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 3-2

Example Manual
ATENA Science

Written by
Vladimír Červenka, Jan Červenka,
and **Zdeněk Janda**

Prague, March 31st, 2021

CONTENTS

1. INTRODUCTION

In this manual, we show commented descriptions of typical analyses, which can be conducted using ATENA software. The descriptions given in this manual do not provide complete step-by-step instruction but instead describes the key aspects of each type of analysis. For more detailed tutorials, the user is kindly referred to see our basic tutorials:

- GiD Tutorial [13],
- GiD FRC Tutorial [14],
- GiD Construction Process Tutorial [15],
- GiD Strengthening of Concrete Structures Tutorial [16].

Furthermore, there are multiple other examples provided with ATENA installation, which do not have any written description. Still, the user can open them to check how the inputs are specified for each analysis type. The full list of example data files provided with ATENA installation can be found in ATENA GiD User Manual [6], and the data files can be found in the following directory:

" %public%**Documents\ATENA Examples\Science\GiD** ".

2. STATIC ANALYSIS

This chapter contains examples of static analysis using the program **ATENA**. Currently, some commands required for static analysis are not supported by the native ATENA graphical environment, and therefore the necessary commands must be entered manually or by using the ATENA-GiD interface. **GiD** (see the Internet address http://gid.cimne.upc.es/) is a general purpose finite element pre and post-processor that can be used for data preparation for ATENA. See the README.TXT file in the ATENA installation for the instructions how to install the ATENA interface to **GiD**.
In order to activate the creep analysis option an appropriate problem type must be selected: **Data | Problem type | ATENA | Static.**

2.1 Example of a Static Analysis with Reinforcement

In this example, we demonstrate the usage of GiD for data generation of a simple structure. The structure is a reinforced concrete L-shaped cantilever. It has fixed supports on one end and is loaded by vertical force near the free end. See Figure 2-1. The first beam adjacent to the fixed end is subjected to the simultaneous action of bending and torsion, while the second beam is only under bending. Complex three-dimensional behavior can be well analyzed by **ATENA**, and for this purpose, the input data can be prepared in **GiD**.

2.1.1 Reinforcement modeling

The longitudinal reinforcement is by bars 4Ø28 that are located long the edges, and by stirrups Ø12 with spacing 100mm in the first beam, (section A) and with spacing 200mm in the second beam (section B).
Since there are different possibilities to model reinforced concrete, we first make a decision about the modeling approach. Concrete shall be modeled by 3D brick elements. For this, we chose the hexahedra elements. The longitudinal reinforcement shall be modeled by discrete bars. The stirrups shall be modeled as a smeared reinforcement within the reinforced concrete composite material. This is a simplified method, by which we avoid an input of detail geometry of stirrups. In the smeared model, the exact position of individual stirrups is not captured, and only their average effect is taken into account.
The resulting model is shown in Figure 2-2. The colors of elements show two types of materials used: the composite material named Cantilever1 in the short beam and Cantilever2 in the longer beam. The discrete bars are modeled by linear elements as shown in Figure 2-3. In the following, we shall treat the generation of the model in more detail. A data file with this example can be found in the ATENA installation under the name **SmallCantileverWithTorsion** in the subdirectory **"%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.Static3D"**.

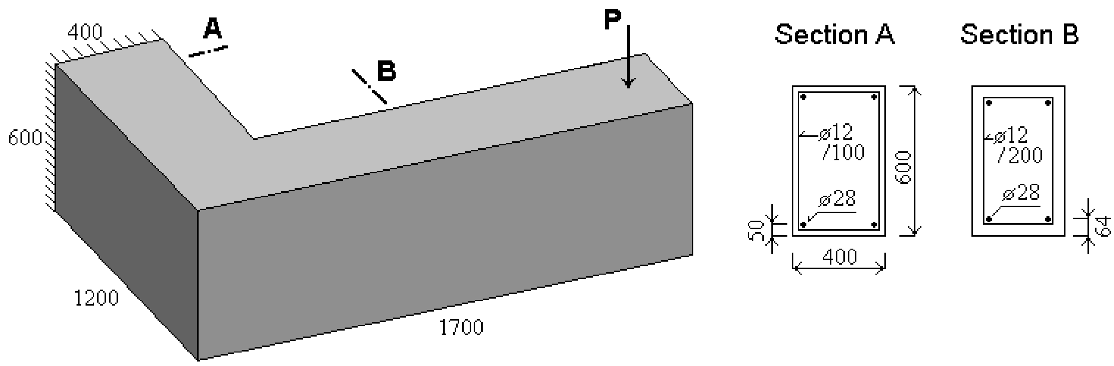


Figure 2-1: L-shaped cantilever beam. Dimensions in mm.

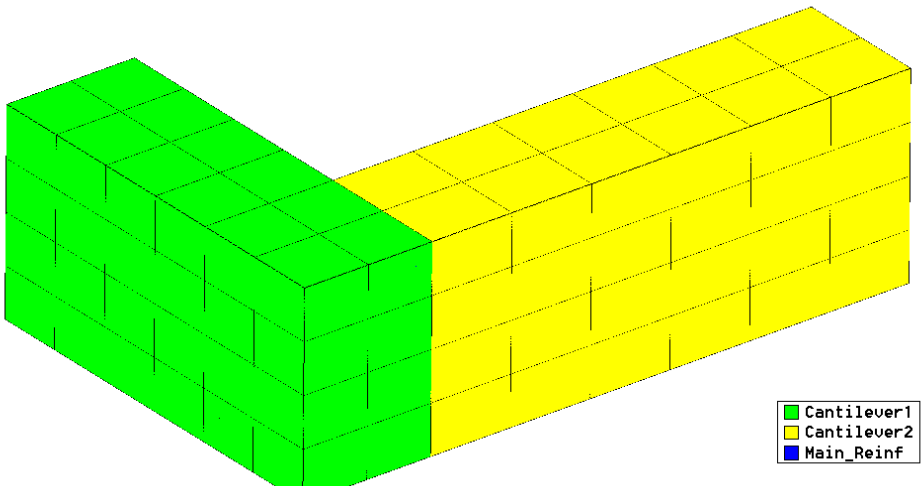


Figure 2-2: The model with two composite materials: Cantilever 1 and Cantilever 2.

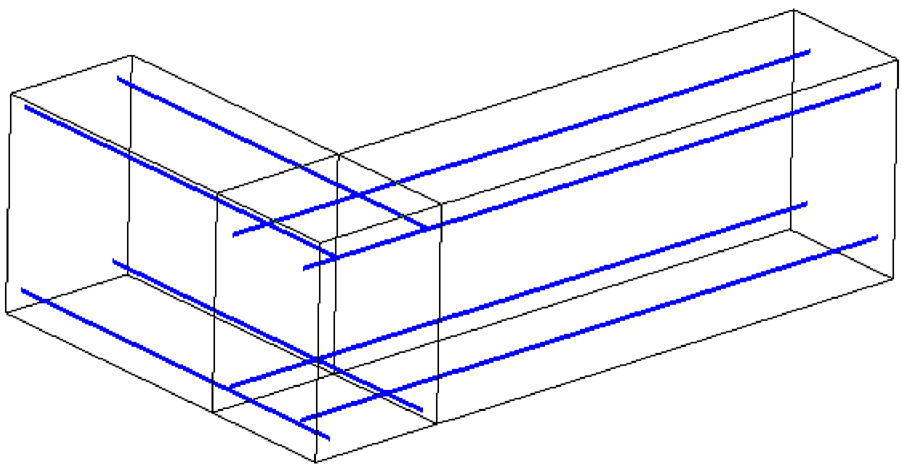


Figure 2-3: The model of the discrete bars.

Since the smeared model of stirrups does not exactly represent their geometry, it is alternatively possible to use discrete bars as well. This case is not described in this manual, but it can be found in the data file **SmallCantileverWithTorsion_DiscreteBars** enclosed in the ATENA installation.

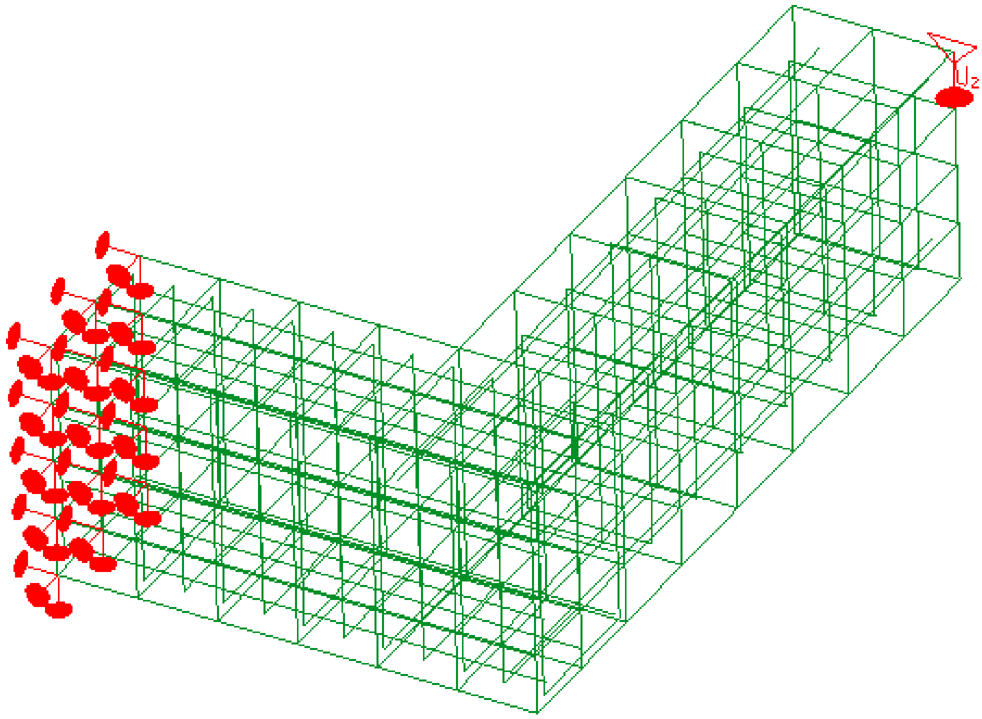


Figure 2-4: Final finite element model with supports and loading.

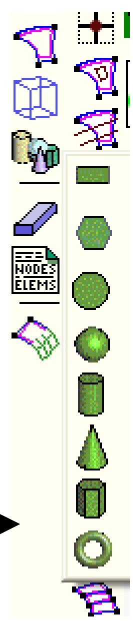
2.1.2 Problem type and data

Typically, the problem definition starts by choosing an appropriate problem type by selecting the menu item **Data | Problem type | ATENA | Static** and then the general solution data in **Data | Problem Data**. Both steps were already described in Chapter 4. However, the parameters of **Problem Data** can also be changed later.

2.1.3 Geometry

The geometry is created using the GiD graphical tools from elementary objects sequentially, starting from points, lines and finally surfaces and volumes. We start with the definition of points. Points are connected to lines. From lines, we can form surfaces, and from surfaces, we can form volumes (solid objects). Details of this input shall be skipped since it belongs to standard GiD functions. The final geometrical model is shown in Figure 2-5. Note that it contains two types of objects: volumes for concrete (and reinforced concrete) and lines for the discrete reinforcement.

In **GiD**, it is also possible to create volumes directly from predefined primitives as shown in the figure on the right, which indicates the available list of predefined primitives such as rectangle, circle, sphere, etc. The volumes can also be created by extrusion, which is activated from the GiD menu **Utilities | Move** or **Copy**. In this dialogue, various copy operations can be selected, such as rotation, translation, sweep. There is also a check box, which activates the extrusion.



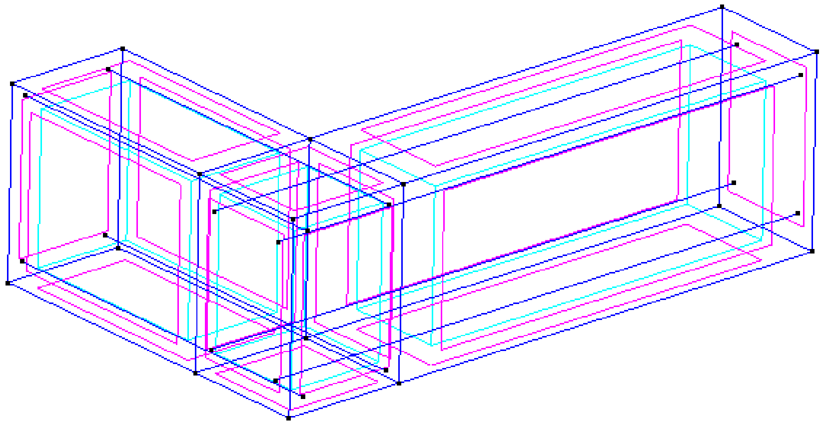
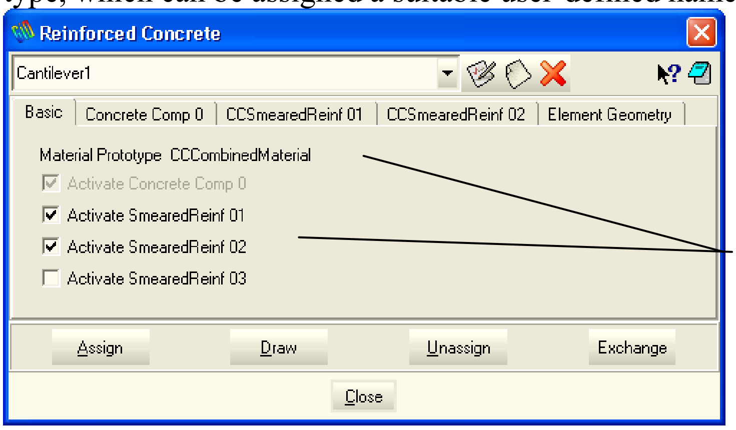


Figure 2-5: Geometrical model.

2.1.4 Materials

The materials can be defined and assigned to the geometry using the menu item **Data | Materials**. The recommended procedure is to keep the default material unchanged for later reference and create any number of user-defined materials. Since we intend to model the vertical stirrups by smeared reinforcement, we shall use the material type **Reinforced concrete**. **CCCombinedMaterial** is a default material, and Cantilever1, Cantilever2 are user-defined composite materials that are created from the default material by pressing the button . This command creates a new material of the same type, which can be assigned a suitable user-defined name (see Figure 2-6).



The smeared reinforcement components are activated using these checkboxes. When selected, new property sheets appear in the dialog

Figure 2-6: Reinforced concrete material. Two composite materials created.

2.1.4.1 Reinforced concrete as composite material

First, we define the parameters of the concrete component. This can be done by selecting the tab **Concrete Component(0)** and modifying its parameters. There are several choices available for the basic material. It is recommended to select the material **CC3DNonLinCementitious2**, which is identical to the same material from the group **Concrete**. The dialog window is extended to allow additional reinforcement components. The buttons  allow changing, adding new, and deleting of materials. When adding a new material with the button , the default material is first copied, then re-named and edited.

The stirrups are modeled by smeared reinforcement as **Component(1)** of the composite material. The first 5 parameters describe the initial elastic modulus, reinforcing ratio, and direction.

The reinforcing ratio of smeared reinforcement is calculated as $p=A_s/A_c$, where A_s , A_c are the section areas of bars and concrete, respectively, in the considered volume. This ratio is different in each part of the cantilever due to different stirrup spacing. The direction of the smeared reinforcement is defined as a unit vector.

The constitutive law of the reinforcement is defined as multi-linear by a sequence of points (stress-strain pairs). The first point is defined by yield strength (and elastic modulus). This gives a bi-linear, elastic-plastic law with unlimited ductility. A general multi-linear function can be defined by additional points. Maximum 4 additional points can be given.

Up to three smeared reinforcements can be defined in one composite material. This limit exists only in the GiD interface. (**ATENA** can define an unlimited number of components for a single composite material, in this case, it is necessary to manually edit the ATENA input, which is generated by **GiD**.)

After the parameter definition, the material can be assigned to the structure. This is done by the button **Assign** and following the appropriate selection by mouse. The process of selection is a general operation, and it allows for selecting of points, lines, surfaces, and volumes. In this case, the material should be assigned to volumes (of geometry), Figure 2-9, Figure 2-10.

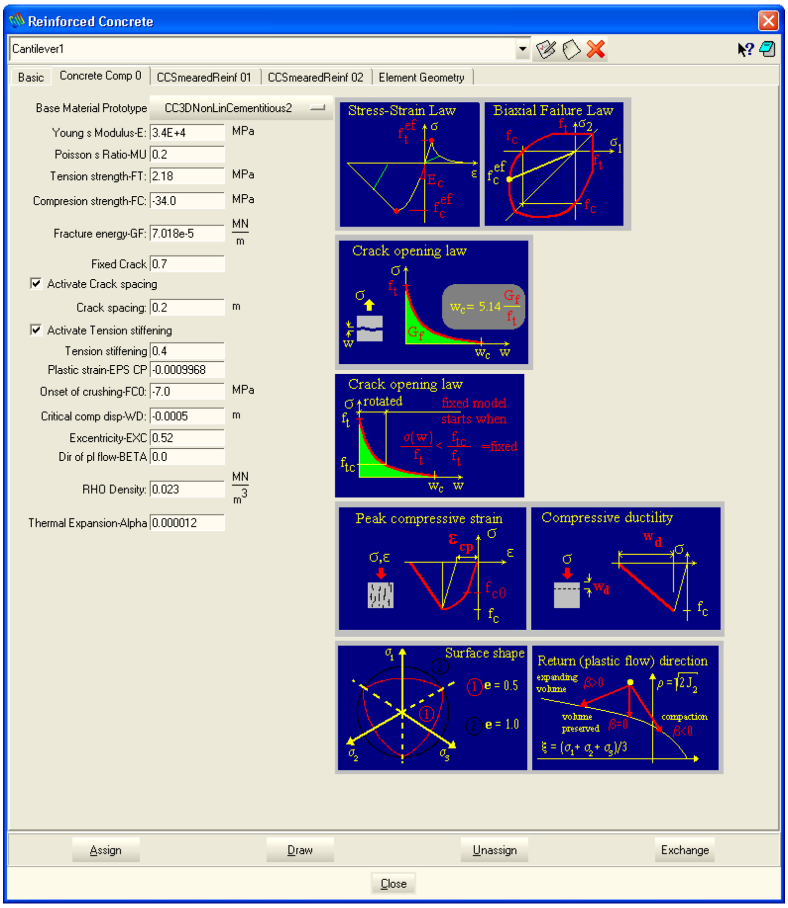


Figure 2-7: Concrete component in the 'Reinforced concrete' material

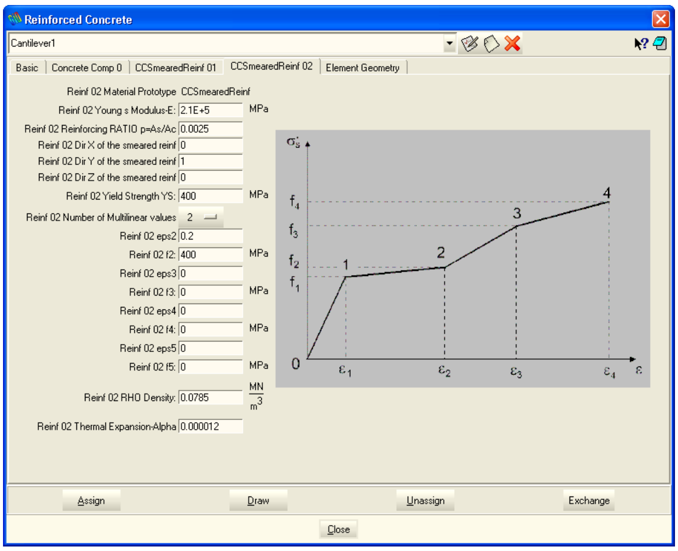


Figure 2-8: Components of smeared reinforcement in the composite material

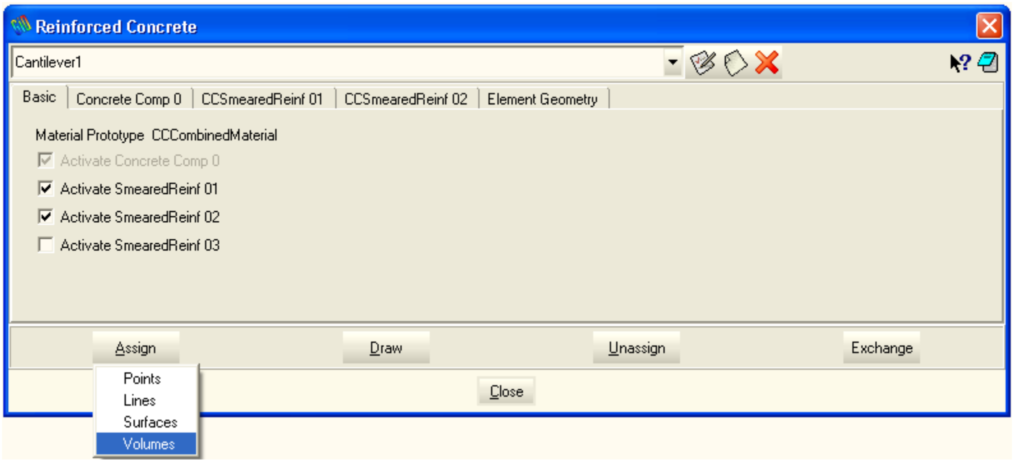


Figure 2-9: Menu item 'Assign | Volumes'.

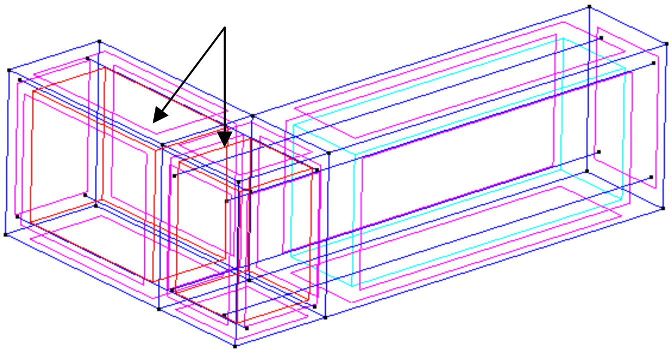


Figure 2-10: Selected volumes are highlighted in red colour.

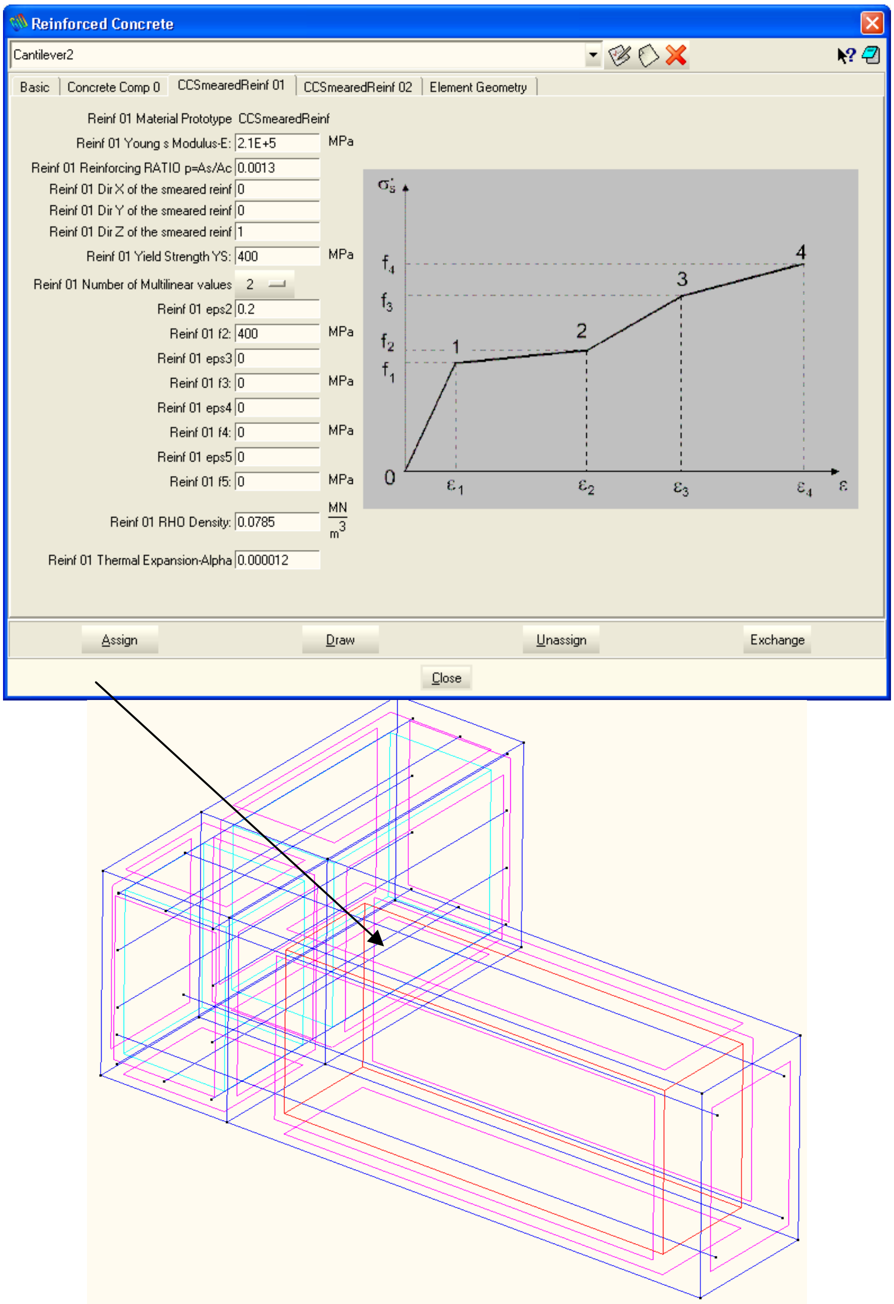


Figure 2-11: Assignment of the material 'Cantilever2.'

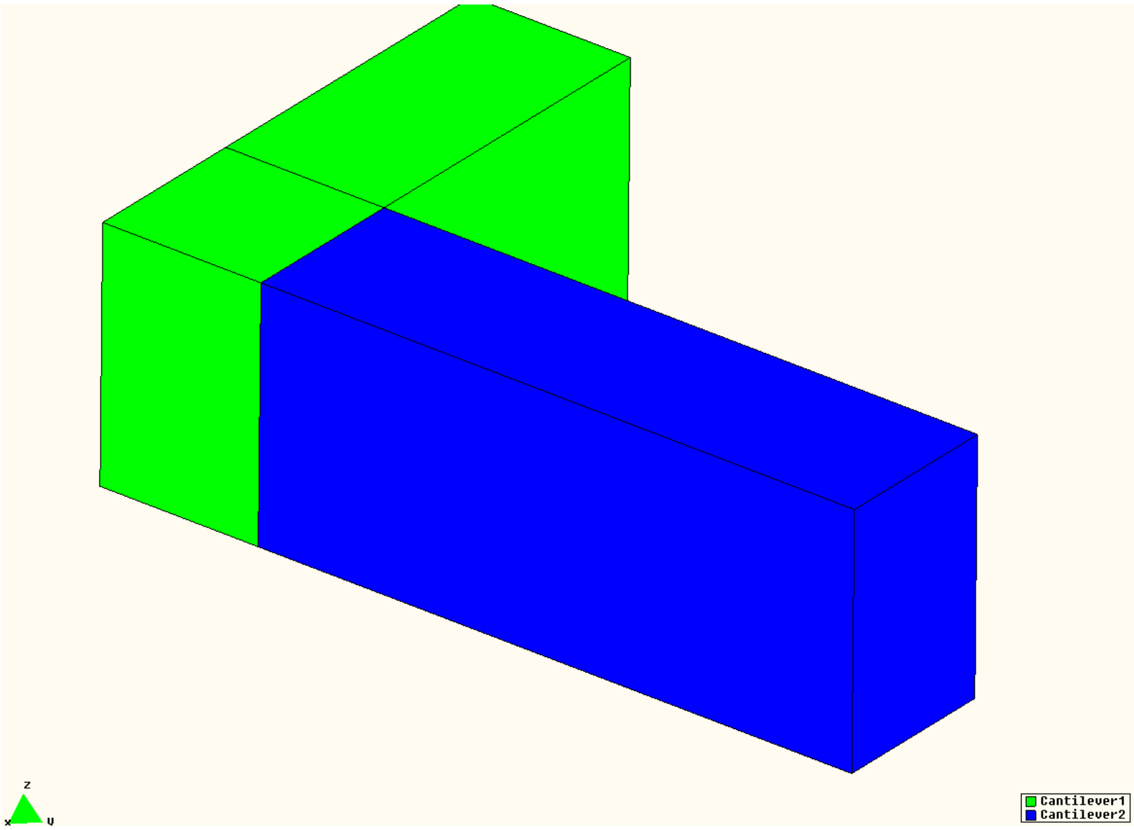


Figure 2-12: Display of the assigned material groups.

Composite material for the second part of the structure (named as Cantilever2) can be defined in a similar way, where the only difference is in the value of reinforcement ratio, Figure 2-11.

2.1.4.2 Bar reinforcement

From the menu **Data | Materials** we select the material **Reinforcement**, which is designated for discrete bars. There we choose from the list the ATENA-model **CCReinforcement** and then click on the button **New reinforcement** and enter the name for the reinforcement material.
After confirmation by OK a dialog for material parameters appears. The parameters include initial elastic modulus, yield strength, and optionally points on the stress-strain curve. The last parameter is the bar cross-section area (see Figure 2-14).
The material is then assigned to the geometry by pressing the button **Assign** and selecting **line** geometric entities by the mouse. The selected bars are marked by red color, Figure 2-15. Applying the command **Draw** at the bottom of the reinforcement material dialogue (see Figure 2-16) can check a correct assignment, which shows the geometry (in this case lines) with the currently assigned material.

In the case of pre-stressed bars, each bar (cable) must have a distinct material (even if its values are identical with other bars). The reason for this is to distinguish among groups of elements for pre-stressing. The pre-stressing is defined in **Conditions | Lines | Initial strains** and is assigned to the lines that model the pre-stressing reinforcement.

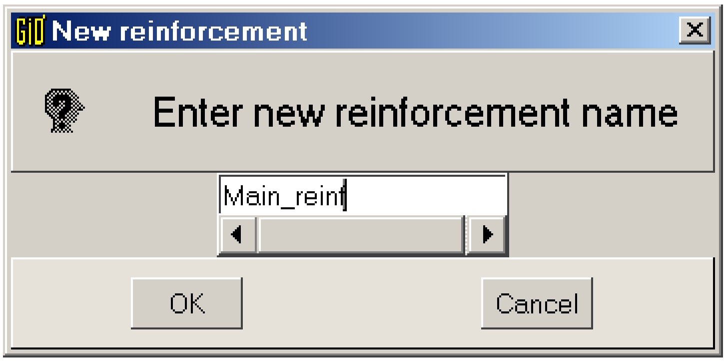


Figure 2-13: New material for bar reinforcement.

Figure 2-14: Material parameters for the 'Reinforcement' model.

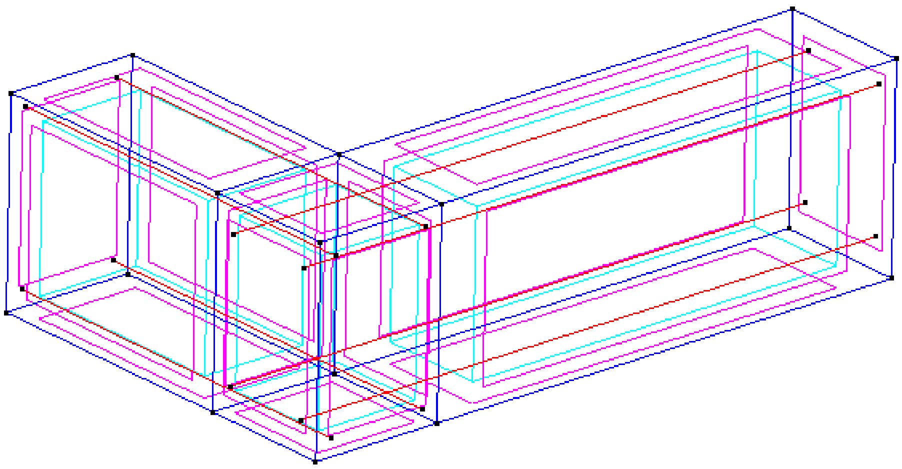


Figure 2-15: Assigning material to the geometry of bars.

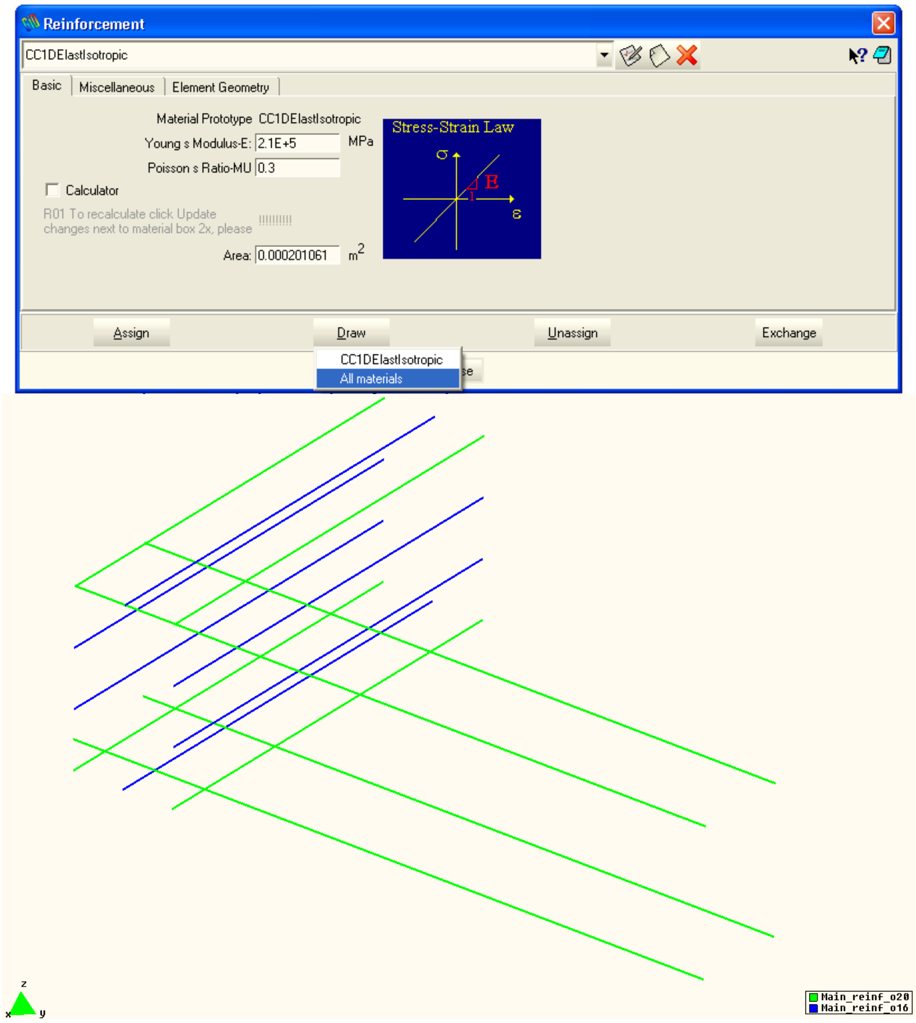


Figure 2-16: Display of the reinforcement material assignment

2.1.5 Supports and loading

The supports and loading can be specified using the menu **Data | Conditions**. We define the fixed nodes by checking X-, Y-, Z-Constrains and the type of geometry **Surface**. Using the command **Assign** we select the end face of the cantilever and finish the assignment of support conditions.

In a similar way, we assign the Point-displacement at the node of the load application. The load is applied as a vertical imposed displacement. Consequently, the force value is a reaction at this node.

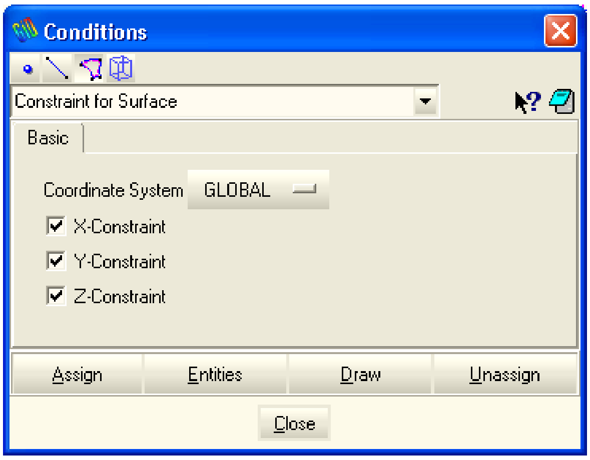


Figure 2-17: Definition of the surface support in all directions

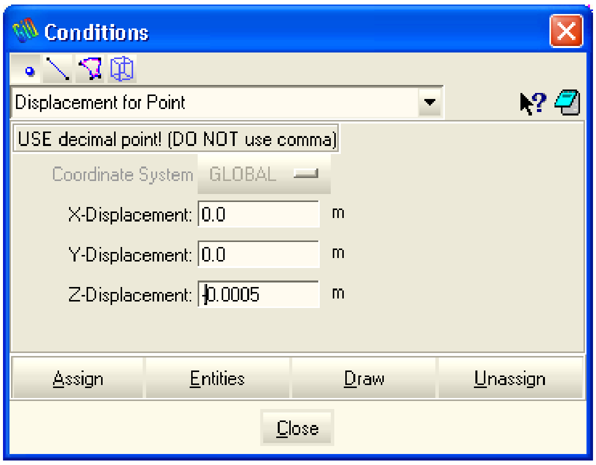


Figure 2-18: Definition of prescribed displacement in vertical direction

The conditions dialog of **GiD** can also be used to define ATENA monitors. These are special types of conditions that do not affect the analysis results. They are merely used to monitor certain quantities during the analysis. In this example, the following monitors will be specified:

- Maximal crack width
- Displacement at the point of load application
- Reaction at the point of load application

The definition process of the above conditions and monitors is described in Figure 2-20. The resulting assignment of the boundary conditions can be checked using the command **Draw | All Conditions | Exclude local axis**, which can be located at the bottom of the **Conditions** dialog. It should be noted that it is also possible to apply these conditions directly on the generated finite element model, but then the applied conditions are lost every time the mesh is regenerated.

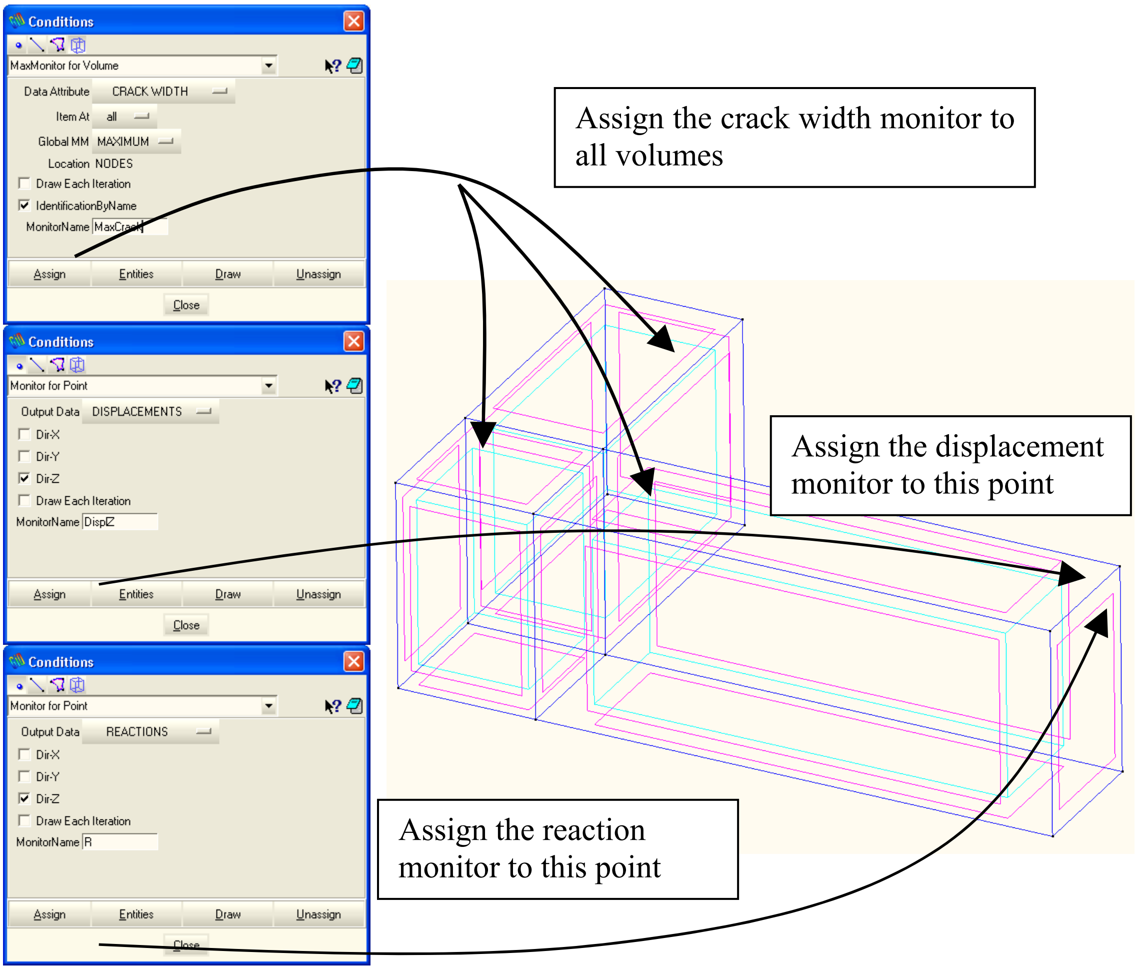


Figure 2-19: Definition of the ATENA monitors

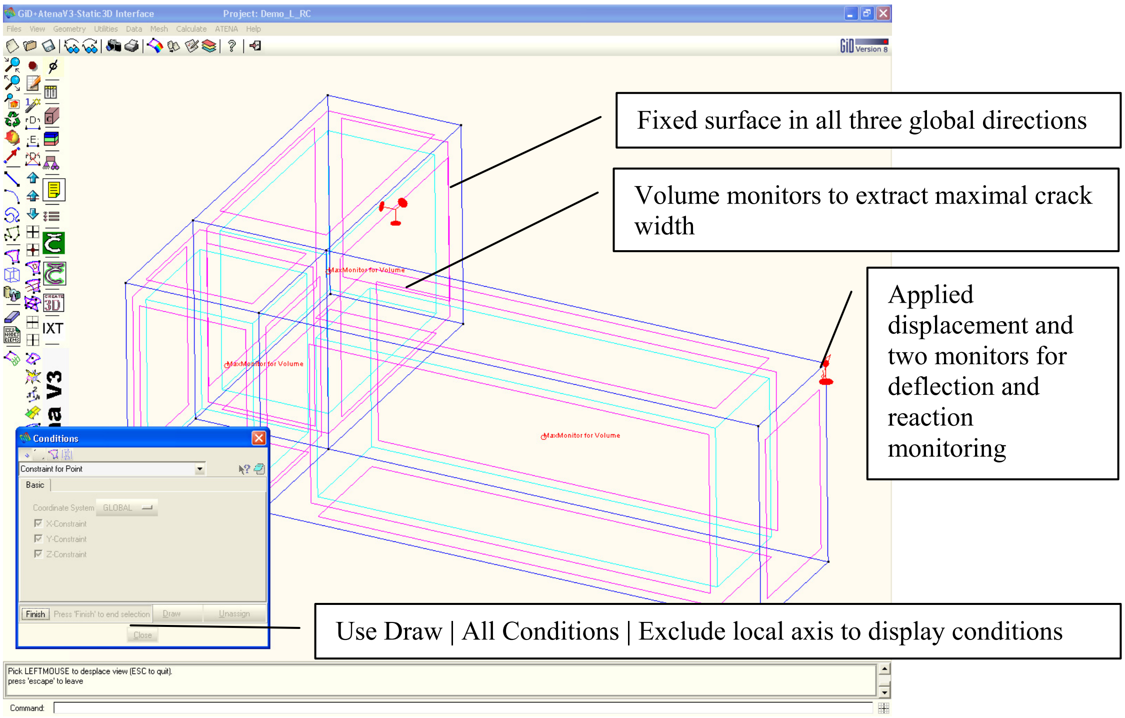


Figure 2-20: Display of assigned conditions

In certain cases, it may be advisable to manually identify which line entities represent reinforcement. By default, the ATENA-GiD interface attempts to treat all lines that are not connected to any surface or volume as reinforcement. This default behavior is activated by the corresponding check box in the **Problem Data** dialog. In certain cases, the automatic identification does not work properly. In this case, it is advisable to deactivate this default behavior, un-assign all reinforcement nodes and element identification, and then assign it again manually.

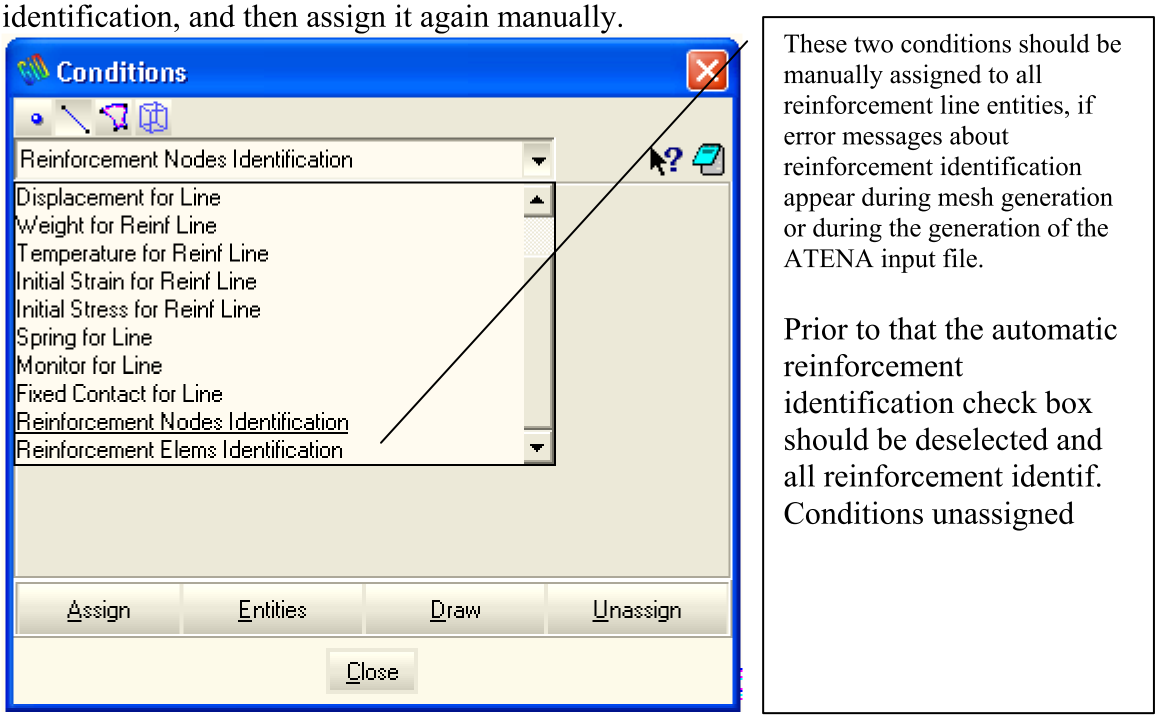


Figure 2-21: Manual identification of reinforcement nodes and elements

2.1.6 Meshing

In the preceding description, the geometry was defined and all properties (material, supports, loading) were assigned to the geometry. Now we shall generate a finite element mesh.

For this, we must set up appropriate parameters in the menu **Meshing**,

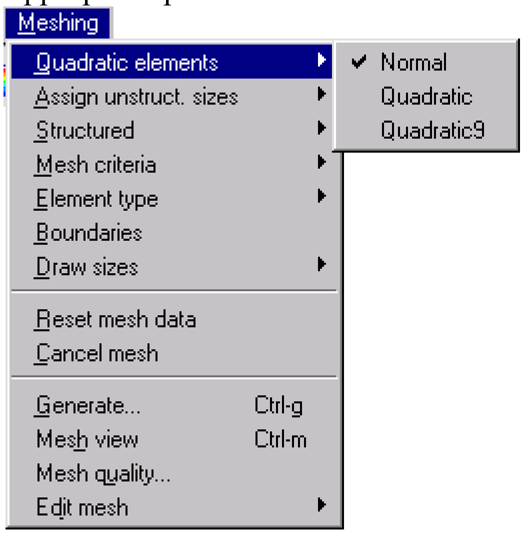


Figure 2-22: Meshing menu.

2.1.6.1 Mesh definition for volumes (concrete)

First, we shall deal with the meshing of volumes (concrete). There are many ways how to define the mesh. In this case, we use a simple method in which divisions on all lines are defined. If opposite lines have the same division, we can create a regular mesh.

- In the item **Quadratic elements,** we define low order elements by checking **Normal**.
- In **Structured,** we define division on all lines. It is always sufficient to select one line. **GiD** automatically assigns the same division to all opposite edges.
- In **Mesh criteria,** we select lines.
- In **Element types,** select **Hexahedra**.

2.1.6.2 Mesh definition for lines (reinforcement)

It is important to realize that lines of reinforcement in **GiD** serve only to export geometry to **ATENA**. The embedded reinforcement will be generated in **ATENA**. This means that we should make the line elements of reinforcement as large as possible. If we use division into a single element, then this single element is then passed to **ATENA** for the generation of the individual bar segments. Finding the intersections of the reinforcement bar with the solid elements generates the segments. In case the reinforcement in **GiD** is modeled using curved lines, then it is recommended to prescribe a certain division to finite elements such that the curved geometry of the bar is properly represented.

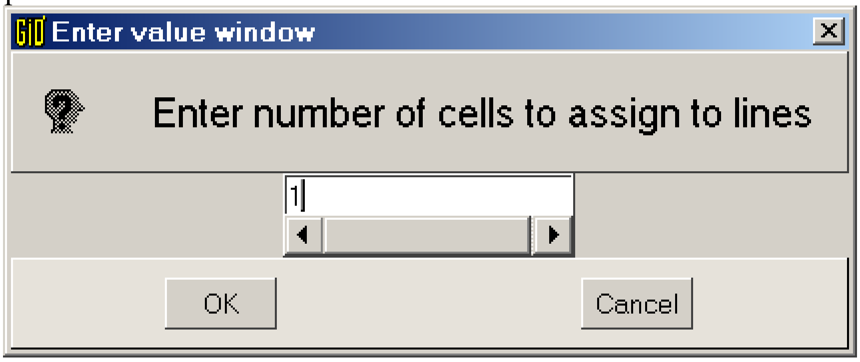


Figure 2-23: One division in lines of reinforcement.

- In the item **Quadratic elements,** we define low order elements by checking **Normal**.
- In **Structured,** define 1 division on lines, Figure 2-23
- In **Mesh criteria,** select **lines**.
- In **Element types,** select **Linear | Lines**.

2.1.6.3 Mesh generation

By selecting the item **Generate...** the mesh is automatically generated. The mesh can be inspected in the items **Mesh view**, **Mesh quality**. To change the mesh, the whole process can be repeated. **GiD** also allows changes by editing the mesh dimensions and properties.

2.1.6.4 Assign conditions to mesh nodes

Now, if needed, it is possible to assign additional conditions or materials directly to finite elements of nodes. Select **Data | Conditions** as shown in Figure 2-24. For this, we must select by mouse the node where the condition should be applied. It is, however, recommended to assign the material properties and boundary conditions on the geometric entities rather than on the mesh, otherwise, it is necessary to reassign such properties every time the mesh is regenerated.

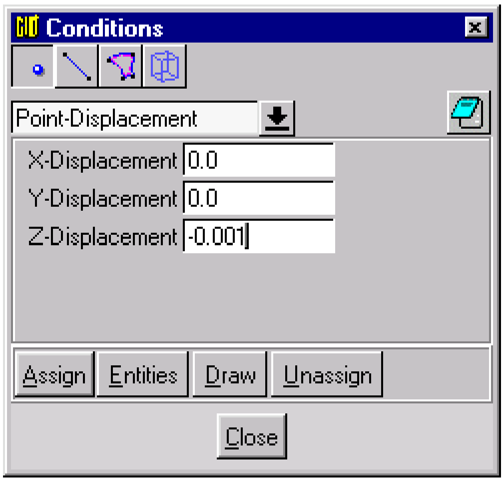


Figure 2-24: Assigning condition of point-displacement to a mesh node.

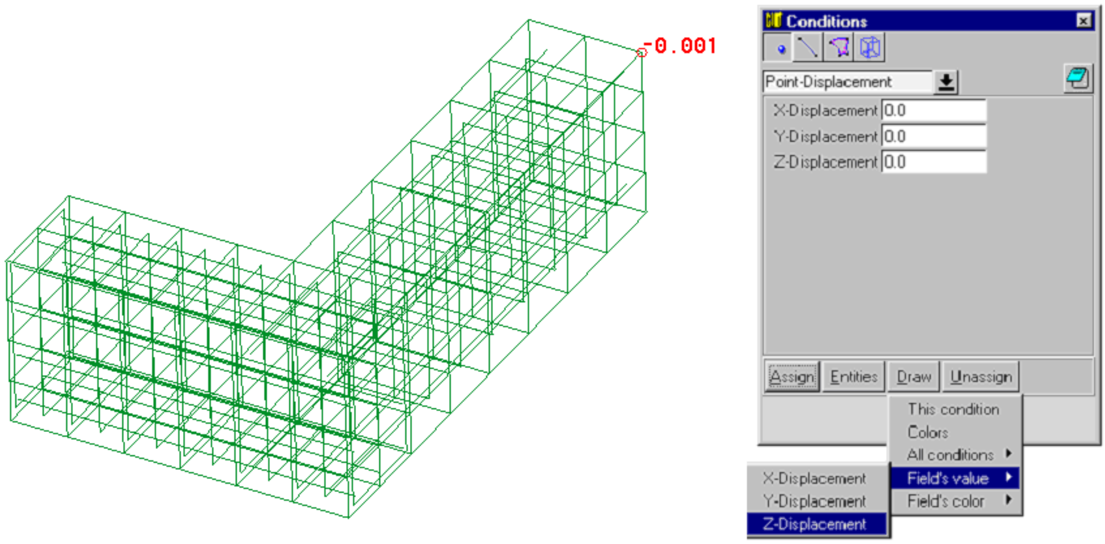


Figure 2-25: View and inspect a condition in a mesh node.

If we want to inspect the assigned values, we can do it by clicking on the button **Draw** and select **Field value | Z-Displacement**. Then the assigned condition value appears at the concerned node. See Figure 2-25.

2.1.7 Monitoring points

Analogically to Section 2.1.5, it is also possible to specify the monitoring points directly on the finite element mesh. The monitoring points are tools to record a structural response, for example, a load-displacement diagram. In **GiD,** we can for instance, specify only force and displacement monitoring at a mesh node. This is also done in **Conditions**. For applied force, we select **Force-Monitor**, for reaction force **Reaction-Monitor**, for nodal displacement **Displacement-Monitor**. The displacement component is selected by checking the appropriate box.

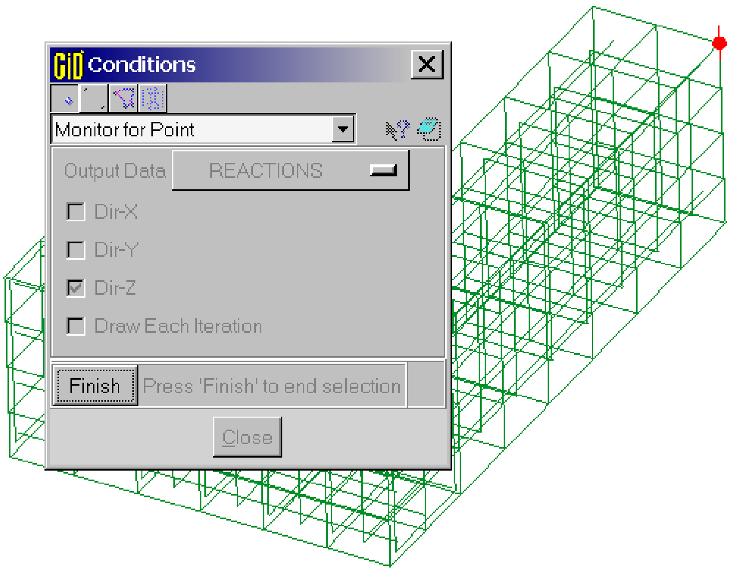


Figure 2-26: Definition of a monitor for reaction at node.

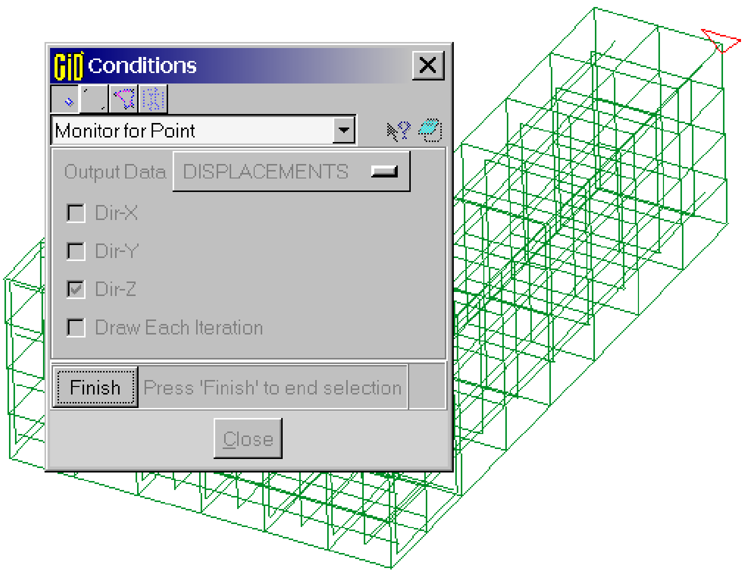


Figure 2-27: Definition of monitor for displacement at node.

Figure 2-26 and Figure 2-27 show the definition of force (reaction) and displacement monitors at a node. An inspection of monitors can be done by the command **Draw** in the same manner as in other conditions.
The monitoring points must be included within **Conditions** of the first load interval in **GiD**. Monitors included in other intervals will not be active in ATENA analysis.

2.1.8 Load history

For analysis in **ATENA** a load history as a sequence of load steps must be defined. The load steps can be proportional or non-proportional. In this example, the load history is simple. We define the first interval, which includes a set of conditions for supports at the fixed end and point-displacement. This can be checked and changed in the menu item **Data | Interval Data**. The next load steps can be done in two ways. The simplest way is to enter the number of repeated load steps and multipliers in the window of **Interval Data**, Figure 2-28, which is a proportional load history. In the case of a non-proportional history, for example, first, a vertical load followed by a horizontal load, we can use **Data | Interval Data**. Default settings of calculation method and global settings are in **Data | Problem Data**.

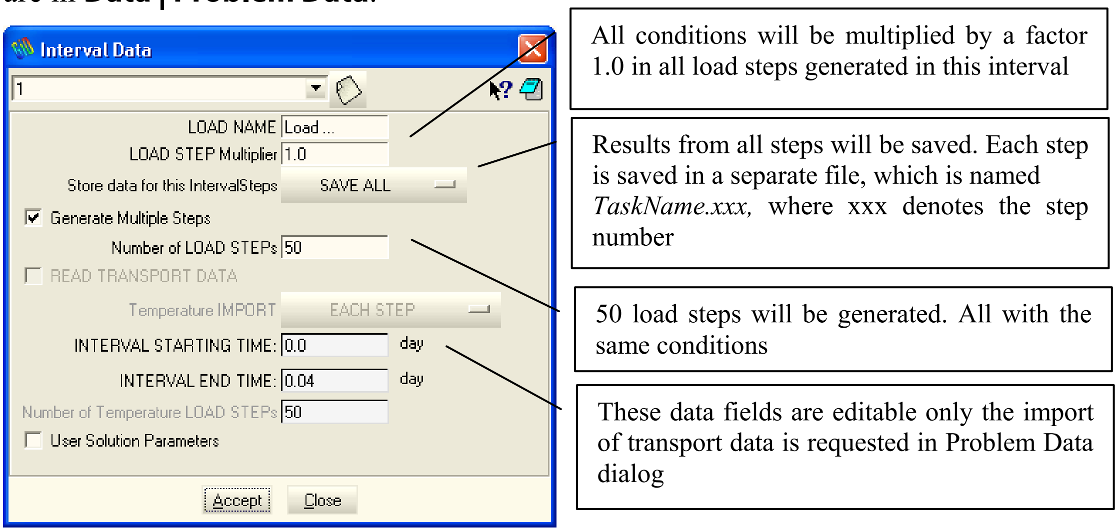
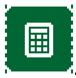


Figure 2-28: Interval data definition

2.1.9 Analysis and post-processing

The non-linear analysis is started by the menu item **Calculate** or icon . This causes the data from **GiD** to be written into an input file for **ATENA** (*.INP), and the program **AtenaStudio** is started.

During the execution of **AtenaStudio** variety of intermediate results can be viewed and inspected. The results of the analysis can be presented in the program **ATENA 3D**. It is necessary to import the binary result files (TaskName.xxx) from the required load steps into **ATENA 3D**. This is accomplished through the ATENA 3D menu **File | Open other | Results by step**.

For the operation of **AtenaStudio**, **ATENA 3D,** or any other details of ATENA software, see the ATENA Documentation.

2.2 Tutorial for Construction Process

2.2.1 Introduction

The objective of this tutorial is to show how the graphical environment of **GiD** can be used to model the construction process. The finite element solution core of **ATENA** supports the possibility to add or remove groups of finite elements. This feature can be used to model the construction process in **GiD**. The ATENA-GiD extension of the GiD graphical environment includes direct support for this feature. This feature can be modeled using the conditions for surface, and it will be demonstrated in this manual on the example of a tunnel (see Figure 2-29:). This example you can find at "%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.Static2D\TunnelWithConstructionProcess.gid"

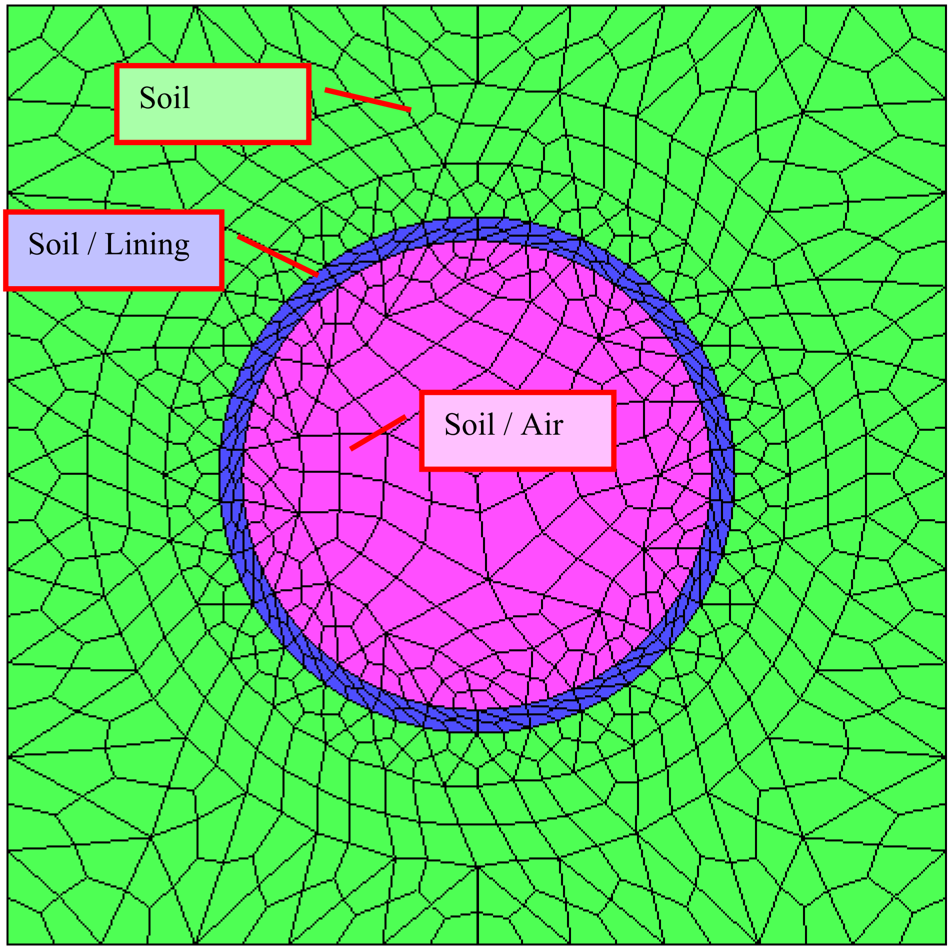


Figure 2-29: Model with three macro-elements.

The basic idea of the construction process modeling in **ATENA** is the following. It is possible to add or remove finite element groups at any time.

2.2.2 Geometry, boundary conditions, and load

We need to analyze a structure of a tunnel. Around the tunnel, there is a concrete lining. Boundary conditions are seen in Figure 2-30.

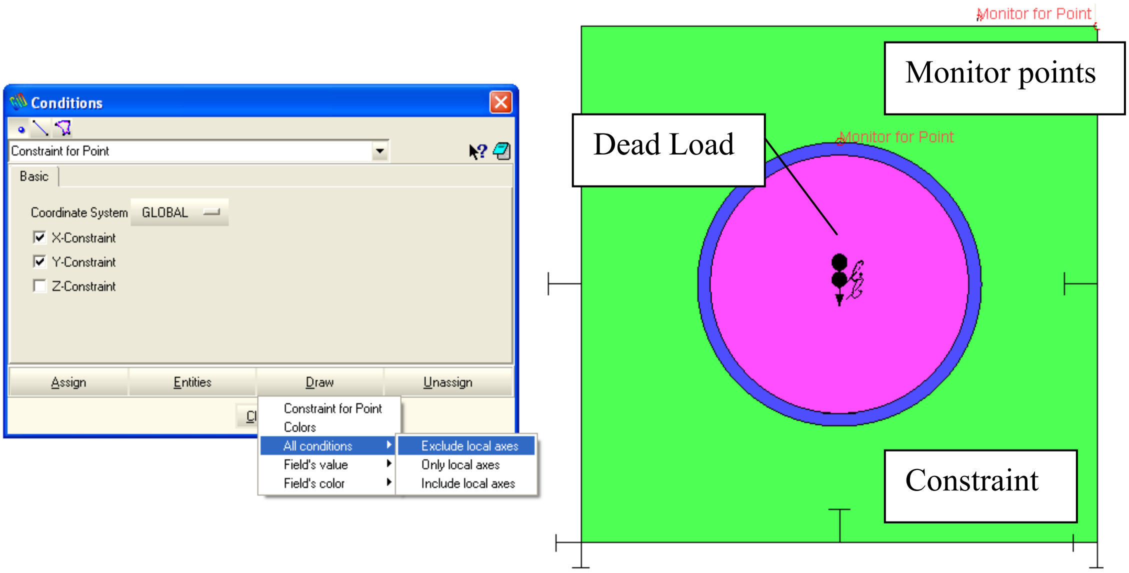


Figure 2-30: Draw all conditions on model.

The construction should proceed as follows:

1. excavation of a circular hole in the soil
2. adding lining (ring)
3. adding load

First, it is necessary to construct the model of the whole structure. Three separate macro-elements will be created for all four intervals. **For each of these macro-elements, it is necessary to have one separate material.**

Interval 1: this interval is used to define the basic boundary conditions to support the model from the bottom and both sides.
Interval 2: this interval is used for the excavation of a circular hole in the soil by deleting two centered macroelements.
Interval 3: this interval is used to add lining (ring shape) with concrete material characteristics around the hole.
Interval 4: this interval is used to add load to the top face of the model.

In the beginning, the whole area consists of soil; however, we must define separate macroelements for future changes (soil, lining, air). We assign the soil material to all these macroelements for the first interval (Figure 2-31:). The additional intervals will be needed for the subsequent phases of the construction process.

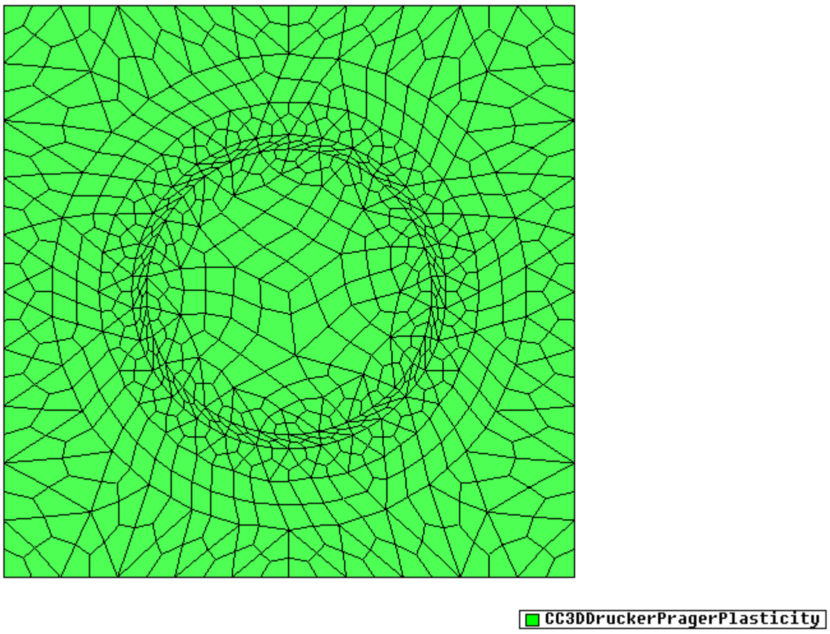


Figure 2-31: Material for interval 1

In the next step (excavation), we need to remove both circles from the center. It can be made using conditions for surfaces. In menu **Data | Interval | Current,** we switch to interval No.2, which we want to edit (Figure 2-32:). In menu **Data | Conditions | Conditions for surface,** we choose **Elements Activity for Surface** and select **Construction (Elements Activity): DELETE** (Figure 2-33). Next, we can **Assign** areas which we want to excavate (Figure 2-34). We can draw all macroelements which have assigned some conditions by choosing **Draw | Colors** (Figure 2-34).

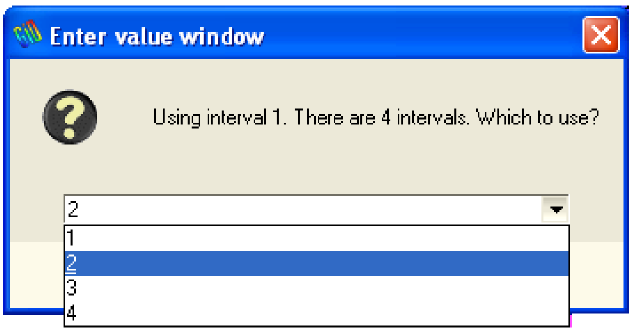


Figure 2-32: Switching current interval.

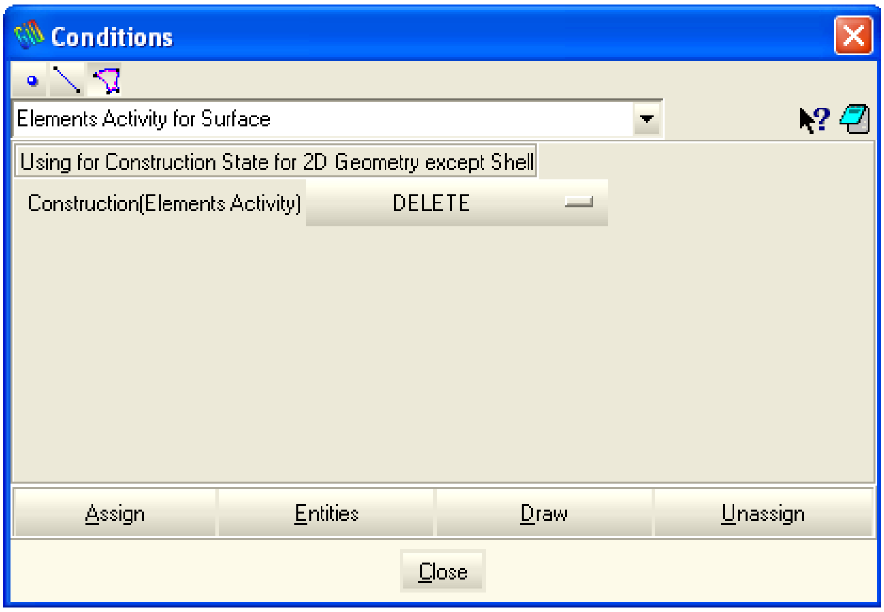


Figure 2-33: Conditions for surfaces

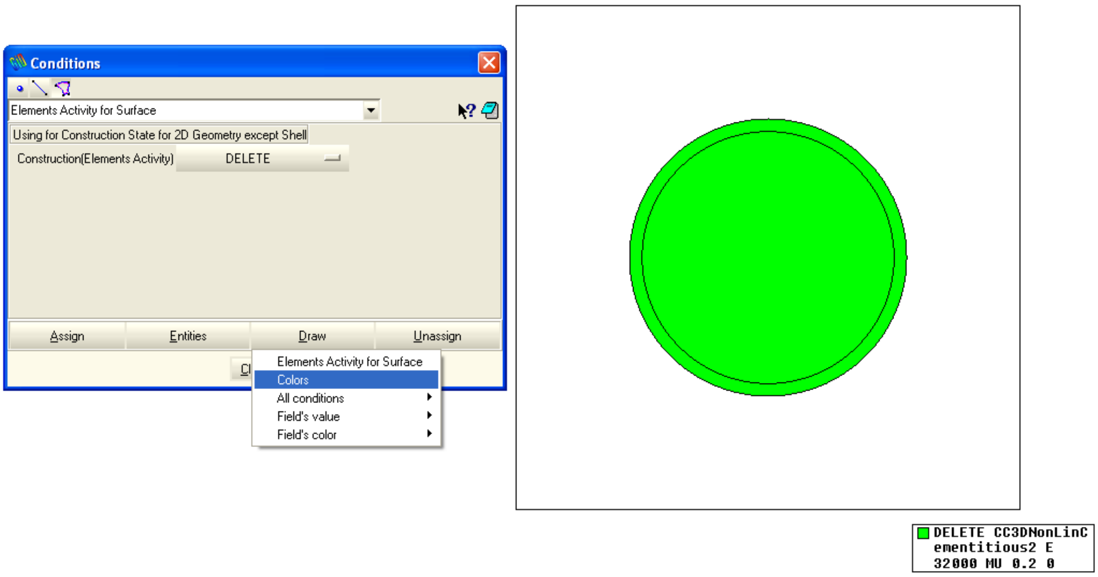


Figure 2-34: Deleting materials in interval 2

In the next step, we need to create the lining with non-linear concrete material. We switch the current interval to No.3. In menu **Data | Conditions | Conditions for surface,** we choose **Elements Activity for Surface** and select **Construction (Elements Activity): CREATE WITH NEW MAT** (Figure 2-35), and choose the **CC3DNonLinCementitious2** material. We can create specific material for this case and assign it to surfaces, which we want to create (Figure 2-36).

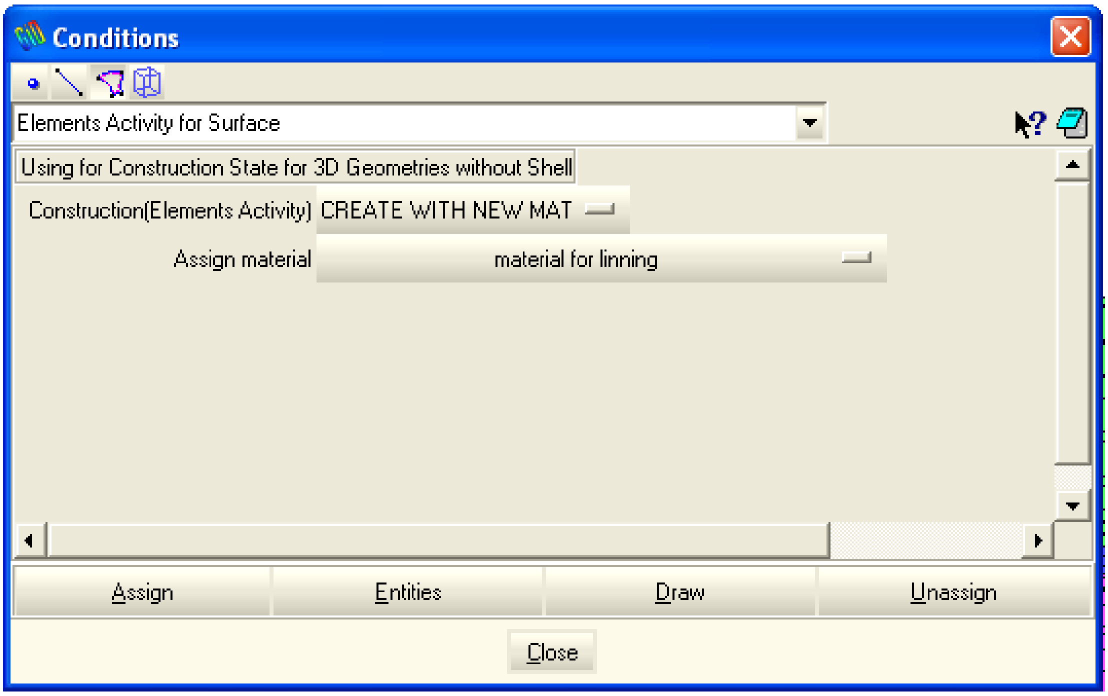


Figure 2-35: Condition for surface, create new material

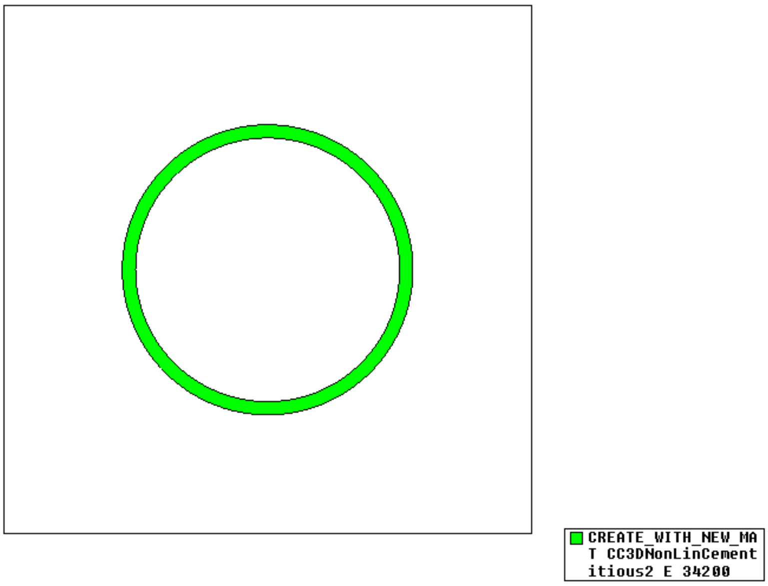
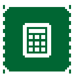


Figure 2-36: Creating concrete lining.

In the last step (Interval No.4), we will only add load to the top of the model (**Data | Conditions | Conditions for line**, **Load for line**).

2.2.3 Running analysis

Analysis can be run by selecting button in menu **Calculate | Calculate** or clicking on the button  “**ATENA Calculate**” and in **AtenaWin** by clicking on button  “**Execute**“.

2.3 Durability Analysis: Chloride-induced Reinforcement Corrosion

2.3.1 Introduction

Corrosion of the reinforcement bars represents one of the most dangerous phenomena affecting the service life of reinforced concrete structures. Chlorides from seawater or de-icing agents may over the years penetrate the concrete microstructure, disrupt the passive oxide film on the surface of the reinforcement bars and at this moment initiate the reinforcement corrosion. The reduced reinforcement cross-section together with the corrosion-induced cracks in the concrete cover compromise the structural serviceability and can even lead to the collapse of the structure in the worst-case scenario.

Typically, a structure subjected to a chloride attack undergoes several stages during its life cycle, as shown in Figure 2-37. During the depassivation stage, the imposed chloride content at the surface of an element causes the transport of chlorides through the concrete porous system towards the embedded reinforcement bars. This process continues until the critical chloride content is reached at a depth of the reinforcement bar, which corresponds to the moment of the vanishing of the protective passive film on the surface of the reinforcement bar. Since this moment, the reinforcement cross-sectional area starts to decrease as a result of the corrosion processes, and the propagation stage begins. Moreover, the larger volume of the corrosion products compared to the initial volume of the steel causes a build-up of the internal stress in the concrete cover, which, once exceeding the tensile stress of the concrete, results in cracks formation. The corrosion cracks further grow and ultimately cause spalling of the concrete cover exposing the reinforcement bar to the environmental conditions at the site of the structure.

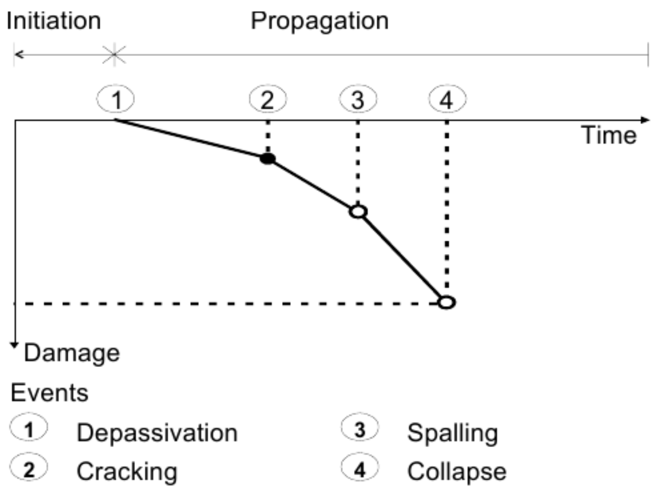


Figure 2-37 Stages during deterioration of a structure.

The problem showed in this example manual demonstrates the general approach towards a durability analysis using ATENA and explains how to set the details of the problem using GiD. At this moment, it is expected that the user already has an understanding of the general static analysis as well as is familiar with the GiD environment.

2.3.1.1 Concept of durability analysis

A general concept of a durability analysis is shown in Figure 2-38. It consists of the following intervals:

1) Application of the self-weight
2) Application of the characteristic load
3) Deterioration of the structure
4) Determination of the ultimate load-bearing capacity

In the first interval, typically, the self-weight and constraints of the model are applied. If the static scheme of the model does not change between different intervals, the constraints can be easily copied to another interval as will be shown later in this example. Next, since the deterioration rate is typically accelerated by the cracks present in the material, a typical service load is applied to the model. Based on that, ATENA calculates crack size distribution in the element, which will be used in the next interval during the chloride ingress (or possible concrete carbonation). During the interval when the deterioration mechanism is active, the chloride content concentration within the concrete material increases due to the high chloride concentration applied at the boundary of the model. If the critical chloride concentration is reached, the corrosion rate is computed, and the corresponding reduced reinforcement area is obtained. The reduction of the reinforcement then affects the structural model and typically increases the deflection compared to the previous interval.

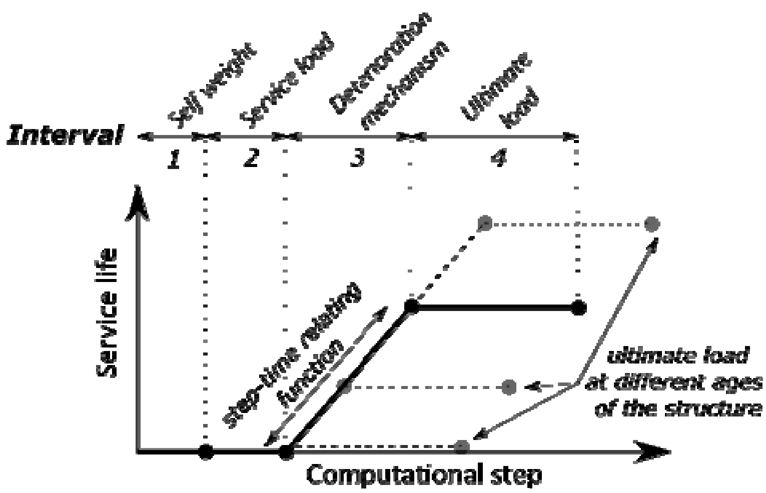


Figure 2-38 A general concept of a durability analysis. The analysis is typically done in four intervals, and the ultimate load-bearing capacity at different moments throughout the service life is obtained by varying the duration of the deterioration phenomenon.

By variation of the length of the step for which the deterioration phenomenon reduces the structural performance, the ultimate load-bearing capacity at a different moment of the life cycle can be obtained. Finally, in the fourth interval, the model is loaded until its failure to get the ultimate load-bearing capacity.

The relationship between time and computational step is given by a step-time relating function generated at the moment of application of the chloride boundary condition. It should be noted that the time dimension is only important for the duration of the deterioration phenomenon and has no relation nor impact on the static intervals of the analysis.

2.3.1.2 Problem overview

We show the chloride deterioration at the example of the structure of the Nougawa bridge in Japan. The tree-span structure of the total length of 131 meters was constructed in 1930 in a coasted area. In 2009, the bridge was experimentally investigated to obtain its ultimate load-bearing capacity and the chloride concentration in the concrete. For more details, the reader of this example is referred to the following paper [12]. In this example, we model an 11-meter-long section of the bridge. The model represents a standard three-point bending test. The geometry of the model is installed with ATENA and can be found in the following directory:
Public%**Documents\ATENA Examples\Science\GiD\Tutorial.Static3D\BeamWithChlorides_60years.gid**

2.3.2 Analysis settings

2.3.2.1 Intervals

As explained earlier, the analysis will be composed of four intervals. The input settings of each interval are shown in figure Figure 2-39. The boundary conditions need to be specified for each interval. The boundary conditions, in general, describe the constraints of the model, such as the supports and the fixed contacts between different parts of the model and the loads acting at the model. The intervals are defined through **Data | Interval data**.

As for other types of non-linear analyses, the loads in each interval should be applied to the model in multiple steps. The application of the self-weight (intervals 1), the load representing the general service load (intervals 2), and the load for determining the maximum load-bearing capacity (interval 4) are applied in 10, 20, and 50 steps, respectively. Similarly, the chloride ingress is not computed in a single step, but the 60-years duration of the process is calculated in 20 steps.

There is a difference in the settings of the number of steps for an interval with a deterioration mechanism and a general mechanical load. The prescribed mechanical loads are divided by the number of load steps, and this proportional part is applied at each load step; therefore, the full value of the prescribed load is reached at the last step of the interval. This approach is not suitable for the chloride boundary condition, where the surface chloride concentration should remain constant and equal to the prescribed value during the entire interval. Therefore, for the interval with chloride load, the option "**Active Step multiplier**" should be activated with "**Step multiplier value"** set to 1 and "**Interval Multiplier**" set to the same value as the number of steps ("**Number of Load Steps**").

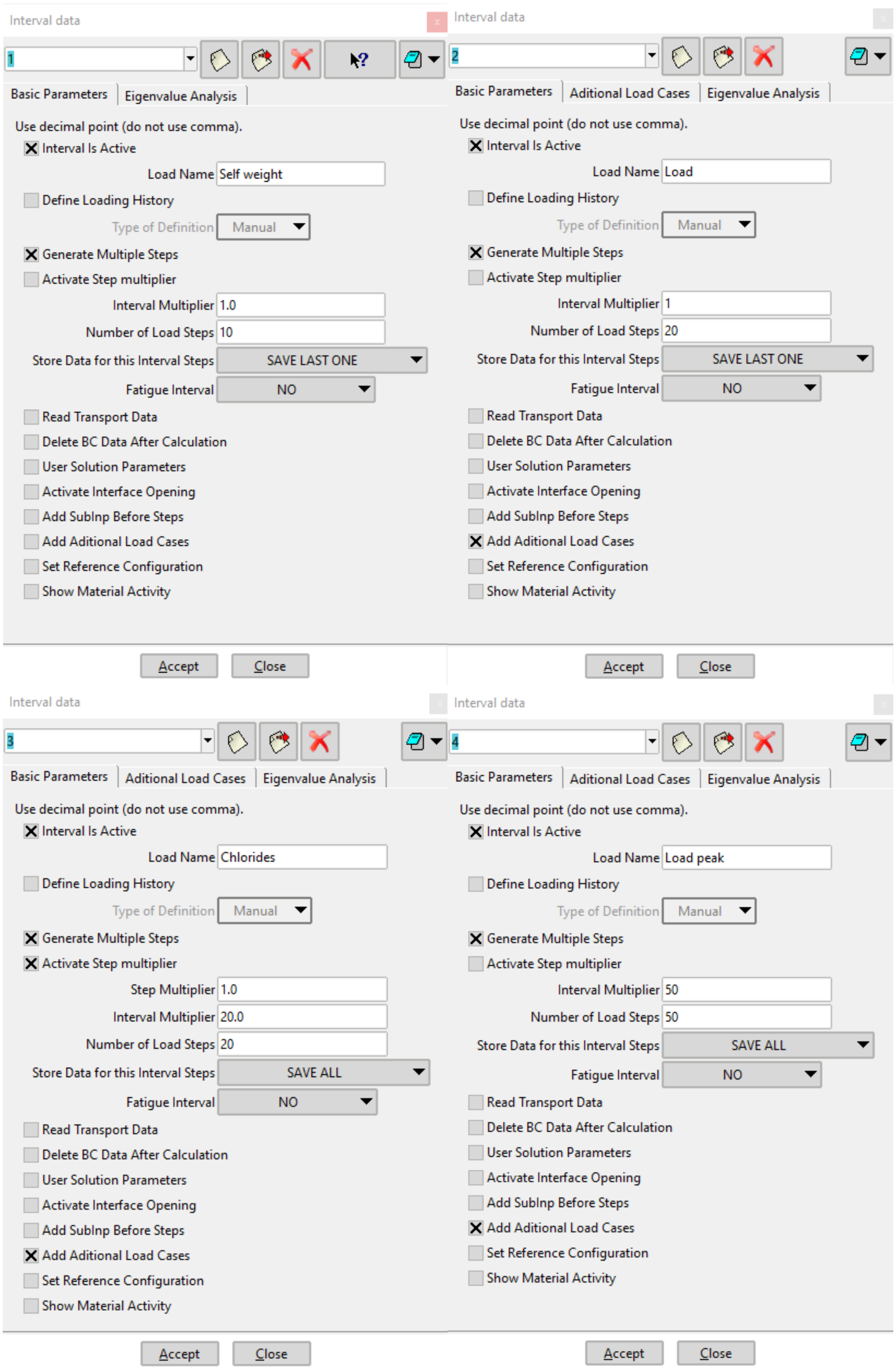


Figure 2-39 Interval settings for the analysis.

The need to repeatedly specify the same boundary conditions (i.e., the boundary conditions, which do not change during the analysis) at different intervals can be overcome by activating **"Add Additional Load Cases"** option in the **"Interval data"** dialogue, as shown in Figure 2-40. By this option, the conditions applied previously to the model can be addressed. The interval where the conditions were previously specified is chosen by writing its number in the "**id**" column. In this case, we select interval number 1, where the supports and master-slave conditions for surfaces are specified. In the "**multiplier**" column, a factor for scaling of the applied loads is given. It should be noted that 0 represents a factor for multiplication of the load in the interval; however, it has no impact on the boundary conditions of supports or master-slave conditions. In this example, the self-weight applied to the model in interval 1 is not used as a load again, but the supports and master-slave conditions between the load plates and the beam are still acting at the model. Then the range of computational steps where the conditions from the addressed interval are active is specified in "**from**" and "**to**" columns. In the case of creep analysis, the additional adjustment of the computational stepping is possible in the "**creep fixed**" column.

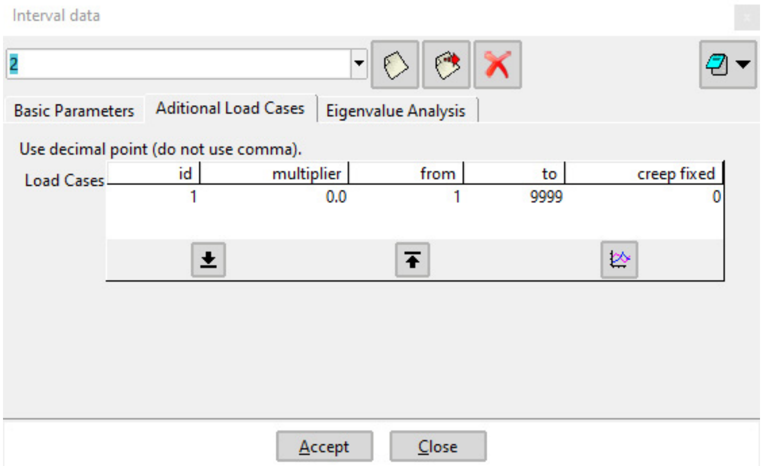


Figure 2-40 Dialogue window for activation of the conditions already specified in a different interval. It should be noted that the 0 multiplier only affects the loads but has no impact on the supports or master-slave conditions.

2.3.2.2 Inputs for chloride ingress calculation

The chlorides act at the model in the third internal. The interval is selected through **Data | Interval data** and selecting its number from the drop-down list. Once this interval is activated by clicking the "**Accept**" button, new boundary conditions can be defined as well as the boundary conditions already prescribed can be checked. The chloride load is applied as a boundary condition at the surface of the model. The input is open through selecting **Data | Conditions**, choosing  tab and selecting **"Chlorides for Surface"** from the drop-down list. By selecting **Draw | Colors**, the applied load can be displayed. The conditions, which has been already used in the model, can be checked by selecting **Entities | Chlorides for Surface**, and copied to the boundary condition window by selecting "**Transfer**" button

The values needed for the calculation can be generally divided into material-related, environmental-related, and computational-related inputs. The material-related parameters describe the properties of concrete, such as its diffusivity for chlorides, strength, or critical chloride concentration for steel depassivation. These values can usually be obtained from the structural design, experimentally or, for typical mixtures, from literature data and code recommendations. The environmental-related parameters, such as chloride content at the surface, decay rate, and rate of corrosion after spalling, describe the conditions at the site of the structure and for typical environments are summarized in the DuraCrete report [9]. Finally, several parameters are related to the computation in the program. All the input values used in this example are shown in Figure 2-41, and their explanations together with the references given in Table 2.3-1. Furthermore, it should be noted that the details of the solution of the transport problem and the description of the implemented corrosion model can be found in the ATENA Theory Manual [1]

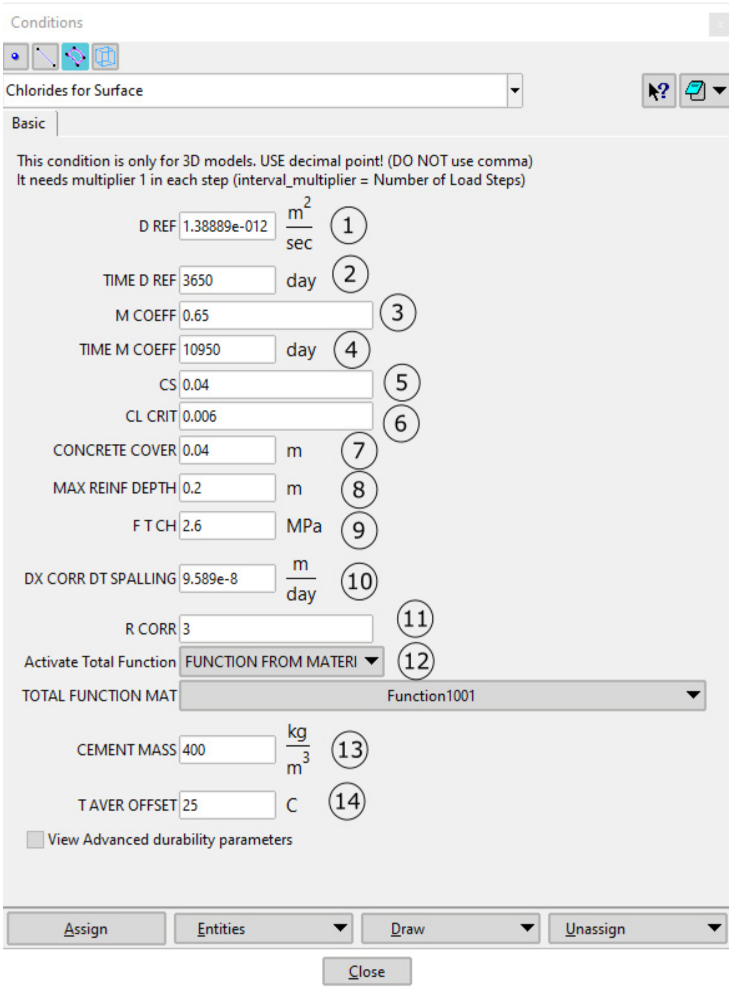


Figure 2-41 Definition of the surface boundary condition for chloride attack. All parameters used in this example are summarised and commented inTable 2.3-1. Please note that the “CONCRETE COVER” parameter differs for horizontal and vertical/inclined surfaces due to the structural design of the element.

Table 2.3-1 Summary of applied input parameters for chloride ingress.

	Input Name	Value and Source	Comment
①	Reference diffusion coefficient of chloride transport	interpolation for D_a from data in chapter 6 of ATENA Theory manual [1] for a given cement type and water/binder ratio, and applying: $D_{ref} = D_a(1\text{-}m) = 1.39{\cdot}10^{-12}\ m^2/s$	Parameter influences the rate of chloride transport through the concrete microstructure.
②	Reference time for diffusion coefficient:	$t_{ref} = 3650\ days$ (for data available in chapter 6 of ATENA Theory Manual [1])	Parameter depending on the time when the diffusion coefficient was measured.
③	Decay rate (time factor)	$m = 0.65$ (based on Table 8.6 of DuraCrete [9])	Parameter depends on the exposure conditions and cement type.
④	Lower bound for constant diffusion coefficient	$t_R = 10\ 950\ days$	It is usually assumed that the reference diffusion coefficient remains constant after 30 years.
⑤	Surface chloride content	$C_s = 0.04\ g_{Cl}/g_{binder}$ (determined experimentally or based on Table 8.6 DuraCrete [9])	Value which defines the boundary condition of the diffusion problem and depends on cement type and exposure condition.
⑥	Critical chloride concentration	$C_{crit} = 0.006\ g_{Cl}/g_{binder}$ (value recommended by fib)	Critical chloride concentration for reinforcement depassivation (in terms of mass of chloride per mass of binder).
⑦	Concrete cover	*0.04 m* for horizontal and *0.09 m* for vertical/inclined surfaces (based on structural design)	Post processing variable for reviewing of the chloride content at the depth of the reinforcement.
⑧	Maximum reinforcement depth	*0.2 m* (should be greater or equal to the distance of the reinforcement from surface)	The maximum depth from the surface of the model for reinforcement detection.
⑨	Characteristic tensile splitting strength	$f_{t,ch} = 2.6\ MPa$ (can be estimated based on the characteristic compressive strength)	Determines the resilience of concrete cover against cracking due to corrosion-induced rebar expansion.
⑩	Corrosion rate after spalling	$dx_{corr}/dt_{spalling} = 9.6{\cdot}10^{-8}\ m/day = 35\ \mu m/year$ (based on EN ISO 9223 [10])	The corrosion rate under environmental exposure conditions used after spalling.
⑪	Pitting factor	$R_{corr} = 3$ (for chloride-induce corrosion [11])	Parameter describing how localized is the corrosion (depending on the type of corrosion).
⑫	Total function	time-step relating function (explained later in the example)	A function generated to relate the computation step with the duration of the chloride attack.
⑬	Cement mass	$m_{cem} = 400\ kg/m^3$ (based on mixture design)	Mass of the cement in 1 m^3 of concrete.
⑭	Average temperature	$T_{aver} = 25\ °C$ (average environmental temperature at the site)	Temperature in the material used for calculation of the corrosion rate.

The boundary conditions defining the chloride attack are applied at 7 surfaces at the bottom of the model. Due to the structural design, the concrete cover differs at different parts of the beam. The distance of stirrups from the vertical surfaces is 90 mm, while the concrete cover from at the horizontal surfaces is only 40 mm. Therefore, it is desirable to apply the chloride boundary condition with different values of the **"CONCRETE COVER"** parameter in order to check the results chloride concentration at the depth corresponding to the depth of reinforcement during post-processing.

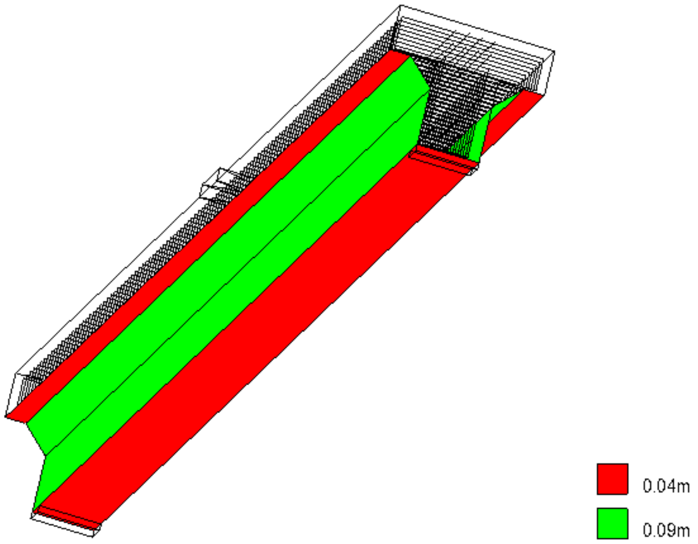


Figure 2-42 Color plot of the concrete cover used in the model.

As shown in Figure 2-42, the parameter of each boundary condition can be checked by plotting it on the model either by **Draw | Field's Value** or by **Draw | Field's Value** and selecting the parameter of interest.

2.3.2.3 Time-step relation and calculation

The relationship between the computational step and the duration of the chloride attack is specified using the function previously generated during the application of the boundary condition. It can be accessed through **Data | Materials | Functions**, and its dialogue window is shown in Figure 2-43. The relationship is defined in a table-like manner, where the computational step is specified in column "**X**" and the time is listed in column "**Y**." The program uses a linear function to interpolate between the given values.

In the case of this example, interval 2 ends at step number 30, which corresponds to the time 0 days of the chloride ingress. The interval, when chlorides are applied (interval 3), is divided into 20 steps, and the time is specified as 21900 days (60 years) by the end of the interval. Therefore, each step corresponds to 21900/20 = 1095 days (3 years).

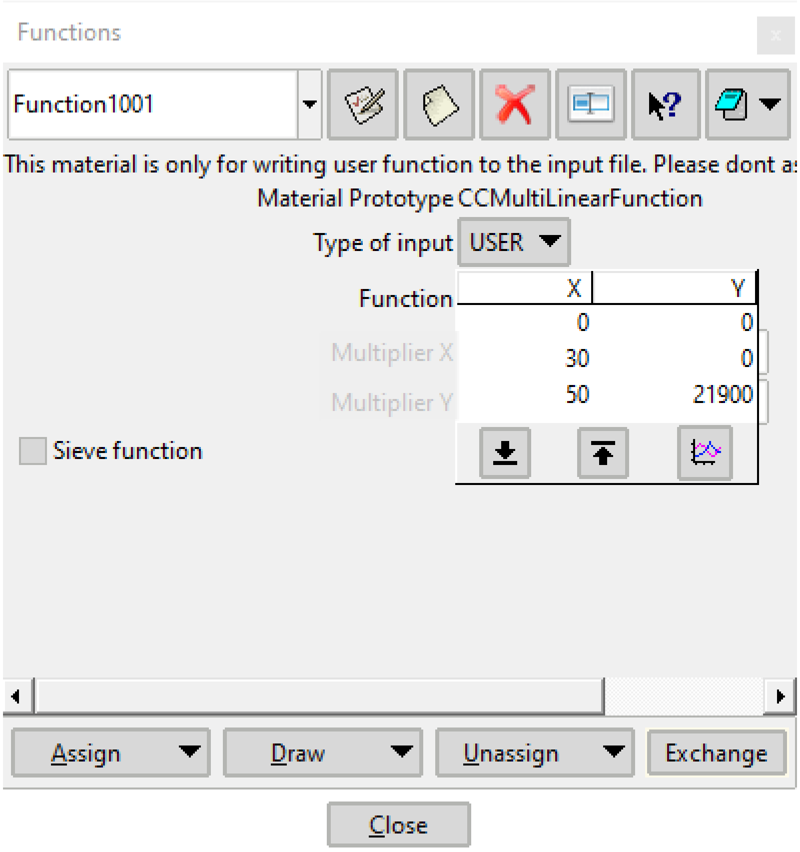


Figure 2-43 Specification of the function which relates the computational step with the duration of the chloride ingress.

By defining the function relating the time with the computational step, the necessary information is given to the model. Finally, the structured mesh is generated through **Mesh | Generate mesh** using the settings already defined in the problem. The solution in ATENA Studio is then started by clicking  icon.

2.3.3 Results

The relationship between the mid-span deflection and the applied load is shown in Figure 2-44. It can be observed that the elastic capacity of the element is reached shortly before the self-weight of the material is fully applied to the model (end of interval 1, step 10). Next, in interval 2 (steps 10 – 30), the deflection linearly increases with the applied service load. The chloride ingress (interval 3) is modeled from step 30 to step 50. Even though no additional load is applied to the model in this interval, the deflection increases as a result of chloride-induced corrosion of reinforcement bars. Finally, since step 50, interval 4 begins where the maximum load-bearing capacity of the corrosion-deteriorated beam is determined. Based on this chart, we can roughly conclude that the 60-years-long chloride attacked caused an increase in the deflection of approximately 39 %, and the ultimate load-bearing capacity of the deteriorated is approximately 208 % of the service load. Furthermore, a decrease in the stiffness of the model can also be observed before and after the modeled chloride attack.

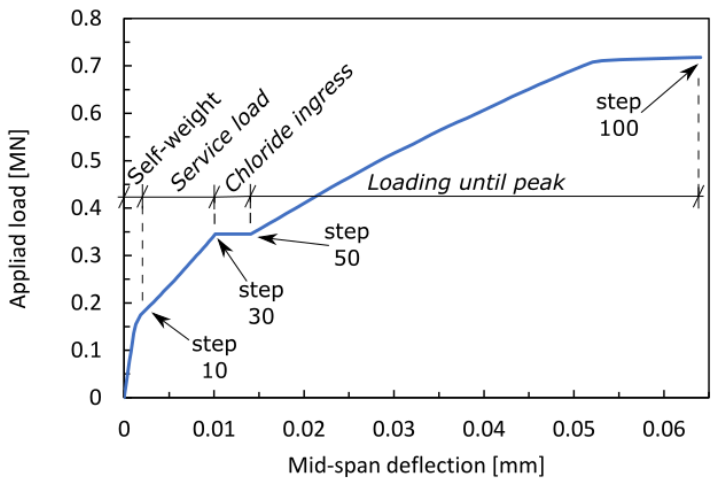


Figure 2-44 Load-deflection curve obtained from analysis with different intervals labeled in the graph.

The comparison of the model before and after the chloride attack is shown in Figure 2-45. Due to the larger number of cracks induced by the service load in the mid-span area of the model, the chloride ingress is accelerated in this region resulting in higher chloride concentration compared to the regions closer to the supporting plates. This naturally speeds-up depassivation of the reinforcement and increases the total reinforcement corrosion after 60 years in the center part of the beam. It can be observed that the edges of the stirrups are corroded up to 91 % in the center region of the beam, and this reduction of the cross-sectional area increases the stress, which they transfer. Also, comparing the models, we can see new mechanical cracks, which developed in concrete due to the chloride attack.

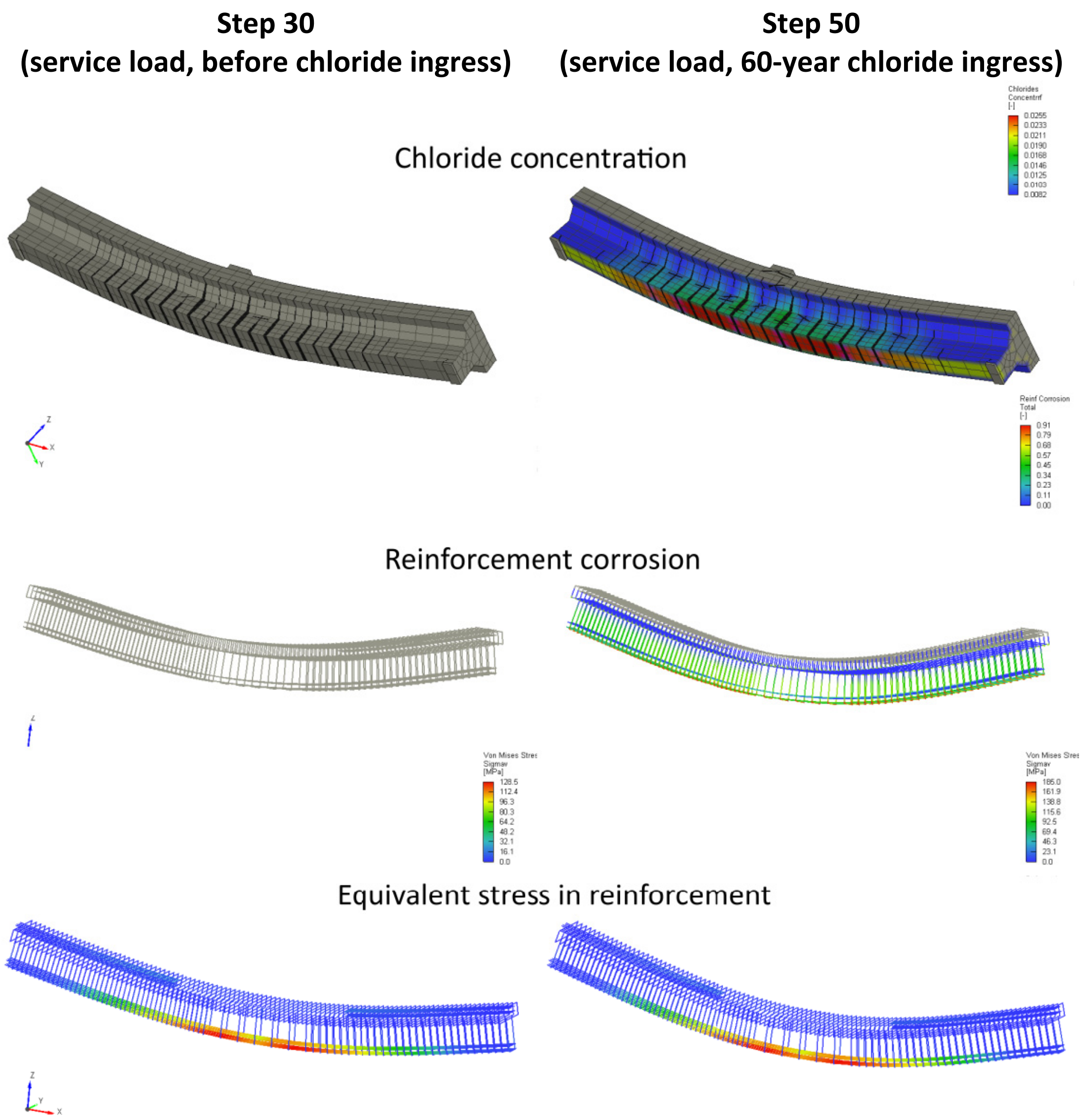


Figure 2-45 Comparison of the results before the chloride ingress (step 30) and after 60-years-long chloride attack. Please notice how the mechanical cracks increase the chloride concentration at the bottom of the beam and how the corrosion increases the stress in reinforcement and the deflection of the beam.

Finally, we show a possible outcome of a durability analysis in Figure 2-46. In this case, the duration of the chloride attack was varied to obtain the load-deflection curves for three models; model not affected by the chloride-induced corrosion and for models subjected to 30-years and 60-years chloride attack. Based on such data, the decrease in the ultimate-load bearing capacity of the structure can be predicted.

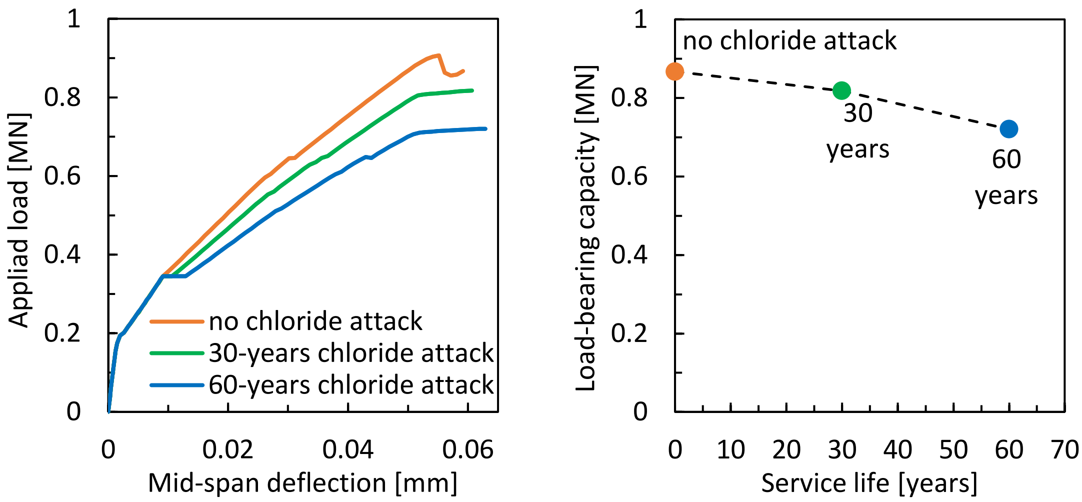


Figure 2-46 (left) Comparison of load-deflection curves for models not subjected to chloride attack with models subjected to 30-years and 60-years of chloride attack, and (right) development of the ultimate load-bearing capacity during the service life of the structure.

2.4 Modeling with 1D Beam and 2D Shell Elements

In this section, we show an example where different types of elements are combined to model the structure. In all cases, the geometry represents a simple frame; however, either 1D or 2D entities are used for modeling the beam and columns. Since the meshes of different geometrical entities do not always share common nodes, they need to be connected with boundary conditions, which will ensure the proper constraints of the model.

2.4.1 Frame using 1D beams

2.4.1.1 Pre-processing

This example shows the frame modeled with 1D beams only. It can be found in the following directory: %public%**\Documents\ATENA\Examples\Science\GiD\Tutorial. Static3D_Frame_examples\Frame1DBamFixedHinge.gid** .
The model is loaded with a distributed load applied to the full length of the beam. It should be noted that if the beam is modeled with multiple lines, which share a common point in the corners, a fixed connection will be automatically generated there. However, if any degree of freedom needs to be released at the connection (e.g., for modeling a hinge), the geometry needs to be created using separate lines (i.e., lines which do not share common) points, and those lines will be subsequently connected with the desired type of connection. For the sake of simplicity, the problem is modeled elastically. The material is defined through **Data | Materials | SOLID Elastic**, and the input is shown in Figure 2-47.

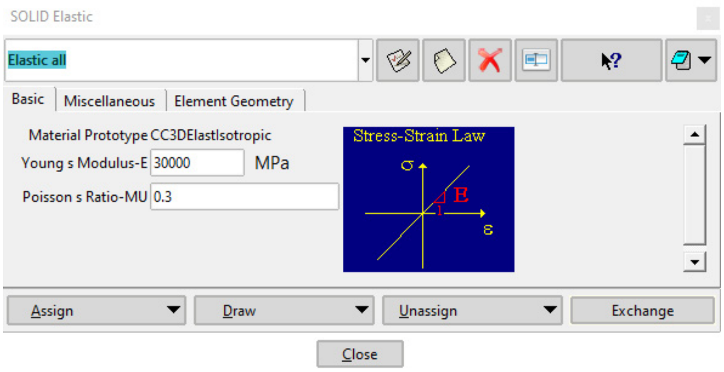


Figure 2-47 Definition of material.

In the next step, the material is assigned to two different beam elements, which will be used of the beam and columns. This is done through **Data | Materials | BEAM Concrete**. All parameters can be seen in Figure 2-48. The cross-sectional dimensions are set as 0.2 x 0.2 m and 0.5 x 0.2 m for the beam and columns, respectively. The beam and columns also differ in the definition of the local coordinate system. It is advisable to match the orientation of the bending axis to conveniently display the bending moments during post-processing (i.e., y-axes of the columns and the beam should have the same orientation; thus, M_y will show the bending moment for the whole model.). Same for the beam and columns, the material is chosen through “**Use Base Material**” as

"**Elastic_all**", which will assign the previously-defined material to the 1D beam elements. Finally, we choose the "**1D Beam**" idealization.

Columns **Beam**

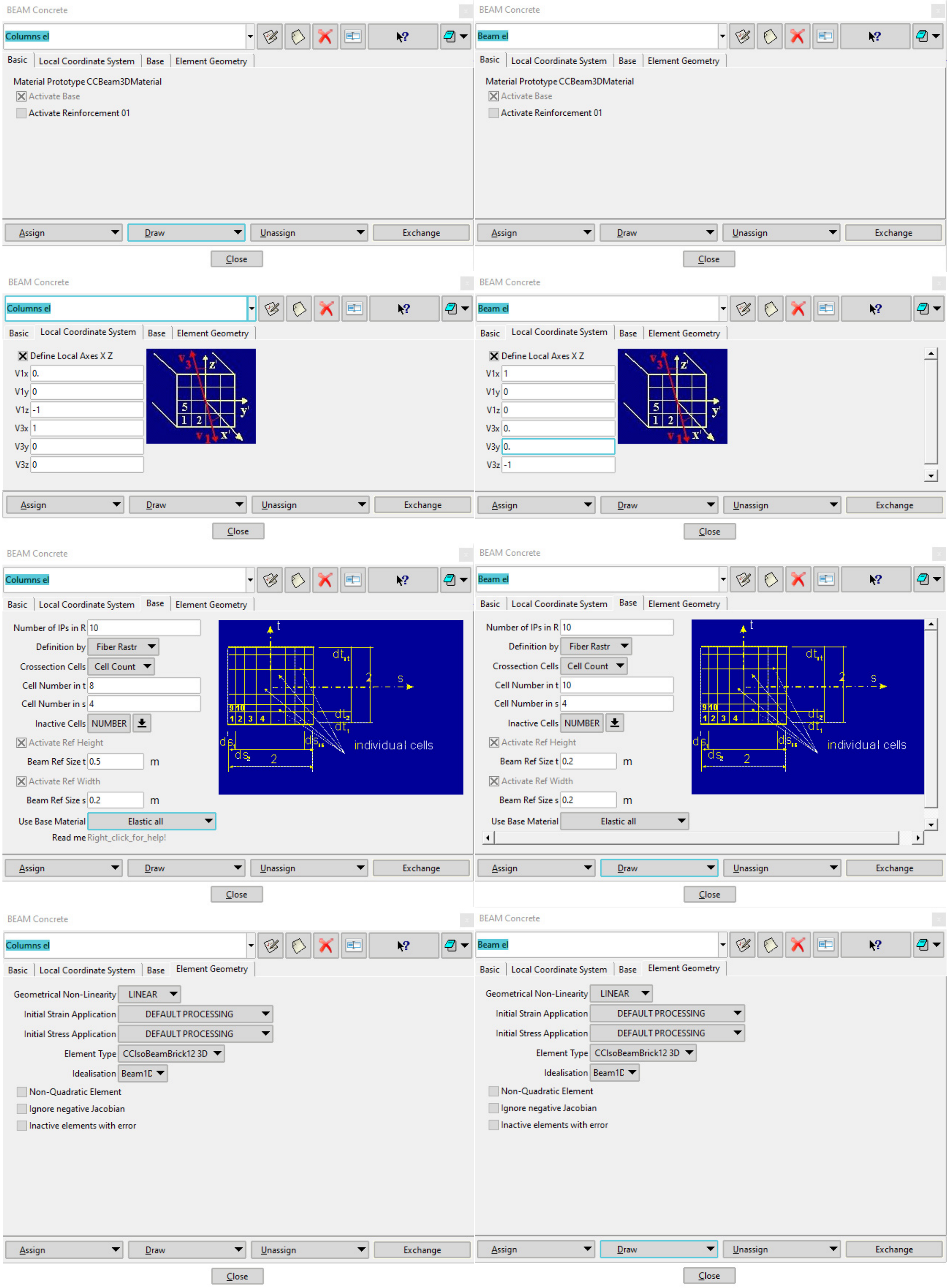


Figure 2-48 Definition of 1D BEAM elements for beam and columns.

Once all parameters are set, the 1D beam properties are assigned to the lines representing the beam and columns through "**Assign ǀ Lines.**"

At the next step, the lines are connected to create a frame. In the case of this example, one corner will represent a fixed connection (i.e., moment-resisting connection), and the other corner will represent a hinge connection, where the rotation degree of freedom around the y-axis will be released. The connection will be defined through **Data | Conditions**. Since the lines will be connected at points, the conditions are given for points, and  category is selected. For the definition of the fixed connection, "**Fixed contact for point**" option from the drop-down list is used. Regarding the "**Type of Cond**" option, the point belonging to one line (e.g., beam) will be defined as "**Master**", and the point belonging to the other line (e.g., column) will be defined as "**Slave**". The connection will be created no matter the master/slave choice, and the points can be defined vice versa in this case. The master-slave pair need to have the same "**ContactName**" to create a connection. It is recommended to assign the lines of the model to different layers and hide the layer, which is not used in the selection, to select the desired point easily.

In general, if it is necessary to allow displacement in a certain direction, the "**Fixed contact for…**" allows releasing this DoF through "**Do no connect selected DoFs**" option. However, this is not applicable for rotation degrees of freedom which need to be treated in a different way. Therefore, for the creation of a hinge connection, the "**Fixed 1D Beam to Solid for point**" option should be used. It works similarly to the master-slave condition; first, a point on one element is selected while "**Type of Cond**" parameter is set as "**1D BEAM**", and then the point on the second line is selected with "**Type of Cond**" parameter set as "**SOLID**". The bending takes place around the y-axis; therefore, selecting "**Do NOT Connect Rotation Y**" under "**Do no connect selected DoFs**" parameter is sufficient for the creation of the hinge. The settings of the input dialogs are shown in Figure 2-49.

The right support of the frame is a fixed support, and the left is a sliding support. These conditions are specified through fixing the given DoFs in "**Constant for Point**" and "**Rotation Constant for Point.**" The details can also be seen in Figure 2-49.

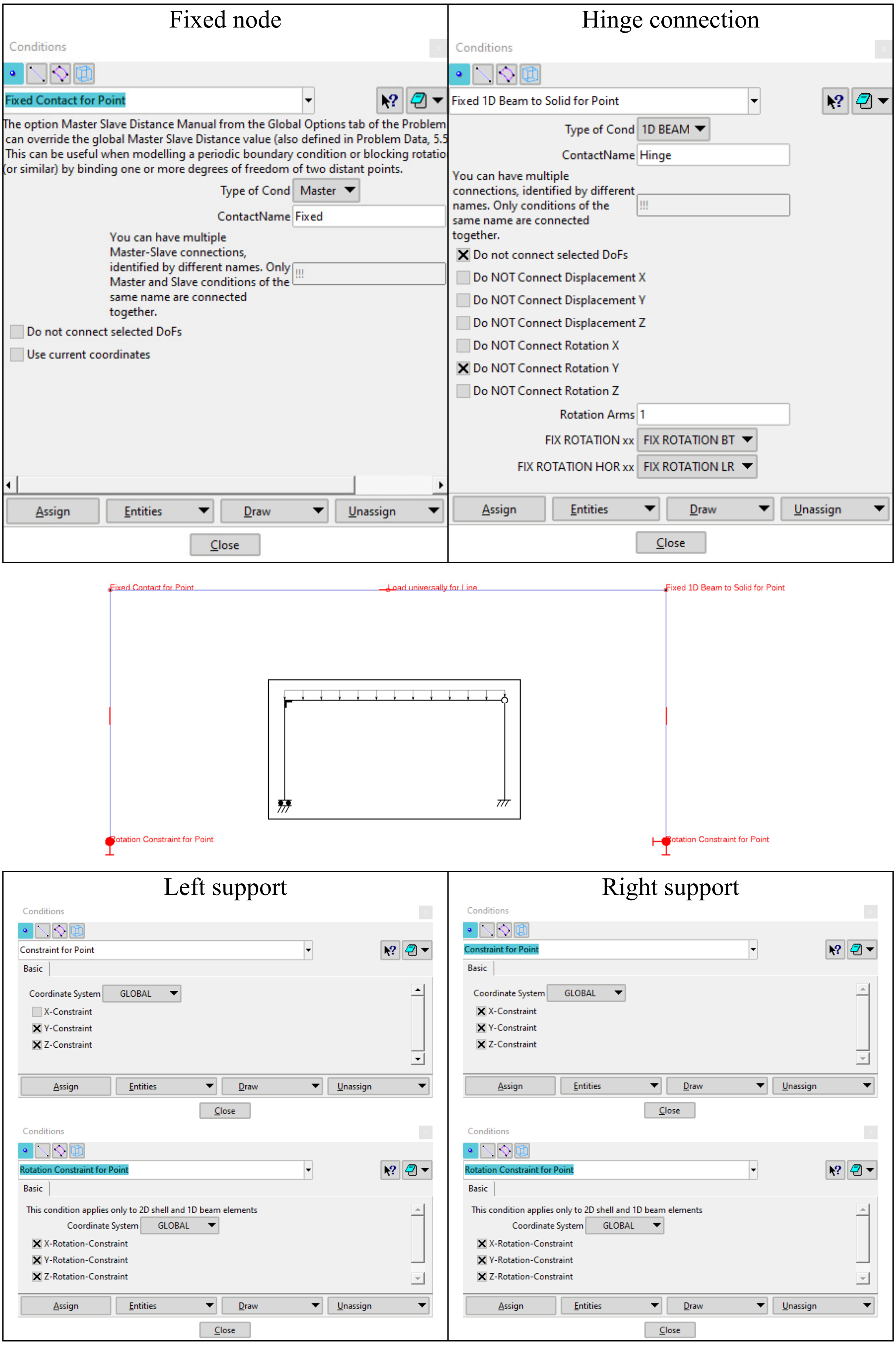


Figure 2-49 Static scheme of the structure, and the details of defining fixed and hinge connection between the column and beam, and supports.

Finally, a distributed load is applied to the beam of the frame using **Data | Conditions** load and choosing "**Load universally for Line**" from a drop-down list of conditions for line objects. The input window is shown in Figure 2-50.

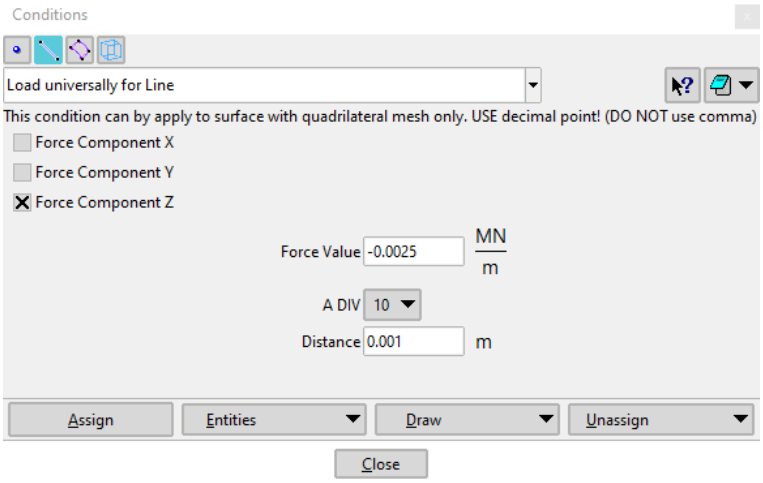


Figure 2-50 Dialog window for the application of distributed load to the beam.

For this example, a structured mesh is applied. This is specified through **Mesh | Structured | Line | Assign size**, and specifying 0.2 m. Then, the mesh is generated through **Mesh | Generate mesh …** It should be noted that any value suggested by GiD for meshing in the subsequent window will be neglected if the mesh size for structured mesh has been defined previously.

The solution in ATENA Studio is initiated by clicking  icon.

2.4.1.2 Results

The deformed shape and distribution of equivalent stress are plotted in Figure 2-52. Furthermore, the results of internal forces on the elements can be plotted by choosing "**Evolution 1D**" from the "**Result**" section of the "**View setting toolbox.**" For the internal forces, "**M n q Beam**" should be selected from the drop-down list. The distributions of the normal force, shear force, and bending moment are shown in Figure 2-52.

Normal force
N_x

Shear force
Q_{xz}

Bending moment
M_y

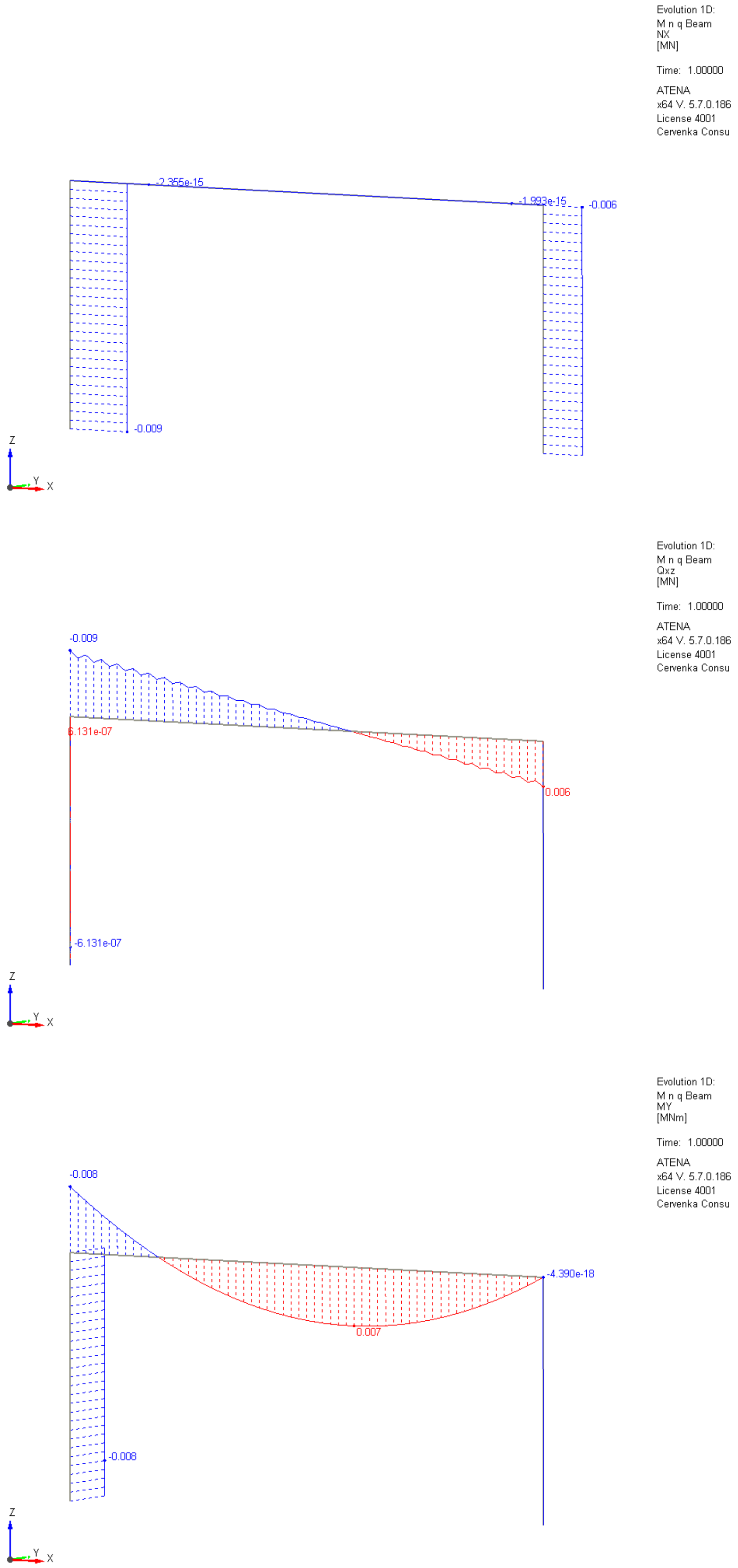


Figure 2-51 Distribution of the shear force and bending moment on the frame.

Deformed shame

Von Misses stress in integration points

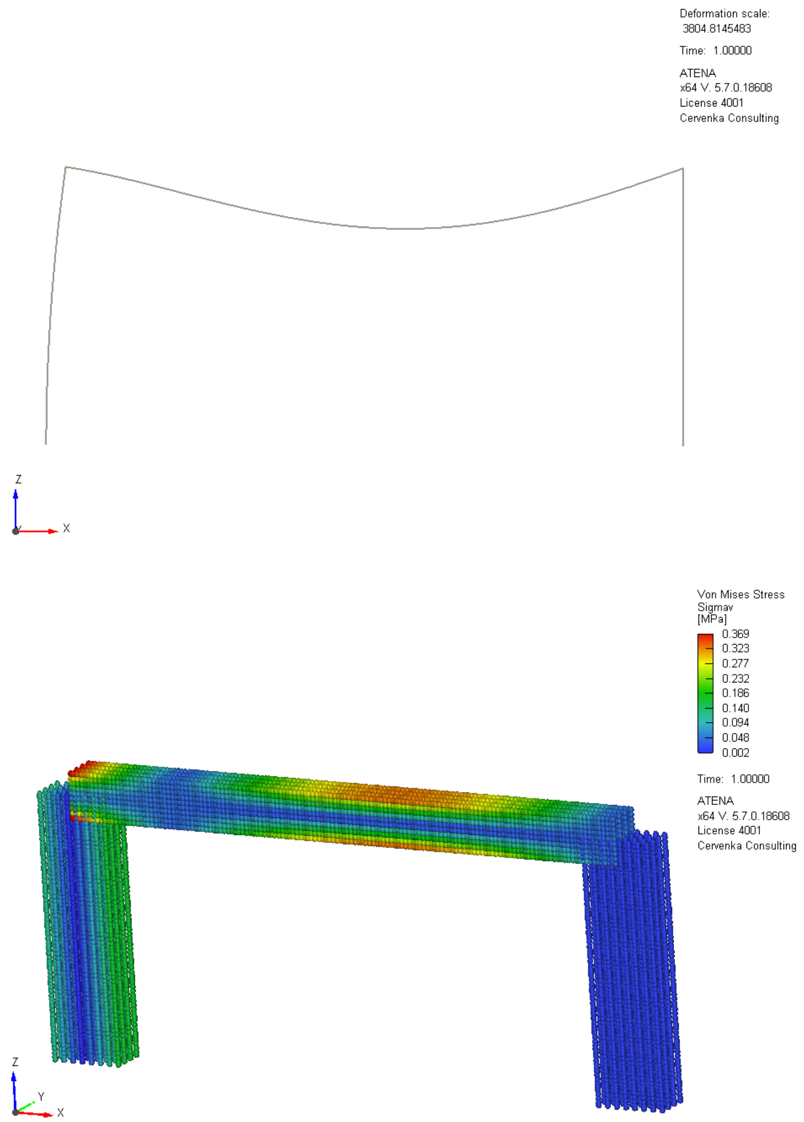


Figure 2-52 Deformed shape and equivalent stress in integral points.

2.4.2 Frame using 1D beam and solids

In this example, we show how to combine 1D beam with a volume meshed with solid elements. This can be particularly useful for models, where one part of the structure needs to be analyzed in more details and therefore will be modelled using 3D solid elements, and 1D/2D elements will be used for the rest of the model, where less precise results are acceptable. We will show examples of two frames; first, where the beam is connected to the columns with moment-resisting connections, and second, where hinge connections are applied.

Both examples are part of the ATENA installation. The frame with **fixed** beam-to-columns connections is located in the following directory:
%public%**Documents\ATENA\Examples\Science\GiD\Tutorial.Static3D_Frame_examples \FrameSolidsAndBeam.gid** ,
and the frame with **hinge** connections:
%public%**Documents\ATENA\Examples\Science\GiD\Tutorial.Static3D_Frame_examples\FrameWithHingesSolidsAndBeams1.gid** .

2.4.2.1 Pre-processing

The model of the simple frame is shown in Figure 2-53. Regarding the supports and loads, through **Data | Conditions**, displacement in all directions is fixed at the bottom surfaces of the volumes to support the frame, and a distributed load (0.002 MN/m) is applied to the line representing the beam of the frame.

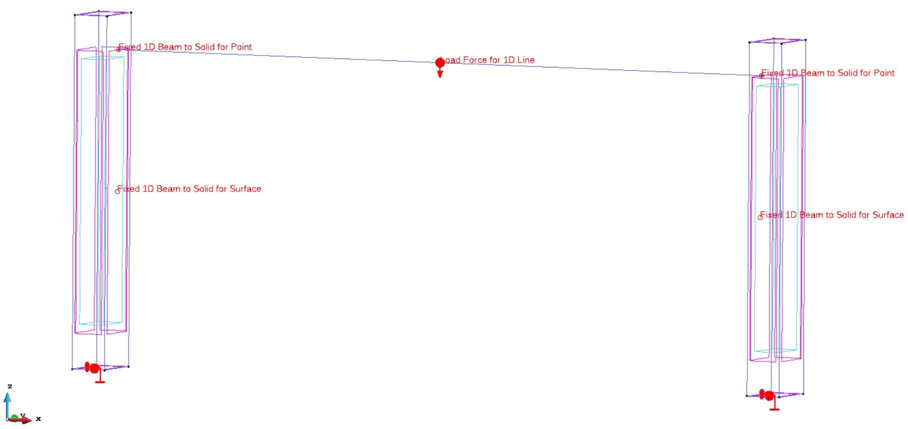


Figure 2-53 Model combining line geometry (1D beam) and volumes (3D solids).

The properties for the columns are set through **Data | Materials | Solid elastic**. First, a new material named "**ElasticColumn**" is created by clicking icon. The material properties used in this example are shown in Figure 2-54. Once the properties are defined and saved using icon, the material can be assigned to both columns through "**Assign | Volumes.**"

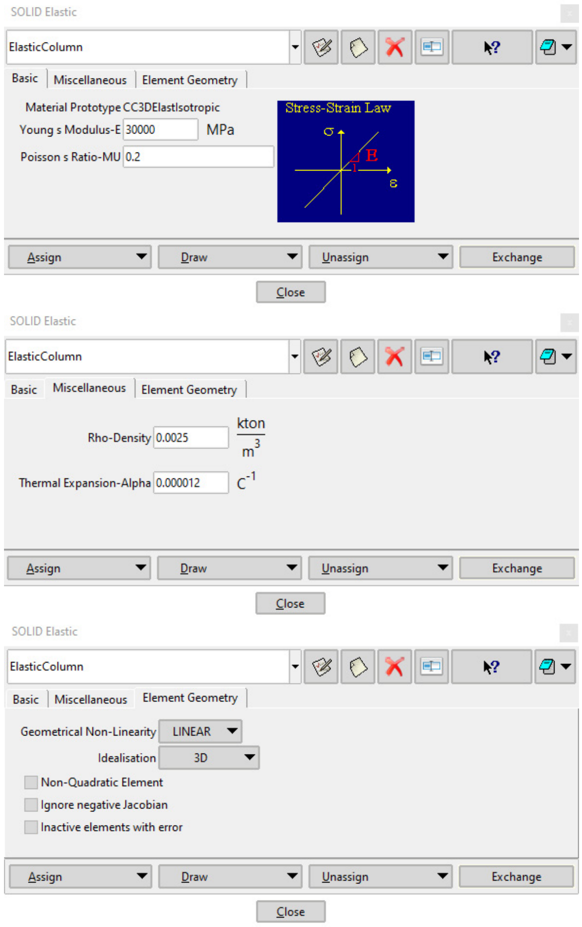


Figure 2-54 Input parameters for 3D solid material used for the columns.

In the case of the beam, its material and cross-sectional characteristics are given through **Data | Materials | BEAM Concrete**. The material properties are defined by addressing the data already specified for the columns. This is done by selecting "**ElasticColumn**" in the "**Use Base Material**" option. For more details about modeling with 1D beam elements, the reader is referred to the previous example of this chapter (2.4.1 Frame using 1D beams).

Then, the connections between the volumes and line need to be created. The conditions window is open through **Data | Conditions**. The dialog windows with complete settings both for fixed and hinge connections are shown in Figure 2-55. The column and beam will be connected through the endpoint of the beam's line, which lies on one of the surfaces of the column's volume. Therefore, the "**Fixed 1D Beam to Solid for point**" condition (located in  category) is applied to the endpoint of the beam's line, and "**Fixed 1D Beam to Solid for surface**" (located in  category) is applied to the surface of the volume. It is important to assign the same "**ContactName**" to both parts of the connection. The difference between fixed and hinge connection comes from releasing the rotational degree of freedom (DoF) when defining the contact. As the rotation takes place around the y-axis, releasing this DoF is sufficient for creating the hinge in case of this frame. The beam is connected to the column with two rotation arms ("**Rotation arms**" = 2), i.e., two force couples are transferred from the beam to the face of the column. Using this setting, one force couple acts at the top/bottom part of the beam cross-section, and the second force couple acts at $h/2$ and $-h/2$ (where h stands for the height of the beam's cross-section). The beam is connected to the column with the entire area of its cross-section as indicated by parameters "**FIX ROTATION xx**" and "**FIX ROTATION HOR xx**" set as "**BT**" (bottom-to-top) and "**LR**" (left-to-right), respectively.

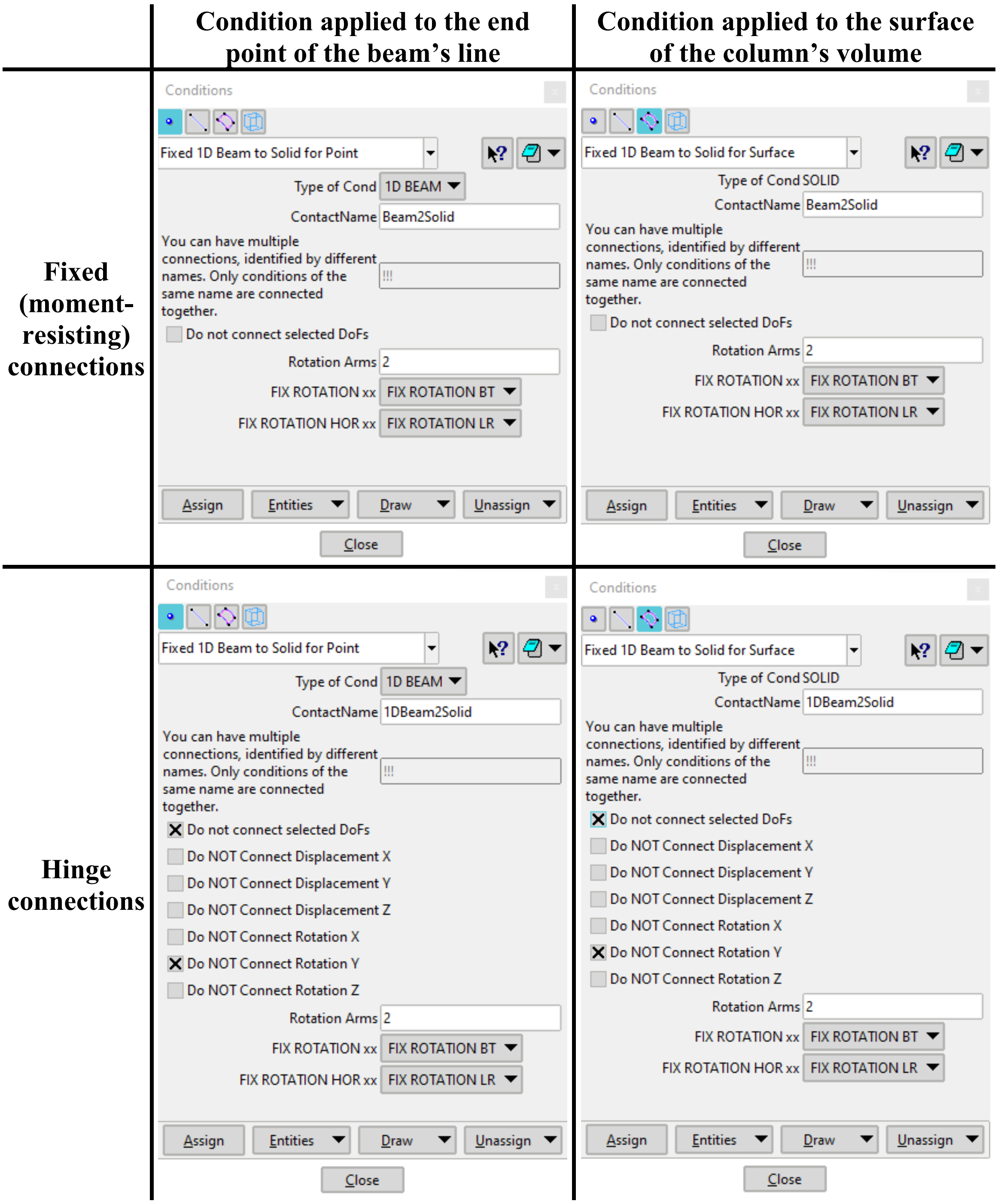


Figure 2-55 Conditions settings for connections of the 1D beam with 3D solids. Two types of settings for creating fixed and hinge connections are shown.

2.4.2.2 Results

The plots of the shear force, deformed shape, the bending moment in the beam, and the equivalent Von Mises stress are shown in Figure 2-56 for the frame with fixed and hinge connection. It can be observed that, although the beam is connected to the column through a hinge, the eccentricity of the connection with respect to the column's centerline (i.e., the shear force of the beam acts at the column's surface) induces banding in the column.

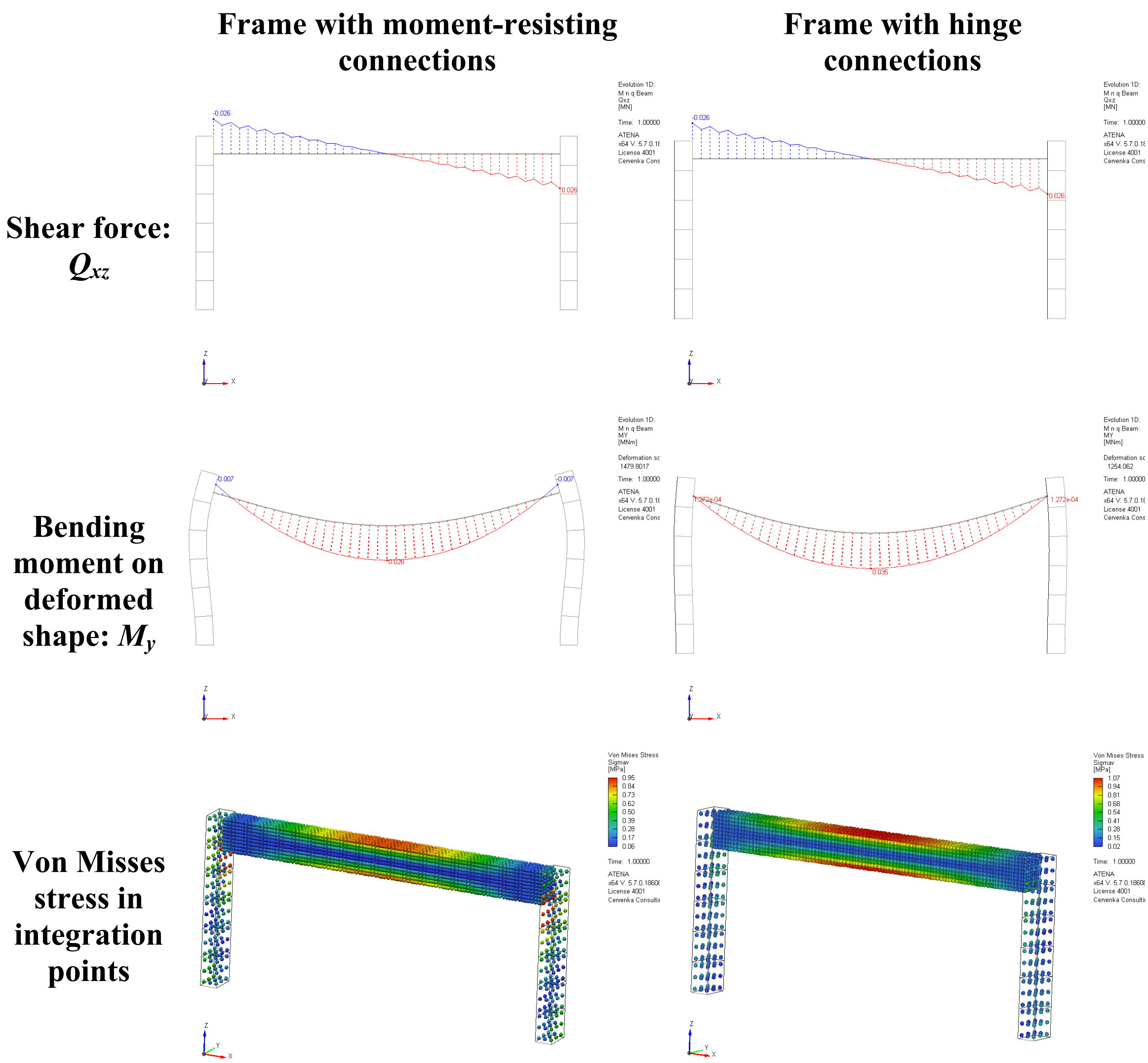


Figure 2-56 Comparison of result for beam connected to the columns through fixed and hinge connection.

2.4.3 Frame using 2D shell and solids

A combination of 2D shell elements with 3D solids can be useful for certain engineering applications. We show three examples of frames, where the columns are modeled using 3D solid elements, and 2D shells are used for the beam. The frames differ in the type of connection between the beam and columns (fixed or hinge) and in the placement of the beam. All examples are part of the ATENA installations:

- beam with **fixed** (moment-resisting) connections (%public%**\Documents\ATENA\Examples\Science\GiD\Tutorial.Static3D_Frame_examples\FrameSolidsAndShell.gid.gid**),
- beam with **hinge** connections (%public%**\Documents\ATENA\Examples\Science\GiD\Tutorial.Static3D_Frame_examples\FrameWithHingesSolidsAndShell.gid.gid**),
- beam connected to the **centerlines** of columns with **hinge** connections (%public%**\Documents\ATENA\Examples\Science\GiD\Tutorial.Static3D_Frame_examples\FrameWithHingesSolidsAndShell2.gid.gid**).

2.4.3.1 Pre-processing

As in previous examples, all frames have fixed supports at the bottom of the columns, and a distributed load is applied to the beam. The models are shown in Figure 2-57.

Models for fixed/hinge connection: the beam is connected to the vertical faces of the columns

Model for hinge connection: the beam is placed to the top faces of the columns

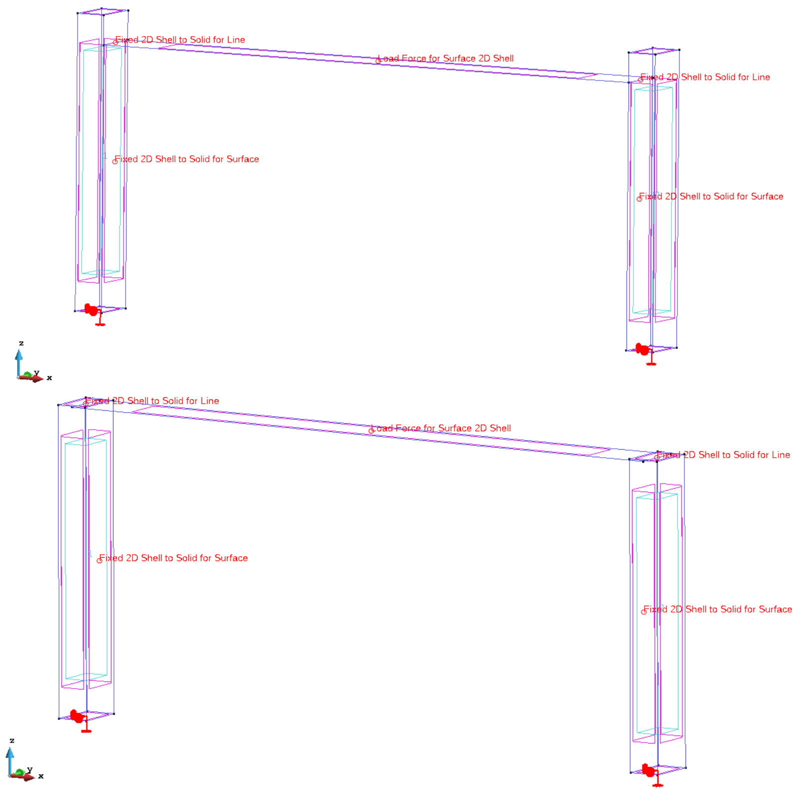


Figure 2-57 Models of the frame with different placements of the beam.

In the case of 2D elements, the distributed load represents a pressure applied to the surface. To apply the load, the loads/constrains dialog window is opened through **Data | Conditions**, then icon is selected, and "**Load Force for Surface 2D Shell**" is chosen from the drop-down list. The load of 0.005 MPa (equivalent to 0.0025 MN/m for the beam with a depth of 0.5 m) is applied, as shown in Figure 2-58.

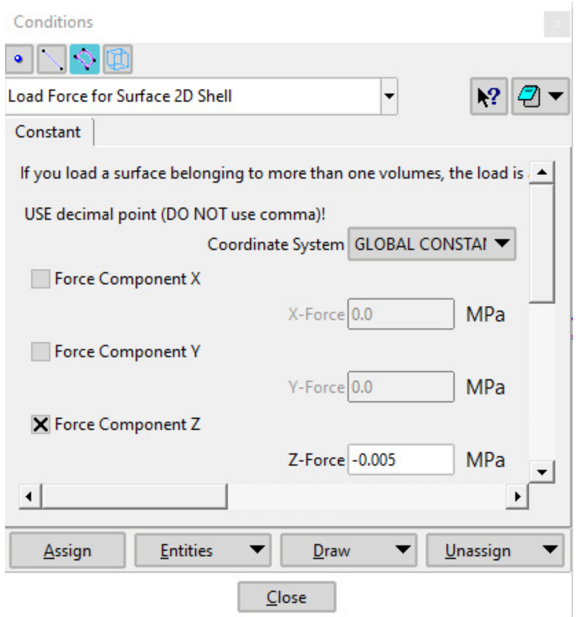


Figure 2-58 Dialog window for a load on the surface of a shell element

Both the beam and columns will be modeled with the same elastic material. This material is first given to the 3D solids used for the columns and later referred to when the 2D shell is defined. The 3D solid elements for columns are set through **Data | Materials | SOLID Elastic**. A new material called "**ElasticColumn**" is created with the Young modulus of 30 000 MPa and Poisson ratio of 0.2. As these settings are the same as in the case of the previous example (2.4.2 Frame using 1D beam and solids), the reader is referred there for more description and dialog windows screenshots.

The properties of the beam are specified for the 2D shell elements and then assigned to the beam's surface. The input window is open through **Data | Materials | SHELL Concrete-Steel**. A new shell called "**MyShell**" is created by clicking icon. Similarly to 1D elements, the local coordinate system needs to be defined for 2D shells. The *x* and *y* axes are the in-plane axes, while the *z*-axis is always parallel to the thickness of the shell. Regarding the dialog window in GiD, the definition by vectors is recommended, for which vectors in global coordinates for *z* and *x*-axes are needed, and the *y*-axis is then defined automatically. In the case of this example, the local coordinate system agrees with the global one. The previously-defined material properties are referred by choosing "**ElasticColumn**" from the drop-down list of "**Use Base Material**" parameter. The thickness of 0.6 m corresponding to the height of the beam is specified in "**Solid Ref Thick**" and "**Thickness Eqn**" parameters. To capture the bending behavior using the shell elements, it is recommended to use at least 6 internal layers over the thickness of the shell element (i.e., the height of the beam). Therefore, the parameter "**Layers**" is set as 6. On the other hand, it is not necessary to place multiple shell elements (surfaces) above each other to capture the bending behavior (see also chapters 5.3.2 Shell Material and 5.7.1 Notes on Meshing in the theory manual [1]). Finally, the element called "**CCIsoShellQuad<xxxxxxxx>**" is assigned to the shell material. Once all the inputs are specified,  icon is selected to update the parameters, and "**Assign**" button is used to assign the material to the surface in the model. All the input parameters used in this example are shown in Figure 2-59.

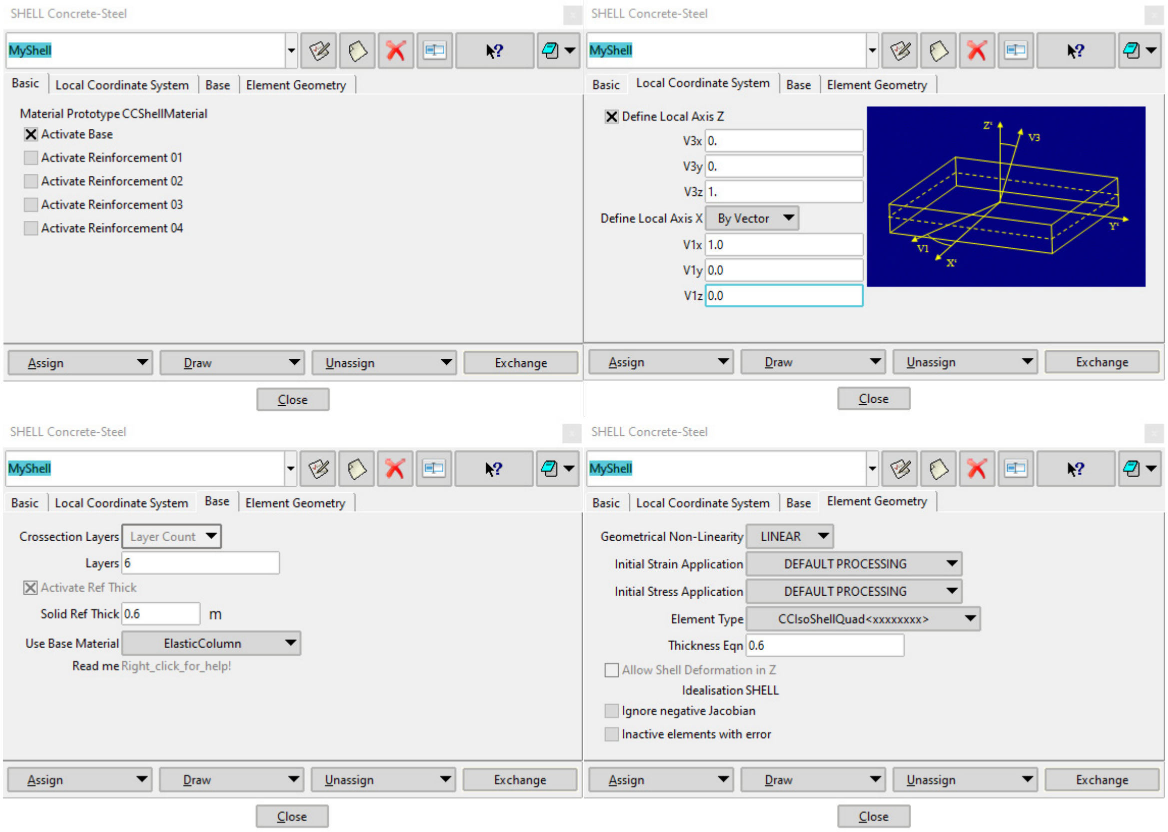
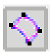


Figure 2-59 Inputs for 2D shell material used for modeling the beam.

Then, the connection between the beam's surface and column's volumes needs to be created. The condition dialog is open through **Data | Condition**. Geometrically speaking, the line belonging to the beam's surface will be connected with the surface belonging to the column's volumes. Therefore, we apply conditions "**Fixed 2D Shell to Solid for Line**" in  category to the line and "**Fixed 2D Shell to Solid for Surface**" in  category to the surface. The parameter "**Type of Cond**" is set as "**2DShell**" when the condition is defined for the surface's line and as "**Solid**" for the surface belonging to the column's volume. It should be noted that it is important to assign the same name to both parts of the contract; otherwise, the connection will not be created. The specification of the contact allows releasing of displacement or rotation degrees of freedom (DoF). In the case of the fixed connections, all DoFs between the beam and the columns will be connected; however, for the hinge connections, the rotation DoF around the y-axis will be released. Please note that we model two frames with hinges; one with the beam connected to the vertical surfaces of the columns and one with the beam placed on the top surfaces of the columns. Therefore, appropriate surfaces should be selected while defining the connections. By setting the "**Rotation arms**" parameter equal to 2, it is specified that the surface will be connected to the volume by two force couples; one acting along the surface's line at the top/bottom part of the shell's cross-section and one acting along beam's line at the height of $h/2$ and $-h/2$, where h stands for the height of the shell's cross-section. Furthermore, by setting "**FIX ROTATION xx**" parameter as "**FIX ROTATION BT**" the entire height (bottom-to-top) of the shell's cross-section will be connected to the surface of the column's volume.

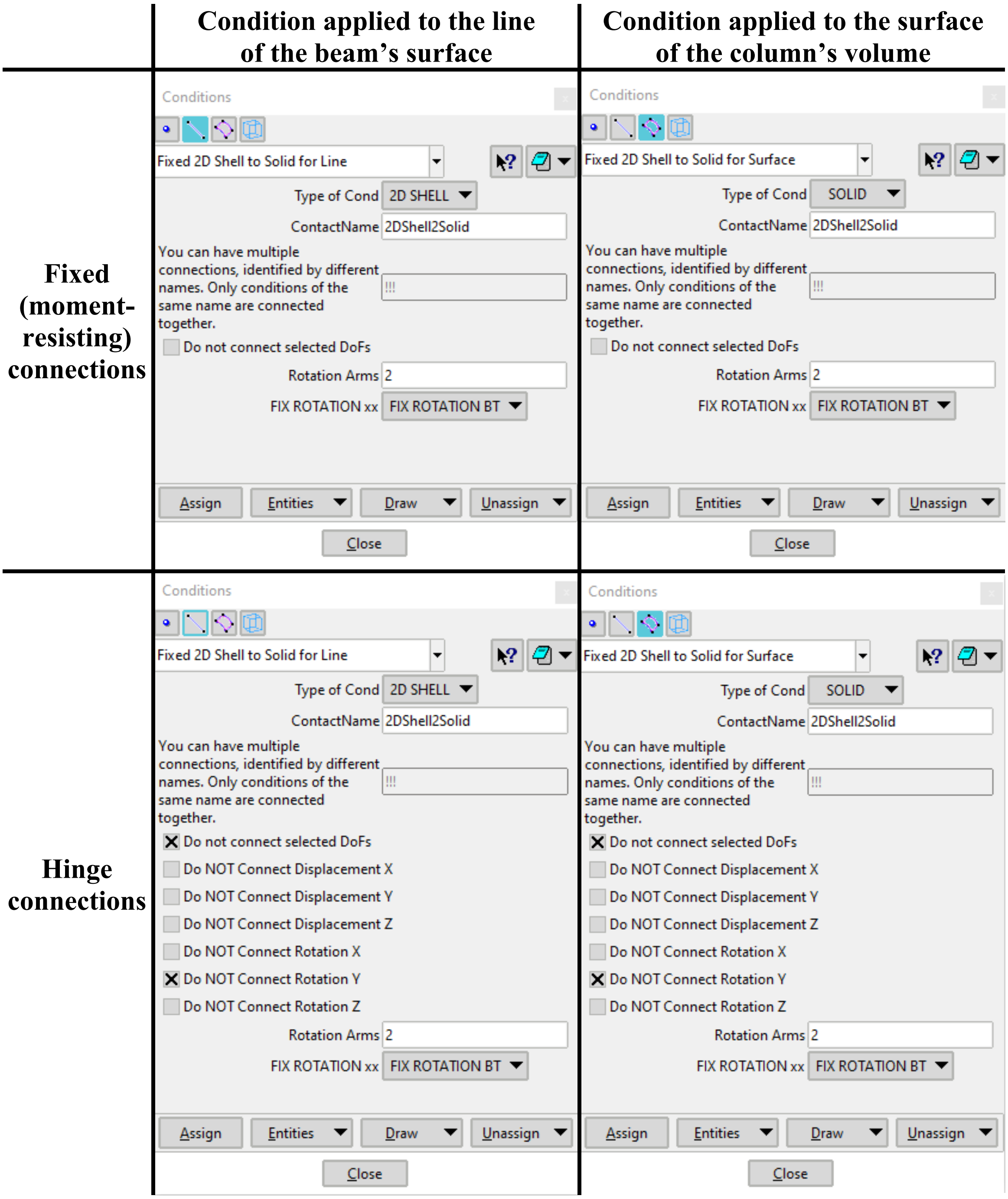


Figure 2-60 Conditions settings for connections of the 2D shells with 3D solids. Two types of settings for creating fixed and hinge connections are shown.

2.4.3.2 Results

The results of all three models are shown together in Figure 2-61. The bending moment in the beam on the deformed shape and equivalent Von Mises stress in integration points are plotted. It can be observed how the placement of the beam affects the result, i.e., the eccentricity of the induces bending in the column by the bending moment by the shear force in the connection.

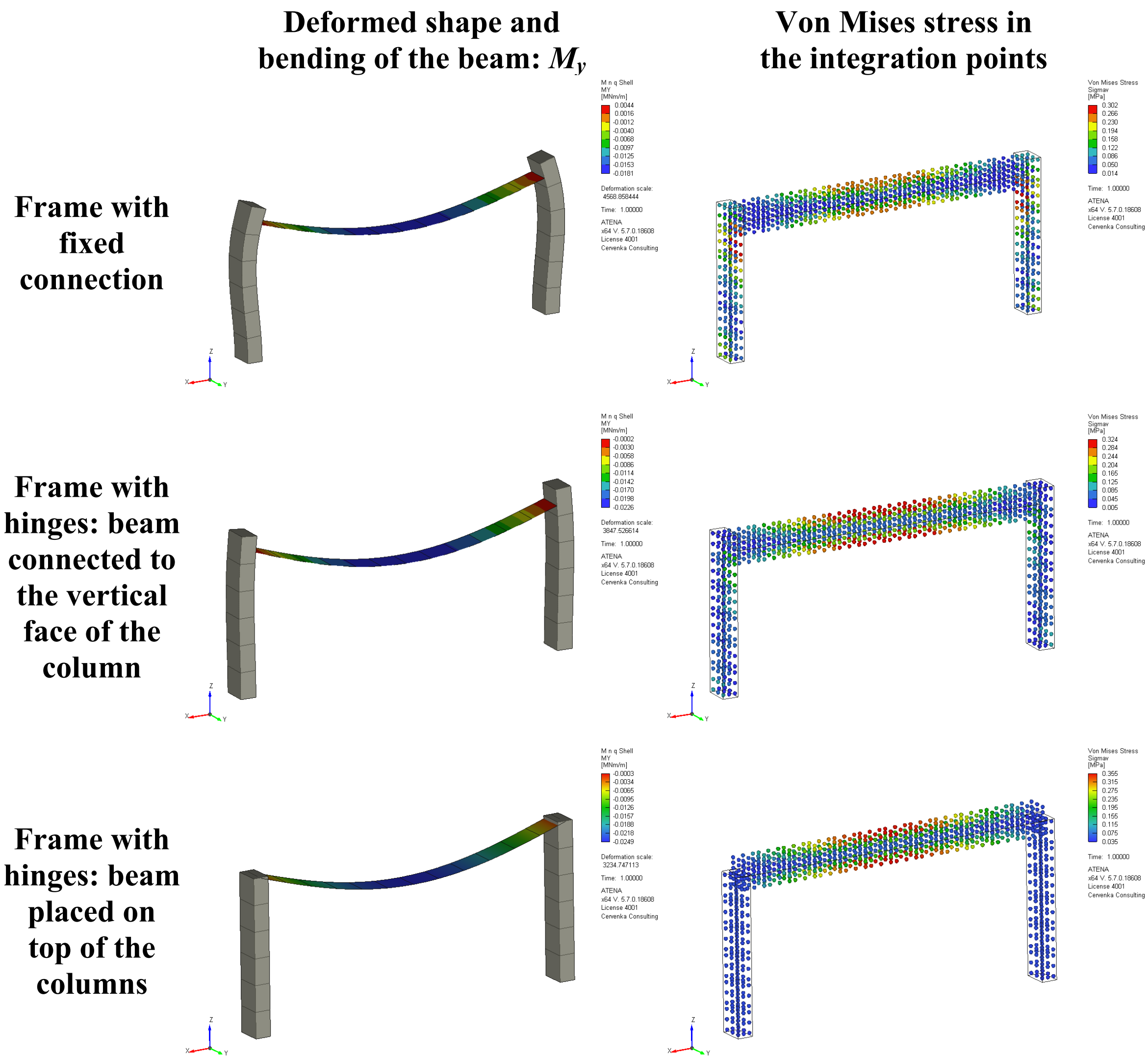


Figure 2-61 Comparison of the results for the frames with fixed and hinge connection, and for the different placing of the beam.

2.5 Modeling of fiber reinforced concrete (FRC)

2.5.1 Introduction

The volume of fiber reinforced concrete (FRC) used for construction processes increases every year. If the fibers are added to the plain concrete, the material performance characteristics such as tensile strength or post-peak ductility improve significantly. Currently, there are multiple material models suitable for FRC modeling in ATENA software. CC3DNonLinCementitious2User model allows tailored calibration of the material model and is particularly useful when the results of experimental tests are available. For this model, a separate tutorial [14], where we show in-depth how to obtain the material characteristics, is part of ATENA documentation. Besides that, in this example, we show the modeling of FRC using CC3DNonLinCementitious2FRC material model. It is based on the standard CC3DNonLinCementitious2 model, where the tension softening branch is modified based on the added fracture energy approach proposed by Juhász [17]. The tension softening law is schematically shown in Figure 2-61.

The GiD model can be open in the following path:
%public%**Documents\ATENA\Examples\Science\GiD\Tutorial.FRC\ FRC_4PBT_2D_force.gid**

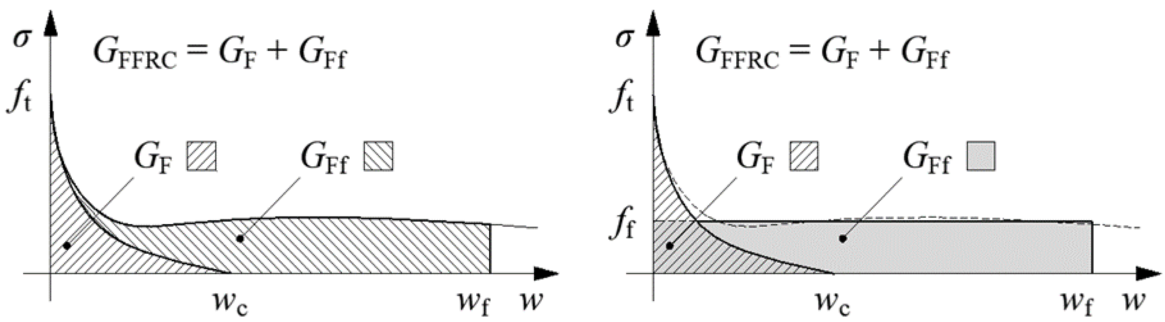


Figure 2-62 Tension softening law modified by the added fracture energy approach used for FRC material model: (left) experimental data, and (right) CC3DNonLinCementitious2FRC material model [17].

2.5.2 Geometry and material characteristics settings

The geometry of the test specimen is with slight modifications taken from the special tutorial dedicated to advanced modeling of fiber reinforced concrete [14]. The problem is simplified as a 2D task with a thickness of 0.15 m. In Figure 2-63, the schematics of the experimental set-up and the GiD model are shown. In detailed observation, it can be noticed that the constrain on the left supporting plate is placed slightly asymmetrically. This introduces a certain eccentricity to the model, which helps better localization of the failure crack in the midspan. Similar to the experiment set-up, the loading of the model is controlled by prescribing forces at two steel plates on the top surface of the beam. For the simulation of a force-controlled experiment, the unloading branch of the load-displacement curve cannot be obtained by the default Newton-Raphson iteration scheme, and the Arc-length method needs to be chosen in the solver settings. The solver method can be selected through **Data | Problem data | Problem Data** in the "**Solution Parameters**" tab.

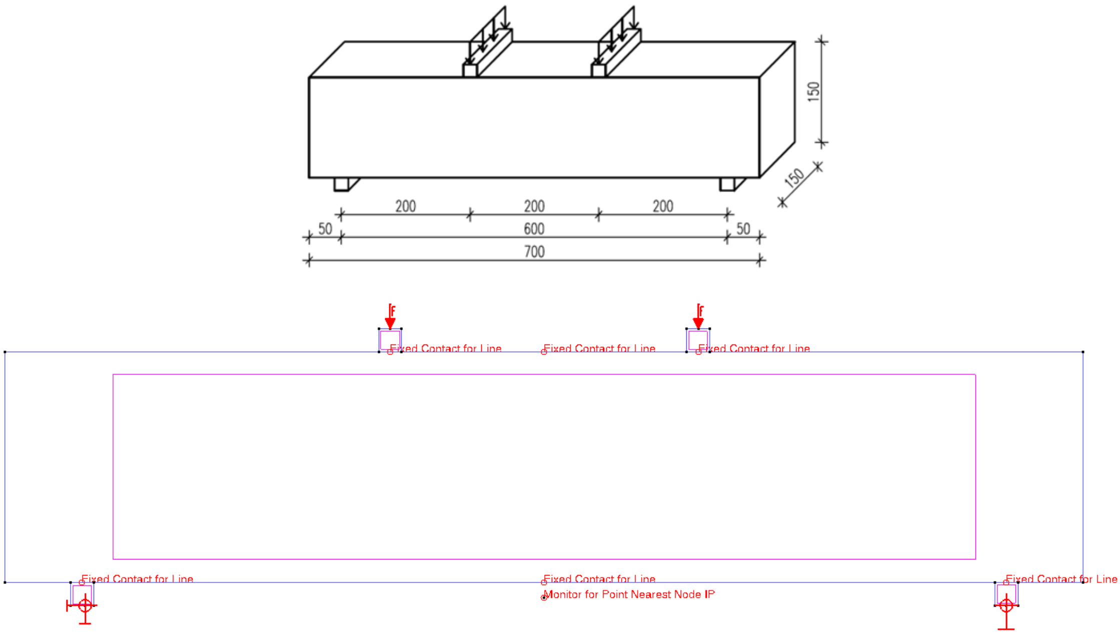


Figure 2-63 (top) Schematics of the experimental set-up, and (bottom) 2D model in GiD.

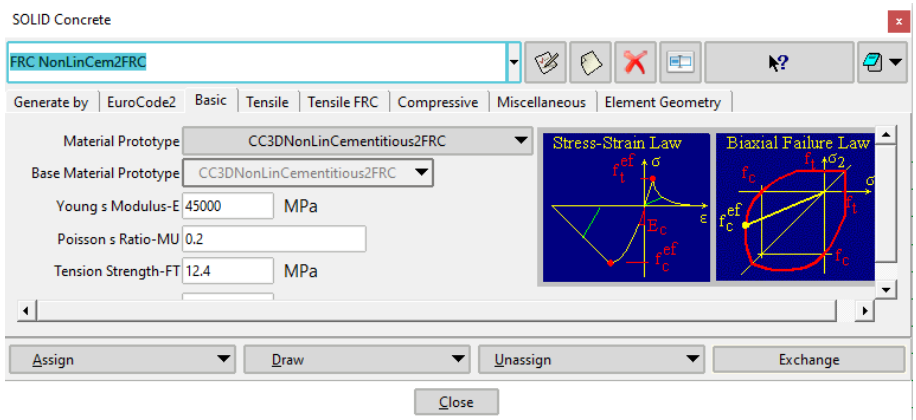


Figure 2-64 Material definition – Basic tab.

Next, the material settings for the FRC concrete are to be set. The material settings are open through **Data | Materials | SOLID Concrete**. Based on the measurements of material characteristics, the compressive strength of the FRC concrete was 125 MPa, the elastic modulus was 45 GPa, and the tensile strength was estimated as 12.4 MPa. The material model "**CC3DNonLinCementitious2FRC**", which we use in this example, can be found as a prototype in the **"Basic"** tab of the **"Cementitotus2"** material, as shown in Figure 2-64. Once selected, the additional tab called **"Tensile FRC"** appears for inputting two parameters describing the added fracture energy. One is the value of the added fracture energy itself, and the second is the maximum crack opening width of the FRC. For the FRC mixture in question, BASF Masterfiber 482 in a volume fraction of 1.5 % were used. Based on the product documentation, the length and diameter of these fibers are 13 mm and 0.2 mm, respectively. In this example, it has been found that the added fracture energy yields to 2300 N/m and the maximum crack width to 0.5 mm, as shown in Figure 2-65. It should be noted that the value of the added fracture energy corresponds to the energy needed for the pull-out of the fibers and the maximum crack opening corresponds to the length of the used fibers.

In a general case, the two parameters needed for the specification of the added fracture energy depend on the mass and length of fibers used in the mixture. For some typical fibers, these values can be automatically generated in the GiD environment based on the type of fibers. Currently, two types of steel fibers and one type of synthetic fibers are supported in the generator.

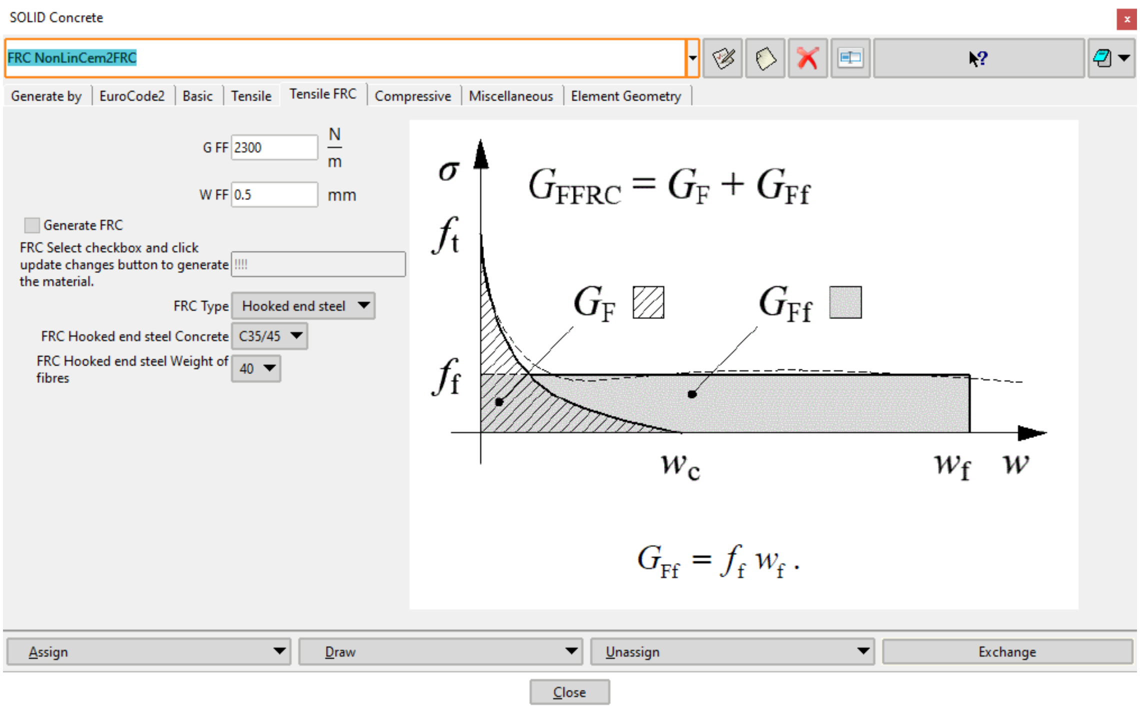


Figure 2-65 Material definition – added fracture energy specification and automatic generator.

Once the material definition is finished, the solution in ATENA Studio can be initiated by clicking icon.

2.5.3 Results

The numerical results are compared with the results of two experimental tests in Figure 2-66. It can be observed that the maximum load-bearing capacity of the model and the L-D diagrams agree reasonably well. This demonstrates the general ability of this simplified material model for describing the behavior of FRC mixtures. Furthermore, in Figure 2-66, the results of the plain concrete are shown to demonstrate how the added fiber modify concrete mechanical performance.

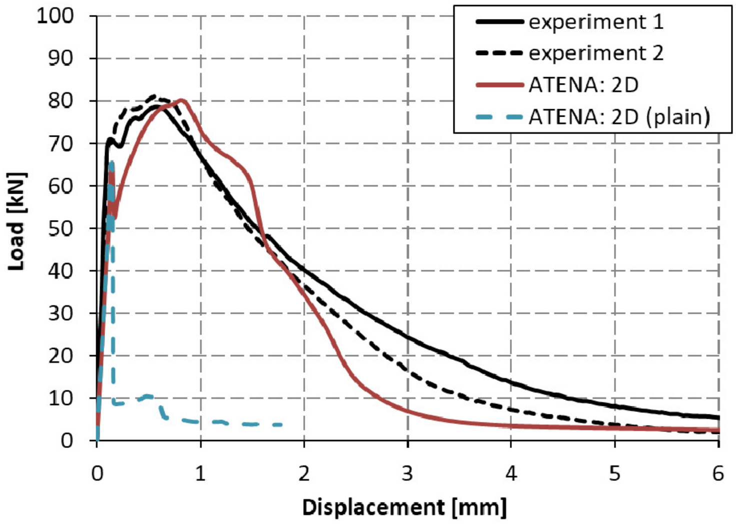


Figure 2-66 Comparison of experimental and numerical results of the 4-point bending test of FRC beams. Numerical results for plain concrete are plotted for comparison.

3. CREEP ANALYSIS

This chapter contains examples of creep analysis using the program **ATENA**. Currently, the commands required for creep analysis are not supported by the native ATENA graphical environment, and therefore the necessary commands must be entered manually or by using the ATENA-GiD interface. **GiD** (see the Internet address http://gid.cimne.upc.es/) is a general purpose finite element pre and post-processor that can be used for data preparation for **ATENA**. See the README.TXT file in the ATENA installation for the instructions how to install the ATENA interface to **GiD**.
In order to activate the creep analysis option an appropriate problem type must be selected: **Data | Problem type | ATENA | Creep**.

3.1 Long-term Deflection of a Reinforced Concrete Beam

Keywords: reinforced concrete, discrete reinforcement, creep

Input files:

%Public%**\Documents\ATENA Examples\Science\GiD\Tutorial.Creep2D\BeamWithCreep.gid**

3.1.1 Introduction

This example demonstrates the application of ATENA system to the creep analysis of a reinforced concrete beam. The analyzed beam was tested by Dr. Jan Vitek from Metrostav corp., Czech Republic.

3.1.2 Comments on FE model preparation

3.1.2.1 General data

The problem is modeled by two models: two-dimensional one and three-dimensional. In both cases, the geometrical model (Figure 3-2 and Figure 3-4) is created in such a way to facilitate the generation of purely structural meshes, i.e., meshes that are composed of only quadrilateral and hexahedral elements in 2D and 3D, respectively.

3.1.2.2 Reinforcement

If program **GiD** is used for pre-processing, the reinforcement can be modeled by discrete bars. Discrete reinforcement bars are modeled as line curves. These lines should be meshed by as few elements as possible. Typically one truss element per line is sufficient. **ATENA** then automatically determines the intersection of these lines with the 3D model and places reinforcement-embedded elements into each segment that is created by this process.

3.1.2.3 Materials

When creep analysis is requested, the material for which creep should be taken into account must be modeled by one of the creep materials (see the ATENA Theory Manual [1] or ATENA Input File Format [2]). Within the creep material a base concrete material is defined, which is one of the standard ATENA materials. Currently only following materials are supported as creep base materials: "**CC3DNonLinCementitious2**", "**CC3DbiLinearSteelVonMises**" or "**CC3DDruckerPragerPlasticity**".

Material properties used in this example are listed in Table 3.1-1 and Table 3.1-2.

3.1.2.4 Topology and loading

The loading history is defined in terms of intervals in **GiD**. In the first interval, the supports are defined as well as the two vertical forces. The first interval should represent the application of the permanent load. In the subsequent interval, this load will be kept constant, and the material will creep, causing the deflections as well as cracking increase. The application of the permanent load is expected to cause some cracking; therefore, it is subdivided into 20 steps. The application of the permanent load will start at the time of 63 days, and it will be completed at 63.02 days.

In the second interval, no additional forces are applied; therefore, only supports are defined for this interval. The interval starts at 63.02 days and ends in 360 days. This interval is represented in **ATENA** by only a single step. **ATENA** automatically inserts substeps if it determines that one load step would not be sufficient for such a long time period.

3.1.2.5 Monitoring points

Monitoring points are chosen in order to describe a load-displacement response as well as the long-term behavior. In ATENA-GiD interface, the monitoring is defined as conditions (**Data | Conditions**). The monitoring defined in this way is considered only if specified in the first interval. The definition of monitoring points in subsequent intervals is ignored. In this example always the applied force is monitored as well as the mid-span deflection.

3.1.2.6 Run

Analysis can be started either directly from GiD or by other options… see ATENA Troubleshooting 4.3.1

3.1.3 Results

The results from the analysis are documented in Figure 3-6, where the calculated long-term mid-span deflection is compared with the experimental data obtained by Dr. Vitek. It shows that without any specific calibration, the model predicts well the long-term deflections.

3.1.4 References

[1] ATENA Documentation, Part 1, ATENA Theory Manual, Cervenka Consulting 2003.

[2] ATENA Documentation, Part 6, ATENA Input File Format, Cervenka Consulting 2003.

[3] ATENA Documentation, Part 8, User's Manual for ATENA-GiD Interface, Cervenka Consulting 2003.

Table 3.1-1 Material properties of concrete

Material type		Creep material: CCModelB3
Concrete type		1
Thickness, i.e., ratio of volume [m^3] to the surface area [m^2] of cross-section		0.05138
Cylindrical compressive strength after 28 days [MPa]	$f_c^{'28}$	46.75
Elastic modulus after 28 days [GPa]	E^{28}	34,2
Humidity [-]		0.6
Density [kg/m^3]	ρ	2 370 kg
Aggregate/cement ratio [-]	AC	4.44
Water/cement ratio [-]	WC	0.5
Shape factor [-]		1.0
Curing [-]		Air
End of curing [days]		7
		Base material: CC3DNonLinCementitious2
Elastic modulus [GPa]	E_c	34 200 MPa
Poisson's ratio -	ν	0.2
Compressive strength [MPa]	f_c	46.75
Tensile strength [MPa]	f_t	3.257
Fracture energy [N/m]	G_f	127
Compressive plast. def. [m]	w_d	-0,0005

Table 3.1-2 Material properties of reinforcement

Material type		Reinforcement	
		bilinear	
Elastic modulus	E	210	GPa
Yield strength	σ_y	400	MPa
Hardening		perfectly plastic	

Table 3.1-3 Finite element mesh

Finite element type	CCIsoQuad – Quadrilateral, isoparametric	CCIsoBrick - Hexahedral isoparametric
Element shape smoothing	N/A	N/A
Optimization	Gibbs-Poole	Gibbs-Poole

Table 3.1-4 Solution parameters

Solution method	Newton-Raphson
Stiffness/update	Tangent/each iteration
Number of iterations	60
Error tolerance	0.01/0.0001/0.01/0.01
Line search	off

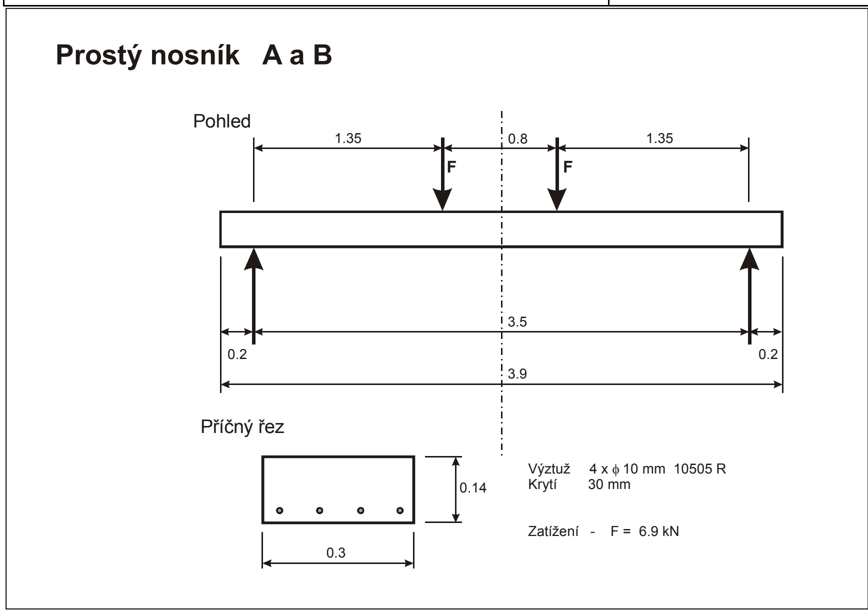


Figure 3-1: Geometry of the reinforced concrete beam.

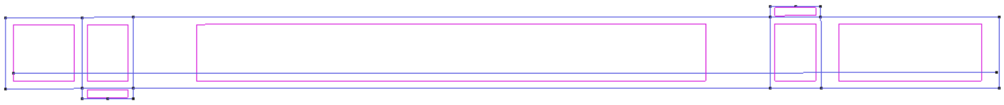


Figure 3-2: Two dimensional geometrical model with reinforcement, loading and boundary conditions in GiD.

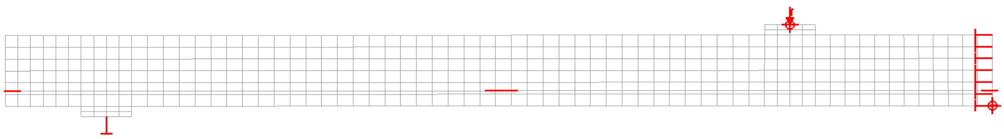


Figure 3-3: Two-dimensional finite element model.

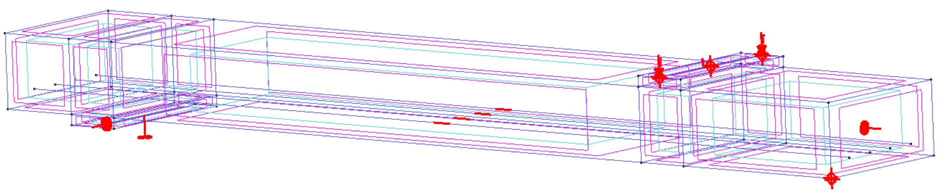


Figure 3-4: Three-dimensional geometrical model with reinforcement, loading and boundary conditions in GiD.

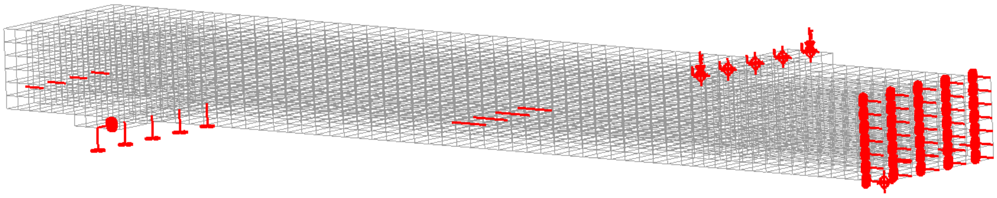


Figure 3-5: Three-dimensional finite element model.

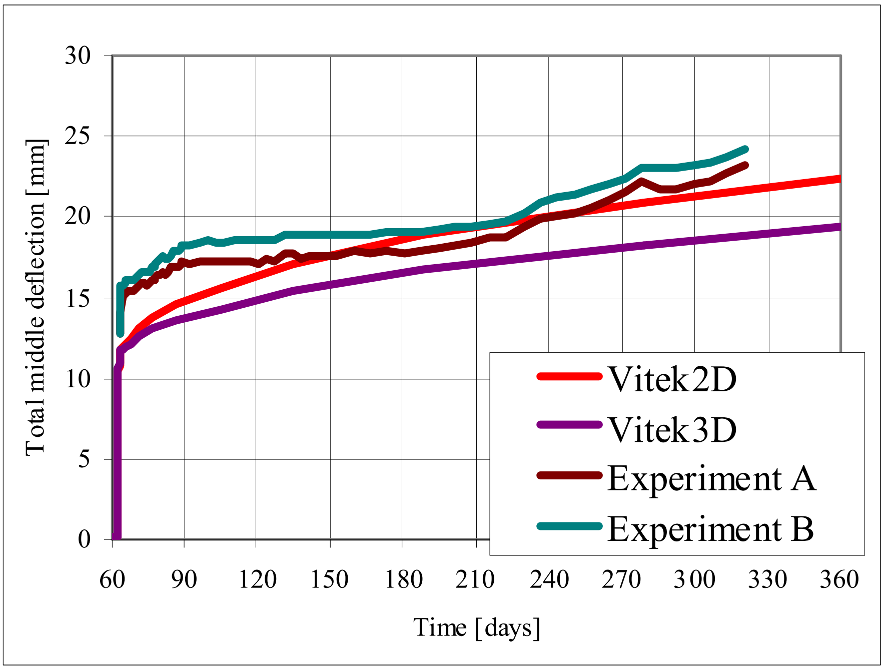


Figure 3-6: Long-term mid-span deflection. Comparison of two- and three-dimensional analysis with experimental data.

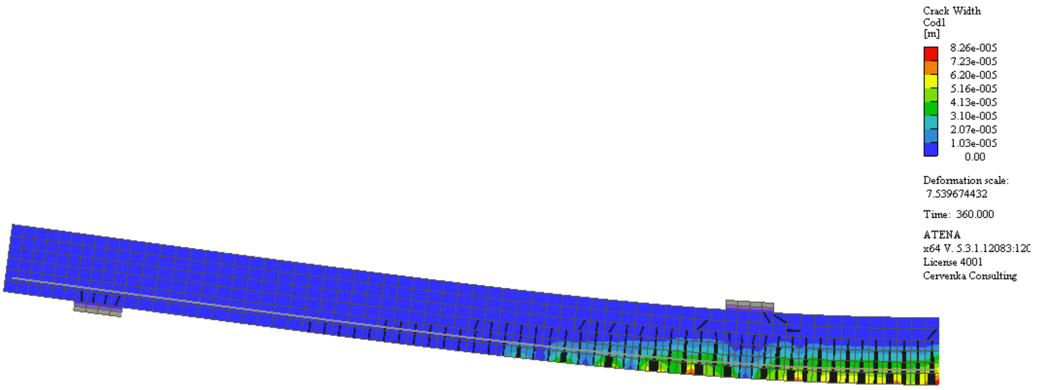


Figure 3-7: Crack Width of beam at 360 days, 2D

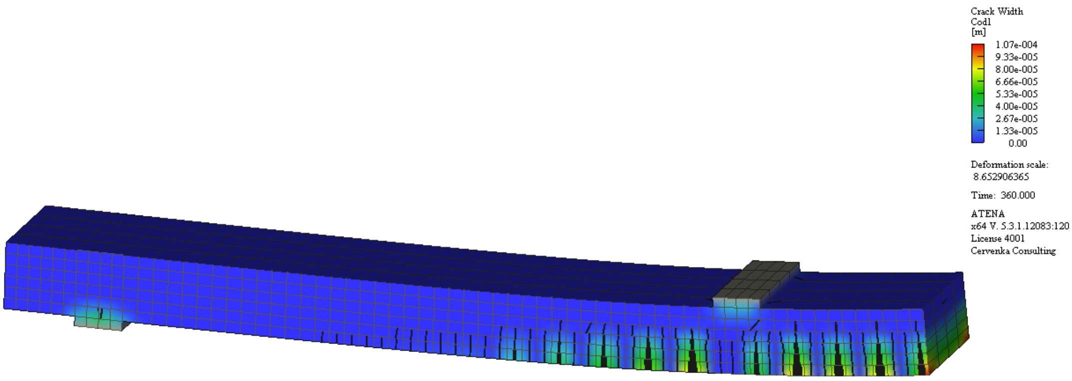


Figure 3-8: Crack Width of beam at 360 days, 3D

4. TRANSPORT ANALYSIS

4.1 Combination of Thermal and Static Analysis

This example shows the coupling of thermal and stress analyses.

4.1.1 Introduction

This document describes an example of a rotationally symmetrical vessel subjected to thermal loading. The analysis is performed using the programs **ATENA** and **GiD**. **ATENA** is used for thermal and static analysis, and the program **GiD** is used for data preparation and mesh generation.

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. Public%**Documents\ATENA Examples\Science\GiD**

4.1.2 Thermal analysis

First, the program **GiD** is started. The recommended version is 9.0.4 or newer (the oldest supported version is 7.7.2b). After starting **GiD**, the user should open the example analysis:

"**Public%\Documents\ATENA Examples\Science\GiD\Tutorial.Temperature2D\‘PipeBTemp.gid**".

This is an existing model demonstrating the combination of thermal and stress analysis. This problem is using the problem type: **ATENA | Transport**. It represents a section of a pipe wall with a thickness of 0.23 m and an internal diameter of 1 m. Taking advantage of the symmetry, only a quarter of the whole cross-section is modeled. The geometry of the model is shown Figure 4-1, and the numerical model is shown in Figure 4-2. Details about ATENA-GiD interface and associated problem types for **ATENA** can be found in the manual [6]. The same mesh size is used for thermal and static analysis. This is however not a strong requirement. The thermal loading can also be exchanged between models with totally different meshes.

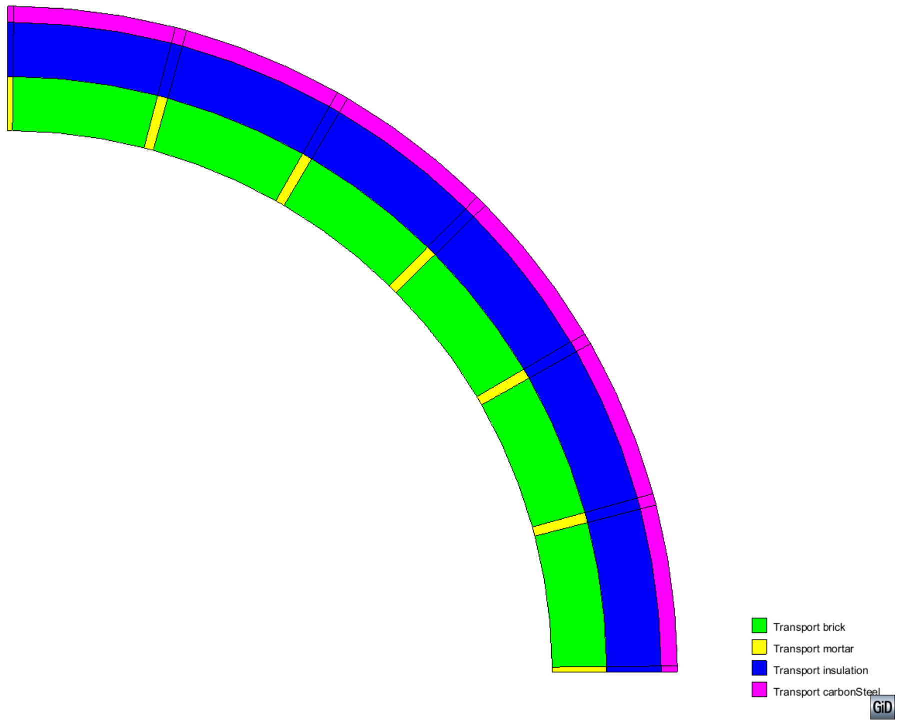


Figure 4-1: Geometry and material properties of the axisymmetrical pipe model.

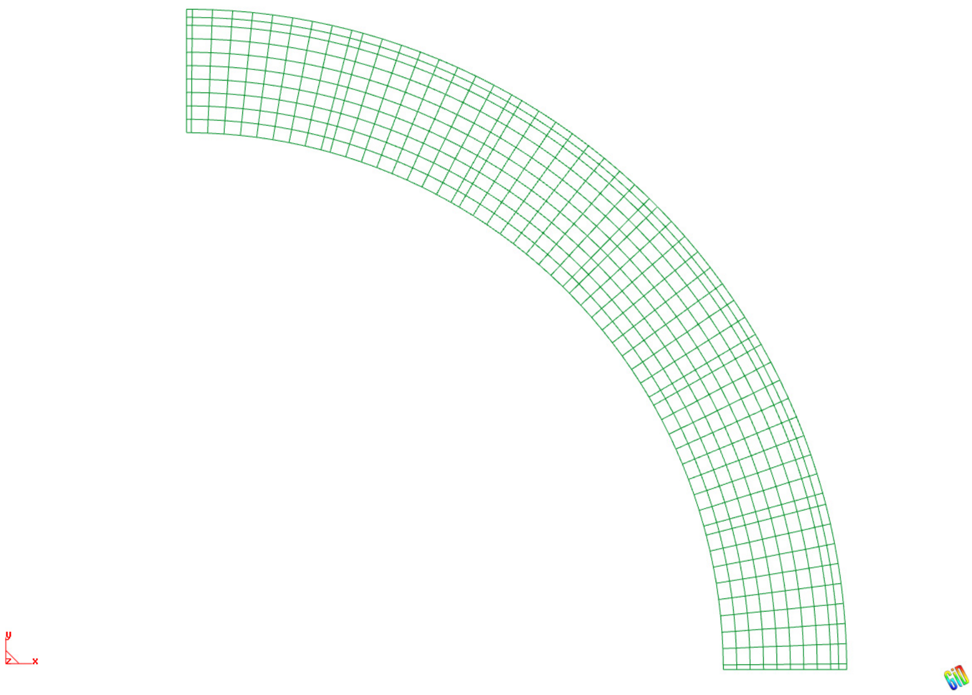


Figure 4-2: Numerical model, finite element mesh.

The loading is subdivided into three intervals. In the first interval, 12 load steps are defined with boundary conditions as described in Figure 4-3. In each step, the temperature on the outer surface is increased by 1 °C (see Figure 4-3). The temperature in the exterior is increased up to 37 °C, starting from the initial uniform at 25 °C. Each step in this interval represents 3000 seconds; thus, the whole interval covers the period 0-36000 seconds (0-10 hours).

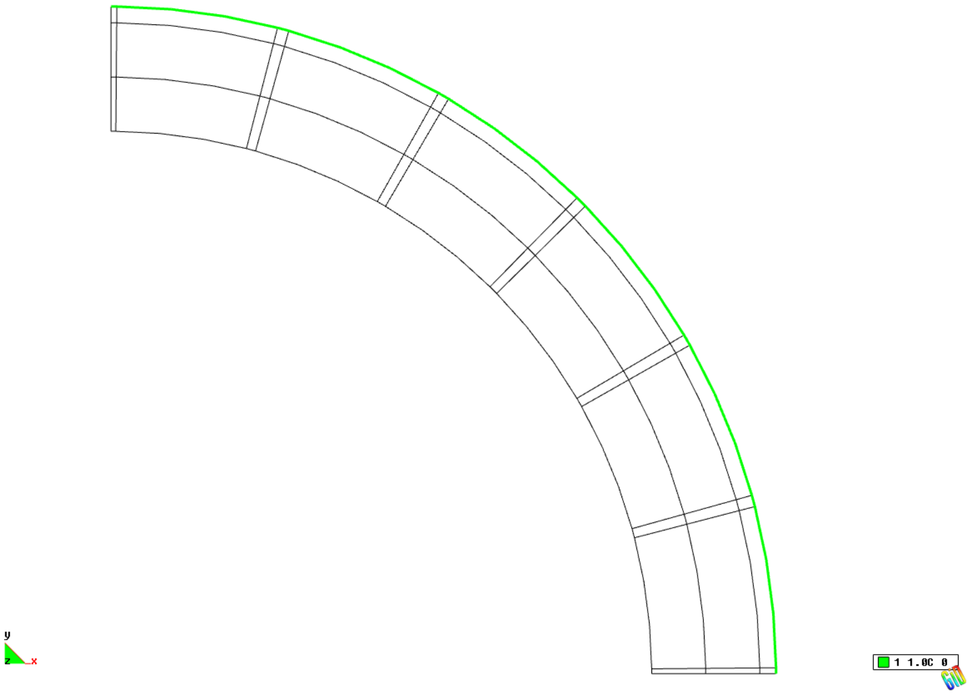


Figure 4-3: Boundary conditions for the interval 1.

In the subsequent interval 2, the temperature at the outer surface is kept constant. This interval contains 10 steps 2880 seconds long. This means the whole interval spans the time period from 36000-64800 seconds (10 hours-18 hours).

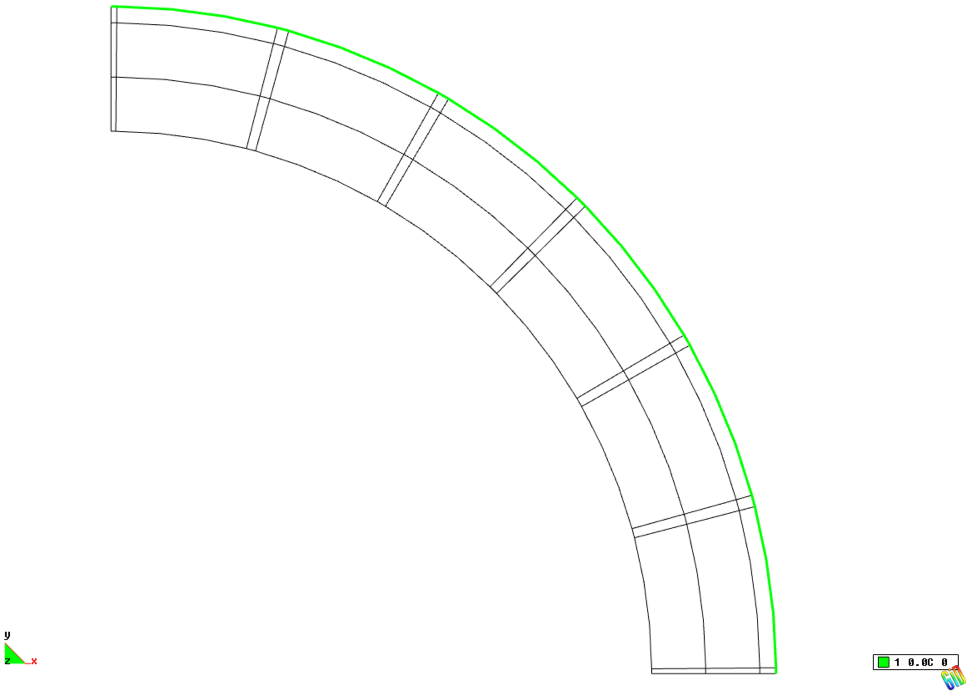


Figure 4-4: Boundary conditions for interval 2.

In the last interval 3, the outer surface is cooled back to 25 degrees. This interval contains 12 steps 1800 seconds long. This means the whole interval spans the time period from 64800-86400 seconds (18 hours-24 hours).

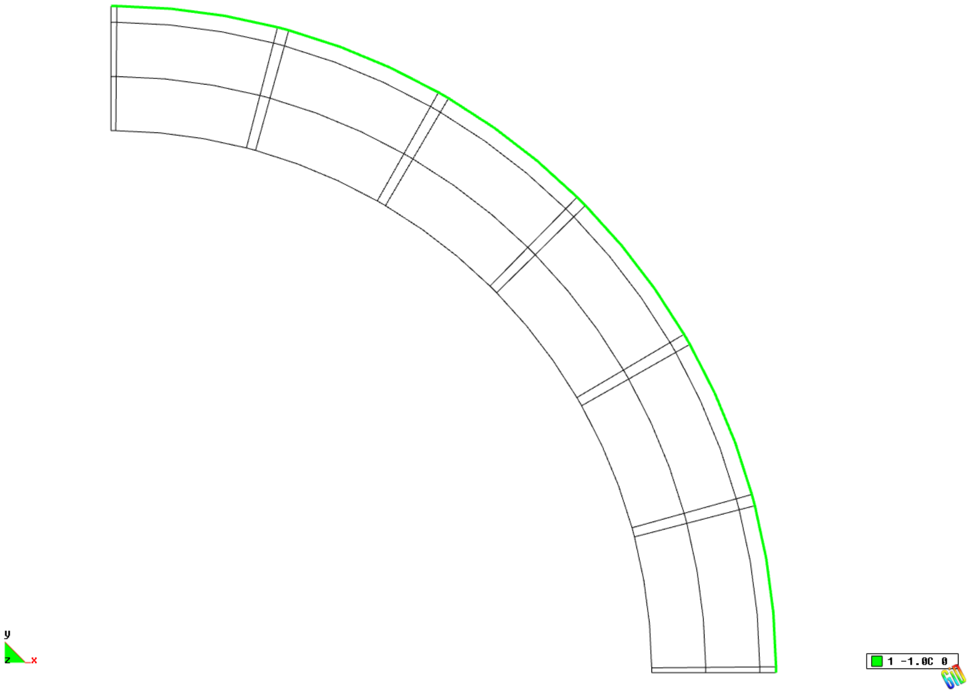


Figure 4-5: Boundary conditions for the interval 3.

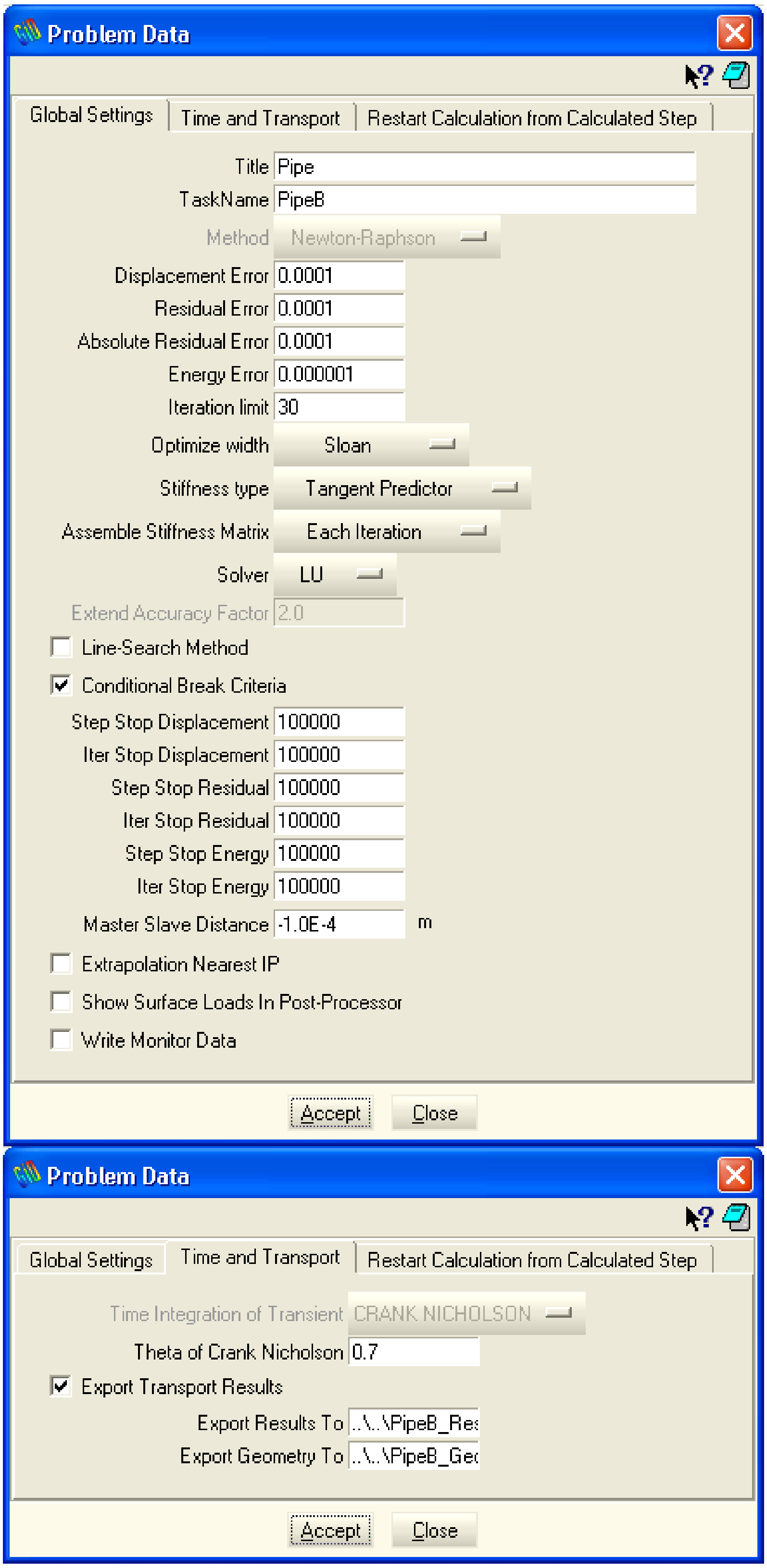


Figure 4-6: Problem data dialog including the definition of temperature exchange files with stress analysis.

The problem data dialog that is shown in Figure 4-6 can be opened via the menu item **Data | Problem data**. This dialog can be used to define the basic parameters for the thermal analysis. The most important fields can be found at the bottom of the **Time and Transport** tab, where the names of two files are to be specified. These files are used for exchanging the temperature fields with the subsequent stress analysis. By default, the files would be stored in the AtenaTransportCalculation subdirectory of the main problem directory, inserting "../../" before the names write them into the Tutorial.Temperature2D directory.

The ATENA calculation is started from the menu **ATENA | ATENA analysis** or by clicking the calculator icon . This will start **AtenaStudio** program, which is a graphical interactive environment for the execution control of ATENA finite element core module.

After executing ATENA analysis, the following window appears on the user's computer (see Figure 4-7).

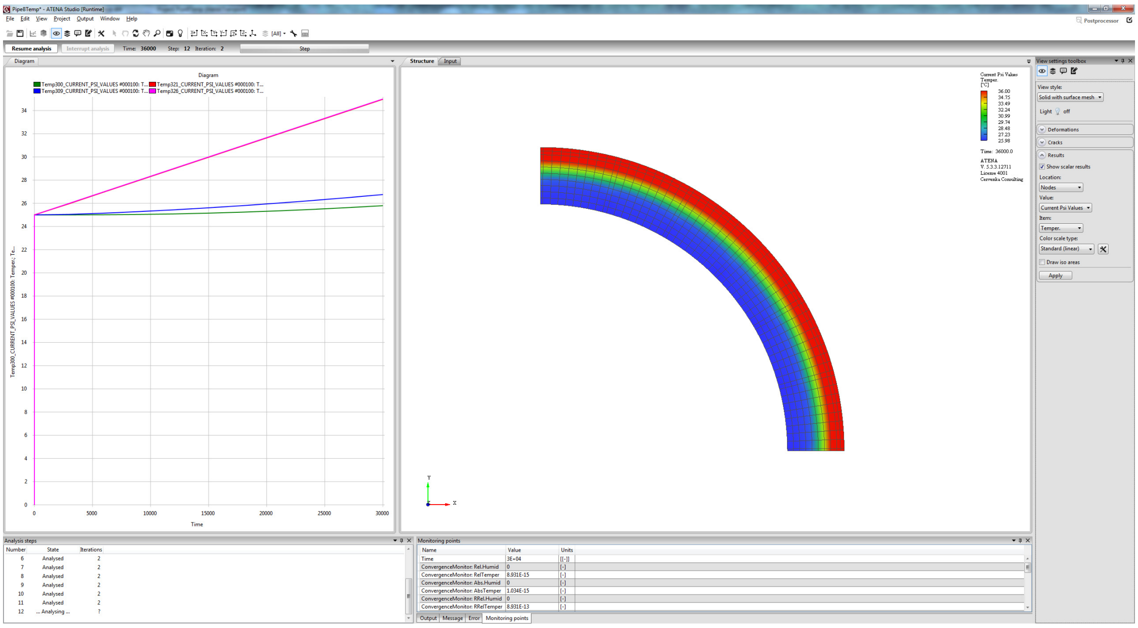


Figure 4-7: The main AtenaStudio window after its activation from GiD.

The ATENA analysis is started automatically (or by clicking the  button). This starts the thermal analysis in AtenaStudio environment. In order to visualize the development of the temperature field Result part can be selected various variables can be selected for display. The temperature fields can be displayed by selecting **CURRENT_PSI_VALUES | Temper**.

After the thermal analysis is completed **AtenaStudio** can be closed. All resulting files are stored in the subdirectory AtenaTransportCalculation of the PipeBTemp.gid directory. In this subdirectory the following files will exist after the completion of the thermal analysis:

PipeBTemp.inp	ATENA input file created by **GiD** and used by **AtenaStudio**
PipeB.00xx	Binary result files created by **ATENA** during the thermal analysis.

These two files are created in the Tutorial.Temperature2D directory:

PipeB_Results.thw	Saved temperatures to be used by stress analysis.
PipeB_Geometry.bin	Saved geometry to be used for the interpolation of temperatures in the stress analysis.

4.1.3 Stress analysis

After the thermal analysis is completed **AtenaStudio** can be closed, and the stress analysis can be performed using the calculated thermal fields. A new GiD problem must be created, or the existing problem PipeBStatic.gid can be used. This model defines the input for stress analysis of the same pipe wall as was used in the thermal analysis.

The geometrical model and boundary conditions are shown in Figure 4-8.

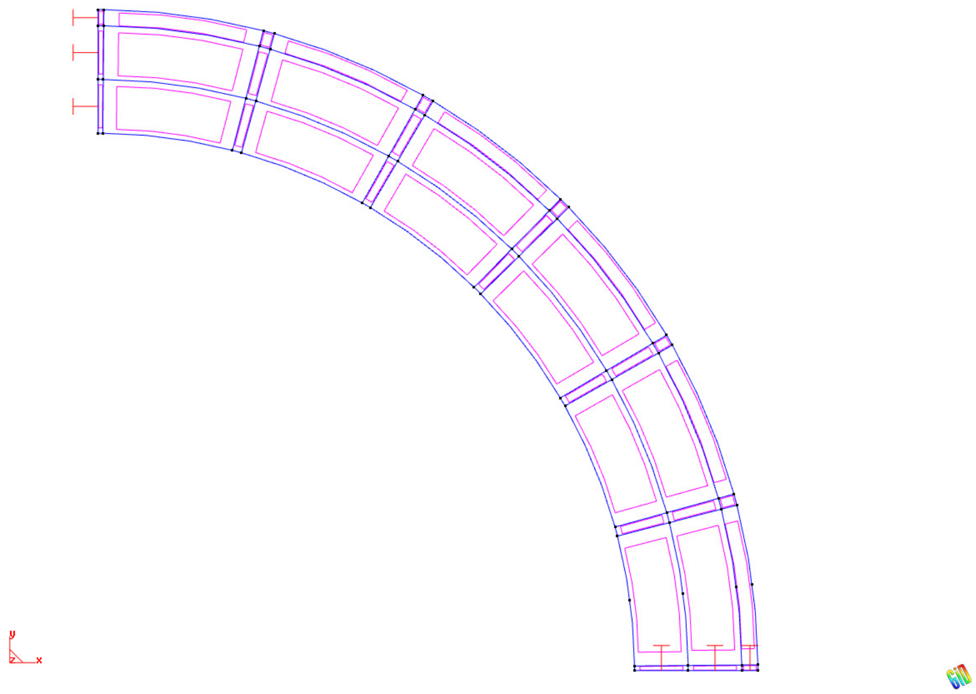


Figure 4-8: Geometrical model and boundary conditions for stress analysis.

In order to be able to utilize the thermal fields calculated during the thermal analysis, the appropriate import files must be specified in the problem data dialog that is activated from the menu **Data | Problem data** and is shown in Figure 4-9. This information is located in the bottom two input fields where appropriate file names are specified, including their path.

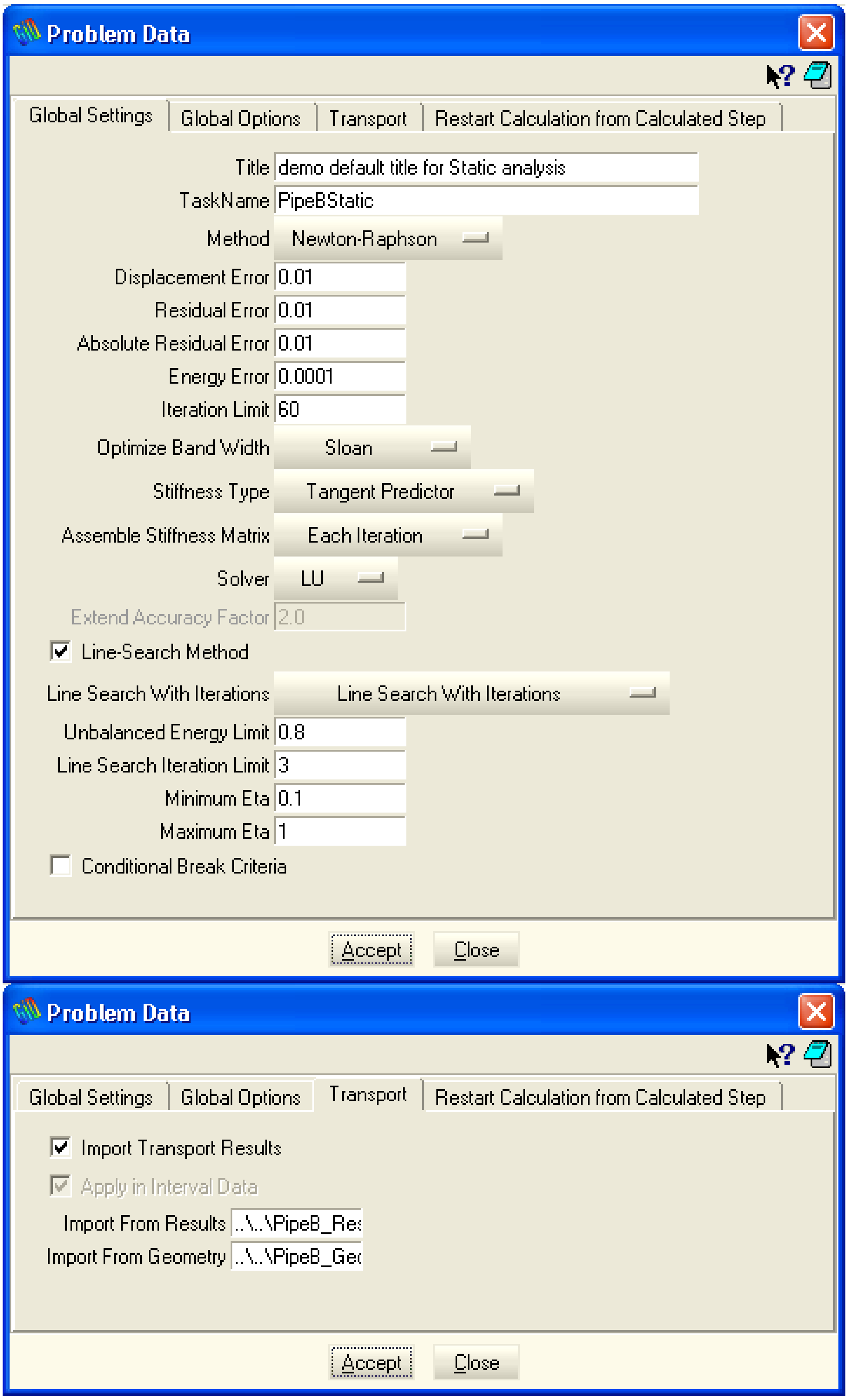


Figure 4-9: Problem data dialog including the definition of temperature exchange files.

The loading history is specified in a single interval. The interval is divided into 50 load steps. In each step, the temperature difference of 720 seconds from the thermal analysis is applied (Figure 4-10). So the interval spans the period of 10 hours, i.e., it only covers the heating phase.

The **Transport Import** switch set to **Interval Beginning** means the thermal analysis results are only imported once, and the temperature values are interpolated for each static analysis step. For complex temperature histories not well approximated by a linear interpolation throughout the interval (e.g., fire analysis), this should be changed to **Each Step**. Then, the thermal results are imported in each load step, and the temperature fields are interpolated from the two nearest transport analysis steps.

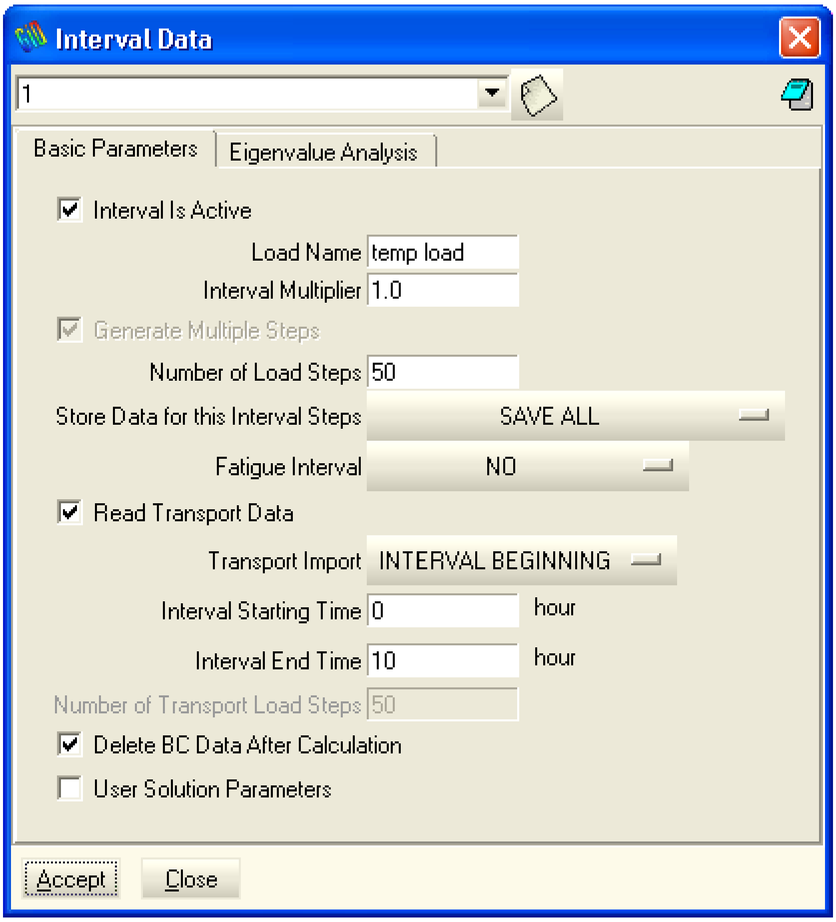


Figure 4-10: Interval data for interval 1.

The stress analysis is again started by selecting the menu item **ATENA | ATENA analysis** or the  icon. This starts the **AtenaStudio** program. The analysis is started automatically or by clicking the  button. This starts the stress analysis in **AtenaStudio** environment. The contour areas of crack width can be displayed by selecting **Elements | CRACK_ATTRIBUTES | COD1**. The resulting computer screen is shown in Figure 4-12.

The imported temperature values can be displayed as **ELEM_TOTAL_TEMPERATURE | TotalTemp**. Please note that only the difference from the reference (initial) temperature is displayed in static analysis.

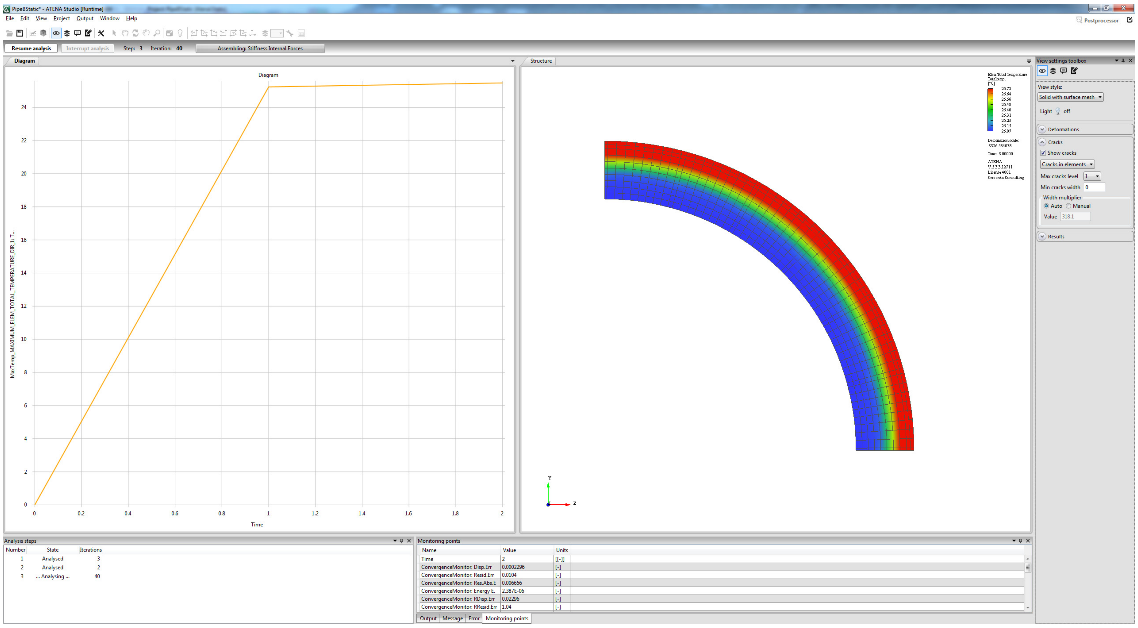


Figure 4-11: Execution of static analysis in AtenaStudio and the selection of crack opening display.

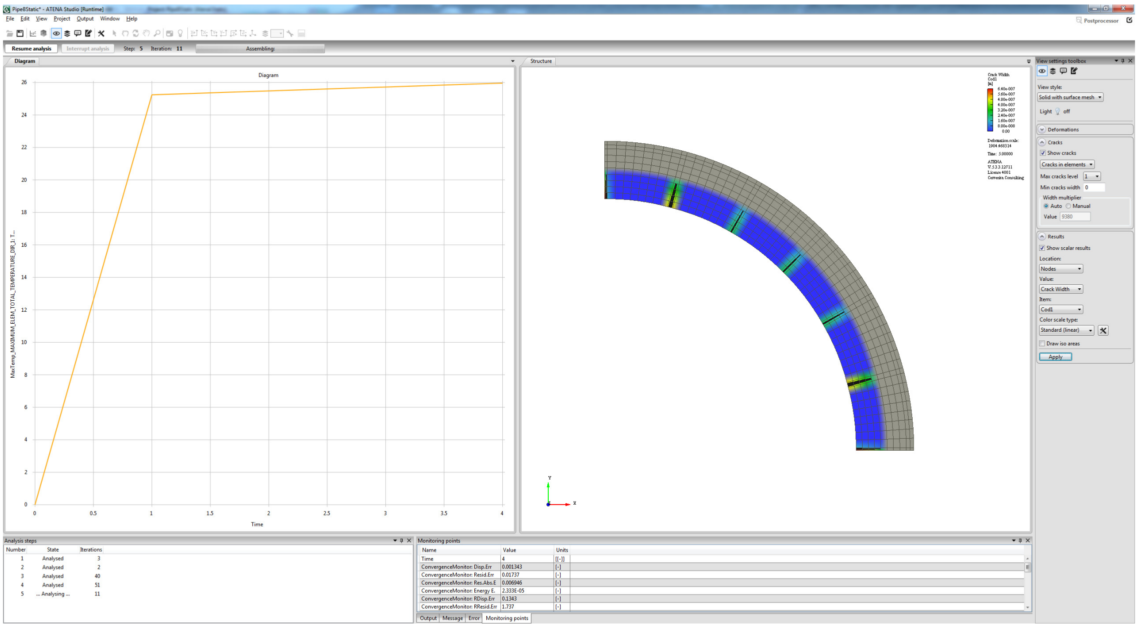


Figure 4-12: Execution process of stress analysis in AtenaStudio showing the crack opening displacements.

After the completion of the analysis, the AtenaCalculation subdirectory of PipeBStatic.gid contains the following files:

PipeBStatic.inp	ATENA input file created by **GiD** and used by **AtenaStudio**
PipeBStatic.00xx	Binary result files created by **ATENA** during the stress analysis.

4.1.4 Post-processing

Post-processing can be done either in **AtenaStudio, AtenaWin**, **GiD**, or **ATENA 3D**. The most recommended option is post-processing in AtenaStudio. More about this post-processing you can find in ATENA Program Documentation Part 12 - User's Manual for ATENA Studio

4.1.4.1 Postprocessing in GiD

To be able to post-process the results in **GiD**, the result quantities must first be made available by selecting them in the **Data | Problem Data | Post Data** dialog, see Figure 4-13.

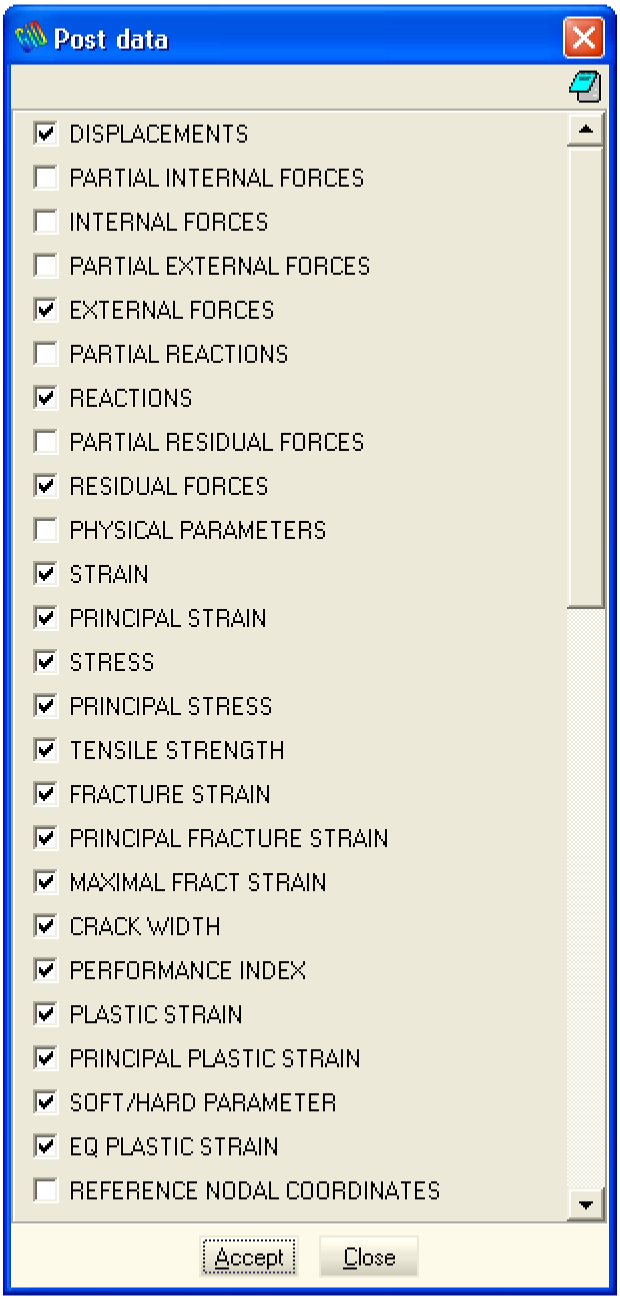
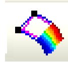


Figure 4-13: Selecting results for post-processing in GiD.

The menu command **ATENA | GiD Post-processing** or the  icon toggles **GiD** between pre- and post-processing. A warning about non-existing .res file may appear, then a console window is started, and the results are converted into a format readable by **GiD**, see Figure 4-14. The conversion can take a few minutes depending on model size, the number of load steps, and the number of quantities selected for post-processing. The results are stored in the *AtenaResults.flavia.res* file in the *AtenaCalculation* respectively *AtenaTransportCalculation* directory for static analysis respectively transport analysis. This file can be opened in **GiD** by the **File | Open** command (see Figure 4-15).

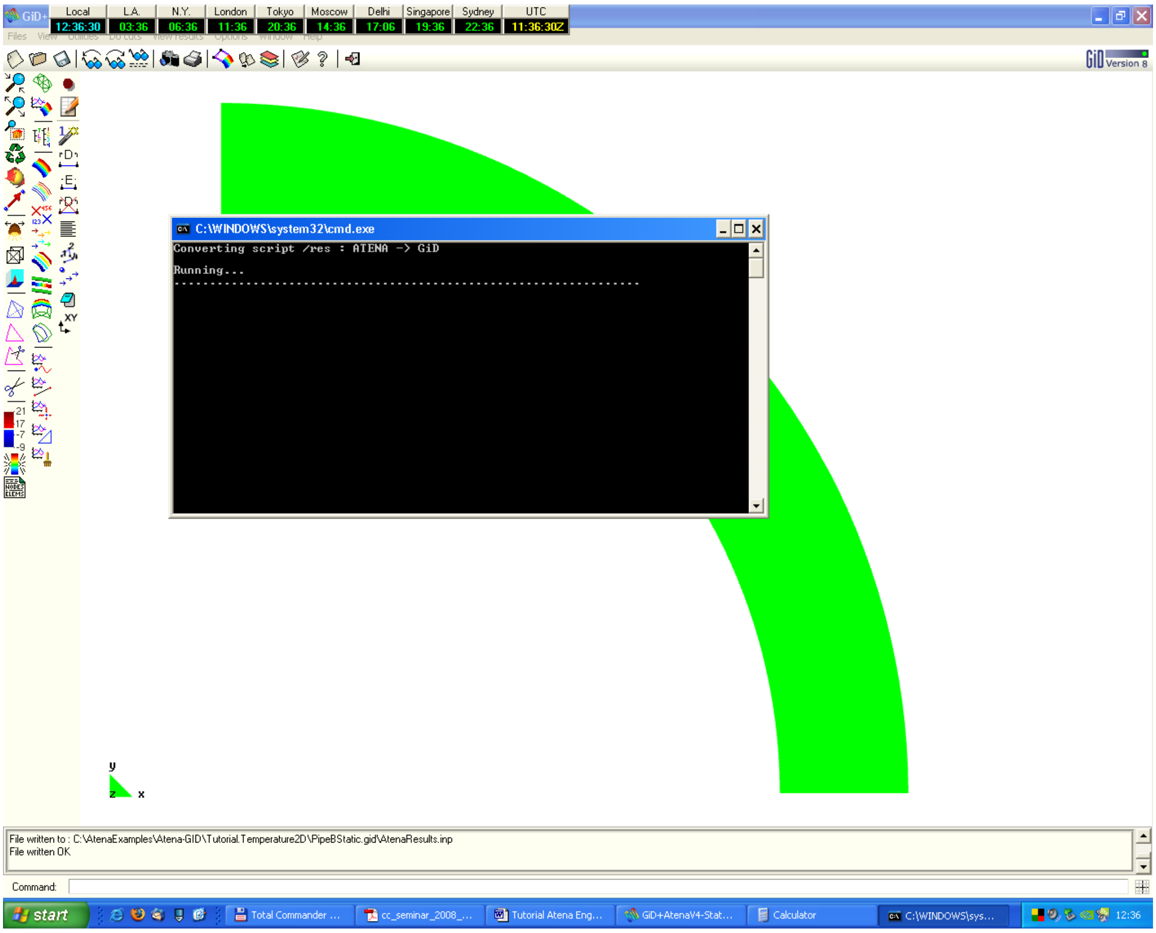


Figure 4-14: Importing results for post-processing in GiD.

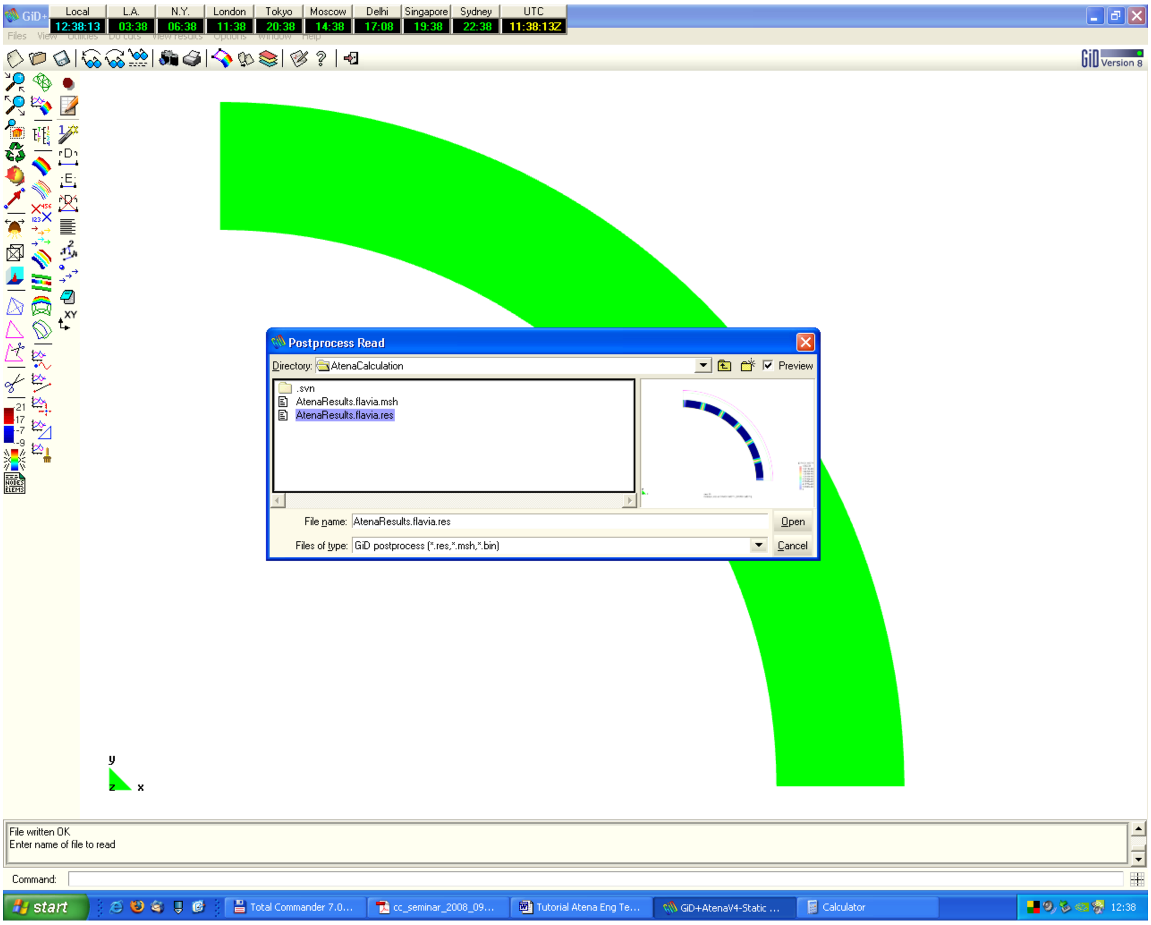
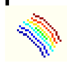


Figure 4-15: Opening results for post-processing in GiD.

The results can then be post-processed. Figure 4-16 shows crack width as contour lines, which can be selected by the menu command **View results | Contour lines | CRACK WIDTH | COD1**. The command can also be accessed from the  icon.

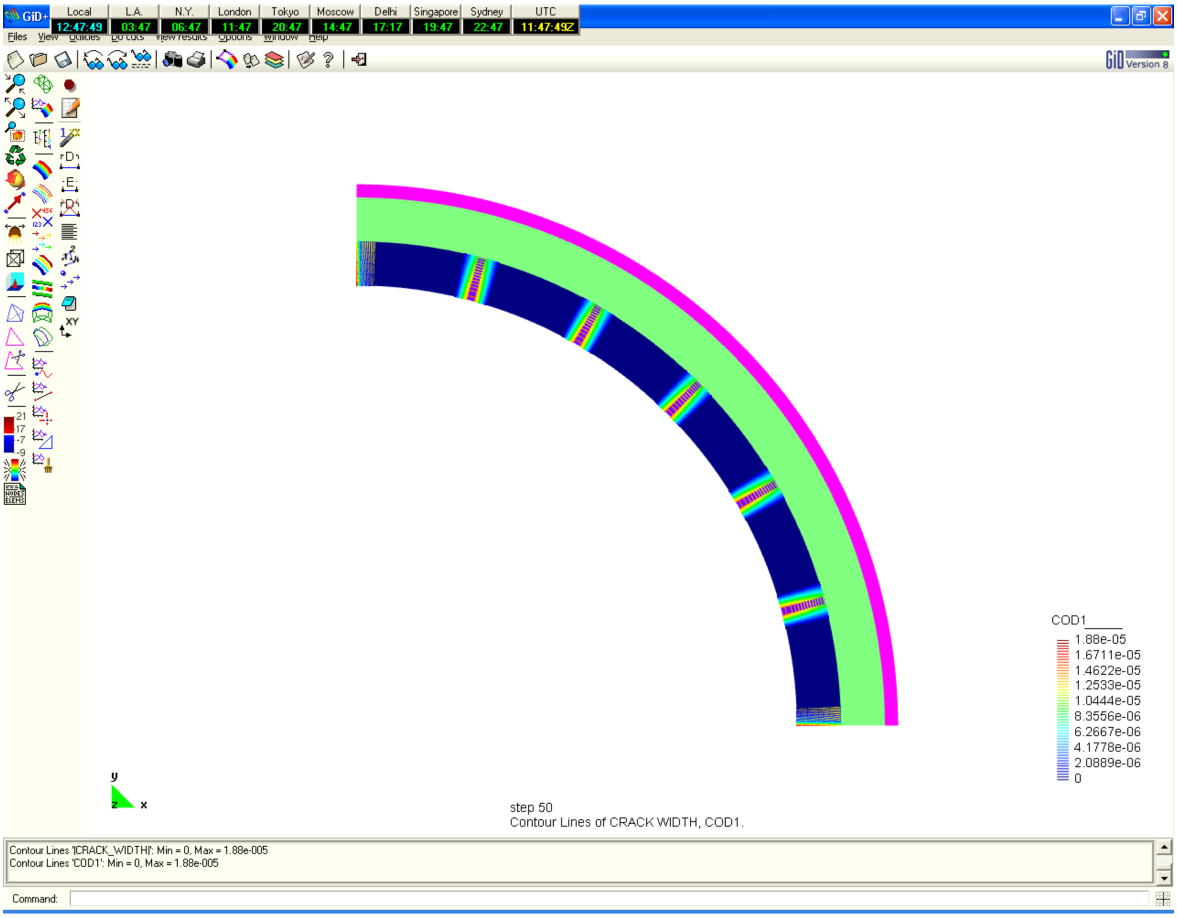


Figure 4-16: Displaying crack opening displacement isolines in GiD.

4.1.4.2 Postprocessing in ATENA 3D

The highest user comfort for post-processing is provided by **ATENA 3D**. After executing **ATENA 3D** the result file for each increment can be loaded into the program by using the menu **File | Open other | Results by step (*) ...** This command activates a dialog that can be used to load ATENA binary files with results to ATENA 3D post-processor. When the user finishes loading the needed result file and closes the dialog, ATENA 3D post-processor is automatically started. It should be noted that it is possible to open only binary result files that come from the same analysis. For further details about post-processing, the user should consult Part 2-2 of ATENA user's manual [2].

4.1.5 Conclusions

This tutorial provided a step-by-step introduction to performing combined thermal and stress analysis using **ATENA** software with GiD preprocessor. The tutorial involves an example of an axisymmetric pipe heated from the outside from 25 to 37°C in 10 hours. Then the heating remains constant for another 8 hours and, after that, is cooled back to 25°C in 6 hours.

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

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

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

4.2 Heat and Moisture Transport Analysis incl. Hydration

This Section brings an example of heat and moisture analysis within ATENA-GiD framework. Its primary goal is to demonstrate how concrete hydration can be considered in the analysis. Both the development of hydration heat and moisture consumption due to hydration are accounted for. The analysis also shows use of **FIRE_BOUNDARY** load condition. It is employed to simulate the radiation of generated heat within the structure to the ambient air.

Heat and moisture transport analysis is carried out to analyze segment 4B of the bridge over the Oparno valley, see Figure 4-17. The bridge is located on the highway between Prague and Dresden. The segment length is 5.6 m, width 7.0 m, and height 2.2 m and falls in the category of the mass concrete element. (That's why the interest in hydration heat!) The casting took place in August 2009.

The multiscale model is used. The microscale analysis of the evolution of hydration heat was accomplished by CEMHYD3D model. Having the results, the affinity hydration model has been calibrated, and the calculated material parameters were applied in the L7 transport material model in ATENA-GiD.

The main problem of casting arches of the bridge was the development of considerable hydration heat. Therefore, at early times the arch's segments had to be cooled down by circulating water in six steel pipes. The cooling to mitigate temperature gradients in the segment was turned off after roughly 40 hours after casting. Figure 4-18 shows the temperature field at 0,20,40,100 hours of the hydration time.

The heat transfer coefficient was estimated as 10 $W/m^2/K$ on the surface of the beam, and the air temperature was assumed constant at 25 ^{o}C. The casting took place during the summer season when high ambient air temperatures contribute to excessive temperatures in the beam. A prescribed temperature 17 °C was imposed on all inner nodes which were in contact with a cooling pipe. After approximately 40 hours, the cooling nodes were incorporated in a vector of unknown temperature field and computed. The results of the analysis show that due to the cooling, it was possible to keep the concrete temperature well below 90°C. This value is considered to be the critical temperature, after which the concrete properties may start to deteriorate and may impair the quality and durability of the structure.

Figure 4-17 Arches of the bridge over Oparno valley. The simulated segment is approximately above the scaffolding.

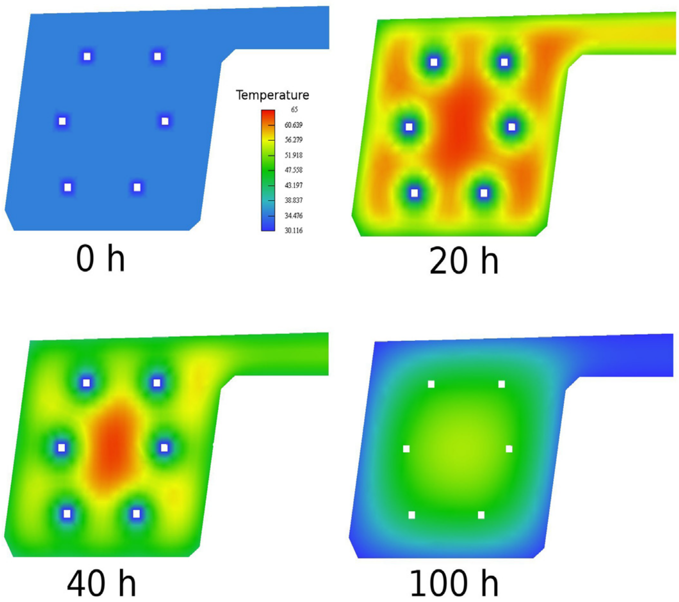


Figure 4-18 Temperature field in the left symmetric part of the beam. The dimensions of the displayed cross-section are 3,5x2,17 m. The beam is cooled down by the water.

Figure 4-19 validates the simulation and shows a relatively good match between the calculated and measured temperature at particular times. Nevertheless, one has to mention that several details regarding conditions of the casting process and other important data were unknown, and they were just estimated in the analysis. This applies particularly to a detailed description of the sequence of the casting procedure.

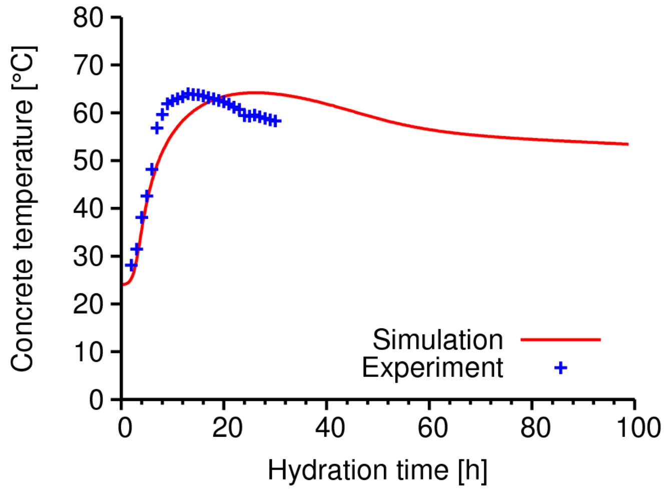


Figure 4-19 Validation of multiscale model on the segment 4B of the Oparno bridge. The cooling was turned off at 40 hours in the simulation.

The above paragraphs bring a description of the presented problem of the Oparno bridge from the engineering point of view, including a brief validation of the calculated results. The next part of this Section explains the sample analysis from the point of ATENA-GiD modelling. It concentrates mainly on the input of transport material parameters used by the L7 material model. The example you can find at Public%**Documents\ATENA Examples\Science\GiDT\utorial.Temperature2D\BridgePierHydratation.gid**

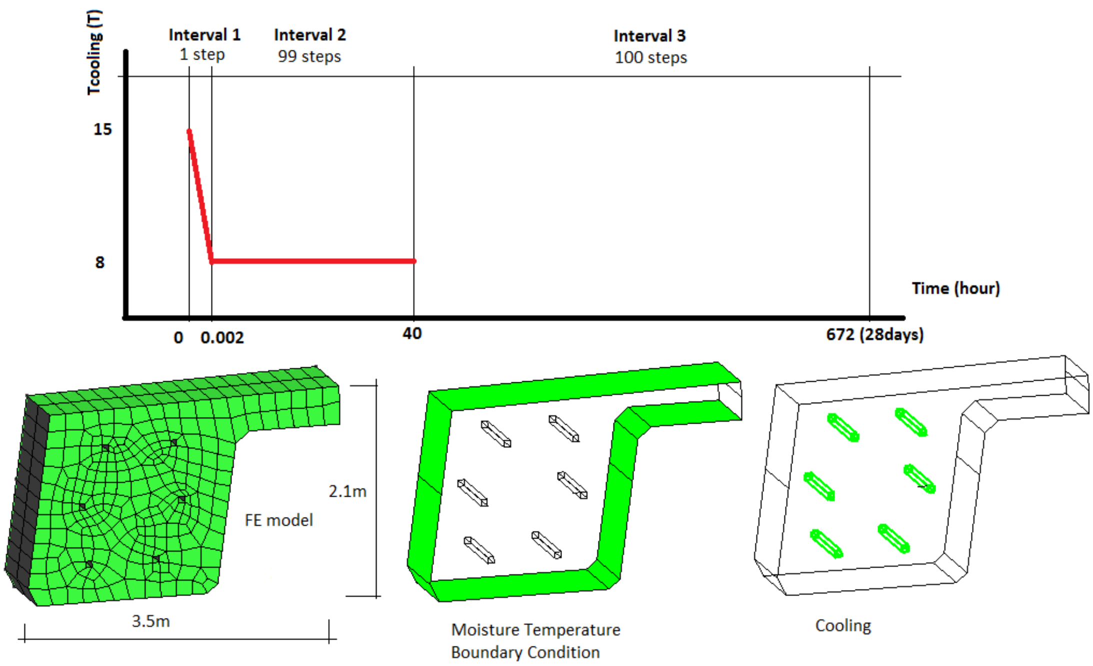


Figure 4-20 Description of example

5. LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2016

[2] ATENA Program Documentation, Part 2-1 and 2-2, ATENA 2D and 3D User's Manual, CERVENKA CONSULTING, 2016

[3] ATENA Program Documentation, Part 3, ATENA 2D Examples of Application, CERVENKA CONSULTING, 2010

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2016

[5] ATENA Program Documentation, Part 7, AtenaWin Manual, CERVENKA CONSULTING, 2015

[6] ATENA Program Documentation, Part 8, User's Manual for ATENA-GiD Interface, CERVENKA CONSULTING, 2016

[7] ATENA Program Documentation, Part 12, User's manual for ATENA Studio, CERVENKA CONSULTING, 2016

[8] ATENA Program Documentation, Part 11, Troubleshooting manual, CERVENKA CONSULTING, 2016

[9] DuraCrete, Probabilistic performance based durability design of concrete structures, The European Union–Brite EuRam III, Final technical report of Duracrete project, Document BE95□1347/R17, 2000

[10] EN ISO 9223, Corrosion of Metals and Alloys-Corrosivity of Atmospheres—Classification, Determination and Estimation, 2012

[11] J. A. Gonzales, C. Andrade, C. Alonso, and S. Feliu, Comparison of Rates of General Corrosion and Maximum Pitting Penetration on Concrete Embedded Steel Reinforcement, Cement and Concrete Research, 25(2), pp. 257-264, 1995

[12] K. Hájková, V. Šmilauer, L. Jendele, and J. Červenka, Prediction of reinforcement corrosion due to chloride ingress and its effects on serviceability, Engineering Structures, 174, pp. 768-777, 2018

[13] ATENA Program Documentation, Part 4-6, ATENA Science – GiD Tutorial, CERVENKA CONSULTING, 2016

[14] ATENA Program Documentation, Part 4-7, ATENA Science – GiD FRC Tutorial, CERVENKA CONSULTING, 2016

[15] ATENA Program Documentation, Part 4-8, ATENA Science – GiD Construction Process, CERVENKA CONSULTING, 2016

[16] ATENA Program Documentation, Part 4-9, ATENA Science – GiD Strengthening of concrete structures, CERVENKA CONSULTING, 2016

[17] K. P Juhász, Modified fracture energy method for fibre reinforced concrete, Fibre Concrete 2013, Prague, Czech Republic, pp. 89-90, 2013