ČERVENKA
CONSULTING

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

ATENA Program Documentation
Part 4-1

Tutorial for Program ATENA 2D

Written by:

Jan Červenka

Prague, June 2, 2015

CONTENTS

1. INTRODUCTION

This tutorial provides a basic introduction to the usage of the program **ATENA 2D** and it is specifically targeted to **ATENA 2D** beginners. This tutorial contains a step by step explanation how to perform a non-linear analysis on an example problem of a reinforced beam without smeared reinforcement. The geometrical and material properties correspond to the experimental setup by Leonhard in 1962. More details about the problem or experiment can be also obtained from the original report [5] or from the program developer or distributor.

The step by step demonstration is performed on an example of simply supported beam, which is loaded by two loads as it is shown in Figure 1. The problem is symmetric around its vertical axis, therefore, only one symmetric half of the beam will be analyzed.

The steps necessary for the data preparation, non-linear analysis and post-processing are depicted on subsequent figures, which show the computer screen for each step and user action. There is always also a short description for each figure. In the post-processing stage only some basic post-processing methods are described. **ATENA** offers many options for viewing results from FE non-linear analysis. These options can be easily accessed from the post-processing window by self-explanatory buttons and toolbars. For more details, it is recommended to consult the **ATENA 2D** user's manual or the hotline desk at the program distributor or developer.

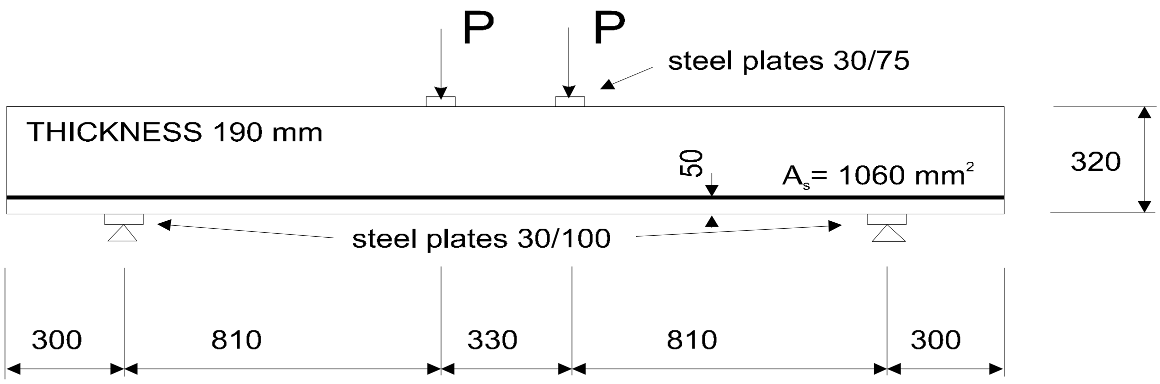


Figure 1: Geometry of the structure.

2. STARTING PROGRAM

The program can be conveniently started from **Start | Programs** menu on your computer desktop. You can double click any **.CC2** model file to open it in **ATENA 2D** (making use of the extension association in the system). Alternatively, you can also directly execute the program **CCAtena2D2_en.exe** (english version; run **CCAtena2D2_cs.exe** for the czech version) from the directory where the program is installed.

3. PRE-PROCESSING

3.1 Introduction

This chapter explains the basic steps, which are to be performed in order to define a complete geometrical and then a finite element model for non-linear FE analysis by **ATENA**. The purpose of the geometrical model is to describe the geometry of the structure, its material properties and boundary conditions. The analytical model for the finite element analysis will be created during the pre-processing with the help of the fully automated mesh generator. The geometrical model is created in the following steps. First, geometrical joints are defined. These joints are later connected into boundary lines. It is possible to create straight, arc or circular lines. The subsequent step is to define macro-elements or regions, by specifying a list of boundary lines, which surround the macro-element.

Before starting the definition of the geometrical model it is a good idea to introduce the graphical user interface of **ATENA 2D** pre-processor. The pre-processing window is shown in the subsequent Figure 2.

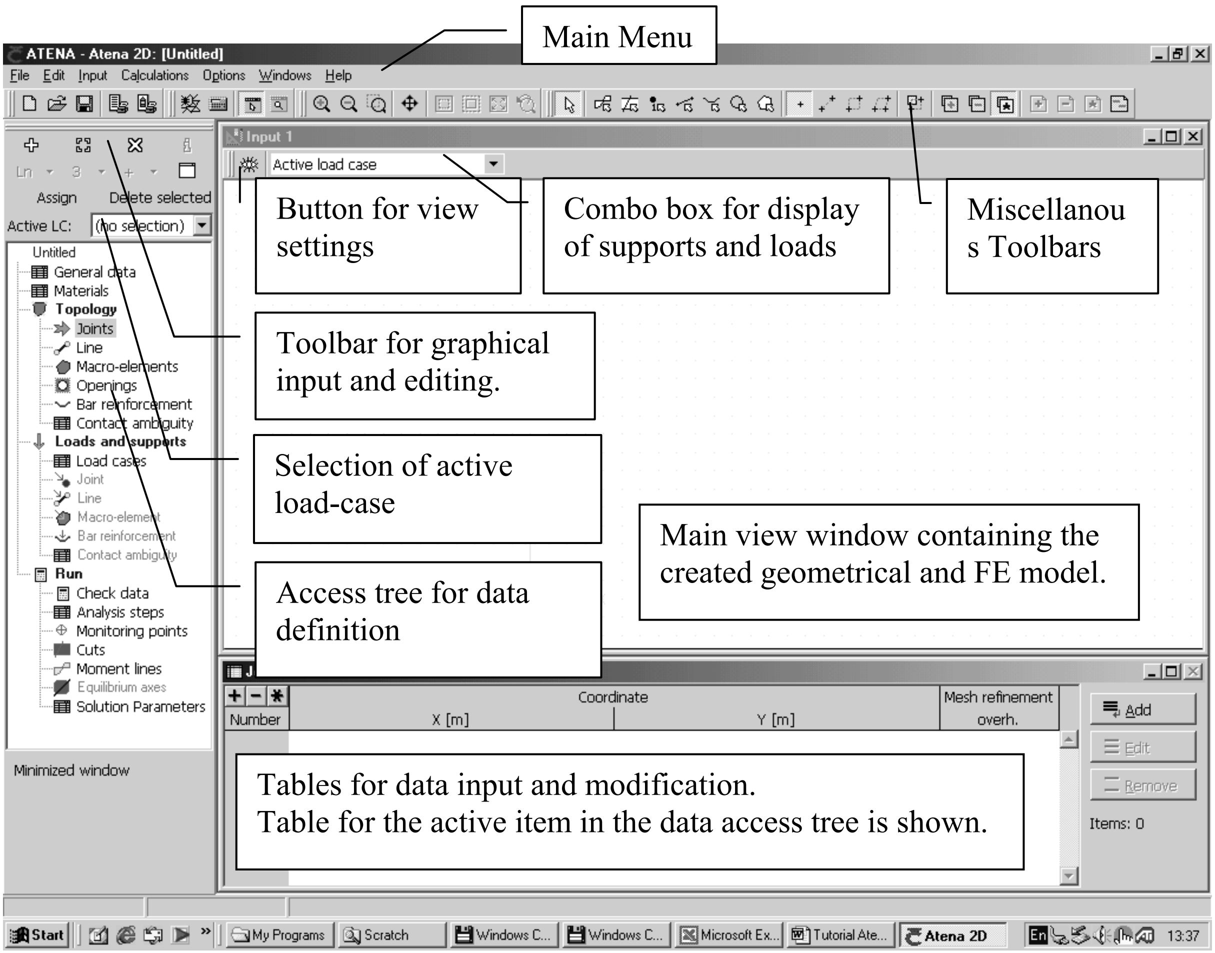
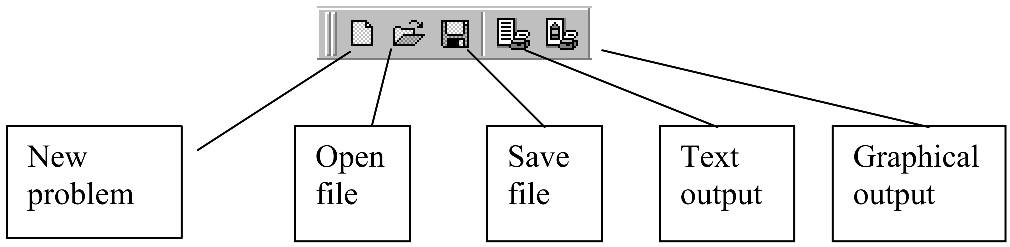


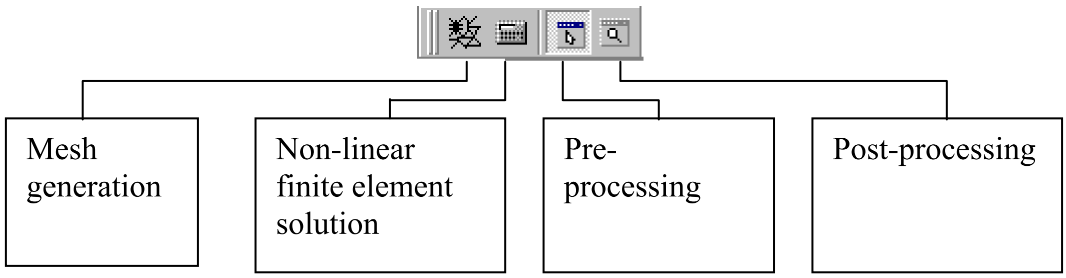
Figure 2: Graphical user interface of ATENA 2D pre-processor.

ATENA 2D contains four main toolbars:

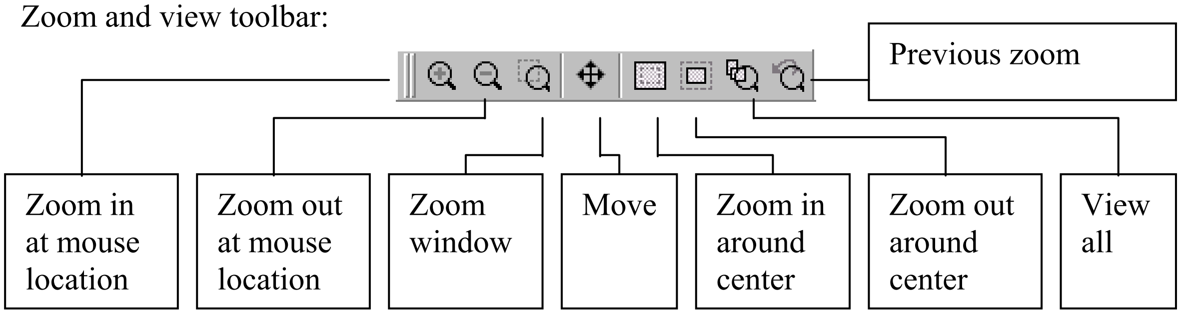
File toolbar:



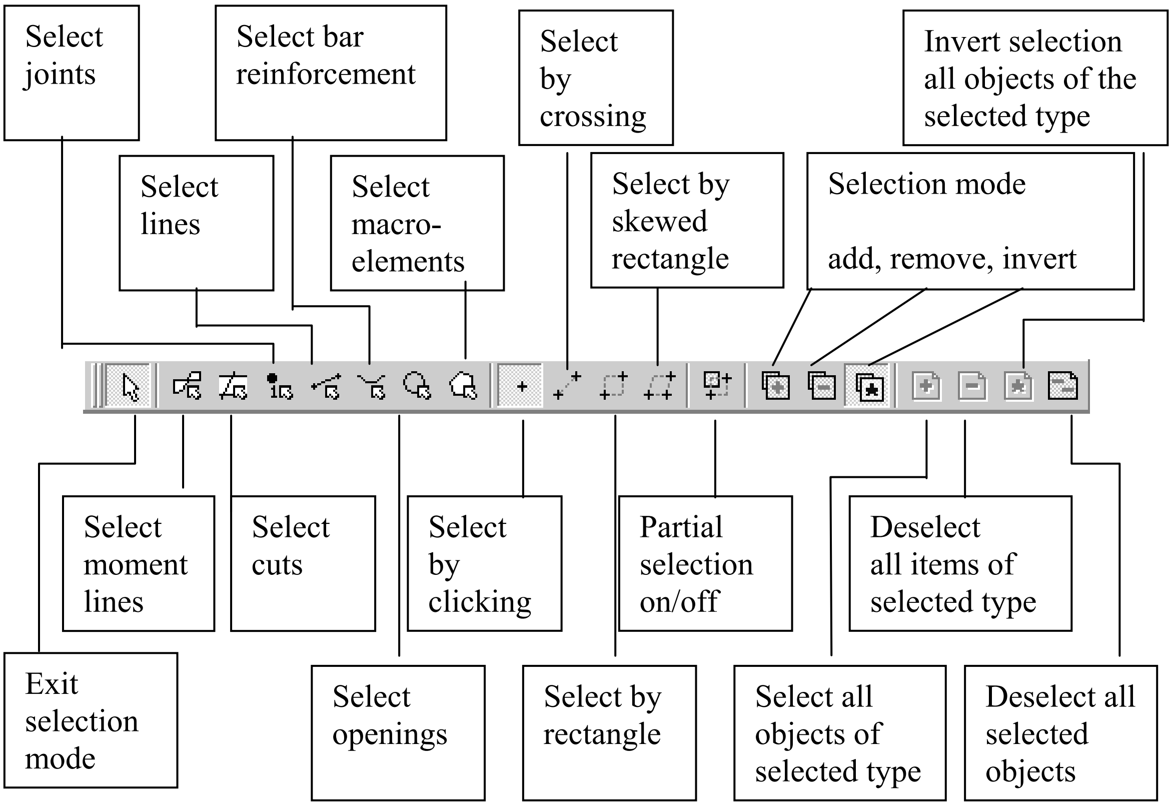
Solution toolbar:



Zoom and view toolbar:



Selection toolbar:



After examination of the user interface layout, it is possible to start with the definition of the geometrical model of the analyzed structure. It is a good practice to provide a short description of the problem to be analyzed. In **ATENA 2D** this can be done be selecting the **General data** item in the data access tree. This opens the following table in the table window.

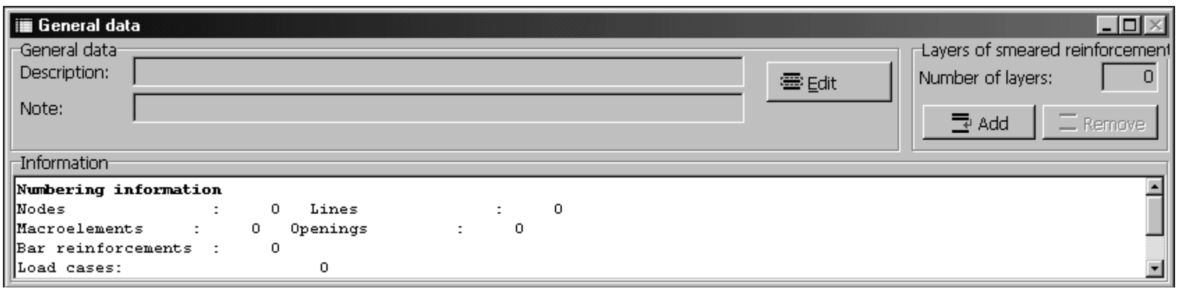


Figure 3: General data table shows general information about the structure.

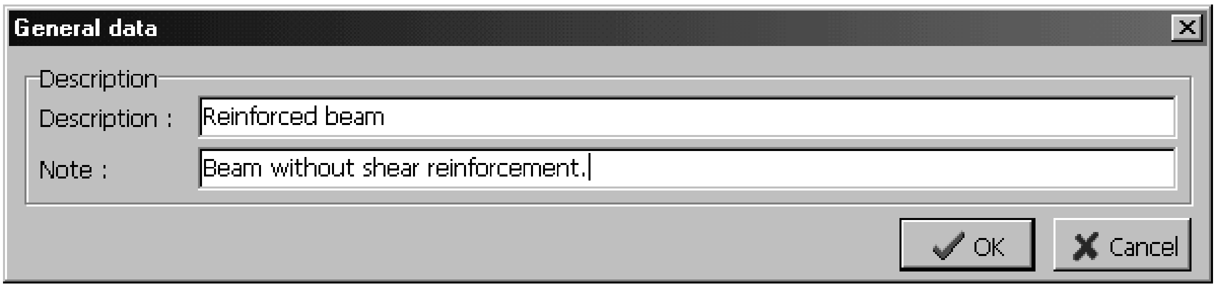


Figure 4: The editing dialog for general data appears after selecting the Edit button from the General data table.

3.2 Material Parameters

Next step should be the definition of material groups and material properties. Selecting the item **Materials** from the data access tree opens the **Materials** table.

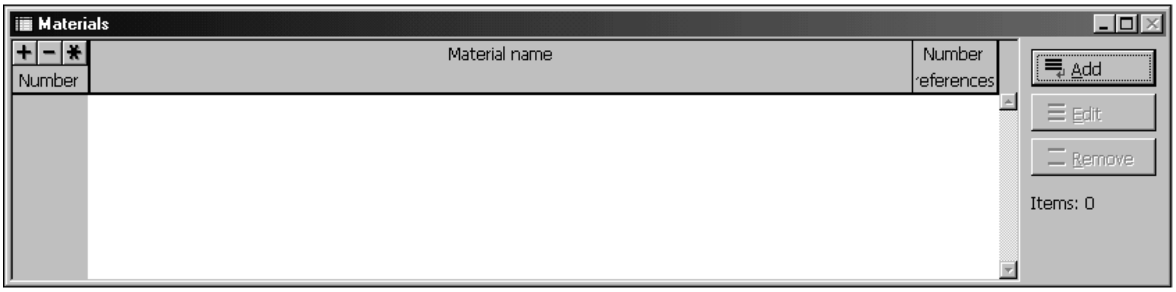


Figure 5: The Materials table, from which new materials can be added or existing materials can be modified or removed.

Clicking the **Add** button on the material table window creates a new material. For the current problem it is necessary to define three material types: one plane stress elastic material for the steel plates at support and loading points, concrete material for the beam and reinforcement material.

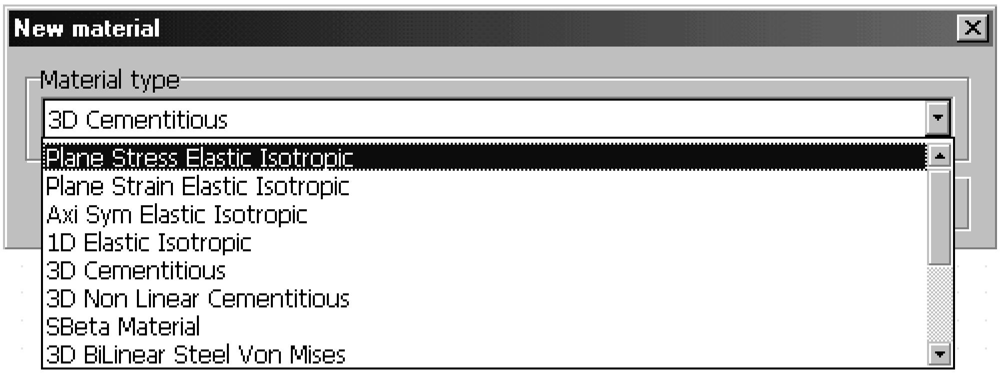


Figure 6: Selection of plane stress elastic isotropic material for the steel plates.

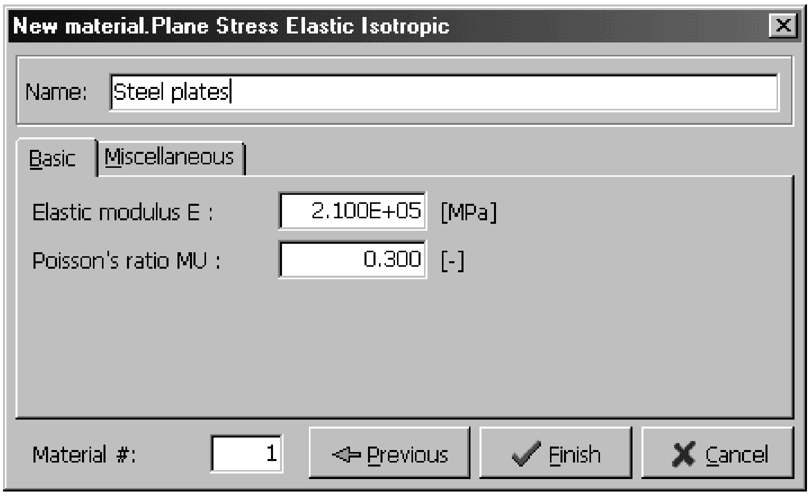


Figure 7: The dialog for the definition of material properties for the steel plates.

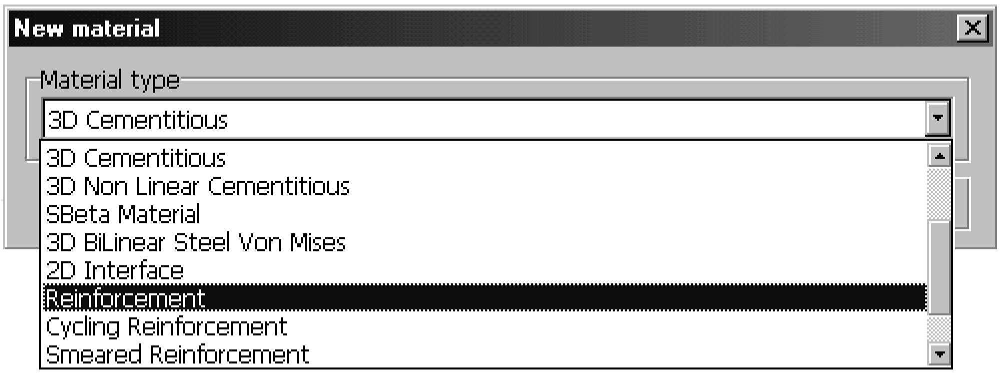


Figure 8: Selection of material model for the bar reinforcement.

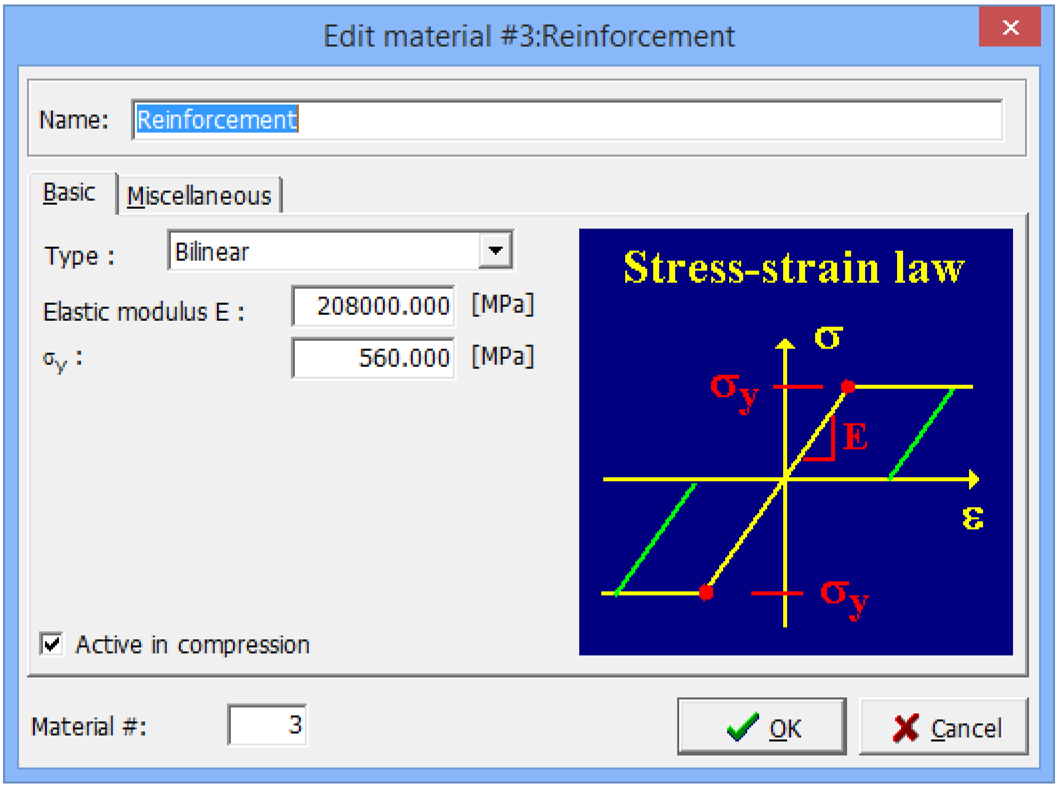


Figure 9: The dialog for the definition of reinforcement material parameters. The bi-linear elastic-perfectly plastic stress-strain diagram is selected for this problem.

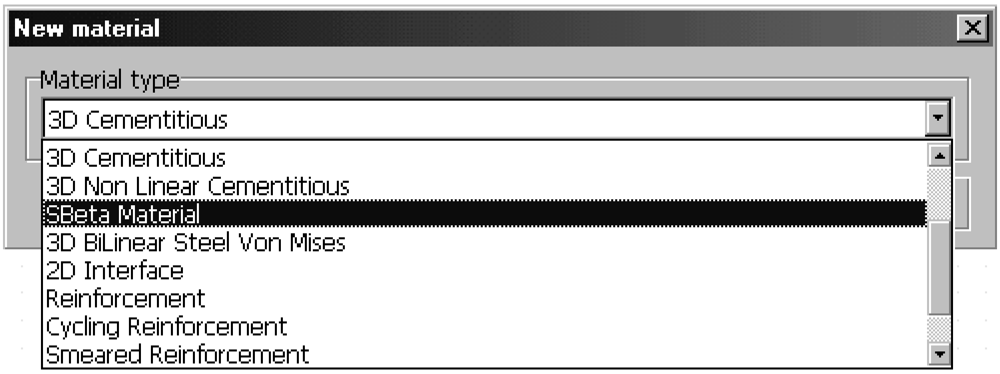


Figure 10: Selection of SBETA material model for the concrete beam. The SBETA model corresponds to the material formulation, which was implemented in the program SBETA. SBETA was a previous DOS version of ATENA.

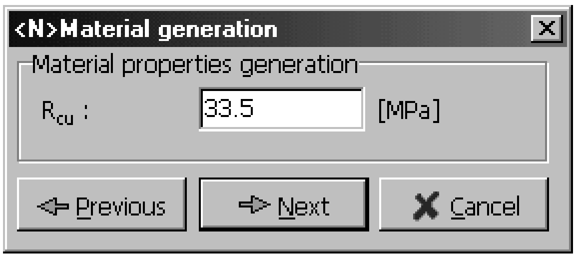


Figure 11: Default values of material parameters are generated based on the cube strength of concrete. For this case, the cube strength should be 33.5 MPa.

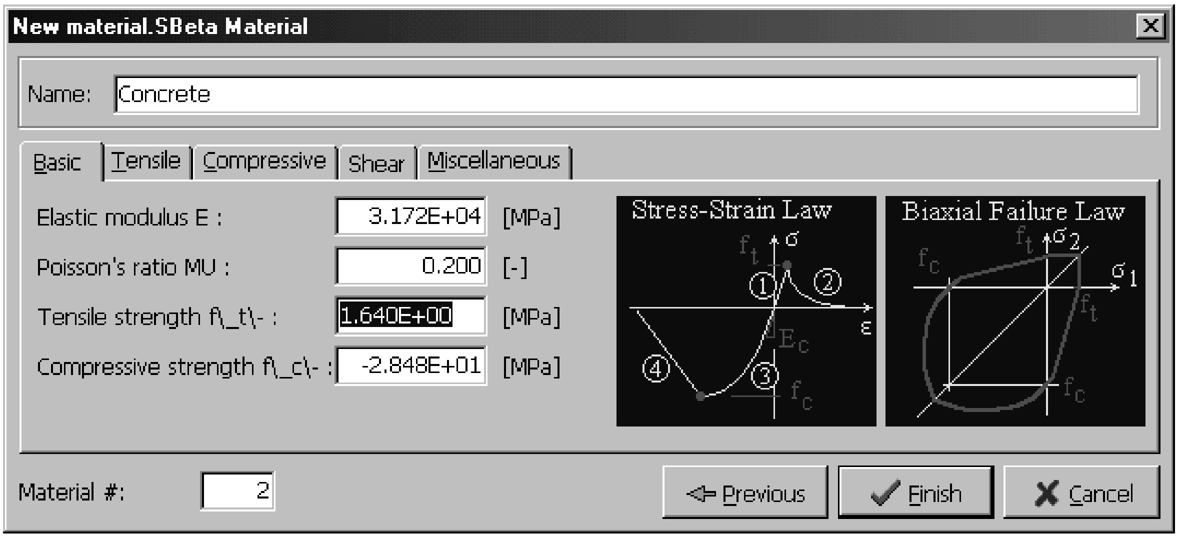


Figure 12: The dialog window for the definition of basic properties for SBETA material. The parameters were generated based on the concrete cube strength. The tensile strength should be edited to 1.64 MPa for the Leonhard's beam.

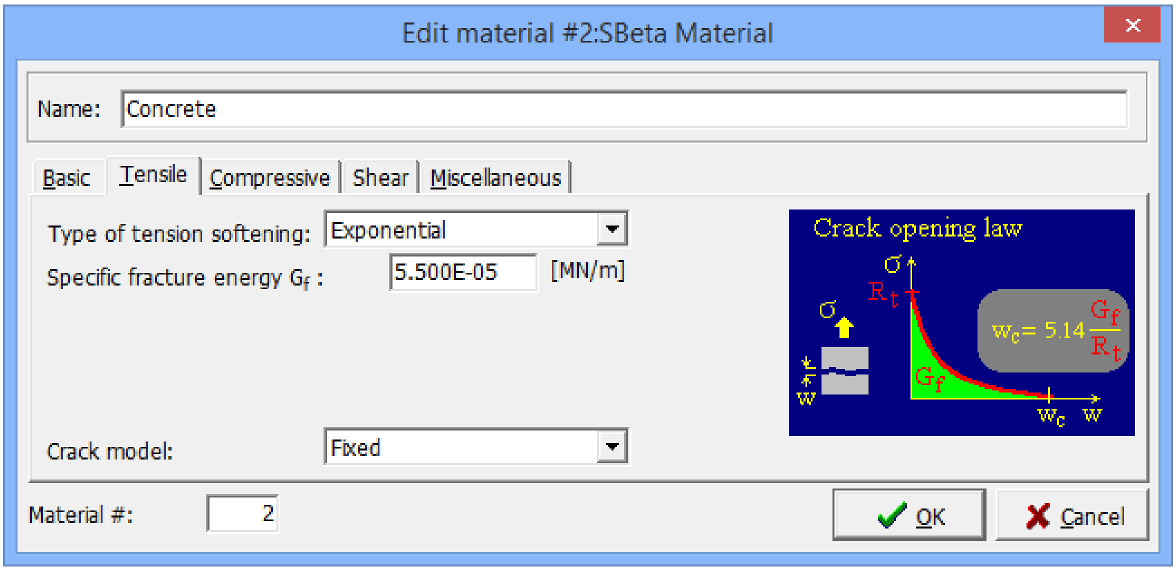


Figure 13: The dialog window for the tensile properties for SBETA material. The fracture energy should be edited to 5.5e-5MN/m.

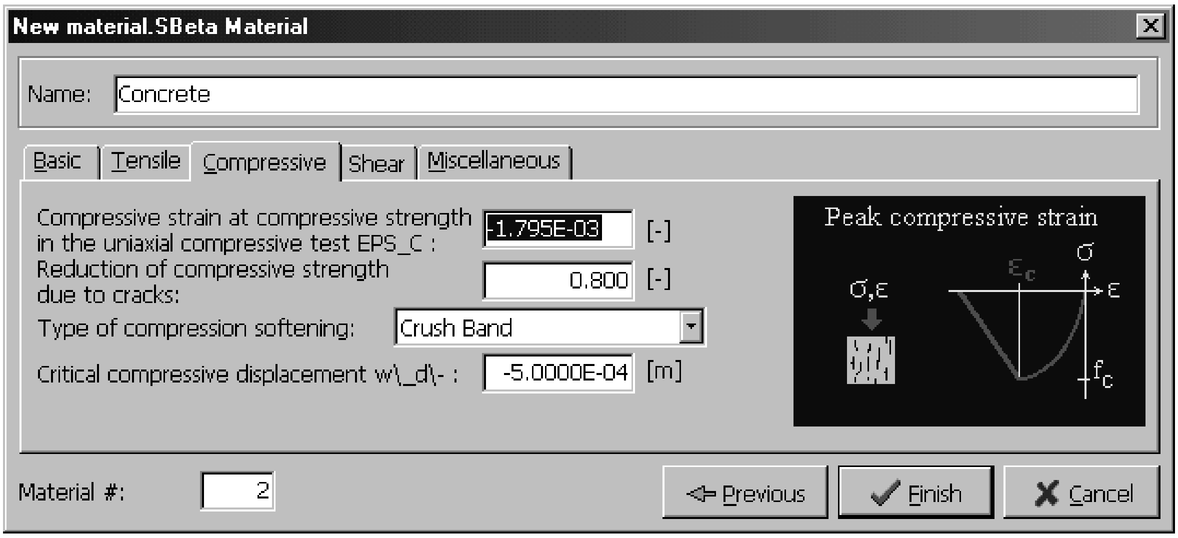


Figure 14: The dialog window for the compressive properties of SBETA material.

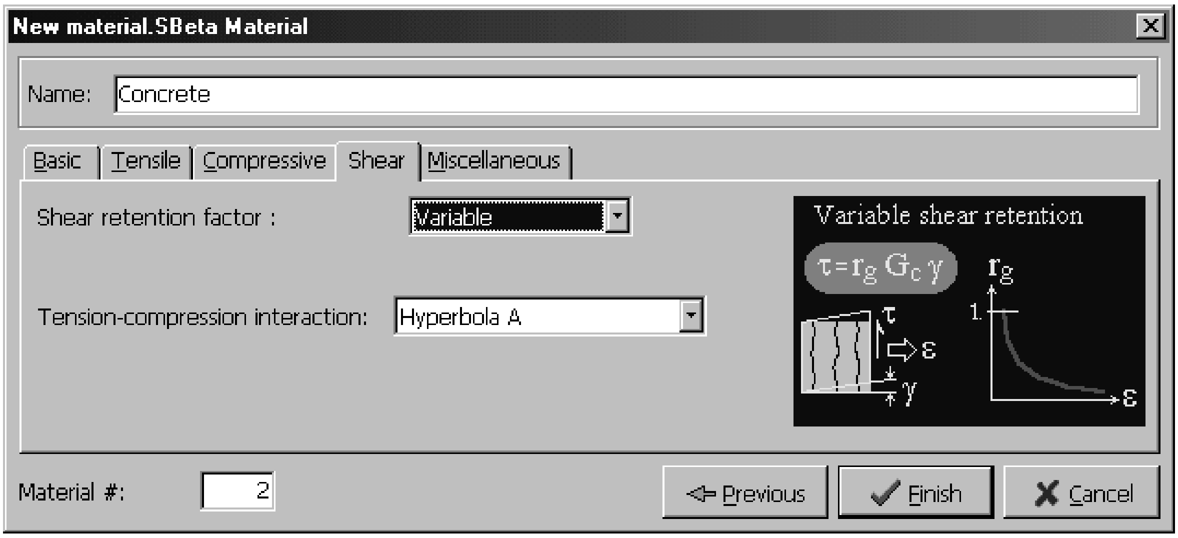


Figure 15: The dialog window for the shear properties of SBETA material.

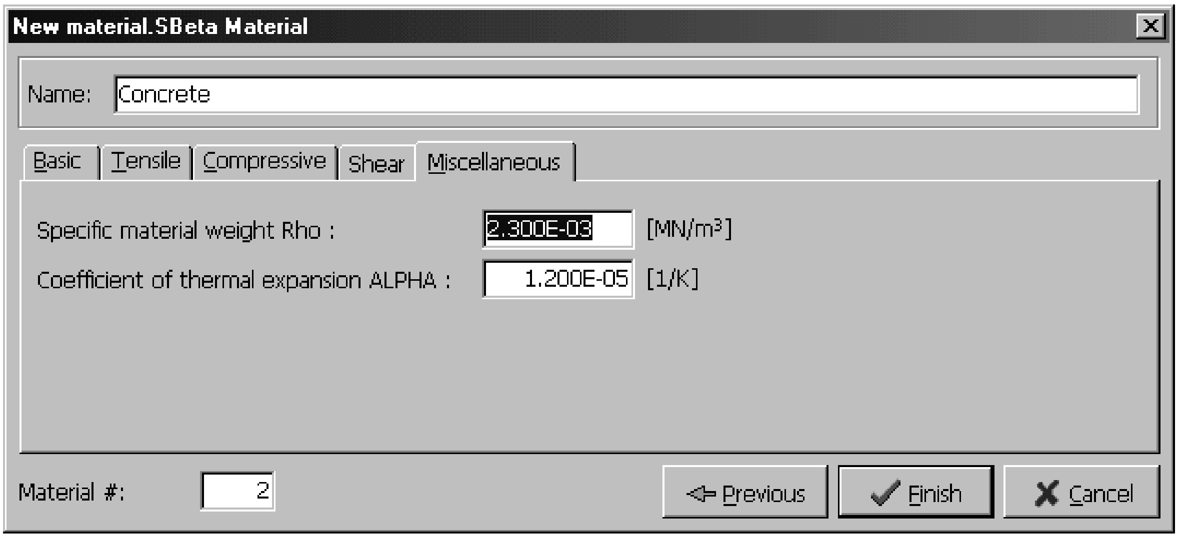


Figure 16: The dialog window for the miscellaneous properties of SBETA material.

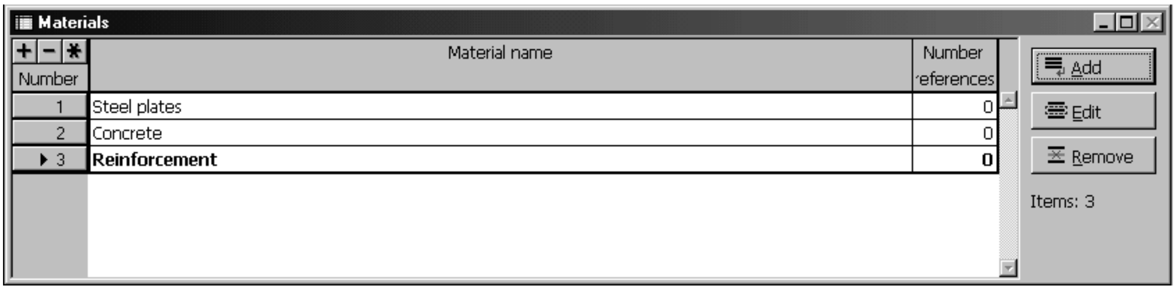


Figure 17: The three materials, which were defined previously, can be viewed or modified from the Material table window.

3.3 Geometrical Joints

Next step in the input data preparation should be the definition of geometrical joints. The geometrical joints will be later connected to geometrical lines and macro-elements (i.e. regions). Selecting the appropriate item (i.e. **Joints**) in the access data tree can start the definition of geometrical joints. After that it is possible to continue in two ways: either by selecting the button , which will be followed by mouse picking at new joint locations, or by clicking the **Add** button in the **Joints table** window.

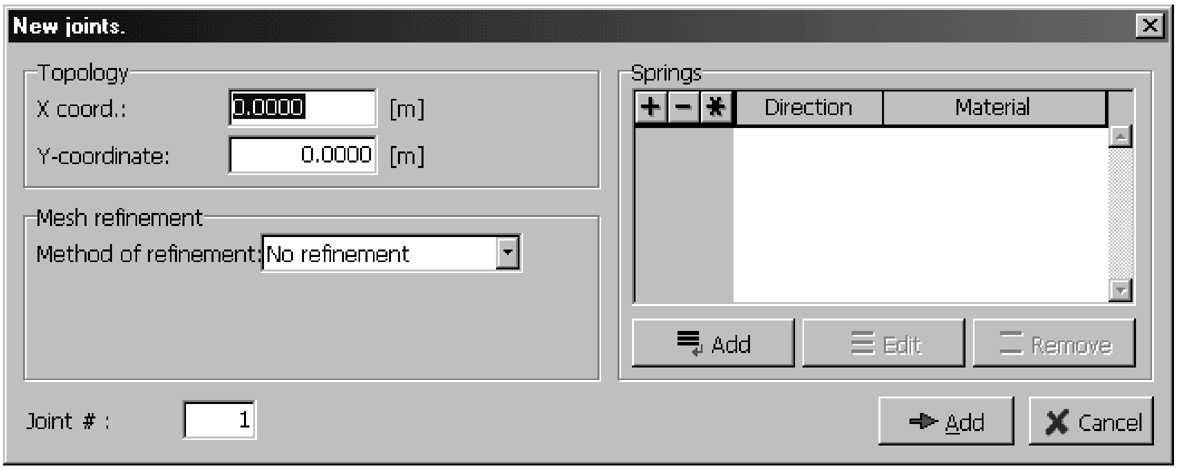


Figure 18: The dialog for specifying the coordinates and properties for the newly created joints.

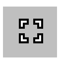
Table 1 contains the coordinates for the geometrical joints, which are necessary to fully define the geometry of the Leonhard's shear beam.

Table 1: Coordinates of the geometrical joints.

Joint Num.	Coordinate x [m]	Coordinate y [m]
1	0.0000	0.0000
2	0.0000	0.3200
3	0.2500	-0.0300
4	0.2500	0.0000
5	0.3000	-0.0300
6	0.3500	-0.0300
7	0.3500	0.0000
8	1.0725	0.3200
9	1.0725	0.3500
10	1.1100	0.3500
11	1.1475	0.3500
12	1.1475	0.3200
13	1.2750	0.0000
14	1.2750	0.3200

In case a typing mistake is made during the input of coordinates, it is possible to edit wrong geometrical joints. There are two possibilities to access joint coordinates and other properties.

The first possibility is to use the **Joints table** window. In this case, the geometrical joint to be edited is selected by double-clicking on it in the table or using the **Edit** button.

The other possibility is to select the joint in the window containing the view of the structure. In this method, the **Joint** item in the data access tree should be highlighted, and the edit button  must be selected from the toolbar for graphical input and editing. Then geometrical joint properties can be modified just by clicking at an appropriate joint. The same philosophy can be used to edit other geometrical entities as lines, macro-elements and reinforcement bars.

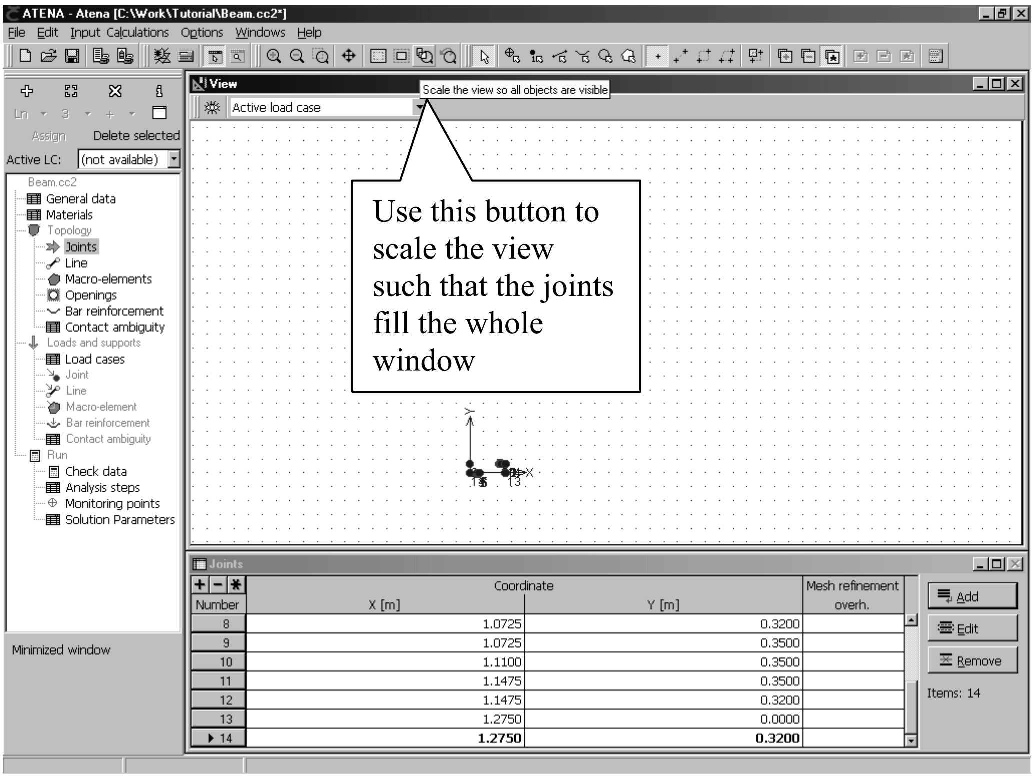


Figure 19: The program window after the definition of all geometrical joints.

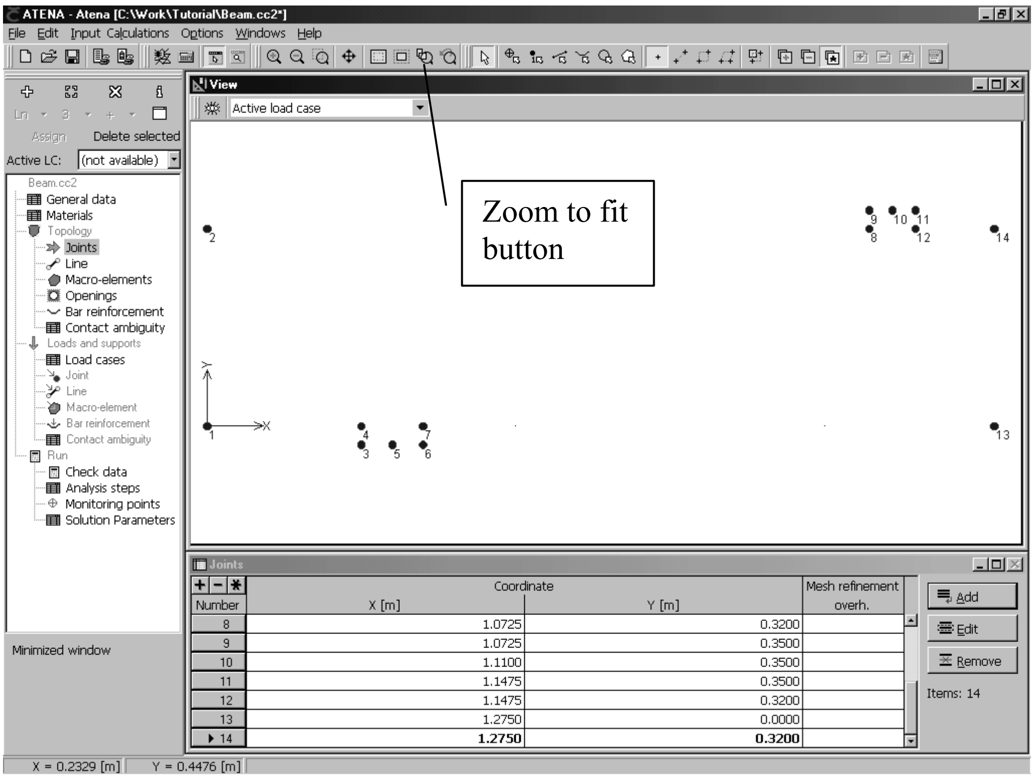


Figure 20: The program window after the selection of the zoom to fit button.

3.4 Geometrical Lines

After the definition of the geometrical joints, it is possible to proceed with the definition of geometrical lines, which will connect the previously specified joints.

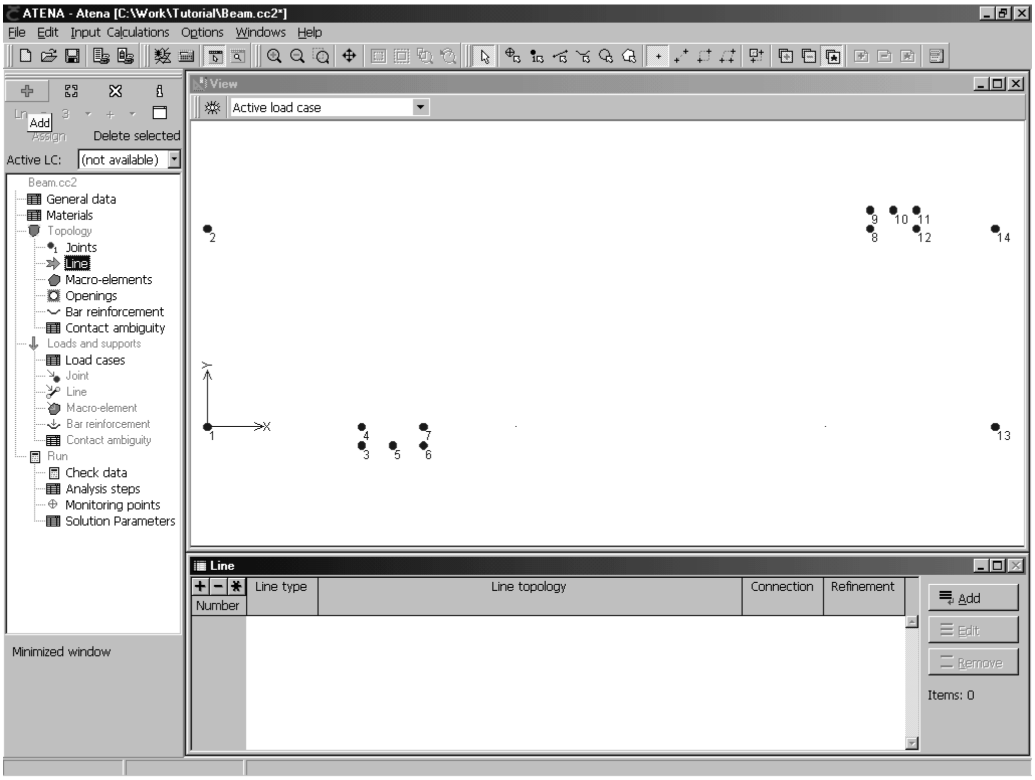


Figure 21: The definition of geometrical lines begins by selecting the Line item in the data access tree. The graphical definition of geometrical lines starts by clicking the button ✚.

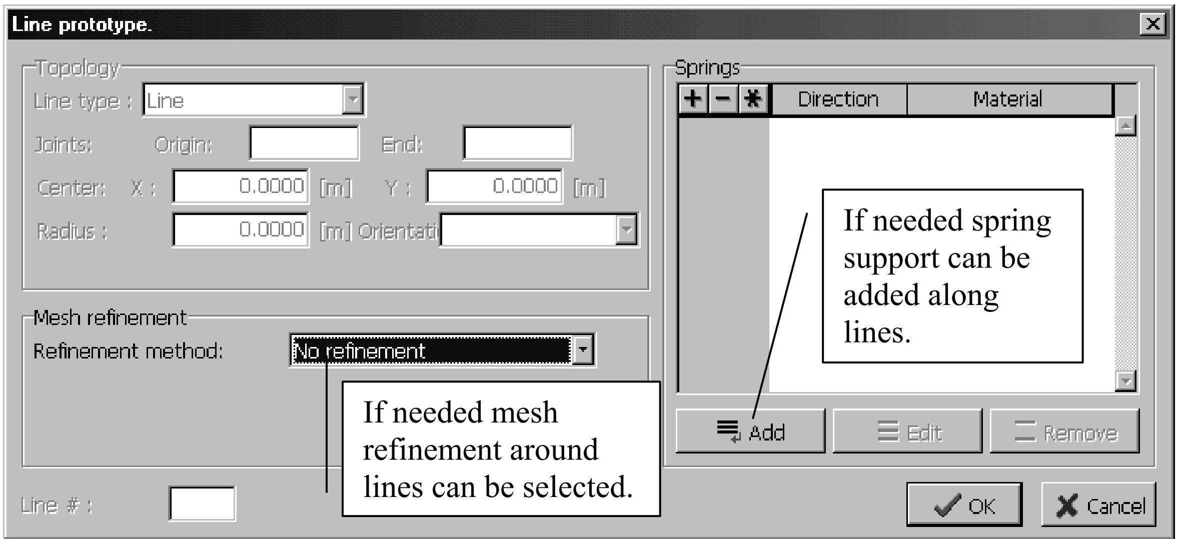


Figure 22: The line prototype dialog box appears after clicking the button ✚. In this dialog a mesh refinement method or line springs can be specified. All subsequently created lines will use this set of prototype properties.

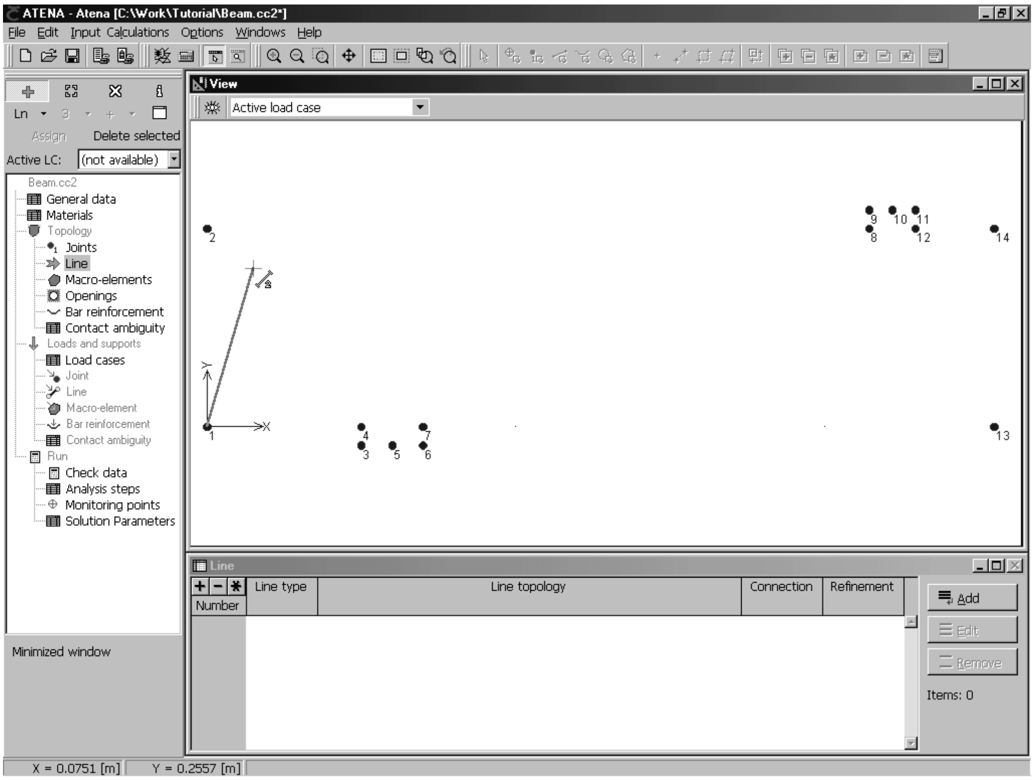


Figure 23: In the graphical mode, a geometrical line is defined by first selecting a line beginning and a line end joint by mouse. The order of end points is not important in ATENA.

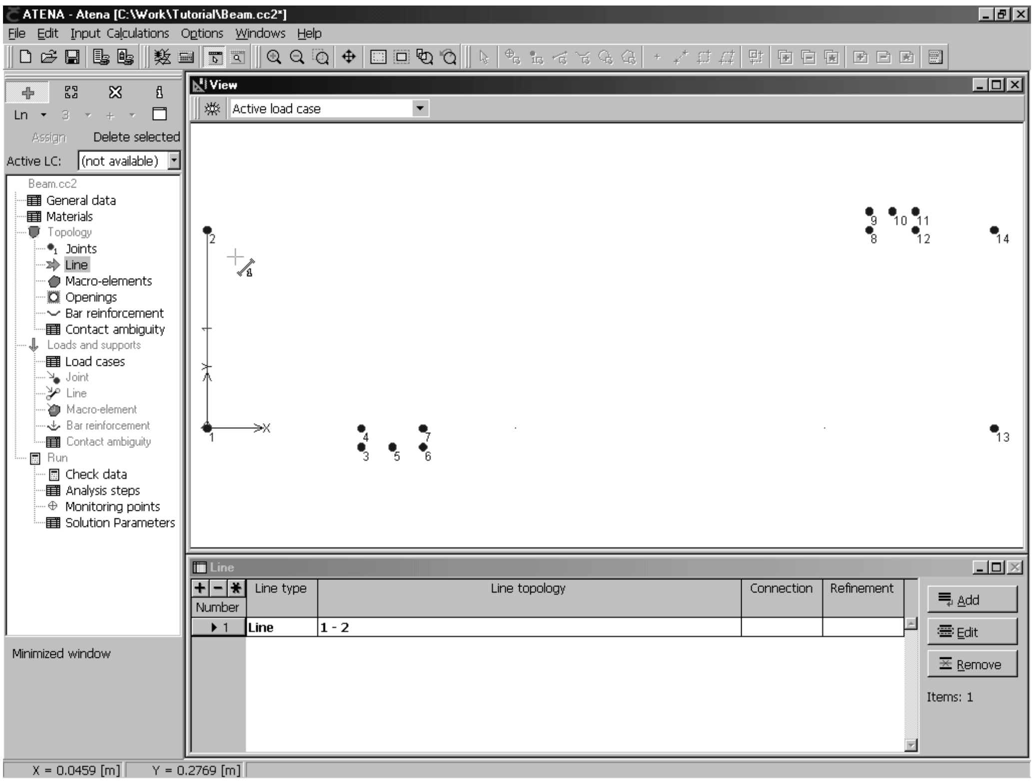


Figure 24: Program display after the definition of the first boundary line.

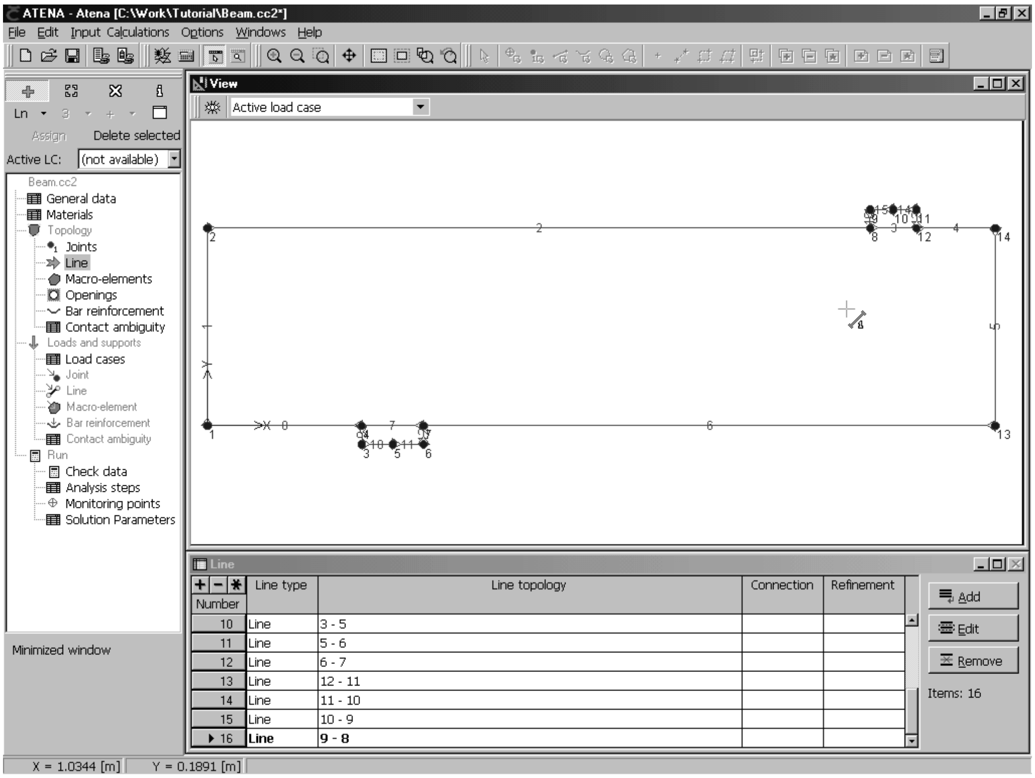
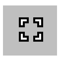
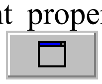


Figure 25: The program display after the definition of all geometrical lines.

3.5 Geometrical Macro-elements

After the definition of geometrical lines, the next step is to connect these lines to form regions. In **ATENA 2D** regions are called macro-elements. The regions can be again defined in two ways: either from the **Macro-element table** window by selecting the button **Edit** and by providing a list of boundary lines, or graphically using the mouse to select the boundary lines for macro-elements.

The second and more convenient approach starts by highlighting the item **Macro-elements** in the data access tree (see Figure 26). Then the button  should be selected. After that a dialog window, which is shown in Figure 27, appears for the specification of macro-element properties. These properties will be used in the subsequently created regions. We will start with the definition of regions for the steel plates that are located at the loading point and at the vertical supports. Mouse clicking selects lines that form a macro-element. It is possible to note that the shape of the mouse pointer changes when it is close to a particular line.

The button  can be used to edit macro-element properties. The button  is for removing macro-elements, and the buttons  and  for getting information and setting new prototype properties respectively.

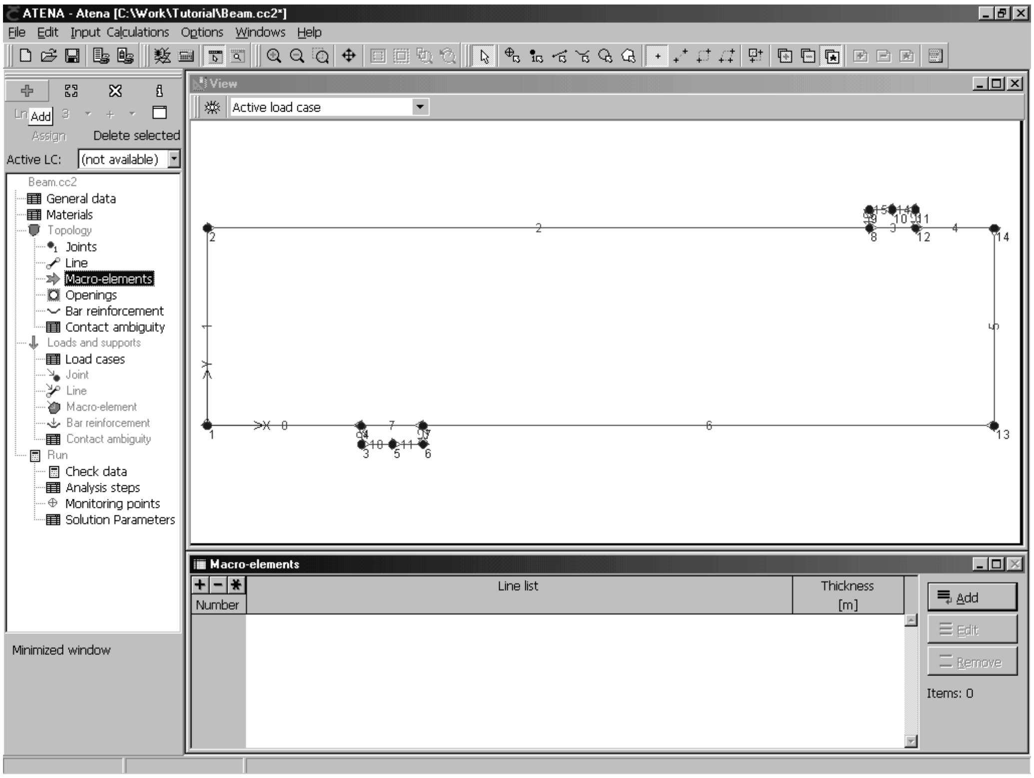
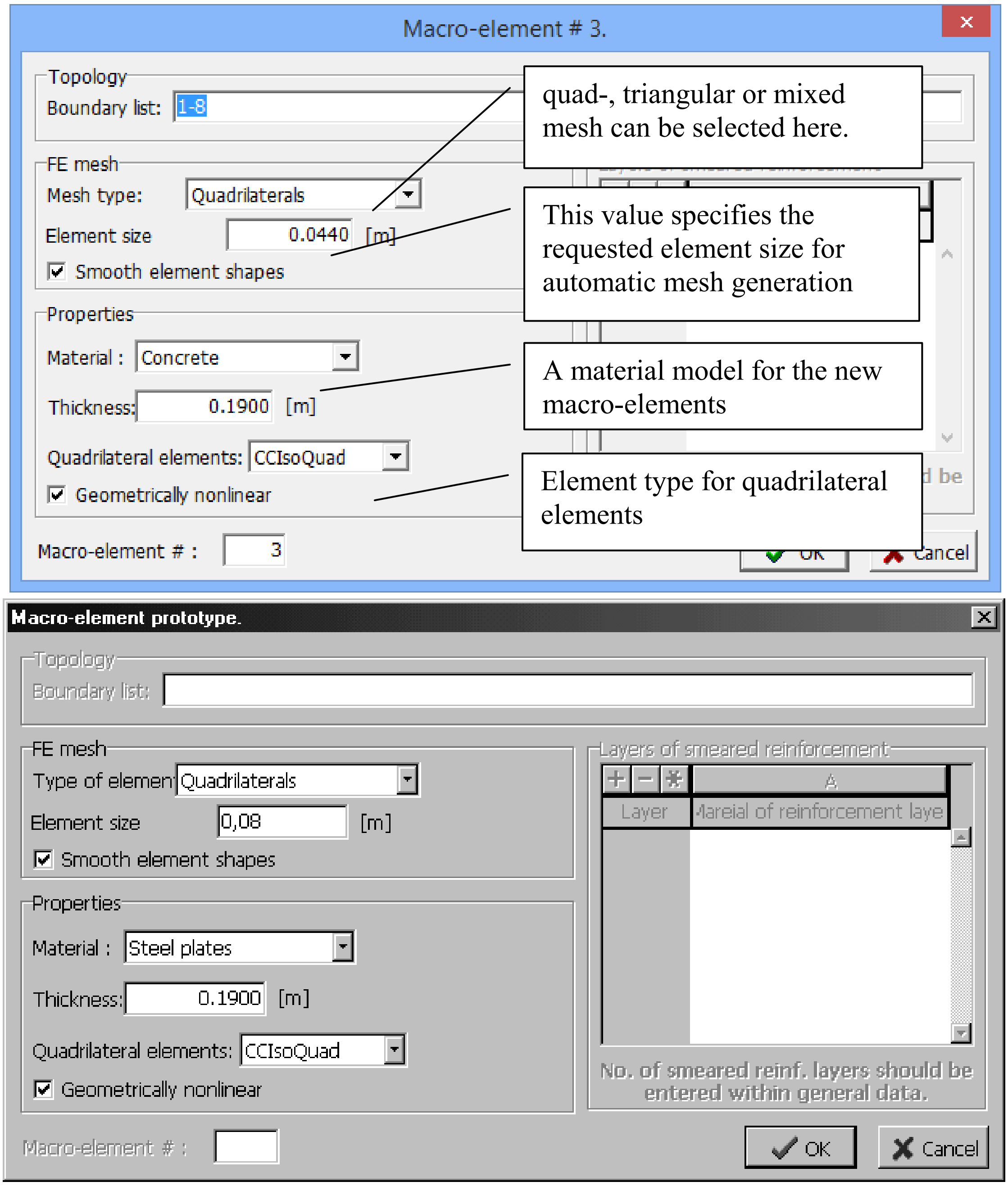


Figure 26: Program display at the beginning of the macro-element definition.



(a)

(b)

Figure 27: The dialog window, which appears after the selection of the button from the toolbar for graphical input and editing. This dialog is used for the definition of macro-element prototype, the properties of which will be used for the subsequently created macro-elements. In this case we will start with the definition of regions (i.e. macro-elements) for the supporting steel plates. The mesh size of 4.4cm (a) gives 8 elements per thickness, for DEMO version you can run with 8cm (b) to prevent exceeding the limited number of elements.

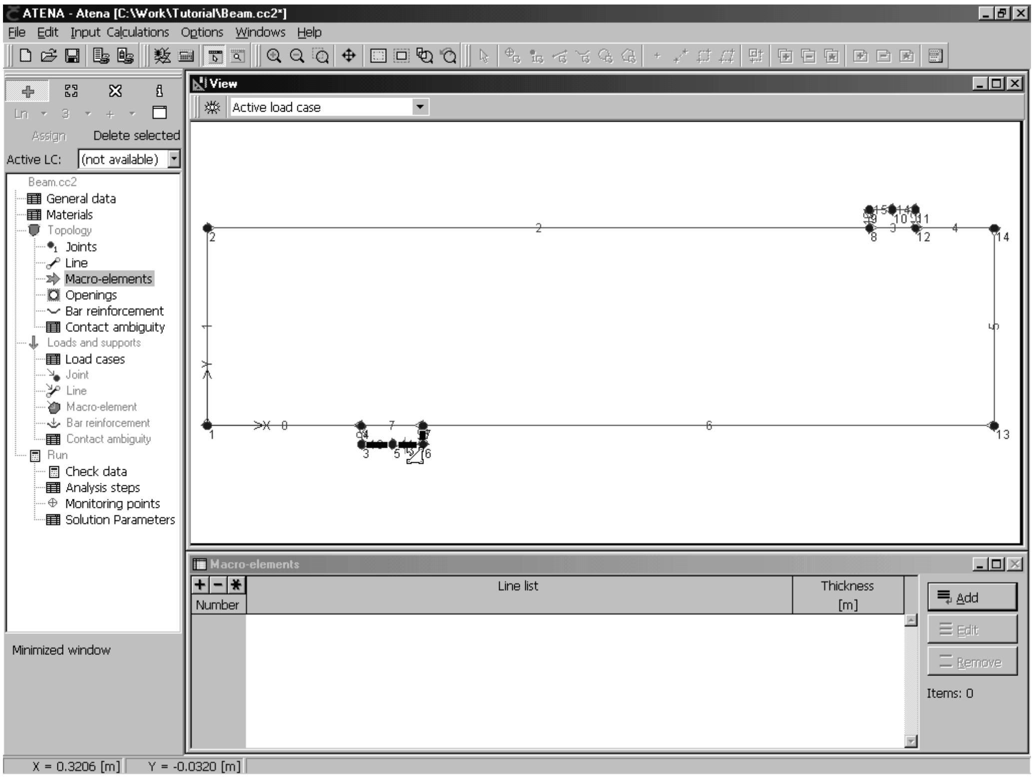


Figure 28: Selection of boundary lines for first macro-element representing the steel plate at the vertical support.

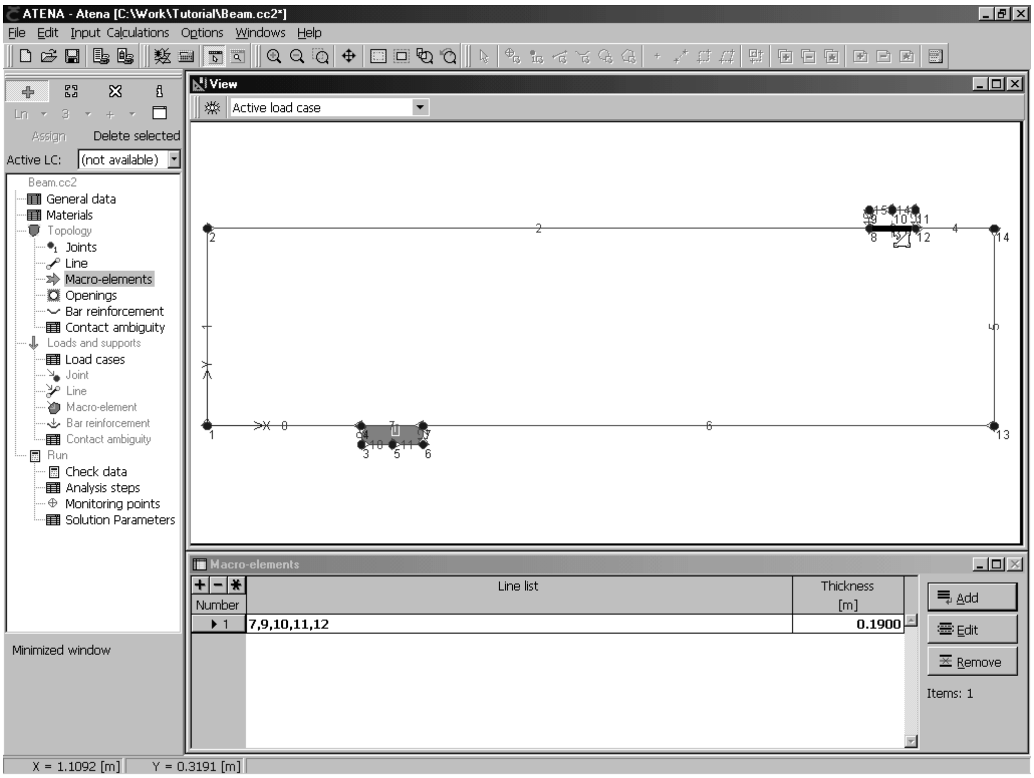
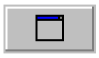


Figure 29: Program display after the first steel plate is defined and during the process of creating the second steel plate, which is located at the point of load application.

After the definition of macro-elements for the steel plates, it is necessary to change the macro-element prototype properties, since for the beam the concrete material is more suitable than the elastic isotropic one. Clicking the button  changes the prototype properties.

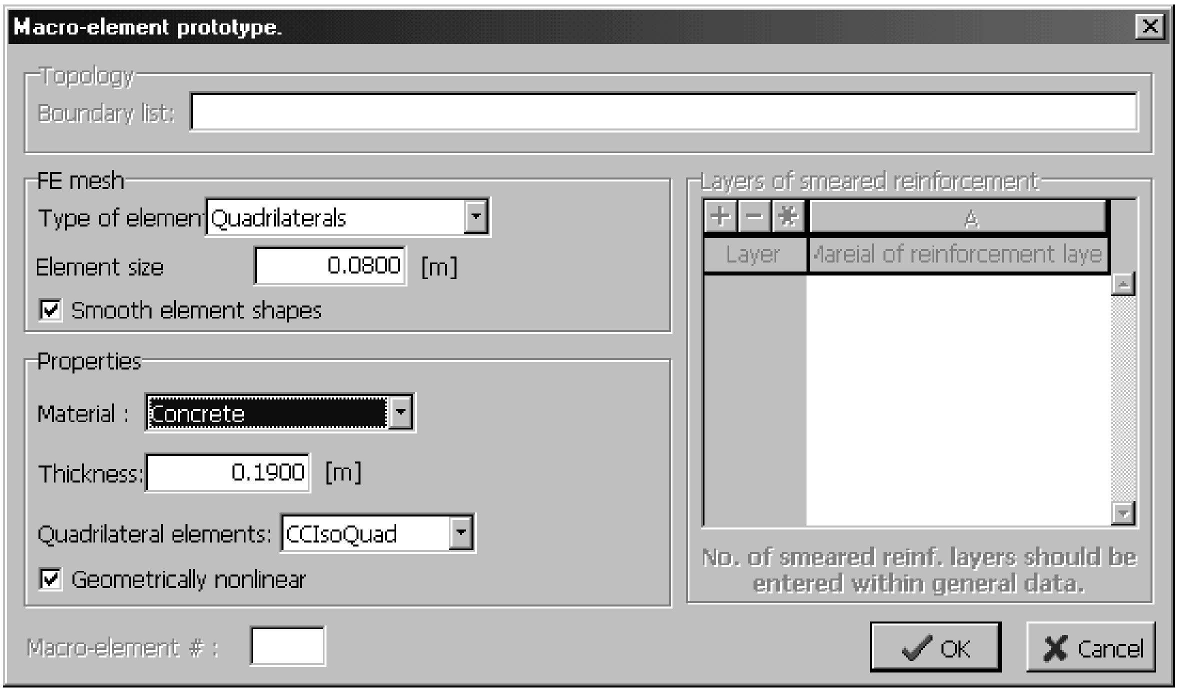


Figure 30: The dialog for changing the macro-element prototype properties for the beam region, where concrete material model should be used.

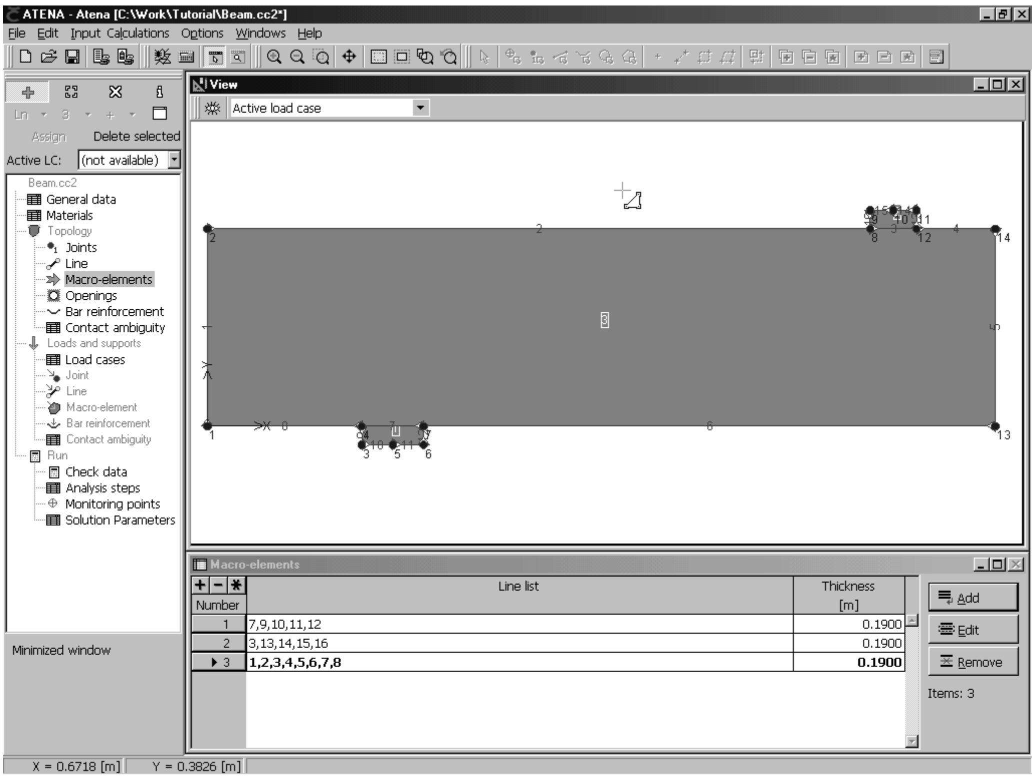
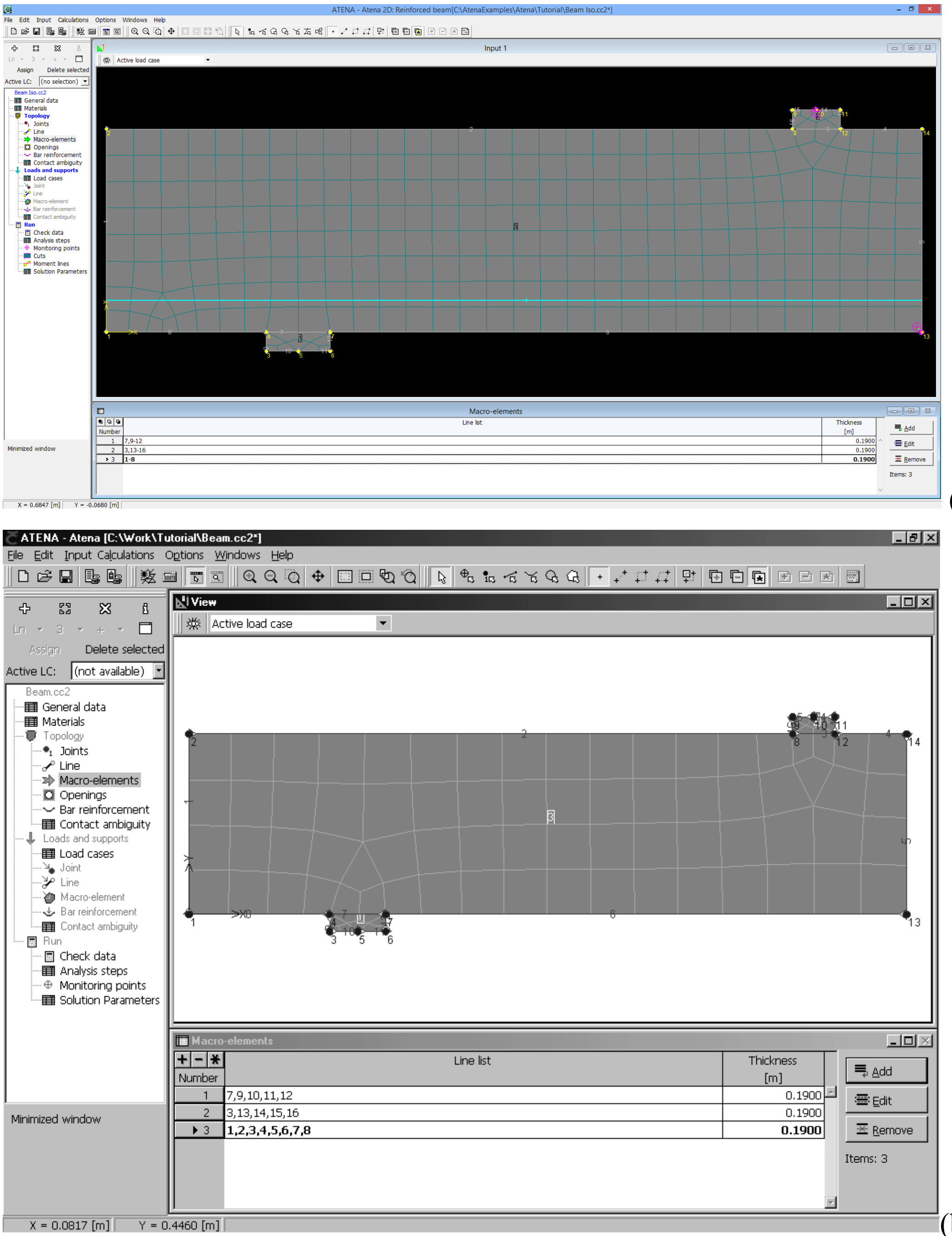


Figure 31: The program display after the definition of the last macro-element with the concrete material.

3.6 Mesh Generation

After the definition of macro-elements is completed it is possible to start an automatic mesh generation. The automatic mesh generation is executed using the button . The mesh generator in **ATENA 2D** is fully automatic. Based on element sizes that are defined for each macro-element a finite element mesh is generated automatically. The created mesh size can be controlled by local refinements around geometrical lines and joints. It is useful to note that when the generator recognizes that the macro-element is composed of four edges with same number of divisions along the opposite edges, it attempts to generate a mesh using the mapping technique. This feature can be useful in some cases when nice and uniform meshes are required. This feature of the mesh generator, however, is not exploited in this example, and we rely on the capabilities of the fully automated meshing.



(a)

(b)

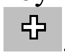
Figure 32: Generated finite element mesh using the element size of 0.044m (a) and 0.08 m (b).

3.6.1 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.

3.7 Bar-reinforcement

In the next step reinforcing bars will be defined. It should be noted that reinforcement bars can be defined any time during the input data preparation. It is not necessary to wait till the macro-elements are defined and mesh is generated. The reinforcement bar definition starts by highlighting the **Bar reinforcement** item in the data access tree (see Figure 33). Then it is again possible to define the bar geometry by mouse or by numerical values. The graphical input can be activated using the button .

In this example the numerical input is used, and it is started by the **Edit** button in the **Bar reinforcement** table window.

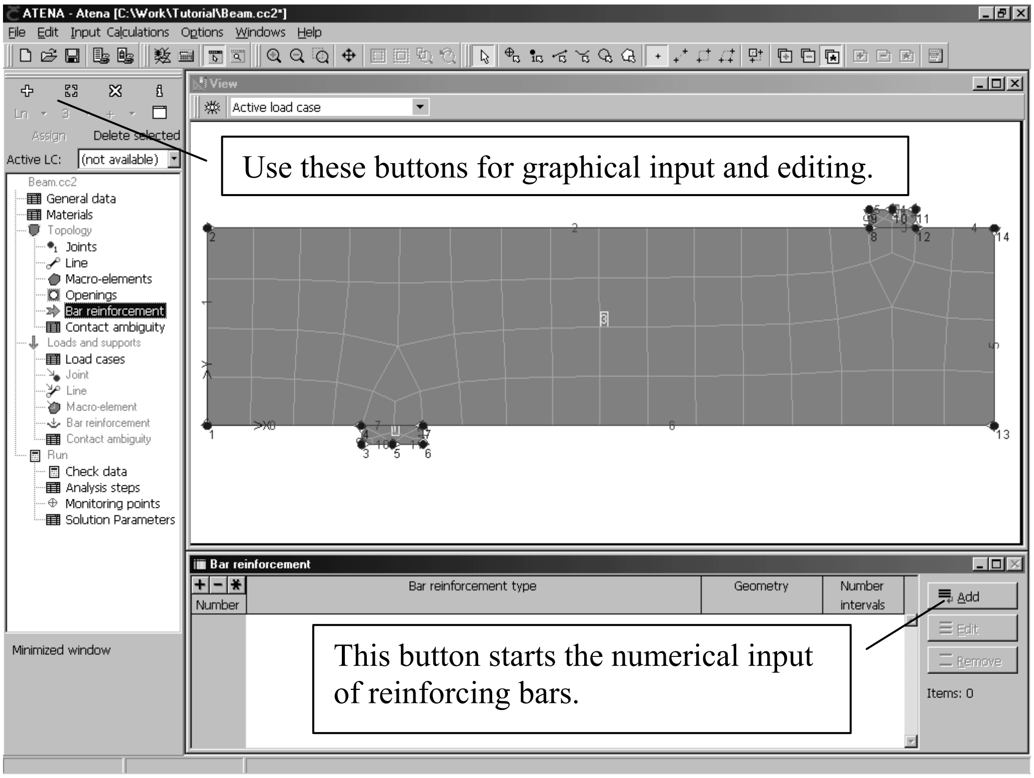


Figure 33: The program window at the beginning of the reinforcement bar definition.

In this example, there is only one reinforcing bar along the bottom side of the beam. The bar distance from the beam bottom surface is 0.05 m. In reality, this bar models two bars with diameter of 26 mm. The steps necessary to create a new reinforcement bar in **ATENA 2D** are documented in the subsequent figures.

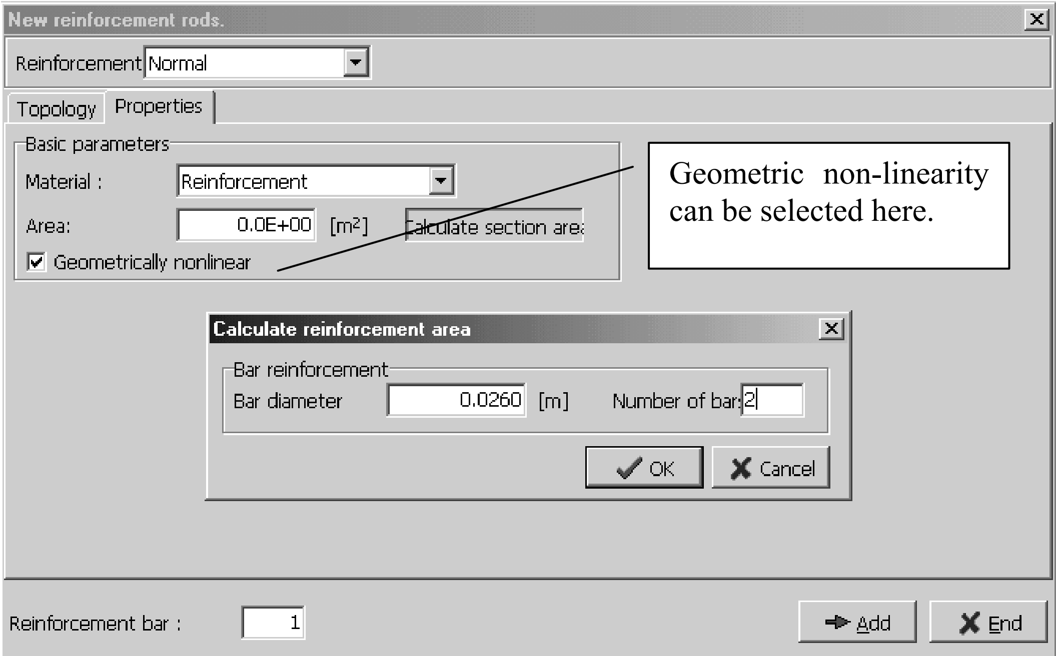


Figure 34: The dialog for the definition of reinforcement bars contains two property sheets. The sheet Properties is used for the definition of material model and reinforcement cross-sectional area.

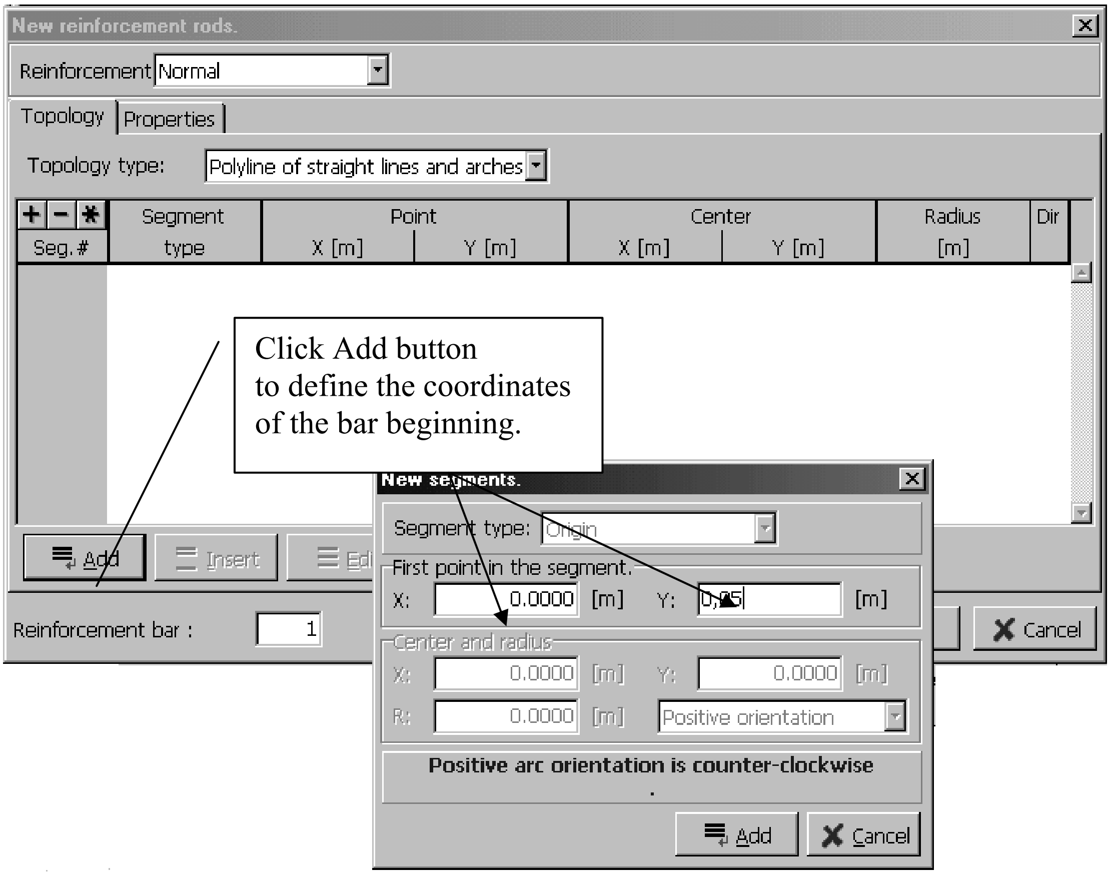


Figure 35: The sheet Topology is used for the definition of bar geometry. A reinforcement bar is composed of segments, and each segment can be a line, arc or a circle.

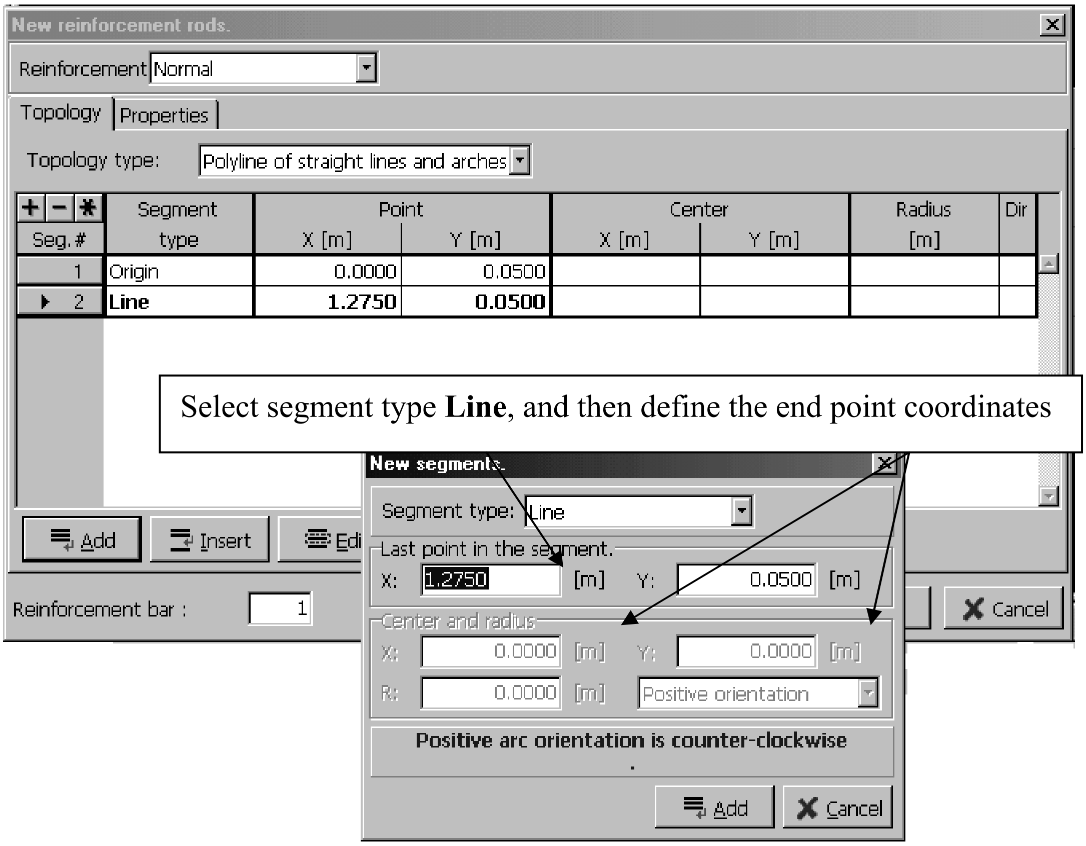


Figure 36: This figure shows the definition of the bar end point.

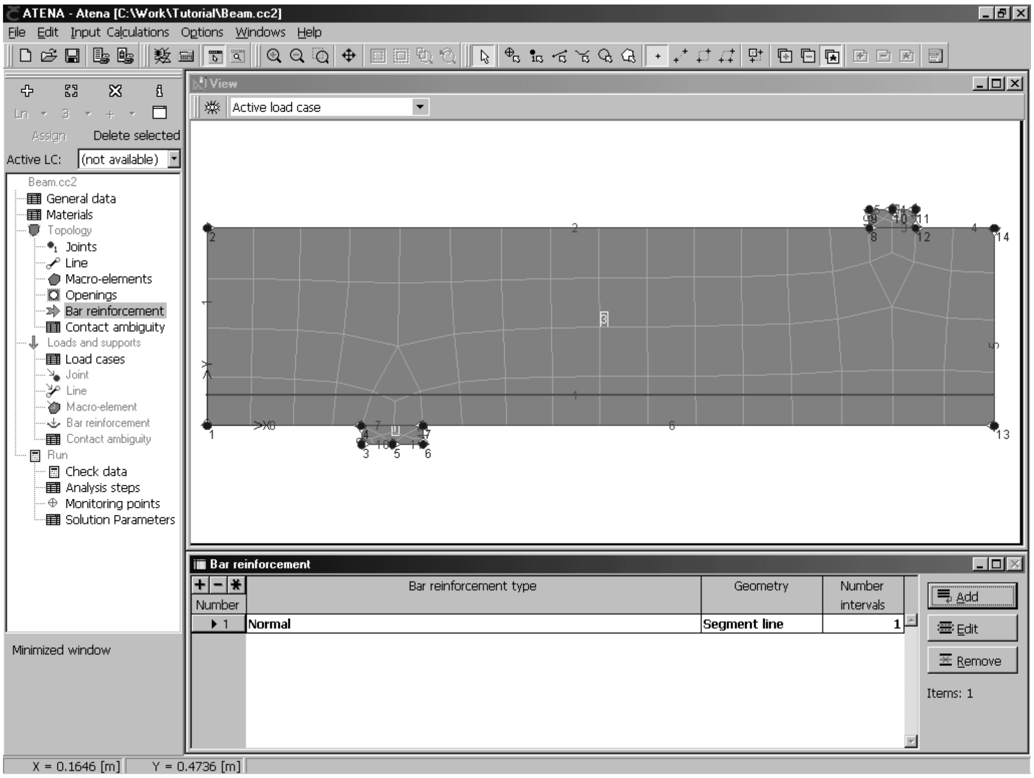


Figure 37: The program display after the definition of the reinforcement bar.

3.8 Supports and Actions

This section describes the definition of supports and loads for our example problem.

The analyzed beam is supported at the bottom steel plate in the vertical direction. Since we are analyzing only a symmetric half of the beam, it is necessary to enforce the axis of symmetry along the line 5. Horizontal displacements along this line should be equal to zero.

The beam is loaded at the top steel plate. We are interested in determining the maximal load-carrying capacity of the beam, which means we want to be able to trace the structural response also in the post-peak regime. The easiest method to accomplish this is by loading the beam by prescribed displacements at the top steel plate. It is possible to apply the loading by vertical forces, which will be increased in each load step. In order to be able to go into post-peak, advanced non-linear solution strategies such as Arc-length method would be necessary. Such techniques are available in **ATENA 2D**, but they will not be used in this example, where Newton-Raphson method and displacement load control is sufficient.

A loading history in **ATENA 2D** is defined in analogy to previous version **SBETA.** This means that first load cases are defined, and then they are combined together to form a loading history for an analyzed structure.

For this example, two load cases are defined: one containing the vertical and horizontal supports, and second with the prescribed deformations at the top steel plate.

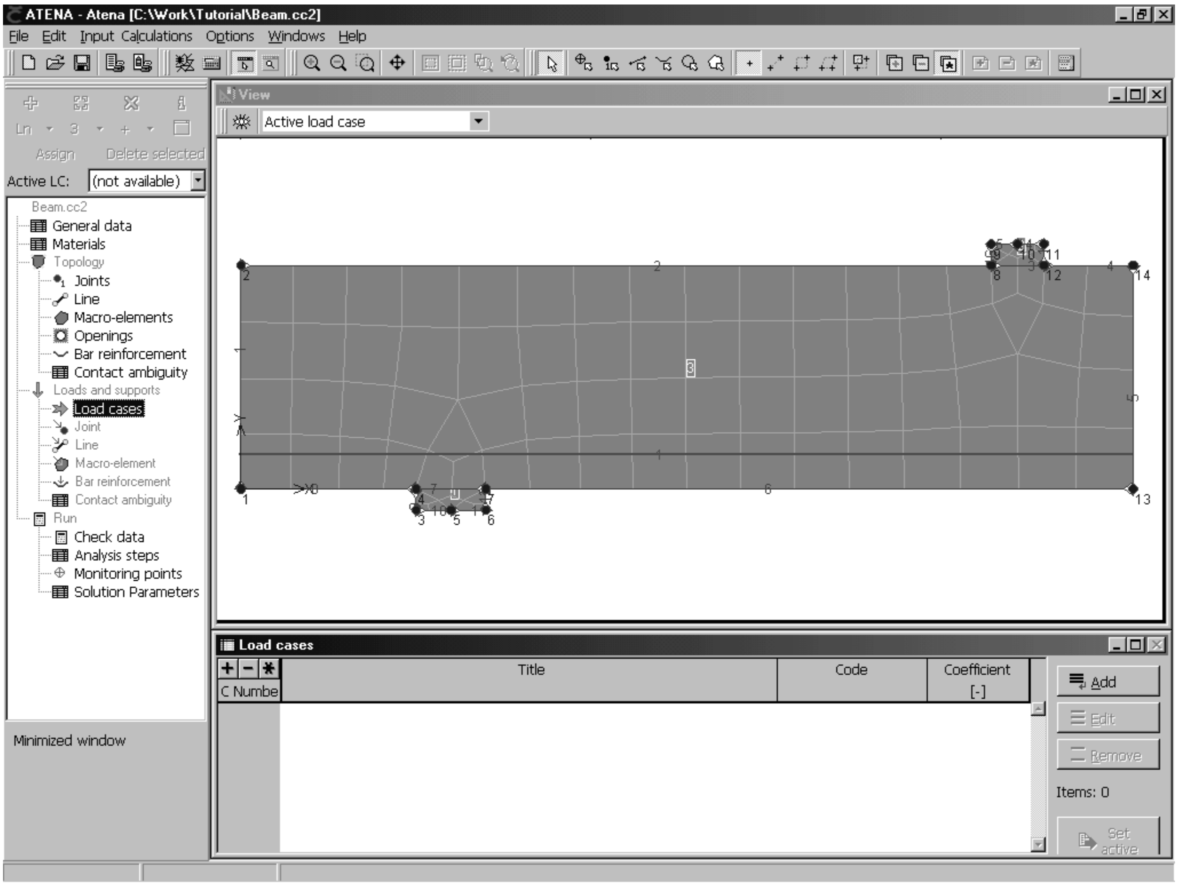


Figure 38: The load-case definition starts by highlighting the Load-cases item in the data access tree and clicking the Add button in the Load cases table.

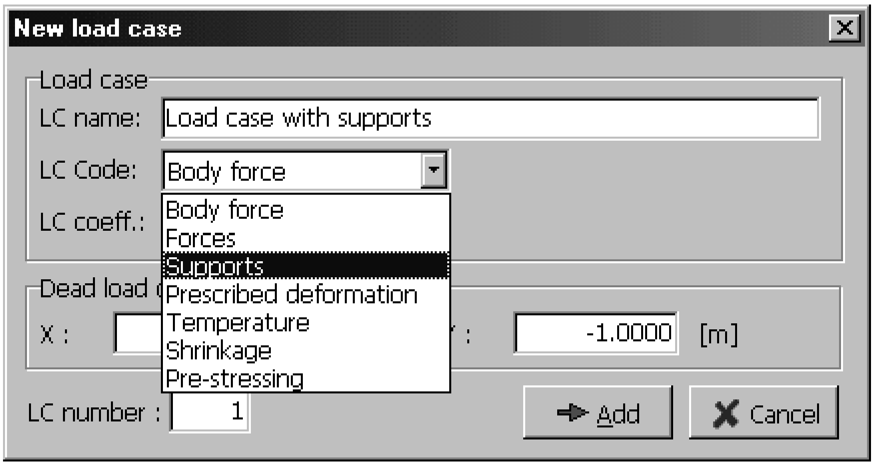


Figure 39: The first load case will contain the horizontal and vertical supports.

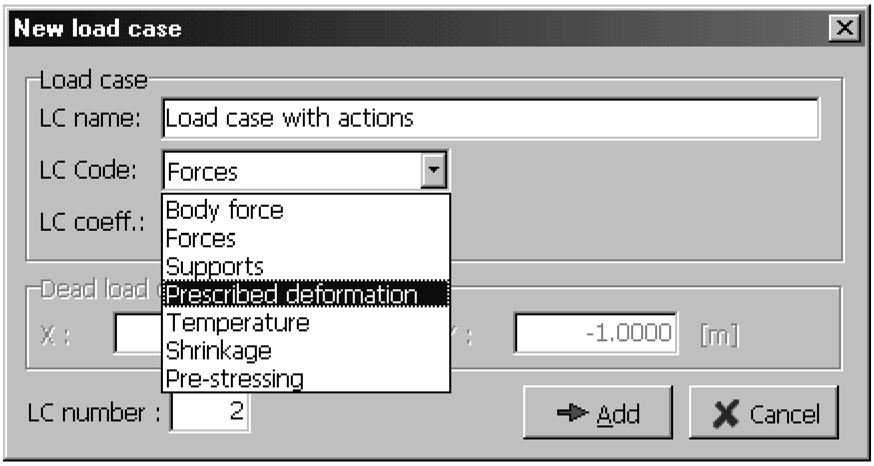


Figure 40: The second load case will contain the prescribed deformation at the top steel plate.

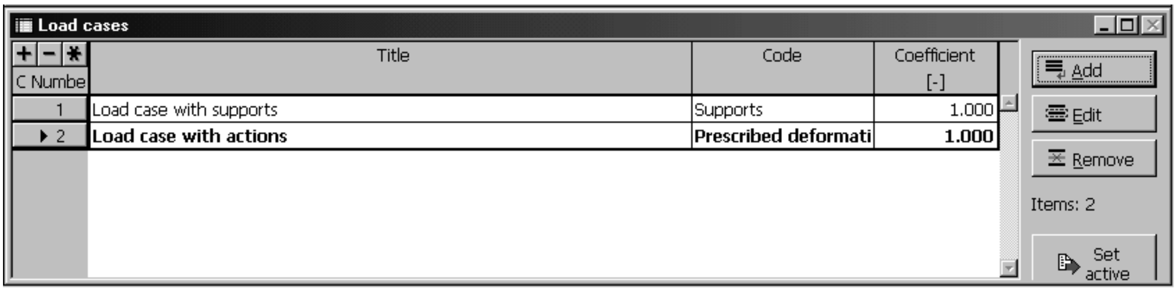


Figure 41: The list of created load-cases in the Load cases table.

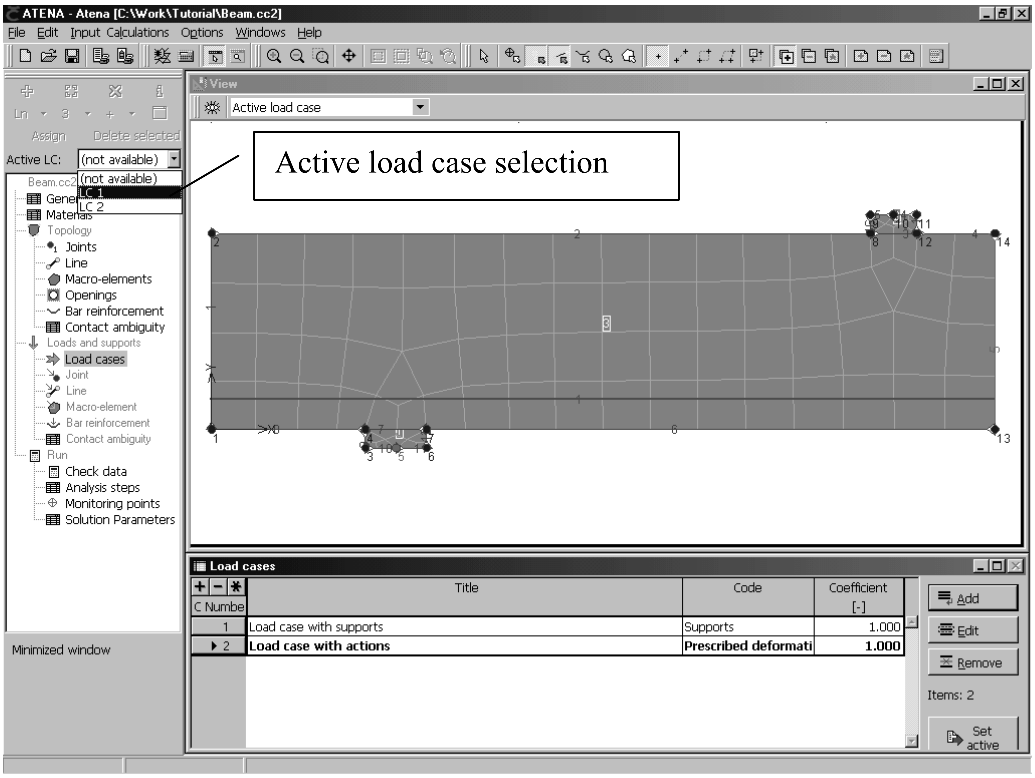


Figure 42: An appropriate active load case must be selected before the definition of supports. Supports should be in the load case 1.

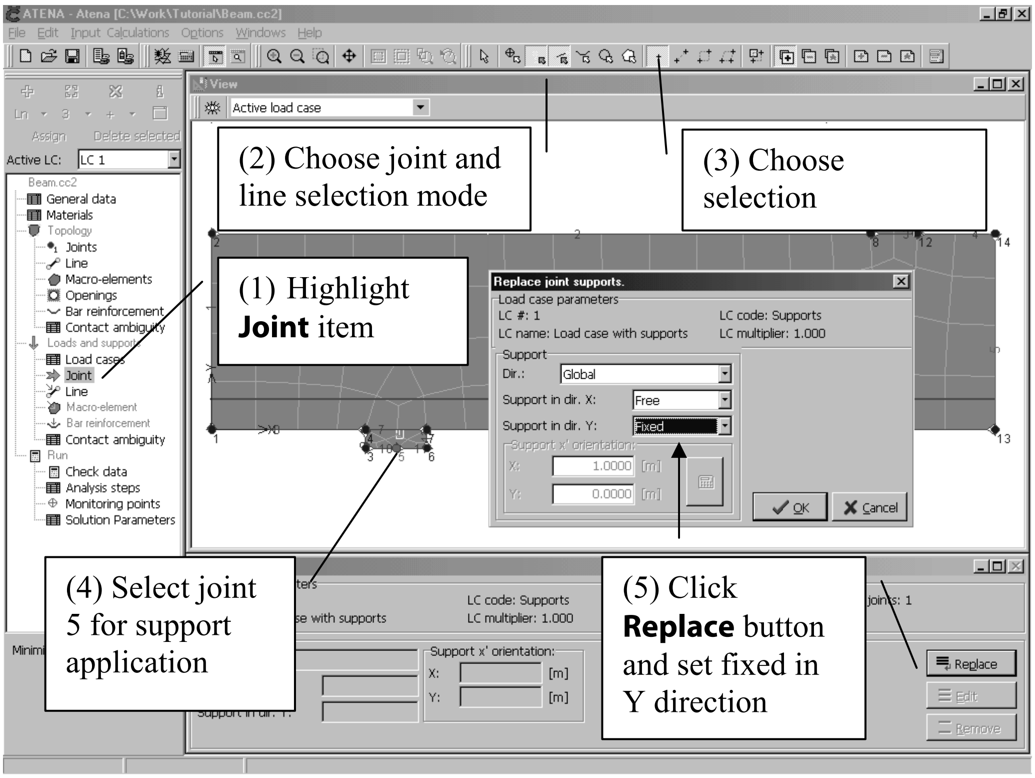


Figure 43: The definition of the vertical support at the bottom steel plate.

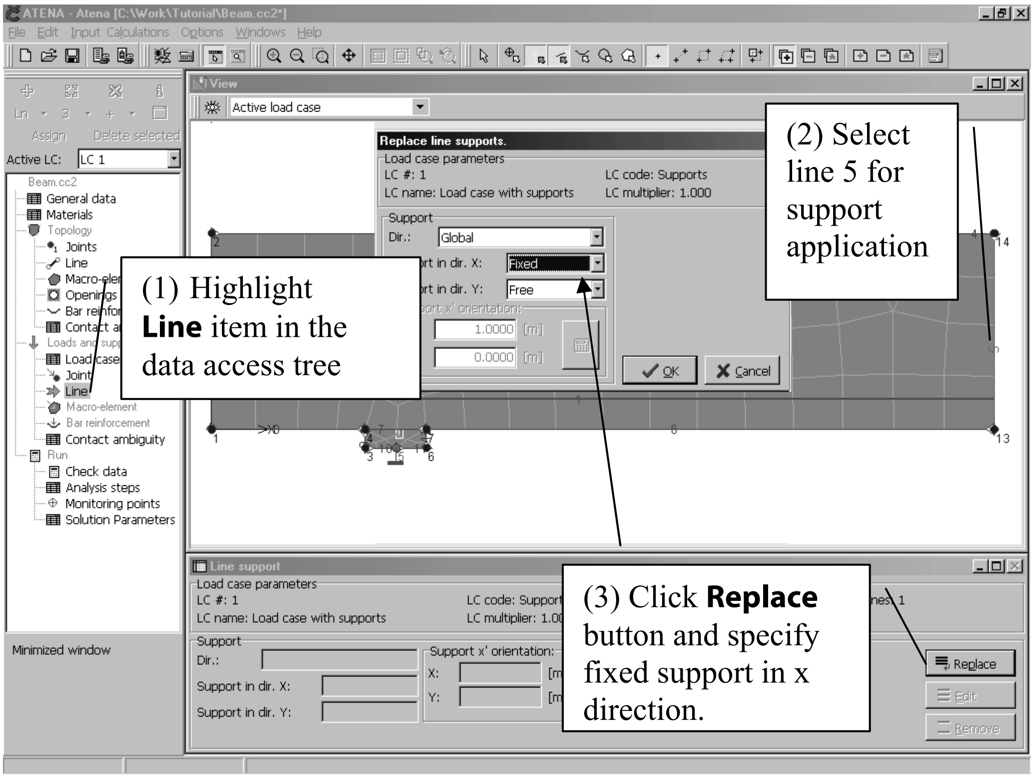


Figure 44: The definition of horizontal support along the line 5.

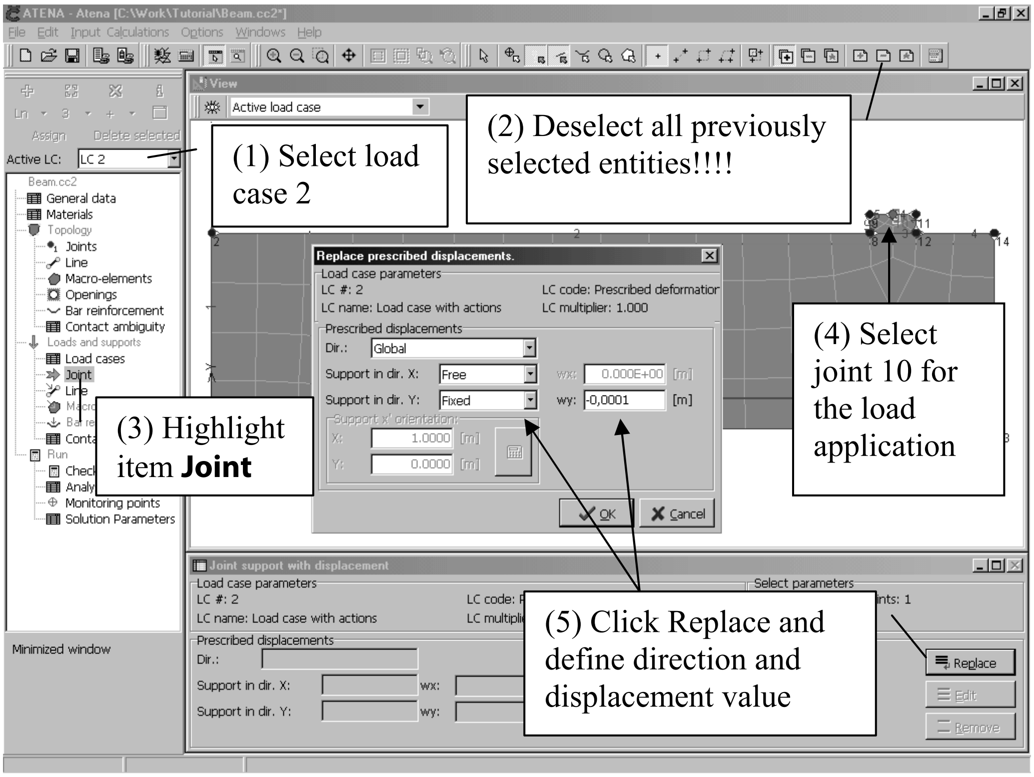


Figure 45: The definition of the prescribed displacement at the top steel plate in load case 2.

3.9 Loading History and Solution Parameters

This section describes the definition of loading history for the analysis of Leonhardt's shear beam. The loading history consists of load steps. Each load step is defined as a combination of load cases, which had been defined previously. Each load step contains also a definition of solution parameters, which define solution methods that are to be used during the load steps. **ATENA 2D** contains a standard set of solution parameters. The standard solution parameters can be examined in the table of **Solution parameters**. This table appears in the table window after highlighting the **Solution Parameters** item in the data access tree.

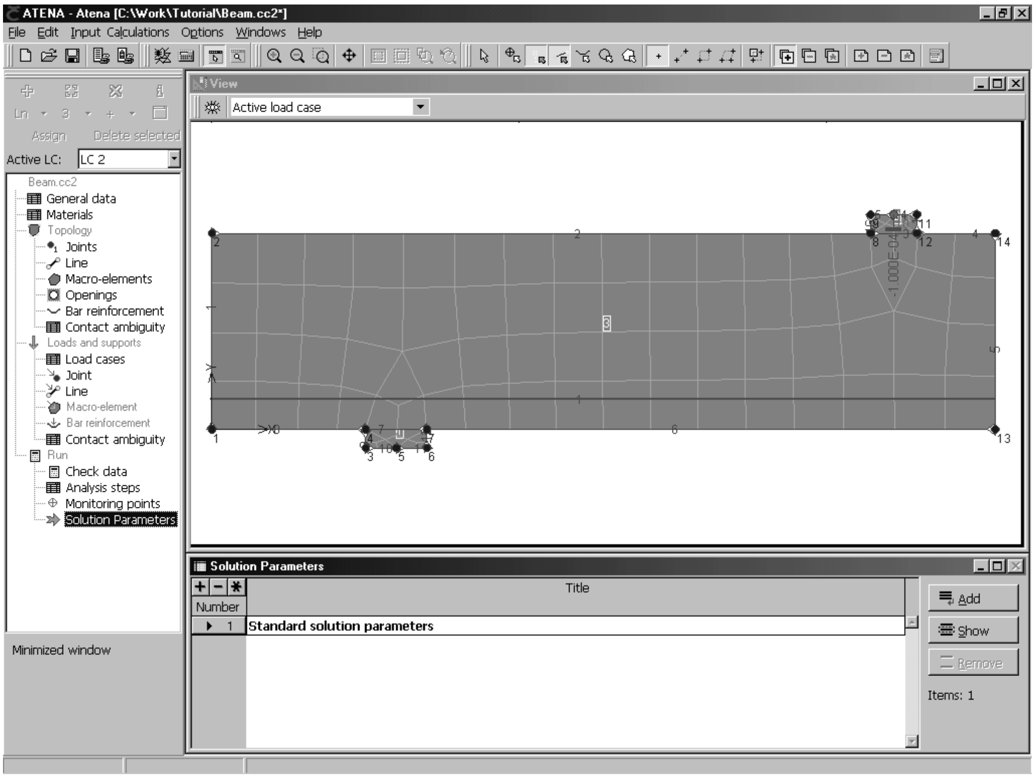


Figure 46: The program display with the table of solution parameters. Standard solution parameters can be examined by clicking the button Show. New set of solution parameters can be created using the button Add.

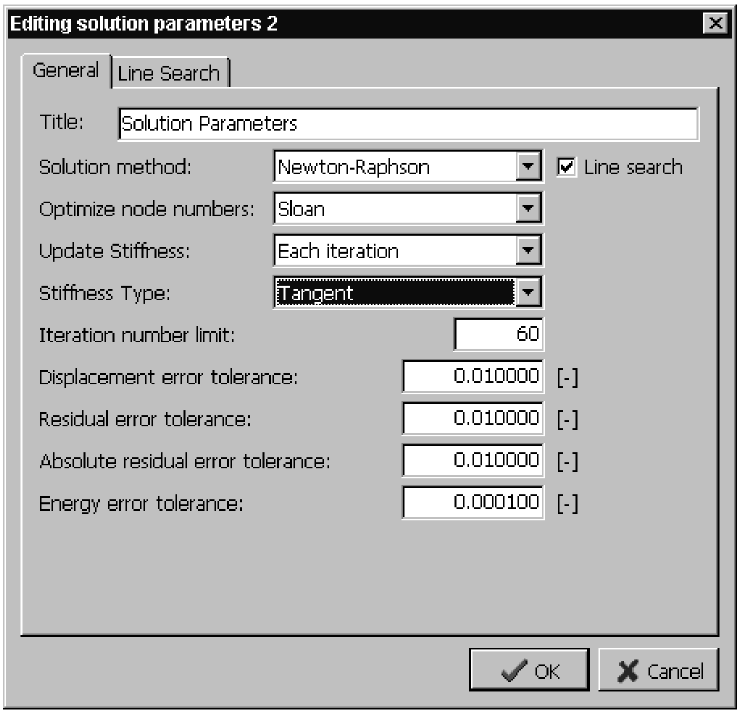


Figure 47: The first property sheet for the new set of solution parameters for Leonhardt's beam analysis.

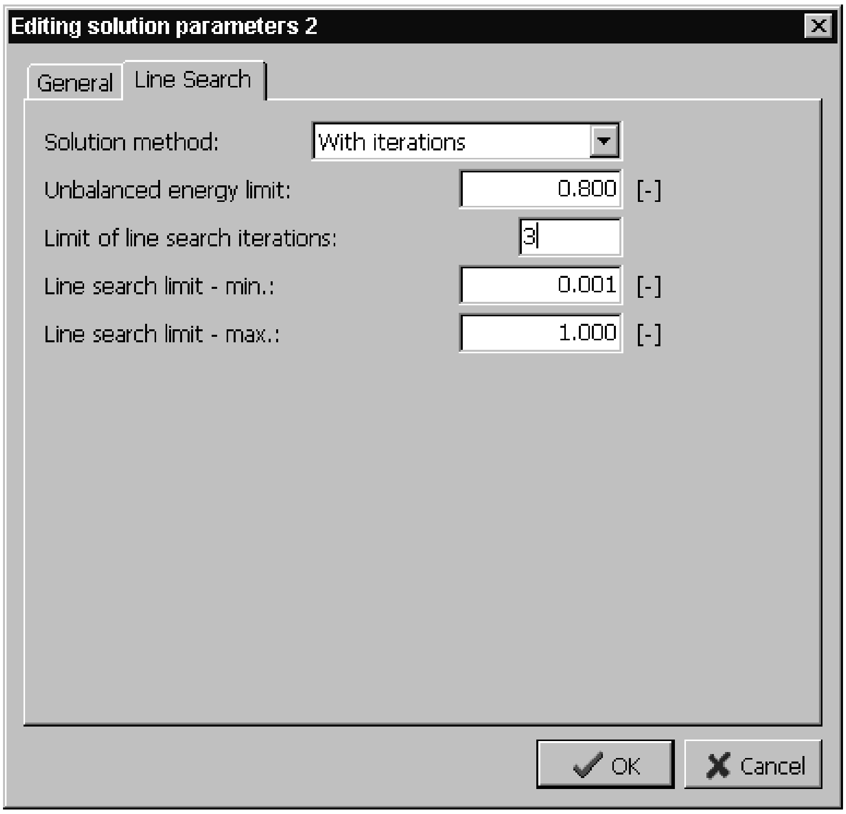


Figure 48: The second property sheet for the new set of solution parameters for Leonhardt's beam analysis.

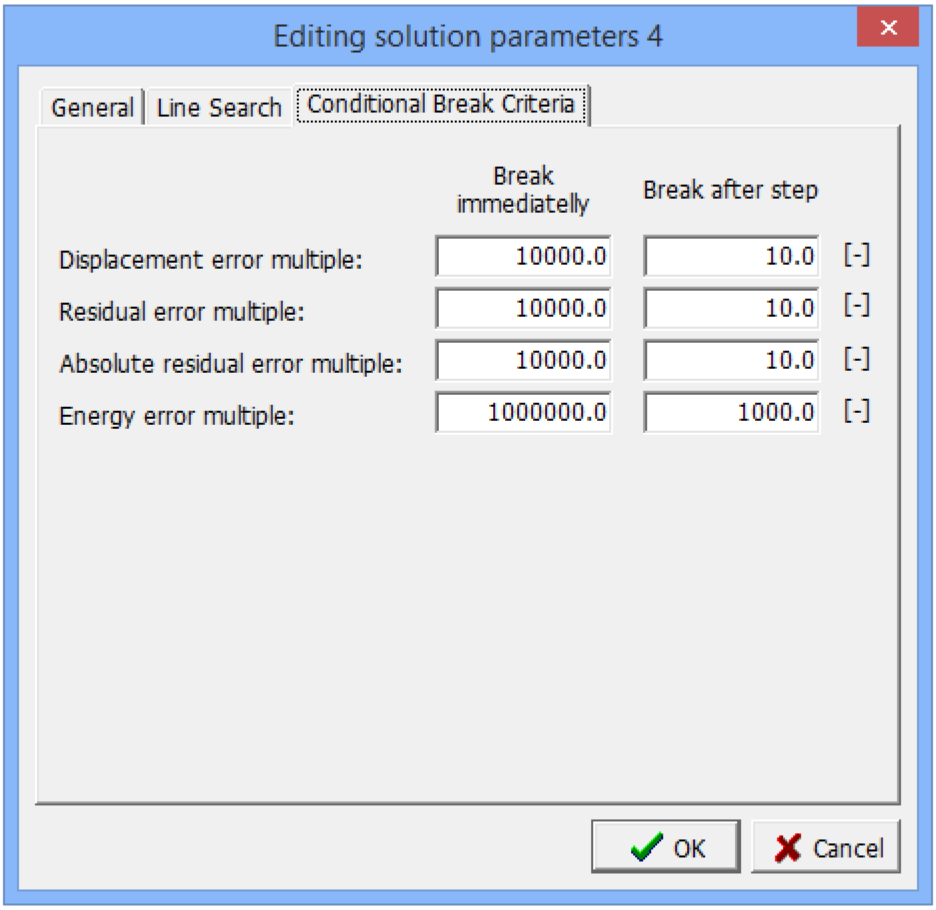


Figure 49: The third property sheet for the new set of solution parameters – Conditional Break Criteria.

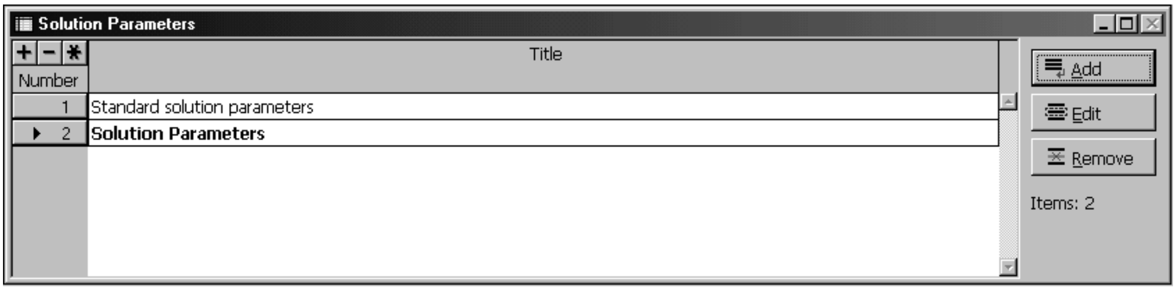


Figure 50: The table with the newly created solution parameters.

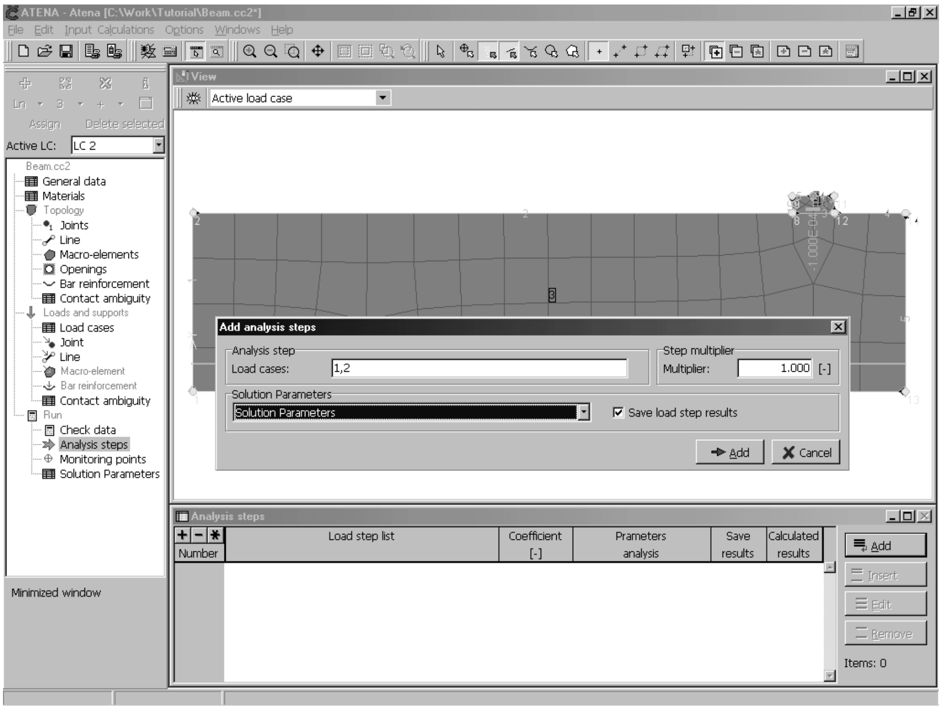


Figure 51: Load steps are specified using the button Add from the table of Analysis steps. This table appears in the table window after highlighting the Analysis steps item in the data access tree.

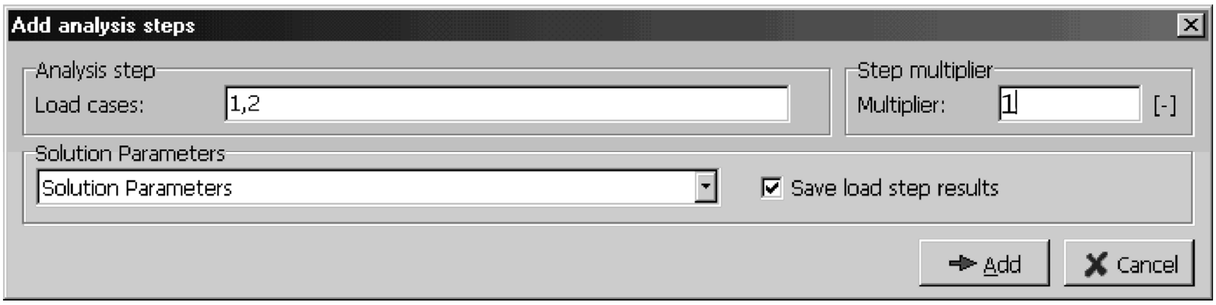
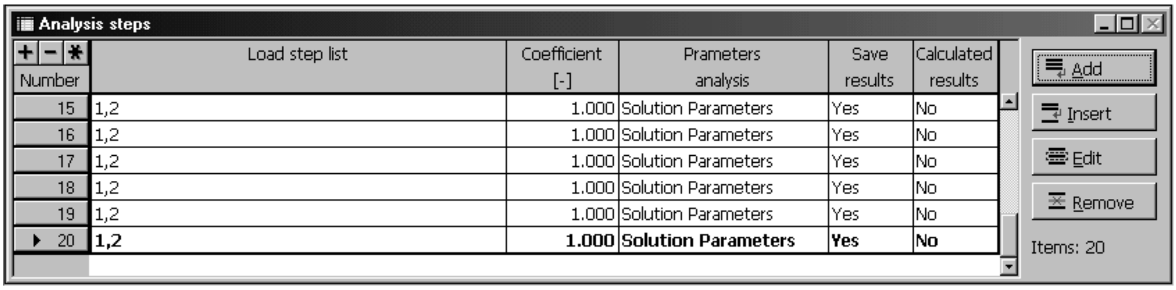


Figure 52: Each step will be composed of load cases 1 and 2. The multiplier 3 will be used to multiply the applied actions and the newly created solution parameters will be used during the load steps.



Number	Load step list	Coefficient [-]	Prameters analysis	Save results	Calculated results
15	1,2	1.000	Solution Parameters	Yes	No
16	1,2	1.000	Solution Parameters	Yes	No
17	1,2	1.000	Solution Parameters	Yes	No
18	1,2	1.000	Solution Parameters	Yes	No
19	1,2	1.000	Solution Parameters	Yes	No
20	1,2	1.000	Solution Parameters	Yes	No

Figure 53: The Analysis steps table after the definition of twenty load steps with the above parameters. It is possible to add more load steps later during the analysis.

3.10 Monitoring Points

During non-linear analysis it is useful to monitor forces, displacements or stresses in the model. The monitored data can provide important information about the state of the structure. For instance from monitoring of applied forces it is possible to determine if the maximal load was reached or not. Monitoring points can be defined by highlighting the **Monitoring points** item in the data access tree. Then it is again possible to use graphical or alpha-numerical specification of the monitoring point location. The graphical input is activated by the button , and follows by the selection of the exact location by mouse. The alpha-numerical input starts by the button **Add** from the table of **Monitoring points**.

For this example, the first monitoring point should be added near the joint where the prescribed displacements are applied. The second component (i.e. y direction) of nodal applied forces should be monitored at this point. It is not necessary to define a location exactly at the finite element node. The program automatically selects the closest FE node. In case monitoring at integration points is required, the closest finite element integration point is selected.

The second monitoring point should be located at the middle of the beam near its bottom surface, where the largest vertical displacements can be expected. The second component (i.e. y-displacement) of nodal displacements should be monitored at this location.

These two monitoring points will allow us to monitor the load-displacement curve during the non-linear finite element analysis. It makes it possible to see the changes of action forces and displacement at each load step and even in each iteration. The program display after the definition of the monitoring points is shown in Figure 56.

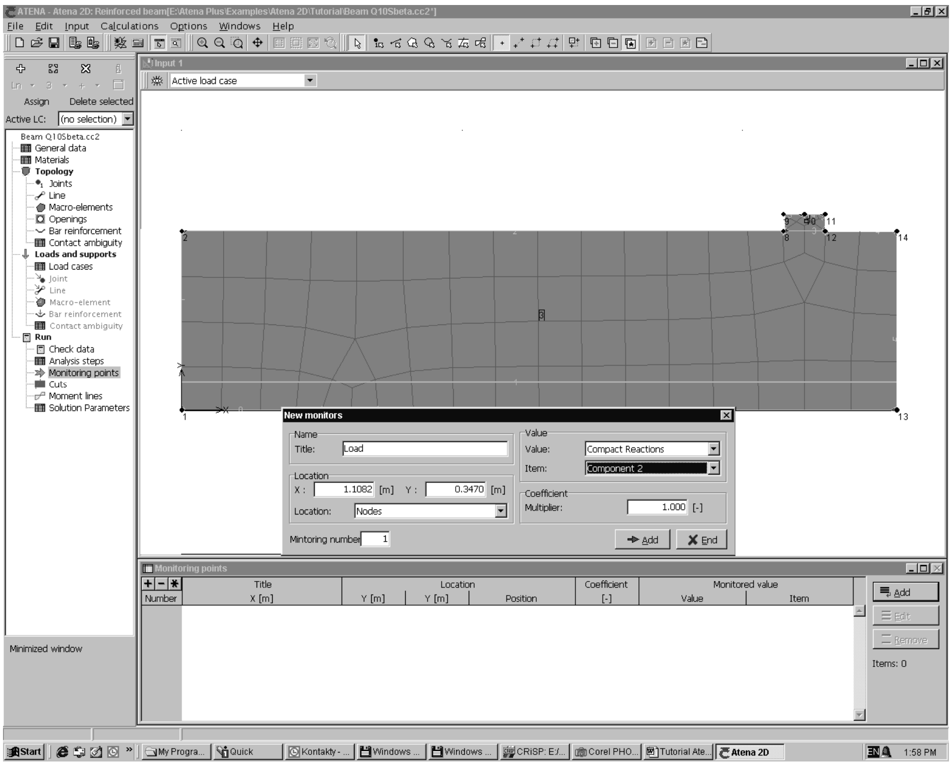


Figure 54: The definition of the first monitoring point.

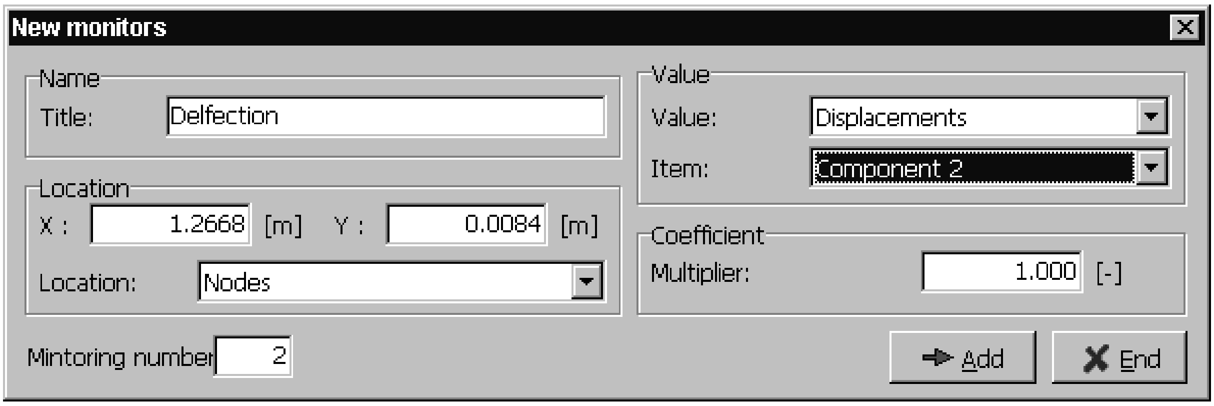


Figure 55: The dialog window for the definition of the second monitoring point.

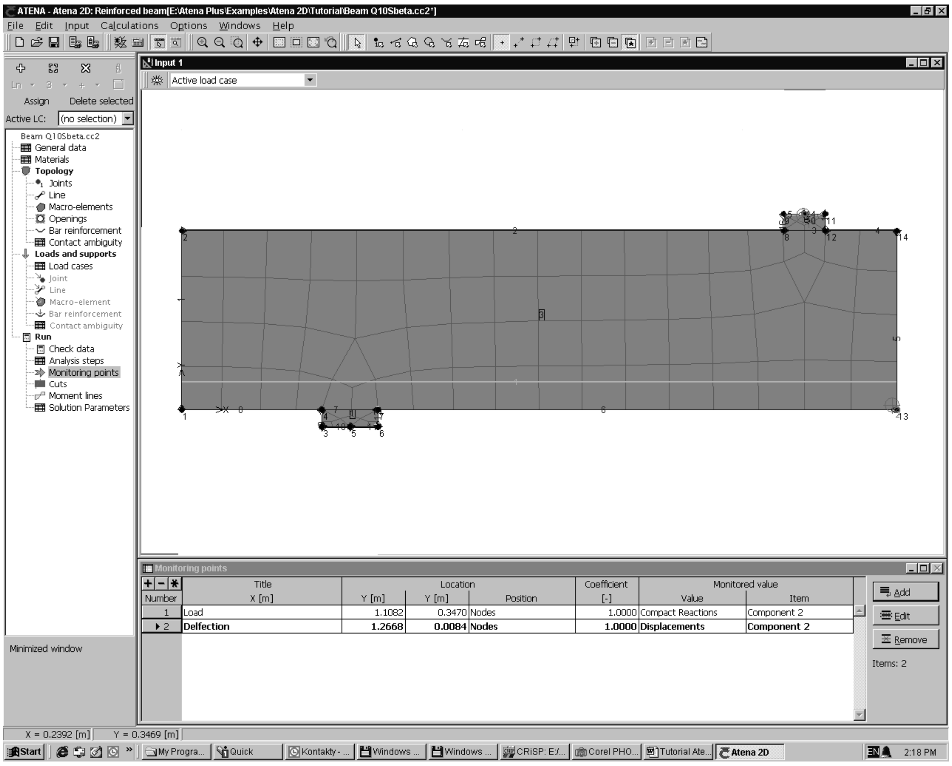


Figure 56: The program display after the definition of monitoring points.

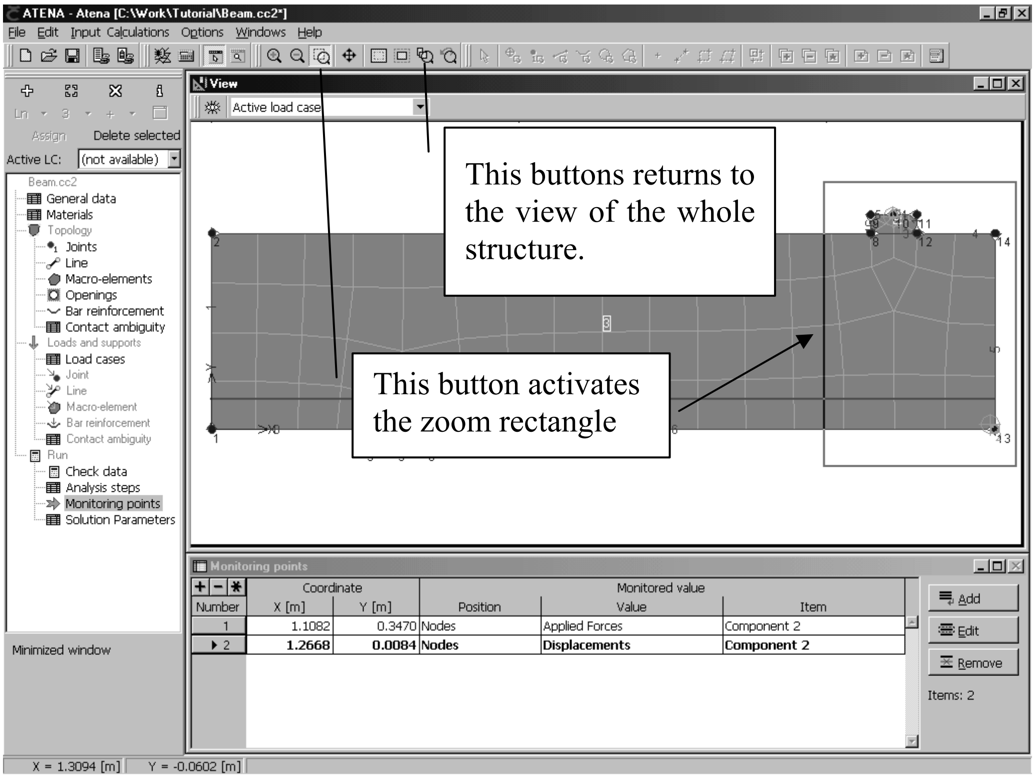


Figure 57: The way, in which the program selects the closest node for monitoring becomes more apparent after zooming at the middle section of the beam. The button returns the view to the state when the whole structure is displayed.

4. FE NON-LINEAR ANALYSIS

4.1 Introduction

This section describes the process of running a non-linear finite element analysis of the Leonhardt beam using the data that have been prepared in the previous sections of this tutorial.

Before finite element analysis it may be useful and interesting to view the finite element mesh numbering. The finite element model numbers can be displayed through the view setting button , which appears at the left top corner of the view window. This actions opens a dialog window (see Figure 58) that can be use to select the data to be displayed in the view window. Among others it is possible to turn on/off numbering of finite elements, nodes or geometrical entities, turn on/off the display of reinforcement bars or monitoring points.

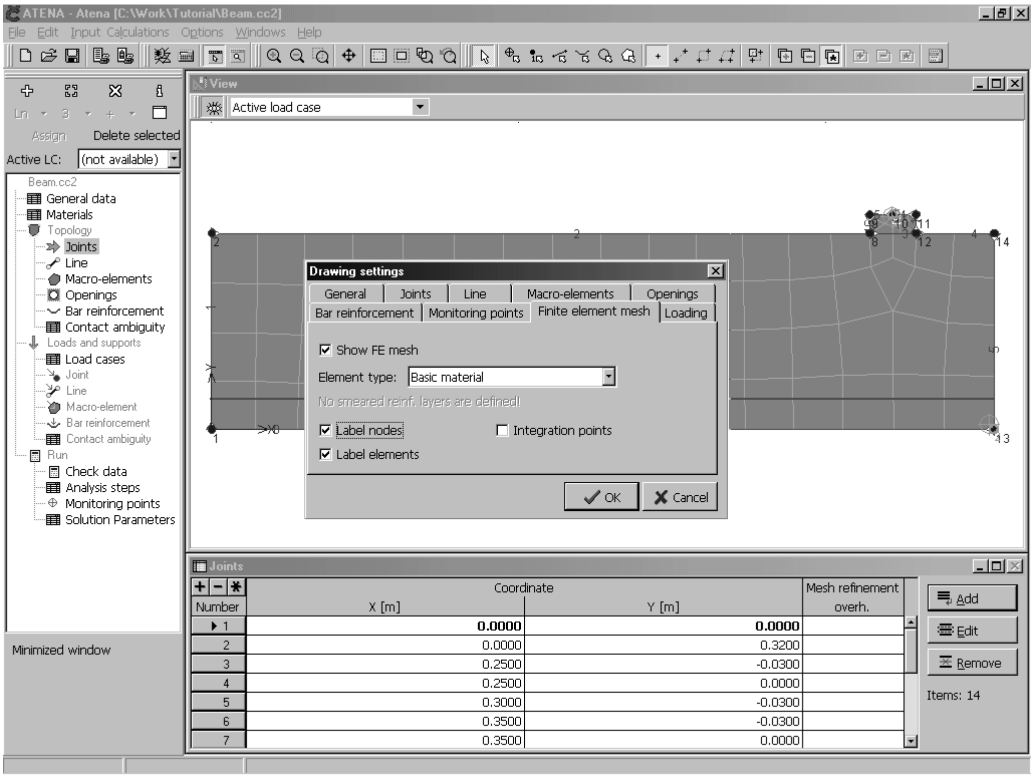


Figure 58: The dialog box for activating the display of finite element node and element numbers.

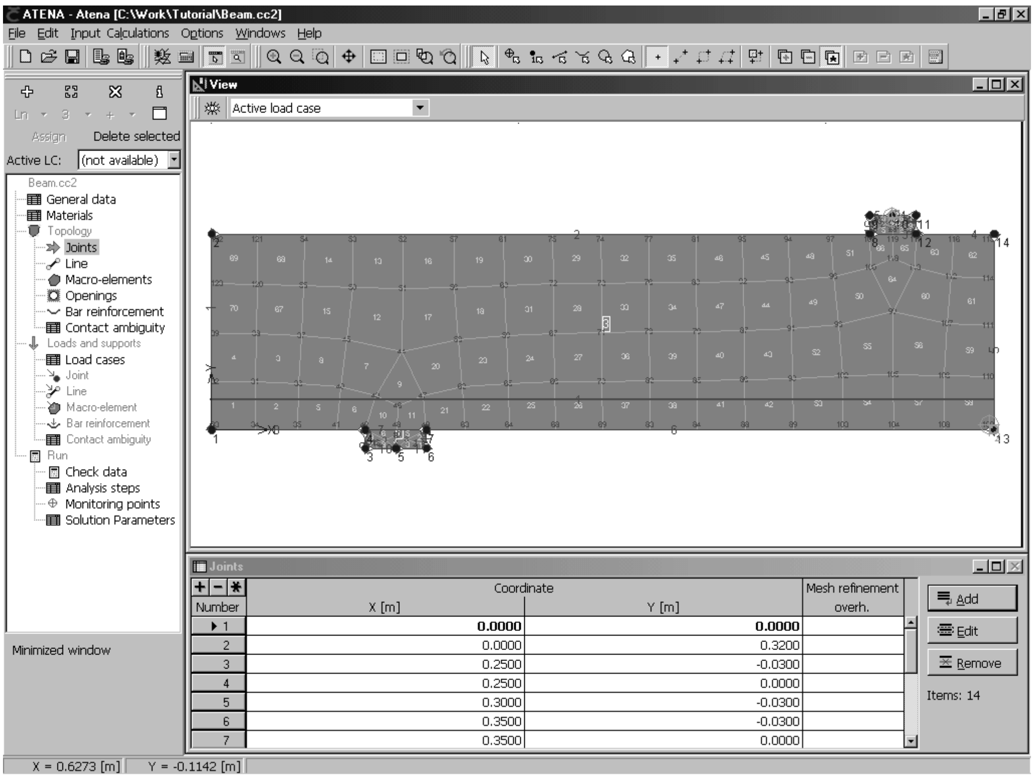
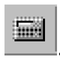


Figure 59: The finite element mesh along with node and element numbers. The size of characters can be modified from the menu item Options | Settings.

4.2 Starting Analysis

The finite element analysis is started using the button . After clicking this button an initialization dialog window appears on the computer screen. This dialog window is shown in Figure 60, and it can be used to select the load step when the analysis will be terminated and data to be displayed in the L-D diagram. In addition, the load steps can be specified, in which results should be saved, in this dialog. Please note that it is not possible to perform a post-processing for the steps, in which the results were not saved.

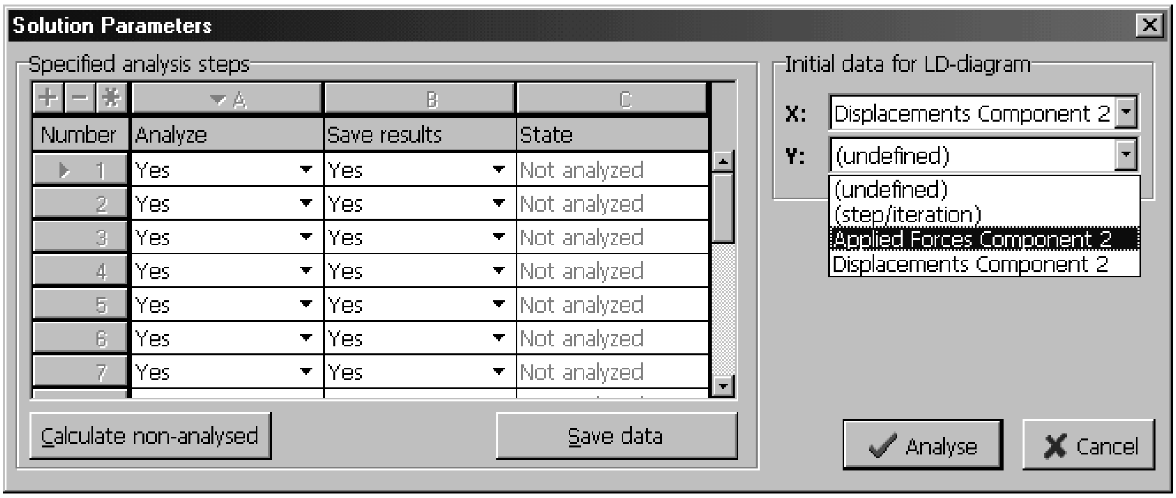


Figure 60: The dialog window before the finite element analysis.

4.3 Interactive Window

After the button **Analyse** in the dialog, which is shown in Figure 60 is selected, the actual finite element analysis is started. The analysis progress can be monitored using the interactive window that is shown in Figure 61.

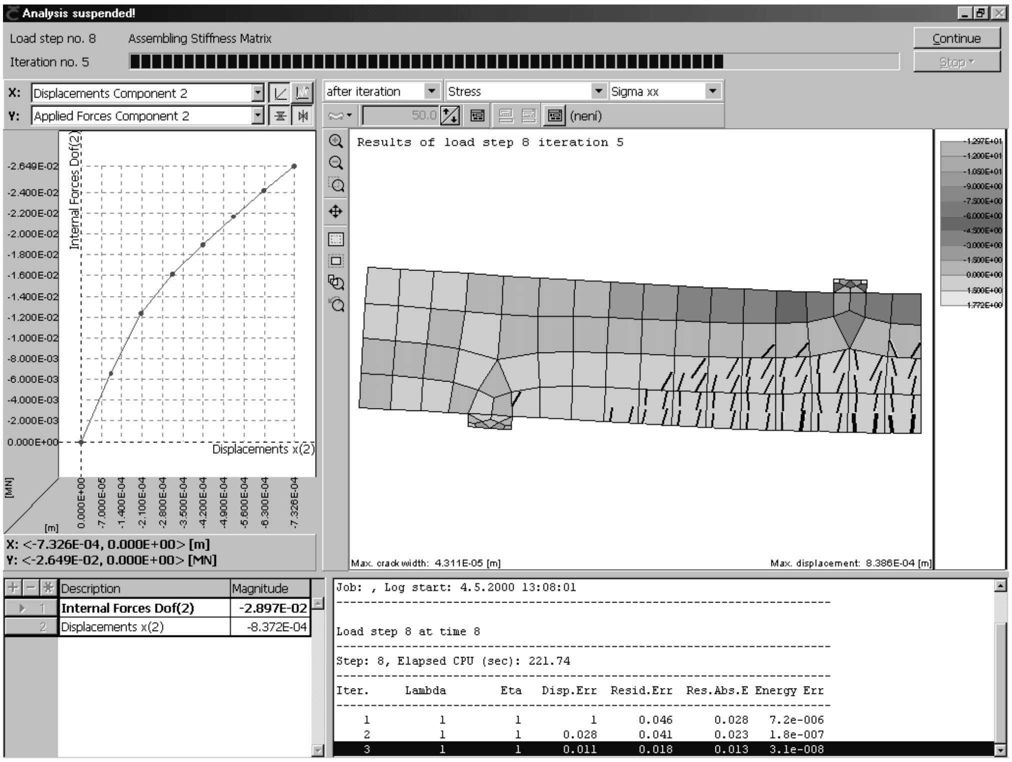


Figure 61: The interactive window for monitoring the progress of non-linear analysis.

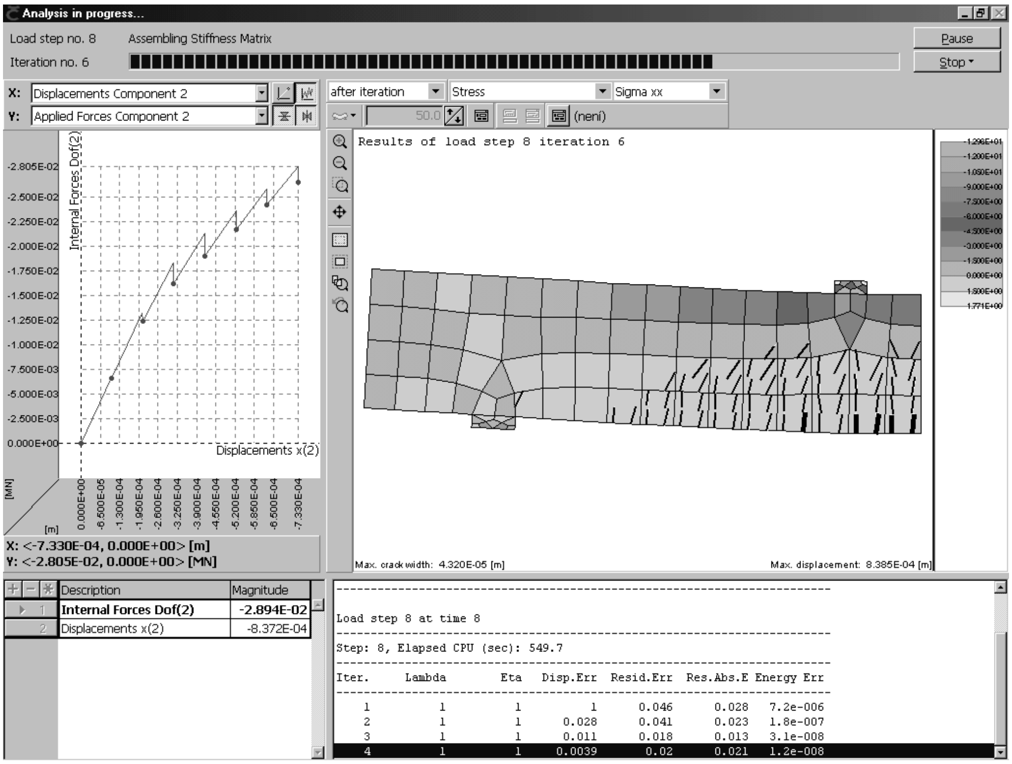
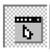


Figure 62: The interactive window after selecting a different format of the L-D diagram. The L-D diagram in this figure shows the iterative changes of the monitored quantities.

4.4 Adding New Load Steps

After the first 20 load steps are completed it is possible to specify additional 20 load steps. In order to define additional load steps it is necessary first to switch to the pre-processing mode using the button . After the specified steps are completed the program automatically enters the post-processing mode. It is necessary to switch into the input mode before new steps can be defined.

The new load steps are defined in analogy to the process described in Section 3.9 by adding new analysis steps in the **Analysis steps** table.

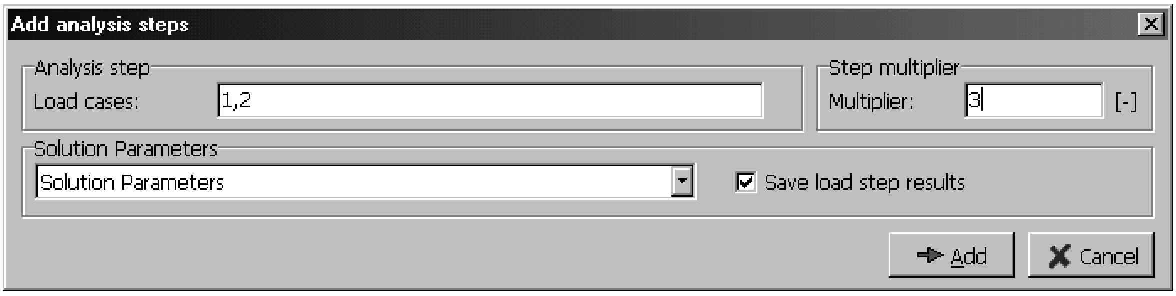


Figure 63: The dialog box for the new analysis steps. Same properties are used as in Section 3.9.

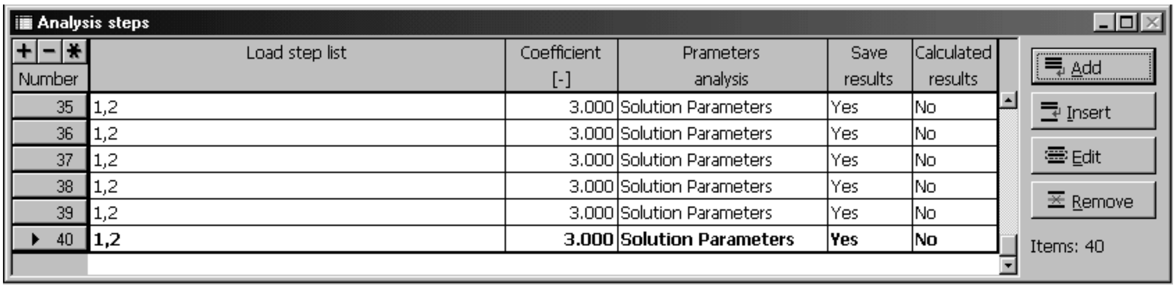
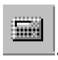
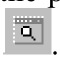


Figure 64: The table with the analysis steps after the definition of additional 20 steps.

After the definition of new load steps, the analysis can be restarted again by clicking the button .

5. POST-PROCESSING

5.1 Introduction

After the finite element analysis is completed or terminated, the program automatically enters into the post-processing mode. The post-processing can be entered also by clicking the button . This action is meaningful only after the analysis has been performed, otherwise there would be no results to visualize.

5.2 Post-processing Window

The layout of the post-processing window is shown in Figure 65. The first step in post-processing is to select the analysis step (i.e. load step), from which the results are requested. The program loads the data for the requested load step into the computer memory and fills in appropriately the lists of available output quantities. The type of analysis and used material models determines the available output data.

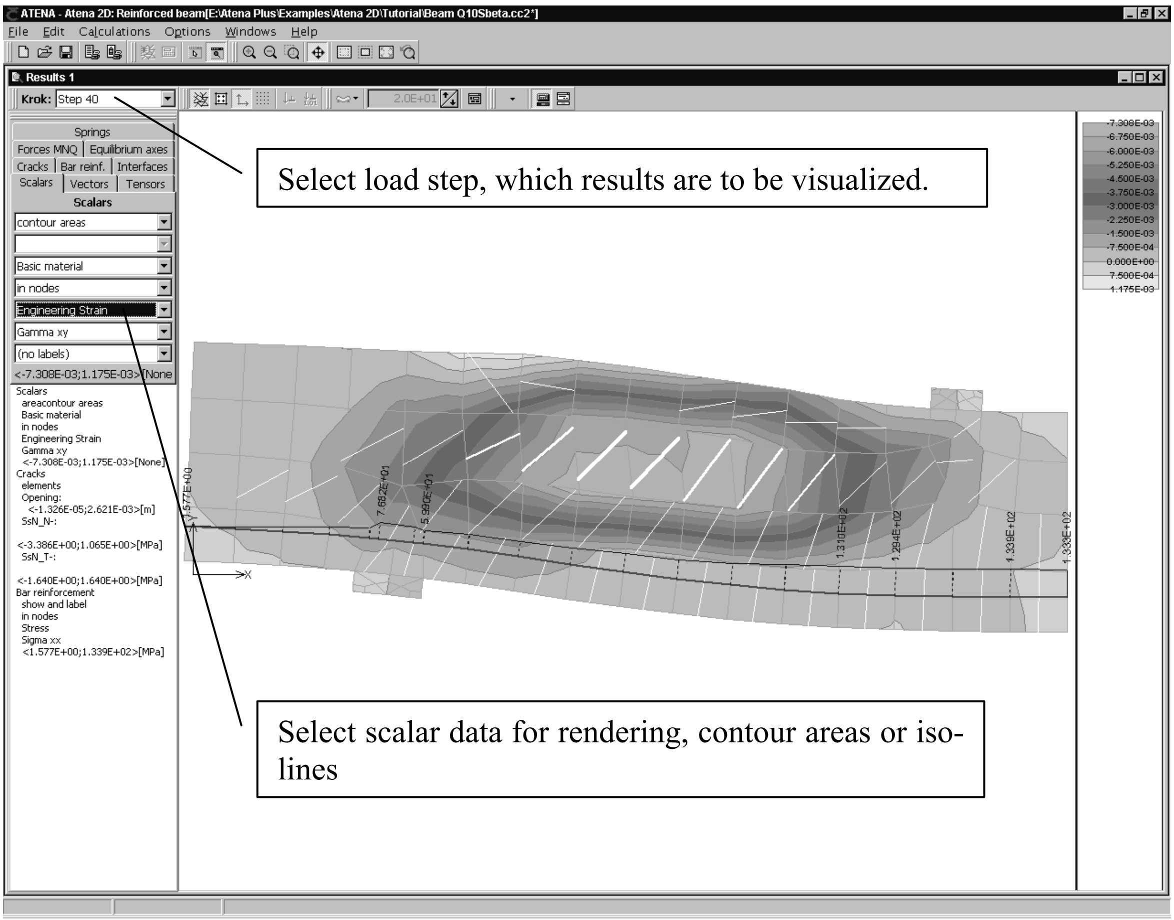


Figure 65: The post-processing window containing contour areas, cracks and reinforcement stresses for the last load step 40.

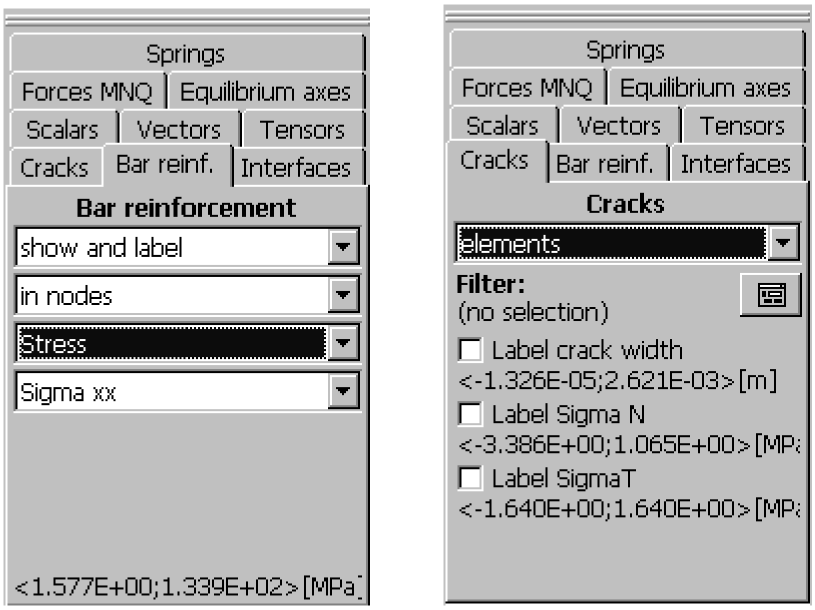


Figure 66: The display of reinforcement bar stresses and cracks is activated by clicking an appropriate label in the toolbar along the left side of the program window.

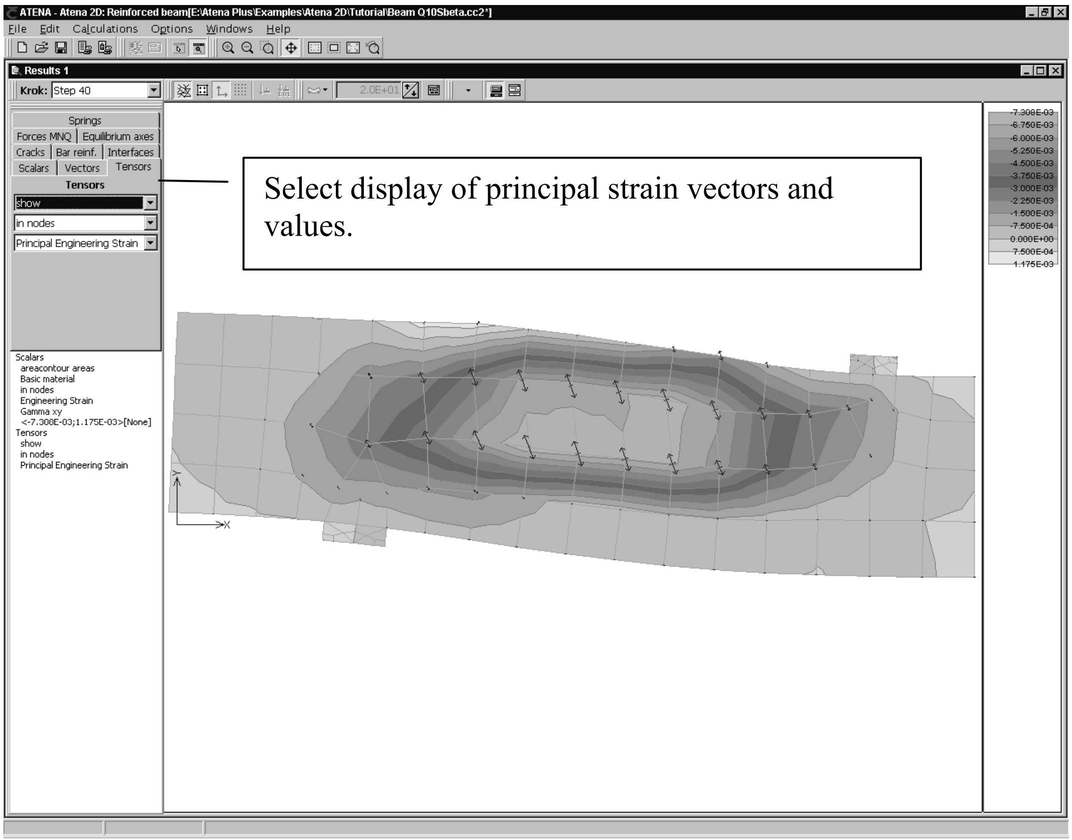


Figure 67: The post-processing window with rendering and principal vectors of maximal principal strains for the last load step 40.

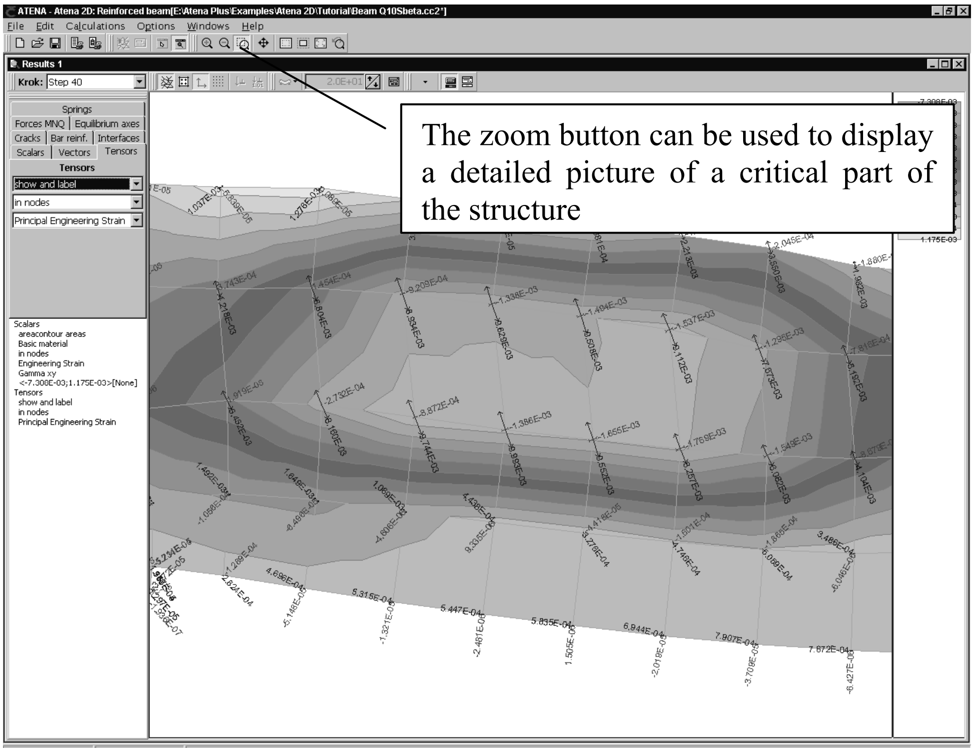


Figure 68: The post-processing window with zoom, rendering and principal vectors of maximal principal strains for the last load step 40. If a user selects the item show with label, the numerical values of the principal strains will be displayed as well.

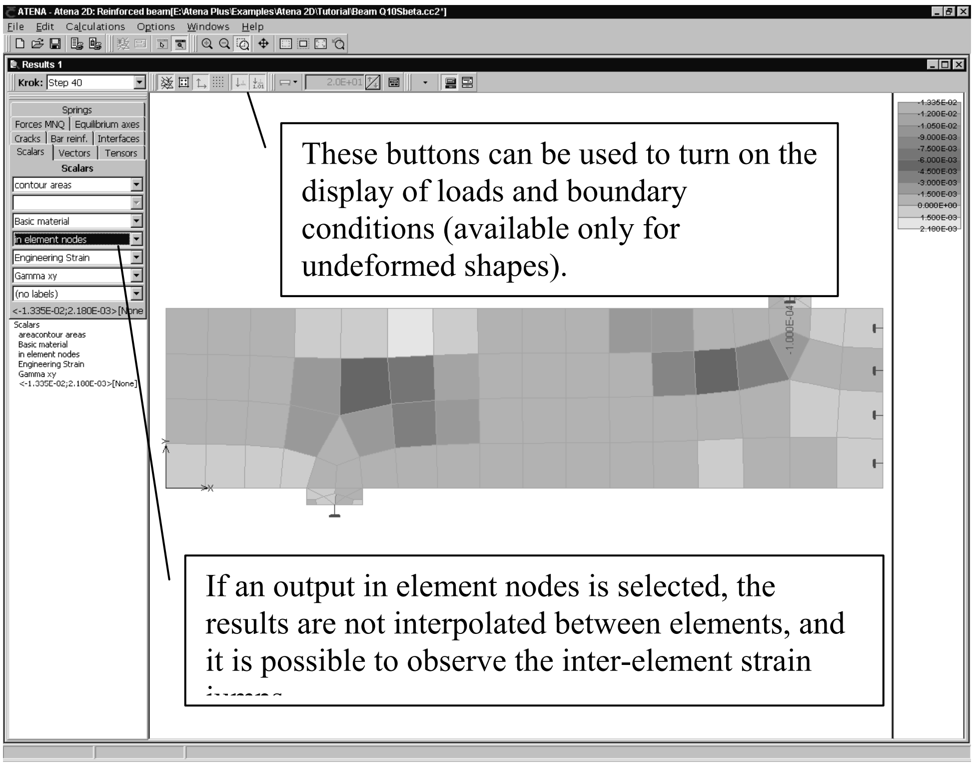


Figure 69: The post-processing window with rendering of element data for the last load step 40. This figure shows also the undeformed configuration with support and loading conditions. The undeformed shape is selected by the following button **.**

It is possible to open several post-processing windows at the same time. Each window can be used to show results from different load steps. A new post-processing window can be opened using the menu item **Windows | New | View**.

The active post-processing window can be printed from the menu item **File | Graphic printout…** or copied to the clipboard from **Edit | Copy picture**. The copied picture can be for instance pasted to a Microsoft Word document. The picture remains in vector format so it can be easily scaled or resized while preserving its resolution for printing.

It is possible to modify some parameters controlling the display on the screen or on paper with the help of the dialog **Options | Settings**. This dialog and some of its features are described in the subsequent figures.

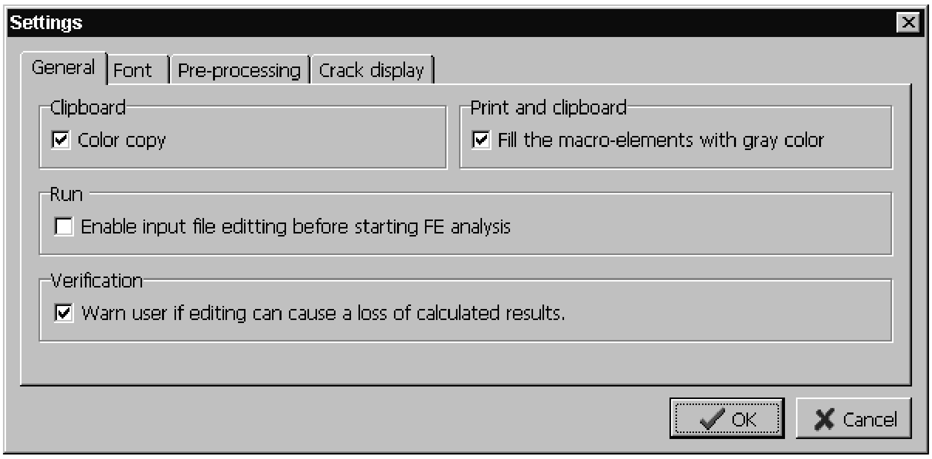


Figure 70: The dialog sheet **General** contains various checkboxes affecting the clipboard and printing functions. The block **Run** enables the user to edit the program input file before the numerical calculation are started. This option should be used only by the experience users, since wrong editing can cause a program crash and can result in a loss of data.

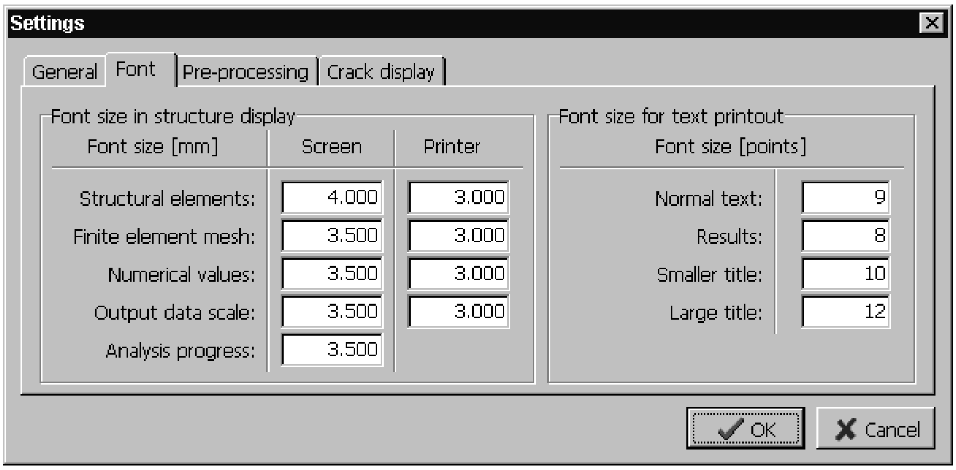


Figure 71: The dialog sheet Font controls the size of labels on the computer monitor and in the printer.

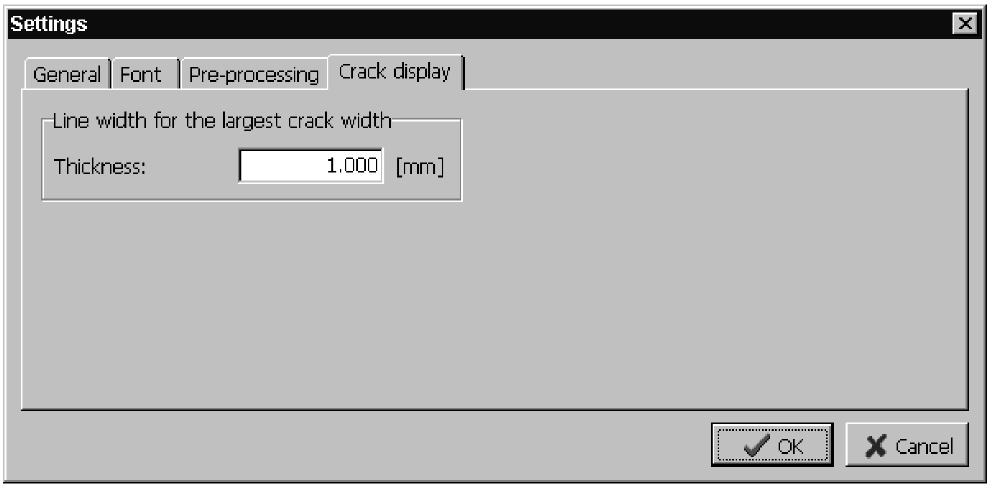


Figure 72: The dialog sheet Crack display enables the definition of line thickness that will be used for drawing cracks with the largest opening.

5.3 Load-displacement Diagrams

The important information about the structural behavior can be obtained from the data collected during the analysis at the monitoring points. In our case the force at the point of load application and the maximal vertical displacement were monitored. The load-displacement diagram can be displayed as another post-processing window from the menu item **Windows | New | Graph**. An empty window appears on the computer screen. Next step is to select which monitored quantities are to be plotted on X and Y-axes.

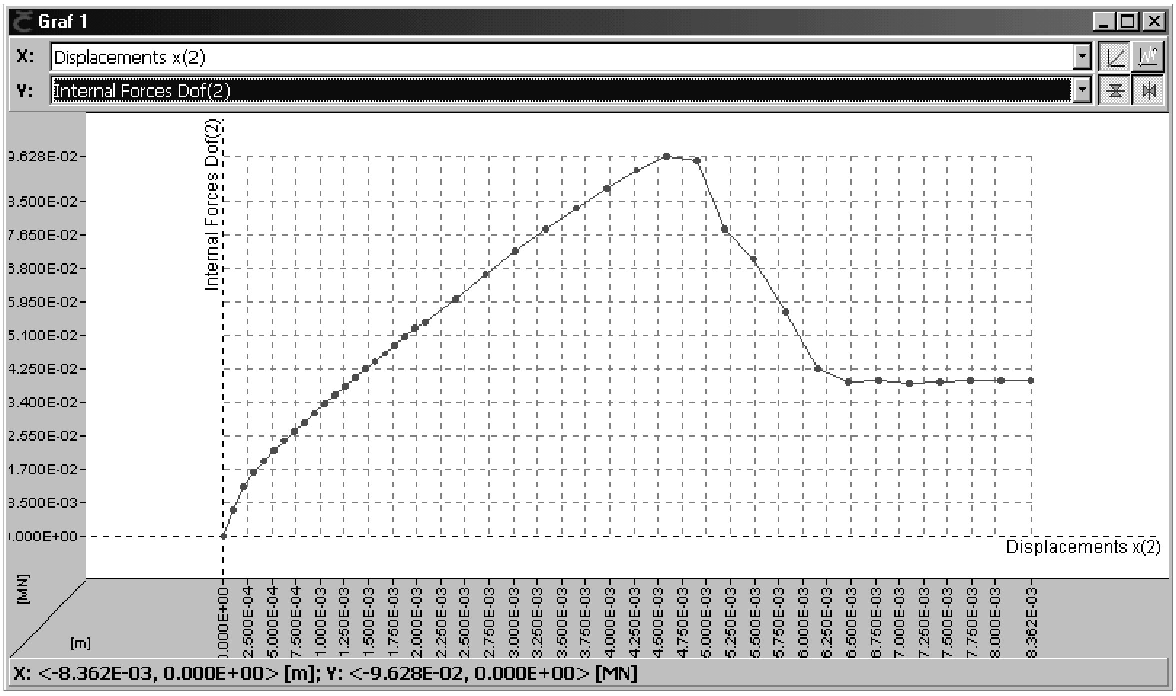
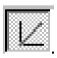


Figure 73: The load-displacement diagram.

The buttons in the top right hand corner of the graph window can be used to modify the diagram appearance. The button . selects the display of monitoring data at the end of load steps, while the button  displays the quantities as they had been changing during the iterative process. The buttons  can be used to change the quadrant for the graph display.

The selected diagram can be printed or copied to the clipboard in the same manner as it was described in Section 5.2. The numerical values of the monitored quantities can be obtained from the text output that is described in Section 5.4.

5.4 Text Output

This section describes another form of output from the program **ATENA 2D**. The text output can be used to obtain numerical data at finite element nodes, elements, integration points or monitoring points. The text output is selected from the menu item **File | Text printout…**. This selection opens the window that is shown in Figure 74. The window is composed of two main sub-windows. The left-hand window contains a tree structure of the available data types. The requested data should be checked in this tree, and then by clicking the button **Generate**, an alphanumerical output will be created in the right-hand window. The contents of this window can be printed, saved to a file or copied to another program using the system clipboard.

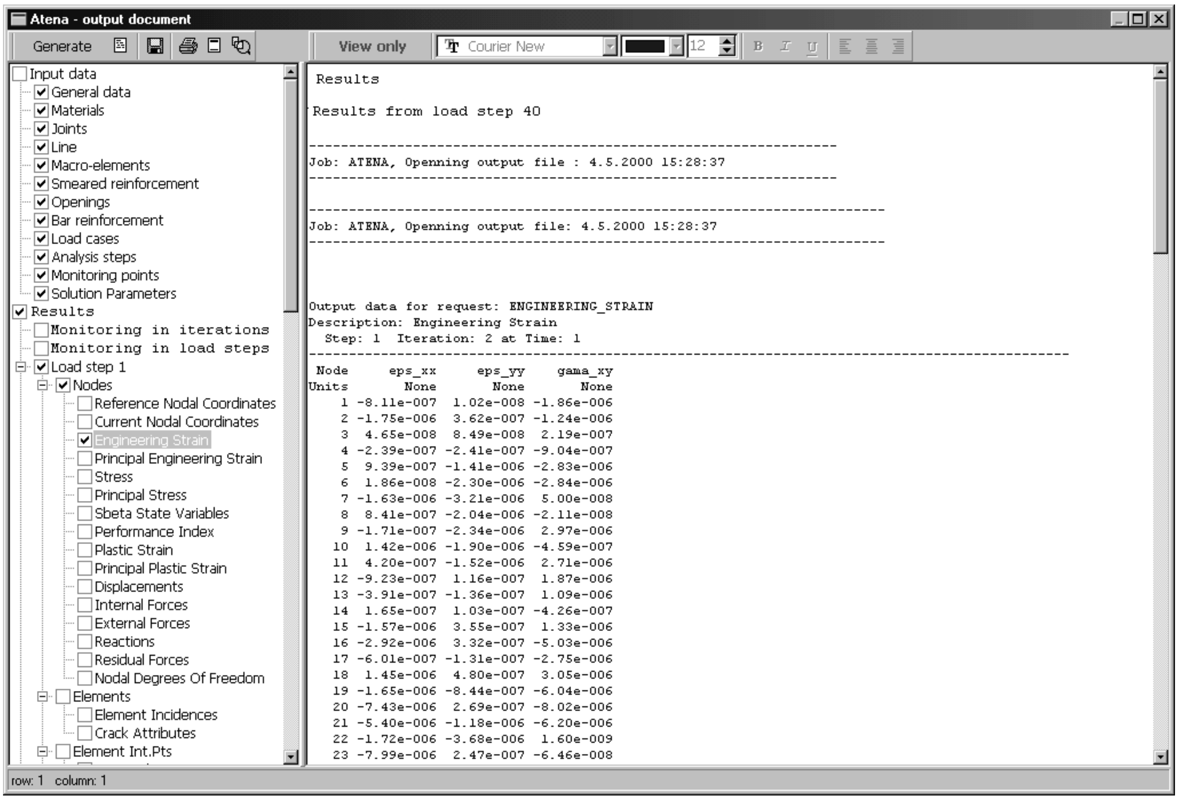


Figure 74: The program window for the definition of alpha-numerical output.

5.5 Analysis Log Files

The program **ATENA 2D** consists of several modules. The two main modules are the graphical user interface (GUI) and the analysis module. These two modules communicate with each other through the Microsoft component object model (COM) interfaces and also through four file streams. The contents of these streams for each analysis step can be examined using the menu item **Calculations | Step information**. This action opens the following window on your computer screen:

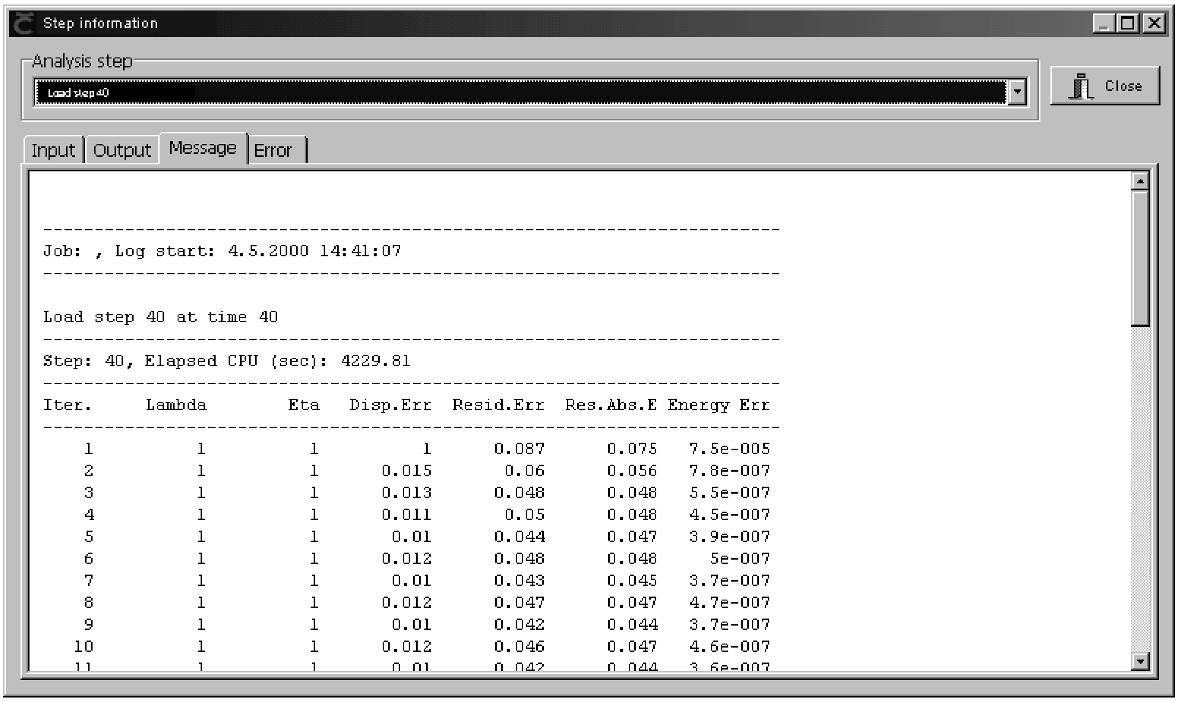


Figure 75: The step information window contains the input and output files from the finite element analysis.

It is possible to view the contents of these data streams for each analysis step, which can be selected from the pull-down list at the top of the window. The content of each data stream can be examined by selecting an appropriate bookmark at the top part of the window.

The input stream contains the commands that were passed from the GUI to the analysis module. In the first step, it contains the definition of the numerical model. In the subsequent load steps it contains the definition of supports, loads and solution parameters. The format of this file is described in the ATENA Input File Format manual [4]. The advanced users can modify the contents of this file before executing the analysis if proper settings are defined in **Options | Settings**. Only users experienced with the program **ATENA** and the format of this file should modify the input file, otherwise they can damage their data, which may then become unusable.

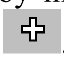
The output stream contains the output from the analysis module. Normally this stream is empty since it is used later when text output is requested.

The message stream contains the information about the analysis progress as they appeared also in the interactive window during the non-linear analysis.

The error stream contains error and warning messages from the analysis modules. This stream should be examined for errors that might have occurred during the numerical calculations.

5.6 Cuts

Starting from version 1.2.0 or younger, the program **ATENA 2D** enables the definition of cuts, along which scalar quantities can be evaluated and displayed. The cut can be a single straight line, an arc, or a polygon consisting of straight lines or arcs. The cuts must be defined in the pre-processing window, but does not have to be defined before the numerical analysis. It is possible to define them even after some results and load steps are calculated, just by switching between the pre- and post-processing windows.

The cut definition starts in the pre-processing window by selecting the item **Cuts** in the data access tree in the left side of the program window. After that the procedure is an analogy to the definition of reinforcement bars (see Section 3.7). It is again possible to define the cut geometry by mouse or by numerical values. The graphical input can be activated using the button .

In this example the numerical input is used, and it is started by the **Add** button in the **Cut** table window, which appears at the bottom of the program window, after the item **Cuts** is highlighted in the data access tree.

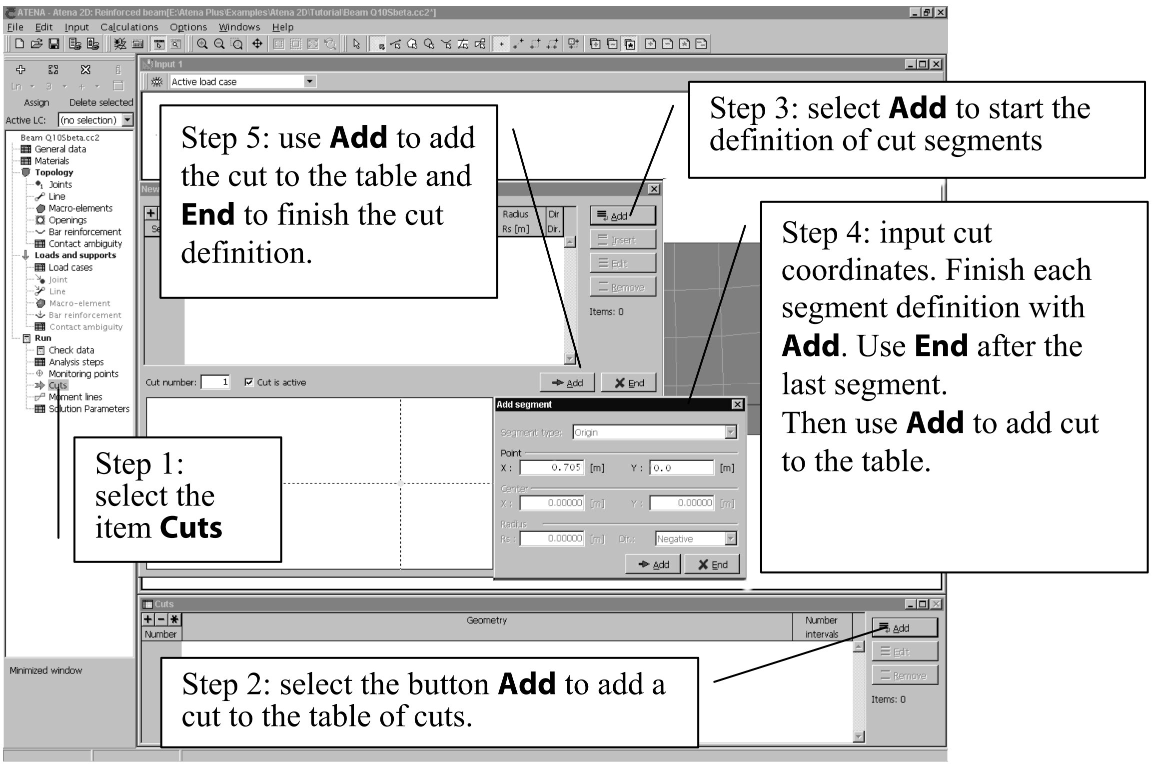


Figure 76: The beginning of the numerical definition of cuts.

Use the procedure above to define two cuts with the following end point coordinates:

Table 2: The coordinates of points for the cut definition.

	Cut 1		Cut 2	
	Beginning	End	Beginning	End
x [m]	0.705	0.705	1.275	1.275
y [m]	0.000	0.320	0.000	0.320

Then the program window shows the structure with two vertical cuts denoted by yellow color.

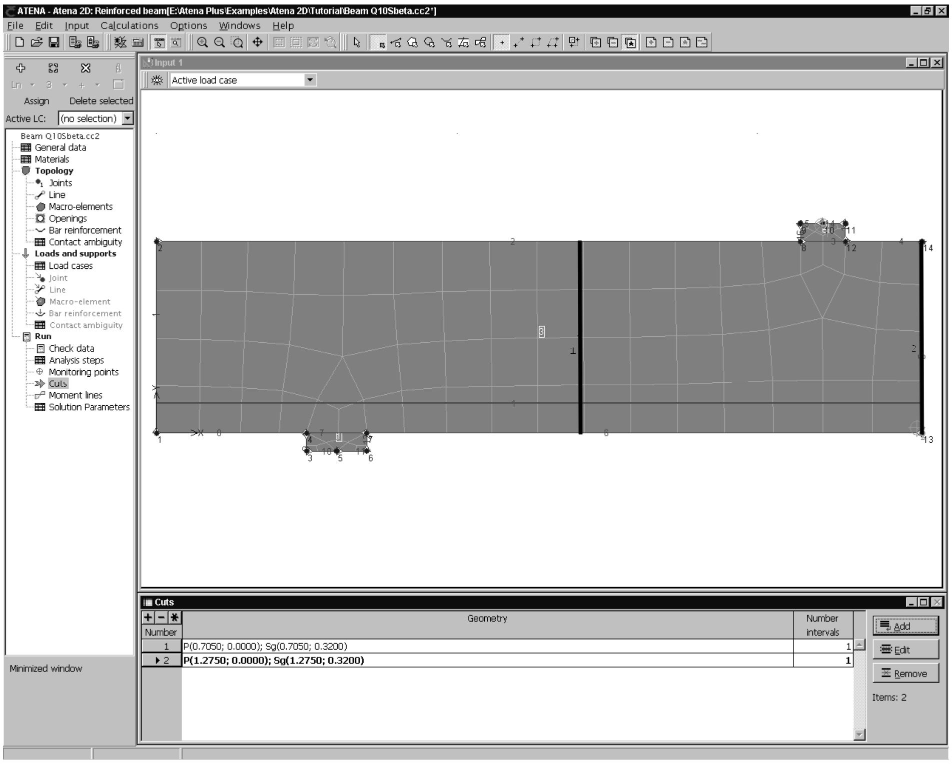
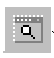


Figure 77: The program window shows two cuts after the cut definition.

When the cuts are defined, it is possible to switch into the post-processing mode (the button ). In this window it is possible to select which quantities are displayed along the cut lines. Same data as in the scalar plot are always displayed along the cuts, therefore the cut display is controlled from the **Scalar** sheet in the post-processing toolbar that is normally located along the left edge of the program window. The combo-box item **Show cuts with labels** must be selected in order to see the display of cut data (see Figure 78).

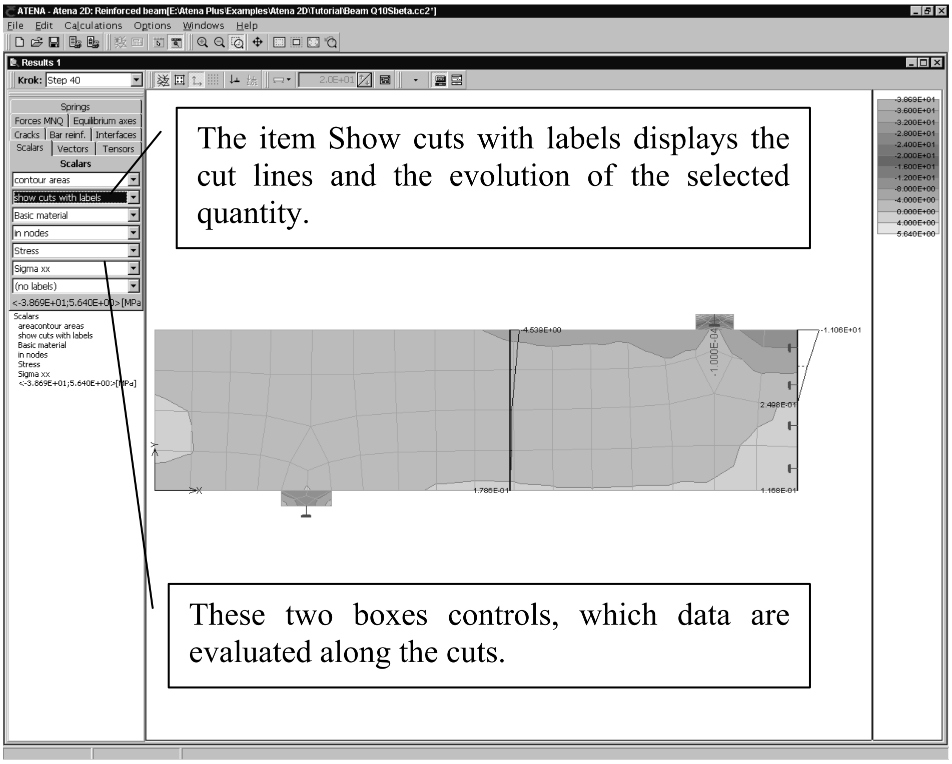


Figure 78: The post-processing window with the display of cut lines.

If necessary the display of certain cuts can be deactivated using the main menu item **Options | Activity**. In this dialog it is possible to define which cuts, reinforcements or moment lines are to be displayed on your computer screen.

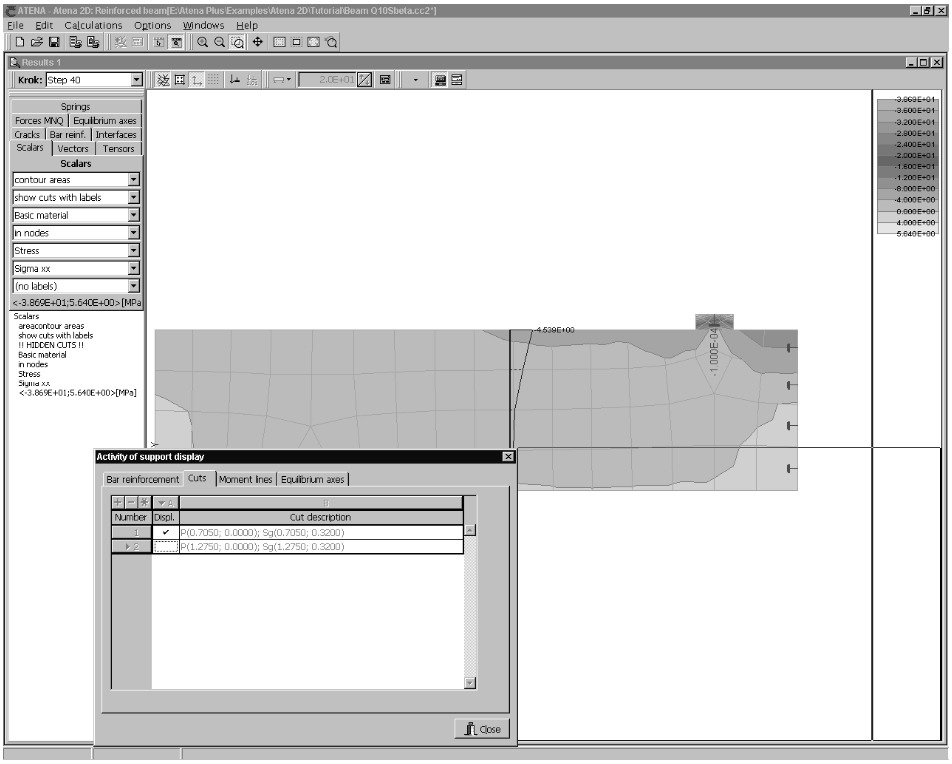


Figure 79: The visibility of cuts, moment lines or reinforcement bars can be specified from the main menu item Options | Activity.

The resulting figure can be copied to the clipboard or printed using the same procedure as it is described in Section 5.2.

5.7 Diagrams of Internal Forces

The ATENA program version 1.2.0 and younger contains a unique feature that enables the calculation of moment, shear and normal forces diagrams for beam-like structures. The user only has to specify the center line, to which the internal forces are to be calculated. This moment line can be only defined in the pre-processing window, but it does not have to specified before the analysis. It can be defined any time during the whole analysis process just by switching into the pre-processing model using the button .

The moment line definition starts in the pre-processing window by selecting the item **Moment lines** in the data access tree in the left side of the program window. After that the procedure is an analogy to the definition of reinforcement bars or cuts (see Section 3.7 or 5.6). It is again possible to define the line geometry by mouse or by numerical values. The graphical input can be activated using the button .

In this example the numerical input is used, and it is started by the **Add** button in the **Moment lines** table window, which appears at the bottom of the program window, after the item **Moment lines** is highlighted in the data access tree.

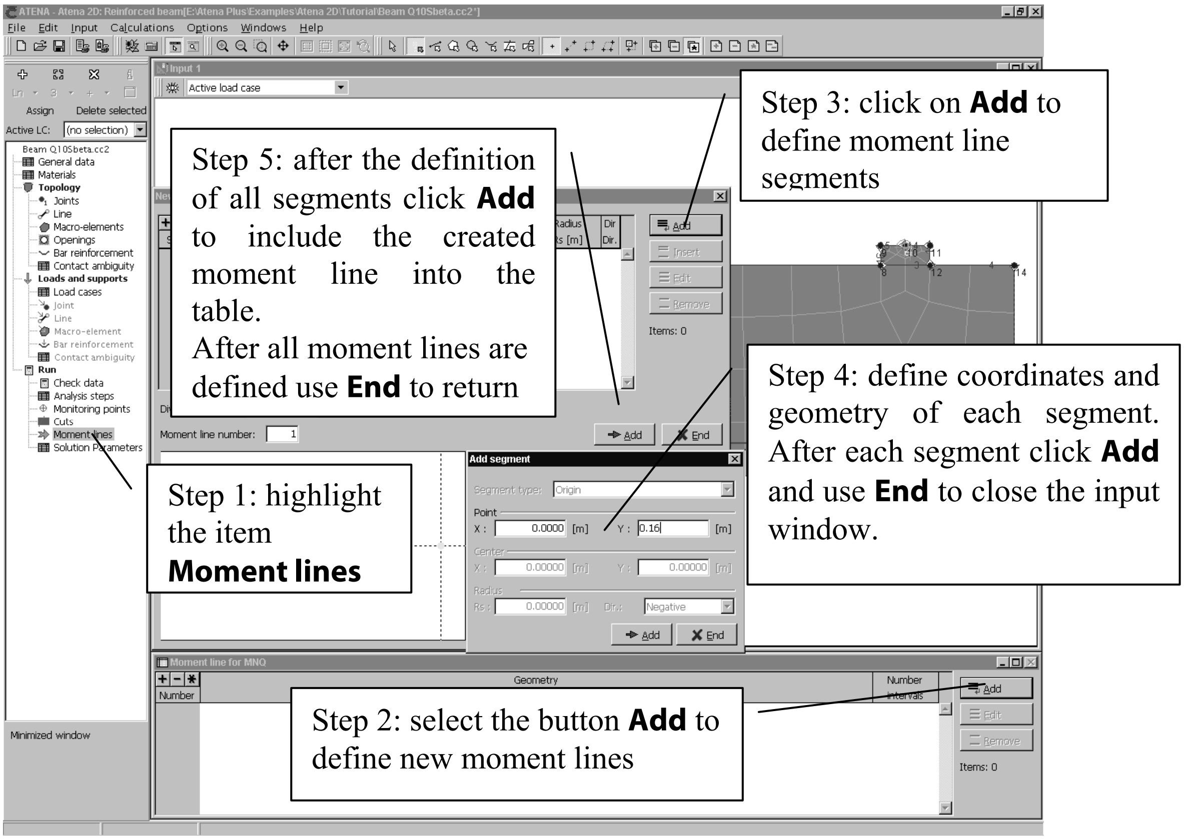


Figure 80: The procedure for the definition of moment lines for the calculation of internal forces.

Use the above steps to define a centerline that is composed of only one linear segment starting at the coordinates (0;0.16) and extending up to the point (1.275;0.16).

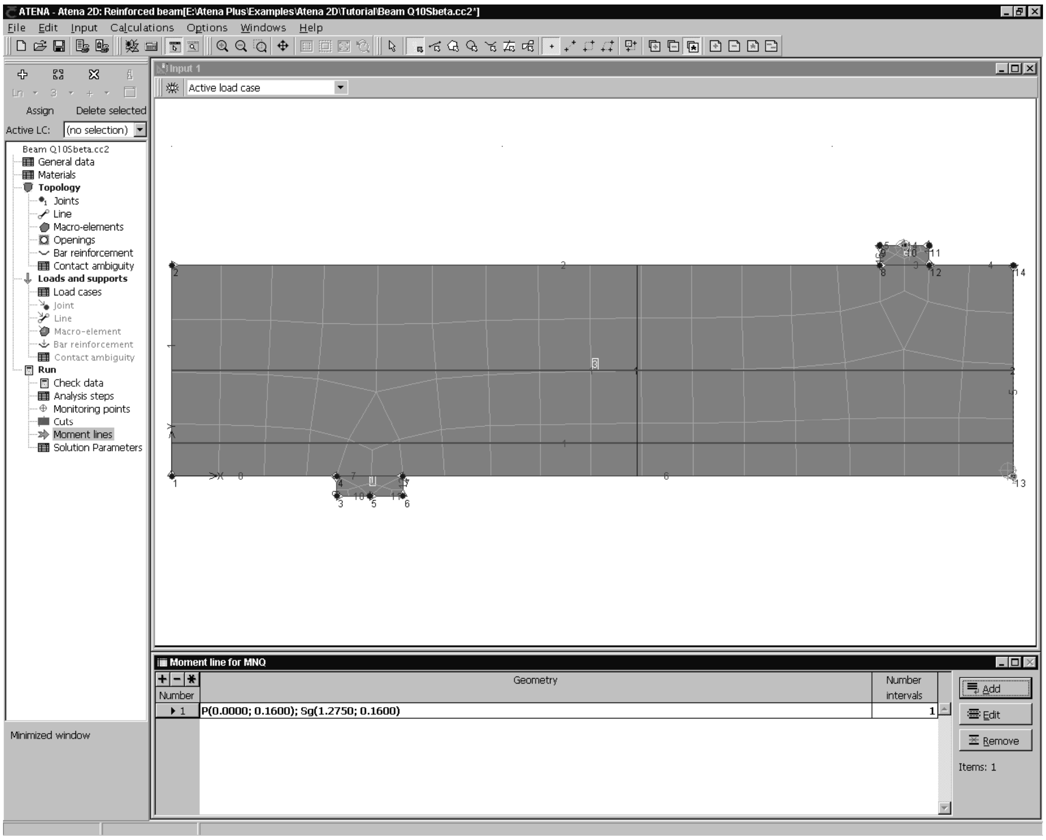


Figure 81: The pre-processing window after the specification of one moment line: coordinates (0;0.16) -> (1.275;0.16).

The diagram of the internal forces distribution can be obtained only when the program is in a post-processing mode, therefore, the next step is to select the button to enter the visualization mode.

There is a special sheet: **Moment lines**, in the post-processing toolbar along the left side of the program window. If it is selected and if the top box shows the note: **show and label** the diagrams of moment, normal and shear forces are displayed on the screen as it is shown in Figure 82. Please note that if a display of iso-lines or vector plot has been previously selected, it remains on the screen, and the internal forces diagram is drawn over the original picture. In many cases this is what the user wants, otherwise the iso-line or vector display must be deactivated using the appropriate sheets in the left side toolbar.

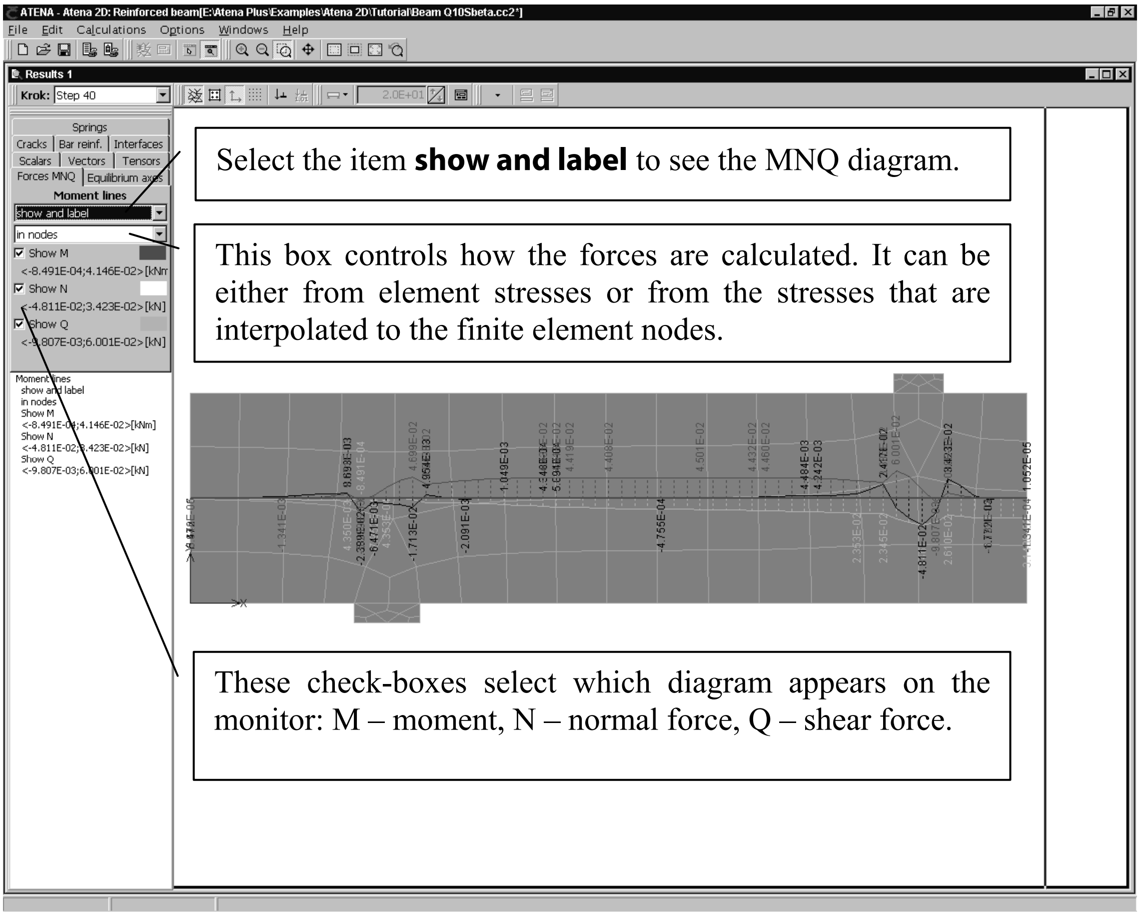


Figure 82: The diagram of internal forces in the post-processing window.

The program **ATENA 2D** offers two methods for the calculation of the internal forces. A suitable method can be selected from the second box in the **Moment lines** sheet in the post-processing toolbar, which is normally located along the left side of the program window. The first method calculates the internal forces from stresses that are interpolated and averaged at nodes. The second approach calculates the forces directly from stresses at element nodes. In this method there is no averaging of stresses between elements. The first method tends to smooth the stress field, and therefore, it can hide some spikes from being considered in the internal forces calculation.

6. CONCLUSIONS

This tutorial provided a step by step introduction to the usage of **ATENA 2D** on an example of a reinforced concrete beam without shear reinforcement. Although this example is relatively simple from geometrical and topological point of view, it is not a simple problem from the numerical point of view. Due to the missing shear reinforcement the beam fails by a diagonal shear crack, which is very difficult to capture using smeared crack approach. This example demonstrates the powerful simulation capabilities of **ATENA** for modeling the brittle failure of concrete structures. Even with very coarse mesh, which was used in this demonstration example, the diagonal shear crack was successfully captured. Further improvement of the results can be achieved by decreasing the finite element size to at least 6-8 elements over the beam height. It is also possible to select different quadrilateral elements: CCQ10 or CCQ10SBeta, which should exhibit even better behavior in shear dominated problems.

The objective of this tutorial is to provide the user with basic understanding of the program behavior and usage. For more information the user should consult the user's manual [2] or contact the program distributor or developer. Our team is ready to answer your questions and help you to solve your problems.

The theoretical derivations and formulations that are used in the program are described in the theory manual [1].

The experienced users can also find useful information in the manual for the analysis module only [4].

7. Program Distributors and Developers

Program developer: **Červenka Consulting s.r.o.**
Na Hrebenkach 55, 150 00 Prague 5, Czech Republic
phone: +420 220 610 018
fax: +420 220 612 227
www.cervenka.cz
email: cervenka@cervenka.cz

The current list of our distributors can be found on our website:
http://www.cervenka.cz/company/distributors/

LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2000-2014

[2] ATENA Program Documentation, Part 2-1, ATENA 2D User's Manual, CERVENKA CONSULTING, 2000-2014

[3] ATENA Program Documentation, Part 3-1, ATENA Engineering Example Manual, CERVENKA CONSULTING, 2000-2010

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2000-2014

[5] Leonhardt and Walther, Schubversuche an einfeldringen Stahlbetonbalken mit und Ohne Schubbewehrung, Deutscher Ausschuss fuer Stahlbeton, Heft 51, Berlin 1962, Ernst&Sohn.