ČERVENKA CONSULTING

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

ATENA Program Documentation
Part 4-6

ATENA Science – GiD Tutorial

Step by step guide for nonlinear analysis
with ATENA and GiD

Written by

**Zdenka Procházková, Jan Červenka,
Zdeněk Janda, Dobromil Pryl , Jitka Mikolášková**

Prague, June 2, 2015

CONTENTS

1. INTRODUCTION

This tutorial provides a basic introduction to the usage of the program **ATENA** and **GiD,** and it is specifically targeted for **ATENA-GiD** beginners. **ATENA-GiD** is a finite element based software system specifically developed for the nonlinear analysis of reinforced concrete structures. **ATENA** is used for the analysis itself and the program **GiD** is used for the data preparation and the mesh generation.

This tutorial contains a step by step explanation how to perform a non-linear analysis on an example problem of a reinforced beam without smeared reinforcement. The geometrical and material properties correspond to the experimental setup by Leonhard in 1962. More details about the problem or experiment can be also obtained from the original report [6] or from the program developer or distributor.

It is possible to create and analyse the example problem described in this tutorial in demo version of **ATENA-GiD**. Because of that a rather coarse finite element mesh is used. It is recommended that in the analysis of real engineering problems, users use sufficiently fine meshes, and if needed a mesh sensitivity study should be performed.

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

It is recommended to print-out this version, in order to easily follow the instructions. In case of printing, it is advisable to use both sided and colour printing.

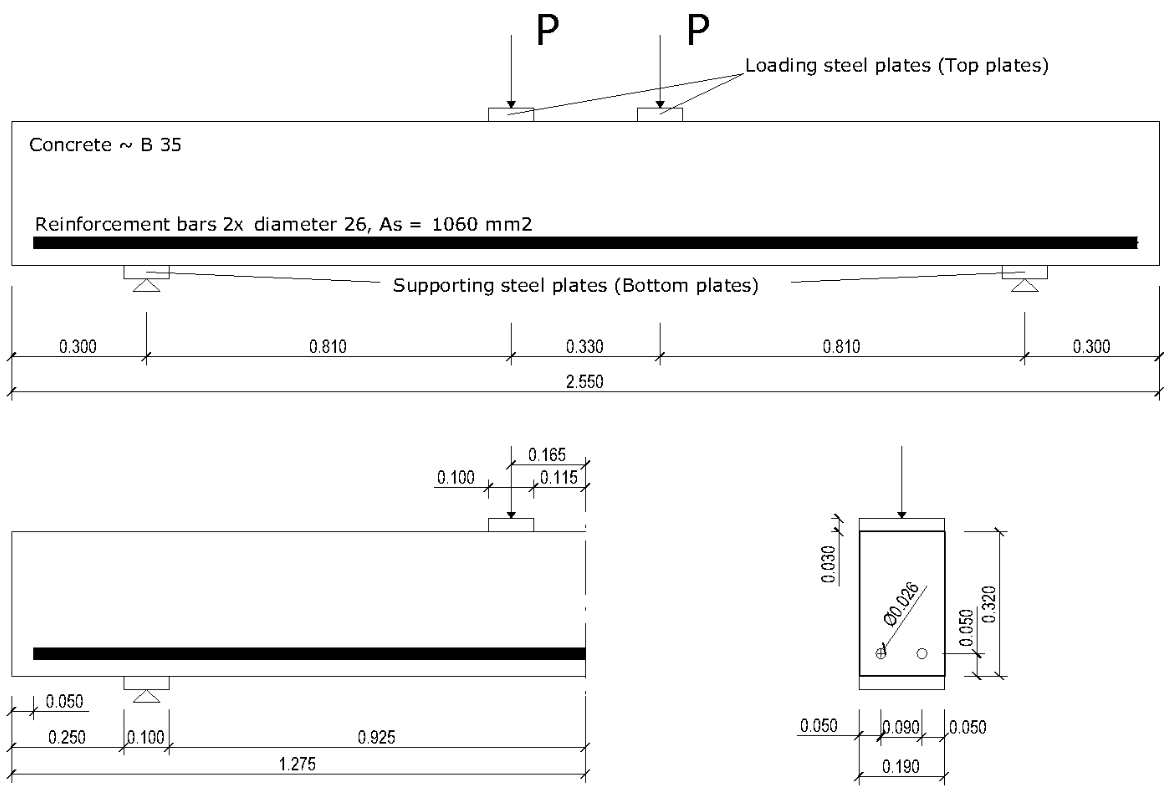


Figure 1: Geometry of the structure

The steps necessary for the data preparation, non-linear analysis and post-processing are depicted on subsequent figures, which show the computer screen for each step and the corresponding user action. There is always also a short description for each figure.

It should be noted that not all features of **ATENA-GiD** system are described in this manual. For more details about the data preparation and post-processing, the user is encouraged to read the manual of the program **GiD** and **ATENA-GiD** manual [2].

2. STARTING PROGRAM

Before using the **ATENA-GiD** system it is necessary to install it on your computer. The programs **GiD** and **ATENA** can be installed using the standard **ATENA** installation. At the end of the installation, the user must select the installation of **GiD** and **ATENA-GiD** interface. After that your computer should be ready to run the example problem described in this document. The installation process is described in detail in ATENA-GiD manual [2].

In order to start a nonlinear analysis in **ATENA-GiD** system, first the program **GiD** is started. The recommended version is 11.0.1 or newer (the oldest supported version is 7.7.2b). The program **GiD** can be started from the start menu of your computer using the following path: **Start | All Programs | CervenkaConsulting | ATENA Science | GiD**.

This opens the program **GiD**, which is used for the preparation of the numerical model of the analyzed structure. This process is described in the subsequent Chapter 3. The execution of the nonlinear analysis is described in Chapter 4 and the post-processing in Chapter 5.

3. PRE-PROCESSING

3.1 Introduction

This chapter explains the basic steps, which are to be performed in order to define a complete geometrical, and then a finite element model for the non-linear FE analysis by **ATENA**.

The purpose of the geometrical model is to describe the geometry of the structure, its material properties and boundary conditions. The analytical model for the finite element analysis will be created during the pre-processing with the help of the fully automated mesh generator.

The definition of the geometry starts with the creation of geometrical points. These points are later connected into boundary lines. The surfaces are defined by selecting appropriate bounding lines. Volumes can be formed either by extrusion of surfaces or manually by selecting all bounding surfaces. Three-dimensional regions are modelled by volumes in **GiD**. The reinforcement is modelled as a line. These reinforcement lines are not usually connected to any surface or volume, but they usually lie inside the volumes entities that form the concrete structure.

After creation of the geometry, material properties should be defined and assigned to individual volumes. Boundary conditions are used to define supports and loads. The boundary conditions and loads are defined in **GiD** with the help of “Intervals”. Interval represents a set of boundary conditions and loads that are applied in a specified number of steps. An appropriate definition of intervals can be used to specify a complete loading history.

In **ATENA** analysis it is always useful to define monitoring points. The monitoring points are used to see the evolution of certain quantities during the analysis. For instance they can be used to follow the development of deflection or forces at given locations. The monitoring points are defined as special conditions that should be specified in the first interval.

3.1.1 Introduction of the Graphical User Interface

Before starting the definition of the geometrical model it is good to introduce the graphical user interface of **ATENA-GiD**. The main window is shown in the Figure 2. It shows the basic layout of **GiD** program right after its start and it explains the basic functionality of the various icons and menus.

This window shows the basic layout of the **GiD** program. At this stage it contains only commands for the creation of geometric objects. In order to activate **ATENA** specific materials and boundary conditions, an appropriate problem type needs to be selected. This is described in the next section.

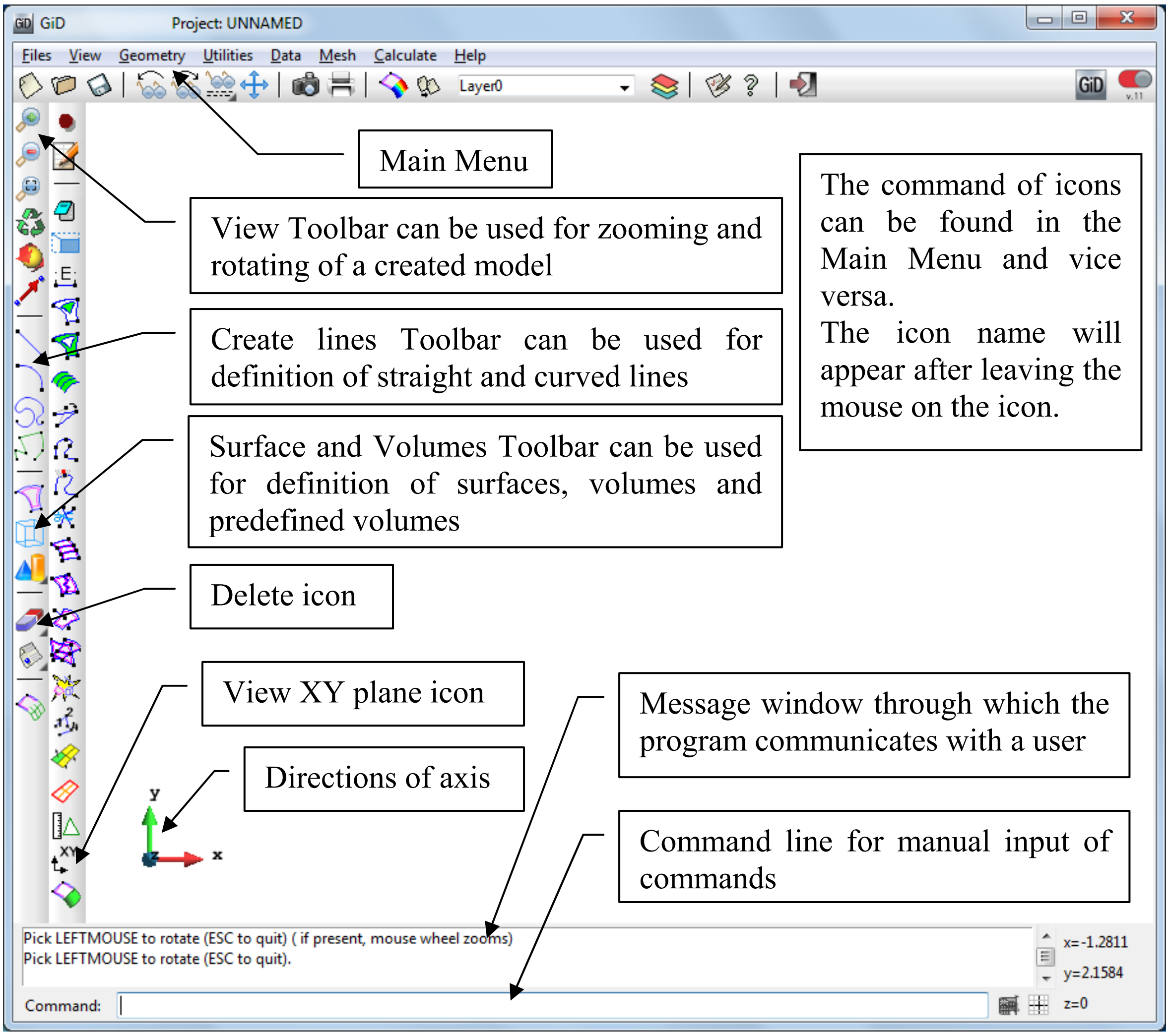


Figure 2: Graphical user interface of ATENA-GiD.

3.1.1.1 Problem Type

The **GiD** is a general-purpose pre- and post-processing tool for variety of numerical problems (and analysis software). The **GiD** can be customized to create input data for basically any finite element software. The customization is done through the definition of various problem types. Each problem type represents certain customization. Therefore it is important to select an appropriate problem type at the beginning of the work.

In this case, **ATENA** problem type has to be selected. The problem type definition must be done before starting input of data. Executing this command later may cause losing of all material and load definition. The problem type is selected from the Main menu **Data | Problem Type | ATENA | Static**. Once this is selected **ATENA** specific icons will appear in the main window (see Figure 3).

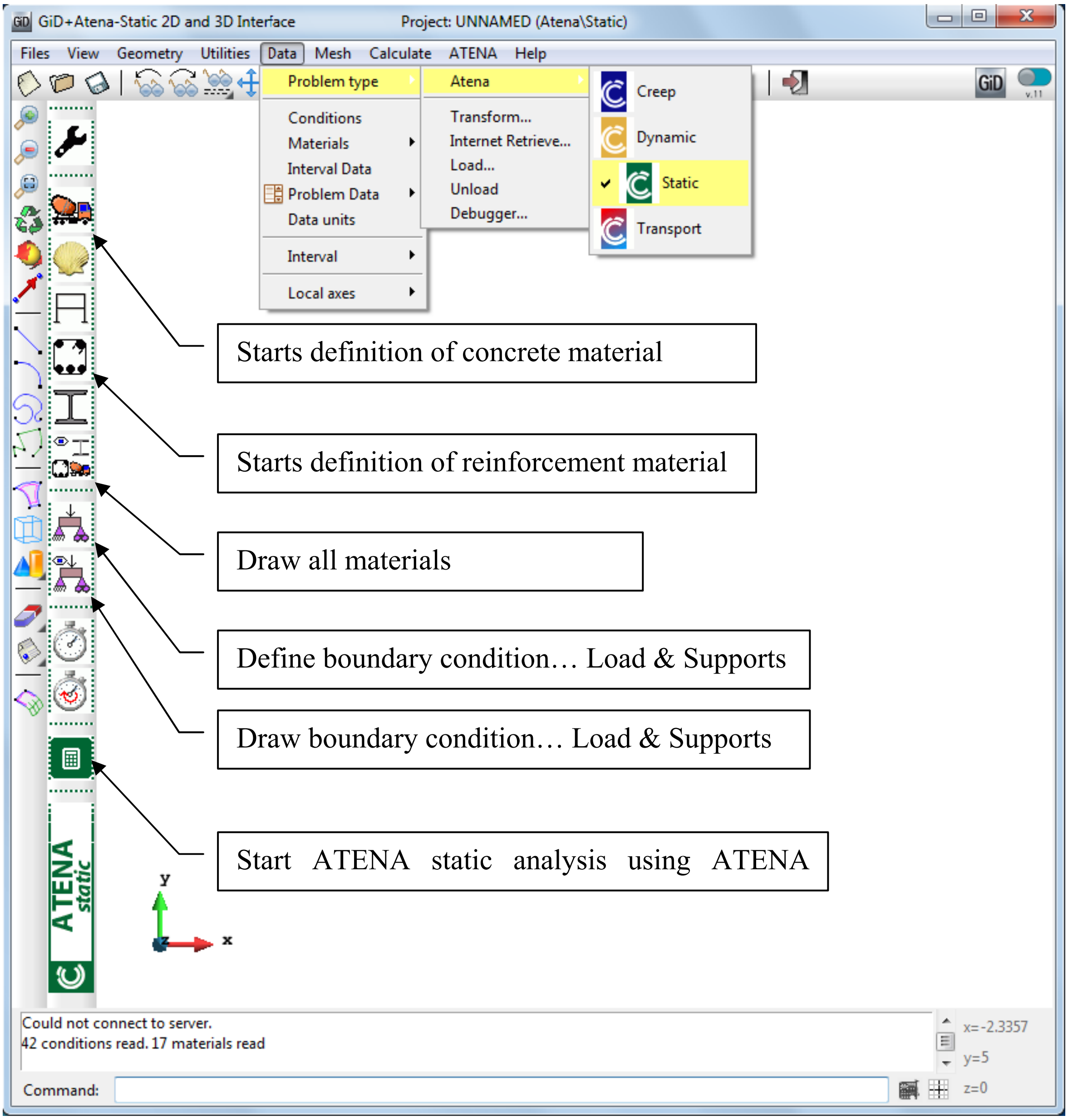


Figure 3: Problem type menu and basic ATENA icons.

It is also recommended to explore the help of the program **GiD**. This can be found in the Main menu or by pressing F1 on your keyboard.

It is also much recommended to save file and also regularly save created model during the formation of the geometrical model. Saving is done by selecting **File | Save** or **Save as**. The name of the document can be chosen for example **3DBeam**.

3.1.1.2 Problem Data

Before starting the model definition it is advisable to define some global analysis parameters. It is done by the command **Data | Problem Data | Problem Data** in the main menu (see Figure 4).

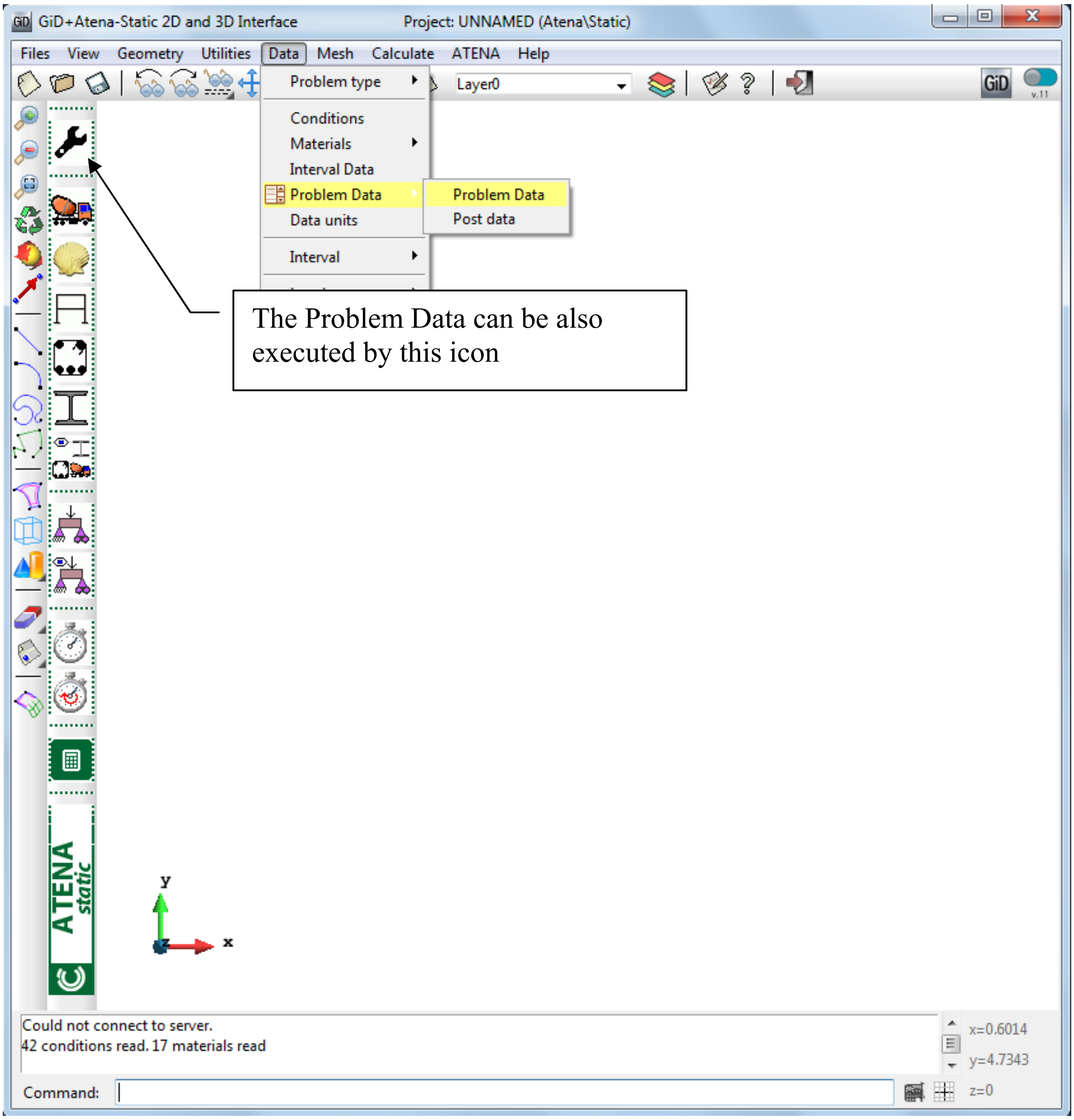


Figure 4: The command Problem Data

After selecting this command the Problem Data window will appear (see Figure 5). There the Title and Task Name should be changed to rename files where the results of the analysis will be saved.

When the analysis is finished, all results are saved in files. From those files, results can be executed and processed lately. Therefore it is useful to rename the title of the files where results should be saved and it is useful to do this saving in the beginning of the any creation of project. Later it could be forgotten.

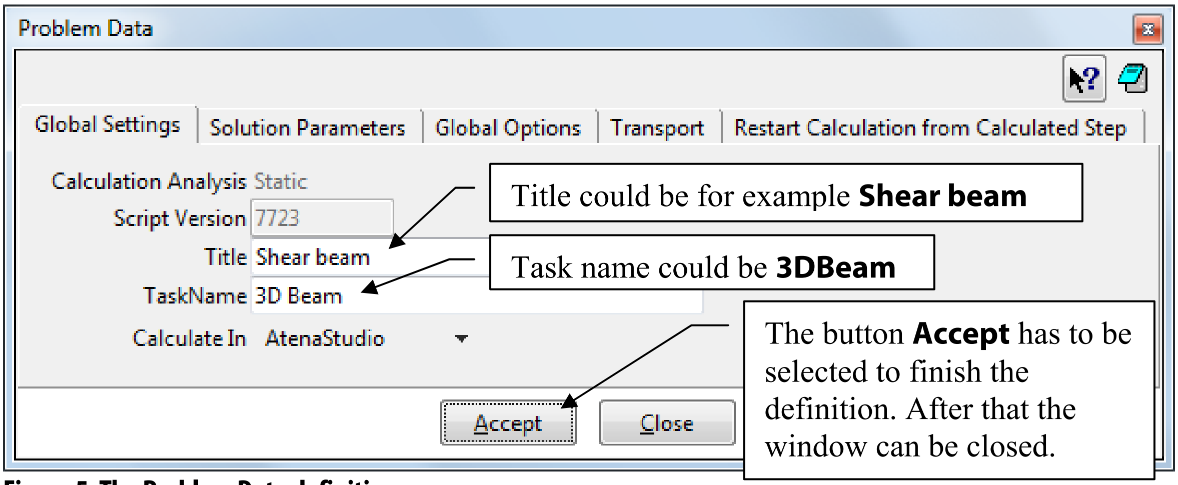


Figure 5: The Problem Data definition

Parameter input:

Title:	Shear beam
TaskName:	3DBeam

3.2 Geometrical Model

This chapter describes definition of the geometrical model. Because the beam is symmetric, only half of the beam will be created in this example.

The geometrical model of this half beam (see Figure 6) is composed of three 3D regions and two reinforcement bars. In **GiD** the 3D regions are called "Volumes". Therefore the geometrical model contains three volumes – beam, loading and supporting plates. The reinforcement is modelled by two straight lines. The definition of these geometrical entities is described in the subsequent chapters.

It is useful to use the layer function for the definition of the geometrical model. It is a function, where particular parts of the model can be placed on different layers and then displayed, hidden or locked etc. In this geometrical model, three separate layers will be created – beam layer, plates layer and reinforcement layer.

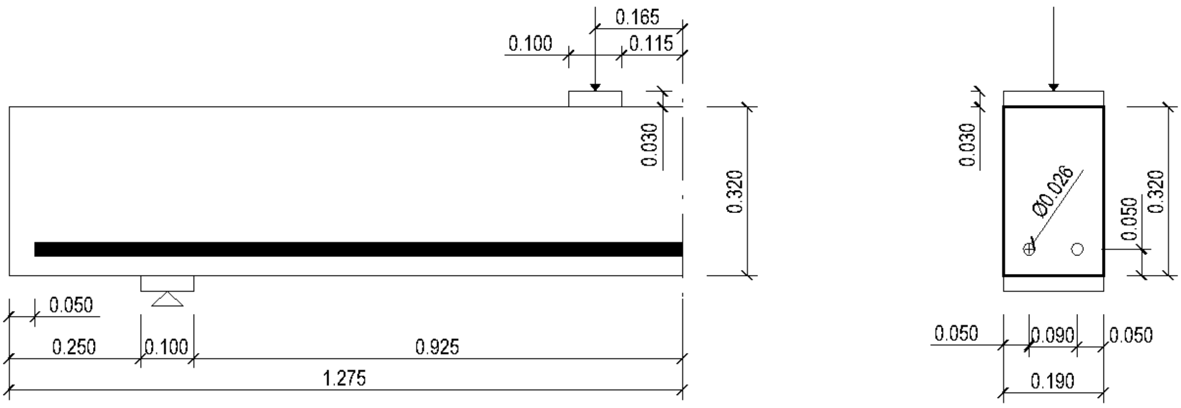


Figure 6: The geometrical model is composed from three volumes – beam and two plates

3.2.1 Concrete Beam

A concrete beam forms the main part of the example. This section describes the definition of the three-dimensional beam geometry. The geometry of the beam will be created by an extrusion of a rectangular surface. That will be defined by four lines.

First step is to create points, which will be later connected into a rectangular surface. A point is created using the command **Geometry | Create | Point** in the Main menu. In order to create a rectangle, four points are needed. Each point is defined by three coordinates (x,y,z). The coordinates of points should be written in the command line in the bottom part of the main window.

The coordinates can be written all together separated by comma. A dot represents a decimal point. The definition of coordinates of each point is completed by ENTER. (In the command line it is very handy to use the key arrow up and down on your keyboard to view previously entered coordinates. These previous coordinates can be changed and entered again.) In this case the following points should be entered:

Parameter input:

Coordinates of points:

1: (0,0,0)

2: (1.275,0,0)

3: (1.275,0.19,0)

4: (0,0.19,0)

NOTE: The table named "**Parameter input:**" will guide you through the whole tutorial. This table shows the parameters, which should be entered. There are predefined default parameters in some dialog windows. The table **Parameter input:** shows only parameters, which should be changed.

After entering coordinates the points appear in the graphical area (see Figure 7). It is useful to enlarge the model such that it fills the whole screen. For that the command **View | Zoom | Frame** in the main menu or the Frame icon  can be used (see Figure 8).

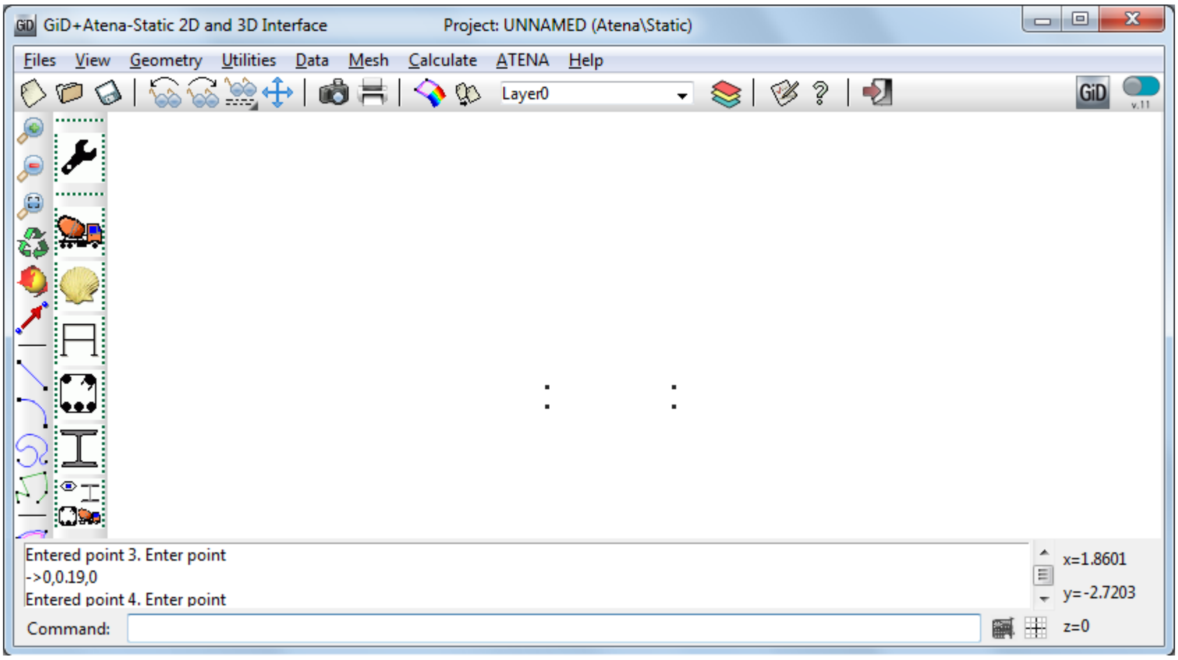


Figure 7: Four created points before zooming.

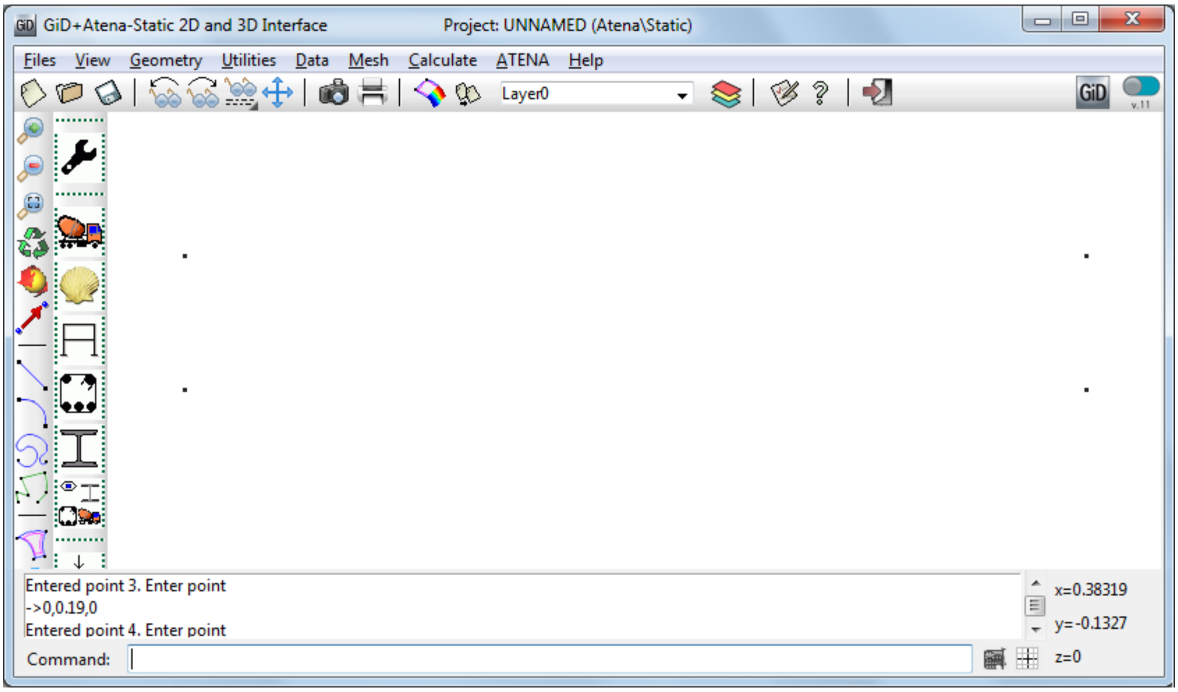


Figure 8: Using of the Zoom frame icon enables to have a better view of the created geometry.

Next step is to connect these points by lines. Lines are created using the command **Geometry | Create | Straight line** in the Main menu or by clicking on the icon . Then the message window at the bottom will show the following sentence:

Enter points to define line (ESC to leave).

The lines can be defined by entering exact coordinates into a command line or it is possible to directly pick the already existing points. In this example the direct picking has been chosen.

The direct picking can be done by selecting **Contextual | Join Ctrl-a** in the Mouse menu. The Mouse menu can be found by clicking on the right button of the mouse in the graphical area (see Figure 9).

Alternatively this option can be activated directly by pressing the key Ctrl and 'a' at the same time.

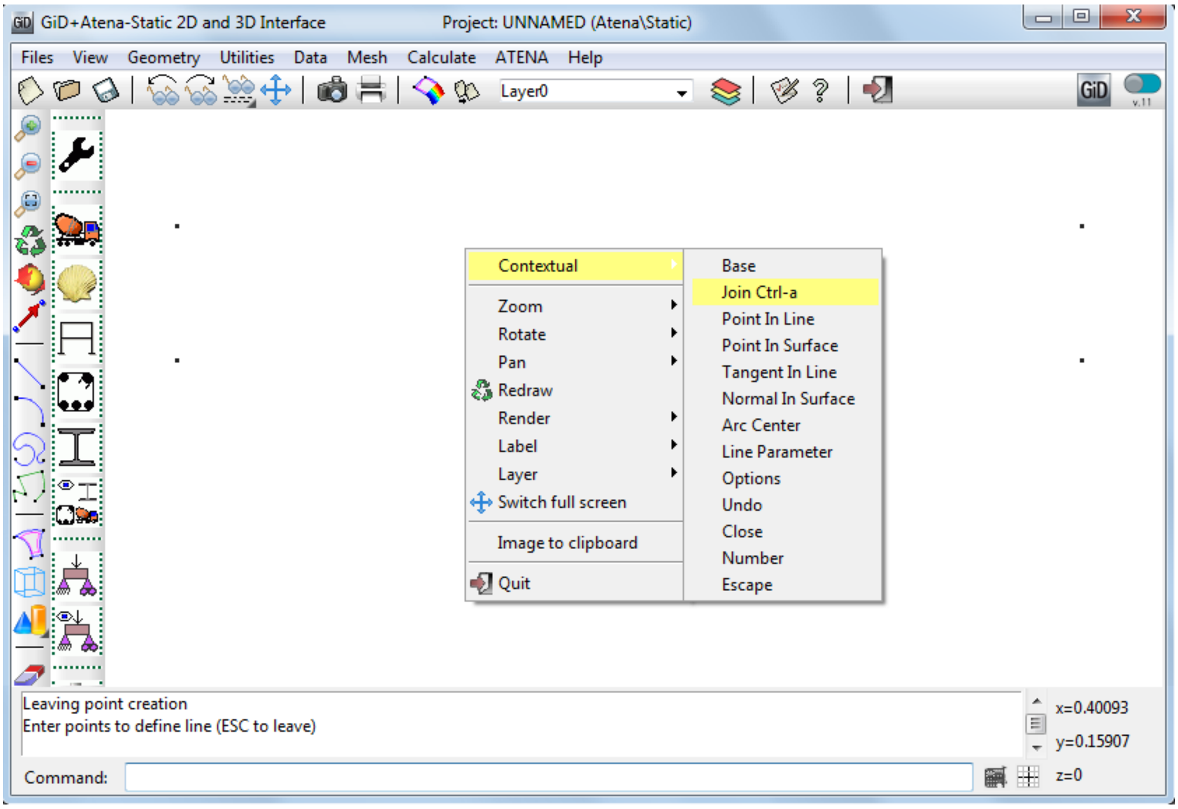


Figure 9: The Join function in the Mouse menu.

After selecting the join function the mouse cursor will change to this shape ▣. Then after clicking into a graphical area the nearest point will be selected. Now all points can be connected by lines into the rectangle (see Figure 10). The create line function should be finished by pressing ESC key.

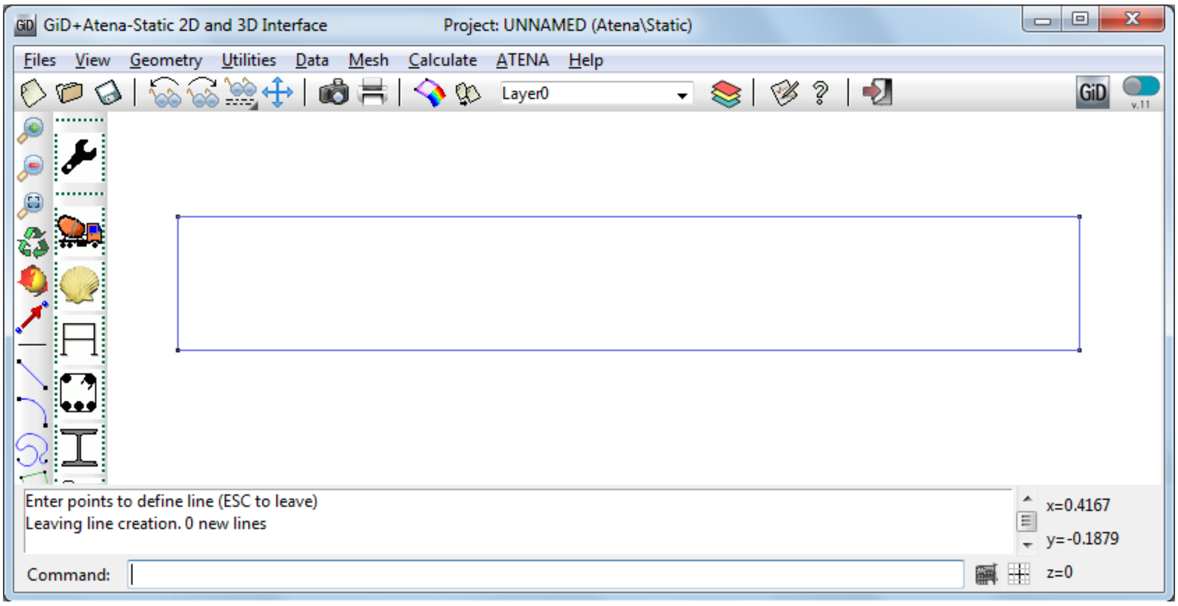


Figure 10: The lines connected into a rectangle.

The **GiD** distinguishes four types of entities – point, lines, surfaces and volumes. In our case there are already two entities - points and lines. Lines define a rectangular boundary but it is not a surface until a surface is defined. Therefore, the next step is to create a surface using the already existing lines.

It is done by selecting **Geometry | Create | NURBS surface | By contour** in the main menu and then selecting all lines defining the required surface in the graphical area (see Figure 11). Clicking on the icon  can also start the **Create surface** function. Next, the lines bounding the surface should be selected, and then it is necessary to press ESC key to complete the surface definition. The newly created surfaces are denoted by a pink colour as seen in the Figure 11.

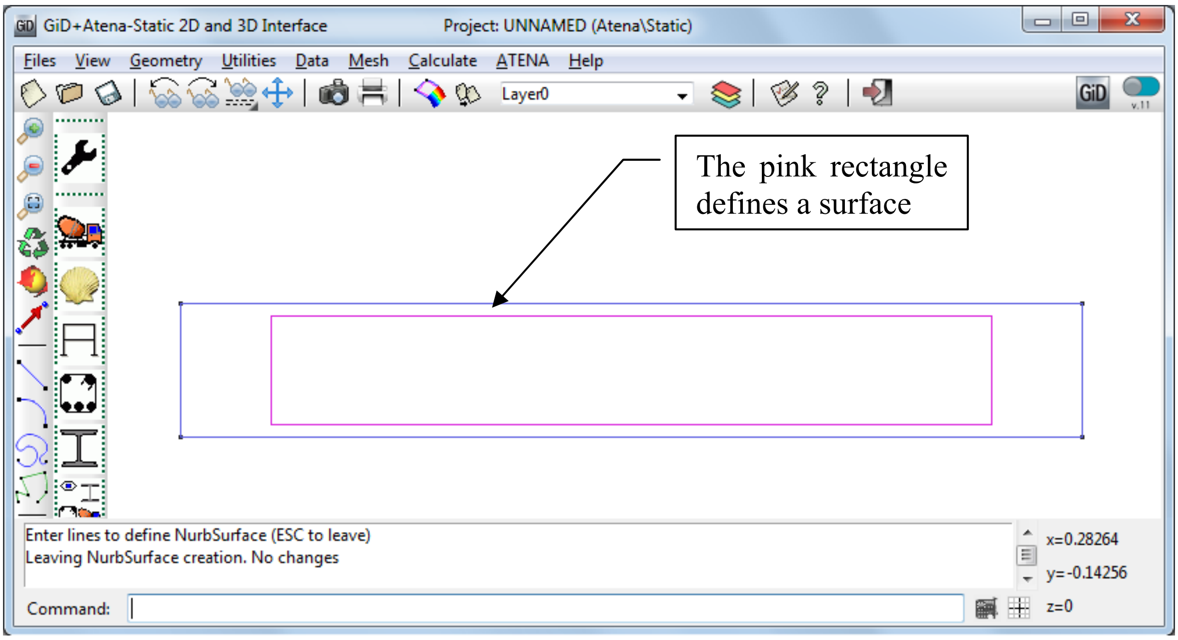


Figure 11: The pink rectangle in the middle of blue lines defines the added surface.

The next step is to extrude the created surface into a volume to obtain the required beam. The extrusion is done by the command **Copy**, which appears after selecting the command **Utilities | Copy** in the main menu (see Figure 12).

In this example, the surface is extruded in the direction of the Z-axis over the beam thickness **0.32 m**. The thickness will be given by a vector that is defined by coordinates of two points in the **Copy** menu. The definition of the extrusion is depicted in the Figure 12.

After the definition of all copy parameters, the **Select** button should be pressed. Then the surface for the extrusion can be selected in the graphical area. The command is completed by pressing **Finish** button.

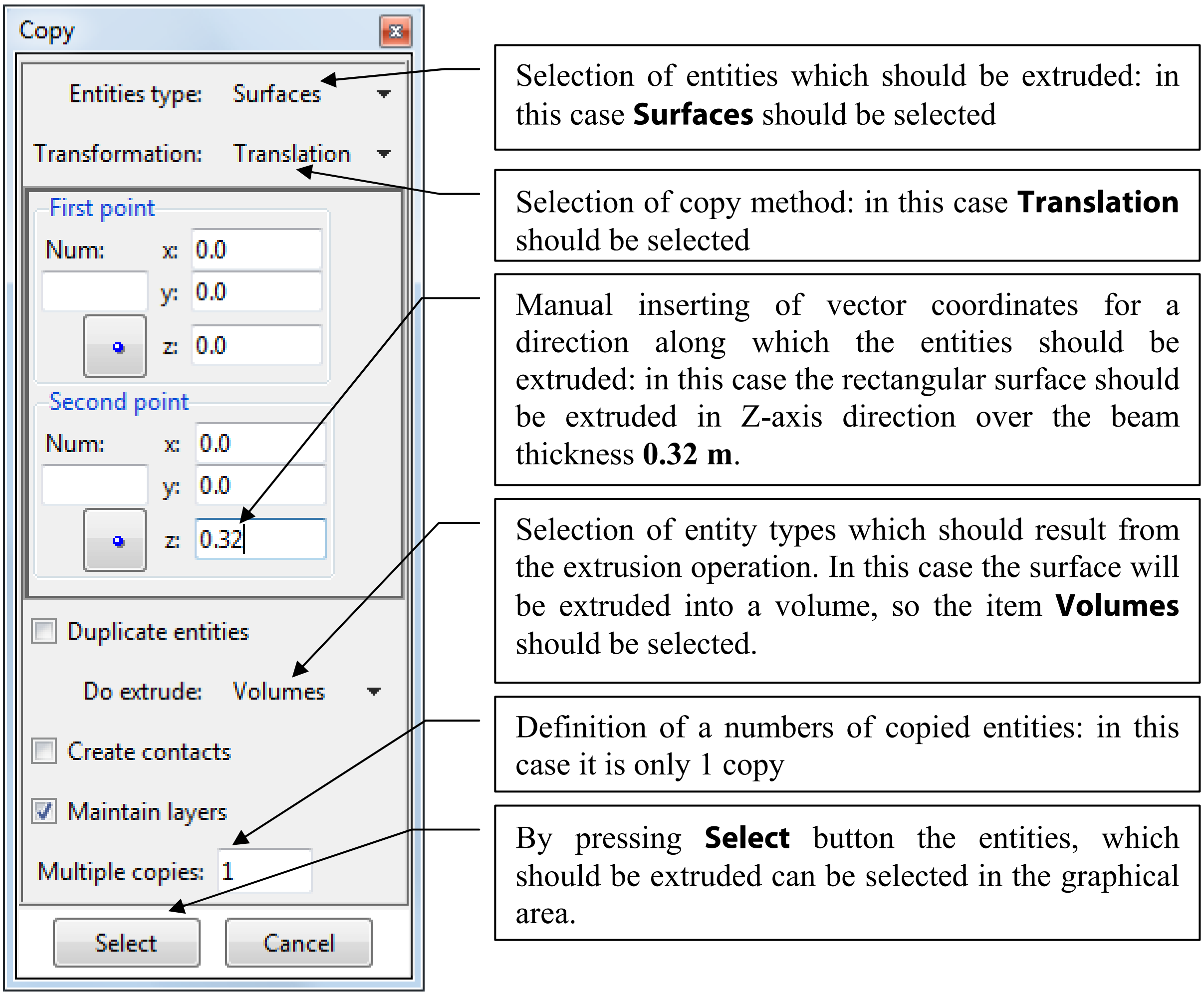


Figure 12: The description of Copy menu

Parameter input:
Entities type: Surfaces
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.0
z: 0.32
Do extrude: Volumes

The selection of the surface can be done by a direct clicking on the pink line, which defines a surface. Another option is to select the surface by holding the right mouse button and by moving of the mouse. The box should cross at least one line of the surface to be selected. After the proper selection the pink selected surface will change to the red colour.

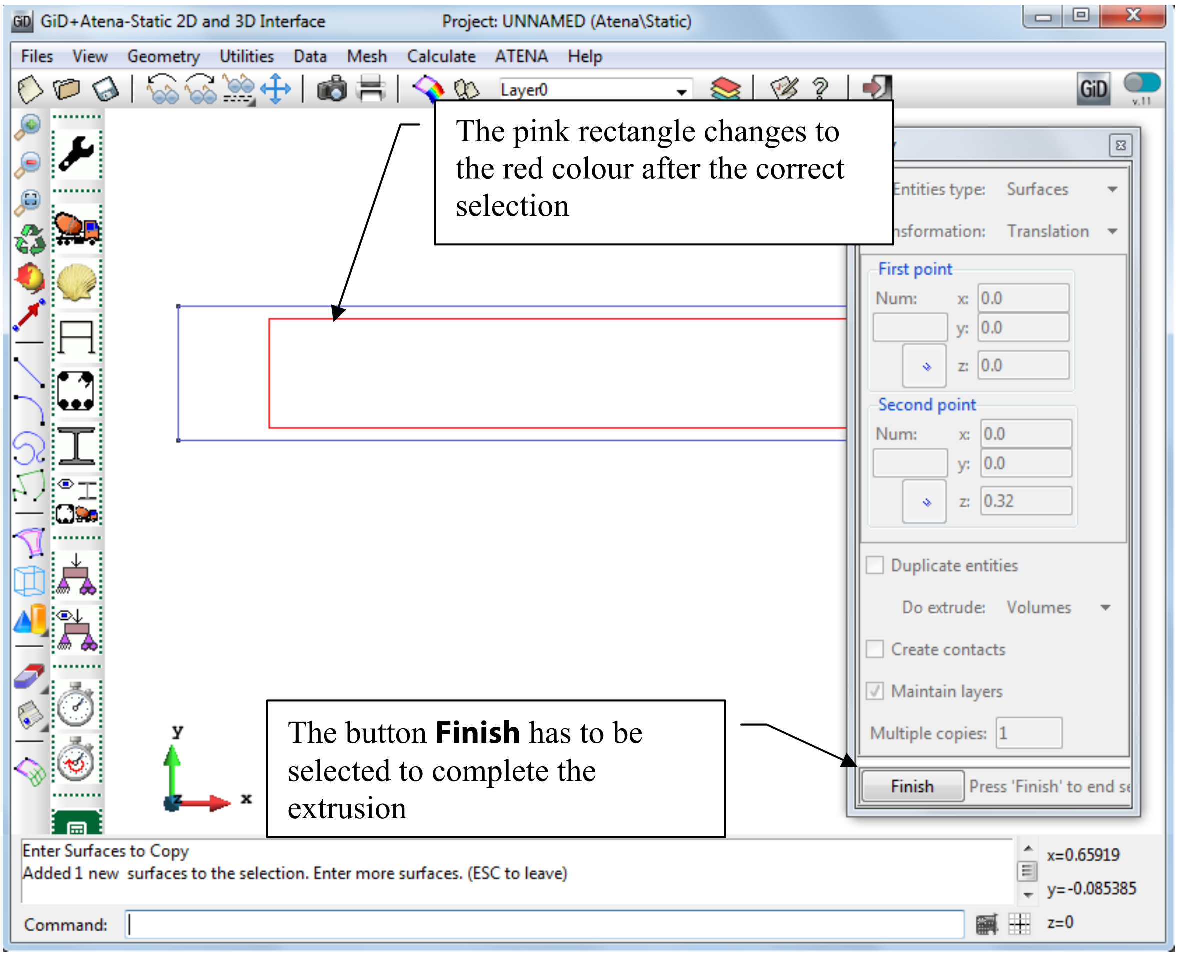


Figure 13: The selection of the surface for the extrusion

To see the extruded volume it is possible to use Rotate Trackball icon  or holding left mouse button + SHIFT key (see Figure 14).

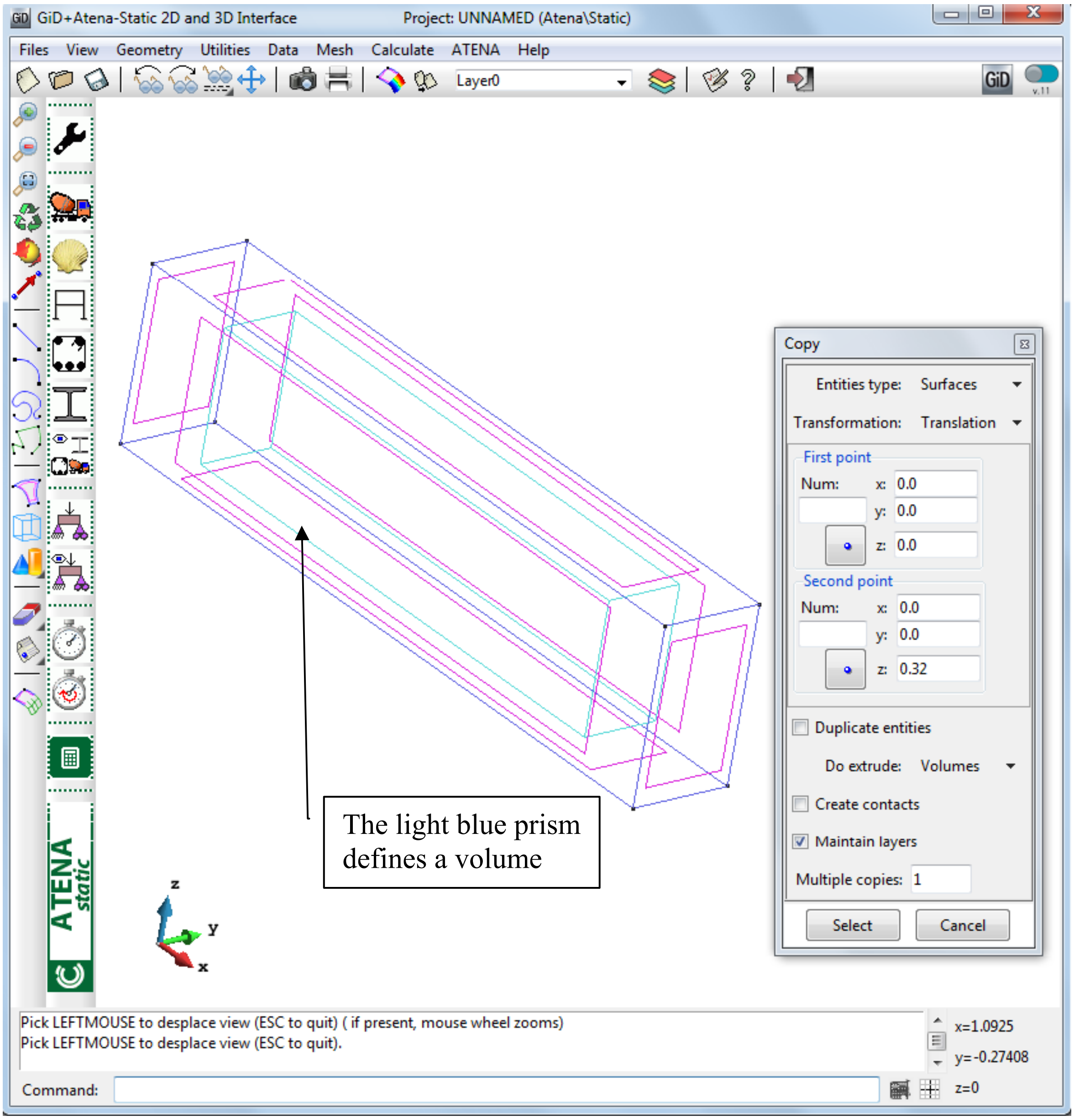


Figure 14: The extruded volume – the light blue prism defines a volume

3.2.2 *Loading and Supporting Steel Plates*

After the creation of the beam geometry loading and supporting plates should be created. The top plate (loading plate) will be created first. The bottom plate (supporting plate) will be created by copying of the top plate.

The top plate will be created with using the commands **Copy** and **Create lines**. These commands should be known from the previous chapter. The dimensions and location of the plates can be seen on Figure 15.

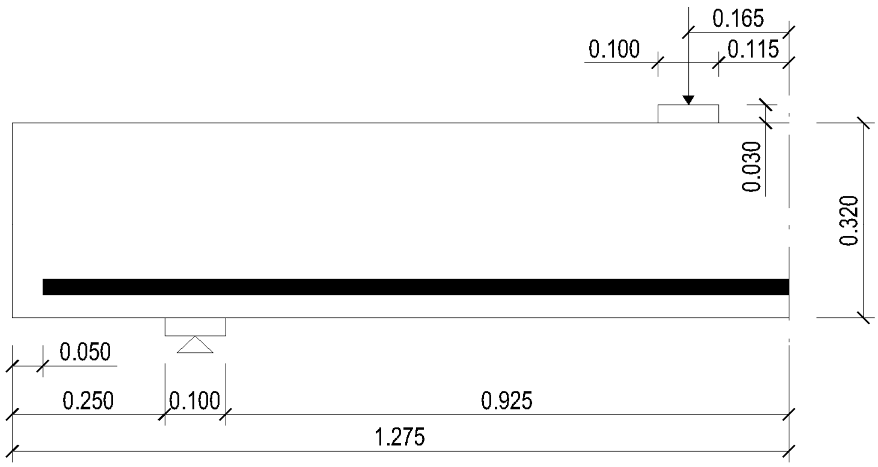


Figure 15: The dimensions of the half beam and location of steel plates

3.2.2.1 Top Plate

It is useful and easier to use existing elements for the creation of a new object. The top plate is located on the right corner of the created beam. Therefore the upper-right edge of the beam can be copied and moved to **0.115 m** from the right end. Then this line will be copied and moved again. The second copy operation should move the line by a distance identical to the width of the steel plates. These two lines will be then connected into a rectangle. The surface will be added to this rectangle and then this surface will be extruded into a volume of the steel plate.

Before starting copying it is better to zoom in the right beam corner (see Figure 16). The **Zoom in** is activated by command **View | Zoom | In** or by clicking on the icon . The command **Zoom in** and **out** can be also activated by holding SHIFT key and using mouse scroll (In that case it is also necessary to move the view of the geometry. It can be done by holding SHIFT + right mouse button.).

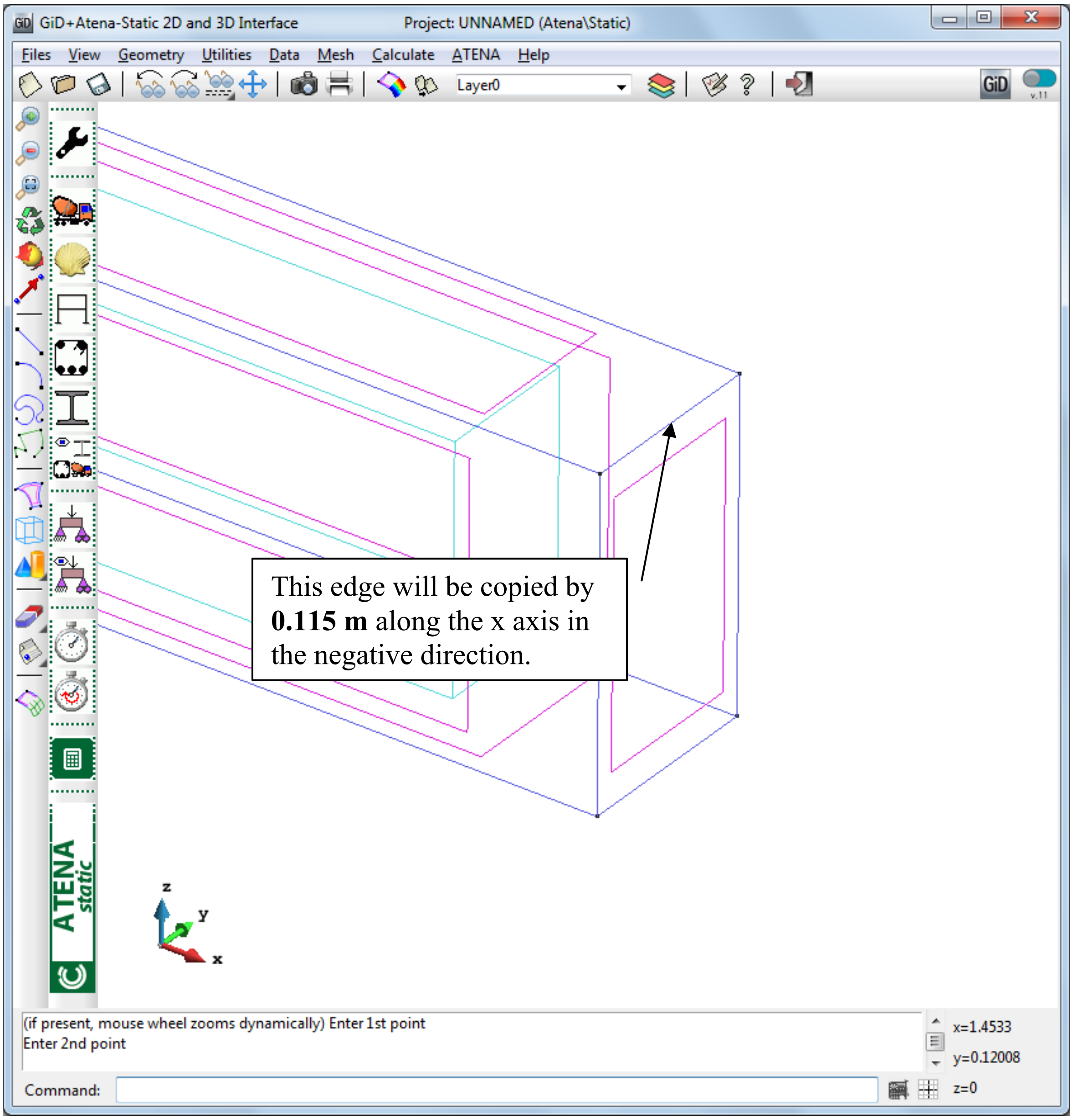
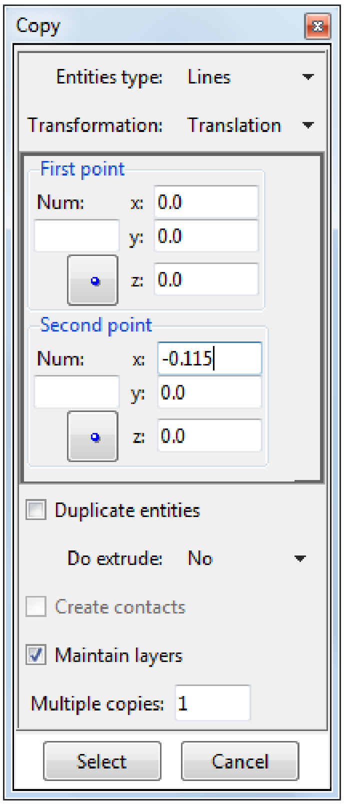


Figure 16: The geometry after Zoom in

The **Copy** menu appears after selecting **Utilities | Copy** in the Main menu. The new line should be in the **0.115 m** distance from the right edge of the beam. The copied entity is line, and there is no extrusion necessary. The parameter definition is depicted in the Figure 17. After the definition of all parameters, the **Select** button should be pressed. Then the line required for the copying can be selected in the graphical area (see Figure 18). After the selection of the line, it is necessary to press **Finish** button to complete the translation (see Figure 19).



Parameter input:

Entities type: Lines

Transformation: Translation

First point: x: 0.0

y: 0.0

z: 0.0

Second point: x: -0.115

y: 0.0

z: 0.0

Do extrude: No

Figure 17: The definition of translation of the line

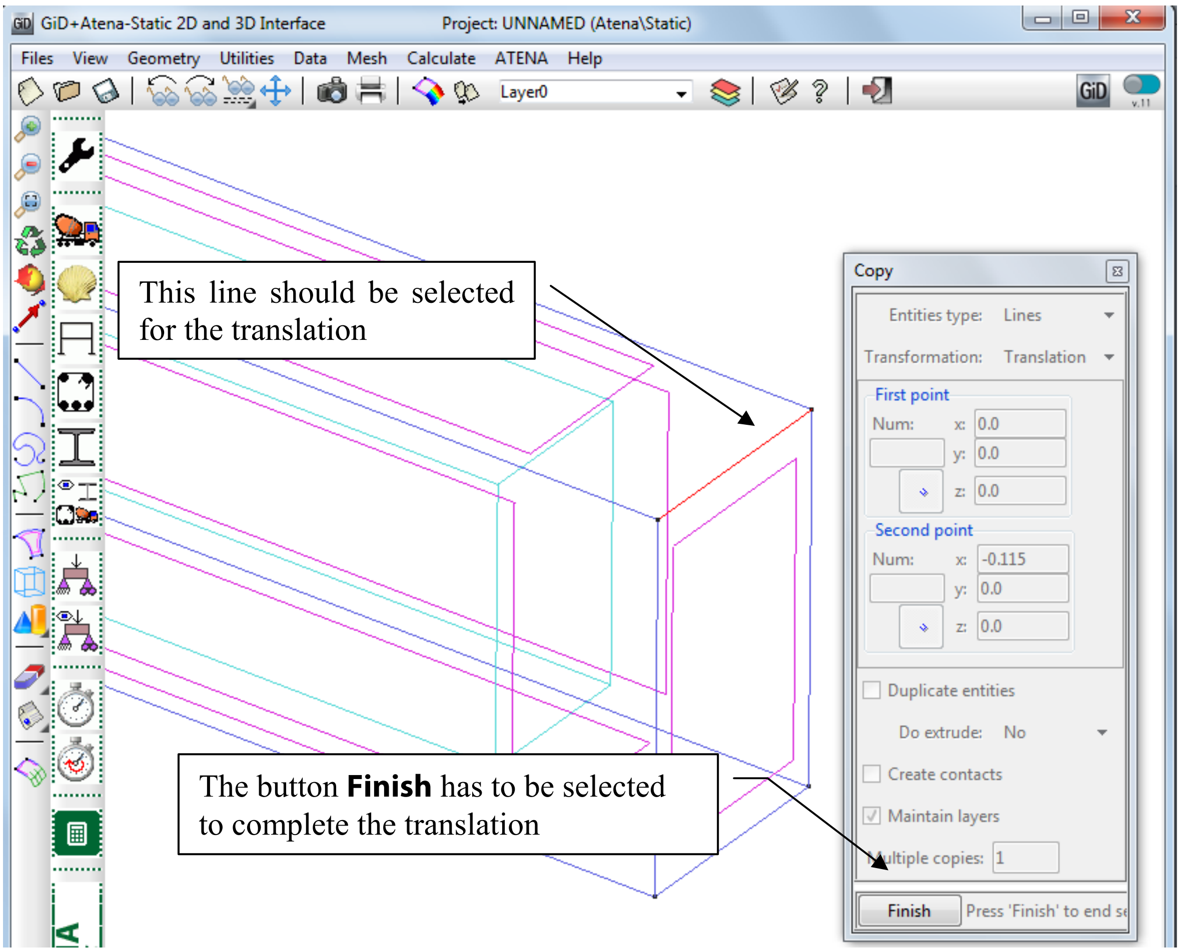


Figure 18: The selection of the line which should be copied

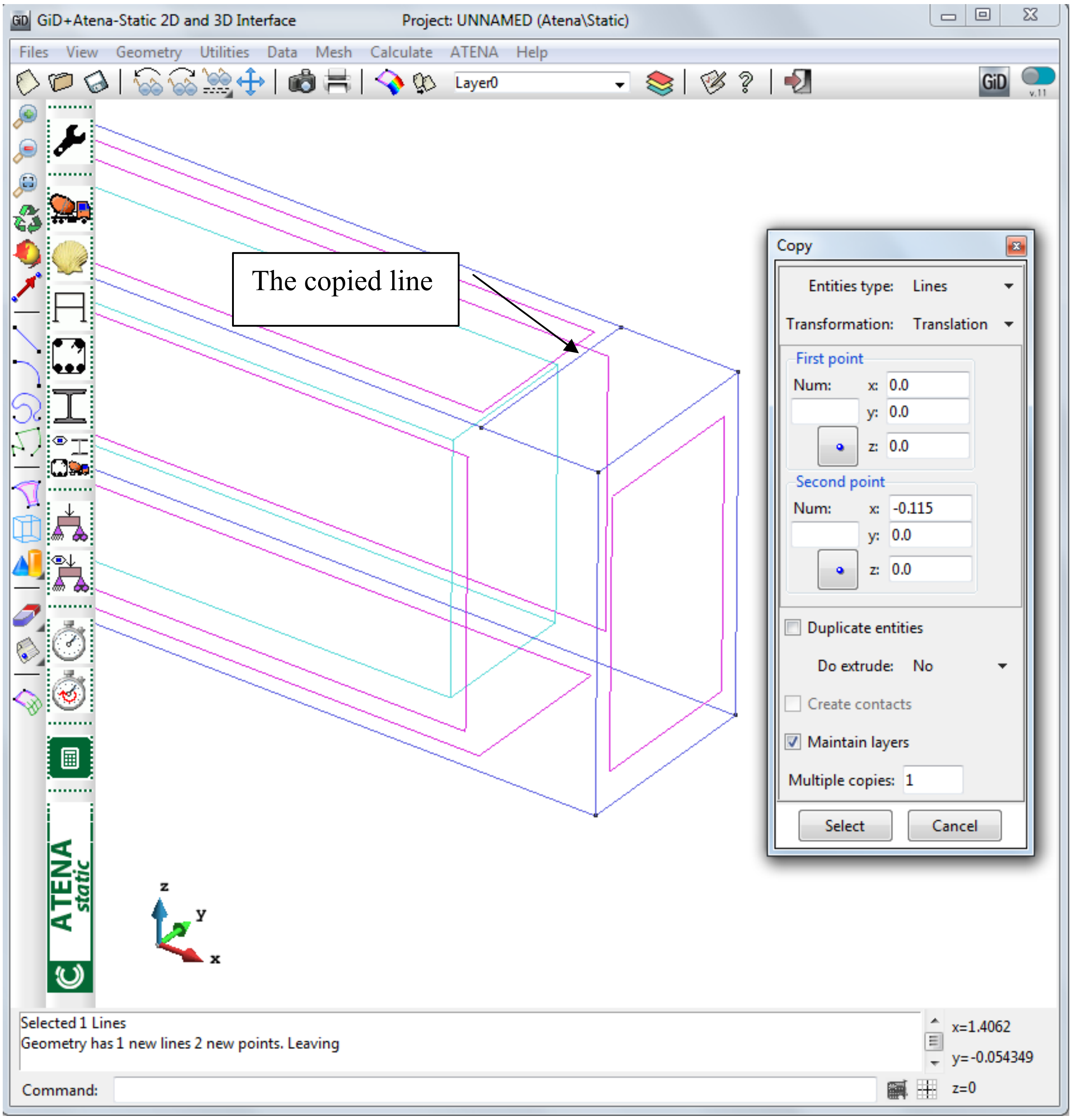


Figure 19: The new copied line

Now the new line will be copied again to create the second edge of the top plate. The width of the plate is 0.100 m. Therefore the second line will be translated by **0.100 m**. The parameter definition is depicted in the Figure 20. After the definition of all parameters the **Select** button should be pressed. Then the line required for copying can be selected in the graphical area (see Figure 21). After the selection of the line it is necessary to press **Finish** button to complete the translation (see Figure 22).

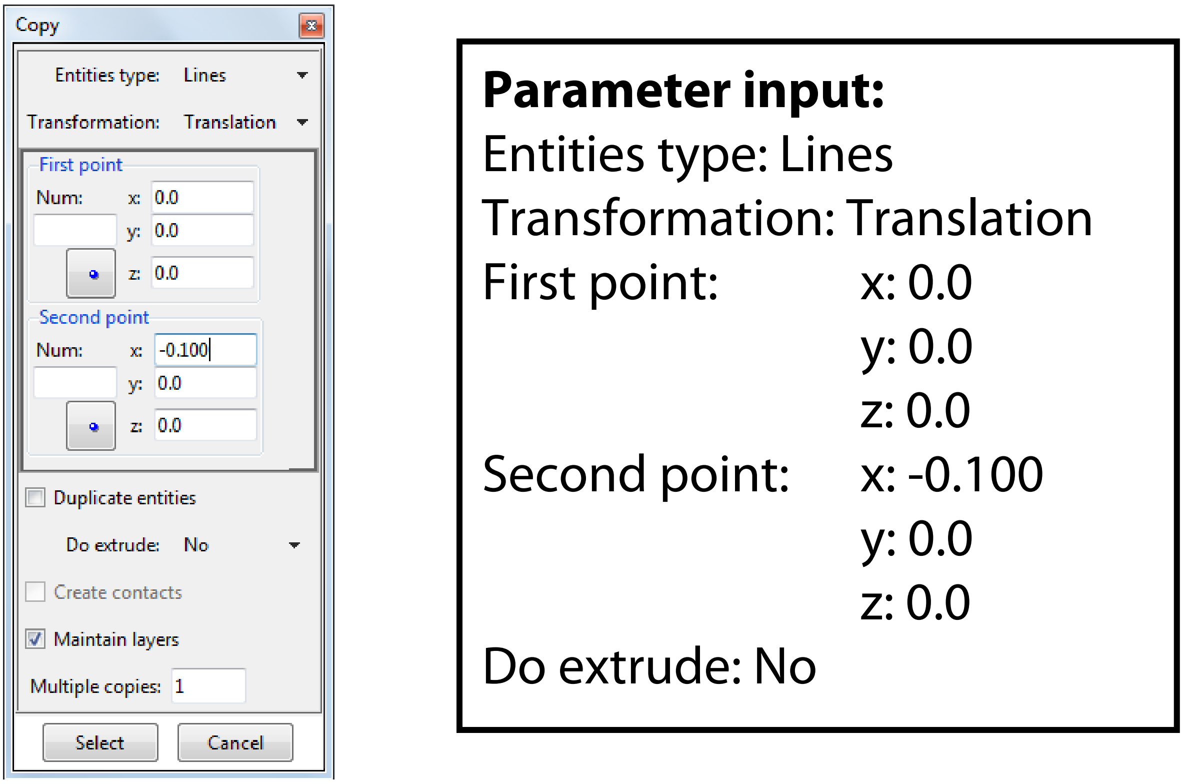


Figure 20: The parameter definition of the second line

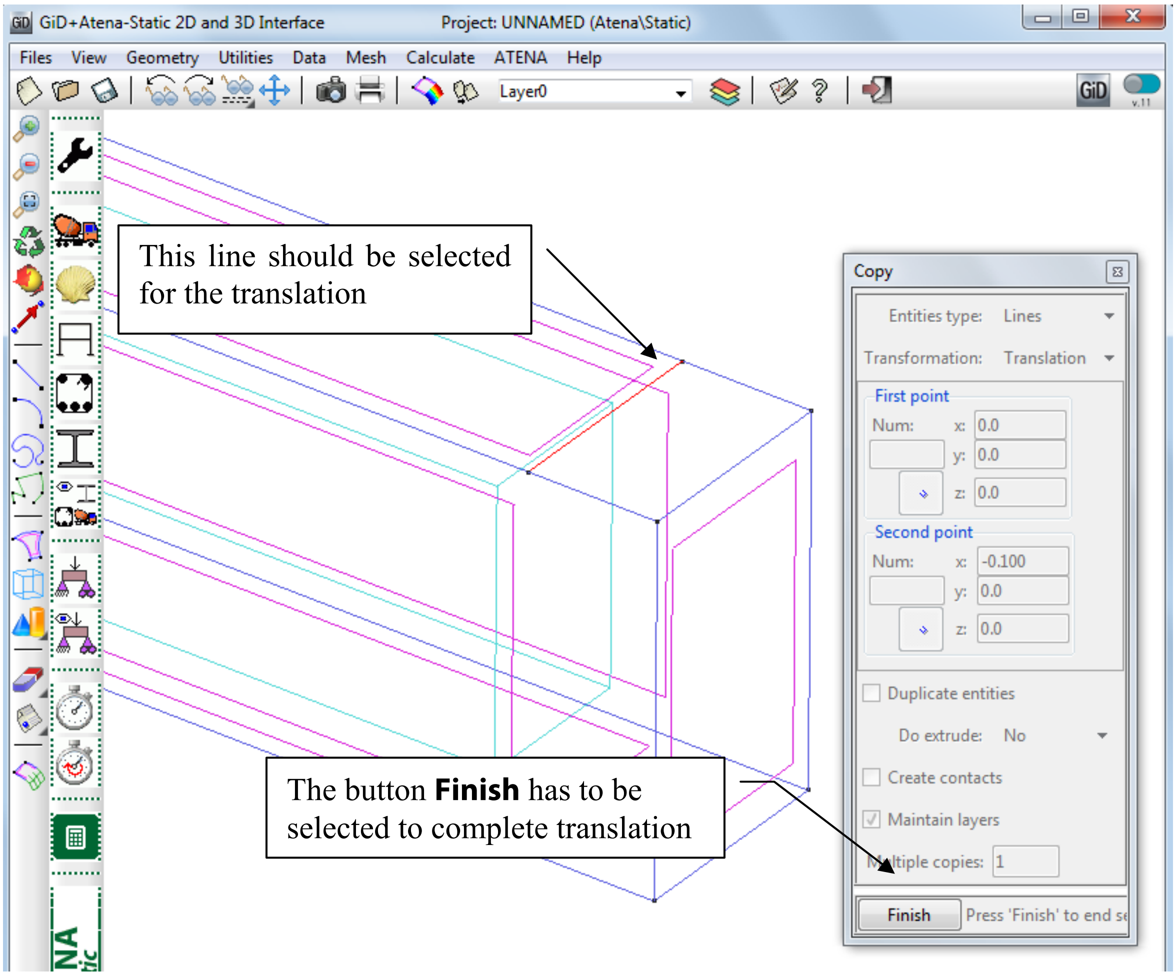


Figure 21: The selection of the line which should be copied

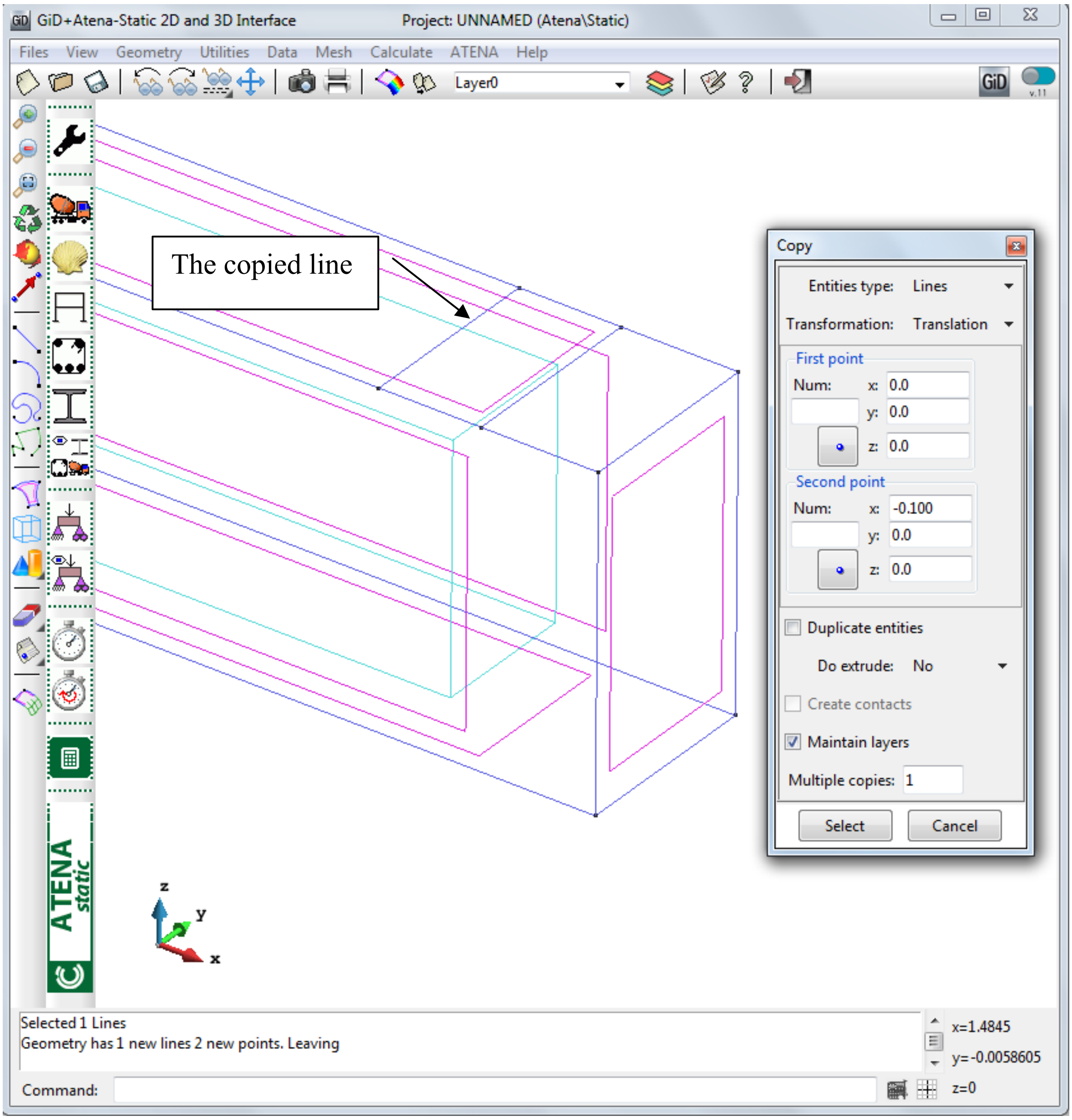
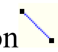


Figure 22: The repeated copy operation to create the second line

The next step is to connect these newly copied lines into a rectangle. This can be done by creation of new lines using the command **Geometry | Create | Straight line** from the main menu or by clicking the icon . Also the **Join** function should be used (Ctrl + a; see chapter 3.2.1). The connection of lines is depicted in the Figure 23 and the Figure 24.

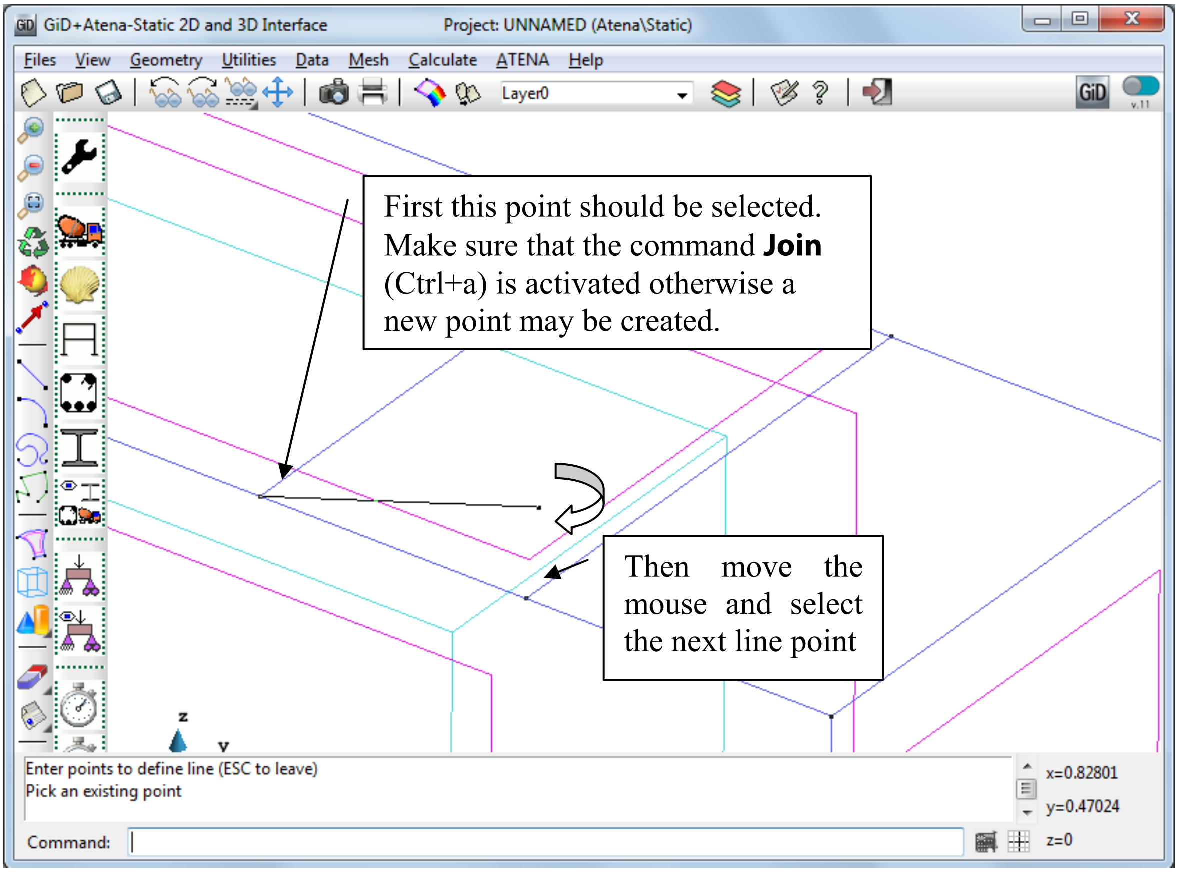


Figure 23: The two lines need to be connected to form a rectangle, i.e. the creation of the bottom line

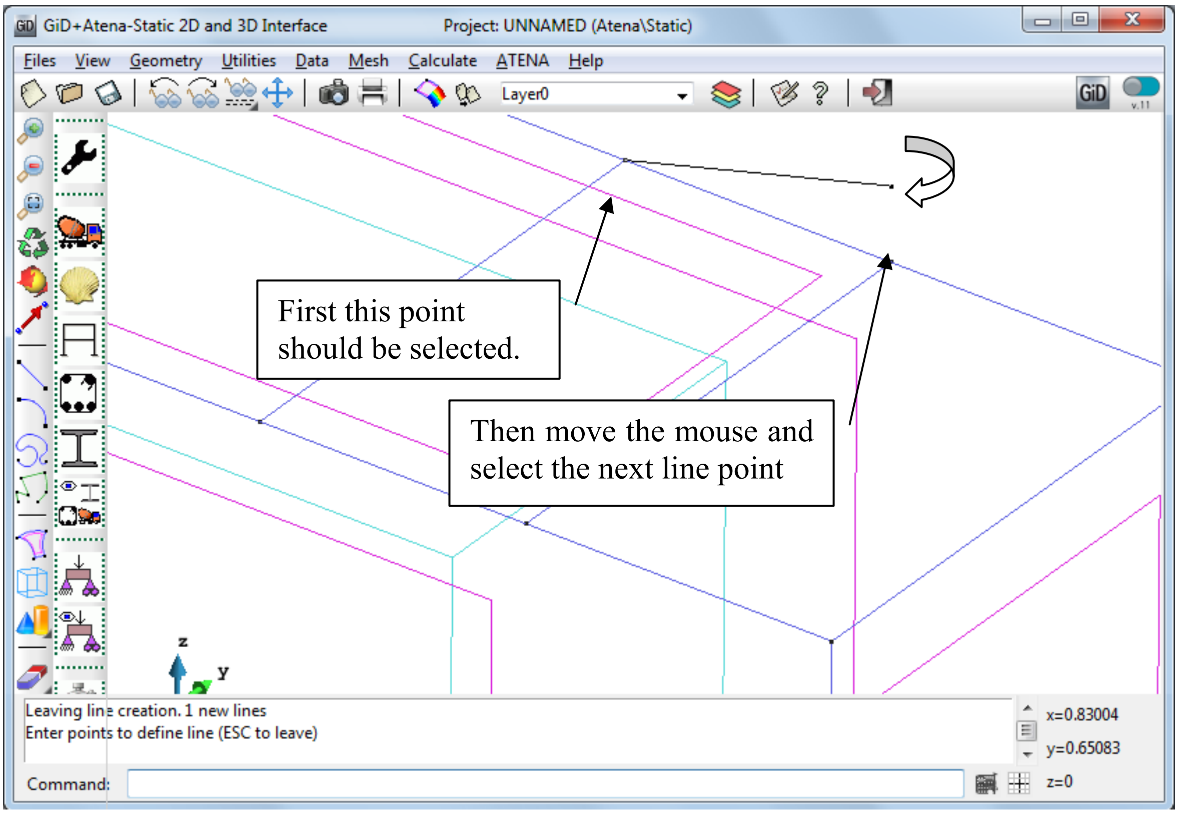
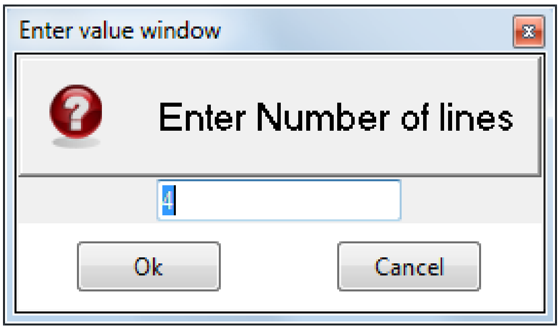


Figure 24: The creation of the top line to finalize the rectangle for the bottom surface of the top plate.

After connecting lines into a rectangle, the surface should be created. For that it is useful to use an automatic surface definition with the command **Geometry | Create | NURBS surface | Automatic**. When this automatic method is used, the program asks for the number of bounding lines (see Figure 25). After definition of this number, the program automatically creates all possible surfaces with the given number of bounding lines.



Parameter input:
Enter Number of lines: 4

Figure 25: The definition of number of bounding lines

After clicking on the **OK** button, the required surface is created (see Figure 26). Then the button **Cancel** should be selected to leave this function.

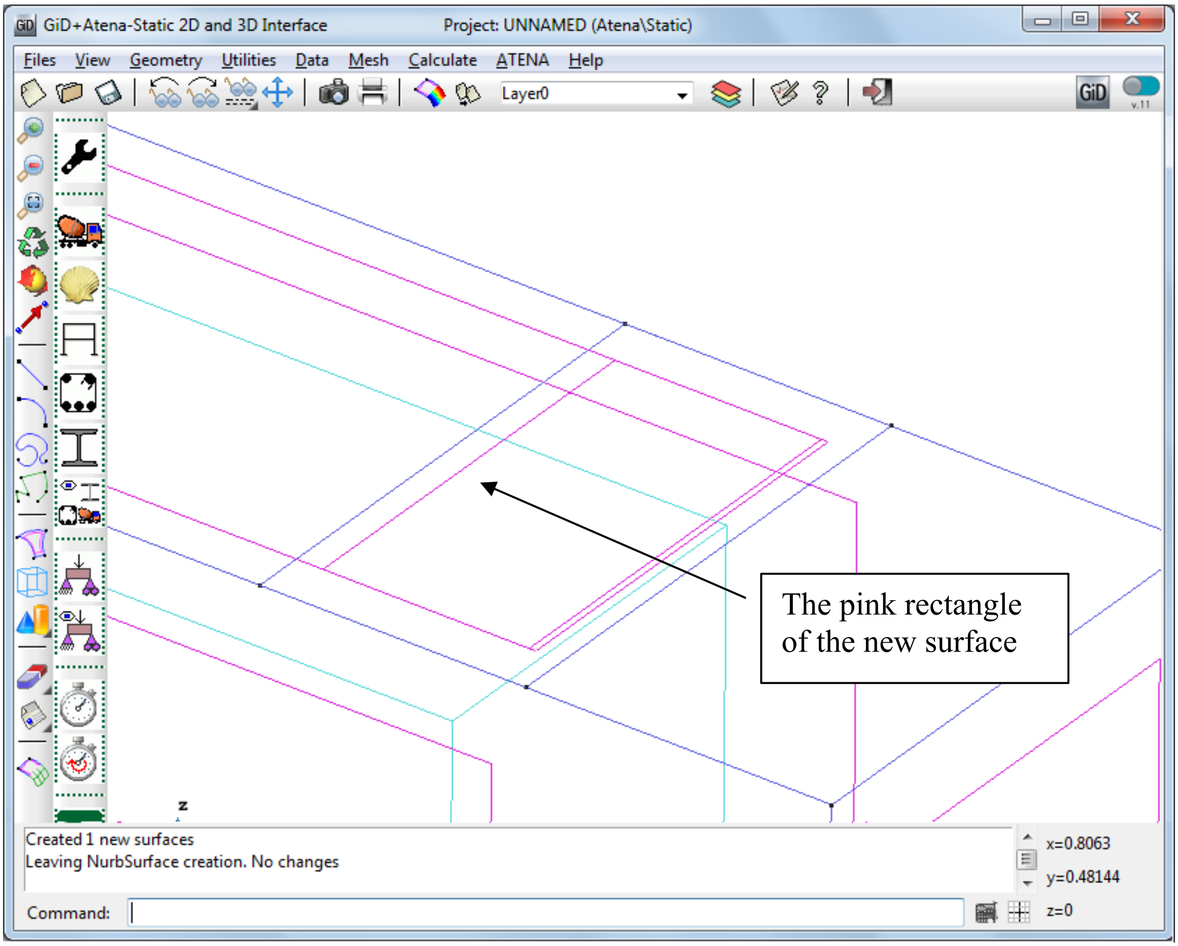
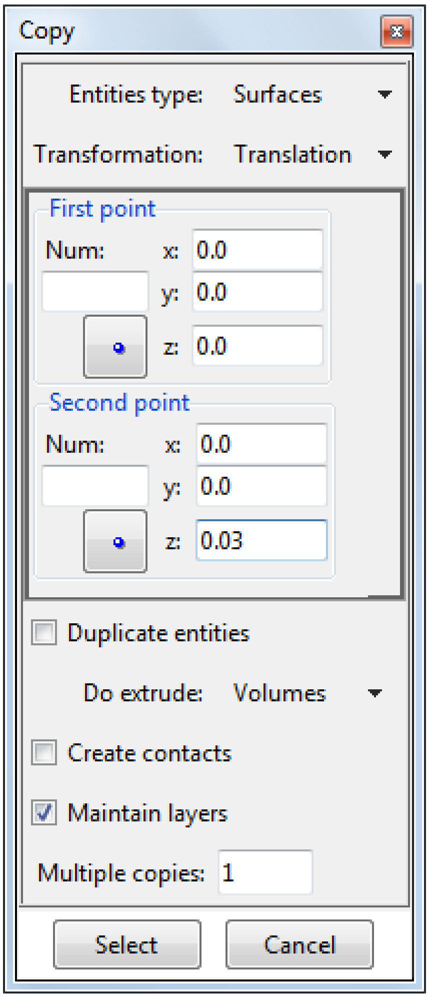


Figure 26: The surface created by automatic surface creation.

The geometry definition of the top plate will be finished by extrusion of the surface. The extrusion is done by the **Copy** command, which appears after selecting item from the main menu **Utilities | Copy**. The height of the steel plate is **0.030 m**. The definition of the extrusion is depicted in the Figure 27. After the definition of all parameters the **Select** button should be pressed. Then the surface required for the extrusion can be selected in the graphical area (see Figure 28). After the selection of surface it is necessary to press **Finish** button to complete the extrusion (see Figure 29).



Parameter input:
Entities type: Surfaces
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.0
z: 0.03
Do extrude: Volumes

Figure 27: The definition of the steel plate extrusion

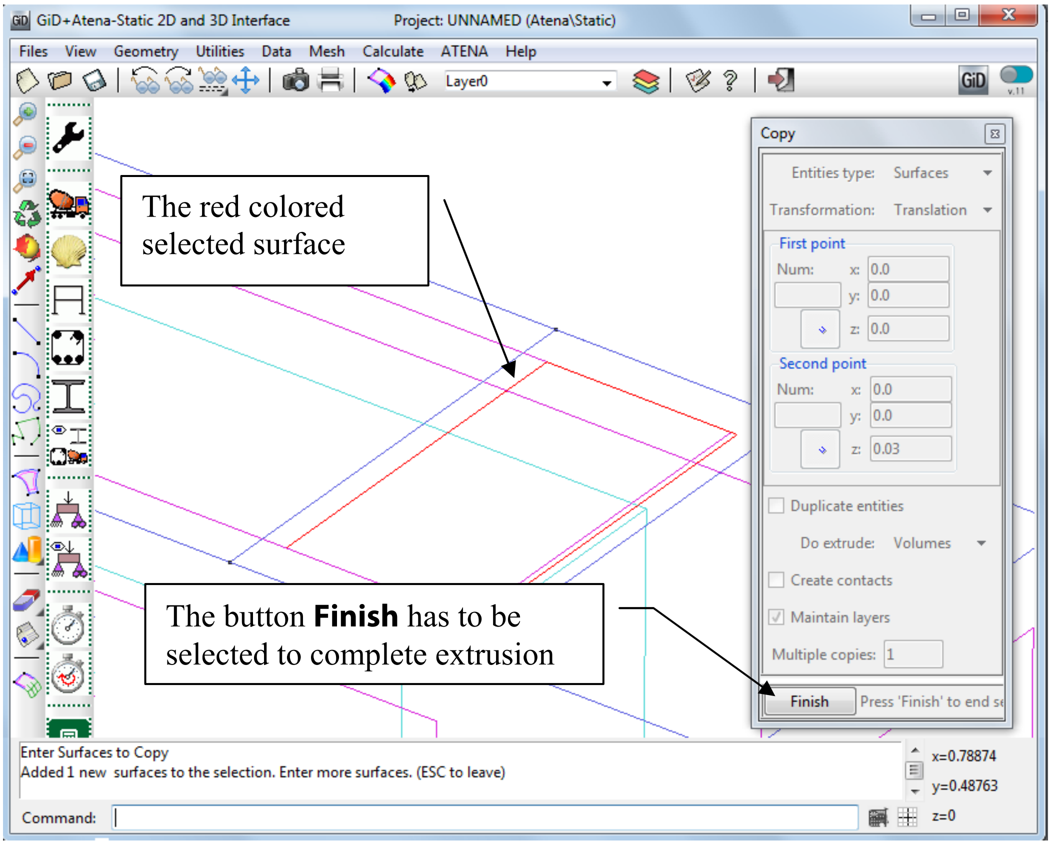


Figure 28: The selection of the surface which should be extruded to obtain steel plate geometry

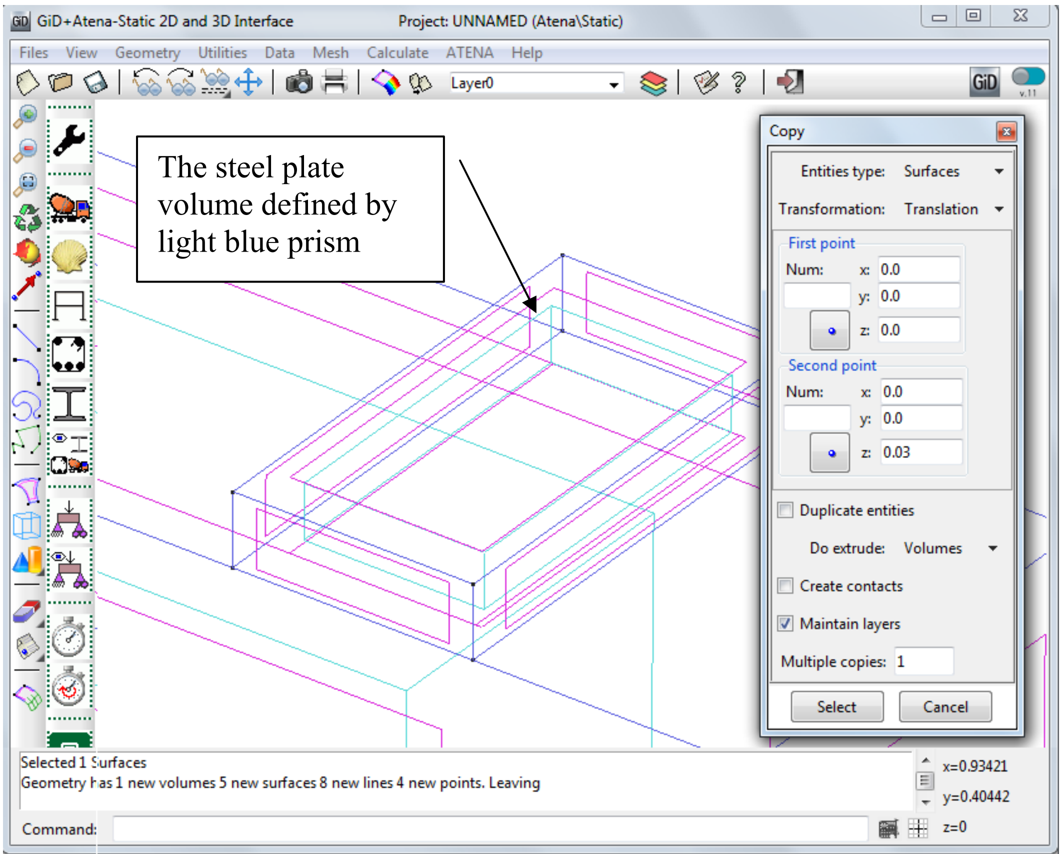
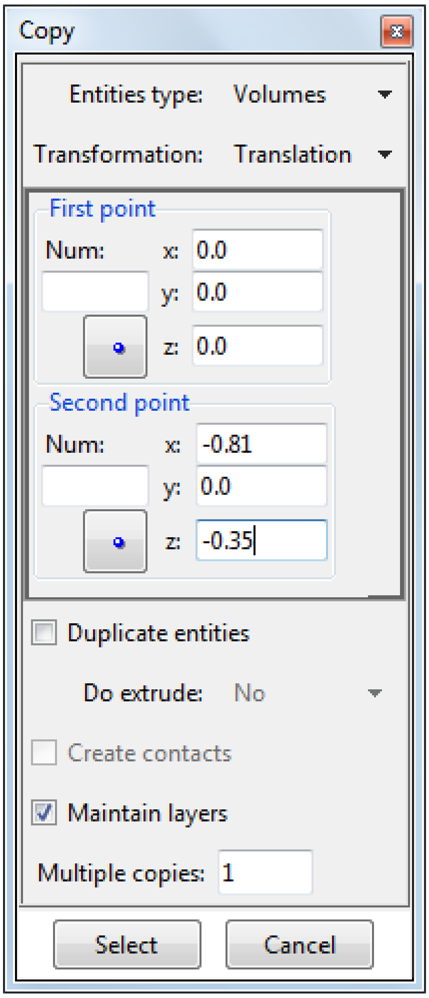


Figure 29: The volume of the top steel plate

3.2.2.2 Bottom Plate

The bottom steel plate will be created by copying of the top plate.

The copy starts by command **Utilities | Copy** in the Main menu. The definition of the extrusion is depicted in the Figure 30. After the definition of all parameters the **Select** button should be pressed. Then the volume required for the translation can be selected in the graphical area (see Figure 31). It is important to select the correct volume representing the top plate. After the selection of volume it is necessary to press **Finish** button to complete the translation (see Figure 32).



Parameter input:
Entities type: Volumes
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: -0.81
y: 0.0
z: -0.35
Do extrude: No

Figure 30: The parameter definition

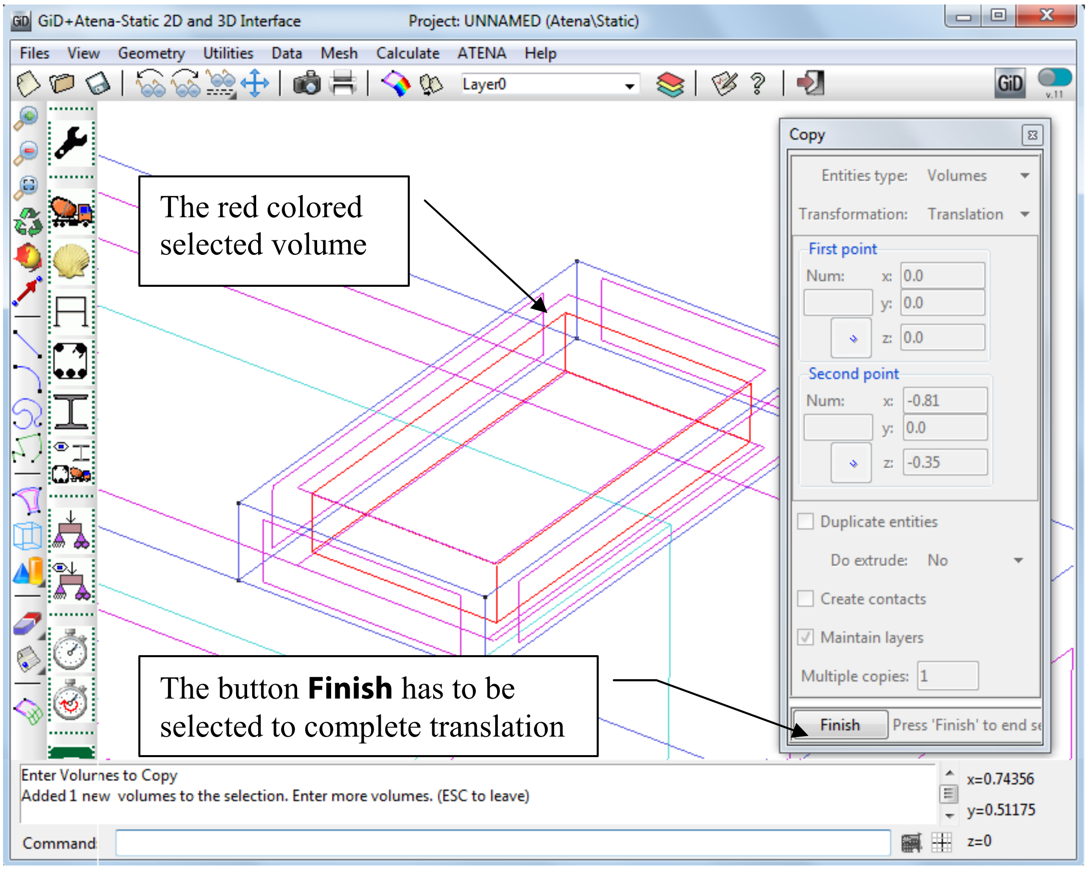


Figure 31: The selection of the volume which should be copied

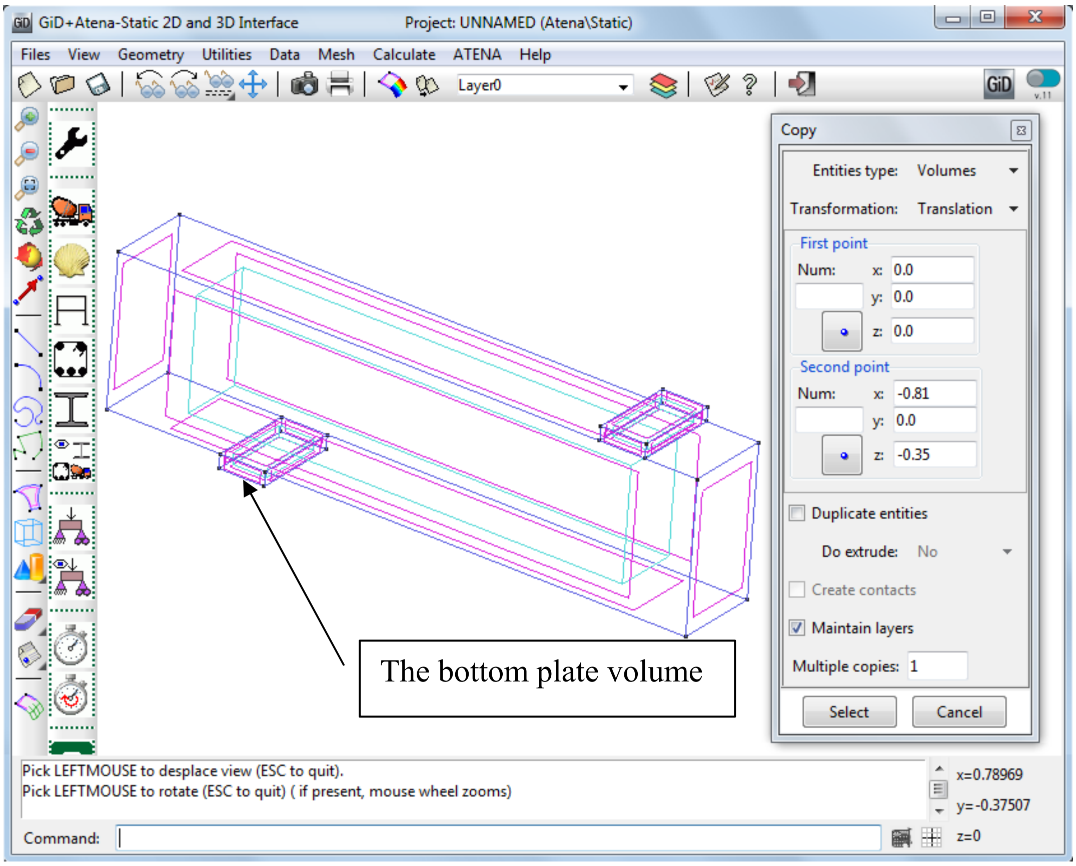
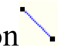


Figure 32: The bottom and top plates

3.2.3 Reinforcement Bars

The geometry of reinforcement bars will be defined only by two lines. The first bar will be created and then the second bar will be copied.

The creation of the first bar starts by clicking the icon  or with the command from the main menu **Geometry | Create | Straight line**. The command line in the bottom of the main window should be used for the coordinates definition. The coordinates of the reinforcement are (**0.05,0.05,0.05**) and (**1.275,0.05,0.05**). See Figure 33.

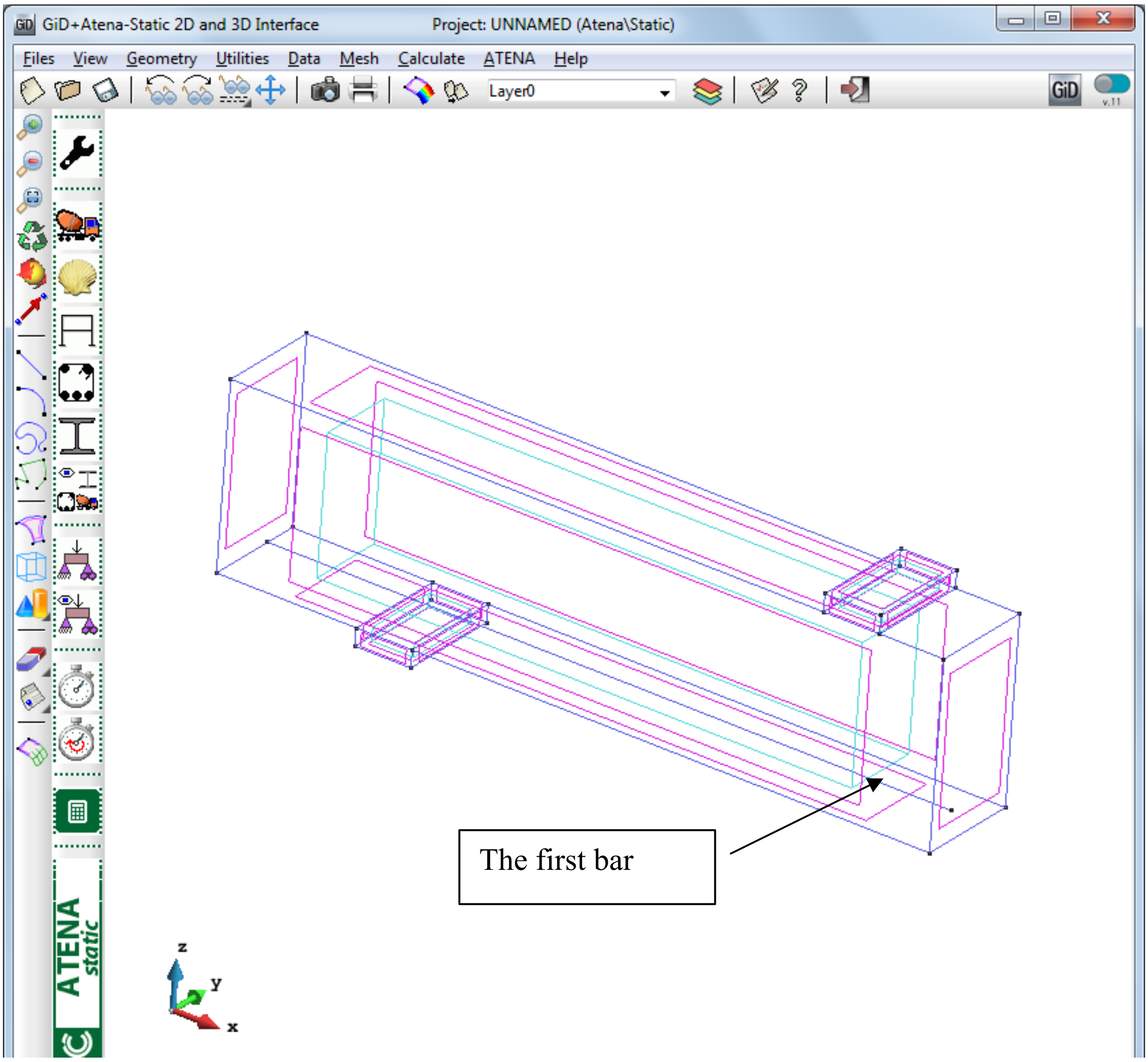
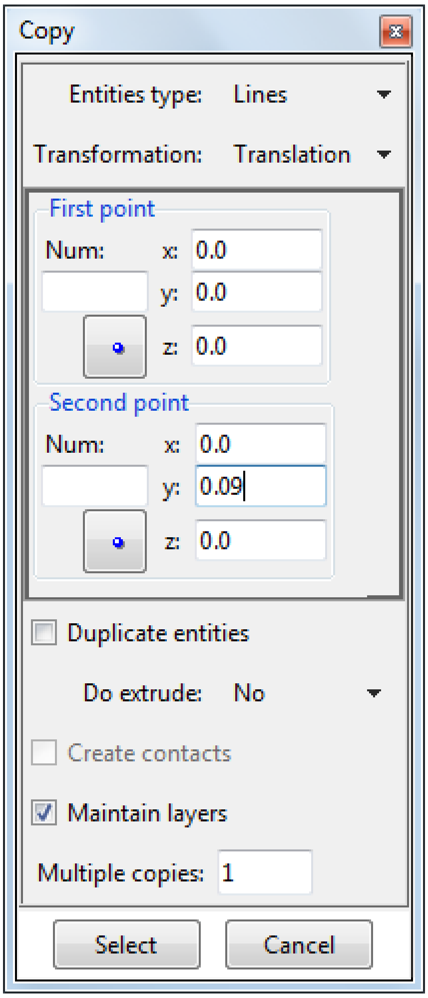


Figure 33: The first reinforcement bar

Parameter input:
Coordinates of the line:
1: 0.05,0.05,0.05
2: 1.275,0.05,0.05

Second reinforcement bar will be created by copying of the first bar. The copy starts by the command **Utilities | Copy** in the main menu. The definition of the translation is depicted in the Figure 34. After the definition of all parameters the **Select** button should be pressed. Then the line required for the translation can be selected in the graphical area (see Figure 35). After the selection of line it is necessary to press **Finish** button to complete the translation (see Figure 36).



Parameter input:
Entities type: Lines
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.09
z: 0.0
Do extrude: No

Figure 34: The parameter definition for the copying of the first bar

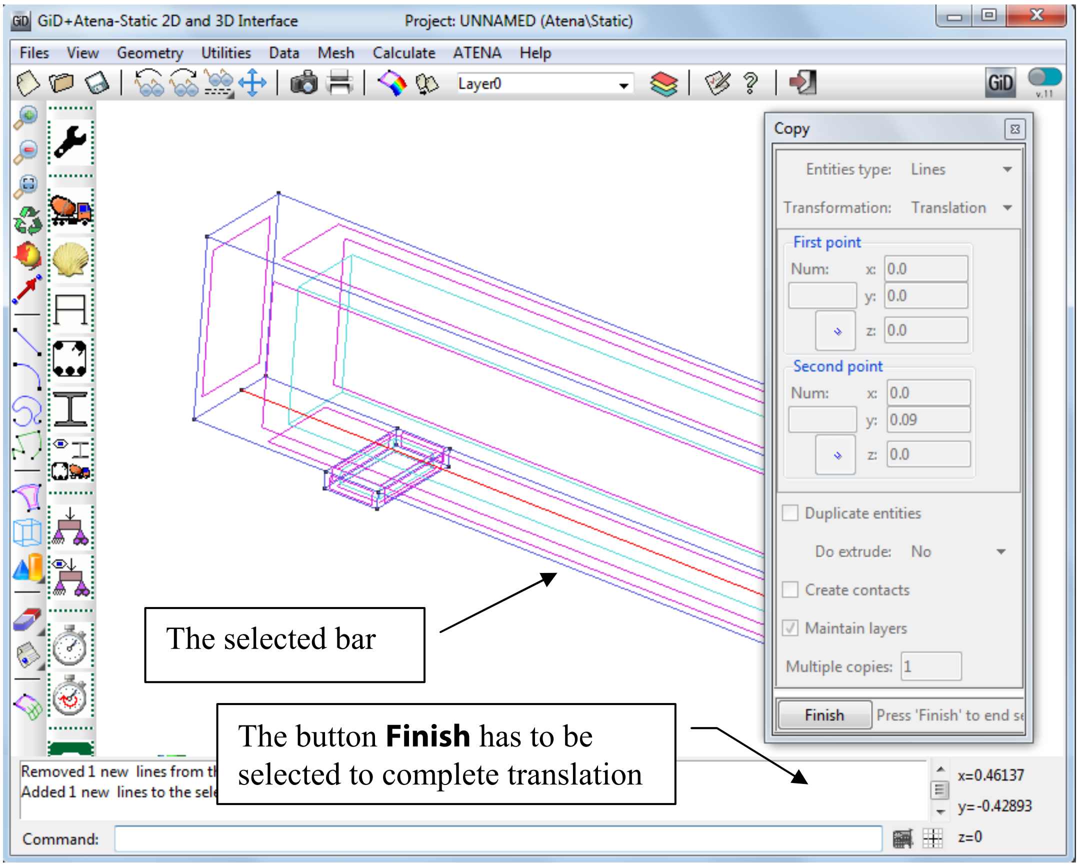


Figure 35: The selection of the first reinforcement bar which should be copied

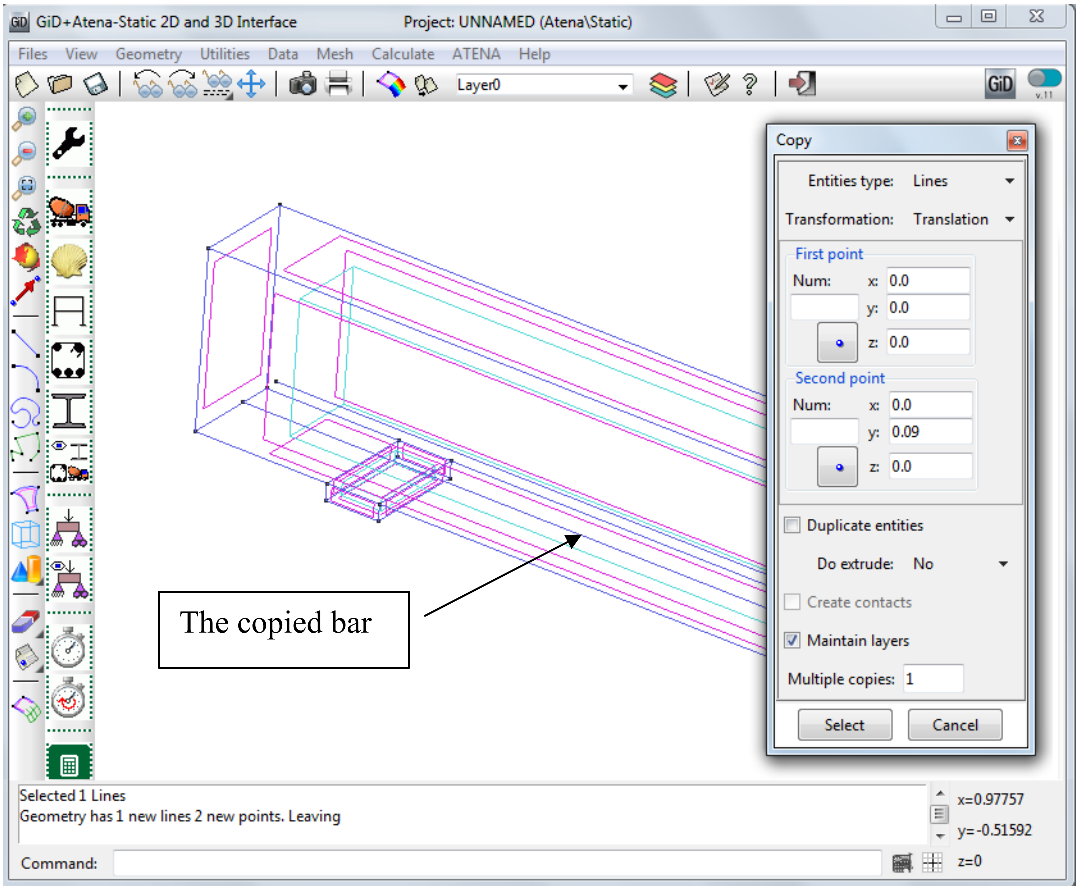


Figure 36: The first and second reinforcement bar

3.2.4 Layers

Layers are useful feature of **GiD**. The individual components of the created geometry can be separated into different layers. In each layer and its components can be selectively displayed, and the user can easily work only with the components of this layer.

In this chapter, three different layers will be created – concrete beam layer, steel plates layer and reinforcement layer.

3.2.4.1 Beam Layer

It is good to start with the definition of concrete beam layer. This is done by the command **Layers**, which appears after selecting **Utilities | Layers** in the main menu. The beam layer will be created by writing **beam** into a window depicted in Figure 37. The new layer will be created after the pressing of the icon . Then the beam layer will appear in the list of the layers.

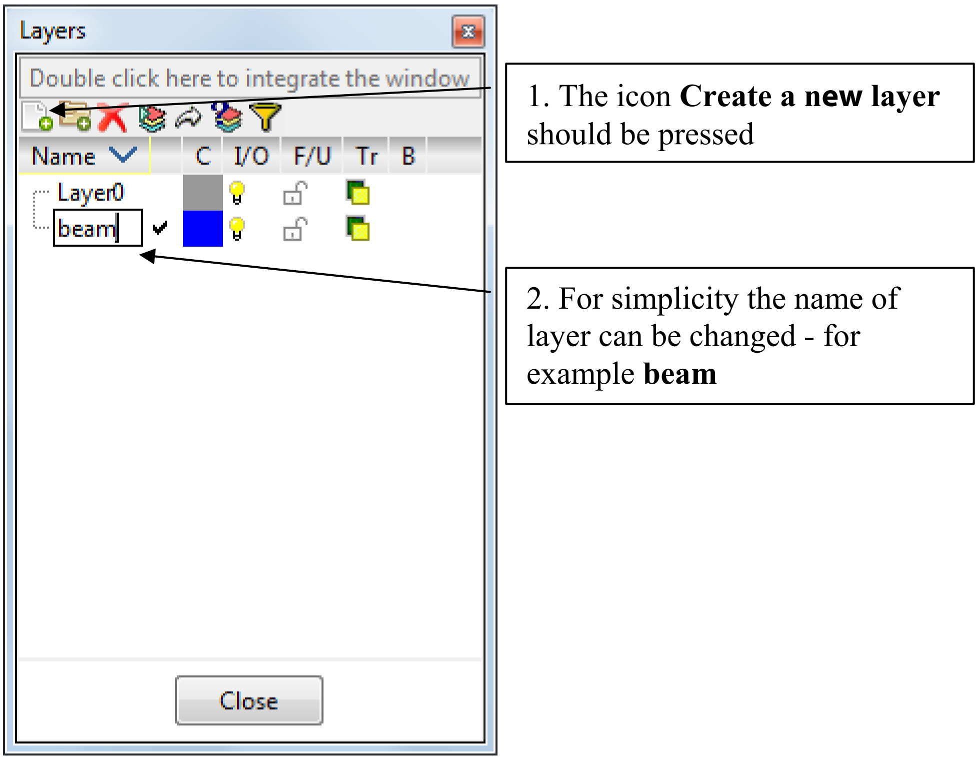


Figure 37: The Layers command

The newly created beam layer is immediately activated. The layer activity is indicated by the sign ✔. The next step is to assign the beam geometry to the beam layer by pressing the icon . Then the pull down menu will open (see Figure 38). The beam geometry contains three types of entities and all of them should be assigned to the beam layer. Therefore the item **Also lower entities** has to be activated and the command **Volumes** should be chosen. After selecting the **Volumes** in the pull down menu, the geometry, which should be send to the beam layer, can be selected (see Figure 39). The pressing of the **Finish** button will complete this command.

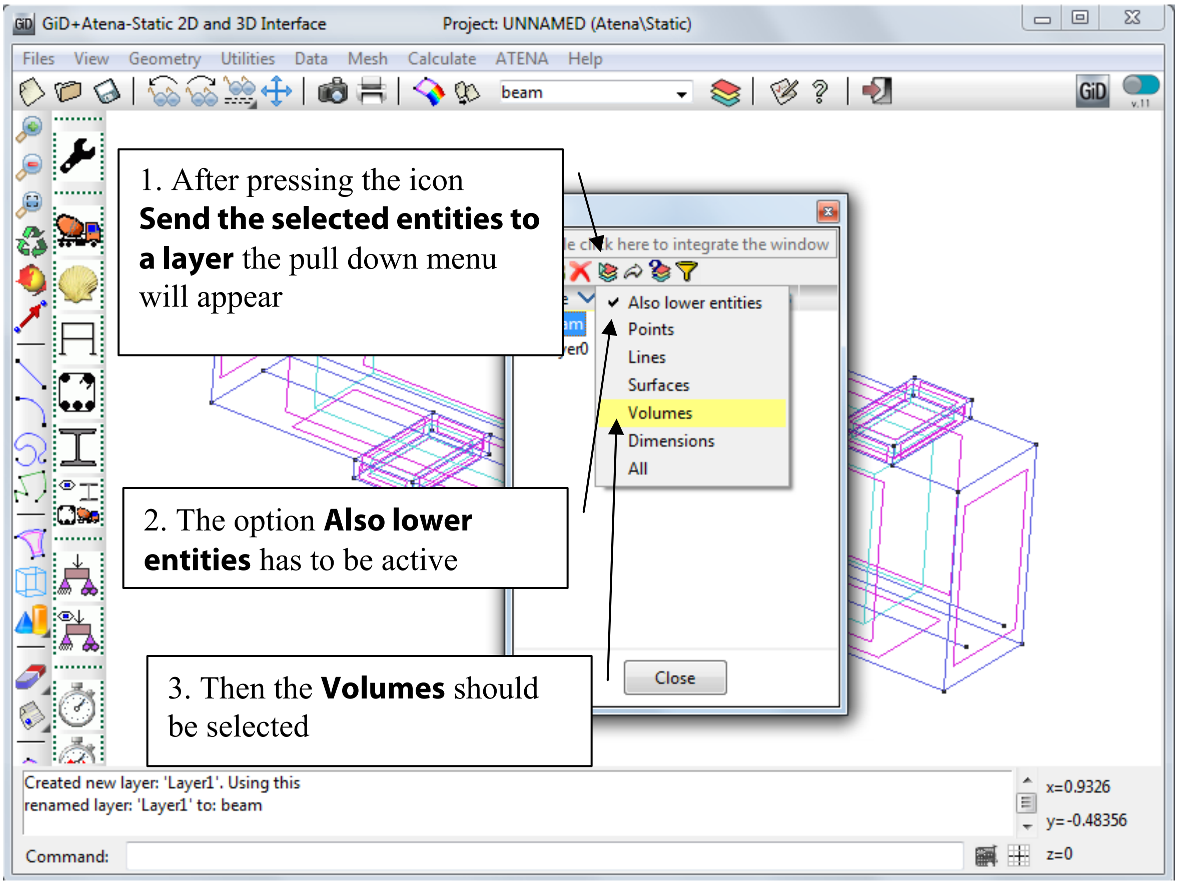


Figure 38: The definition of the Send to command for the beam layer

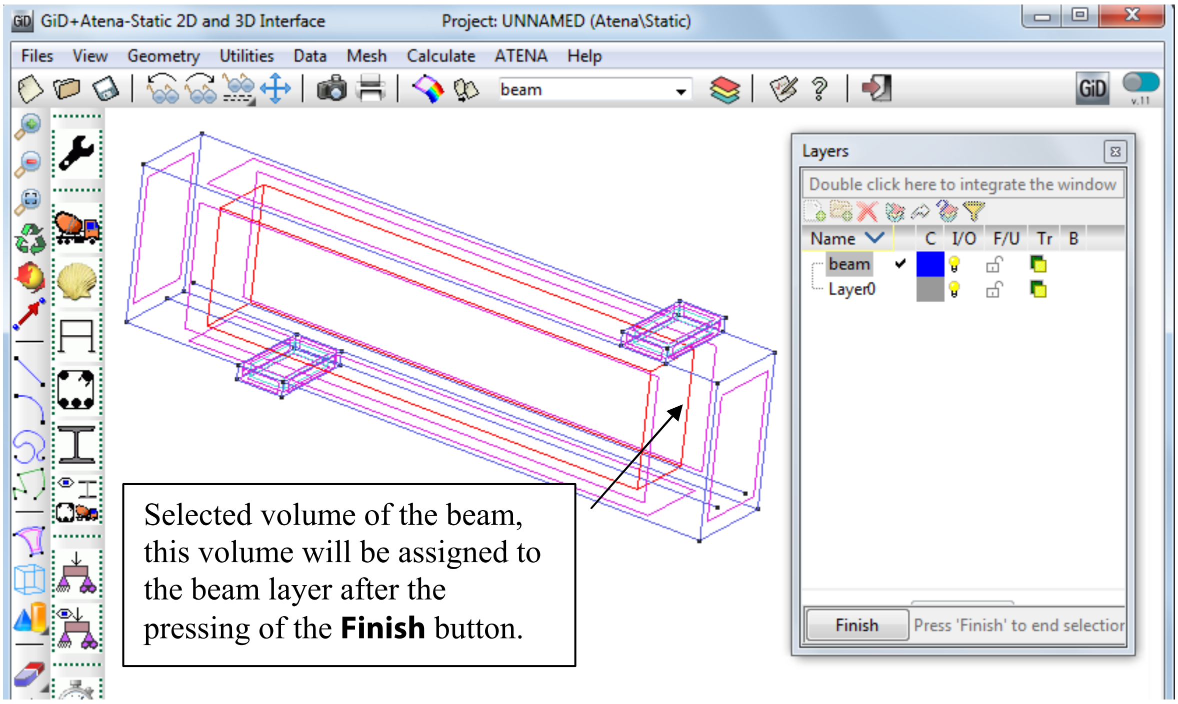


Figure 39: The selection of the volume, which should be sent to the beam layer

The content of the chosen layer can be seen or hidden. The yellow bulb next to the name of the beam indicates the display status of the layer. Also direct clicking on the bulb for an individual layer can switch between the display modes. The Layer0 (the layer which was already there before creating the beam layer) should be selected and then the yellow bulb. The yellow bulb will change to the grey colour . It means that all its content should not be displayed. The Layer0 still contains the geometry of steel plates and reinforcement. Therefore these geometries should disappear in the graphical area after deactivation of the Layer0 (see Figure 40). It should be possible to see only the beam and it assures that the beam geometry was successfully sent to the beam layer.

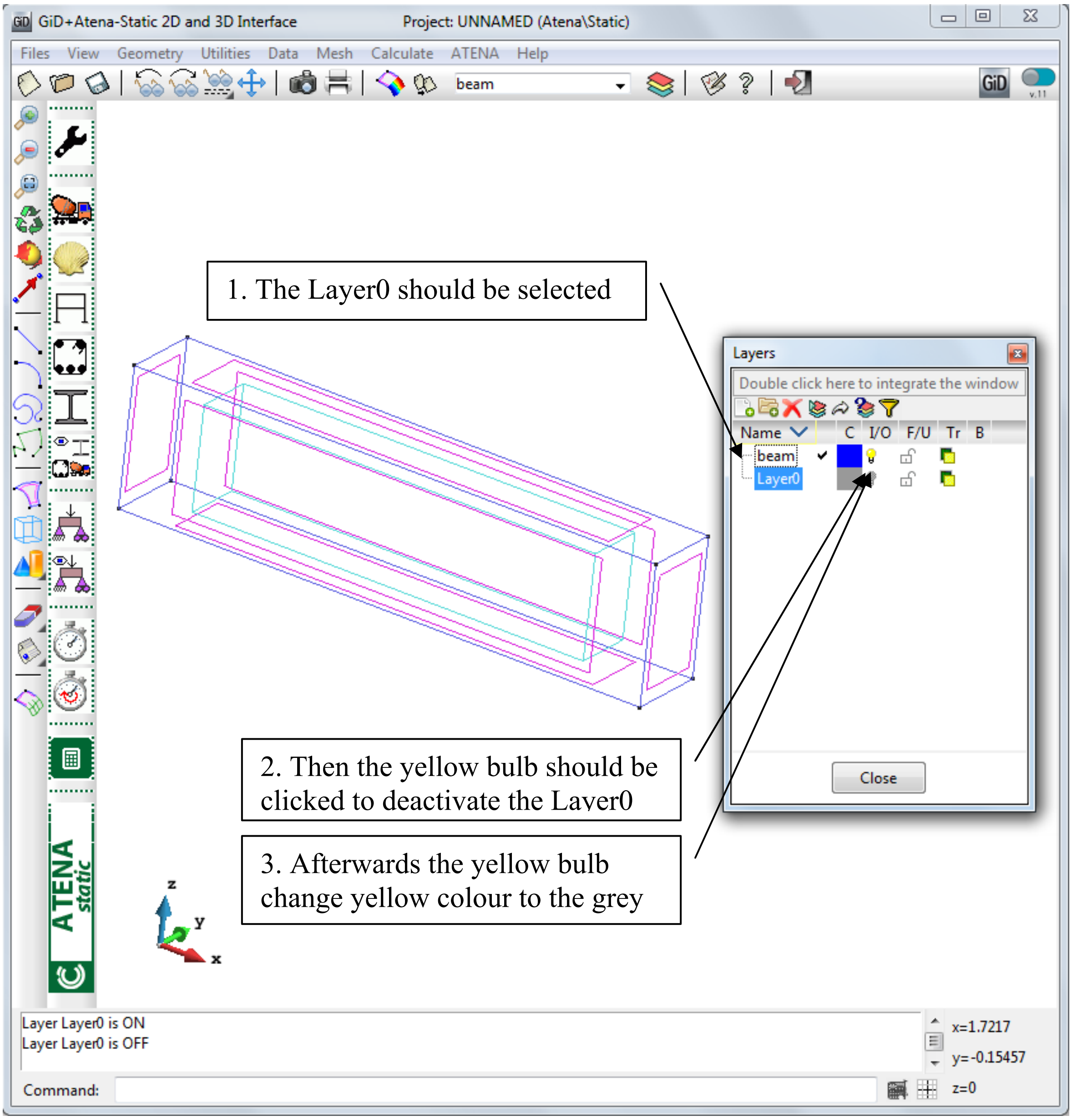


Figure 40: The steel plates and reinforcement geometry will disappear after deactivating of the Layer0

3.2.4.2 Bar Layer

The next step is to create a bars layer. This layer will be created with the same procedure like for the previous beam layer.

First the beam layer should be hidden and Layer0 should be displayed. It is done by selecting the beam layer and pressing the yellow bulb**.**Layer0 is displayed by selecting this layer and then by pressing the grey bulb. Afterwards the beam geometry will disappear and the reinforcement and steel plates will appear in the graphical area (see Figure 41).

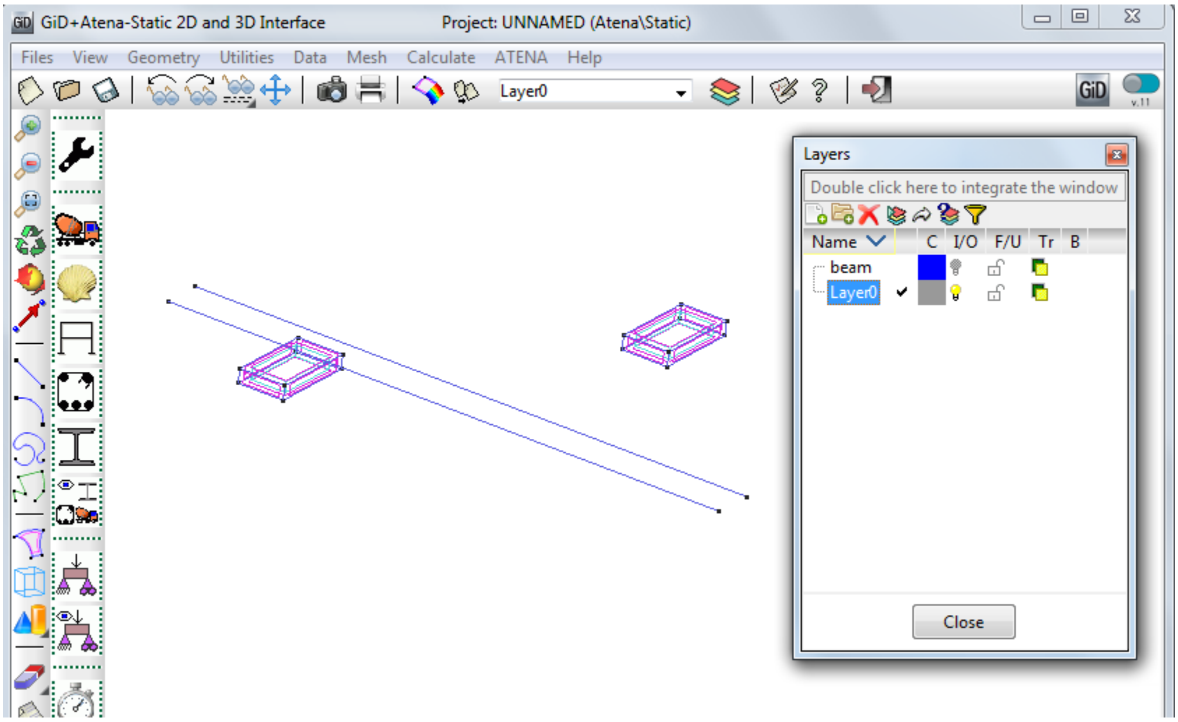


Figure 41: The Layer0 is activated and reinforcement and steel plates will appear in the graphical area

The reinforcement layer is created by pressing the icon . Then the reinforcement layer will appear in the list of layers and the name **bars** can be written. The newly created bars layer is automatically activated. The activation is indicated by the checkbox symbol ✔.

The reinforcement geometry is assigned into the bars layer by pressing of the icon . Then the pull down menu will open (see Figure 42). The reinforcement geometry contains two types of entities and all of them should be moved into the bars layer. Therefore the item **Also lower entities** has to be activated and the command **Lines** should be chosen.

After selecting the **Lines** in the pull down menu the geometry, which should be send to the bars layer, can be selected (see Figure 43). **Finish** button completes the layer assignment.

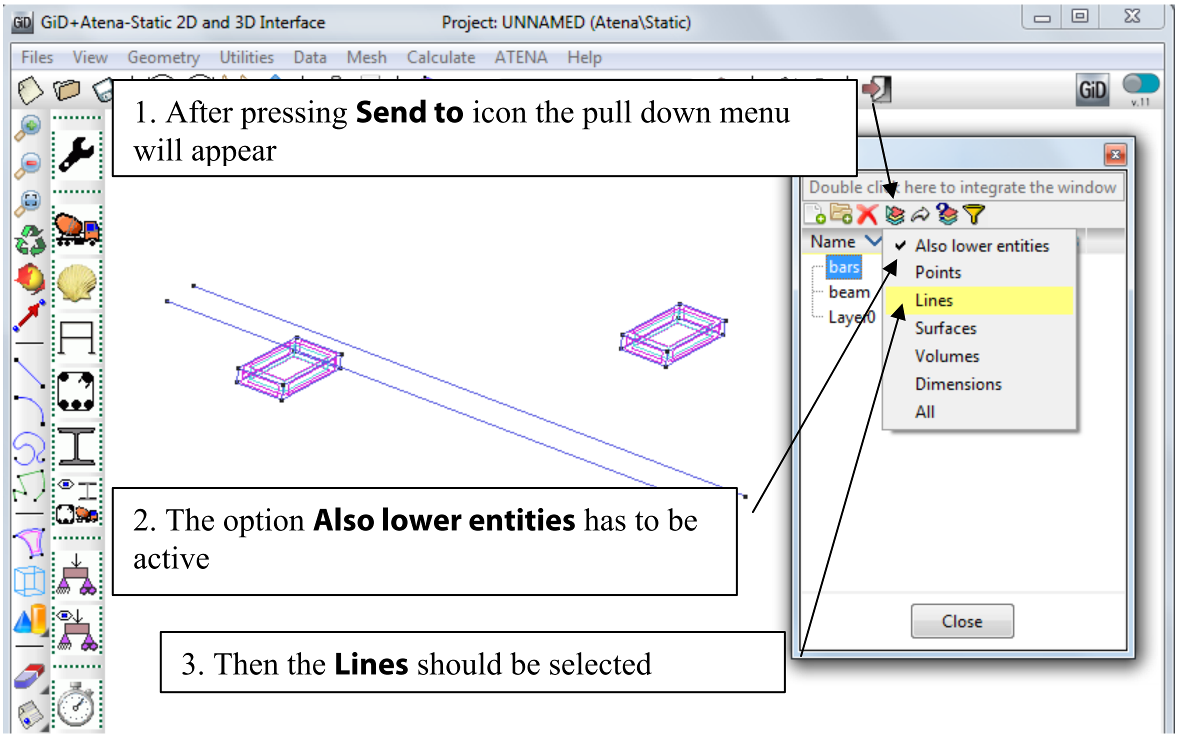


Figure 42: The definition of Send to command for the reinforcement layer

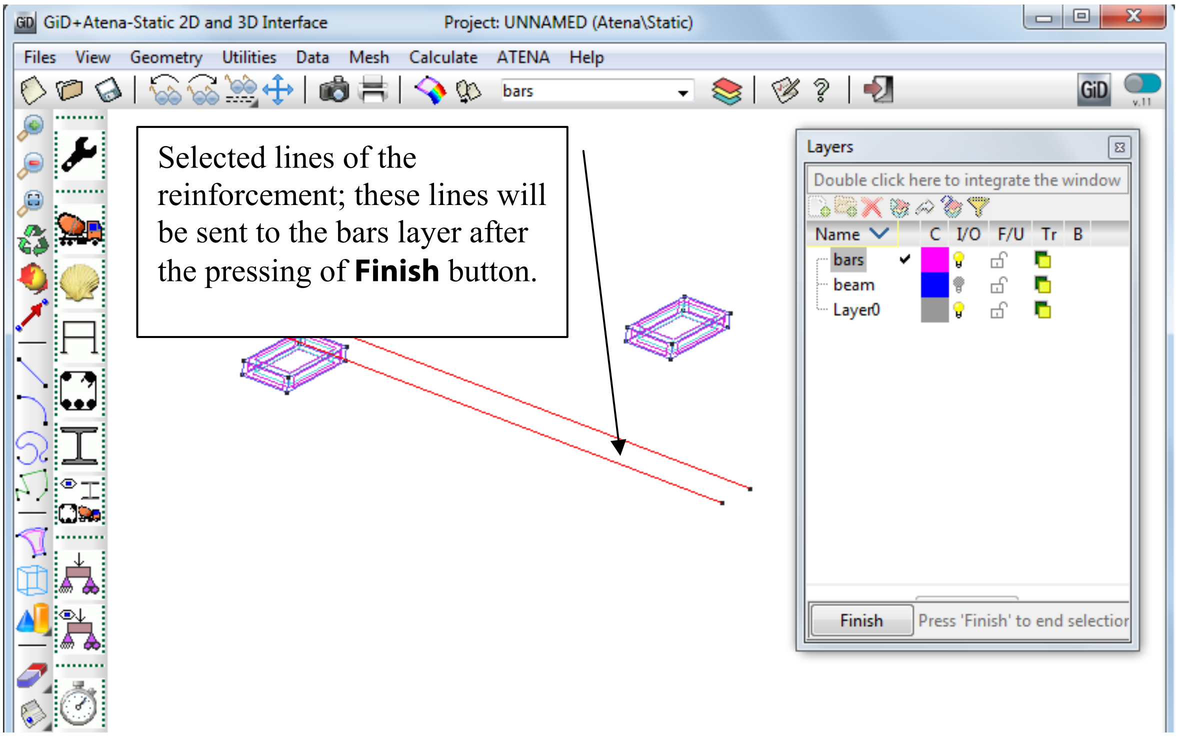


Figure 43: The selection of the lines, which should be sent to the bars layer

3.2.4.3 Plate Layer

It is useful to deactivate of the display of the bars layer by click the appropriate yellow bulb (see Figure 44). The reinforcement lines should disappear.

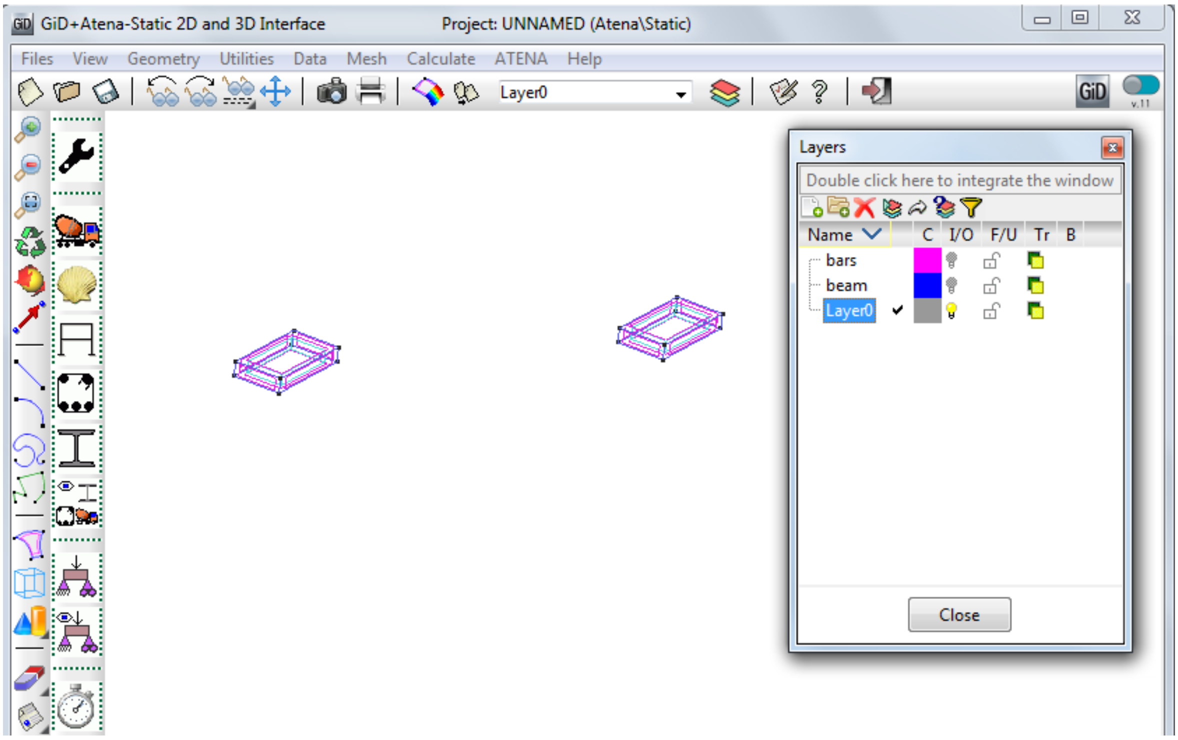


Figure 44: The reinforcement disappear after deactivating of the reinforcement layer

The last step is to create a plate layer. Like in previous two layers it is done by pressing the icon and name the new layer for example **plates**. Then the plate layer will appear in the list of layers. The newly created plate layer is automatically activated. The activation is indicated by the checkbox symbol ✔.

The moving of the steel plate geometry into the plate layer can be started by pressing of the icon. Then the pull down menu will open (see Figure 45). The reinforcement geometry contains two types of entities and all of them should be moved into the bars layer. Therefore the option **Also lower entities** has to be activated and the command **Volumes** should be chosen.

After selecting the **Volumes** in the pull down menu, the geometry, which should be assigned to the bar layer, can be selected (see Figure 46). **Finish** button will complete this command.

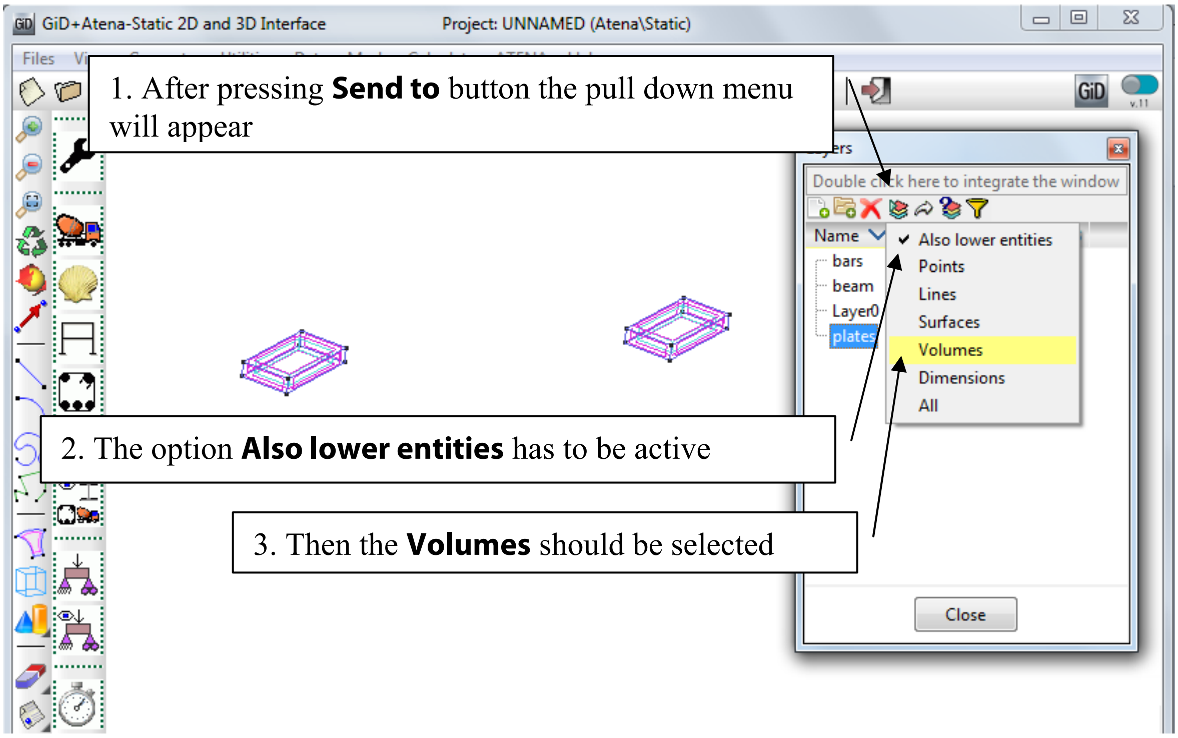


Figure 45: The definition of Send to command for the plates layer

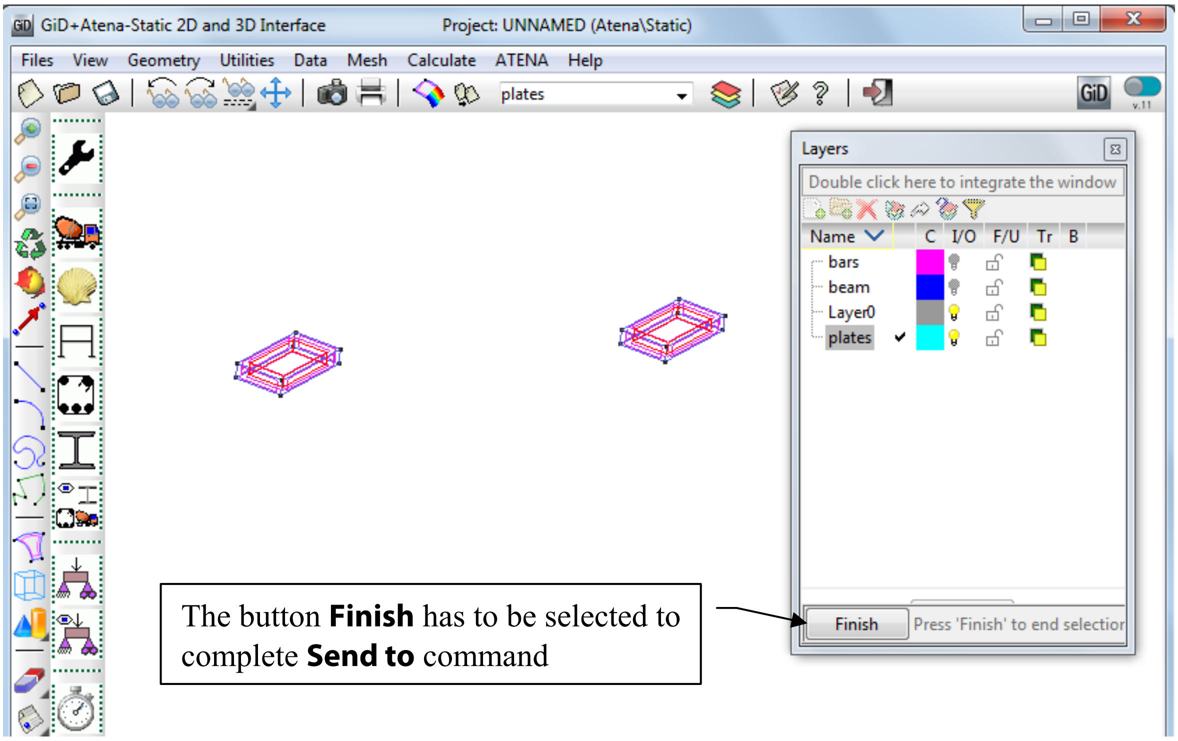


Figure 46: The selection of the volumes, which should be sent to the plates layer

If the display of the plate layer is deactivated, the volumes of the steel plates should disappear. Deactivation is done by selecting the plate layer in the list of layers and then pressing the yellow bulb (see Figure 44).

The Layer0, which is now active, is empty. It does not contain any geometry and therefore this layer can be deleted. It is done by selecting this Layer and by pressing the icon . After that the Layer0 will be deleted (see Figure 47).

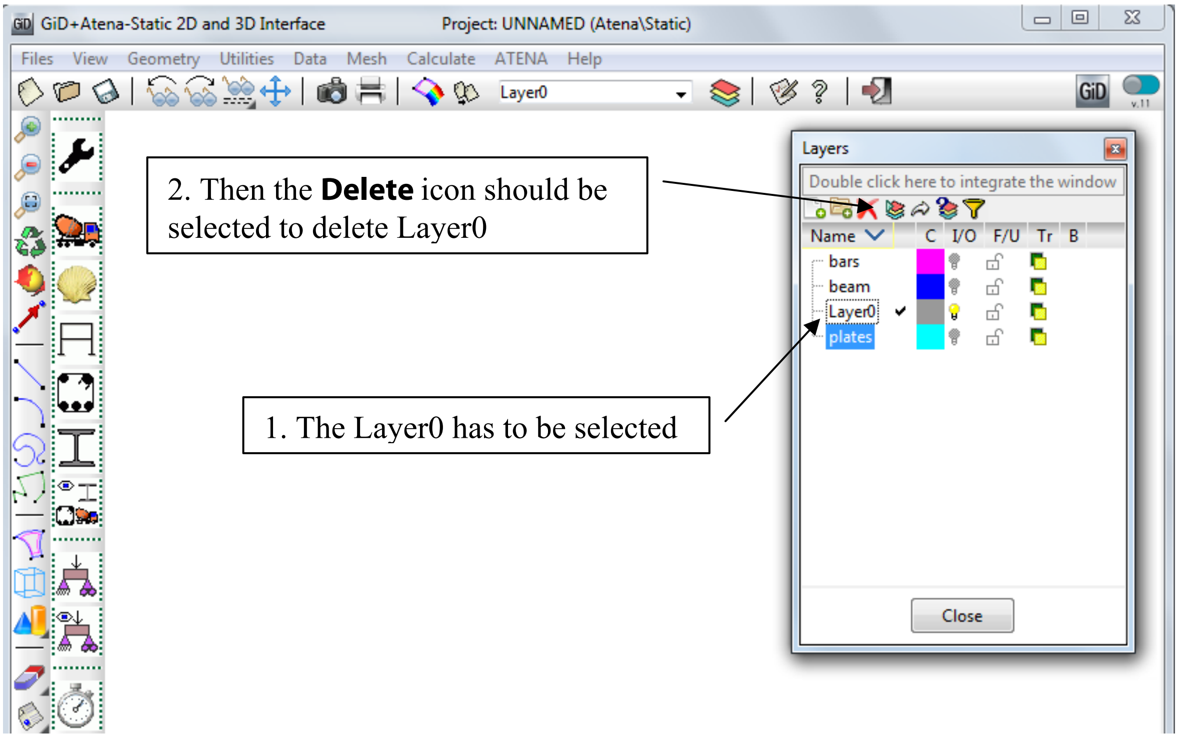


Figure 47: After deactivation of the plates layer the graphical area will stay empty. The Layer0 is active and it does not contains any geometry therefore it can be deleted.

It is recommended to try to display each layer separately to verify that they contain all required geometry. The correct results are shown in Figure 48, Figure 49 and Figure 50.

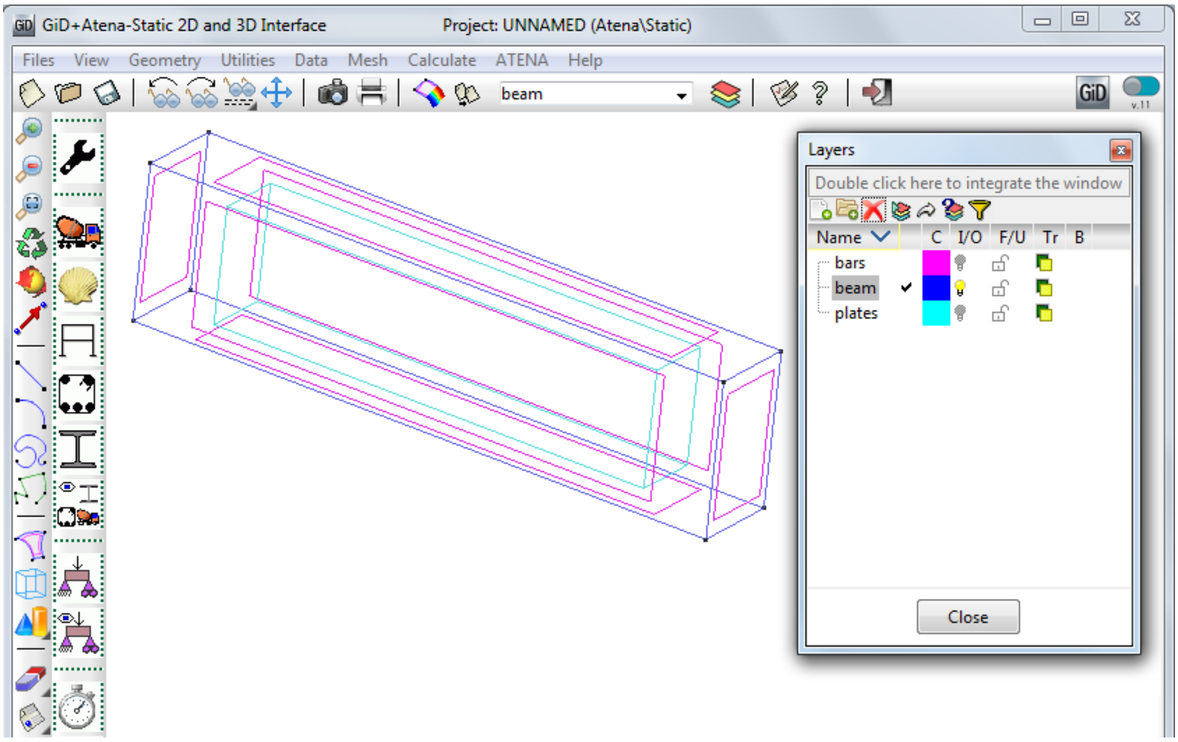


Figure 48: The displayed beam layer – contains beam volume

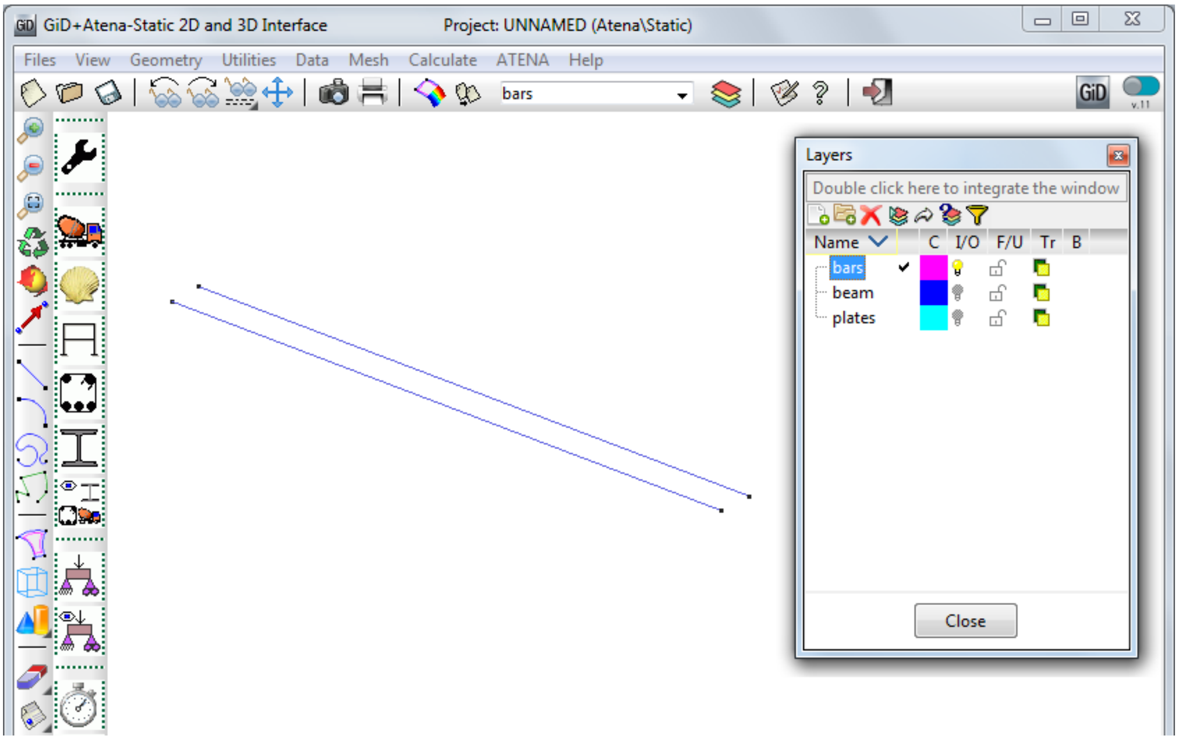


Figure 49: The displayed bar layer – contains reinforcement lines

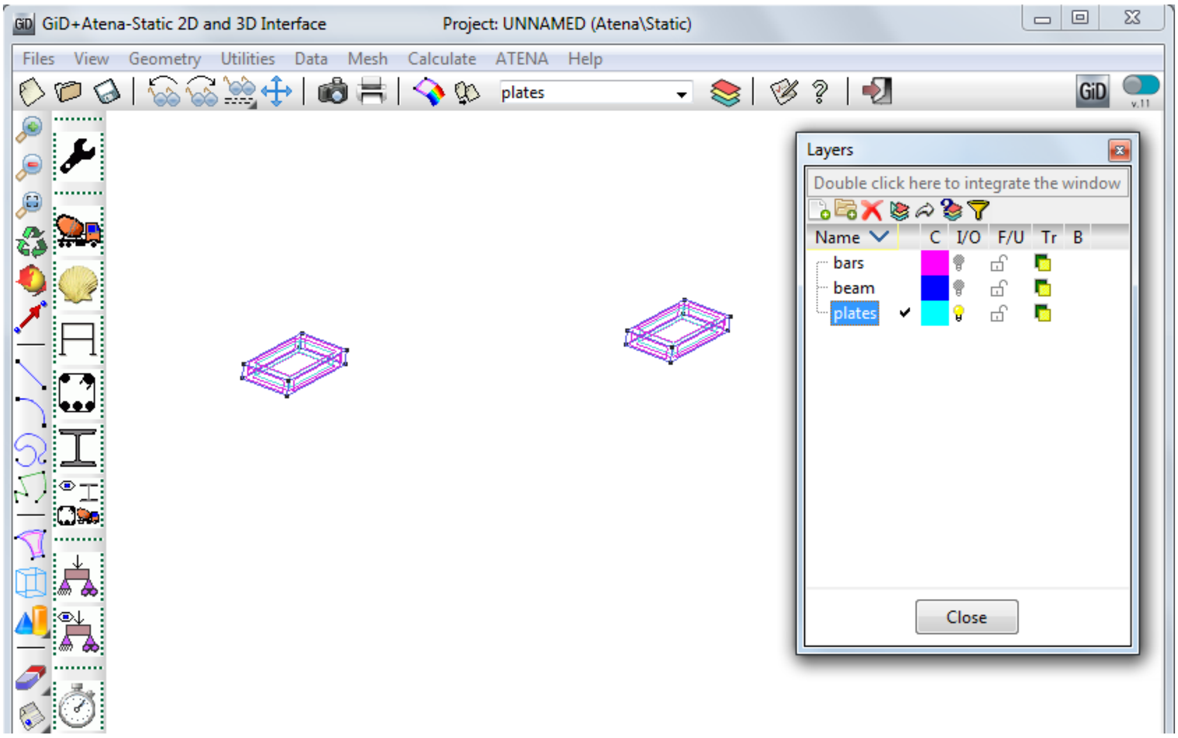


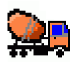
Figure 50: The displayed plate layer – contains plate volumes

3.3 *Material Parameters*

This tutorial example contains three entities, which are made from three different materials. These three entities are concrete beam, steel plates and reinforcement bars. In this chapter the characteristics of materials will be defined and then the material will be assigned to an appropriate geometrical entity.

3.3.1 *Concrete Beam*

Before definition of the concrete beam material it is good to display only the beam layer.

The material definition of the beam starts by selecting the icon  or with the command **Data | Materials | SOLID Concrete** in main menu (see Figure 51).

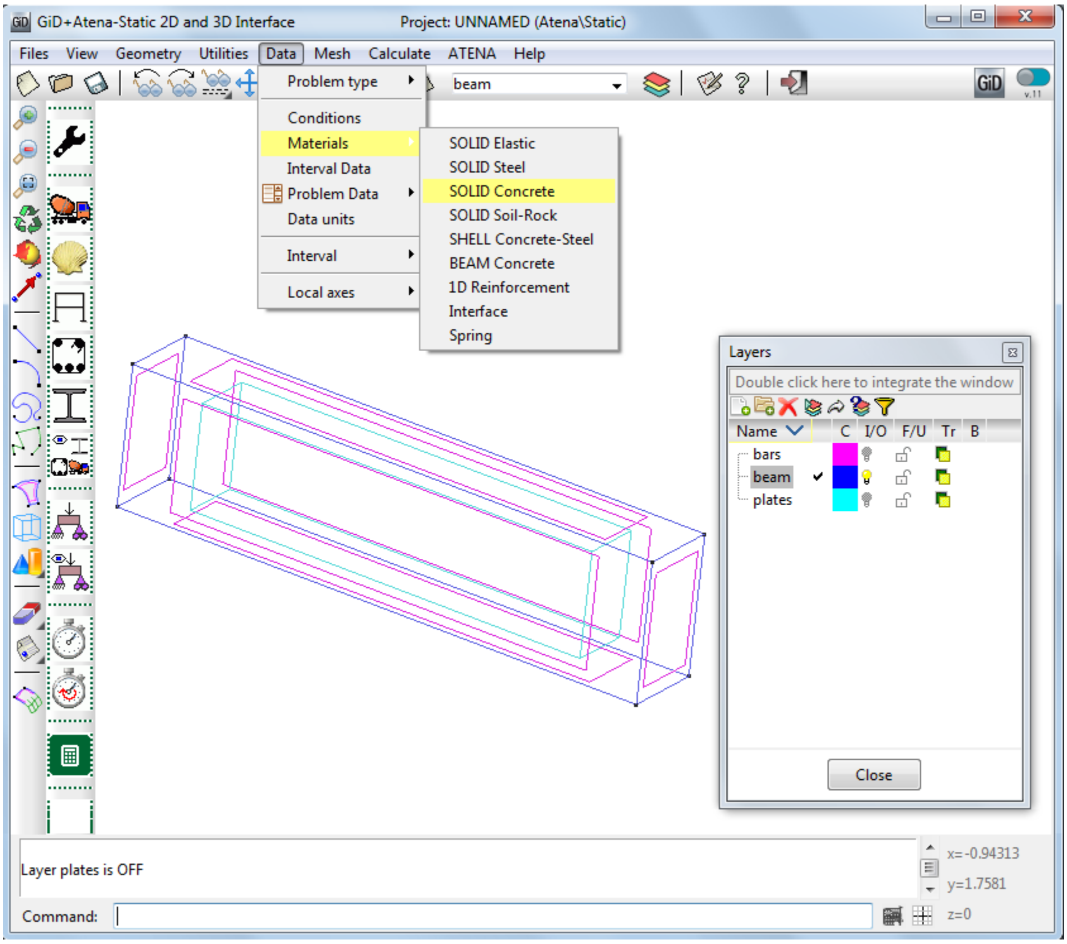


Figure 51: The selection of the command for the definition of the concrete material

After the selection of this command, the window for the definition of the SOLID Concrete appears (see Figure 52).

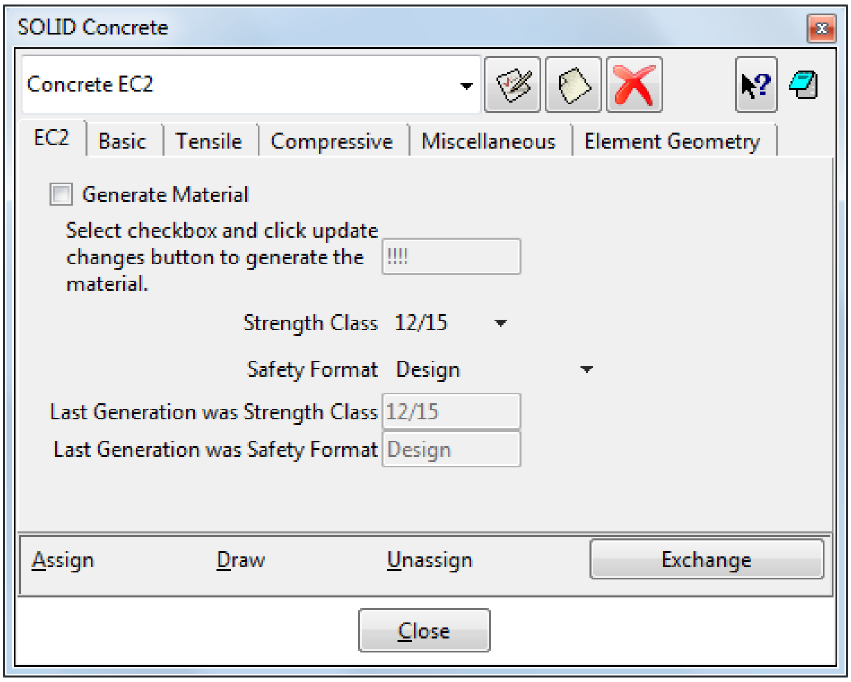


Figure 52: The window for the definition of the SOLID Concrete

First, it is important to copy material definition of the already existing material and save it under a new name. In this case, the new name shall be **Beam** and it should be created based on the predefined material Concrete EC2. After the selection of the predefined material (i.e. material Concrete EC2) the icon New SOLID Concrete  should be selected.

The selection of this material and selection of the New SOLID Concrete icon are depicted in the Figure 53. After the selection of the New SOLID Concrete icon the new window for the definition of the new material name will appear (see Figure 54). Here the name **Beam** should be written, and **OK** button is used to complete this command.

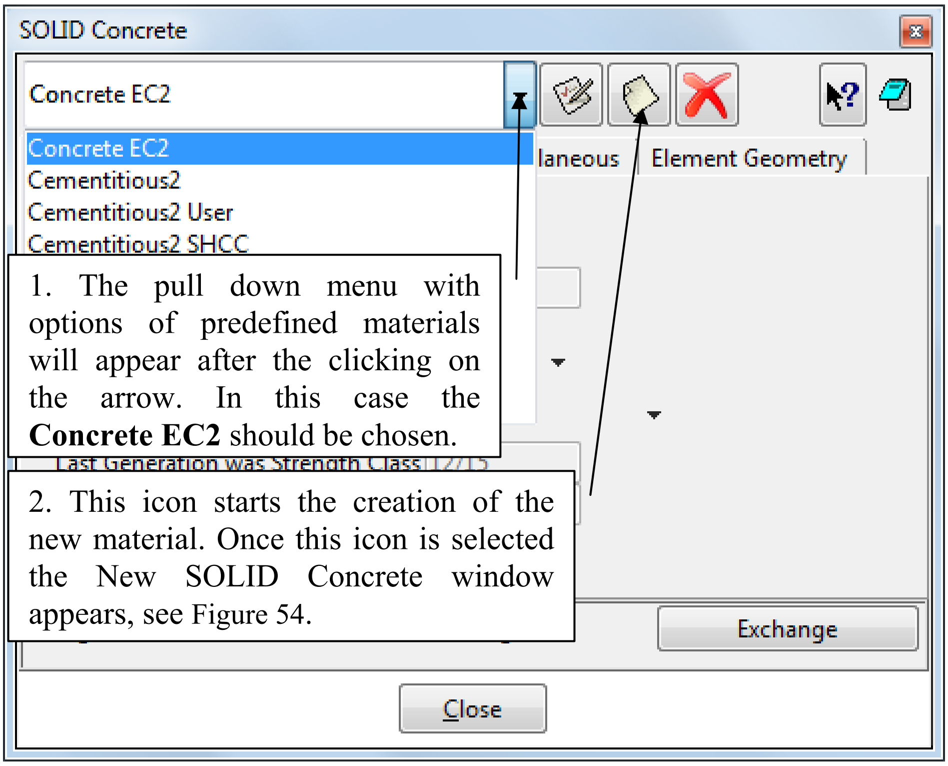


Figure 53: Description of the new material creation

Figure 54: The window for the definition of the New SOLID Concrete

Parameter input:
Enter new SOLID Concrete name:
Beam

When the new material is created, its name will be offered in the pull down menu (see Figure 55). This new material should be selected, and then its parameters can be changed.

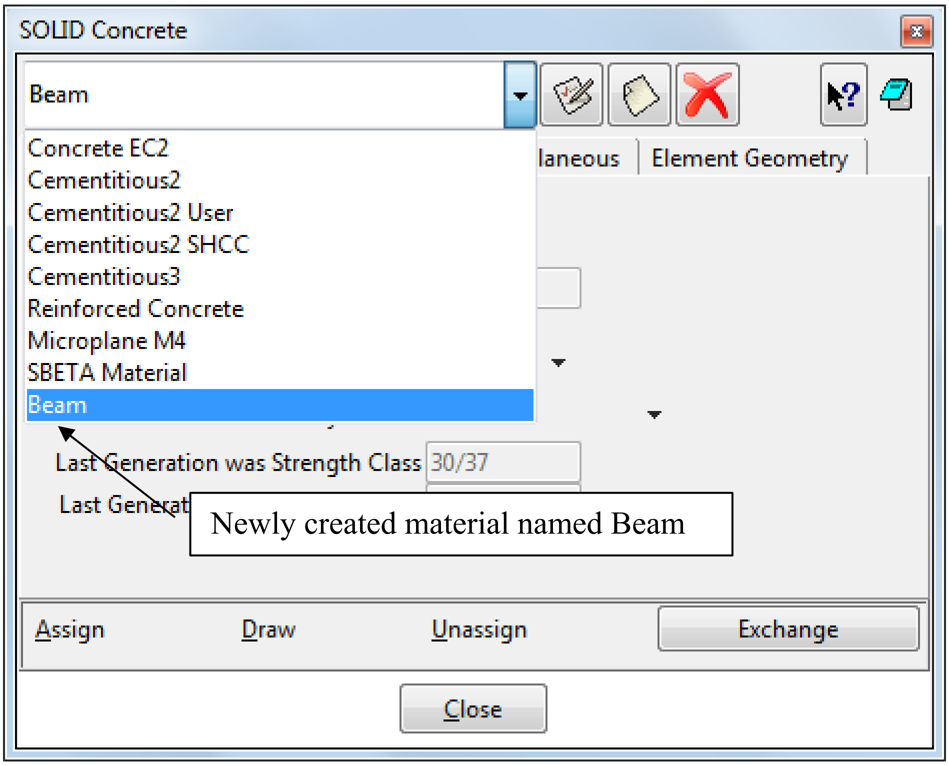


Figure 55: The selection of the New SOLID Concrete material

The parameters of the new material Beam are predefined according to Eurocode 2. In this example it is necessary to have parameters of concrete class **20/25** and Safety Format should be **Mean**. It can be done by selecting this class parameter and safety format in the material window. The process of the class and safety format definition is depicted in the Figure 56. It is very important to select checkbox Generate Material otherwise no parameters will be updated. All parameters definition is completed by clicking on the Update Changes icon .

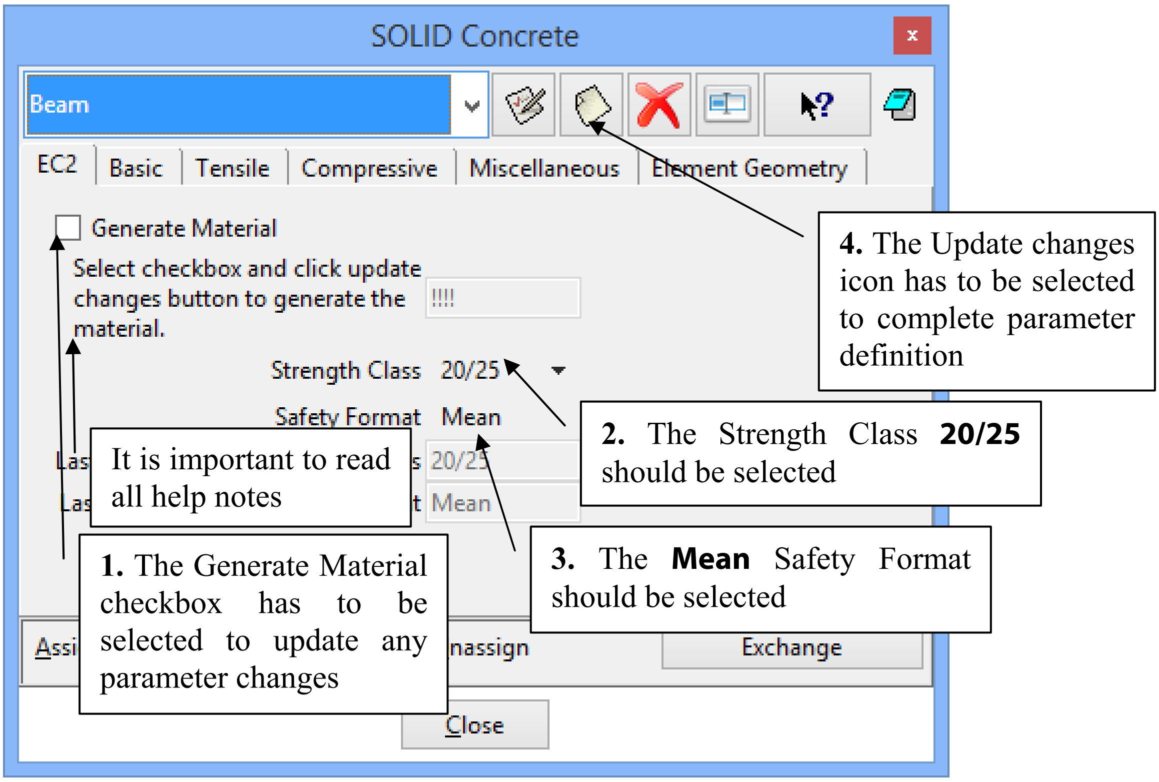


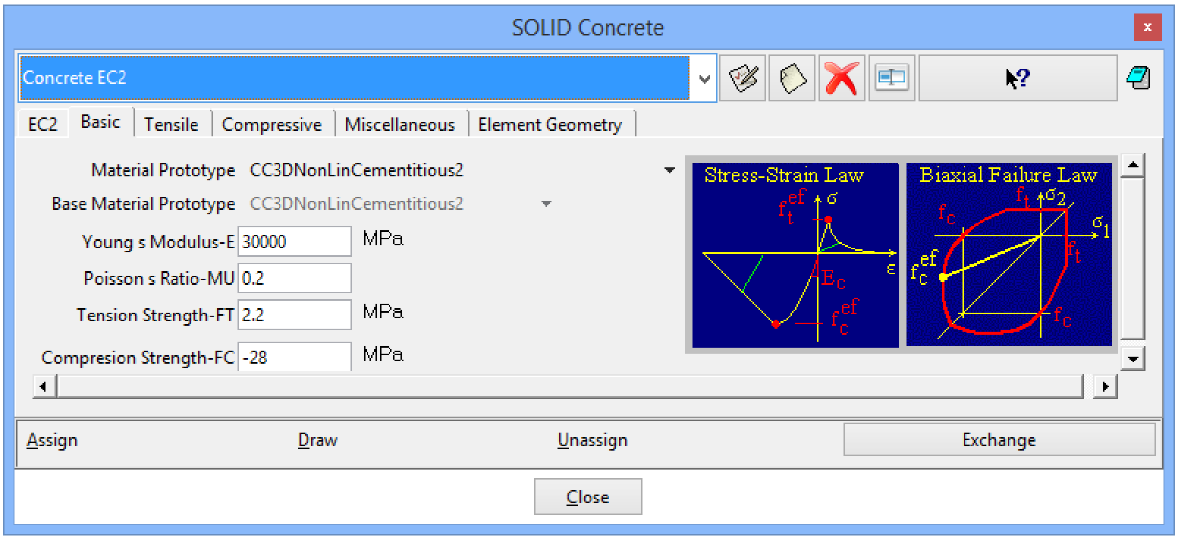
Figure 56: The description of the class definition

After updating of EC 2 parameters, the rest of parameters will change automatically. The following pictures show the generated material parameters of concrete class 20/25.

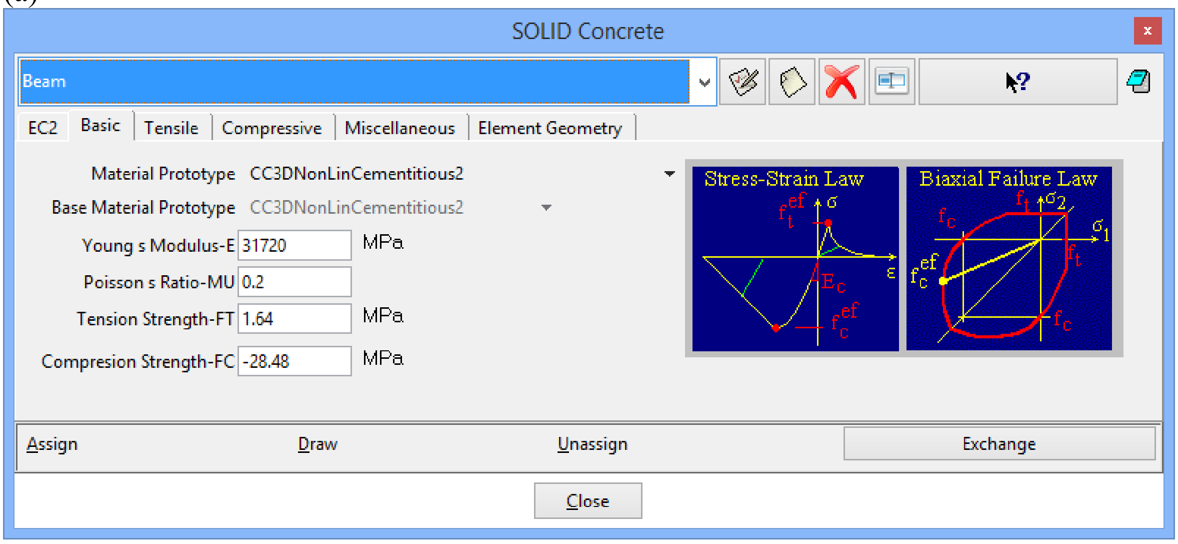
See Figure 57, Figure 58, Figure 59, Figure 60, and Figure 61.

If needed it is possible to modify these generated default parameters. However, it should be understood that the manual definition changes every time the Update changes button is selected. It is not recommended to modify these default parameters unless the user is an expert in nonlinear modelling and simulation.

In this tutorial problem, the generated parameters will be modified to get consistent with the original material properties and with the other versions of this Tutorial. The tensile strength is reduced to account for Shrinkage.



(a)



(b)

Figure 57: The default Basic parameters of the concrete class 20/25 before (a) and after (b) adjustment

Parameter input:
Young's Modulus-E: 31720 MPa
Tension Strength-FT: 1.64 MPa
Compression Strength-FC: -28.48 MPa

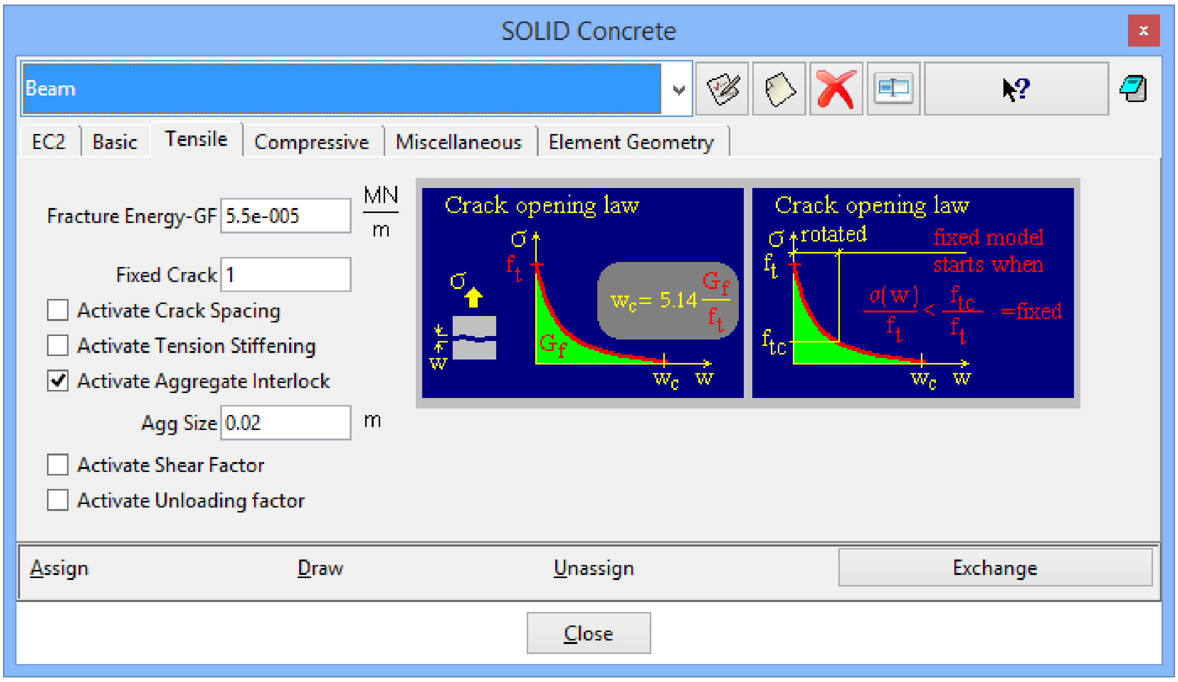


Figure 58: The default Tensile parameters of the concrete class 20/25

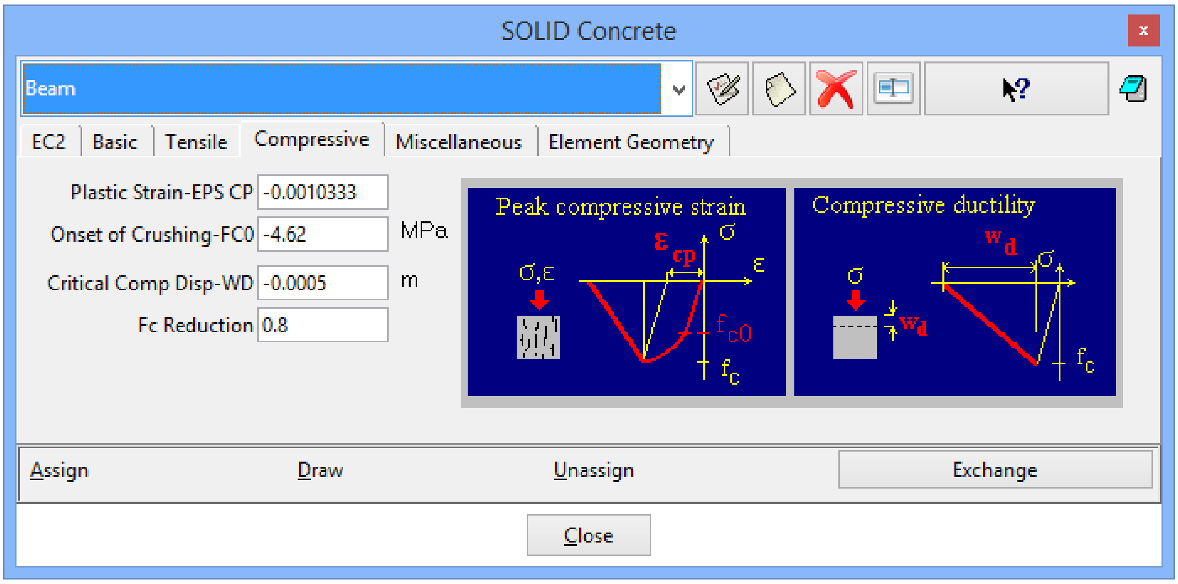


Figure 59: The default Compressive parameters of the concrete class 20/25

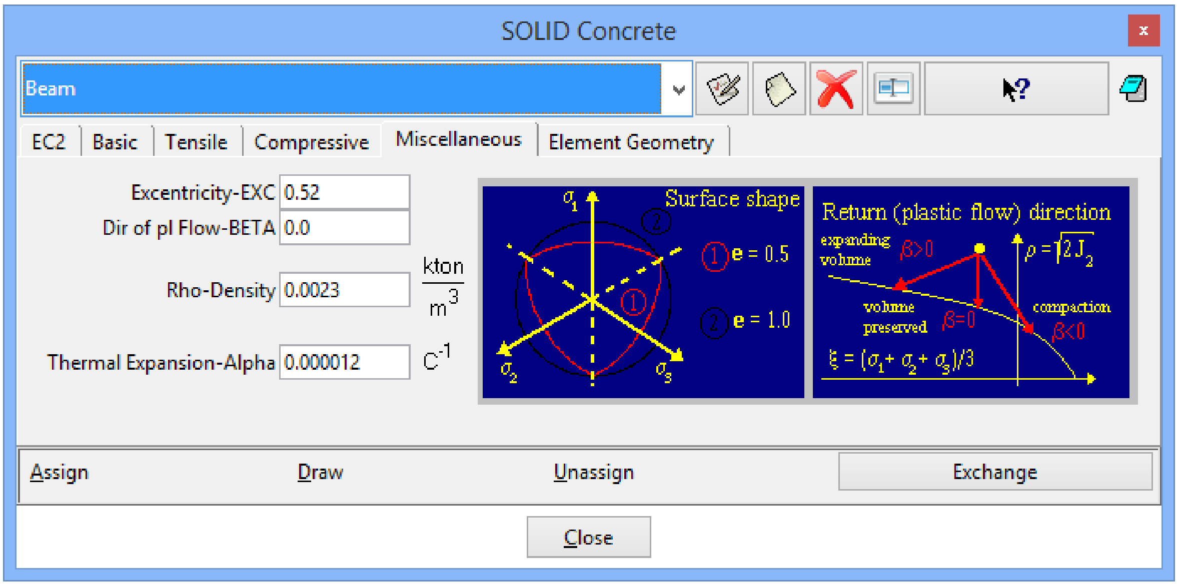


Figure 60: The default Miscellaneous parameters of the concrete class 20/25

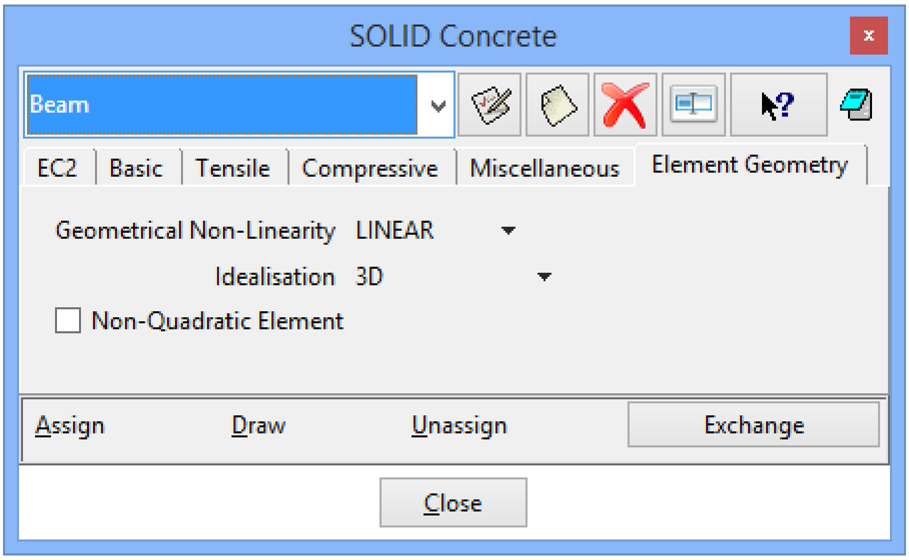


Figure 61: The default Element Geometry parameters of the concrete class 20/25

When the Beam material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **Assign** in the bottom of the material window. After this the several options will appear. In this case the Beam material will be assigned to the beam which is a volume. Therefore the option **Volumes** should be selected. Then the volume of the beam geometry can be selected in the graphical area, and the button **Finish** has to be pressed to complete the assignment.

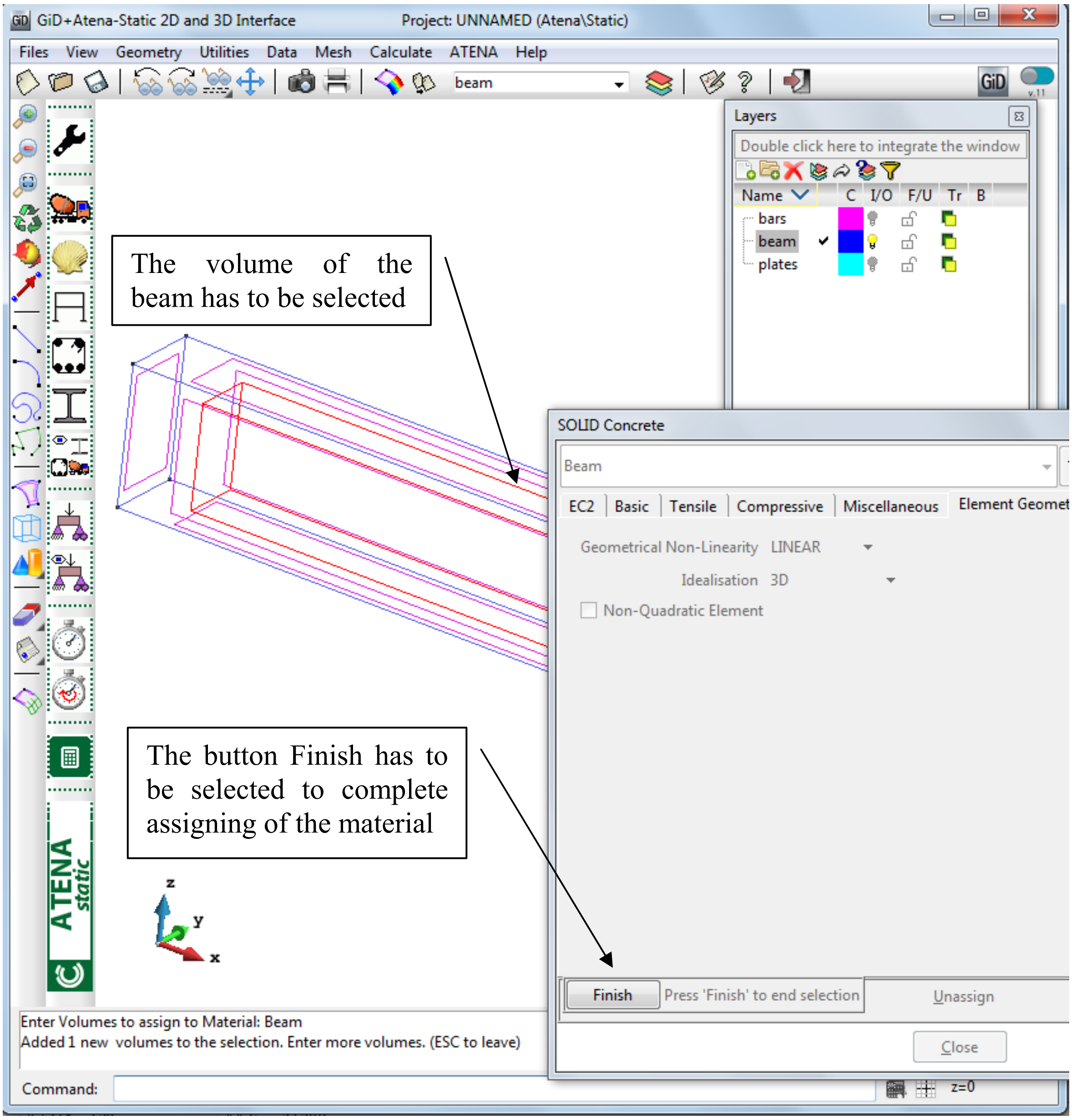


Figure 62: The assigning of the CONCRETE material to the volume

The beam material was created and assigned. Now, in the following section the steel plate material will be created.

3.3.2 *Loading and Supporting Steel Plates*

Before definition of the loading and supporting plate material it is a good idea to display only the plate layer.

Loading and supporting steel plates are made from steel material. It is assumed that the load level will not be so high to cause any plastic deformation in the plates. Because of that an elastic material will be used for the steel plates. The material definition starts with the command **Data | Materials | SOLID Elastic** in the main menu (see Figure 63).

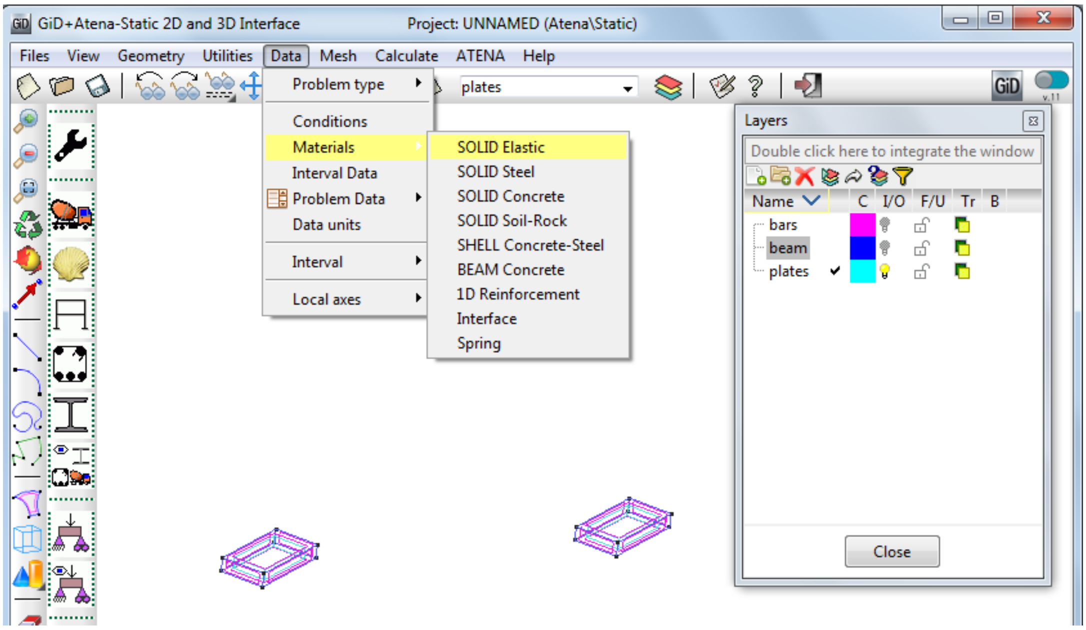


Figure 63: The selection of the command for the definition of the plates material

After the selection of this command the window for the definition of the SOLID Elastic will appear (see Figure 64).

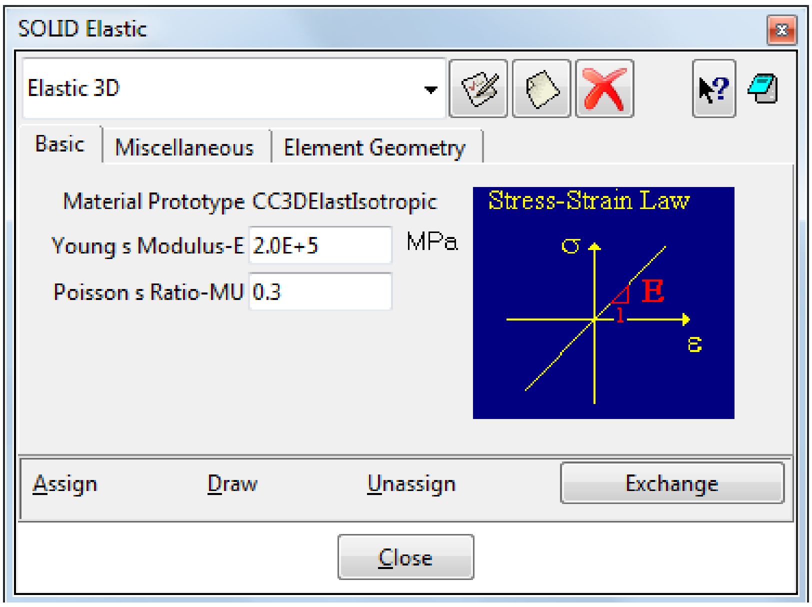


Figure 64: The window for the definition of the SOLID Elastic

The process of the Elastic material creation is very similar to the creation of the Concrete material. First, it is important to copy the material definition of the already existing material, and save it under a new name. There is only one elastic material and it will be chosen to be copied for the material of this example. The Elastic 3D should be selected and then the icon New SOLID Elastic  should be pressed. The selection of this material and selection of the New SOLID Elastic icon are depicted in the Figure 65.

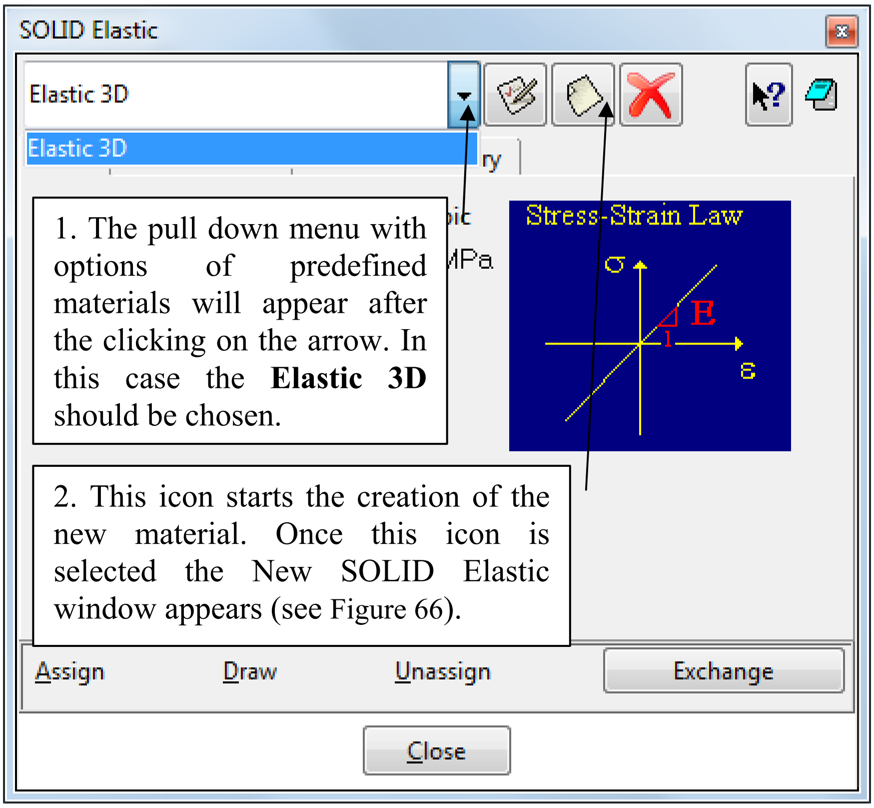


Figure 65: Description of the new elastic material creation

After the selection of the icon New SOLID Elastic the new window for the definition of the new material name will appear (see Figure 66). Here the name **Plates** should be written and then it is necessary to press **OK** button to complete this command.

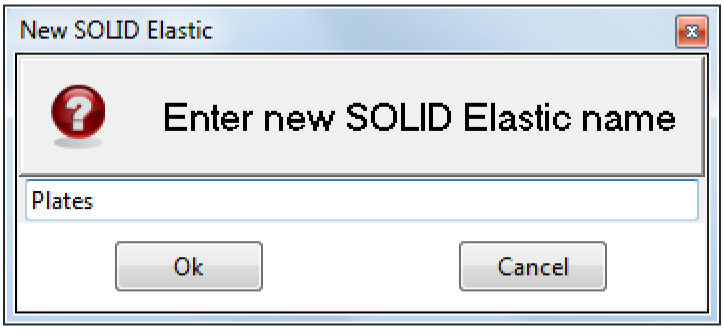


Figure 66: The window for the definition of the New SOLID Elastic material

Parameter input:
Enter new SOLID Elastic name:
Plates

Then the new material should be selected and then the parameter definition can be changed by clicking on the icon . In this case of the elastic material, the default parameters will be left unchanged.

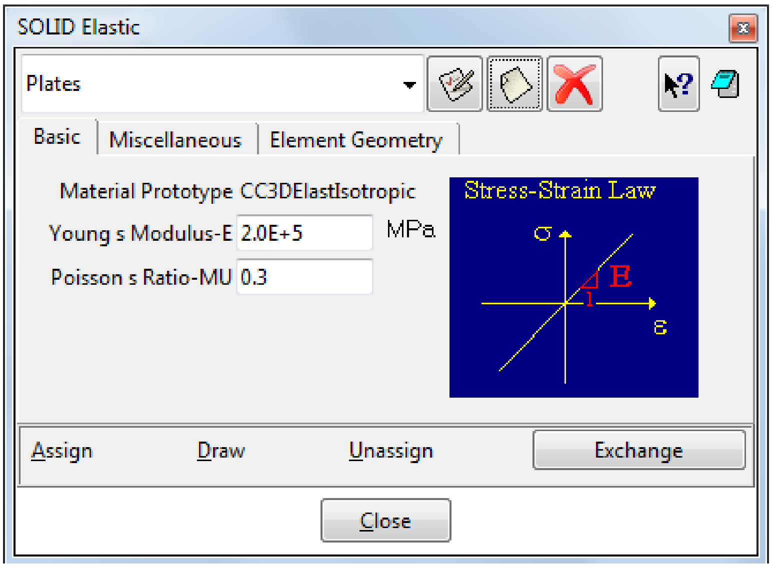


Figure 67: The default Basic parameters of the elastic material

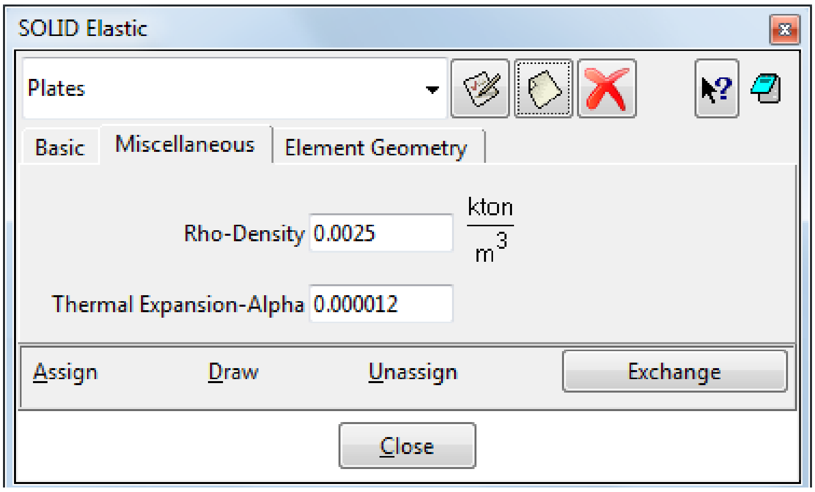


Figure 68: The default Miscellaneous parameters of the elastic material

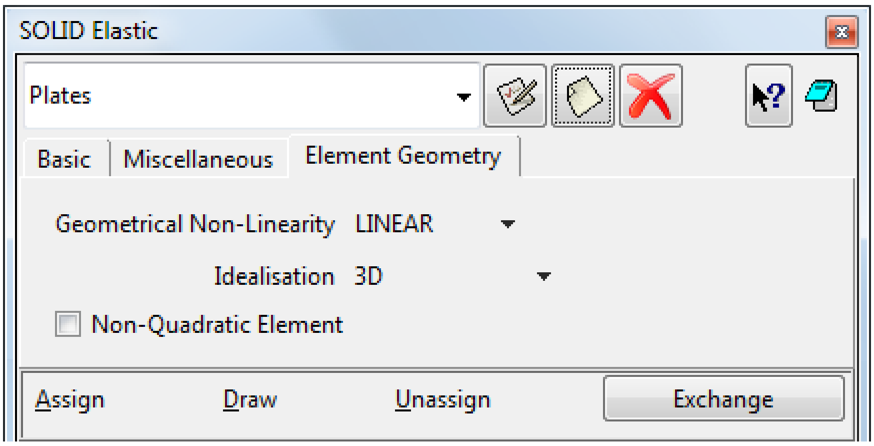


Figure 69: The default Element Geometry parameters of the elastic material

When the elastic material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **Assign** in the bottom of the material window. After selecting this button the several options will appear. In this case the **Plates** material will be assigned to the loading and supporting steel plates, which are represented by volume entities. Therefore the option **Volumes** should be selected. Then the volumes of the plates can be selected in the graphical area and the button **Finish** has to be pressed to complete the assignment.

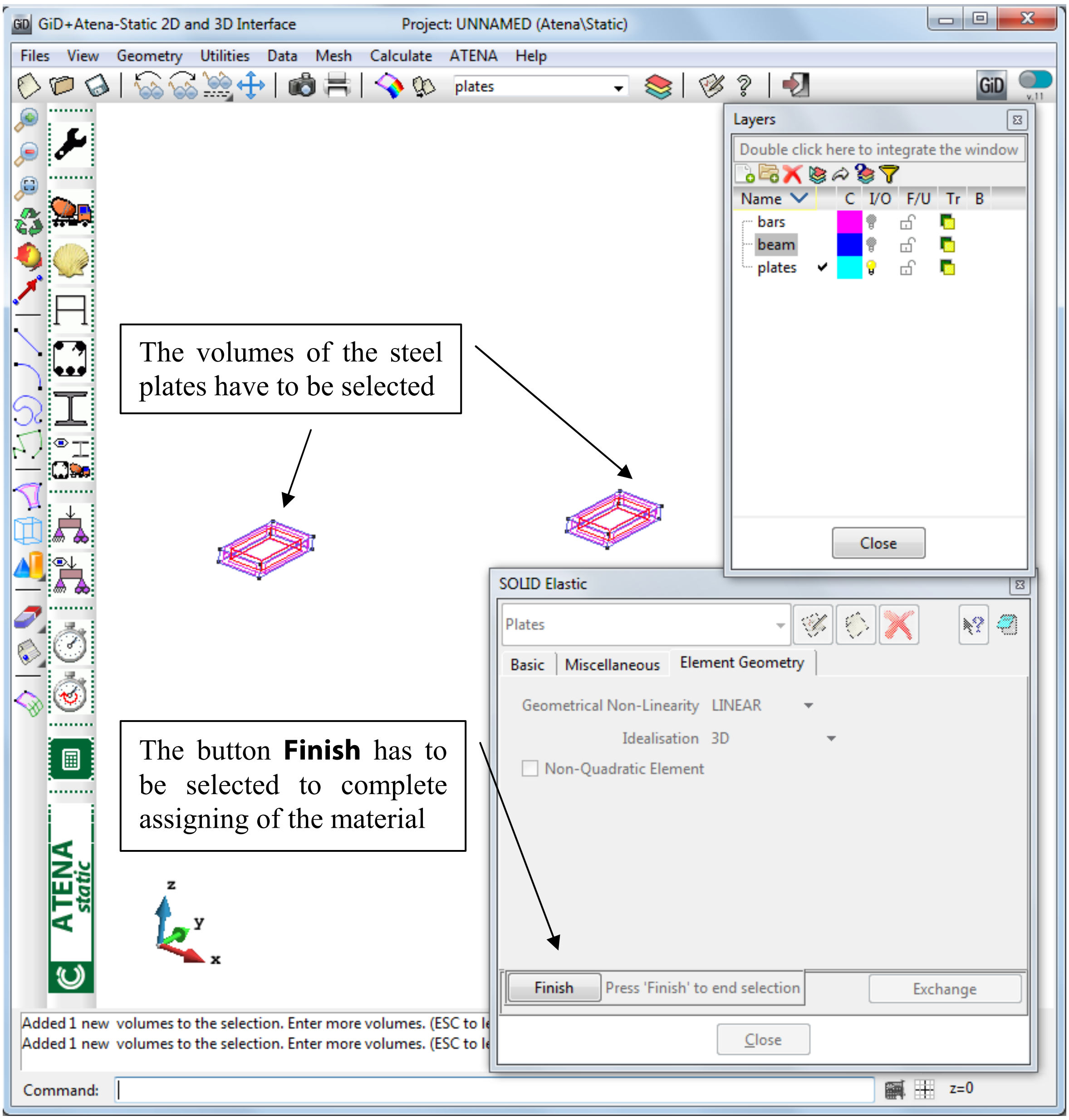
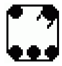


Figure 70: The assigning of the Plates material to the volumes

The steel plate material was created and assigned. In the last section, the reinforcement material will be created.

3.3.3 *Reinforcement Bars*

Before definition of the reinforcement material it is good to display only the Bar layer.

The material definition of the reinforcement starts by selecting the icon  or with the command **Data | Materials | 1D Reinforcement** (see Figure 71).

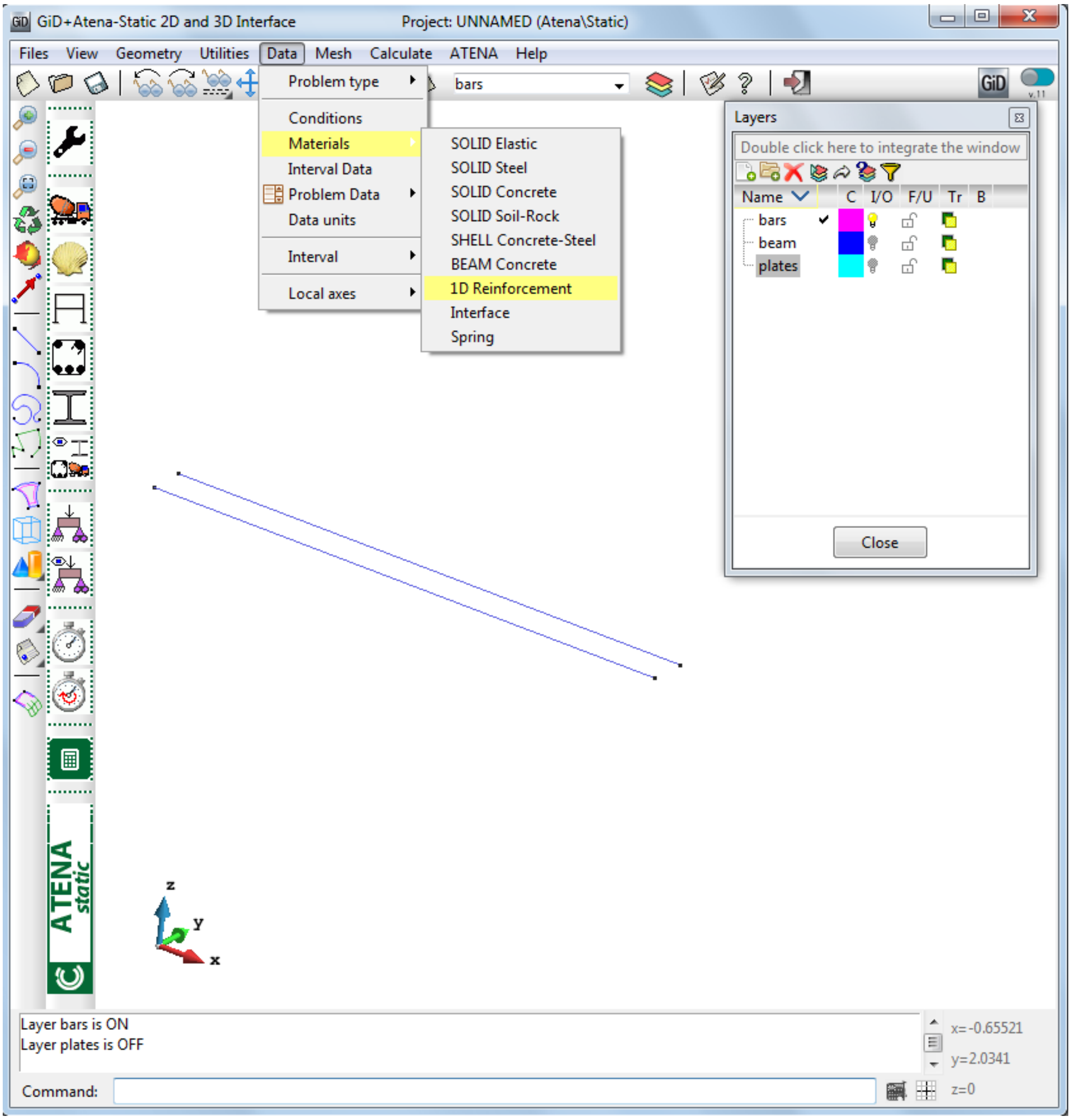


Figure 71: The selection of the command for the definition of the reinforcement material

After the selection of this command the window for the definition of the 1D Reinforcement will appear (see Figure 72).

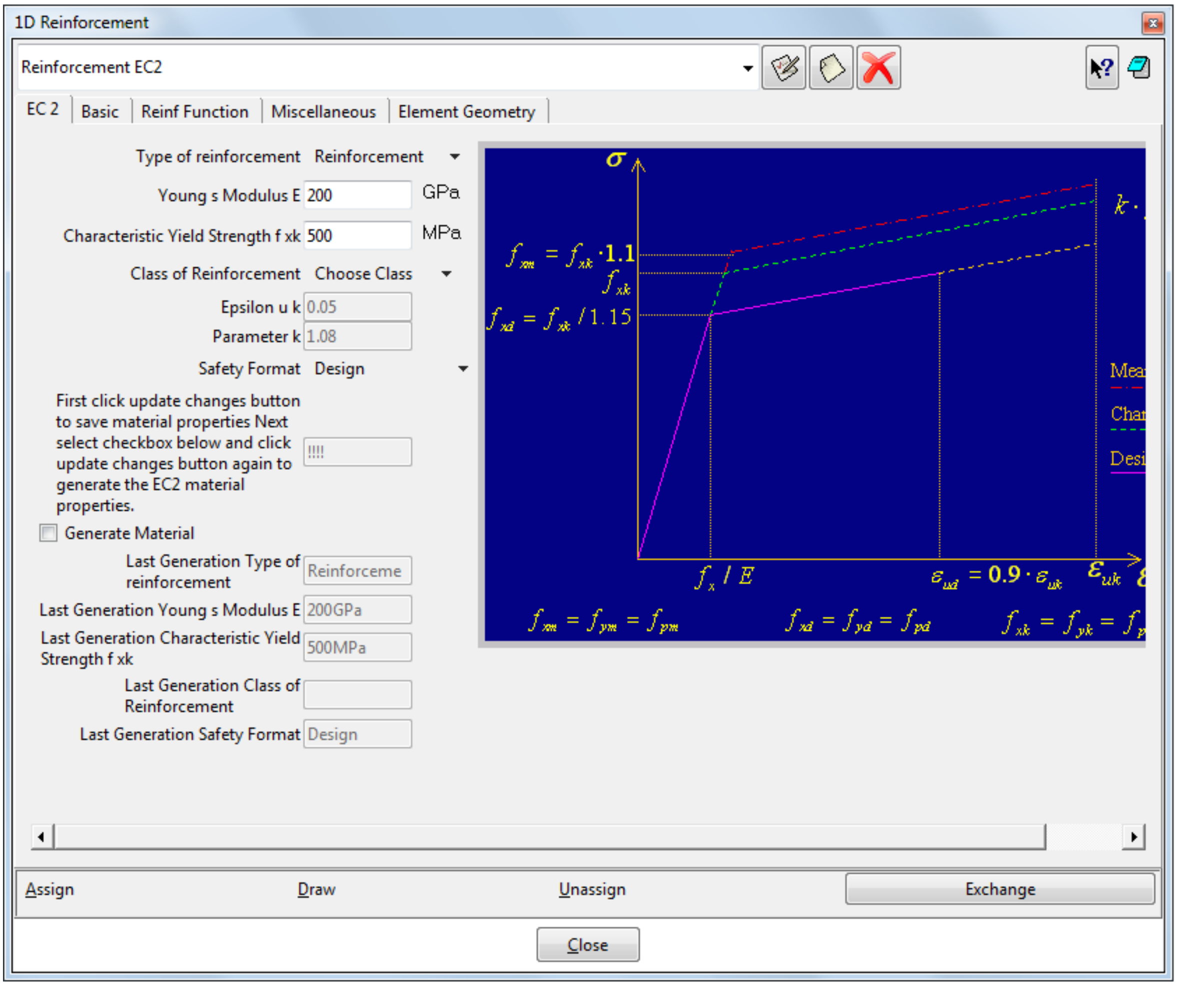


Figure 72: The window for the definition of the 1D Reinforcement

First, it is important to copy material definition of the already existing material and save it under a new name. In this case, the new name will be **Bars**. The predefined material Reinforcement EC2 should be chosen for the copying. After the selection of the predefined material the icon New 1D Reinforcement  should be selected. After the selection of the New 1D Reinforcement icon, the new window for the definition of the new material name will appear (see Figure 73). Here the name **Bars** should be typed, and then it is necessary to press **OK** button to complete the command.

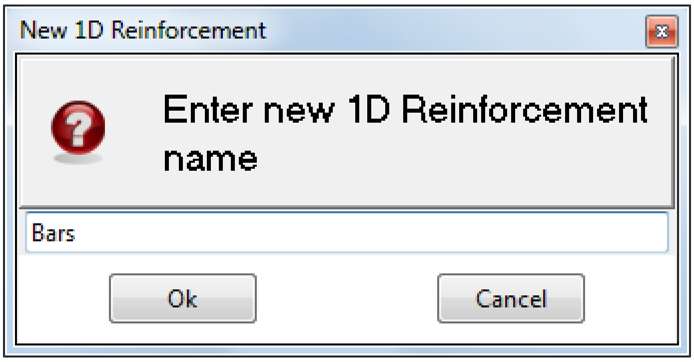


Figure 73: The window for the definition of the New 1D Reinforcement

Parameter input:
Enter new 1D Reinforcement name: Bars

This new material should be selected, and then it is possible to change the parameter definition. The parameters of the new material **Bars** are predefined according to Eurocode 2. In this example the **Mean Yield Strength** should be **560 MPa** and **Class of Reinforcement** should be **A**. The parameter definition is depicted in the Figure 74. It is very important after all changes are updated to select checkbox Generate Material and do update again. Otherwise no parameters will be updated. All parameters definition is completed by clicking on the Update Changes icon .

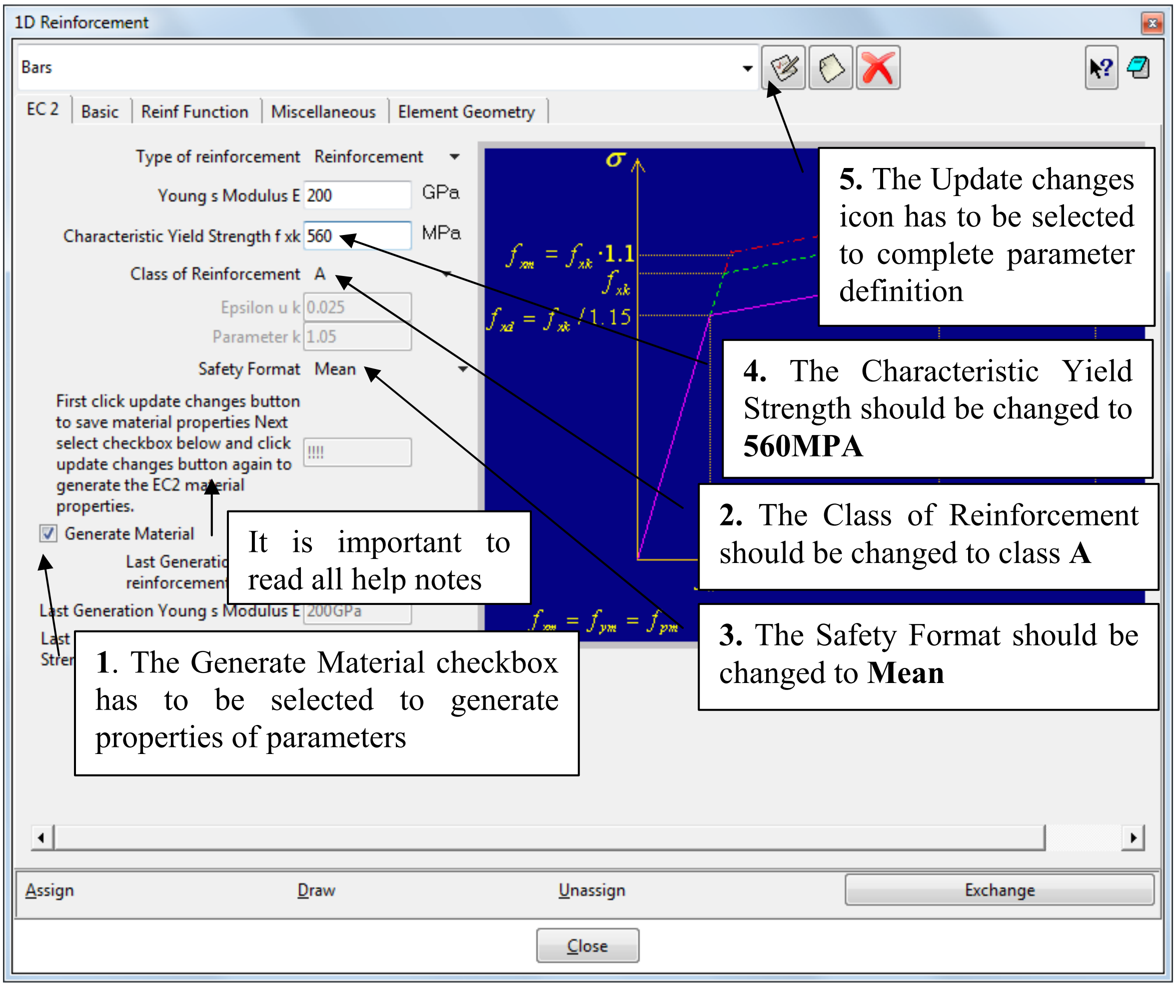


Figure 74: The description of the reinforcement definition

In the Basic properties the bar diameter and number of bars can be defined. By checking the checkbox Calculator, dialogs for the profile definition will appear. In this tutorial example, the Profile should be 26 mm and number of profiles will stay 1. Then the Update changes icon  has to be clicked to recalculate the reinforcement area. Then it is necessary to click on the Update changes icon again to save all changes into the material (see Figure 75).

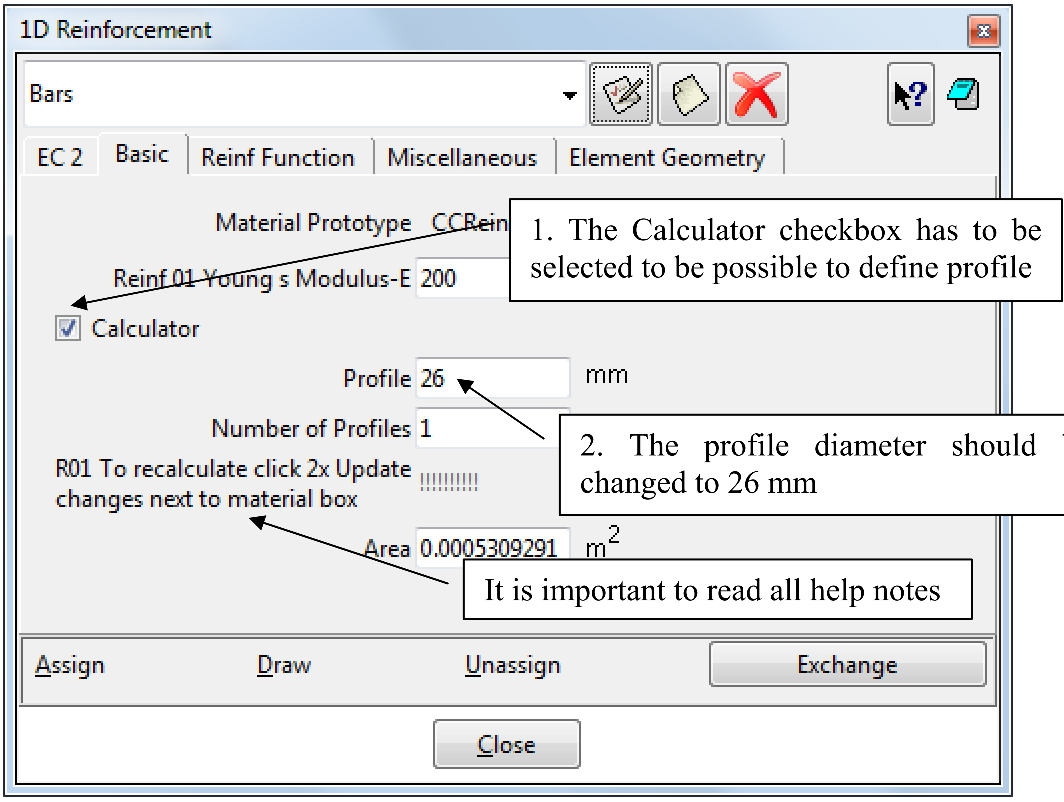


Figure 75: The default Basic parameters of the reinforcement, the icon Update changes has to be clicked 2x to change parameters

The rest of the reinforcement parameters will be default. There is no change necessary (see Figure 76, Figure 77, and Figure 78).

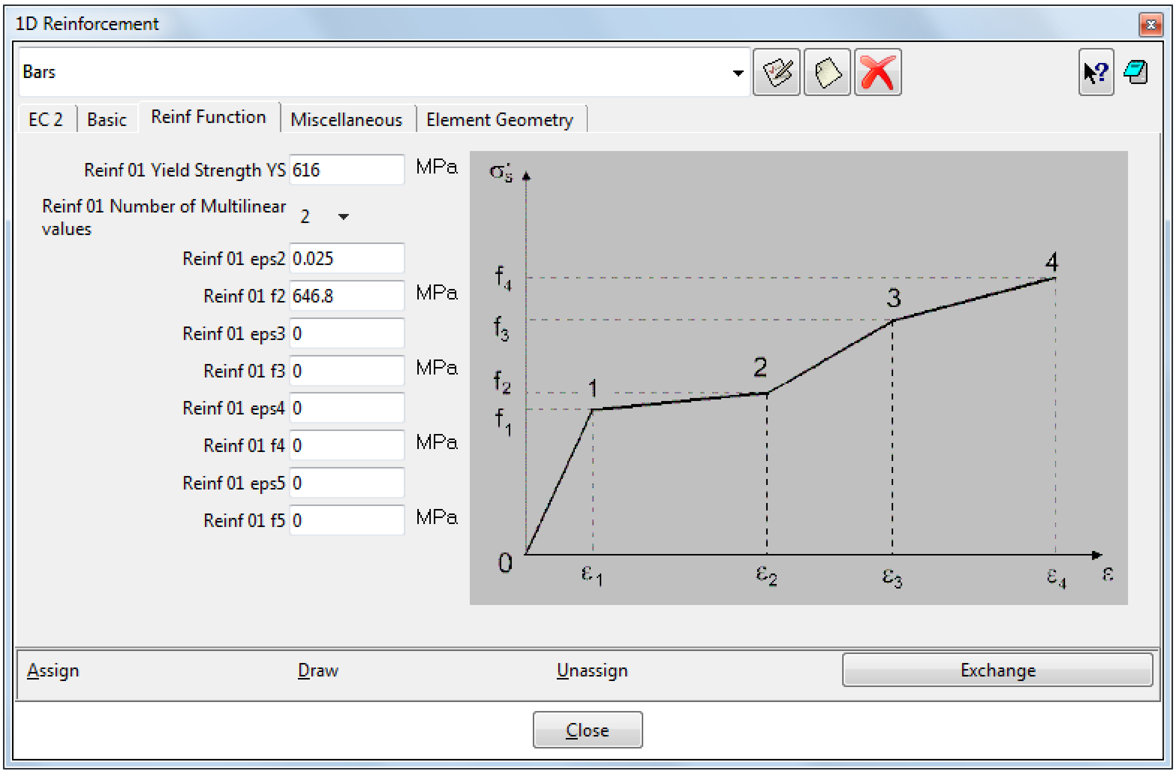


Figure 76: The default Reinf Function parameters of the reinforcement

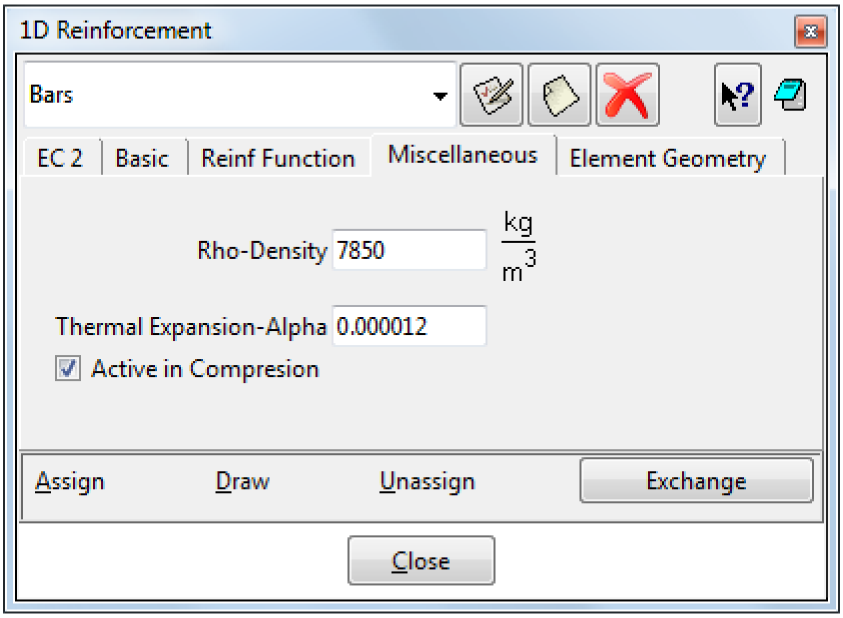


Figure 77: The default Miscellaneous parameters of the reinforcement

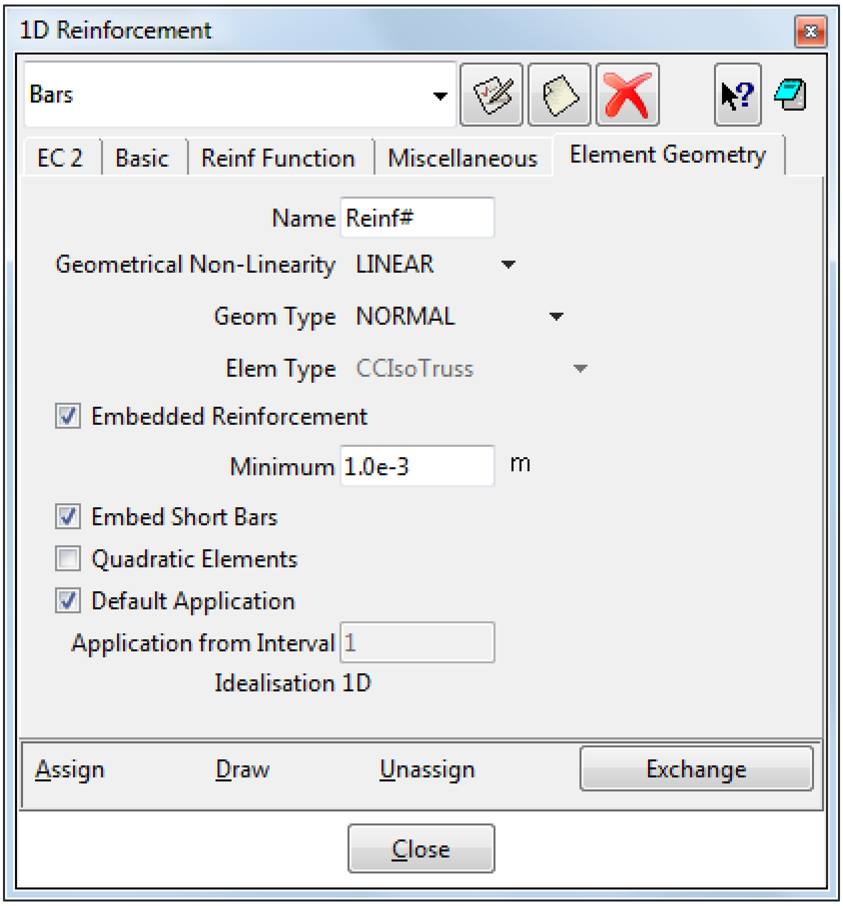


Figure 78: The default Element Geometry parameters of the reinforcement

When the bar material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **Assign** in the bottom of the material window. After this the several options will appear. In this case the Bars material will be assigned to two straight lines. Therefore the option Lines should be selected. Then the lines of the reinforcement can be selected in the graphical area and the button **Finish** has to be pressed to complete the assignment (see Figure 79).

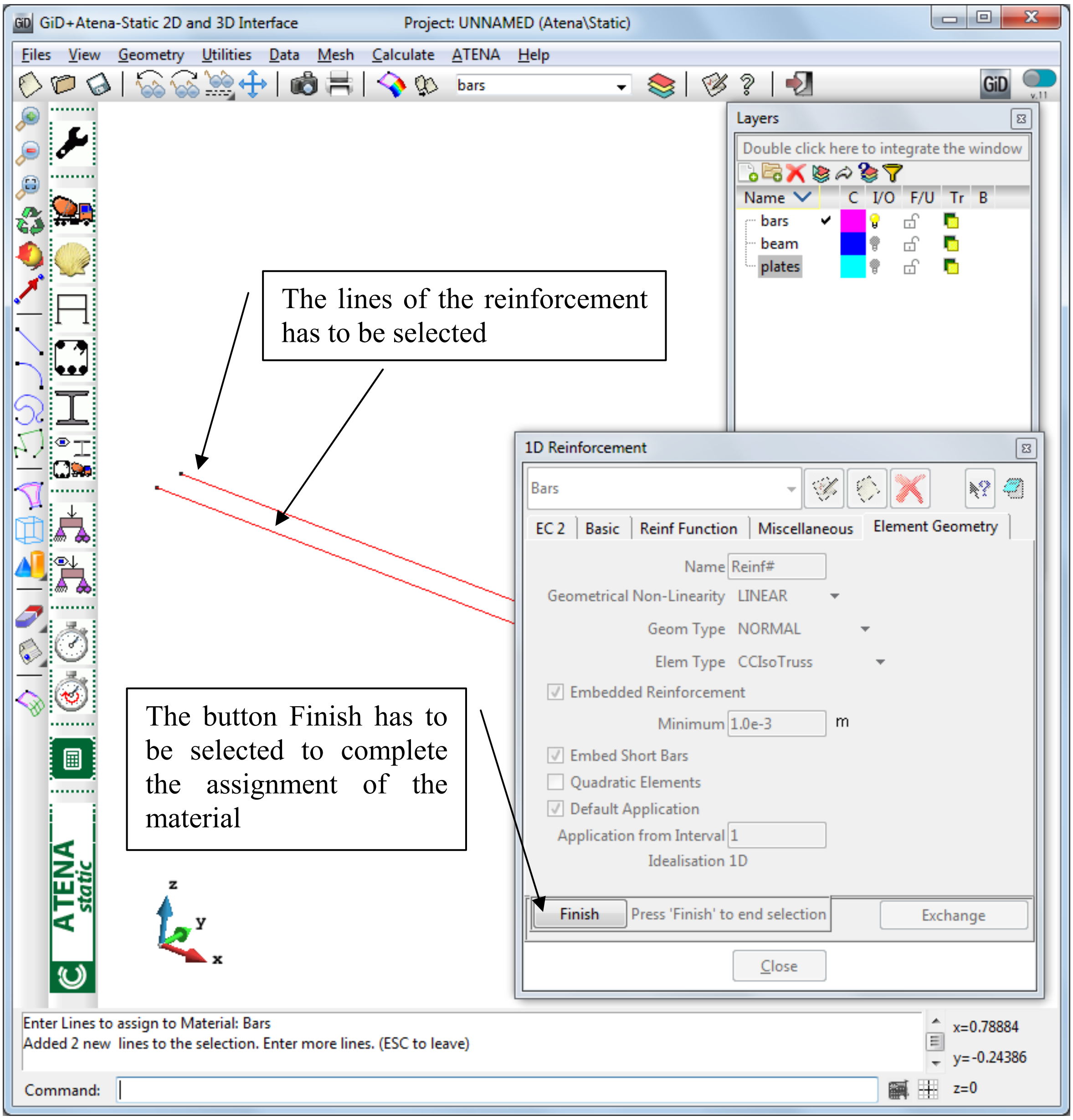
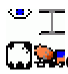


Figure 79: The assigning of the Bars material into lines

All materials are created and assigned. The icon Draw all materials can be used to check if all materials are correctly assigned. But before that it is important to display all layers and their content. It is simply done by clicking on the grey bulb which should change to the yellow after the clicking. Then the Draw all materials icon can be used. See Figure 80.

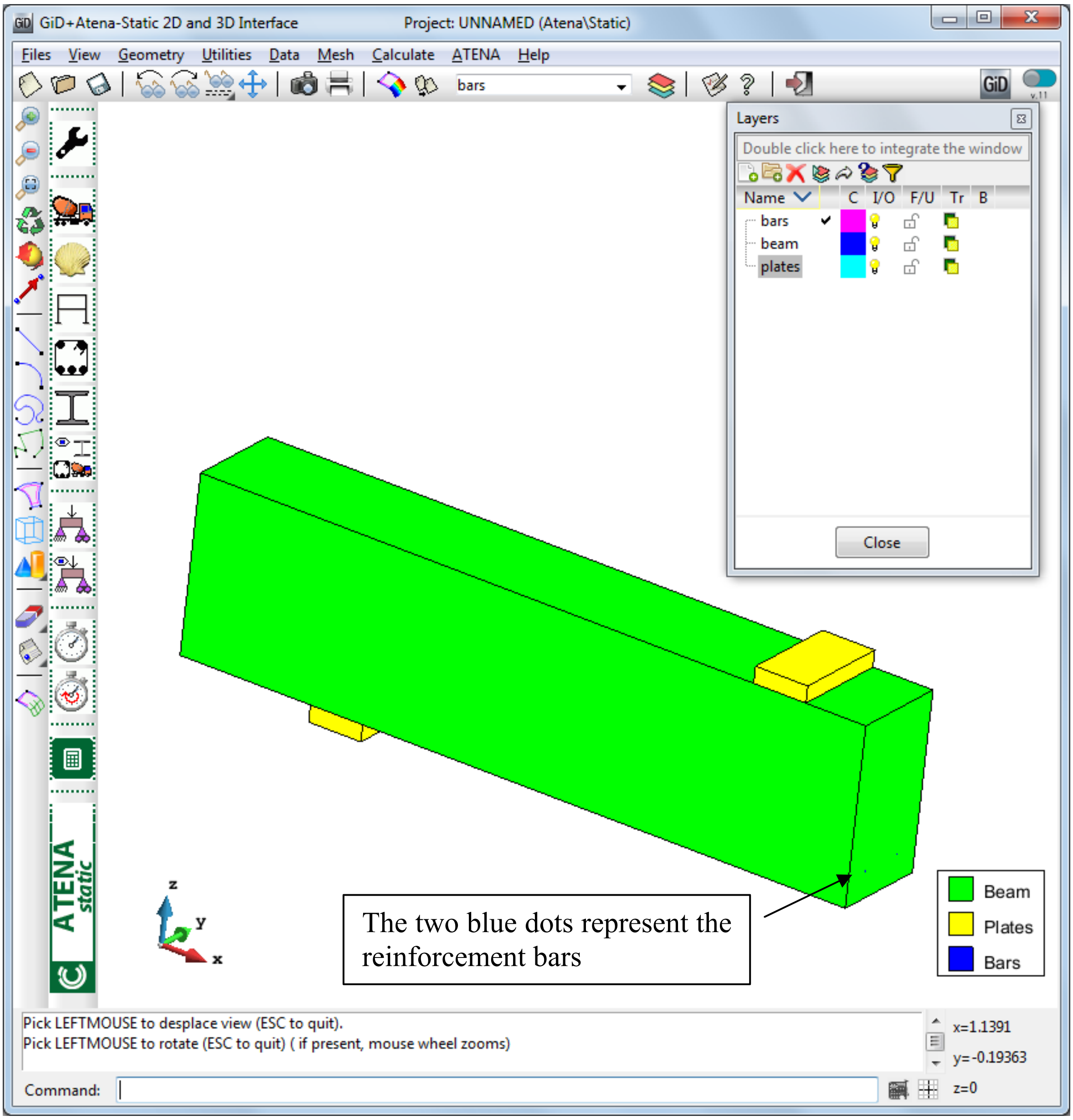


Figure 80: The drawn assigned materials

3.4 Boundary Conditions

In this chapter the boundary condition are described. The analyzed beam is supported at the bottom steel plate in the vertical direction. There the support condition will be defined. Since only a symmetric half of the beam is analyzed, it is necessary to enforce the fixed condition along the right side of the beam. It means that the horizontal displacements along x-axis should be equal to zero.

The beam is loaded at the top steel plate. The object of this example is to determine the maximal load-carrying capacity of the beam. It means that it should be possible to trace the structural response also in the post-peak regime. The easiest method to accomplish this is by loading the beam by prescribed displacements condition at the top steel plate.

It is important to monitor forces, displacement or stresses during the non-linear analysis. The monitor data are important information about the state of the structure. For instant from monitoring of applied forces, it is possible to determine if the maximal load was reached or not.

In summary, there are four types of the boundary conditions in this example – monitors, support, displacement and symmetry conditions.

3.4.1 Support

The analyzed beam is supported at the bottom steel plate in the vertical direction. The support condition should be applied to the line. This line has to be added into the bottom plate geometry. It will be done by dividing the bottom plate surface.

The steel plates are assigned into the plate layer. Therefore the plate layer should be activated and displayed. The bar layer can be hidden but the beam layer is better to keep displayed to be able recognize the bottom surface. It is also recommended to zoom at the bottom plate. Make sure that the zoomed surface is the bottom surface of the bottom plate (see Figure 81 and Figure 82).

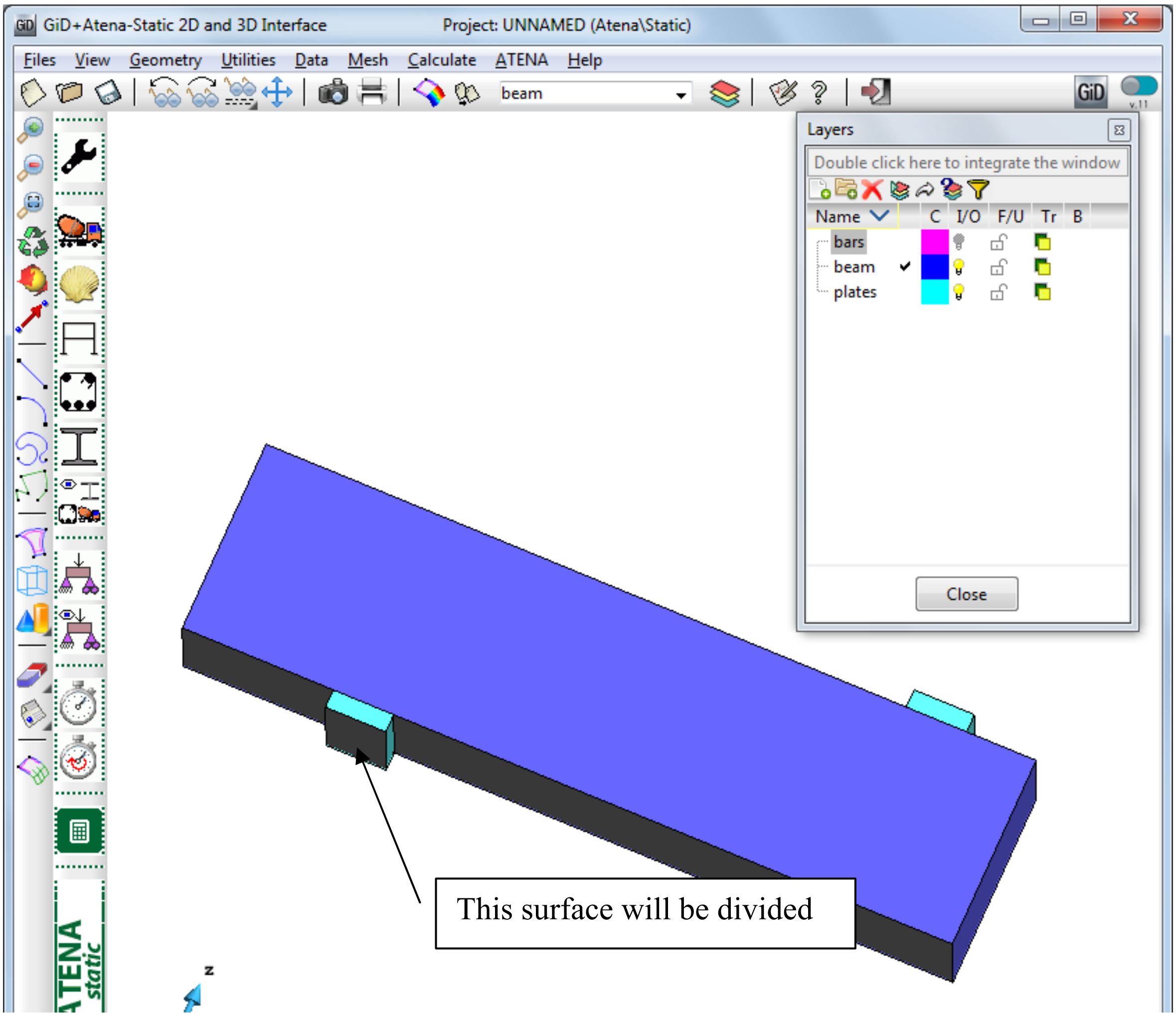


Figure 81: The bottom surface of the bottom plate

The division of the surface starts with the execution of the command from main menu **Geometry | Edit | Divide| Surfaces | Num Divisions** or by selecting of the Divide surface icon  (see Figure 82).

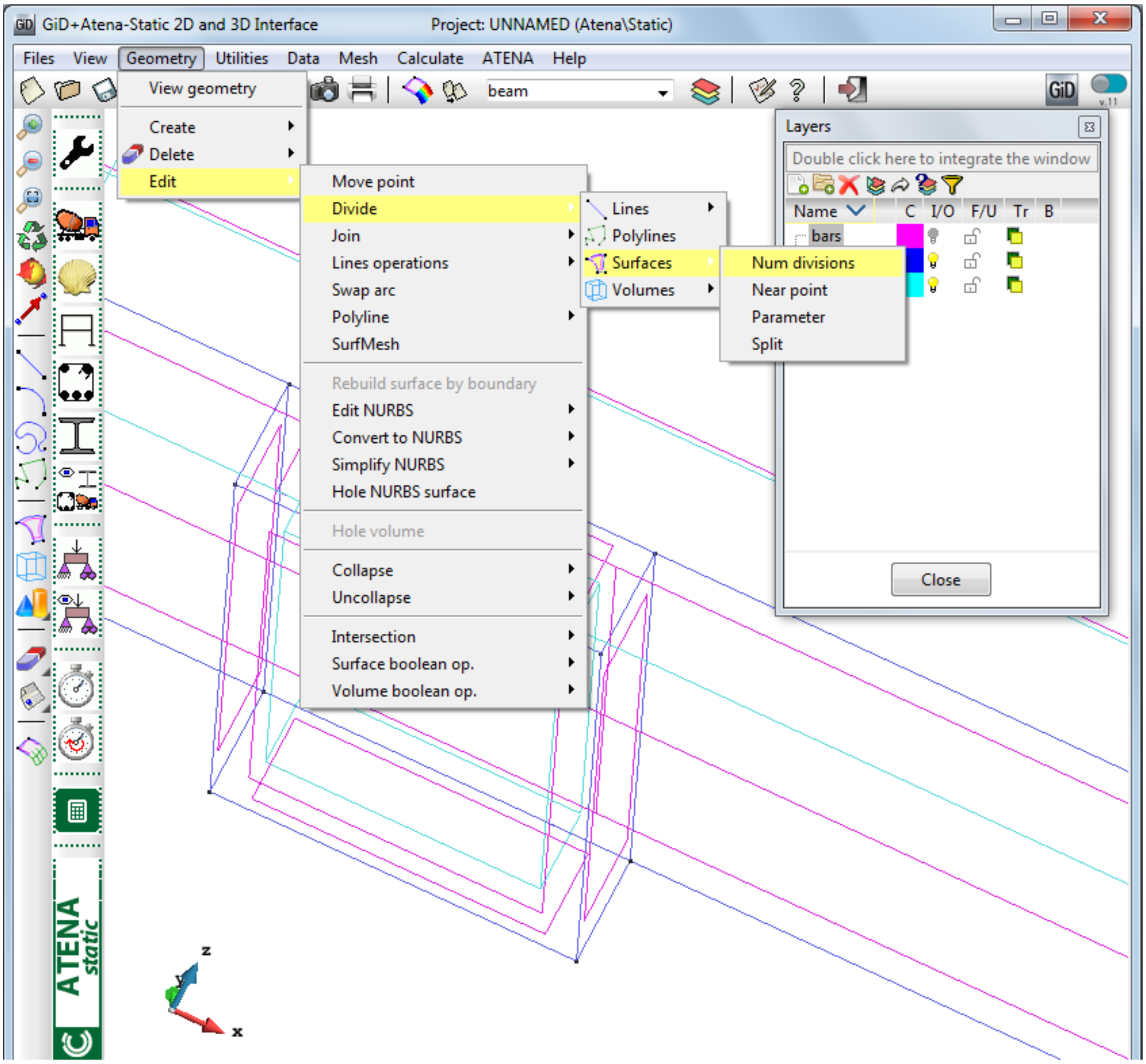


Figure 82: The executing of the division command

After the execution of the divide command the cursor will change into this  shape, and the surface required for dividing should be selected. Once the surface is selected the dialog window will appear on the screen (see Figure 83).

This dialog asks for the direction, along which the surface should be divided. There are U and V direction, and in the graphical area it is possible to see green axis representing U and V direction. In this case U Sense should be chosen. Once the U Sense button is chosen the program asks for the number of the divisions. Bottom surface should be divided into two parts (see Figure 84).

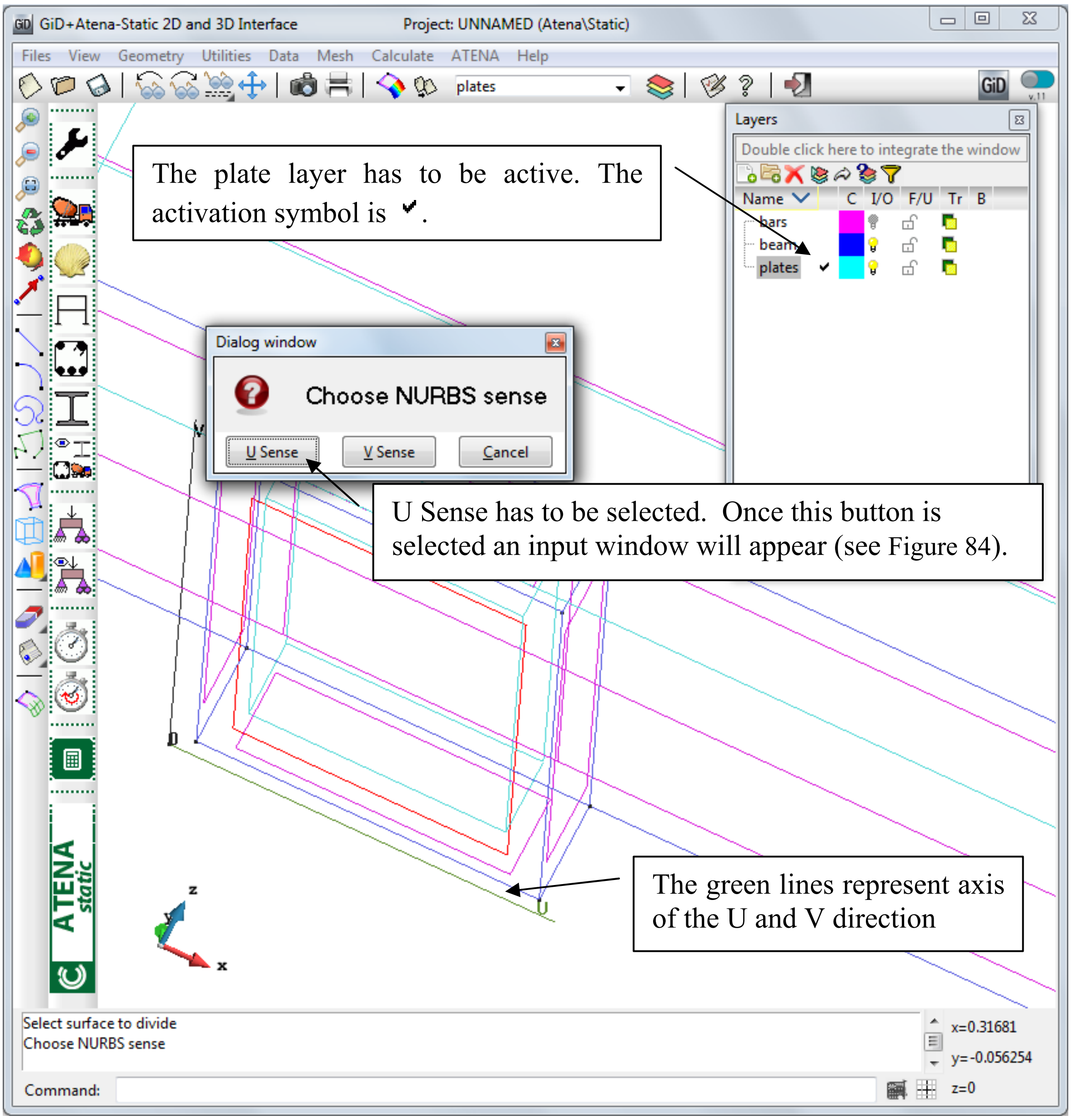


Figure 83: The dividing of the surface

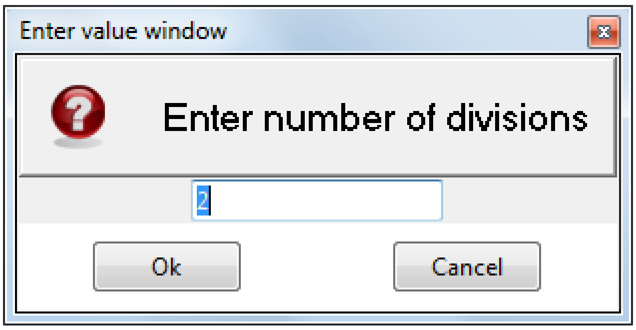


Figure 84: The enter value window

Parameter input:	
Enter number of divisions:	2

The button **OK** should be pressed in the above dialog. After that the surfaces is divided into two parts (see Figure 85).

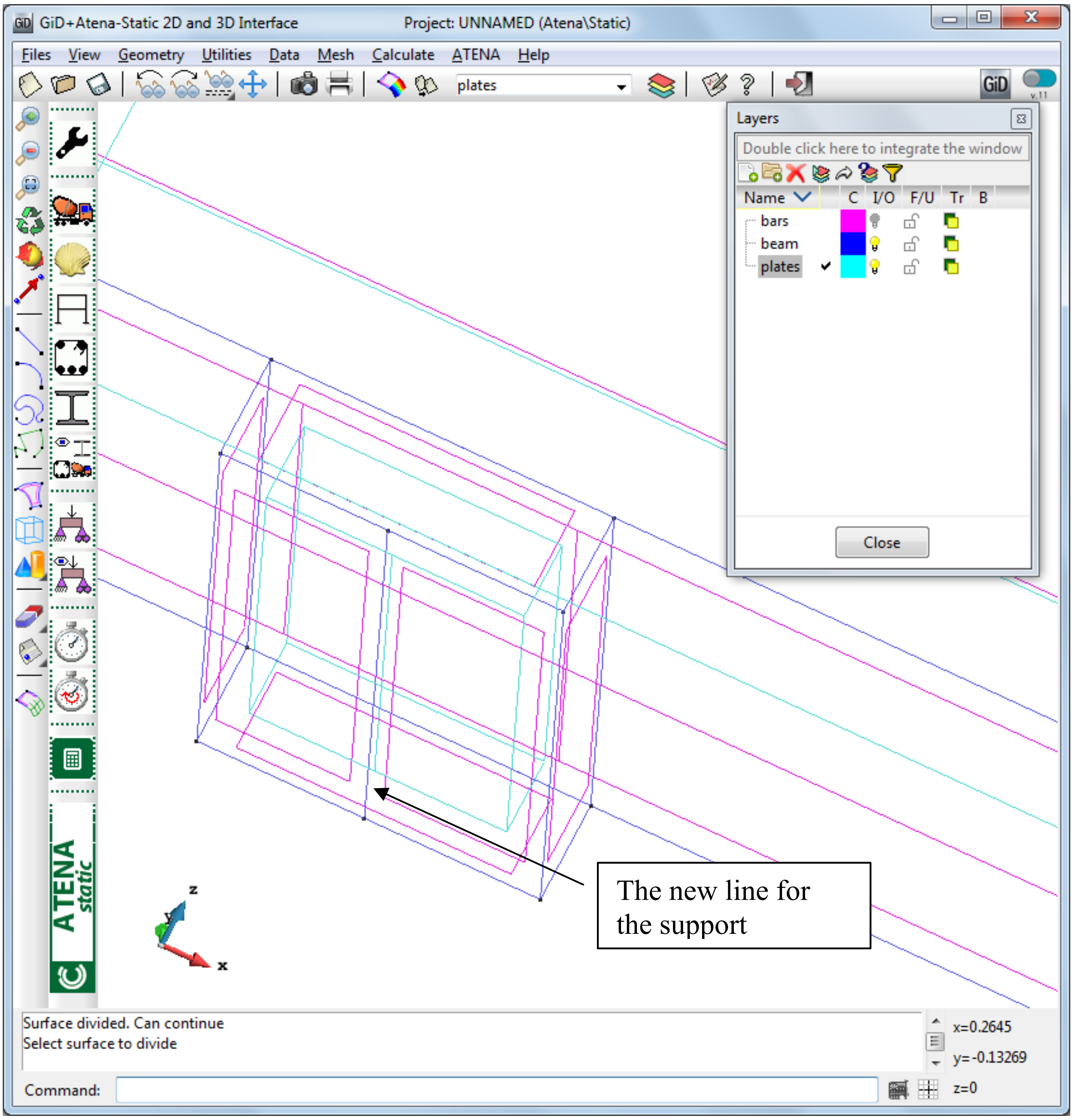
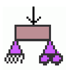
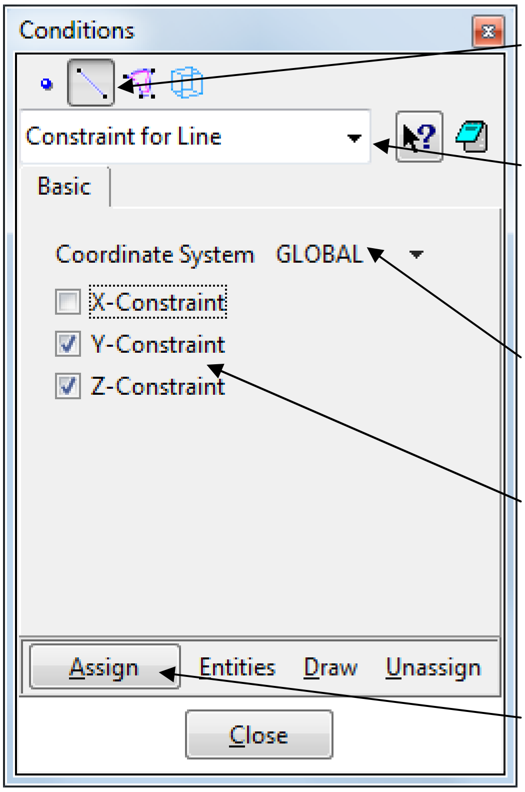


Figure 85: The divided top surface

When the geometry for the support is created the support condition can be defined. Conditions command can be executed by the **Data | Conditions** in the main menu or by the icon . The support condition definition is depicted in the Figure 86.



The support condition is to be applied on a line, therefore this icon should be selected.

By clicking on the arrow, the list of available line conditions will appear. The option **Constraint for Line** should be selected.

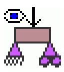
By clicking on this button the several options will appear. The option **GLOBAL** coordinate system has to be selected.

The support is in the vertical direction. Therefore the **Z-Constraint** has to be selected.
In order to prevent any rigid displacement the **Y-Constrain** should be selected too.

By this button the monitor can be assigned to the geometry (see Figure 87).

Figure 86: The support condition definition

Parameter input:
Constraint for Line
Coordinate System: GLOBAL
Y-Constraint
Z-Constraint

By clicking on the icon  the created condition can be drawn. After clicking on that icon the support condition will be displayed on the assigned lines (see Figure 88).

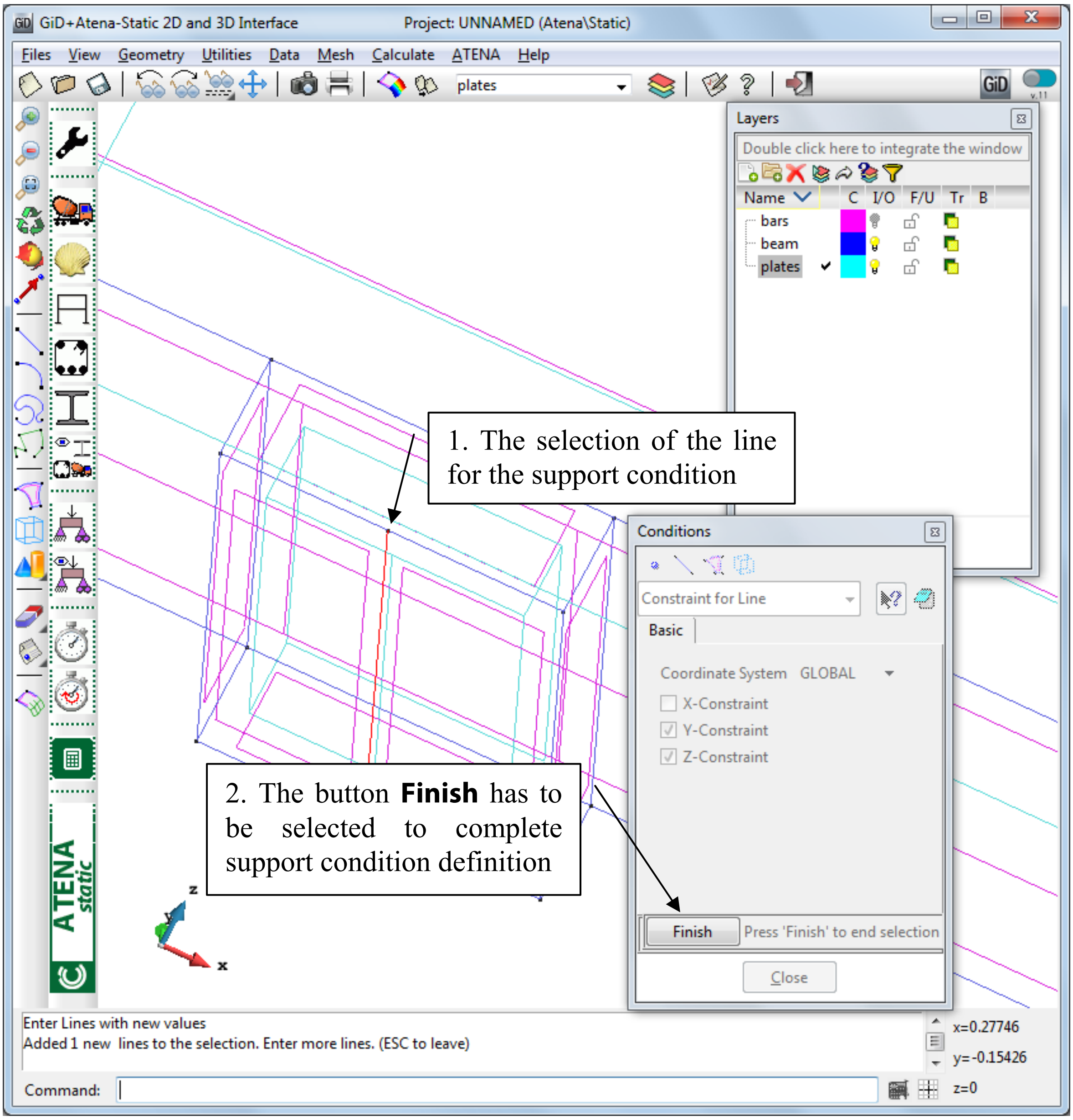


Figure 87: The selection of the support line

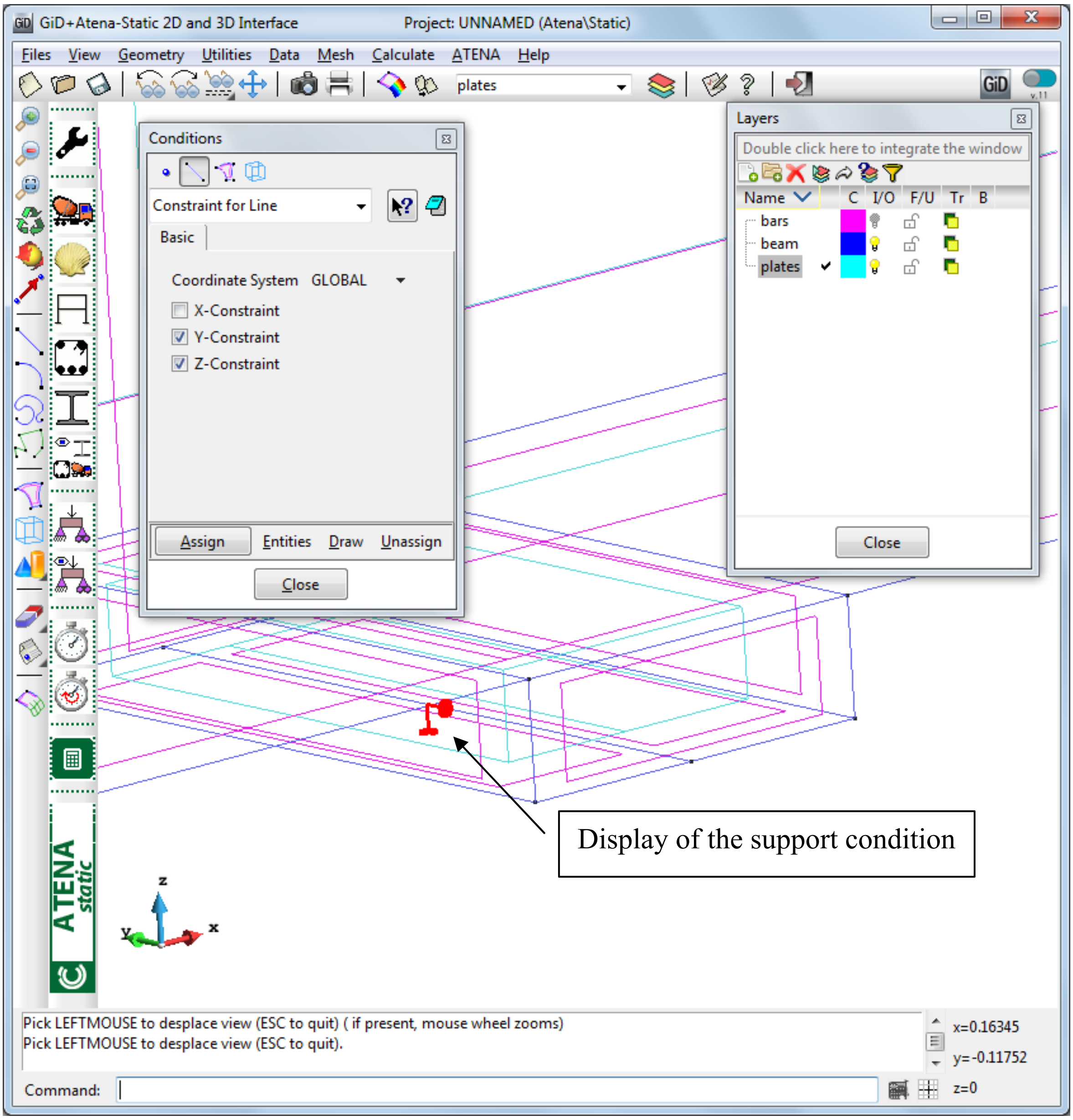


Figure 88: The support condition

3.4.2 Displacement

On the top plate a predefined displacement should be specified. This displacement will be located in the middle of the loading plate (top plate) and the displacement should be defined -0.0001m in the z direction.

This load should be applied on a point in the middle of the top plate. However, this point does not exist yet. Therefore, first the geometry of the top plate has to be modified.

The point should lie in the centre of the top surface. This point has to be part of the top plate geometry. It cannot be simply created on the surface. Therefore, the top surface will be divided into two surfaces, and then the line which separates these surfaces will be also divided into two parts. Then the middle point can be used to for the application of the prescribed displacement.

The steel plates are assigned into the plate layer. Therefore, the plate layer should be activated and displayed. The beam and bar layers can be hidden. It is also recommended to zoom at the top plate (see Figure 89).

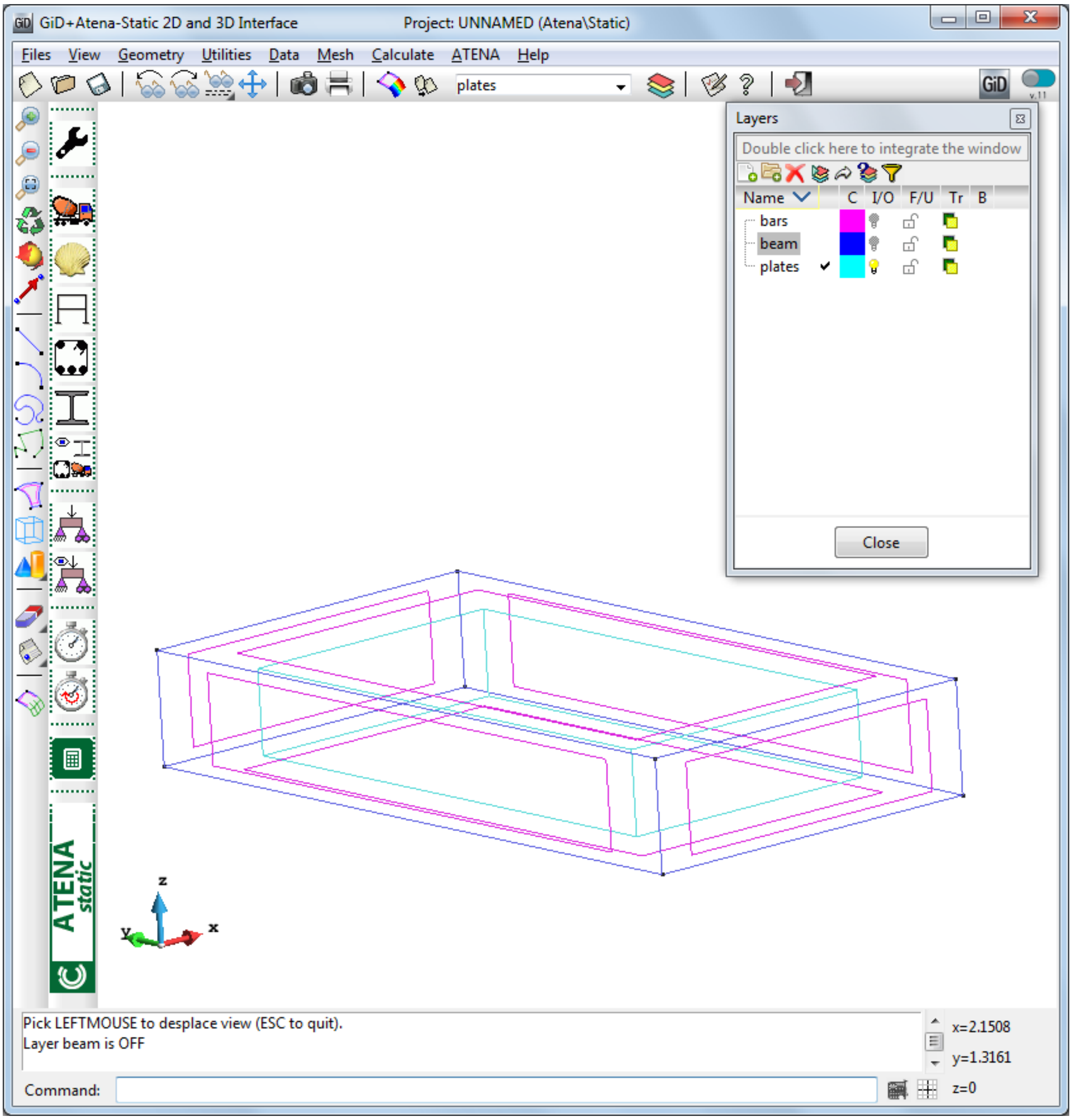
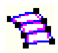


Figure 89: The activated plate layer and zoom view of the top plate

The top surface will be divided using the command from the main menu **Geometry | Edit | Divide| Surfaces | Num Divisions** or by selecting Divide surface icon  (see Figure 90).

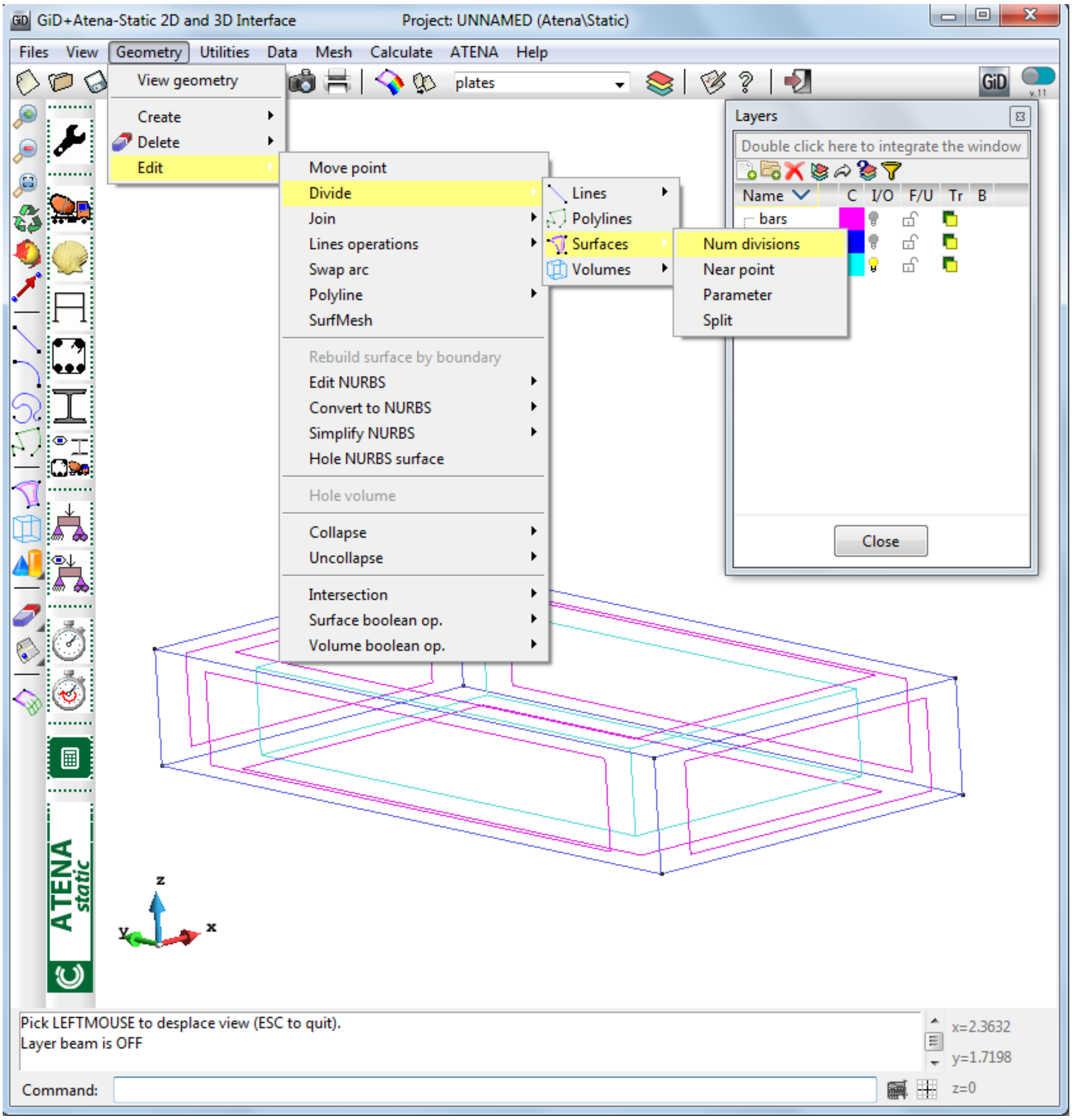


Figure 90: The execution of the division command

After the executing the divide command the cursor will change into this  shape, and the appropriate surface should be selected. Once the surface is selected a dialog window will appear on the screen (see Figure 91).

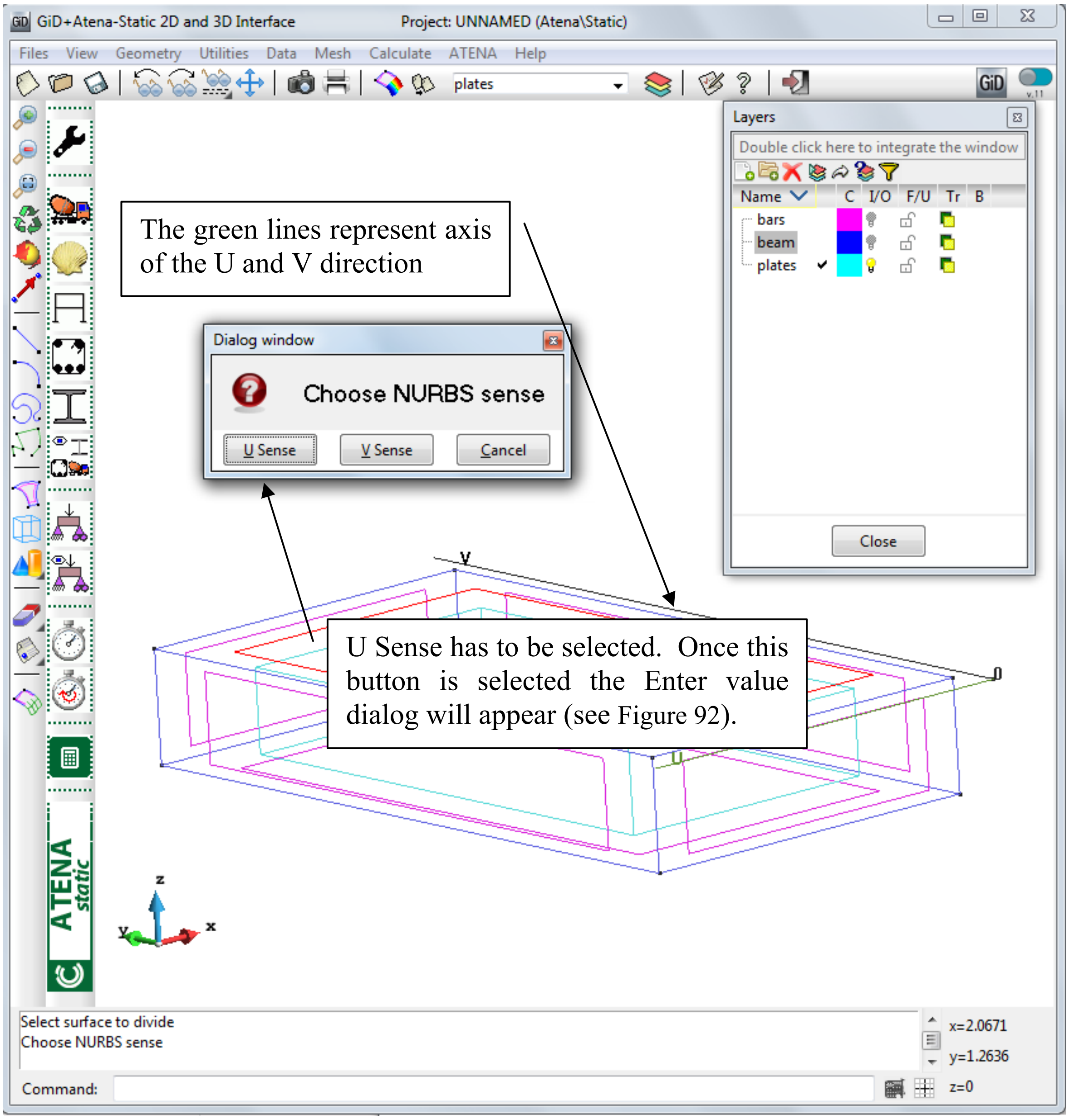


Figure 91: The dividing of the surface

The dialog window asks for a direction, along which the surface should be divided. The possible directions are denoted as U and V, and they are represented as green axes in the graphical area. In this case U Sense should be chosen since it is necessary to divide the surface along the U direction. Once the U Sense button is chosen the program asks for the number of the divisions. Top surface should be divided into two parts (see Figure 92).

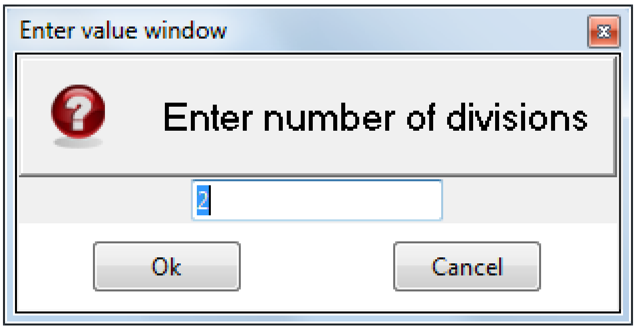


Figure 92: The enter value window

Parameter input:	
Enter number of divisions:	2

The button **OK** should be pressed in the enter value window. After that the surfaces is divided (see Figure 93).

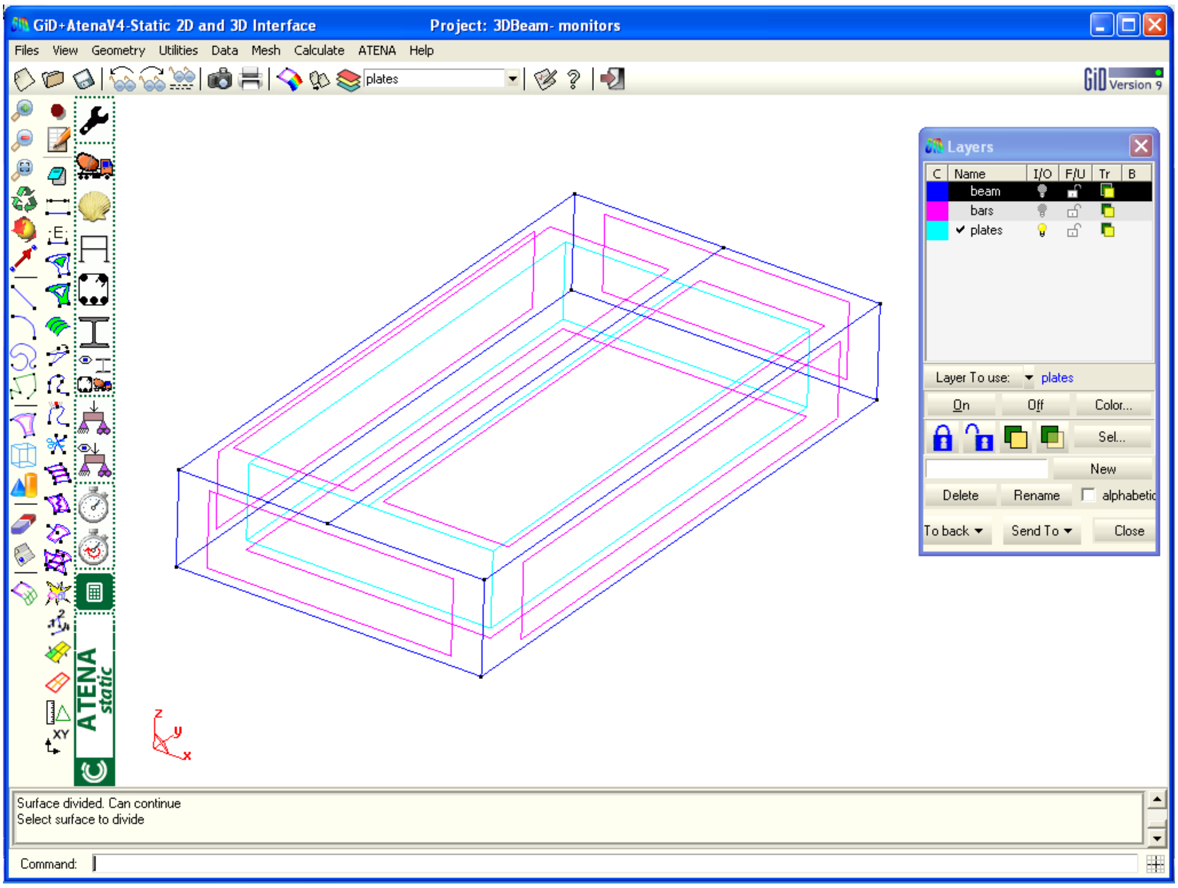


Figure 93: The divided top surface

Now the middle line can be divided into two parts. It can be done by executing command **Geometry | Edit | Divide| Lines | Num Division** or by the icon. After the execution of this command the enter value dialog will appear. Here the number of required divisions is to be written. The line should be divided in two divisions (see Figure 95).

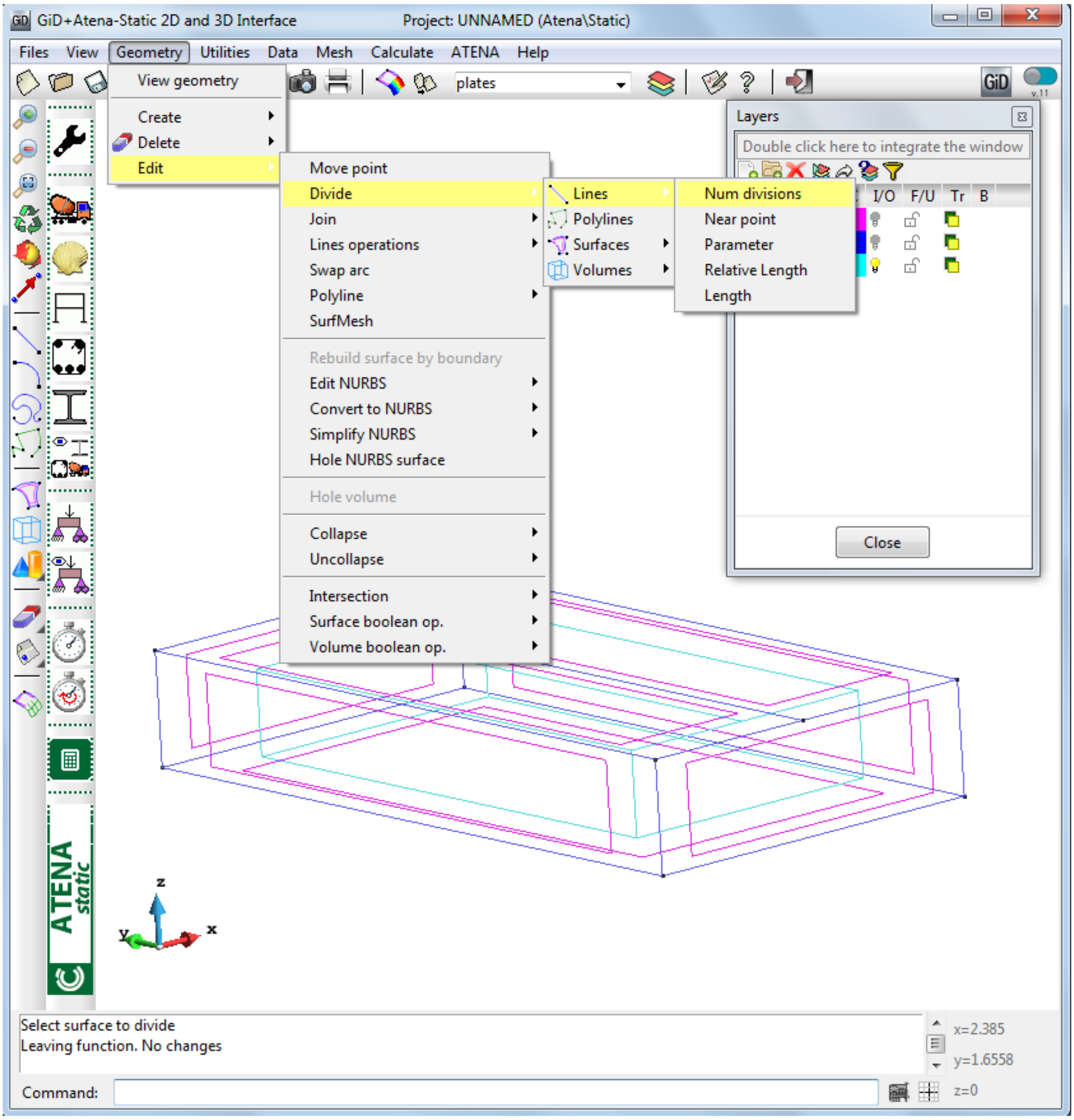


Figure 94: The dividing of the line

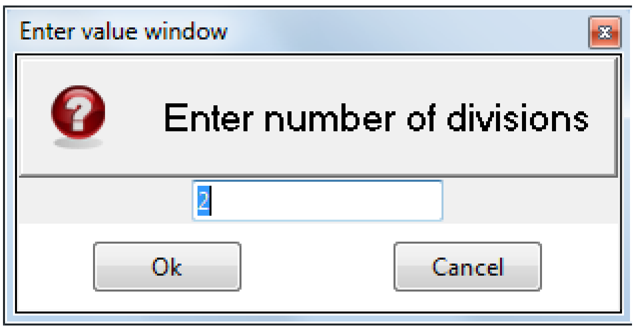


Figure 95: The enter value window

Parameter input:
Enter number of divisions: 2

After the specification of the required division the button **OK** has to be pressed and the appropriate line should be selected (see Figure 96). The line selection is completed by pressing the ESC key has (see Figure 97).

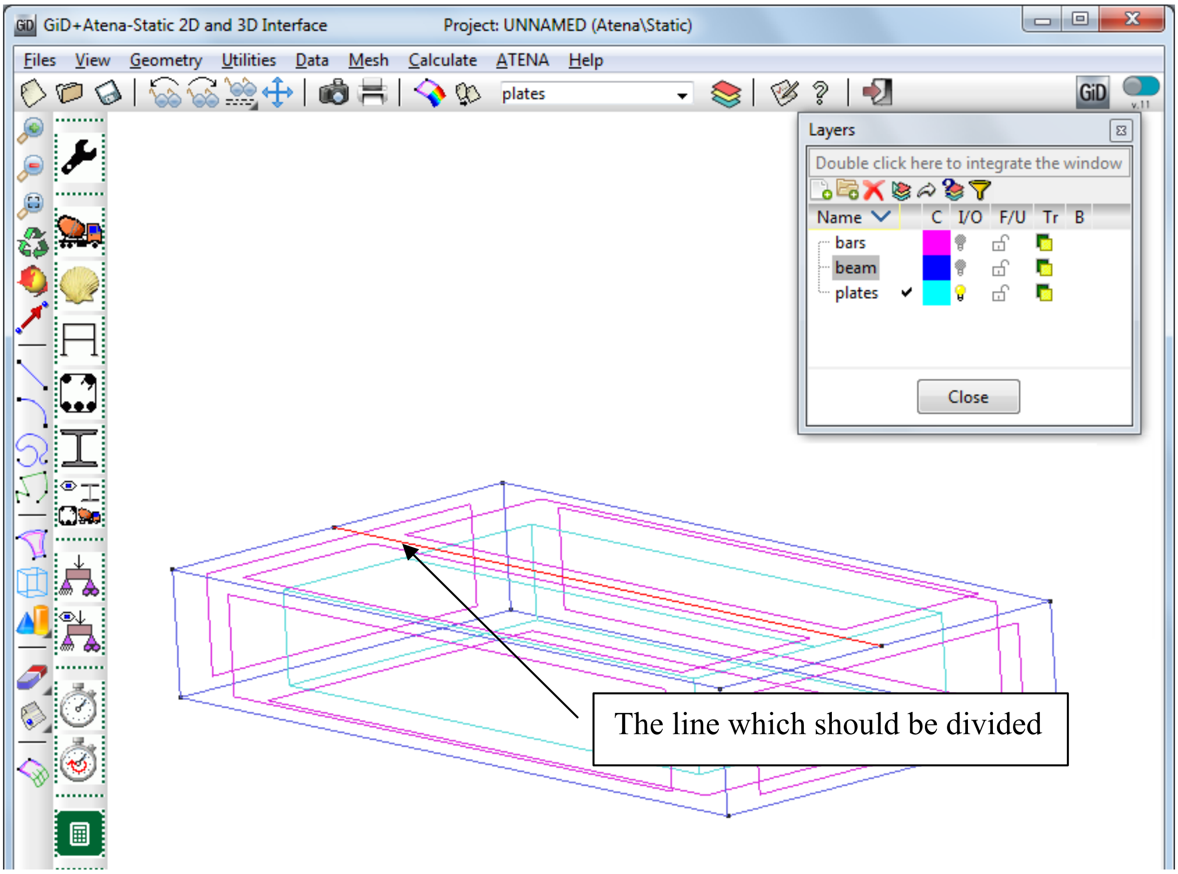


Figure 96: The selection of the line

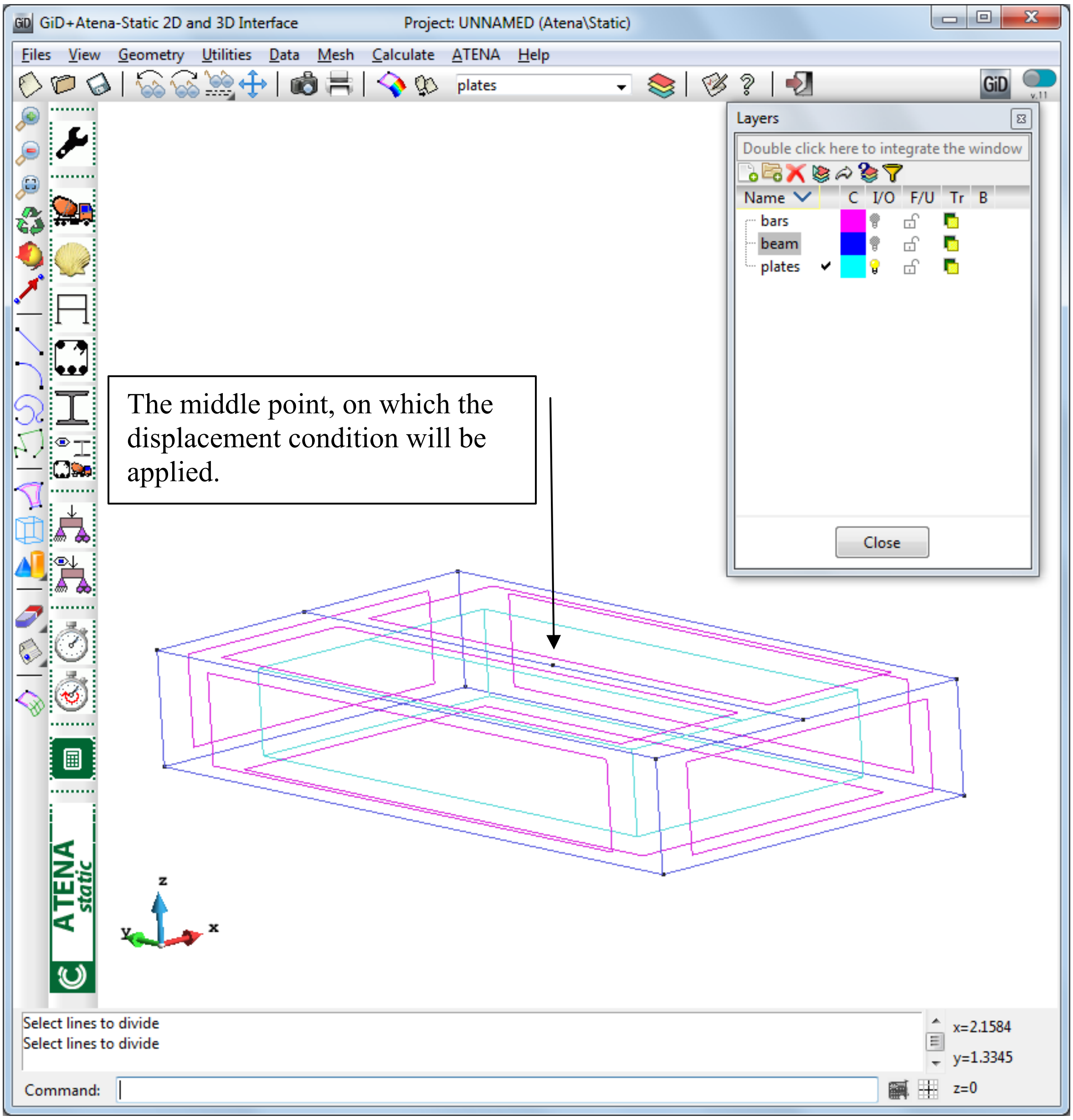
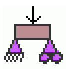


Figure 97: The divided line

Now the necessary point for the displacement condition is already created. Boundary conditions automatically belong to the same layer as the geometry, onto which they are assigned. Therefore it is not necessary to control which layer is activated.

The condition command can be executed by the icon  or by **Data | Conditions** in the main menu. The displacement condition definition is depicted in the Figure 98.

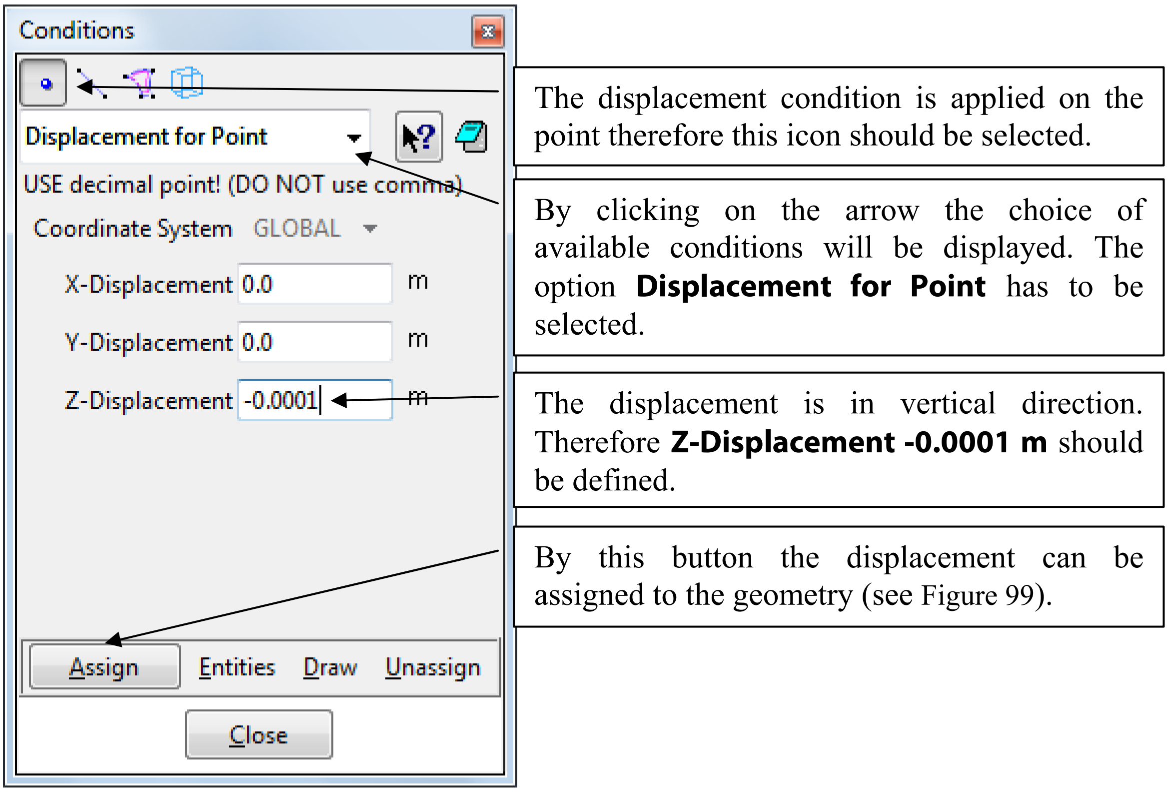
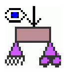


Figure 98: The displacement condition definition

> **Parameter input:**
> Displacement for Point
> Z-Displacement: -0.0001 m

By clicking on the icon  the created condition can be displayed and can be used to verify if it is correctly applied at the right locations. After clicking on that icon the displacement condition should be displayed at the point in the middle of the top plate (see Figure 100).

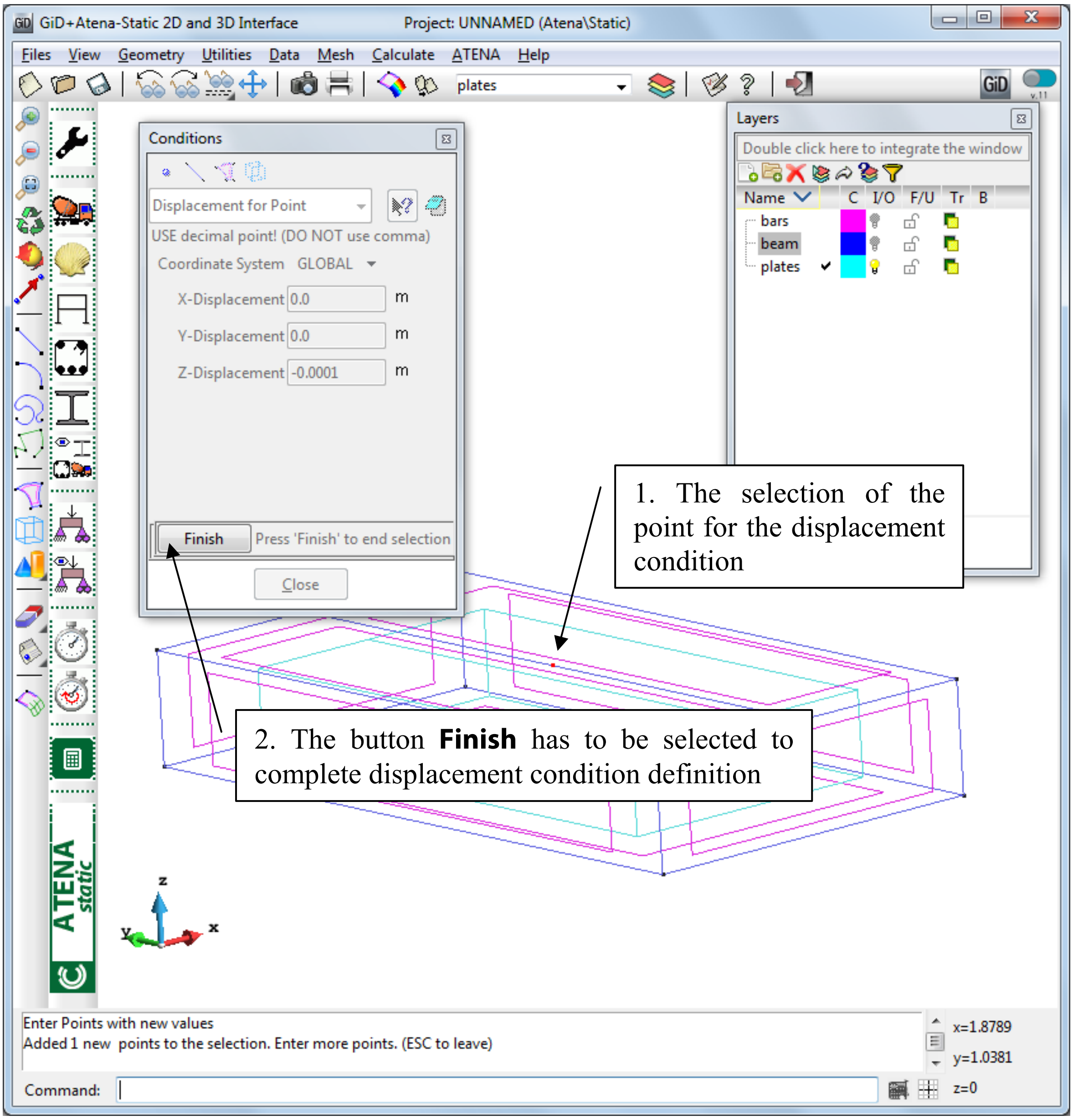


Figure 99: The selection of the point for the displacement condition

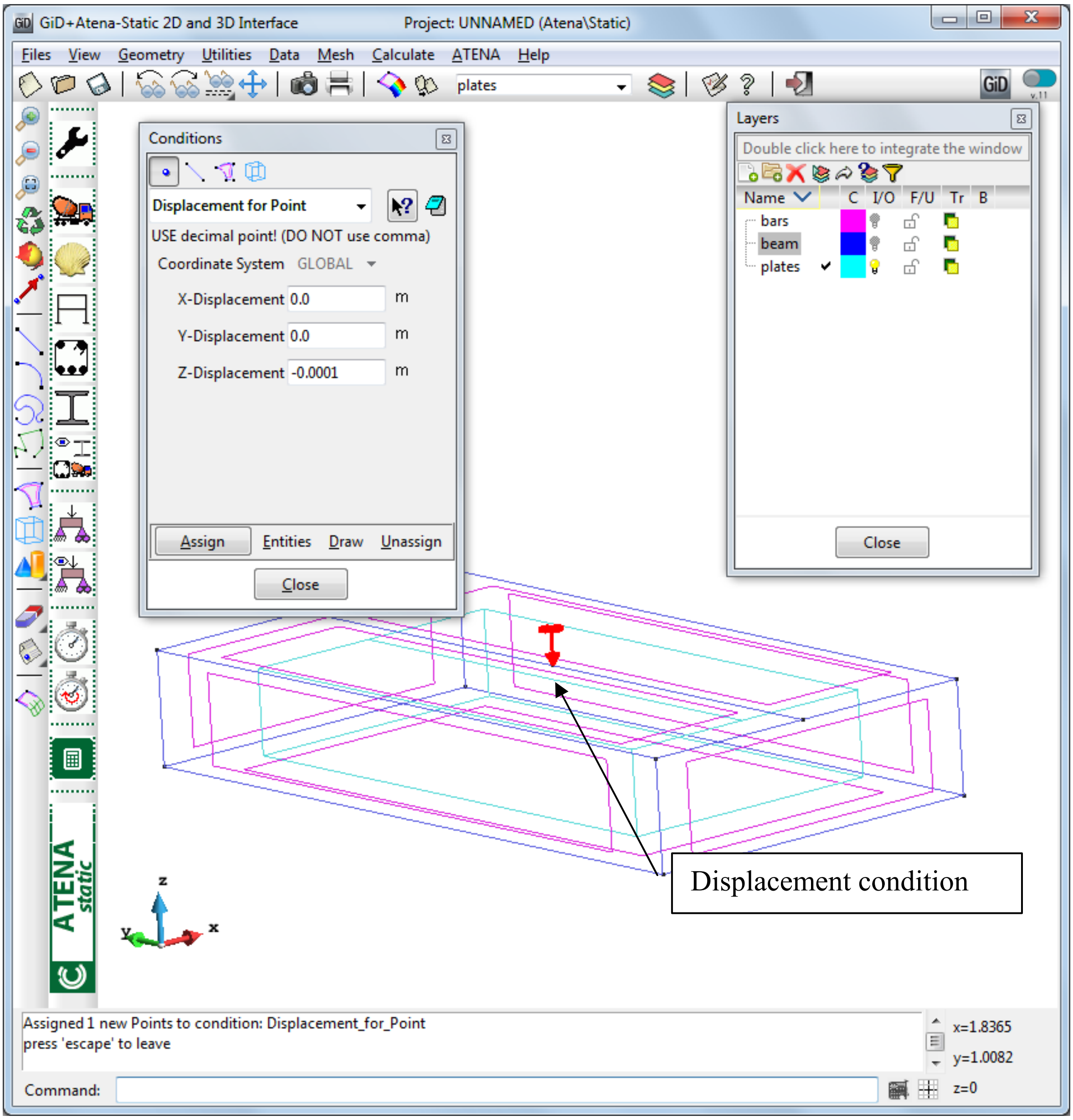
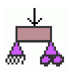


Figure 100: The visual display of the displacement condition

3.4.3 Symmetry Condition

The beam of this example is symmetrical. Therefore, only half of the beam is analysed and it is necessary to enforce the axis of the symmetry along right side of the beam. This means that the horizontal x-displacements along this side should be equal to zero. It can be done by definition of the boundary condition on the surface (see Figure 101).

Condition command can be executed by the icon  or by the **Data | Conditions** in the main menu. The symmetry condition definition is depicted in the Figure 102.

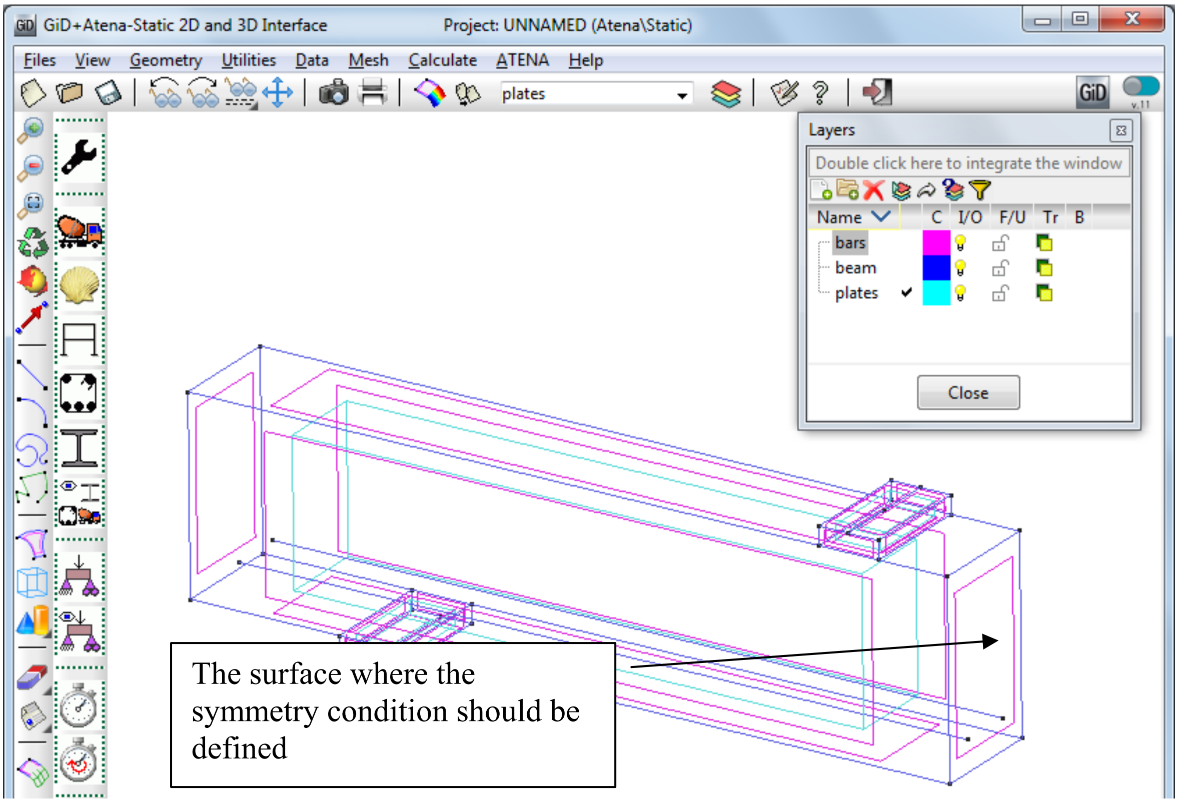


Figure 101: The surface for the symmetry condition

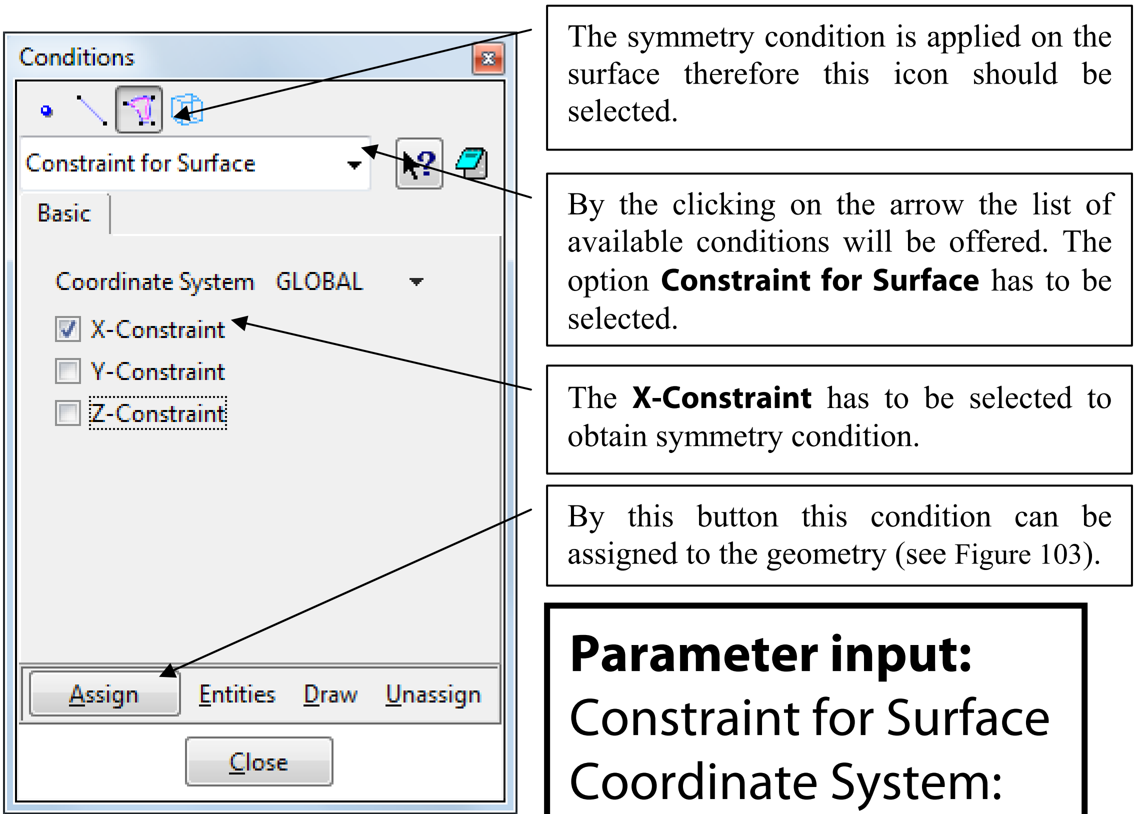
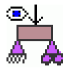


Figure 102: The symmetry condition definition

Parameter input:
Constraint for Surface
Coordinate System:
GLOBAL
X-Constraint

By clicking on the icon  the created condition can be shown in the graphical area. After clicking on that icon the symmetry condition will be displayed on the middle surface of the beam (see Figure 104).

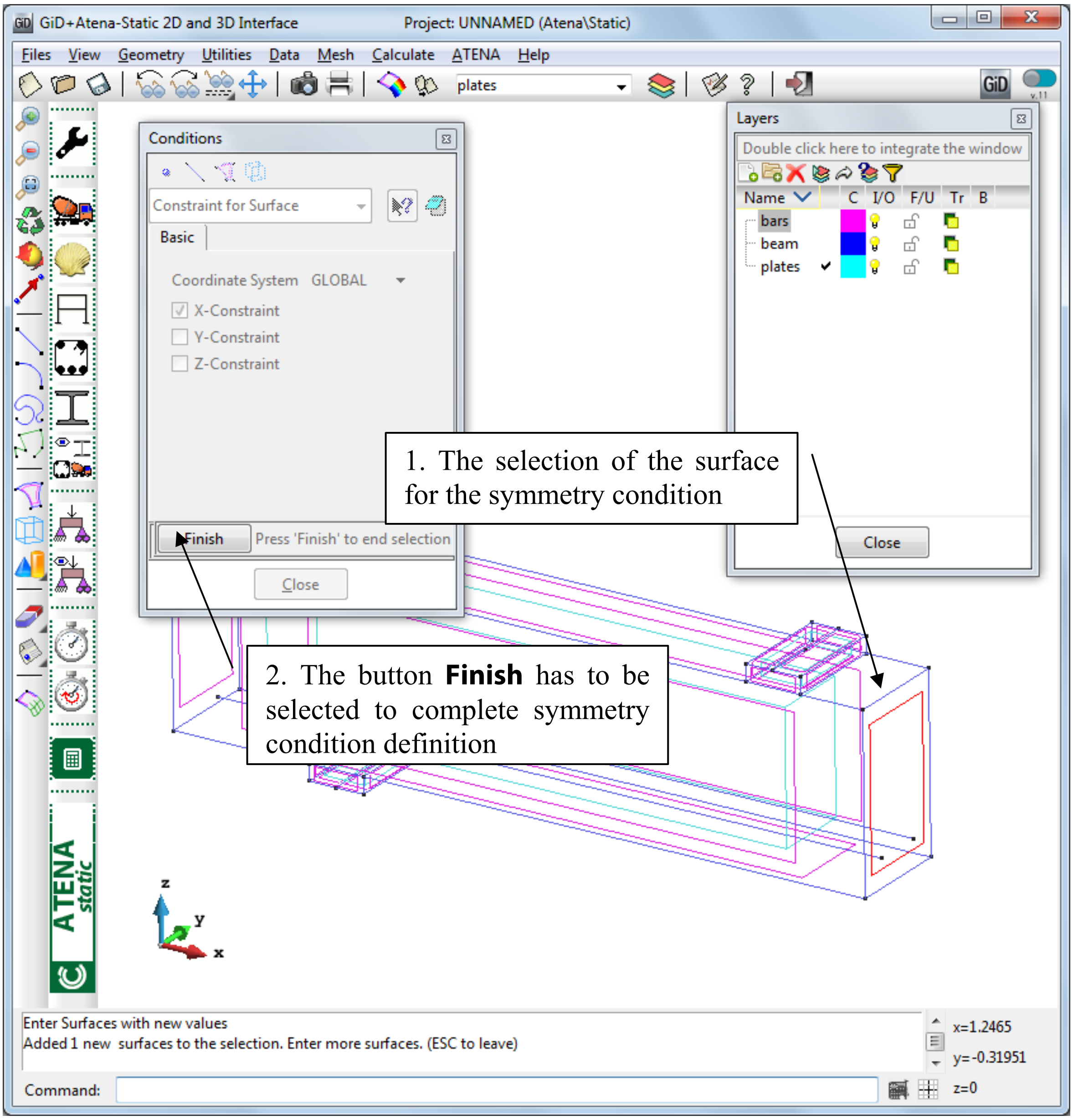


Figure 103: The selection of the surface for the symmetry condition

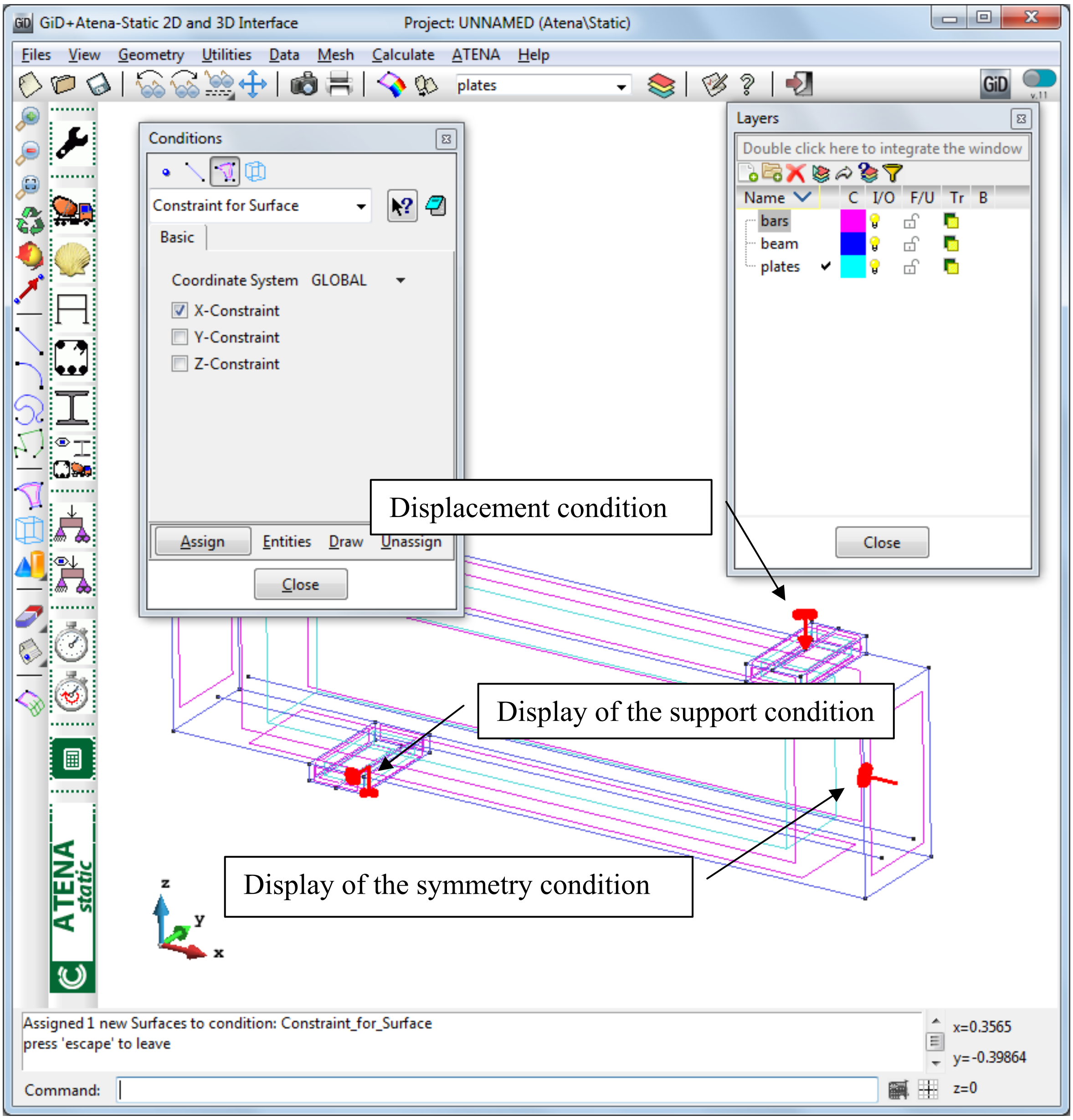


Figure 104: The visualization of the applied symmetry condition

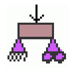
3.4.4 Monitors

Monitors provide important information about state of the structure. They can be used to monitor various important quantities during the analysis. For instance it may be interesting to monitor the development of deflections or strains at certain critical locations during the nonlinear analysis.

In this example, two monitors will be defined. One monitor will be monitoring loads on the top plate and second one will monitor deflections in the middle of the beam. The monitors are represented in **GiD** as a special condition that needs to be applied in the first interval. In this example, the monitors will be defined as a point condition at the top plate and in the middle of the beam.

3.4.4.1 First Monitor

The first monitor should be located on the top plate and it will be used to monitor the loads that are applied onto the structure. It will be applied on the point where the displacement condition is also defined. Since the loading is applied as prescribed displacement, the applied forces are represented in the finite element analysis as reactions. This means that the reaction in the z-direction should be evaluated at this monitor.

The definition of the monitor condition starts by the icon  or by executing command **Data | Conditions** in the main menu. The monitor condition definition is depicted in the Figure 105.

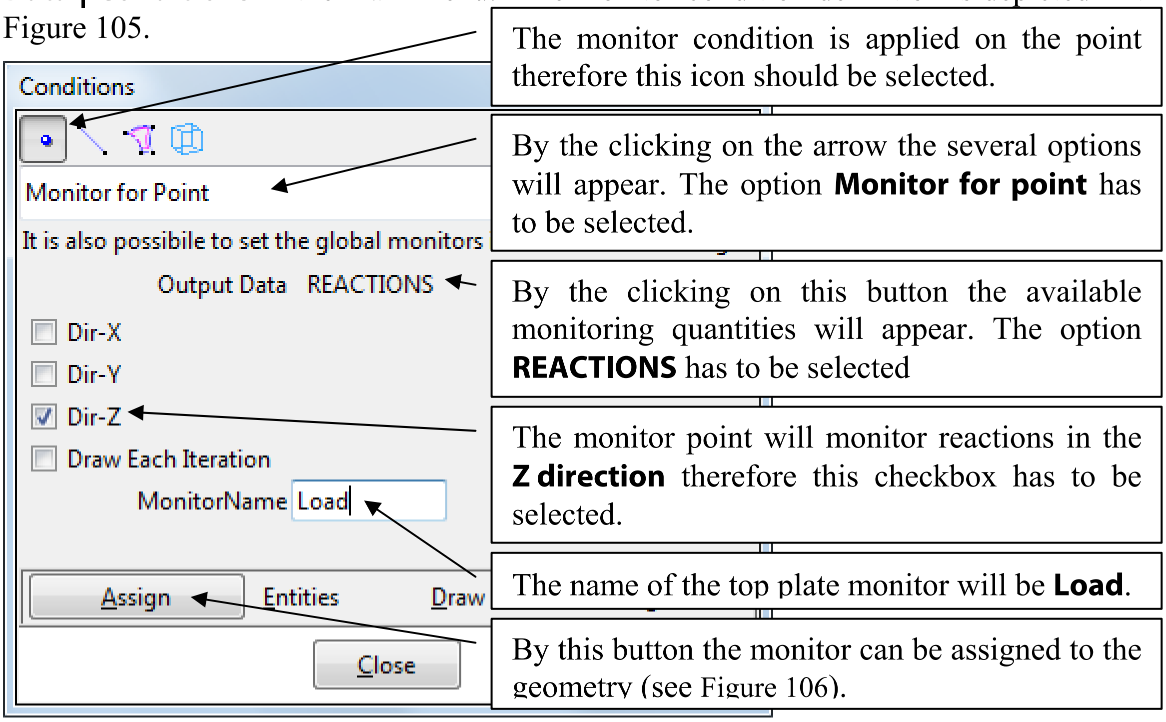


Figure 105: The first monitor condition definition

Parameter input:
Monitor for point
Output Data: REACTIONS
Dir-Z
Monitor Name: Load

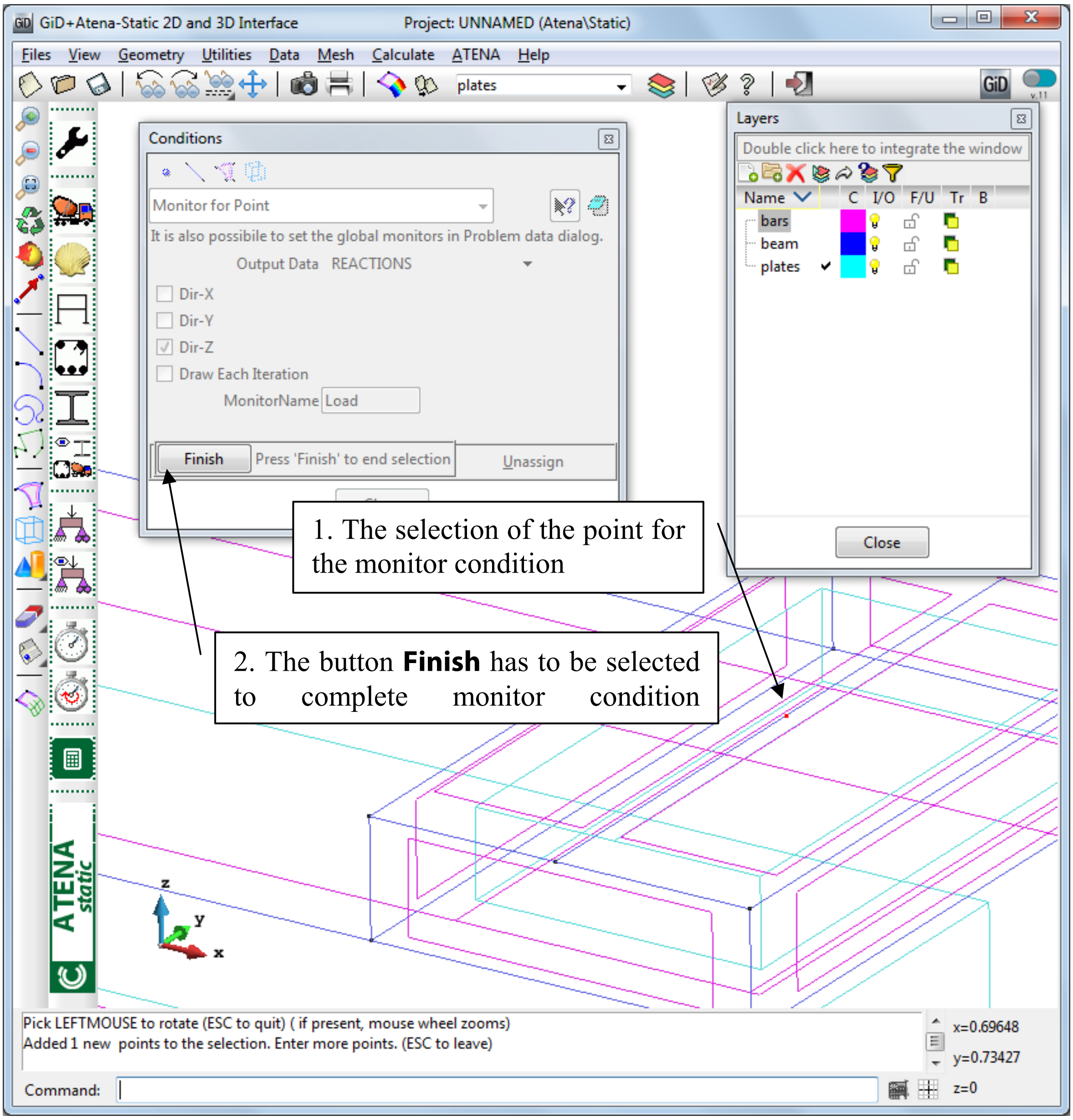
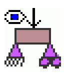


Figure 106: The selection of the first monitoring point

By clicking on the icon  the created condition can be drawn. After clicking on that icon the monitor condition will be displayed at the point in the top plate (see Figure 107).

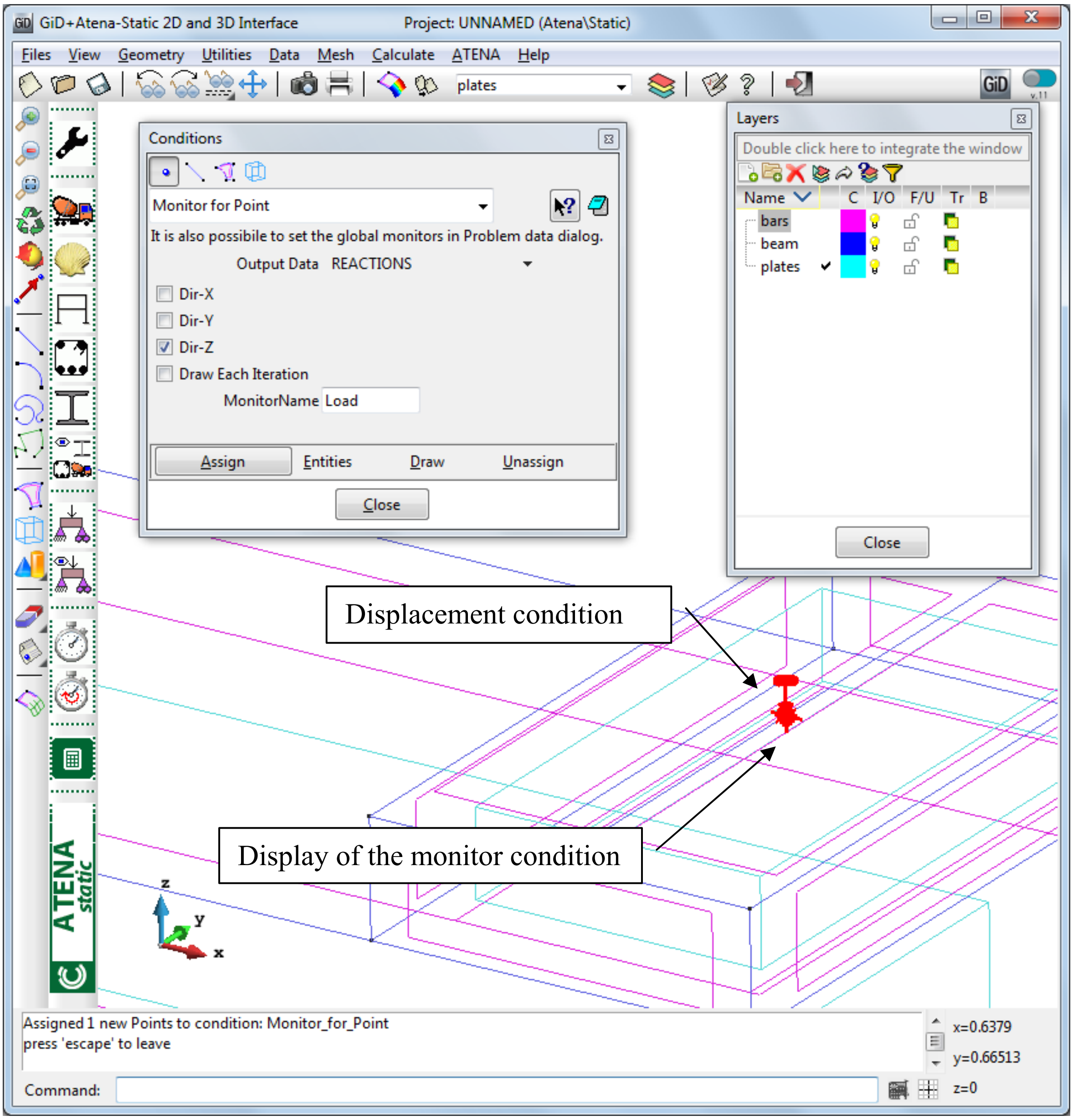
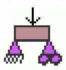


Figure 107: The first monitor condition

3.4.4.2 Second Monitor

The second monitor point should be located at the middle of the beam near its bottom surface, where the largest vertical displacement can be expected. The displacement in the z-direction should be evaluated at this location.

The conditions command can be executed by the **Data | Conditions** in the main menu or by the icon . The second monitor condition definition is depicted in the Figure 108.

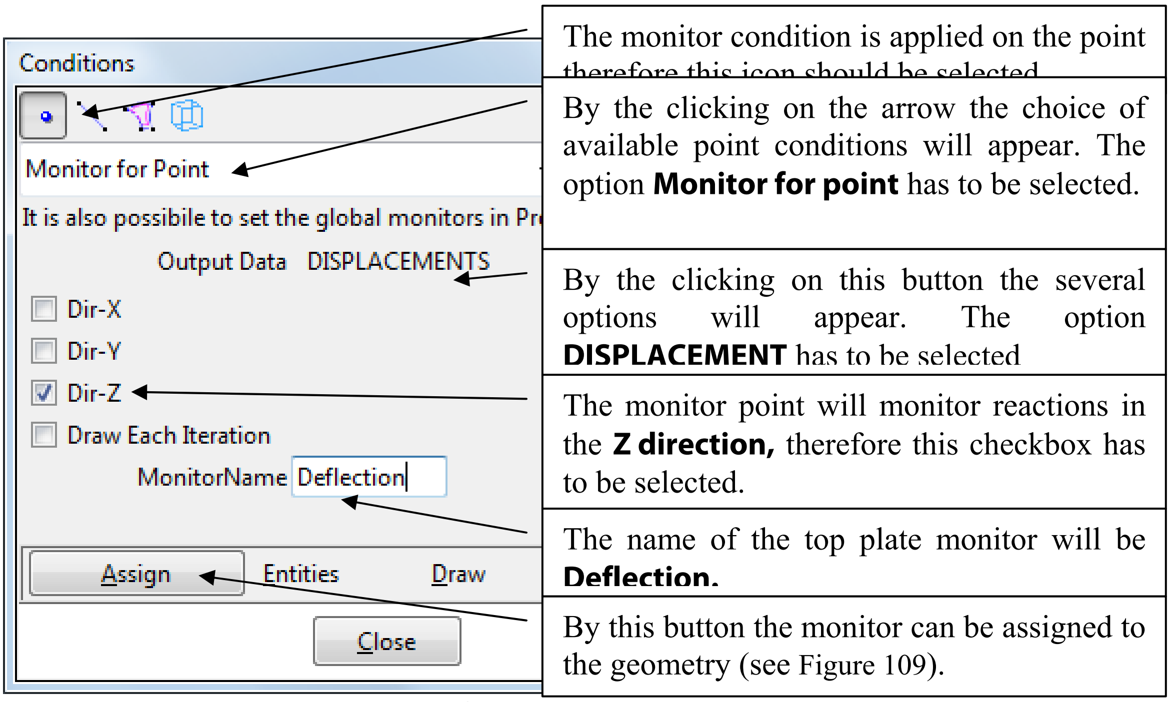
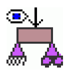


Figure 108: The second monitor condition definition

Parameter input:
Monitor for point
Output Data: DISPLACEMENT
Dir-Z
Monitor Name: Deflection

By clicking on the icon  the created condition can be drawn. After clicking on that icon the monitor condition will be displayed in the middle of the beam (see Figure 110).

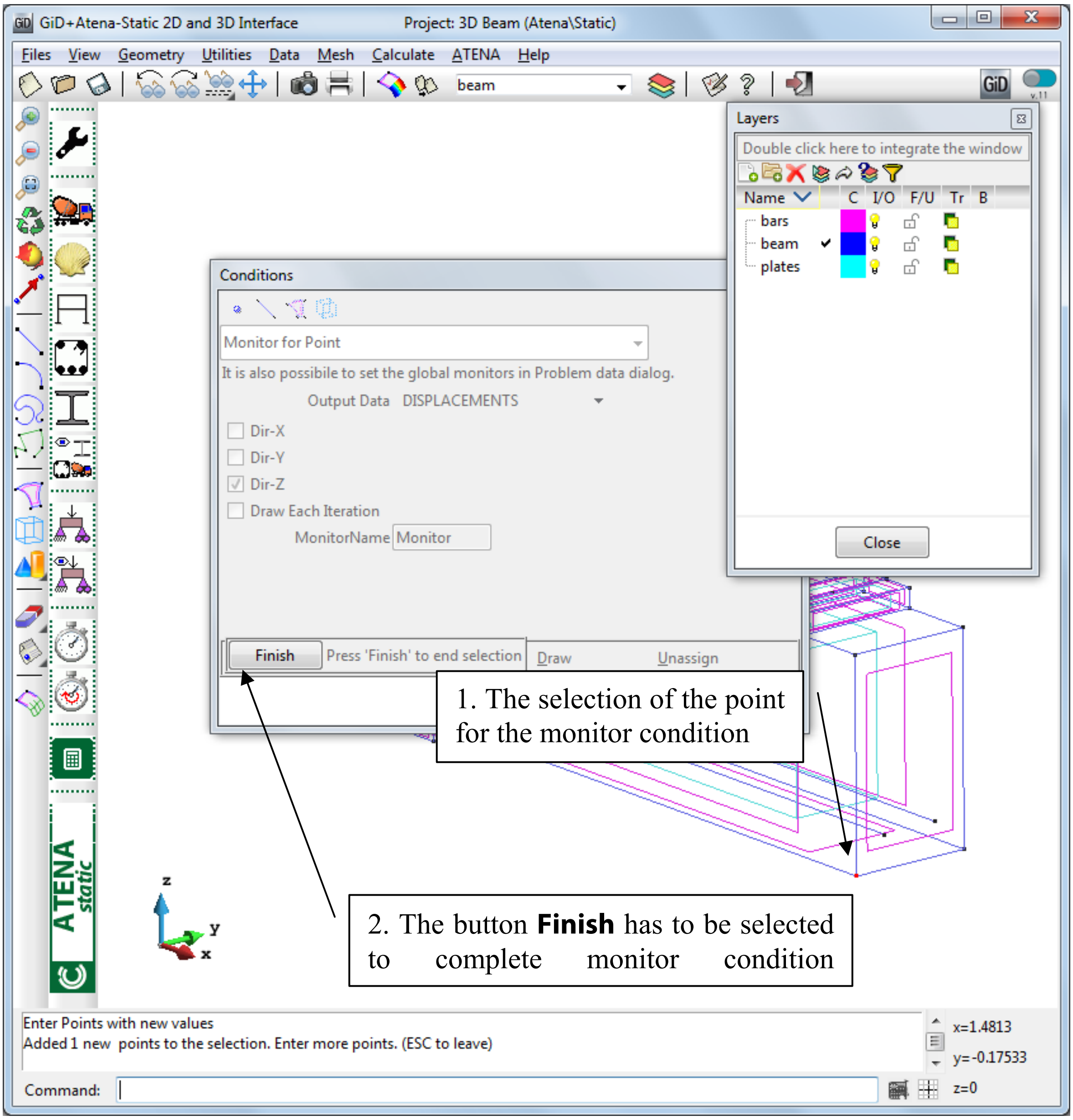


Figure 109: The selection of the second monitoring point

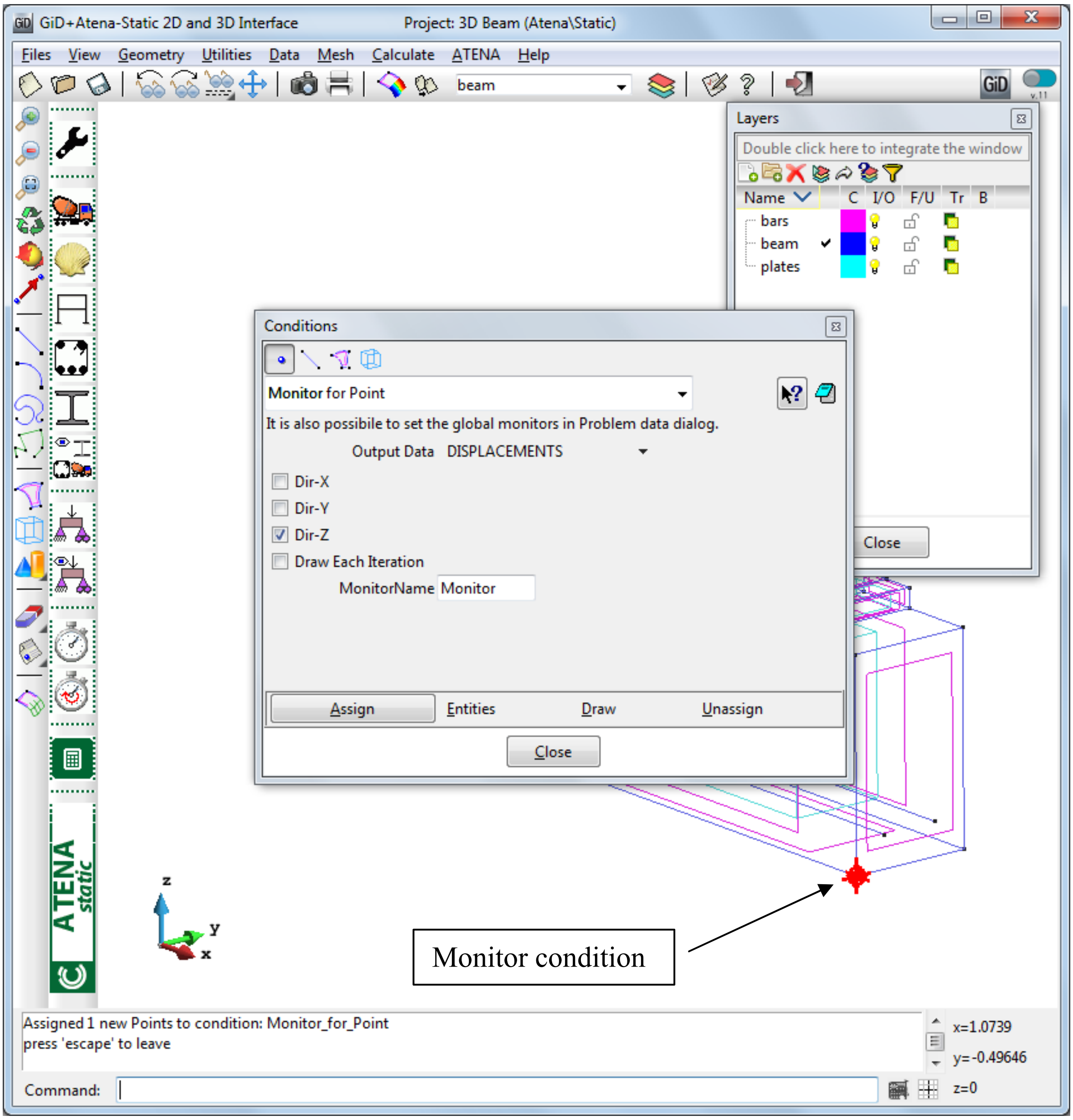
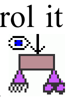


Figure 110: The second monitor condition

Now, all boundary condition should be defined. For control it is recommended to display boundary condition. It can be done by clicking on the icon  (see Figure 148).

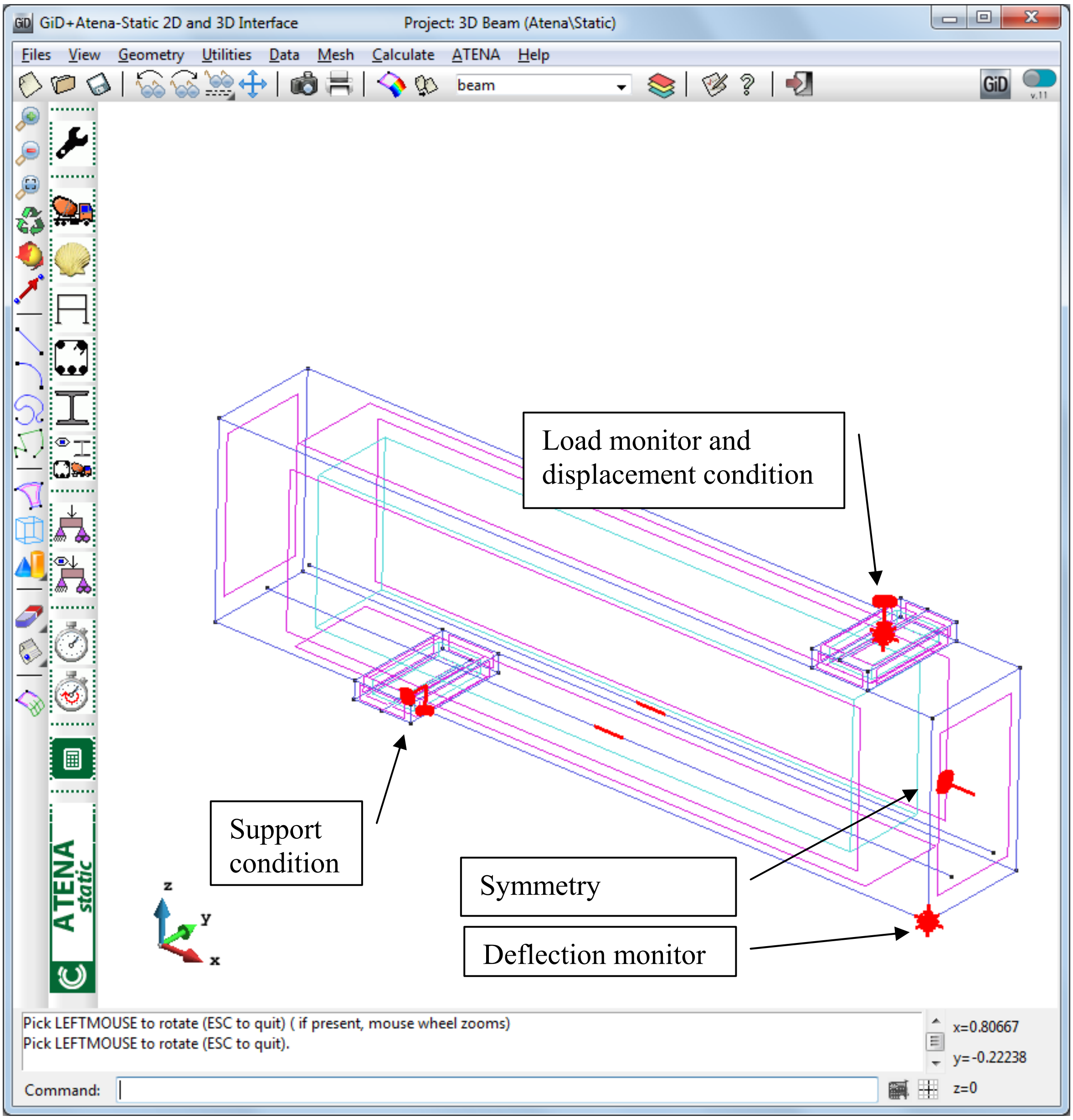


Figure 111: All boundary conditions

3.5 Intervals – Loading History

This section describes the definition of the loading history for the analysis of Leonhardt's shear beam. The loading history in **GiD** consists of intervals. Each interval is divided into load steps. Because in this case the structure is loaded by only one type of force (defined displacement), only one interval will be used. Then this interval will be subdivided in several steps.

The objective is to gradually increase the load up to failure. Very often before an analysis is started it is difficult to estimate the required loading level that would lead to failure. The maximal load level however, can be often estimated either by simple hand calculation or by performing an initial analysis with a very small load level. Then from the resulting stresses it is possible to estimate how much the load must be increased to fail the structure.

In this example, it is known from the experimental results that the beam should fail at the deflection of about 0.003 m. In previous section the prescribed displacement of 0.0001 m was applied at the top plate. This means that the predefined displacement should be multiplied approximately 30 times to reach the failure. Base on this assumption, the Load interval will be multiplied by 40. Naturally, such a load should not be applied to the structure in one moment. Therefore it is necessary to subdivide the interval in several load steps. In this case the interval will be divided in 50 load steps.

The loading history can be prescribed by selecting item **`Data | Interval Data`** in the main menu (see Figure 112). After selection of this command the Interval data window will appear and the data which should be defined are depicted in the Figure 113.

Detailed information about Loading history you can find in (ATENA GiD User's Manual - Interval data - Loading history). Some examples are in (Example Manual ATENA Science - Tutorial for Construction Process)

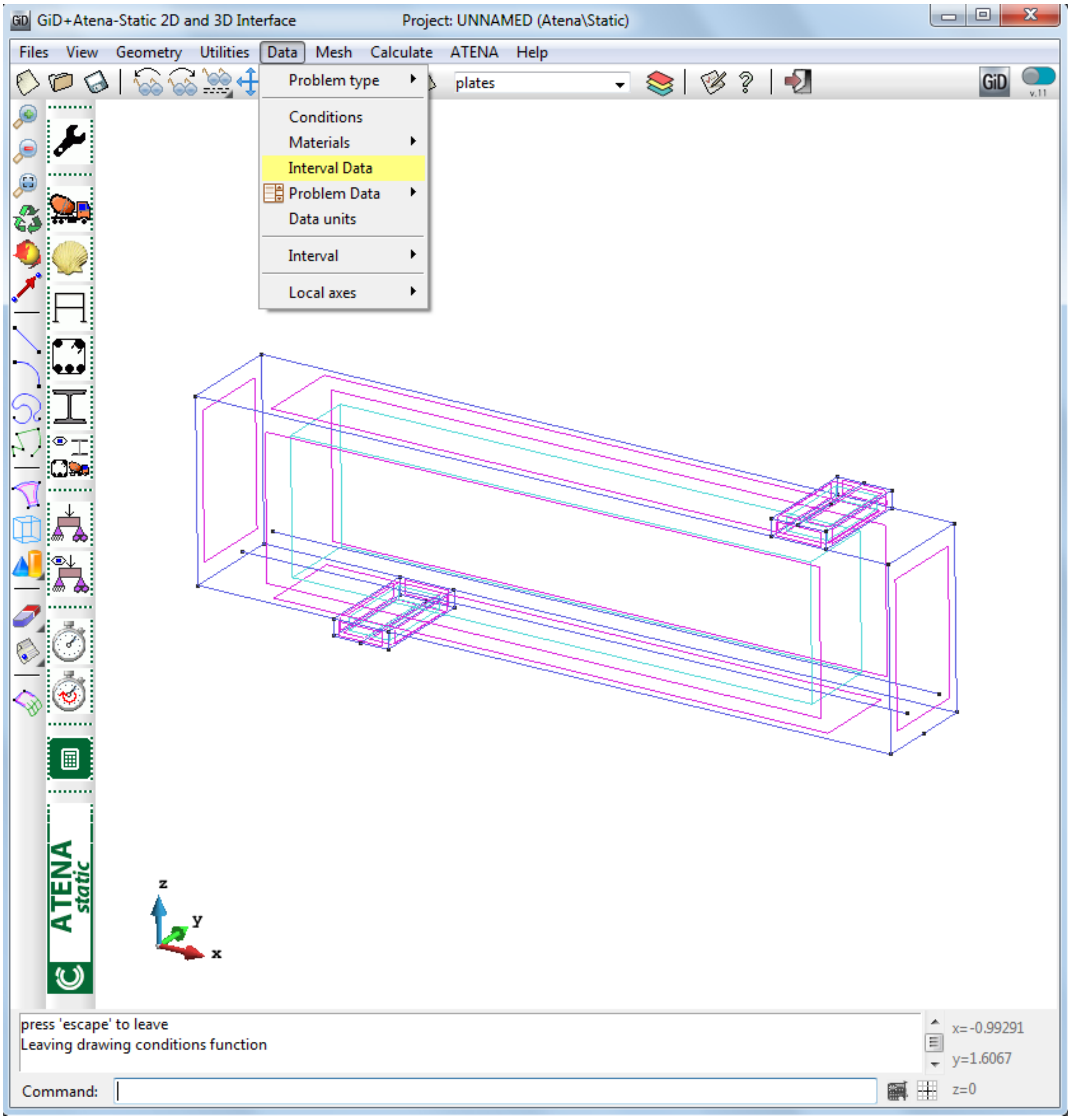


Figure 112: The Interval data command

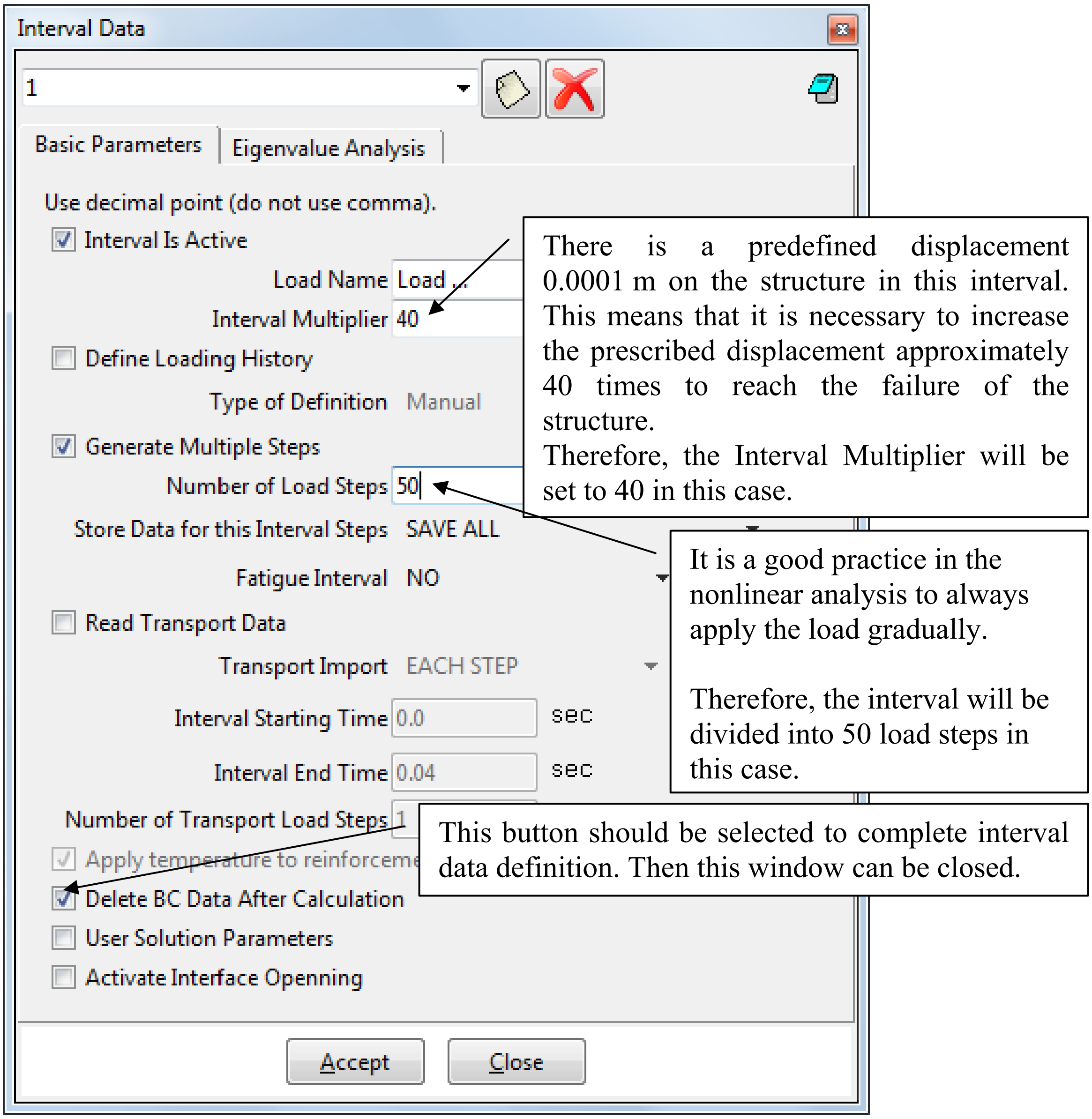


Figure 113: The contact conditions

Parameter input:

Interval Multiplier:	40
Number of Load Steps:	50

3.6 Mesh Generation

The generation of a finite element mesh is the last step in pre-processing. Because it should be possible to create this tutorial example in demo version of **ATENA** and **GiD** it is necessary to use a rather coarse mesh. The demo version of **ATENA** is limited to 300 elements so the generated model should satisfy this limit.

The easiest way of the mesh definition is to use automatic generation. Program will automatically define the smallest suitable mesh. This command can be executed by selecting **Mesh | Generate mesh** (see Figure 114) or this option can be activated directly by pressing the key Ctrl and 'g' at the same time. Then the program asks for the definition of the size of the generated mesh (see Figure 115). The default size of the mesh can be used. By the selecting **Ok** button the mesh will be generated and the list of elements and nodes of the mesh will appear (see Figure 116).

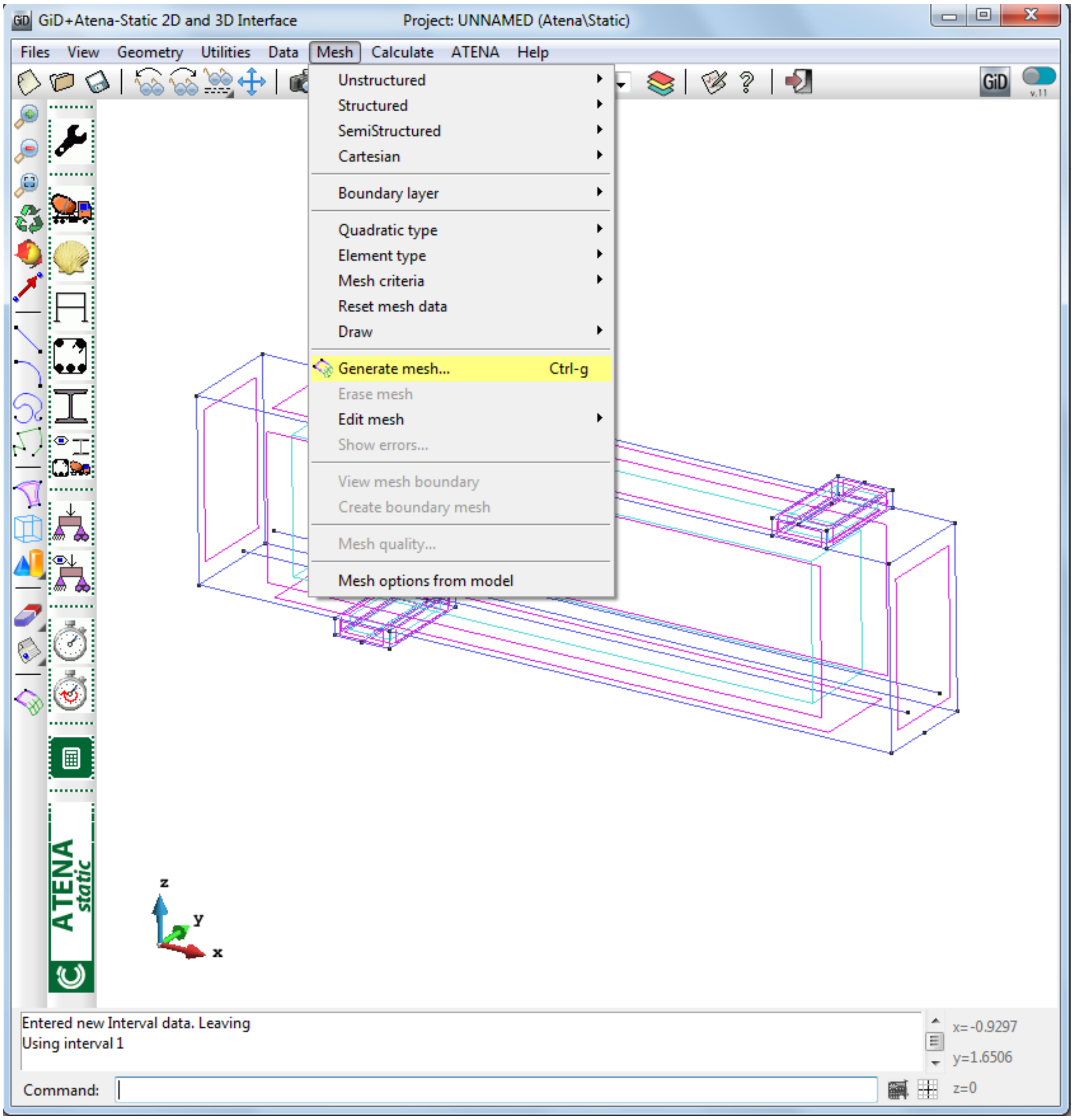


Figure 114: The Generate mesh command

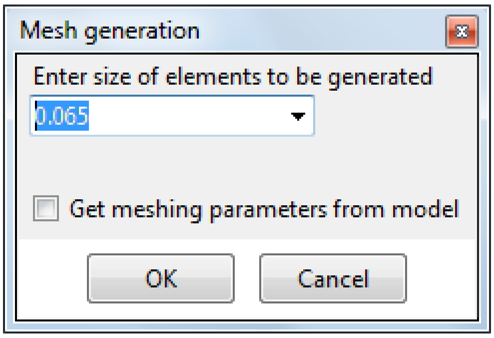


Figure 115: The program offer the size of mesh

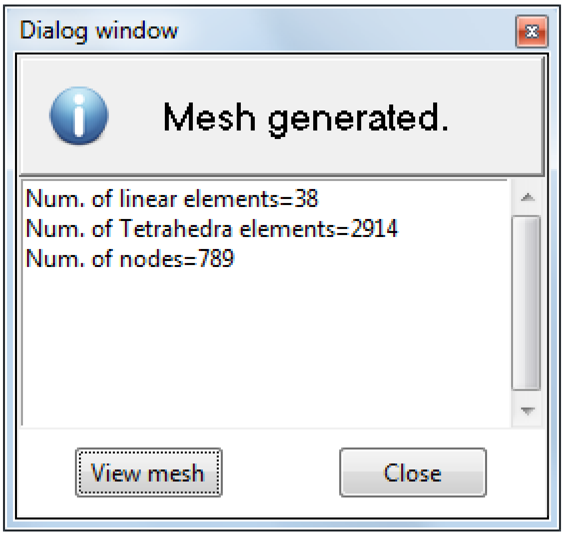


Figure 116: The numbers of elements and node of this geometrical model

The demo version of the **GiD** is limited to 1000 nodes. The example of this tutorial contains 789 nodes. Therefore the automatic sized mesh could be generated (see Figure 117).

But the demo version of **ATENA** is limited to 300 elements (see Figure 118). And this mesh contains almost 3000 elements; therefore this mesh will not be functional in **ATENA** and the number of element should be decreased.

It can be done by using the structured mesh option, which allows better control about the number of generated elements. Also in structural analysis it is usually preferred to use brick elements. Therefore, in the next steps of the mesh generation the option to create six side brick element will be described.

In this case the structured mesh will be specified only for the beam volume because it is an important part of the structure for the structural analysis.

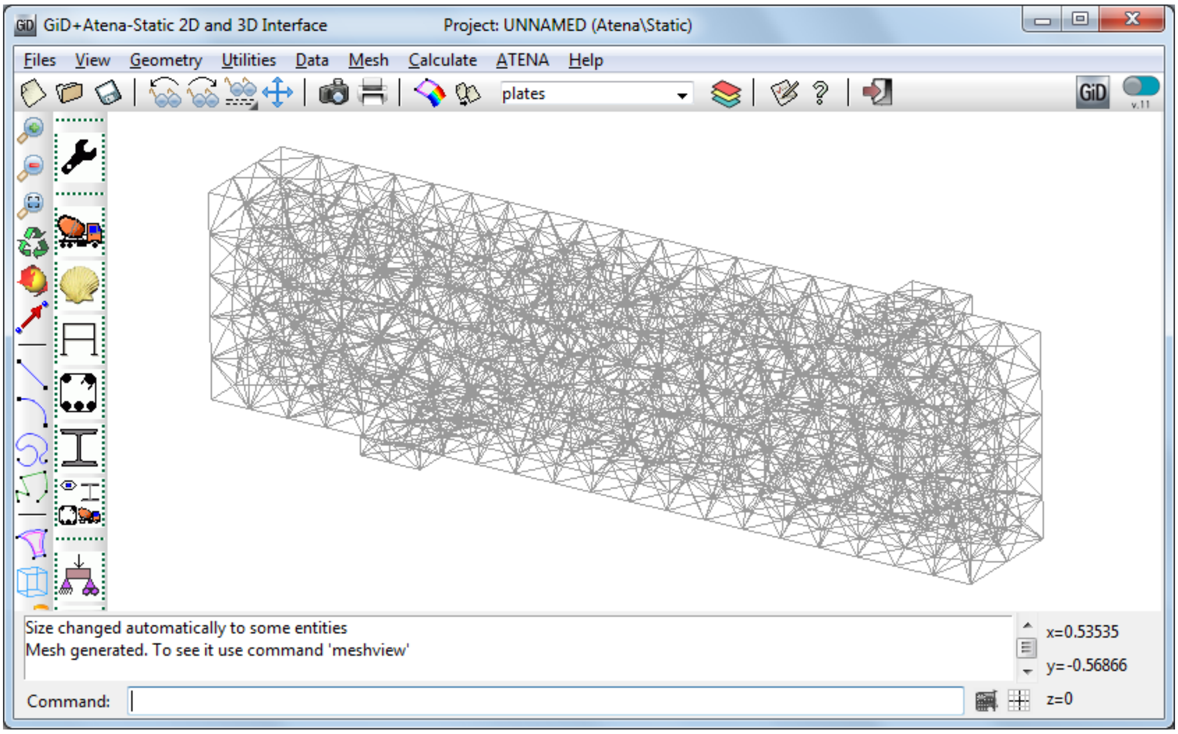


Figure 117: The generated mesh

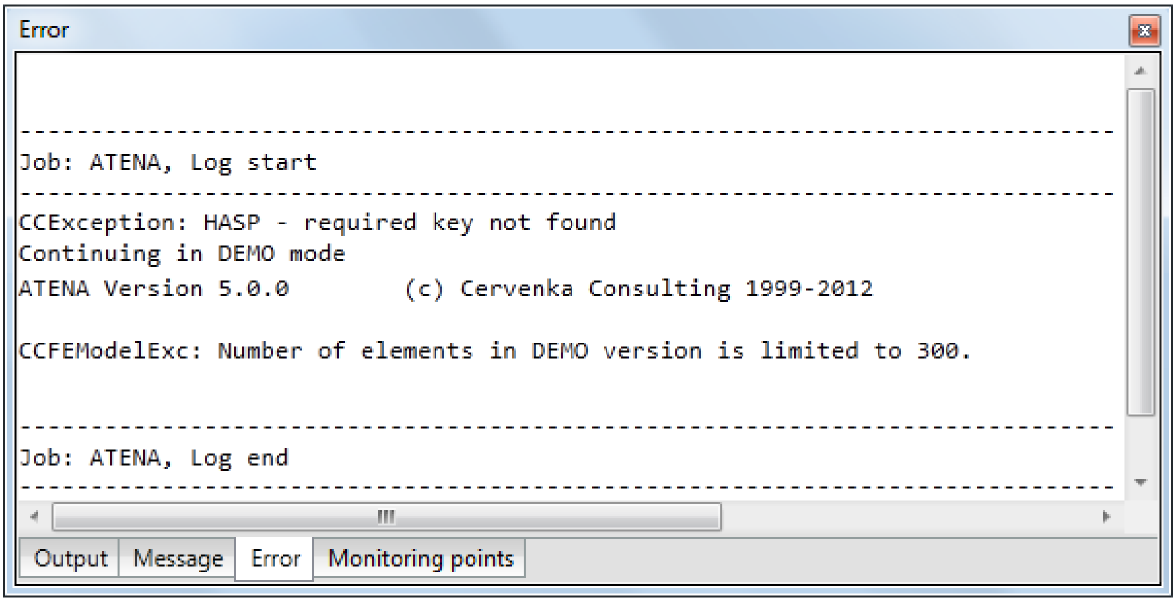


Figure 118: The limitation of demo version of ATENA

In this example the beam will have structured brick mesh, and steel plates will be meshed with tetrahedral elements.

3.6.1 Notes on Meshing

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

A bad mesh, like a single layer of volume elements in a region where bending plays a significant role, can produce very wrong results – see the "Mesh Study" example in the **ATENA Engineering Example Manual**. A minimum of 4-6 elements per thickness is recommended for at least qualitative results in bending. Alternatively, shell elements may be used (see section *Shell Material* in the **User's Manual for ATENA-GiD**).

3.6.2 Structured Mesh

Because this example should be possible to create in demo version, the mesh of the beam volume will be structured and limited to 300 elements. The finite element size should be 5 elements over the beam height, 2 elements over the beam width and 16 elements over the beam length. It should be noted that such a mesh is not an optimal one for this problem type, but our mesh size is limited by the capacity of the demo version of the program. In real structural problems finer meshes should be used.

The structured mesh is done by command **Mesh | Structured | Volumes | Assign number of cells** in the main menu.

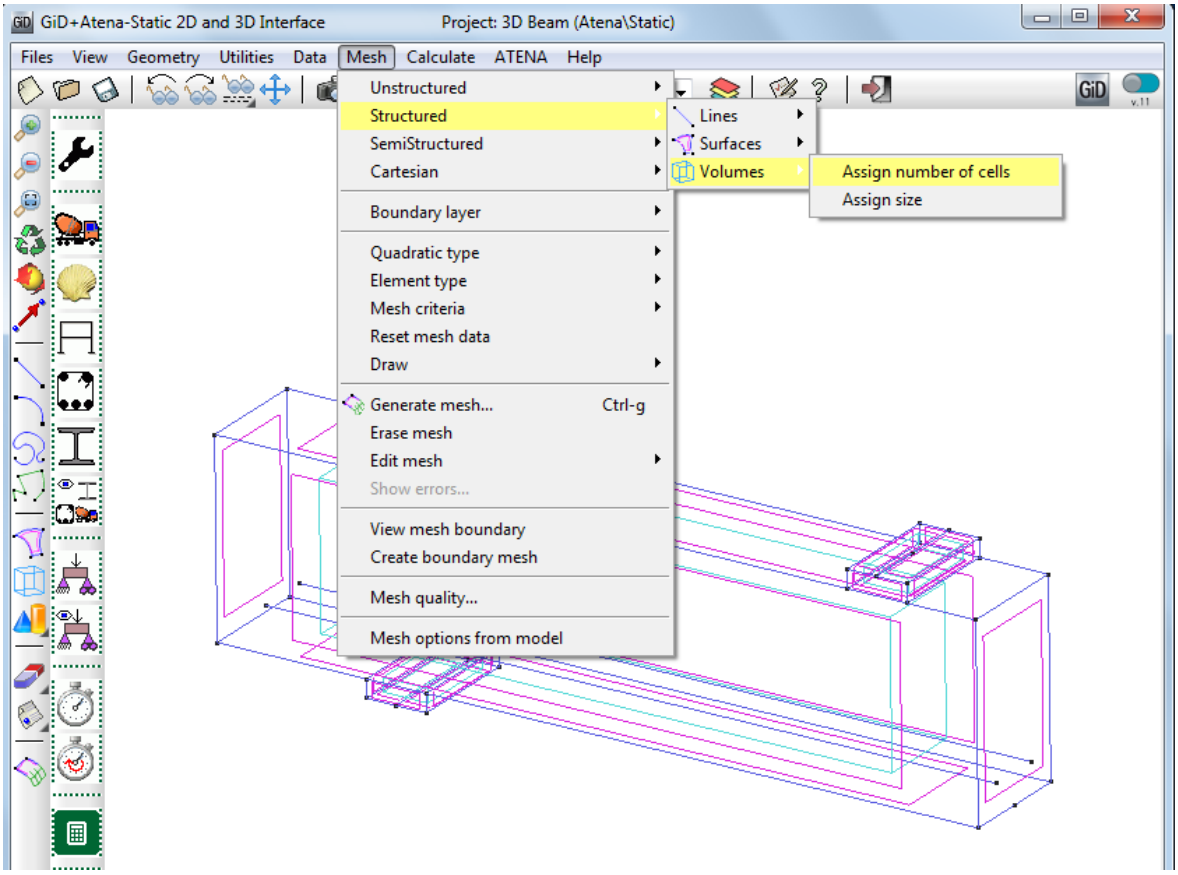


Figure 119: The Structured mesh command

Once this command is executed the volume, which should be structured has to be selected (see Figure 120). After the selection the program asks for the number of cells which should be assigned to the lines (see Figure 121).

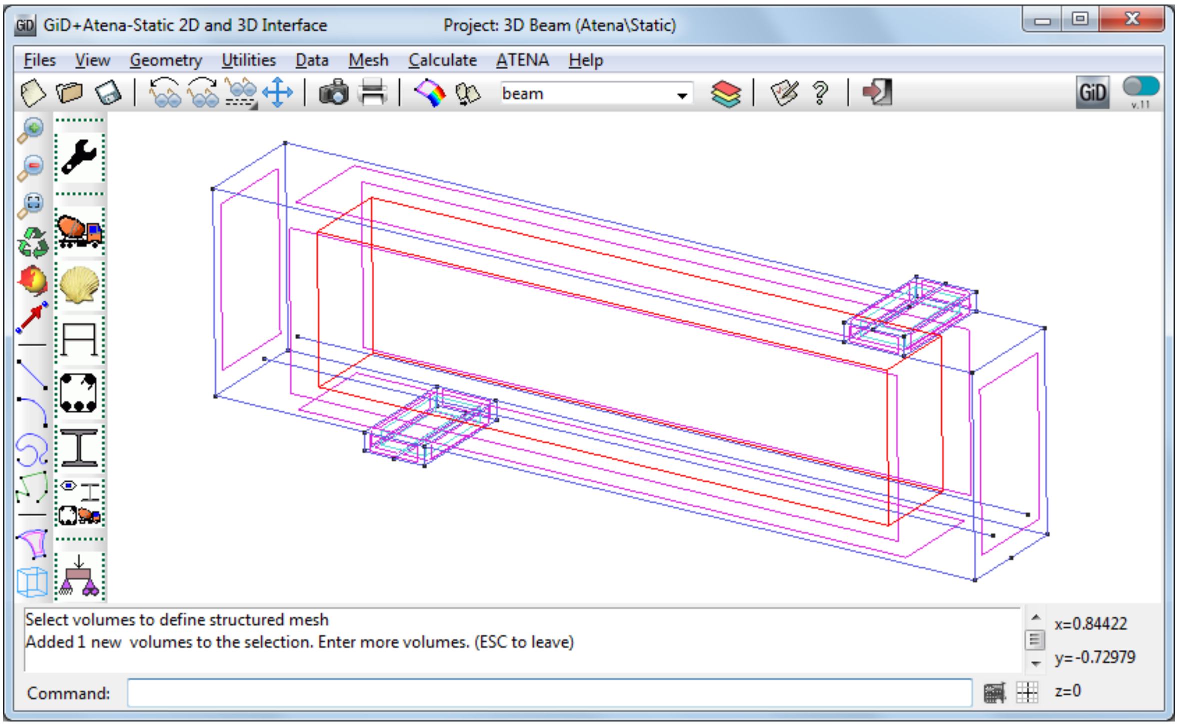


Figure 120: The selection of the beam volume which should be structured

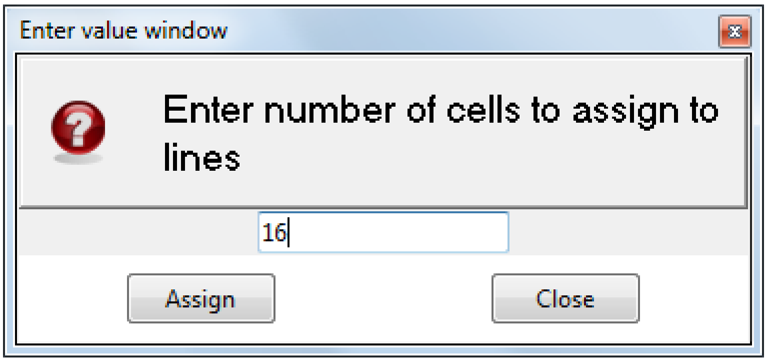


Figure 121: The number of cells for length of the beam

Parameter input: Enter number of cells to assign to lines: 16

When the number of cells is defined, the button **Assign** has to be pressed to select lines which should be structured. The 16 cells will be assigned to the beam length. When one of the longitudinal edges of the beam volume is selected, the program automatically detects which lines should have the same number of cells to guarantee the generation of a structured mesh. (see Figure 122).

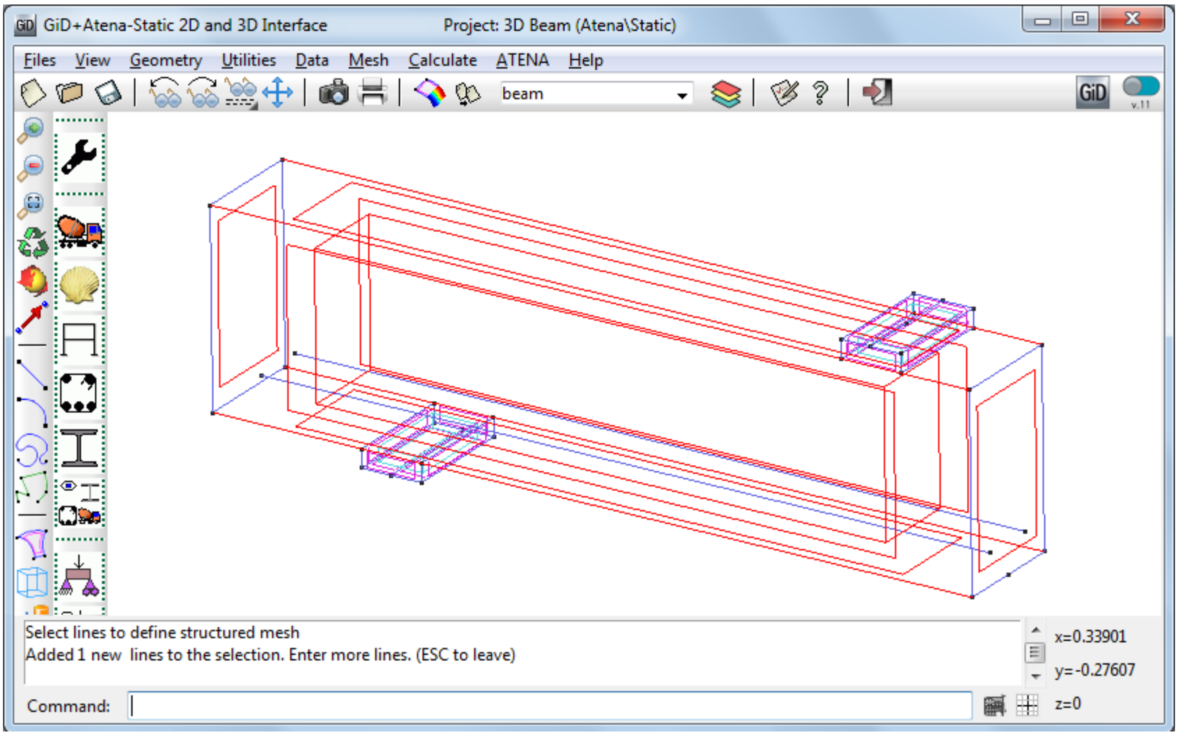


Figure 122: The selection of the length lines

After the selection the ESC key should be pressed to return to the definition of number of cells. Then the 5 cells should be defined and assigned to the height of the beam (see Figure 123). Then the selection can be done by selecting the button **Assign**, and one of the vertical beam edges shall be selected. (see Figure 124).

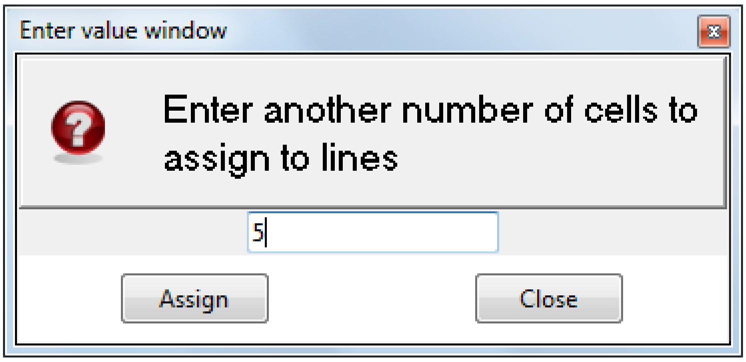


Figure 123: The number of cells for height of the beam

Parameter input:
Enter number of cells to assign to lines: 5

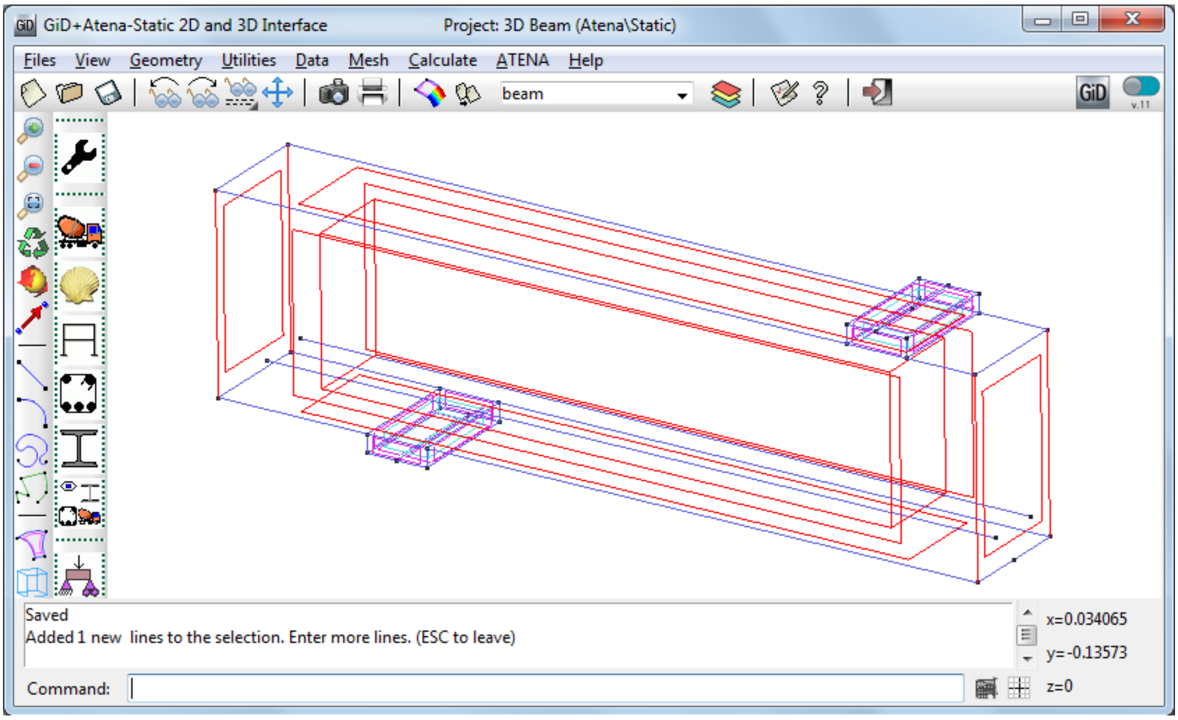


Figure 124: The selection of the height lines

Next step is to assign the number of element along the width of the beam. The 2 cells should be defined to these lines. Procedure is same like in previous two examples.

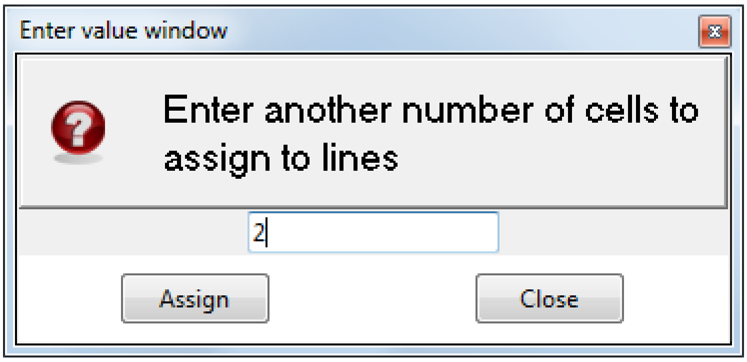


Figure 125: The number of cells for width of the beam

Parameter input: Enter number of cells to assign to lines: 2

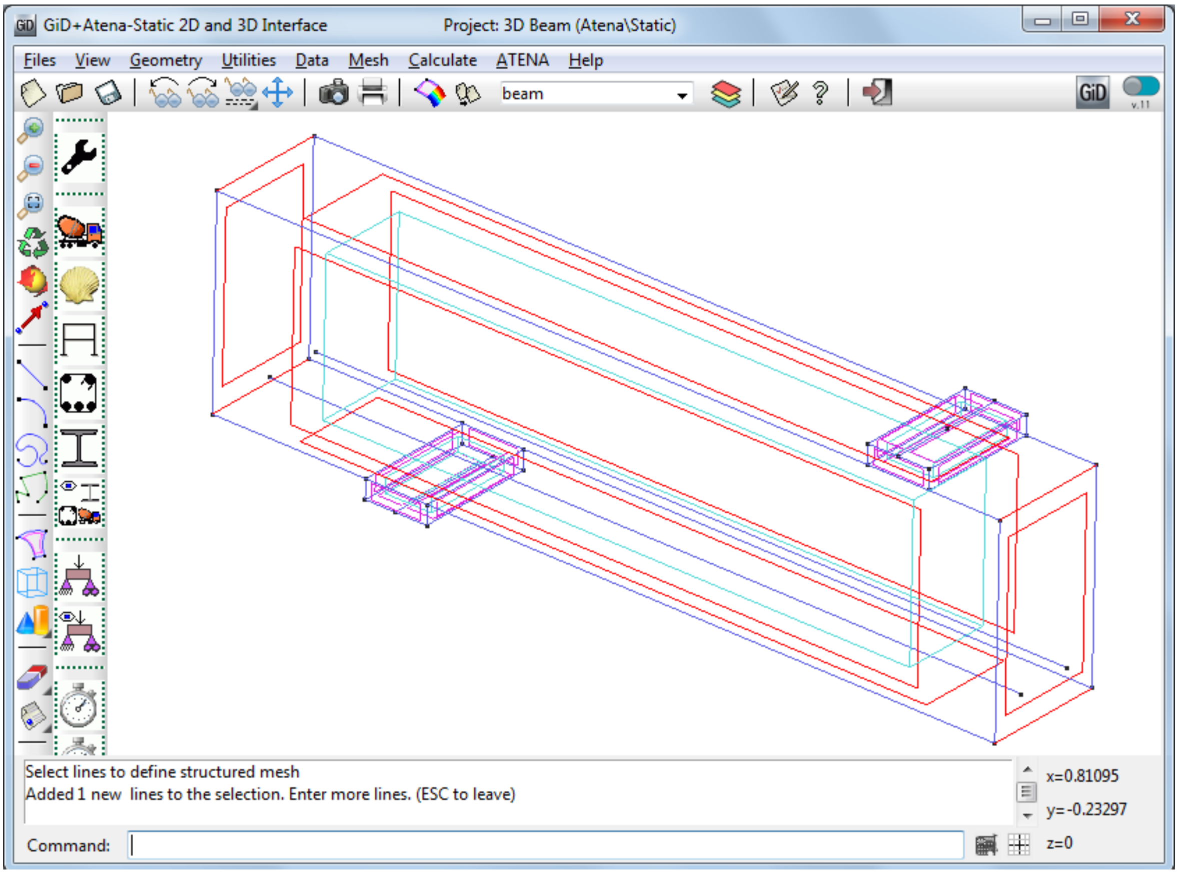


Figure 126: The selection of the width lines

Now all necessary divisions are defined and the command is completed by selecting the **Close** button in the Enter value window above.

After the structured mesh definition the element type have to be changed. Predefined element type is tetrahedra. It is better to use hexahedra mesh. It is done by command **Mesh | Element type | Hexahedra** (see Figure 127). Then the volume of beam has to be select as on the Figure 128. Use Escape button to finish.

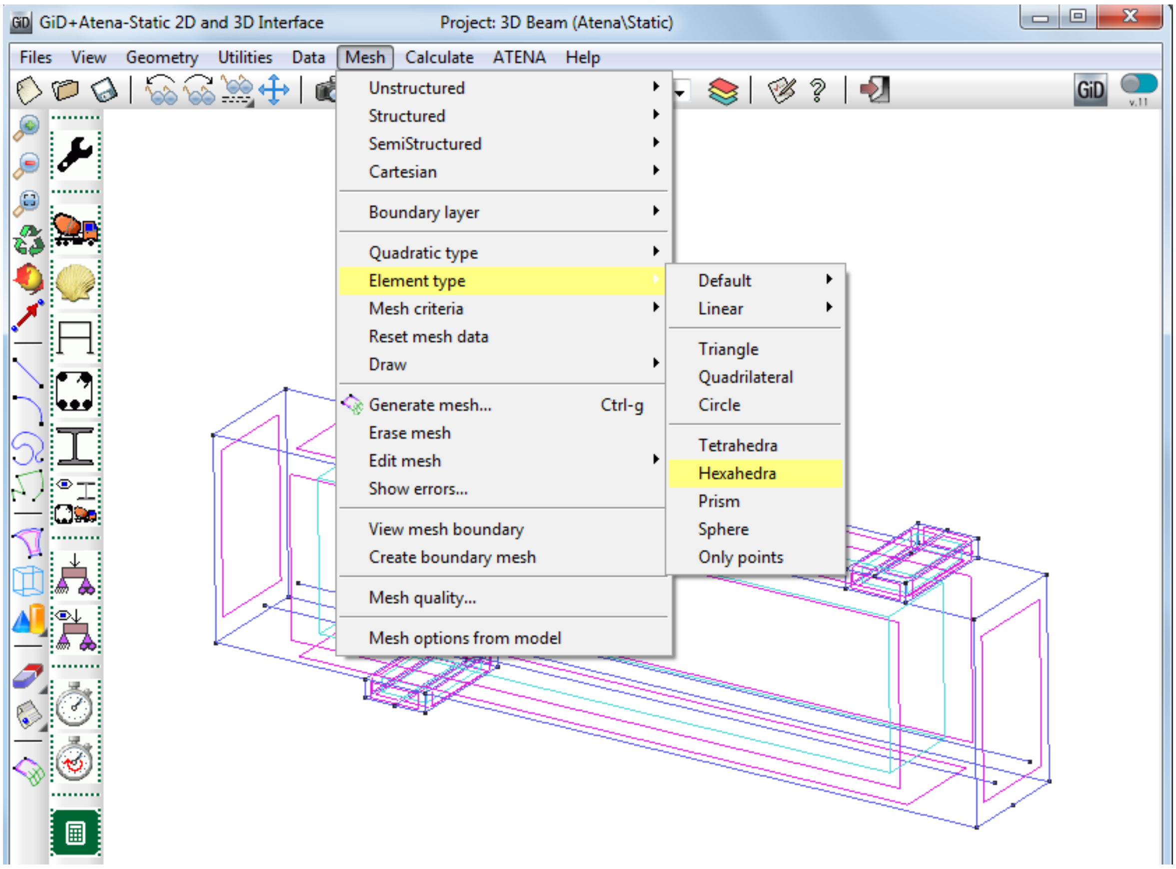


Figure 127: The selection of the hexahedra type

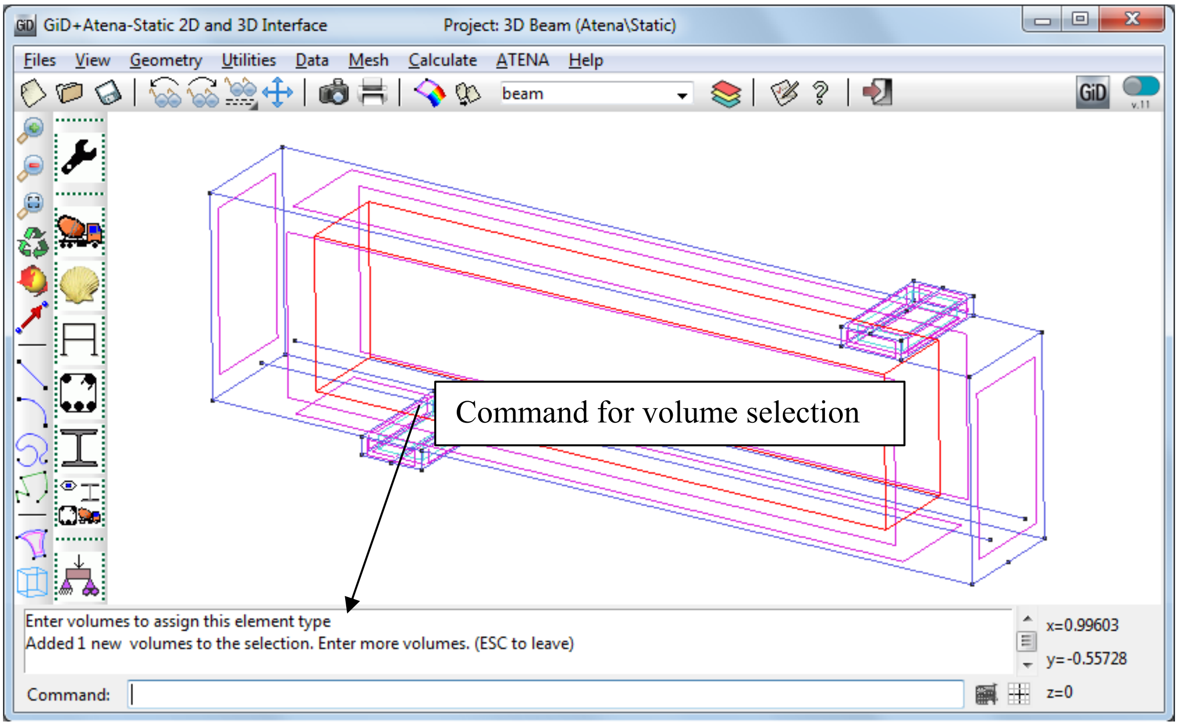


Figure 128: The selection of the beam volume

Now the mesh can be generated. It is done by command **Mesh | Generate mesh** or it can be activated directly by pressing the key Ctrl and 'g' at the same time. After that the enter value window will appear (see Figure 129). Here it is necessary to define the default element size for the volumes that are not mesh using the structured option. There the value 0.065 can be left and the button **Ok** can be pressed. The generation of the mesh will start and then the list of elements will appear. The number of elements can be checked in that list (see Figure 130). It is necessary that the total number of elements is below 300. This limit is necessary only for the demo version of **ATENA**. If a full version of ATENA is available it is recommended to use more elements. The generation of the mesh is finished by selecting button **Ok** (see Figure 131).

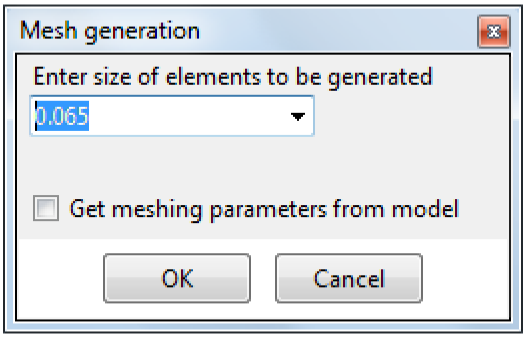


Figure 129: The enter value window

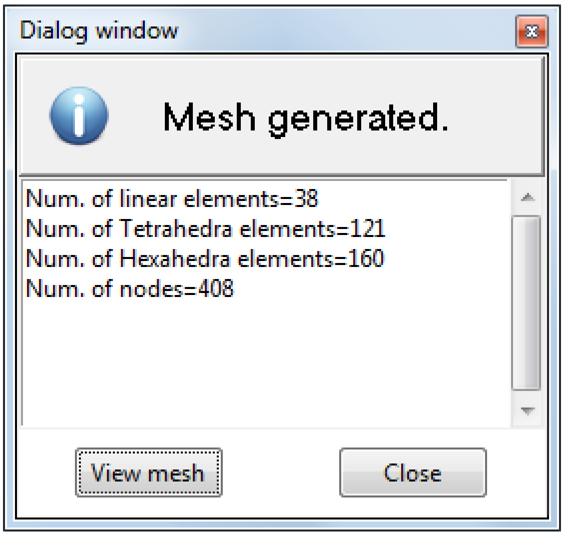


Figure 130: The list of the elements of the mesh

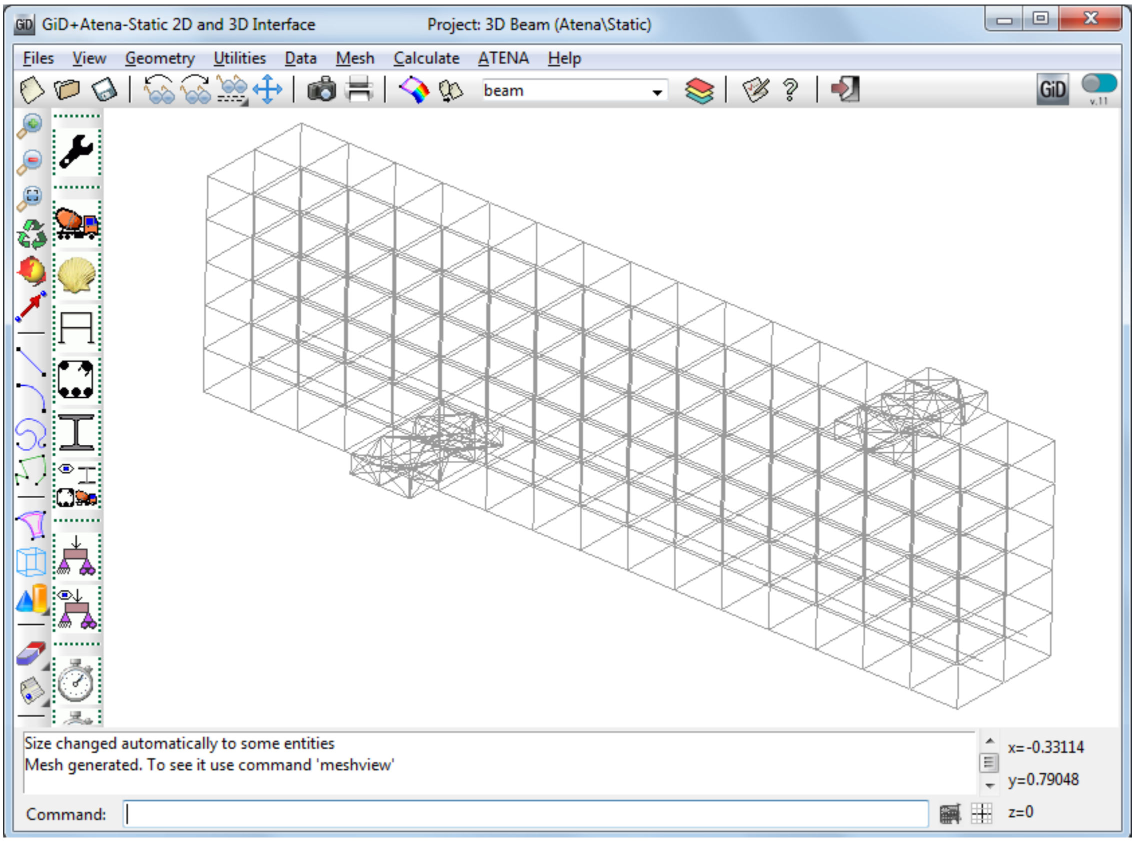


Figure 131: The generated structured mesh

Number of linear elements is 38. It means that the reinforcement bars were also divided. This is not necessary and it is better to reduce number of elements. The procedure is same as for dividing the beam. Use the command **Mesh | Structured | Lines | Assign number of cells** in the main menu.

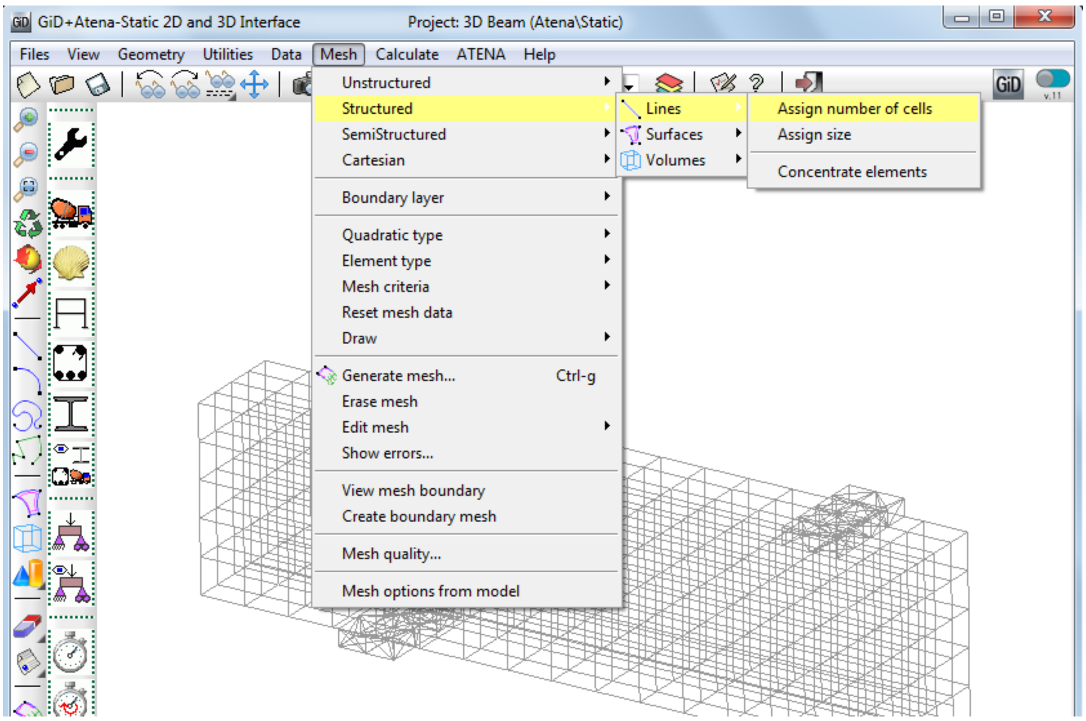


Figure 132: The structured mesh command

Once this command is executed the program asks for the number of cells which should be assigned to the lines. Write number one. The reinforcement bars have to be selected (see Figure 133Figure 120).

Parameter input:

Enter number of cells to assign to lines: 1

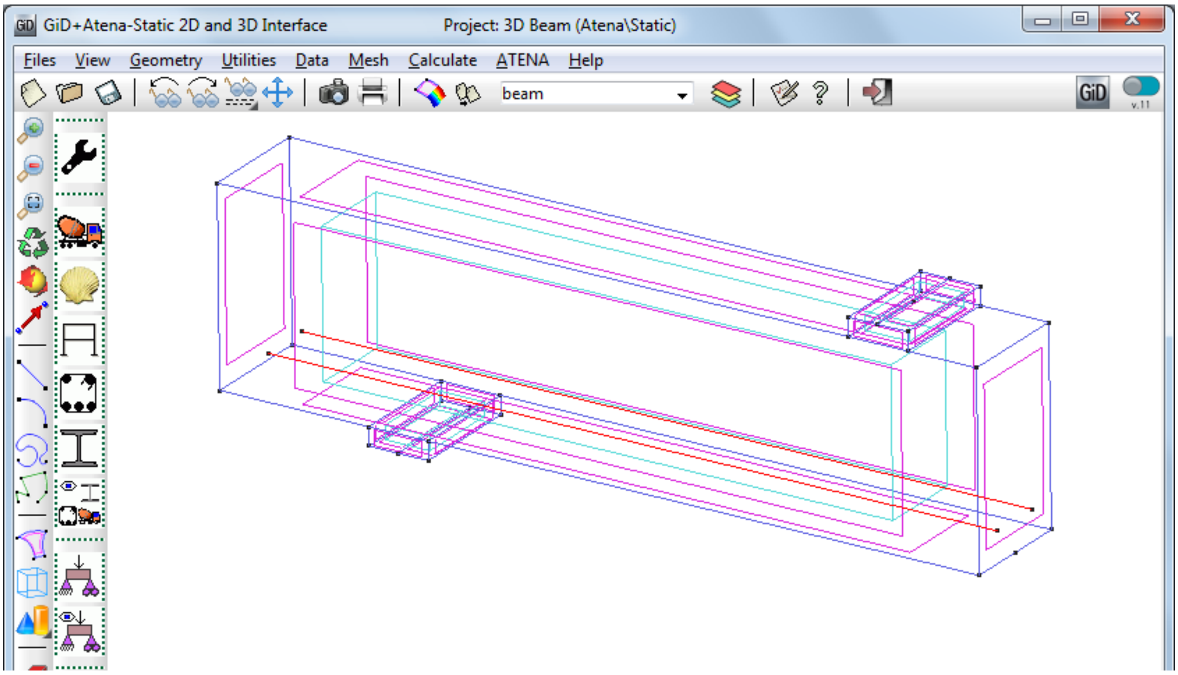


Figure 133: The lines selection

After that a new mesh can be generated. It is done by command **Mesh | Generate mesh** or it can be activated directly by pressing the key Ctrl and 'g' at the same time. Number of elements is default. As can be seen on Figure 134 the number of linear elements is only 2.

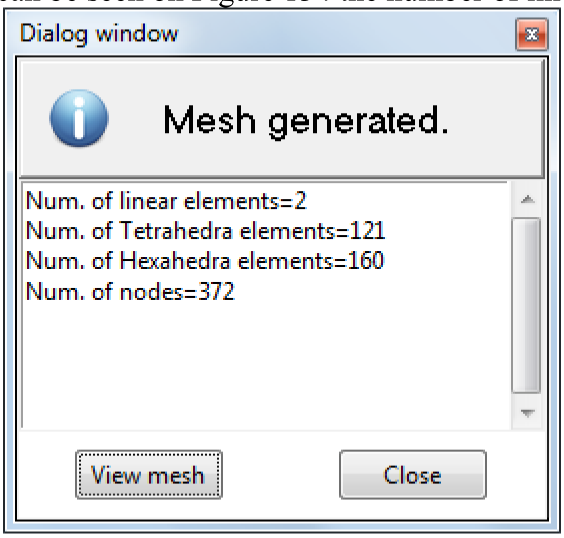


Figure 134: The dialog window

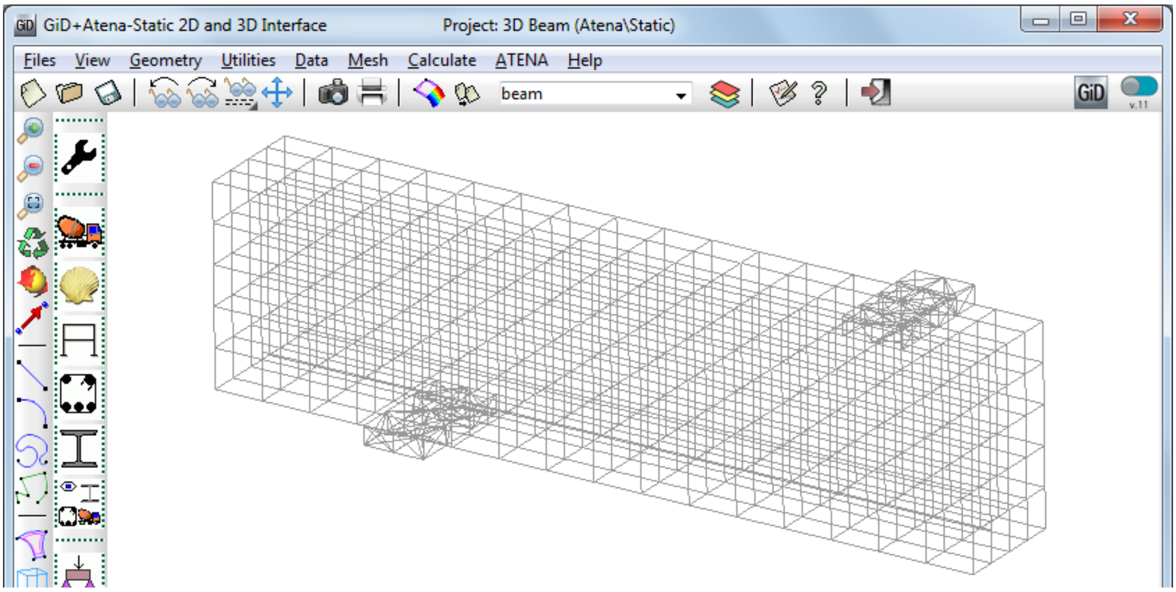


Figure 135: The new mesh

For better view of the structured mesh the created model can be rendered. It is done by selecting Render in the Mouse menu which appears after clicking on the right-mouse button (see Figure 136).

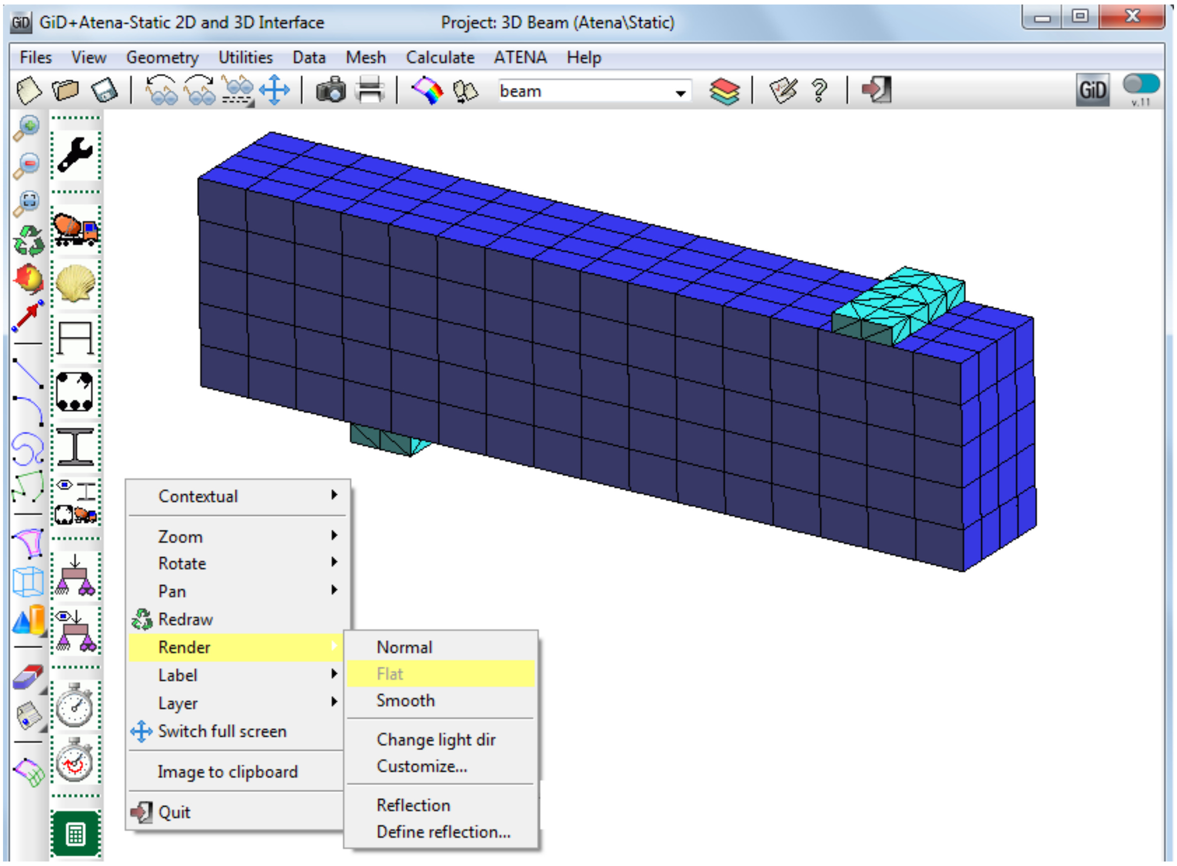


Figure 136: The flat rendered geometrical model

When the mesh is correctly generated the geometrical model definition is finished and calculation can be started. See following chapter 4.

4. FE NON-LINEAR ANALYSIS

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

The finite element analysis is started by the clicking on the icon  or by the using of command **Calculate | Calculate**. After selecting this command, the program will start to generate the input files for each step of the non-linear analysis. This process is indicated by the dialog box (see Figure 137). And then the **ATENA Studio** window will appear and analysis will be in progress (see Figure 138).

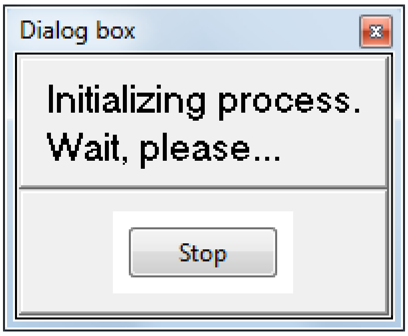


Figure 137: The initializing of the analysis

If the creation of the geometrical model and definition of the boundary conditions were done right, the static analysis should be finished in one minute. Then in the Geometry window it is possible to see that the loading steel plate strangely distorted and shifted (see Figure 139) by the applied loads. Due to that the structure could not be calculated correctly. It is because there is no connection defined between the concrete beam and the steel plates. Program does not automatically detect possible contact between volumes. Contacts have to be added manually by boundary special conditions.

Therefore, it is necessary to return back to **GiD** graphical interface and defined fixed contacts. **ATENA Studio** can be simply closed without any savings of data and then it is necessary to return back to the **GiD** graphical interface and define the missing contacts.

It should be noted that this problem is a direct consequence of the modelling approach that was chosen in the previous section.

In this tutorial, the geometry is created by three individual and separated volumes. In such a case contacts have to be added manually. If the corresponding surfaces of the steel plates would be also parts of the geometry of the beam, all parts of the structure would be connected and no special condition would have to be defined.

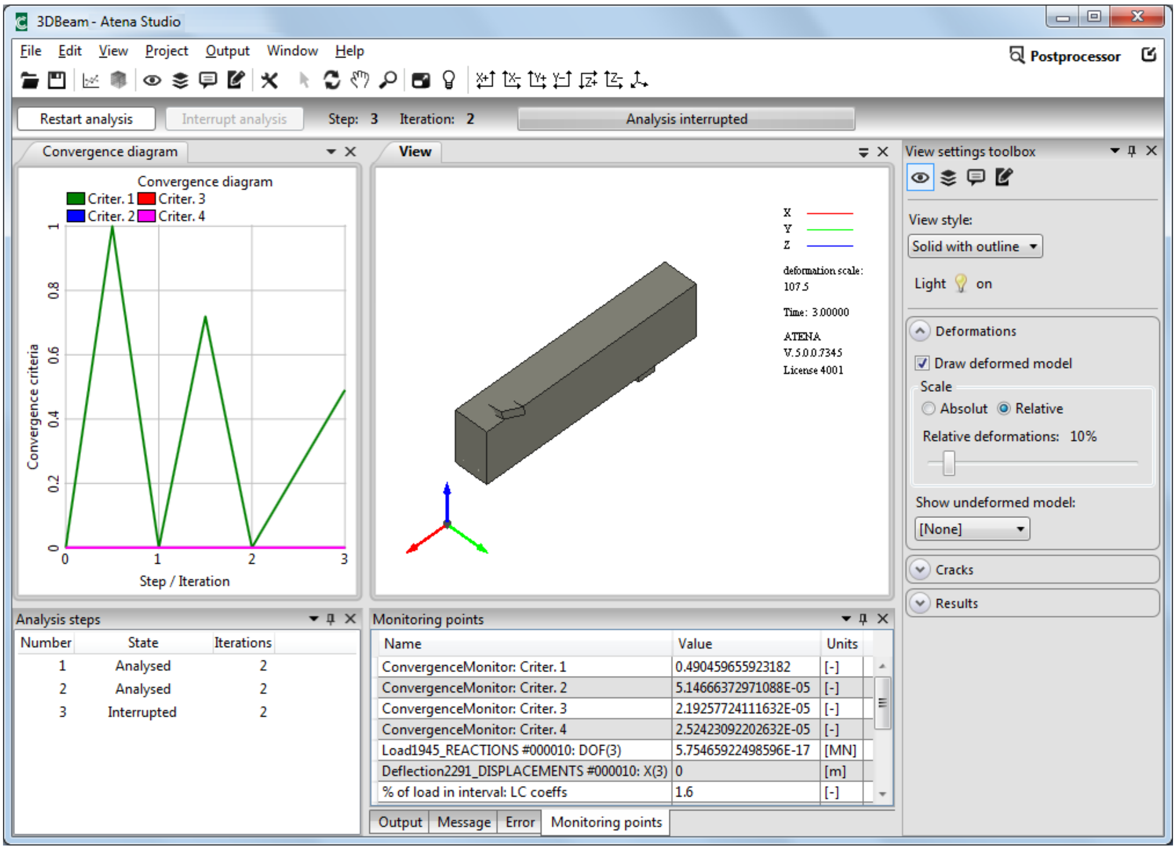


Figure 138: The ATENA Studio interface window

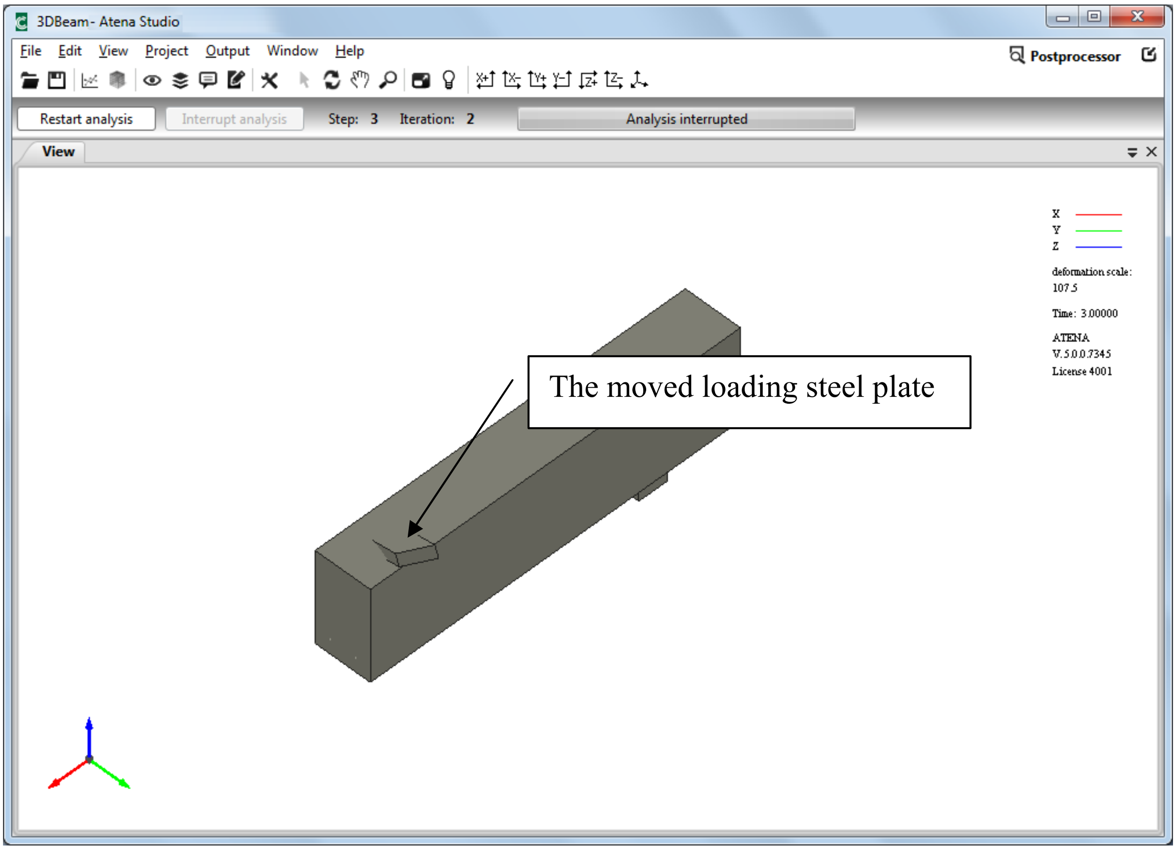


Figure 139: The moved loading steel plate.

There should be Info window in the GiD (see Figure 140). This informative window can be closed and the definition of the missing contacts can be started (see 4.1).

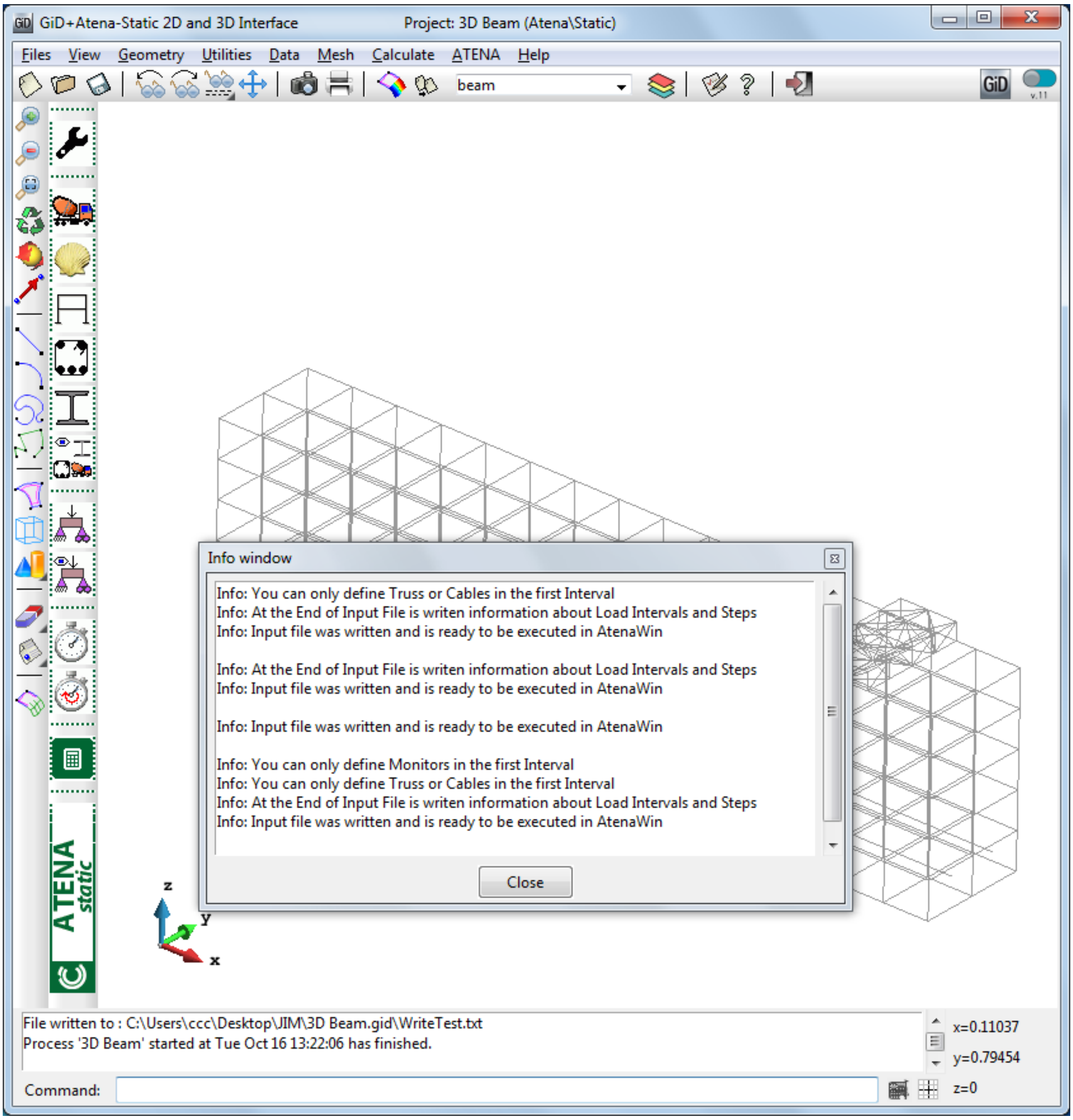


Figure 140: The GiD interface after analysis

4.1 Missing Contacts

The geometry is composed from three 3D regions – concrete beam and two steel plates. These regions should be connected together. However in this example there is no connection yet. Therefore, suitable contact conditions have to be added. In **ATENA** a suitable condition for connecting independent surface together is called Fixed Contact.

Fixed contact condition distinguishes Master and Slave conditions. In this case, the beam surfaces will be masters and plates will be slaves. Therefore, four contact conditions have to be added – two master conditions on beam (top and bottom) and two slave conditions on plates (top and bottom).

The conditions should be applied on the geometrical model and not on the mesh itself, otherwise it would be lost during next mesh generation. Therefore, if the mesh is displayed in the graphical area of the program the icon should be selected to switch between the mesh view and geometry view. This can be alternatively also accomplished by selecting the command **Geometry | View geometry** in the main menu.

4.1.1 Master Top Beam Condition

Conditions command can be executed by the selection of the icon or by the selecting the command **Data | Conditions** in the main menu. The contact condition definition for master top beam is depicted in the Figure 141.

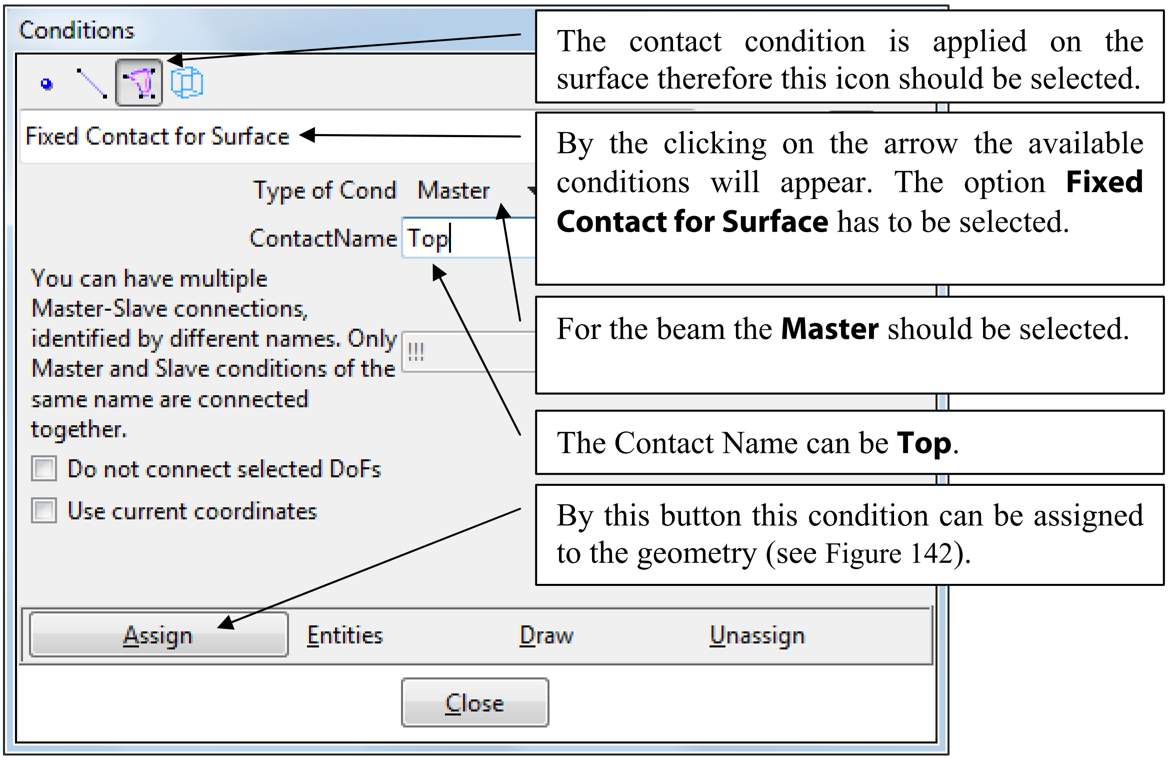


Figure 141: The master top beam contact condition

Parameter input:
Fixed Contact for Surface
Type of Cond: MASTER
Contact Name: Top

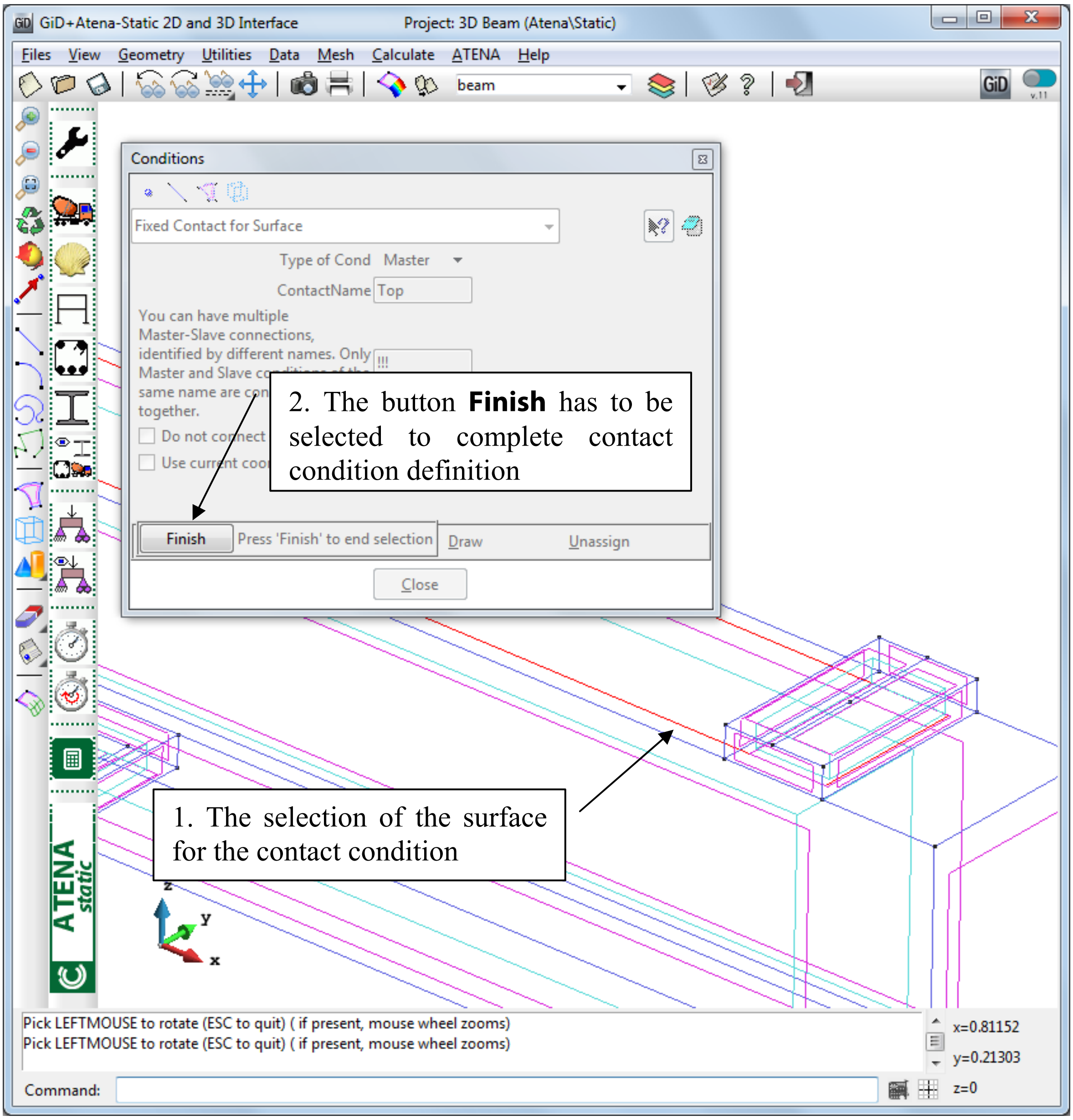


Figure 142: The selection of the surface for the master top beam contact condition

Next the command draw condition can to be selected to display and verify condition definition. The button **Draw** should be selected in the bottom of the Conditions window. After clicking on that button several options will appear (see Figure 143). For example the **Colors** can be selected and the master contact condition will be drawn (see Figure 144).

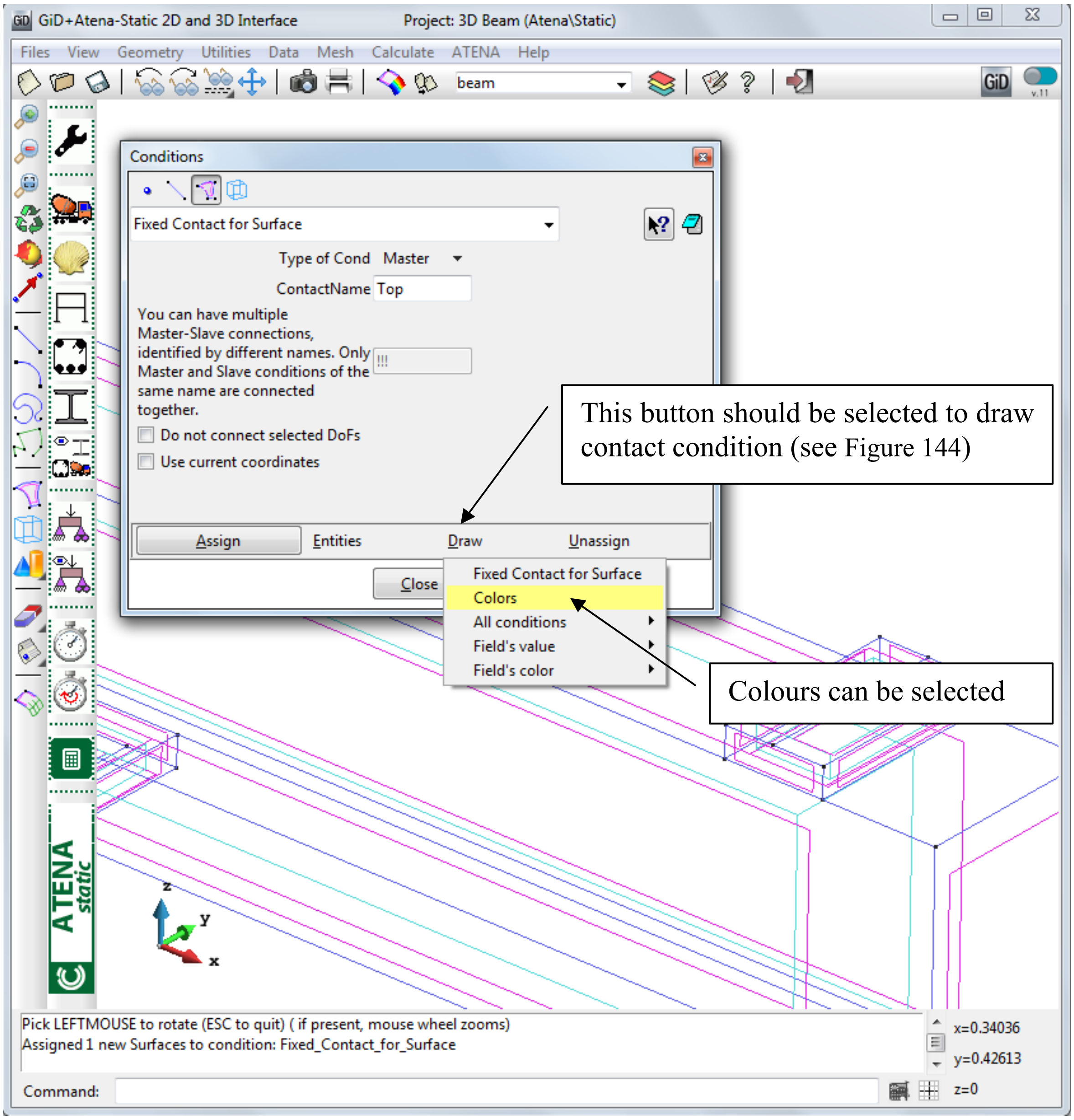


Figure 143: The draw coloured contact condition command

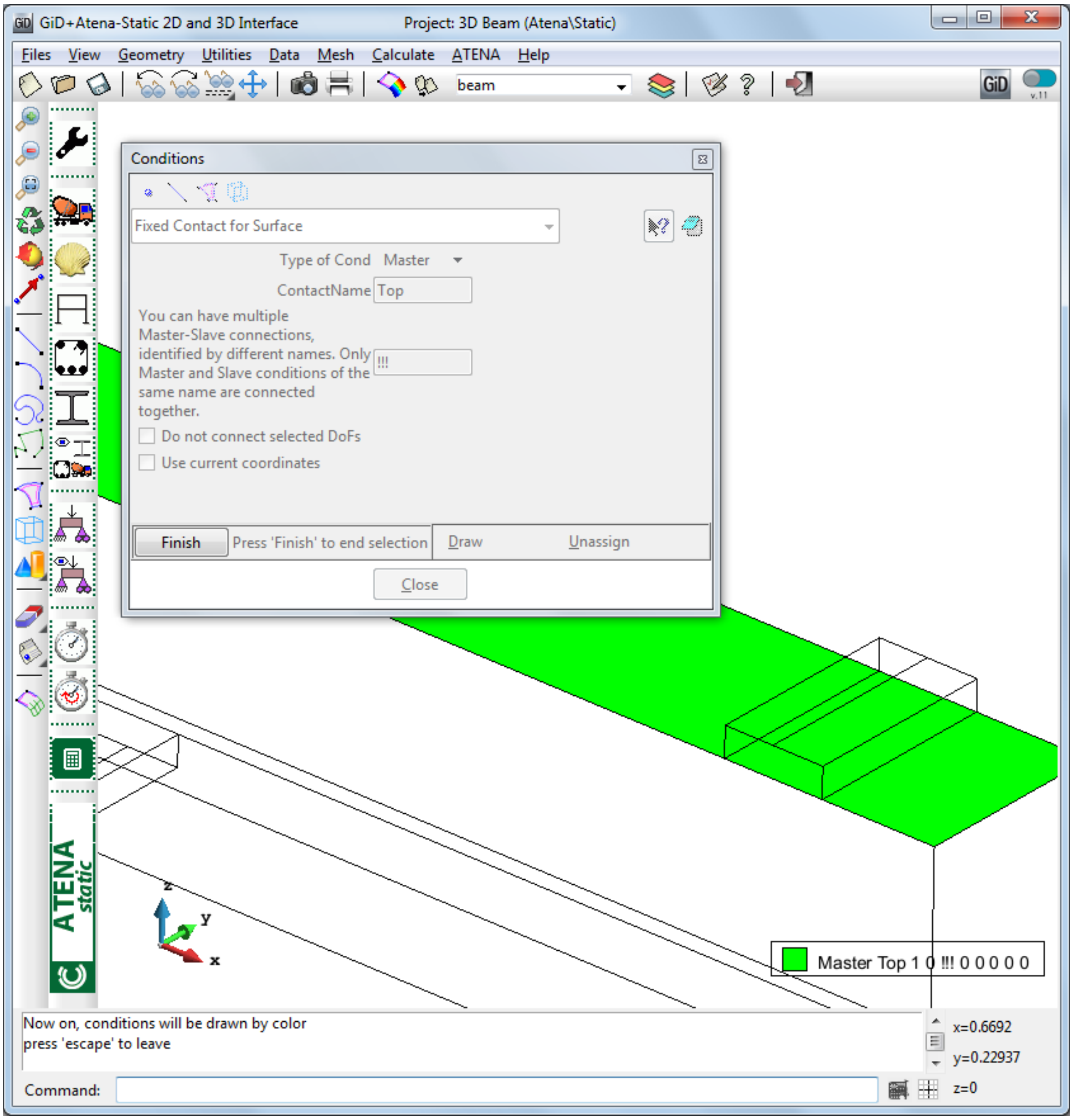
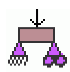


Figure 144: The Master Top beam condition

4.1.2 Slave Top Plate Condition

Conditions command can be executed by the selection of the icon  or by the selecting the command **Data | Conditions** in the main menu. The contact condition definition for master top beam is depicted in the Figure 145.

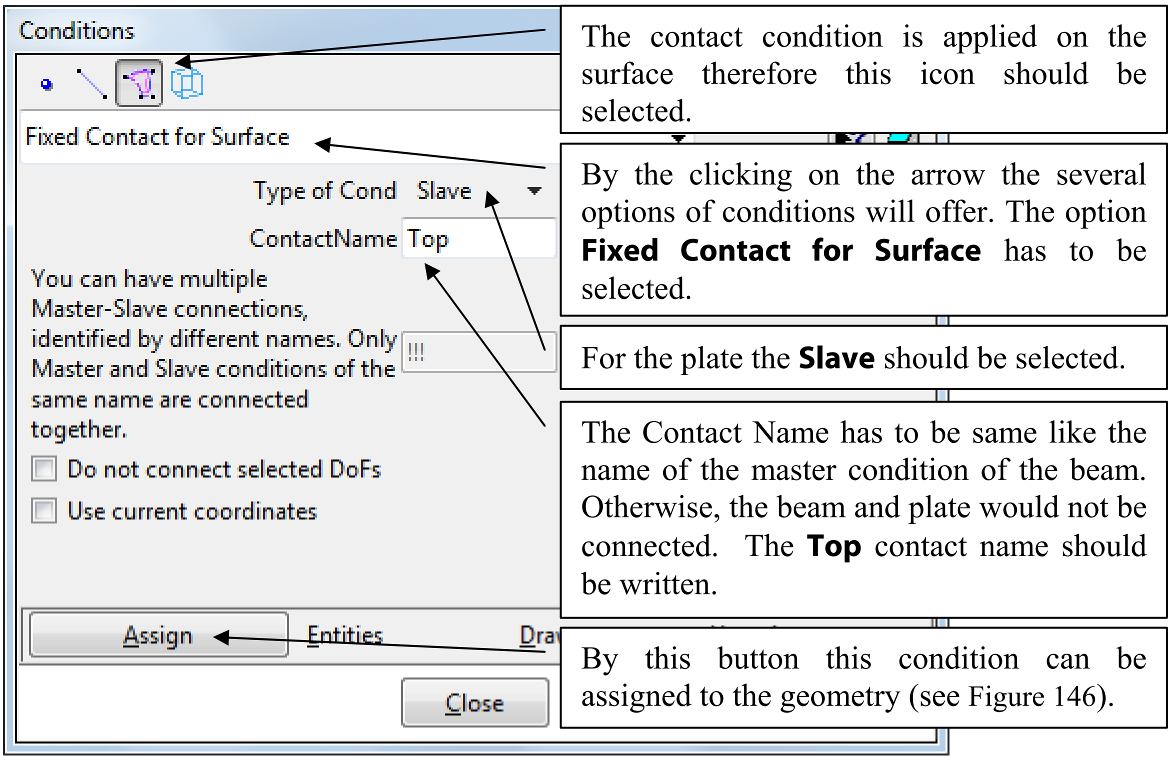


Figure 145: The slave top plate contact condition

Parameter input:
Fixed Contact for Surface
Type of Cond: SLAVE
Contact Name: Top

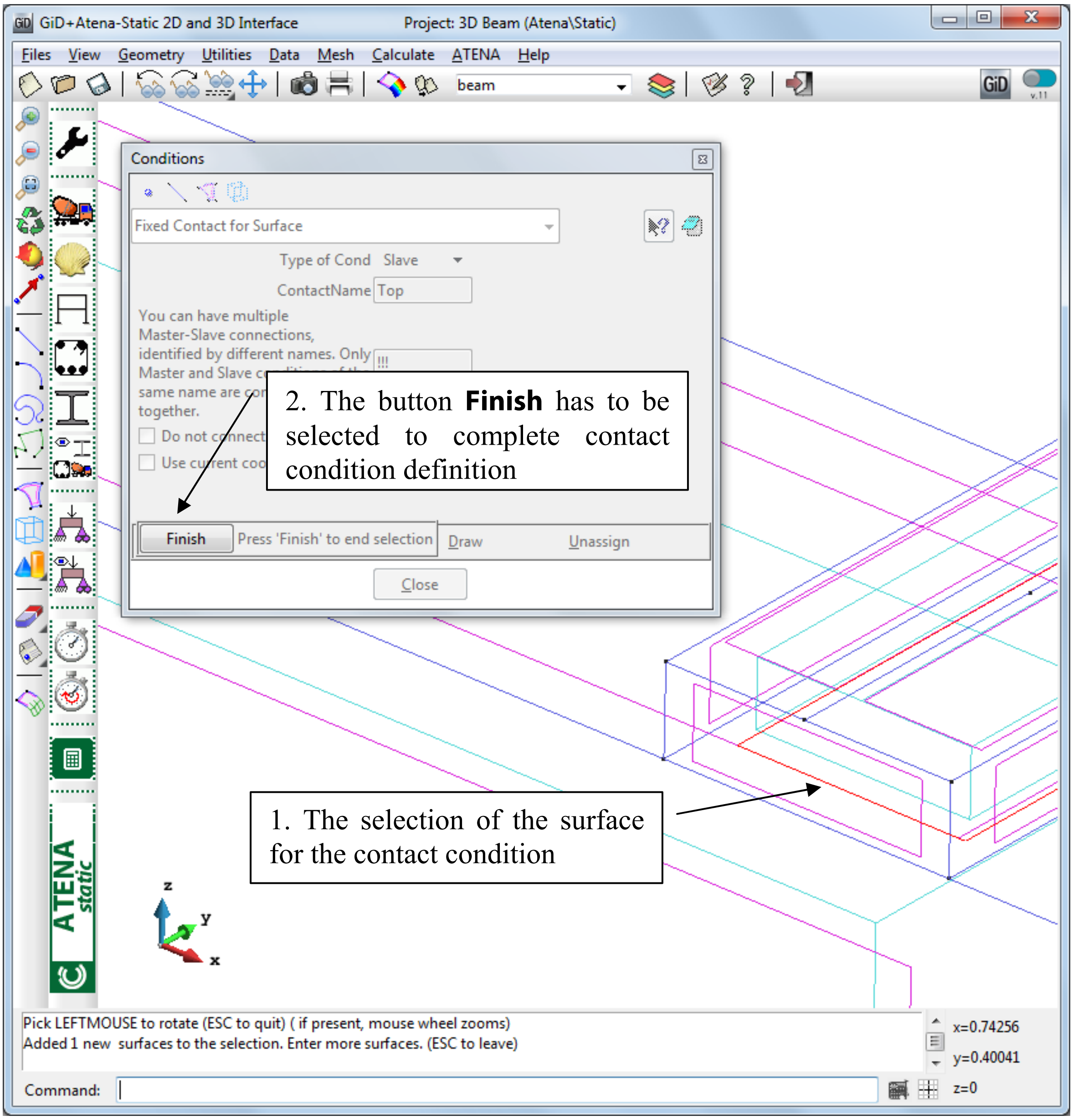


Figure 146: The selection of the surface for the slave top plate contact condition

4.1.3 Master Bottom Beam and Slave Bottom Plate Conditions

The bottom conditions will be done by the same procedure like in the case of top contact conditions. Only the name has to be different. It is recommended to use contact name Bottom. The Figure 147 shows the right definition of bottom contact conditions.

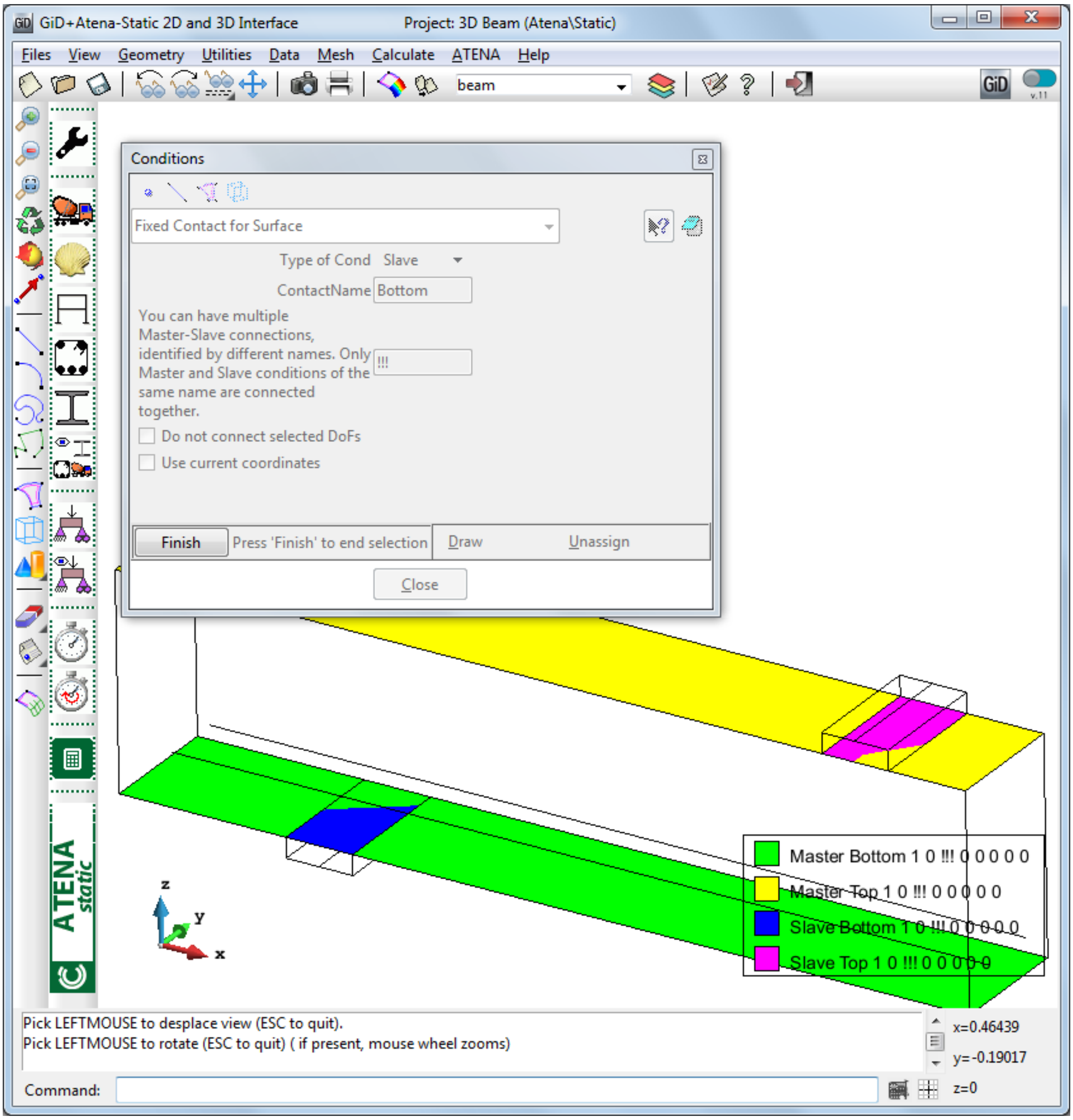
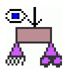


Figure 147: The contact conditions

B the clicking in the icon  all boundary condition can be displayed. It is a good method for checking if all conditions were properly defined (see Figure 148).

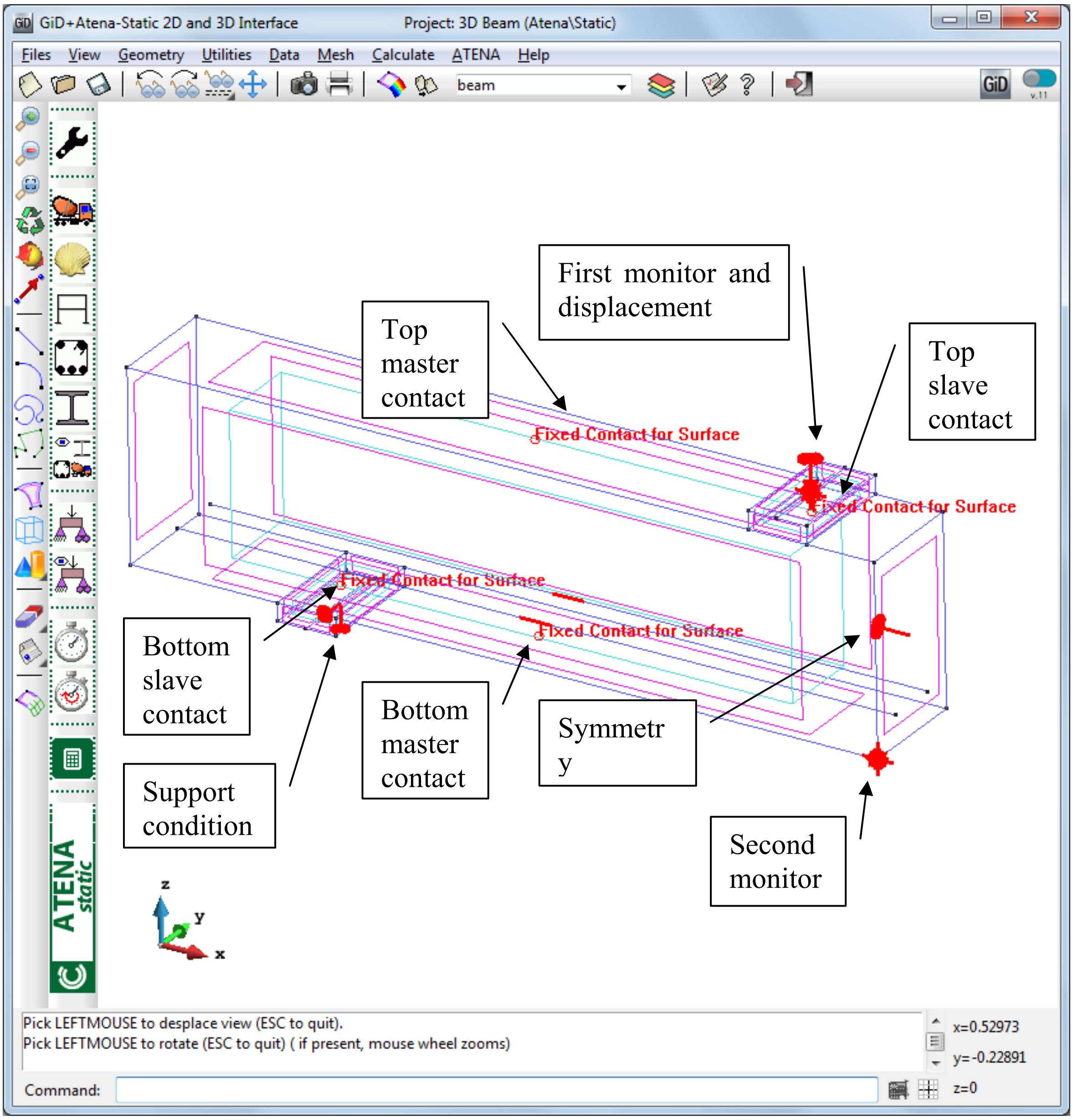


Figure 148: All boundary conditions

When the contact conditions are finished it is important to generate mesh again. After any change of boundary condition and geometry the mesh has to be generated again. It is done by the command **Mesh | Generate mesh** in the main menu or by pressing the key Ctrl and 'g' at the same time.

If the new mesh is generated, the analysis can be started again. It is done by **using the command Calculate | Calculate** or by the clicking on the icon . After selecting this command, the program will first generate **ATENA** input file for the non-linear analysis and then the **ATENA Studio** window will appear and analysis will be in progress (see Figure 149).

4.2 ATENA Studio Interface Description

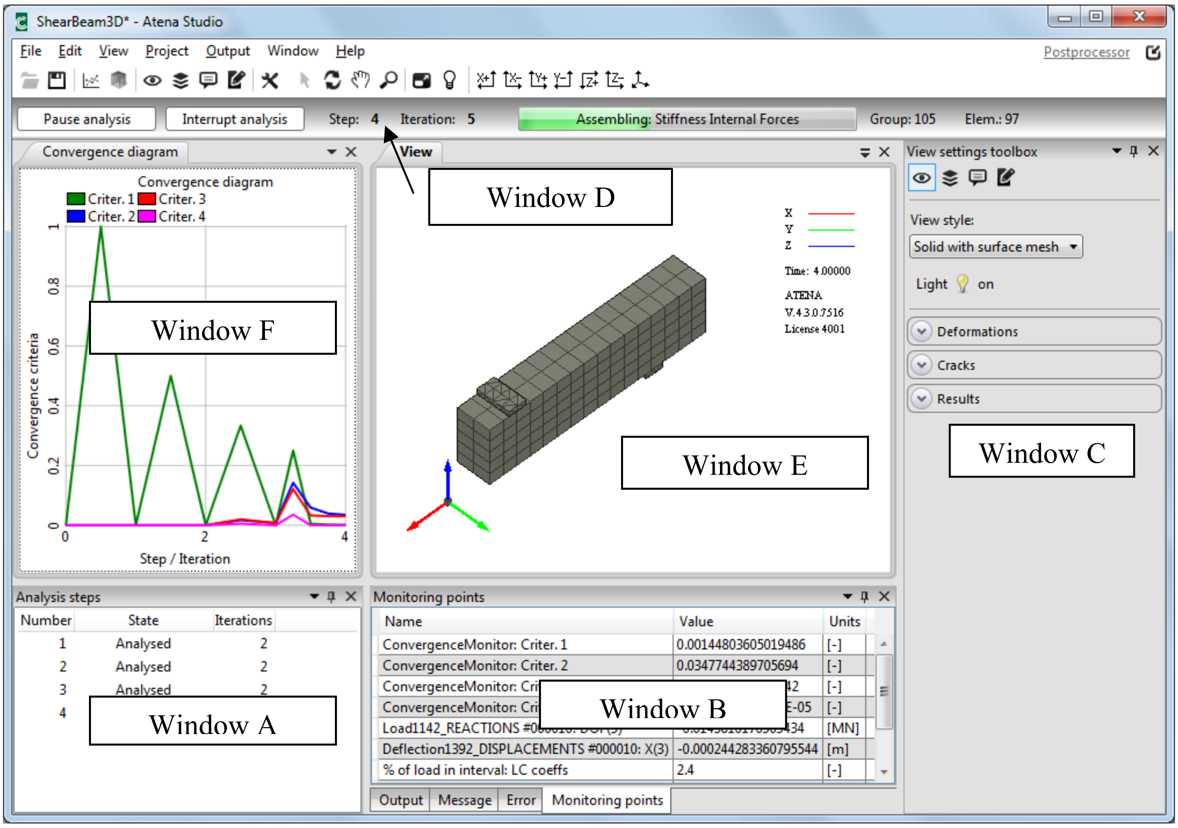


Figure 149: The analysis in progress

Basic description of the ATENA Studio interface:

Window A: contains analysed steps and iterations

Window B: contains important messages from ATENA kernel sent during analysis

Window C: contains settings for displayed results

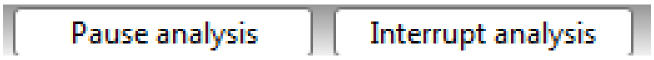
Window D: contains progress of analysis, number of steps and iterations

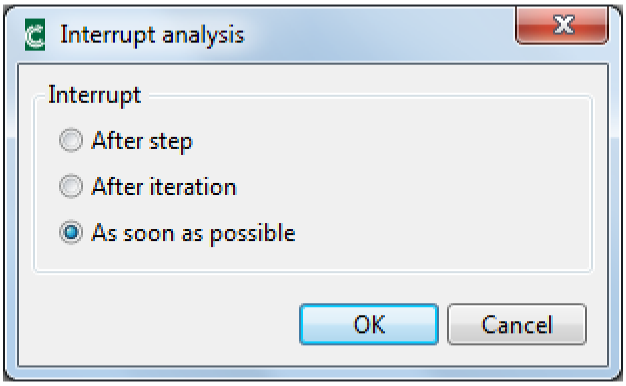
Window E: contains graphical representation of the analysed structure

Window F: contains default convergence diagram or other user-defined diagram

When the analysis is running it is possible to stop or suspend the calculation. However, it is not recommended to do it in this first tutorial example.

For that it is possible to use **Project | Pause analysis/Interrupt analysis | After step/After iteration/As soon as possible** command in the main menu or buttons of the Analysis control toolbar:





For detailed description of the ATENA user interface it is recommended to read ATENA Manual [7].

4.3 *Load-Displacement Diagram*

During the running analysis it is very useful to see the evolution of the applied load and beam deflections. The progress of the load and deflection is available in the monitors that were defined in the previous Section 3.4.4. Now, it will be described how to visualize these monitors during the nonlinear analysis.

The first step in the visualization of monitors is to open a new diagram window by the clicking on the icon . The empty window for the diagram and the diagram settings will appear (see Figure 150). The new diagram is defined by diagram settings dialog (see Figure 151).

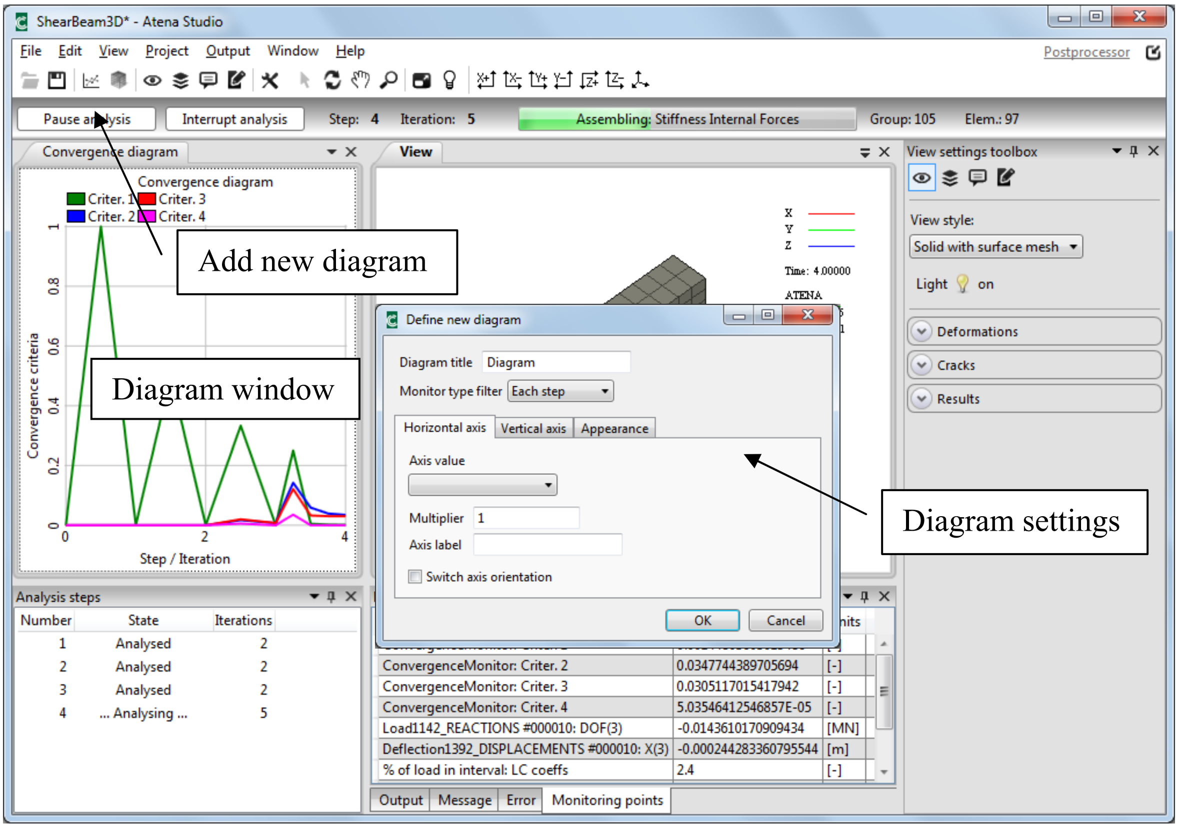
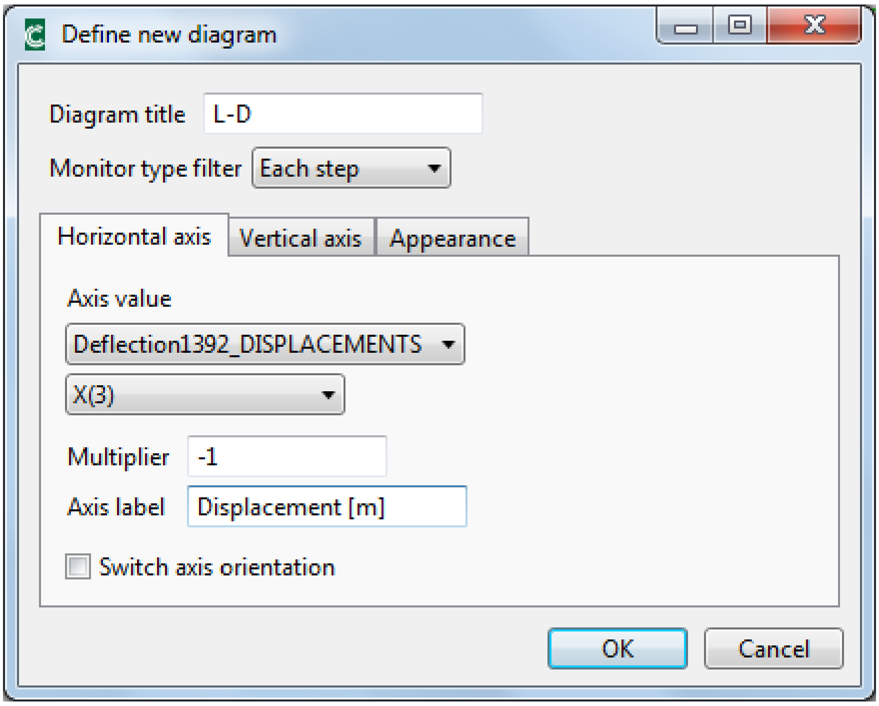
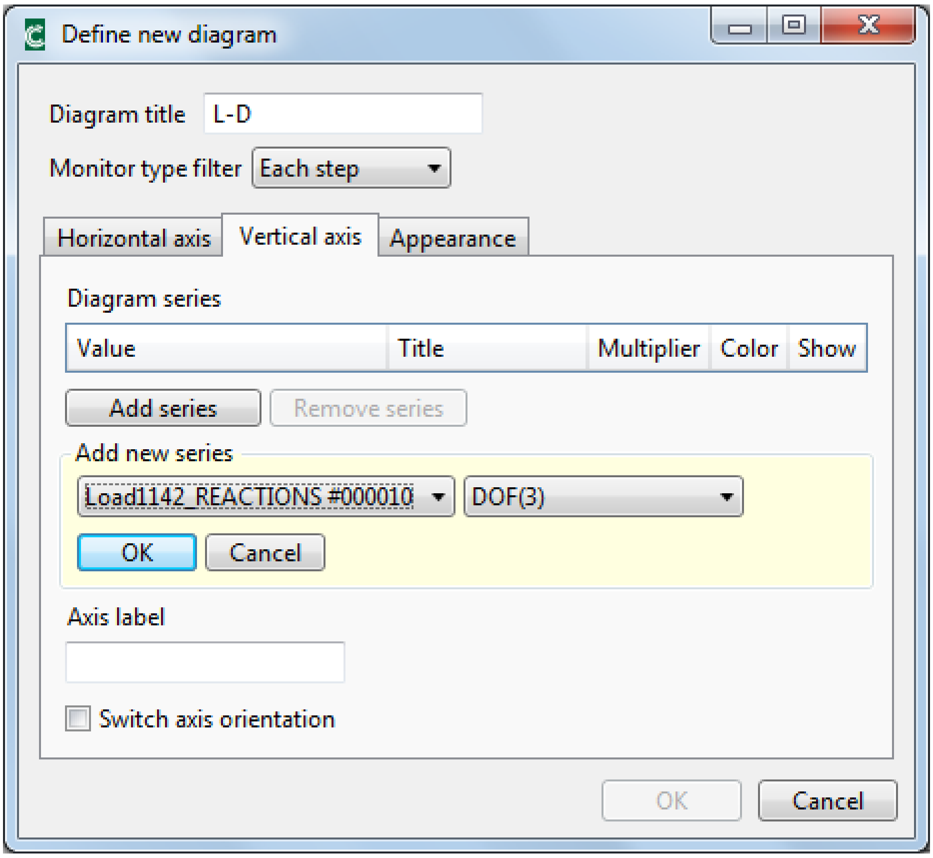


Figure 150: The execution of the graph

The diagram title can be **L-D** and the monitor type filter should be **Each step**. For the horizontal value the name **Deflection_DISPLACEMENT** should be selected. The name of axis should be **Displacement [m]** and orientation should be switched. The vertical axis can display more series. Add new series, the name for the vertical value should be **Load_REACTIONS** and axis label can be **Load [MN]**. The series definition must be applied by the **OK** button above Axis label. The definition of the diagram parameters is finished by clicking on the **OK** button. After this, the L-D diagram is shown on the left side of the **ATENA Studio** interface. This diagram is showing actual stage of the running analysis and it changes as the analysis progresses based on the current loads and deflections.


Define new diagram
Diagram title
L-D
Monitor type filter
Each step
Horizontal axis
Vertical axis
Appearance
Axis value
Deflection1392_DISPLACEMENTS
X(3)
Multiplier
-1
Axis label
Displacement [m]
Switch axis orientation
OK
Cancel


Define new diagram
Diagram title
L-D
Monitor type filter
Each step
Horizontal axis
Vertical axis
Appearance
Diagram series
Value
Title
Multiplier
Color
Show
Add series
Remove series
Add new series
Load1142_REACTIONS #000010
DOF(3)
OK
Cancel
Axis label
Switch axis orientation
OK
Cancel

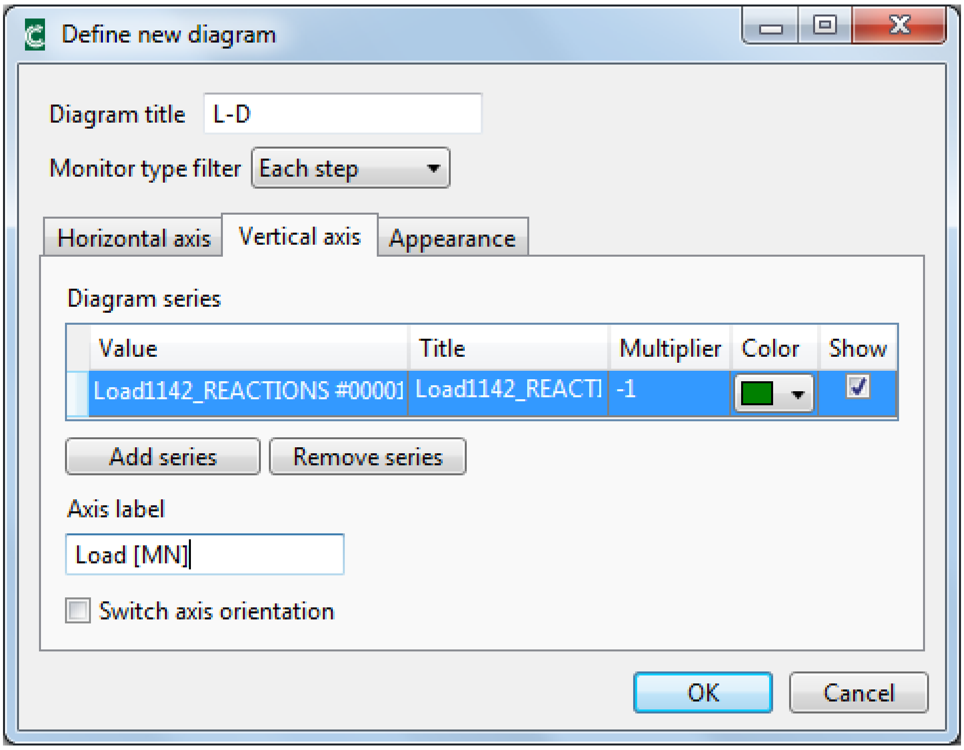


Figure 151: The diagram definition

Parameter input:

Diagram title: L-D

Horizontal axis

Axis value: Deflection_DISPLACEMENT

Multiplier: -1

Axis label: Displacement [m]

Vertical axis

Add new series: Load_REACTIONS

Multiplier: -1

Axis label: Load [MN]

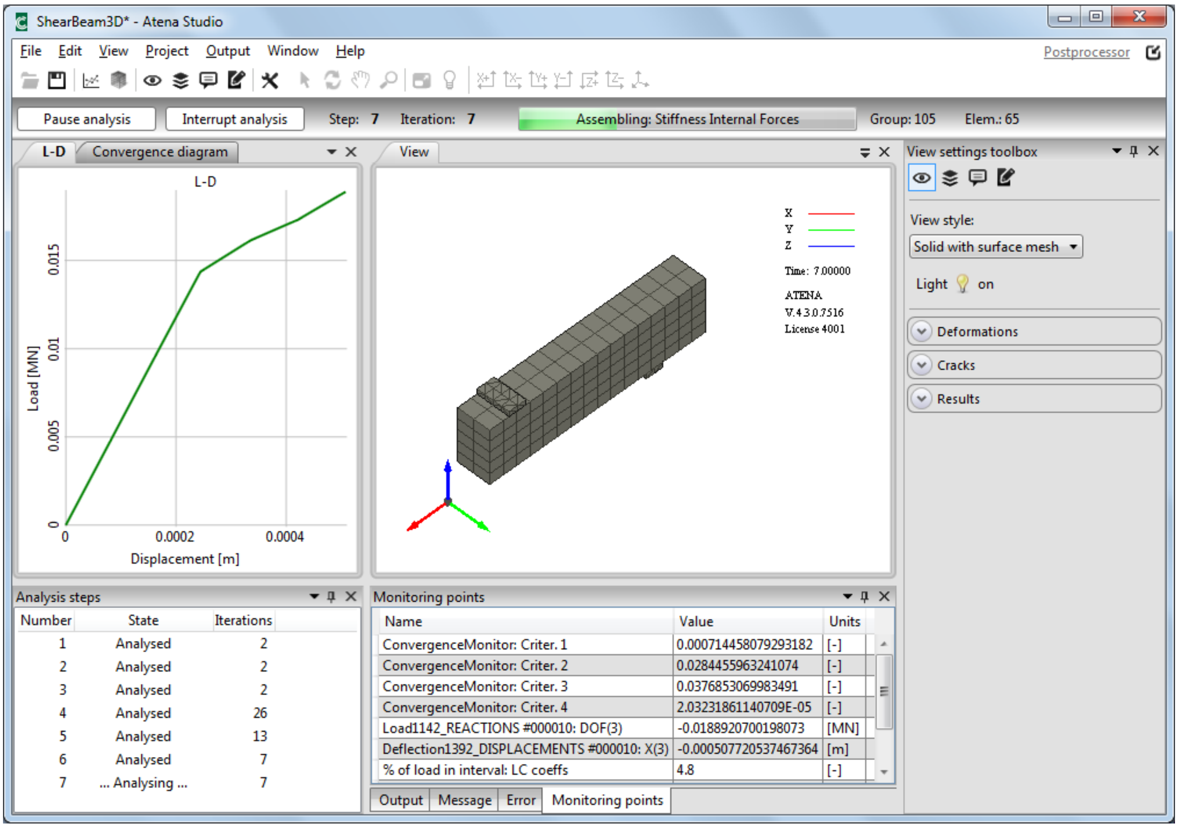


Figure 152: The L-D diagram showing stage of the running analysis

The diagram parameters were defined. Now, the diagram properties should be set. It is done by the selecting of the Properties icon . After that the graph property window will appear and properties can be described. The window is the same as for adding new diagram. There can be changed names of both axes and can be added new series. It can be useful to show a legend for series. This option is can be found in Appearance tab. The diagram properties dialog is depicted in the Figure 153 and the definition of new diagram is finished by pressing the **OK** button (see Figure 154).

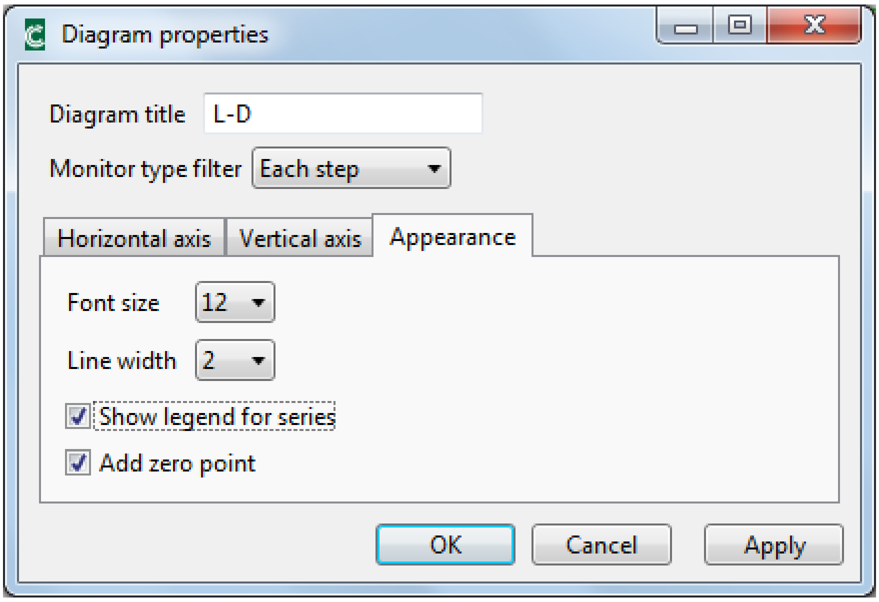


Figure 153: The diagram properties definition

Parameter input:
Tick: Show legend for series

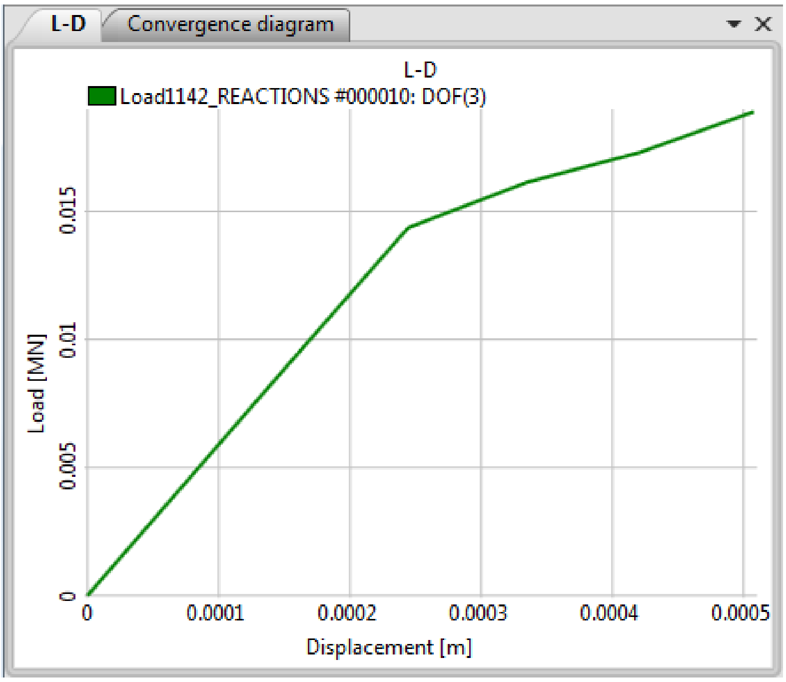


Figure 154: The defined L-D diagram

Detailed description of the L-D diagram creation can be found in the ATENA Studio Manual [7] chapter 3.5.

When the new diagram is created the tab-window is added to **ATENA Studio** layout. Default layout command in the main menu can be used to organize all windows and restore original window appearance. After selecting the option Default layout (**Window | Default layout** in the main menu), all user defined widows will be closes. Only one window with structure and convergence criteria diagram stay open (see Figure 155). But user-defined diagrams are not lost. It is possible to open previously defined L-D diagram using command **View | All diagrams | L-D.**

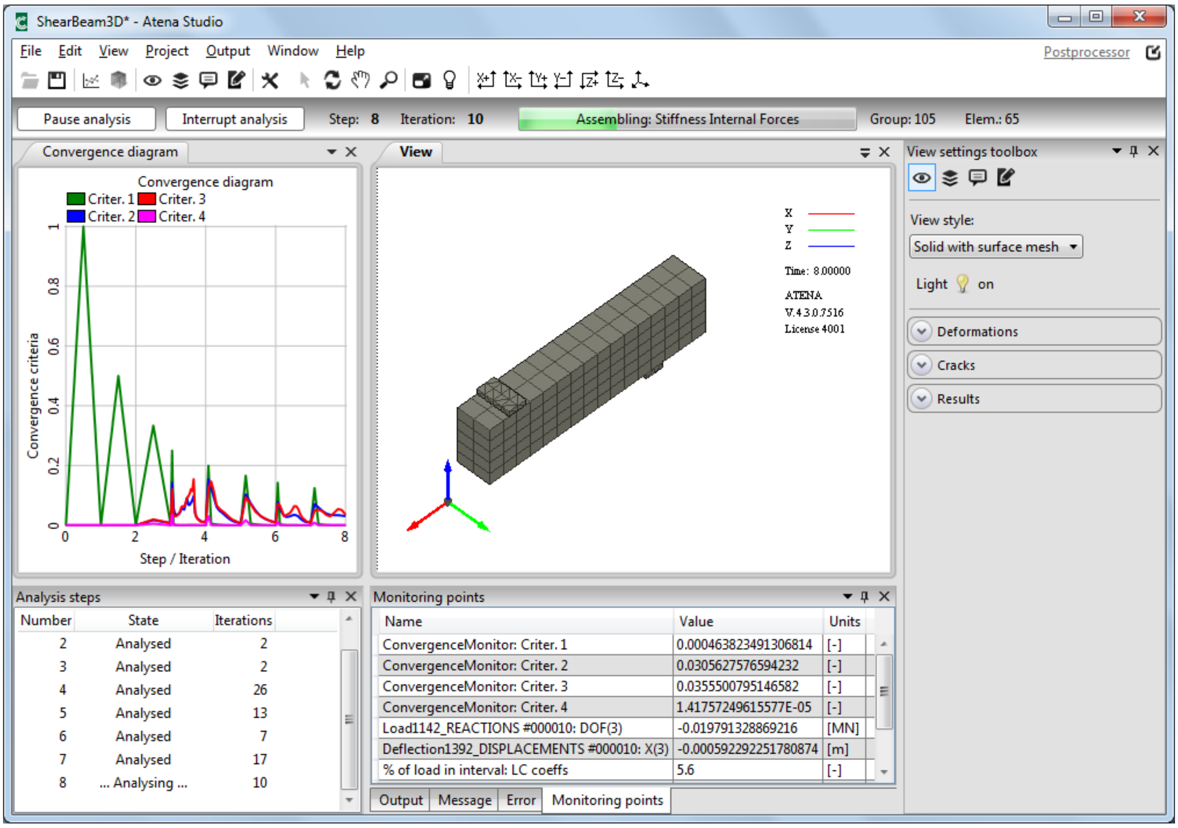


Figure 155: The Default layout

4.4 Crack Width Display

During the running analysis it can be also useful to display crack width in the Structure window. When this window is active, all icons of the Graphics Toolbar are active too. There is Structure settings toolbox on the right side of **ATENA Studio** window. This toolbox can be used to activate the display of various result quantities. Before selecting result data the displayed activity should be selected. Use the icon  (Visible domain toolbox) and select 3D activity (seeFigure 156). After that the result data can be set in settings toolbox. Click on the icon  and choose crack width from the list (see Figure 157).

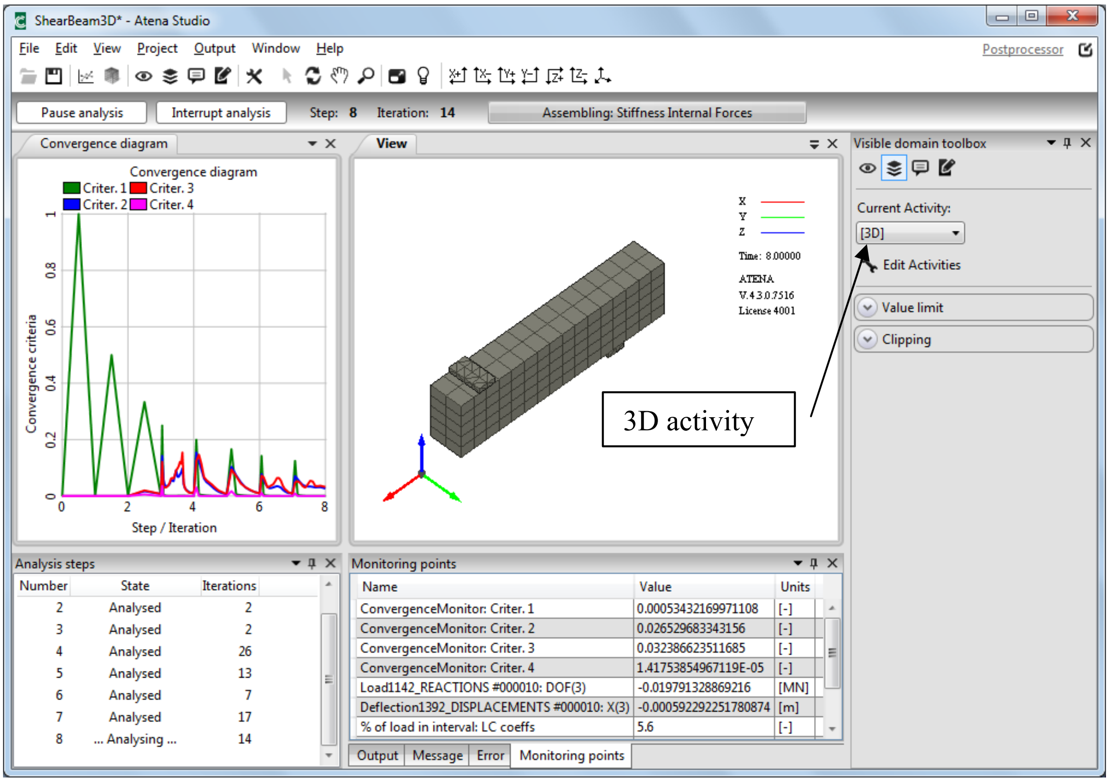


Figure 156: Visible domain toolbox with activity settings.

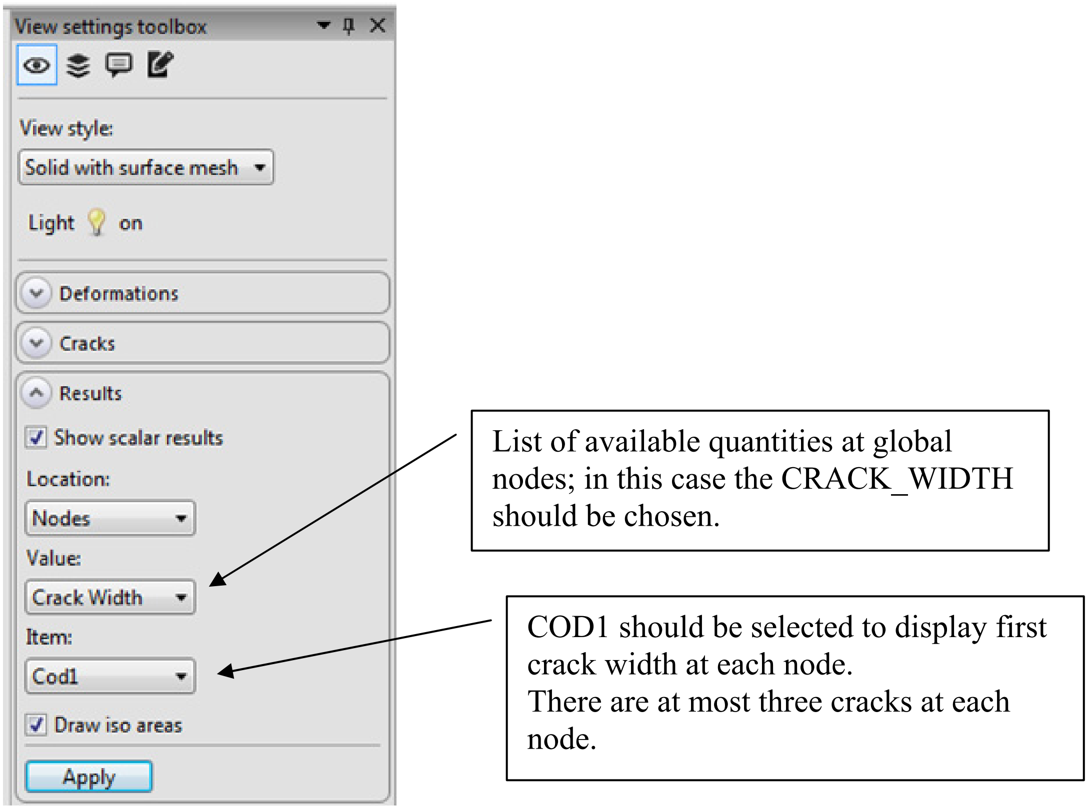


Figure 157: View settins toolbox with results panel.

To update structure according to selected result press the button **Apply** (see Figure 158).

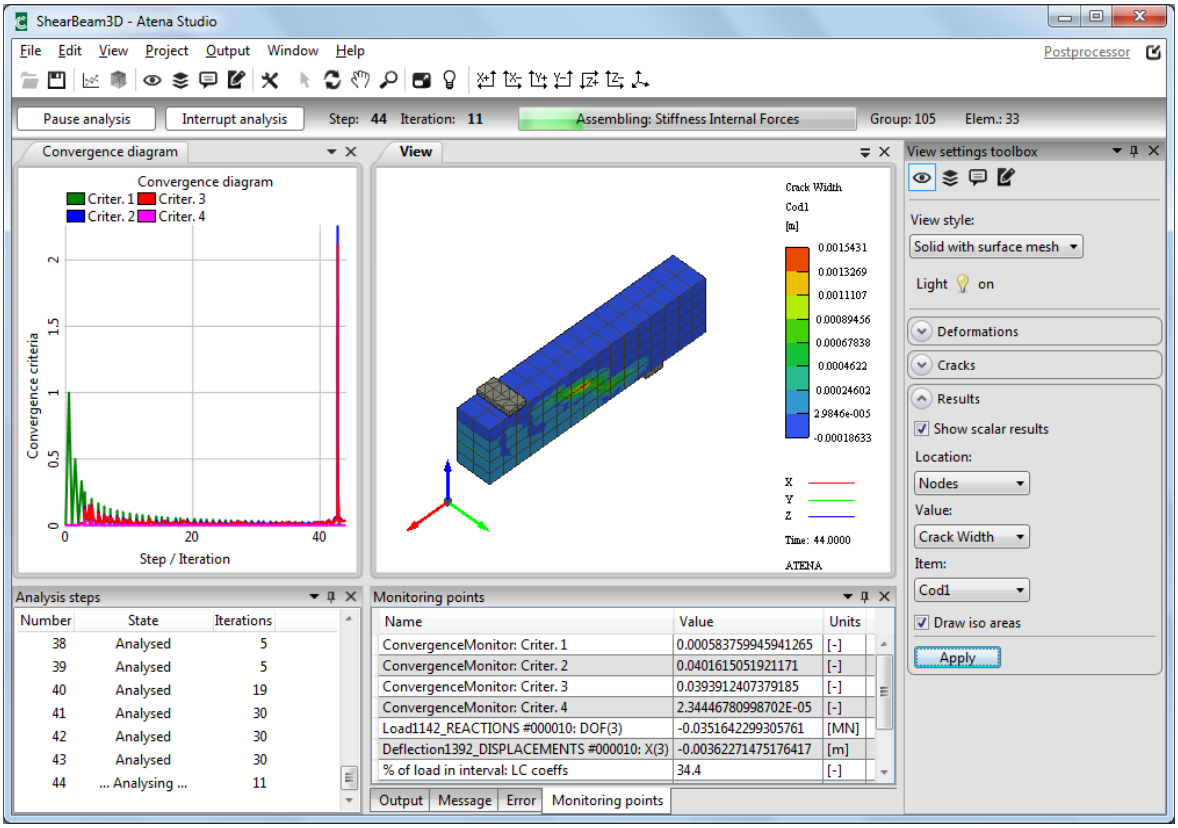


Figure 158: The crack width shown in the geometry window

For better view, the model can be rotated. To activate rotation click on icon or press and hold the Shift key and move the mouse with left button pressed (see Figure 159).

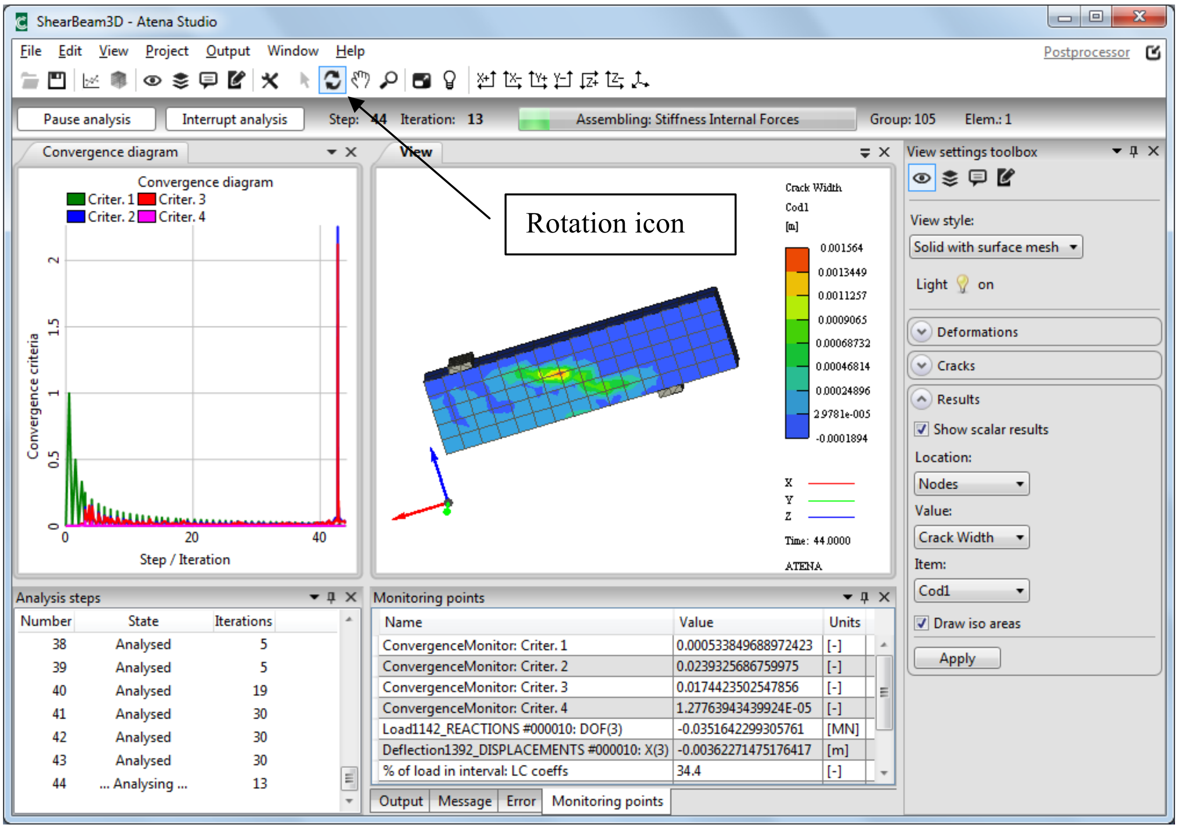


Figure 159: The crack width display and the rotation of the model

Also for better view the model can be displayed deformed. It is done by the clicking on the Deformations in setting toolbox on the right side of the window and then by the checking the option Draw deformed model (see Figure 160). There can also be set scale of deformations (relative or absolute) but it isn't necessary (see Figure 161 and Figure 162).

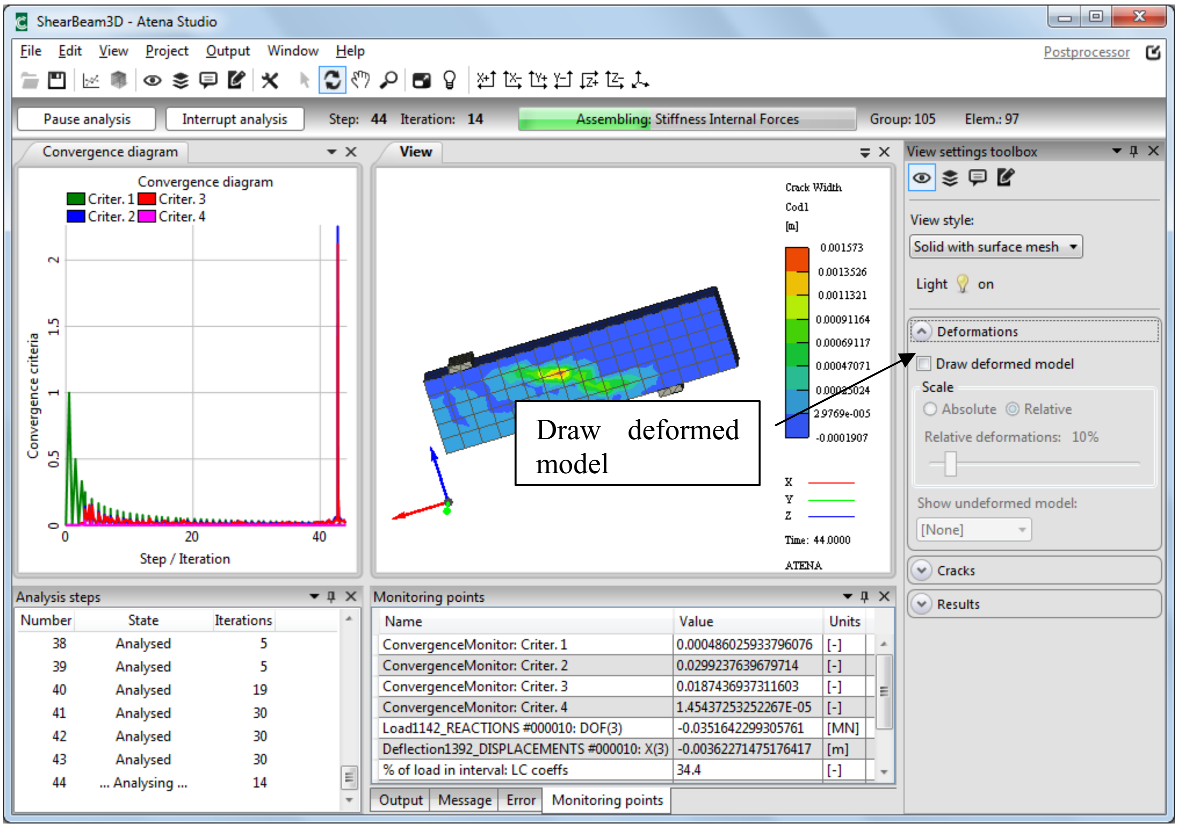


Figure 160: View settings toolbox with Deformations settings panel

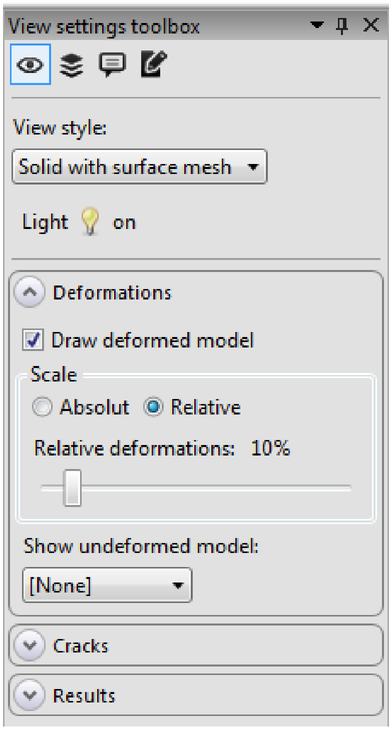


Figure 161: Deformed model settings

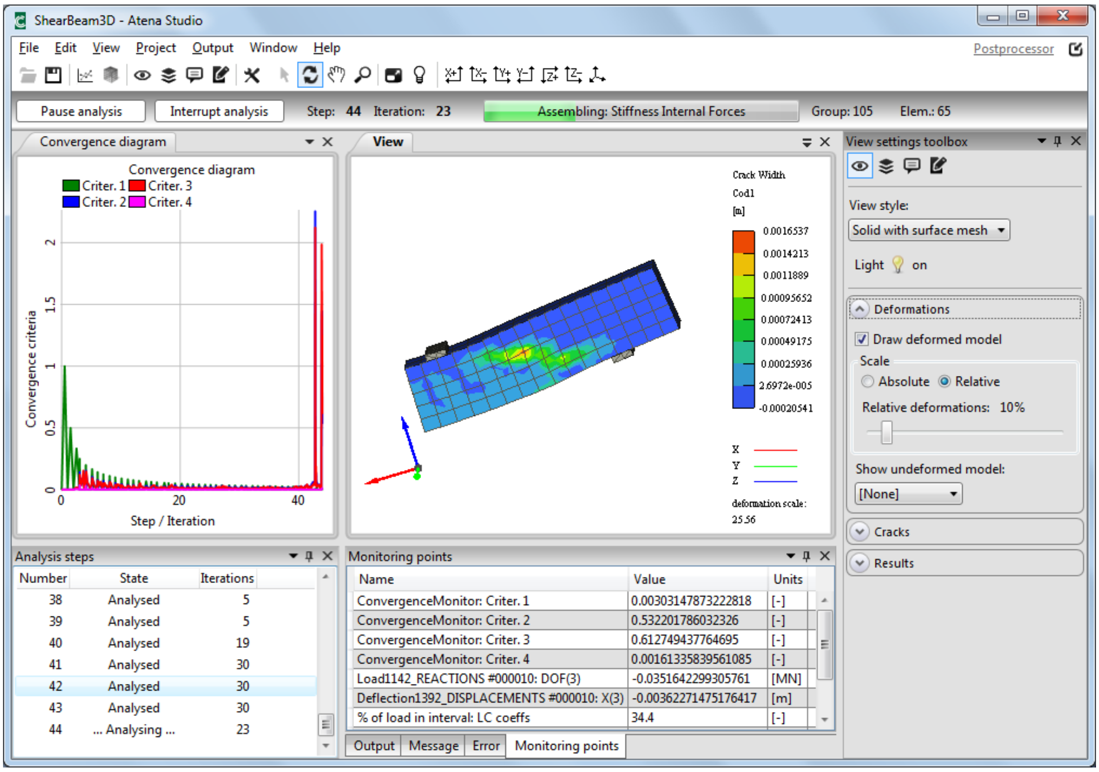


Figure 162: Deformed structure

Besides colour scale indicating distribution of crack width on the structure the actual cracks can also be displayed directly on the surface. It is done by opening Cracks box in View settings toolbox (see Figure 163) and then by the setting the minimal crack width to show and multiplier of crack width (see Figure 164).

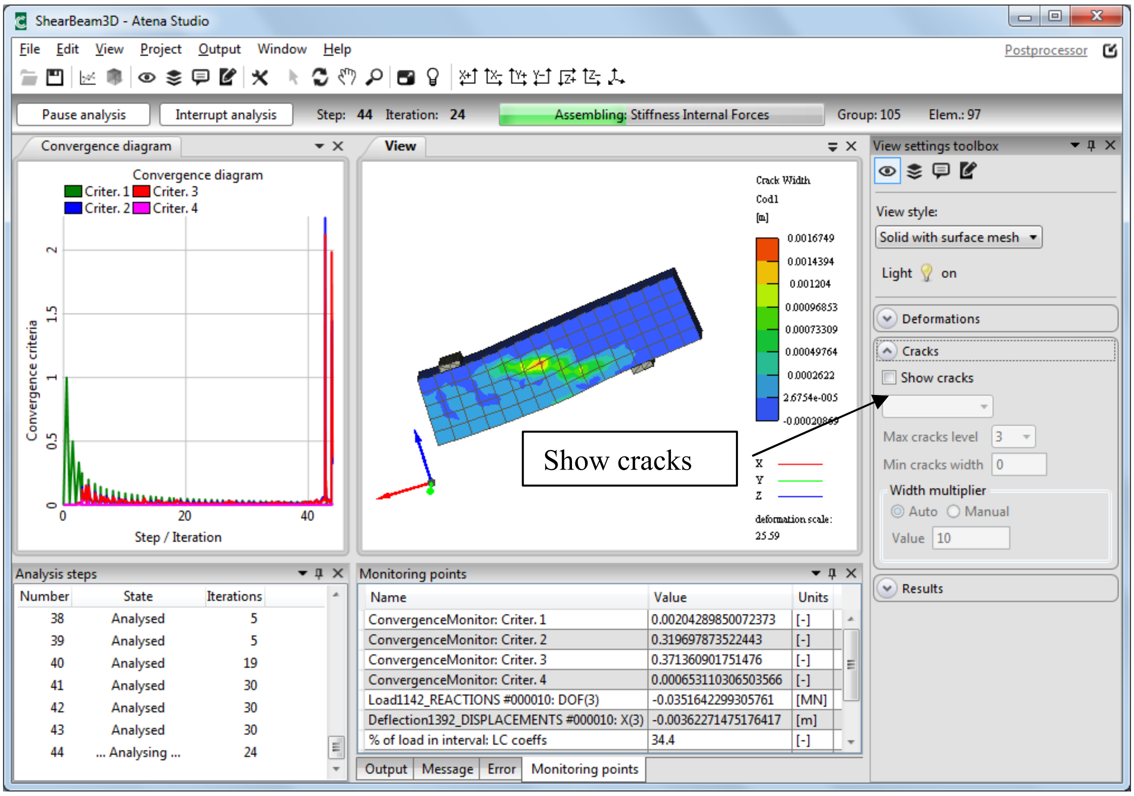


Figure 163: View settings toolbox with Cracks panel

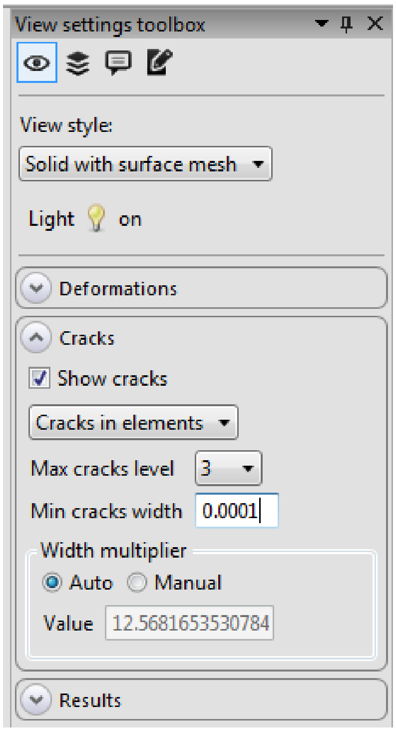


Figure 164: Cracks panel options

Parameter input:
Cracks in elements

Max cracks level:	3
Min cracks width:	0.0001
Width multiplier:	Auto

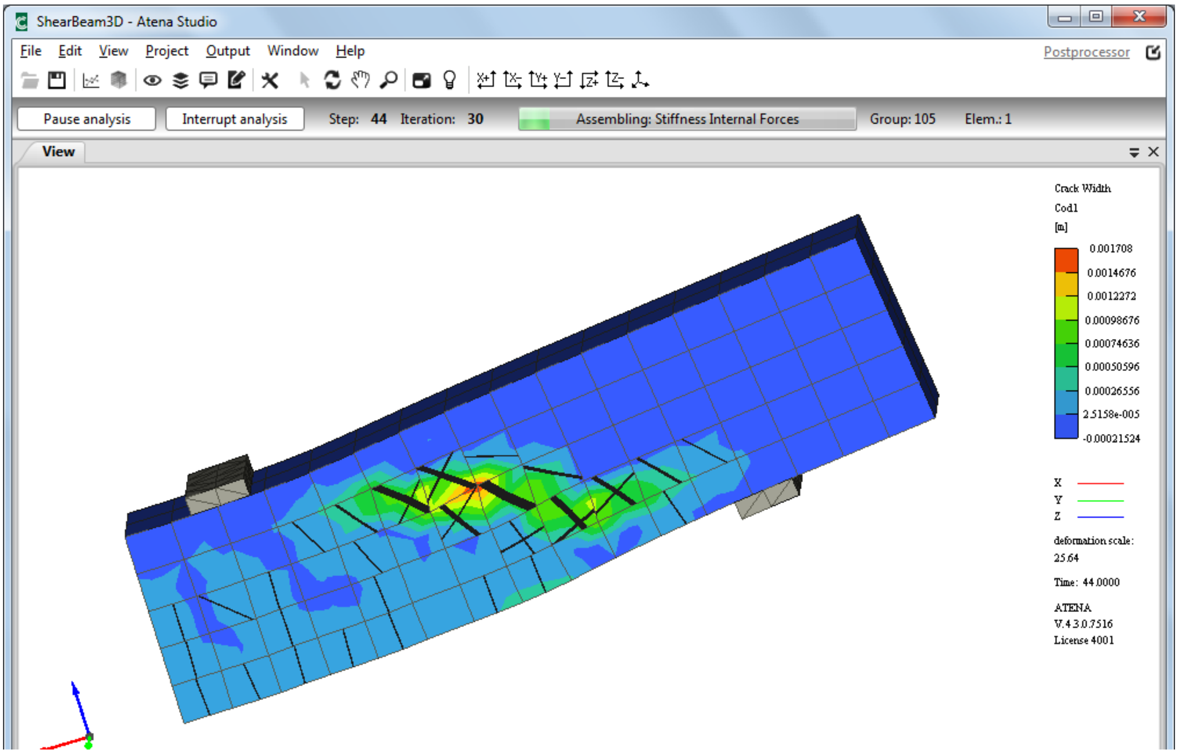


Figure 165: The crack width value on structure surface with cracks drawn as lines

All of these options of drawing cracks and results on the structure are in fact post processing features of **ATENA Studio**. But they can also be used during the execution of the nonlinear analysis. This is one of the unique features of **ATENA** software. During analysis execution all **ATENA** post-processing capabilities are available. For more information it is recommended to study ATENA Studio Manual [7].

5. POST-PROCESSING

The created model can be post-processed in the **ATENA Studio** or in the **GiD**. **ATENA** post-processing was already briefly described in the previous Section 4.4.

5.1 GiD Post-processing

After finishing the nonlinear analysis, **ATENA Studio** window can be closed. The program asks if all changes should be saved. Then button **Yes** should be selected in all cases.

Then back in the **GiD** interface the process info will appear. Through this dialog the program asks if the process of the analysed problem is finished or if the post-processing should be started. The button **Postprocess** should be selected (see Figure 166).

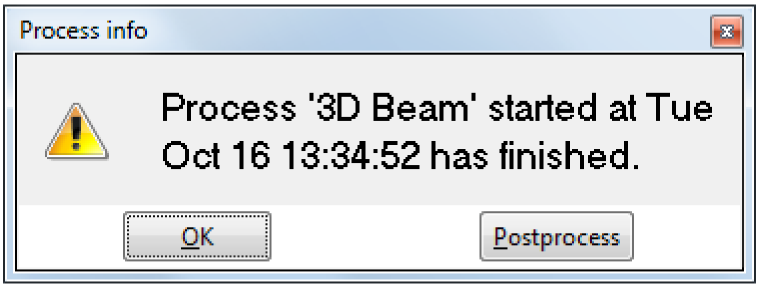


Figure 166: The button Postprocess should be pressed

But before any post-processing features can be used the results from the **ATENA** have to be imported into **GiD**.

It is done by the clicking on the Import results from **ATENA** icon  . Then the process of importing will start (see Figure 168) and when it is finished the model changes its colours (see Figure 169).

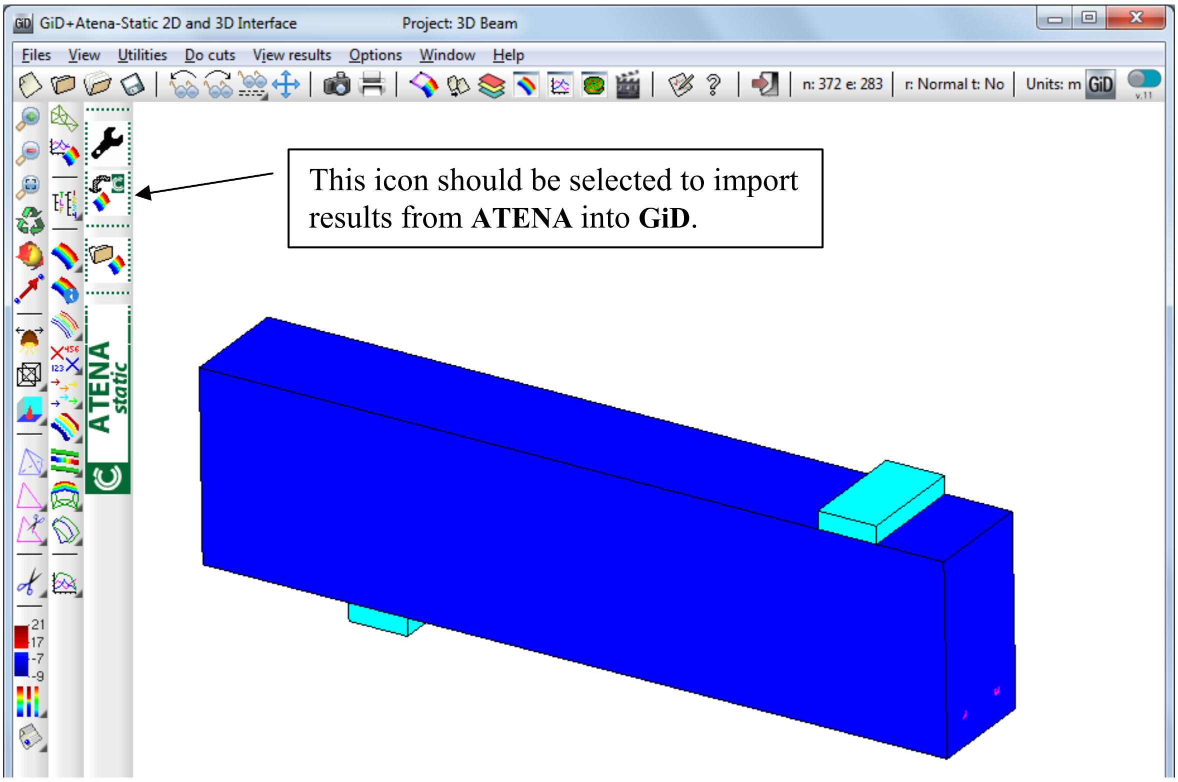


Figure 167: The GiD postprocessor interface

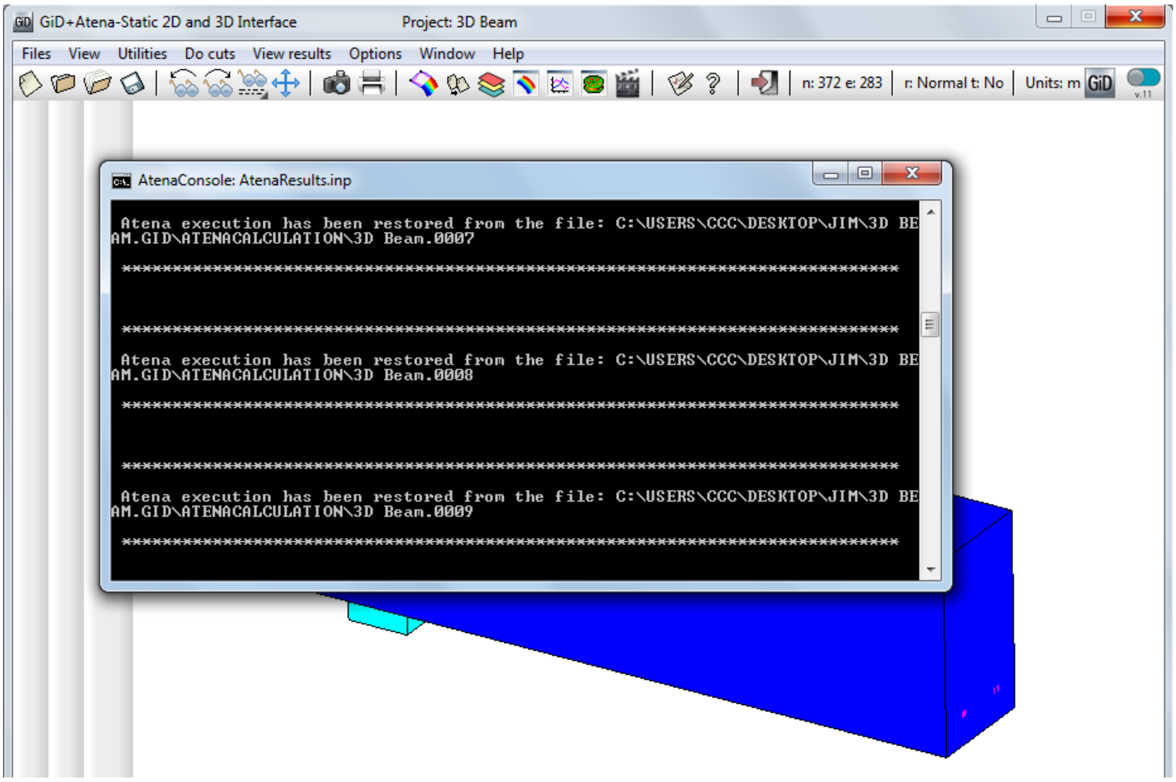


Figure 168: The importing of the results from ATENA into GiD

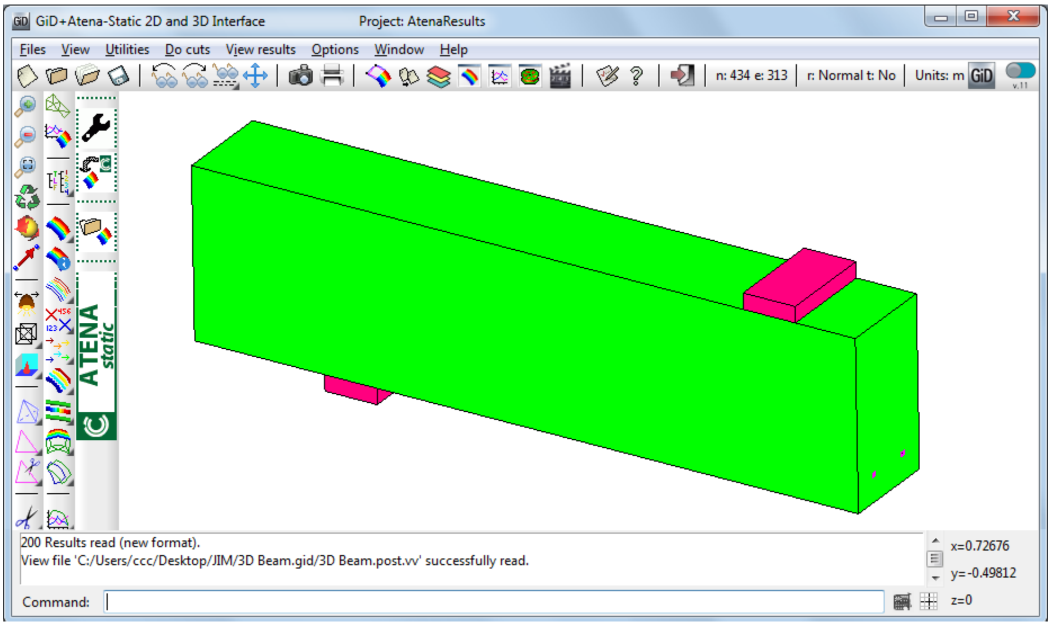


Figure 169: The importing of the results from ATENA were finished

After importing data from **ATENA**, the post-processing can be started. Let's display cracks like in previous chapter 4.4 of FE non-linear analysis in **ATENA Studio**.

First of all it should be checked which step will be post-processed. It is done by selecting **View Results | Default Analysis/Step | ATENAResults2GiD** in the main menu or by the Default Analysis/Step icon . From the L-D graph (Figure 154) it is possible to see that structure failed after 50th step, therefore it is good to post-process for example step 35 (see Figure 170).

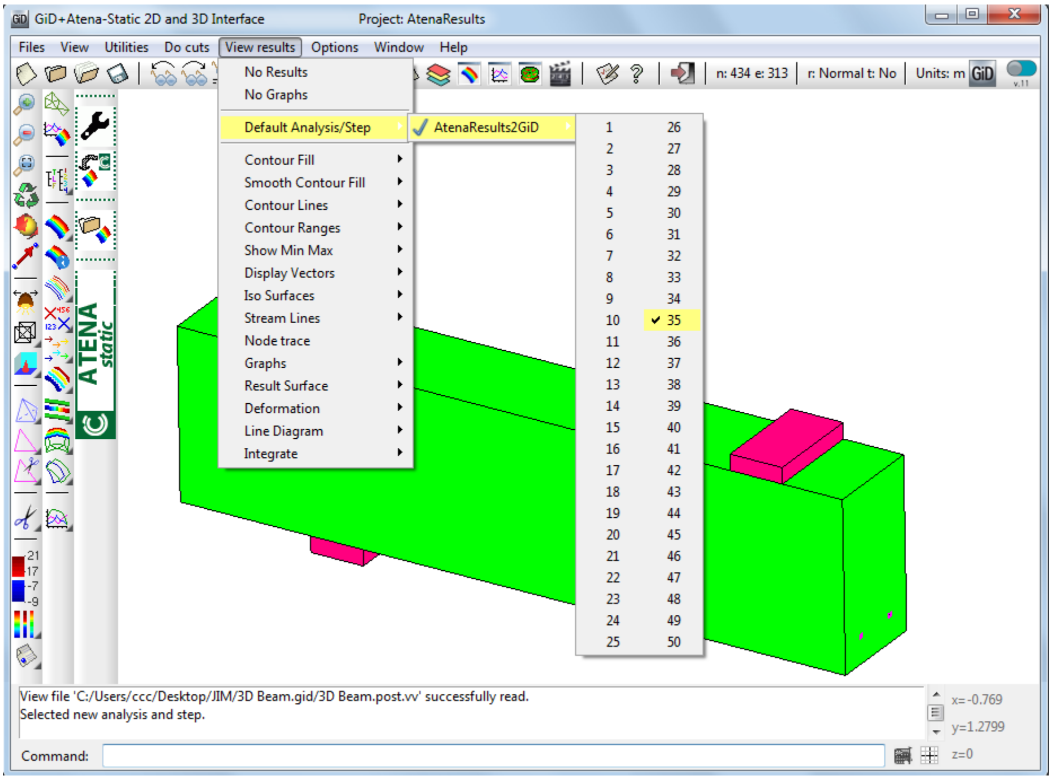


Figure 170: The selection of the step which should be post-processed

By the clicking on the Contour fill icon or by the selecting the command from main menu **View results | Contour Fill | CRACK WIDTH | COD1** crack width can be displayed like in previous chapter (see Figure 171).

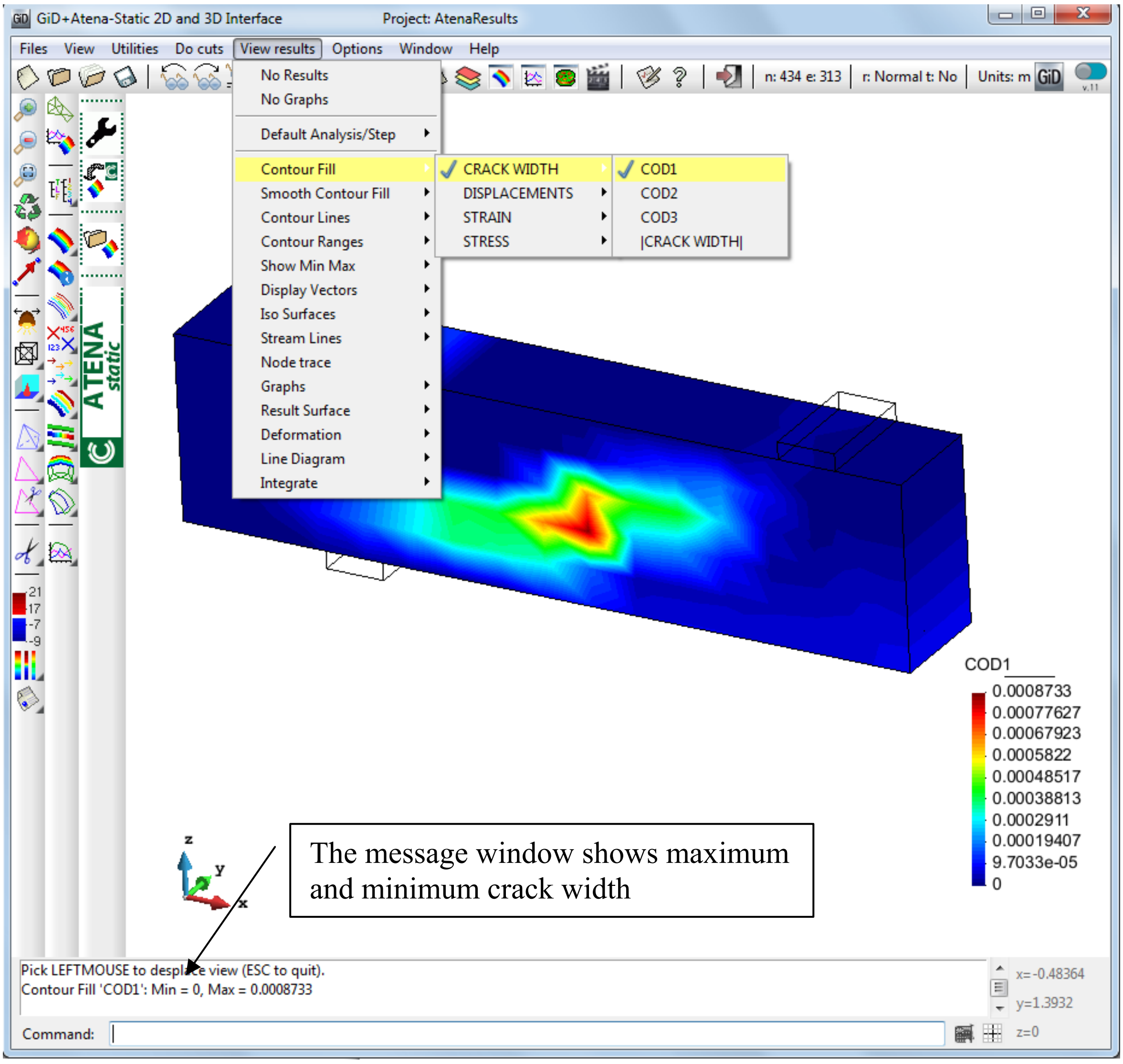


Figure 171: The display of the crack width

In the command Contour Fill, the pull down menu offers options which can be displayed. Currently rather limited set of quantities is available, however, much more result types are available in **ATENA Studio**. To be able to visualize these additional quantities, the program has to be switched to pre-processing.

It is done by selecting icon Toggle between pre and postprocess (see Figure 172). After that a dialog window appears and the button **OK** should be pressed. The program switches into pre-processing. Then the command **Data | Problem Data | Post Data** can be selected in the main menu and a window for the definition of the post data will appear (see Figure 173). This dialog you can run directly by clicking to icon in postprocessor.

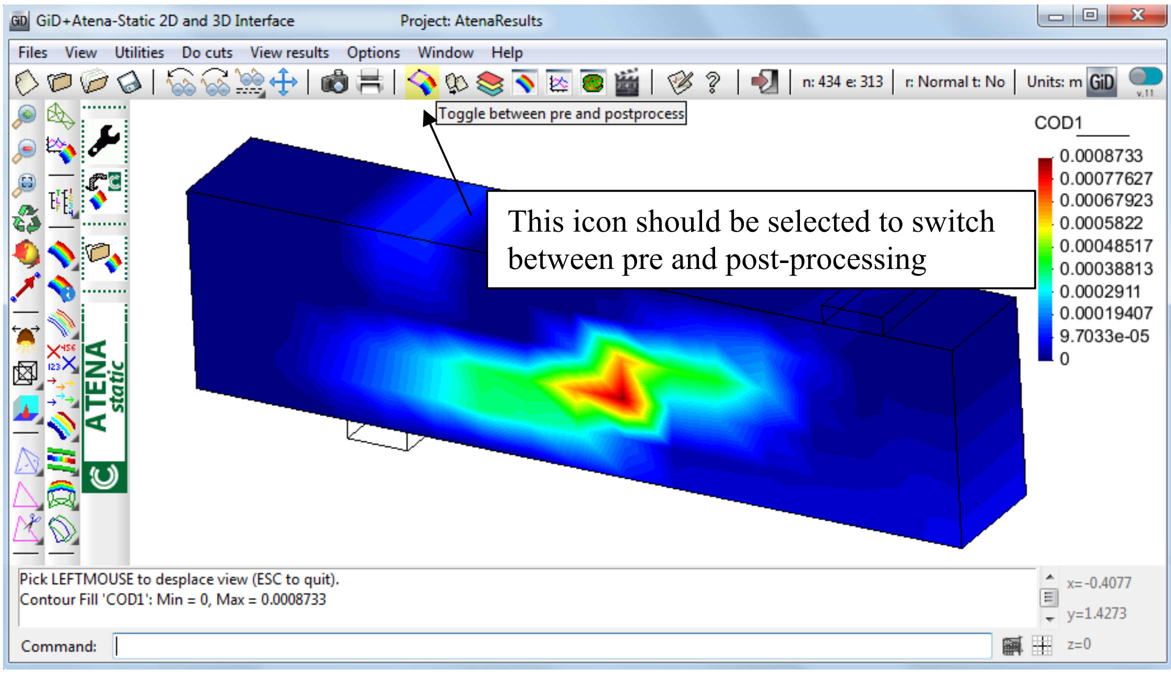


Figure 172: Switch between pre and postprocessing

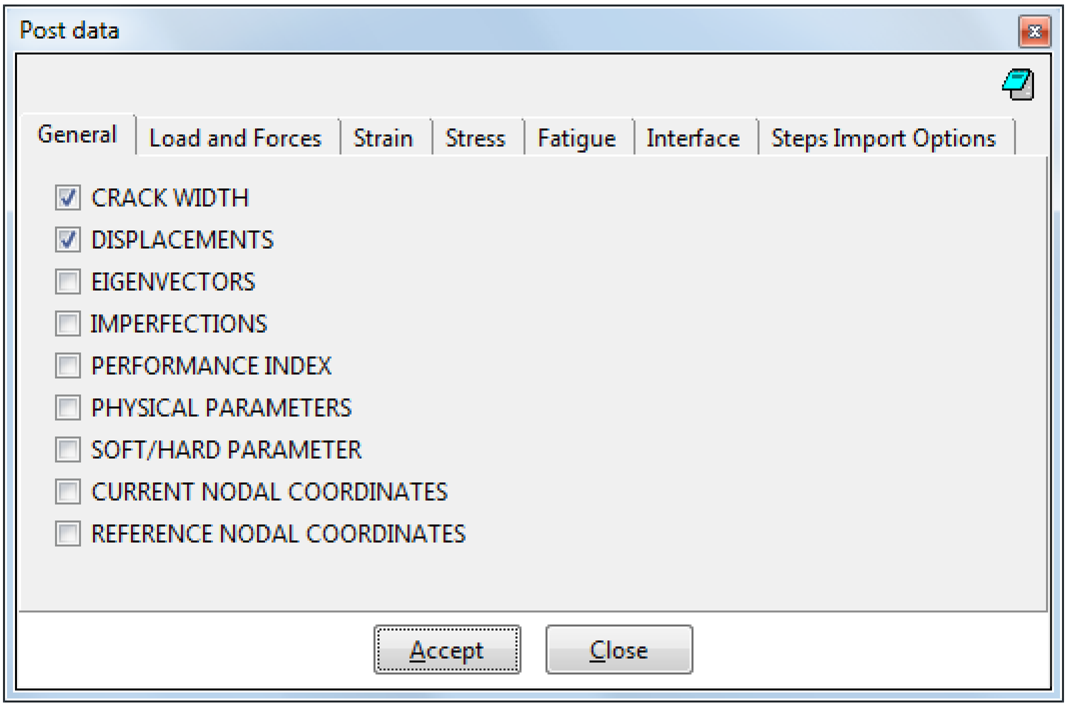


Figure 173: The selection of the data which should be available for the post-processing

For example the FRACTURE STRAIN can be chosen. The definition of post data is completed by selecting **Accept** button (see Figure 174). Then the button **Close** can be pressed and the **GiD** will switch to post-process automatically. But there in the post-process the data from **ATENA** has to be imported again.

It is done by the clicking on the **ATENA** icon  . Then the FRACTURE STRAIN can be found in the options for the post processing (see Figure 175, to obtain this figure the 35th step has to be selected again).

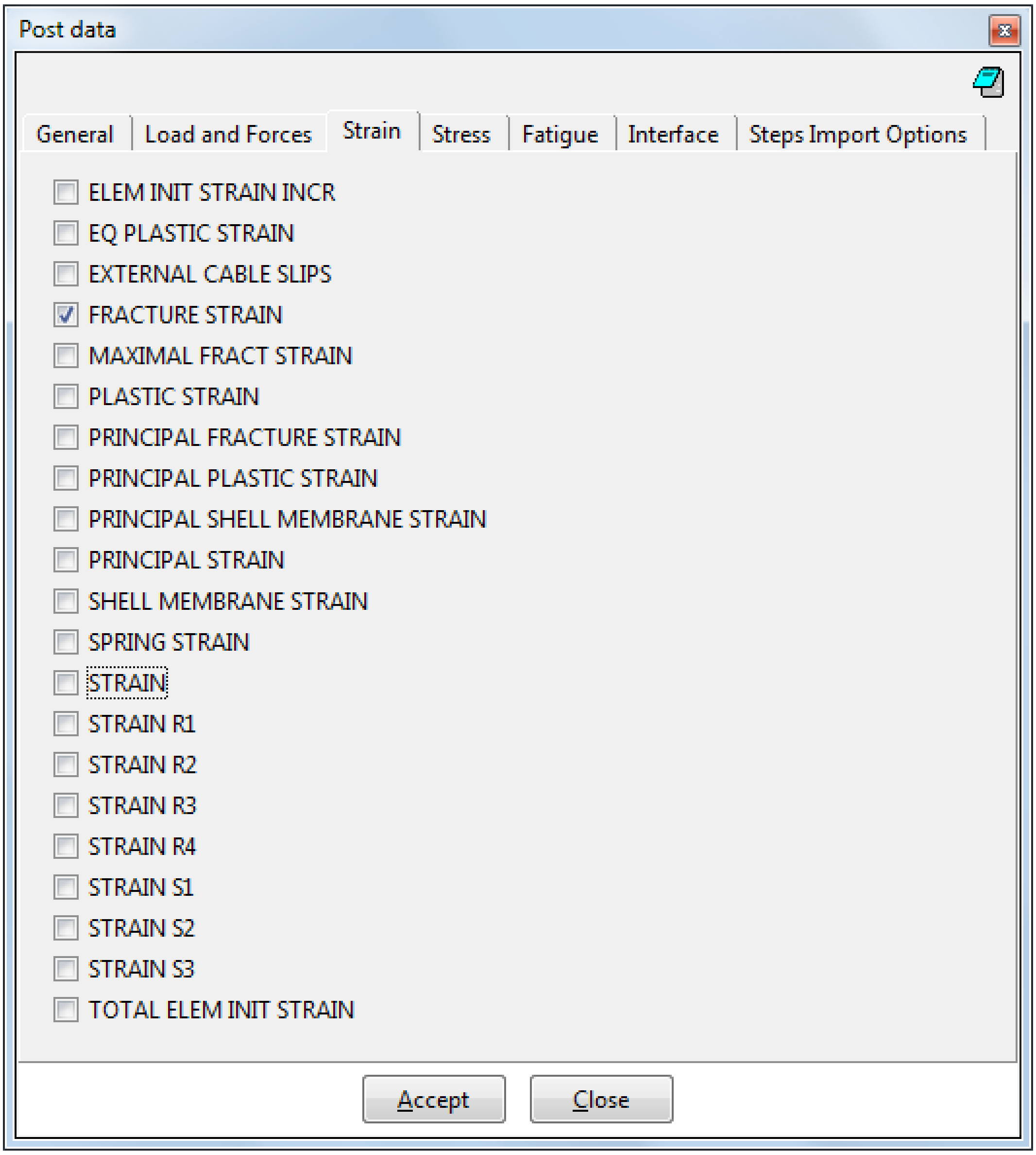


Figure 174: The selection of the FRACTURE STRAIN

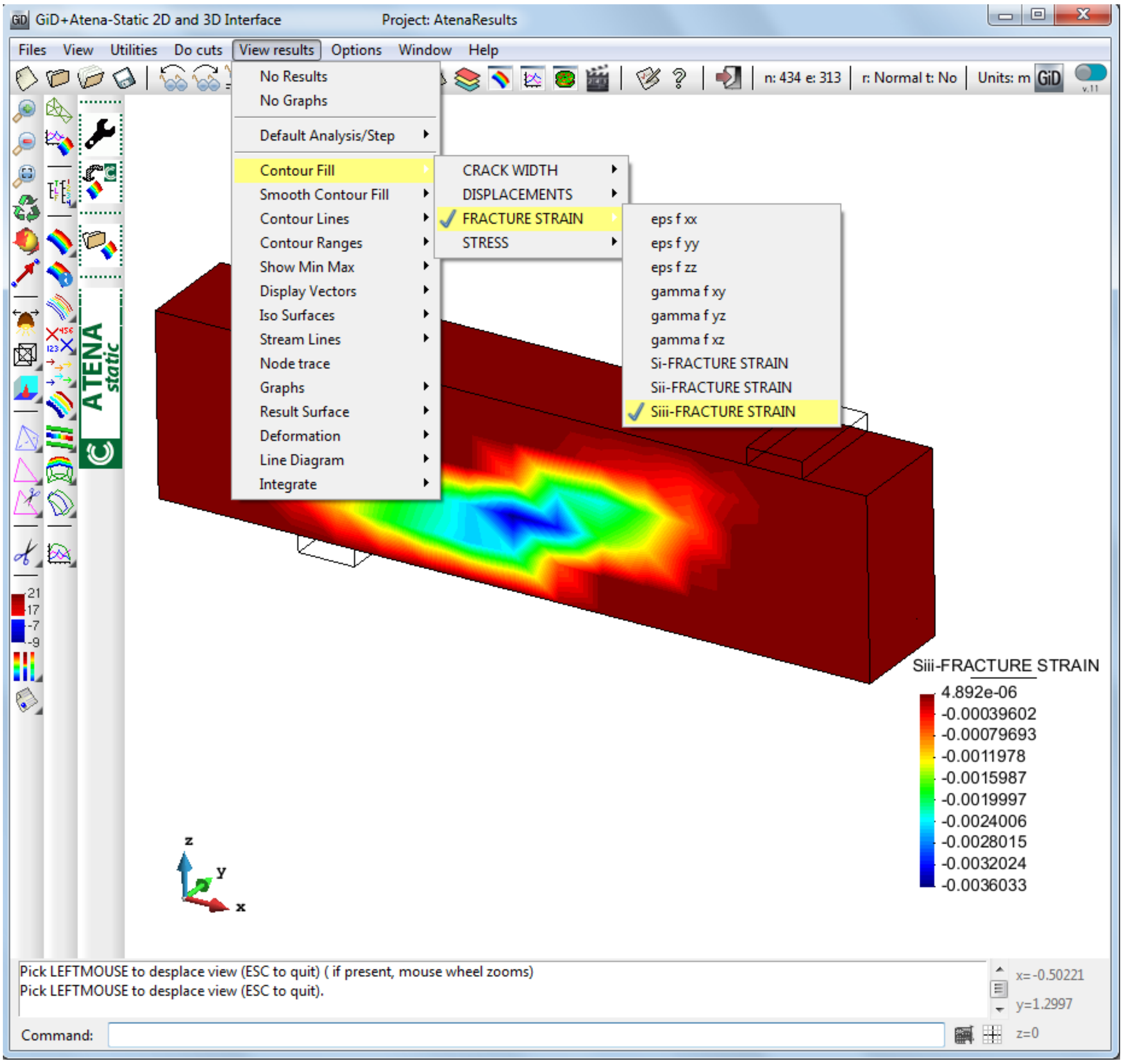


Figure 175: The displayed FRACTURE STRAIN

More post-processing capabilities can be found in the Help of the **GiD** or in the GiD manual [5].

5.2 ATENA Studio Post-processing

Results can be post-processed also in **ATENA Studio**. The L-D diagram and Crack width, which have been explained in the chapter 4 (section 4.3 and 4.4) are the few of the many possibilities of post-processing in **ATENA**.

For post-processing in **ATENA Studio** it is important to know how to open results. First of all **ATENA Studio** should be started from the Start menu on your computer. Then create new project from result files (see Figure 176).

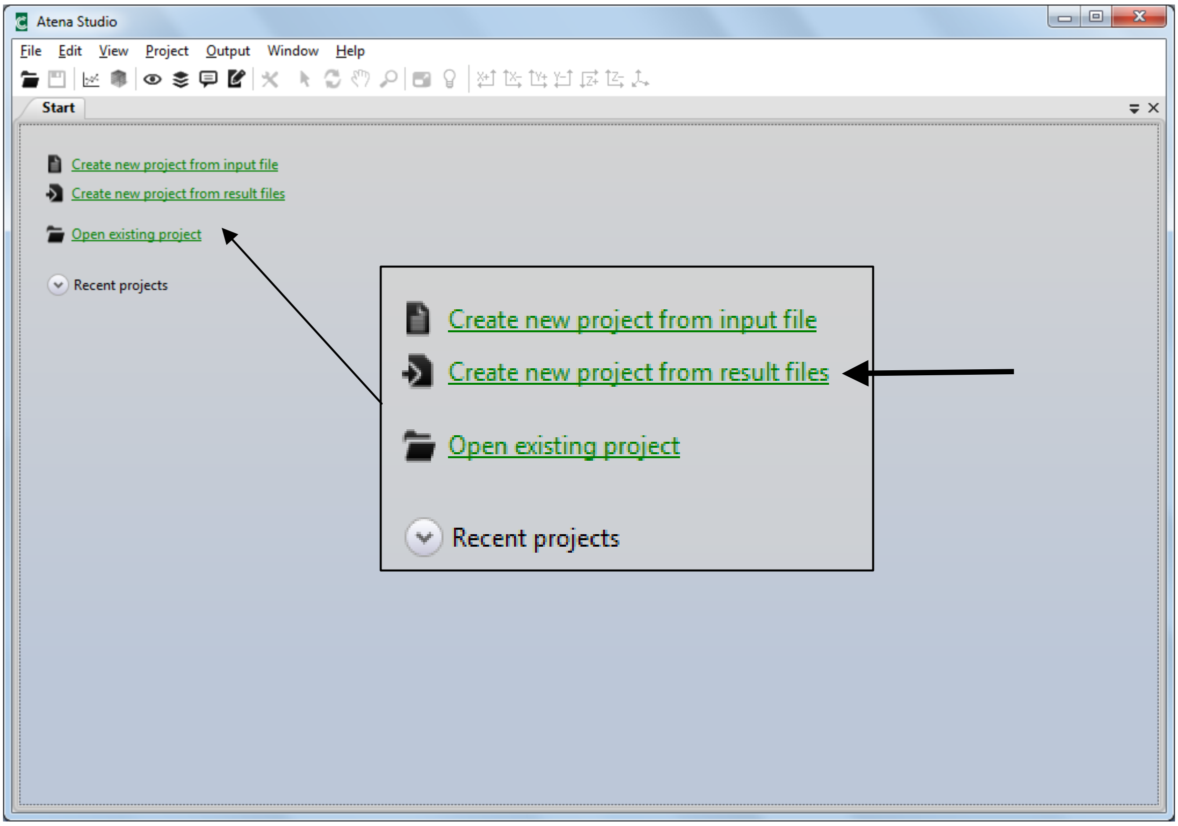


Figure 176: Starting of the ATENA Studio project

The step data file name should be **"3DBeam.0xx"**, where **3DBeam** is the task name as it was defined in **GiD** in Section 3.1.1.2. The suffix **0xx** represents the load step number, which should be post-processed. In this case for example the 25th step can be chosen (see Figure 177). Then the project properties are displayed (see Figure 178). Click **OK** button and then the display crack width can be defined (see Figure 179). The process of displaying of the crack width is described in the chapter 4.4.

More information about post-processing can be found in ATENA Studio Manual [7].

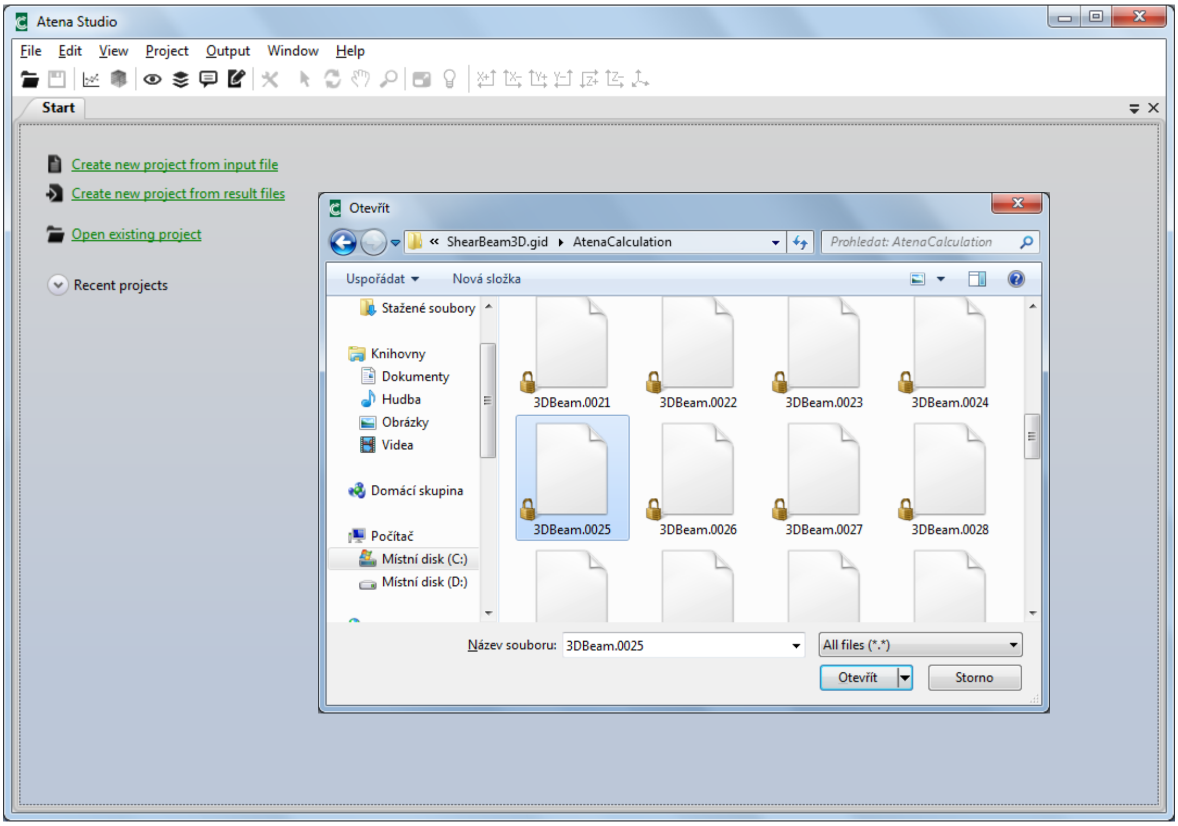


Figure 177: The result file opening

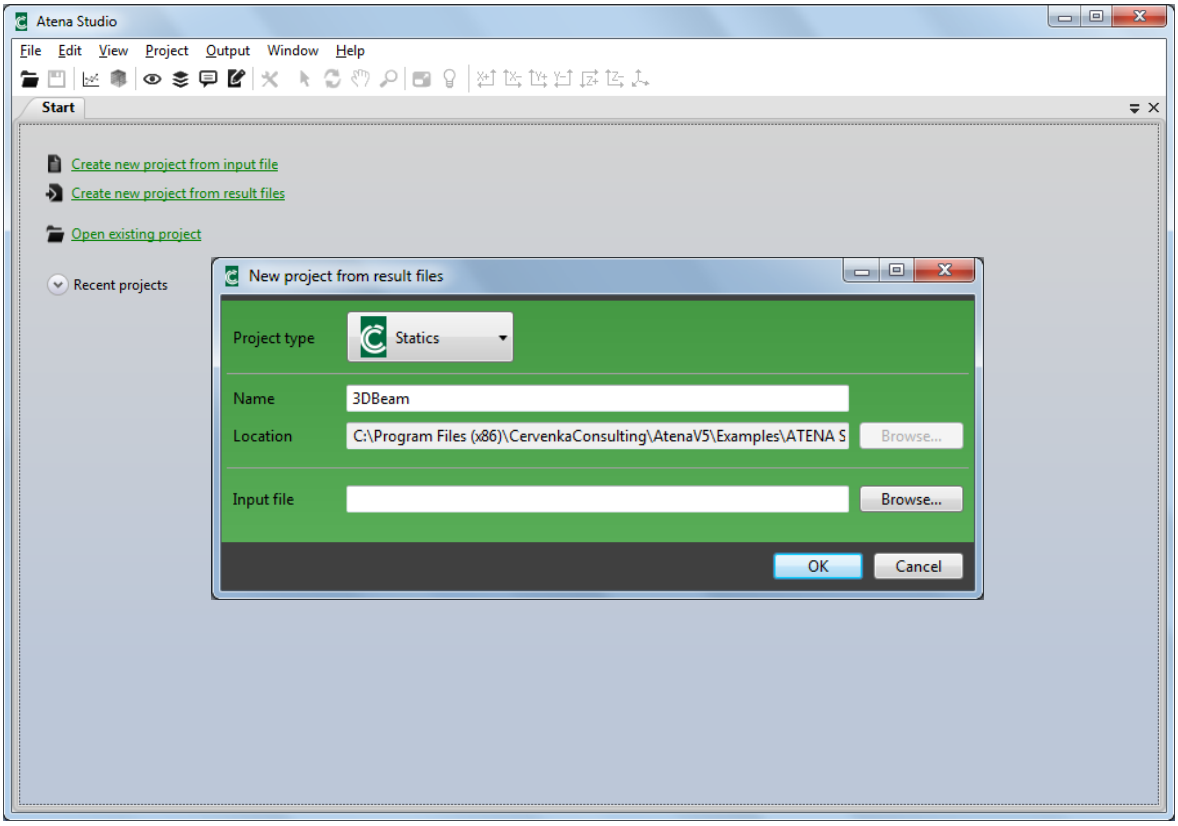


Figure 178: The project properties

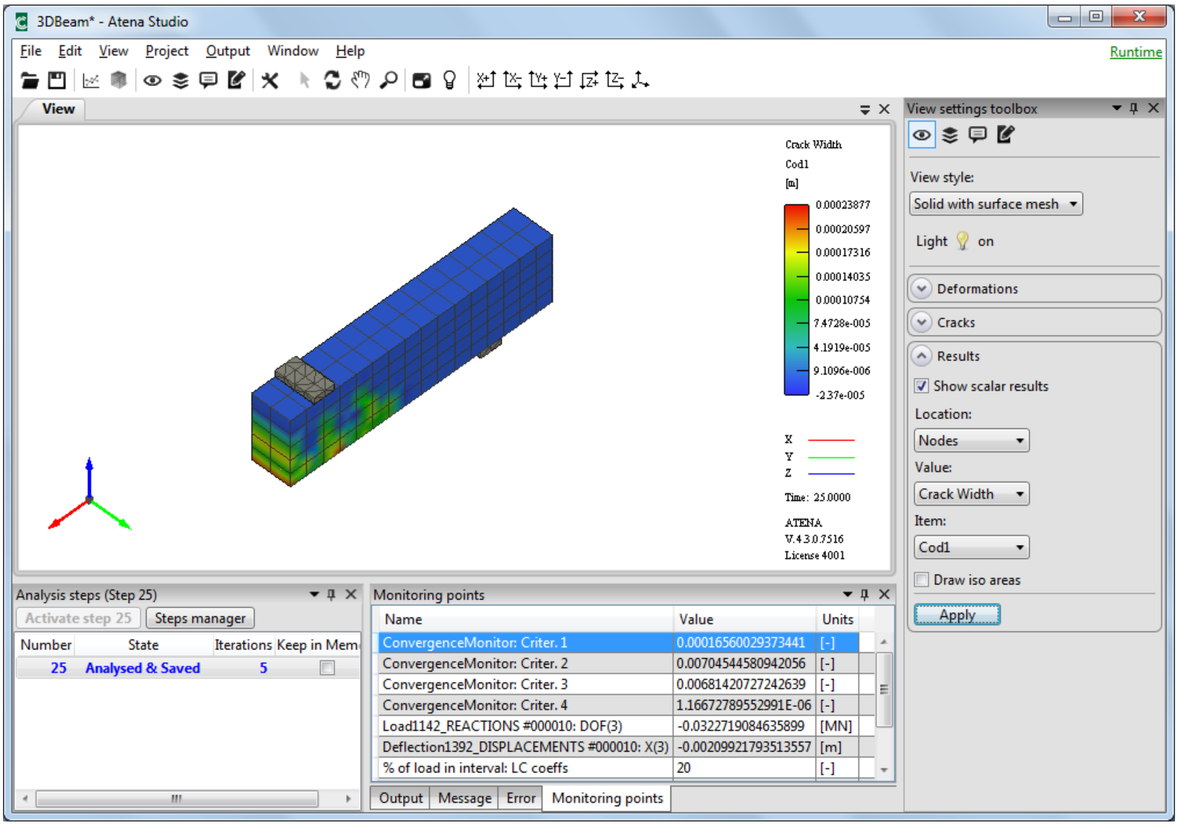


Figure 179: The crack width display of the 25th step

6. CONCLUSION

This tutorial provides a step by step introduction to the usage of **ATENA-GiD** on an example of a reinforced concrete beam without shear reinforcement. Although this example is relatively simple from geometrical and topological point of view, it is not a simple problem from the numerical point of view. Due to the missing shear reinforcement the beam fails by a diagonal shear crack, which is very difficult to capture using smeared crack approach.

This example demonstrates the powerful simulation capabilities of **ATENA-GiD** for modelling the brittle failure of concrete structures. Even with a coarse mesh, which was used in this demonstration example, the diagonal shear crack was successfully captured. Further improvement of the results can be achieved by decreasing the finite element size to for instance 8 elements over the beam height, 4 elements over the beam width and 25 elements over the beam length.

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

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

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

7. PROGRAM DISTRIBUTORS AND DEVELOPERS

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

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

8. LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2009

[2] ATENA Program Documentation, Part 8, User's Manual for ATENA-GiD Interface, CERVENKA CONSULTING, 2009

[3] ATENA Program Documentation, Part 3, ATENA Examples of Application, CERVENKA CONSULTING, 2005

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2009

[5] GiD Reference Manual, version 9.0.4, International Center For Numerical Methods In Engineering (CIMNE), 2009

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

[7] ATENA Program Documentation, Part 12, ATENA Studio Description, CERVENKA CONSULTING, 2012