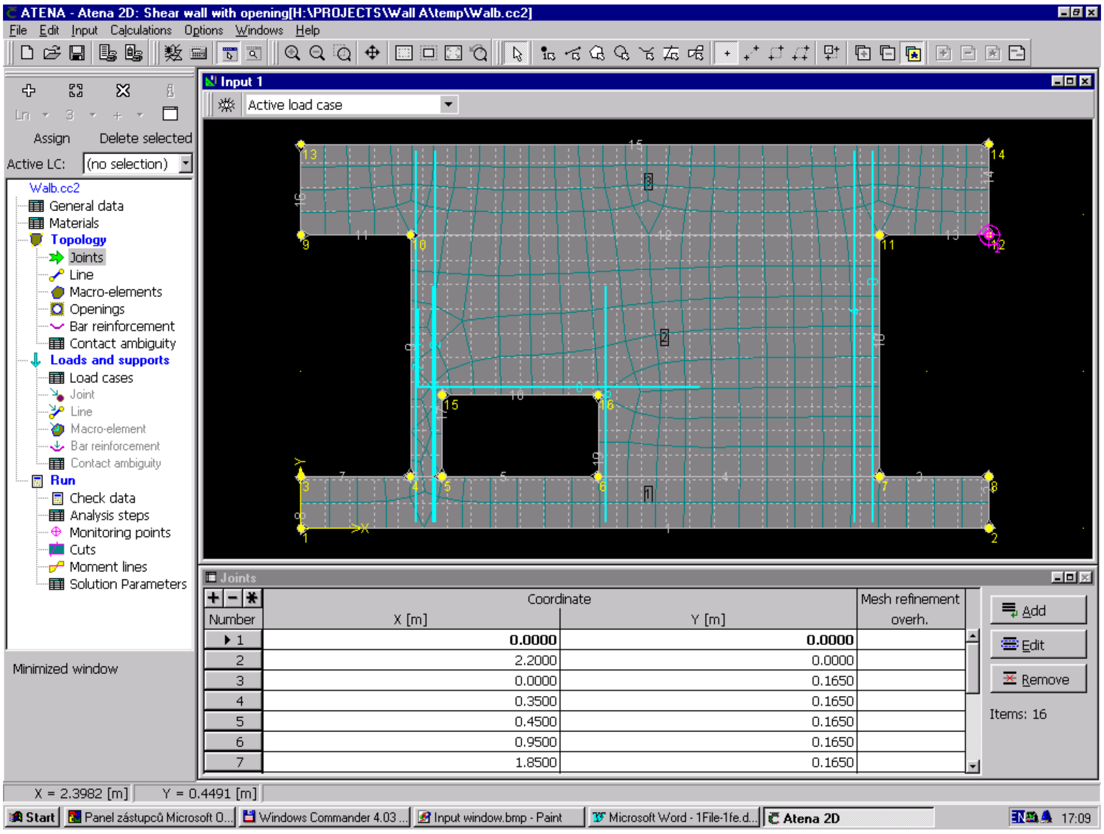


Fig. 4-2 Pre-processing window.

The input data are structured as follows:

1. **General data**. Task names, layers of smeared reinforcement.
2. **Materials**. Defining material types (models) and their parameters.

3. **Topology**. Building geometrical model from basic components: Joints, lines, macroelements, openings, reinforcing bars, interfaces, springs.
4. **Loads and supports**. Defining basic load cases: supports, loading forces, prestressing, temperature, shrinkage, etc.
5. **Run** (analysis parameters). These data include: Definition of load history (load step array), selecting data to be monitored, selecting solution methods and their parameters.

The above sequence of steps is a recommended order in which the model should be built. However, it is possible, whenever necessary, to change the sequence, or to go back to steps and change some parameters. Obviously, some actions can be performed only after completion of other actions. For example, a loading can be assigned to a joint only after the joint is defined, or a material can be assigned to the macro-element only after a definition of such material is made.

All input data are related to the object to be analyzed and not to its finite element model. Finite element model is created automatically by the program. The entities of the model, such as geometry, loading, etc. are not bound to the finite elements, but to the geometrical model. This has advantage that the input need not to be changed if the new mesh is created.

Before going to the details of input tools and commands, it is useful to describe some common features, which are general to the input environment. This is done in the next section.

4.2 Properties of Input Environment

4.2.1 General Description of Input Window (Pre-processing)

The input window environment includes a wide range of graphical and numerical tools. This section describes the common features and rules of the input window. Its organization can be seen from Fig. 4-2.

The main graphical window shows a view of the numerical model. Several such windows can be opened simultaneously. Each window is identified in the top blue bar by a name, such as **Input i**, where **i** is the identification number of the graphical input window. All opened windows can be active simultaneously and can be used for data input. They can be minimized, maximized or cancelled using the standard windows buttons in the right top corner. The open, but minimized windows are indicated on the list in the left bottom field. The arrangement of windows can be also changed from the menu item **Windows** in the main menu.

The tools for data input and related actions are organized in several groups in form of menu bars and trees. This organization shall be described next.

4.2.2 Select Tools

The tool bar on the top, just under the main menu bar extends through the full screen length. It includes the tool buttons for data and screen manipulation as described in Fig. 4-3. These tools serve for manipulating objects already generated by input functions. Some of these tools are general and are active also during the post-processing mode.

The tool bar **File** contains three buttons for file access (from left: **open new file**, **open existing file** and **save into a file**) and two buttons for printing (**print text** and **graphic output**). These tools have already been treated in detail in Chapter File.

The tool bar **Analysis** contains switches for chosing ATENA processing mode: **generate finite element mesh**, **start analysis**, **pre-processing**, **post-processing**. The tool bars **File** and **Analysis** are active at all times. The tool bar **Scale** is used to zoom or move the graphic objects in the window.

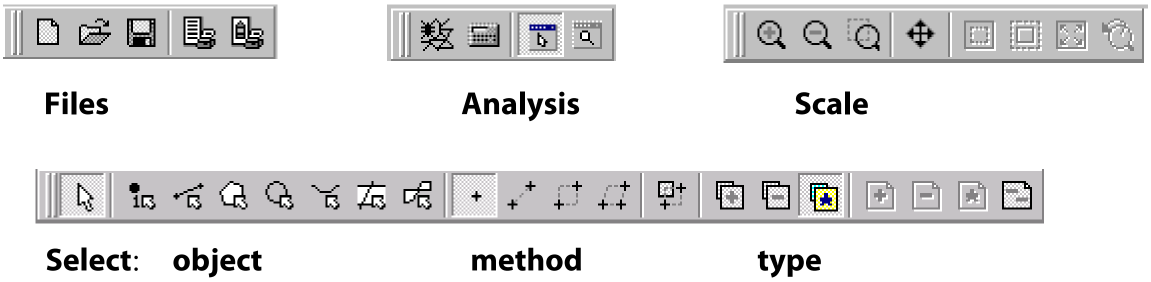


Fig. 4-3 Tool bars

The tool bar **Select** can be used to select geometrical objects. This bar is present only in the input window (pre-processing). The selected objects are depicted by green color. The way how to select objects in the window is defined by combination of several functions as follows.

Defining the object. There is a group of seven buttons for selecting various objects: geometrical points (joints), lines, macro-elements, openings, reinforcing bars, cuts and moment lines. The first button (arrow) cancels any chosen object.

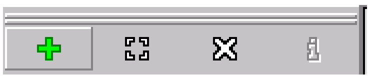
Method of selection. The selection can be done by pointing to an object, by drawing a line across the object to be selected or by drawing a quadrilateral field enclosing the objects.

Type of selection indicates selection or its inverse action, un-selection. The most right button performs an un-selection of all selected objects. This button can be conveniently used if all kinds of selection should be cancelled.

4.2.3 Input Tools

Input tools serve to generate input data. They can be accessed through a tree located on the left side of the main window, see Fig. 4-2. It includes the five groups of data as described in Sect. 4.1. There are two ways how to generate data:

1. Graphical way using pointing device – mouse.
2. Numerical way, by typing-in numerical values using the key board.

The input by the mouse is controlled by the tools  located above the input access tree. These three buttons, from left to right activate the actions add, edit, and delete for input of a *single object*.

The input from the keyboard is done from the field located under the bottom side of the main graphic windows, which is devoted to the numerical data. This field contains a table of data and on the right side the command buttons Add, Edit, Remove, which serve to open dialog windows for numerical data input. In most cases the both ways, graphical or numerical input, can be used as equal alternatives. One method can be superior to the other in specific cases as will become clear in practical applications. For example, it is better to enter the precise coordinate values of a joint from the keyboard, while it is very difficult to do it by a mouse. Entering a line connecting two joints is more efficient by the mouse rather then by the keyboard.

Group operations can be made for deleting of objects and for assigning of common properties to objects. These operations are possible only for properties of joints, lines and macro-elements. A group of objects must be first selected. Giving properties to objects is made possible by pressing the button Assign, deleting a group of object is done by pressing the button Delete selected.

Example of group operations shall be treated separately.

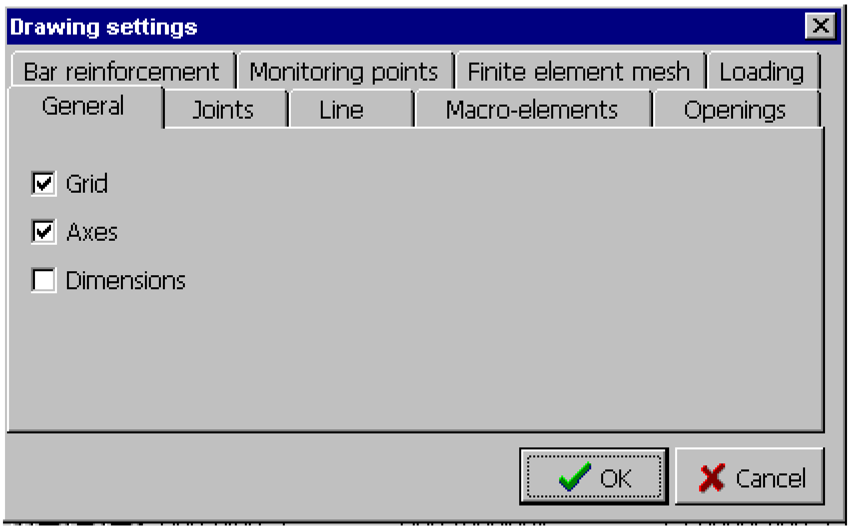
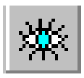


Fig. 4-4 Setting input window parameters.

4.2.4 Set Window

The view of the input window can be adjusted by setting its parameters. The setting can be accessed by clicking on the icon  in the left top corner of the input window. This opens the menu shown in Fig. 4-4, which offers a variety of graphical settings. Its main purpose is to switch on only the features, which should be visible and to eliminate those, which need not be displayed. Each input window can be set in a way, which suits to the user needs.

4.3 General Data

4.3.1 Description

This dialog allows to enter and to view general data information. After clicking on the item General data in the access tree, the dialog window shown in Fig. 4-5 will appear. The same can be done by selecting the item **Input | General data** from the main menu.

This window has two fields. The first field described as **General data** contains a task name (Description:) and additional text (Note:). The text in the field, **Description** appears always in the top blue field of the **ATENA** window and serves to indentify the current task in this window. Text **Description** can be modified by clicking on the button **Edit**. These data are only descriptions, can be changed at any time and have no effect on the analysis.

4.3.2 Smeared Reinforcement Layers

The dialog window **General data**, Fig. 4-5, is also used to set the number of smeared reinforcement layers. The number of smeared reinforcement layers is increased by clicking on the add button 

and is decreased by clicking on the button 

.

The smeared reinforcement is modeled as uni-directional continuum material, with stiffness in direction of reinforcement only. This material is included in a separate finite element, which is in form and coordinates identical with the underlying concrete element. Each component (direction) of smeared reinforcement is in a separate element layer. The final reinforced concrete material is composed of a basic concrete element and layers of smeared reinforcement elements. The number of reinforcement layers must be equal to the number of smeared reinforcements used in the structure. The number of layers is defined for the whole structure and must not be less then the maximal number of layers in any macroelement. In each macro-element the contents of the layers can be different, as required by its reinforcement.

Example: There are two smeared reinforcement types, one vertical other horizontal, Each type is used in one macroelement (in two different macroelements). It is necessary to define one smeared reinforcement layer and to fill this layer in each macroelement with an appropriate smeared reinforcement type.

A consequence of this method is, that the number of finite elements is increased. For example in case of one smeared reinforcement layer, the total number of elements is doubled. (It concerns only the elements used for concrete.) The nodal displacements in all layers are identical. The relation between the degree of freedoms in different layers is made by the master-slave method.

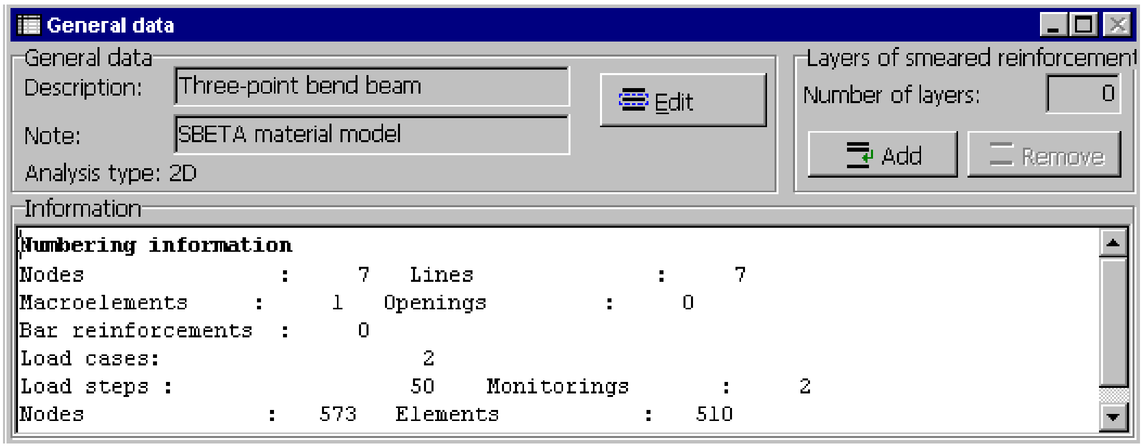


Fig. 4-5 Dialog window 'General data'.

The bottom part of the window shows a summary of global data about the numerical model. The initial numbers are zeros and are changed as the model is defined. This window is typically viewed in order to check the model size.

4.4 Materials

4.4.1 Selecting Material Model

The input of material types and their parameters – material properties is done choosing the menu item Materials in the access tree. The input window in the bottom part will change into the form shown in Fig. 4-6. To add a new material type click on the button Add. This will open a dialog window **New material** in which the list of available material models can be unfolded by pointing on the arrow button, Fig. 4-7. A material is selected by clicking on a desired line and the chosen material name shall appear in the field **Material type**, Fig. 4-8. A click on button Next will open a dialog for the input of material properties of chosen model.

The input of material properties is accompanied by explaining comments and pictures. All material models offer a set of default parameters. These default parameters of models for concrete-like materials are derived form cube compressive strength. The relations for material parameters (Elastic modulus, tensile strength, etc.) are taken mainly from the FIB Model Code 90 and other sources. In cases where user does not know all parameters, the default values can be used. If precise values of material properties are known the default parameters can be edited and changed to the desired ones.

Details of material models are described in documentation ATENA Part 1 - Theory.

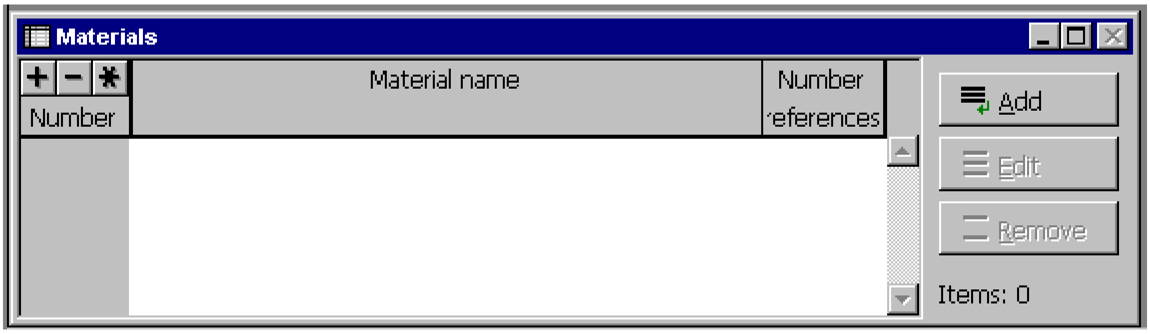


Fig. 4-6 Window 'Materials'.

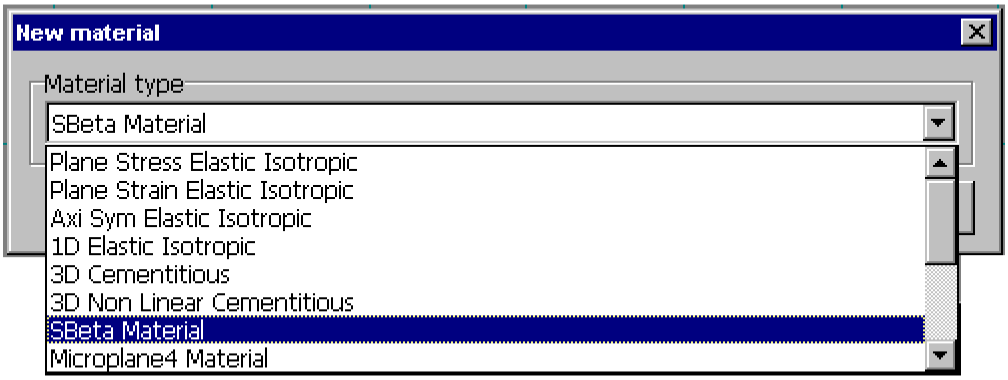


Fig. 4-7 Selecting material model in 'New material'.

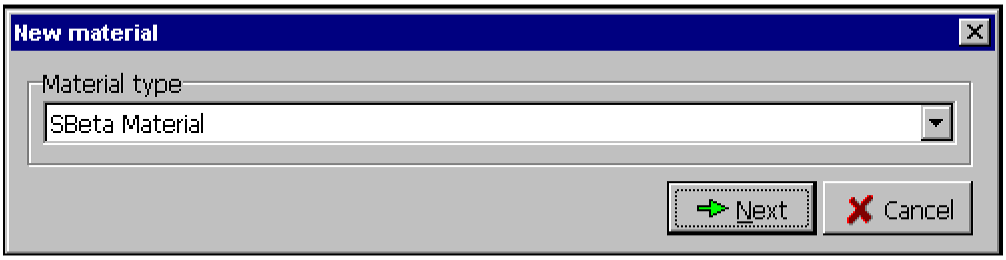


Fig. 4-8 Window 'New material'.

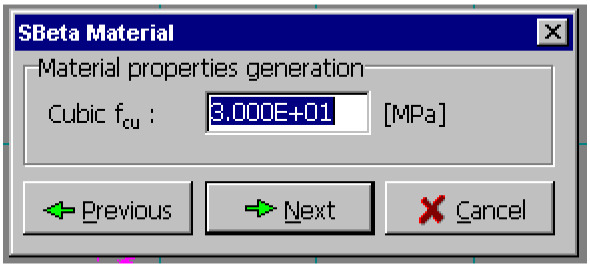


Fig. 4-9 Input of basic parameter-cubic strength for the material SBETA.

An example of a starting dialog window for the SBETA material is shown in Fig. 4-9. By clicking on 'Next' a set of default material parameters is generated, Fig. 4-10.

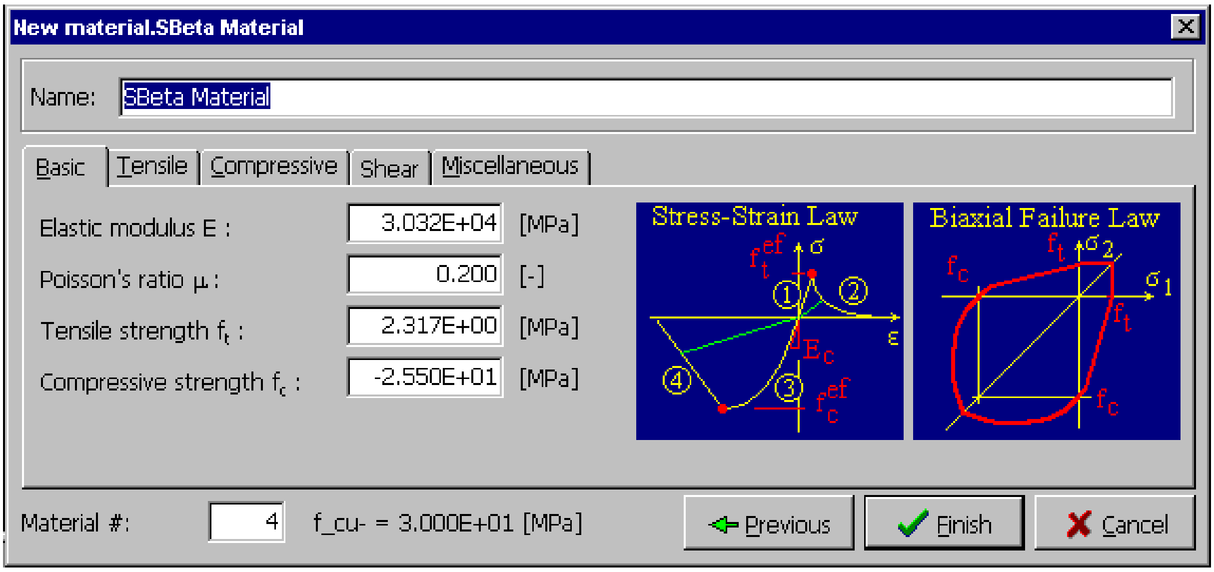


Fig. 4-10 Default set of parameters for material SBETA.

By selecting various sections of material properties (Tensile, Compressive, Shear, Miscelaneous) a complete set of material parameters can be accessed.

> ***Warning:*** The material parameters are important input values. They decide about the model response, failure load and mode. Therefore, it is strongly recommended not to change the default values of material parameters unless rational reasons for other values then those generated by the program exists.

4.4.2 Material Models in ATENA

Material models available in ATENA are listed in Table 4-1 .

Table 4-1 Material Models in ATENA .

Material Type	*Description*	*Material*
Plane Stress Elastic Isotropic	Linear elastic material for 2D-plane stress state.	Any
Plane Strain Elastic Isotropic	Linear elastic material for 2D-plane strain state.	Any
Axi Sym Elastic Isotropic	Lin. elastic for axially symmetrical stress state.	Any
1D Elastic Isotropic	Linear elastic material for 1D-reinforcement.	Reinforcement
3D Cementitious	Fracture-plastic, linear compression.	Concrete
3D Non Linear Cementitious	Fracture-plastic, non-linear compression.	Concrete
3D Non Linear Cementitious 2	Same as above but fully incremental both in tension and compression. **Recommended model for 2D and 3D concrete**.	**Concrete**
3D Variable Non Linear Cem.	Same as above but certain material parameters can vary during analysis.	Concrete
3D Non Linear Cem. User	Same as 3D Nonlinear Cem. 2, but the user can specify stress-strain relationships in tension, compression, shear and tension-compression interaction	Concrete, Fiber reinforced concrete

Sbeta Material	SBETA material model, orthotropic smeared crack model. Suitable for plane stress 2D analysis.	Concrete
Microplane4 Material	Bazant Microplane M4	Concrete
3D Bilinear Steel Von Mises	Von Mises plasticity.	Steel
2D Interface	Iterface, cohesive, dry friction, gap.	Contact
Reinforcement	1D non-linear, bi-linear, multi-linear.	Steel
Cycling Reinforcement	1D Cyclic, Pinto-Menegoto.	Steel
Smeared Reinforcement	1D non-linear, bi-linear, multi-linear.	Steel
Spring	1D linear, multi-linear,	Support
Bond for Reinforcement	Bond-slip relation.	Reinforcement
3D Drucker-Prager Plasticity	Drucker-Prager plasticity.	Soil, Rock
Material With Random Fields	Material for random field generation. Can be used together with any of the above materials and in connection with the SARA program to generate a material with spatial distribution of selected material parameters.	Any

Table 4-1 shows the standard materials for which the models are developed in the most right column. In cases of application to other material types, the default material parameters can not be used, and it is up to user to define a set of appropriate material parameters.

The material models are used in combination with specific finite elements. Some models are specially written for one type of element only. Other models are more versatile and can be used for several elements. This property of material models is automatically recognized by the program and it is allowed to use only material models compatible with chosen finite elements. Thus, for example, the spring material is offered only for spring elements, reinforcement material only for reinforcement elements, SBETA material can be used only for 2D plane stress elements, etc.

The more general 3D materials (3D cementitious, 3D Drucker Prager) can be used in continuum elements of different dimensions: 2D plane stress, plane strain, axial symmetry and general 3D. The functionality of 3D models in 2D (plane stress, plane strain, axial symmetry) is automatically provided by the program. Although it is possible to use full 3D material models in **ATENA 2D**, a general 3D stress state analysis is not available within **ATENA 2D** program packages. However, the confinement features of 3D models can be fully utilized within the frame of axially symmetrical or plane strain analysis.

4.4.3 Special Material Models

The material models are formulated as stress-strain relations. Thus all other properties, which are not part of such formulation and can be considered as geometrical dimensions, are defined in other input entry points. We shall describe these materials briefly.

4.4.3.1 Spring Material

The spring material defines the stress-strain relation in a spring. The purpose of springs is to model supports which reflect a response of supporting sub-structures. Spring is a structural element and its response depends on its dimensions as well as material properties. Only the material properties of spring, the stress-strain relation is defined within the material input. Other properties related to spring dimensions are entered as properties of spring elements.

Two types of spring materials are offered in the input dialog window, Fig. 4-11. They can be selected after unfolding the menu **Type:**. 'Elastic' material property represents a modulus of elasticity. On the example in Fig. 4-11 the elastic modulus of spring is 1000 MPa. Such spring is linear and has the same stiffness in tension and compression.

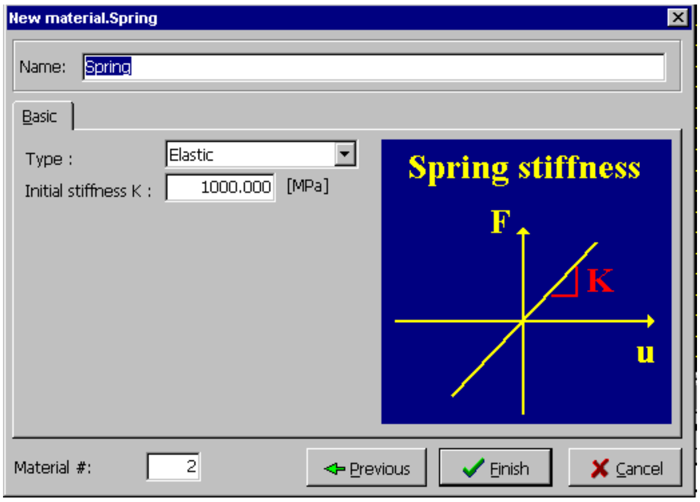


Fig. 4-11 Elastic spring material input

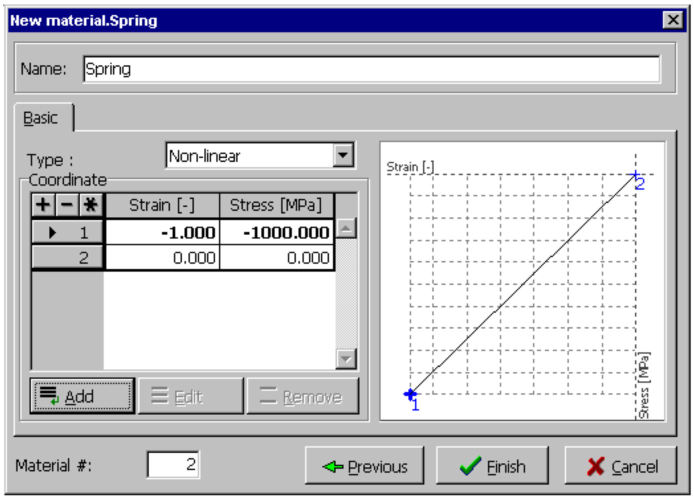


Fig. 4-12 Nonlinear spring material input.

'Non-linear' material is described by a piecewise linear stress-strain law. The function of this law can be entered using the dialog shown in Fig. 4-12. Points of the line can be entered in the field 'Coordinate' by clicking on the button 'Add'. The points are automatically ordered from minimal to maximal values. The law works exactly in the form as defined. All points within the range of the line represent the stress-strain relation and all points outside the line mean zero response (or broken spring material). In the example in Fig. 4-12 we have a case of so called 'contact spring'. The spring is active only under compressive strains up to the strain value –1. In tension the stress is zero, which can simulate an opening of contact between the ground and a structure.

Note: In case of nonlinear spring material and especially in case of nonsymmetrical properties in tension and compression it is important to realize, that the tension and compression states in spring are decided by a local coordinate system of the spring. This is effected by the geometry definition of spring element – spring orientation.

4.4.3.2 Interface Material

Interface material describes the physical properties of contact between two surfaces. The model in **ATENA**, covers following properties of the contact: Frictional properties as recognized in model of dry friction (Mohr-Coulomb) defined by shear cohession *c* and the friction coefficient φ. The maximum shear stress is limited by the linear relation $\tau = c + \varphi\sigma$, where σ is the amplitude of the interface compressive stress (positive value). In addition to the shear properties there is also the tensile cohesion f_t representing a tensile strength of the interface.

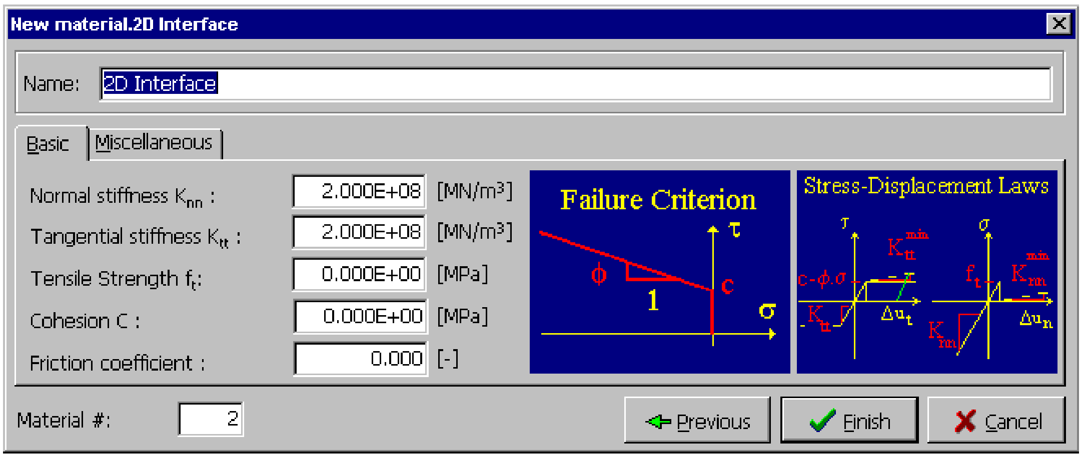


Fig. 4-13 Input of interface material.

The above model holds for a contact behavior of perfectly rigid bodies. However, the ATENA interface model is based on a stiffness concept, which is compatible with the underlying finite element approach. In this model the interface interaction is accomplished by an artificial stiffnesses. The model is illustrated in the input window, Fig. 4-13 and is described by stress-displacement relationships of shear and normal stress components of the interface. The displacements correspond to the relative normal and shear displacements of interface.

The model has two types of parameters. First set of parameters is describing the real physical properties of interface: tensile strength f_t, shear cohesion *c*, and friction coefficient φ. They must correspond to real material properties.

The second set of parameters is stiffness coefficients, which serve purely for numerical purposes. There are two stiffness coefficients, K_{nn} (normal), K_{tt} (shear) and each has two values: basic (in closed state, included in menu **Miscellaneous**), minimal (in open state, included in menu **Miscellaneous**). The unit of these stiffness coefficients is stress per unit displacement (MPa/m, or MN/m^3).

The basic stiffness represents the stiffness of the interface model in closed state. Instead of rigid connection, the interface in closed state undergoes displacements according to the stiffness. The higher the stiffness the smaller are these displacements. Under compressive stresses these displacements generate a slight overlapping of interface lines. The minimal stiffness serves only for the numerical purposes as 'predictor' within the nonlinear iterative solution method.

Theoretically, the basic stiffness should be very high in order to represent well the rigid body and the minimum stiffness should be near zero in order to represent the open contact. However, in practical numerical solutions these values must be reasonably set to allow a stable convergence. In case of difficulties, following rules may be applied: (1) The basic (maximal) stiffness should be about 10 times of the stiffness of adjacent finite elements. (2) Minimal stiffness should be 0.001 times of the maximal stiffness.

4.4.3.3 Reinforcement Material for Bars

Reinforcement material properties for discrete bars are included in the material **Reinforcement**. This material offers an uni-axial law for stress strain relation in reinforcing bars. Three types of law are available: linear, bi-linear, multi-linear. Only a tension part of the law is defined in input. However a complete symmetric form for tension and compression is considered in the program. In case of the multi-linear form the points of the law are entered within a dialog in the field **Coordinate** by clicking on the button **Add**.

The other parameters of bars, such as diameter, number of bars, are entered at the input level of reinforcement elements.

4.4.3.4 Smeared Reinforcement

The smeared reinforcement material law has the same options of three forms of stress-strain law as the discrete reinforcement. In addition it is extended for input of two parameters specific to the smeared reinforcement. The field 'Ratio' should be filled with the value of reinforcing ratio $p=A_s/A_c$ (where A_s is the reinforcement cross section area and A_c is the concrete cross section area). The field 'Reinforcement direction' should be filled with the direction parameters. Two ways of entry are possible: (1) Components of unit vector, (2) Reinforcement direction angle in degrees.

4.4.3.5 Cyclic Reinforcement for Bars

Its properties are similar to the model for bars and extended into the range of cyclic loading. The model is according to Menegoto-Pinto.

4.4.3.6 Bond of Reinforcement

The bond properties for slip between reinforcement and concrete can be specified in this material law. It defines a relation between the bond stress and the slip between the bar and concrete. Default law is defined according to CEB-FIP Model Code 1990 (see the theory manual). In the dialog one of the three bond quality can be chosen, (Fig. 4-13).

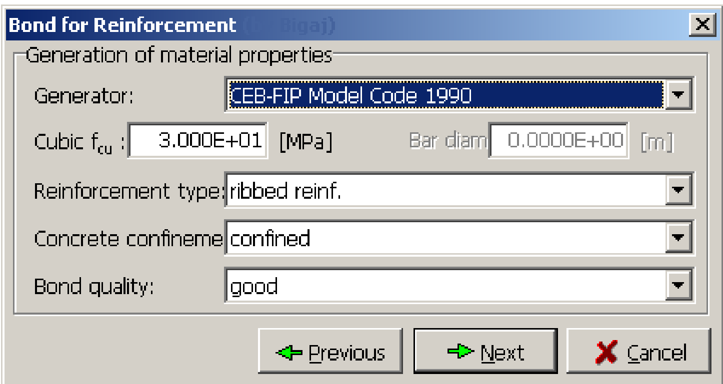


Fig. 4-14 Bond properties input window.

The bond law according to Bigaj requires four parameters: Concrete strength f_{cu} (cube strength), reinforcement type **ribbed reinf., cold drawn wire, hot rolled bar**, concrete confinement (**confined, unconfined**) which indicates the amount of stirrups or other confinement reinforcement, and the quality of bond (**poor, good**). For details see the theory manual.

Another options of bond properties are: Bond model according to Bigaj and user defined bond material. The shape of the bond law can be seen in the graphical window on the right, Fig. 4-15. The entry of the bond law points can be ended by clicking on the This dialog can be ended by clicking on the button 
.

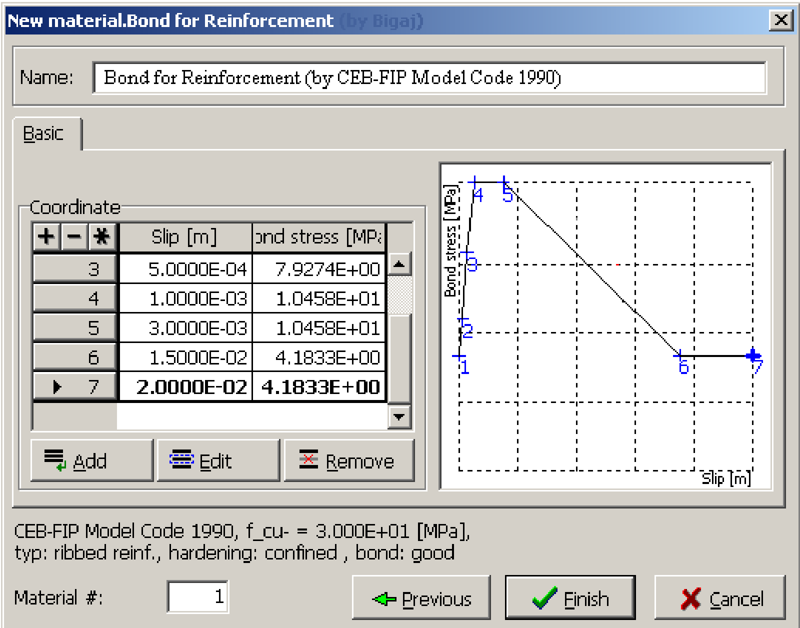


Fig. 4-15 Bon-slip law according to CEB-FIP Model Code 1990. In the program the slip value is set to an extremely high value as it is specified in CEB-FIP model. If the slip value for the last point is set to 0.02 the shown figure is recovered.

Any other law approximated by a polygon of lines can be entered by editing, adding/removing the points. The name of the law must be changed appropriately in the top box.

4.5 Topology

4.5.1 *Rules for Input of Objects*

This section provides tools for building of the geometrical model. This process includes definition of following topological objects of the model:

- Joints (geometrical points)
- Lines
- Macroelements
- Reinforcements

(Note: Objects ‘Openings’ and ‘Contact ambiguity’ are closely related to Macroelements and will be treated in that context.)

The input data of these objects are divided into two groups from practical reasons:

- Specific data unique to each object.
- Prototype data common to a group of objects.

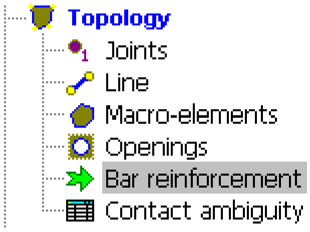
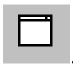


Fig. 4-16 Topology access tree

The input of geometrical data can follow the procedure described below, which is common for all geometrical objects.

1. Object type is selected from the access tree, Fig. 4-16, by mouse. The active object is marked by a green arrow in the access tree. (Bar reinforcement in Fig. 4-16.) In response to this selection the numerical field in the bottom is changed and appropriately labeled (Joints, Lines, etc.)
2. Prototype data should be defined first using the prototype menu. This menu is opened by clicking on the icon . The prototype menu is also automatically opened at the first attempt to make an input of any geometrical object. The contents of prototype data depends on the type of geometrical object (joint, line, etc.). The prototype data typically includes the data on FE mesh, spring supports and materials, which are common to a group of objects.
3. Specific data of an individual object is entered using either of two input methods, mouse or numeric input, respectively. A sequence of objects of the same type can be entered, for example Joints. At this input each the prototype is also automatically assigned to each object.

From the practical point of view, if prototype data is properly set, the input of those data is automatically made without need to repeat their input for each object. However, if data are not correct or known from any reason, a default prototype data can be also used and later changed. The summary of specific and prototype data for objects is given in Table 4-2.

Table 4-2 Specific and prototype data of objects.

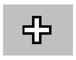
Object	Specific data	Prototype data
Joint	Coordinates	Refinement, spring
Line	Joints	Refinement, spring
Macroelement	Lines	Finite element mesh, materials, thickness…
Reinforcement	Coordinates	Type, material, geometrical nonlinearity

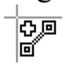
4.5.2 Joints

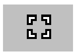
Object 'Joints' must be selected as active for input and marked by a green arrow in the access tree. This activates the access to the tools for input, edit and delete of joints.

When starting the input first time a joint prototype input dialog will appear. It should be preformed as described in section 'Joint prototype'.

4.5.2.1 Joint Location Input



Joint input by mouse is started by clicking on this button. The input is made by pointing at a desired location and pressing the left mouse button. By pressing the right button the input is terminated. The pointer location on the screen is marked as: 



Changes in joint coordinates can be made by clicking on this button. They can only be made by changing numerical values of coordinates in the opened dialog menu. At the same time any other properties of joint can be edited.

Deleting of existing joints can be started by this button. The selected join is marked by a blue square. Each delete action must be confirmed in a special dialog.

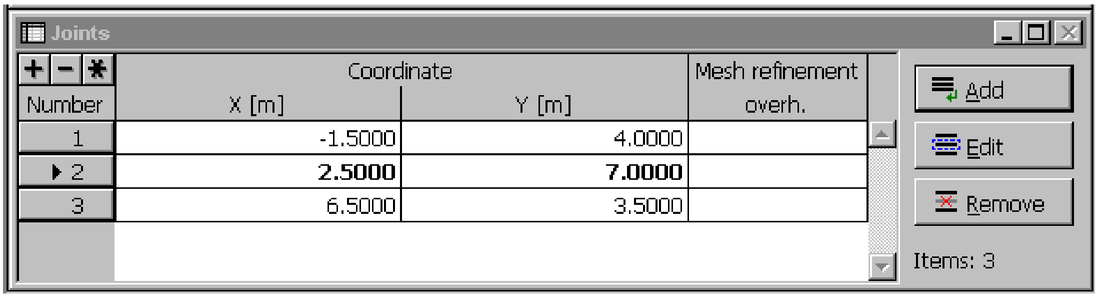


Fig. 4-17 Joints numerical window.

The generated joints appear in the list of the numerical window as shown in Fig. 4-17.

Snap-to-grid: The mouse pointer snaps from one grid point to another as the mouse moves when in the 'Add' mode. This is a default setting and has advantage if a set of regularly spaced joints should be generated. In cases of joints, which do not coincide with the grid, this option is not useful and can be switched off by pressing the 'CTRL' key while moving the mouse. The 'Snap-to-grid' property can be set in the menu **Options | Settings | Pre-processing | Snap to grid**. It can be also accessed by the button .

Numerical input can be made as an alternative to the mouse input. The buttons of the numerical window, 'Add', 'Edit', 'Remove' can be used to open corresponding dialogs for this purpose, Fig. 4-17.

An example of joints with various data types is shown in Fig. 4-18 .

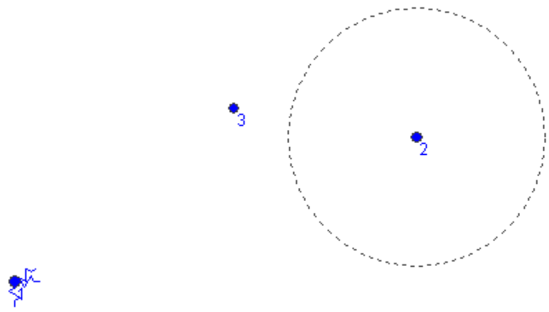


Fig. 4-18 Joints. 1- with sprigs, 2-with refinement, 3-without additional options.

4.5.2.2 Joint Prototype

The additional data on mesh refinement and springs can be entered in prototype menu and used of the next group of joints. The same data can be entered for each joint individually by its editing at any time.

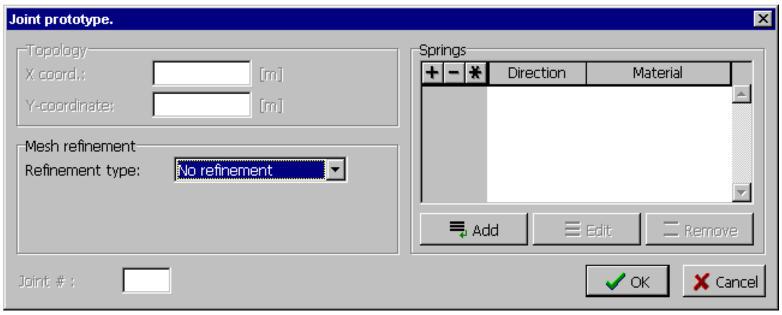


Fig. 4-19 Joint prototype window.

4.5.2.3 Mesh Refinement at Joint

The mesh density can be increased near a joint by specifying the radius of the region and the mesh density at the joint, see joint no.2 in Fig. 4-18. The refinement region is depicted by a circle. The menu is shown in Fig. 4-20.

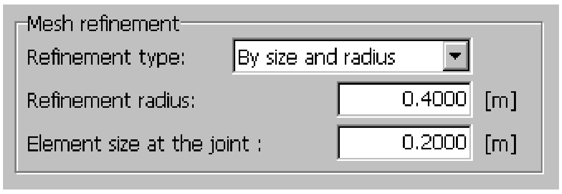


Fig. 4-20 Menu form for definition of mesh refinement near a joint.

4.5.2.4 Spring Supports at Joint

The joint can be supported by any number of discrete springs. The spring is a 1D finite element with only axial stiffness and has the same behavior as a truss or a bar. Its one end is attached to a joint and the other end is supposed to be fixed. Any number of spring can be attached to a joint. The menu for adding springs to a joint is shown in Fig. 4-21. This menu can be accessed either from the joint prototype or from the joint edit menu. By a click on

button the menu **New joint spring** opens, Fig. 4-20, and spring data can be entered.

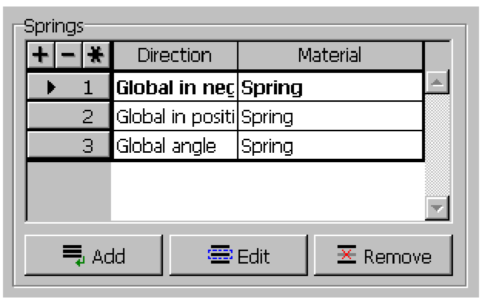


Fig. 4-21 Spring menu for joint.

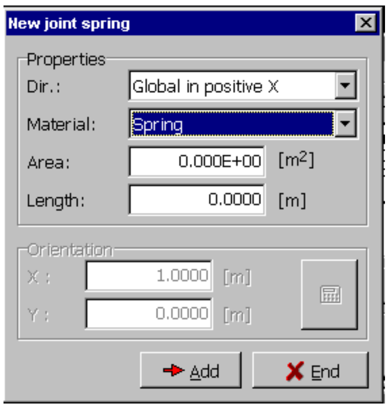
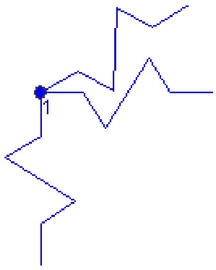


Fig. 4-22 Data form for joint spring.

An example of springs is shown in Fig. 4-23, where the spring pointing downward is at direction ‘Global in negative Y’. The horizontal spring is in ‘Global in positive X’. The inclined spring is in the ‘Global angle’ 30°.



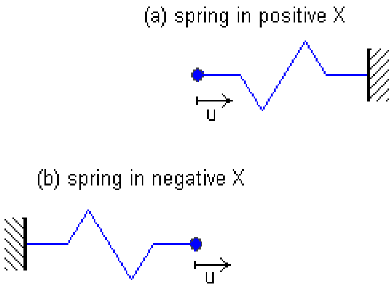


Fig. 4-23 Example of joint with three springs.

Fig. 4-24 Spring element direction.

The spring direction **Dir** is important for nonlinear springs sensitive to tension or compression. This is demonstrated on the example of two horizontal spring attached to a joint, Fig.4-22, (a) in positive and (b) in negative direction. A positive displacement of the joint will generate different stresses in each spring: in the spring (a) compression and in the spring (b) tension. This may have serious consequences on a response. In case of no-tension spring the stress in the case (b) is zero, while in (b) it is a finite negative value.

Material of the spring must be first defined in the Material menu. Such material can be chosen from a list offered in the field **Material**.

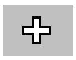
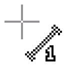
Two dimensions of a spring must be defined: The cross section **Area** and the spring **Length**.

Note: Since spring usually represents a surrounding structure it is reasonable to give such values for its dimensions, that may be similar to those of a real structure and not several orders of magnitude different. If such unrealistic dimensions are chosen the spring may have unrealistic or geometrically non-linear behavior. For example, if expected deformation of spring is in the order of millimeters, then the spring length should be chosen in the order of meters.

4.5.3 Lines

Object ‘Line’ must be selected as active for input and marked by a green arrow in the access tree. This activates the access to the tools for input, edit and delete of lines.

4.5.3.1 Line Input

 Line input by mouse is started by clicking on this button. The pointer location on the screen is marked as: 

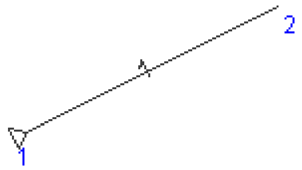
When starting the input first time a line prototype input dialog will appear. It should be preformed as described in section ‘Line prototype’.

Each line is defined by two joints. The line input is made by the following sequence of mouse moves:

1. Put pointer on the first joint and click the left mouse button.

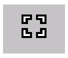
2. Draw the line from the first joint to the second joint.
3. Click again the left button in the location of the second joint.

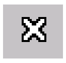
The above sequence completes the input of a line. This means, when a series of lines connecting a chain of joints is generated, a double-click must be made on all intermediate points. The reason for this procedure is, that each line is treated as a separate object and, their chain connection in group is not remembered by the program.



The line orientation (local coordinate system) is defined from the first joint to the second joint. This orientation is indicated by an arrow in the first joint as shown in Fig. 4-25. The joint markers are switched off in this example.

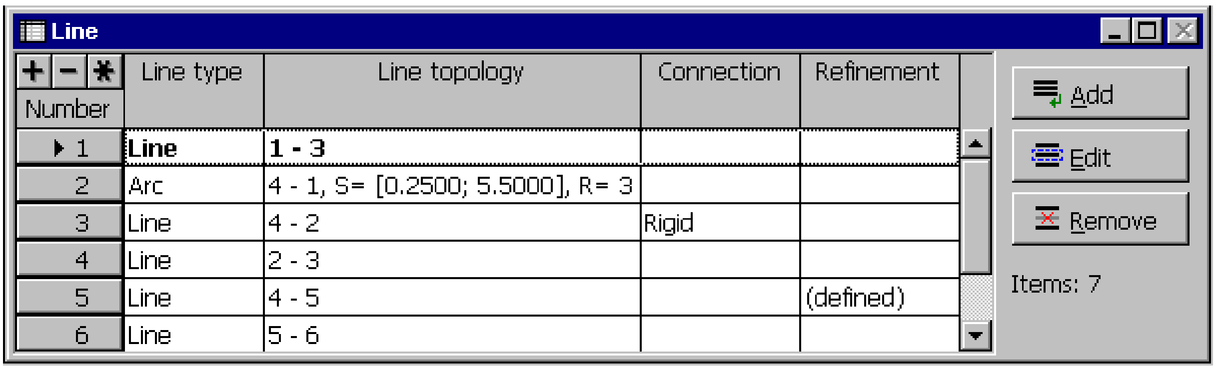
Fig. 4-25 Line direction.

 Changes in line data can be made by clicking on this button. This opens a numerical dialog menu. All line data can be edited.

 Deleting of existing line can be started by this button. The selected line turns to blue color. Each delete action must be confirmed in a special dialog.

Numerical line input can be made as an alternative to the mouse input. The buttons of the numerical window, ‘Add’, ‘Edit’, ’Remove’ can be used to open corresponding dialogs for this purpose. Example of numerical window for lines is shows in Fig. 4-26. The table contents by columns:

Number: Line number.
Line type: Line, Arc, Circle.
Line topology: Joint numbers and arc data.
Connection: Connection of lines on macro-element border.
Refinement: Prescribed refinement on line.



Number	Line type	Line topology	Connection	Refinement
1	Line	1 - 3		
2	Arc	4 - 1, S= [0.2500; 5.5000], R= 3		
3	Line	4 - 2	Rigid	
4	Line	2 - 3		
5	Line	4 - 5		(defined)
6	Line	5 - 6		

Fig. 4-26 Line numerical window.

4.5.3.2 Arc and Circle Line

The arc and circle forms of line can be also defined. They can be selected by special tools which are located near the graphical input buttons as shown in Fig. 4-27. They are active only when object ‘Line’ is selected and simultaneously the graphical input is active.

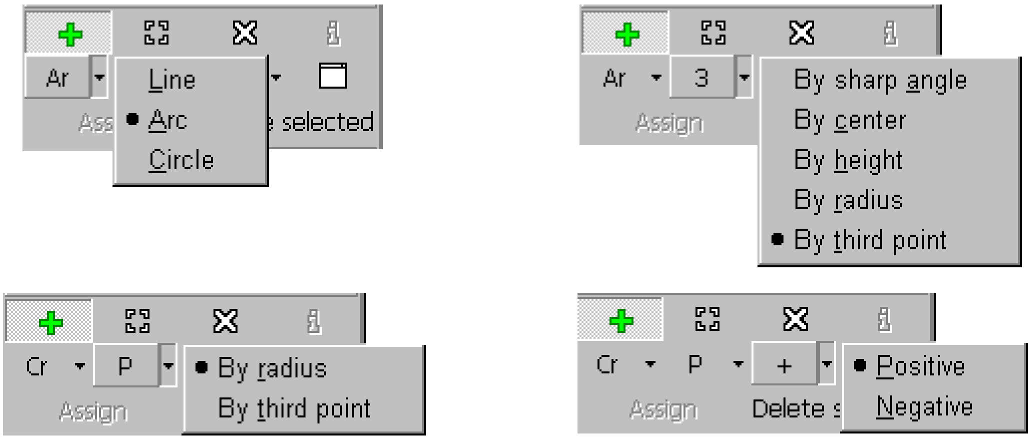


Fig. 4-27 Arc and circle tools for definition.

A definition of an arc and a circle can be also entered numerically in a dialog through the numerical menu. This input method can be initiated by click on the button Add in the numerical window (right, bottom). A dialog box for input of lines opens, Fig. 4-28.

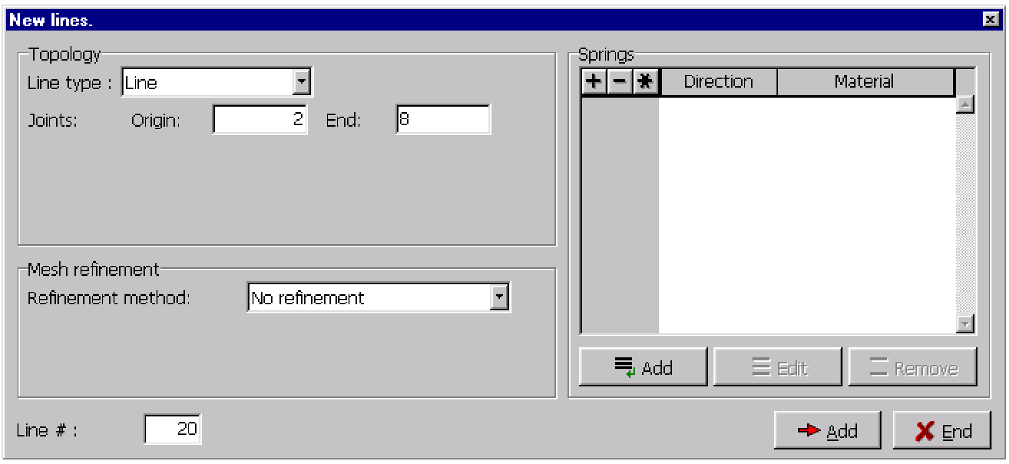
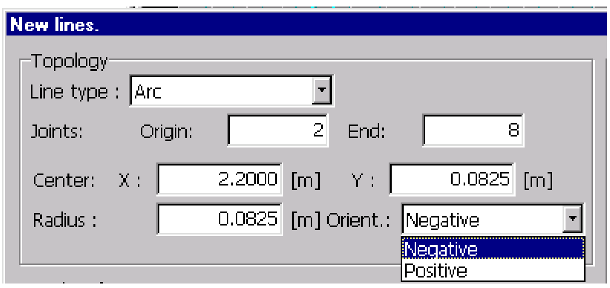


Fig. 4-28 New lines, numerical input.



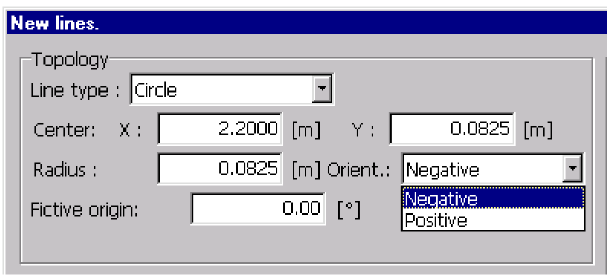


Fig. 4-29 Arc line numerical input. Fig. 4-30 Arc line numerical input.

Line type menu can be unfolded by a click on the arrow and a type can be chosen. Dialog boxes for input of arc and circle are shown in Fig. 4-29.

4.5.3.3 Line Data Prototype

A window for line prototype data input is shown in Fig. 4-31. Mesh refinement parameters and spring supports can be entered through this dialog. They will be used for all lines entered in a next input sequence.

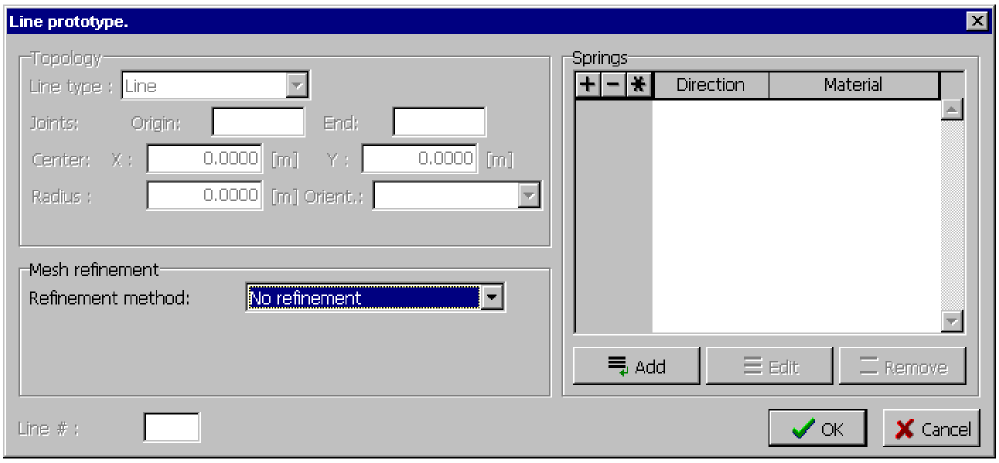


Fig. 4-31 Line prototype window.

4.5.3.4 Line mesh refinement

The mesh refinement can be set in the dialog shown in Fig. 4-32. Two methods are available as described further.

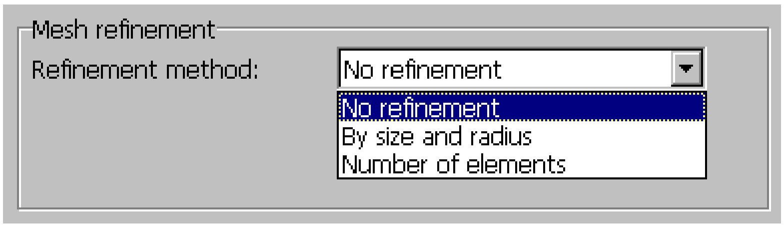
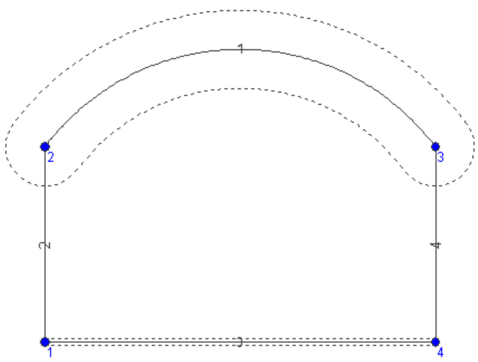


Fig. 4-32 Line mesh refinement method.

(1) Refinement by size and radius.



In this case ‘size’ is the element size along the line and ‘radius’ is the distance from the line where the element size given by of macro-element is resumed. An example of such refinement is shown in Fig. 4-33, where the arc line no.1 has prescribed mesh refinement by size and radius.

Fig. 4-33 Mesh refinement along line.

(2) Refinement by number of elements.

N number of elements (divisions) is prescribed on the line. This type of refinement is marked by dashed lines running on both sides along the line. Example of such refinement is shown in the bottom line in Fig. 4-33.

4.5.3.5 Line Spring Supports

A line can be supported by springs. The springs element is in this case distributed along the line. The model is almost identical with the model of discrete springs except of some special properties related to lines. The procedure is similar to joints with discrete springs as described in Section 4.5.2.4 and will not be repeated here. Only special features for lines will be described. New spring is added by clicking Add button in the input field **Springs**. Then a dialog window for a **New line spring** is open, Fig. 4-34.

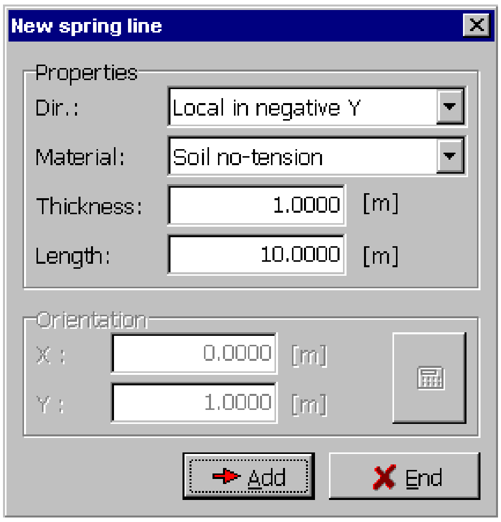


Fig. 4-34 New line spring input.

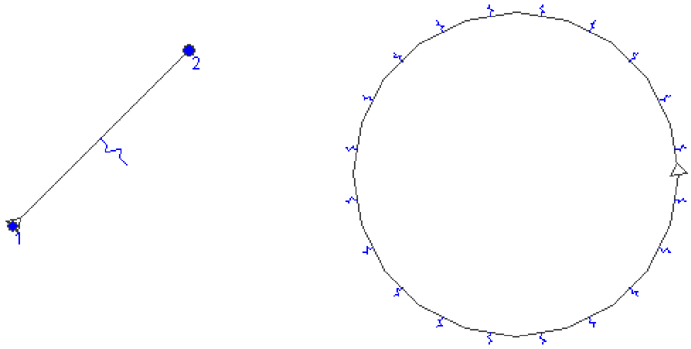


Fig. 4-35 Examples of line springs.

Spring direction has more options in case of line springs. In addition to global directions there are also local directions. Local directions are oriented in local coordinates of lines. Examples of springs defined with respect to local coordinates are shown in Fig. 4-34.

In both case the spring direction defined as 'Local in negative Y'. The inclined line in Fig. 4-34 has positive local X' direction oriented from joint 1 to 2 and consequently the direction of spring is pointing in negative local Y' as shown. Only one spring mark per line is used to depict the line springs before mesh is generated.

The circle line has a positive local direction X' contra-clockwise and thus the local negative Y' direction is pointing out in radial direction.

The feature of local direction is useful for definition of springs representing pressure normal to structural surface.

Cross section area of the line spring is defined as a product of the line length and spring thickness. Thickness is a width of the strip supported by springs (analogy to the thickness of concrete element). The thickness must be entered as one of the line spring dimension. The length of spring has the same meaning as in the discrete springs in joints.

4.5.4 *Macroelements*

Macro-elements The object Macroelements in the access tree must be selected as active for input and marked by a green arrow.

4.5.4.1 Macroelement Input

Macroelement *input by mouse* is started by clicking on this button. The pointer location on the screen is marked as:

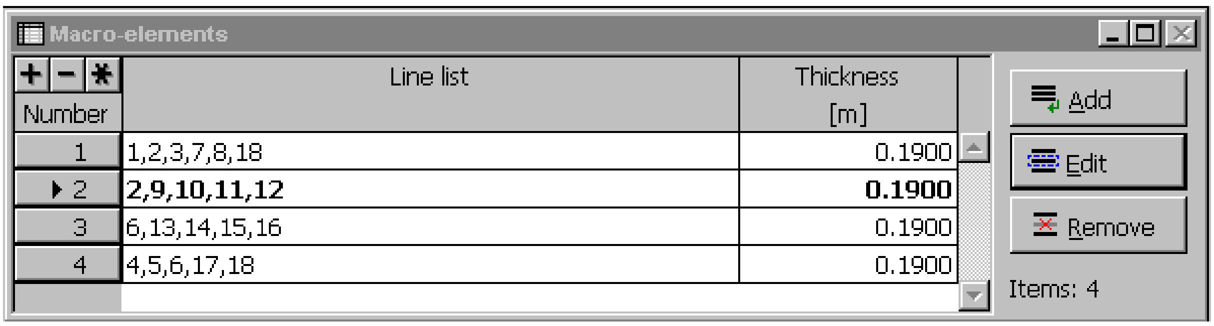
When starting the input first time a prototype input dialog will appear. It should be preformed as described in Section 4.5.4.2 Macroelement prototype.

Each macroelement is defined by lines. The line must enclose a connected region. A line select by mouse is made by placing pointer on line to be selected and a left mouse button click. Selected lines change color to blue. When the last line is selected the area of complete macroelement is filled by gray color and a macroelement mumber appears in the numerical box.

Changes in macroelement data can be made by clicking on this button, putting the pointer on a macroelement to be edited and a click on the left mouse button. Selected macroelement is marked by the blue hatch pattern. This opens a numerical dialog menu. All macroelement data can be edited.

Deleting of existing macroelement can be started by this button. The selected macroelement is marked by the red and blue hatch pattern. Each delete action must be confirmed in a special dialog.

A *numerical macroelement input* can be made as an alternative to the mouse input. The buttons of the numerical window Add, Edit, Remove can be used to open corresponding dialogs for this purpose. The input window has the same appearance as the one for prototype, Fig. 4-37, except that the top field for topology is active. In case of numerical input the line numbers of lines enclosing the macroelement must entered from keyboard in the field **Topology | Boundary list**. Example of numerical window for macroelement is shows in Fig. 4-36. The table includes topology data about macroelement: macroelement number, list of lines forming the macroelement and its thickness.



Macro-elements

Number	Line list	Thickness [m]
1	1,2,3,7,8,18	0.1900
2	**2,9,10,11,12**	**0.1900**
3	6,13,14,15,16	0.1900
4	4,5,6,17,18	0.1900

Fig. 4-36 Window for macroelement numerical data.

4.5.4.2 Macroelement Prototype

A window for macroelement prototype data input is shown in Fig. 4-37.

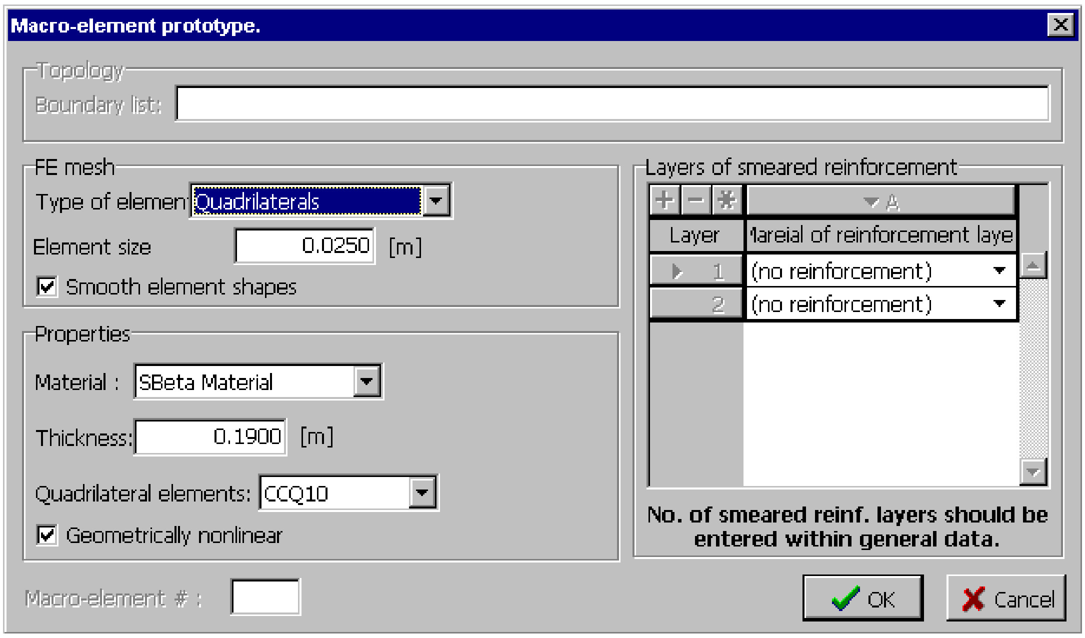


Fig. 4-37 Prototype window for macroelement.

Finite element mesh properties are entered in the field **FE mesh**, Fig. 4-38. The type of mesh option serves for the purpose of selecting a proper meshing method (quadrilateral, triangular or mixed). It does not yet say anything about a finite element type.

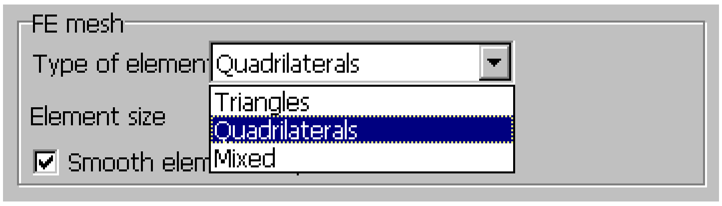


Fig. 4-38 FE mesh menu in macroelement.

Further parameters for meshing are **Element size** and **Smoothing**. The former decides about the mesh density and the later about its form. Smoothing avoids sharp angles in elements.

Material can be selected from a list as shown in Fig. 4-39. These materials must be first defined as described in the section on Materials. Only materials compatible with the given macroelement are offered.

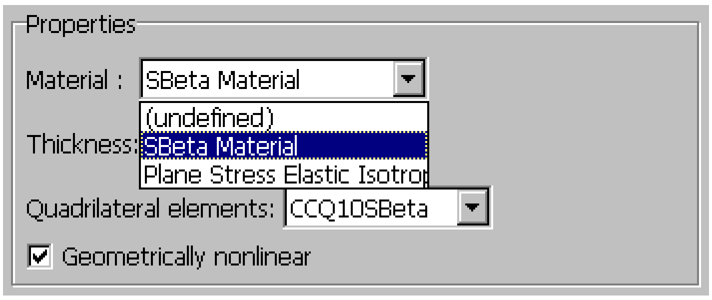


Fig. 4-39 Material properties menu in macroelement.

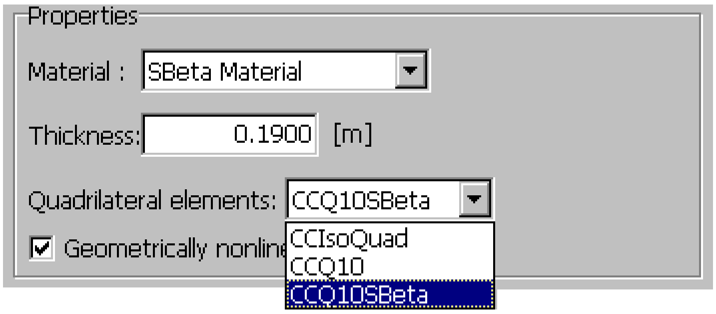


Fig. 4-40 Type of finite element in macroelement.

A finite element type can be chosen in the field **Quadrilateral elements**. For example, if a quadrilateral form of mesh is chosen one of several quadrilateral finite elements can be used for this mesh.

Geometrically nonlinear analysis, if activated, has two consequences in the analysis: (1) Large strains and geometrically nonlinear terms are considered in the finite element formulation, (2) Updated coordinates (of deformed structure) are used for equilibrium analysis. If not checked, only updated coordinates are considered. Equilibrium on a deformed structure is always considered.

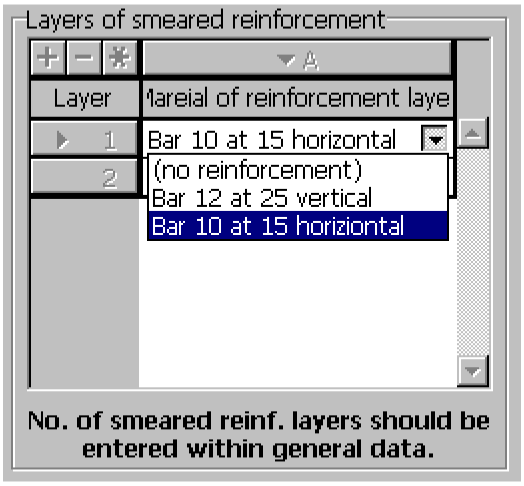


Fig. 4-41 Smeared reinforcement.

Smeared reinforcement can be entered for macroelement in the field **Layers of smeared reinforcement**. Only the number of layers chosen in **General data** can be used for this purpose. Each layer can be filled with one smeared reinforcement type. Example: there is an orthogonal reinforcing mesh. Each bar direction must be entered as one smeared reinforcement layer. This can be done by clicking on the small black arrow in the field of one layer. After this a list of already defined reinforcement is unfolded as shown in Fig. 4-41.

4.5.5 Delete Group of Objects

First select objects using tools from the toolbar shown in Fig. 4-3 (type of object and method). Selected objects are marked by green color. Then click on **Delete selected** button, see Fig. 4-42 (left).

Fig. 4-42 Delete and assign to a group of selected objects.

4.5.6 Assign Properties to a Group of Objects

The objects must be first selected using the numerical list window by click on the item number button. (A multiple selection can not be done by mouse.) The selected objects are marked by blue color in the list and also by green on the model.

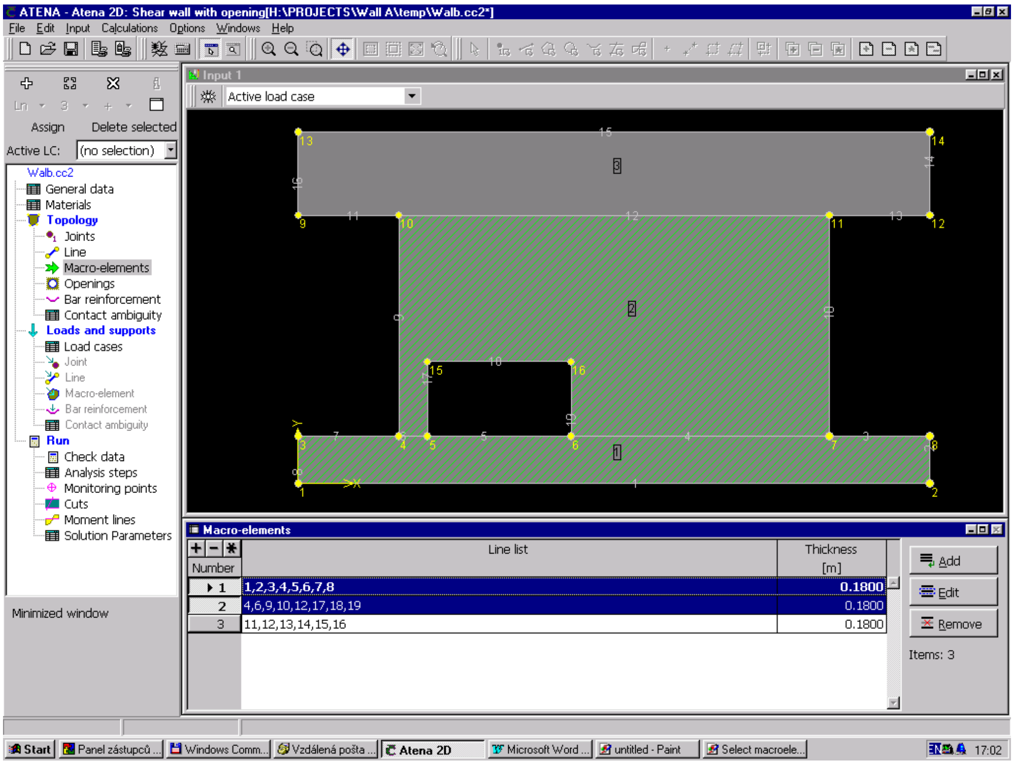


Fig. 4-43 Selected group of macroelements.

Macroelements 1 and 2 are selected in the example in Fig. 4-43.

By pressing the button Assign in the input tools, Fig. 4-42 (right), a pop-up input box appears as shown in . A number of a master macroelement can be entered, properties of which will be used for a group of macroelements.

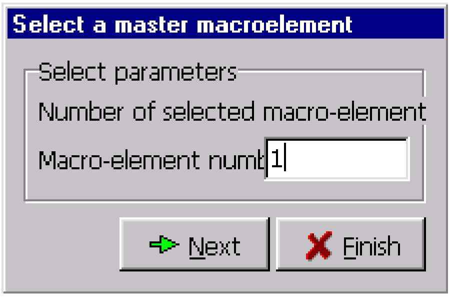


Fig. 4-44 Master macroelement for a group.

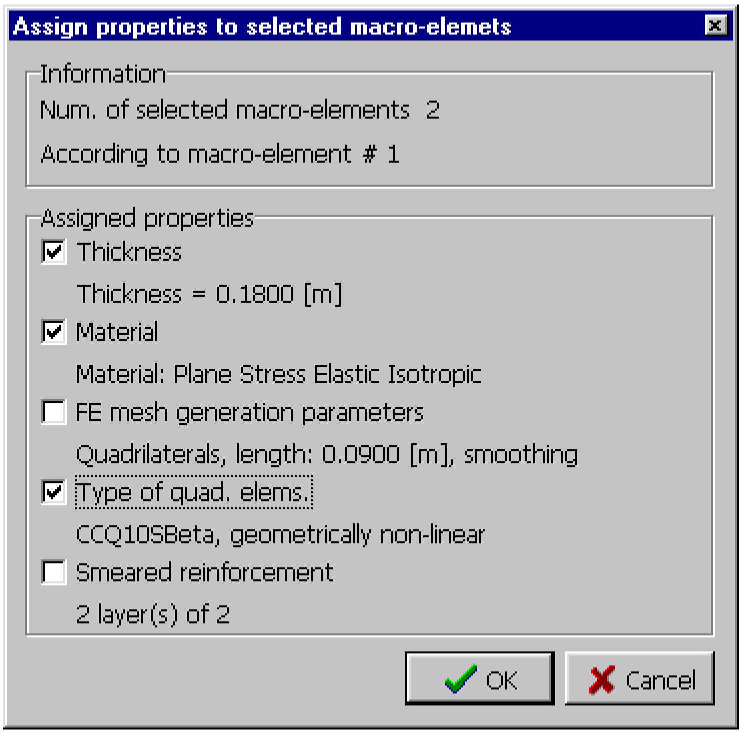


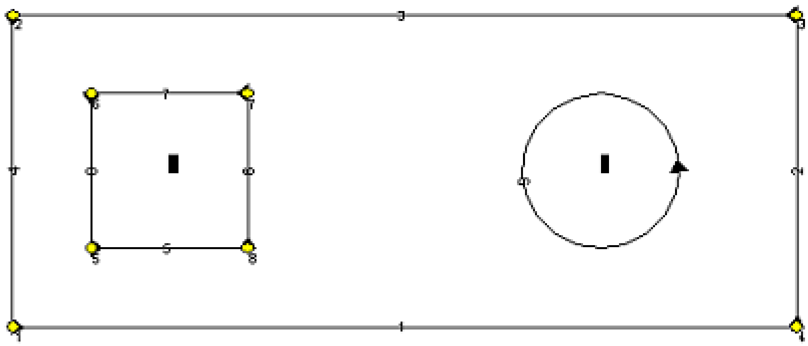
Fig. 4-45 Assign selected properties to a group of macroelements.

Next, a window as shown in is opened, in which the properties to be assigned can be selected. In the above example only the thickness, material and element type are assigned.

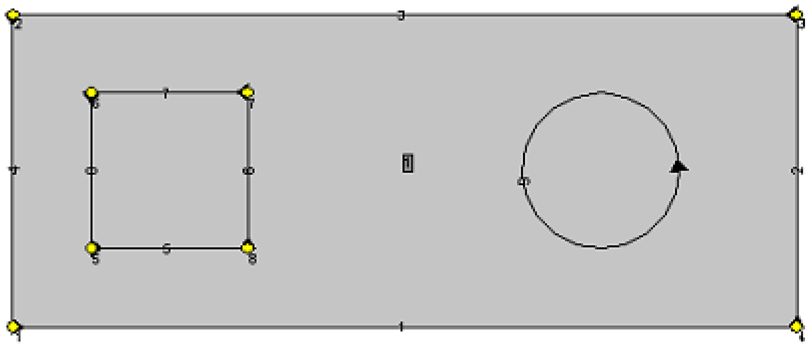
Similar procedure can be used also for a group editing of joint and line properties.

4.5.7 Opening

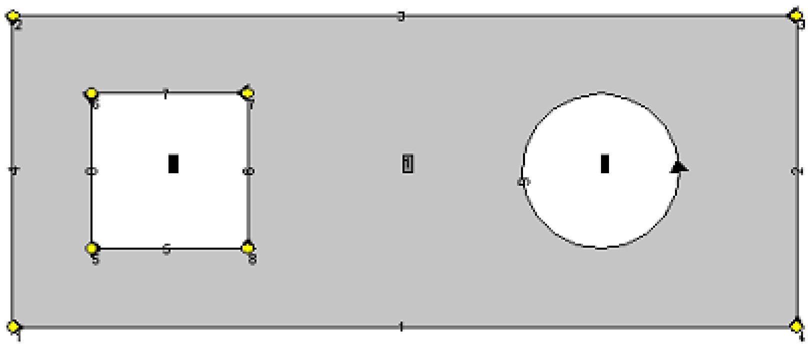
Openings are similar objects as macroelements. They are formed by a chain of simply-connected lines. An opening object located within a macroelement is considered as empty space. Example of openings shown in Fig. 4-46 can be described in the following sequence of steps:



The lines are entered by using tools for input of joints and lines.



The macroelement without opening is defined using the macroelement input tools.



The openings are entered by choosing the tool for opening and selecting all lines forming each opening.

Fig. 4-46 Example of Openings.

4.5.8 Reinforcement

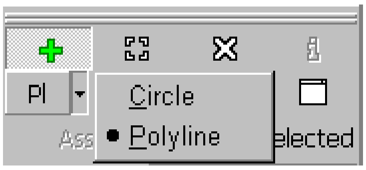
Object Reinforcement must be selected as active for input and marked by a green arrow in the access tree. This activates the access to the tools for input, edit and delete of reinforcement bars. Each bar is defined by a polyline consisting of chain of connected segment lines. The segments can be straight lines or arches.

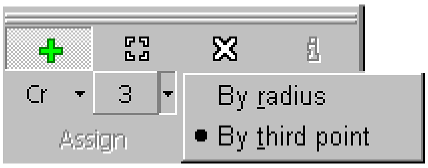
4.5.8.1 Reinforcement Bar Input

Reinforcement bar input *by mouse* is started by clicking on this button. The pointer location on the screen is marked according to the definition method of the segment:

Line   Arc

Circle by center and radius   Circle by 3 points





The setting of the segment geometrical form can be made in the field of tools for mouse input shown in left. These tools are located in the second row under the input buttons. By clicking on the first black arrow the menu for bar form is unfolded. Two options are available: **Circle** and **Polyline**. A click on the second arrow unfolds a menu for the circle definition: **By radius** and **By third point**.

In the polyline, a segment form is selected by pressing the 'Shift' key. This change is visually confirmed by changing of pointer icon from Line to Arc.

The segments are generated by pressing the left mouse button at segment joints. A double click ends the bar input. Arc segments generated by mouse are constructed under a condition of equal tangent at (only) first joint. This makes the bar form on transition from line to arch an arc to arc (not from arc to line) a smooth one.

Changes in reinforcement data can be made by clicking on this button, putting the pointer on a reinforcement bar to be edited and a click on the left mouse button. Selected reinforcement is marked by the blue color. This opens a numerical dialog menu for data changes.

Deleting of an existing reinforcement bar can be started by this button. The selected reinforcement bar is marked by the red color. Each delete action must be confirmed in a special dialog.

Numerical input of reinforcement can be made as an alternative to the mouse input. The buttons of the numerical window Add, Edit, Remove can be used to open corresponding dialogs for this purpose. An example of numerical window for reinforcement is shows in Fig. 4-47. The data include: bar number, type of reinforcement (normal, eternal cable), geometrical form and number of segments. To start an input of a new bar the button Add must be clicked. The input window for a new bar is shown in Fig. 4-48. It shows the data of a bar from Fig. 4-50. The bar geometry can be of entered when selecting **Topology** and clicking on the button Add. An input dialog for a segment is shown in Fig. 4-50. The properties of bar (material, diameter, number of bars) can be entered in the menu **Properties**, Fig. 4-49.

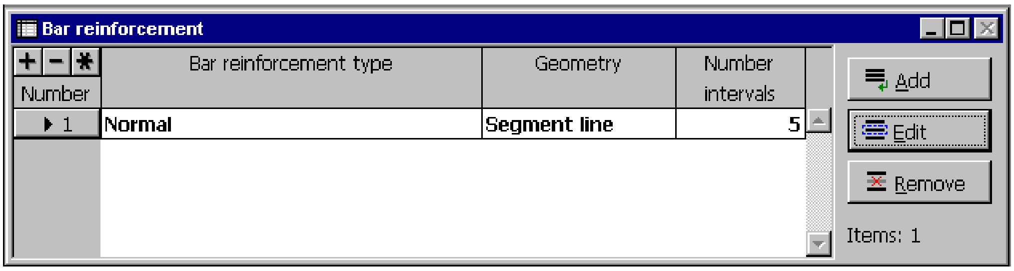


Fig. 4-47 Numerical window for bar reinforcement.

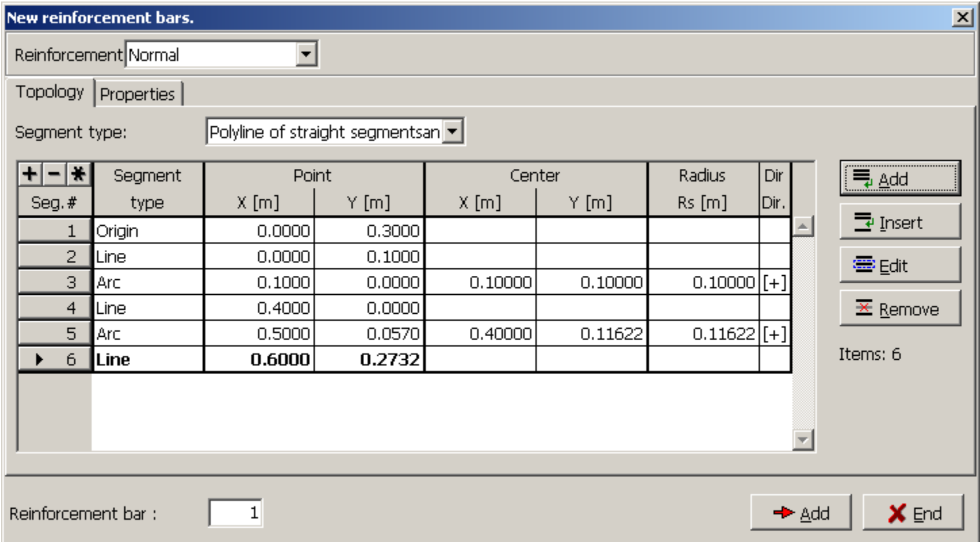


Fig. 4-48 Input window for new bars.

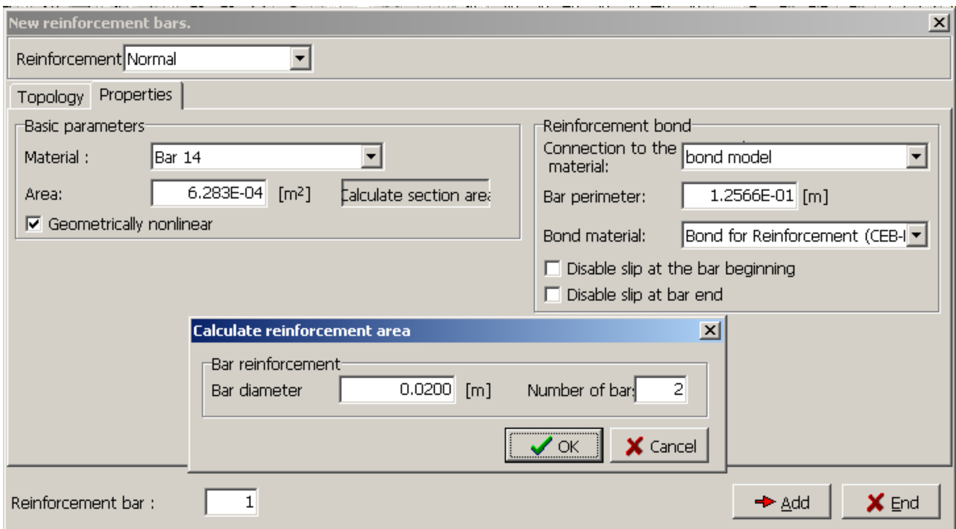


Fig. 4-49 Properties of new bar.

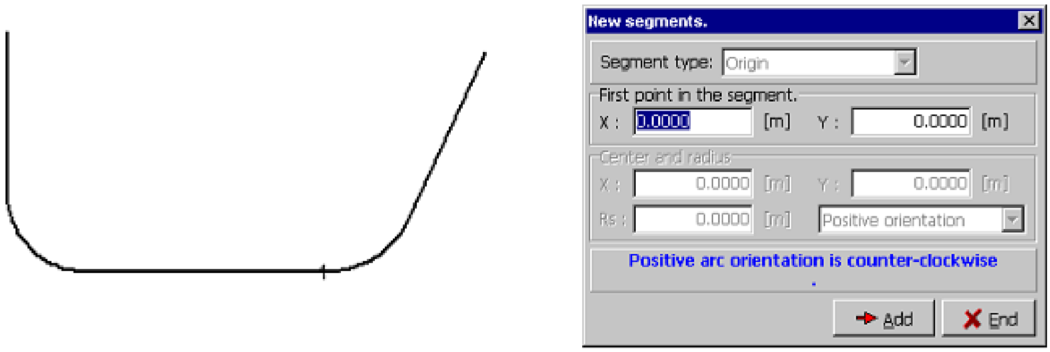


Fig. 4-50 Example of bar and definition of a new segment.

4.5.8.2 External Cables

The external cables can be specified in the menu **New Reinforcement | Reinforcement**. Then the input menu appears as shown in Fig. 4-51. Only straight line is allowed as a geometrical form of a segment in external cables.

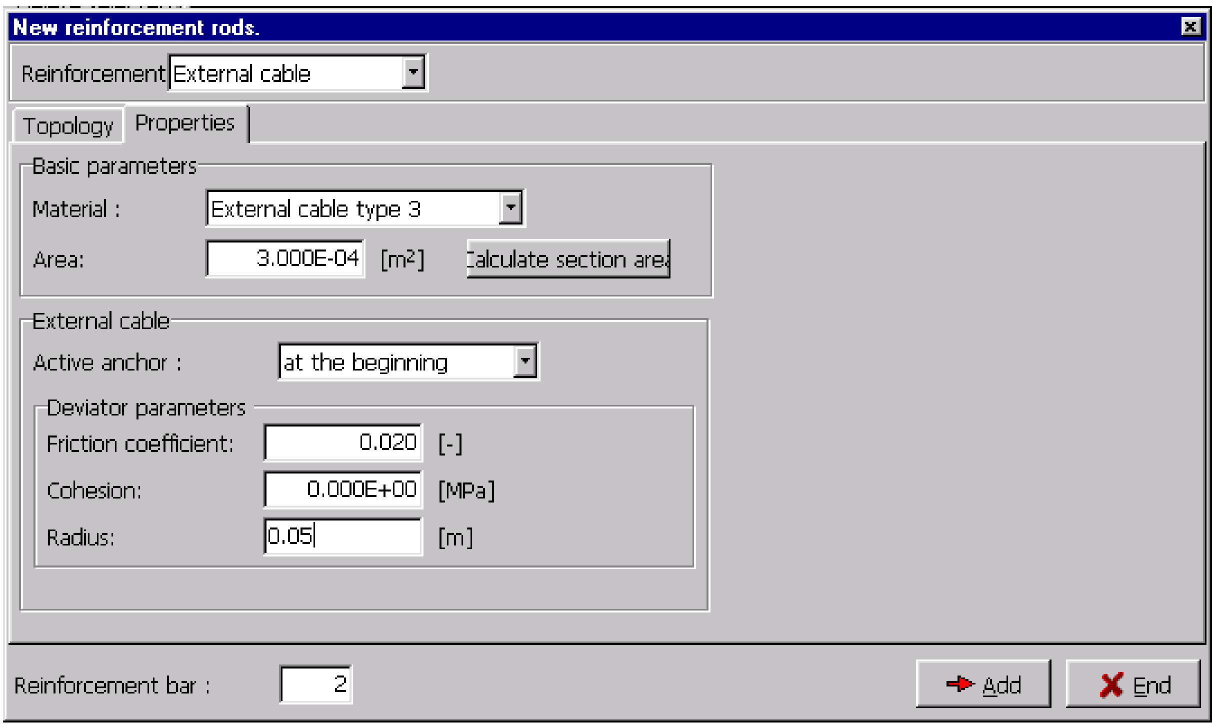


Fig. 4-51 Properties of external cable.

The data for basic parameters are the same as for normal reinforcement. Additional data specific for external cables can be entered in the field **External cable**. An **active anchor** is the anchor at which a prestressing force is applied. Its position is important for calculation of frictional losses of prestressing. The beginning of the cable is at the first point. It is depicted by an arrow. **Deviator parameters** are for the calculation of frictional forces. External cables are depicted on screen by dashed lines.

4.5.8.3 Contact Ambiguity of Lines

A line on a border of two macroelements is always a double line. Connections of these lines can be of three types. (The lines styles for screen graphics for connections are written in parenthesis.):

1. *Rigid.* The lines are identical and rigidly connected. (Single line)
2. *No connection.* The lines have no relations. They behave like two independent lines. (Double line).
3. *Interface.* Behavior of two lines is governed by an '2D interface' constitutive law. This law must be defined in Materials. (Dashed double line)

The line is entered first as a single line. After definition of a macroelement all lines between macroelements are automatically generated as double lines and by default set as rigidly connected. Other types of connection can be chosen by editing of lines. For this it is necessary to go to the menu Lines Line and Edit . Then a dialog window, as shown in Fig. 4-52, appears. The section **Connection** shows information only on double lines located between two macroelements.

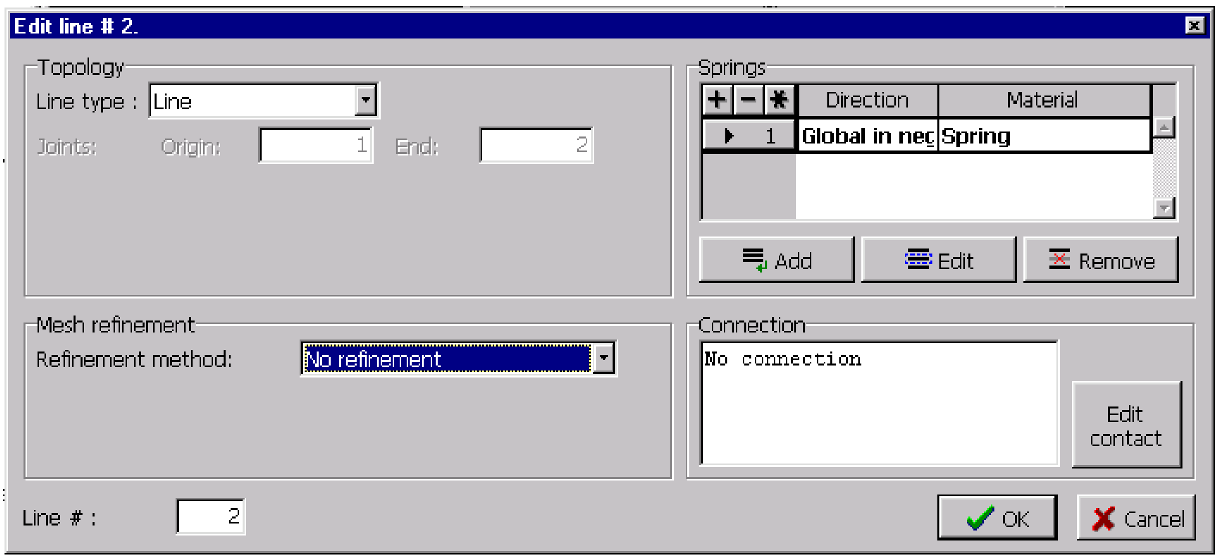


Fig. 4-52 Edit line connection.

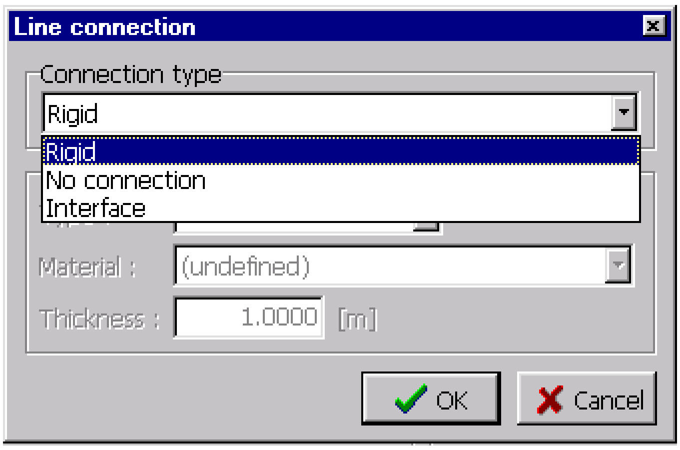
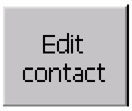


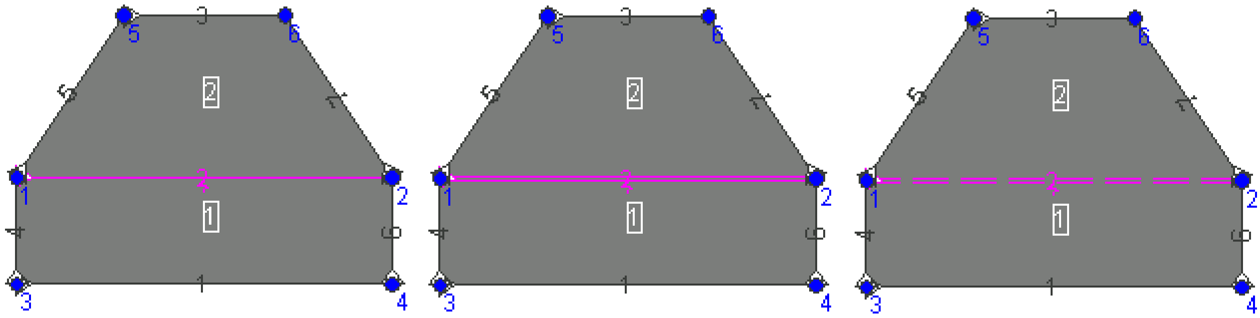
Fig. 4-53 Line connection menu.

By clicking on the button 

a dialog shown in

Fig. 4-53 is opened. An appropriate type of connection can be selected from the menu. In the box ‘Material a pre-defined interface material can be assigned. In the box ‘Thickness’ a thickness of the interface must be specified. This thickness can be different comparing to the adjacent macroelements if desired.

Each type of connection is marked by a special line style as mentioned above and illustrated in Fig. 4-54. An example shows two macroelements connected by line no.2. The figure shows three types of connection.



Rigid **No connection** **Interface**

Fig. 4-54 Three types of line connection between macroelements.

4.5.8.4 Contact Ambiguity of Springs

Spring attached to one of the double lines on a macroelement border line. As first macroelement must be constructed and spring must be entered. The line connection type must be either ‘No connection’ or ‘Interface’. Then a contact ambiguity of spring can be chosen as follows.

Initially a spring can be assigned to a single line. After forming a macroelements, if the line happens to be a border line it is transformed into a double line and it must be decided which of the lines (macroelements) are to be supported by the springs. The menu for Contact ambiguity can be used for this purpose.

An object Contact ambiquity must be activated for input by a click in the access tree and marked by a green arrow. Only input through numerical window is possible. All disconnected lines will be listed in the numerical window as shown in Fig. 4-55. Initially the spring assignment is not determined and indicated as ‘undefined’ in the most right column.

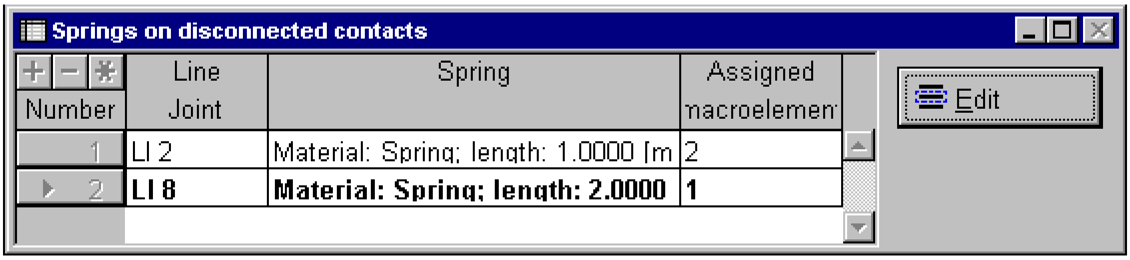


Fig. 4-55 Numerical window with a list of disconnected lines.

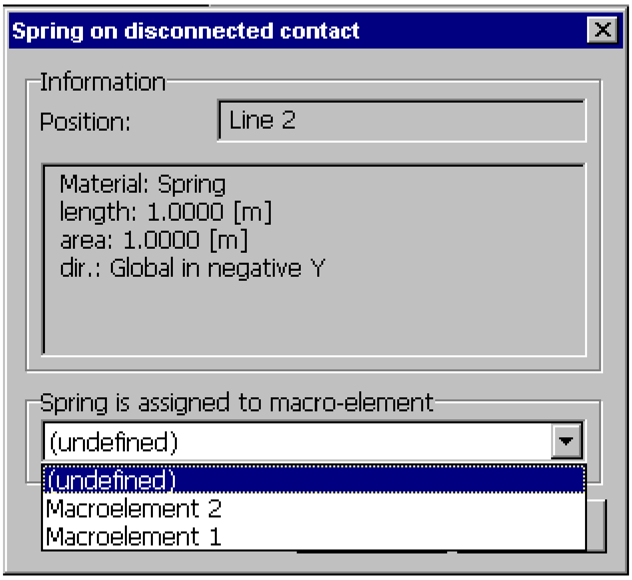


Fig. 4-56 Spring assignment to macroelement.

Initially the spring assignment is not determined and indicated as **undefined** in the most right column.

A disconected line can be edited by clicking on the button 'Edit', which opens a dialog as shown in Fig. 4-56. One of two macroelement can be chosen. The line spring will be attached to this macroelement.

4.6 Load Cases

4.6.1 Load Case Types

Load cases are objects for definition of external actions on the structure. Prior to an input of load cases, geometry and properties of the structure must be defined as described in sections on Materials and Topology. Only one type of loading can by defined within one load case. (For example, supports and loading forces must be defined in separate load cases.) Later, any number of load cases can be combined in load steps and form a history of loading.

A load case input can be started by clicking on the Load cases icon in the access tree. An active load case input is marked by a green arrow. After this the numerical window changes to a form of **Load cases** as shown in Fig. 4-57.

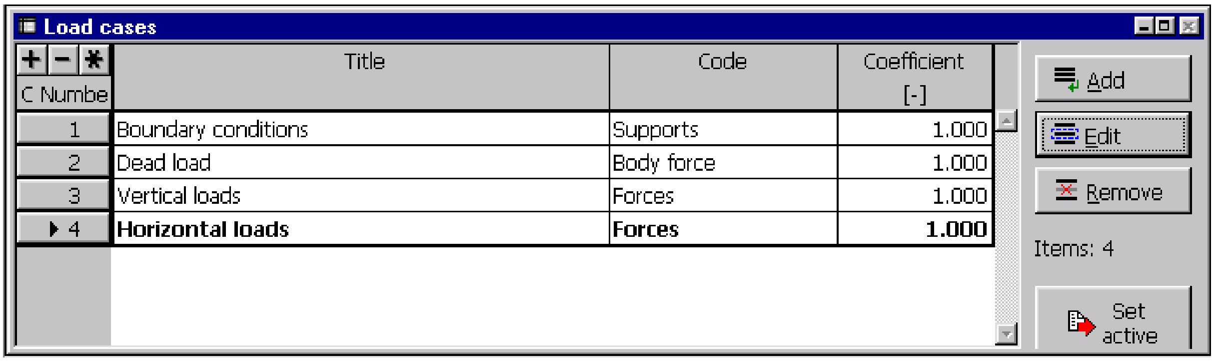


Fig. 4-57 Load cases numerical window.

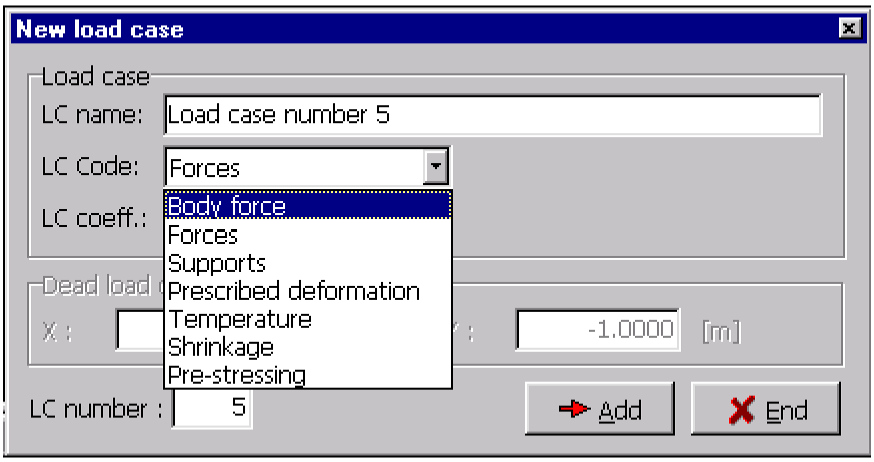


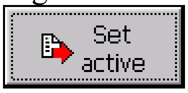
Fig. 4-58 New load case.

By clicking on the button 

load cases can be added. Then a new load case dialog appears. Following data related to load cases can be defined within this window:

- **LC Name:** Default name is 'Load case no. i'. It can be changed to any description.
- **LC Code:** Possible options are: **forces**, **supports**, **prescribed deformations**, **temperature**, **shrinkage**, **pre-stressing**. One of them must be chosen.
- **LC Coefficient:** Numerical values in the load case are multiplied by this coefficient.

So far only a load case code was defined. Now, before any input of the load case data, a load case must be activated. This can be done by two ways:

1. Selecting load case in the numerical window. It is marked by a small black arrow before the LC number. For example in Fig. 4-57 the selected load case is no. 4. A click on the button in the bottom right corner 

will activate the load case.

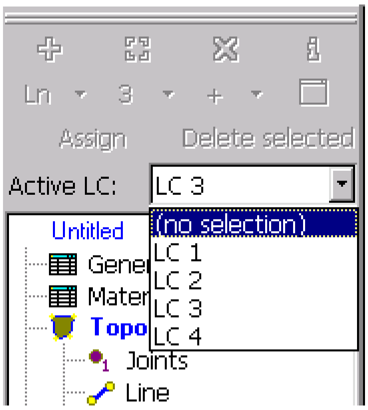


Fig. 4-59 Active load case.

2. Selecting an active load case from a list in the menu above the access tree as shown in Fig. 4-59. The active load case can be always checked in this field.

When the active load case is set a loading data for the load case can be entered as will be described in the following sections.

4.6.2 *Changing Load Case Data*

Editing of load case data has some limitations. The load case must be selected first and then by clicking on the button 

the **Edit load case** is opened. The window is similar to the **New load case**. In this dialog all data except load type (**LC code:**) can be changed. A change of a load code can be done only by deleting existing load case and introducing a new load case.

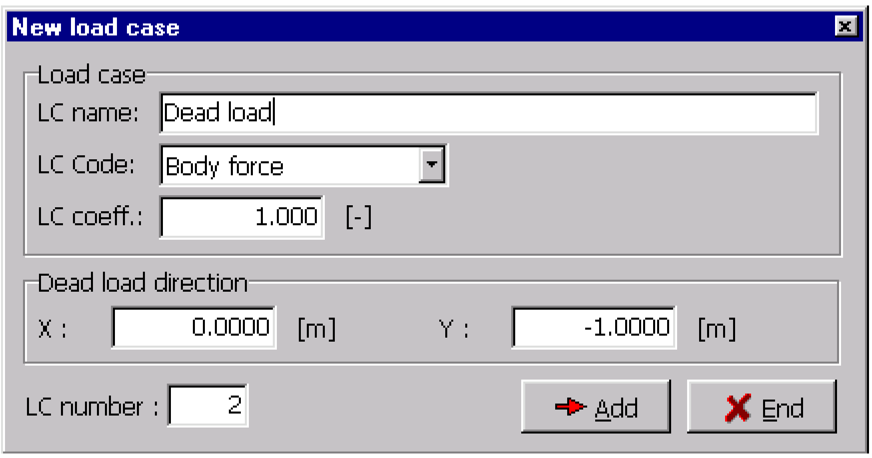


Fig. 4-60 Body force direction.

4.6.3 Body Force

The load code 'Body force' does require only one input data: direction, see Fig. 4-60. This is done in the field 'Direction' by entering components of direction vector. No other data are required. A default direction is –Y, i.e. vertical direction in case of a usual global coordinate system.

4.6.4 Force at Joint

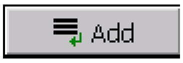
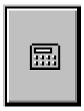
Discrete forces can be applied to joints. Load case code 'Forces' must be set active. Input of load actions is done in two steps: (1) Selects joints; (2) Apply forces.

4.6.4.1 Select Joints

Joints can be selected using select tools. As first an object type is selected by clicking on the joint button  in the select menu bar. Then a selection method is chosen from by clicking on one of these buttons  and one or a group of joints are selected. The selected joints are marked by green color.

4.6.4.2 Apply Forces

The object must be activated for input by clicking on **Joints** in the access tree in the section **Load cases**. The active joint for force input is shown by green arrow. After this the numerical window changes to **Loading force in joint**, Fig. 4-63.

By clicking on the button  an input menu appears as shown in Fig. 4-61. Two parameters of the force must be entered: direction and magnitude. In the above example the force is vertical, pointing downwards. One of three options in the field **Dir.** can be chosen after clicking on the arrow : **Global X**, **Global Y** and **Global angle**. A direction angle can be entered in the field **Force orientation** in two ways: (1) Components of unit vector **X, Y**; (2) Direction angle in degrees. For later way the button Calculate orientation  must be pressed. An example is shown in Fig. 4-62 for a force in direction –60°.

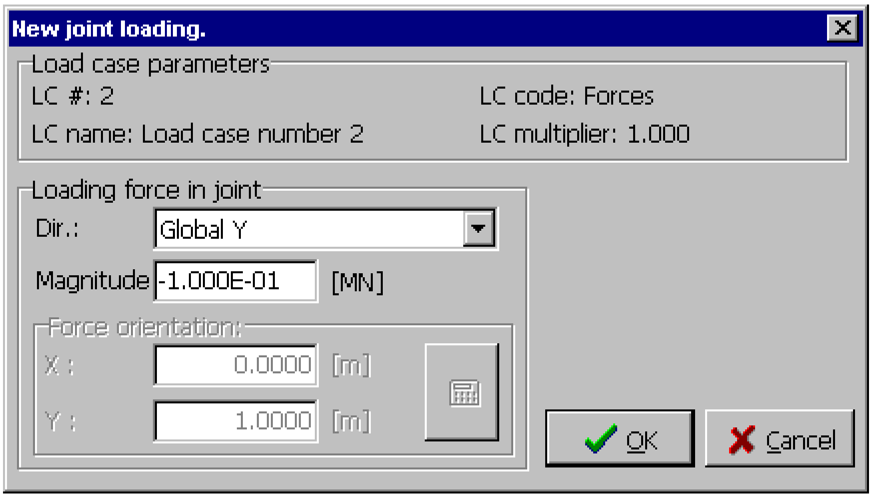


Fig. 4-61 New force at a joint

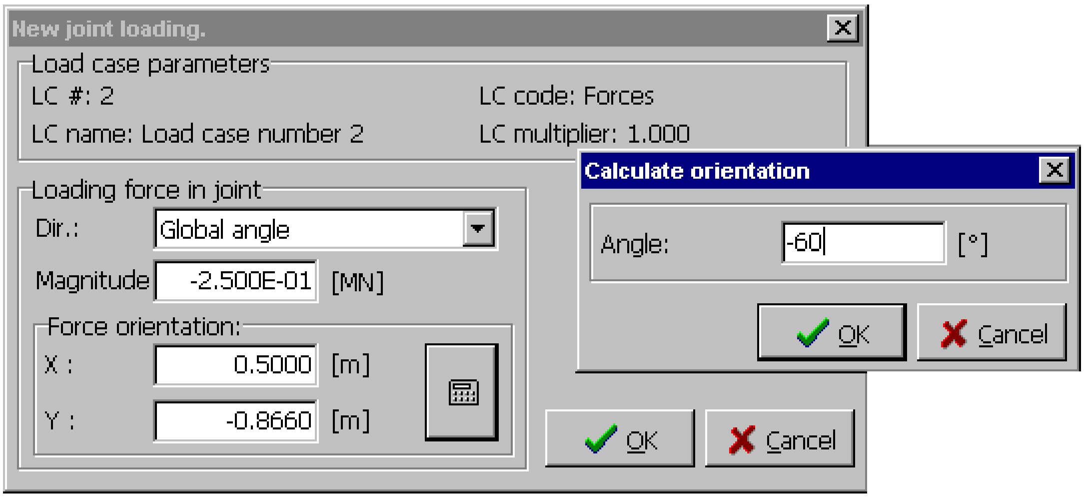


Fig. 4-62 New force at -60°.

The data of forces at selected joints are shown in the list of the numerical window as shown in Fig. 4-63.

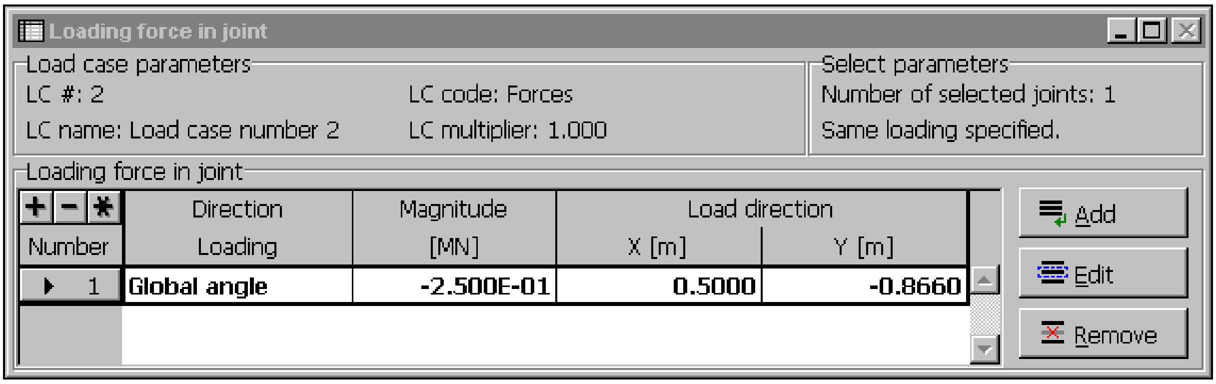


Fig. 4-63 List of forces at joint.

If one or more joints with the same set of forces are selected the list shows these force If more joints with different forces are selected (see example in Fig. 4-64) then the list is not shown and the note: **Different loading specified** appears in the field **Select parameters**.

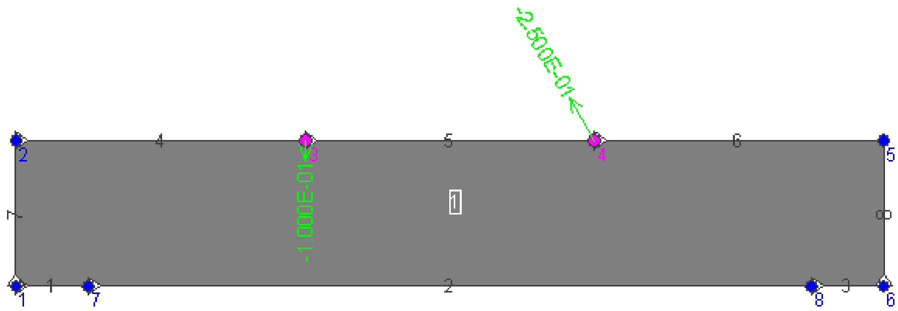


Fig. 4-64 Loading case with two forces.

Changing of already entered forces can be done in the numerical window. First the force must be selected. This can be done by using select tools in the same manner as in case of input. By clicking on the button  an edit menu similar to the one for input, Fig. 4-61, Fig. 4-62 will be opened and any parameters of the loading force can be changed.

Note, that only an uniquely defined force can be edited. For example, if two forces with different magnitude or direction are selected, such as shown in Fig. 4-64, they will not appear in the list and obviously cannot be edited as a group. They must be selected and edited one by one.

Removing of already entered forces is done by selecting in the same way as described above and by a clicking on the button .

4.6.5 Force Along Line

Distributed forces can be applied to lines. The load case code 'Forces' must be set active. Input of load actions is done in two steps: (1) Selects lines; (2) Apply forces to lines.

4.6.5.1 Select Lines

Lines can be selected using select tools. As first, an object type is selected by clicking on the line button in the select menu bar. Then a selection method is chosen from by clicking on one of these buttons and one or a group of joints are selected. The selected lines are marked by green color.

4.6.5.2 Apply Forces to Line

Line The object **Line** must be activated for input by clicking on it in the access tree section **Load cases**. The active line for force input is marked by the green arrow. After this the numerical window changes to Line load, Fig. 4-65. By clicking on the button Add in the numerical window an input menu appears as shown in Fig. 4-66. The properties and data of line load can be entered through the dialog boxes **Type** and **Dir.** These input types will be treated in the following sections.

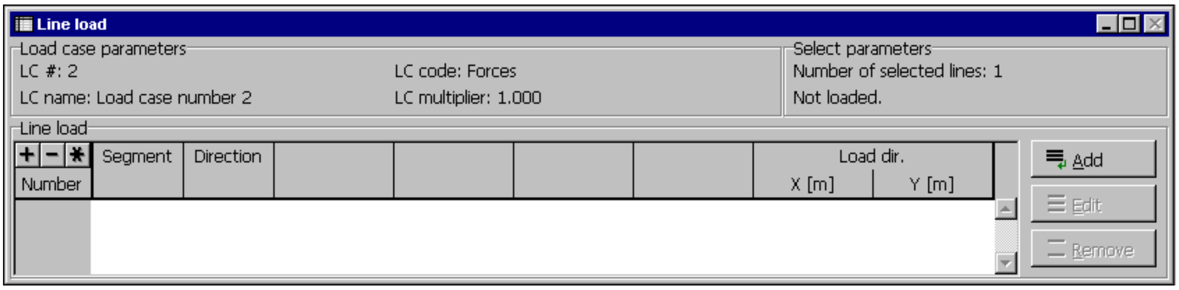


Fig. 4-65 Line load input window.

4.6.5.3 Continuous Line Load

The menu for the load type **Continuous full length** is shown in Fig. 4-66. Example of this load is shown in Fig. 4-67. The load intensity value *f* and its direction should be entered.

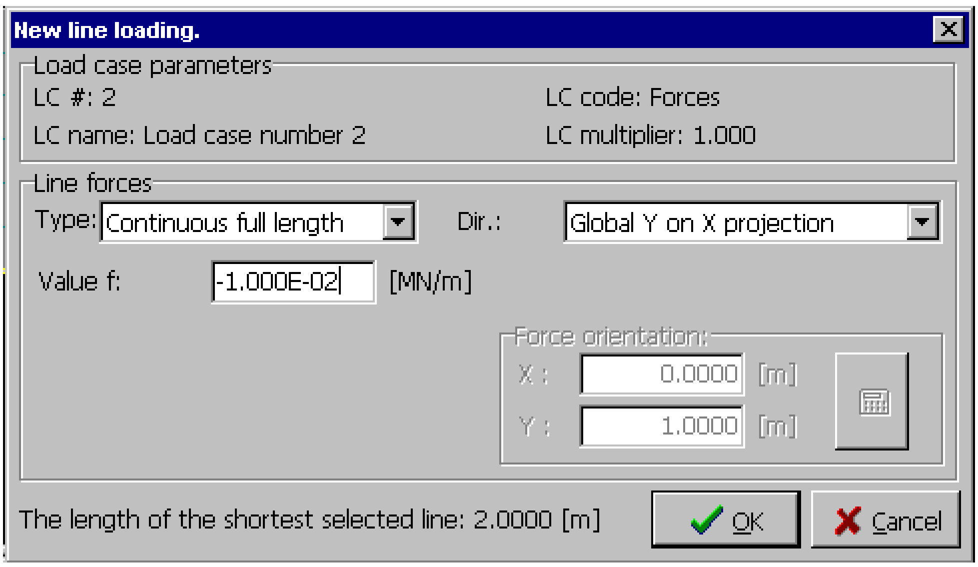


Fig. 4-66 Continuous line force menu.

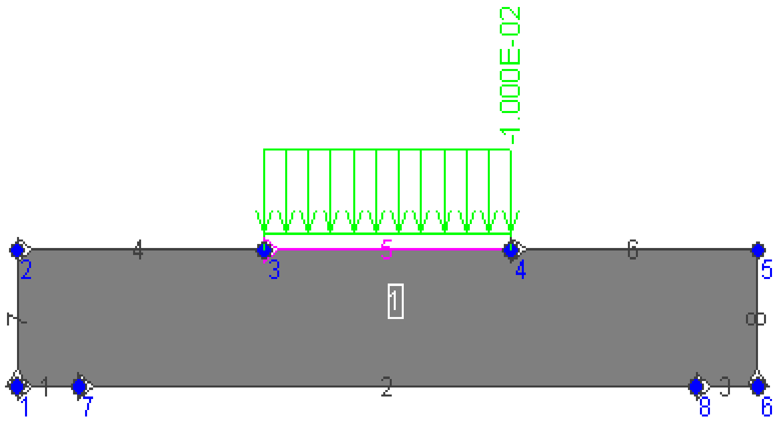


Fig. 4-67 Continuous line force example.

4.6.5.4 Force Within Line

The menu for loading type **Point load** is shown in Fig. 4-69 and iths example in Fig. 4-68. In this loading type two parameters of force can be entered: the force magnitude ***f*** and its distance ***a*** from the first point of the line.

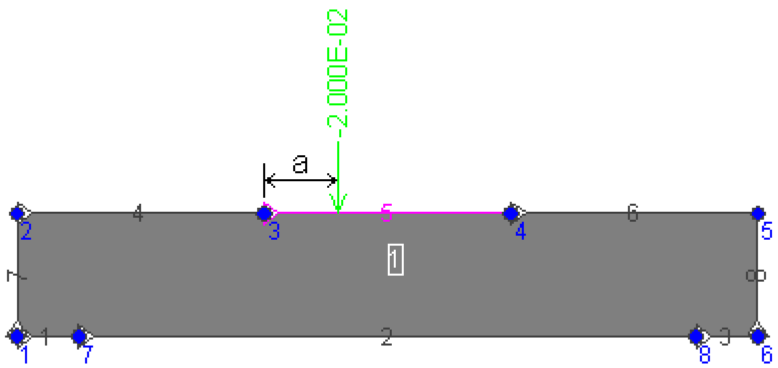


Fig. 4-68 Force within a line - example.

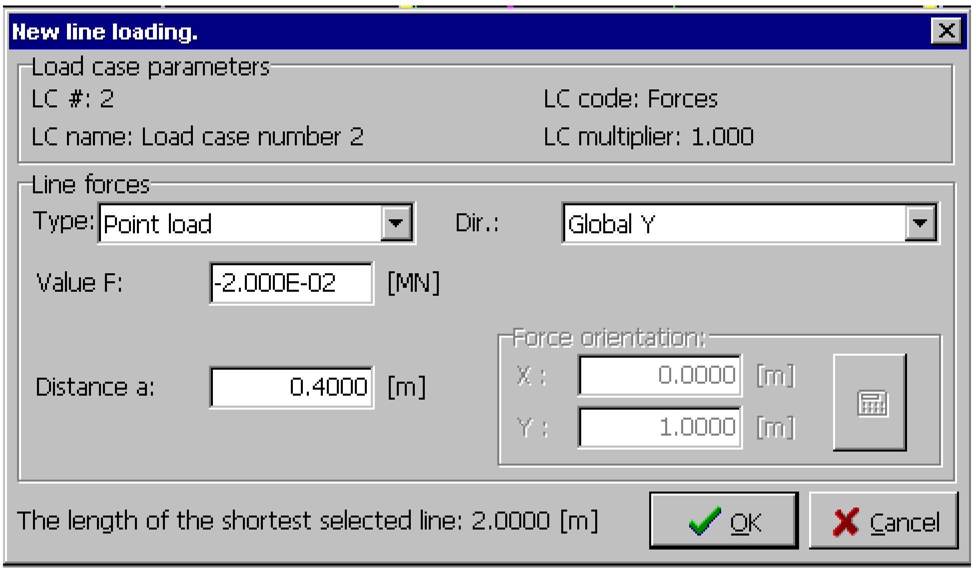


Fig. 4-69 Force within line menu.

4.6.5.5 Line Load Partial

The menu for loading type **Partial and quadrilateral** is shown in Fig. 4-70.

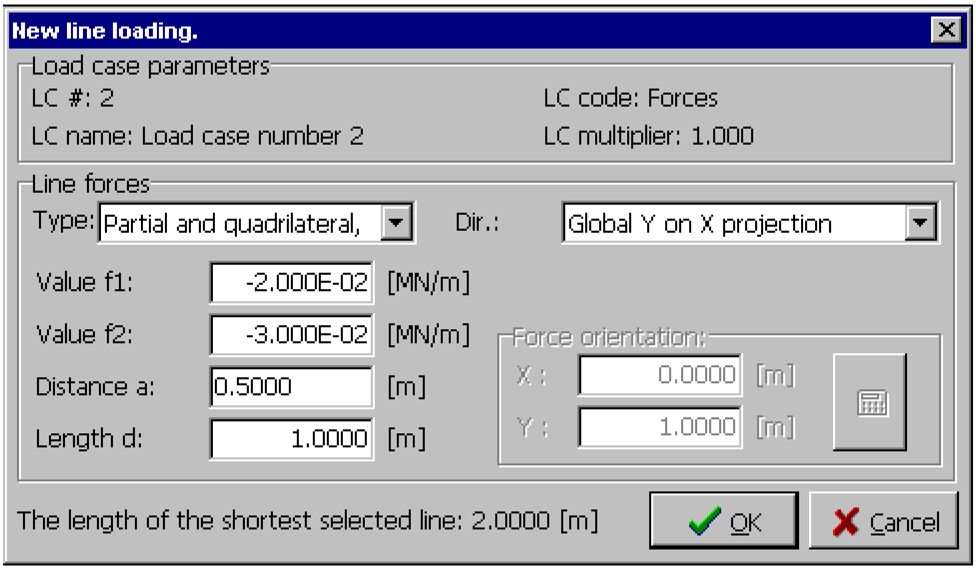


Fig. 4-70 New line load. Partial, quadrilateral.

The parameters illustrated in Fig. 4-70 can be entered: force magnitude f_1 at the starting point of partially distributed load, f_2 at the end point, distance a from the first point of the line and d length of the line load.

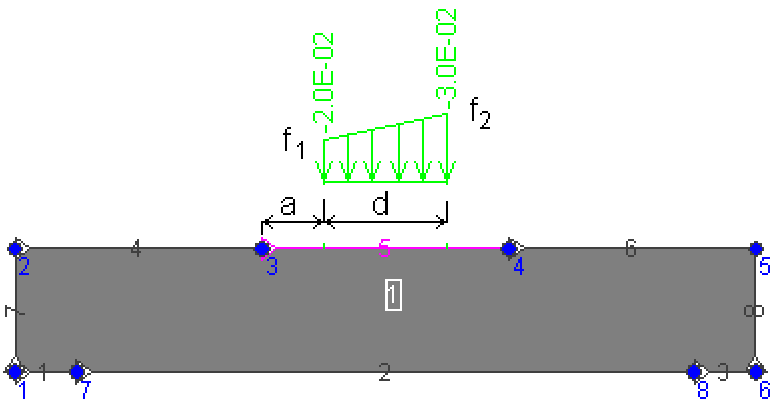


Fig. 4-71 Line load distributed on part of line – example

4.6.5.6 Line Load Quadrilateral

The menu for loading type **Quadrilateral** is shown in Fig. 4-72. In this loading type the parameters illustrated in Fig. 4-73 can be entered: force magnitude f_1 at the starting point of partially distributed load, f_2 at the end point.

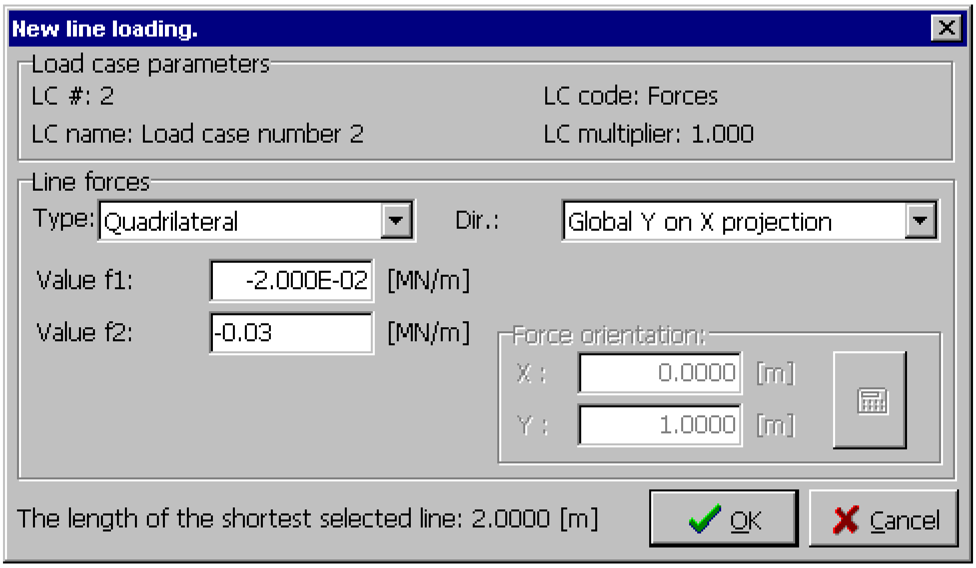


Fig. 4-72 Line load quadrilateral – menu.

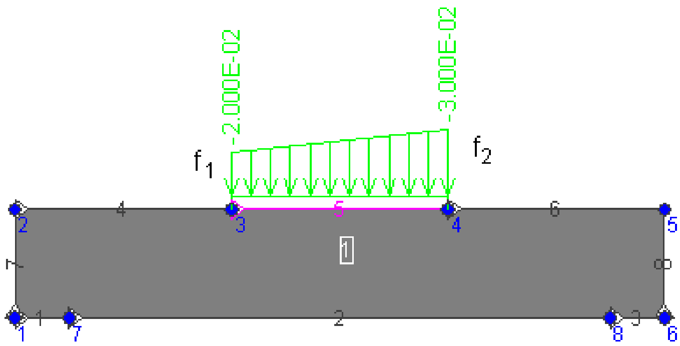
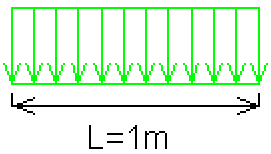


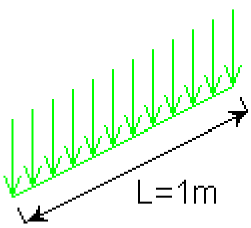
Fig. 4-73 Line load quadrilateral – example.

4.6.5.7 Direction and Projection of Line Load

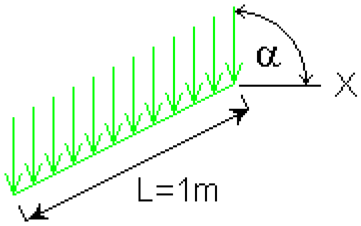
Methods of direction and projection of the line load is described in the next illustrations. The local coordinate system of line has the local axis X' in direction of line and pointing from the first to second point.



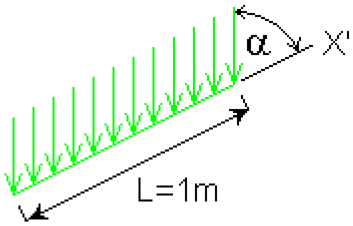
Global Y on X projection.



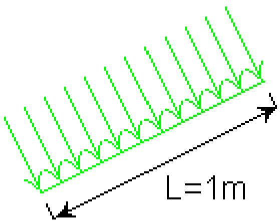
Global Y along line.



Global angle along line.



Local angle along line.



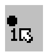
Local Y along line.

The above description shows the cases of line loads of intensity -1 in global Y (local Y') directions. In a similar manner is defined the line load in a given angle.

4.6.6 Supports

Supports can be applied to joints and lines. The load case code **Supports** must be set active. Input of supports is done in two steps: (1) Selects joints or lines; (2) Apply supports.

4.6.6.1 Select Joints or Lines

The objects 'Joint' or 'Line' must be selected by the clicking on the buttons   in the select menu bar. Then a selection method is chosen by clicking on one of selection method buttons . The selected objects are marked by green color.

4.6.6.2 Apply Supports to Joint

The object **Joint** must be active and marked by green arrow in the section 'Load cases of the access tree. After this a numerical window changes to a form shown in Fig. 4-74.

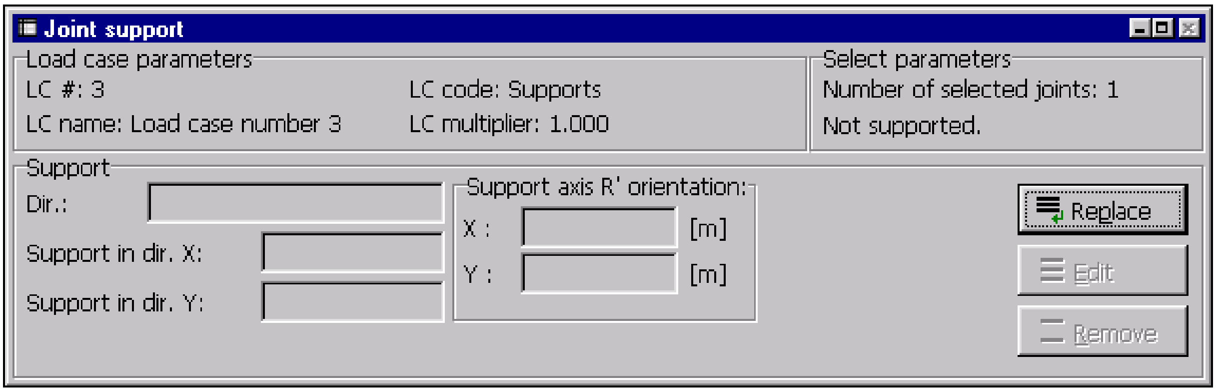


Fig. 4-74 Supports in joints, numerical window.

By clicking on the button 

the input window for supports at joint is opened and support conditions can be entered, Fig. 4-75.

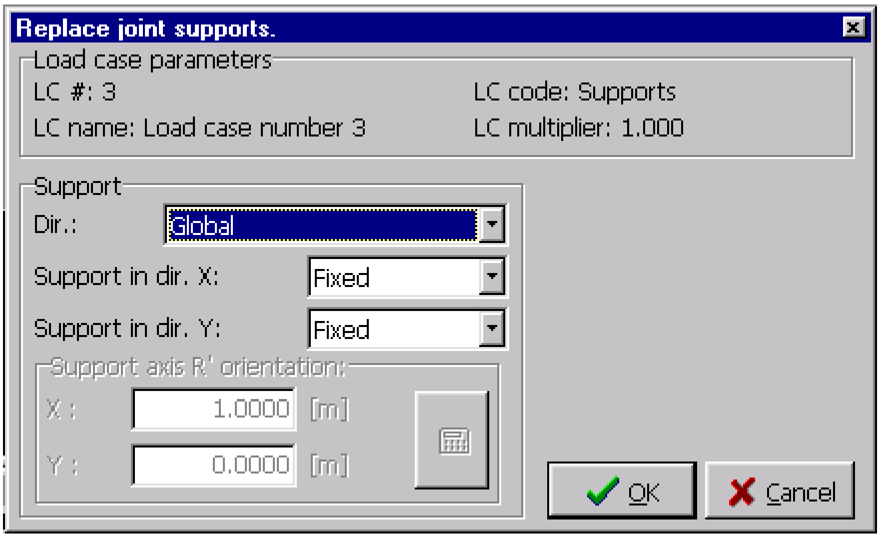


Fig. 4-75 Replace joint supports.

The direction of supports is oriented to global coordinated system and is chosen in the field **Dir.**. An inclined support can be imposed by chosing **Global angle**, see Fig. 4-76. The angle of support direction can be entered either by components X, Y of a unit direction vector, or by an angle.

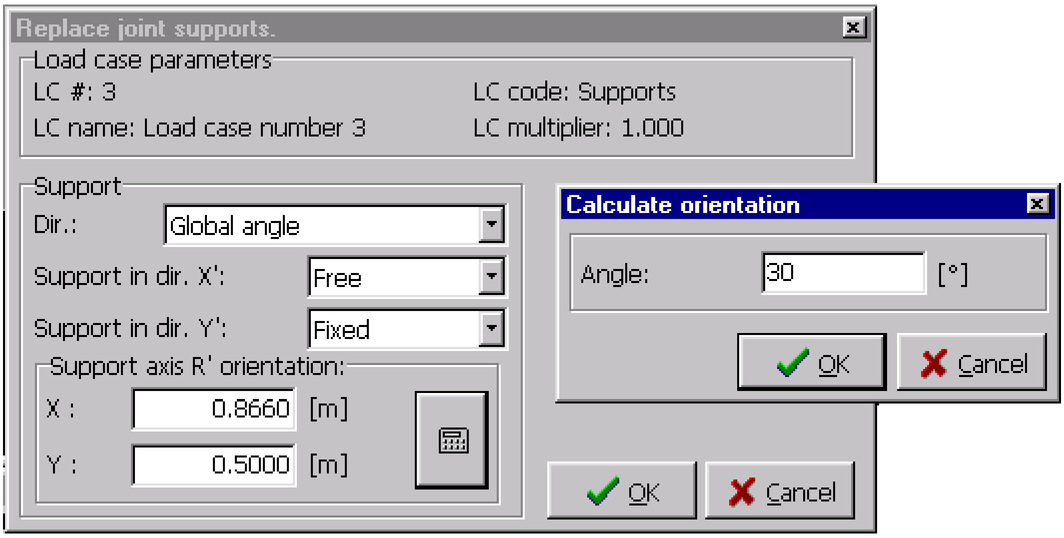


Fig. 4-76 Replace support at global angle 30°.

An example of supports is shown in Fig. 4-77, where the left joint is supported in global X, Y directions and the joint on the right is supported in local direction Y', where the local axes X' is rotated by 30°.

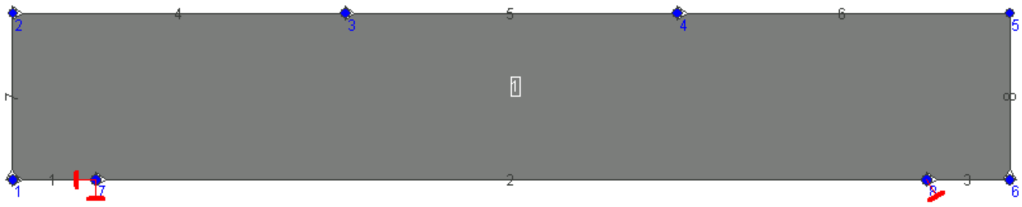


Fig. 4-77 Supports at joints, example.

4.6.6.3 Apply Support to Line

Line The object **Line** must be activated by clicking on it in the access tree section **Load cases**. The active item **Line** for support input is shown by green arrow. The numerical window changes to **Line support**, Fig. 4-78.

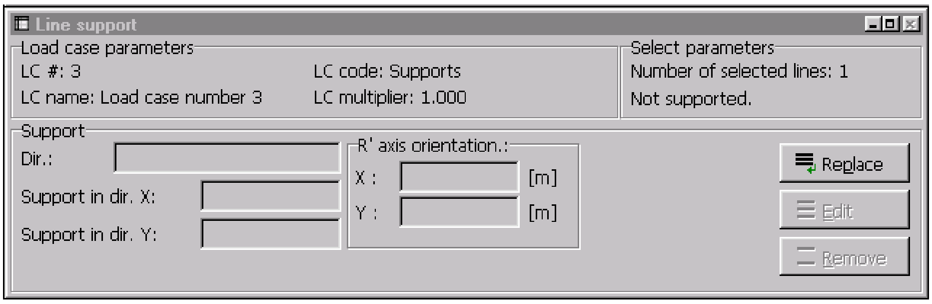


Fig. 4-78 Line support numerical window.

By clicking on the button Replace in the numerical window an input menu appears as shown in Fig. 4-79.

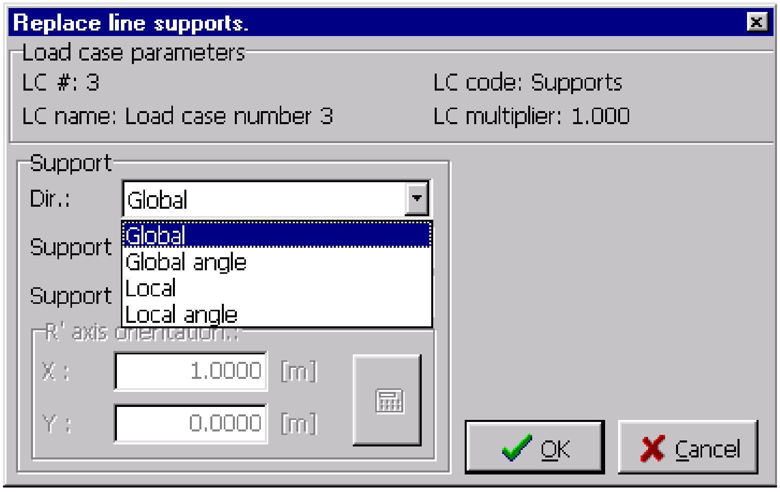


Fig. 4-79 Replace line support.

The input of supporting condition is the same as for joints. In addition the line offers a possibility to define the support direction in global as well as in local coordinate system. The X' axes of line local system is defined by the sequence of the line joints. Example of a line with vertical support is shown in Fig. 4-80 (bottom middle line). The structure will be provided with supports in all mesh nodes nodes within the line. However, before a mesh of nodes is generated only one support icon is shown in the middle of the line. After a mesh generation supports in all nodes will be indicated.

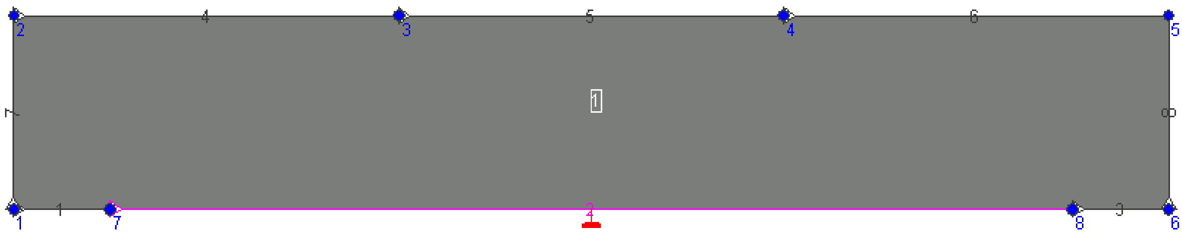


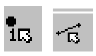
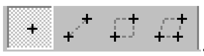
Fig. 4-80 Line support example.

4.6.7 *Prescribed Deformations*

Prescribed displacements can be applied to joints and lines. Load case code **Prescribed deformations** must be set active. Input of prescribed deformations supports is done in two steps: (1) Selects joints or lines; (2) Apply displacements to selected joints or lines.

The process is similar to the input of supports and is extended by the input of prescribed displacements.

4.6.7.1 Select Joints or Lines

The objects **Joint** or **Line** must be selected by clicking on the buttons  in the select menu bar. Then a selection method is chosen by clicking on one of selection method buttons . The selected objects are marked by green color.

4.6.7.2 Apply Displacements to Joint

Joint The object **Joint** must be active and marked by green arrow in the section Load cases of the access tree. After this the numerical window changes to a form shown in Fig. 4-81. By clicking on the button Replace the input window for prescribed displacement in support is opened, Fig. 4-82.

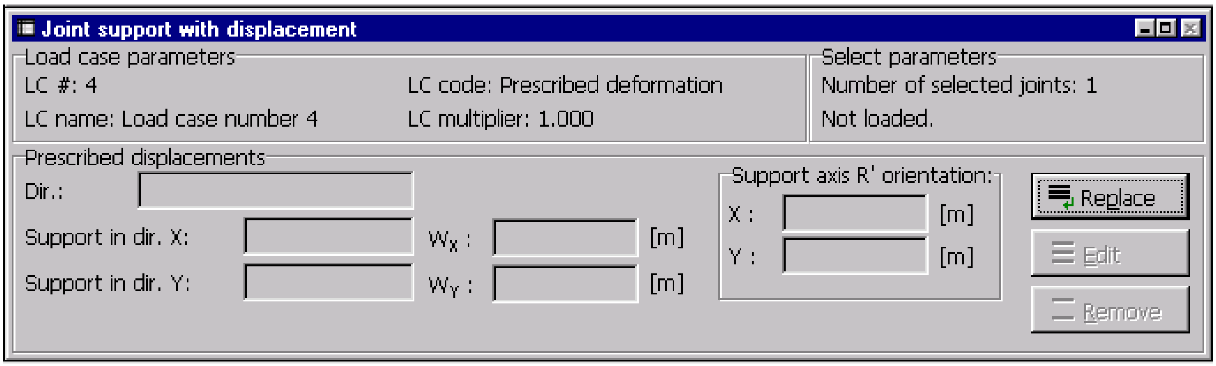


Fig. 4-81 Joint support with displacement window.

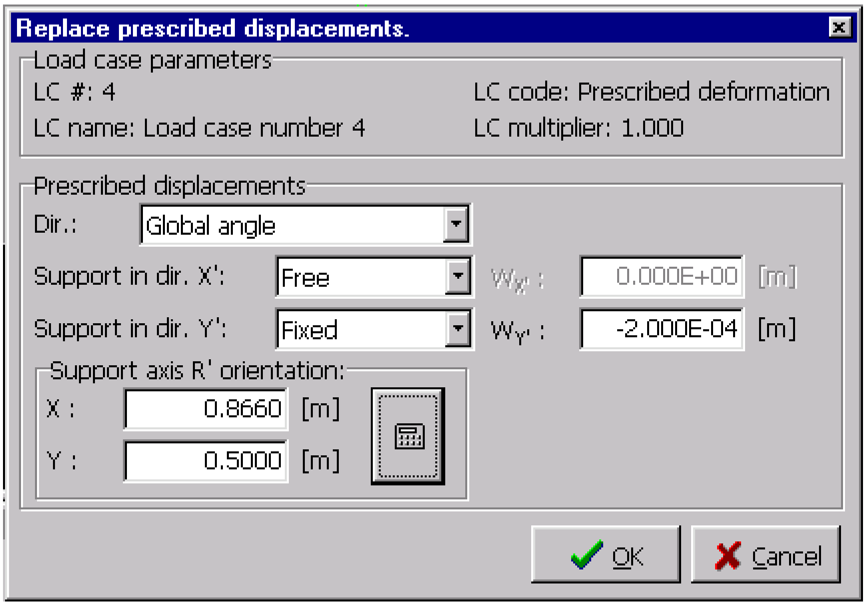


Fig. 4-82 Replace displacement at joint.

One or two supports in a node are defined by choosing **Dir.** (direction) and the components **Support in dir. X'** (**Y'**). Then the prescribed displacements $\mathbf{W}_{X'}$, $\mathbf{W}_{Y'}$ can be entered in the field right from the supported component.

An example of displacement prescribed to one of the joints in a structure is shown in Fig. 4-83. The joint is marked as support and labeled with a numerical value of prescribed displacement.

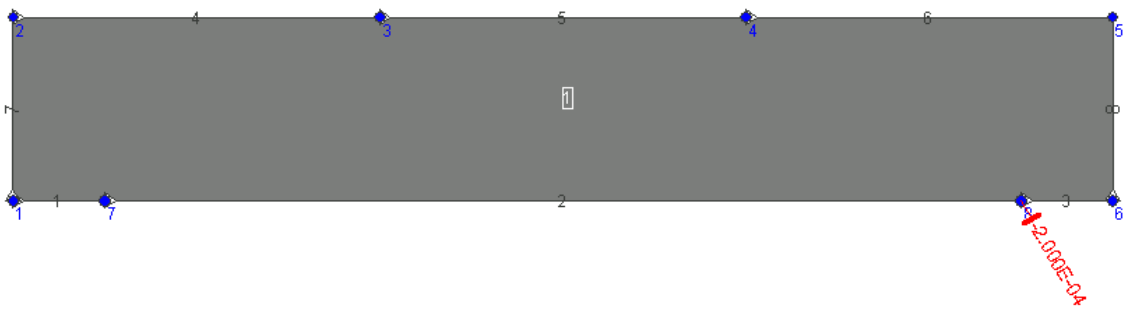


Fig. 4-83 Prescribed displacement at joint – example.

4.6.8 *Temperature*

Temperature can be applied to macorelements and to bar reinforcement. Load case code **Temperature** must be set active. Input of temperature is done in two steps: (1) Select macroelements; (2) Apply temperature to selected macroelements.

4.6.8.1 Temperature in Macroelements

Macro-elements The object **Macroelement** must be active and marked by a green arrow in the section **Load cases** of the access tree. After this the numerical window changes to a form shown in Fig. 4-84.

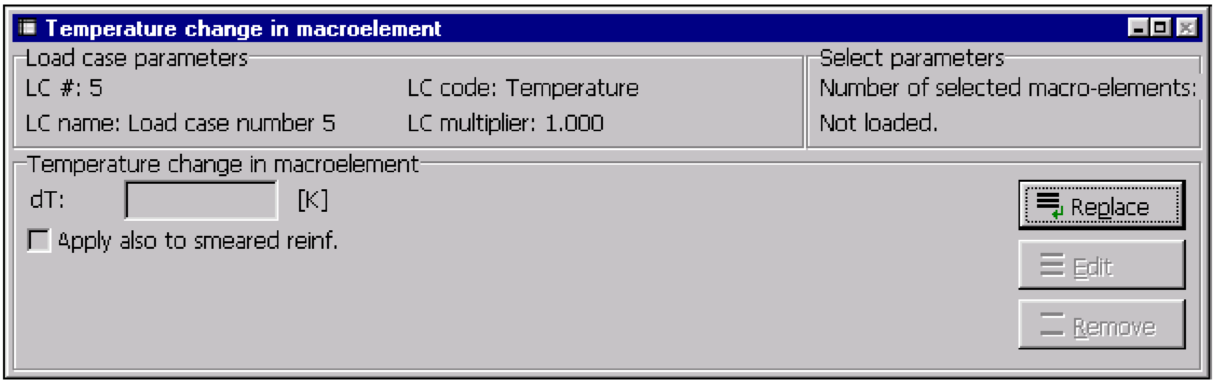


Fig. 4-84 Temperature input numerical window.

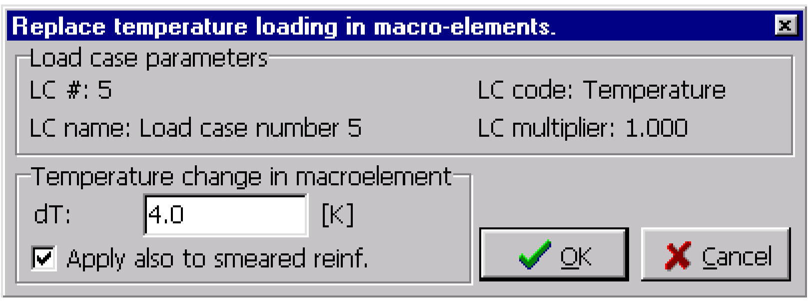


Fig. 4-85 Replace temperature in macroelement.

By clicking on the button Replace the input window for temperature is opened, Fig. 4-85. Temperature increment **dT** in the macroelement can be enetered. This is a change of temperature with respect to the previous load step. The temperature applies also to the smeared reinforcement if the box in the corner (left, bottom) is checked.

Macroelements with temperature loading are marked by hatch pattern as shown in Fig. 4-86. The temperature value is written within the macroelement.

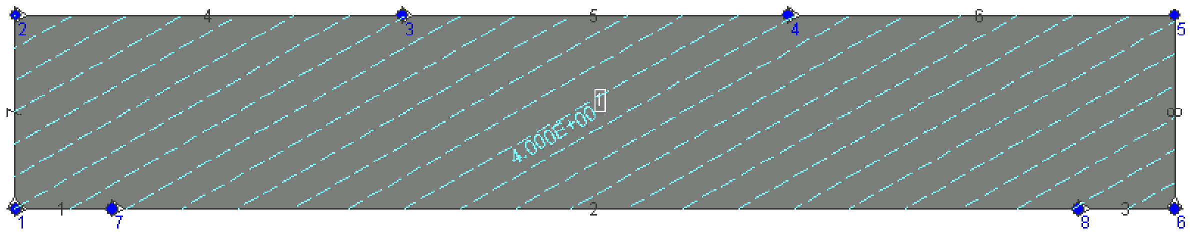


Fig. 4-86 Temperature in macroelement – example.

4.6.8.2 Temperature in Bar Reinforcement

Bar reinforcement The object **Bar reinforcement** must be active and marked by the green arrow in the section **Load cases** of the access tree. After this the numerical window changes to a form shown in Fig. 4-87.

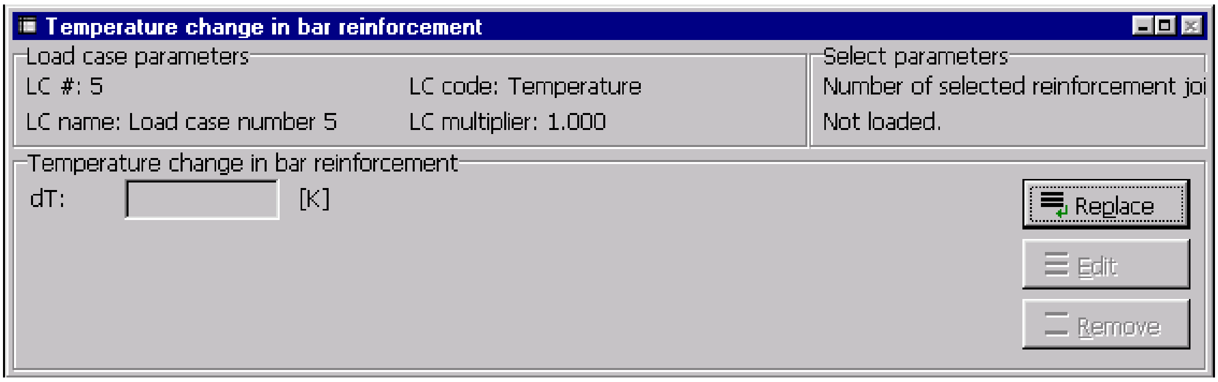


Fig. 4-87 Temperature in bar reinforcement input window.

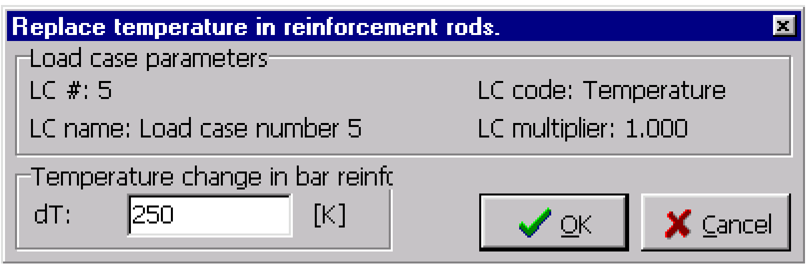


Fig. 4-88 Replace temperature in bar reinforcement.

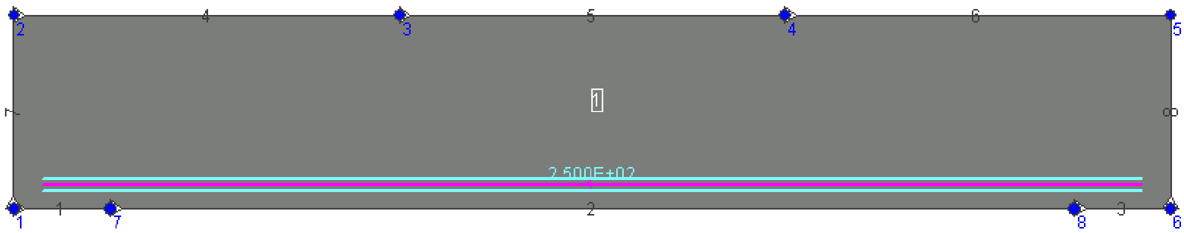


Fig. 4-89 Temperature in bar reinforcement.

By clicking on the button  the input window for temperature in bar is opened, Fig. 4-88. In the field **dT** temperature in reinforcement should be entered. *This temperature will be applied to the bar only and not to the macroelement material.* The bar with temperature loading is marked by two light blue lines and the the temperature value is written in the middle of bar.

4.6.9 Shrinkage

Shrinkage can be applied to macorelements. Load case code **Shrinkage** must be set active. The input of shrinkage is done in two steps: (1) Select macroelements; (2) Apply shrinkage. The process is almost identical with the temperatue input.

 The object **Macroelement** must be active and marked by green arrow in the section 'Load cases' of the access tree. After this the numerical window changes to a form shown in Fig. 4-90.

By clicking on the button  the input window for shrinkage is opened, Fig. 4-91. Shrinkage (volume) strain should be entered in the field **Esp**. This volume strain will be applied to the macroelement material. The shrinkage value is written in the middle of bar.

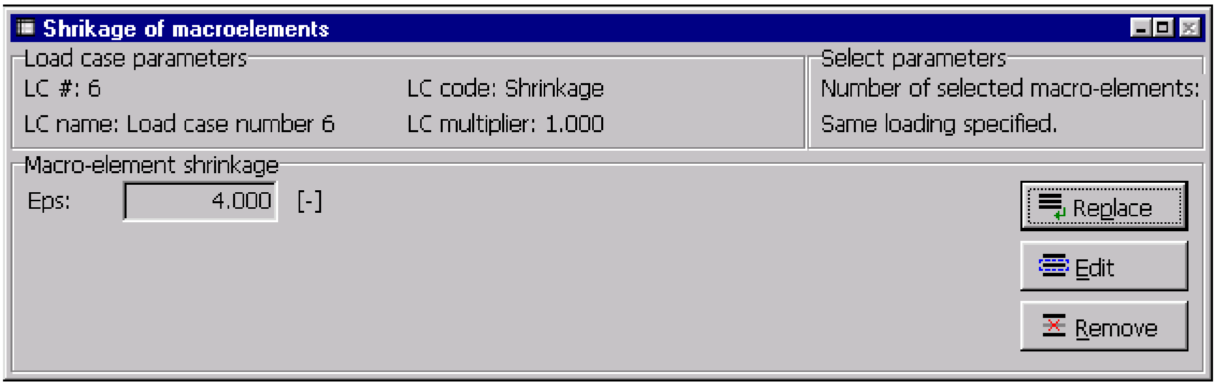


Fig. 4-90 Shrinkage numerical input window.

Macroelements with shrinkage loading are marked by hatch pattern as shown in Fig. 4-92, which is different then the one due to temperature.

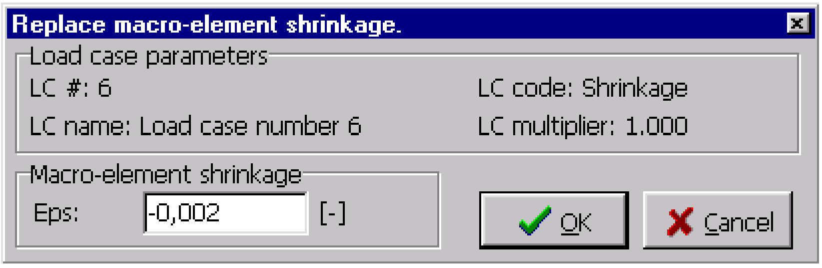


Fig. 4-91 Replace shrinkage in macroelement.

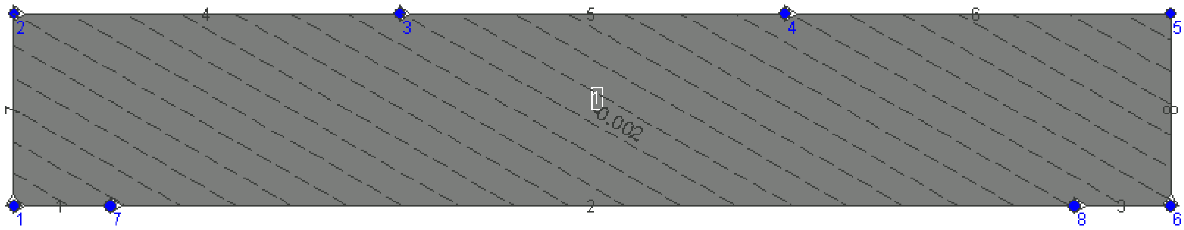
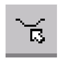


Fig. 4-92 Shrinkage in macroelement – example.

4.6.10 Pre-stressing

Pre-stressing can be applied only to bar reinforcement. The load case code **Pre-stressing** must be active. Input of pre-stressing is done in two steps: (1) Select reinforcing bars; (2) Apply pre-stressing.

4.6.10.1 Select Bar Reinforcement

The **Bar reinforcement** selection must be activated by clicking on the buttons  in the select menu bar. Then a selection method is chosen by clicking on one of the selection method buttons . The selected objects are marked by green color.

4.6.10.2 Apply Pre-stressing

The object **Bar reinforcement** must be active and marked by green arrow in the section **Load cases** of the access tree. After this the numerical window changes to a form shown in Fig. 4-93.

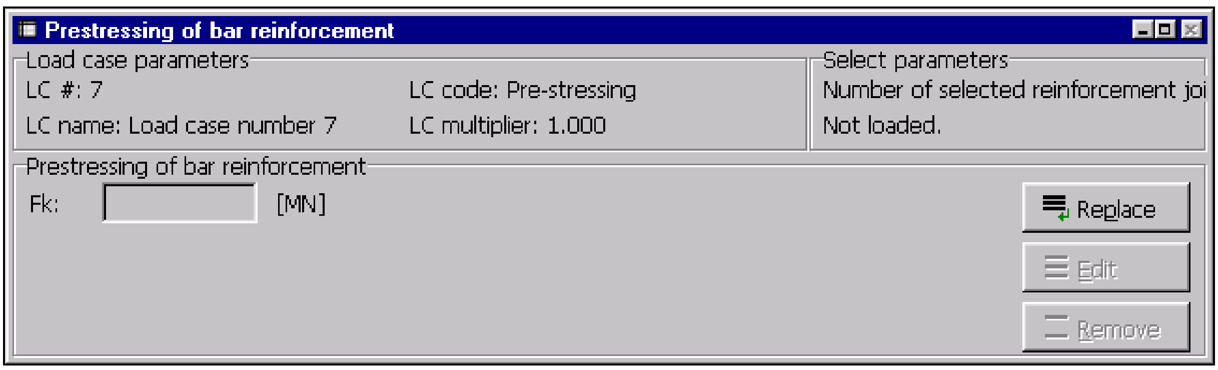


Fig. 4-93 Prestressing of bar reinforcement numerical input window.

By clicking on the button Replace the input window for prestressing of bar is opened, Fig. 4-94. Prestressing force should be entered in the field **Fk**. The prestressing force will be introduced according to the reinforcement type.

In *normal reinforcement* (with full bond) prestressing is applied by the method of pre-tensioning, where it is introduced as the initial strain. In *external reinforcement* (unbonded) it is applied as a force the active anchor. the value of pre-stressing force is shown at the bar end. In case of external cable it is at the active anchor.

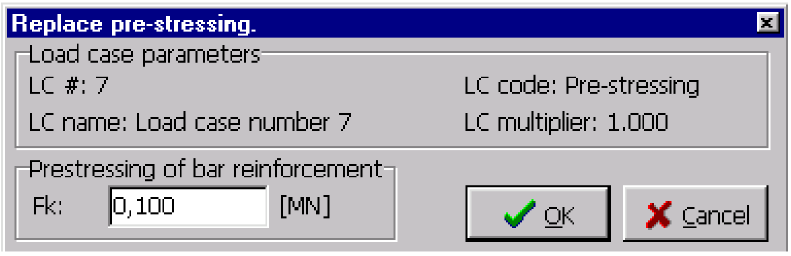


Fig. 4-94 Replace pre-stressing of a bar.

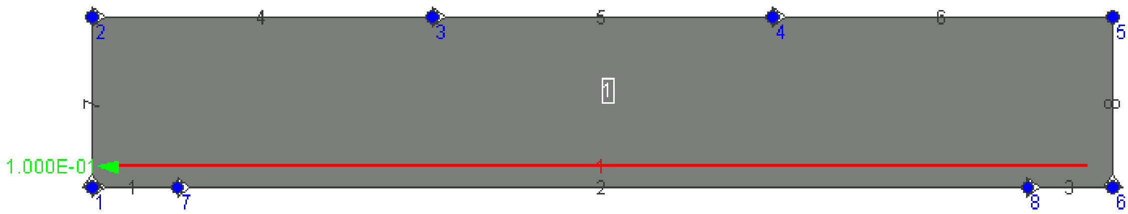


Fig. 4-95 Pre-stressing of a bar, example.

4.6.11 Contact Ambiquity of Loading

Several types of loading can be assigned to one of the double lines on a macroelement border line. As first a macroelement must be constructed and load must be entered. The load type on the double line can be: *force, line load, support and line support*. The line connection type must be either **No connection** or **Interface**. A concerning load case must be set active. For example, for line loading or force at a joint it must be the load case code **Force**. Example of such load is shown in

Fig. 4-96. Then a contact ambiquity can be resolved as follows.

Contact ambiguity The object **Contact ambiquity** within the load cases partition of the access tree must be activated for input and marked by a green arrow. This opens the access to the tools for connecting lines. All potential lines within the chosen load case and requiring adjustment of contact ambiquity will be listed in the numerical window as shown in Fig. 4-97.

Line must be selected in the list and after clicking on the button Edit a window for contact ambiquity appears as shown in Fig. 4-98. One of the two possible macroelements can be chosen and the choice must be confirmed by OK. The macroelement number assigned to the loading appears in the data table in the column **Assigned macroelement**.

A similar procedure is applied to the supports located on double lines and will not be repeated here.

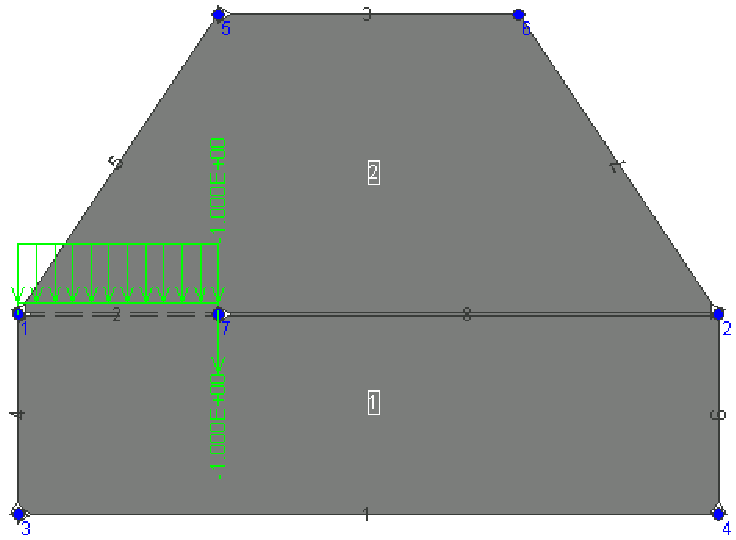


Fig. 4-96 Load on macro-element border line, example.

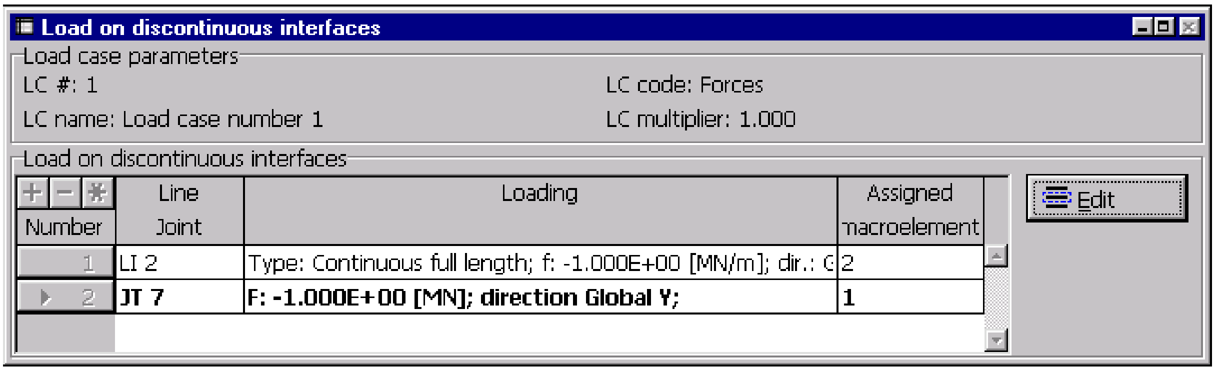


Fig. 4-97 Loads on a discontinuous line.

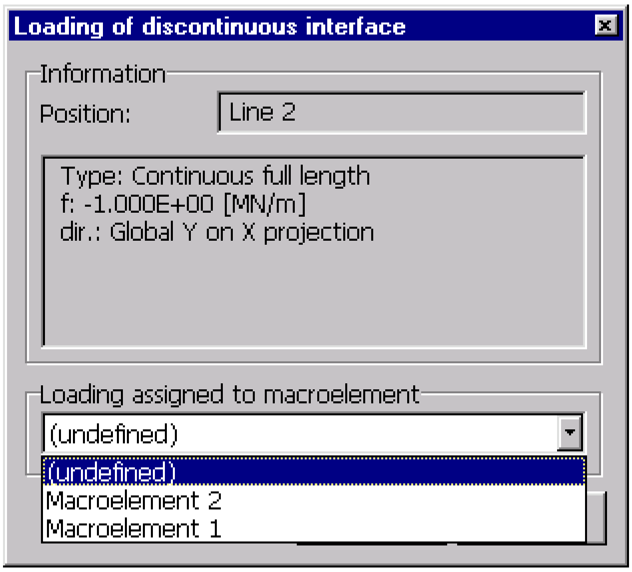


Fig. 4-98 Load assignment to a macroelement.

4.7 Run

This section describes the input of data needed to run analysis and some additional data for post-processing, which are out of scope of topology and loading. The input of this data group can be accessed from the access tree partition **Run**, Fig. 4-99.

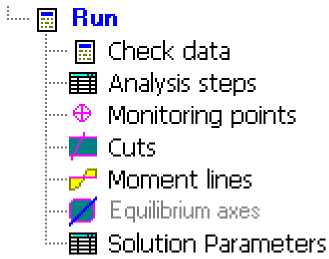


Fig. 4-99 Run access tree.

4.7.1 Check Data

This option performs a data check. Errors and inconsistencies in data are reported in the list of the numerical window.

4.7.2 Analysis Steps

The load history is generated by this process.

Input of analysis steps can be started by clicking on the **Analysis steps** icon in the access tree. An active input of analysis steps is marked by a green arrow. After this the numerical window changes to a form shown in Fig. 4-100.

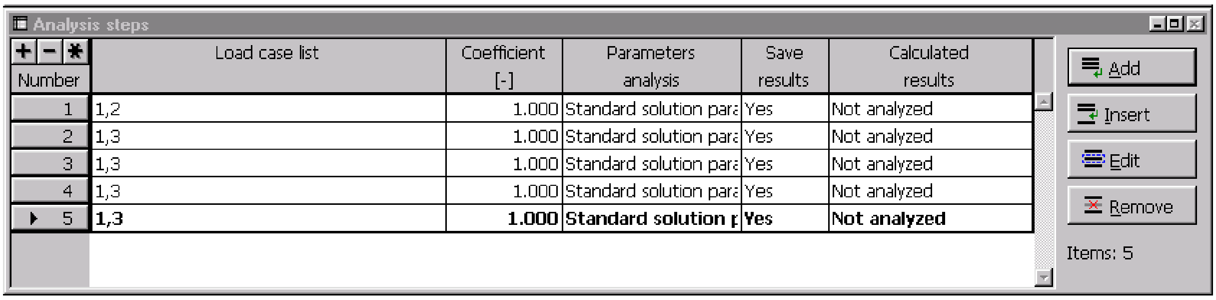


Fig. 4-100 Analysis step input window.

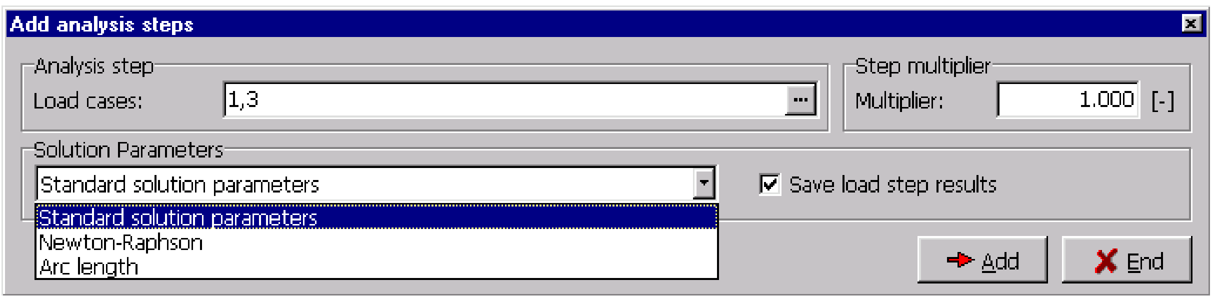


Fig. 4-101 Add analysis step.

Data for a single step can be generated by clicking on the button Add in the numerical window. After this an input window for analysis step opens as shown in Fig. 4-101. The load case numbers combined in this analysis step are entered in the field **Load cases**. **Multiplier** is used as a multiplying factor for all numerical loading data in this load step. The field **Solution Parameters**, when unfolded by clicking on the black arrow ▾, displays a list of already defined solution parameter sets. One of these sets should be chosen. The box **Save results** should be checked in order to ensure data saving from this step.

The parameters in this menu can influence analysis time and data volume. For example, some solution methods are faster then others and saving data in all steps takes some time and a large volume of storage.

4.7.3 Monitoring Points

In monitoring points a development of interesting data can be recorded and displayed.

Monitoring points Input of monitoring points must be started by clicking on the icon **Monitoring points** in the access tree. An active input of monitoring points is marked by a green arrow. After this the numerical window changes to a form shown in Fig. 4-102. Data for a monitoring point can be generated by clicking on the button Add. After this an input window for new monitoring point opens as shown in Fig. 4-103.

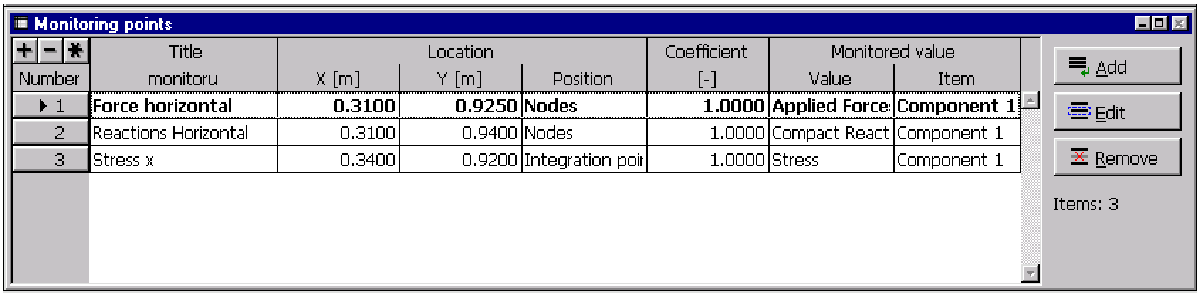


Fig. 4-102 Monitoring point input window.

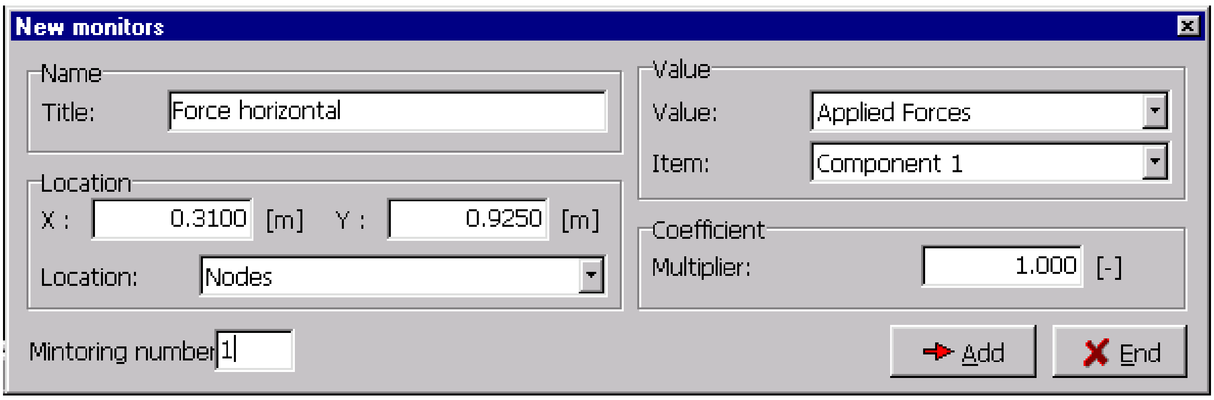


Fig. 4-103 New monitoring point.

In the field **Location** the coordinates of monitoring point should be entered and monitoring location either in **Nodes** or **Integration points** can be selected.

Value. In nodes following options are available: **Displacements**, **Applied forces**, **Reactions**. In integration points the available options are: **Stress**, **Strains**.

Item is a component of the value. Notation of stress and strain components: 1 - *x*, 2 – *y*, 3 - *xy*. Example of monitoring points is shown in Fig. 4-104. In this example the points listed in Fig. 4-102 are shown. The points 1 and 2 are in the node and point 3 is in the (middle) integration point. The position of a monitoring point need not to be given exactly. The point is related to the nearest node or nearest integration point, respectively. The relation of monitoring points to nodes and integration points is shown graphically by a violet dashed line.

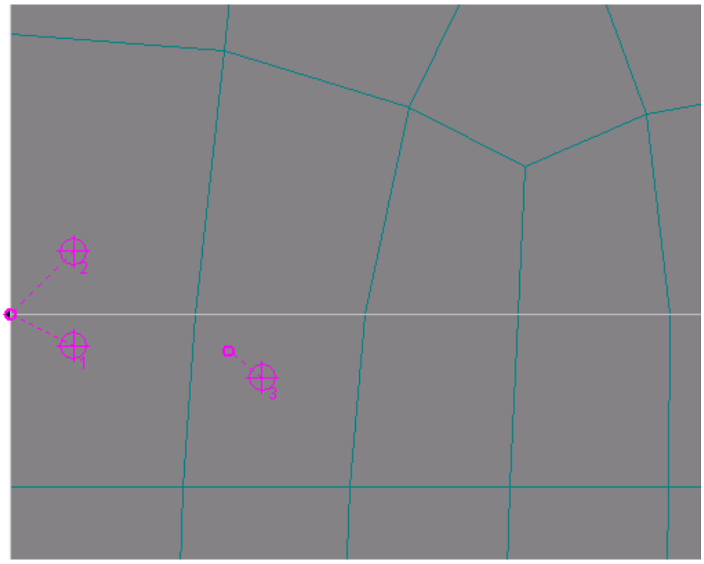


Fig. 4-104 Monitoring points – example. Relation to node and integration point.

The monitoring point can be also entered by mouse. This is done by clicking on the button in the toll bar for the mouse input and then a followed by a click on the desired location of the monitoring point. A monitoring point entered by mouse can be edited using the numerical window.

4.7.4 Cuts

The cuts are sections, which serve in the post-processing for evaluation of state variables (stresses, strains, etc.). The cut path must be entered by user. Failing to define cuts causes that no internal variables distribution along cuts can be evaluated in the post-processing.

Cuts Input of cuts is activated by clicking on this item in the access tree. Then numerical window changes to the form as shows in Fig. 4-105. Cuts entered either by mouse or numerically are listed in the table. Each item (line) in the table contains one cut. A list of segments is shown in the column **Geometry** and the last column shows the number of segments (intervals) in the cut.

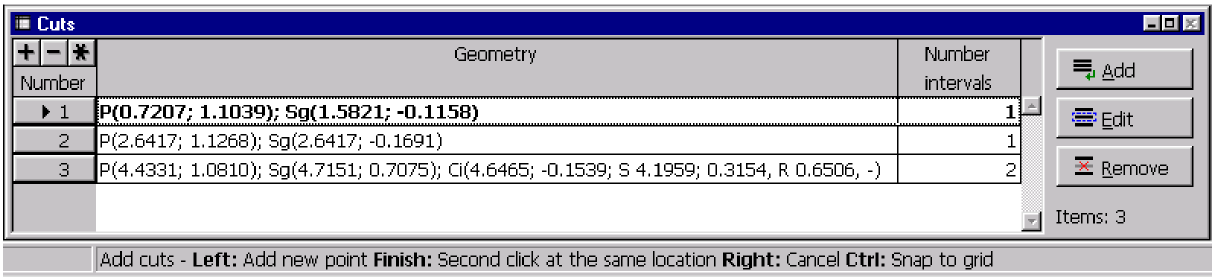
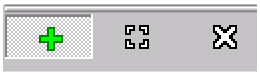


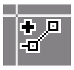
Fig. 4-105 Numerical window for cut input.

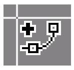
Cut is composed of a chain of segments. the segment can have a form of line or arc. In the post-processing the state variables are drawn along cuts in form of a diagram as will be shown later in chapters on post-processing.

The geometry of a cut can be entered either by mouse pointer or by numerical values.

4.7.4.1 Cut Input by Mouse

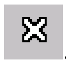
 An input by mouse is activated by clicking on the add button , providing that the **Cut** item is active. The input is ready to generate line segments.

This is indicated by the following screen pointer: 

The segment geometry can be changed to a circle by holding down the key Shift and is indicated by the following screen pointer: 

The end point of segment is generated by pressing the left mouse button at its location. The double click on a point ends the cut input. Pressing the right mouse button cancels the cut input.

The geometry of circle segments is defined under the assumption of equal tangents of the segments at their joints. Therefore, there is a smooth transition from line to circle and from circle to circle. (But, not from circle to line.)

Cut geometry can be edited or deleted using the mouse tools  and .

4.7.4.2 Cut Numerical Input

in the Cut numerical window. This initiates the input dialog as shown in Fig. 4-106.

The coordinates of segment points are entered via the **Add segment** window, where **Line** or **Arc** type of geometry can be chosen and a point or an arc coordinates can be entered. For better control of numerical input the window shows in its bottom part a geometrical form of the generated cut.

Using the button 

a new segment can be inserted into an exiting cut.

The buttons 

, 

serve for editing and deleting of segments.

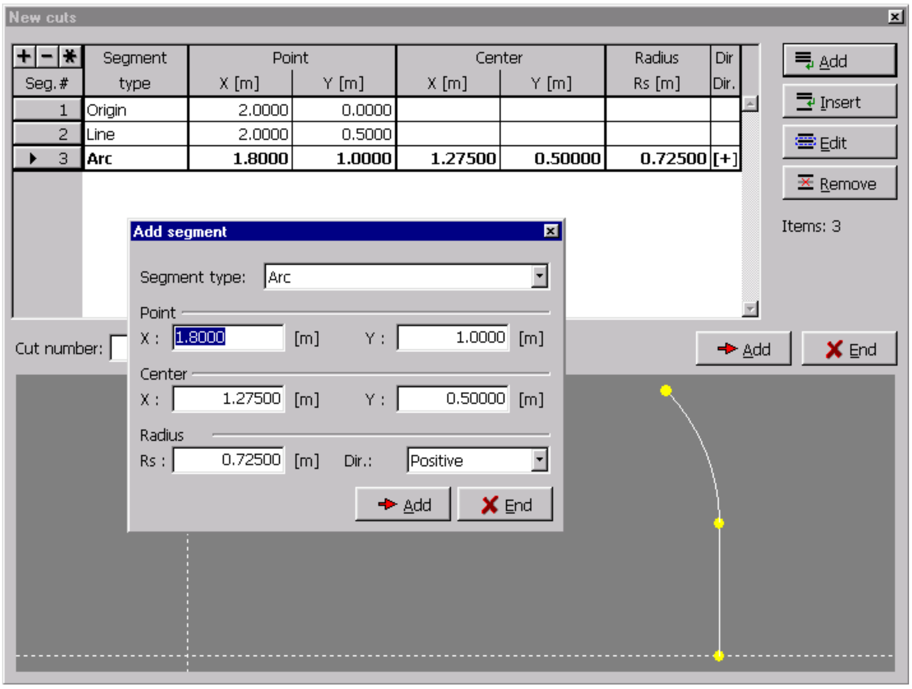


Fig. 4-106 New cut input dialog.

Example of three cuts as they appear in the input screen is shown in Fig. 4-107. The two cuts are made of one-line segments and the third cut is made of two segments, line and arc. As shown, the cuts may, but need not to coincide exactly with the model border.

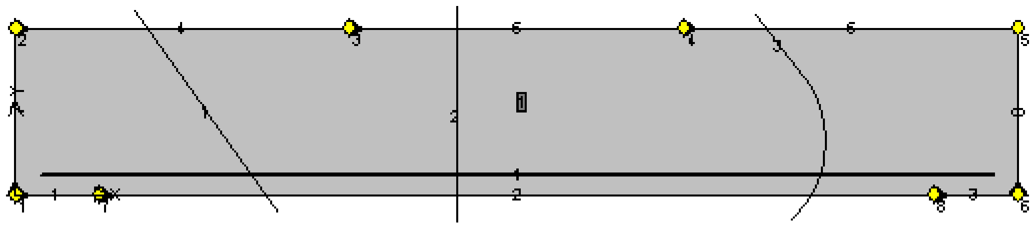


Fig. 4-107 Example of cuts in a model of beam.

4.7.5 Moment Lines

This feature enables to define the lines for evaluation of internal forces, which will be performed in the post-processing. Such lines can be center lines or any other lines and must be entered by user. Failing to define moment lines causes that no internal forces can be evaluated in the post-processing.

The moment lines are formally constructed in a way identical with cuts. Therefore, the input of moment lines shall not be repeated here while the same procedure as in cuts is used. The only additional input parameter, which appears in moment lines is the number of intervals in line. Three internal forces (moment, normal and shear forces) are calculated at each interval point along the moment line. Although three internal force are calculated the line is called for brevity as ‘moment line’.

A note on arc-shaped moment lines

Please note that only those parts of a moment line that pass through the interior of a finite element can deliver results. This becomes especially important with arc-shaped moment lines near an arc-shaped macroelement boundary - the finite elements have straight boundaries, while the moment line is an arc (resp. has finer division than the FE mesh). If the moment line is close to an outside boundary, it may happen that some parts of the moment line fall outside of the elements and therefore there can be no results.

4.7.6 Solution Methods

A solution method includes the techniques and parameters for the iterative nonlinear solution of load steps. It is associated with the solution of equilibrium equations.

Solution Parameters Input of solution parameters is activated by clicking on this item in the access tree. An active item is marked by green arrow and the numerical window changes to solution parameters. A default is a set of **Standard solution parameters**. This set can not be changed and can be used as a most reliable set.

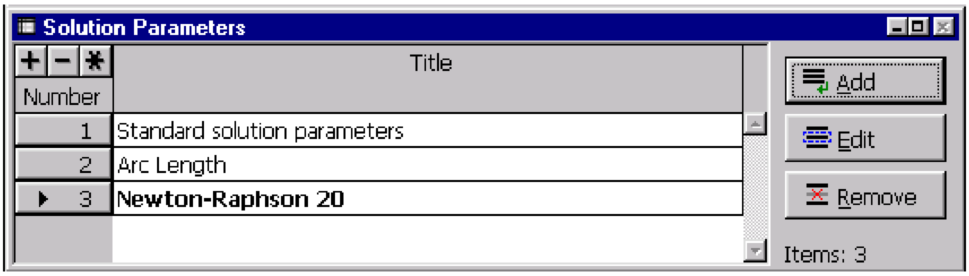


Fig. 4-108 Solution parameter sets.

By clicking on the button Add a dialog window for input of parameters opens, Fig. 4-109. Three basic methods can be chosen. **Newton-Raphson** and **Arc Length** are two basic iterative schemes for elimination of unbalanced forces and restoring equilibrium state. For force loading up to near peak load or in postpeak, the **Arc Length** method has to be used (please note it changes both displacements and forces). The **Newton-Raphson** method is recommended in all other cases.

Line search can be used in combination with both of them to accelerate a convergence rate.

Title of the set describes the methods and parameters used as shown in Fig. 4-109. Any text can be used.

Solution method can be chosen form the list of two: **Newton-Raphson**, **Arc Length**. The Newton-Raphson method is chosen in the example on Fig. 4-109.

Optimize node numbers. Two methods of numbering are available: **Sloan**, **Gibs-Pole**.

Update Stiffness. Two update methods are available: **Each iteration**, **Each step**.

Stiffness Type. Two stiffness types are available: **Tangent** and **Elastic** stiffness.

Iteration number limit. Iteration stops when this limit is reached and load step is finished regardless of other criteria.

Four **error tolerances** are the limits for various criteria. Iteration stops when all criteria are satisfied.

If **Arc Length** solution method is chosen the mask of the window changes to the form shown in Fig. 4-110. Arc Length method has its own set of parameters, which can be accessed by clicking on its tab and a dialog box shown in Fig. 4-111 will appear. The **Line Search** tab is employed if the box Line search is checked. Line search parameters can be accessed by clicking on its tab and a dialog window such as shown in Fig. 4-112 will be opened.

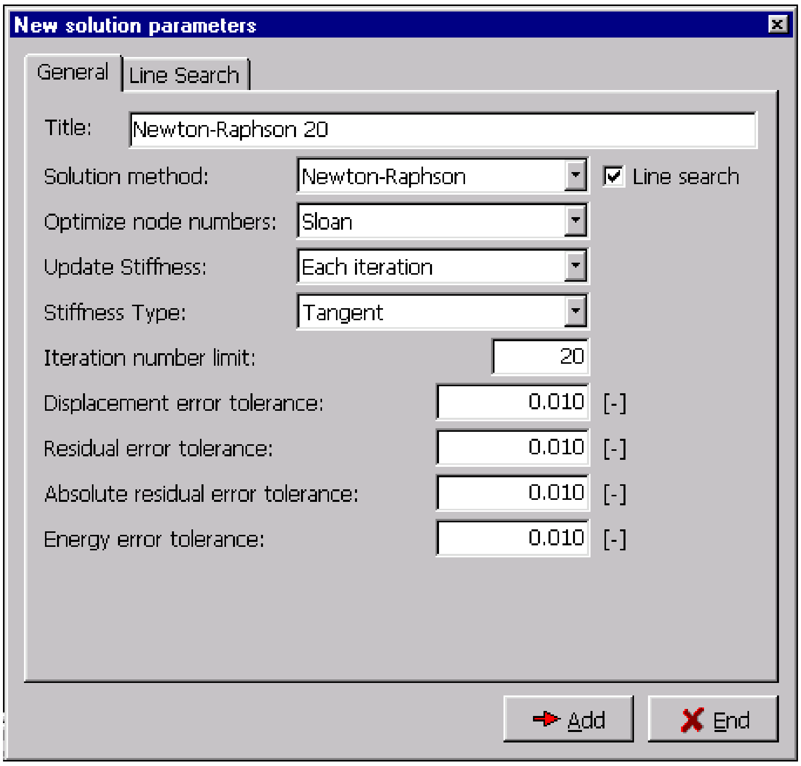


Fig. 4-109 New solution parameters. Newton-Raphson method.

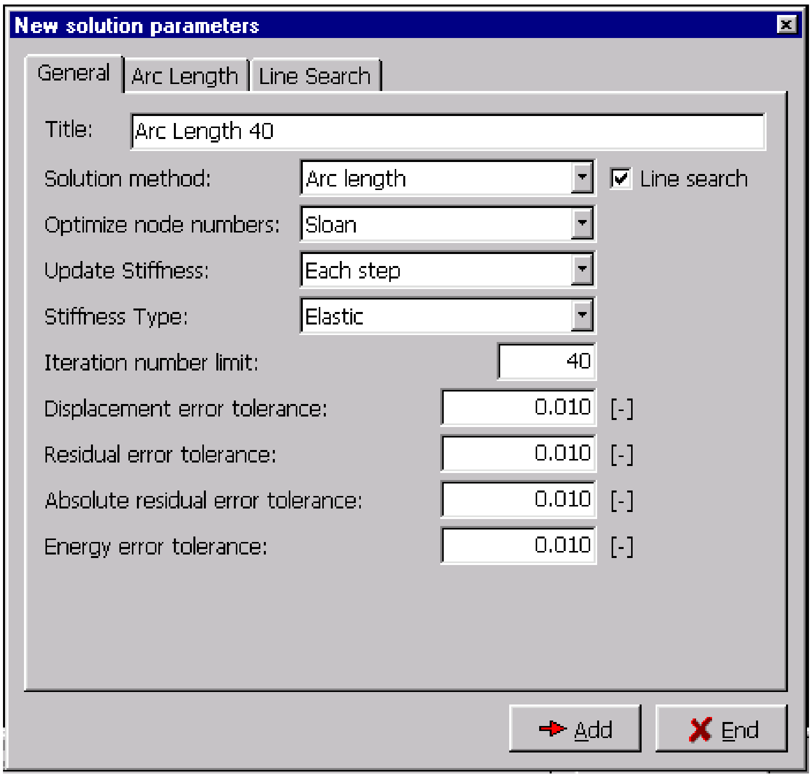


Fig. 4-110 New solution parameters for Arc Length method.

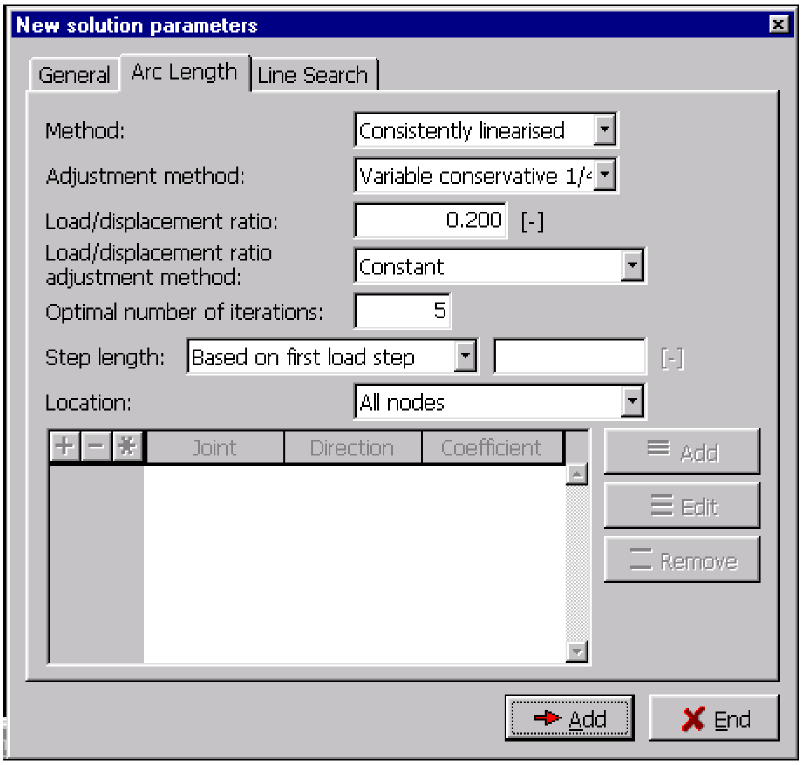


Fig. 4-111 Arc length parameters.

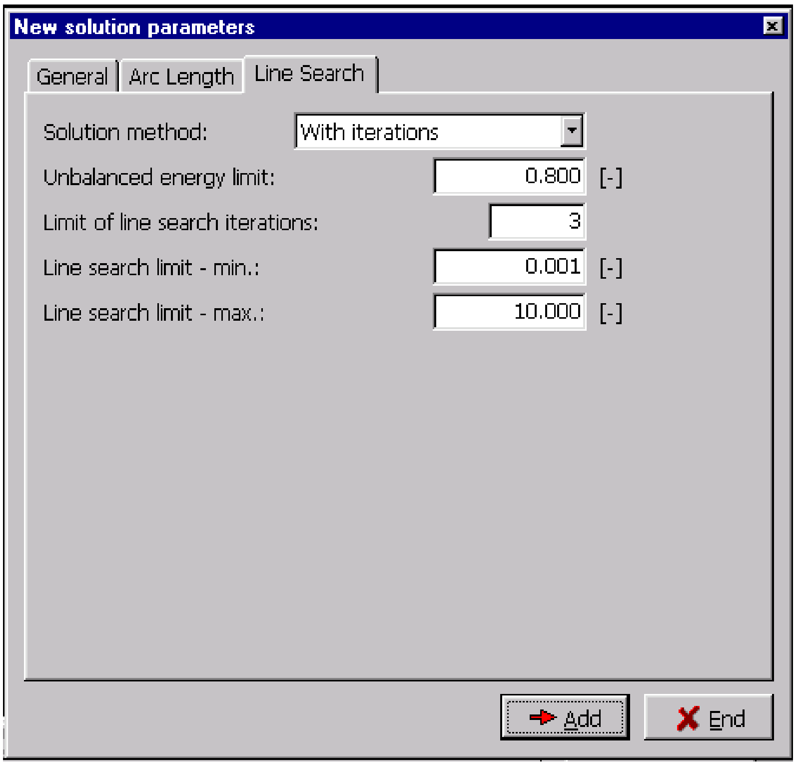


Fig. 4-112 Line search parameters.

A description of the theoretical background of the solution methods and their parameters is included in the documentation volume ATENA Theory. It is strongly recommended not to change default setting of solution methods, unless its meaning is fully understood.

Notes to solution methods:

Newton-Raphson method keeps the load increment unchanged and iterates displacements until equilibrium is satisfied within the given tolerance. This means that this method should be used in cases when load values must be exactly met. It should be used in case of following types of loading: body forces, temperature, shrinkage, pre-stressing. However, it should not be used near local and global peaks, followed by load decrease (descending branch) in case of force loading. In other words, it should not be used for analysis of ultimate load, when force loading is prescribed.

Arc length method keeps the solution path constant (in the load-displacement space) and iterates both increments of displacements and forces. Therefore, it changes both, displacements and forces. It is more general then Newton-Raphson method, however, it is not useful for some type of problems, such as those with exactly define intensities (body forces, pre-stressing, shrinkage, temperature). For example in case of body force it would change the self weight.

Notes to line search parameters:

1. The Line Search iterations are performed within the line search method only. They are iterations within one iteration cycle (Newton-Raphson or Arc Length). It is not necessary to use more than 5 iterations.
2. The Line Search method can accelerate or slow down Newton-Raphson or Arc Length predictor-corrector solution. The **max. limit** of line search iterations is for the acceleration and **min. limit** is for the slow down. In case of non-convergent performance it may be

useful to increase the min. limit and to decrease the max. limit. For example, a set of limits (min.=0.1, max.= 2.0) can be chosen to reduce the line search effect.

Notes to update stiffness and stiffness type:

1. The **stiffness update in each iteration** causes that the structural stiffness matrix is calculated and assembled in each iteration. It is reasonable to use this update only in connection with the **tangent stiffness**, because it may change in each iteration.
2. The **stiffness update in each step** causes that the stiffness matrix is calculated and assembled only in the initial iteration of the step and stays unchanged during the rest of iterations. This update should be used in combination with the **elastic stiffness**.

   General comment:

   It is difficult to make a general recommendation for a solution method. Nevertheless, at least some hints can be made. In stable and convergent situations the tangent stiffness with update in each iterations can be used. In unstable situations the elastic stiffness with update in load step can be recommended. In this case the iteration limit should be set high (for example 100).

For both **Newton-Raphson** and **Arc Length** methods, **conditional break criteria** can be set to stop the computation if an error exceeds the prescribed tolerance (as set in Fig. 4-109 or Fig. 4-110) multiplied by the prescribed factor during the iterations or at the end of an analysis step. Example settings are shown in Fig. 4-113.

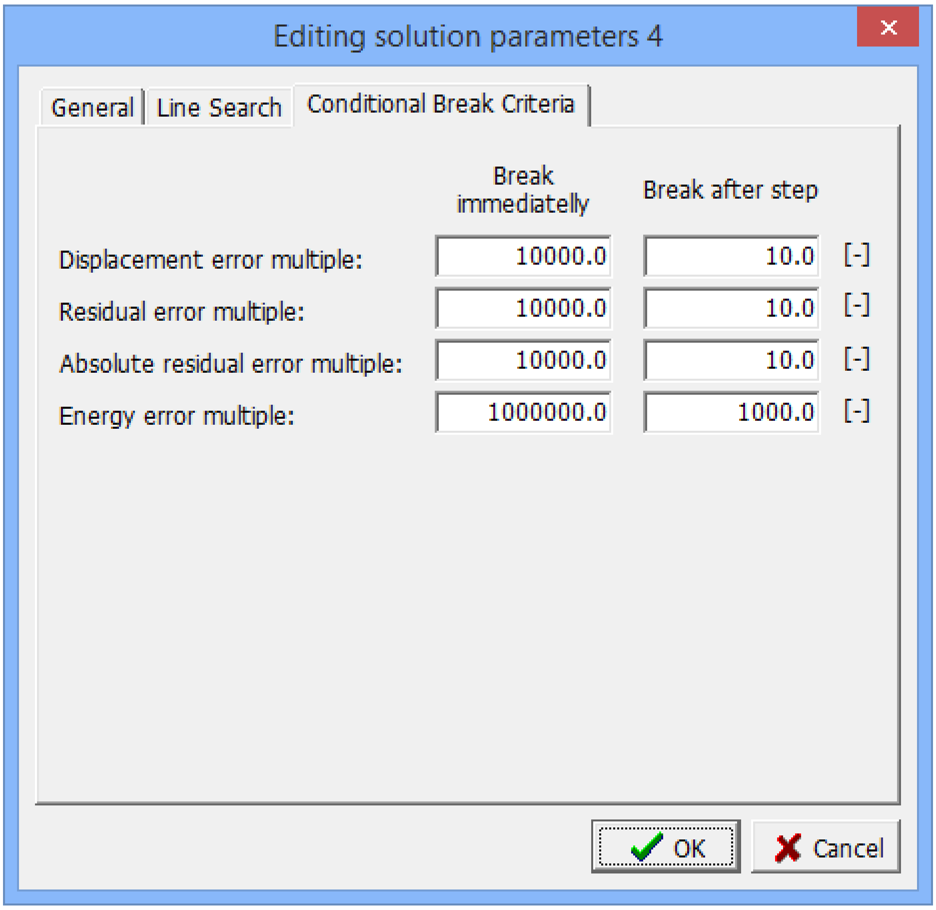


Fig. 4-113 Conditional break criteria.

5 CALCULATIONS

5.1 General Description

Menu **Calculations** provides access to all main modes of **ATENA** operation. The menu,

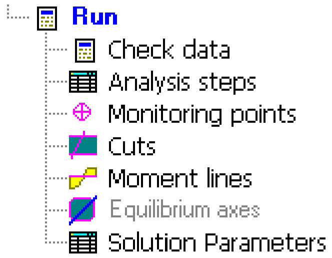


Fig. 4-99 is divided in three parts: Part 1 – mesh generation and finite element analysis, Part 2 – pre- and post-processing, Part 3 – Analysis progress information. All items (except the last one) can be alternatively accessed directly form the menu bar.

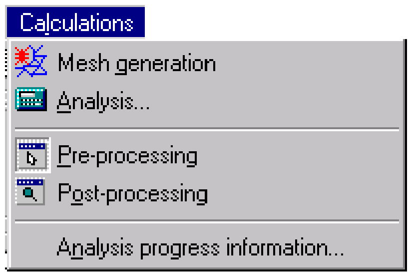
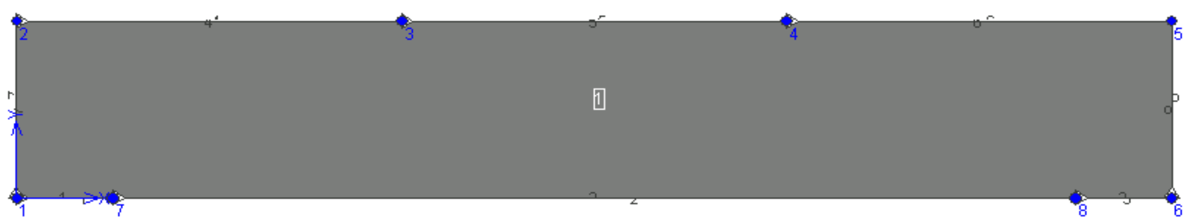


Fig. 5-1 Menu Calculations.

5.2 Mesh Generation

This item is open to access only in a pre-processing mode. It generates the finite element mesh based on input data entered by the input of **Topology**. Mesh is constructed in two phases. In the first phase a mesh is generated in macroelements one by one, in the sequence of their numbering. In the second phase the smoothing is performed to improve the mesh form. The smoothing reduces sharp mesh angles. The mesh is always generated in the whole structure. Thus, before mesh generation all data for mesh must be entered. In case of incomplete data, or errors, a message shall be displayed and the mesh generation is not performed. Example of a mesh generation is shown in Fig. 5-2. Mesh generator tries to make an optimal mesh for given geometry, element size and mesh refinement (if any).

Before mesh generation:



After mesh generation:

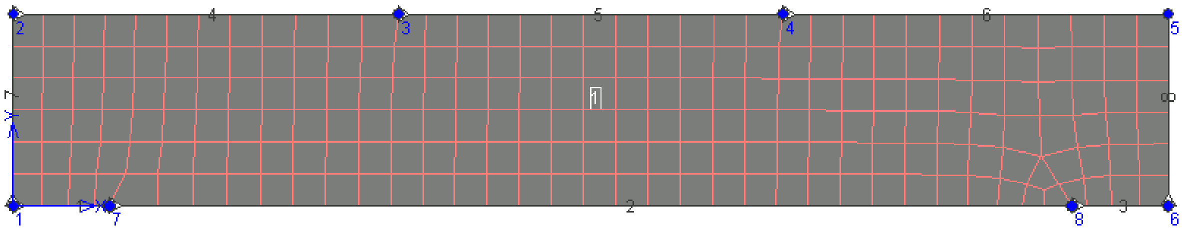


Fig. 5-2 Macroelement of a beam before and after mesh generation.

5.2.1 Notes on Meshing

The finite element mesh quality has a very important influence on the quality of the analysis results, the speed, and memory requirements. Refining only the important parts can save a lot of processor time and disk space.

A bad mesh, like a single layer of volume elements in a region where bending plays a significant role, can produce very wrong results – see the "Mesh Study" example in the **ATENA Engineering Example Manual**. A minimum of 4-6 elements per thickness is recommended for at least qualitative results in bending.

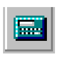
Another frequent example of a problematic mesh are elements with extreme aspect ratios, in other words, the ratio of element edge lengths = longest to shortest edge of an element. A maximum of 3:1-4:1 is recommended. The higher the aspect ratio, the worse the conditioning of the system matrix, which can lead to numerical problems in the solver.

5.3 Analysis

5.3.1 Setting Analysis Mode

Nonlinear finite element analysis can be started after completing input of all relevant data. It is essential, that all data are correctly entered and include: geometrical model, finite element mesh and analysis steps (load history). Analysis can be started (re-started) only from the pre-processing mode.

If some data are missing or inconsistent an error message appears and analysis does not start. However, **ATENA** makes only a formal check of data. It may happen that data are formally consistent, but not correct. In such a case the results can be effected and it is up to the user to check the correctness of input data.

Analysis mode is initiated by selecting the item **Analysis** from the menu, or by pressing the button  in the toolbar. After this the window shown in Fig. 5-3 appears where the parameters of analysis mode can be set.

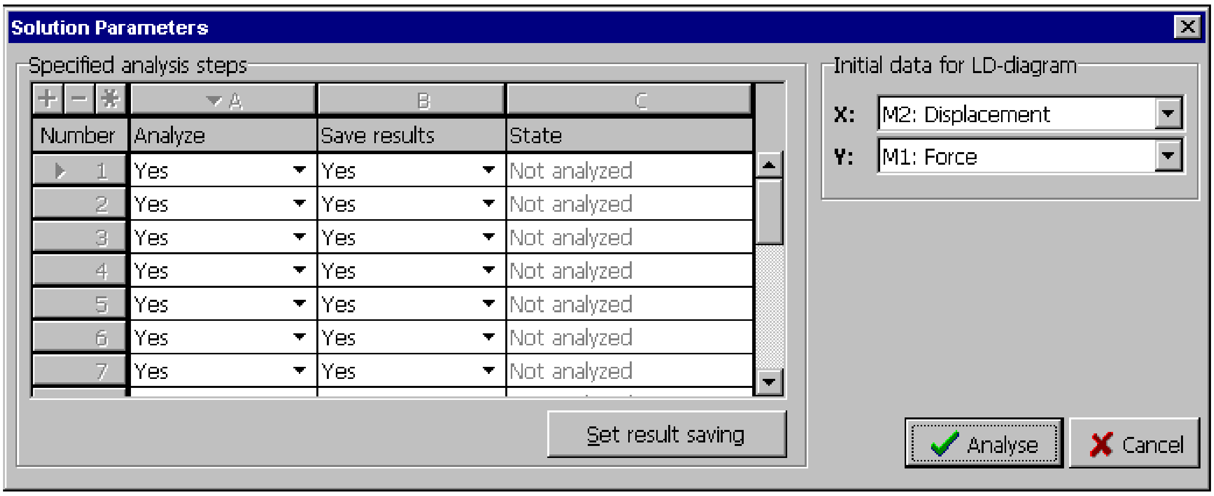


Fig. 5-3 Setting analysis mode.

5.3.2 Setting Response Diagram

Response diagram typically represents a load-displacement diagram and reflects the structural behavior. However, it can be any other relationship of two monitoring points.

Monitoring points for the response diagram can be selected from the menu located in the top right corner of the window. In the above example a displacement monitoring point is selected for the X-axis and a force is selected for the Y-axis. Although arbitrary number of monitoring points can be defined, the response diagram to be monitored during the analysis can show only one diagram. The monitoring points for the response diagram can be changed during analysis after user-interrupt.

In case no monitoring points are specified during the pre-processing no response diagram can be monitored during analysis, but the analysis can be performed.

5.3.3 Setting Load Steps for Analysis

Load steps to be analyzed can be set in the column **Analyze** of the table **Specified analysis steps**. The steps selected for analysis must form a continuous array starting from the step 1. This is required by consistency of analyzed data and has some consequences as will be explained below.

The load step can be in one of the two states: **not analyzed** or **analyzed** (before and after analysis, respectively). In the initial state the load steps are not analyzed and are all marked **Yes** for analysis. Now, if user changes one step to **No** (not to be analyzed), all steps with higher number shall be set to **No**. Similarly, if all steps are set **No** for not to be analyzed and one step is changed to **Yes**, then all preceding steps are set to **Yes**.

Example: The above procedure is typically used for a restart of analysis at some preceding already analyzed load step. The parameter of the load step from which it should be restarted is changed to **Yes**.

5.3.4 Saving Load Step Results

Saving of load step results is very important activity, with high impact of the analysis performance. Saving of load step results makes possible a post-processing. Therefore, all load steps where post-processing is foreseen must be saved. On the other side, saving considerably increases analysis time and data storage requirements. Saving of load step results was pre-set in the input of **Analysis steps**. This setting can be changed at this stage for each step individually, by modifying the parameter in the column **Save results**.

A group setting can be made by pressing the button 

where saving of each nth step can be defined.

5.3.5 Input File Editing

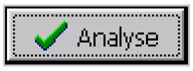
Input file is an interface between the Graphical User Interface and ATENA solution core. It is a text file, which can be edited. The input file can be accessed if the following setting is made. First, an access to the file is enabled in the menu **Options | Setting**, where the box “Enable input file editing before starting FE analysis” is checked, Fig. 5-4.

Fig. 5-4 Box to enable input file editing in ‘Option/Settings’ menu.

In case of active access the solution window shown in Fig. 5-5 displays a box for input file editing. If the Edit-option is checked the input file is opened before analysis starts. The file can be edited or viewed. After its closing the analysis continues with the changed input file. The format of input file is described in a separate volume of ATENA documentation. Note that the input file does not exist as a disk file. However, its contents can be copied and saved in a text file on a disk at the time when it is open for viewing and editing.

5.3.6 Analysis

5.3.6.1 Starting analysis

Pressing the button 

in the solution window (Fig. 5-5) starts the calculation. The finite element analysis is performed by the ATENA solution core. This module communicates with the ATENA Graphical User Interface during the calculations and makes possible a unique monitoring of solution process – real time graphics. During the solution mode the screen changes to a form shown in Fig. 5-6.

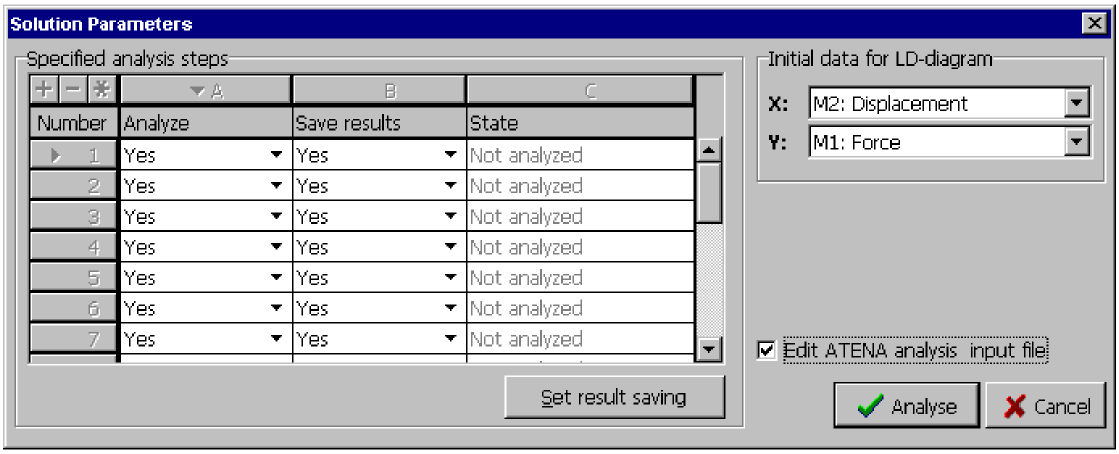


Fig. 5-5 Solution menu with checked option for input file editting.

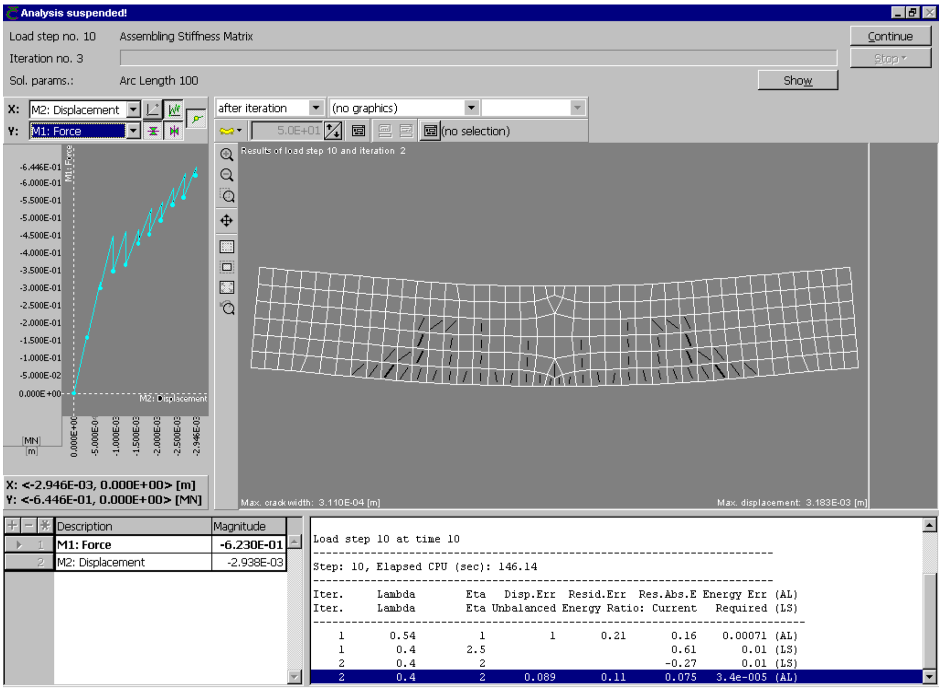


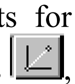
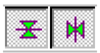
Fig. 5-6 Screen at the solution mode.

The window in the analysis modes includes four partitions for monitoring:

– Response diagram (load-displacement) located in top-left.
– Current vatues of monitoring points located in bottom left.
– Stress states and crack patterns located in top rights.
– Convergence text file located in botton rigth.

These information enable to evaluate main features of the structural behavior in the course of nonlinear analysis.

5.3.6.2 Monitoring of Response Diagram

The nonlinear response can be monitored by the diagram in the left part of the window. The tools for controlling the diagram are located above the diagram. Monitoring points for diagram axes X,Y can be selected from the list after pressing the button . The buttons ,  are for showing the converged response (load steps only) and the iterative response (load steps and iterations), respectively. The buttons  are to flip the diagram vertically and horizontally and the button  is to mark the load steps.

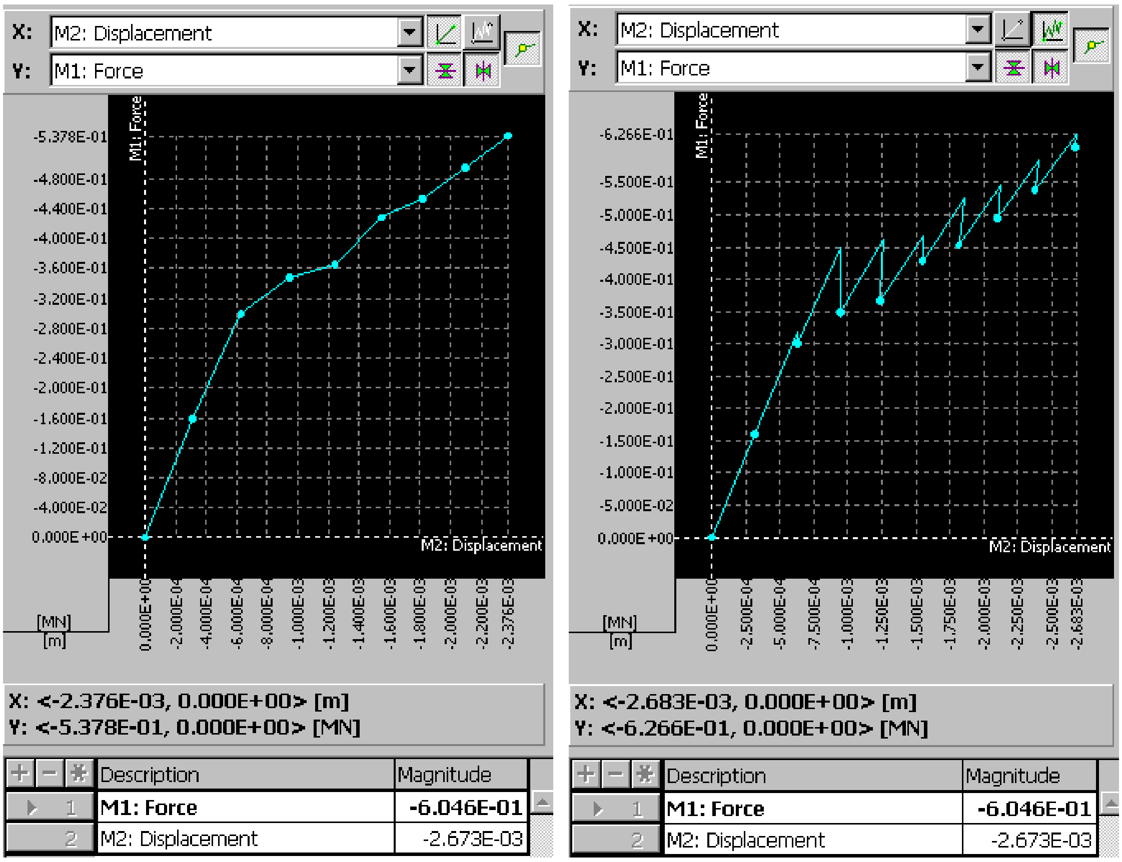


Fig. 5-7 Load-displacement diagram. Left - load steps only. Right – iterations and load steps.

A monitoring of response offers a possibility to observe the non-linear response during solution. In the above example, Fig. 5-7, the solution method is Arc Length. The diagram with iterations (right) enables to observe the convergence path of the solution. In the first iteration the force is well above the converged curve and reflects residual forces, which violate equilibrium. Through the iterative process the response curve is descending and approaches the final converged point. The magnitude of the applied force is adjusted by the Arc Length method.

5.3.6.3 Real-Time Graphics

Damage state of the structure, namely, deformations, cracks and stress can be observed during the nonlinear analysis in the large window on the right. It is called ‘real-time graphics’ because it reflects stress state during the analysis (contrary to the post-processing graphics.) Initially the window is in ‘no graphic’ mode as shown in Fig. 5-8.

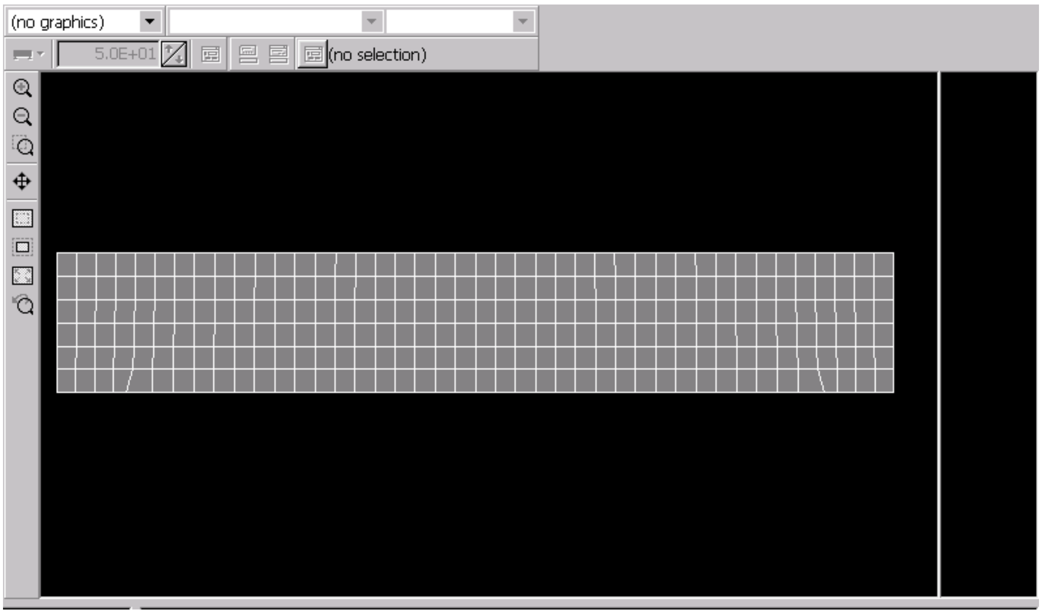
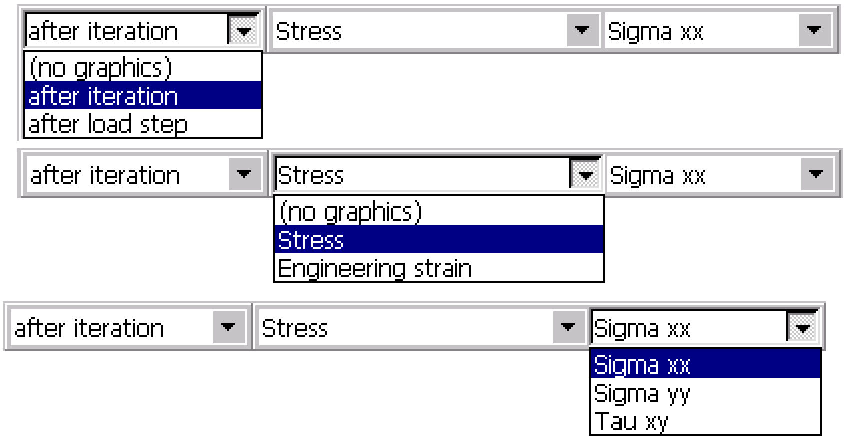


Fig. 5-8 Real-time graphics window in 'no-graphics' mode.

The properties of the window can be set in three list-boxes in the top. The menu of the list boxes can be unfolded by pressing the arrow button . The contents of the list boxes is shown in



Refresh

Stress, strain

Component

Fig. 5-9 List boxes of the real-time graphics window.

If the refresh mode is in 'no graphics' state the current data are not show. In this case the analysis is somewhat faster, since the graphic data retrieval requires some processor time. This mode can be used in case when user is not observing the analysis, or when analysis runs in a hidden window.

In case of refresh 'after iteration' the graphic data are updated after each iteration and in the refresh 'after load step' data are updated only after completing a load step. In these both cases the graphics shows cracks.

In addition to cracks either stress or strains can be shown. One value of stress (strain) is shown in element element. The required component can be selected in the list box. The color scale representing numerical values is generated automatically and is shown on the right. It can not be changed by user.

The object in the window can be shown either in undeformed or in deformed state. This can be selected using the 'deformed' toolbar:

Undeformed object

Deformed object

Example of a view of the undeformed beam with cracks and stresses σ_x is shown in Fig. 5-10. The same beam with deformed mesh is shown in Fig. 5-11. Deformations should be normally multiplied by a multiplying factor in order to make them visible.

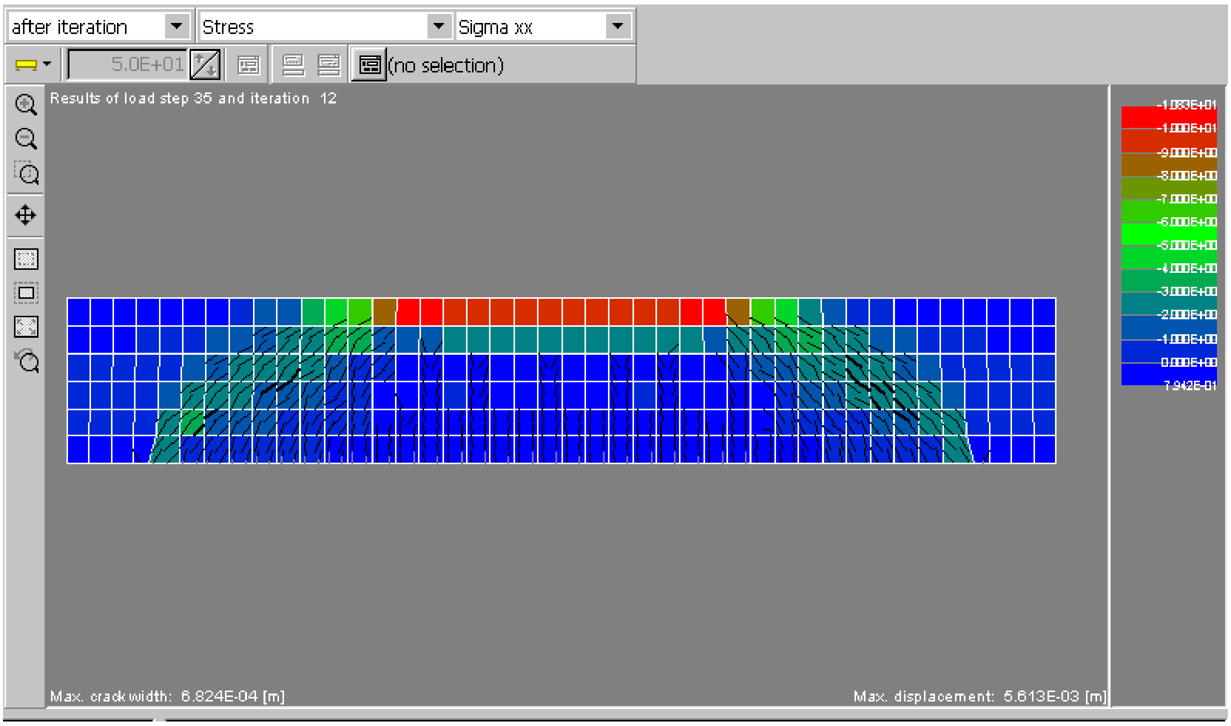


Fig. 5-10 View of cracks and stresses in undeformed mesh.

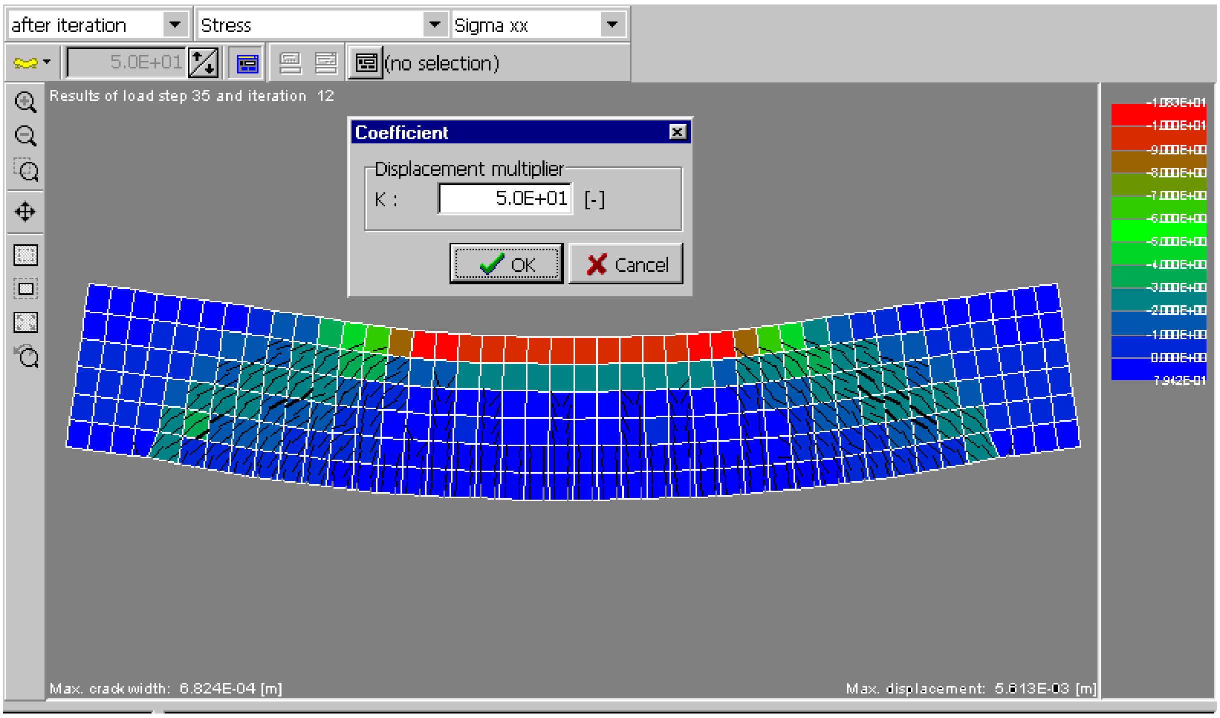


Fig. 5-11 View of cracks and stresses in deformed mesh.

The multiplier for displacements can be entered in two ways. By clicking on one of the arrow buttons the existing factor can be increased/decreased, or, by pressing the numerical input button a dialog can be opened as shown in Fig. 5-11. The cracks are shown in each material point (integration point). The number of material points depends on element type. The range of crack width for display can be defined by pressing the numerical input button for the cracks filter . This opens the input dialog shown in Fig. 5-12.

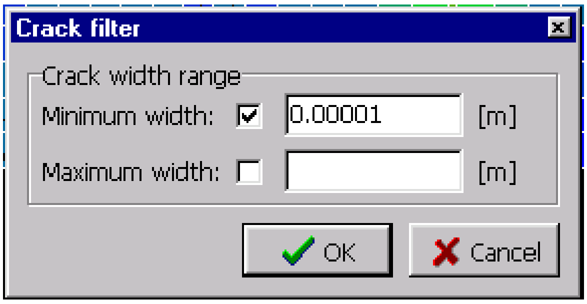


Fig. 5-12 Crack filter input.

Only the cracks with widths within the given range are displayed. This option is useful to show only the wide-open cracks indicating failure, or to suppress micro-cracks which are normally not visible. The values of the crack range are displayed in the top bar as shown in Fig. 5-13. Cracks parallel to the X-Y plane (R-Z plane in axial symmetry) are shown as circles.

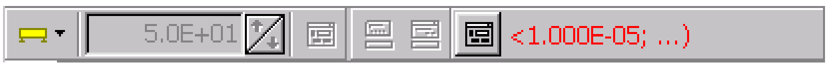
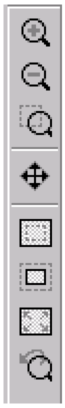


Fig. 5-13 Display of the crack width filter values.

View of the object in the window can be manipulated by scaling tools located within the left margin of the window, see Fig. 5-14.



- zoom-in at a point
- zoom-out at a point
- select viewed area
- move object
- zoom-in frame
- zoom-out frame
- make all objects visible
- previous view

Fig. 5-14 Scaling tools in real-time graphics.

5.3.6.4 ATENA Solution Log

The window in the bottom right corner shows the message stream log coming from ATENA solution core. This message is a record of convergence characteristics. Its contents is later available from the menu item **Calculations | Analysis progress information | Message**. The head of the record for load each contains the load step number. The items in the table have the following meaning:

First row:

Lambda – Factor for load modification in Arc Length method
(=1 if Newton-Raphson method).
Eta – Factor for displacement modification in line search.
Displ.Err – Displacement error, based on vector norms.
Resid.Err. – Residual force error, based on vector norms.
Resid.Abs.E – Maximum error of residual forces, based on residual forces.
Energy Err. – Unbalanced energy error.
(AL) – Arc Length or Newton-Raphson iteration method.

Second row:

Lambda – same as in the first row.
Eta – same as in the first row.
Unbalanced Energy Ratio: Current – Current value.
Required – Required value for the line search criterion.
(LS) – Line search iteration.

The rows denoted in the last column by (AL) correspond to the convergence criteria of the a used method (Newton-Raphson or Arc Length). This row appears always in the end of iteration loop. The second row denoted in the end by (LS) appears only if the line search is employed. If the line search is switched on it is employed only if required by the algorithm.

5.4 Pre- and Post-processing

Pre- and post-processing items of this menu, see , change the mode of graphical user interface in appropriate modes of operation. They are treated separately in self-contained chapters. The top menu is adjusted according to the current mode of GUI. If **Pre-Processing** is active the menu item Input appears in the main menu. Its usage is described elsewhere. If **Post-Processing** is active the window changes to another form and only tools for graphical post-processing are active.

5.5 Analysis Progress Information

This is the last item in the menu **Calculations**. It offers text records of various messages from ATENA core solution, see Fig. 5-15.

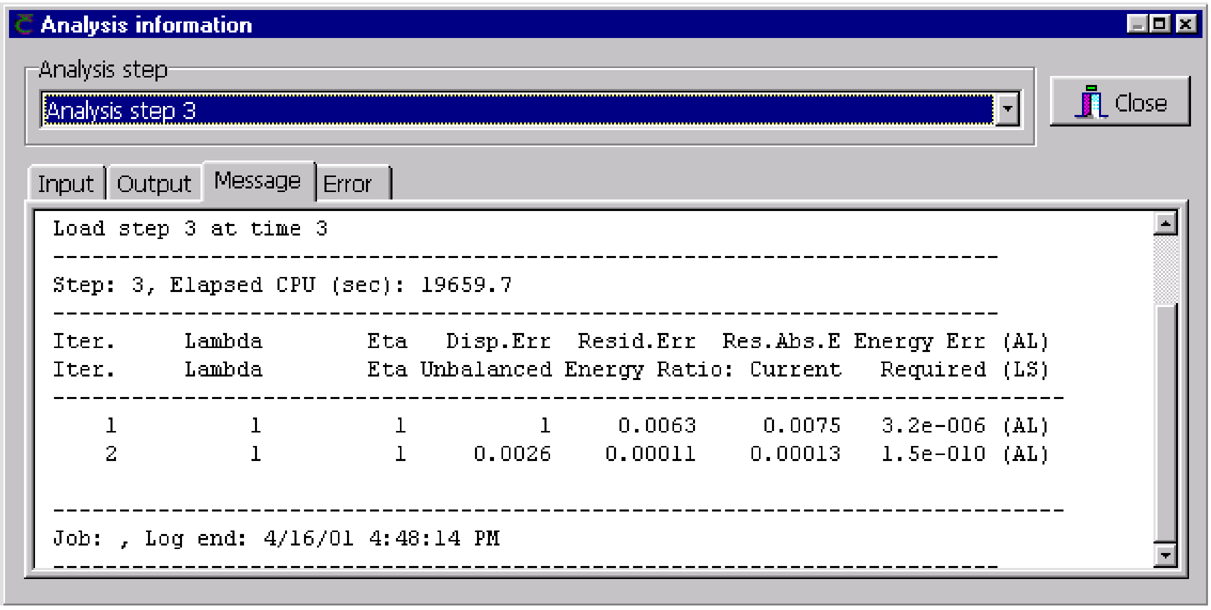


Fig. 5-15 Box for display of text ATENA streams.

By pressing the tabs an appropriate text output can be visualized. The texts can be also selected and copied to other documents in a usual way. The files are structured for each load step separately. They are generated during analysis and maintained for all executed load steps, also for the steps without saved results. Following type of output can be accessed:

Input – Input text file for **ATENA** analysis. This is the file with control commands and data for analysis. The same file can be open at the time of analysis begin.

Output – Output file text from **ATENA**. This file is usually not used if **ATENA** is executed by GUI. All output data can be accessed through the menu **File | Text printout**.

Message – Record of convergence parameters. It is very important file. Example is shown in Fig. 5-15.

Error – Error and warning messages.

The analysis information is important source of data for user. It is typically used to check the convergence of solution. In case of not converged iteration the error text appears in the Error file such as:

`‘Warning: Convergence criteria not satisfied‘.`

The error values can be found in both ‘Error’ and ‘Message’ lists.

6 OPTIONS

6.1 General Description

Menu **Options** offers several general settings as shown in Fig. 6-1.

Fig. 6-1 Options menu.

Application language. It is defined in the program and can not be changed.

Output language have three options: application language, Czech, English. Output language can be chosen independent of the application language.

Tool bars 'Files', 'Analysis' and 'Scale and shift' can be included or removed from the tool bar.

Output. This option enables configuration of headers, footers and pages for printed text and graphics.

Window display setting is not active.

6.2 Activity

This item enables to select objects for display. Three types of object can be handled in this selection: reinforcing bars, cuts and moment lines. Dialog window is shown in Fig. 6-2.

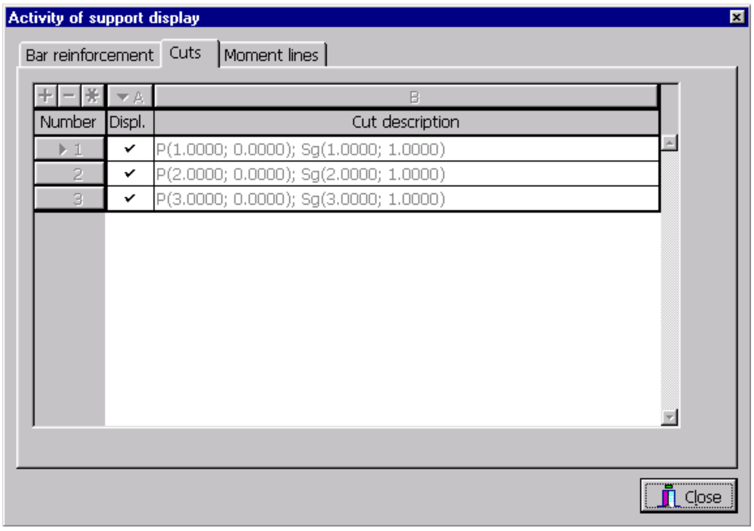
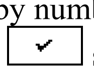


Fig. 6-2 Activity dialog window.

The appropriate type of object (bars, cuts, moment lines) is chosen by clicking on the tab. Then a list of items appears. The objects (bars, cuts, moment lines) are identified by numbers. These numbers can be related to the objects in pre-processing. The checked items  shall be displayed. The check mark can be changed by clicking on it.

6.3 Settings

This is an access to the setting of various graphical properties. A view of the dialog window is shown in Fig. 6-3. It contains four partitions, which are accessed by click on their headers.

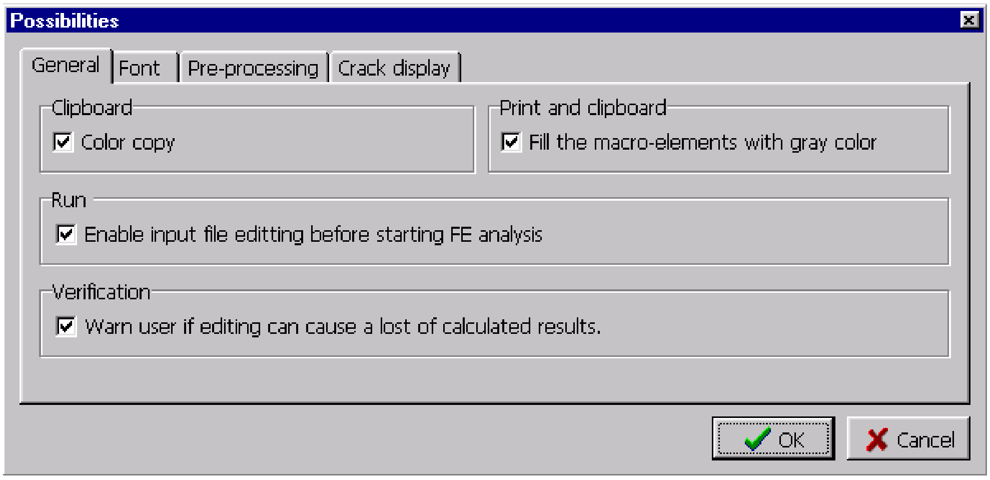


Fig. 6-3 Settings dialog window

The meaning of various parameters is clear from their descriptions. Only a brief comment shall be made to some items.

General

Clipboard

In the box **Clipboard** the color of the lines can be set. It concerns only lines and text. If the box is checked, the lines of object on the clipboard are the same color as on the screen. The clipboard contents is transferable to other documents in Windows by the "Paste" command. If nor checked, the lines and texts are black.

Print and clipboard

The macroelements on the screen are filled gray. This filling can be eliminated from the exported pictures if the box is not checked.

Run

If the box is checked the input text file for **ATENA** is made available for editing before running ATENA analysis.

Verification

If this item is checked the program gives a warning in case of editing data, which shall cause lost of already calculated results. (This may happen, for example, if material parameters are changed in a material, which was already used for analysis.)

Font

This dialog allows to change the font size of various output text on screen as well as in printed and graphical forms.

Pre-processing

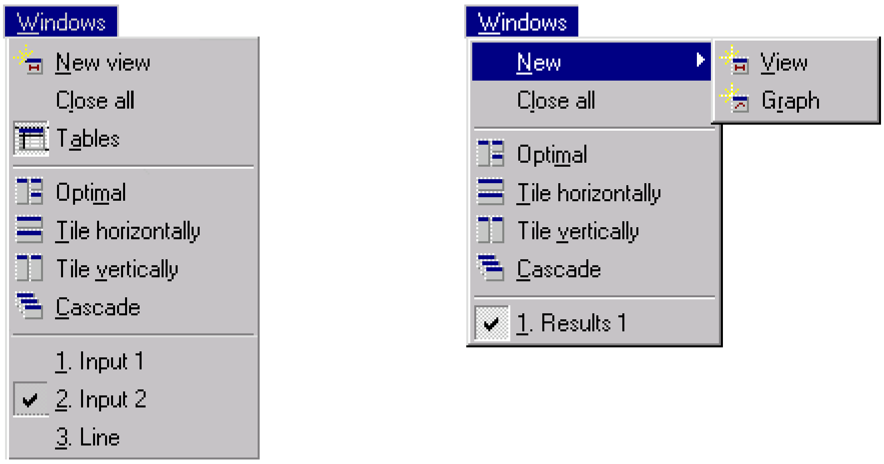
Predefined values of grid, which is used to control pointer positioning. If **Snap to grid** box is checked the pointer is controlled by grid.

Crack display

Maximal thickness of the line representing cracks can be set.

7 WINDOWS

Windows menu contains commands for windows opening and arrangement. Its contents is shown in Fig. 7-1. The contents of menu depends on the mode of GUI. In a pre-processing mode the menu contains items as shown in Fig. 7-1, left.



Pre-processing Post-processing

Fig. 7-1 Windows menu.

In the top section of the menu new windows are opened. In the middle section the window arrangement on the screen can be made. The bottom section contains a list of opened windows.

New window can be started in the item **New**. In the post-processing two types of window can be generated. By choosing **View** another window with object is opened. Any number of windows can be opened simultaneously. In case of uncertainty the bottom list can be inspected and not neede windows can be closed.

Window **Graph** in the post-processing makes possible to display an X-Y graphical view of monitored data. Example of such graph is shown in Fig. 7-2.

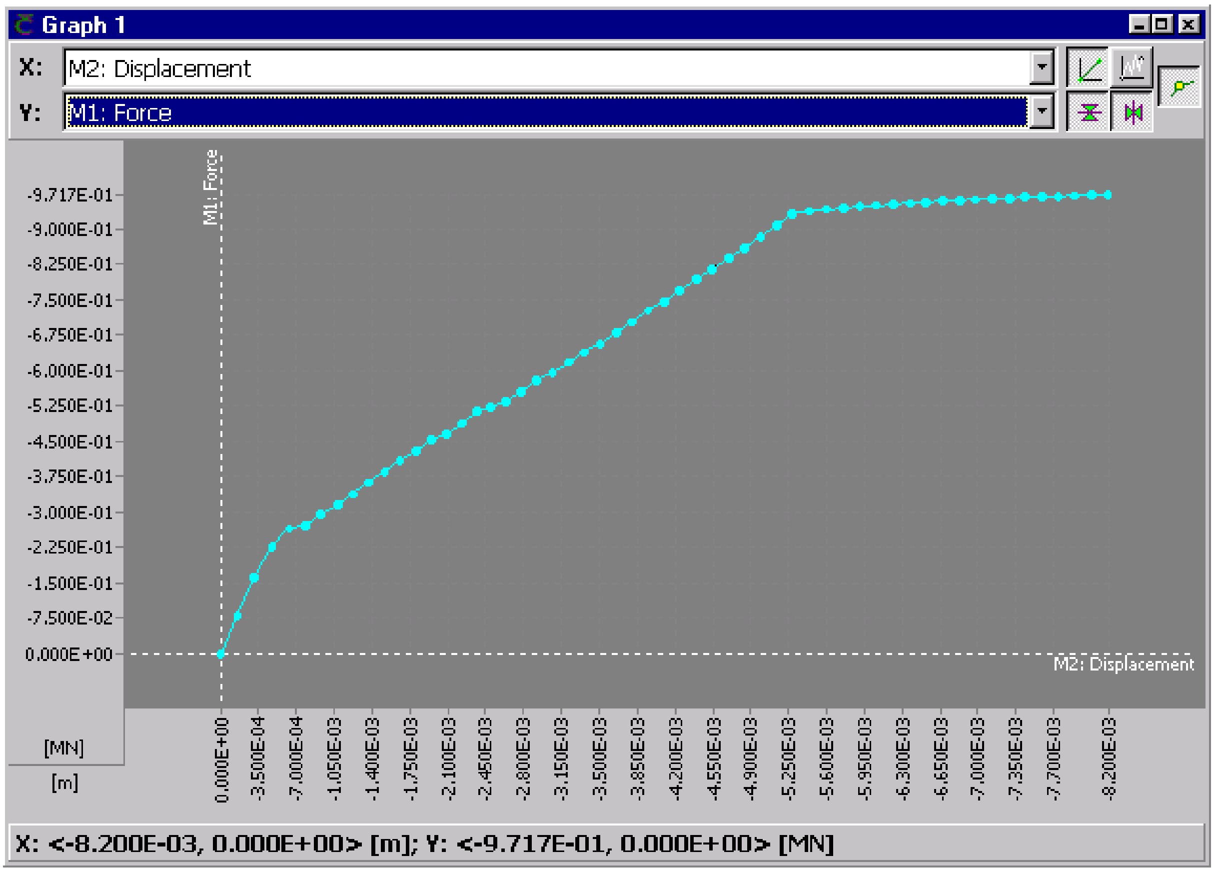


Fig. 7-2 Graph window with monitored load-displacement diagram.

The variables for X- and Y- axes can be selected from the lists, which are unfolded after clicking on the arrow button . The functions of this window are the same as described in the section Analysis, 5.3.6.3 Real-Time Graphics.

8 POST-PROCESSING

8.1 Starting Post-processing

Post-processing can be started by a click on the button in the **Calculation** menu or tool bar. Then the ATENA GUI is changed to the post-processing mode. Location of various tools and settings in the post-processing mode is shown in Fig. 8-1.

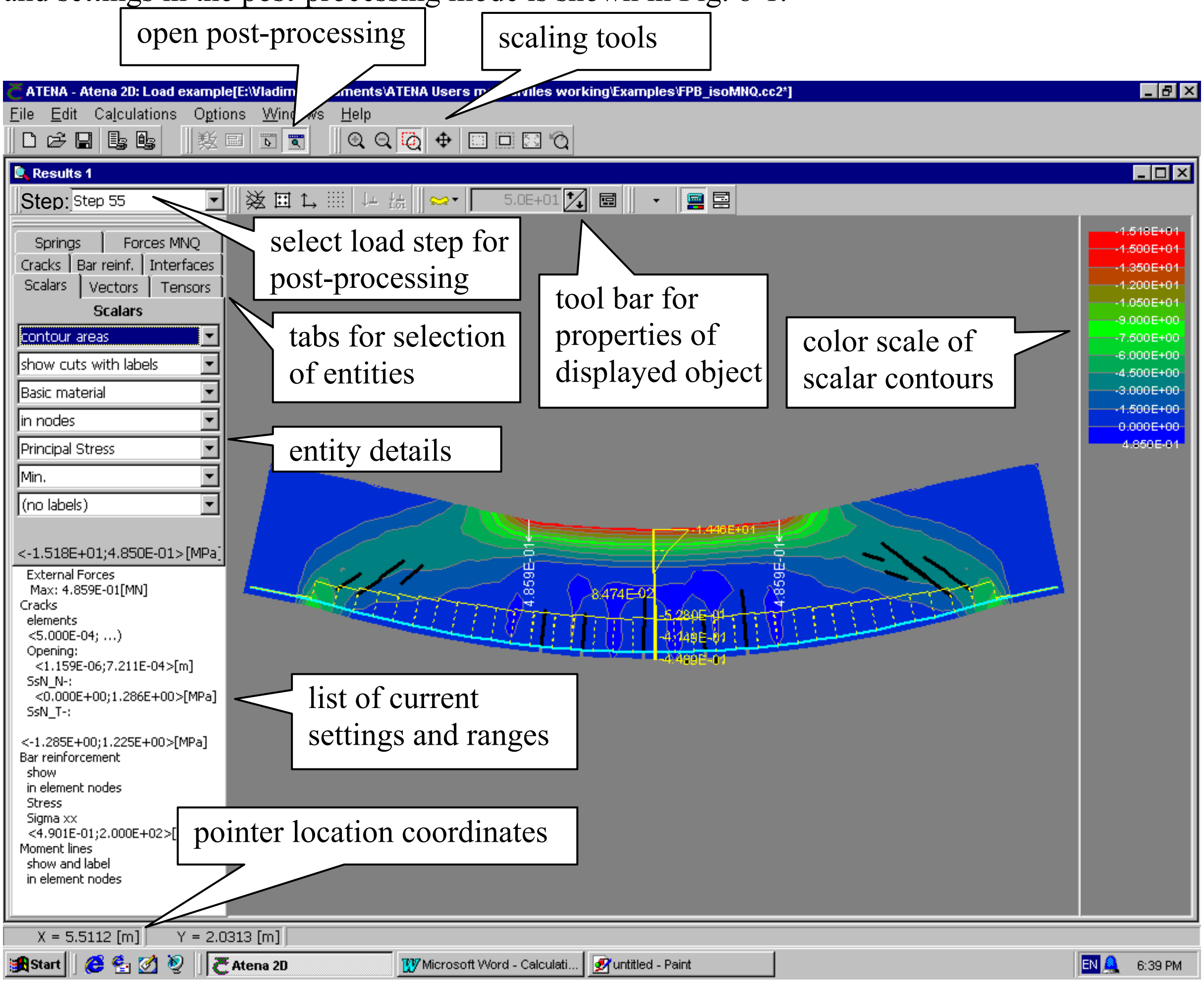


Fig. 8-1 View of the post-processing mode of the ATENA GUI.

Post-processing can be started at any time providing there are any saved results from solution. It can be data loaded from disk files *.cc2 resulting from analysis preformed previously, or the data existing in a current ATENA window after an end or interrupt of a solution.

The post-processing can be started in following sequence of steps:

1. Initiate the pros-processing mode by pressing the post-processing button in the man tool bar in the top.
2. Select the load step for post-processing from the pull-down menu located in the top-left corner of the window.

3. Select the entity to be displayed (such as scalars, cracks, etc.) from the set of tabs. Tabs are located in the top-left corner below steps. More entities can be show simultaneously.
4. Select detail parameters of the entity (such as contour type, component, label, etc.) form the pull-down menus, which are located below tabs.
5. Define the properties of the graphical window using the tool bar above the window (mesh on/off, boundary conditions on/off, deformed form, crack filter, etc.).
6. Choose appropriate section of the object in order to make all details visible using the scale tools located in the tool bar just under the main menu (zoom-in, -out, move, make all objects visible, etc.)

A list of defined entities is shown in the filed left of the window including the ranges of values. This may help to find quickly the maximum values.

The way of selecting detail properties is similar for all entities and can be described on the example of scalars. It includes three steps:

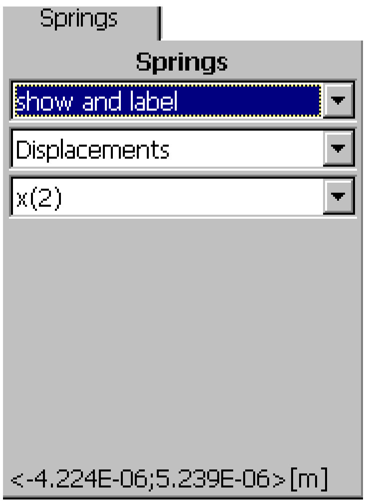
8.1.1 Type of Display

Type of display can be chosen from the pull-down menu as shown for tensors in Fig. 8-28.

8.2 Springs

The values along the lines supported by springs can be shown by choosing the tab 

. After this the folder 'Springs' is moved to the top and the pull-down menus for details are made available as shown below. Details of the display should be selected form the offered pull-down menus.



Display type:

no graphics - Display of spring state is not shown.
show – Only graphics is shown.
show and label – Graphics and labels are shown.

Type of variable:

Displacements – Displacements of the spring.
Partial Internal forces – Nodal force in the spring only.
Internal forces – Sum of the nodal forces in springs and adjacent macroelement.
Reactions – Nodal forces in the springs.

Component:

x(1) – Component in X-direction.
x(2) – Component in Y-direction.

The spring post-processing is demonstrated on a beam resting on contact foundation. The model is shown in Fig. 8-2. The beam is loaded by uniformly distributed load in the middle and the contact springs are distributed along the bottom line. The post-processing of the springs is shown in Fig. 8-3 and Fig. 8-4. Positive deformation of springs, (contact opening) is observed near beam ends and negative deformation (contact) is in the beam center.

Reaction force distribution corresponds to spring deformation. There are zero reaction forces in the end regions, which are not in contact.

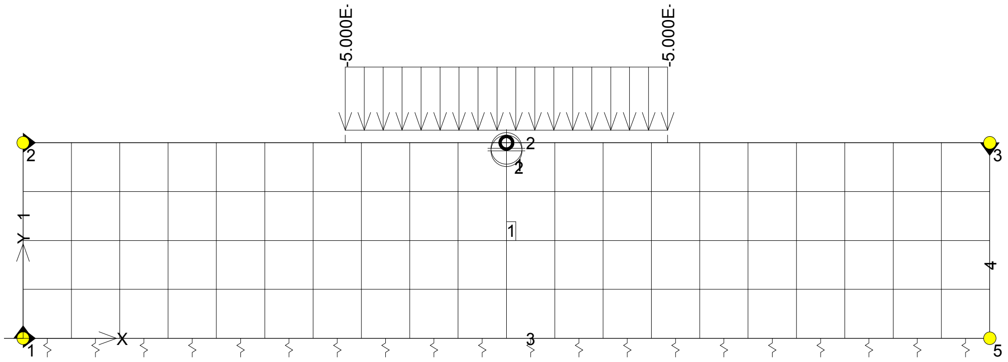


Fig. 8-2 RC beam on contact springs.

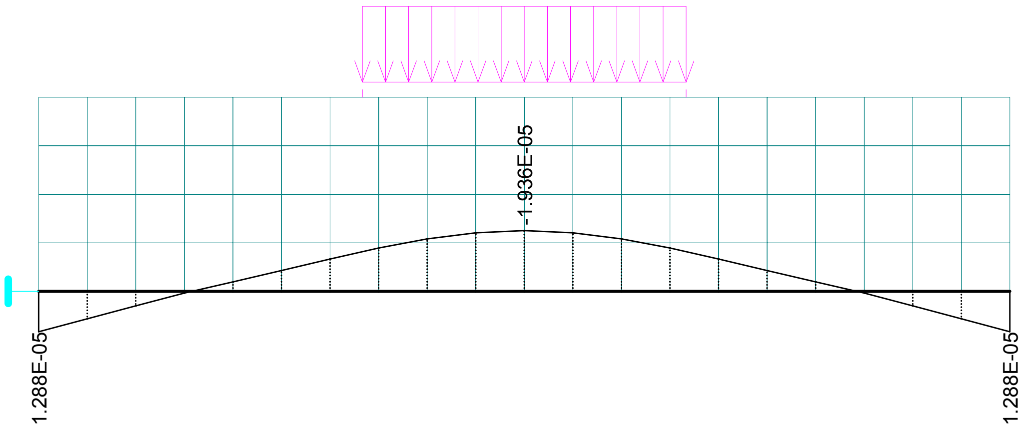


Fig. 8-3 Beam on springs – deformations of springs.

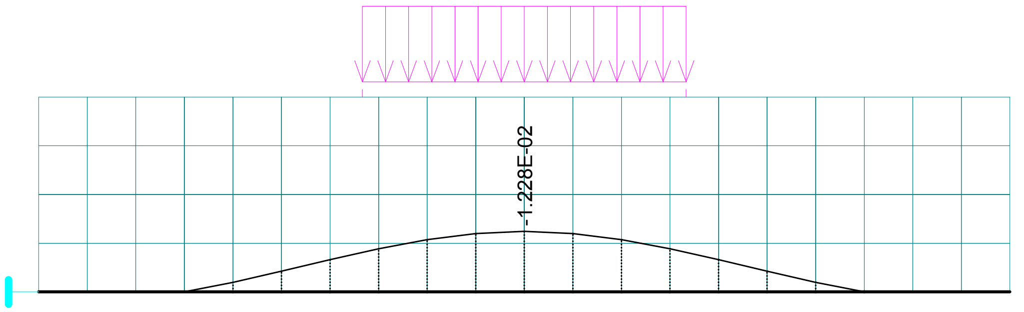
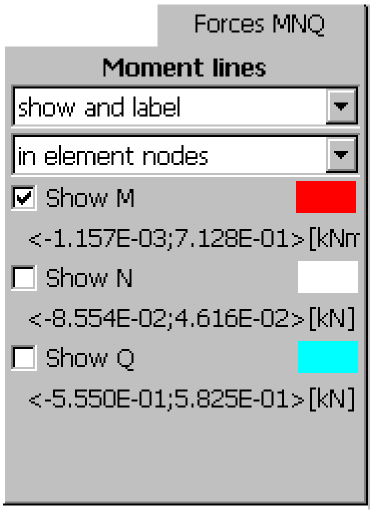


Fig. 8-4 Beam on contact springs – reaction forces in springs.

8.3 Internal Forces

Internal forces can be evaluated along pre-defined lines. Thy can be included by choosing the tab Forces MNQ. Folder 'Forces MNQ' is opened and the pull-down menus for details are made available as shown below. Details of the display should be selected form the offered pull-down menus.



Display type:

no graphics – Display of internal forces not shown.
show – Only graphics is shown.
show and label – Graphics and labels are shown.

Type of stress averaging:

in nodes – Average stresses in nodes are considered in the numerical integration of internal forces.

in element nodes – Stresses in element nodes are considered in the numerical integration of internal forces. (Stresses are not averaged before integration).

Internal force:

Show M – Moment with respect to line is calculated if this box is checked.

Show N – Normal force resulting from integration of normal stresses on a section normal to the line is calculated if this box is checked.

Show Q – Shear force resulting from integration of shear stresses on a section normal to the line is calculated if this box is checked.

Internal forces are demonstrated on a beam subjected to a four point bending at the load step 55. The model is shown in Fig. 8-2. The beam is loaded by two forces with the magnitude of 0.4858 MN at the load step 55.

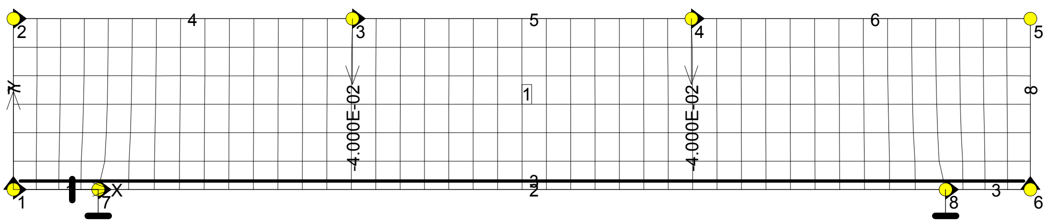


Fig. 8-5 Four-point-bending RC beam.

Internal forces are calculated with respect to lines pre-defined in pre-processing. They are results of integration of stresses in sections. Spacing of sections is defined in pre-processing.

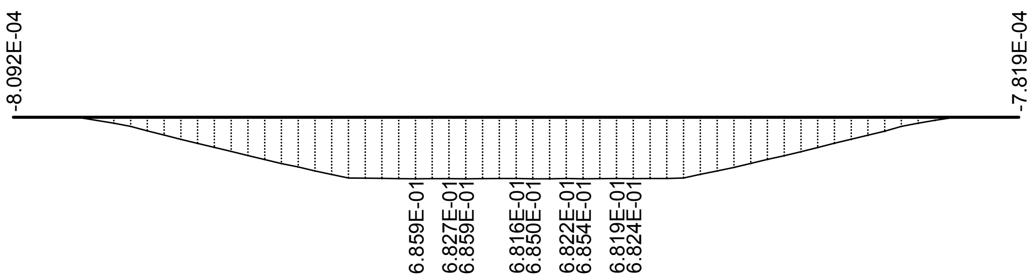


Fig. 8-6 Moment distribution. Integration in nodes (average stresses).

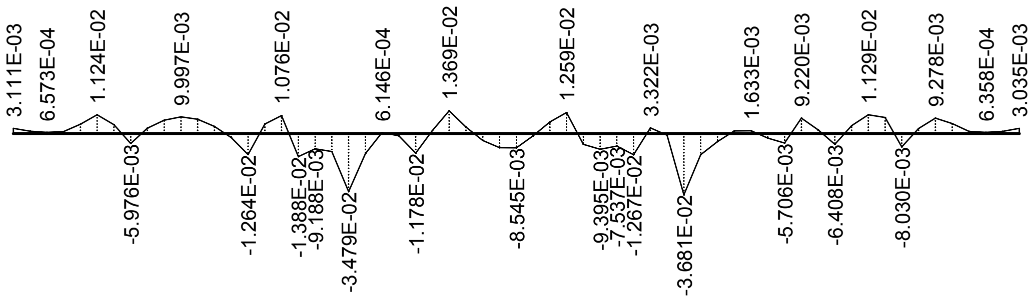


Fig. 8-7 Normal force distribution.

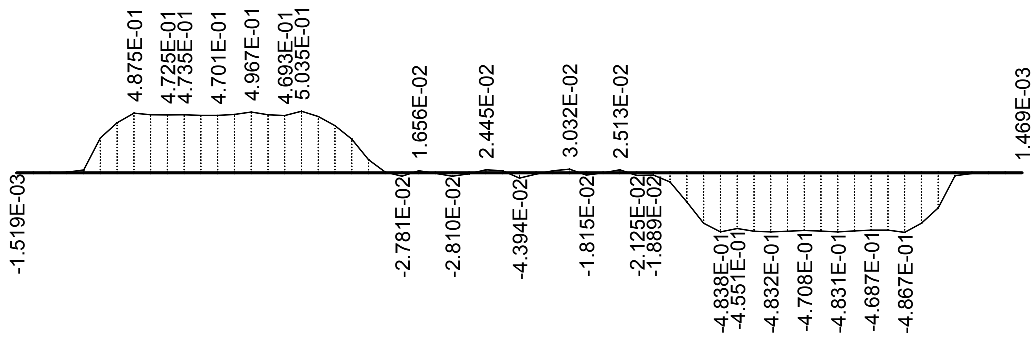


Fig. 8-8 Shear force distribution.

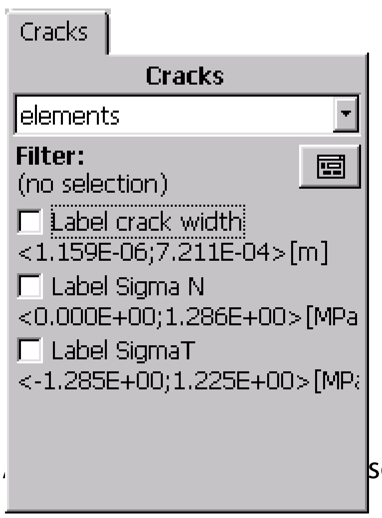
Notes:

- Internal forces can be used to check the equilibrium of internal forces and loading. In the above example the acting shear force of 0.4858 MN is constant between reaction and loading, which is well reflected by the analysis. The acting moment is 0.7229 MNm and calculated one is about 0.685 MNm. A better agreement can be found without averaging, by using option 'in element nodes'.

- Moment lines/internal forces can only work for areas with the same 2D idealisation of the continuum, i.e., combining plain stress and plane strain macroelements is not possible.

8.4 Cracks

Cracks can be displayed by choosing the tab Cracks. Folder is opened and the pull-down menus for crack menus are made available as shown below. Details of the display should be selected form the offered pull-down menus.



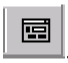
Display type:

no graphics – Display of cracks is not shown.

elements – Cracks are displayed. One crack per element.

integration points – Cracks are displayed. One crack per integration point.

Crack filter:

Filter – Values of crack filter can be given. Pop-up dialog is opened after pressing the numerical button . Only cracks within chosen range of crack width are displayed.

Cracks entities:

Label crack width – Numerical value of the crack width is shown if this box is checked.

Label Sigma N – Numerical value of the normal stress on the crack face is shown if this box is checked.

Label Sigma T – Numerical value of the shear stress on the crack face is shown if this box is checked.

Example of a crack post-processing of a R.C. beam under four-point-bending is shown in Fig. 8-9. The post-processing is done without labels and with no crack filter.

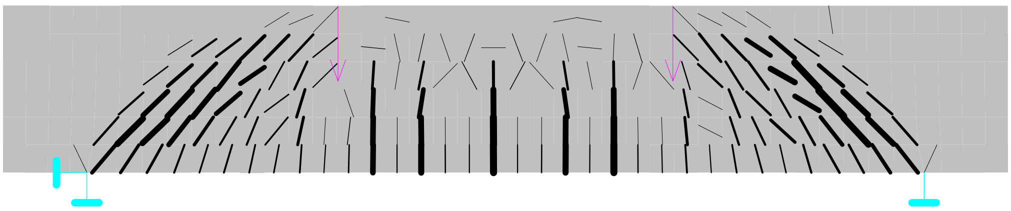


Fig. 8-9 Cracks in R.C. beam without crack filter and labels.

Fig. 8-10 shows the same beam where the minimum value of the crack width is set as 0.0004m. Only cracks larger then this value are displayed.

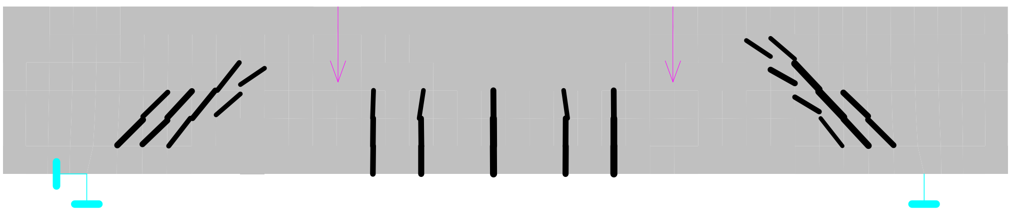


Fig. 8-10 Cracks in R.C. beam with crack filter and without labels.

Fig. 8-11 shows the crack pattern and deformed mesh in a severely damaged shear slab (plane strain, symmetric half). Cracks parallel to the X-Y plane (or R-Z plane in axial symmetry) are displayed as circles.

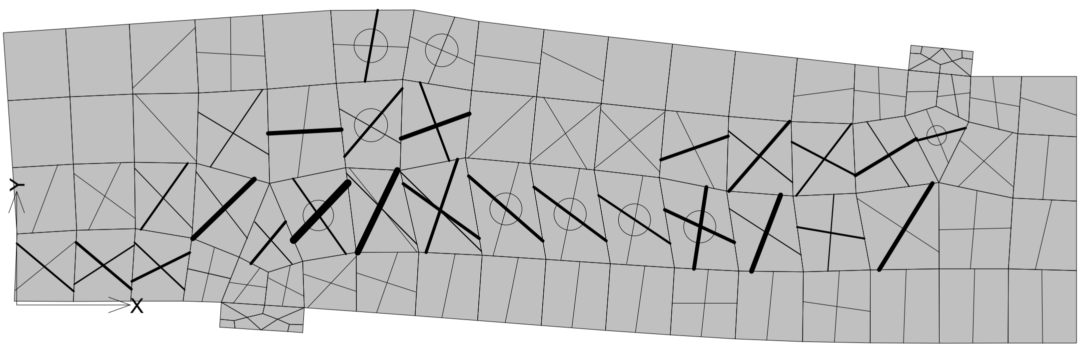


Fig. 8-11 Cracks in failing shear slab. Cracks parallel to the X-Y plane are shown as circles.

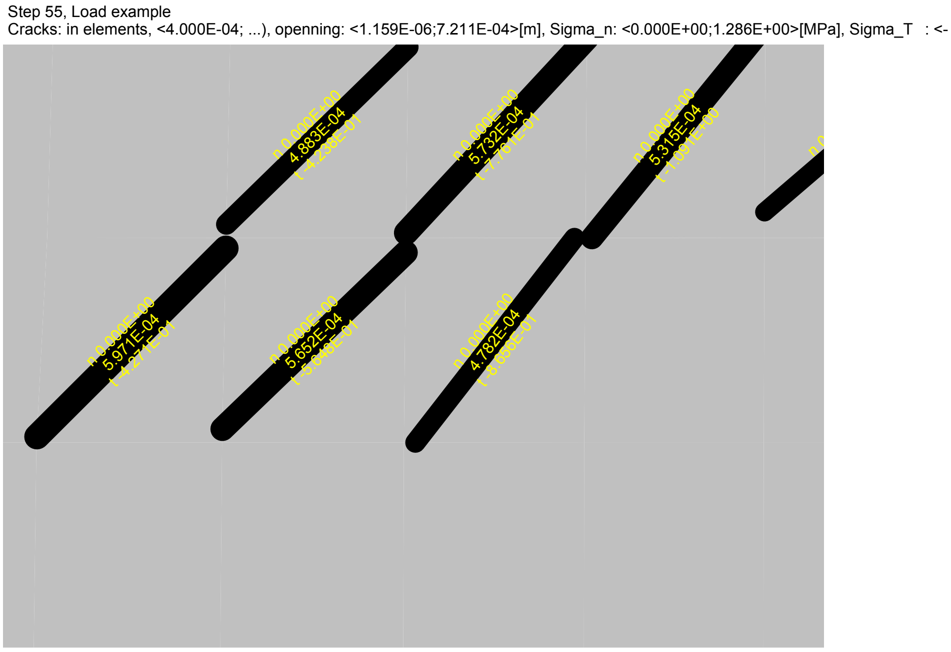


Fig. 8-12 Cracks in R.C. beam. Enlarged view of cracks with labels.

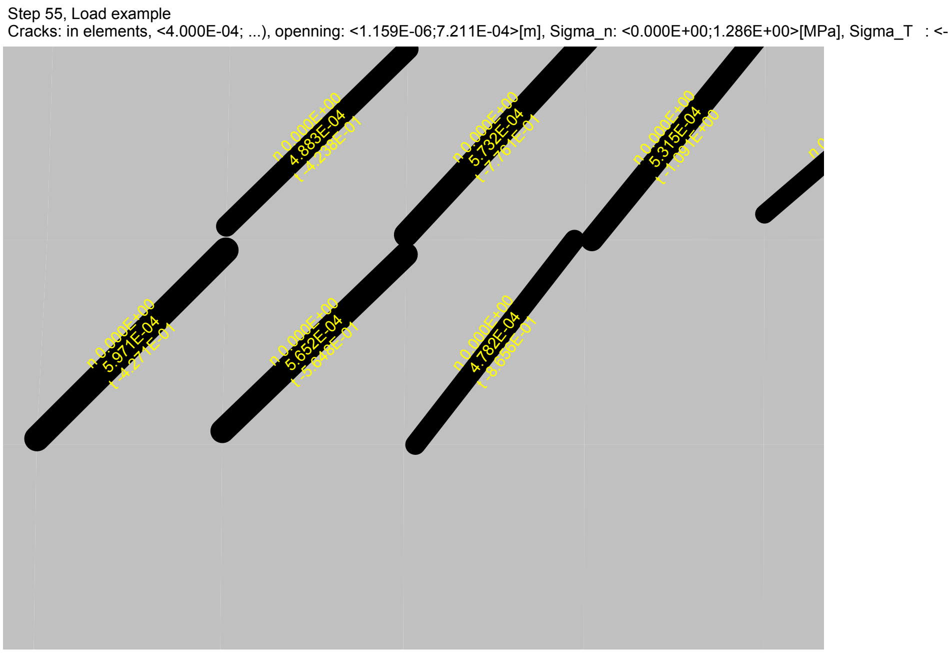


Fig. 8-12 shows a detail view of a cracked region with labels. The label contains in the first row normal stress (denoted by **n** before the number), in the second row the crack width and in the third row the shear stress (denoted by **t** before the number). The stresses are concrete stresses on the crack face.

8.5 Bar Reinforcement

Stress and strains in bar reinforcement can be displayed by choosing the tab Bar reinf.. Folder is opened and the pull-down menus for bars are made available as shown below. Properties of display should be selected form the offered pull-down menus.

Display type:

no graphics – Display of bars is not shown.

show – Bars are displayed.

show and label – Bars are displayed with labels.

Averaging method:

in nodes – Average values are shown in element nodes.

in element nodes – Values are shown in element nodes. No averaging is made.

Entity:

Engineering Strain – Engineering strain in bar in axial direction.

Principal Engineering Strain – Same as Engineering strain.

Stress – Stress in bar in axial direction.

Principal Stress – Same as Stress.

Plastic Strain – Plastic strain in bar.

Principal Plastic Strain – Same as Stress Plastic strain.

Component:

The desired component can be chosen from a list. The list contents changes with the entity type.

Example of a bar post-processing is shown in Fig. 8-13. It shows stress distribution in bar of the RC beam under four-point bending load. The average stresses in nodes are shown.

The same bar, but with values in element nodes (without averaging) is shown in

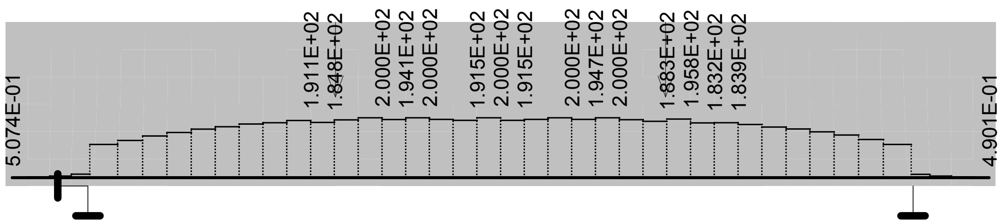


Fig. 8-14. It can be noted, that element stress reaches the yield stress of 200MPa in this case. In case of nodal values with averaging the stresses are slightly smaller due to averaging.

Plastic strain in element nodes (no averaging) is shown in

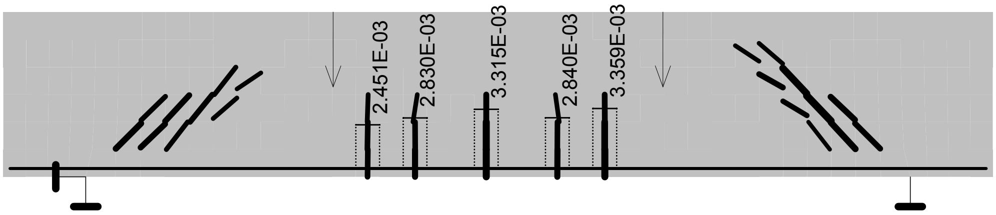


Fig. 8-15. In this picture the cracks are also shown in order to demonstrate that the reinforcement yielding indicated by plastic strains takes place in cracks.

Note: If there are more reinforcement bars the picture showing all bars can be unclear. For this situation only one, or selected number of bars, can be displayed. This can be done using the menu item **Option | Activity | Bar reinforcement***.* ***In this dialog the bars to be displayed can be selected. The bars can be identified only by numbers. Their numbers can be found in the preprocessing.***

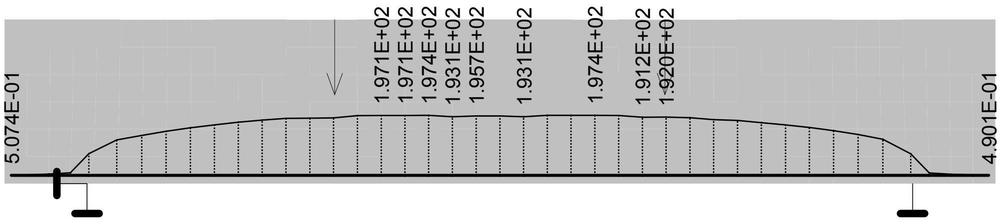


Fig. 8-13 Bar post-processing. Stress in nodes.

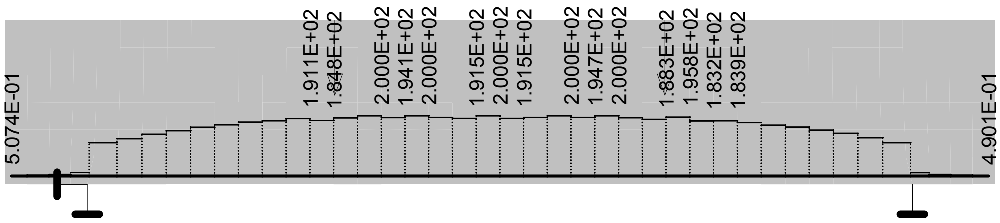


Fig. 8-14 Bar post-processing. Stress in element nodes (no averaging).

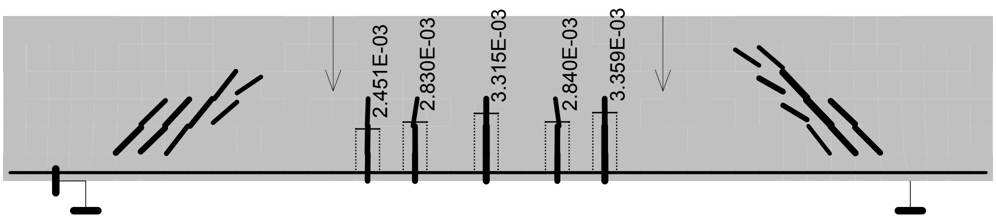
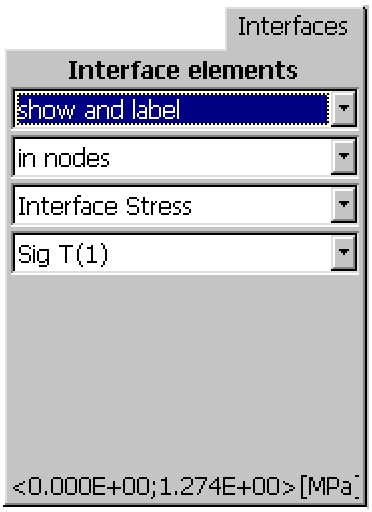


Fig. 8-15 Bar post-processing. Plastic strains in element nodes (no averaging).

8.6 Interface

Interface can be displayed by choosing the tab Interfaces. Folder is opened and the pull-down menus for interface are made available as shown below. The properties should be selected form the offered pull-down menus.



Display type:

no graphics – Display of interfaces is not shown.

show – Interfaces are displayed.

show and label – Interfaces are displayed with labels.

Averaging:

in nodes – Average values are shown in element nodes.

in element nodes – Values are shown in element nodes. No averaging is made.

Entity:

List depends on the entity type.

In nodes:

Performance index – Error in stresses in material model. Zero means no error.

Interface Stress – Stress in interface.

Interface Displacements – Interface displacement (relative).

Interface Plastic Displacements - Interface plastic displacement (relative).

Displacements – Displacements of interface nodes.

Partial Internal Forces – Forces in interface nodes equivalent to interface stresses.

Internal Forces – Sum of forces in interface nodes.

Reactions – Same as Internal forces.

Note: In nodes average values of adjacent element nodes are calculated.

In element nodes:

Performance index – Error in stresses in material model. Zero means no error.

Interface Stress – Stress in interface.

Interface Displacements – Interface displacement (relative).

Interface Plastic Displacements - Interface plastic displacement (relative).

Note: In element nodes no averaging is performed.

Component:

Components of entities can be selected from a list. The contents of the list changes according to the entity type. Directions are denoted by numbers: (1) parallel to interface, (2) normal to interface.

Example of interface is shown in Fig. 8-16, where two blocks are connected by interface. The bottom block is supported by fixed supports on the bottom face. The top block is first loaded vertically to generate normal stress in the interface and then loaded by prescribed displacement on the left top edge.

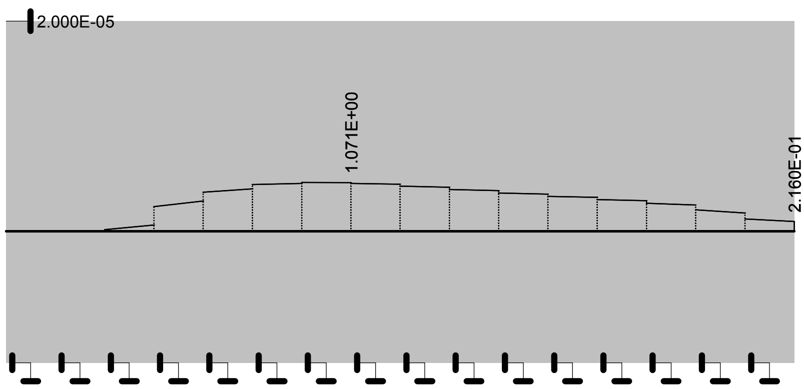


Fig. 8-16 Shear stress distribution in interface. Load step 3, undeformed object.

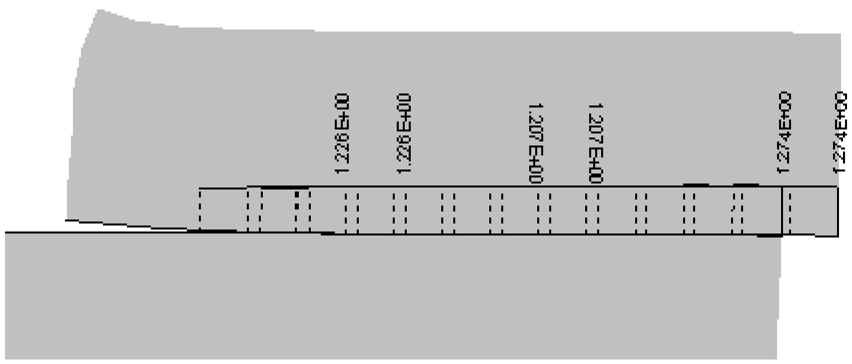


Fig. 8-17 Shear stress distribution. Load step 10, deformed object.

Deformed object shows opening of interface near the left edge, which is caused by the force action. The stress in element node is plotted on each interface side by different color. In this case both stresses are identical. However, different node values can occur in other variables, such as displacements.

8.7 Scalars

8.7.1 Scalar fields

Scalar menu can be opened by selecting the tab Scalars. The properties of scalars can be defined using the set of pull-down menus as shown in Fig. 8-18.

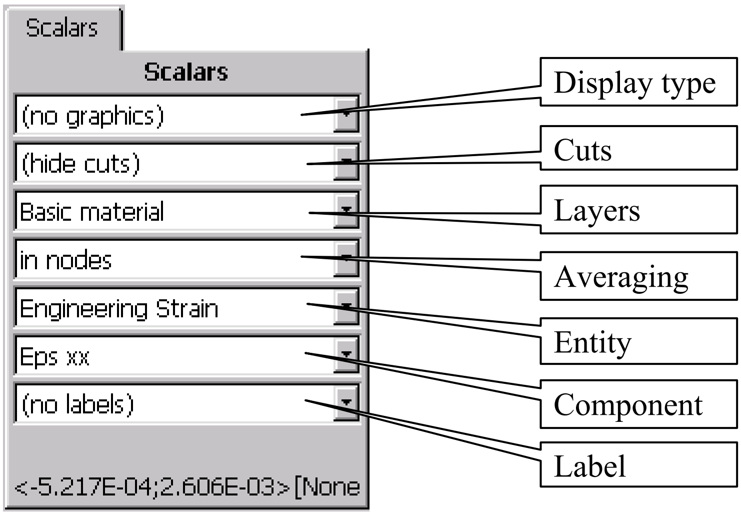


Fig. 8-18 Set of pull-down menus for scalars.

Display type:

no graphics – Display of scalars is not shown.

rendering – Values are shown by a colors. Continuous transition between colors is used. The color scale is defined automatically.

contour areas – Values are shown by a contour color areas. A discrete set of color bands is used to define the numerical scale. The color scale can be defined automatically or manually.

contour lines – Values are shown by a color contour lines. A discrete set of color lines is used to define the numerical scale. The color scale can be defined automatically or manually.

Cuts:

hide cuts – Cuts are not displayed.

show cuts – Cuts are displayed.

show cuts with labels – Cuts are displayed with labels showing numerical values.

Averaging:

in nodes – Average values are shown in element nodes.

in element nodes – Values are shown in element nodes. No averaging is made.

Layer:

Basic material – Scalars in basic material layer are shown.

sm. reinforc. layer (i) – Scalars in smeared reinforcement layer are shown.

Entity:

List of variables in this menu corresponds to the material model chosen. Example of variables: stresses, engineering strains, plastic strains, principal values.

Component:

A component of tensor or vector can be chosen from this list. The contents of the list corresponds to the type of the entity.

Label:

Numerical value in a label can be included.

Example of a scalar post-processing is shown on the RC beam. Fig. 8-19 shows principal strains in rendering iso-areas. The same is shown with contour areas in Fig. 8-20 and with contour lines in Fig. 8-21.

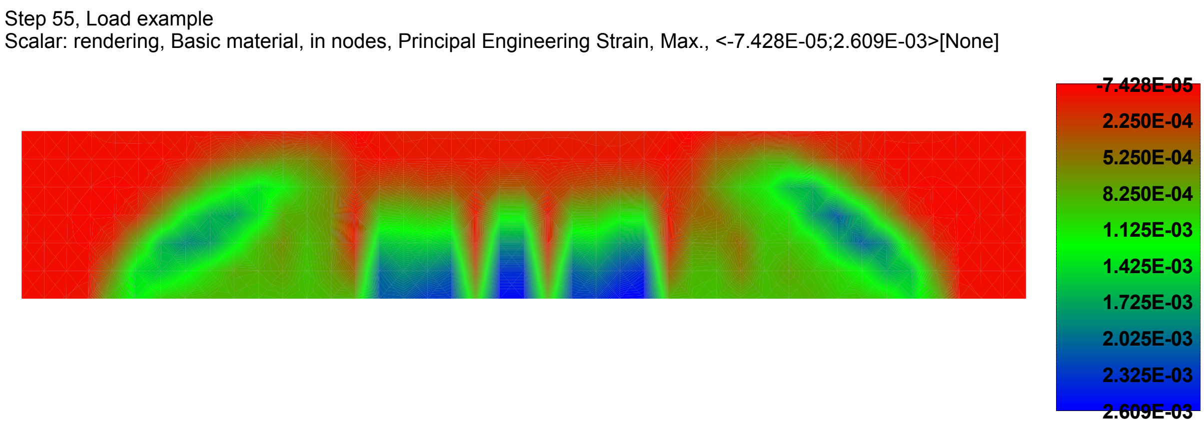


Fig. 8-19 RC beam. Maximal principal strains by rendering.

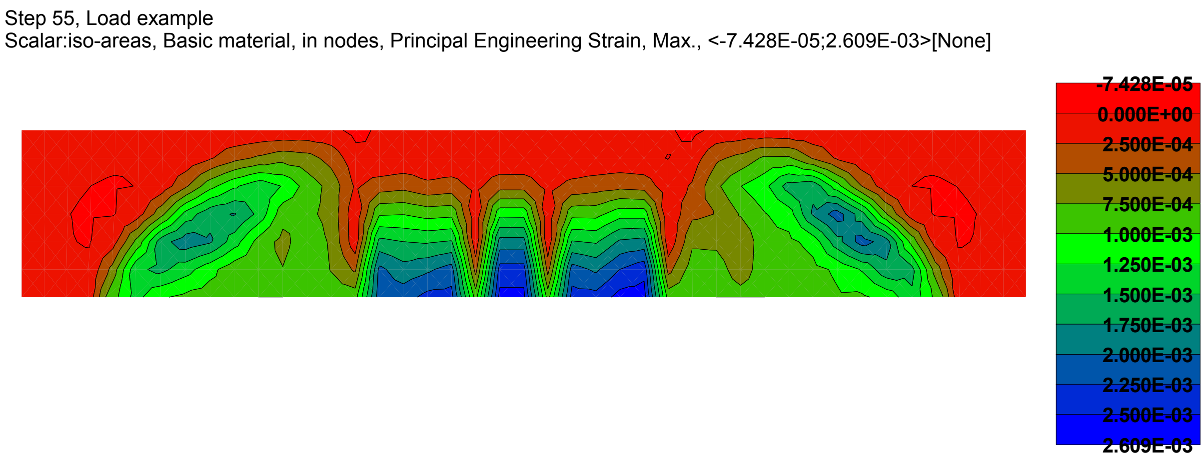


Fig. 8-20 RC beam. Maximal principal strains by contour areas.

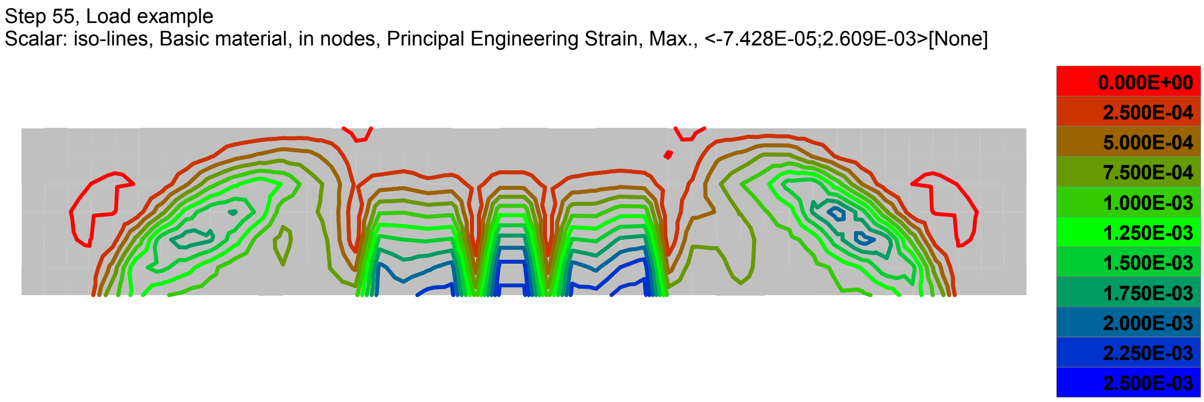


Fig. 8-21 RC beam. Maximal principal strains by contour lines.

The color scale is made automatically by the program. However, it can be also set manually by user by clicking on the ‘scale manual’ button  located in the top tool bar.

8.7.2 Scalars on Cuts

Cuts show distribution of variables along pre-defined cuts. Geometry of cuts can be defined in pre-processor. Display of cuts is initiated by selecting ‘show cuts’ or ‘show cuts with labels’ from the **Cuts** menu. If many cuts are defined and their display may interfere and become unclear. Then one or several cuts can be selected for display using the menu item **Option | Activity | Cuts**. In this dialog the cuts to be displayed can be selected by numbers. Their numbers can be found in the preprocessing. Example of cuts is shown Fig. 8-22 in and Fig. 8-23.

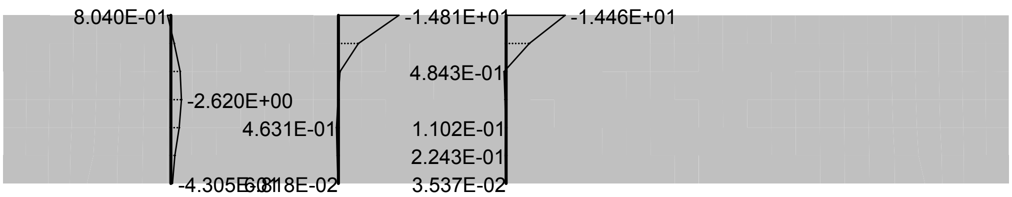


Fig. 8-22 Cuts. Normal stress distribution in concrete sections. Nodal (average) values.

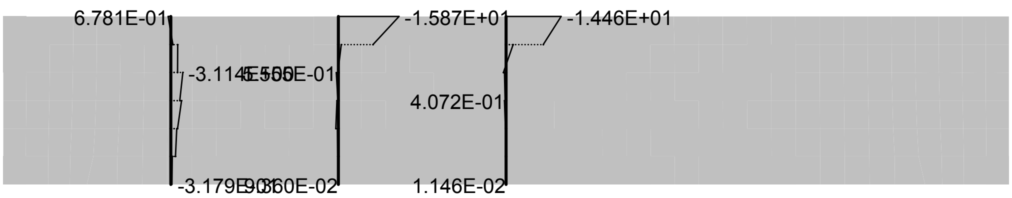


Fig. 8-23 Cuts. Normal stress distribution in concrete sections. In element nodes (not averaged).

8.7.3 Tools for Post-Processing Graphics

The properties of window and of the object inside can be manipulated by tools located on the top of the window, see Fig. 8-24.

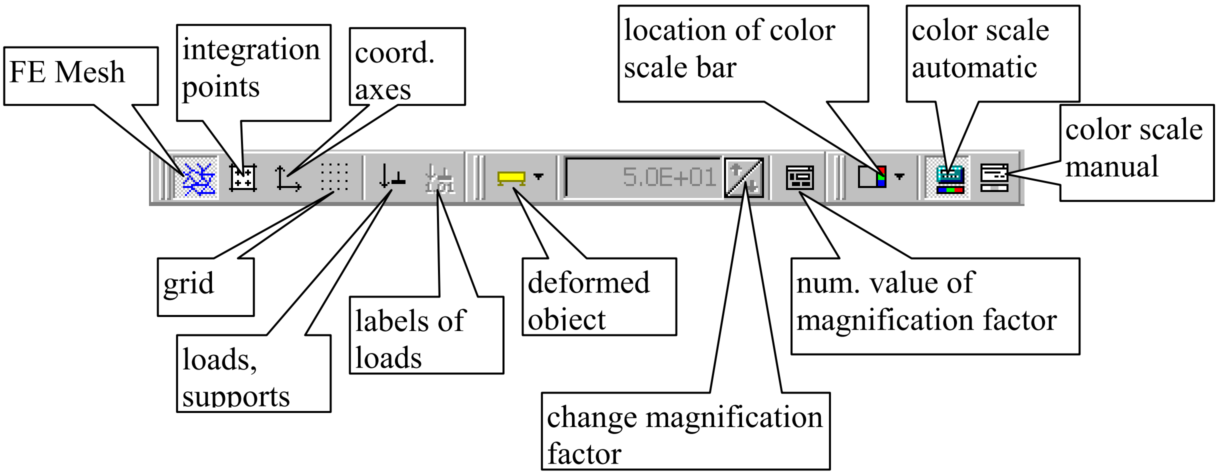


Fig. 8-24 Post-processing window tools.

The post-processing window can be displayed with or without axes and grid.

The object can be displayed with or without finite element mesh and integration points. The object can be also displayed in undeformed or deformed state. The magnification factor can be increased/decreased by pressing the arrows. It can be also given by a numerical value by pressing. Loads and supports can be displayed only on an object in undeformed state.

The color scale can be changed manually by clicking. The manual scale option is only available for contour areas and lines.

8.8 Vectors

Vectors display menu can be opened by selecting the tab Vectors. The properties of vectors can be defined in the menu as shown in Fig. 8-25.

Type of display:

no graphics – Vectors are not displayed.
show – Vectors are displayed graphically.
show and label – Vectors are displayed with labels.

Layer:

everywhere – shows all layers simultaneously.
basic material – shows only basic material (such as concrete, steel, soil, etc.).
bar reinf. – shows bar reinforcement.

Entity:

Displacements, **partial forces**, **partial reactions** – Nodal values in macroelements.
Forces, **reactions** – Compact values of nodal forces.

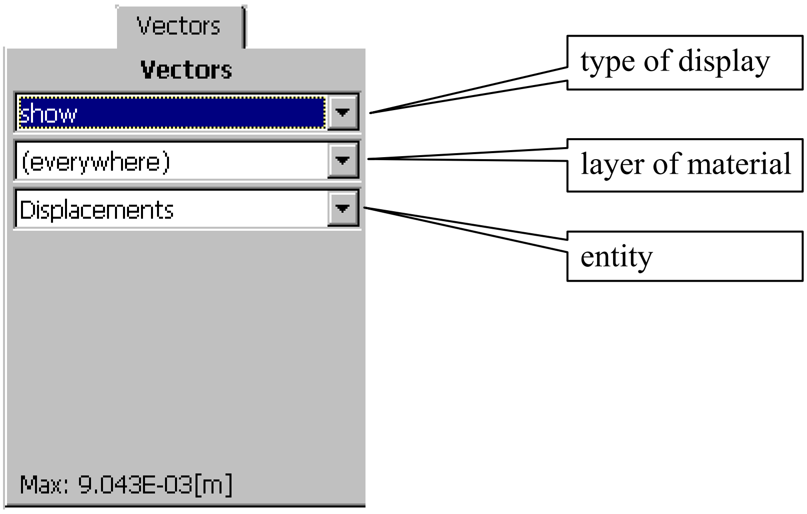


Fig. 8-25 Vector details.

Example of vector plot is shown in Fig. 8-26 and Fig. 8-27. Vectors indicate nodal displacements in the load step 55.

Vector plot can be used to show residual forces indicating errors of iterative solution, see

Fig. 8-27. Residual forces indicate unbalanced forces. In given example maximal residual force is 0.00417 MN which can be compared with the total load in the load step 55, 2P=0.972 MN.

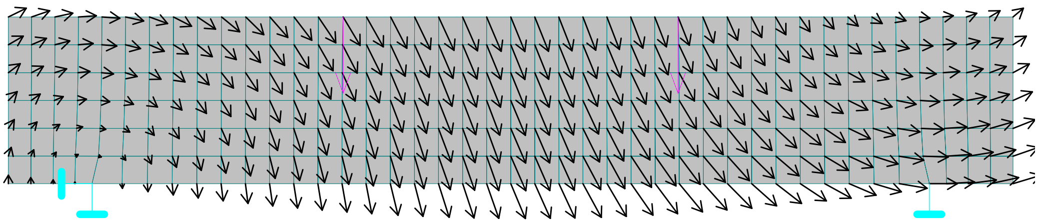


Fig. 8-26 Vector plot of displacements.

(The used iteration criterion error was 0.01). Note, that partial (non-compact) forces reflect forces in element groups and thus need not to show global equilibrium.

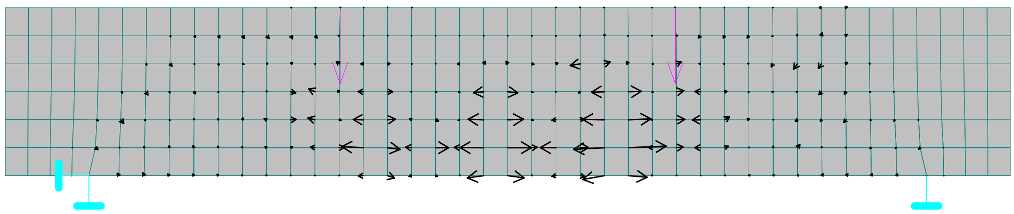


Fig. 8-27 Vector plot of residual forces.

8.9 Tensors

type of display

Tensors display menu can be opened by selecting the tab Scalars . averaging f scalar graphics can be defined in the menu as shown in Fig. 8-28.

variable

Type of display:

no graphics – Tensors are not displayed.
show – Tensors are displayed graphically.
show and label – Tensors are displayed with labels.

Averaging:

in nodes – average values in nodes are displayed.
in integration points – values in integration points are displayed.

Variable:

Principal engineering strain – Tensor of principal engineering strain.
Principal stress – Tensor of principal stress.

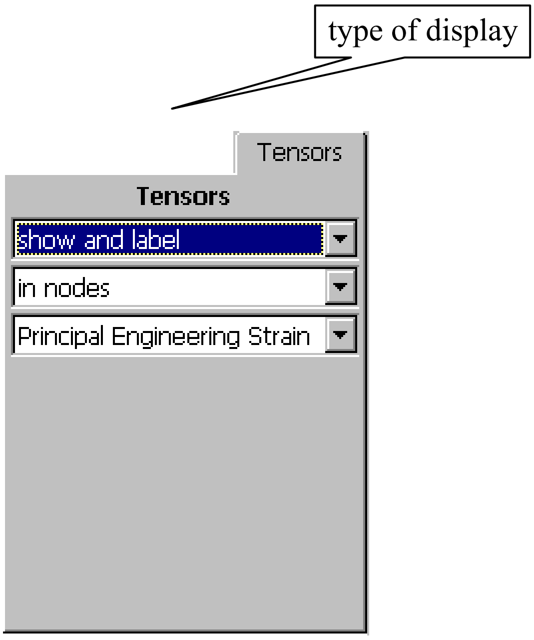


Fig. 8-28 Tensor details.

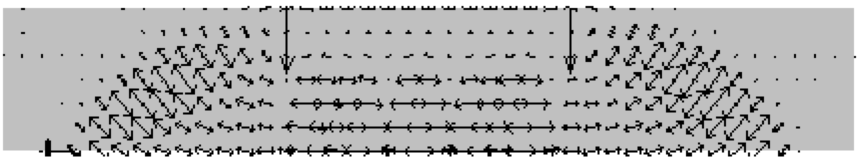


Fig. 8-29 Engineering principal strain tensors. Average values in nodes.

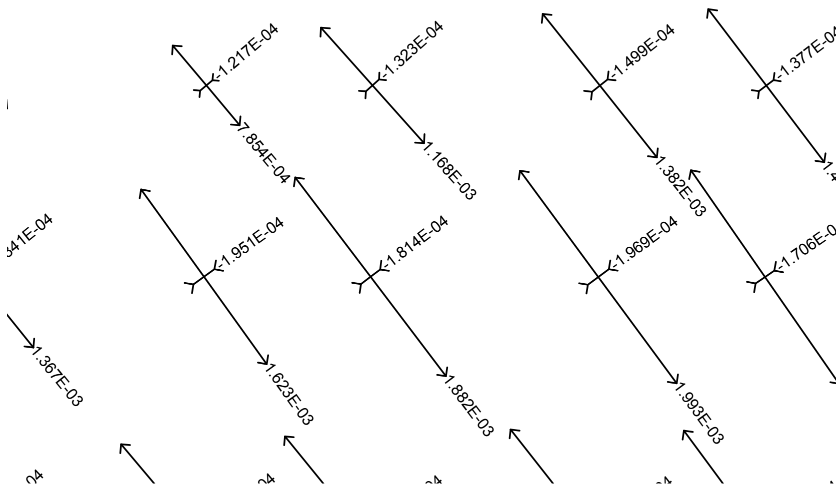


Fig. 8-30 Engineering principal strain tensors in integration points with labels.

The tensor graphics shows principal directions and magnitudes. Example of tensor display is shown in Fig. 8-29. Principal engineering strains are given in mesh nodes. Average values of adjacent element nodes are calculated. Not averaged values can be displayed in integration points. Example of display in integration points with labels is shown in Fig. 8-30. The mesh is not included in the picture.