ATENA Program Documentation
Part 12

User's Manual for ATENA Studio

Version 5.9.0

Written by:

Štěpán Beneš, Jitka Mikolášková, Tomáš Altman

Prague, September 02, 2021

CONTENTS

1 INTRODUCTION

This document describes the usage of the program **ATENA Studio** (ATENA Studio Graphical User Interface), which is a part of the ATENA program system. This document is compatible with the ATENA version 5.9 released in May 2021.

The ATENA program, which is determined for nonlinear finite element analysis of structures, offers tools specially designed for computer simulation of concrete and reinforced concrete structural behavior.

ATENA program system consists of a solution core and several user interfaces. The solution core offers capabilities for variety of structural analysis tasks, such as: stress and failure analysis, transport of heat and humidity, time dependent problems (creep, dynamics), and their interactions. Solution core offers a wide range of 2D and 3D continuum models, libraries of finite elements, material models and solution methods. User interfaces are specialized on certain functions and thus one user interface need not necessarily provide access to all features of ATENA solution core. This limitation is made on order to maintain a transparent and user friendly environment in all specific applications of ATENA.

ATENA Studio is simple and intuitive user interface for ATENA solution core. It is unified environment for all analysis types: 2D, 3D, statics, dynamic, creep, heat & moisture transport. The program has two main modules:

1. Runtime module. It makes possible a real time monitoring of results during calculations.

2. Post-processor module. Access to a wide range of graphical and numerical results.

This manual is devoted mainly for the description of all program functions. The questions of the ATENA solution core and the theoretical background are mentioned only marginally and are included in the other volumes of the ATENA documentation: ATENA Part 1 - Theory, and Part 6 - ATENA Input file. Although some examples are included in this manual to support description of some functions, a systematic treatment of examples is not covered. It is also a subject of a separate document ATENA Part 5 – Examples.

2 ATENA STUDIO IN GENERAL

2.1 ATENA Studio Window

ATENA Studio user interface consists of two main modules, through which a user can control two phases of execution: Runtime (calculation) and Post-processor. For pre-processing some other ATENA interface program must be used such as for instance **GiD**. **ATENA Studio** has a modular structure. Calculation or post-processing phases are processed by these program modules separately by opening a corresponding window. This can be done after opening a project from input or result file by clicking on the program button located on the top-right of the main menu bar.
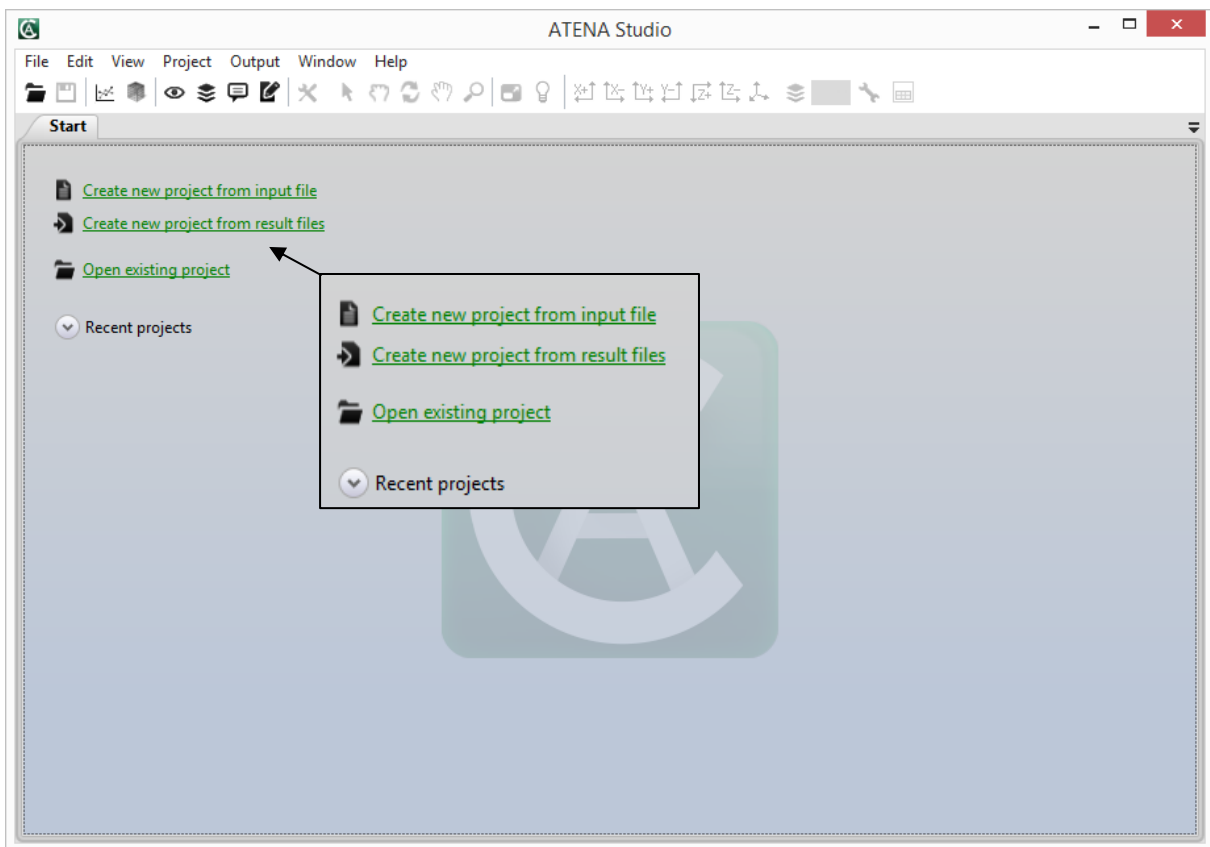
Fig. 2-1 Start window.

The content of the window slightly changes by running the program module. See chapters 3 and 4 for the descriptions of the Runtime and Post-processor windows.

By switching the programs, ATENA replaces one program module by another and all data related to the specific task are also loaded, if available. For example, if the program is in a runtime mode and switches to post-processor, the data with results are made available. This may take some time in proportion to the task size.

2.2 Geometrical and Numerical Models

ATENA recognizes two models, geometrical and numerical. Data of these models are treated strictly separately. **Geometrical model** represents dimensions, properties and loading. It

consists of an assembly of macro-elements (solids). Macro-elements maybe connected by contacts. Each macro-element is an independent object defined by joints lines and surfaces.

Reinforcement can be modeled in two ways. First way is a discrete bar, which is defined as a geometrical multi-linear object. It is embedded in solid objects. Its geometry is defined independently of macro-elements. Thus one reinforcing bar can be embedded in several number of macro-elements. Second way is a smeared reinforcement, defined as a composite material, which has no effect on geometrical model.

Geometrical model is supplemented by the definition of loading (load cases and load history) and construction cases. **ATENA Studio** in its current version does not support the definition of the geometrical model, it operates strictly only on the numerical model, which is created from the geometric model using a suitable pre-processor.

Numerical model is generated based on a geometrical model and represents a numerical approximation of the structural analysis problem. Numerical model is a result of discretisation made by the finite element method. The mesh generator in **ATENA** makes possible to generate automatically meshes for solid and reinforcing objects.

2.3 Starting ATENA Studio

ATENA can be started at any stage of the analysis progress, depending on data available. The data of a new task must be imported from input file. The current project at any stage of processing can be saved and re-started later. The details of the access to various functions shall be described in the context of each module.

Typically, the program is automatically started from the ATENA-GiD interface or can be run manually from the system menu or command line. If it started manually, there is a list of command line parameters, which can be additionally used:

/silent - do not display any message boxes while running execution

/batch_execute – forces to automatically execute given problem without user intervention. After execution data is saved and program is terminated.

/execute – forces to automatically execute given problem without user intervention. After execution program remains running.

/turbo – run execution in turbo mode

/m – set ATENA module (CCStructures, CCStructuresDynamic, CcStructuresTransport, CcStructuresCreep)

/o – overwrite protocol files (.out, .msg, .err)

/catch_fp_instructs – catch floating point exceptions

/inp – create new project from input file

/d – default directory path

/threads – number of threads to be used for execution

- threads > 0 … number of threads to be used for execution (limited by current processor)
- threads == 0 … sequential execution (one thread)
- threads < 0 … number of threads to be left out of execution

2.4 Terminology

Material – Constitutive model of material used in ATENA to represent a certain material type.

Geometrical model – Solid structure described as an assembly of basic geometrical objects (macro-elements, their contacts, reinforcing, springs, etc.)

Macro-element – basic geometrical 3D object defined by surfaces.

Surface – planar face of object circumvented by lines.

Line – line connecting two joints.

Joint – basic geometrical entity.

Reinforcement bar – multi-linear object embedded in macro-elements.

Contact – contact plane of two macro-elements.

External cables – reinforcement connected to structure in limited number of joints only and otherwise not embedded in solid macro-elements.

Spring – object representing a spring support.

Construction case – assembly of geometrical objects (macro-elements, contacts, reinforcement bars, etc.) representing certain construction phase.

Load case - load action defined on assembly of macro-elements.

Load step – increment of load action.

Analysis steps – loading history defined as a sequence of load increments.

Finite element – basic element used for approximation of deformation and stress state of a solid.

Node – point in finite element mesh.

DoF – degree of freedom used as independent variable in finite element analysis.

3 RUNTIME

3.1 Introduction

To start analysis an input file has to be imported. The input file can be produced by ATENA interface in other programs, such as for instance **GiD**. The input is shown in window and can be edited. After executing command Runtime (located top right on the window toolbar), Runtime module interface appears as shown in Fig. 3-1 and Fig. 3-2.
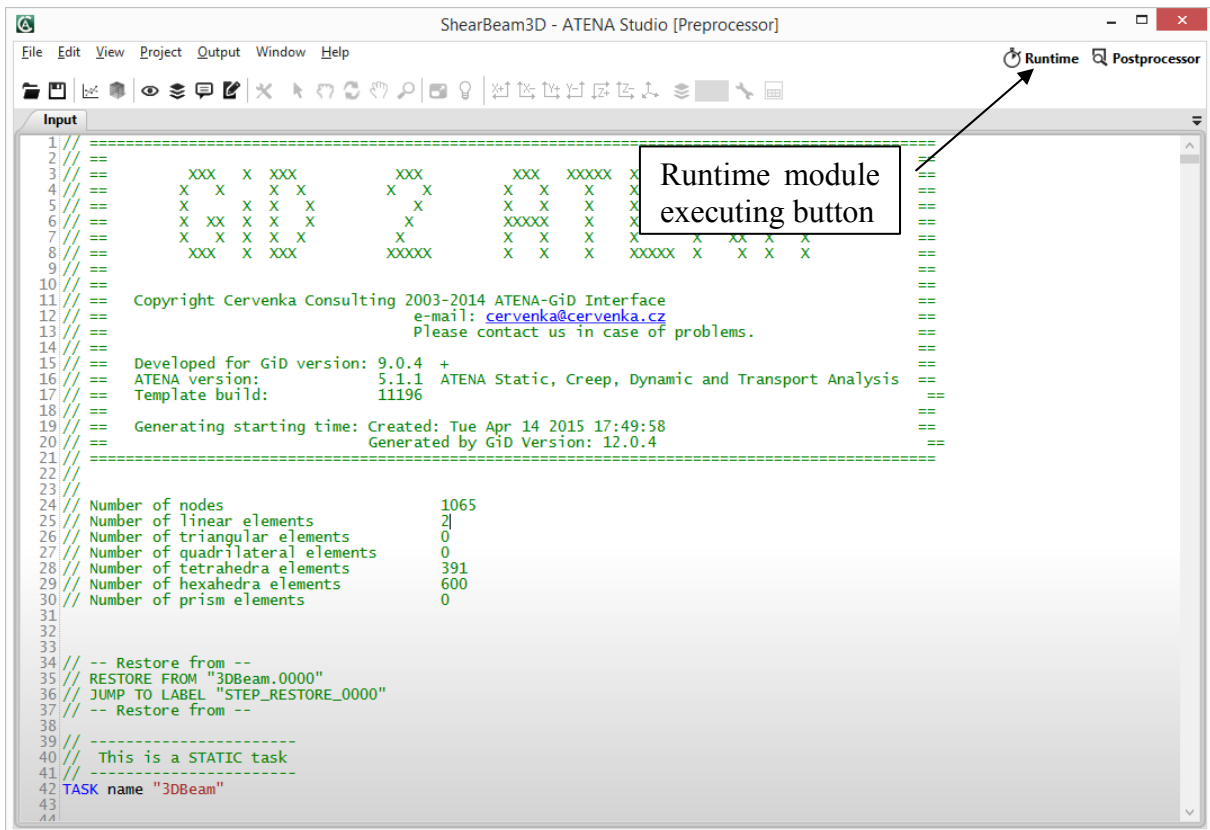
Fig. 3-1 Imported input file
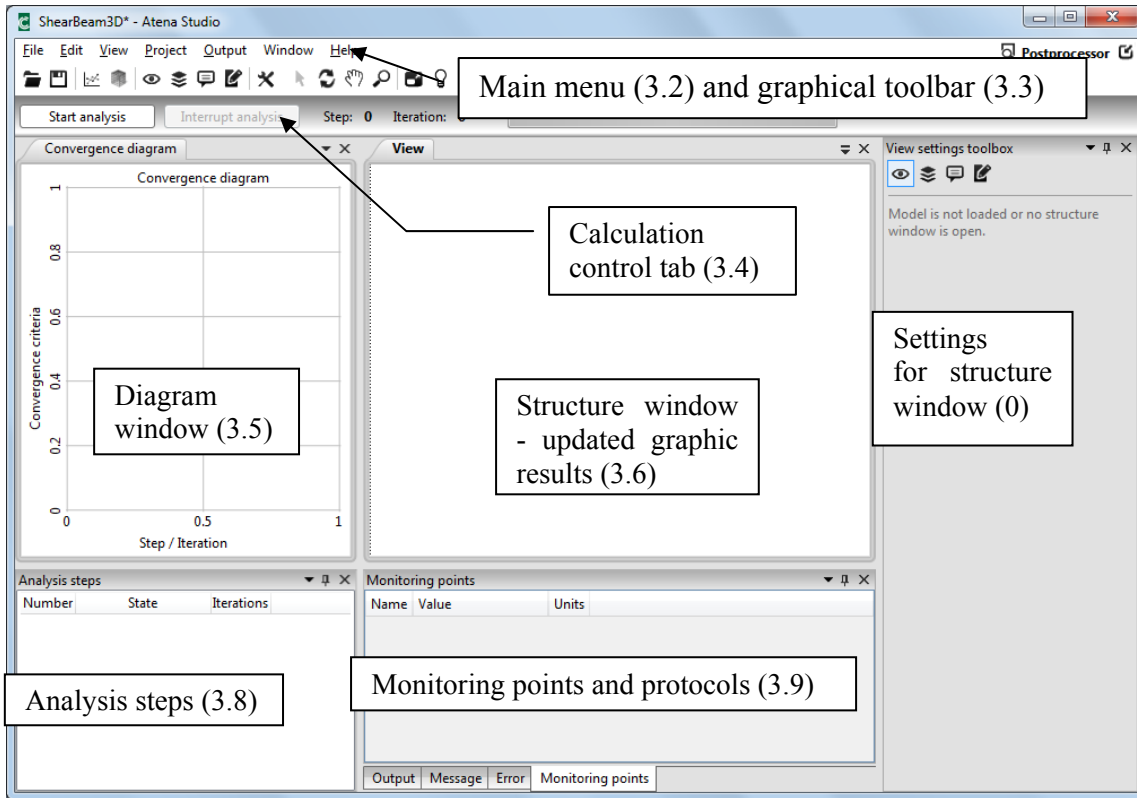
Fig. 3-2 Runtime module

The main function of Runtime module is to control calculation progress (start, stop or suspend analysis) and to monitor intermediate results. To that are used windows which are indicated in Fig. 3-2 and described in following sections.

3.2 Main Menu

The main menu contains all important functions to control the program. The items are same as in the Post-processor module except menu **Output** and some other. There is basic description of them in following sections. The links to more detailed description can be found in boxes.

3.2.1 File
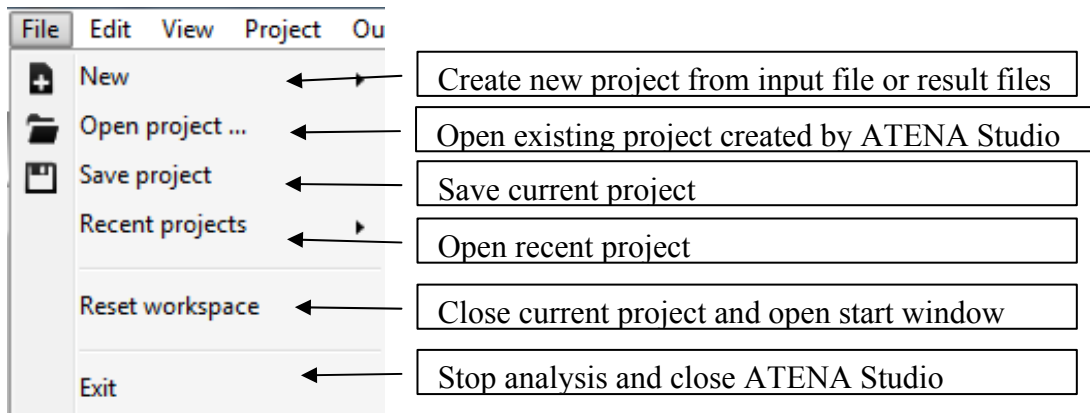
Fig. 3-3 Items in menu "File"

3.2.2 Edit
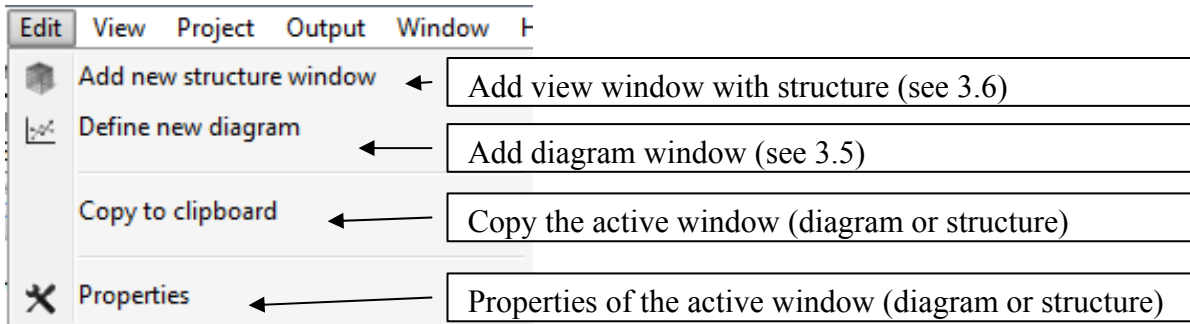
Fig. 3-4 Items in menu "Edit"

Copy to clipboard - supports multiple data types (it can be used to insert values to spreadsheet or picture to image editor), copies only active window - diagram or structure

Properties - open: diagram properties (horizontal and vertical axis, font size, ...)

                       structure properties (light, axis, font size, transparency, color scale)

Detailed description of diagram properties is in 3.5 and structure properties in 3.6
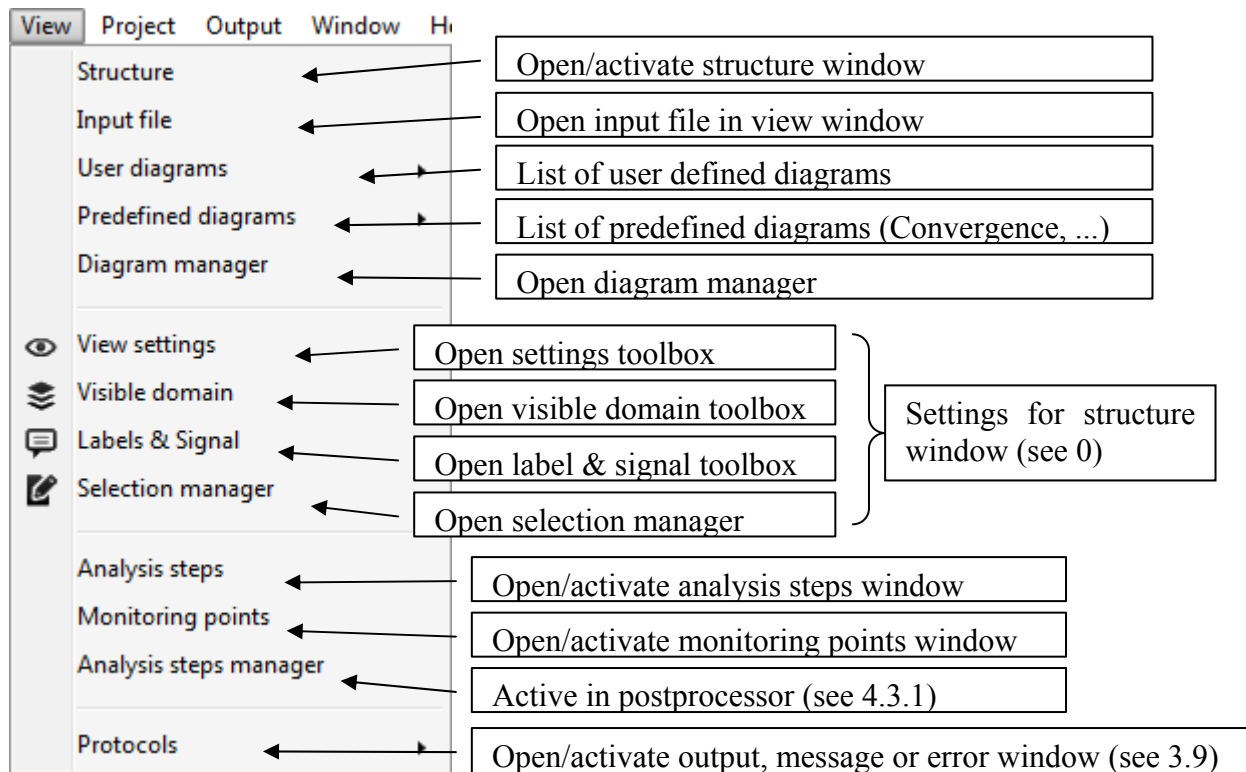
Fig. 3-5 Items in menu "View"

Diagram manager - there you can manage all created diagrams (remove, edit, show)

Analysis steps - number and state of analyzed steps and numbers of iterations (3.8)

Monitoring points - values of all monitoring points (predefined and user defined) updated after each iteration (3.9)

3.2.3 Project
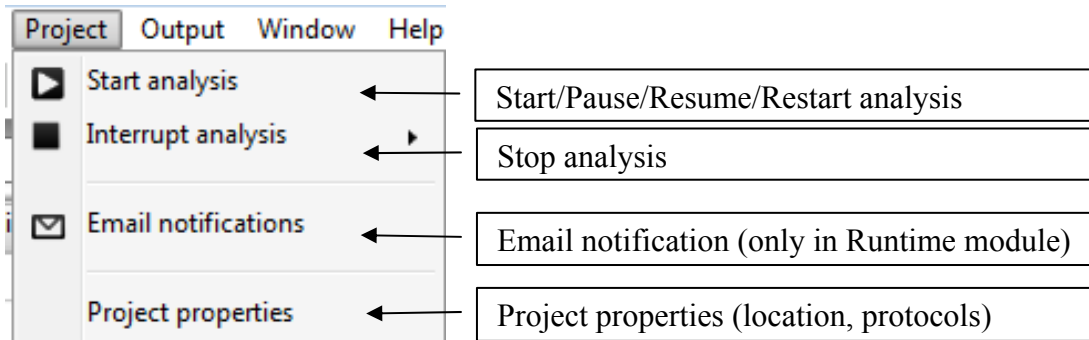
Fig. 3-6 Items in menu "Project"

Start analysis - After starting the analysis, this button changes to **Pause analysis**, which temporarily stops calculation, and then it is possible to resume the analysis with the same button. If the analysis is interrupted, the button changes to **Restart analysis**.

Interrupt analysis - Analysis can be stopped by three ways: after step, iteration or as soon as possible; for details see 3.4

Email notification - if enabled and if email address is properly filled in, **ATENA Studio** will send a report about interrupted or finished analysis by email.

Project properties - here the project type can be selected (statics, dynamics, transport or creep) and path to input file (.inp) or to protocol files (.out, .msg, .err)
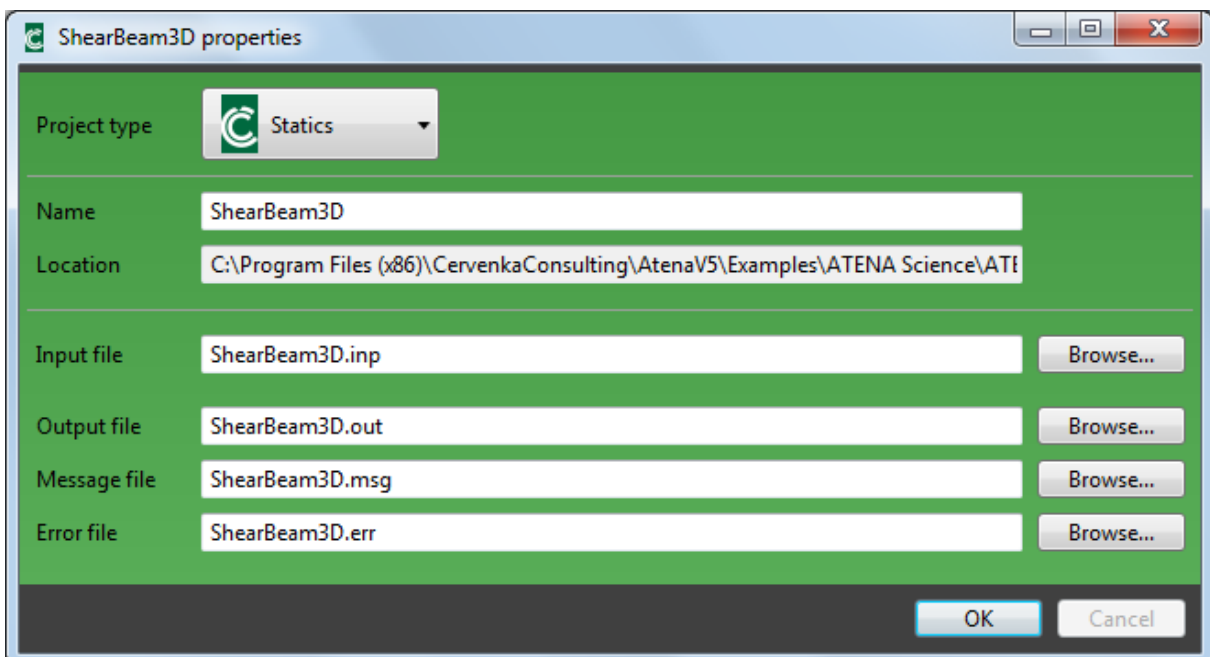
Fig. 3-7 Project properties

3.2.4 Output

Output menu is active only during post-processing (see 4.3.2).

3.2.5 Window
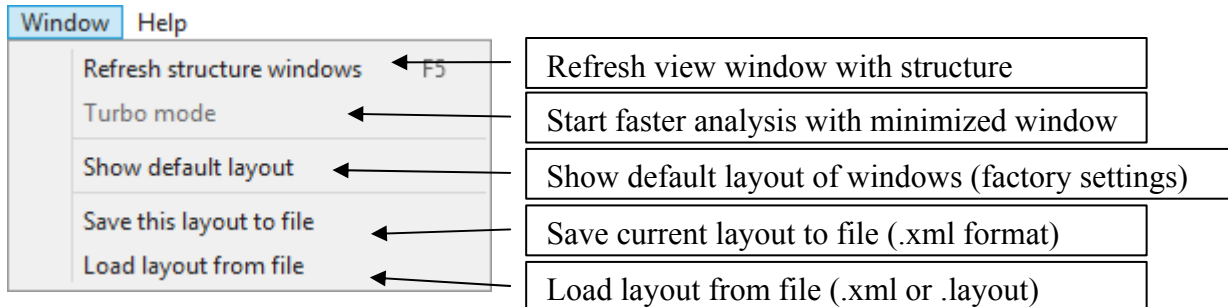
Fig. 3-8 Items in menu "Window"

Turbo mode - It can be enabled during the calculation. All the graphical windows will be closed, and the analysis runs faster. The runtime window is collapsed to minimal size so monitoring of calculated values is not possible
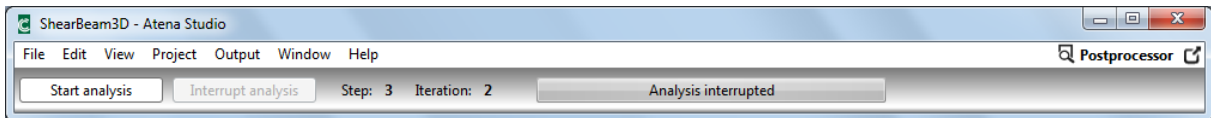
Fig. 3-9 Turbo mode window

3.2.6 Help
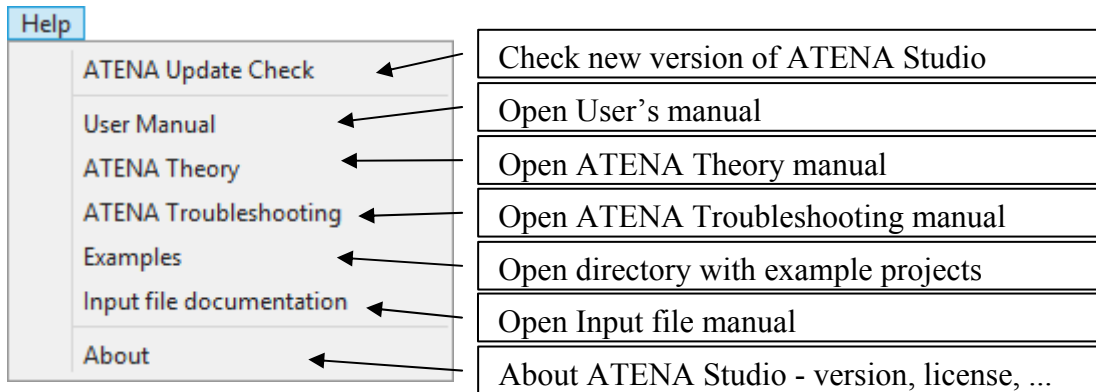
Fig. 3-10 Items in menu "Help"

3.3 Graphical Toolbar

Toolbar icons are shortcuts to the most used functions in **ATENA Studio** which can be also found in the main menu (see 3.2).

File toolbar

Fig. 3-11 Items in menu "File toolbar"

Edit toolbar

| Add new diagram window | → | Add new structure window |

Fig. 3-12 Items in menu "Edit toolbar"

View toolbar

| Open setting toolbox | → | Open selection manager (see 3.7.4) |

| Open visible domain toolbox | | Labels & signals toolbox |
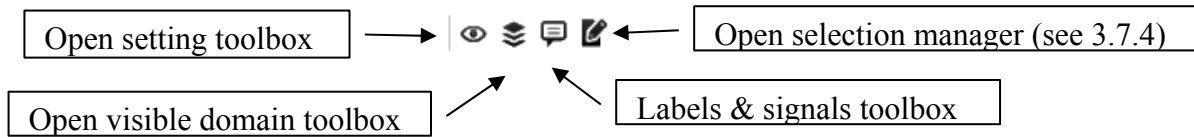
Fig. 3-13 Items in menu "View toolbar"

Structure window toolbar

Properties of active window	Set light position
Select - active when defining new selection with selection manager	Zoom to fit view window
Drag – active when showing shear forces on cut plane	Zoom window
	Move
Rotate	
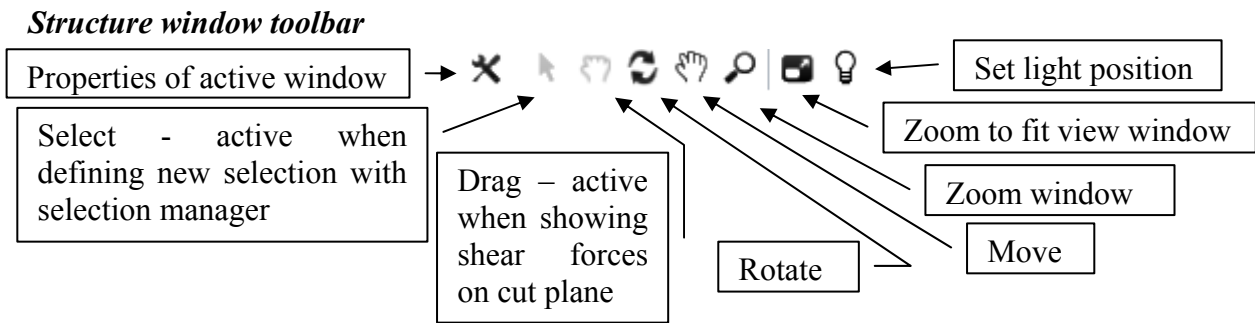
Fig. 3-14 Items in menu "Structure window toolbar"

View direction toolbar

View in direction X+	Diagonal view (iso projection)
View in direction X-	View in direction Z-
View in direction Y+	View in direction Z+
	View in direction Y-
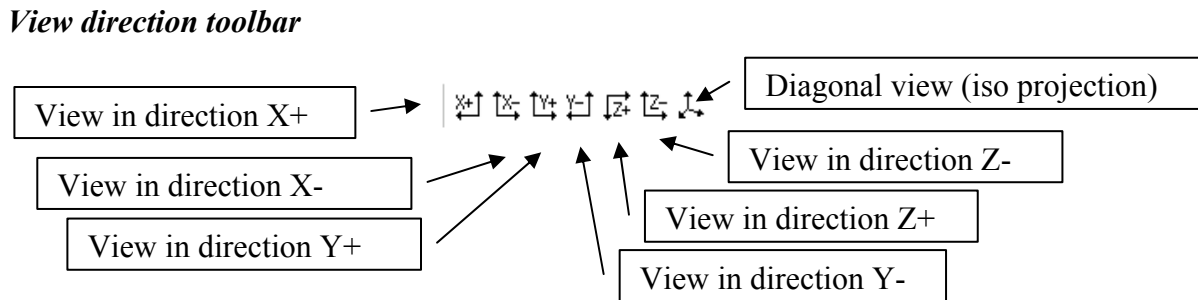
Fig. 3-15 Items in menu "Structure window toolbar"

3.4 Calculation Control

Analysis starts by clicking the button [Start analysis] located on the left side of the calculation control tab. After starting the calculation, this button changes to [Pause analysis] and calculation can be temporarily stopped by pressing it. The same button changes again to [Resume analysis] and the calculation can be continued again. To stop analysis click the button [Interrupt analysis]. It is possible to choose tree ways of interruption: after load step, iteration and as soon as possible, as shown in Fig. 3-16.
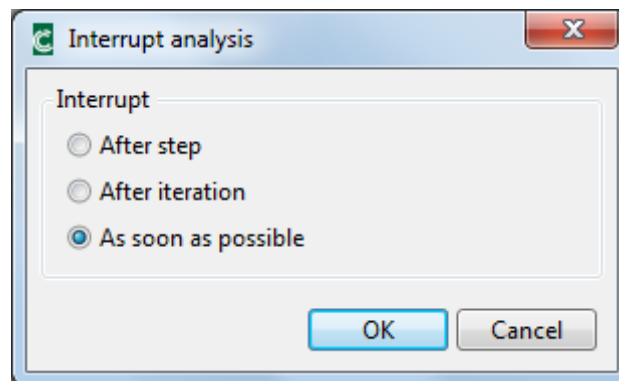
Fig. 3-16 Interruption of analysis.

After interruption it is possible to proceed with following actions:

- Inspect results in Runtime module and restart or continue analysis.

- Go to post-processor and inspect results. Eventually, return to Runtime and restart or continue analysis.

- Save analysis and restart or continue any time in the future.

If you want to restart analysis from some specific step it is necessary to edit input file for continuing analysis. This can be done quite easily if the input file was created by **GiD**. View input file, find two lines at the beginning of the file (highlighted in the following picture), delete slash marks and type number of last saved step. Save and continue analysis using button 　Restart analysis 　. The calculation will restart from the provided step number.

Fig. 3-17 Edited input file, analysis starts from step 26.

Next to the button 　Interrupt analysis 　 there is a line with information about current processing: step number, iteration number and a calculation phase under processing. The progress bar indicates the speed of calculation. One run of progress bar corresponds to one loop through the finite element set, or through the equation solver.

3.5 Diagram Window

Graphic record of monitoring is shown in the diagram window. The default monitors are convergence criteria (i.e. errors in the iterative process), which are of the greatest importance in nonlinear analysis, because they indicate solution accuracy. These are evaluated and shown automatically and need not to be defined by user. If desired, other monitors, forces, displacements, stresses and other results must be defined in the input file by the user.
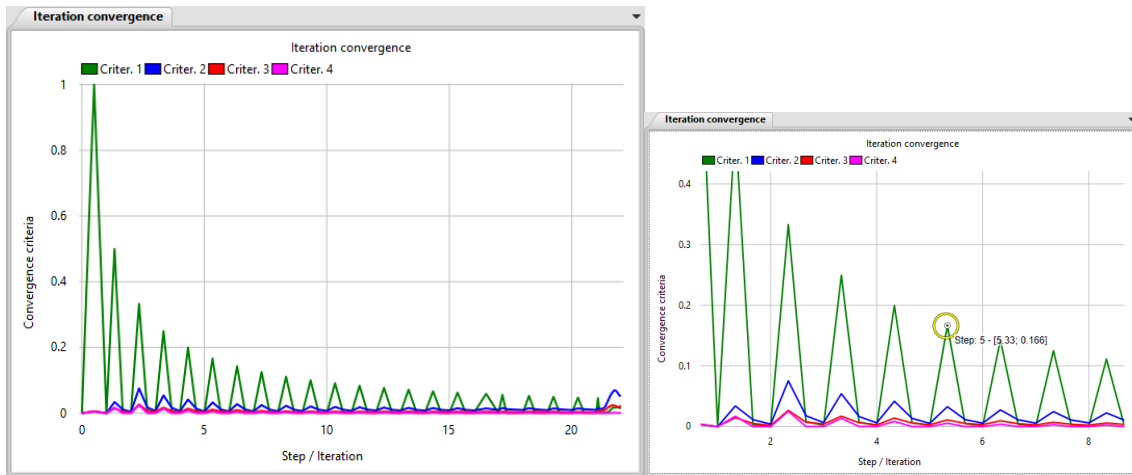
Fig. 3-18 Monitoring of convergence criteria. Full range - left; selected range – right

Monitoring of convergence criteria is shown in Fig. 3-18. Four solution errors serve to check following criteria: (1) displacement increment, (2) normalized residual force, (3) absolute residual force, (4) energy dissipation.

The diagram window can be controlled by mouse. The left button is used for moving, right button for scaling and mouse scroll wheel for zooming. To fit the diagram to window, click the scroll wheel or turn the mouse wheel until it zooms out. Values of monitoring points can be found in diagram by moving mouse cursor. The points are marked with yellow ring and labeled.

The diagram properties can be changed using the icon ![icon], which opens a dialog as shown in the following pictures. They show a default settings of these parameters.
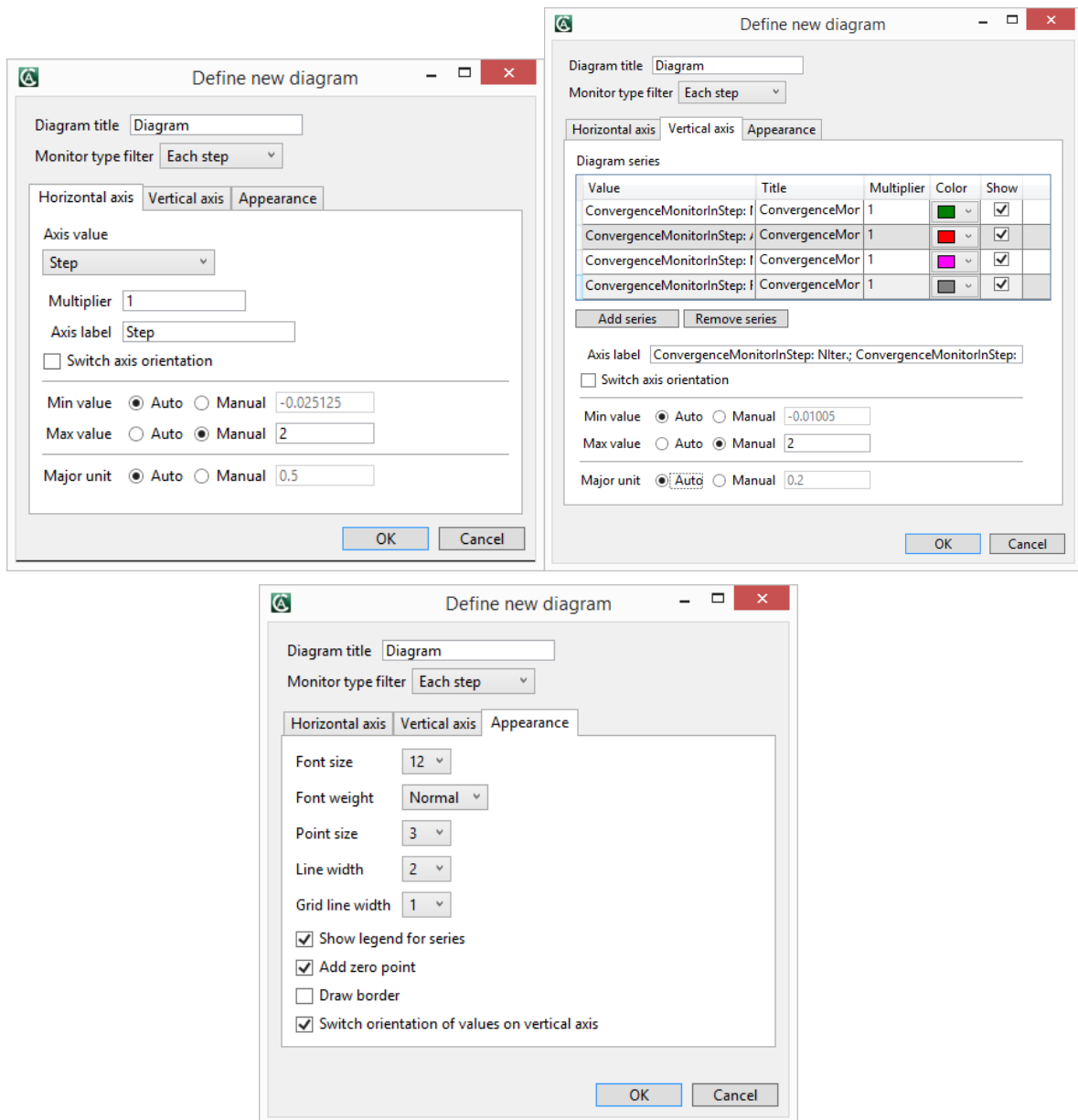
Fig. 3-19 Diagram parameters for convergence criteria.

A load-displacement diagram is a useful tool to illustrate the response of a structure. Monitors must be defined by user in the input file (input file is generated mostly by tool such as **GiD**). Such monitors can be displayed during the calculation. They can be selected in diagram properties of existing diagram but better is to create new diagram. The procedure of graph setting is as follows:

- Open the dialog for new diagram by ⌨.
- Select monitor type filter - each step or each iteration. It depends on how the monitor is defined in input file.
  - Note: Each iteration monitors are now available in each step list too, where are filtered per step. If it is switched monitor filter, monitors with both forms (each step and each iteration) are automatically switched to the other form.

Horizontal axis:

From the list in section **Axis Value**, which unfolds after clicking on ▾, select pre-defined monitor **Displacement**.

Change multiplier if required (for example -1)

Write axis label, for example "Displacement [m]".

Switch axis orientation if needed.

Change min value and max value to limit range of the diagram

Change major units to manual if needed (for example from 3 to 1 to better precision).

Vertical axis:

Click on button **Add series**.

From the list in section **Add new series**, which unfolds after clicking on ▾, select predefined monitor, for example "Reaction". Click OK.

Write axis label, for example "Reaction [kN]".

It can be changed a multiplier, for example -1000 and color of line.

Switch axis orientation if required.

Change min value and max value to focus on particular part of the diagram

Change major units to manual if needed (for example from 0.1 to 0.2 to more transparent diagram).

Note that several values can be shown on Vertical axis (multiple diagrams).
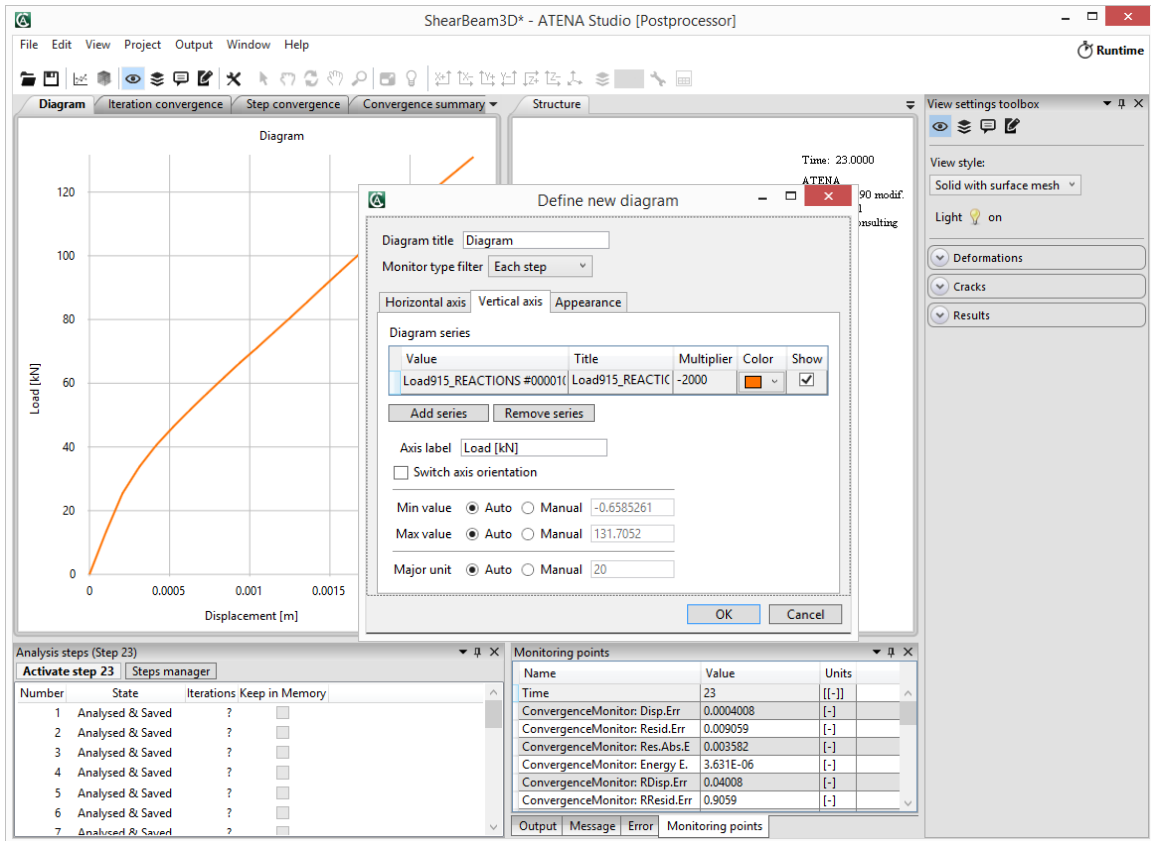
Fig. 3-20 Diagram parameters of load-displacement diagram.
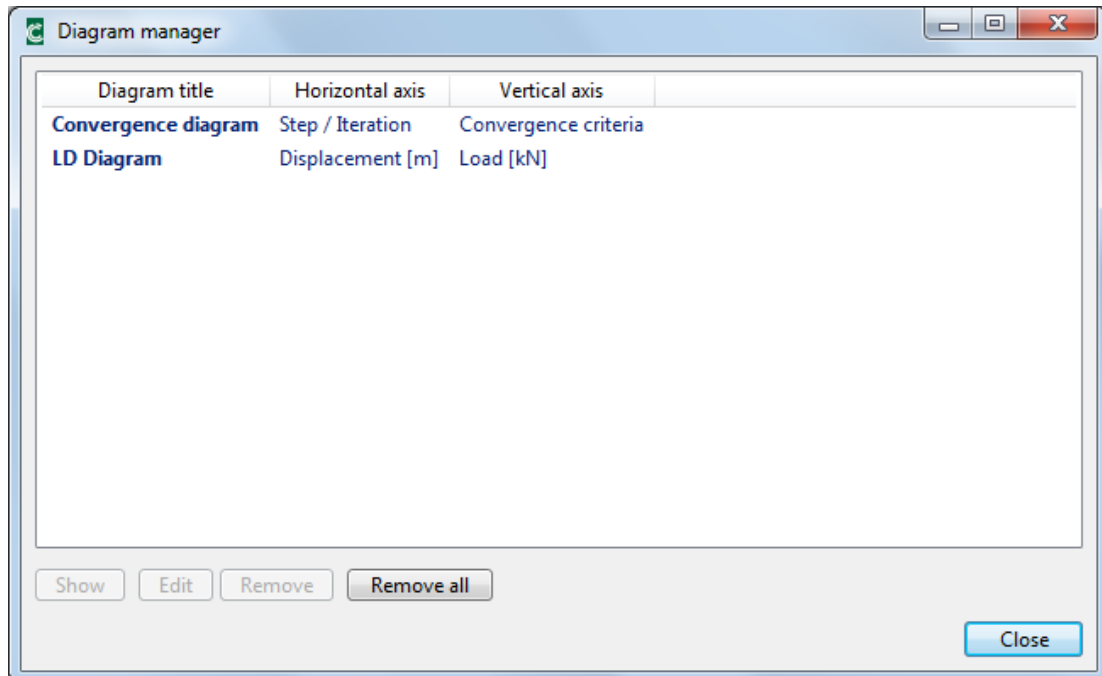
Fig. 3-21 Diagram manager.

It is possible to see both diagrams at one time. Grab bar diagram window and move it. An arrow locator will appear as shown in the following picture. Move the cursor on the selected arrow or square in the middle and place the window. The location can be vertical, horizontal or in the center.
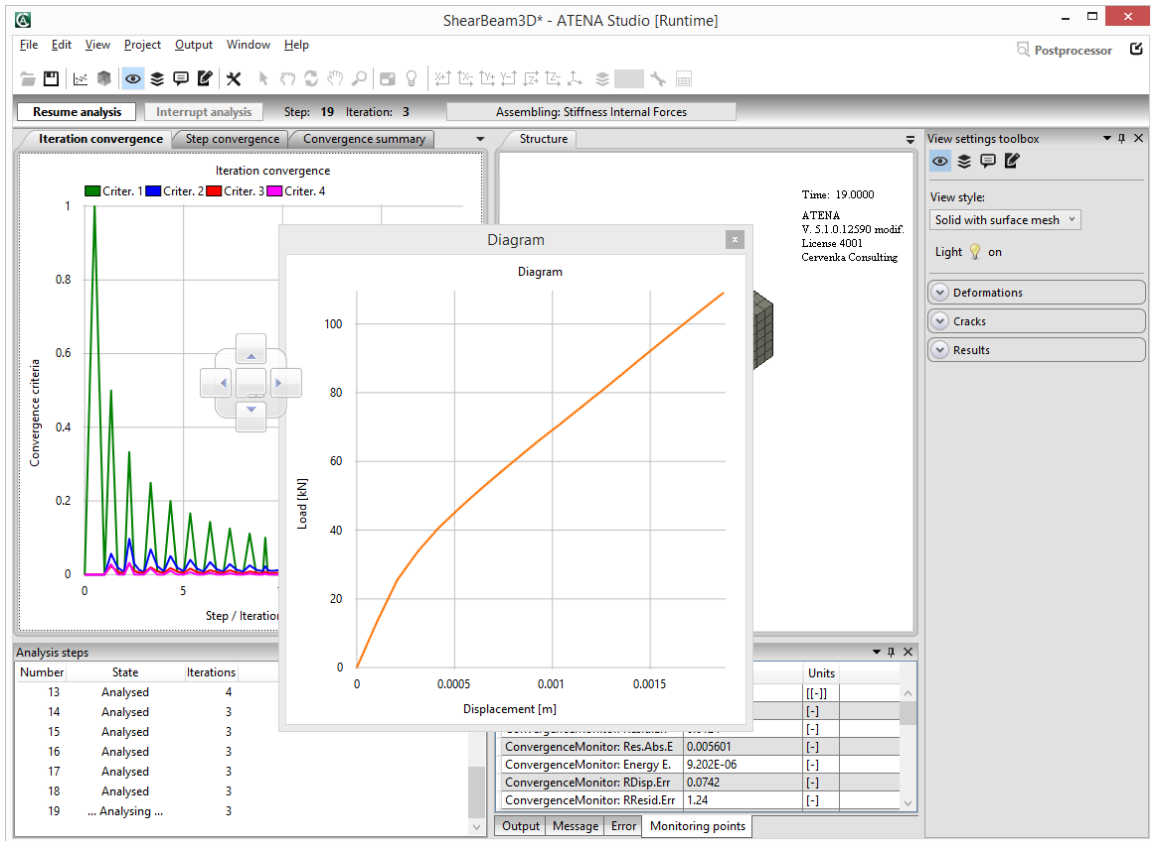
Fig. 3-22 Diagram moving and locator arrows.
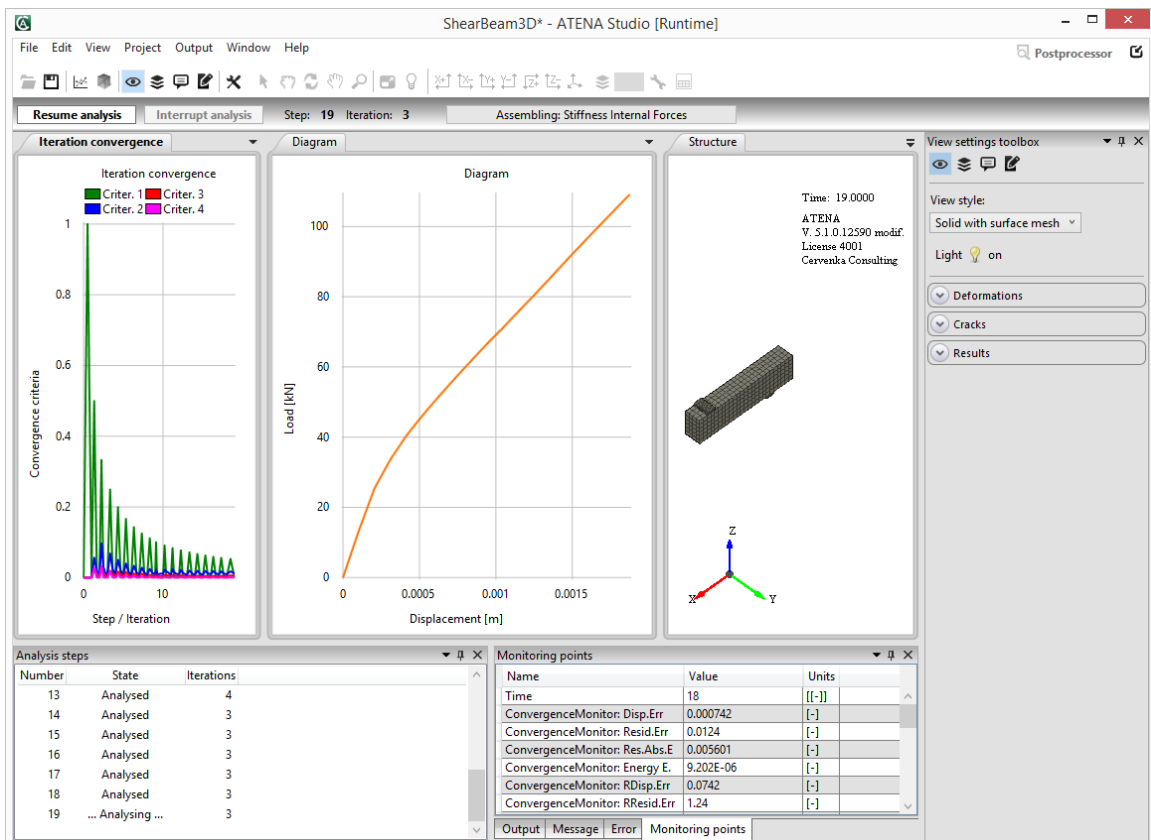
Fig. 3-23 Horizontal location of diagrams.

ATENA Science Studio User´s Manual   19

Monitors are not available for display before analysis starts. Therefore, it is necessary to wait with definition of graph parameters, until at least one cycle of analysis is completed (one iteration of solution).

3.6 Structure Window

The graphical window shows real-time results during calculations. This graphics contains the same possibilities as the final post-processing, but it should be understood that it may take significant resources of your computer, thus slowing down the analysis. It is possible to open several structural windows, with different views at the analyzed structure. Add new structure window by icon ▦ and move it the same way as the diagram window. For details about moving see 3.5. The display parameters are set by using the View settings toolbox on the right hand side of the ATENA Studio window (0). This dialog contains several dialogs with different settings and parameters that control the visualization of results in the Structure window. An example is shown in Fig. 3-25 where the deformations in x3 (i.e. z direction) are shown as contour lines.
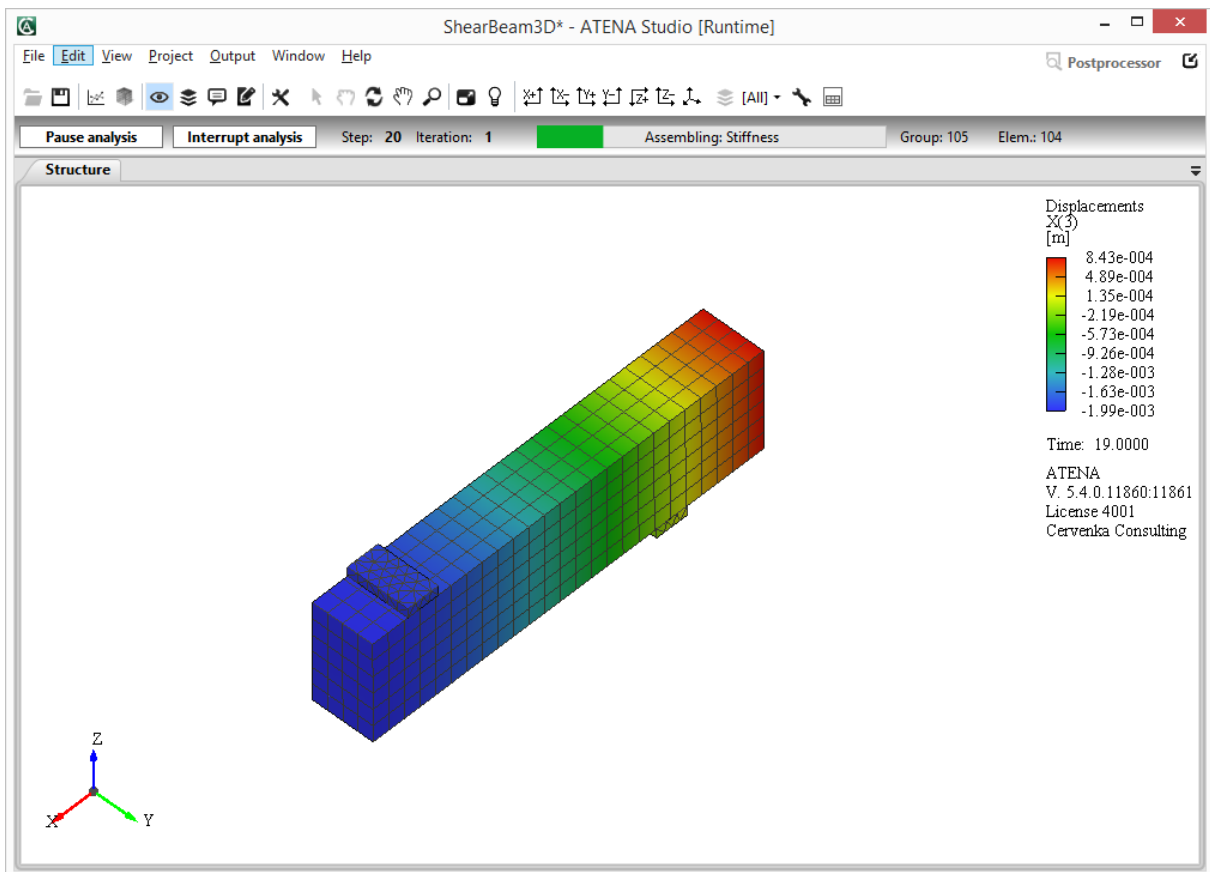
Fig. 3-24 Graphic window showing real time results during analysis.

Structure window can be controlled by holding Shift key and mouse:

shift + left button - rotation,

shift + right button - move,

mouse scroll wheel - zoom,

click scroll wheel – fit structure to whole window.

Structural appearance can be changed by using the icon , which opens the dialog
structure properties (see Fig. 3-25)

3.6.1 General Properties

- general settings of the structure (see Fig. 3-25)

Light on - colors showing result values are different from color scale, if the light is on. The position of light can be moved using tool with icon ▽ .

Quadratics size - size of displayed nodes in the structure (nodes can be shown using **Labels** panel (3.7.3.1))

Enable force to redraw – some actions (display activity, results, cuts...) need to redraw the whole structure, which can be very slowly for large scenes. If it is not neccessary to display changes after each action, uncheck Enable force to redraw, execute a batch of these actions and redraw whole structure only once at the end (using a key F5 or check the Enable force to redraw and confirm).

Element transparency – there are three possibilities to select: opaque, semi-translucent and transparent surfaces, which is a special case where surfaces are culled such that only back surfaces of solid from viewer stay visible, i.e. inner reinforcements are visible too.
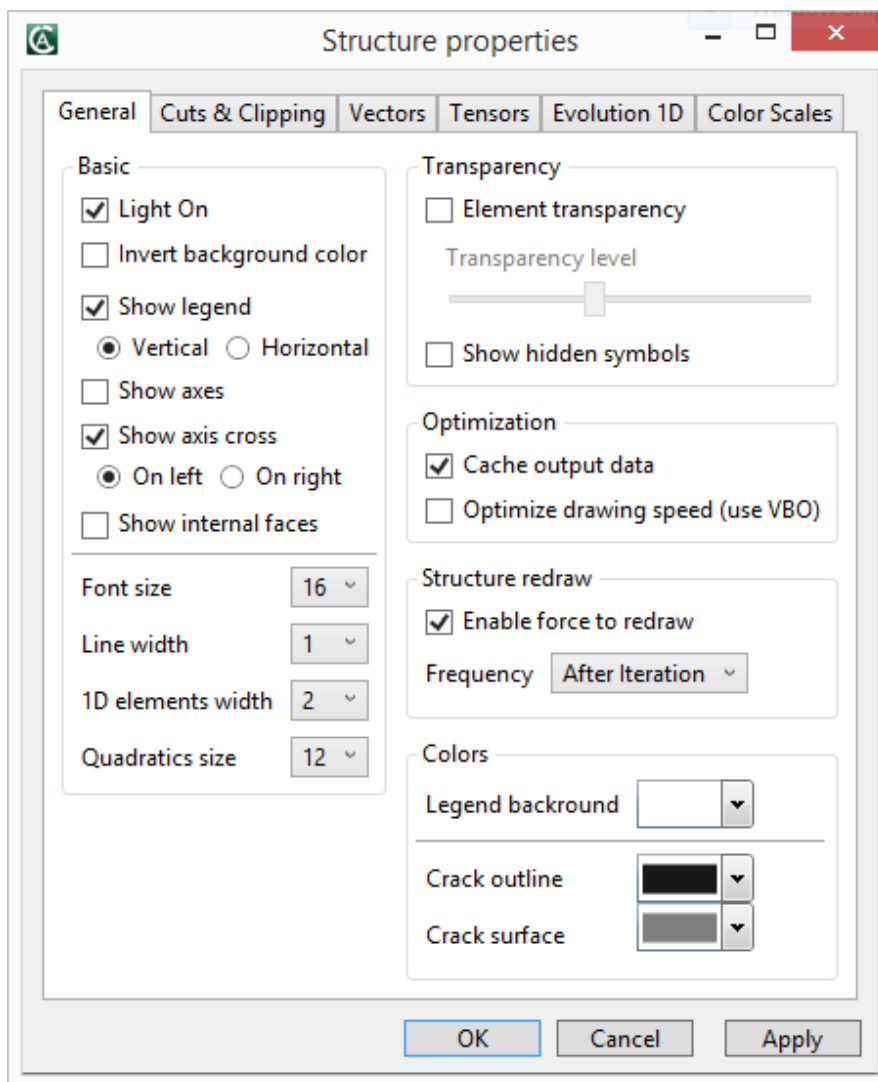
Fig. 3-25 Structure properties. General

3.6.2 Cuts & Clipping Properties

- global settings of cuts and clipping planes (see Fig. 3-26)

- definition of cuts is described in section 3.7.2.1 and definition of clipping planes in section 3.7.2.3

Update automatically position – if it is checked, when a plane coordinate is changed, the plane is automatically redrawn (automatically is pushed Apply button).

Increment delta – plane coordinates can be manually incremented or decremented by clicking on adjacent arrows ⌃⌄ about a some delta. The delta is either relative to a structure or absolute. With checked **Update automatically position**, it allows to go easily through the structure.

View style – cuts are displayed in Solid or Solid with wireframe or Wireframe style
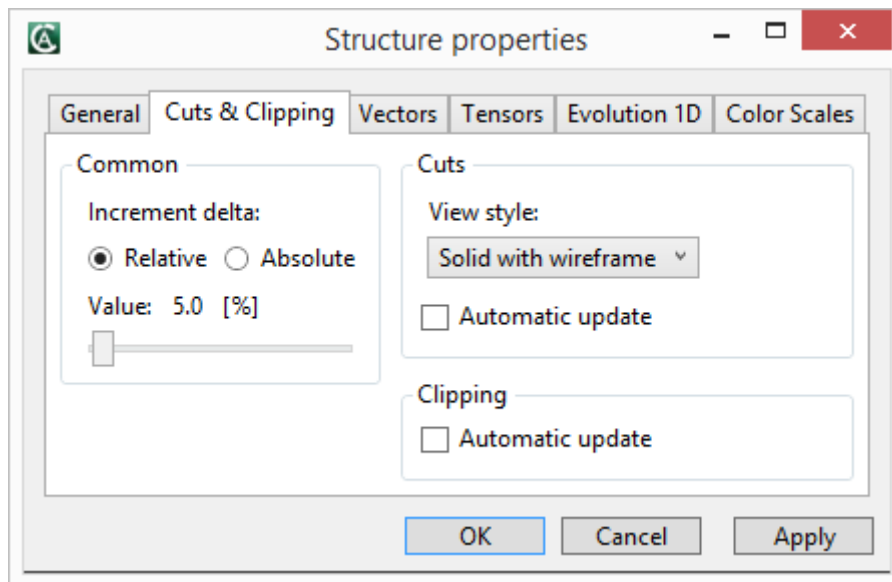
Fig. 3-26 Structure properties. Cuts & Clipping

3.6.3 Vectors Properties

- settings of results displayed as vectors (see Fig. 3-27)
- definition of vectors is described in section 3.7.1.3

Vectors range - displayed vectors can be a huge amount, so it is possible to filter them either moving thumbs on a range slider or manually. If the **Inverse range** is checked, vectors outside slider thumbs are displayed and inside are filtered. That is highlighted by a background color too.

Color settings – vectors can be colored by a **Color scale**, where its self definition is under the tab Color Scales, or by **One color**

Scale – length and size of vector arrows can be scaled as necessary.

Show labels – there is option to show a number of levels of max values, where one level means labels with the same value. Similarly to the **Vectors range**, there is the **Values range** which serves to filter labels by a some range. The **Value format**, where Automatic format switches between Scientific and Fixed format, and the **value precision** can be set as you wish.
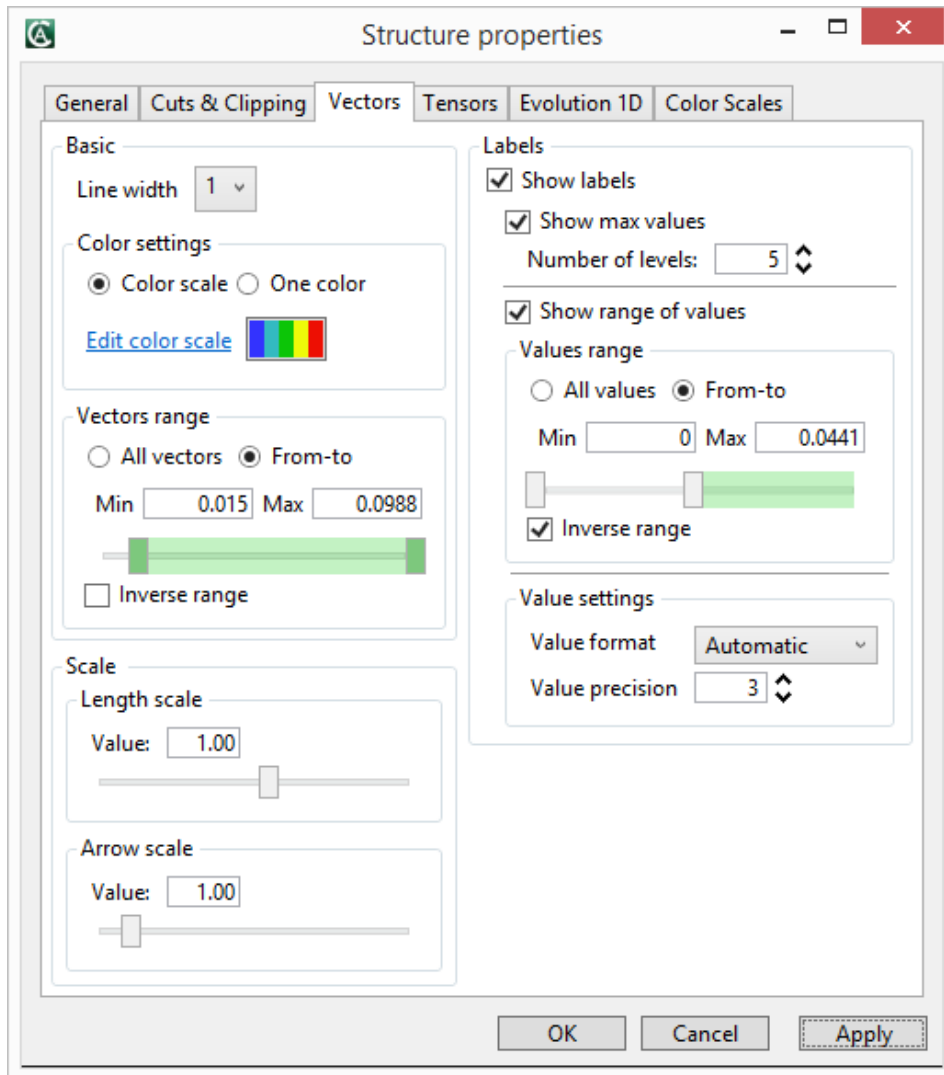
Fig. 3-27 Structure properties. Vectors

3.6.4 Tensors Properties

- settings of results displayed as tensors (see Fig. 3-28)

- very similar to the Vectors properties (see 3.6.2)

- definition of tensors is described in section 3.7.1.3

- tensor can be colored by a **Color scale**, where its self definition is under the tab Color Scales, or by two colors – the first one used for **Positive values** and the second one for **Negative values**

- often is desirable to filter tensor values close to zero which can be done by moving thumbs on a range slider in the **Tensors range** box with checked **Inverse range** (see Fig. 3-28)
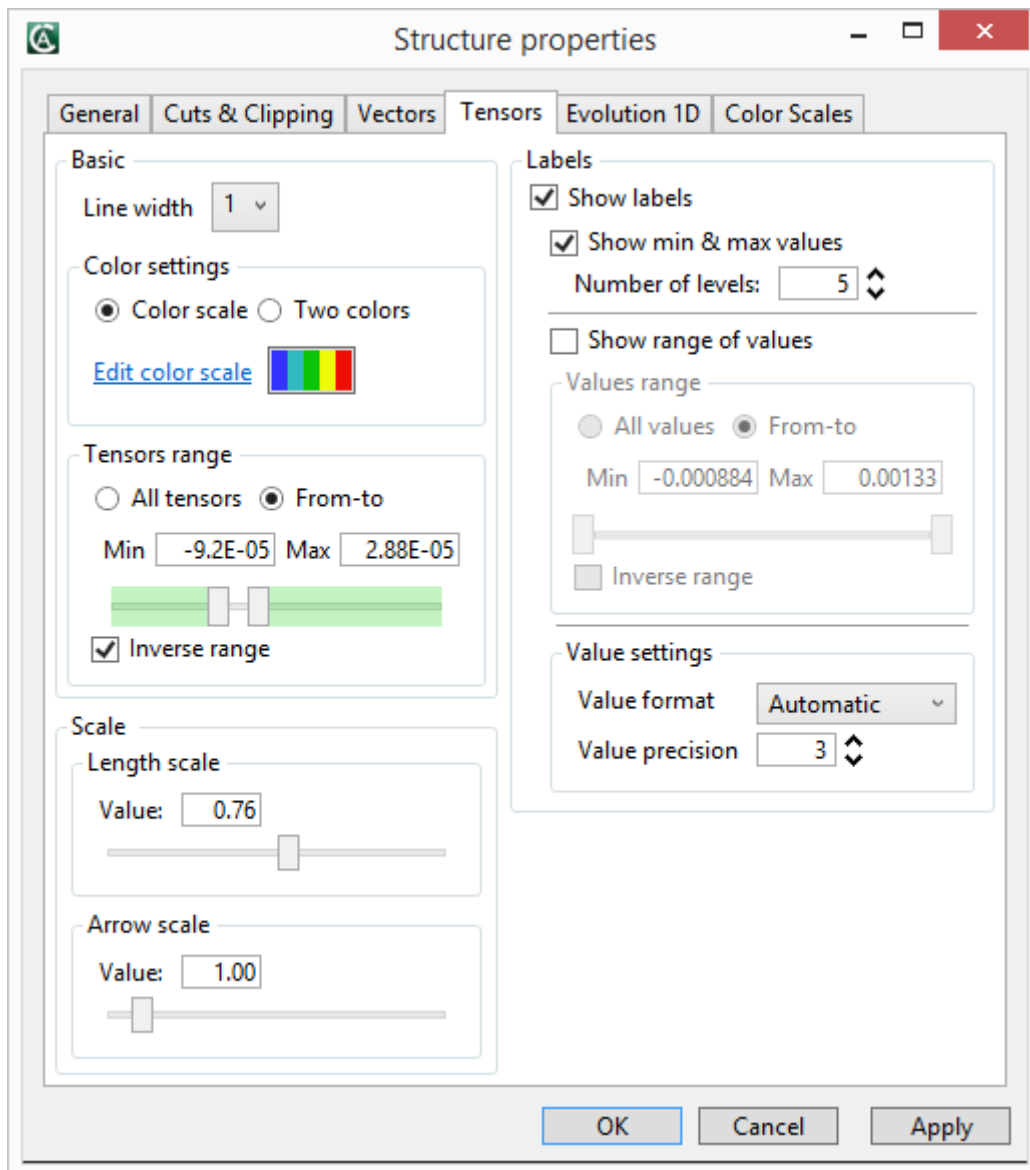
Fig. 3-28 Structure properties. Tensors

3.6.5 Evolution 1D Properties

- settings of an evolution above 1D elements (definition in section 3.7.1.3) and centers of cuts (definition in section 4.2.1) see Fig. 3-29.

Displacement direction – when Auto mode is selected, a displacement direction of the evolution is defined for each 1D element. Unlike for Manual mode it is set a global displacement direction.

Labels – there are options to show min & max value, first & last value, local extremes limited by a number of neighbouring items, which are all less or greater than these extremes.

Set forces (moments) comparable – If checked, displayed forces (moments) can be easily compared with each other according to their common max displacement.
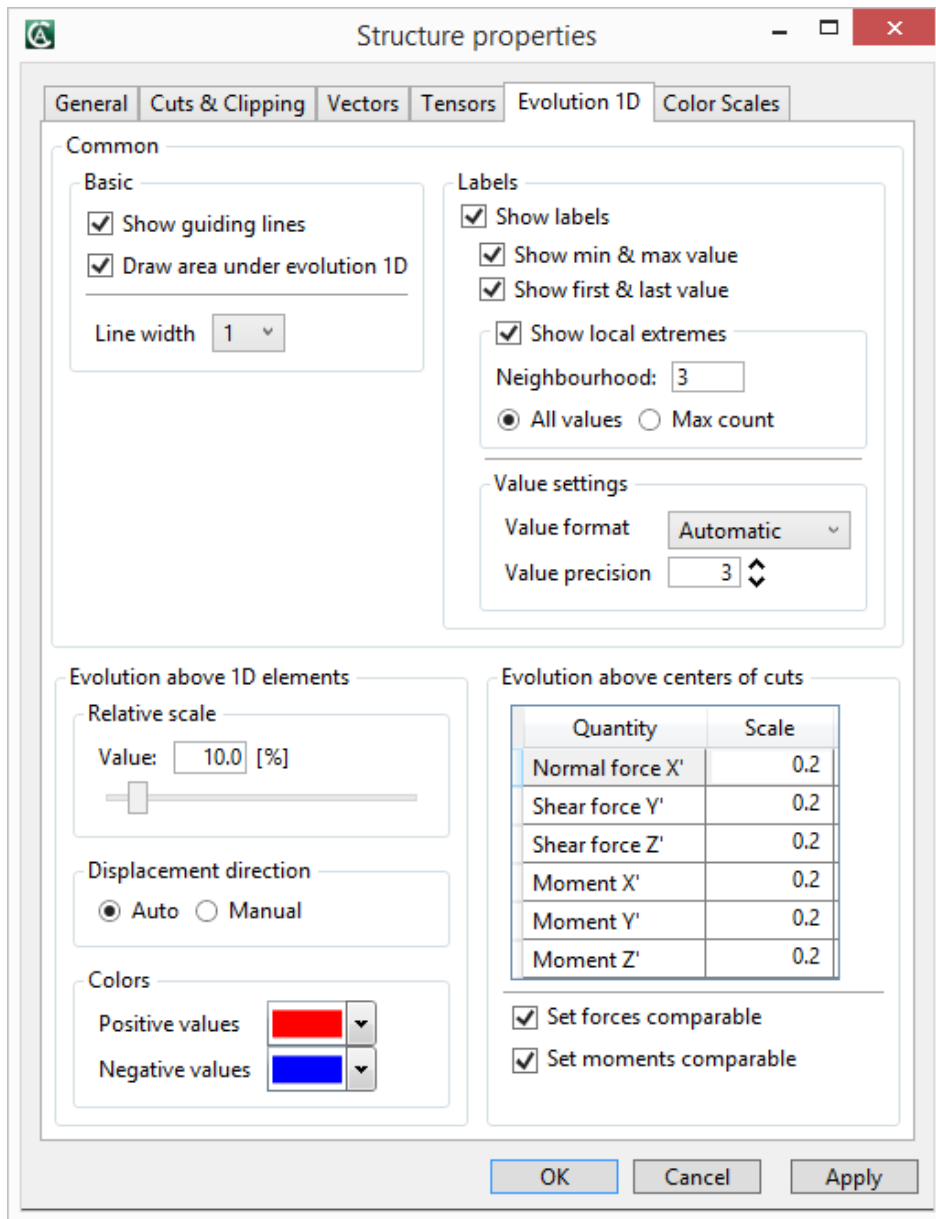
Fig. 3-29 Structure properties. Evolution 1D

3.6.6 Color Scales Properties

- a map linking colors with scalar, vector or tensor results which is dependent on selected **Result type** (see Fig. 3-30)

- there are three types of the color scale (standard - linear, separated by zero and user-defined). In case of the „user-defined" type size of the color scale can be dynamically increased or decreased. Checked values and colors can be edited and intervening items between the checked ones are linearly interpolated.

- in case of **Draw iso areas** one color is disabled because a color on row determines range from value on this row to a value below.

- to change format of a value (**Value format**) it is possible to choose one of the pre-defined format type: "Automatic", "Scientific" and "Fixed". "Automatic" format switches between Fixed and Scientific format according to "Value precision". If set **Auto value precision,** value precision is set such that neighboring values are distinct with minimal used digits.

- user can change color of checked values and align values on left or right

- for difficult color scale, that you want to use multiple times, there is an option to save a current color scale into a special file and load it again in another project.
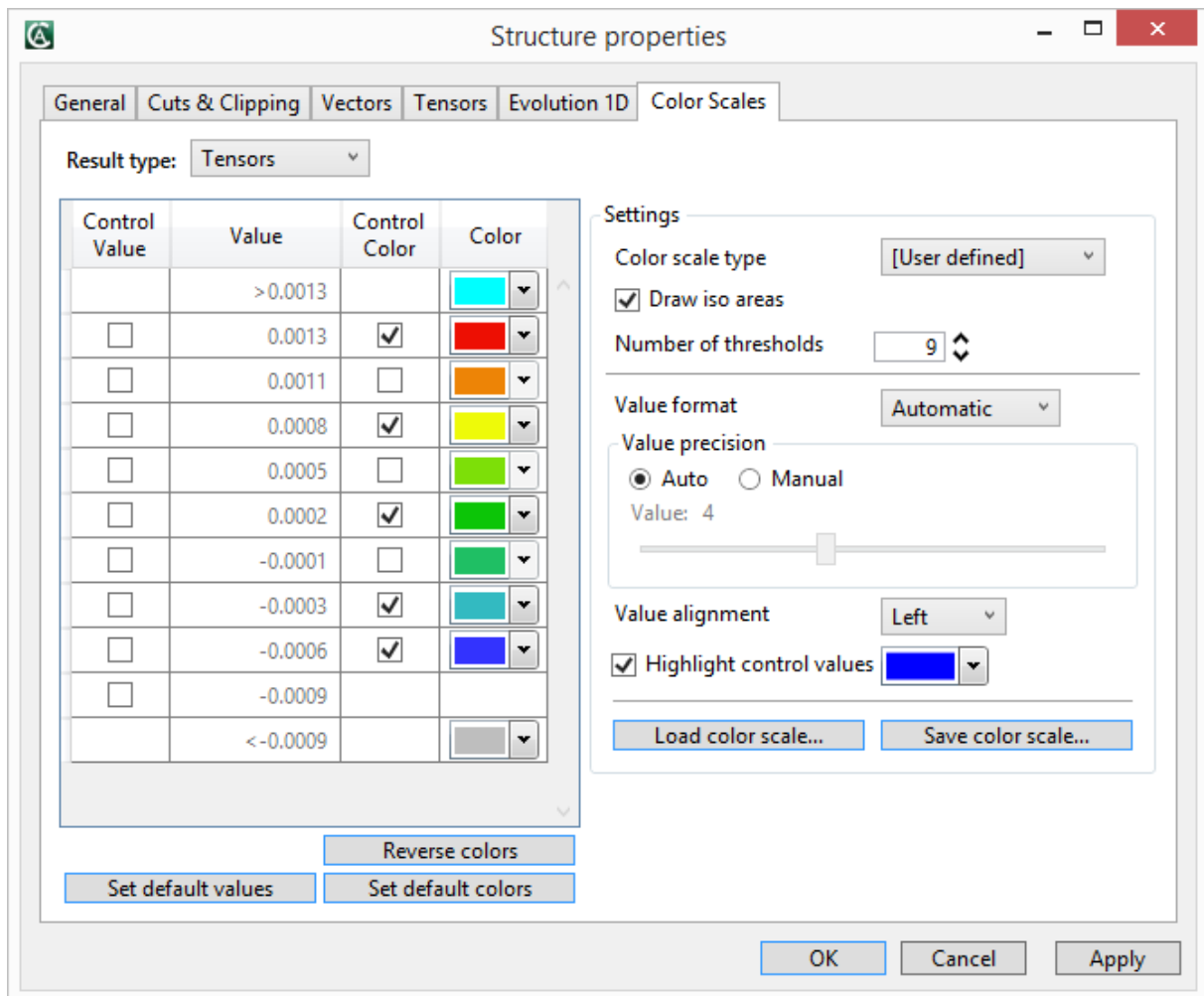
Fig. 3-30 Structure properties. Color scales

3.7   Settings for Structure Window

Real-time results can be set by four toolboxes on the right side. They can be also activated by command in **View** menu if they are closed.

3.7.1 View Settings Toolbox

View settings toolbox is used for showing results in real-time and also there can be set view style and light. The light function was explained in 3.6. There are five view styles of construction (solid model, wireframe model, outline, solid with surface mesh and solid with outline).

3.7.1.1 Deformations

It is possible to draw deformed model. Deformations are usually very small so they can be scaled. There are two types of scale, absolute and relative. Absolute scale uses number which multiplies the deformation. Relative scale uses the size of construction. For example: the deformation is 3mm and the beam's height is 100mm, using relative scale 10%, the max deformation in structure is drawn as 30mm. Using absolute scale 5, the max deformation is shown as 15mm.

Undeformed model can be shown together with deformed model. There are the same view styles as for deformed model.
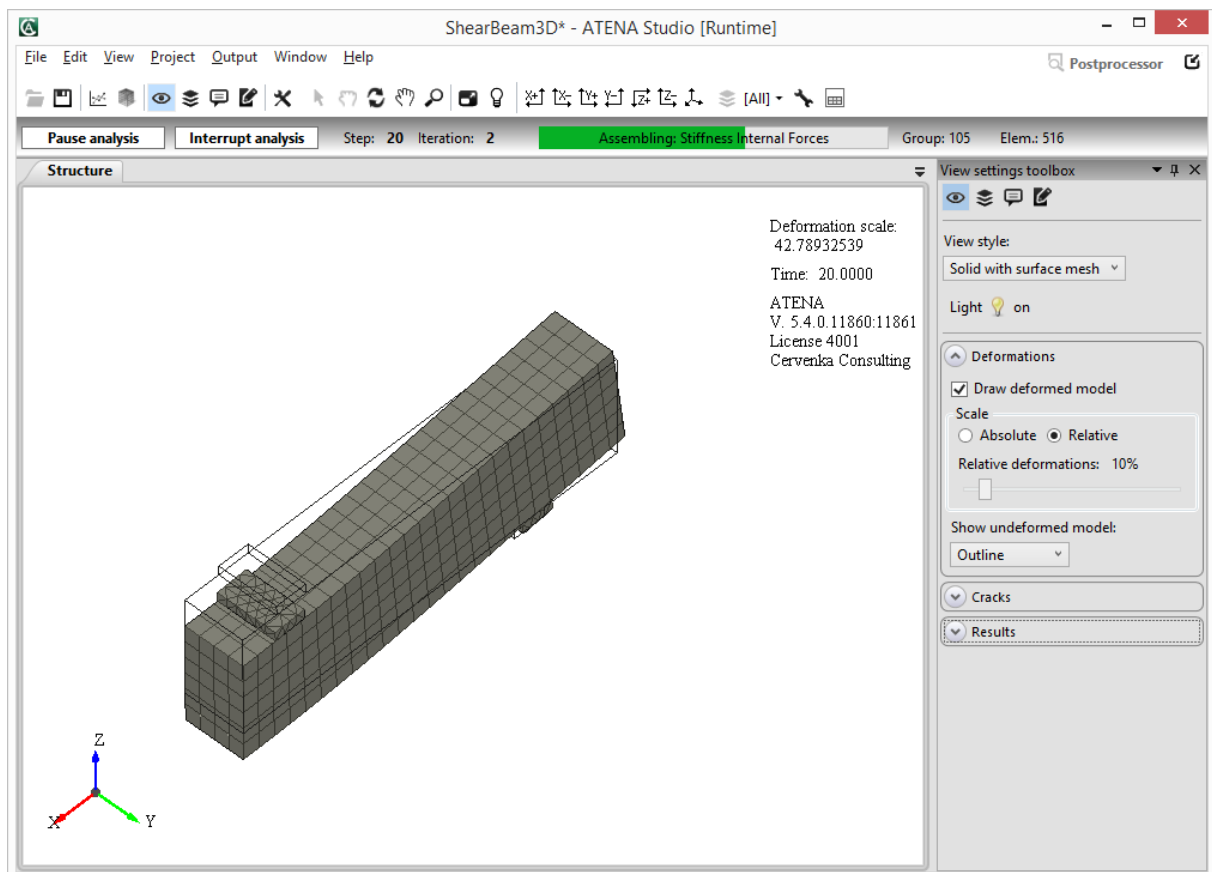
Fig. 3-31 Deformation settings and construction window (deformed model with undeformed outline).

If you are executing dynamic analysis, it will be interesting for you to display **eigenfrequencies** and **eigenshapes/vector**. It can be found in window below named as **Data for deformed model** (see **Fig. 3-32**).
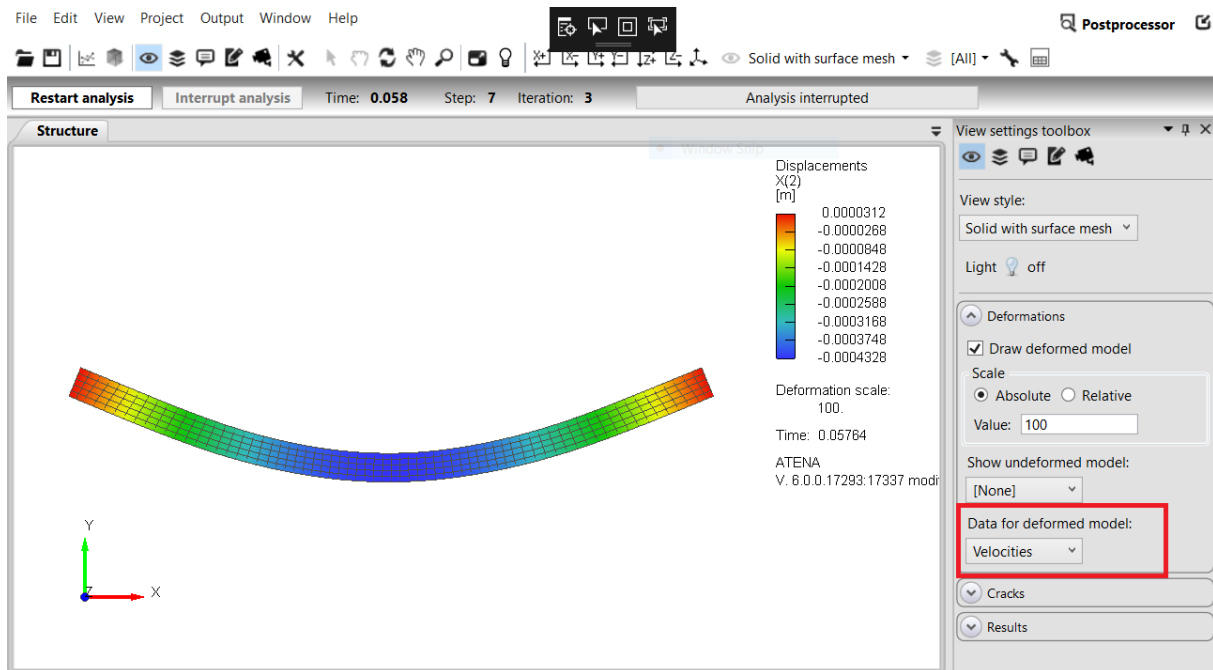
Fig. 3-32 Dynamic analysis - velocities shown as deformations.

3.7.1.2 Cracks

Cracks can be shown in elements or at integration points (IPs). There can be at most three cracks in each element but the count of drawn cracks can be set with Max cracks level. Width multiplier increases the width of crack lines on screen. This enables a better representation of the crack width. Option Auto tries to set optimal value of this parameter. When the construction is ragged there are many small cracks. It is good to set minimal crack width to filter the minor cracks. When the mouse hovers over the textbox the tooltip shows the range of possible input values.
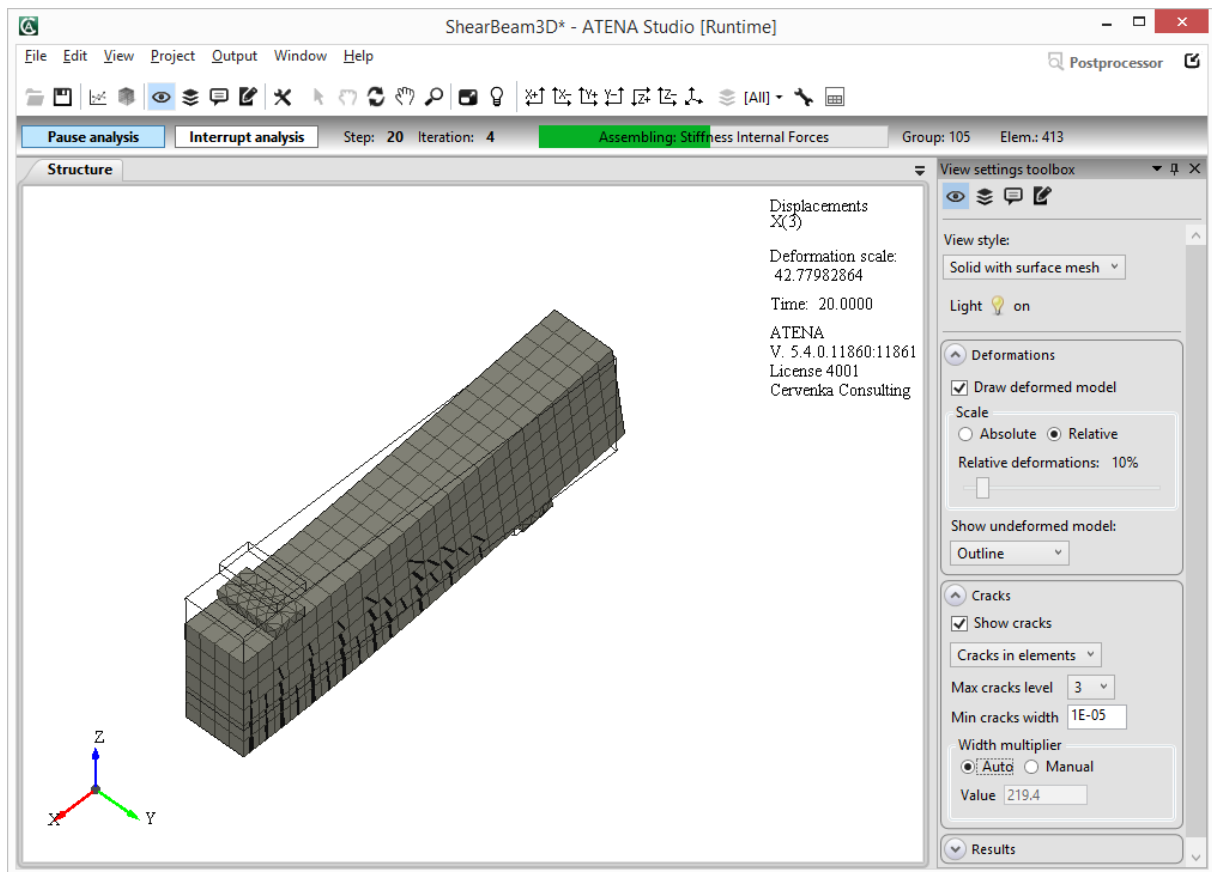
Fig. 3-33 Cracks settings and construction window.

3.7.1.3 Results

This part of the View settings toolbox controls the display of various results; by contour areas (Scalars), by vectors (Vectors), by principal tensors (Tensors) or by evolutions above 1D elements (Evolution 1D). Vectors are available for three dimensional results as e.g. Reactions, External Forces… By tensors it is typically meant Principal Stress or Principal Strain. Vectors, tensors or evolutions 1D can be edited in Structure Properties (see 3.6.3, 3.6.4 or 3.6.5). Displayed data results are dependent on location. There are four types of location (nodes, element nodes, elements and element integration points (IPs)). First choose location and then the resulting value which should be shown (for example: displacement, crack width, stress, strain, etc.). The displayed values are also dependent on shown activity (1D, 3D, user-defined…). For details about activities see 3.7.2. It is necessary to the confirm changes using the button **Apply**. There are some examples of result post-processing presented in the following pictures.
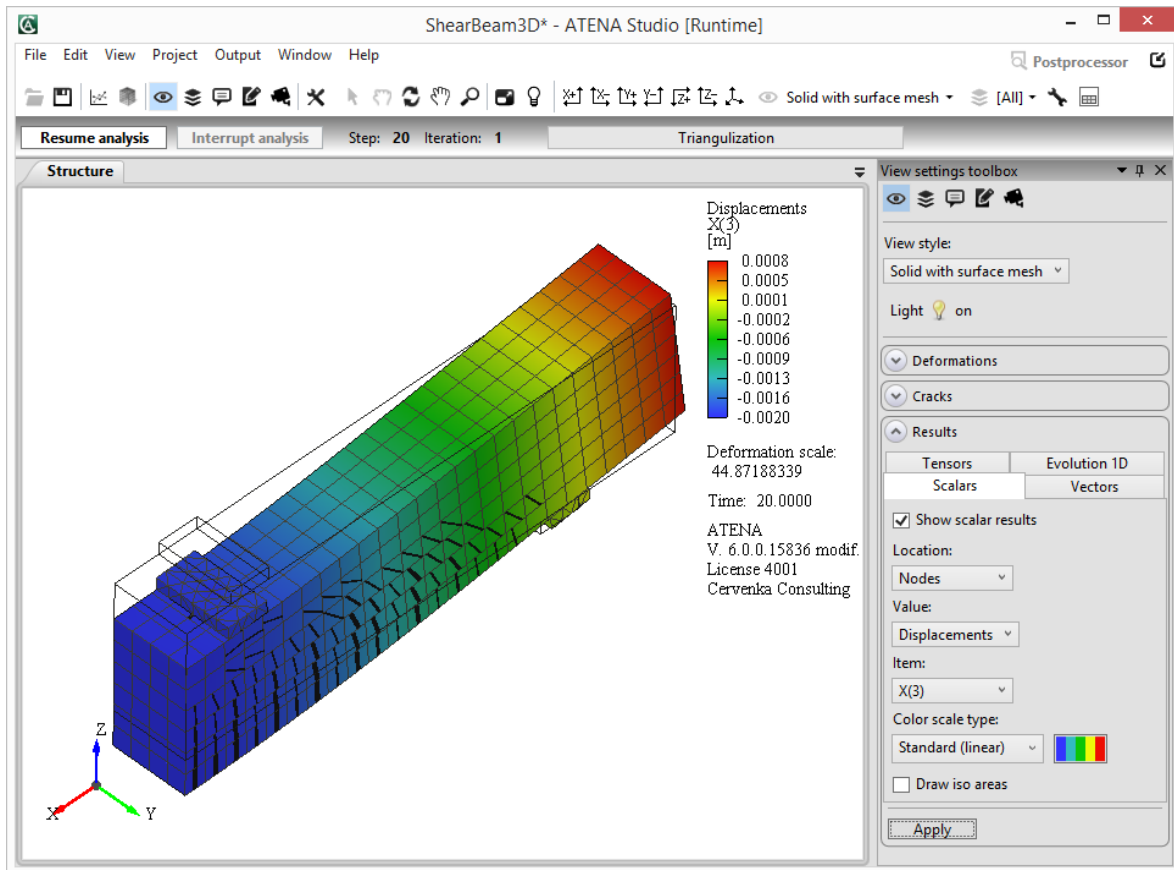
Fig. 3-34 Displacement without iso-areas.

Draw iso-areas
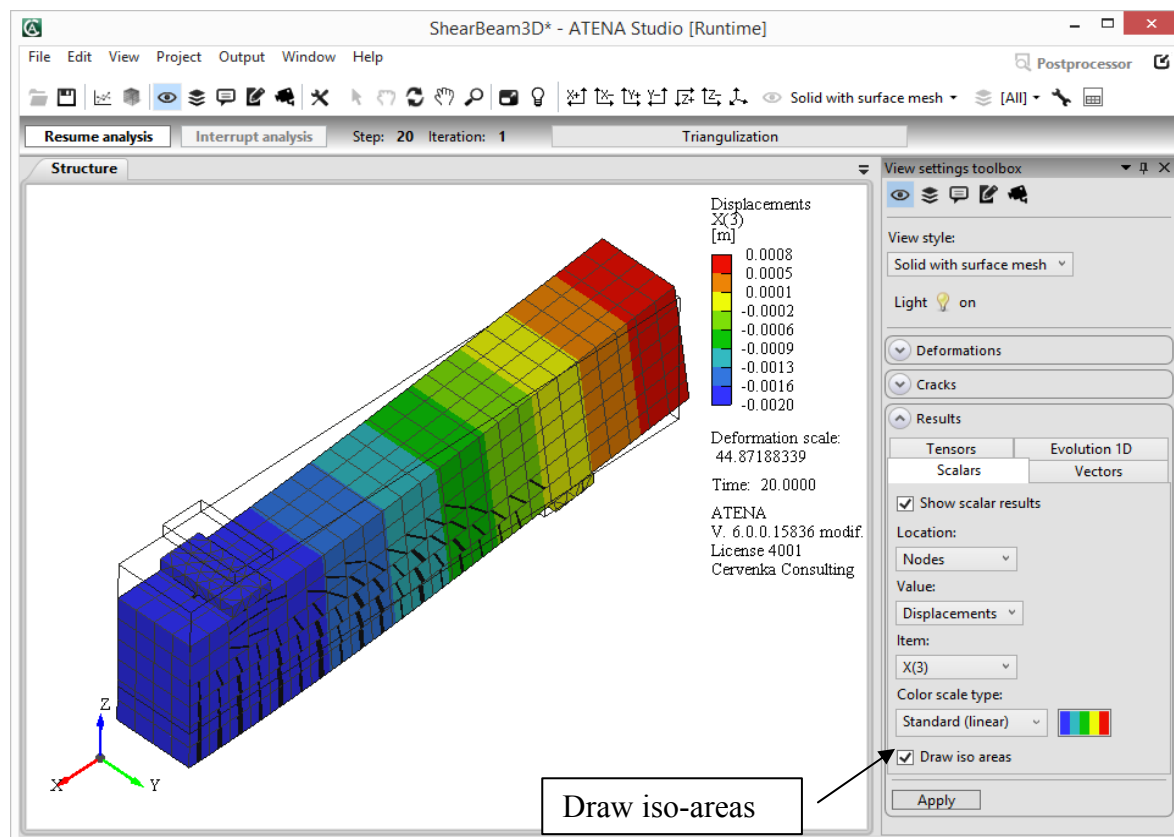
Fig. 3-35 Displacement with iso-areas.
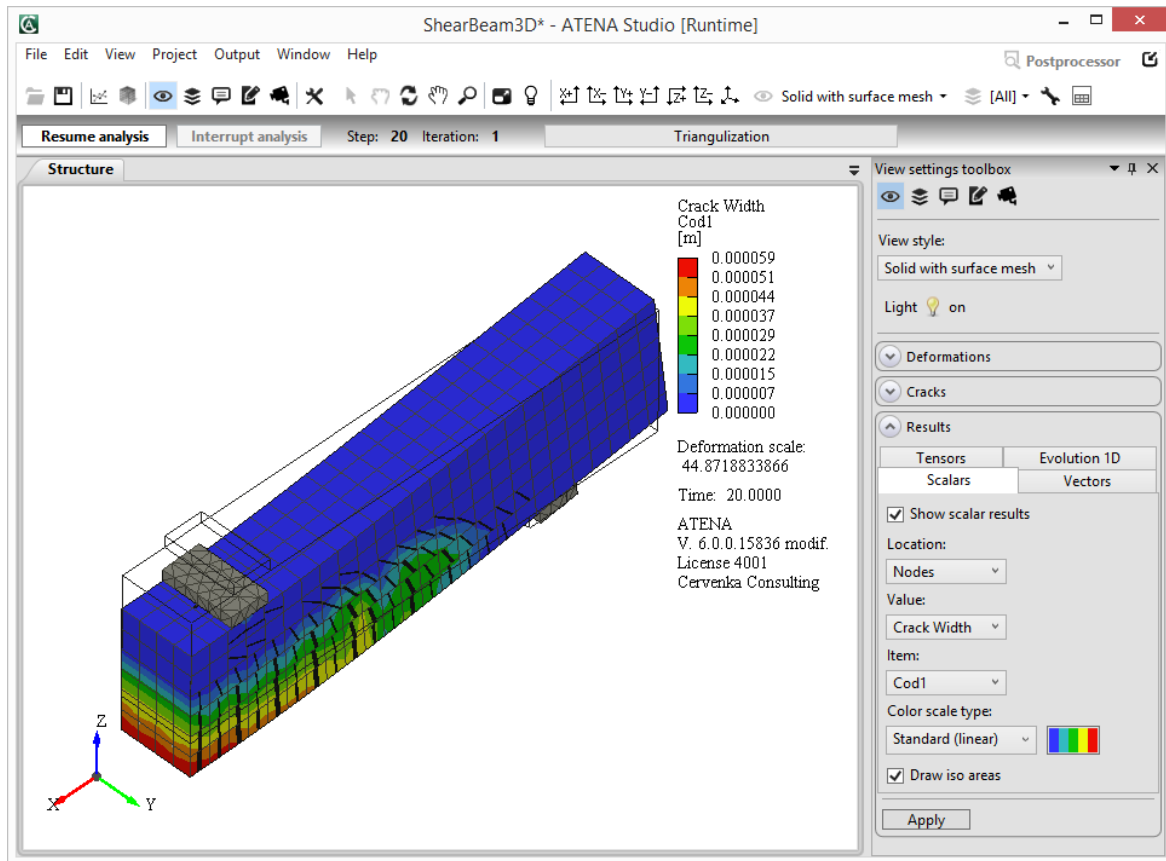
Fig. 3-36 Crack width.
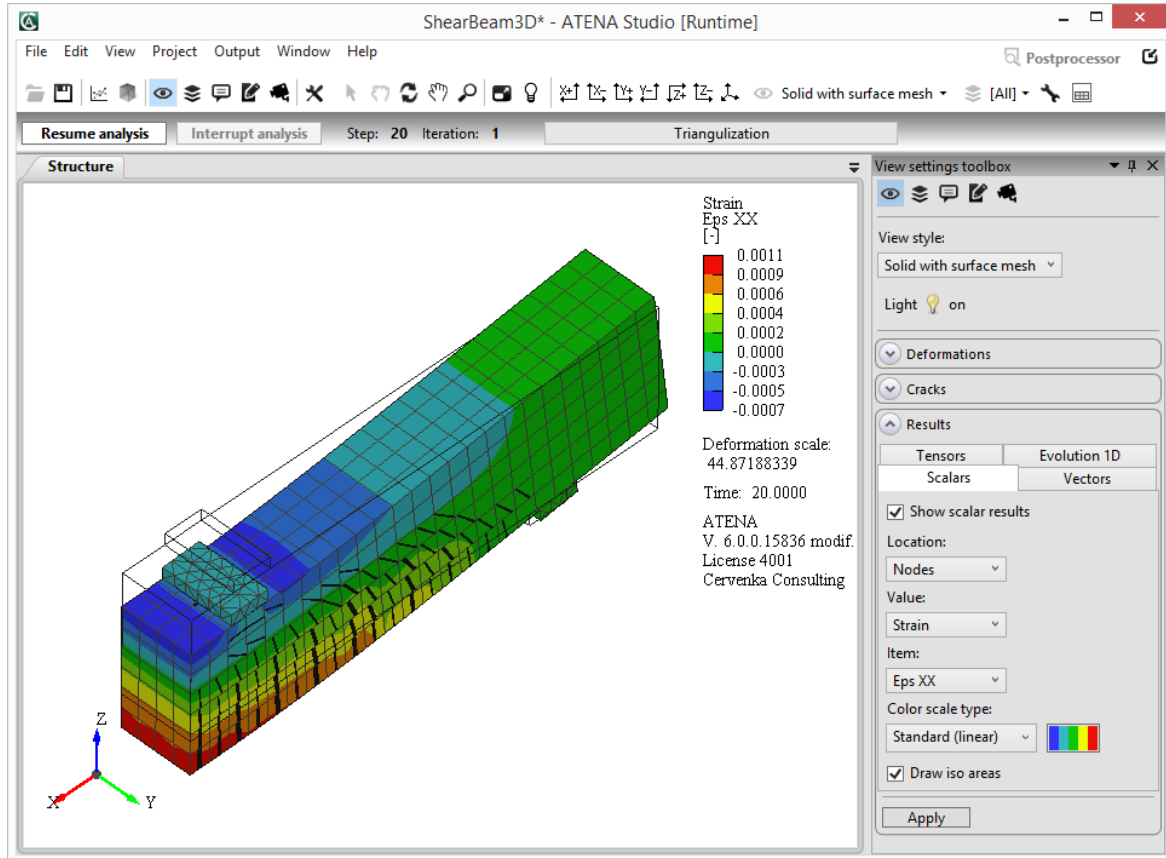
Fig. 3-37 Strain epsilon XX.
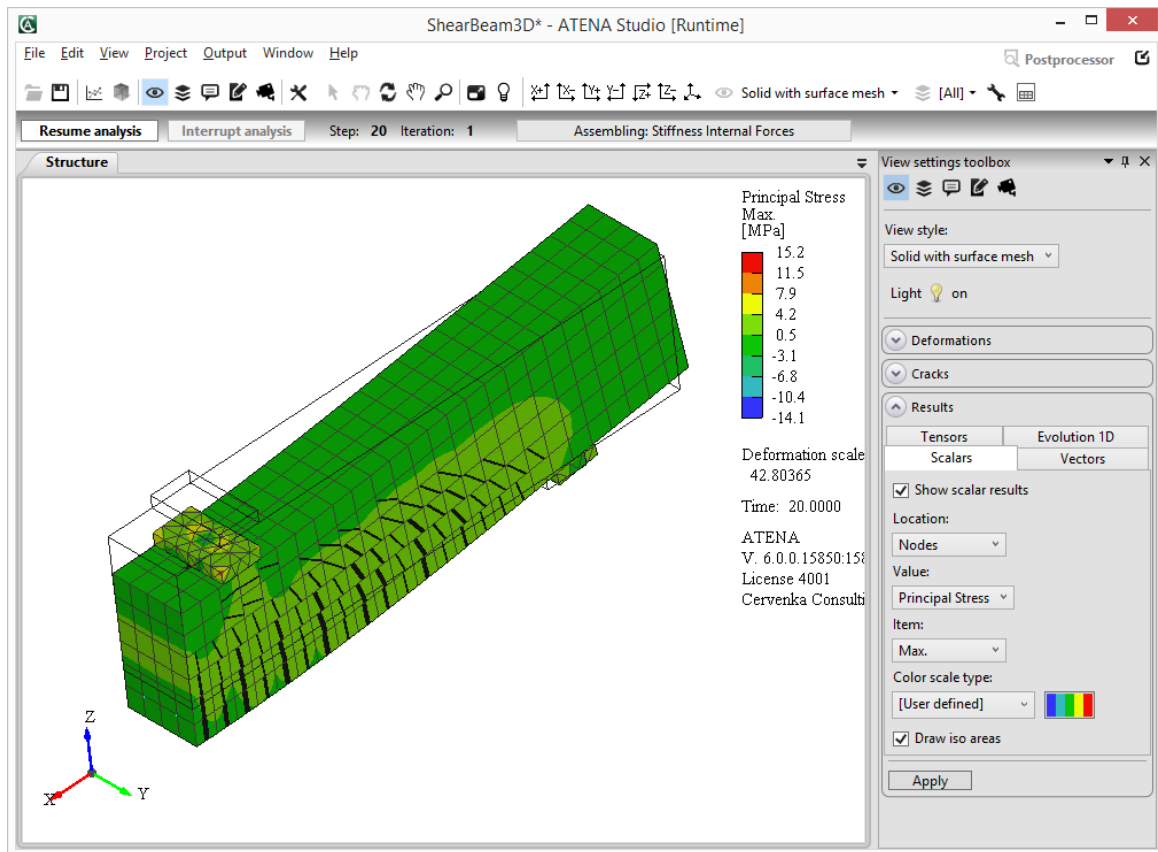
Fig. 3-38 Principal tensile stress.
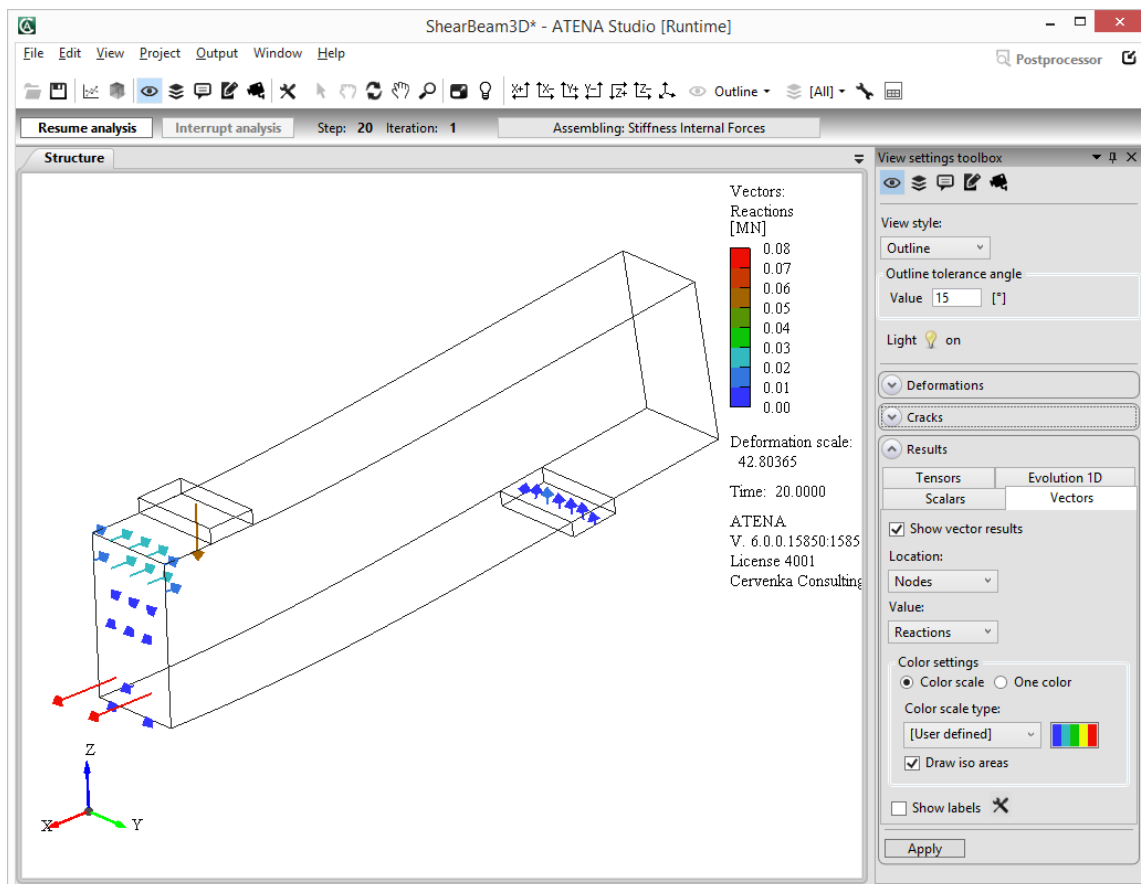
Fig. 3-39 Reactions in nodes displayed by vectors.
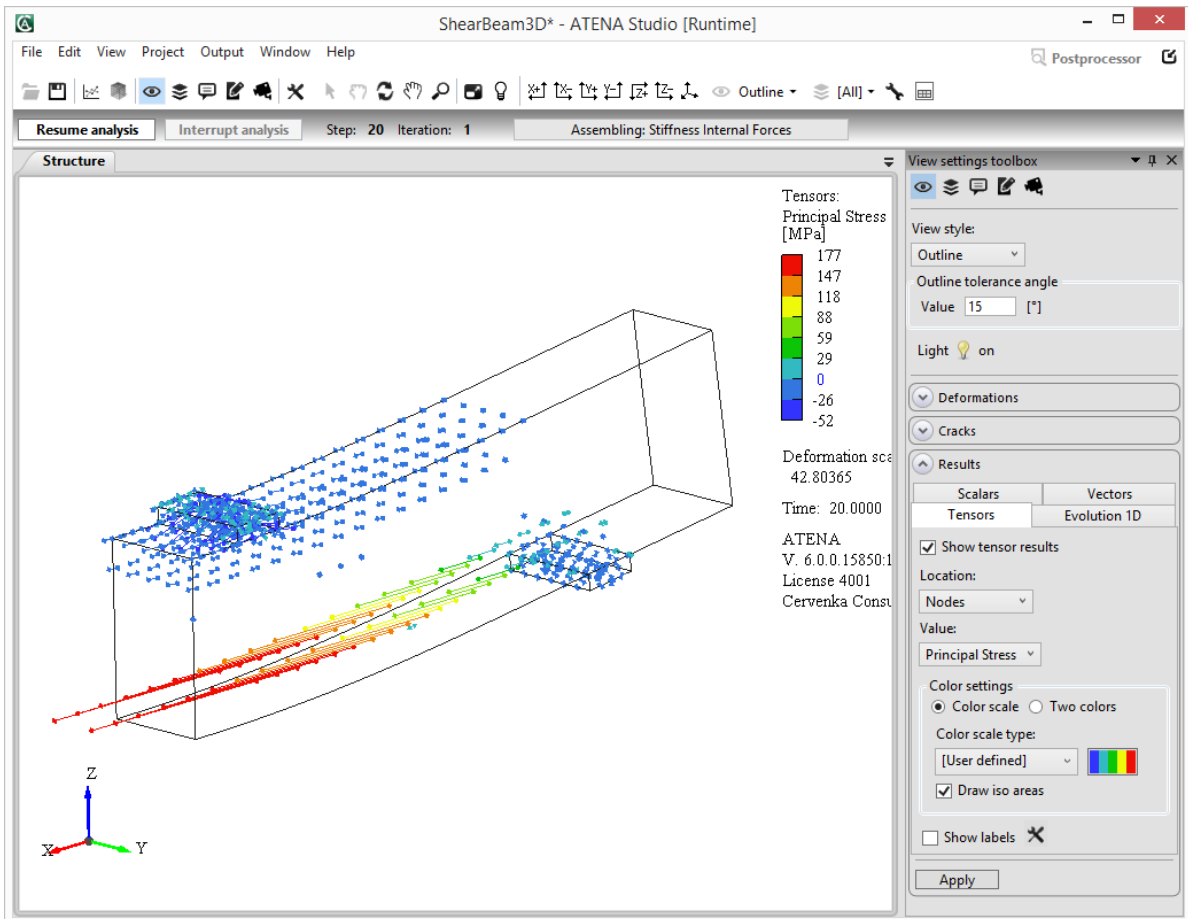
Fig. 3-40 Principal stress displayed by tensors.
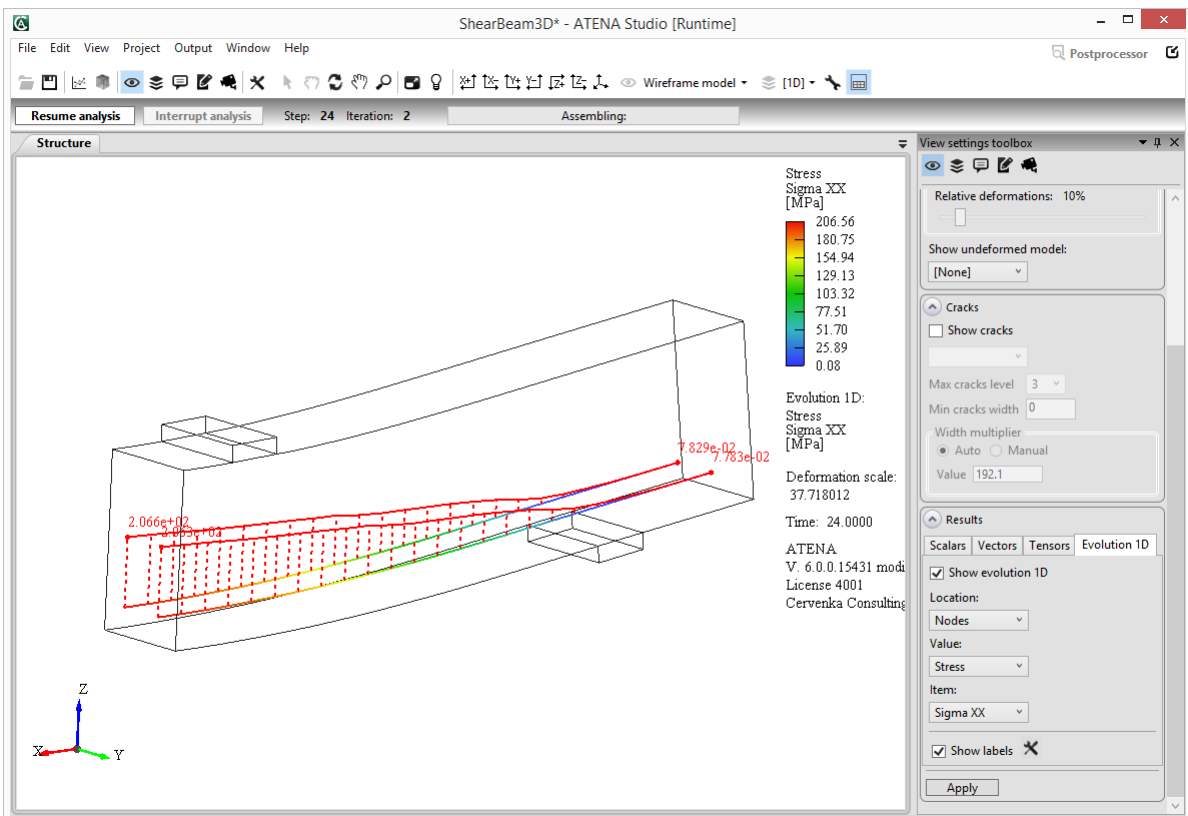
Fig. 3-41 Stress in reinforcement displayed by evolution 1D.
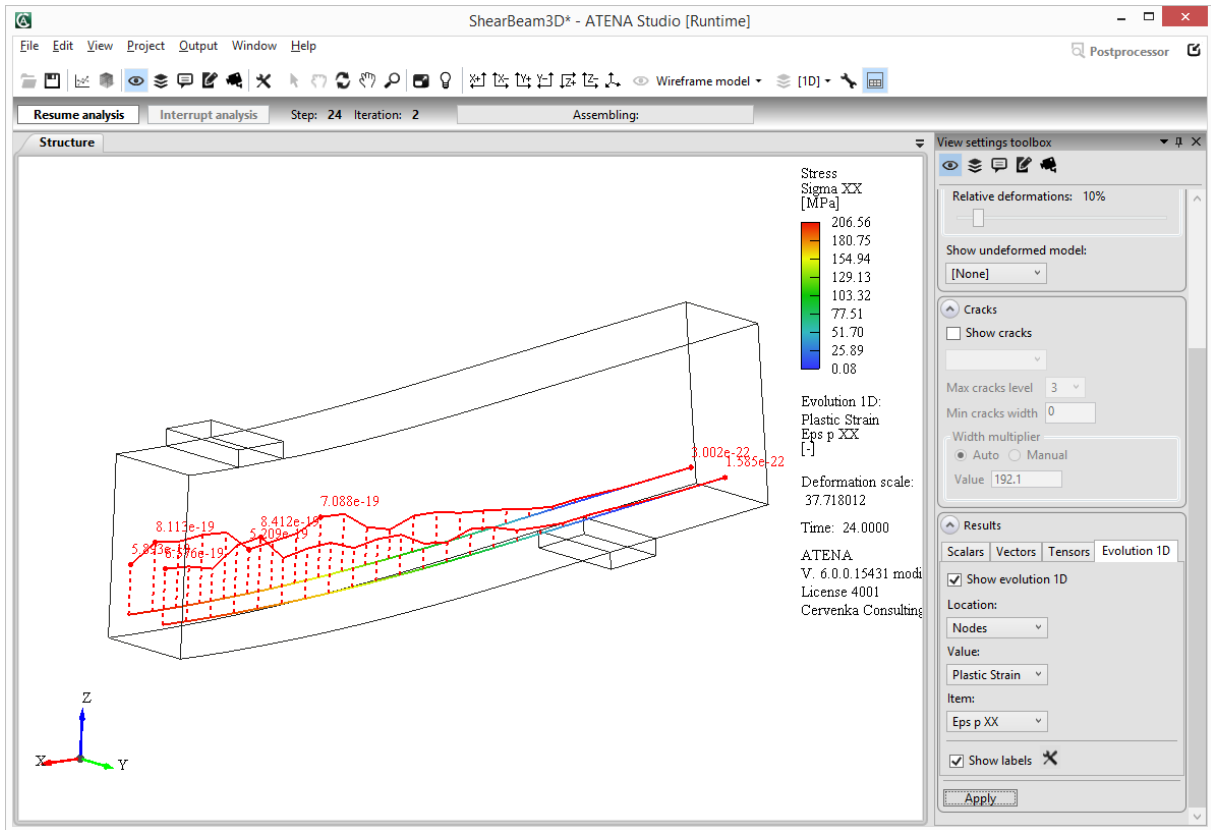
Fig. 3-42 Plastic strain in reinforcement as evolution 1D.
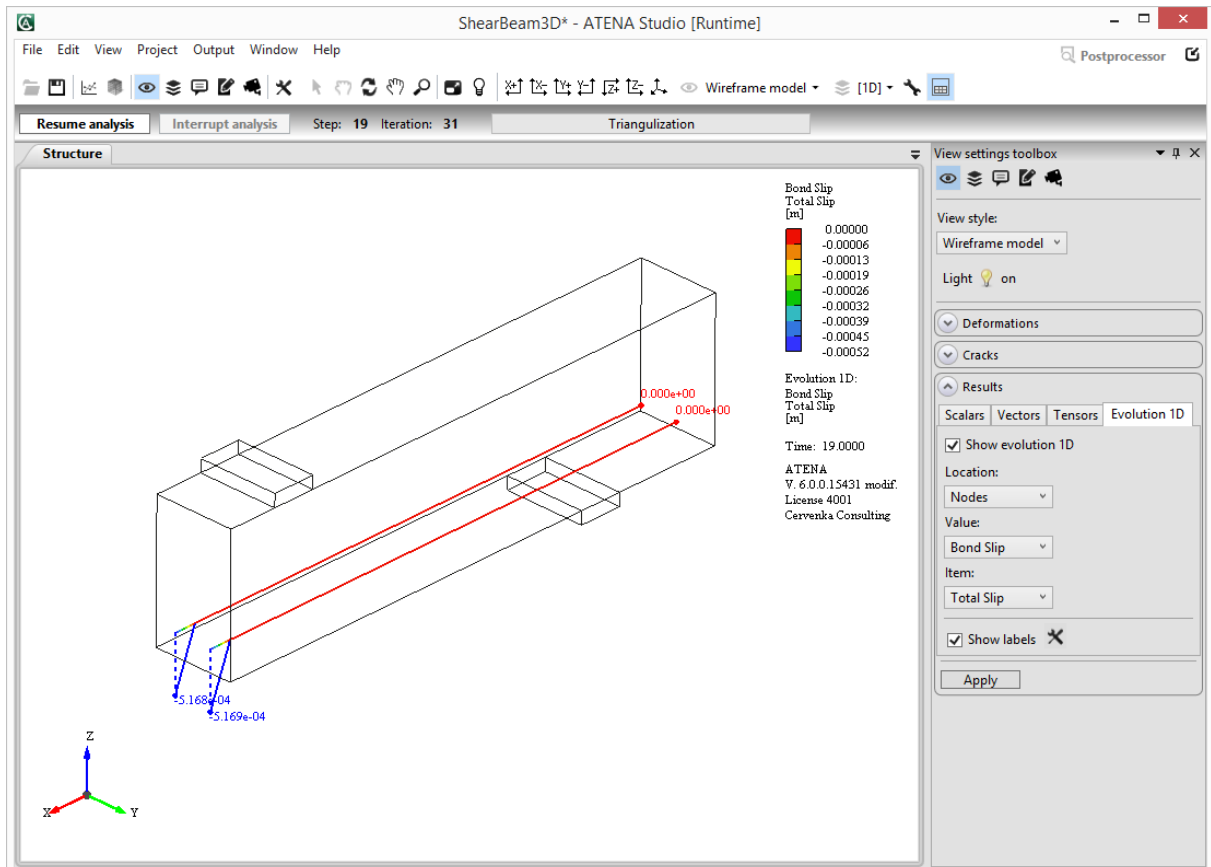
Fig. 3-43 Bond slip in reinforcement as evolution 1D.

3.7.2 Visible Domain Toolbox

Visible domain toolbox is used for the definition of activity sets, which control the groups and elements that will be shown in the **Structure** window. These settings can be applied only for some part of the structure, for example all 3D or 1D elements or for some user-defined subset of elements. This feature is similar to the "Layers" option known in CAD software. The only difference is that certain group or element can be a member of several activities and not just one layer, which is the typical situation for layers option in CAD programs. This named set of elements is called **Activity** in **ATENA Studio**. There are predefined activities such as **[All]**, **[3D]**, **[2D]** and **[1D]**. It is possible to create new activity using the button **Edit Activities**, which opens the **Activity manager** (see Fig. 3-44). Here new activity can be created using the button **Add new activity**, copied, removed or its name can be specified. The element groups that should be part of the new activity can be selected by ticking the appropriate checkbox or pressing key Space. More refined definition can be achieved with the help of the Selections dialog. The final activity is the intersection of the selected element groups and the named selections, where even individual elements can be selected (see 3.7.4 in "Selections in activity").
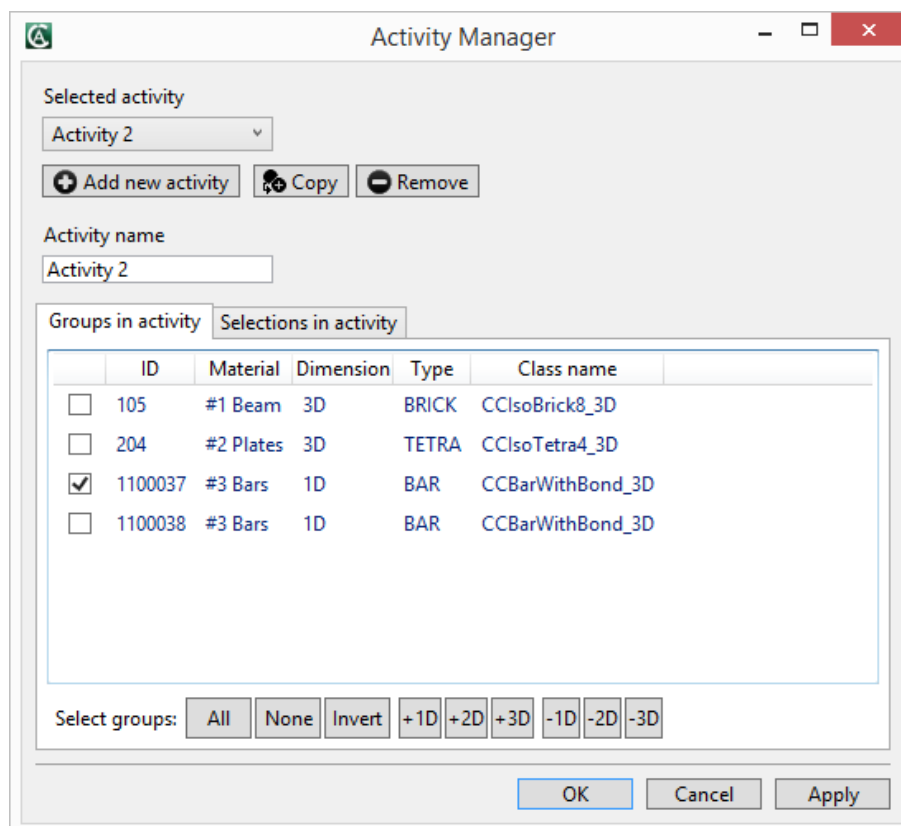
Fig. 3-44 Activity Manager.

3.7.2.1 Cuts

The cut option enables the user to define an arbitrary number of cut planes that can be used to visualize the results inside the analyzed structure. The cut plane is defined by three points or by a point and a normal vector orthogonal to the requested plane. For both cases it is necessary to define a tangent vector, which defines a direction of an integrated shear force Vy'. Point coordinates can be inserted manually or grabbed from the structure windows by picking existing nodes. To that press the button **Select** and then select node in the structure window. After that point coordinates are automatically grabbed and can be adjusted manually if needed. For faster manipulation it can be checked the **Update automatically position** checkbox in Structure Properties (see Fig. 3-26), which allows to redraw the cut plane after any change of coordinates.

The cut plane is then drawn inside the structure (see Fig. 3-45). On the structure surface a contour is visible only. To see the full cut, it is necessary to change the view style on the top toolbar or in the **View settings** toolbox to outline or wireframe. The cut view style can be changed in Structure properties dialog, section Cuts & Clipping (see 3.6.2), directly accessed by a tool icon in the cut definition dialog. A cut of 1D elements (e.g. reinforcement bars) is represented by a sphere.
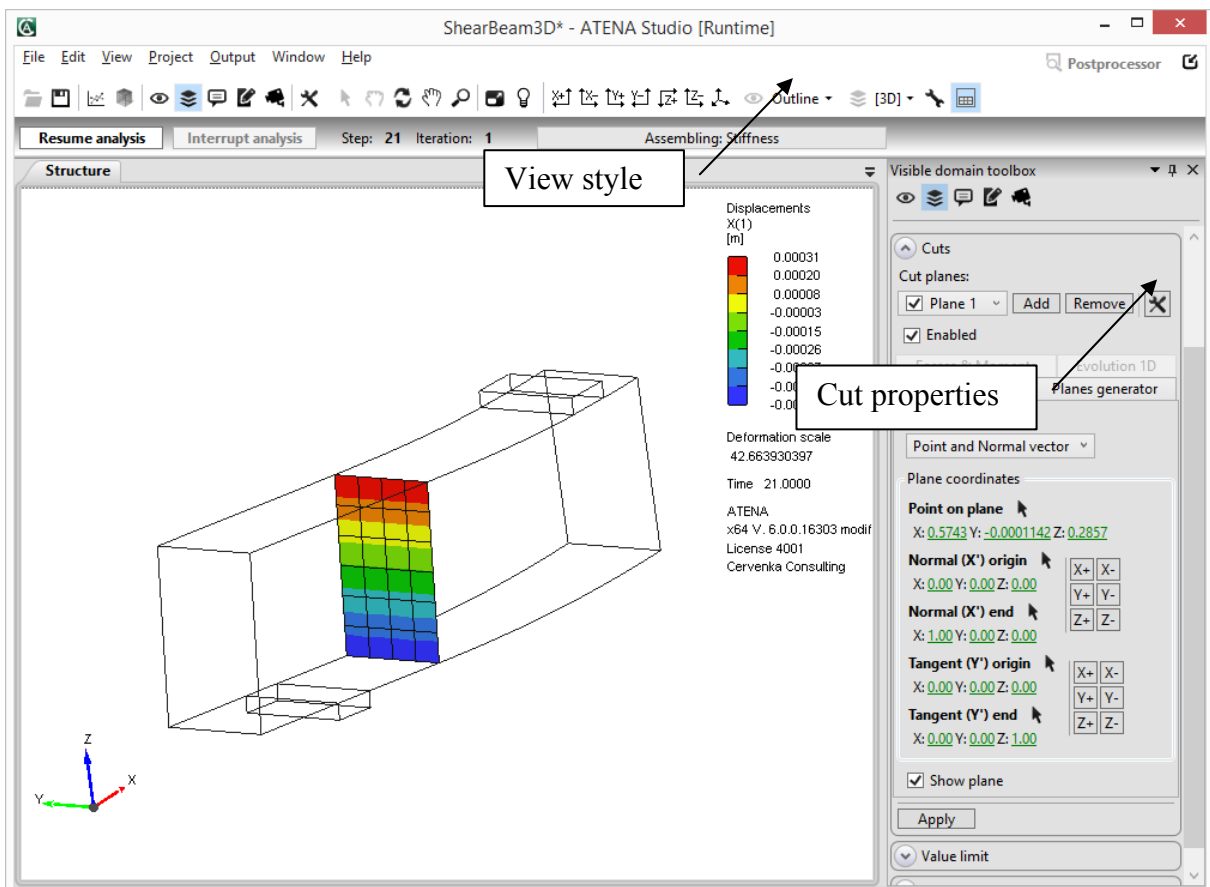
Fig. 3-45 Cut plane. Solid with wireframe.
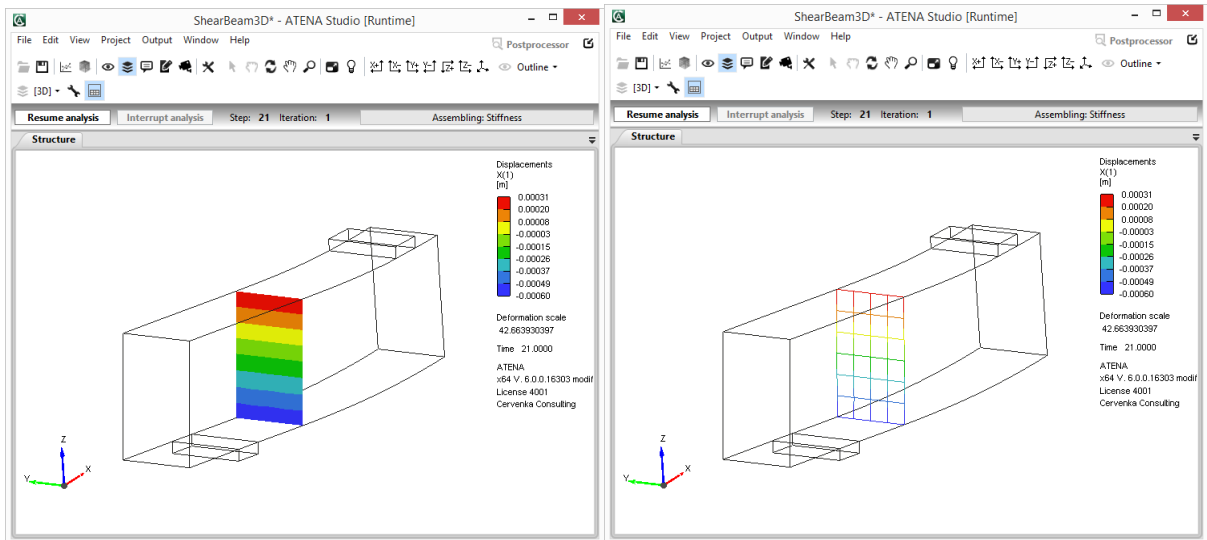
Fig. 3-46 Cut plane. Solid faces - left, wireframe - right.

In various cases one cut plane is not enough and it is required to have more parallel cuts and above them to see evolution of forces and moments. To that there is **Planes generator** section, where can be set a number of planes to generate and boundary of generation; either over whole model, where must be set direction of generation, or from one point to other one. After pushing Apply new cuts are generated. They can be hidden using the **Show planes** checkbox and above them it can be shown evolution of integrated forces, which is enabled only in Post-processor (see 4.2.1).
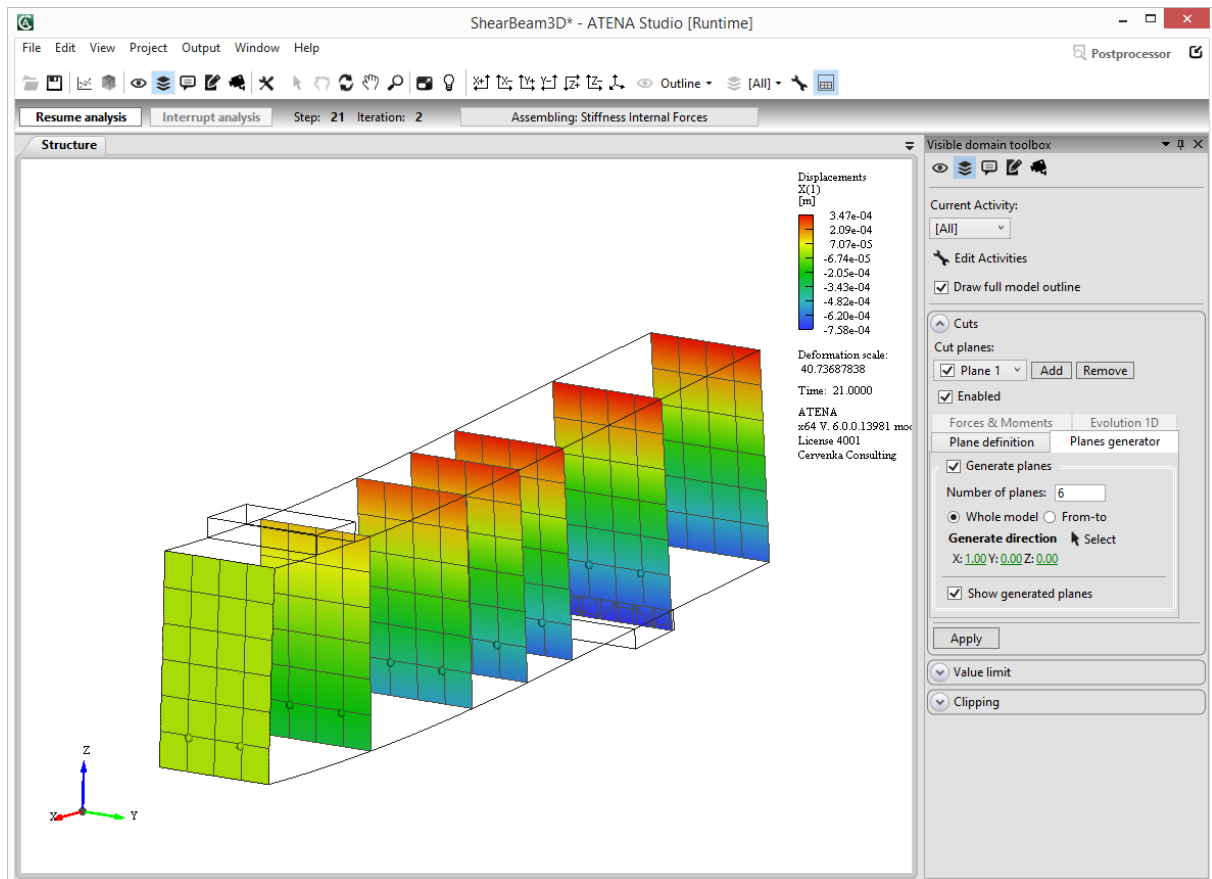
Fig. 3-47 Generated cut planes.

3.7.2.2 Value Limit

The displayed elements can be also filtered by output data value which is currently active in **Results** box. In this case a maximum and minimum value for the current data is to be specified. The program then displays only elements which satisfy the condition that the output quantity lies inside the provided interval of max. and min. values. If values in element node do not comply, the element isn´t shown. For invert display the option **Inverse** can be used.
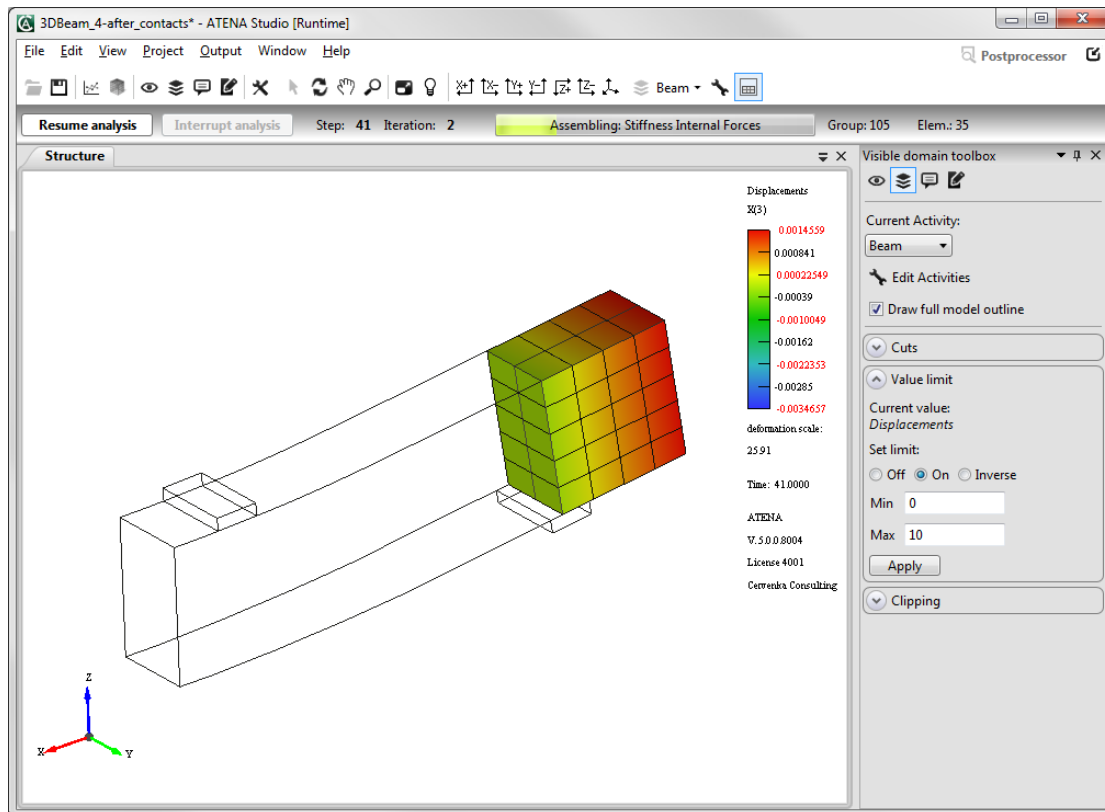
Fig. 3-48 Value limit.

3.7.2.3 Clipping Planes

This is an additional option for defining cuts. The previous **Cuts** feature creates cutting planes inside the structure, while the clipping plane feature removes parts of the structure outside the provided clip planes. The clipping plane is defined by three points. There can be up to 6 planes and only the part of the structure will be shown, which is defined by the intersection of all these planes. The point coordinates can be inserted manually or grabbed from the structure window. To do that press the **Select** button and then select a node in the structure window. Coordinates are grabbed automatically and can be adjusted by the user if needed. The above steps should be repeated for the next 2 points, and finally click **Apply** to show the clipped structure.
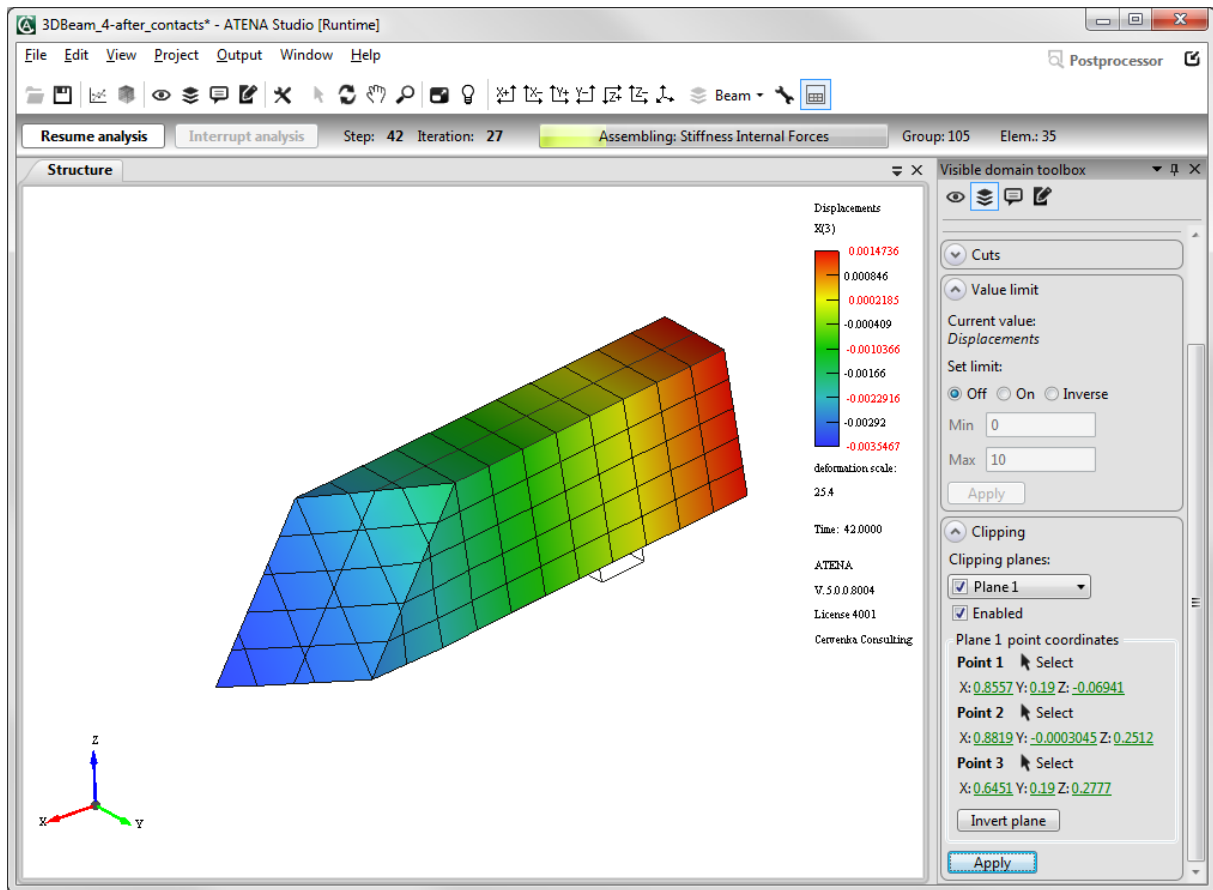
Fig. 3-49 Clipping plane.

3.7.3 Label & Signal Toolbox

3.7.3.1 Labels

Purpose of Labels box is to display additional information describing entities in the structure window, e.g. Node IDs, Element IPs etc. When you choose desired entity click on **Show all ...** to display all entities in the structure or you can specify the area where the labels will be shown by pressing **Select to show** button. When your selection is done, you can add more entities to the current selection by checking the option **Add to selection** in the yellow box.

It is possible to mark more entities at once, the command **Clear all** will erase all defined labels.
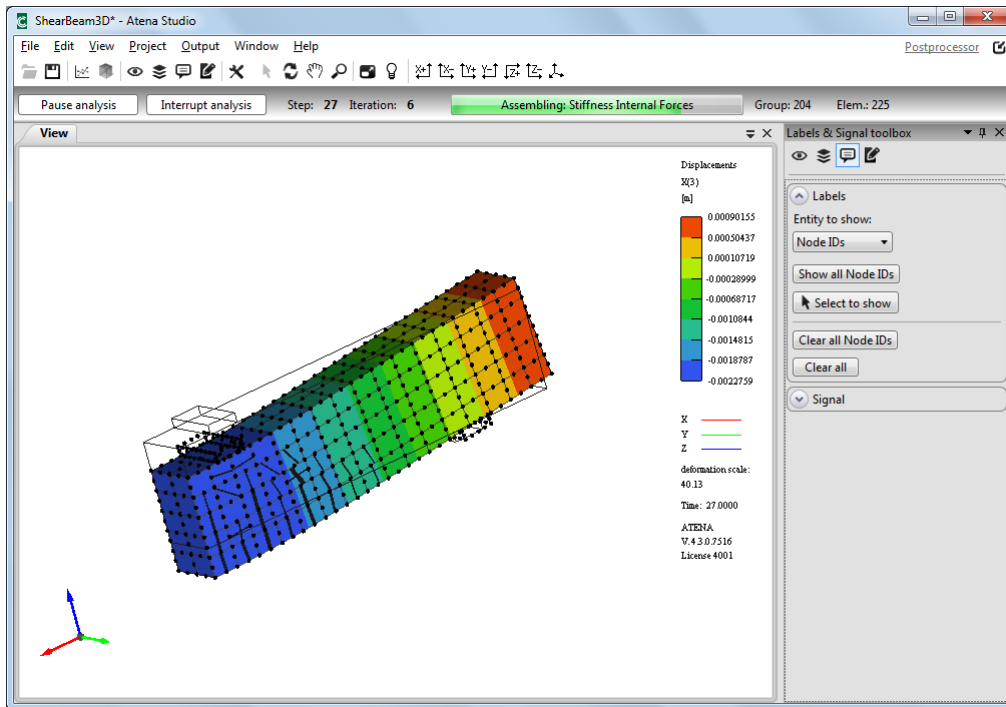
Fig. 3-50 Node location is marked.

3.7.3.2 Signals

Signal is a useful feature that enables the user to find the location of certain node, macro-node or element according to its id. It is possible to show only one entity at the same time.
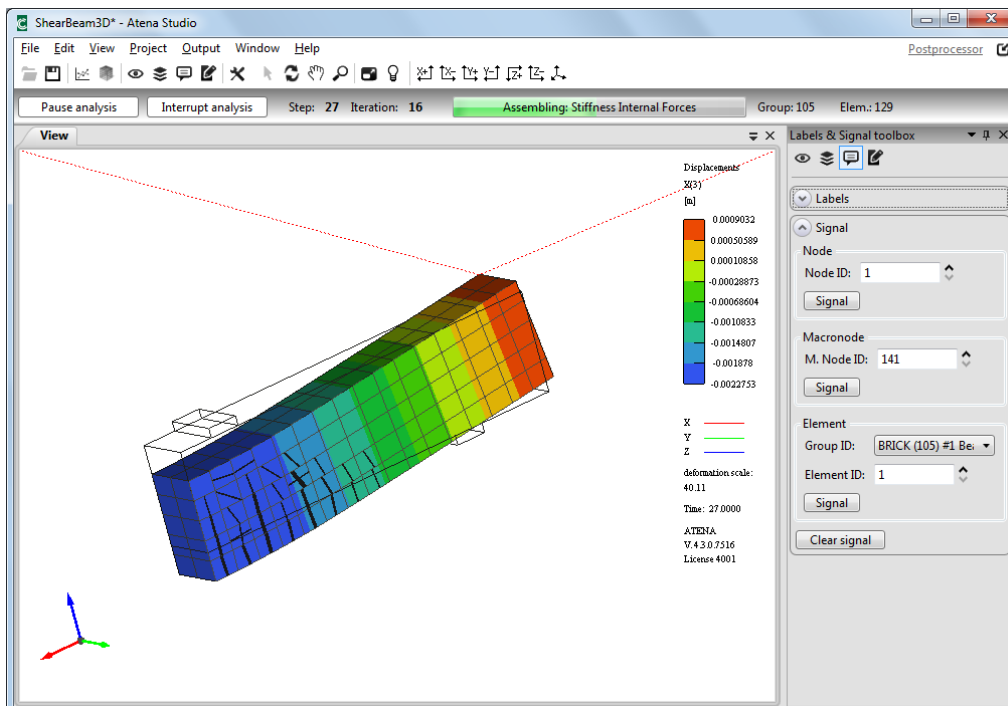
Fig. 3-51 Node ID 1 is signaled by the intersection of the red lines.

3.7.4 Selection Manager

A selection is a user-defined set of nodes, elements or element IPs. It has a name, and it can be used later on in other commands. For instance, it is useful for text output of results at selected nodes, elements or element IPs. It can be also used for the definition of displayed activities in combination with element groups (see Activity manager 3.7.2). Its selection can have a name, can add other selections and its content is defined by the selection of nodes, elements or element IPs(see 3.7.4.1). The created selection can be modified if needed using the button Show/Edit selected entities , which opens the selection editor where you can define entities by its id. The new selection is created by clicking the button **Save selection**, and the new selection then appears in the list of available selections.
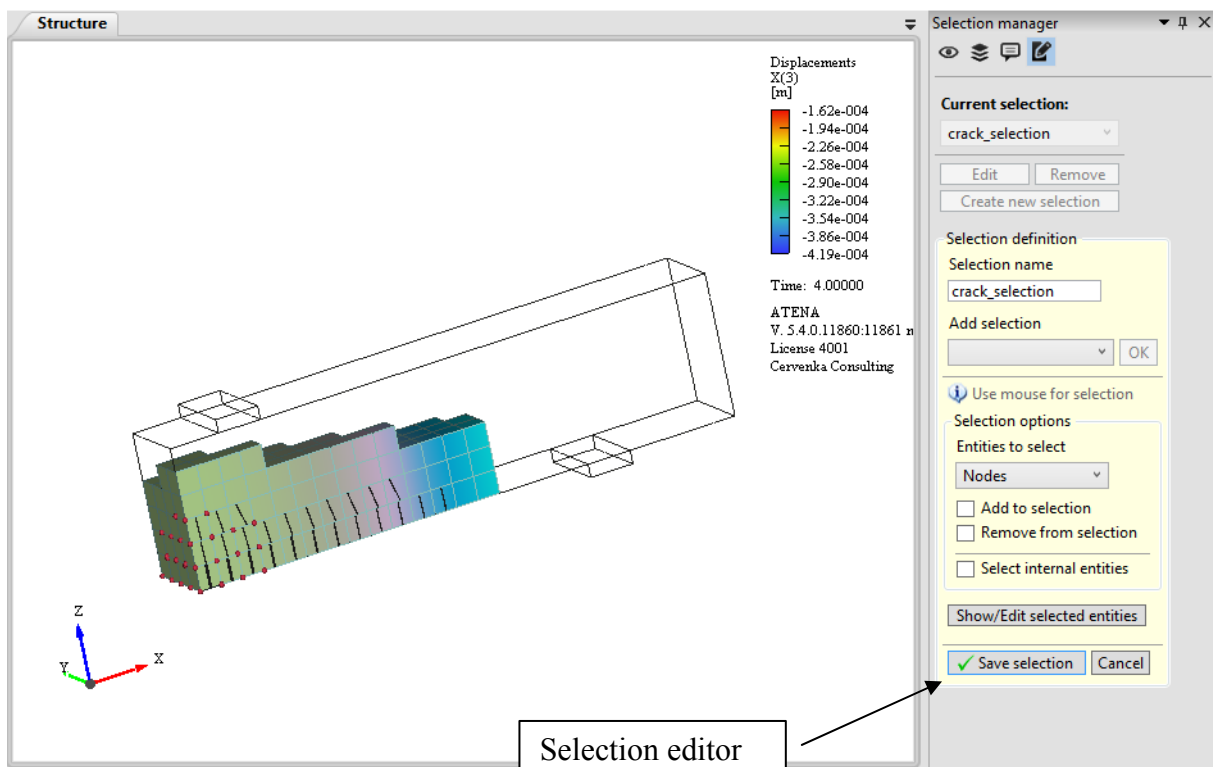
Fig. 3-52 Selection used for activity and outline of undeformed model.

3.7.4.1 Selecting nodes, elements or element IPs

Easy way to select nodes, elements or element IPs is by mouse. One item – just click on it, more items – click and drag encapsulating them into selection rectangle (see Fig. 3-53).
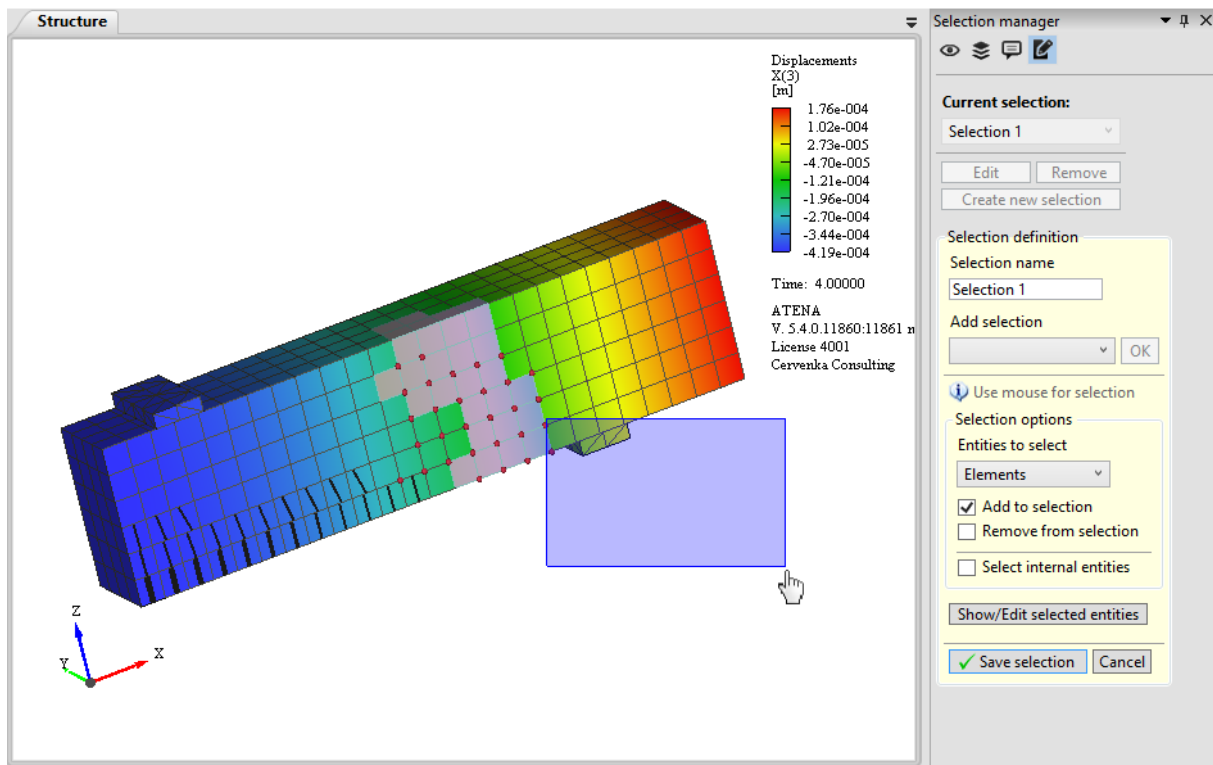
Fig. 3-53 Selecting nodes and elements.

3.7.5 View Manager

Whole or some parts of structure settings are often same as in other structures and some settings can be useful in every structure. For these purposes serves the View manager (see Fig. 3-54). Its main functions are **Create new view**, which creates a view from a current structure and saves in project file, **Apply**, which uses a chosen view on active structure and **Import**, which imports views from selected project file. Every view has own name and a list of options, which will be applied on active structure.

Common structure views, which are offered in every project, have a name surrounded with brackets ("[..]"). They are saved in local application folder with following path, e.g. "C:\Users\$User$\Cervenka_Consulting\Atena\ViewTemplates". There you can add new views or modify current ones.
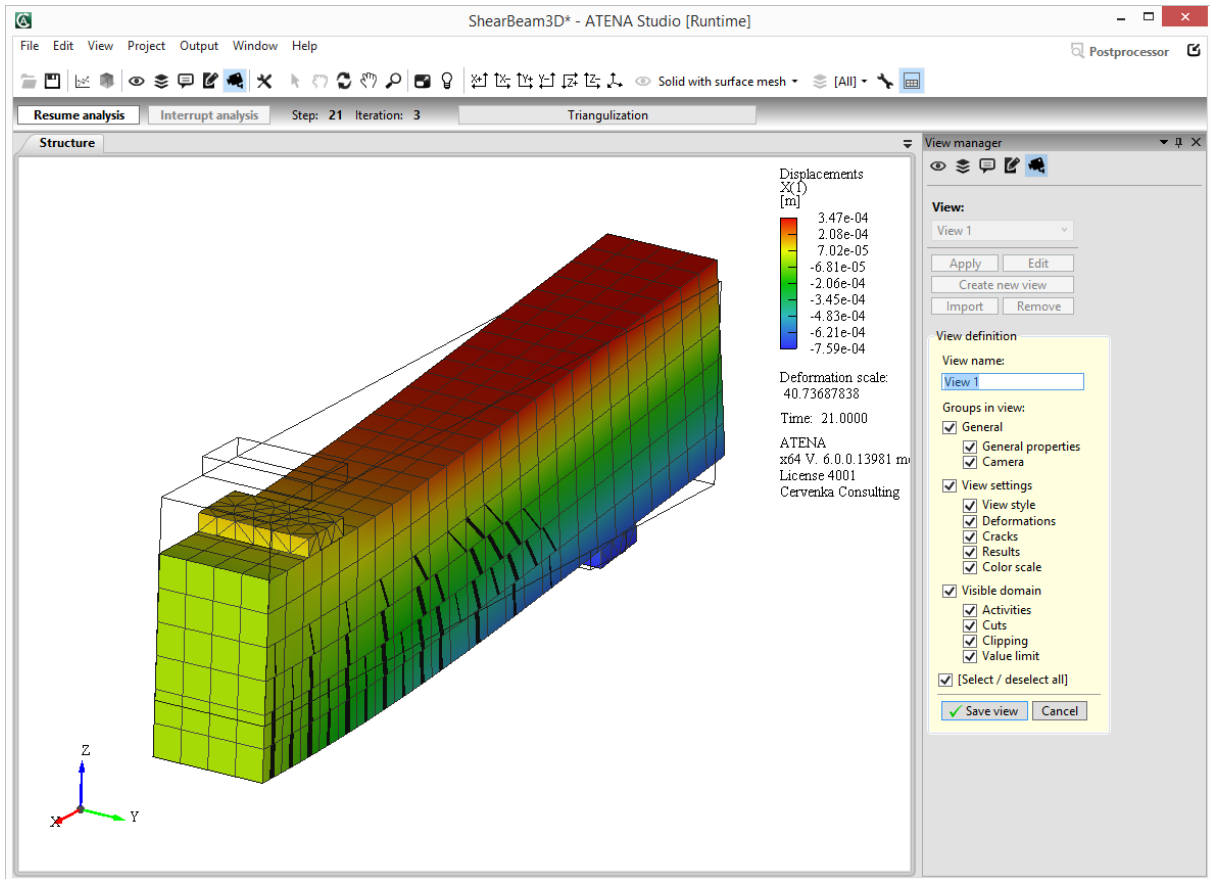
Fig. 3-54 View manager.

3.8 Analysis Steps

Typical ATENA analysis consists of several steps. The completed and saved steps are shown in the Analysis steps window. There are three columns: number of step, state of step and number of iterations. In the post-processor the Analysis steps window has several important additional functions to control displayed results (see 4.8).

Number	State	Iterations
25	Analysed	5
26	Analysed	8
27	Analysed	30
28	... Analysing ...	4

Fig. 3-55 Analysis steps.

3.9 Monitoring Points and Protocols

The information about the progress of the calculation is shown in the section **Protocols**. The information is displayed as it is reported by program. The text contains all messages issued by ATENA during an analysis. If number of lines in this text exceeds the window size, it is possible to move the text using the sliding bar on the right side of window. A type of output is chosen by clicking on appropriate tab. Each protocol window is connected with its source file located in project directory (with the appropriate extension - .msg, .out, .err)

Fig. 3-56 Message text output.

3.9.1 Message

The message text output (see picture above) shows the convergence information about progress of the nonlinear analysis. It is the same information as the one shown in the graphical monitoring of convergence criteria. In addition it shows also progress of line search iterations. Codes in parenthesis at the end of each line indicate to which method the information is related:

(NR) – Newton-Raphson method.

(AL) – Arc length method.

(LS) - Line search method.

Numerical data exceeding convergence criteria are provided by exclamation character (!). Thus converged state is indicated by data without "!".

3.9.2 Output

Output window displays information about global solution parameters, such as matrix size, boundary conditions, etc. And also shows output results printed by Output request command in the post-processor module.

Fig. 3-57 Output text.

3.9.3 Error

The error window includes convergence information supplemented by error and/or warning messages. There are two types of this message: Warnings and errors. Warnings are not fatal and usually only indicate that data need to be checked. Calculation usually proceeds after warnings and error messages are issued. It is a user responsibility to take appropriate actions (change input data and repeat analysis if necessary) in order to prevent error messages.

Fig. 3-58 Error messages.

3.9.4 Monitoring Points

This window shows the current values of convergence monitors (predefined) and monitors defined by user. These values are then used for the definition of diagrams (see 3.5).

Fig. 3-59 Monitoring points values.

4 POST-PROCESSING

4.1 Starting Post-Processor

This chapter describes the usage of ATENA Studio post-processor, which is a part of the ATENA program system. ATENA post-processing can be started from the runtime module after the analysis is completed or it can be directly activated from the start window (see Fig. 2-1) by importing result files. The Post-processor button is located in the tool bar (top-right). Post-processor is almost identical to the Runtime module. Differences and new functions are described in the following text.

4.2 Post-Processing Window

The main tools and functions of the post-processing window are shown in Fig. 4-1.
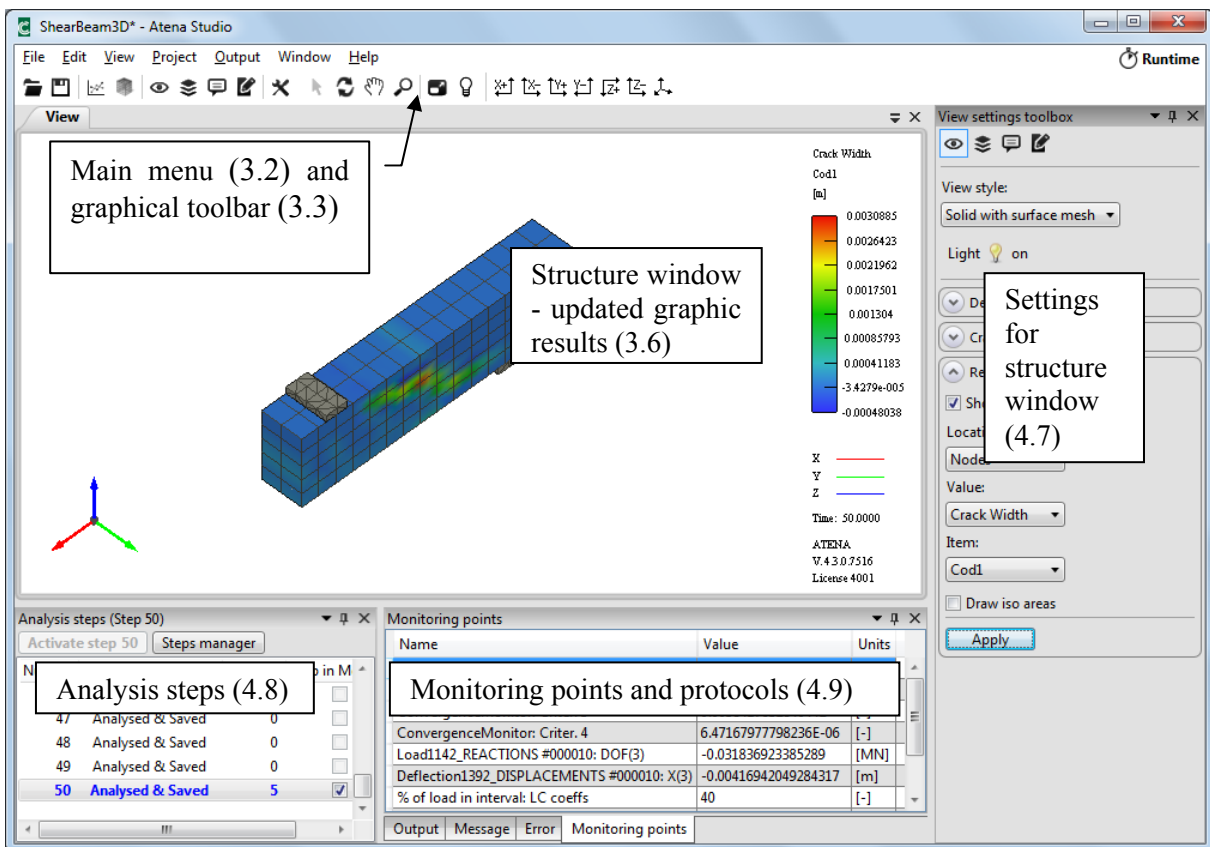
Fig. 4-1 Post-processing window.

The main menu is located at the top of the window. The items of the main menu are modified for the purpose of post-processing. On the right, the button to switch to Runtime module is located. Under the main menu, there are graphic toolbar supporting functions for file and view management. The details are described in section Runtime (3.3).

On the right side of the program screen is a vertically positioned panel controlling the visualization of the ATENA results. Here the user can select *what* and *how* it should be displayed in the structure window (for details see section 4.7).

4.2.1 Cuts – Forces and moments

Creating a cut and then generating more cuts parallel with this reference cut is described in 3.7.2.1.

In Post-processor is added new feature, which enables to integrate forces and moments on the cut plane. In **Structure toolbox** panel under **Cuts** (see 3.7.2.1) is **Forces & Moments** tab and if **Show Forces & Moments** is checked, confirmed by **Apply**, it is displayed an axis cross representing directions of particular forces on cut plane. Near the cross is a table with values of integrated forces and moments (see Fig. 4-2). The table with values can be dragged, if **Drag legend** button is pushed, and axis arrows scaled by slider below.
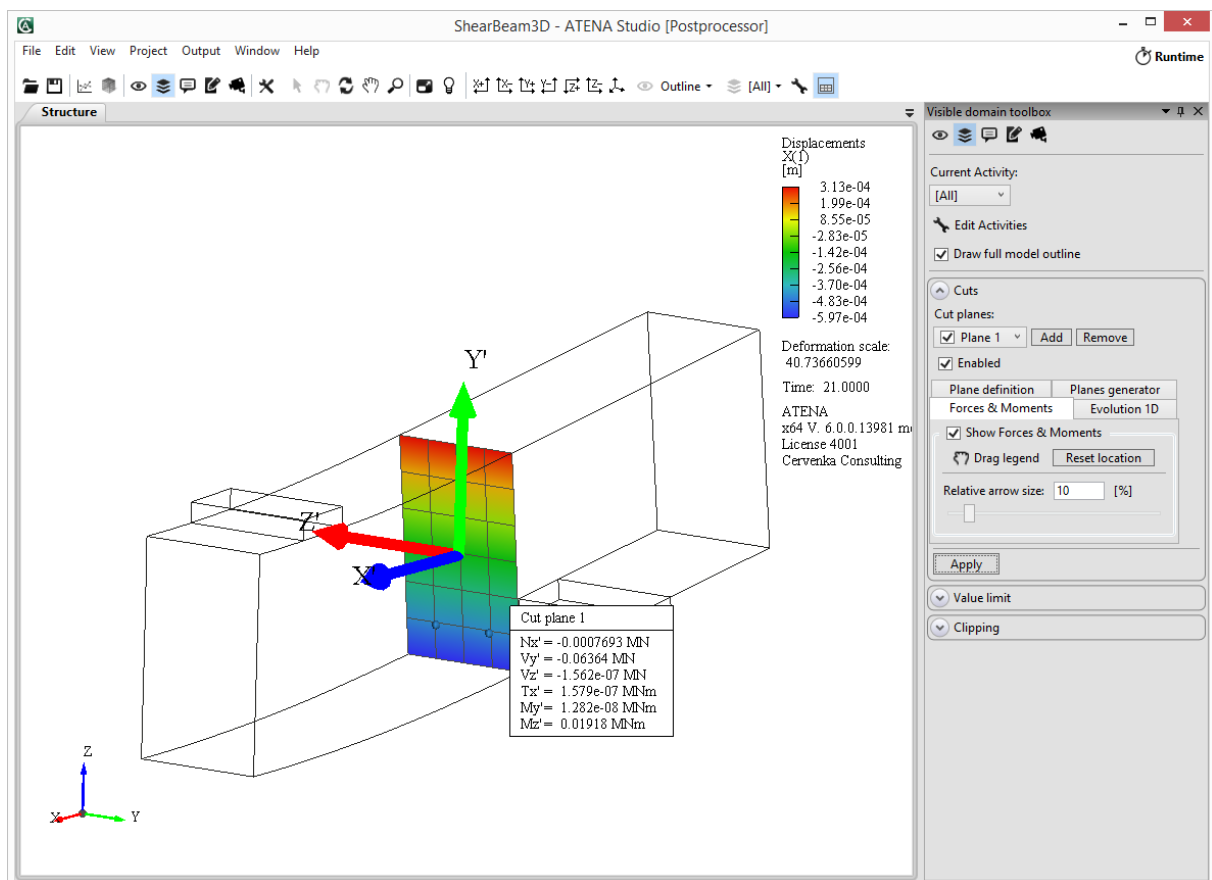
Fig. 4-2 Forces and moments on cut plane.

For more cuts it can be previous way confused and thus is convenient to use evolution 1D**.** In this case it is need to generate more cuts using **Planes Generator** (see 3.7.2.1) and then in section **Evolution 1D** check **Show Evolution 1D,** confirmed by **Apply**, which shows evolution 1D for each force and moment (see Fig. 4-3). Forces and moments can be variously displayed or hidden and also color changed. For later processing values of displayed quantities can be copied to clipboard (in table and csv format).
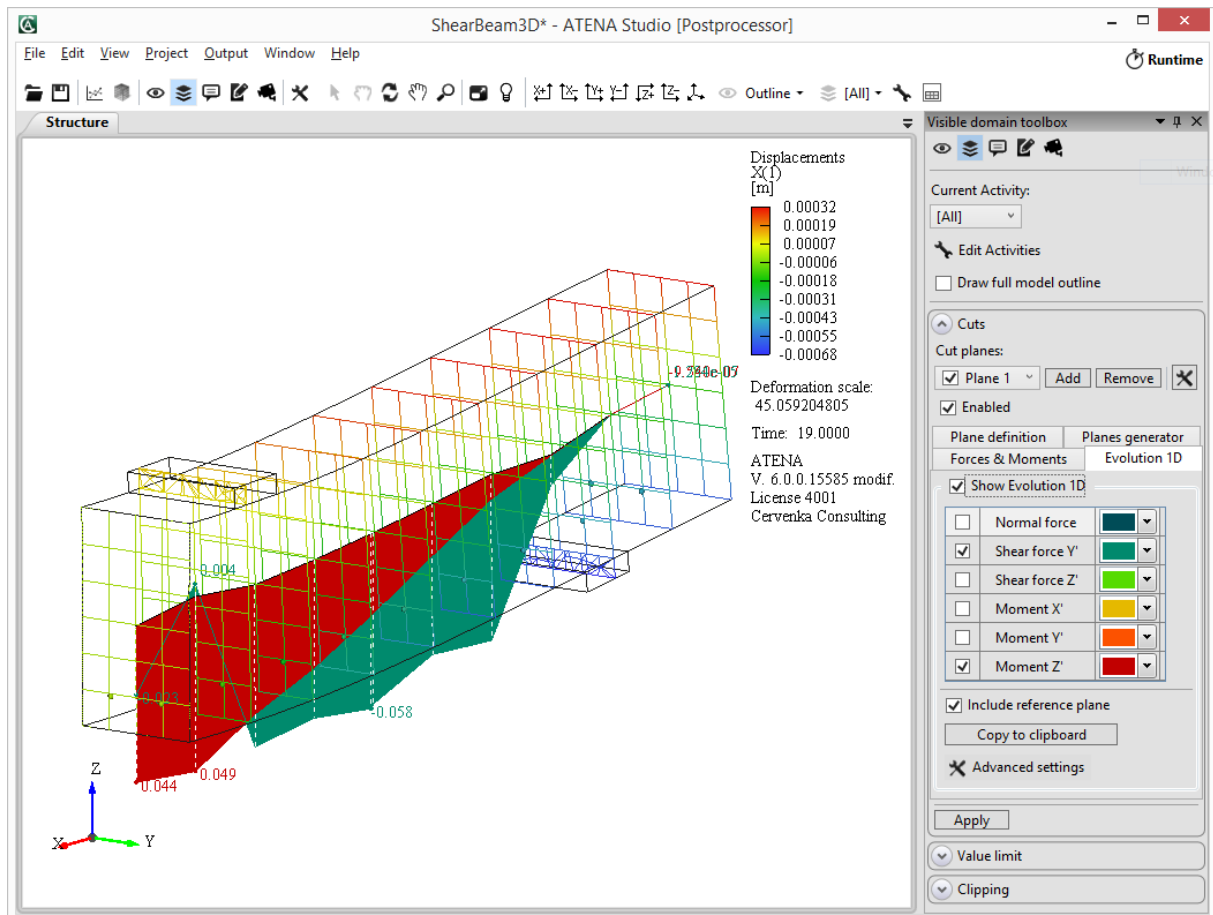
Fig. 4-3 Evolution 1D of shear force and moment on cut planes.

4.3   Main Menu

Functions in menu **File**, **Edit**, **Project**, **Window** and **Help** are the same as in the Runtime. For the detailed description see **Runtime** - Main menu (3.2).

4.3.1 View

Menu View includes one new active item. The rest is same as in Runtime module (see 3.2.3).
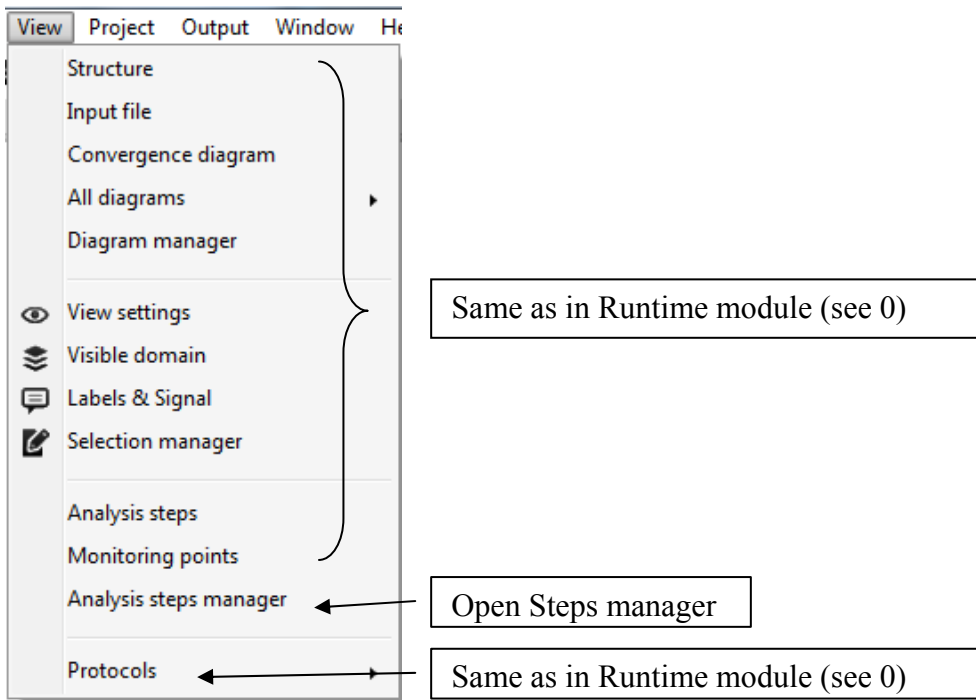
Fig. 4-4 View menu.

Item **Analysis steps manager** will be described later in the "Analysis steps" 4.8.1.

4.3.2 Output
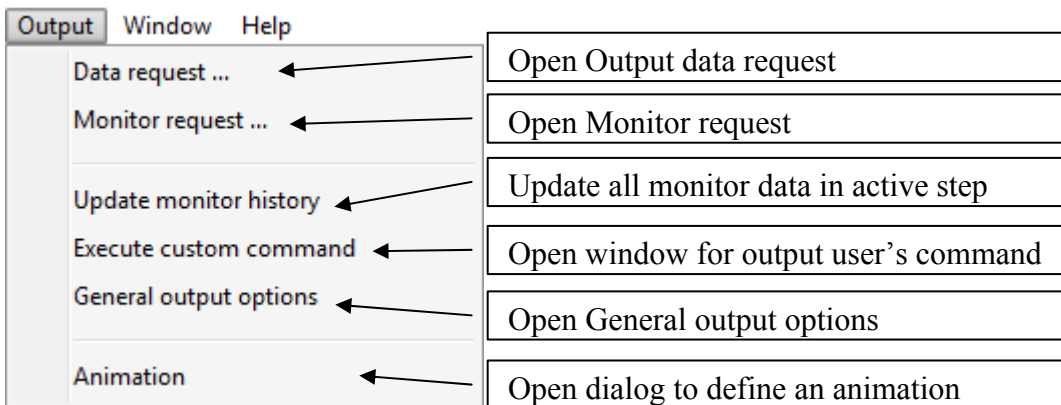
Fig. 4-5 Output menu.

4.3.2.1 Data Request

It should be noted that, as described in section 4.5

Diagram Window, the easiest method to export the data plotted in the active ATENA Studio diagram is via Ctrl+c shortcut or via **Edit | Copy to clipboard**.

Data request is an important command that can be used for textual output of any analysis data. It is possible to list data for the whole structure or only for some of its parts, which are defined by the Selection manager (see 3.7.4). In the first part of this dialog, the data location is selected. This action will change the list of available output quantities at this location (see and the ATENA Input File Format, ATENA Engineering 2D User's Manuals). At the bottom of the dialog in Fig. 4-6, it is possible to limit the output only for certain selections including selections from ATENA Core (for example imported from GiD). On the right side of the dialog, it is possible to specify an item range for output quantities that have more than one item (for instance $\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, \dots$). There are several advanced processing options that can calculate average, or min. max. values for the specified output request.

The second part of the dialog is accessible from the tab Output command, and it shows the actual ATENA commands for the specified output request. These commands could be directly copied to the Custom command dialog (4.3.2.4), edited and executed there.

When the output request is performed (the button Apply) the output data are listed to the Output window, which is located at the bottom of the ATENA Studio window (see Fig. 4-1). If it is closed, the window can be opened by command **View | Protocols | Output window**.
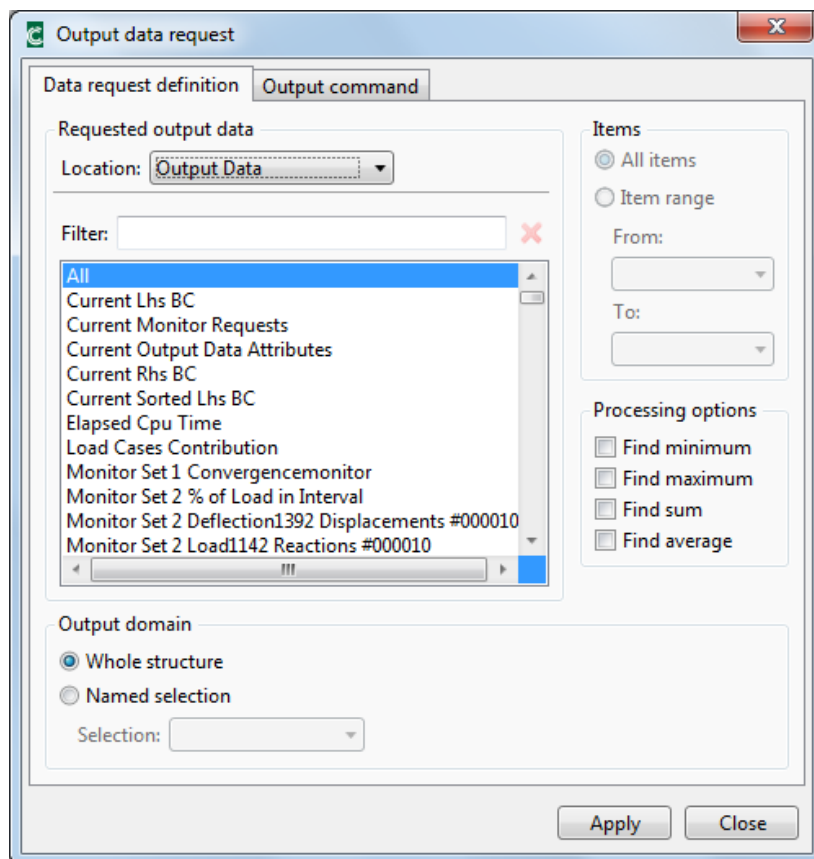
Fig. 4-6 Data request for user defined monitors.

Users often need to print the results at the monitoring points for further processing in a spreadsheet software. The monitoring points are located under the Output Data as shown Fig. 4-6. Their names are starting with the text "Monitoring Set .....".

4.3.2.2 Monitor Request

After the analysis is completed, it is often useful to define new monitoring points. This can be done with the help of the **Monitor request** window, which is almost identical to the **Data request** window (see 4.3.2.1). There is one new tab, **Monitor definition**, which is used for defining the name and type for the new monitor..

First the monitor name and monitor type should be selected. After the analysis is completed, it makes no sense to monitor results each iteration, since these data are not available. In most cases, it is therefore recommended to select **Monitor each step**. The monitor request definition involves two steps. In the first step the monitor is defined, and in the next step, the requested data needs to be extracted from all calculated results. These too steps can be performed at the same time. This is controlled by the checkbox **Import data from previous steps**. This checkbox should be therefore selected only when the goal is to define just one or two new monitors. In case of large problem sizes with many steps, and if it is clear that several new monitors will be needed, it is better to first define all needed monitoring points, and then separately extract their values from the stored binary result files. This can be done by a separate command **Update Monitor History** (see 4.3.2.3).
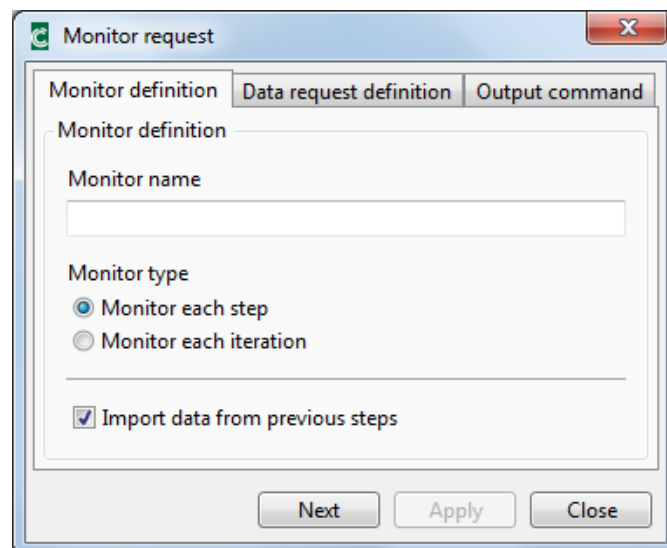
Fig. 4-7 Monitor request.

4.3.2.3 Update Monitor History

This function updates all monitors including the new created monitors. Data will be loaded to new monitors. It can take several minutes. This function should be called when you have created some monitors without checking the option **Import data from previous steps**.

4.3.2.4 Execute Custom Command

This custom command feature can be used by advanced users. A user can define or edit data request alone. This window is same as **Output** command in **Data request** and **Monitor request**.
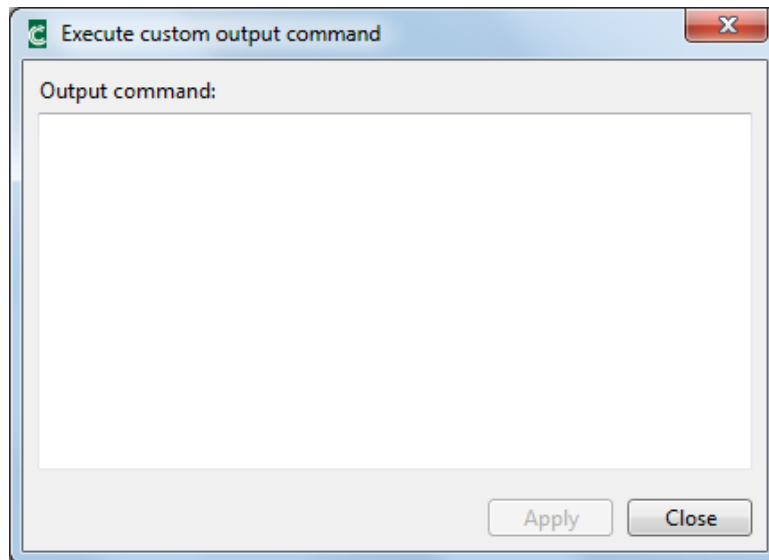
Fig. 4-8 Custom command.

4.3.2.5 General Output Options

This setting is for advanced users. Here it is possible to define output options and recovery strategy for transforming values from element integration points (material points) to element nodes.
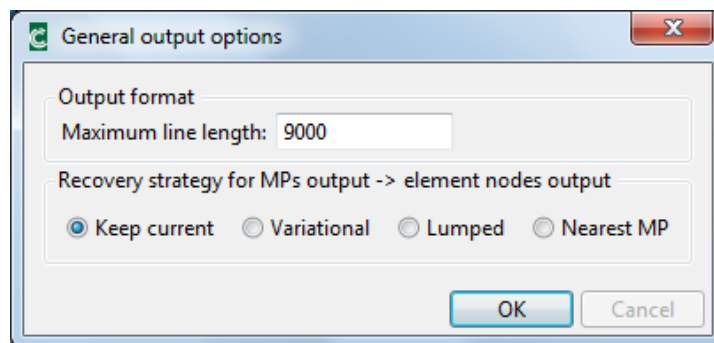
Fig. 4-9 Output options.

4.4 Output Data Attributes

This section deals with the result data, which come out of the finite element analysis. They are displayed in a graphical form on the screen in the post-processing mode and can be also presented in a text form.

See also sections 3.9.2, 4.3.2.1. You can also find additional information about output data attributes in Section 3.9 Output Command of the ATENA Input Manual.

4.4.1 Result Data of Monitored Points

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

4.4.2 Result Data Evaluated in Nodes

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

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

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

		Contents
	Acceleration	Nodal accelerations within dynamic analysis.
Nodes	Beam Forces	Nx, Vy, Vz, Kx, My, Mz beam forces for CCBeam3D element.
		1 – Nx – normal force
		2 – Vy – shear force in local y-direction
		3 – Vz – shear force in local z-direction
		4 – Kx – torsional moment
		5 – My – moment around local y-axis
		6 – Mz - moment around local z-axis
	Bond Slip	Reinforcement displacement relative to surrounding material (bar with bond, external cable).
		Delta Slip – slip increment in the current step
		Total Slip – the current bond slip
	Bond Stress	Stress in the reinforcement contact (bar with bond, external cable).
		1- Force – bond force along the bar per unit length
		2 - Stress – the current bond stress along the bar
	Cable Force	Force in reinforcement.
		1 – Force in the reinforcement bar/cable
	Carbonation	Data about concrete degradation due to carbonation propagation.
		1 – Carbdepth – depth of carbonation from the loaded surface
		2 – Inducttime – induction time for carbonation-induced corrosion at the defined CONCRETE COVER depth

	Chlorides	Data about concrete degradation due to chlorides propagation. 1 – Concentrrf – chloride concentration at the defined CONCRETE COVER depth 2 – Inducttime – induction time for chloride-induced corrosion at the defined CONCRETE COVER depth
	Crack Width	Crack opening displacement (COD) order from the first crack (usually the maximum one) to the last one. 1 – COD1 2 – COD2 3 – COD3
	Current Nodal Coordinates	Coordinates of each node in the current deformed configuration. 1 – x 2 – y 3 – z
	Displacements	Displacements of each node (Current minus Reference Nodal Coordinates). 2D 1 – displacement in x-direction 2 – displacement in y-direction 3D in addition 3 – displacement in z-direction If shell or beam elements are used: 4 – rotation around x-axis 5 – rotation around y-axis 6 – rotation around z-axis
	Eigenvectors x	Structure eigenvectors of the mode x, e.g. EIGENVECTORS_1 to print the 1st eigenvector.
	Elem Humidity Incr	Current element incrementally applied humidity (total from all loads for the current time step).
	Elem Init Strain Incr	Current element initial strain increment (total from all loads for the current time step).
	Elem Init Stress Incr	Current element initial stress increment (total from all loads for the current time step).
	Elem Mass Accel Load Incr	Element load increments due to the element's acceleration, (for a particular step), transformed into nodal concentrated forces.

Elem MP Humid Temper[1]	Humidity and temperature imported from the corresponding transport analysis into creep material history.
Elem Temperature Incr	Current element incrementally applied temperatures (total from all loads for the current time step).
Elem Total Humidity	Total humidity. 1 – Totalhumid. – current relative humidity
Elem Total Temperature	Thermal load. 1 – Totaltemp. – current temperature
Element Crack Volume	Coordinates of shell's volume with cracks
Element Orientation	Element orientation for bricks, Ahmad and beam elements. Useful especially for checking reference depth vectors of shells and beams.
Eps Mi	Value of internal creep variables.
Eq Plastic Strain	Equivalent plastic strain for the Cementitious fracture-plastic material
(Compact) External Forces	External forces acting in nodes. See also Partial External and Internal Forces. In older **ATENA** versions this output type was called Compact External Forces.
Fc Reduction	Reduction of compressive strength due to concrete cracking in Cementitious fracture-plastic models
Fracture Strain	Strain corresponding to crack opening.
Imperfections	Incremental values of imperfect structural geometry (with regards to its reference coordinates).
Integ Stress	Cross sectional forces and moments for bended elements
(Compact) Internal Forces	Nodal forces. Summation of forces includes all elements regardless of the group. See also Partial Internal Forces and Residual Forces. In older **ATENA** versions this output data type was labeled Compact Internal Forces.

---

[1] Supported since version 5.8.0

	Interface Displacements	Relative displacements at contact elements. The components depend on problem dimension:
		2D
		1 – Dv(1) – shear displacement
		2 – Du – normal displacement
		3D
		1,2 – Dv(1,2) – shear displacements
		3 – Du – normal displacement
	Interface Plast Displacements	Nonlinear part of the relative displacements at contact elements. The components depend on problem dimension:
		2D
		1 – Dv P(1) – shear displacement
		2 – Du P – normal displacement
		3D
		1,2 – Dv P(1,2) – shear displacements
		3 – Du P – normal displacement
	Interface Stress	Stress on contact elements. The components depend on problem dimension:
		2D
		1 – SigT(1) – shear stress
		2 – SigN – normal stress
		3D
		1,2 – SigT(1,2) – shear stresses
		3 – SigN – normal stress
	Maximal Fract Strain	The maximum fracture strain during the loading history so far.
		1 – Eps_ft(1) – fracturing strain corresponding to the first crack (see Crack width COD1)
		2 – Eps_ft(2) – fracturing strain corresponding to the second crack (see Crack width COD2)
		3 – Eps_ft(3) – fracturing strain corresponding to the third crack (see Crack width COD3)

	M n q Beam	Integral values for 1D and 3D Beam elements, in local element coordinates (X corresponds to the beam axis). The components:
		1 – NX – normal axial force
		2 – Qxy – shear force in the XY plane
		3 – Qxz – shear force in the XZ plane
		4 – Tyz – torsional force in the YZ plane
		5 – MY – moment around the Y axis
		6 – MZ – moment around the Z axis
		For more information, see ATENA Theory, 3.21 Integrated forces and moments for beams.
	M n q Shell	Integral values for 2D and 3D Shell elements, in local element coordinates (Z corresponds to the thickness direction). The components:
		1 – NX – normal force in X direction
		2 – NY – normal force in Y direction
		3 – NZ – normal force in thickness direction
		4 – Qxz – shear force in the XZ plane
		5 – Qxy – shear force in the XY plane (in-plane shear force)
		6 – Qyz – shear force in the YZ plane
		7 – MY – moment around the Y axis
		8 – MX – moment around the X axis
		9 – Txy – torsional force in the XY plane
		For more information, see ATENA Theory, 3.20 Integrated forces and moments for shells.
	Nodal Degrees Of Freedom	A location in the global stiffness matrix.
		This list relates original nodal numbers to the DoF numbers in the global stiffness matrix line numbering. DOF(1), DOF(2), and DOF(3) are degrees of freedom in X, Y, and Z directions, respectively.
		A minus signs in front of a DOF number indicates a DoF with kinematic constraints, i.e., BC. They include supports and master-slave relations.
	Partial External Forces	External forces acting in nodes. The external forces are due to given loading.

	Partial Internal Forces	Nodal forces. Summation of forces includes only elements of the same group. The sums of nodal force in nodes of adjacent elements are shown. In nodes without external loading and kinematic constrains, such as supports and slaves, in an equilibrium state these values should be near to zero within the tolerance specified by convergence criteria.
	Partial Reactions	Reactions in nodes. Partial reactions are listed in all nodes with kinematic constrains. The kinematic constrains can be of two types: (a) fixed supports or prescribed displacements in nodes, (b) slave nodes. Since the slave nodes are kinematicaly related to the master nodes, they are regarded as supports to master nodes and thus provide reaction forces to master nodes. This occurs, for example in nodes on the border of element groups and in reinforcing bars. Partial Reactions are treated in a way similar to Partial External Forces (see above).
	Partial Residual Forces	Residual forces in nodes (sum of external and internal forces acting in node). The residual force shows the unbalanced force in nodes within element groups. Partial residual forces describe equilibrium state. They are calculated as a sum of internal forces, external forces and reactions in a node under consideration and within the element group only. In an equilibrium state the residual forces should be zero. Any nonzero residual forces indicate an error in the non-linear solution (apart from unbalanced forces occurring on the group border).
	Performance Index	Relative error in the evaluation of the material model at each node. 1)
	Plastic Strain	Plastic strain of concrete, steel, or bar reinforcement in nodes. Plastic strains are calculated only in models based on the theory of plasticity. Examples of such materials are: 3D Cementitious2, Von Mises, Drucker-Prager, reinforcement. Therefore, in some materials (e.g., Elastic) plastic strains are not presented at all. In 2D or 3D material models 3-6 plastic strain components can appear. See (Engineering) Strain below for details about the components.
	Plastic Strain (s*j)*	Plastic strain of smeared reinforcement. 2)

	Principal (Engineering) Strain	Principal strains and a direction vector in nodes.
		Max, Mid, Min – the principal values of the strain tensor,
		(Vmax X, Vmax_Y, Vmax_Z),
		(Vmid X, Vmid_Y, Vmid_Z),
		(Vmin X, Vmin_Y, Vmin_Z) – the corresponding direction vectors. For details see the documentation *ATENA Theory, section 2.1.1 Basic assumptions*.
	Principal Fracture Strain	Principal components of strain corresponding to crack opening.
	Principal Plastic Strain	Principal plastic strain of concrete, steel, or bar reinforcement in nodes.
		Number of principal components depends on the material model. For example, the '3D Nonlinear Cementitious2' can have all three principal components (See also Plastic Strain above). The principal directions are described by unit vector components (See also Principal Engineering Strain above).
		In case of reinforcement in a uniaxial stress state, only one component is shown. This strain is in a local direction of the truss element. Furthermore, the principal and normal values of stresses and strains in reinforcement are identical.
		1 – Max.
		2 – Vmax X
	Principal Shell Membrane Strain	The principal components of the strains averaged through the shell thickness (see also Shell Membrane Strain below).
	Principal Strain (s*j)*	Principal engineering strain of smeared reinforcement. 2)
	Principal Stress	Principal stresses in nodes.
		Max, Mid, Min – the principal values of the stress tensor,
		(Vmax X, Vmax_Y, Vmax_Z),
		(Vmid X, Vmid_Y, Vmid_Z),
		(Vmin X, Vmin_Y, Vmin_Z) – the corresponding direction vectors. For details see the documentation *ATENA Theory, section 2.1.1 Basic assumptions*.
	Principal Stress (s*j)*	Principal stress of smeared reinforcement. 2)
	Principal Total Strain	Principal components of strains including initial and thermal strains.

	Rate factor	Not used, equal to 1.0
	Rate factors	Not used, equal to 1.0
	(Compact) Reactions	Reactions in nodes. Reactions are listed in all nodes with kinematic constrains. In reactions only external kinematic constrains such as supports and prescribed displacements are considered. Reactions are treated in a way similar to External Forces (see above). If a reaction is in a double-node the value in a master node is shown. In older **ATENA** versions, this data type used to be called Compact Reactions.
	Reference Border Coordinate	Coordinates for drawing 2D plots showing evolution of certain quantity along the path given by these values
	Reference Nodal Coordinates	Initial nodal coordinates (original nodal coordinates in the undeformed state).
	Reinf Corrosion	Data about reinforcement degradation due to chlorides and carbonation. The portion of corroded cross-section area defined as: $1 - \dfrac{A_{remain}}{A_{initial}}$, where A_{remain} and $A_{initial}$ are the remaining and initial cross-section areas, respectively. 1 – Total – total reinforcement corrosion 2 – Chlorides – corrosion due to chloride ingress 3 – Carbonat. – corrosion due to carbonation
	(Compact) Residual Forces	Residual forces in nodes (sum of external and internal forces acting in node). The residual force shows a global unbalanced force in the node, summed for all elements adjacent to the node. Residual forces describe equilibrium state. In an equilibrium state the residual forces should be zero. Any nonzero residual forces indicate an error in the non-linear solution.
	Soft/Hard parameter	Hardening/softening parameter for the plastic model in Cementitious fracture-plastic materials. It corresponds to the ratio of current compressive strength to the one specified in the material definition: $f_c\left(\varepsilon_{eq}^p\right)\Big/ f_c$

	Sbeta State Variables	State variables of 'SBETA Material' in each node. They contain a complete set of data available in the model and include some data available in other sections (principal stress and strain) as well as additional data (tension stiffening, shear retention factor, etc.). Units of SBETA variables are not shown. They have units of stress and strain. 1)
	Shell Membrane Strain	Strains averaged over the shell thickness.
	Spring Strain	Relative deformation of the spring. There is only one component, in the spring direction: 1 – Du
	Spring Stress	There is only one component, in the spring direction: 1 – Sigma
	(Engineering) Strain	Strains (ε, γ), sometimes also called relative deformations. The components and their numbering depend on model and element types. *3D Element* – in global coordinates: 1 – EpsXX, 2 – EpsYY, 3 – EpsZZ, 4 – GammaXY, 5 – GammaYZ, 6 – GammaXZ *2D Plane Stress*: 1 – EpsXX, 2 – EpsYY, 3 - GammaXY *2D Plane Strain*: 1 – EpsXX, 2 – EpsYY, 3 – EpsZZ=0, 4 – GammaXY *Axisymmetry*: 1 – EpsRr, 2 – EpsZZ, 3 – EpsTt, 3 – GammaRz, 4- GammaZT, 5 – GammaRt *Shell, Beam* – in local coordinates: 1 – EpsXX, 2 – EpsYY, 3 – EpsZZ, 4 – GammaXY, 5 – GammaYZ, 6 – GammaXZ *1D Reinforcement* – axial strain: 1 – EpsXX *2D Membrane* – in local coordinates: 1 – EpsXX, 2 – EpsYY, 3 - GammaXY
	Strain (s*j)*	Engineering strain of smeared reinforcement. 2)

	Stress	Stress components (σ, τ).
		The components and their numbering depend on model and element types.
		3D Element – in global coordinates:
		1 – SigXX, 2 – SigYY, 3 – SigZZ,
		4 – TauXY, 5 – TauYZ, 6 – TauXZ
		2D Plane Stress:
		1 – SigXX, 2 – SigYY,
		3 – TauXY
		2D Plane Strain:
		1 – SigXX, 2 – SigYY, 3 – SigZZ,
		4 - TauXZ
		Axisymmetry:
		1 – SigRr, 2 – SigZZ, 3 – SigTt,
		3 – TauRz, 4- TauZT, 5 – TauRt
		Shell, Beam – in local coordinates:
		1 – SigXX, 2 – SigYY, 3 – SigZZ,
		4 – TauXY, 5 – TauYZ, 6 – TauXZ
		1D Reinforcement – axial stress:
		1 – SigXX
		2D Membrane – in local coordinates:
		1 – SigXX, 2 – SigYY,
		3 – TauXY
	Stress (s*j)*	Stress of smeared reinforcement. 2)
	Tensile Strength	The current tensile strengths (in principal directions).
		1 – Sig_T(1) – current tensile strength in the material direction 1, i.e. corresponding to the direction of the first crack
		2 – Sig_T(2) – current tensile strength in the material direction 2, i.e. corresponding to the direction of the second crack
		3 – Sig_T(3) – current tensile strength in the material direction 3, i.e. corresponding to the direction of the third crack
		4 – Tau_T(1,2) – shear strength on the local material plane 1,2
		5 – Tau_T(2,3) – shear strength on the local material plane 2,3
		6 – Tau_T(3,1) – shear strength on the local material plane 3,1
	Total Elem Body Load	Typically used for dead weight.

Total Elem Init Strain	Initial strains (typically used to represent pre-stressing or shrinkage).
Total Elem Init Stress	Initial stresses (typically used to represent pre-stressing).
Total Strain	Strains including initial and thermal strains. See Strain above for details about the components.
Velocities	Nodal velocities within dynamic analysis.
Von Mises Stress	Equivalent plastic stress for the von Mises yield criteria.

Notes:

1) Displayed only if 'SBETA Material' is used. The variables are explained in the following table (see the ATENA 2D User's manual for details.):

SBETA State Variables

Variable	Contents
Node	Node number.
Y1	Secant modulus in direction 1 (max).
Y2	Secant modulus in direction 2 (min).
Rc1	Biaxial compressive strength in direction 1 at failure. See Theory 2.1.5.1.
Rc2	Biaxial compressive strength in direction 2 at failure. See Theory 2.1.5.1.
Rt1	Biaxial tensile strength in direction 1 at failure. See Theory 2.1.5.2.
Rt2	Biaxial tensile strength in direction 2 at failure. See Theory 2.1.5.2.
s1	Concrete stress in direction 1.
s2	Concrete stress in direction 2.
Rec	Compressive strength factor due to the biaxial stress. See Theory 2.1.5.1.
Ret	Tensile strength factor due to the biaxial stress. See Theory 2.1.5.2.
Sc(1)	Concrete stress normal component in dir. x, σ_{cx}.
Sc(2)	Concrete stress normal component in dir. y, σ_{cy}.
Sc(3)	Concrete stress shear component in dir. xy, τ_{cxy}.
Eps(1)	Strain component, normal, in dir. x, ε_x.
Eps(2)	Strain component, normal, in dir. x, ε_x.
Eps(2)	Strain component, shear, in dir. xy, γ_{xy}.
Sd(1)	Smeared reinf. stress. NOT USED in this version.
Sd(2)	Smeared reinf. stress. NOT USED in this version.
st(1)	Tension stiffening stress. NOT USED in this version.
st(2)	Tension stiffening stress. NOT USED in this version.
Sx(1)	Total stress component, normal, in dir. x, σ_x. (Here same as σ_{cx})
Sx(2)	Total stress component, normal, in dir. y, σ_y. (Here same as σ_{cy})
Sx(3)	Total stress component, normal, in dir. xy, τ_{xy}. (Here same as τ_{cxy})

Yd(1)   Smeared reinf. modulus. NOT USED here.

Yd(2)   Smeared reinf. modulus. NOT USED here.

Angle   Angle of principal strain or material axis 1.

Tevol   Volumetric strain due to temperature or shrinkage.

Redg   Shear retention factor.

Rsm(1)   Yield stress in smeared reinforcement. NOT USED here.

Rsm(2)   Yield stress in smeared reinforcement. NOT USED here.

Ns(1)   Material state number in dir. 1. (See Theory, 2.1.2.1)

Ns(2)   Material state number in dir. 2. (See Theory, 2.1.2.1)

Ncr   Crack state number.

2) The index 'j' stands for layer number. See 4.4.8 for more information.

4.4.3 Result Data Evaluated in Elements

		Contents
Elements	ASR Ksi	ASR load data, such as ASR extend and reduction coefficients for material performance characteristics; Young modulus (E), tension strength (f_t), compressive strength (f_c), fracture energy (G_f). 1 – Ksi – extend (degree) of ASR reaction 2 – Prevecoeff – value of E/E_0 at the previous step 3 – Curecoeff – value of E/E_0 at the current step 4 – Preftcoeff –value of f_t/f_{t0} tension strength 5 – Curftcoeff – value of f_t/f_{t0} at the current step 6 – Prefccoeff – value of f_c/f_{c0} at the previous step 7 – Curfccoeff – value of f_c/f_{c0} at the current step 8 – Prevgfcoeff – value of G_f/G_{f0} at the previous step 9 – Curgfcoeff – value of G_f/G_{f0} at the current step 10 – Aver.Epsv – average ASR strain
	Beam NL Midpoint Params	Several parameters describing element state/conditions for CCBeam3D element at its middle point, (only for beam with a material derived from CCBeamBaseMaterial).
	Element Age[2]	Element age in the digital printing of the structure

---

[2] Not available in ATENA version 5.7.0 and older

	Crack Attributes	Crack attributes of elements. The crack attributes define the crack direction and length within the element, crack opening and stress components, normal and shear, on the crack plane. The attribute values are average values of all integration points.
		Group - Element group (macroelement number)
		Elem. - Element number
		N.Cracks - Number of cracks in element
		Dim - Dimensionality of the model
		N.attr - Number of crack attributes
		Ni (1,2,3) - Crack normal direction unit vector components (i=1,...N.Cracks)
		Size i - Crack length
		CODi - Crack opening displacement – crack width
		Sig_n i - Normal stress on the crack plane
		Sig_t i - Shear stress on the crack plane
	Element Incidences	Node numbers belonging to each element.
		Group - Element group (macroelement number)
		Elem. - Element number
		Type - Element type number
		Geom. - Geometry type number, normally equal to Element group (macroelement number)
		Mater. - Material number
		No. # - Element node number
	Element Material Types	The material types used for each element.
	Element Output Options	Output-related settings for each element (activity, interpolation, etc.)

4.4.4 Result Data Evaluated in Element Integration Points

These data have the same type of information as in the nodes, see sections 4.4.5 and 4.4.2. It should be realized, that the integration points are the 'true' locations, where the response of material model is evaluated (except for true nodal quantities like Displacements, Reference/Current Nodal Coordinates, Bond Slip, Compact/Partial Internal/External/Residual Forces, Reactions). The material state variables displayed in other locations, such as element nodes, can be somewhat changed due to extrapolation.

		Contents
Element IPs	Cracking Moduli	Cracking moduli $\mathbf{E}^{'cr}$ of the Cementitious fracture-plastic material.
	Direction Status	Status variable in the Cementitious fracture-plastic materials indicating material fracture state: 1-virgin, 2-crack, 4-fully cracked.
	Displacements At Ips	Displacements of the integration points
	IP Coordinates	Coordinates of the integration points.
	Material Transformation Matrix	Transformation matrix used in Cementitious fracture-plastic materials for transformation from global to local coordinate system of the cracked material
	Reference Ip Coordinates	Initial coordinates of the integration points.
	Yield/Crush info	Status variable in the Cementitious fracture-plastic material indicating crushing state: 1-virgin, 2-plastic hardening, 4-softening

4.4.5 Result Data Evaluated in Element Nodes

These data have the same type of information as in integration points, see section 4.4.4. Their values are extrapolated from integration points to element nodal points. The values of material variables (stresses, strains) in the same node, but in different element, can be different. (However, in mesh nodes, sect. 4.4.2, only one average value is shown in nodes belonging to the same element group.)

4.4.6 Global Data

		Contents
Global	Arc-Length Params	Arc Length parameters
	Convergence Criteria	Convergence status.
	Eigenvalue Characteristics	Parameters used for the eigenvalue solution
	FE model Characteristics	Numbers of: nodes, groups, element types, materials, dimensions.
	Line Search Params.	Line Search method parameters.
	Solution Characteristics	Parameters of used solution methods.

	Step Convergence	Convergence status at step end.
	Step ID	Load step number.
	Task Name	Task name.
	Task Title	Title.

4.4.7 Load Cases Data

		Contents
Load cases	Load Slave Nodes	List of load slave nodes (loaded nodes).
	Master Slave Nodes	List of Master-Slave constraints
	Support Master Nodes	List of support master nodes. See Support Slave Nodes above.
	Support Slave Nodes	List of support slave nodes. Additional information about boundary conditions with details about master-slave relations, useful for cases of more complex kinematic constrains.

4.4.8 Element Groups and Smeared Reinforcement

The data are grouped according to their locations in a basic group and smeared reinforcement layers. In this classification the basic group includes elements with the basic material (for example Cementitious2, SBETA, Von Mises, etc.) and discrete reinforcement. For example, if one checks a data box for results in 'stress', this makes a request for data of all concrete elements and all truss elements of reinforcing bars. The stress data will include three stress components for concrete and only one stress component for bars.

The additional layers include smeared reinforcement. Each Macroelement can contain one or more smeared reinforcement layers. Description of the data box is composed of the data name (e.g., Stress) and the layer number (e.g., 2). Example: **Stress (s2)** means that request is made for stress in the 2nd layer of smeared reinforcement.

For smeared reinforcement (and also for discrete reinforcement bars), the values are in a local direction of the reinforcement. As the reinforcement material is in an uniaxial stress state, the principal strains (and stresses) are identical with strains (stresses).

4.5 Diagram Window

A new diagram can be added by the icon , which opens the window for definition of the new diagram. The diagram window in Post-processor has same functions as in the Runtime module, for detail see 3.5. When switching between runtime and post-processor module, basic layout is loaded and defined diagrams are closed. But they are not lost. User-defined diagram can be opened any time using menu **View | Diagram** or using the **Diagram manager**. The diagrams are also saved into the project file using the command **Save project** from the main menu.

The diagram can be copied to clipboard and inserted to spreadsheet as values or to image editor as a picture using function **Copy to clipboard** from the menu **Edit**.

4.6 Structure Window

The structure window shows results such as displacement, crack width, stresses, strains etc. It is possible to opened several Structure windows, and show different results in each of them. However, they all have to be from the same task and from the same step. The step number can be changed in Analysis steps window as described later in (4.8). The available options for the **Structure** window are described in more detail in Section 3.6.

The structure can be also copied to clipboard and inserted into other programs as a picture.

4.7 Settings for Structure Window

This dialog is used for the definition of various visualization options, and is identical to the Runtime module. For detailed description see 3.6.1.

4.8 Analysis Steps

The step whose results should be shown in the **Structure** window can be chosen in the **Analysis steps** window. The step must be activated. There are two ways how activate the step: (1) select step number and click the button **Activate step**; (2) click right mouse button and tick **Activate step on select immediately**. In this latter case, the step is immediately loaded into the memory and activated when it is selected with the left mouse button. The appearance for the both modes is slightly different as shown in the figure below.
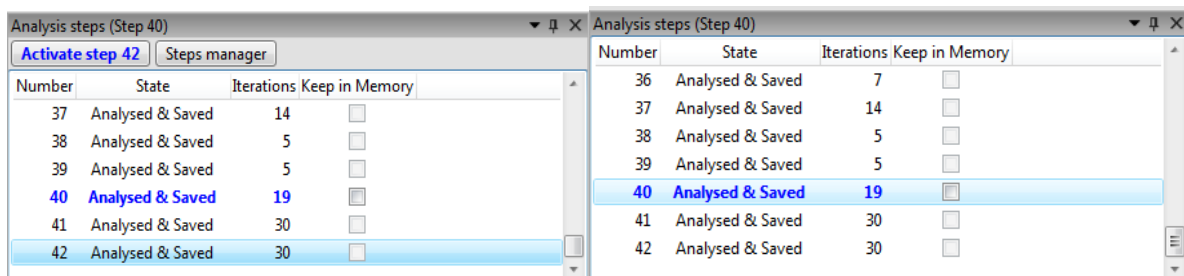

Fig. 4-10 Two states of Analysis steps window.

After selecting the step, results change. If the checkbox **Keep in memory** is enabled, the switching between various steps will be faster, but it will require more memory and is not suitable for very large tasks.

4.8.1 Steps Manager

The **Steps manager** can be opened using button **Steps manager** or right mouse button. Steps can be activated or deleted in the **Steps manager**. It is possible to save the last step which is in the memory. **Import results** can be used when the steps are saved in several directories. **Scan for results** finds result files in the active directory, which belong to the same run and are missing in the current project. For example: a structure is analyzed by some other program and **ATENA Studio** is used only for post-processing. The result steps are updated by this button. From the selected steps can be created an animation or a report.
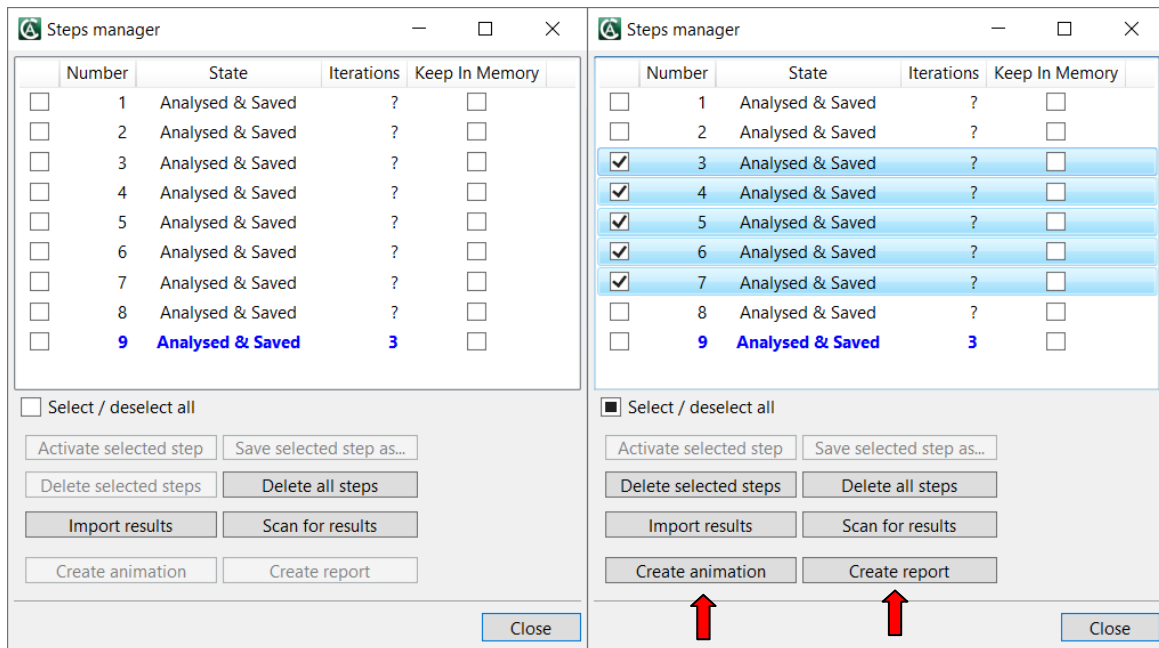
Fig. 4-11 Steps manager.

4.8.2 Animation

In post-processing, it is often requested to produce a sequence of pictures, such as animation showing the crack propagation for a series of load steps. In this case, it is often preferable to keep all view settings constant and change only the load step results. For this we activate first the load step with the most representative range of data (usually the highest load step). In this step we define all settings according to our needs. Then in the **Animation dialog** is the section **Fixes in active step,** where we can take some or all settings from active step and use for all processed steps (i.e same color scale for all frames..). Usually we have checked all settings. To better demonstrate process of load steps it can be add one or more **Diagram displays**. Each diagram display contains a **diagram**, which is selected from a list of available diagrams which corresponds to opened diagram windows. Here can be directly adjusted basic diagram appearance such as font size, font weight and others. During animation the diagram can have one of three display forms called **Display type**. **Normal**, where displayed diagram is same as diagram in processed step, i.e. has same limits. The type **Growing** means, that diagram limits are taken from an active step and the diagram gradually grows. And finally **Moving point,** where the displayed diagram is same as a diagram from the active step, but the currently processed step is highlighted by a circle. The **Location** of the diagram can be

inside or outside the structure window. If two diagrams have same location, combobox **Flow direction** is enabled, where user can decide, in which direction the second diagram will be placed after the first one. Finally each diagram display has own resolution where its ratio can be locked. Before creating animation is useful show first and last frame whether diagram is well placed relative to the construction. If it's not convient, there is a **Structure camera** section, where the camera can be moved by arrows or zoomed by a slider. To faster control smaller structures it can be checked **Automatic update** under frame preview, which allows updating the frame preview after any user change. After all, a new animation can be created, saved and shown later in media player that supports the format .avi.
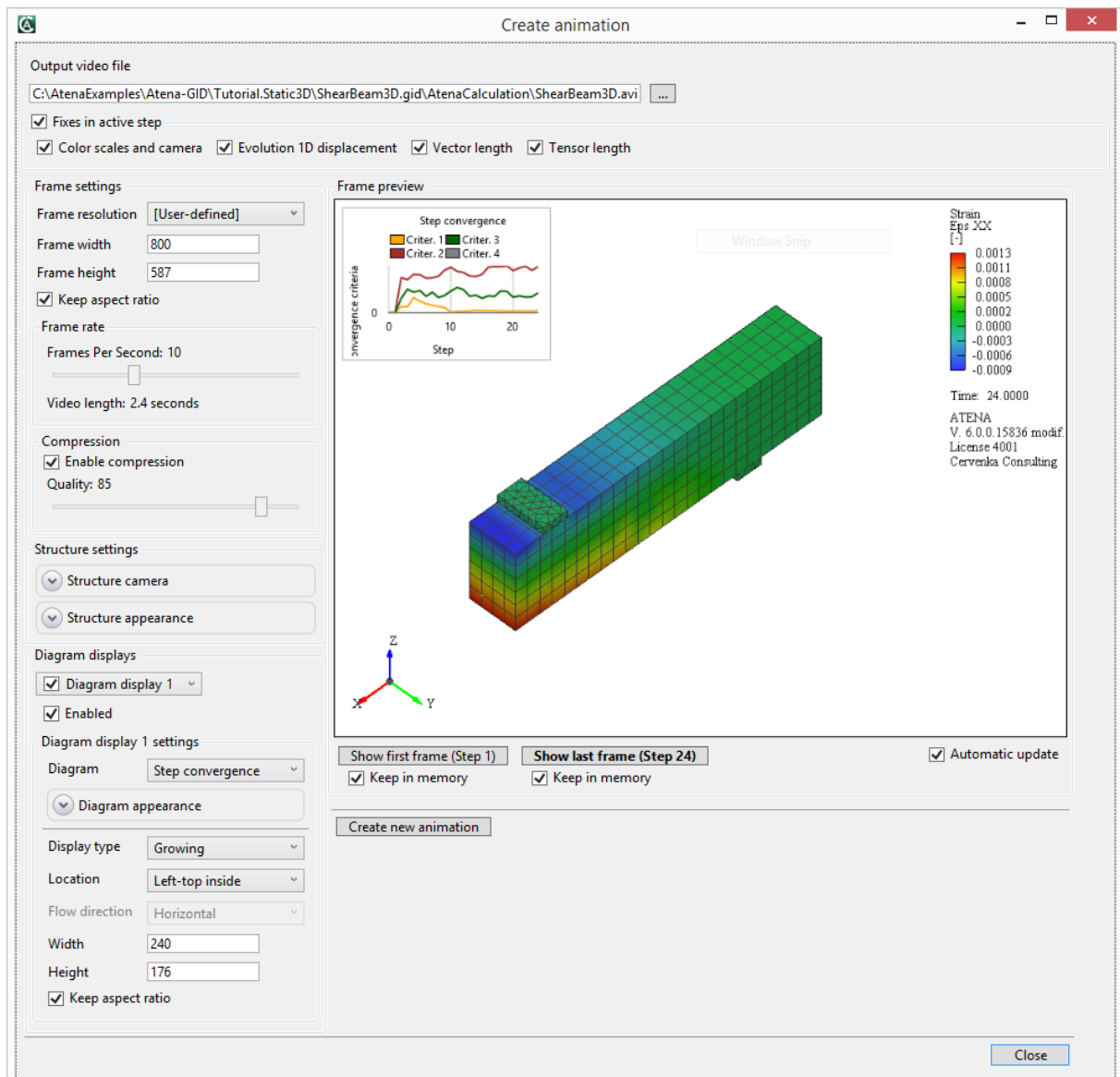
Fig. 4-12 Creating animation.

4.8.3 Report

An alternative way to create output of your execution is generate a report from selected steps. The report is saved in the **.rtf** format. You can decide which input parameters of execution are saved to file (default all) and which structure or diagram images will be displayed at the start of report. Then for each step you can request an arbitrary number of output commands and which structure and diagram images will be saved. After you have set all options, click on **Create new report**. Generating report goes through selected steps and after it has been finished, you click on the **Show report** button to see the generated report in some external program.
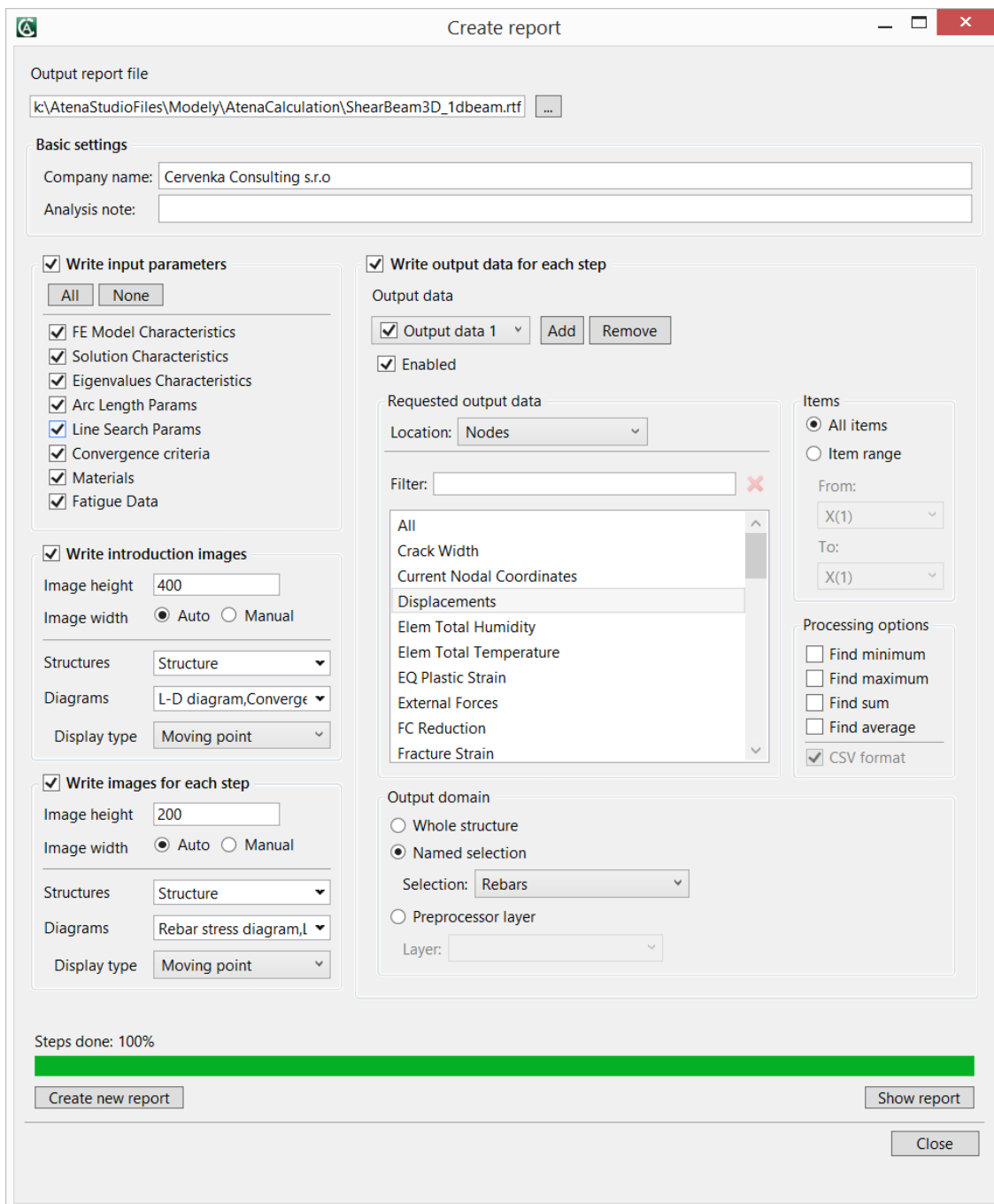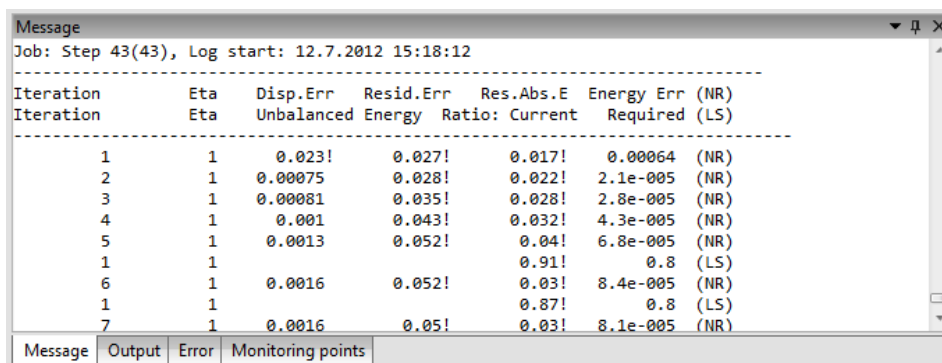
Fig. 4-13 Creating report.

4.9 Monitoring Points and Protocols

Text information about calculation is shown in the section **Protocols**. The text contains all messages issued by during an analysis. A type of output is chosen by clicking on appropriate tab.
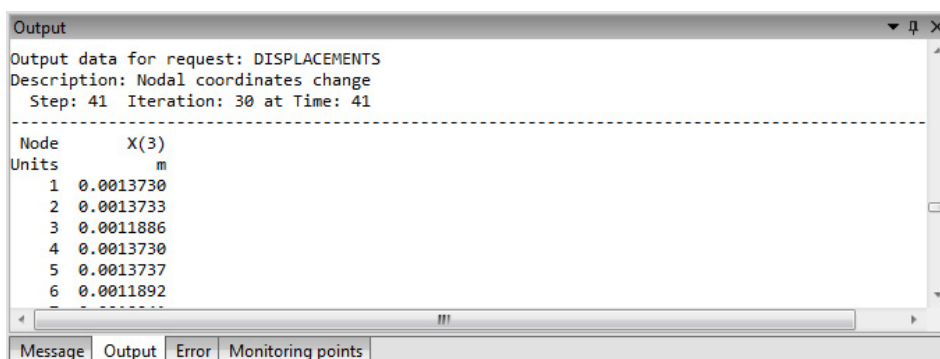

Fig. 4-14 Message text output.

4.9.1 Message

Message text output (see picture above) is described in the previous chapter (3.9.1)

4.9.2 Output

Output text displays information about global solution parameters, such as matrix size, boundary conditions, etc. The requested data can be also shown there. How to display requested data see 4.3.2.


Fig. 4-15 Output text with requested displacement.

4.9.3 Error

Error text includes convergence information supplemented by error messages. There are two types of this message: Warnings and errors. Warnings are not fatal and usually only indicate that data need to be checked. Calculation usually proceeds after warnings and error messages are issued. It is a user responsibility to take appropriate actions (change input data and repeat analysis if necessary) in order to prevent error messages.
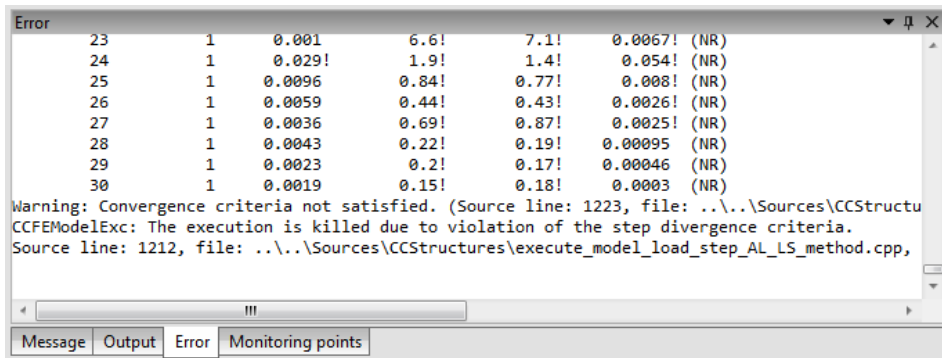
Fig. 4-16 Error messages.

4.9.4 Monitoring Points

This tab shows values of convergence monitors (predefined) and monitors defined by user. The values are changing with steps.

Fig. 4-17 Monitoring points values.

5 Some Useful Hints for ATENA Studio

5.1 Processing of Existing Data

There are many ways how to inspect and process ATENA data. It is possible to save entire project in form of *.ccs file at any stage of processing and open these data later. Project file contains references to all files associated with current task and keeps all settings in one compact environment.

It is also possible to use **ATENA Studio** only for post-processing of data saved by other ATENA programs: **ATENA 2D**, **ATENA 3D**, **AtenaWin** and **AtenaConsole**. Such data must be stored in binary load step files *name.i*, where *i* is number of step. Using command **Create project from result files** you create new project associated with these already computed steps.

5.2 Large Tasks

It should be realized, that the finite element analysis and especially in 3D can grow to large data sizes, which in turn can require long time of processing. However, the large size has more pronounced effect only on some functions. The post-processing functions, which are strongly affected by size are those included in the tools shown in the Structure view settings. In contrast, the tools for graphic manipulations (graphic toolbar depicted in

Fig. 3-14 and Fig. 3-15) are not so much sensitive to data size, since they deal with graphical images only and are efficiently processed by graphic functions.

The above effect should be considered while working with post-processor. For example, it may require some time to change the contents of the image from results of 3D to 1D layer or from stresses to strains, but it is not a problem to look at the same result from different views, move and rotate etc.

The amount of files produced during analysis (and as a consequence, the general response during runtime and post-processing) can be significantly reduced by not saving the full results for each load step, i.e., only selecting steps which should later be post-processed before starting the analysis.