

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

ATENA Program Documentation Part 4-1

Tutorial for Program ATENA 2D



Written by:

Jan Červenka

Prague, June 2, 2015

Trademarks:
ATENA is registered trademark of Vladimir Cervenka.
Microsoft and Microsoft Windows are registered trademarks of Microsoft Corporation.
Other names may be trademarks of their respective owners.

Copyright © 2000-2015 by Červenka Consulting s.r.o.

CONTENTS

1.	INTRODUCTION1
2.	STARTING PROGRAM
3.	PRE-PROCESSING
3.1	Introduction
3.2	Material Parameters
3.3	Geometrical Joints
3.4	Geometrical Lines
3.5	Geometrical Macro-elements
3.6	Mesh Generation
3.7	Bar-reinforcement
3.8	Supports and Actions
3.9	Loading History and Solution Parameters
3.10	Monitoring Points
4.	FE NON-LINEAR ANALYSIS
4.1	Introduction
4.2	Starting Analysis
4.3	Interactive Window
4.4	Adding New Load Steps
5.	POST-PROCESSING
5.1	Introduction
5.2	Post-processing Window
5.3	Load-displacement Diagrams
5.4	Text Output
5.5	Analysis Log Files
5.6	Cuts
5.7	Diagrams of Internal Forces
6.	CONCLUSIONS
7.	PROGRAM DISTRIBUTORS AND DEVELOPERS60
LITE	RATURE

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.

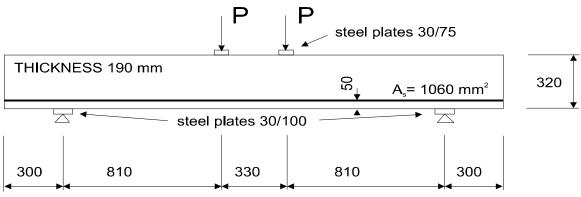


Figure 1: Geometry of the structure.

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 macroelements or regions, by specifying a list of boundary lines, which surround the macroelement.

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.

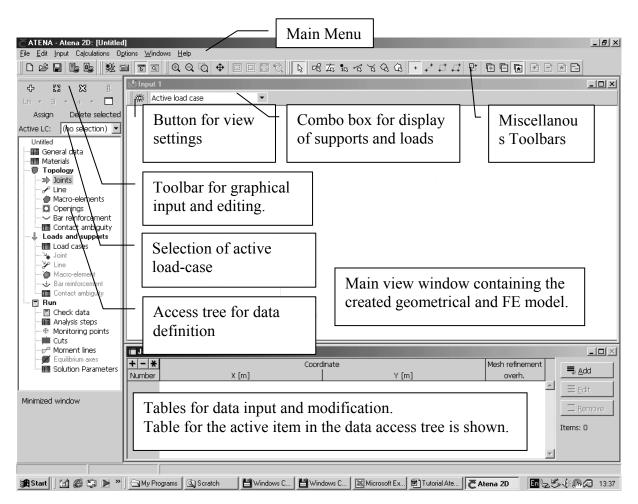
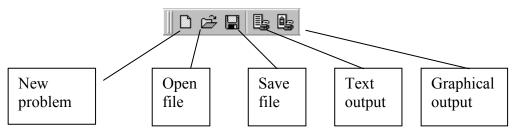


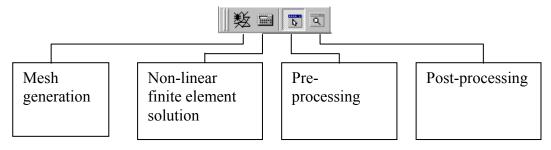
Figure 2: Graphical user interface of ATENA 2D pre-processor.

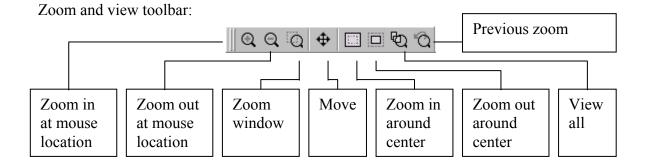
ATENA 2D contains four main toolbars:

File toolbar:

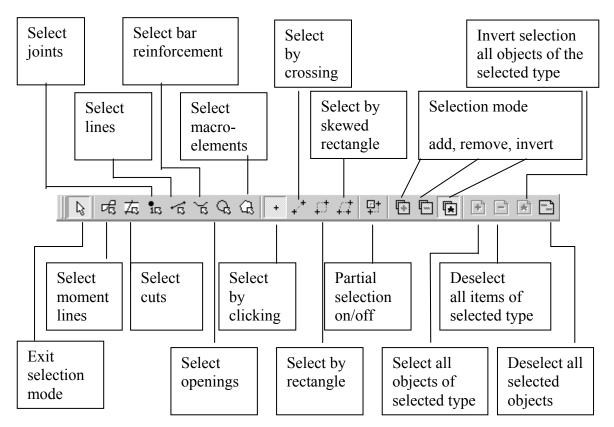


Solution toolbar:





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.

🏢 General data			
General data			Layers of smeared reinforcemer
Note:			Add Remove
Numbering information	0 1 (max		
Nodes : Macroelements :	0 Lines 0 Openings	: 0 : 0	
Bar reinforcements : Load cases:	0		

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

G	ieneral data		1
	Description		
	Description :	Reinforced beam	
	Note :	Beam without shear reinforcement.	
		✓ OK X Cancel	

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.

iii Materi	als		
+ - * Number	Material name	Number references	Add
			E Edit
			<u> </u>
			Items: 0
			v

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.

ew material	<u>></u>
Material type	
3D Cementitious	•
Plane Stress Elastic Isotropic Plane Strain Elastic Isotropic Axi Sym Elastic Isotropic 1D Elastic Isotropic 3D Cementitious 3D Non Linear Cementitious SBeta Material 3D BiLinear Steel Von Mises	

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

New material.Plane Stress Elastic Isotropic	×
Name: Steel plates	
Basic Miscellaneous	
Elastic modulus E : 2.100E+05 [MPa]	
Poisson's ratio MU : 0.300 [-]	
Material #: 1	🗙 <u>C</u> ancel

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

vew material	:
Material type	
3D Cementitious	
3D Cementitious 3D Non Linear Cementitious SBeta Material 3D BiLinear Steel Von Mises 2D Interface	
Reinforcement	
Cycling Reinforcement Smeared Reinforcement	

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

Edit material #3:Re	inforcement ×
Name: Reinforcement	
Basic Miscellaneous	
Type : Bilinear 💌	Stress-strain law
Elastic modulus E : 208000.000 [MPa]	
σ _y : 560.000 [MPa]	$-\uparrow^{\sigma}$
	\longrightarrow
	3 / /
	$- \sigma_v$
Active in compression	
Material #: 3	✓ <u>O</u> K X <u>C</u> ancel

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.

N	ew material	×
Гг	Material type	
	3D Cementitious	
	3D Cementitious - 3D Non Linear Cementitious -	
	SBeta Material 3D BiLinear Steel Von Mises	
	2D Interface	
	Reinforcement	
	Cycling Reinforcement	
	Smeared Reinforcement	

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.

<n>Material generation</n>					
-Material proper	Material properties generation				
R _{cu} :	33.5	[MPa]			
<> Previous	d⇒ <u>N</u> ext	X <u>C</u> ancel			

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.

New material.SBeta Material Name: Concrete	
Basic Image:	Stress-Strain Law $f_t \uparrow^{\sigma}$ $f_t \uparrow^{\sigma}$ $f_c \uparrow^{\sigma}$ $f_c \uparrow^{\sigma}$ $f_c \uparrow^{\sigma}$ $f_t \uparrow^{\sigma}$ $f_c \uparrow^{\sigma}$ $f_t \uparrow^{\sigma}$
Material #: 2	⊲⊨ Previous ✓ Einish ✗ Cancel

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.

Edit material #2:SBeta Material	×
Name: Concrete	
Basic Tensile Compressive Shear Miscellaneous	
Type of tension softening: Exponential \checkmark Specific fracture energy G _f : 5.500E-05 [MN/m]	Crack opening law
Crack model: Fixed	
Material #: 2	✓ <u>O</u> K X Cancel

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

New material.SBeta Material	×
Name: Concrete	
Basic Tensile Compressive Shear Miscellaneous	
Compressive strain at compressive strength in the uniaxial compressive test EPS_C : Reduction of compressive strength due to cracks:1.795E-03[-]Type of compression softening:Crush BandImage: Crush BandImage: Crush BandCritical compressive displacement w_d\- :-5.0000E-04[m]	Peak compressive strain σ, ε ε τ ε ε τ ε τ ε τ τ ε
Material #: 2	s 🖌 Einish 🗶 Cancel

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

New material.SBeta Material
Name: Concrete
Basic] Tensile] Compressive Shear] Miscellaneous]
Shear retention factor : Mariable Variable shear retention Tension-compression interaction: Hyperbola A Image: Comparison of the shear retention Tension-compression interaction: Hyperbola A Image: Comparison of the shear retention
Material #: 2 <-> Previous ✓ Einish ✗ Cancel

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

New material.SBeta Material
Name: Concrete
Basic Tensile Compressive Shear Miscellaneous
Specific material weight Rho : 2.300E-03 [MN/m ³]
Coefficient of thermal expansion ALPHA : 1.200E-05 [1/K]
Material #: 2

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

iii Materi	als		
+ - * Number	Material name	Number references	Add
	Steel plates	0	Edit
2	Concrete	0	 <u> </u>
▶ 3	Reinforcement	0	<u> </u>
			Items: 3
ļ			1

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 \clubsuit , which will be followed by mouse picking at new joint locations, or by clicking the **Add** button in the **Joints table** window.

New joints.	×
Topology X coord.: 0.0000 Y-coordinate: 0.0000	Springs
Mesh refinement Method of refinement	
	Add Edit Remove
Joint # : 1	→ <u>A</u> dd X Cancel

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.

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

Table 1: Coordinates of the geometrical joints.

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 ⁵³ 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.

ATENA - Atena [C:\Work\Tu Ele Edit Input Calculations (C) 口 皮 目 隆 隆 隆 繁 節	D <u>p</u> tions <u>W</u> indows <u>H</u> elp)	₽ <i>₽</i> ४ २	<mark>・</mark> * t <i>t</i> ₽ @		_B×
+ C3 XX 1	№ View	Scale the view so all (bjects are visible			_ 🗆 🗙
Ln = 3 = + =	🛛 🗰 Active load case	1				
Assign Delete selected		/ 🔨				
Active LC: (not available)		/				
Beam.cc2						
🛄 General data		Use this but	ton to			
Materials						
→ >>> Joints		scale the vie	W/			
≁ Line						
Macro-elements		such that the	e joints			
Bar reinforcement			2			
Contact ambiguity		fill the whol	e			
 Loads and supports Load cases 		window				
Joint		window				
>P Line						
→ Macro-element 						
Contact ambiguity						
🛄 Run						
 Check data Analysis steps 		1	X · · · · · · · · ·			
Monitoring points						
Solution Parameters						
	<u></u>					
	🛄 Joints					
	+ - *	Coord	inate		Mesh refinement	🚍 Add
	Number	X [m]		Y [m]	overh.	
	8	1.0725		0.3200	·	i ≣ Edit
Minimized window	10	1.0/25		0.350		Ξ Remove
	11	1.1475		0.3500	-	
	12	1.1475		0.3200		Items: 14
	13	1.2750		0.0000	-	
	▶ 14	1.2750		0.3200		

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

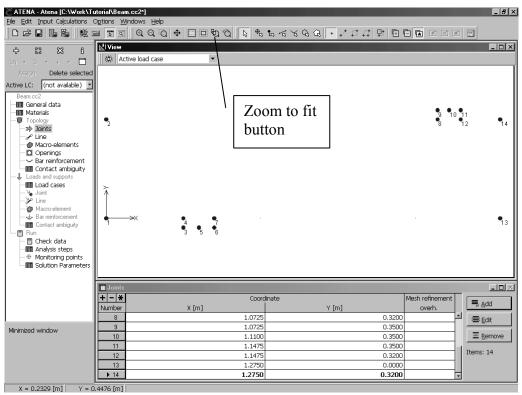


Figure 20: The program window after the selection of the zoom to fit button.

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.

ATENA - Atena [C:\Work\Tt File Edit Input Calculations						_ 8 ×
		\$ 0 0 0 ¢	j ↓ ⊕ • ~ ~ ~	<i>G G</i> I + + <i>I D D</i>		
+ 33 × 1	M I View					_O×
	🛛 🗱 🛛 Active load case	•				
Add Assign Delete selected						
Active LC: (not available) 💌						
Beam.cc2 General data						
Materials					9 1 0 1 1	_
Topology	•2				8 12	1 4
Openings						
Bar reinforcement						
Loads and supports						
You Joint	⊼					
Bar reinforcement	↓ →×	4 . 7				1 3
🗊 Run		4 7 3 5 6				
Check data 						
→ ⊕ Monitoring points → ■ Solution Parameters						
	im Line					
	+ - * Line type		Line topology		Connection Refinement	Add
	Number					Edit
, Minimized window						
						<u>R</u> emove
					Iten	ns: 0

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

Line prototype.	×
Topology Line type : Line	Springs
Joints: Origin: End: Center: X : 0.0000 [m] Y : 0.0000 [m]	
Radius : 0.0000 [m] Orientati Mesh refinement Refinement method:	/ If needed spring support can be added along lines.
Line # :	Add Edit Remove

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

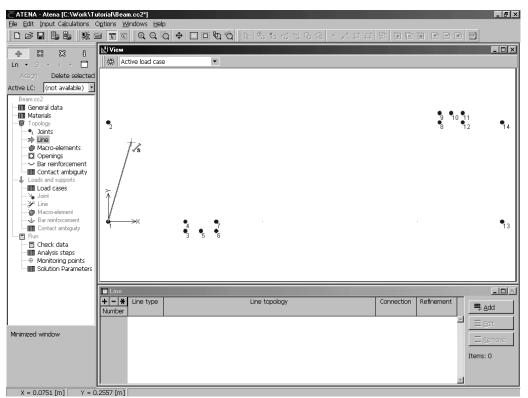


Figure 23: In the graphical mode, a geometrical line is defined by first selecting a line beginning and a line end joint by mouse. The order of end points is not important in ATENA.

CATENA - Atena [C:\Work\T			- 8 ×
Eile Edit Input Calculations		\$;;<;<,<,, +, ≠ ≠ ₽ ₽ ₽ ₽ ₽ ₽	
	🔆 🗰 Active load case 🗸		
Assign Delete selected			
Active LC: (not available)			
Beam.cc2			
		9 10 11	
Topology • 1 Joints		8 12	1 4
→>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>			
Openings			
Contact ambiguity	-		
	<u>↓</u>		
¥o Joint ¥o Line	↑		
Macro-element	↓ >× • •		
Contact ambiguity			1 3
🔤 🖩 Run 🔤 Check data	3 3 0		
→ ● Monitoring points			
Solution Parameters			
	Line		
	+ - * Line type	ine topology Connection Refinement	1
	Number 1 - 2		
Minimized window			
			ove
		Items: 1	
X = 0.0459 [m] Y = 0	2769 [m]	×.	

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

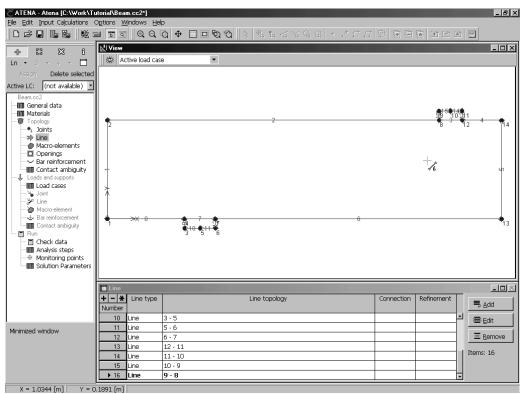


Figure 25: The program display after the definition of all geometrical lines.

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

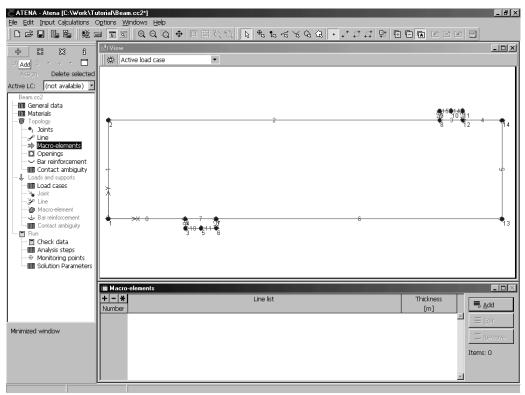


Figure 26: Program display at the beginning of the macro-element definition.

Macro-elen	nent # 3.
Topology Boundary list: 1-8	quad-, triangular or mixed mesh can be selected here.
FE mesh Mesh type: Quadrilaterals ▼ Element size 0.0440 [m] ✓ Smooth element shapes Properties Material : Concrete ▼ Thickness: 0.1900 [m] Quadrilateral elements: CCIsoQuad ▼ ✓ Geometrically nonlinear Macro-element # : 3	This value specifies the requested element size for automatic mesh generation A material model for the new macro-elements Element type for quadrilateral elements
Macro-element prototype.	(a
Boundary list:	
FE mesh Type of elemen Quadrilaterals Element size 0,08 [m] I Smooth element shapes	Layers of smeared reinforcement
Properties Material : Steel plates Thickness: 0.1900 [m] Quadrilateral elements: CCIsoQuad	No. of smeared reinf. layers should be
Geometrically nonlinear Macro-element # :	entered within general data.

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.

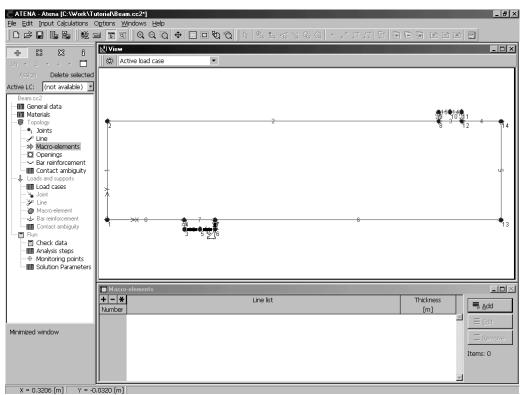


Figure 28: Selection of boundary lines for first macro-element representing the steel plate at the vertical support.

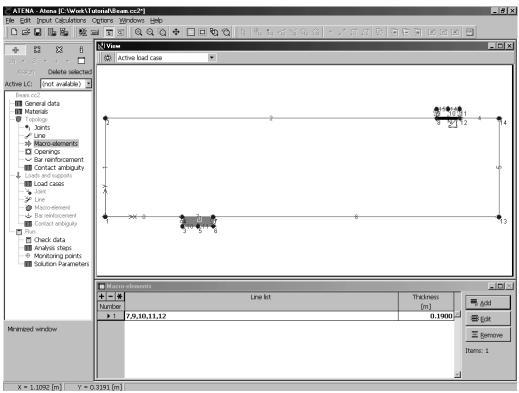


Figure 29: Program display after the first steel plate is defined and during the process of creating the second steel plate, which is located at the point of load application.

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.

Macro-element prototype.	
Topology Boundary list:	
FE mesh Type of elemen Quadrilaterals Element size 0.0800 [m] Smooth element shapes	Layers of smeared reinforcement
Properties Material : Concrete Thickness: 0.1900 [m]	
Quadrilateral elements: CCIsoQuad	No. of smeared reinf. layers should be entered within general data.
Macro-element # :	V OK X Cancel

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

Elle Edit Input Calculations	Options Windows Help	. 8 ×
□ C B B 数 :	■ ▼ 3 Q Q Q ♦ □ □ ♥ 6 \ % % % % % % % # # # # # # # # # # # #	
	X View	
Active LC: (not available)		
Beam.cc2 General data Materials Topology Topology Cline Macco-elements Coopenings Coopenings		¶14
Jose reinforcement Contact ambiguity Tortact ambiguity Oreck data Oreck Oreck data Oreck data Oreck data Oreck data	Macro-elements	•13
	Image: Second	
Minimized window	2 3,13,14,15,16 0.1900 ▶ 3 1,2,3,4,5,6,7,8 0.1900	ive
X = 0.6719 [m] X = 0	Tems: 3	

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.

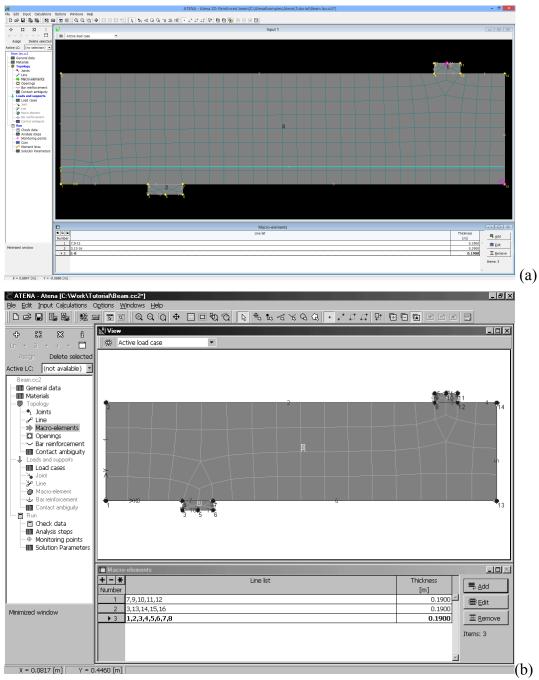


Figure 32: Generated finite element mesh using the element size of 0.044m (a) and 0.08 m (b).

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.

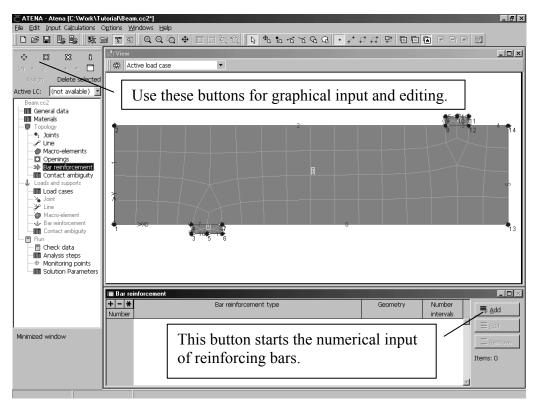


Figure 33: The program window at the beginning of the reinforcement bar definition.

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.

w reinforcement rods.	×
einforcement Normal	
consistent of center of products copology Properties Basic parameters Geometric non-linearity can be selected here. Area: 0.0E+00 [m²] Geometrically nonlinear Geometrically nonlinear Image: Calculate reinforcement area Image: Calculate reinforcement area Bar reinforcement Bar diameter 0.0260 [m] Number of bar 2	
OK ★ Cancel	
inforcement bar : 1 ▲dd ▲dd	1

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.

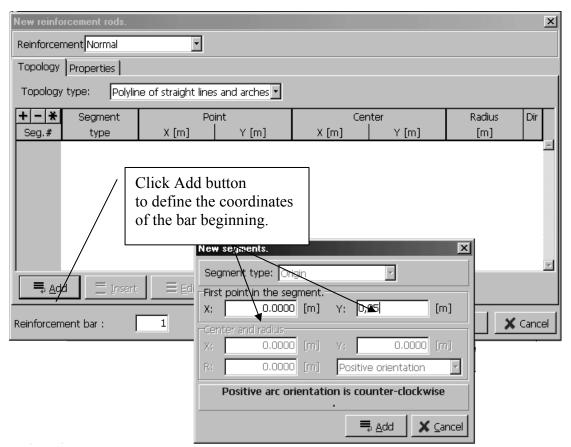


Figure 35: The sheet Topology is used for the definition of bar geometry. A reinforcement bar is composed of segments, and each segment can be a line, arc or a circle.

New reinforcement rods.							×
Reinforcement	•						
Topology Properties							
Topology type: Polylin	e of straight lines	and arches 💌					
+-* Segment	Poir	nt		Center		Radius	Dir
Seg.# type	X [m]	Y [m]	X [m]	Y [m]]	[m]	
1 Origin	0.0000	0.0500					<u> </u>
▶ 2 Line	1.2750	0.0500					
Select		segments.		efine the er	nd point	coordi	nates
📮 <u>A</u> dd 📑 Insert		iment type: Lin		/		(
Reinforcement bar :	1 X:	t point in the se 1.2750 Iter and radius—	[m] Y:	0.050	00 [m]	×	Cancel
	X	0.0000	[m] Y:	0.000	[m] 00	F-	
	R:	0.0000	[m] Po	sitive orientatio	n 🔽		
Positive arc orientation is counter-clockwise							
				≡ , <u>A</u> dd	🗙 <u>C</u> ancel		

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

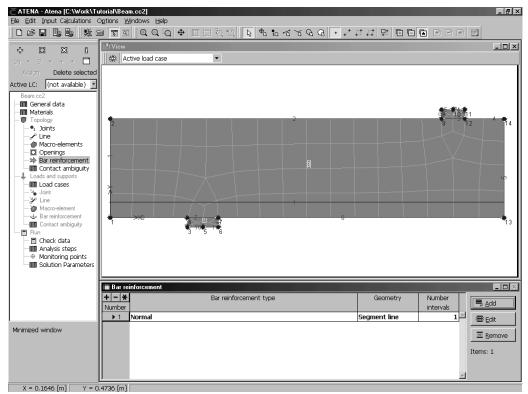


Figure 37: The program display after the definition of the reinforcement bar.

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.

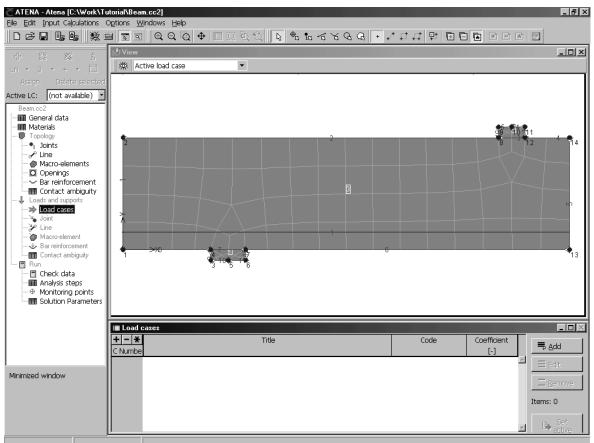


Figure 38: The load-case definition starts by highlighting the Load-cases item in the data access tree and clicking the Add button in the Load cases table.

New load ca	se			×
Load case	Load case with supports			
LC hame.				
LC Code:	Body force			
LC coeff.:	Body force Forces			
	Supports Prescribed deformation Temperature	r		
X:	Shrinkage Pre-stressing	': 	-1.0000	[m]
LC number	: 1		-> <u>A</u> dd	🗙 Cancel

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

Ne w load ca	se			×
-Load case-				
LC name:	Load case with actions			
LC Code:	Forces	•		
LC coeff.:	Body force Forces			
	Sunnorts			
	Prescribed deformation			
X (Temperature Shrinkage	1	-1.0000	[m]
	Pre-stressing			
LC number	2		→ <u>A</u> dd	🗙 Cancel

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

iiii Load c	Cases			
+ - *	Title	Code	Coefficient	📮 Add
C Numbe			[-]	
1	Load case with supports	Supports	1.000	⊞ <u>E</u> dit
▶ 2	Load case with actions	Prescribed deformati	1.000	
				<u> </u>
				Items: 2
			Y	Set active

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

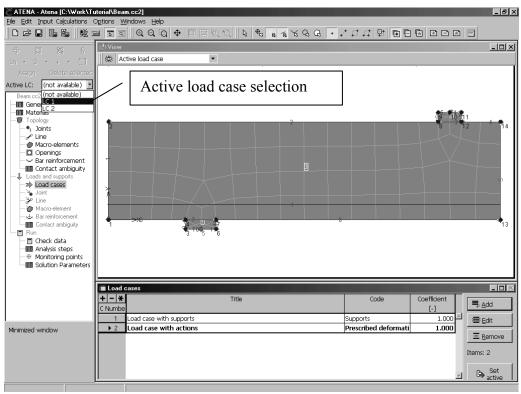


Figure 42: An appropriate active load case must be selected before the definition of supports. Supports should be in the load case 1.

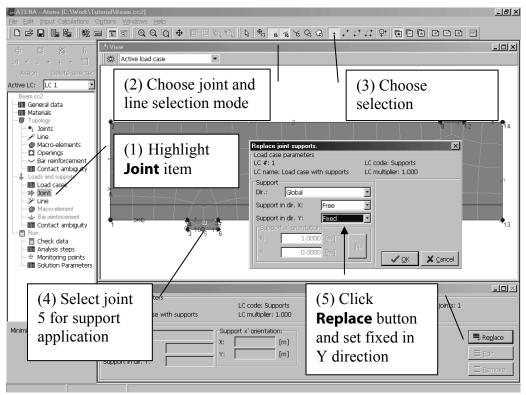


Figure 43: The definition of the vertical support at the bottom steel plate.

Ele Edit Input Calculations Ogtions Windows Help	18 ×
	٥×
Active load case Active load case Active load case Beam co2 Beam co2 Beam co2 Beam co2 Concepts	14
Minimized window Image: Log as parameters LC code: Support (3) Click Replace Minimized window Support as with supports LC code: Support (3) Click Replace Support Support in dir. X: Support in dir. Y: (3) Click Replace Support in dir. Y: Support in dir. Y: Support in dir. Y:	

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

Ele Edit Input Calculations	Options <u>W</u> indows <u>H</u> elp	_ <u>-</u> X
C C C C C C C C C C C C C C C C C C C	Active load case	<u>- 0 ×</u>
	Replace prescribed displacements Load case parameters Load case parameters Load case parameters LC code: Prescribed deformation LC rame: Load case with actions LC multiplier: 1.000 Prescribed displacements UC name: Load case with actions Dr:: Global Support in dir. X: Free Support in dir. Y: Fixed	1 4
Minimized window	Ident support with displacement Lod case parameters LC #: 2 LC code: # LC c	i .

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.

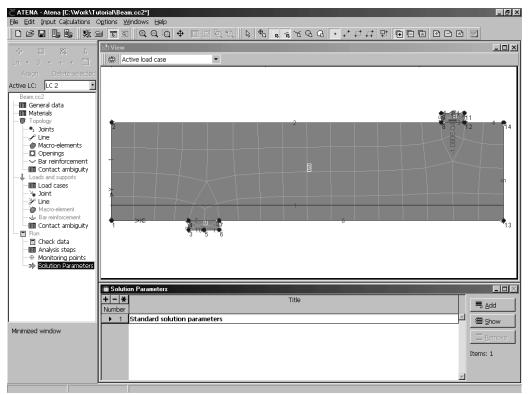


Figure 46: The program display with the table of solution parameters. Standard solution parameters can be examined by clicking the button Show. New set of solution parameters can be created using the button Add.

Editing s	olution parameters	2		×
Genera	al Line Search			
Title:	Solution Paramet	ters		
Soluti	on method:	Newton-Raphs	son 💌	☑ Line search
Optim	ize node numbers:	Sloan	•	
Updat	e Stiffness:	Each iteration	•	
Stiffn	ess Type:	Tangent	-	
Iterat	ion number limit:		60	
Displa	cement error tolerar	nce:	0.010000	[-]
Residu	ual error tolerance:		0.010000	[-]
Absol	ute residual error tol	erance:	0.010000	[-]
Energ	y error tolerance:		0.000100	[-]
			V 0	K X Cancel

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

Editing solution parameters	2
General Line Search	
Solution method:	With iterations
Unbalanced energy limit:	0.800 [-]
Limit of line search iteration	ons: 3
Line search limit - min.:	0.001 [-]
Line search limit - max.:	1.000 [-]
	✓ OK 🗙 Cancel

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

Editing solu	tion parameter	rs 4 ×
General Line Search Conditional	Break Criteria	
	Break immediatelly	Break after step
Displacement error multiple:	10000.0	10.0 [-]
Residual error multiple:	10000.0	10.0 [-]
Absolute residual error multiple:	10000.0	10.0 [-]
Energy error multiple:	100000.0	1000.0 [-]
		✓ OK X Cancel

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

iiii Solutio	on Parameters	
+ - * Number	Title	🚍 Add
1	Standard solution parameters	⊠ ≣ Edit
▶ 2	Solution Parameters	
		<u> </u>
		Items: 2
		T

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

EILE Edit Input Calculations		_ <u>8 ×</u>
-t+ S3 X A Ln + 3 + + □ Assian Delete selected	View Active load case	.o×
Active LC: LC 2	Add analysis steps Add analysis steps Analysis step Load cases: 1.2 Step multiplier Multiplier: 1.000 [-] Save load step results Selution Parameters Selution Parameters Selution Parameters Add X Cancel	₽ • • •
Minimized window	Analysis steps Load step lst Coefficient Prameters Save Calculated index is index is index is index is index i	
	E Eensi O	ove

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.

Add analysis steps					×
Analysis step Load cases: 1,2		Step multiplie Multiplier:	er 1		[-]
Solution Parameters Solution Parameters	🔽 Sav	e load step resul	lts		
		+	<u>A</u> dd	X (Cancel

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.

🔳 Analysis steps							_ _ _ ×
+ - *	Load step list	Coefficient	Prameters	Save	Calculated	Γ	
Number		[-]	analysis	results	results		<u>A</u> dd
15 1,2		1.000	Solution Parameters	Yes	No		🖃 Insert
16 1,2		1.000	Solution Parameters	Yes	No		
17 1,2		1.000	Solution Parameters	Yes	No		🚟 <u>E</u> dit
18 1,2		1.000	Solution Parameters	Yes	No		
19 1,2		1.000	Solution Parameters	Yes	No		<u> </u>
▶ 20 1,2		1.000	Solution Parameters	Yes	No		Items: 20

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 \clubsuit , 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.

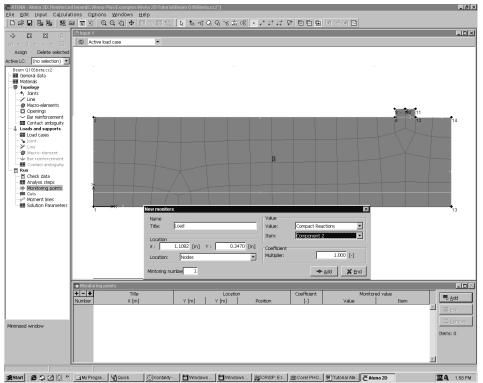


Figure 54: The definition of the first monitoring point.

New monitors		×
Name	Value	
Title: Delfection	Value:	Displacements 💌
Location	Item:	Component 2
X: 1.2668 [m] Y: 0.0084 [m]	Coefficient	
Location: Nodes	Multiplier:	1.000 [-]
Mintoring number 2		➡ <u>A</u> dd 🗙 End

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

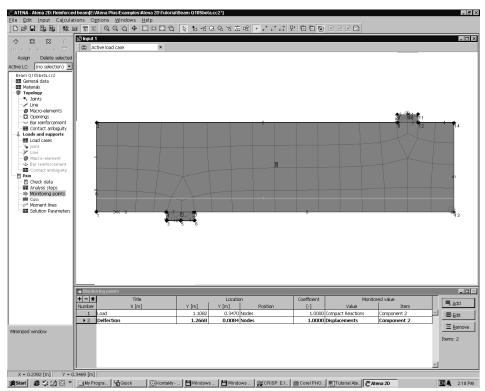


Figure 56: The program display after the definition of monitoring points.

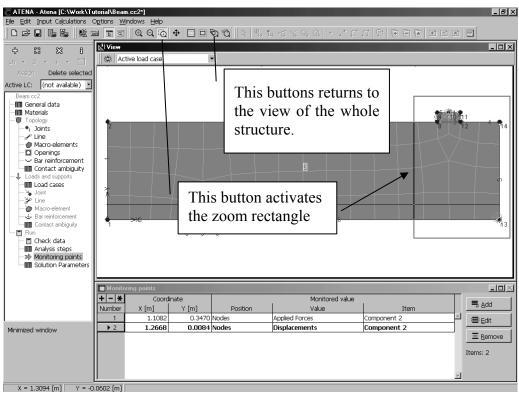


Figure 57: The way, in which the program selects the closest node for monitoring becomes more apparent after zooming at the middle section of the beam. The button returns the view to the state when the whole structure is displayed.

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.

ATENA - Atena [C:\Work\Tu File Edit Input Cajculations (_ 8 ×
		<u>- </u>
Active LC: (not available) Beem cc2 - Marchalable) Beem cc2 - Marchalable - Marchalable	Drawing setting: Image: Comparing setting: General Joints Line Macro-elements Openings Bar reinforcement Monitoring points Finite element mesh Loading Image: Show FE mesh Bernent type: Basic material No smeared reinful layers are defined! Image: Label elements Image: Comparing setting: Image: Comparing setting: </td <td>14 to</td>	14 to
	Image: Second	
Minimized window	▶1 0.0000 0.0000 ▲ 2 0.0000 0.3200 ▲ 3 0.2500 -0.0300 ▲ 4 0.2500 0.0000 ■	nove
	5 0.3000 -0.0300 6 0.3500 -0.0300 7 0.3500 0.0000	4

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

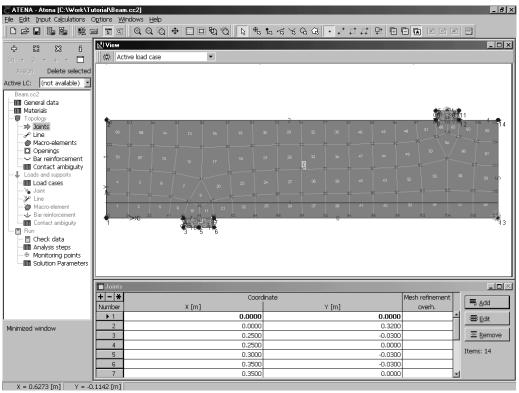


Figure 59: The finite element mesh along with node and element numbers. The size of characters can be modified from the menu item Options | Settings.

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.

F - ¥	analysis steps	В	С	ТΙ	-Initial data for LD	-
Number	Analyze	Save results	State	1		nts Component 2
▶ 1	Yes 🔻	Yes	▼ Not analyzed	-	Y: (undefined)	
2	Yes 🔻	Yes	▼ Not analyzed		(undefined) (step/iterati	
3	Yes 🔻	Yes	▼ Not analyzed			es Component 2
4	Yes 🔻	Yes	▼ Not analyzed		Displacemen	its Component 2
5	Yes 🔻	Yes	▼ Not analyzed			
6	Yes 🔻	Yes	▼ Not analyzed			
7	Yes 🔻	Yes	▼ Not analyzed	.		

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.

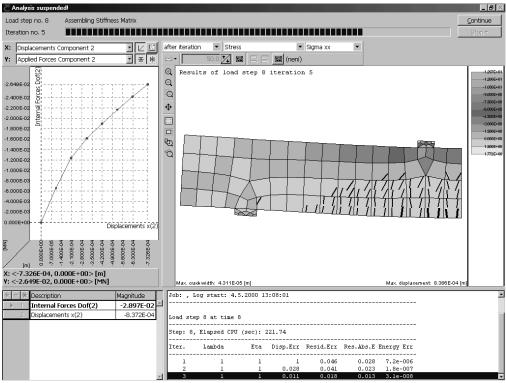


Figure 61: The interactive window for monitoring the progress of non-linear analysis.

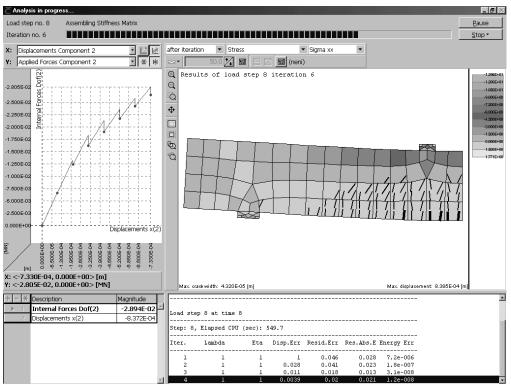


Figure 62: The interactive window after selecting a different format of the L-D diagram. The L-D diagram in this figure shows the iterative changes of the monitored quantities.

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 **b**. 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.

Add analysis steps	×
Analysis step Load cases: 1,2	Step multiplier Multiplier: 3 [-]
Solution Parameters Solution Parameters	☑ Save load step results
	→ <u>A</u> dd X Cancel

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

+ - *	Load step list	Coefficient	Prameters	Save	Calculated	
Number		[-]	analysis	results	results	, <u>■</u> , <u>A</u> dd
35 1,2		3.000	Solution Parameters	Yes	No	🛋 🚍 Insert
36 1,2		3.000	Solution Parameters	Yes	No	
37 1,2		3.000	Solution Parameters	Yes	No	i 🚟 <u>E</u> dit
38 1,2		3.000	Solution Parameters	Yes	No	
39 1,2		3.000	Solution Parameters	Yes	No	<u> </u>
▶ 40 1,2	2	3.000	Solution Parameters	Yes	No	Items: 40

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 postprocessing 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.

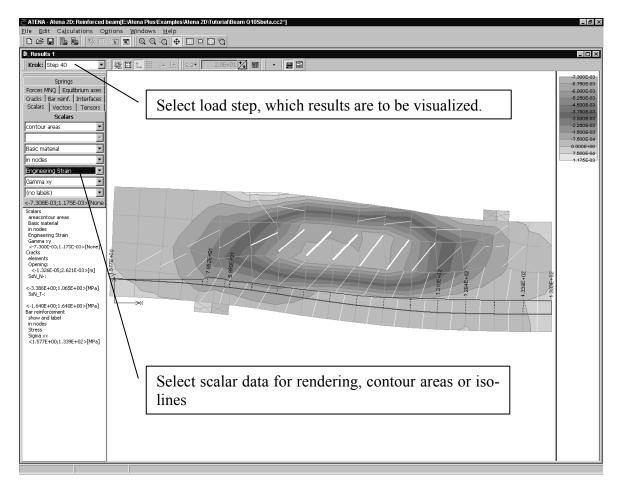


Figure 65: The post-processing window containing contour areas, cracks and reinforcement stresses for the last load step 40.

Springs	Springs
Forces MNQ Equilibrium axes	Forces MNQ Equilibrium axes
Scalars Vectors Tensors	Scalars Vectors Tensors
Cracks Bar reinf. Interfaces	Cracks Bar reinf. Interfaces
Bar reinforcement	Cracks
show and label	elements
in nodes 💌	Filter:
Stress 🔽	Label crack width
Sigma xx 👻	<-1.326E-05;2.621E-03>[m]
	Label Sigma N
	<-3.386E+00;1.065E+00>[MP; Label SigmaT
	<-1.640E+00;1.640E+00>[MP(
<1.577E+00;1.339E+02>[MPa]	

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.

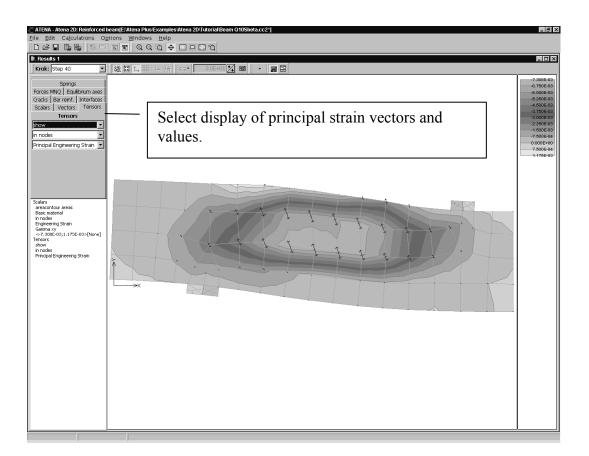


Figure 67: The post-processing window with rendering and principal vectors of maximal principal strains for the last load step 40.

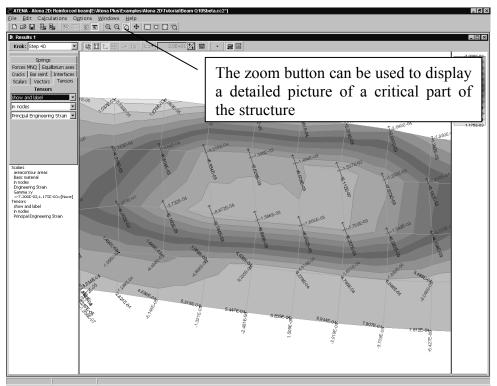


Figure 68: The post-processing window with zoom, rendering and principal vectors of maximal principal strains for the last load step 40. If a user selects the item show with label, the numerical values of the principal strains will be displayed as well.

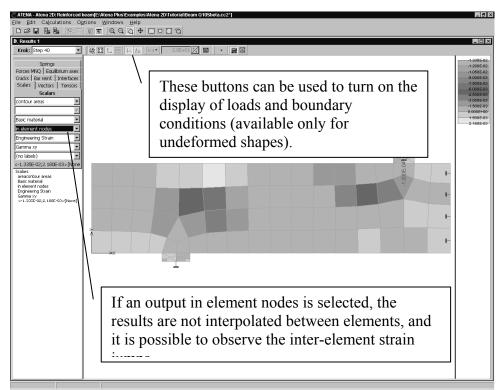


Figure 69: The post-processing window with rendering of element data for the last load step 40. This figure shows also the undeformed configuration with support and loading conditions. The undeformed shape is

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.

Settings	×					
General Font Pre-processing Crack display						
Clipboard Print and clipboard Color copy						
Run Enable input file editting before starting FE analysis						
Verification Verification Verification Verification						
	Cancel					

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.

Settings X							
General Font Pre-processing Crack display							
Font size in structure dis	olay Screen	Printer	Font size for text printout Font size [points]				
Structural elements:	4.000	3.000	Normal text: 9				
Finite element mesh:	3.500	3.000	Results: 8				
Numerical values:	3.500	3.000	Smaller title: 10				
Output data scale:	3.500	3.000	Large title: 12				
Analysis progress:	3.500						
			OK Cancel				

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

Settings	×
General Font Pre-processing Crack display	
Line width for the largest crack width	
Thickness: 1.000 [mm]	

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.

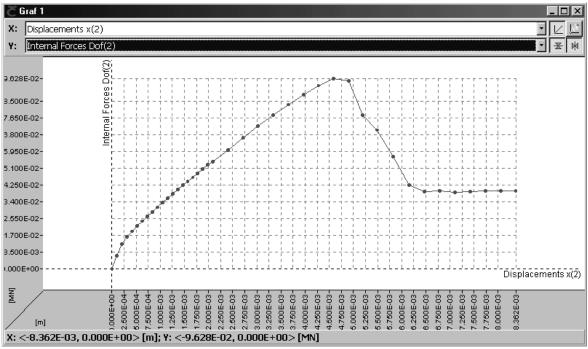


Figure 73: The load-displacement diagram.

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

🖬 Atena - output document		×
Generate 🔄 🔚 🎒 🗖 🖏	View only The Courier New P 12 🕃 B I I I 🖺 E	
☐ Input data	Results Results from load step 40	*
Vine Macro-elements Smeared reinforcement Openings Re reinforcement	Job: ATENA, Openning output file : 4.5.2000 15:28:37	
Garrennotennate Vicad cases Analysis steps Monitoring points Solution Parameters	Job: AIENA, Openning output file: 4.5.2000 15:28:37	
Results Monitoring in iterations Monitoring in load steps VLoad step 1	Output data for request: ENGINEERING_STRAIN Description: Engineering Strain Step: 1 Iteration: 2 at Time: 1 	
Nodes Reference Nodal Coordinates Current Nodal Coordinates WEngineering Strain Principal Engineering Strain	Node eps_xx eps_yy gama_xy Units None None None 1 -8.11e-007 1.02a-008 -1.86e-006 2 -1.75e-006 3.62e-007 -1.24e-006 3 4.65e-008 8.49e-008 2.19e-007 4 -2.39e-007 -2.41e-007 -9.04e-007 5 9.39e-007 -1.81e-007 -2.88e-006	
- Stress - Principal Stress - Sbeta State Variables - Performance Index - Plastic Strain	6 1.86e-008 -2.30e-006 -2.84e-006 7 -1.63e-006 -3.21e-006 5.00e-008 8 8.41e-007 -2.04e-006 -2.11e-008 9 -1.71e-007 -2.34e-006 2.97e-006 10 1.42e-006 -1.90e-006 -4.59e-007 11 4.20e-007 -1.52e-006 2.71e-006	
Principal Plastic Strain Displacements Displacements External Forces Reactions	12 -9.23e-007 -1.32e-006 2.71e-006 12 -9.23e-007 .1.16e-007 1.87e-006 13 -3.91e-007 -1.36e-007 1.09e-006 14 1.65e-007 1.03e-007 -4.26e-007 15 -1.57e-006 3.55e-007 1.33e-006 16 -2.92e-006 3.32e-007 -5.03e-006	
Residual Forces Nodal Degrees Of Freedom Elements Element Incidences	17 -6.01e-007 -1.31e-007 -2.75e-006 18 1.45e-006 4.80e-007 $3.05e-006$ 19 -1.65e-006 $-8.44e-007 - 6.04e-006$ 20 -7.43e-006 $2.69e-007 -8.02e-006$ 21 -5.40e-006 -1.8e-006 -6.20e-006	
Crack Attributes Crack Attributes Element Int.Pts row: 1 column: 1	22 -1.72e-006 -3.68e-006 1.60e-009 23 -7.99e-006 2.47e-007 -6.46e-008	T

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:

Step inform Analysis ste Load skep 40	ep						L Close
nput Out	tput Message	Error					
	Log start: 4. ep 40 at time	40					
Step: 40	O, Elapsed CP						
Iter.	Lambda	Eta	Disp.Err	Resid.Err	Res.Abs.E	Energy Err	
1	1	1	1	0.087	0.075	7.5e-005	
2	1	1	0.015	0.06	0.056	7.8e-007	
3	1	1	0.013		0.048		
4	1	1	0.011	0.05	0.048	4.5e-007	
	1	1	0.01	0.044	0.047		
5		1	0.012	0.048	0.048	5e-007	
6	1	-					
6 7	1	1	0.01	0.043	0.045	3.7e-007	
6 7 8	1 1 1	1	0.01 0.012	0.043 0.047	0.045 0.047	4.7e-007	
6 7 8 9	1 1 1	1	0.01 0.012 0.01	0.043 0.047 0.042	0.045 0.047 0.044	4.7e-007 3.7e-007	
6 7 8	1 1 1 1	1	0.01 0.012	0.043 0.047 0.042 0.046	0.045 0.047 0.044 0.047	4.7e-007	-

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 \clubsuit .

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.

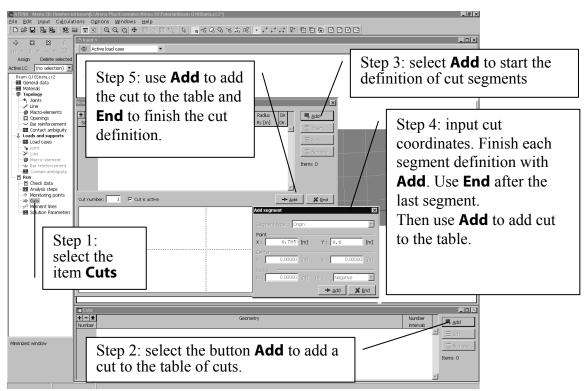


Figure 76: The beginning of the numerical definition of cuts.

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

	Cu	t 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	

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

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

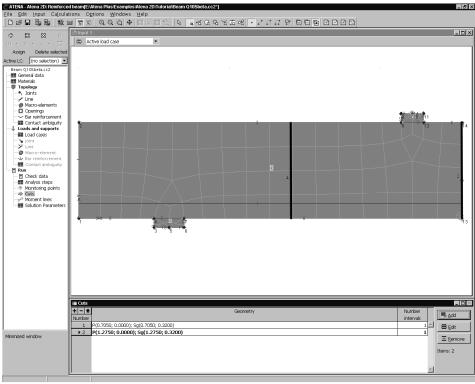


Figure 77: The program window shows two cuts after the cut definition.

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).

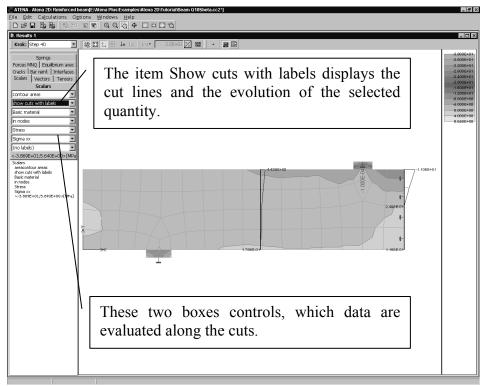


Figure 78: The post-processing window with the display of cut lines.

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.

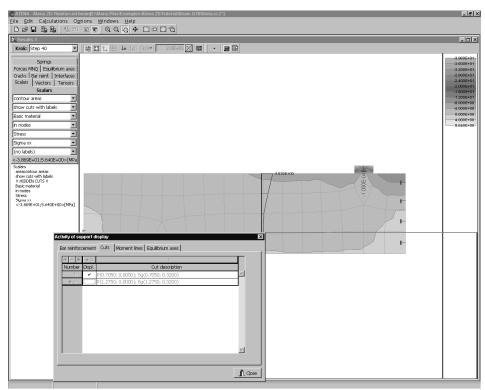


Figure 79: The visibility of cuts, moment lines or reinforcement bars can be specified from the main menu item Options | Activity.

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 5.

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 \clubsuit .

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.

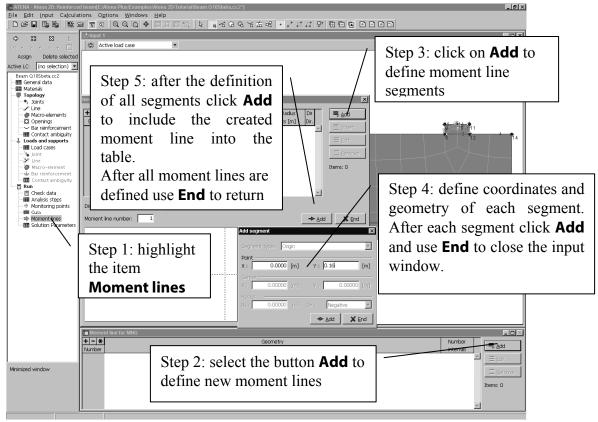


Figure 80: The procedure for the definition of moment lines for the calculation of internal forces.

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).

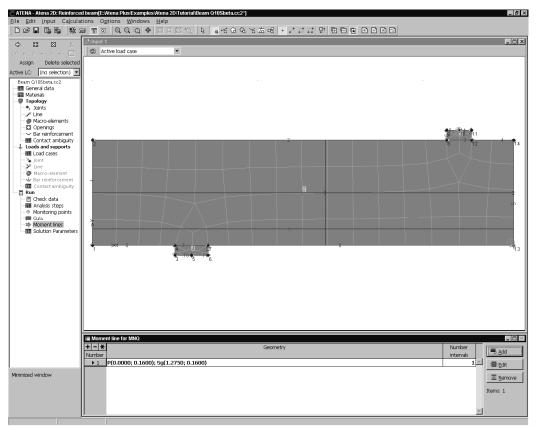


Figure 81: The pre-processing window after the specification of one moment line: coordinates (0;0.16) -> (1.275;0.16).

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 is 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.

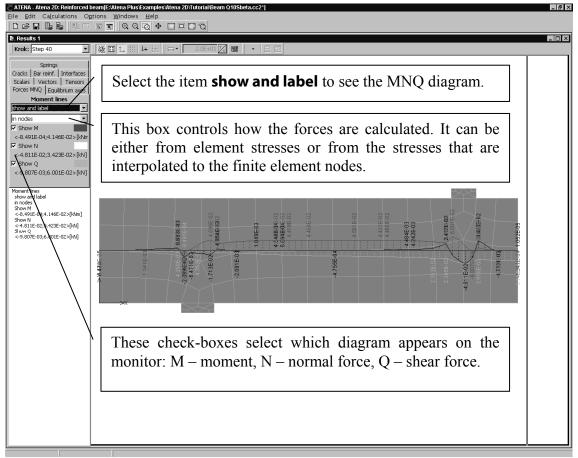


Figure 82: The diagram of internal forces in the post-processing window.

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.