Červenka Consulting, s.r.o
Na Hrebenkach 55/2667
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

ATENA Program Documentation
Part 8

User's Manual for ATENA-GiD Interface

Written by
Vladimír Červenka, Jan Červenka,
Zdeněk Janda, and **Dobromil Pryl**
Prague, December 10th, 2015

CONTENTS

1 INTRODUCTION **1**

2 OVERVIEW **3**

2.1 Working with GiD 3

2.2 Limitations of ATENA-GiD Interface 3

3 GID INSTALLATION AND REGISTRATION **5**

3.1 GiD Network Floating Licenses 6

4 ATENA-GID INSTALLATION **7**

4.1 Manual Installation of the ATENA-GiD Scripts 7

5 ATENA - SPECIFIC COMMANDS **9**

5.1 Problem Type 9

5.2 Conditions 9

5.3 Materials 21

5.3.1 Solid Concrete Material 26

5.3.2 Shell Material 36

5.3.3 Beam Material 42

5.3.4 Reinforced Concrete 45

5.3.5 1D Reinforcement Material 46

5.3.6 Interface Material 52

5.3.7 Spring material 57

5.3.8 The Material Function 58

5.3.9 Material from file 59

5.4 Interval Data - Loading History 59

5.4.1 Fatigue 63

5.5 Problem Data 66

5.6 Units 69

5.7 Finite Element Mesh 70

5.7.1 Notes on Meshing 70

5.7.2 Finite Elements for ATENA 71

5.8 ATENA Menu 75

6 STATIC ANALYSIS 76

7 CREEP ANALYSIS (AND SHRINKAGE) 78

7.1 Boundary Conditions and Load Cases Related Input 79

7.2 Specific Creep Boundary Conditions 80

7.3 Material Input Data 80

8 TRANSPORT ANALYSIS (MOISTURE AND HEAT) 84

8.1 Material Input Data 84

8.1.1 Material CCTransport (CERHYD) 84

8.1.2 Material Bazant_Xi_1994 (only included for backward compatibility of old models).... 91

8.2 Other Settings Related to Transport Analysis 92

8.3 Specific Transport Boundary Conditions 95

9 DYNAMIC ANALYSIS 98

9.1 Specific Dynamic Boundary Conditions 100

10 POST-PROCESSING IN ATENA-GID 104

11 USEFUL TIPS AND TRICKS 112

11.1 Export IXT for ATENA 3D Pre-processor 112

12 EXAMPLE DATA FILES 114

13 CALCULATION OF ATENA IDENTIFICATION NUMBERS 116

REFERENCES 118

1 INTRODUCTION

Program **GiD** can be used for the preparation of input data for ATENA analysis. The program **GiD** is a universal, adaptive and user-friendly graphical user interface for geometrical modelling and data input for all types of numerical simulation programs. It has been developed at CIMNE (The International Center for Numerical Methods in Engineering, http://www.cimne.upc.es) in Barcelona, Spain. When using **GiD**, for some graphic cards it may be necessary to switch off "graphical acceleration".

Several scripts are created, which enables to interface **GiD** with **ATENA**. Selecting an appropriate problem type in the **GiD** environment activates these scripts:

Problem types are compatible with **GiD** ver.7.7.2b and newer, version 10 or 11 is recommended):

- ATENA/Static, - static 2D and 3D analysis
- ATENA/Creep, - creep 2D and 3D analysis
- ATENA/Transport, - transport 2D and 3D analysis
- ATENA/Dynamic - dynamic 2D and 3D analysis

These problem types make it possible to define a finite element model within **GiD** including specific data needed for **ATENA** analysis. **ATENA Studio** [5] can be launched directly from **GiD**, and the non-linear analysis can be performed. Visualization of **ATENA** results is also possible in **GiD**, but it can be done also in the Pre/Post-processor of **ATENA 3D** [3], which is a powerful **ATENA** postprocessor. However, this option is available only if **ATENA** Engineering is installed on your computer. The recommended post/processing environment is **ATENA Studio** [5].

The problem types with the label **ATENA** can be used with **ATENA** version newer than 5.0.0. These problem types support **ATENA** analysis with two- and three-dimensional models (including axi-symmetrical models). In addition it is possible to perform stress, creep, thermal (i.e. transport) and dynamic analyses.

A demo version of **GiD** is limited to 3000 elements (or 1010 nodes). It can be downloaded free of charge from http://www.gidhome.com/, or from our web pages www.cervenka.cz .

This document describes the way how **GiD** can be used to generate data for **ATENA** analysis. The emphasis is on **ATENA**-oriented commands. More details about the general use of **GiD** for the development of the geometric model can be found in the **GiD** documentation.

2 OVERVIEW

2.1 Working with GiD

The procedure of data preparation for **ATENA** analysis with the help of **GiD** can be summarized in the following work sequence:

- Select one of the problem types for **ATENA**.
- Create a geometrical model.
- Impose conditions such as boundary conditions and loading on the geometrical model.
- Select material models, define parameters and assign them to the geometry.
- Generate finite element mesh.
- Change or assign supports and loading conditions to the mesh nodes (if necessary).
- Change or assign materials to individual finite elements (if necessary).
- Create loading history by defining interval data.
- Execute finite element analysis with **ATENA Studio** or **AtenaConsole**.

Some of the above actions are general and not dependent on **ATENA** (geometry definition, finite element mesh), while the others are more or less specific for **ATENA** (material parameters, solution methods). This manual is focused on the later features.

The description of the general features of **GiD** (menu items **View**, **Geometry**, **Utilities**, etc.) can be found in the **GiD** documentation. There is an extensive online help available in **GiD**, which is accessible from the menu **Help** as well as some online tutorials. For example the information how to create geometry is not included in this manual, and can be found in the **GiD** menu **Help | Contents | Geometry**.

In the **ATENA**-specific dialogs (materials, conditions, etc.), help is also available with detailed description and additional information by clicking the right mouse click or the **help** icon .

The practical aspects of the **GiD** use can be exercised on the examples described in Chapter 12. It is also recommended to go through the **ATENA-GiD** Tutorial [6] before starting with one's own modelling.

2.2 Limitations of ATENA-GiD Interface

It should be noted that **ATENA-GiD** interface supports the most common features of the **ATENA** software. However, the direct modification of the **ATENA** input file may be sometimes useful, and it allows the user to exploit all the features of the **ATENA** software. Detailed syntax of all **ATENA** commands is described in the **ATENA** documentation [4]. This **ATENA** command file typically with the extension ".inp" is generated by **GiD**, but it is a readable text file that can be further modified manually if needed.

3 GiD Installation and Registration

GiD installation can be performed during **ATENA** installation or **GiD** can be separately downloaded from the **GiD** developer at http://www.gidhome.com/.

In order to use **GiD** without the limitations of the trial version (30 days or, e.g., 1000 nodes), it is necessary to obtain a user license by purchasing the program from **GiD** distributors in your country, from Cervenka Consulting, or directly from the **GiD** web page http://www.gidhome.com. With a valid license number, it is necessary to obtain a password for the computer (please note the difference between *GiD License Number* and *GiD Password*), on which the **GiD** will be operated, or a USB flash disk (recommended). The same procedure is also used to obtain a free 30-days trial password.

The registration process is activated by starting **GiD** and proceeding to the menu **Help | Register**. Please understand **GiD** needs to be run with Admin rights ("Run as Administrator" once) to allow storing the registration information for next sessions. It should also be noted that there are two possibilities how to operate the **GiD** program. Normally, the **GiD** password is specific to a certain PC configuration. In this case, the full version of **GiD** can be operated only on this computer. Alternatively, it is possible to license **GiD** to a portable USB memory flash disk (please note the HASP USB key for **ATENA** is NOT a memory flash disk). Then, it is possible to operate **GiD** on every computer, to which this registered flash disk is attached. The license price for USB protection is slightly different than the one for PC protection, so it is important to choose this option during the program purchase. If the USB protection is desired, it is necessary to attach the USB flash disk to the computer[1]. Then, the item **Help | Register** should be selected. If a supported flash disk is attached to the computer the following dialog appears, in which the proper choice of the protection mechanism is to be selected. Please, make sure that the correct choice is made here. It is difficult to change the protection method in the future.

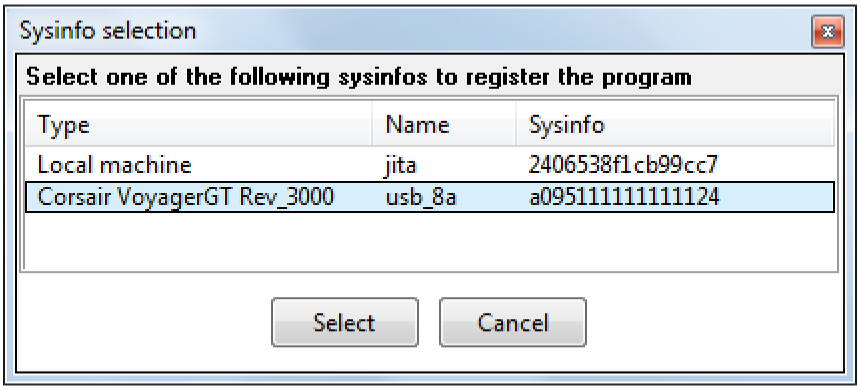


Fig. 3-1: Choice of USB or PC protection

After making the appropriate selection and clicking the button Select, the following dialog appears depending on the previous choices:

[1] Note the HASP hardware keys for **ATENA** do NOT work as a flash disks; on the other hand, most common USB memory flash disks can be used to register **GiD**

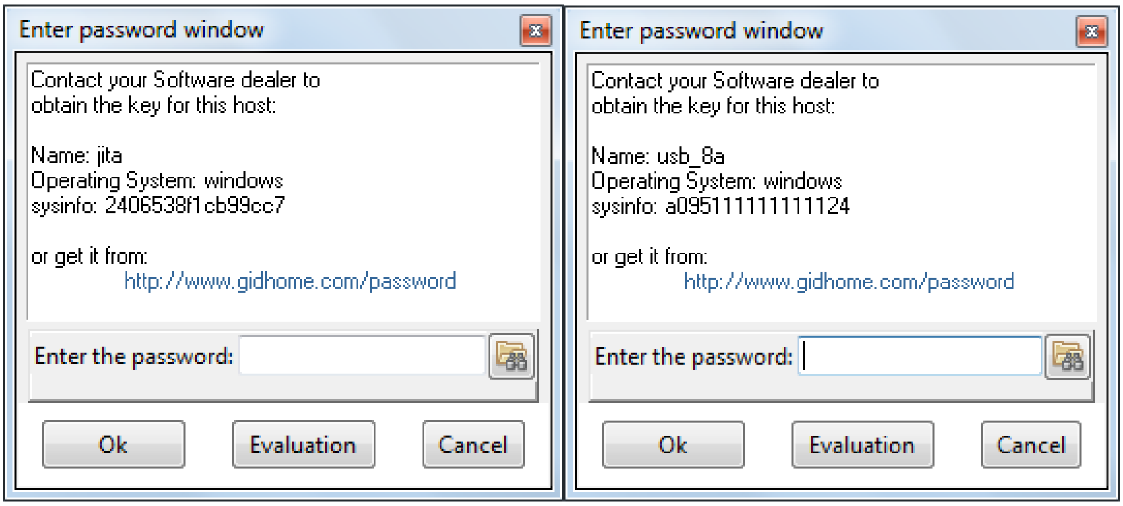


Fig. 3-2: GiD register window (PC protection left, USB protection right)

If **GiD** have been registered previously (the same official version of **GiD**), the password can be reloaded by clicking  and selecting the folder where the old password is.

The new password is obtained by clicking the web address or pasting it into the web browser. In this website, the user then should follow the instructions to obtain the password, which should be typed or copied into the bottom line in the above dialog (do NOT enter the *GiD License number* into the box for *Password*). In order to obtain the final password, the user will need to provide some information such as for instance the email address. The most important information, however, are the “Name”, “Operating System” and “sysinfo”, as shown in Fig. 3-2. Please also note that the “Name” refers to the label of your USB flash disk or your PC hard drive. It is not your personal name.

After registering either a permanent or temporal password, it is possible to generate and post-process an unlimited number of nodes and elements.

3.1 GiD Network Floating Licenses

If you have a network floating license for **GiD**, install **PasServer** on the computer that will work as license server. Follow the instructions from the **GiD** web http://www.gidhome.com/documents/passerver/Tabla%20de%20Contenidos to get the vendor key based on the “sysinfo” corresponding to the server and your network license number and enter it in the **PasServer**. When starting **GiD** on your workstation, enter the IP address of the **PasServer** in the password box. Make sure no firewall is blocking the communication between **GiD** and the **PasServer**.

4 ATENA-GiD Installation

The installation of **ATENA-GiD** interface can be performed using the **ATENA** installer. Please make sure the **ATENA-GiD** interface is selected for installation. During this process, the user needs to confirm the location of the **GiD** directory.

New problem types related to **ATENA** should appear in the **GiD** menu. The problem types are available under the **GiD** menu **Data | Problem type**. If the **ATENA** problem types are not shown there, most likely, you have installed a new **GiD** version after **ATENA** has been installed, or have multiple **GiD** versions installed, and have installed the **ATENA-GiD** scripts into another one than you are using. To fix the issue, you can re-run the **ATENA** setup and select the **ATENA-GiD** interface to be installed for the **GiD** version you wish to work with.

4.1 Manual Installation of the ATENA-GiD Scripts

Alternatively, the **ATENA-GiD** interface can be also installed manually as it is described in the following paragraphs.

1. Download the **ATENA-GiD** version corresponding to your **ATENA** version from the **Downloads** section of www.cervenka.cz and unpack the archive to your hard disk.

1.a You can also find the scripts in the installation directory of another **GiD** version, e.g., if you have just installed a new **GiD** version and were using **ATENA** with an older **GiD** version previously.

2. Copy the Atena directory tree into the Problem types directory of the **GiD** version you like to use with **ATENA**. On most computers, the **GiD** is installed in the directory:

C:\Program Files\GiD\GiDx.x

e.g., if you use GiD 10.0.9, copy the Atena tree into

C:\Program Files\GiD\GiD10.0.9\problemtypes\Atena

3. Start **GiD** and check if the new problem types appear in the **GiD** menu.

In order to be able to directly launch ATENA analysis and ATENA post-processing directly from GiD the following environmental variables are to be defined on your computer:

32bit

SET AtenaWin="%programfiles%\CervenkaConsulting\AtenaV5\AtenaWin.exe"

SET AtenaConsole="%programfiles%\CervenkaConsulting\AtenaV5\AtenaConsole.exe"

SET AtenaStudio="%programfiles%\CervenkaConsulting\AtenaV5\AtenaStudio.exe"

SET AtenaResults2GiD="%programfiles%\CervenkaConsulting\AtenaV5\A2G.exe"

64bit

```
SET
AtenaWin64="%programfiles%\CervenkaConsulting\AtenaV5x64\AtenaWin64.exe"
SET
AtenaConsole64="%programfiles%\CervenkaConsulting\AtenaV5x64\AtenaConsole64
.exe"
SET
AtenaStudio64="%programfiles%\CervenkaConsulting\AtenaV5x64\AtenaStudio.exe"
```

Where the path should point to the appropriate location, where the programs are installed.

5 ATENA - SPECIFIC COMMANDS

5.1 Problem Type

The program **GiD** is a general-purpose pre- and post-processing tool for variety of numerical problems (and analysis software). In this menu it is possible to define a problem type, which in our case is **ATENA** analysis. This is done by selecting for example the menu item **Data | Problem type | Atena | Static** as shown in Fig. 5-1. By this command, **GiD** is configured to create data for analyses, which are compatible with **ATENA** input format (units, materials, conditions, etc.). The data resulting from the **GiD** modelling will be later transferred to **ATENA** via an input file usually called *name.inp*.

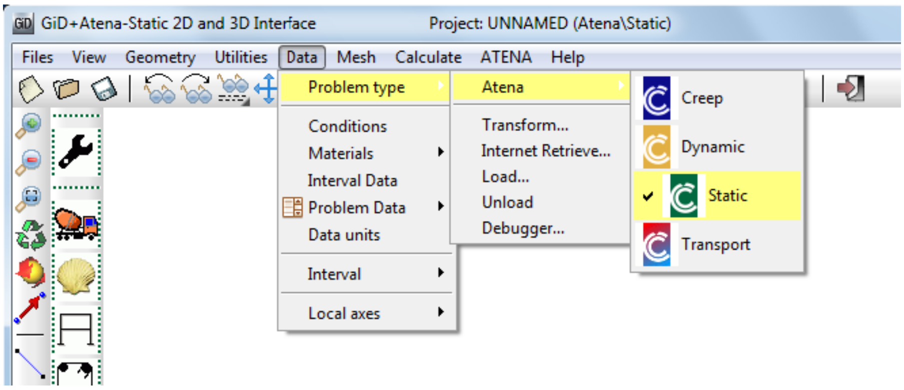
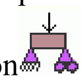
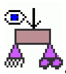


Fig. 5-1 Problem type menu.

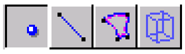
The problem type definition must be done before starting input of any data. Executing this command later may result in the loss of some of the existing data.

5.2 Conditions

The supports and loading conditions for **ATENA** can be defined in a way, which is compatible with **ATENA** through the menu **Data | Conditions**, Fig. 5-2, left or by icon. You can view all currently defined conditions in current interval by clicking to icon .It should be noted that the loading and boundary condition definition is closely related to the definition of Interval data (see Chapter 5.4). The specified boundary conditions are always defined in the current interval. Information about global and local coordinate systems for each element load you find in Theory manual [1] in chapter 3.14.

Loads are incremental in **ATENA**, with just a few exceptions like fire in transport analysis or ground acceleration in dynamics. In other words, unless you unload (by applying a negative force), the load stays there during the following steps (Intervals). A surface with no condition applied corresponds to zero increment of external forces.

The conditions can be assigned to four kinds of geometrical objects: geometric points (finite element nodes), lines (finite element edges), surfaces and volumes (finite elements). The object dimension is selected by choosing one of the buttons

. For each geometric entity an appropriate list of possible conditions can be unfolded and a required type of condition can be selected. An example of the point condition is shown in Fig. 5-2. For each condition the appropriate parameters can be defined as shown in Fig. 5-2, right.

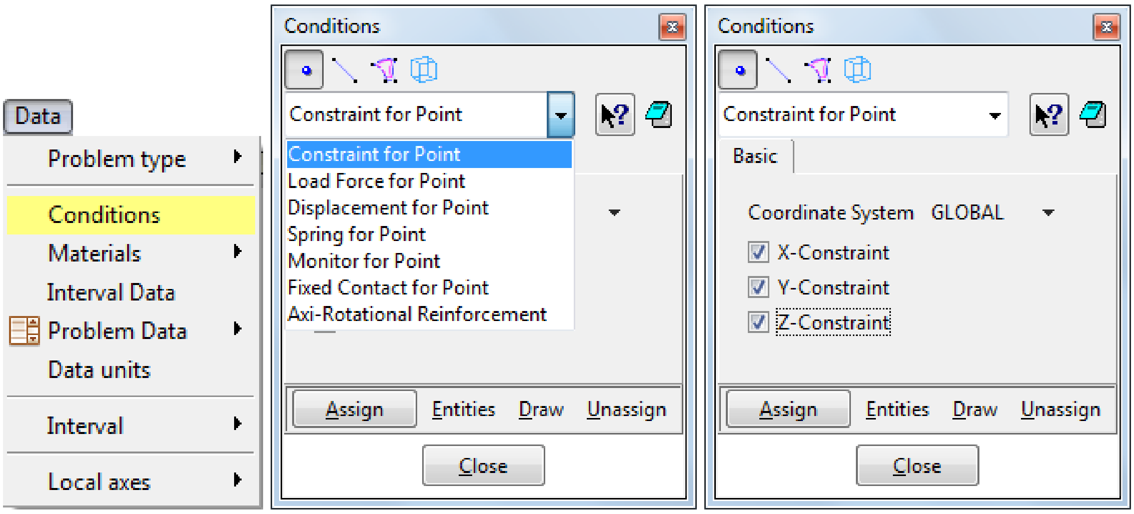


Fig. 5-2 Conditions: menu, list at Point, applied at Point.

At the bottom of the conditions dialog the following buttons are available:

Assign - The target of assignment command depends on the condition type. In case the geometry is displayed, then geometrical objects (point, line, surface) can be selected and condition can be assigned to these entities. In case the finite element mesh is displayed, the condition can be assigned to elements or nodes. If you don't know what should be selected, look at command line. There is always a hint what kind of action is required from the user.

Fig. 5-3 Hints at the Command Line at the bottom of GID Window

Entities – Shows a list of entities with assigned conditions.

Draw – Display of assigned conditions. There are various visualization modes possible in this command. You can draw all defined conditions or only one. If you use the option draw colors, the entities with this condition are colored and a legend with applied values is shown.

Unassign – Reverse operation. It cancels existing assignment of the selected condition type (for selected or all entities).

If it is necessary to modify the parameters of certain already assigned condition, it has to be first unassigned and created again with the new parameters.

There are certain conditions in the following paragraphs, which are strongly **ATENA** specific.

Constraint - This is a boundary condition for modeling supports and can be defined for point, line and surface. The simplest way how to set the condition is to choose the global coordinate system and select directions to be fixed. The inclined coordinate system enables rotated support conditions.

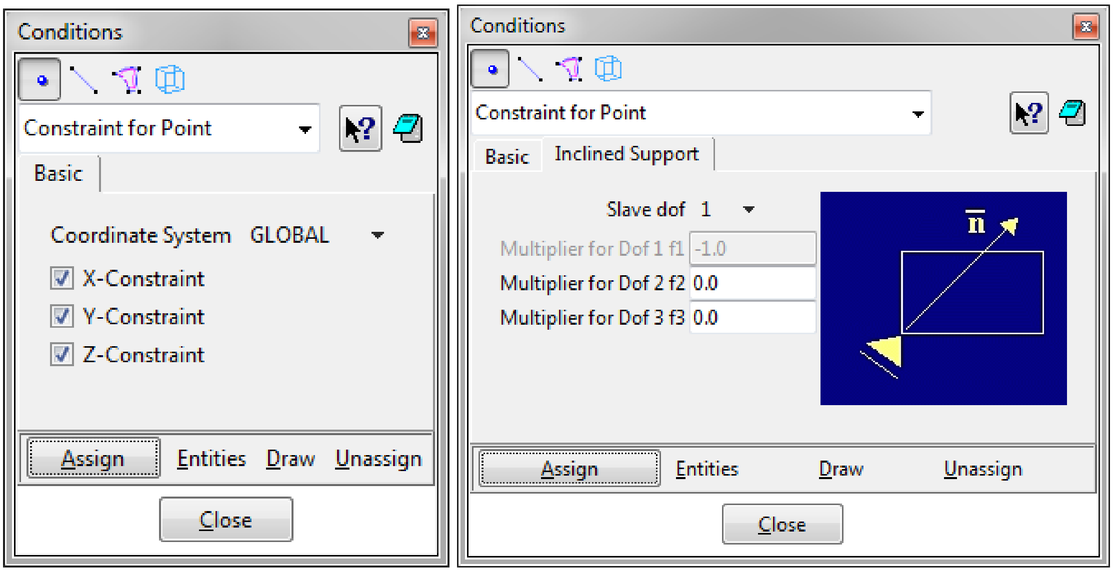


Fig. 5-4 Conditions: Constraint for …

Good to know:

If you use two conditions (of one type) for surfaces (e.g., support in X and support in Y direction), at the edge where they meet only one of them is applied. Therefore, it is necessary to correct the condition manually by defining the corresponding condition also for the line in between (e.g., assigning both X and Y supports to the edge shared by the 2 surfaces - see figure below Fig. 5-5). Similarly, condition for point needs to be applied to each point where different conditions for the line of the same type are intersecting.

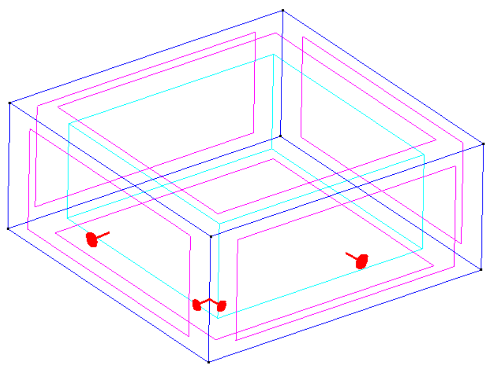


Fig. 5-5: Proper Support Assignment at the Edge of Two Surfaces

Load force - Loading conditions can be prescribed for point, line and surface. When entering the force magnitudes for each component, it is possible to select suitable units. When the **ATENA** input file is created, the load values will be converted to the default unit type (see the menu **Data | Data units**). The value can be entered in several types of units. If the units are changed, the value is recalculated. Load force for point can be defined by three components in each coordinate direction. The loading for line can be prescribed only for 2D elements. Local coordinate system can be used to apply loading normal to the line. The projection can be used for example for the snow or wind load. The loading can be constant or linear. The load force for surface can be obviously defined only for 3D entities. The possible coordinate systems options are similar to the line condition.

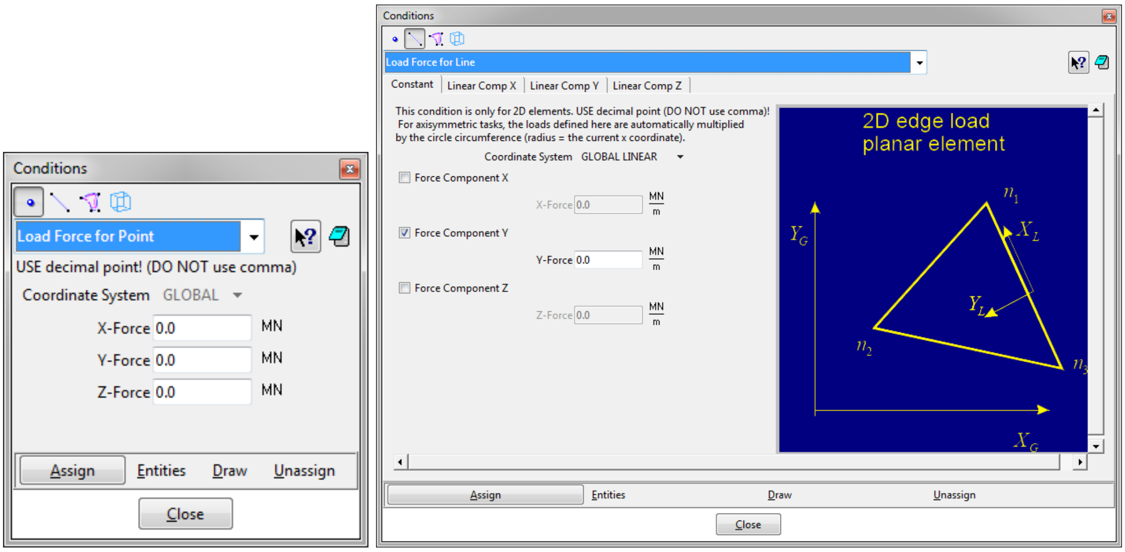


Fig. 5-6 Conditions: Load Force for …

Displacement - This condition can be defined for point, line and surface. The coordinate system is only global and the components are similar as for Load force.

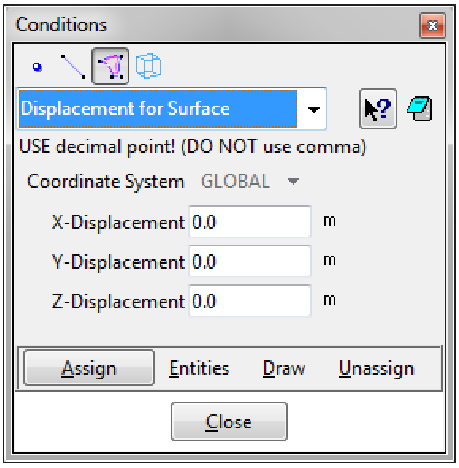


Fig. 5-7 Conditions: Displacement for …

Springs – Spring support can be defined either as conditions (Spring for Point, Spring for Line, Spring for Surface) or as a special layer of line of surface elements along the boundary of the analyzed structure. It is possible to define non-linear spring properties, in this case it is necessary to define the relationship between the force "f" and the relative spring elongation "eps" in the Nonlinear Parameters list. Each spring is defined by its direction and area. If the length of the spring direction vector is 1 and the spring area is also 1, then the "f" and "eps" have the units of force and length. If other values are specified then the "f" has units of stress and "eps" units of strain. The vector defining the spring direction should be oriented away from the line or surface to have the proper meaning of compression and tension.

Important note: Since version 4.3.1, it is recommended to use the special layer of line or surface elements with the spring material. The **Spring for … Conditions** are only available for backward compatibility. Please follow the recommendations in the Help texts of the input dialogs.

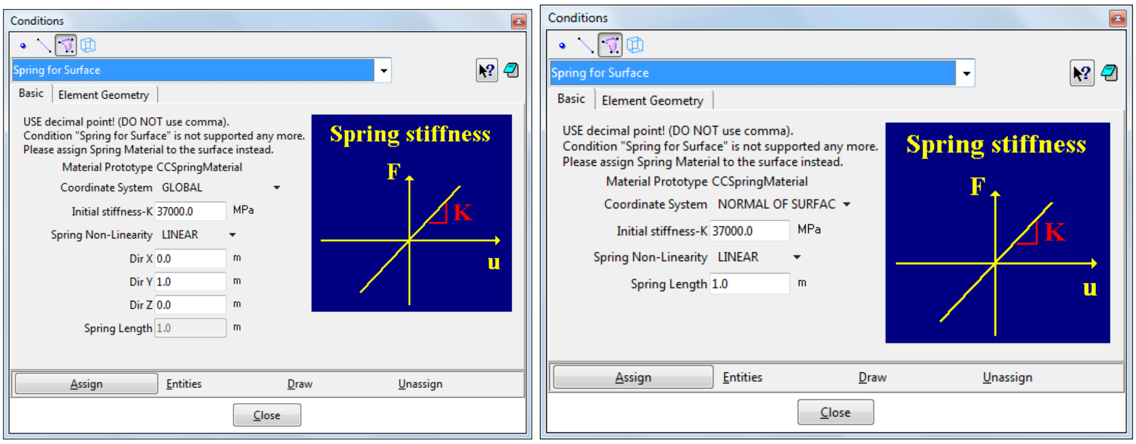


Fig. 5-8 Conditions: Spring for …

For instance in order to define a surface spring with 5kN/m^2 pressure at 15mm displacement:

1. set the spring length to 1m, then 15mm displacement corresponds to relative displacement (elongation/shortening) 0.015 m.

2. set the spring material stiffness to 0.005 [MN] / 0.015 = 0.3333333 MPa ($\sigma = E \times \varepsilon$)

Monitors - It is a special condition that is neither a boundary condition nor a loading; but it makes it possible to record certain quantities during the analysis, such as load-displacement diagrams. It is therefore reasonable to include their definition only in the first Interval data (see Chapter 5.4). The monitors defined in the intervals other than the first one are ignored. It is also possible to enter the global monitors in Problem data dialog (see Section 5.5).

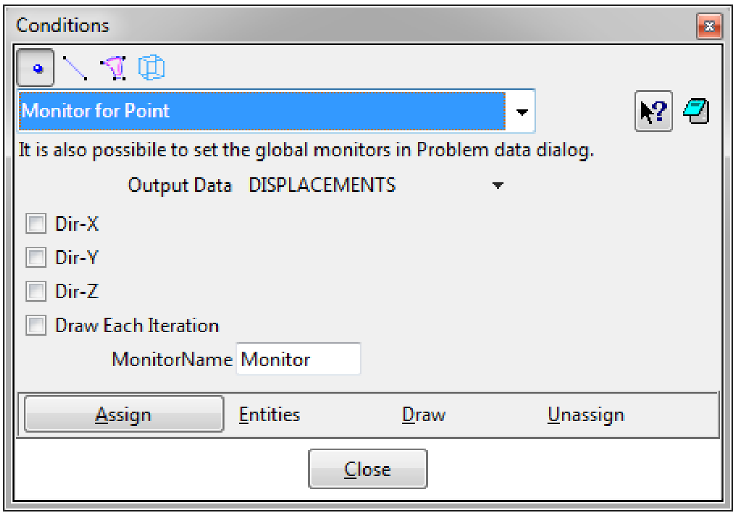


Fig. 5-9 Conditions: Monitor for …

Monitors for Reinforcements – To record values on reinforcement bars and cables, the “Monitor for Reinforcement” condition is to be used instead of the general **Monitor** condition.

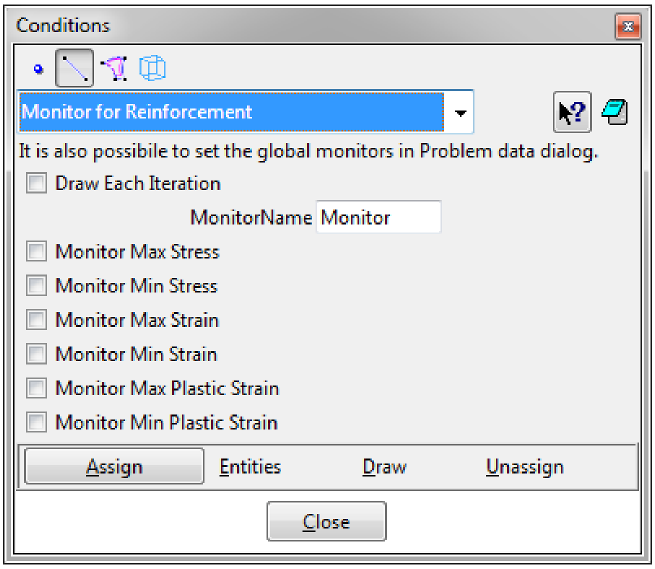


Fig. 5-10 Conditions: Monitor for Reinforcements

Max Monitors – This condition is a special monitor type, which allows users to trace extreme values or sums over some region, e.g., the maximum crack width in a volume or the total reaction from surface support.

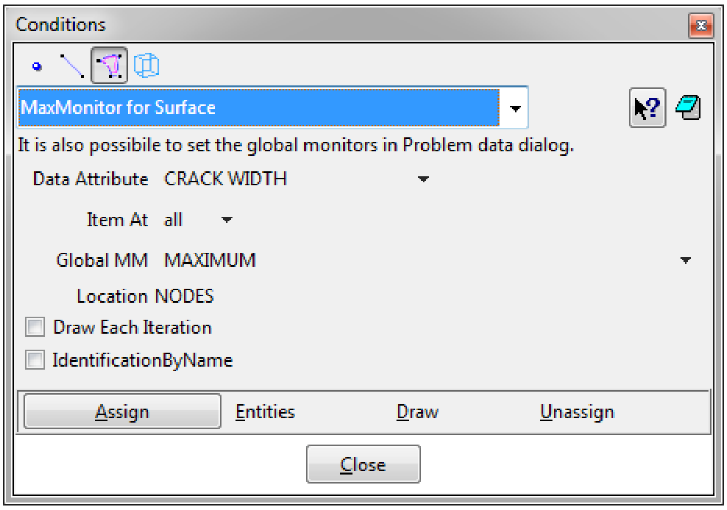


Fig. 5-11 Conditions: Max Monitor for …

Fixed contact – This condition also does not impose any actions on the structure, but it can be used to connect together two parts of the model, which are separated by duplicated entities. You can have multiple Master-Slave connections, identified by different names. Only Master and Slave conditions of the same name are connected together. The meshes on the contact entities do not need to be compatible. ATENA creates special master/slave conditions that enforce the compatibility of displacements.

The side with the coarser mesh (i.e., larger finite elements) should be the Master and the other side (finer mesh, smaller finite elements) the Slave.

The option **Master Slave Distance Manual** from the **Global Options** tab of the **Problem Data** dialog can override the global **Master Slave Distance** value (also defined in **Problem Data**, 5.5). This can be useful when modelling a periodic boundary condition or blocking rotation of a loading plate (or similar) by binding one or more degrees of freedom of two distant points.

Please note **Fixed contact** is different from Interface elements, sometimes also called Contact elements. See Section 5.3.6 for information on Interface (GAP) elements.

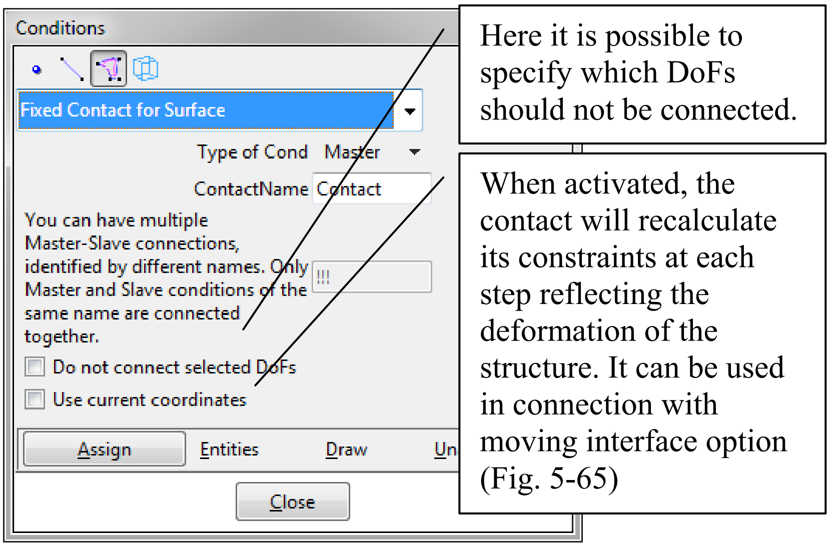


Fig. 5-12 Conditions: Fixed Contact for …

Selection Nodes - This condition can be used for the definition of nodal selections that can be later used by other conditions. Now mainly for experimental use.

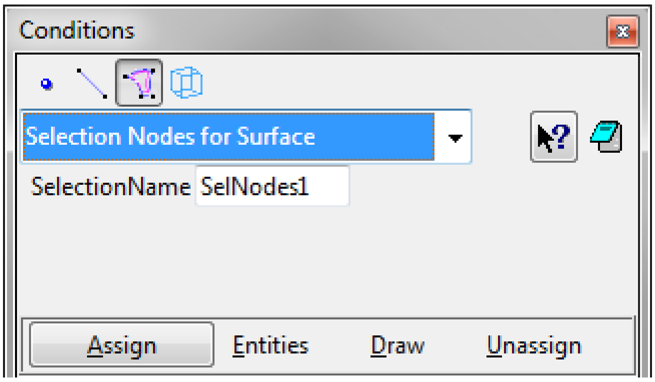


Fig. 5-13 Conditions: Selection Nodes for …

Axi-rotational reinforcement (condition for point) - This condition is aimed mainly for modelling of structural circumferential reinforcement in axi-symmetric analysis. The material is assigned to reinforcement by this condition. The CCCircumferentialTruss has one node only. For proper function of this condition it is necessary to set (Mesh -> Mesh criteria -> Mesh -> Points) to all Points which we want to use with this condition. Look at the example Tutorial.Static2D\axisym.gid to better understand this problem.

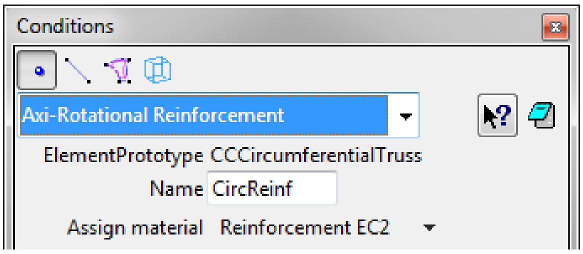


Fig. 5-14 Conditions: Axi-Rotational Reinforcement

Weight - The weight can be defined for reinforcement line, 2D elements surface and volume. Typically is used to consider dead weight load, because the dead load is not analysed automatically in **ATENA**.

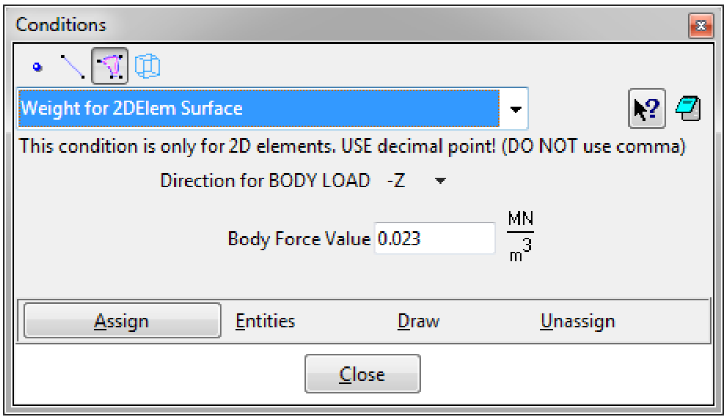


Fig. 5-15 Conditions: Weight for …

Temperature – This condition applies a temperature increment. This way, only a simple constant temperature or a linear gradient over the line/surface/volume can be applied as a load in static analysis. For more complex temperature fields, use the Transport analysis module (see Chapter 8).

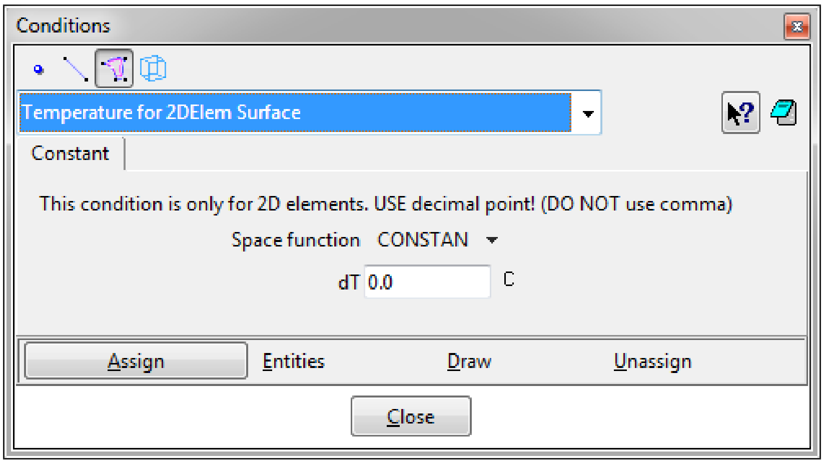


Fig. 5-16 Conditions: Temperature for …

Initial Strain – This condition is used to apply pre-stressing or shrinkage. In both cases, negative strain values are to be assigned.

In the case of pre-stressing, the required value of prescribed initial strain can be calculated from the applied pre-stress σ_p and the elastic modulus E of the reinforcement as $\varepsilon_{ini} = \sigma_p / E$. You may need to correct (increase) the calculated strain to compensate for the losses due to the elastic deformation of the structure resulting from the pre-stress applied (or add an additional “compensation” interval to apply the lost pre-stress).

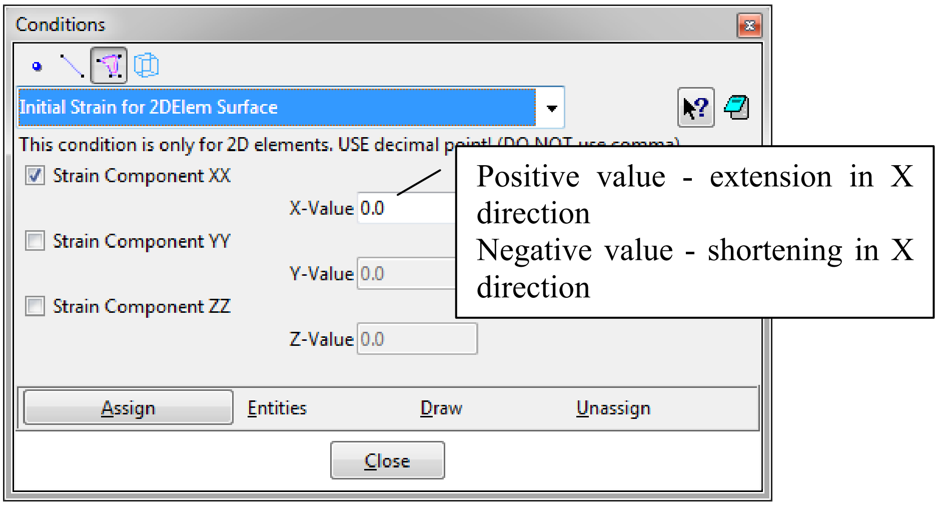


Fig. 5-17 Conditions: Initial Strain for …

Initial Stress – This condition can be used to model pre-stressing. Unlike **Initial Strain**, the stress (force) remains constant. This corresponds to a situation with pre-stressing cables repeatedly post-tensioned to compensate for the losses. Positive stress means tensile pre-stressing.

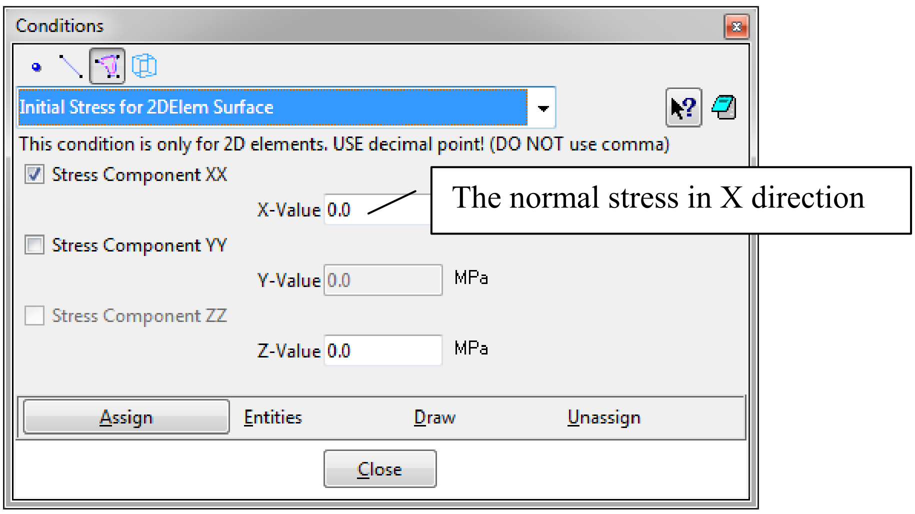


Fig. 5-18 Conditions: Initial Stress for …

Shell Solid Contact – This is a special condition, useful in some situations when shell and volume elements are connected to each other. It does NOT connect the elements, only applies specific handling to the shell. Please see section 5.3.2.1 for details.

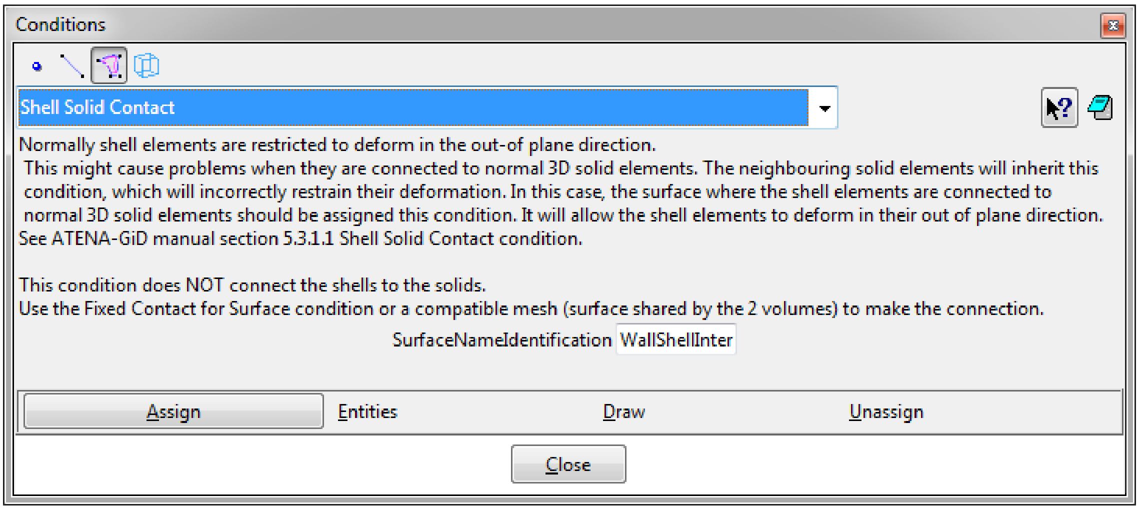


Fig. 5-19 Conditions: Shell Solid Contact for Surface

Reinforcement (Nodes, Elems) identification (condition for line) – This condition is used to identify that certain line entities should be treated as **ATENA** discrete reinforcement bars. The truss elements, which will be generated along these entities, will be embedded into the **ATENA** model as discrete reinforcement bars. This means that they will be further subdivided depending on their intersections with the solid finite elements. By default, the **GiD** program automatically detects lines, which are not connected to any volume or surface and treats these lines as reinforcement. This default behaviour can be controlled by the corresponding check-box in **Problem data** dialog. If this check box is deactivated, it is necessary to manually assign these conditions to any line that should be modelled by embedded reinforcement elements (it has to be assigned twice, for nodes and for elements). The lines, which are not identified as reinforcement, are treated as standard truss elements. In this case, the user is responsible to ensure that the mesh along each line is compatible with the rest of the model.

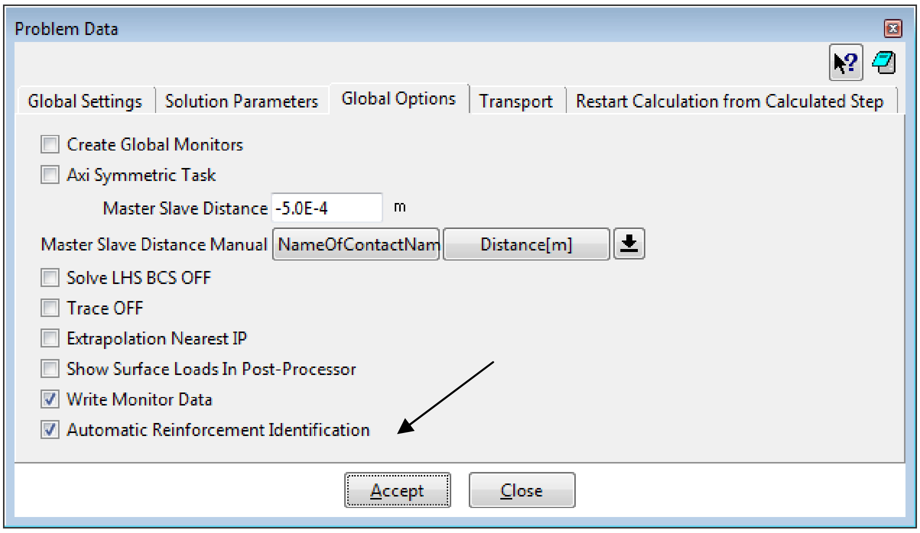


Fig. 5-20: Automatic reinforcement identification in the Problem Data dialog

Initial Gap Load for Volume – This load is used for gaps that are initially open. See material Interface, Section 5.3.6.

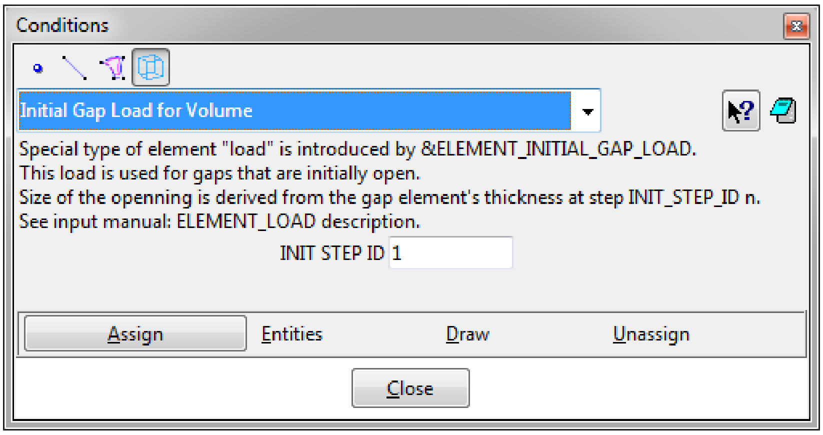


Fig. 5-21: Initial Gap Load for Volume

Elements activity – Used to model construction process. See the ATENA Science Example Manual [8], section 2.2 Tutorial for Construction Process for an example.

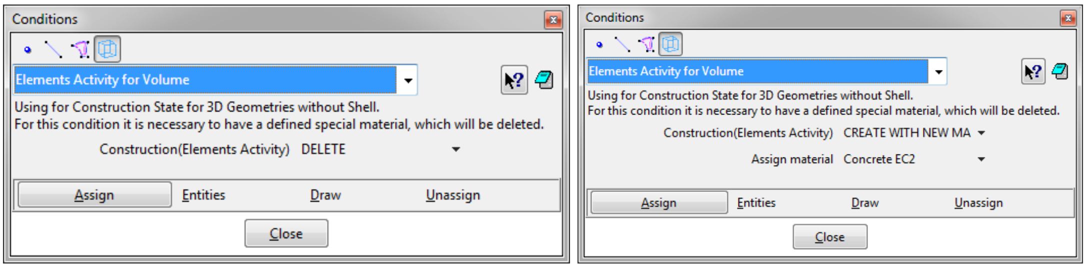


Fig. 5-22: Elements activity for …

Reinforcement Inactivity – By this condition you can inactive and active reinforcement.

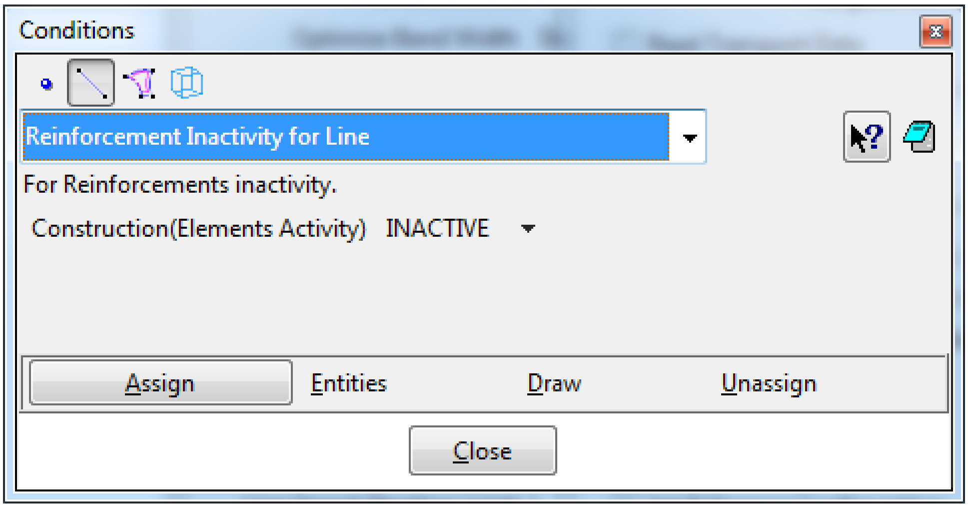


Fig. 5-23: Reinforcement Inactivity for line

Reinforcement Prestressing – By this condition you can define the prestressing of the reinforcement.

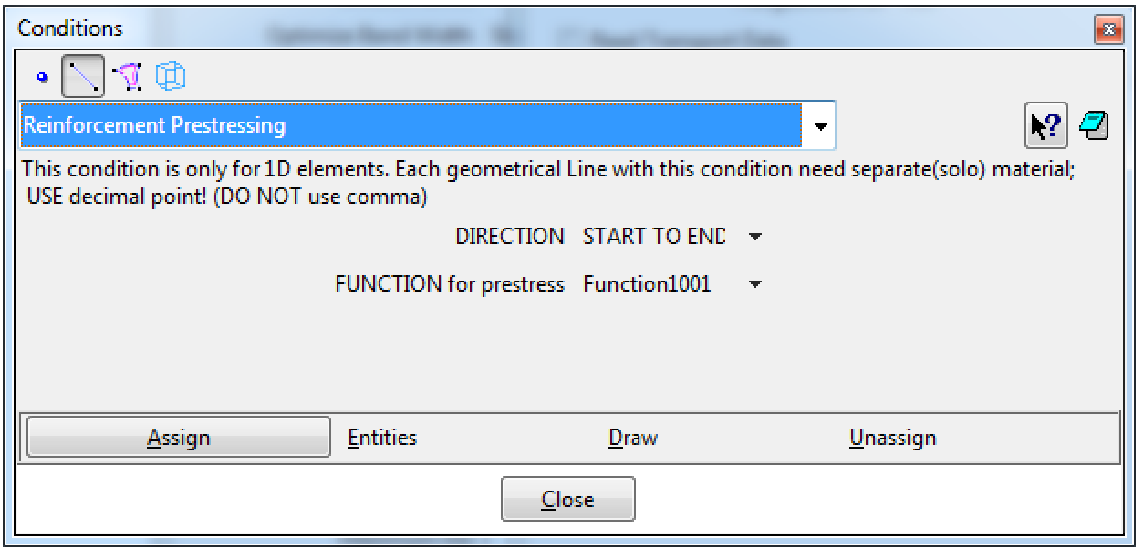


Fig. 5-24: Reinforcement Prestressing

Boundary Reactions for … – Support for the new Austrian tunnelling method. The user can define the activation or removal of parts the structural model to simulate the various construction cases. The redistribution of the forces between the removed parts and the new ones can be controlled through user defined parameters.Example how to use this condition you can find in

AtenaExamples\Tutorial.Creep2D\ TunnelWithConstructionProcessNew.gid.

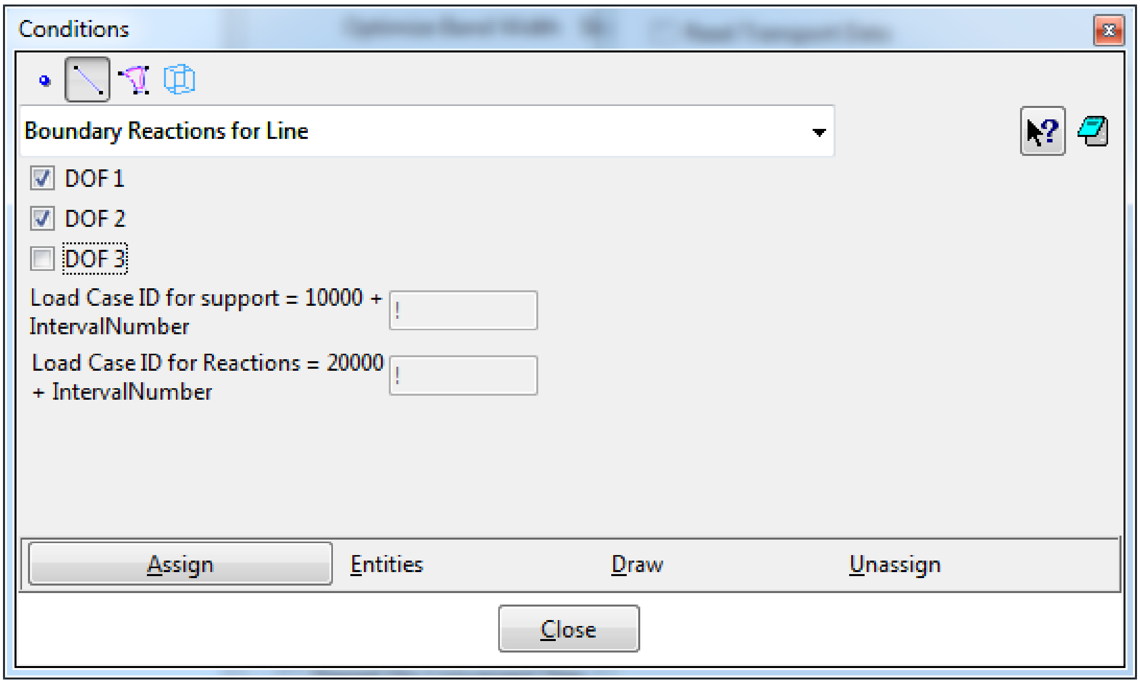


Fig. 5-25: Boundary reactions for …

5.3 Materials

The materials are first defined and then assigned to the model. The later can be done in two ways. In the first and most convenient way, the material is assigned to a geometrical entity. This is usually a volume in 3D or a surface in 2D. On the other hand, reinforcement properties are usually assigned to line entities. After the element generation, the material is automatically assigned to finite elements that are generated on the corresponding geometric entity. The second possibility is to assign materials directly to the finite elements. The material assignment and definition is activated either from the menu item **Data | Materials** or by the icons .

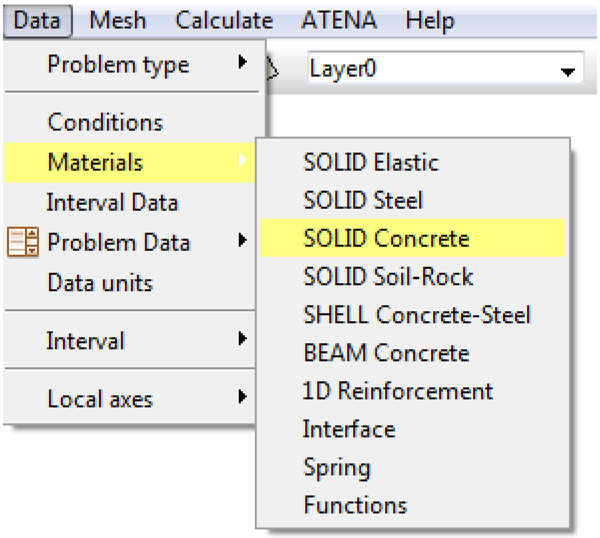


Fig. 5-26 Example of available material categories for static analysis.

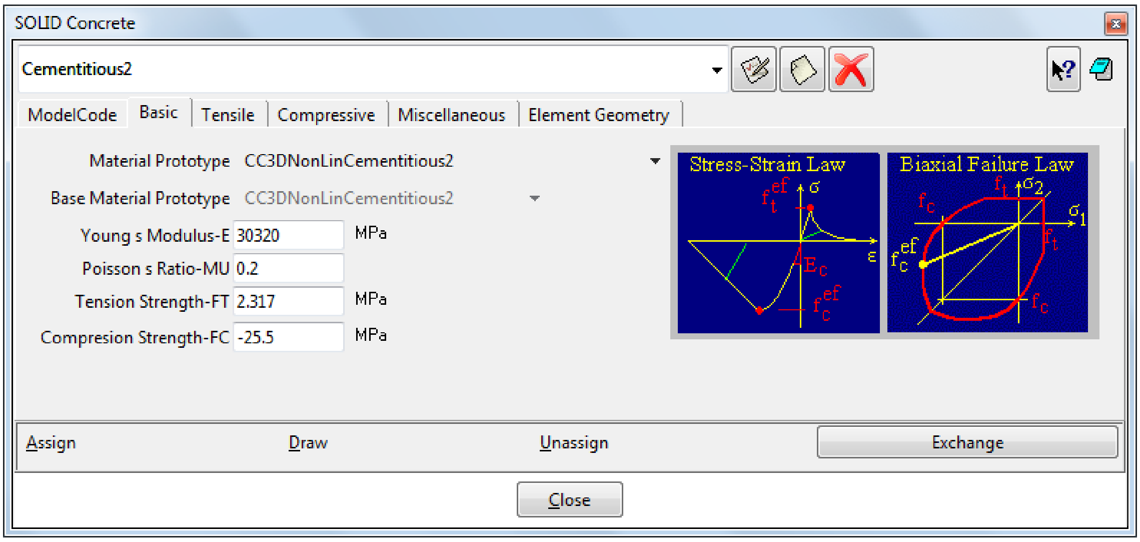


Fig. 5-27 Example of menu window for the material concrete.

Each material can be defined in a special dialog window. Example of such a window for concrete material is shown in Fig. 5-27. Each material offers default parameters. They can be changed to any desired values. After definition of material parameters, the material can be assigned to the numerical model. Operations for material assignment are done with the buttons in the bottom of the dialog.

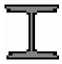
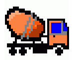
Assign - The target of assignment command depends on the display type. In case that geometry is displayed, then geometry type is to be selected (line for reinforcement, volume for concrete), and material can be assigned to the geometric entities. In case that the finite elements are displayed, the material can be directly assigned to individual finite elements. It should be noted that if a material is assigned directly to finite elements, the assignment is lost every time the mesh is regenerated.

Draw – displays the material assignment to volumes or elements.

Unassisgn – Reverse operation to **Assign**. It deletes the material assignment.

Exchange – Open material database from other GiD project and import your created material to your new project. It is also possible to import material from new project to the other project (exchange).

Table 1: Materials supported by GiD interface to ATENA

GiD name	*ATENA name (INP command)*	*Description*
	SOLID Elastic	
Elastic 3D	CC3DElastIsotropic	Linear elastic isotropic materials for 3D
	SOLID Steel 	
Steel Von Mises 3D	CC3DBiLinearSteelVonMises	Plastic materials with Von-Mises yield condition, e.g. suitable for steel.
Steel Von Mises 3D	CC3DBiLinearVonMisesWithTempDepPropertiess	This model is to be used to simulate change of material properties due to current temperature. The temperature fields can be imported from a previously performed thermal analysis.
	SOLID Concrete 	
Concrete EC2	CC3DNonLinCementitious2	Material is like Cementitious2. You can generate material properties according the EC2
Cementitious2	CC3DNonLinCementitious2	Materials suitable for rock or concrete like materials. This material is identical to 3DNONLINCEMENTITIOUS except that this model is fully incremental.
Cementitious2	CC3DNonLinCementitious2Fatigue	This material is based on the CC3DNonLinCementitious2 material, extended for fatigue calculation.
Cementitious2	CC3DNonLinCementitious2WithTempDepProperties	This model is to be used to simulate change of material properties due to current temperature. The temperature fields can be imported from a previously performed thermal analysis.

Cementitious2 User	CC3DNonLinCementitious2User	Materials suitable for rock or concrete like materials. This material is identical to CC3DNonLinCementitious2 except that selected material laws can be defined by user curves (5.3.1.4).
Cementitious2 SHCC	CC3DNonLinCementitious2SHCC	Strain Hardening Cementitious Composite material. Material suitable for fiber reinforced concrete, such as SHCC and HPFRCC materials. Identical to CC3DNonLinCementitious2User except for the shear response definition.
Cementitious3	CC3DNonLinCementitious3	Materials suitable for rock or concrete like materials. This material is an advanced version of CC3DNonLinCementitious2 material that can handle the increased deformation capacity of concrete under triaxial compression. Suitable for problems including confinement effects.
Reinforced Concrete	CCCombinedMaterial	This material can be used to create a composite material consisting of various components, such as for instance concrete with smeared reinforcement in various directions. Unlimited number of components can be specified. Output data for each component are then indicated by the label #i. Where i indicates a value of the i-th component. Described in section 5.3.4.
Microplane M4, M7	CCMicroplane4, CCMicroplane7	Bazant Microplane material models for concrete
SBETA Material	CCSBETAMaterial	Older version of the basic material for concrete, only suitable for 2-D plane stress models
only for Transport PROBLEM TYPE		
Bazant_Xi_1994	CCModelBaXi94	Material for transport analysis (Transport3D PROBLEMTYPE) – only supported for backward compatibility since ATENA 5.0 (CCTransportMaterial is now recommended), see section 8.1.2 for details.
CCTransportMaterial	CCTransportMaterial	Material for transport analysis, see section 8.1.1.
SOLID_Creep_Concrete (only for Creep PROBLEM TYPE)		
ModelB3	CCModelB3	Bazant-Baweja B3 model
ModelB3Improved	CCModelB3Improved	model same as the above with support for specified time and humidity history
ModelBP_KX	CCModelBP_KX	creep model developed by Bazant-Kim, 1991.
ModelCEB_FIP78	CCModelCEB_FIP78	creep model advocated by CEB-FIP 1978
ModelCSN731201	CCModelCSN731201	model recommended by CSN731202
ModelBP1	CCModelBP1	full version of the creep model developed by Bazant-Panulla
ModelBP2	CCModelBP2	simplified version of the above model
ModelACI78	CCModelACI78	creep model by ACI Committee in 1978.

SOLID Soil-Rock		
Drucker Prager	CC3DDruckerPragerPlasticity	Plastic materials with Drucker-Prager yield condition.
SHELL Concrete-Steel		
Shell Concrete-Steel	CCShellMaterial	Shell geometry with support Ahmad elements, described in section 5.3.2. These elements are reduced from a quadratic 3D brick element with 20 nodes. The element has 9 integration points in shell plane and layers in direction normal to its plane. The total number of integration points is 9x(number of layers). Important feature of shell element is, that its local Z axis must be perpendicular to the top surface of shell plane. The top surface is the surface on which the positive Z-axis points out of the shell. Other two axes, X and Y, must be in the shell plane. Such orientation must be ensured by user. In each shell node there are 3 displacement degrees of freedom and corresponding nodal forces. However, some DOFs are not free due to introduction of kinematic constrains ensuring shell displacement model. For more details see Theory Manual. **Shell material can be used only on 3D quadratic brick elements (5.7.2).**
BEAM Concrete		
Beam Concrete	CCBeam3DMaterial	Special material, which activates the usage of special fiber beam element suitable for large scale analysis of complex structures with large elements (see 5.3.3). The element is based on a similar beam element from BATHE(1982). It is fully nonlinear, in terms of its geometry and material response. It uses quadratic approximation of its shape, so it can be curvilinear, twisted, with variable dimensions of the cross-sections. Moreover, beam's cross-sections can be of any shape, optionally even with holes. The element belongs to the group of isoparametric elements with Gauss integration along its axis and trapezoidal (Newton-Cotes) quadrature within the cross-section. The integration (or material) points are placed in a way similar to the layered concept applied to shell elements, however, the "layers" are located in both "*s,t*" directions. **Beam material can be used only on 3D quadratic brick elements (5.7.2).**

	1D Reinforcement	
Reinforcement EC2	CCReinforcement	Material is like “Reinforcement”. You can generate material properties according the EC2
Reinforcement	CCReinforcement	Material for discrete reinforcement – bars and cables (5.3.5)
Reinforcement	CCReinforcementWithTempDepProperties	This model is to be used to simulate change of material properties due to current temperature. The temperature fields can be imported from a previously performed thermal analysis.
Reinforcement	CC1DElastIsotropic	One dimension elastic material (only supported for backward compatibility since ATENA 4.3.0)
Reinforcement	CCCyclingReinforcement	Material for cyclic reinforcement
	Interface	
Interface	CC2DInterface, CC3DInterface	Interface (GAP) material for 2D and 3D analysis. Please see section 5.3.6 for description and important advice how to create contact elements.
	Spring	
Spring Material	CCSpringMaterial	Material for spring type boundary condition elements, i.e. for truss element modeling a spring.

The following table summarizes, which material types are available in the various **ATENA-GiD** problem types. **GiD** versions older than 7.4 may have compatibility problems with the newer problem types. Similarly, older versions of ATENA prior to the version 3.x.x may have problems with the newer problem types.

Table 2: Available ATENA material types in various GiD-ATENA problem types.

Materials for problem type: ATENA (inp) name	GiD Name	Static	Creep	Transport	Dynamic
CC3DElastIsotropic	Elastic 3D	X	X		X
CC3DBiLinearSteelVonMises	Steel Von Mises 3D	X	X		X
CC3DBiLinearVonMisesWithTempDepProperties	Steel Von Mises 3D	X			
CC3DNonLinCementitious2	Concrete EC2, Cementitious2	X	X		X
CC3DNonLinCementitious2User	Cementitious2 User	X	X		X
CC3DNonLinCementitious2SHCC	Cementitious2 SHCC	X			X
CC3DNonLinCementitious2Fatigue	Concrete EC2, Cementitious2	X			
CC3DNonLinCementitious2WithTempDepProperties	Concrete EC2, Cementitious2	X			

CC3DNonLinCementitious3	Cementitious3	X	X		X
CCCombinedMaterial	Reinforced Concrete	X	X		X
CCCombinedMaterialWithTempDepProperties	Reinforced Concrete	X			
CCMicroplane4	Microplane M4	X	X		X
CC3DInterface	Interface	X	X		X
CC2DInterface	Interface	X	X		X
CCPlaneStressElastIsotropic	-				
CCPlaneStrainElastIsotropic	-				
CCSBETAMaterial	SBETA Material	X			X
CC1DElastIsotropic	Reinforcement EC2	X	X		X
CCReinforcement	Reinforcement EC2	X	X		X
CCReinforcementWithTempDepProperties	Reinforcement EC2	X	X		X
CCSmearedReinf	Reinforced_Concrete	X	X		X
CCCyclingReinforcement	Reinforcement EC2	X	X		X
CC3DDruckerPragerPlasticity	Drucker Prager	X	X		X
CCSpringMaterial	Spring Material	X	X		X
CCShellMaterial	Shell Concrete-Steel	X	X		X
CCBeam3DMaterial	Beam Concrete	X	X		X
CCModelB3	ModelB3		X		
CCModelB3Improved	ModelB3Improved		X		
CCModelBP_KX	ModelBP_KX		X		
CCModelCEB_FIP78	ModelCEB_FIP78		X		
CCModelCSN731201	ModelCSN731201		X		
CCModelBP1	ModelBP1		X		
CCModelBP2	ModelBP2		X		
CCModelACI78	ModelACI78		X		
CCModelBaXi94 – NOT SUPPORTED	Bazant_Xi_1994			X	
CCTransportMaterial	CCTransportMaterial			X	

The selected materials are described in more detail in the subsequent sections.

5.3.1 Solid Concrete Material

The **Solid Concrete** menu contains material models applicable for modeling concrete, rocks, and similar quasi-brittle materials. The most important models and variants are described here.

5.3.1.1 Cementitious2

Check **Generate Material**, select cube or cylinder strength, enter the strength value (e.g., 30 MPa) and the safety format (e.g., mean), and click the **Update Changes** icon  (Fig. 5-28). The generated values are displayed in a window (Fig. 5-29). Pressing the **Update Changes** once more stores the generated material parameters. The values can be checked and adjusted at the tabs **Basic**, **Tensile**, **Compressive**, **Miscellaneous**, and **Element Geometry**.

5.3.1.1.1 Adjusting generated values

If no detailed data are available from tests or from the manufacturer, generating all properties for the corresponding concrete class or cube strength is typically the best option. When precise values are available for some of the parameters (e.g., tensile strength from an experiment, or elastic modulus from a manufacturer's table), the recommended procedure is to first generate the material data for the closest concrete class or compressive strength, and only then adjust the parameters for which better data are available. If you generate values for very different class and then change many values significantly, it can easily happen that you end up with an inconsistent set and as a result, some numerical issues and/or problematic results may appear.

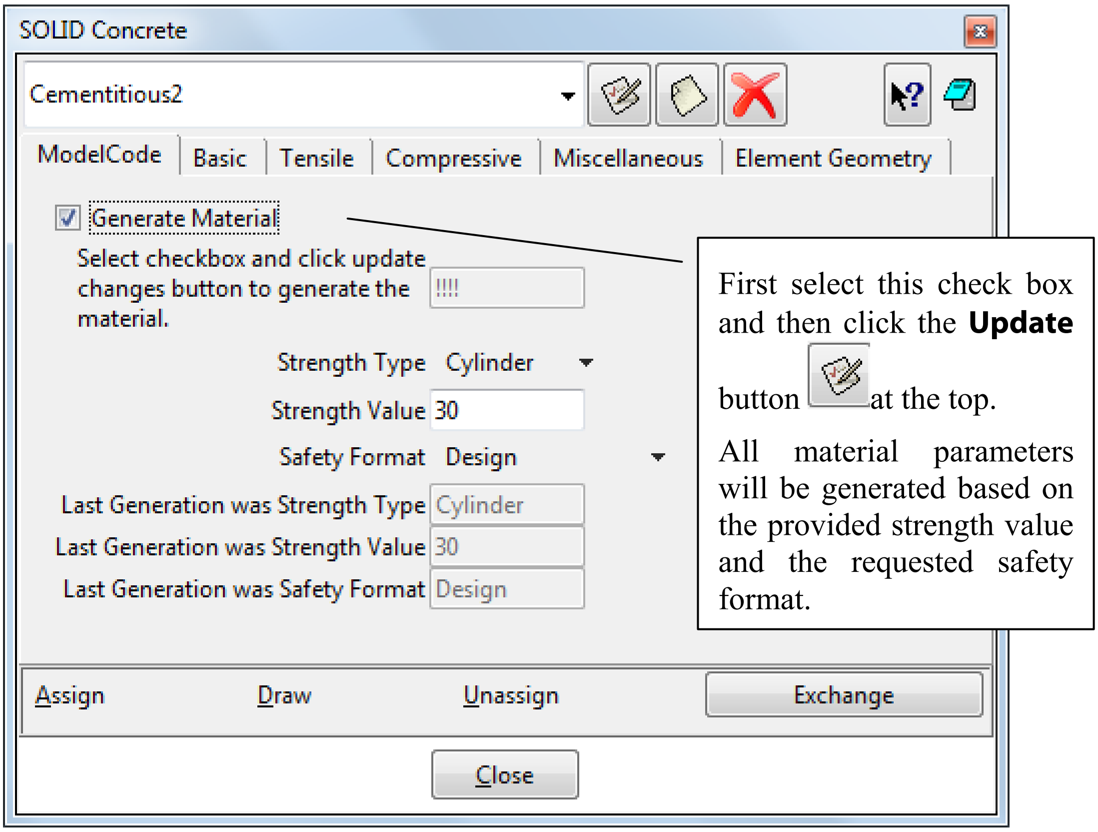


Fig. 5-28: Cementitious2 – Model Code

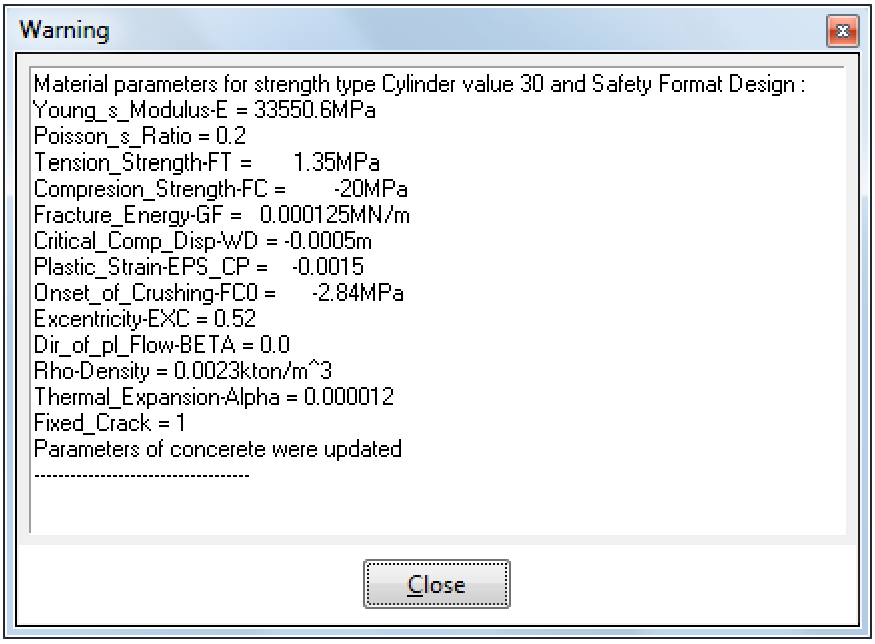


Fig. 5-29: Concrete EC2 – Generated values

The material prototype list box from the **Basic** tab (Fig. 5-30) allows to select the basic CC3DNonLinCementitious2, or CC3DNonLinCementitious2WithTempDepProperties, where some of the material values can depend on temperature, or CC3DNonLinCementitious2Fatigue for modelling high-cycle tensile fatigue (5.3.1.3).

The basic material parameters are defined in the **Basic** dialog – the Young's modulus of elasticity E, the Poisson's coefficient of lateral expansion, the strength in direct tension Ft, and the cylinder compressive strength Fc.

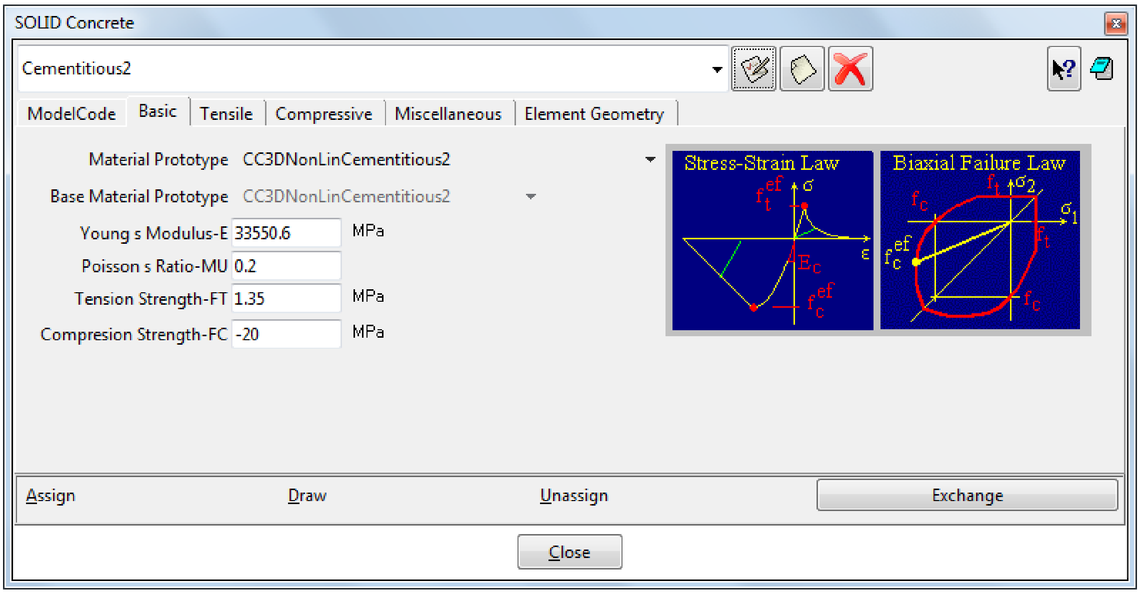


Fig. 5-30: Cementitious2 – Basic

The advanced parameters related to tension are defined at the **Tensile** tab (Fig. 5-31): Fracture energy Gf, Fixed Crack coefficient (0 = rotated, 1 = fixed, more details you can find in ATENA Theory in section"2.1.6 Two Models of Smeared Cracks"), Crack Spacing, Tension Stiffening, Aggregate Interlock, manual definition of Shear Factor, and Unloading Factor (0 = the default unloading to origin, 1 = unloading parallel to the initial elastic stiffness). The meaning of the parameters should be clear from the figures in the dialog and the help texts. For details on these (and also other) parameters, see the ATENA Theory Manual [1].

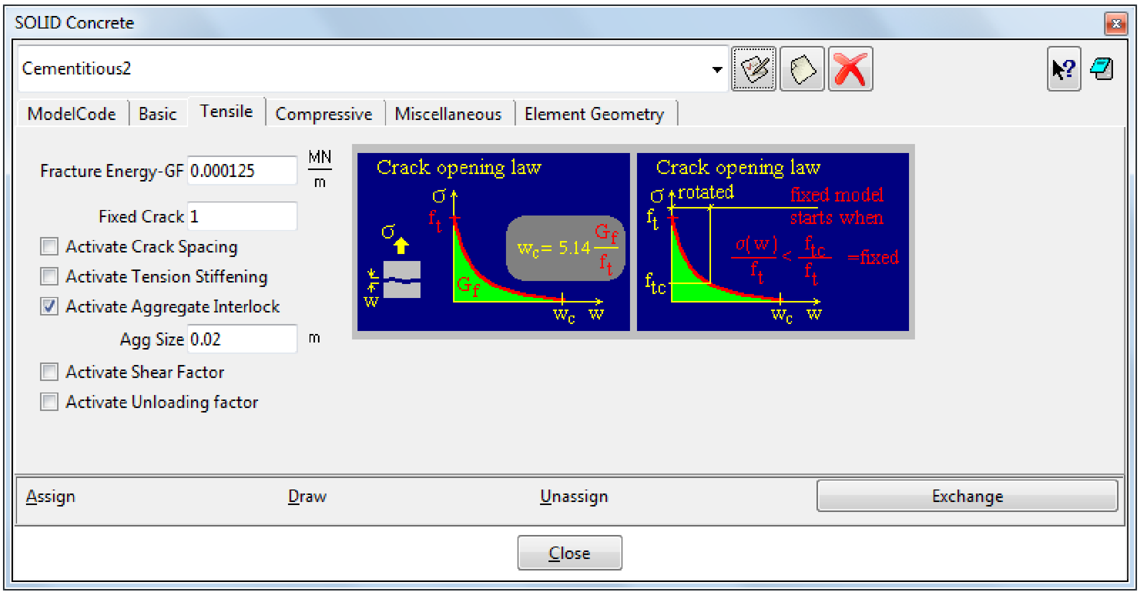


Fig. 5-31: Cementitious2 – Tensile

Crack Spacing option should be used when the element size is larger than the expected crack width. Typically, it should be used in reinforced concrete elements, and is equal to the expected crack spacing. In the simplest case, the spacing of ties or stirrups can be used to estimate its value.

Tension Stiffening - should be used only if reinforcement is present in the model. It defines a relative tensile stress minimal limit for cracked concrete. This means the tensile stress in the cracked concrete cannot drop below this relative level (i.e., ft times tension_stiffening).

Aggregate size for the calculation of **aggregate interlock** based on the modified compression field theory by Collins. When this parameter is set the shear strength of the cracked concrete is calculated using the modified compression field theory by Collins. The input parameter represents the maximal size of aggregates used in the concrete material.

Shear factor that is used for the calculation of cracking shear stiffness. It is calculated as a multiple of the corresponding minimal normal crack stiffness that is based on the tensile softening law.

Unloading factor, which controls crack closure stiffness.

The advanced parameters influencing the compressive response are defined at the **Compressive** tab (Fig. 5-32): Plastic Strain at peak load eps_cp, Onset of Crushing Fc0 (linearity limit), Critical Compressive Displacement wd, and the relative limit for reduction of compressive strength due to cracking Fc Reduction.

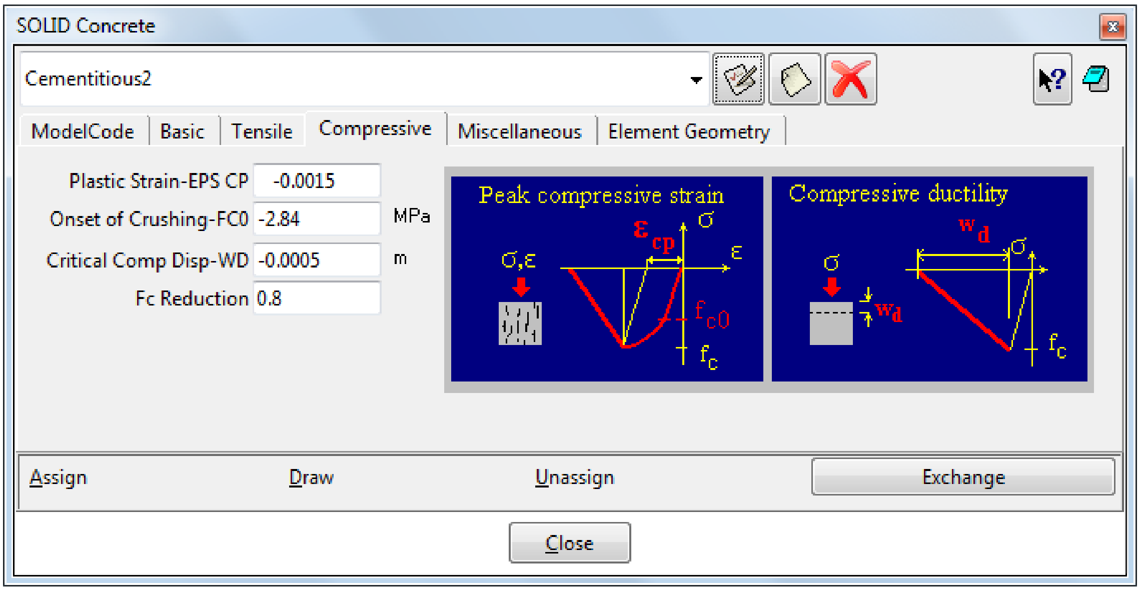


Fig. 5-32: Cementitious2 – Compressive

The **Miscellaneous** tab (Fig. 5-33) contains two additional plasticity-related parameters, the Eccentricity Exc defining the shape of the failure surface, and the Direction of Plastic Flow Beta, determining volume compaction (Beta<0) or expansion (Beta>0) during crushing, i.e. plasticisation, and two general parameters: Density Rho (only used in dynamic analysis) and the coefficient of Thermal Expansion Alpha (only used when thermal load is applied).

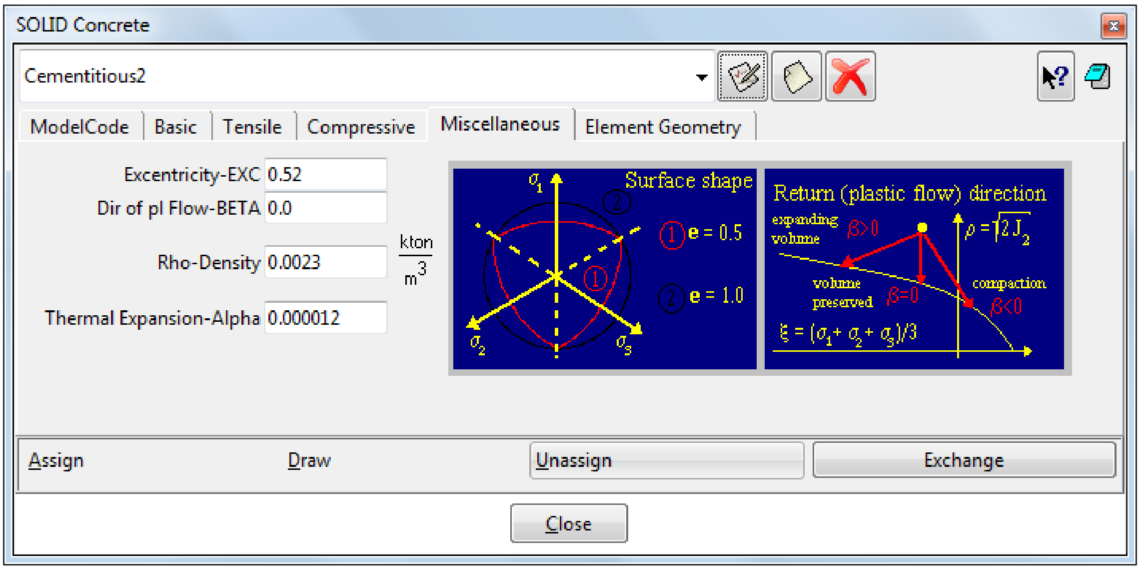


Fig. 5-33: Cementitious2 – Miscellaneous

The settings at the **Element Geometry** tab (Fig. 5-34) are related to the finite elements to be generated for the volumes or surface with the material assigned. The Geometrical Non-Linearity option decides if the nonlinear effects due to deformed geometry are considered in each iteration (NONLINEAR), or if the deformed shape from the end of the previous step is used (LINEAR). Idealisation has to be set corresponding to the type of the analysis (3-dimensional, 2-dimensional plane stress or plane strain, rotational symmetry). If the Non-Quadratic Element checkbox is selected, linear elements are used for the finite elements with this material even if Quadratic elements are selected in the

GiD preferences. This makes it possible to combine quadratic and linear finite elements in a single analysis, for instance, shells for a plate and linear bricks for a column.

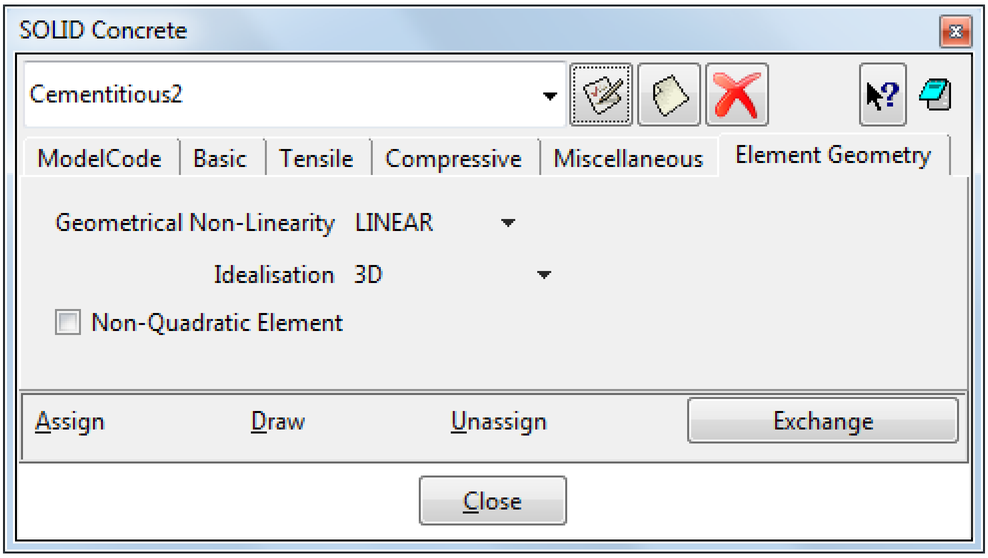


Fig. 5-34: Cementitious2 – Element Geometry

5.3.1.2 Concrete EC2

Concrete EC2 is the same material model as Cementitious2 (5.3.1.1), but allows generating the material parameters based on Eurocode 2. Check **Generate Material**, Select the concrete strength class (e.g., 30/37) and the safety format (e.g., mean) and click the **Update Changes** icon  (Fig. 5-35). The generated values are displayed in a window (Fig. 5-36). Pressing the **Update Changes** once more stores the generated material parameters. The values can be checked and adjusted at the tabs **Basic**, **Tensile**, **Compressive**, **Miscellaneous**, and **Element Geometry**, which are identical to the Cementitious2 material (and therefore not repeated here), and the recommendations from section 5.3.1.1.1 also apply.

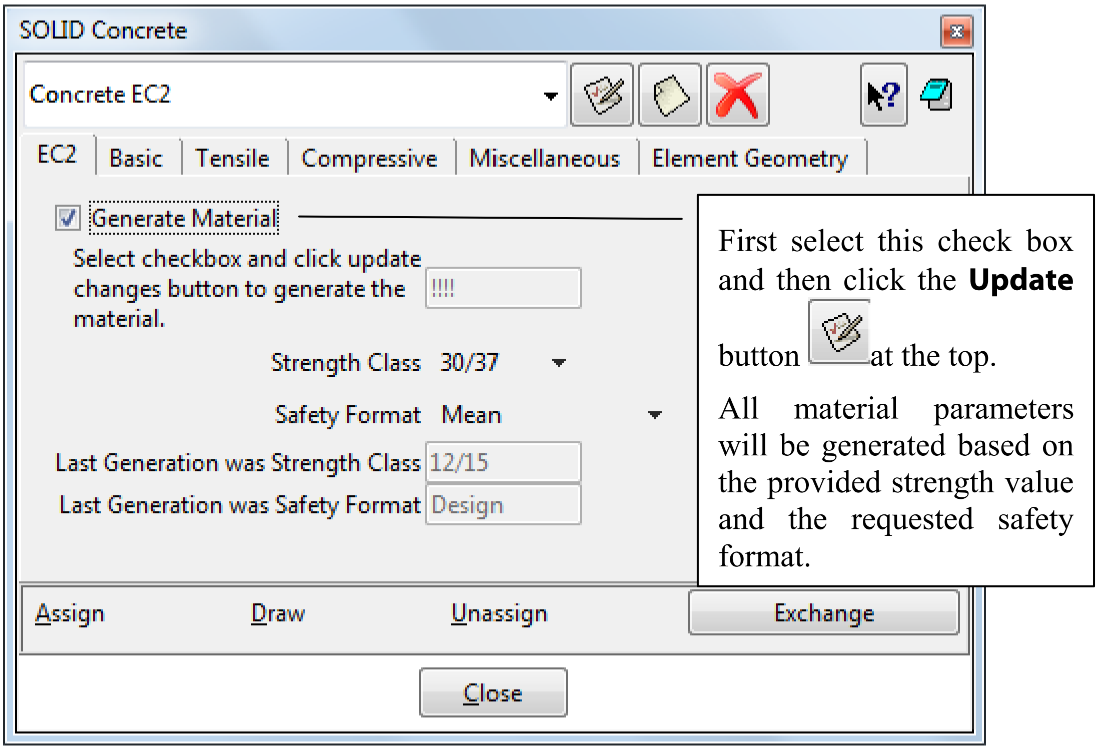


Fig. 5-35: Concrete EC2 – Generation parameters

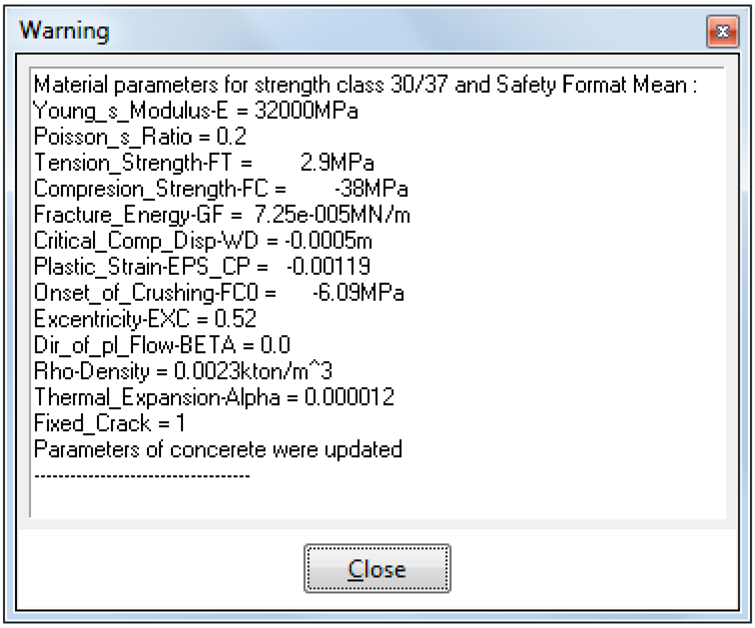


Fig. 5-36: Concrete EC2 – Generated values

5.3.1.3 CC3DNonLinCementitious2Fatigue

The CC3DNonLinCementitious2Fatigue material prototype can be selected at the **Basic** tab of Cementitious2 (5.3.1.1) and Concrete EC2 (5.3.1.2) materials. Then, two additional parameters appear in the dialog (Fig. 5-37):

- Beta Fatigue, β, determining the slope of the Wöhler (S-N) curve for the stress-based contribution, and

- Ksi Fatigue, ξ, defining the growth of existing cracks which repeatedly open and close during the load cycles (ΔCOD).

See also section 5.4.1 for related **Interval Data** settings, and ATENA Theory [1] for details of the fatigue model.

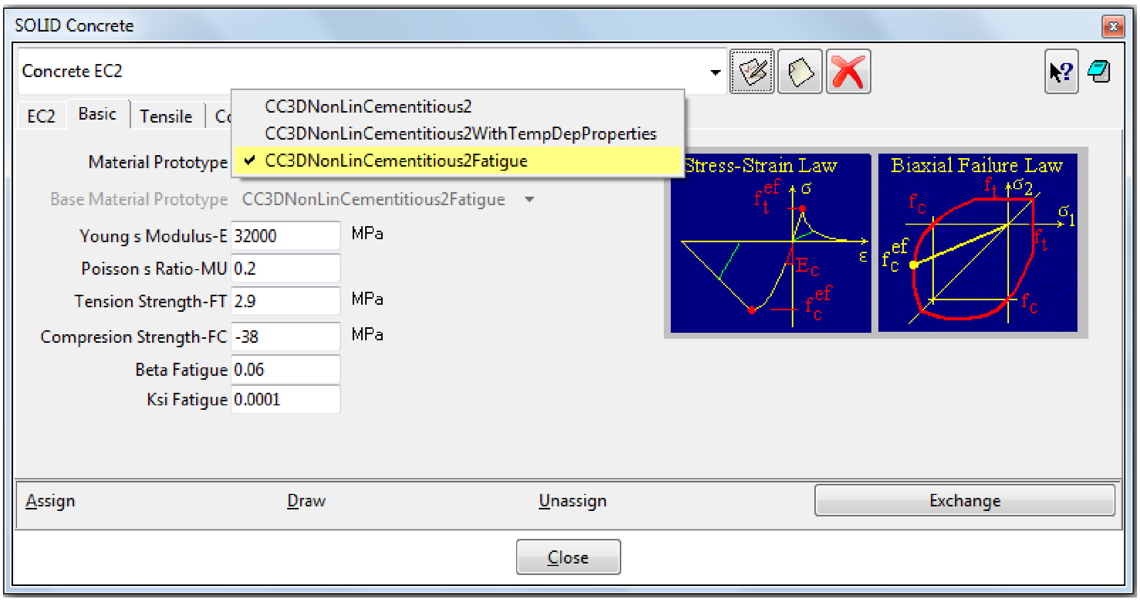


Fig. 5-37: Cementitious2 Fatigue – Basic

5.3.1.4 Cementitious2 User

Cementitious material with user-defined response functions. The tabs with the basic concrete properties, miscellaneous and geometry settings are identical to the Cementitious2 material (5.3.1.1).

On the tabs **Tensile** (Fig. 5-38), **Compressive** (Fig. 5-39), **Shear** (Fig. 5-40), and **Tension-Compressive** (Fig. 5-41), the corresponding user functions and the localization parameters are to be defined. For instructions how to define the user material response functions see ATENA Theory Manual [1] or ATENA Troubleshooting [9], section “2.1.9 I want to use the user-defined stress-strain law of concrete to replace that used in ATENA program. How can I do it?”.

In most cases, the user functions are complemented by the characteristic size and localization onset. These two parameters are used to scale the provided user-defined material functions for different element sizes. This is important when the material exhibits softening, in which case the softening should be dependent on the element size. The characteristic size then represents the size, for which the provided material function is valid. Typically, it is related to the length over which the strains are measured in the experiment. The localization onset typically defines the strain values, when the provided user function starts to exhibit softening, i.e. negative slope.

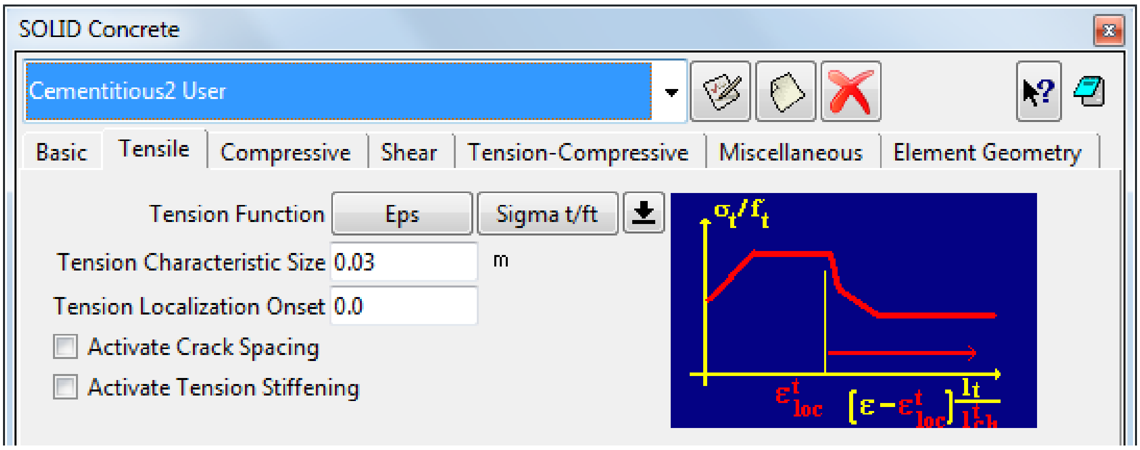


Fig. 5-38: Cementitious2 User – Tensile

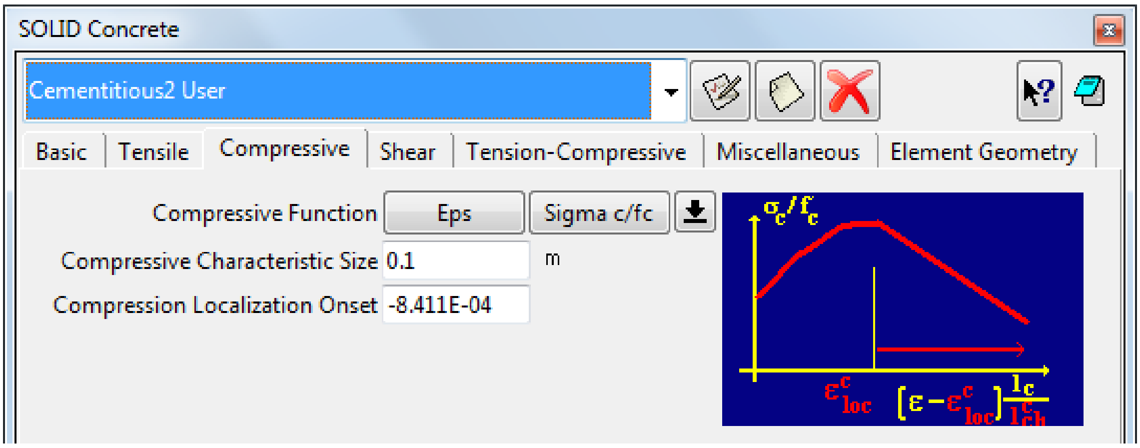


Fig. 5-39: Cementitious2 User – Compressive

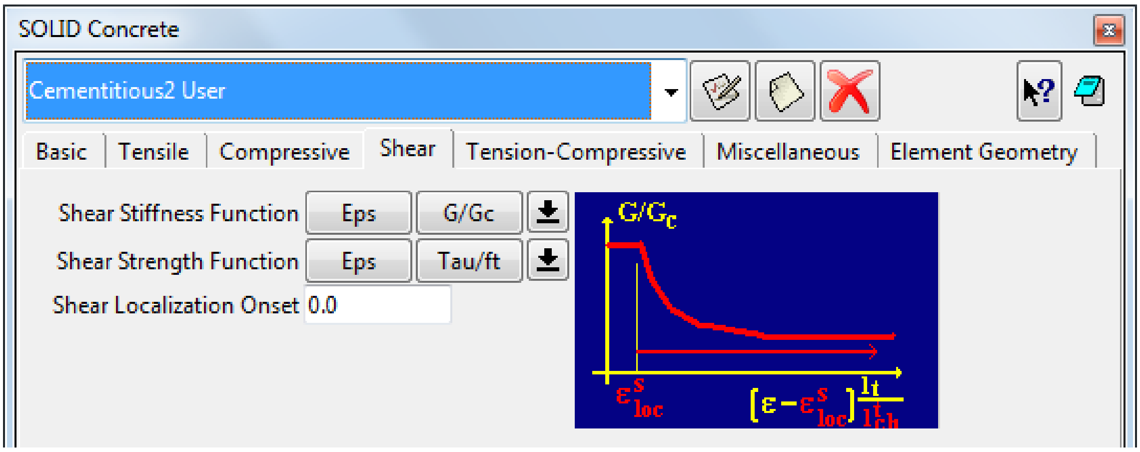


Fig. 5-40: Cementitious2 User – Shear

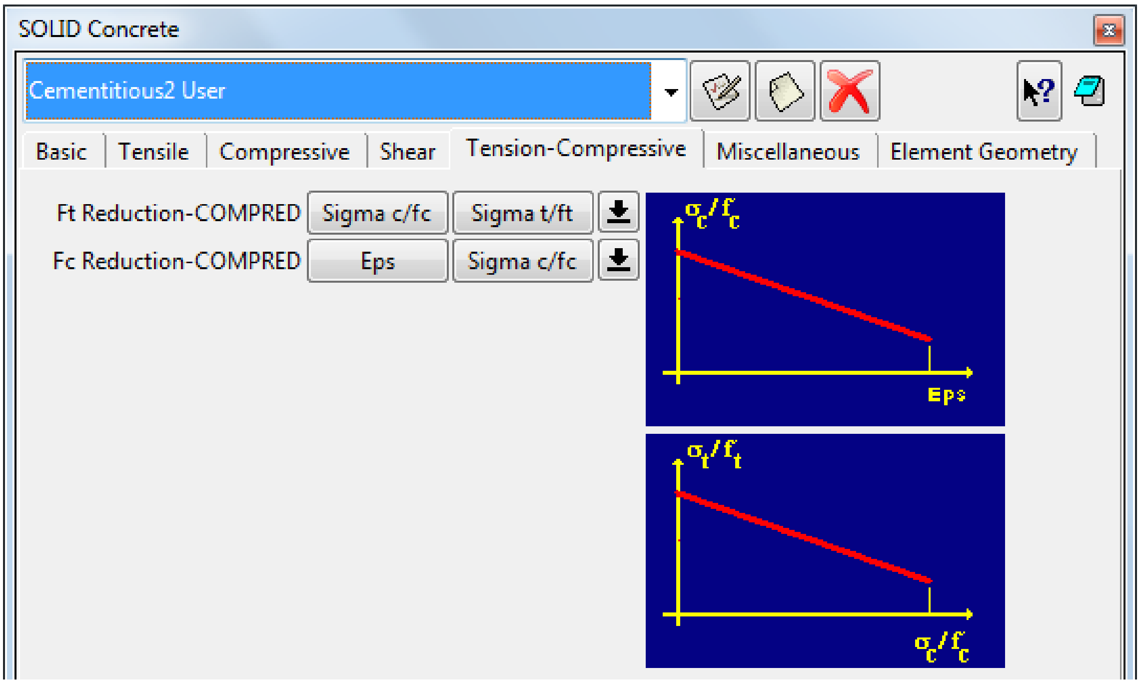


Fig. 5-41: Cementitious2 User – Tension-Compressive

5.3.1.5 Cementitious2 SHCC

Cementitious2 SHCC is a special material for strain hardening cementitious composites (e.g., special mixtures with addition of plastic fibers). The only difference from Cementitious2 User (5.3.1.4) is the **Fibre Reinforcement** tab replacing the **Shear** tab (Fig. 5-42). The settings from this tab are only considered for shear response, i.e., all the remaining functions need to be defined the same way as for the Cementitious2 User material.

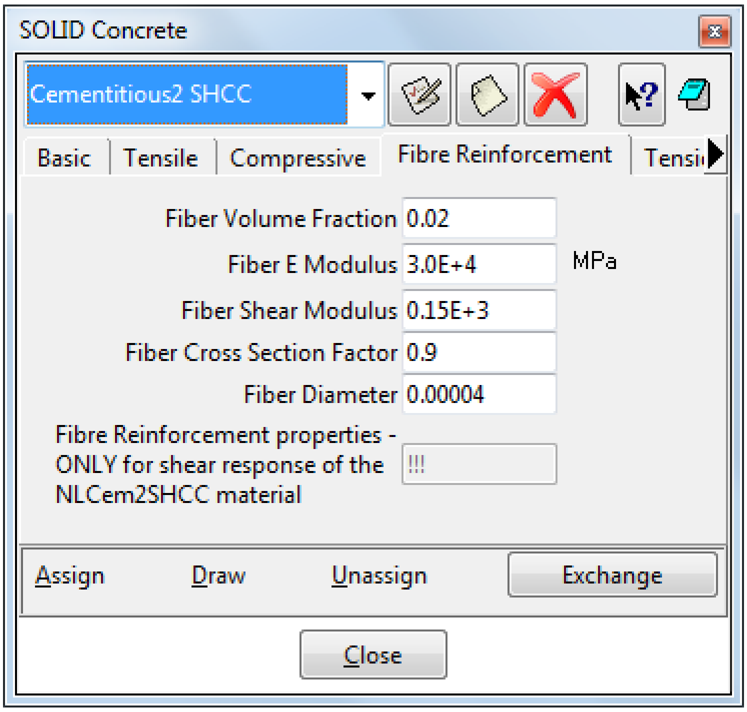


Fig. 5-42: Cementitious2 SHCC – Fibre Reinforcement

5.3.2 *Shell Material*

In this section, shell material is described. In **ATENA-GiD**, this material has to be assigned to volumes where shell (plate) elements are to be used (unlike **ATENA Engineering 3D**, where one switches between volume and shell elements in Macroelement definition). Shell material has geometry which supports **Ahmad** elements (**CCAhmadElement**) and **IsoBrick/IsoWedge** elements (**CCIsoShellBrick**, **CCIsoShellWedge**). These elements are reduced from a quadratic 3D brick (wedge) element with 20 (15) nodes. The element has 9 (6) integration points in shell plane and layers in direction normal to its plane. The total number of integration points is 9x(number of layers) for the bricks, or 6x(number of layers) for the wedges.

An important feature of shell element is that its local Z axis must be perpendicular to the top surface of shell plane. The top surface is the surface on which the positive Z-axis points out of the shell. Other two axes, X and Y, must be in the shell plane. Such orientation must be ensured by user. In this local system, smeared reinforcements are defined and also all results in post-processing are output in this coordinate system. Therefore, it is critical to define the Z direction. For neighboring volumes, it is important to prevent "orientation jumps", i.e., to have the local Z point to the same side (Fig. 5-43, Fig. 5-44). It is also recommended to set the local X direction such that the in-plane directions are continuous over neighboring elements. See Fig. 5-48 showing the corresponding dialog.

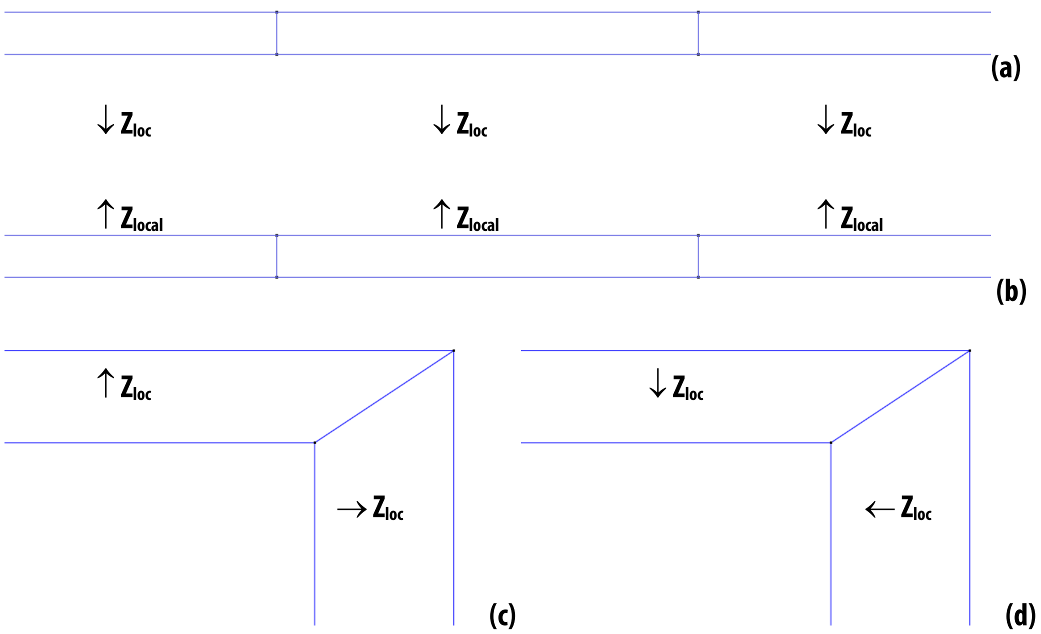


Fig. 5-43: Shell - recommended local Z orientation

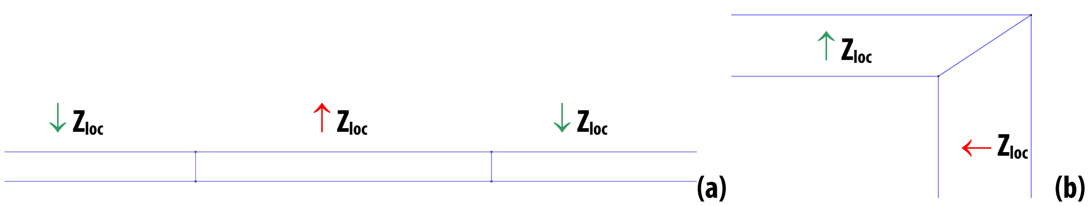


Fig. 5-44: Shell – problematic local Z orientation with orientation jumps

In each shell node there are 3 displacement degrees of freedom and corresponding nodal forces. However, some DoFs are not free due to introduction of kinematic constrains ensuring shell displacement model. For more details see Theory Manual [1].

Shell material can be used only on 3D quadratic brick elements. Unlike volume elements, a single shell per volume thickness works well in bending. In other words, placing 2 or more shell elements above each other ("above" refers to the shell thickness direction) is not a good idea. Instead, use a single shell per thickness with more internal layers to improve precision.

With **Ahmad** shell elements, the best connection at edges is to cut both at 45 degrees, or a different corresponding angle if the thicknesses are not the same, or if connected at other than right angle, see Fig. 5-45 (a). Another option is to use a volume brick element at the corner, (i.e., not using compatible meshes, see also 5.3.6.1), which is the only feasible way when more than two shells are connected, see Fig. 5-45 (b). The **Shell Solid Contact** condition (see 5.3.2.1) has to be assigned on the shell surface connected to the volume element for correct behaviour. Connecting like in Fig. 5-46 is not recommended, as the master-slave relations induced by the fixed thickness of the shell may cause numerical problems.

With the **Iso** shell elements, which can also deform in the local Z direction, the easiest and recommended way of connecting is the one from Fig. 5-46. However, connections from Fig. 5-45 can also be used.

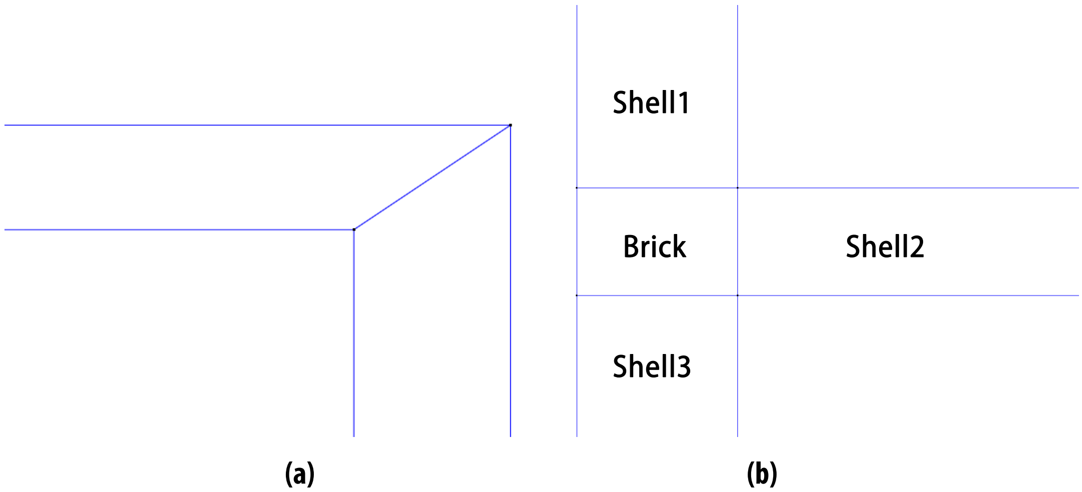


Fig. 5-45: Shell - recommended connection (a) 2 shells (b) 3 shells

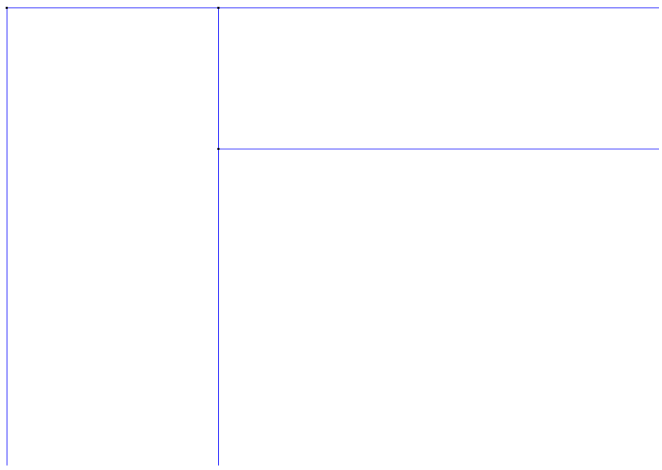


Fig. 5-46: Shell - recommended connection for Iso, not recommended for Ahmad

The **ATENA** implementation of the **Ahmad** and **IsoBrick/Wedge** shell elements supports embedding of smeared reinforcement layers. In this concept, reinforcement bars with the same coordinate *z*, material and the same directions are replaced by a layer of smeared reinforcement. Such a layer is placed at the same elevation *z* as the original reinforcement bars and its thickness is calculated so that sum of cross sectional area of the bars and the replacing smeared reinforcement layer is the same. The layer is usually superimposed over existing concrete layers and it employs **CCSmearedReinforcement** material law (see also section 5.3.5), which makes it possible to account for the original reinforcement bars' direction.

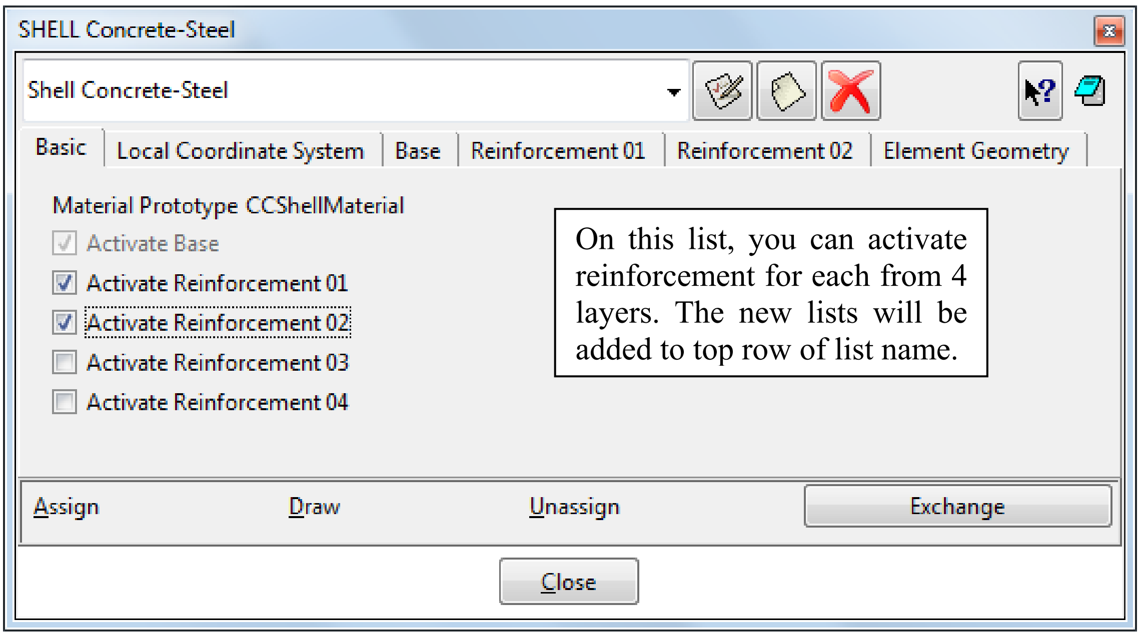


Fig. 5-47: Shell material properties - Basic

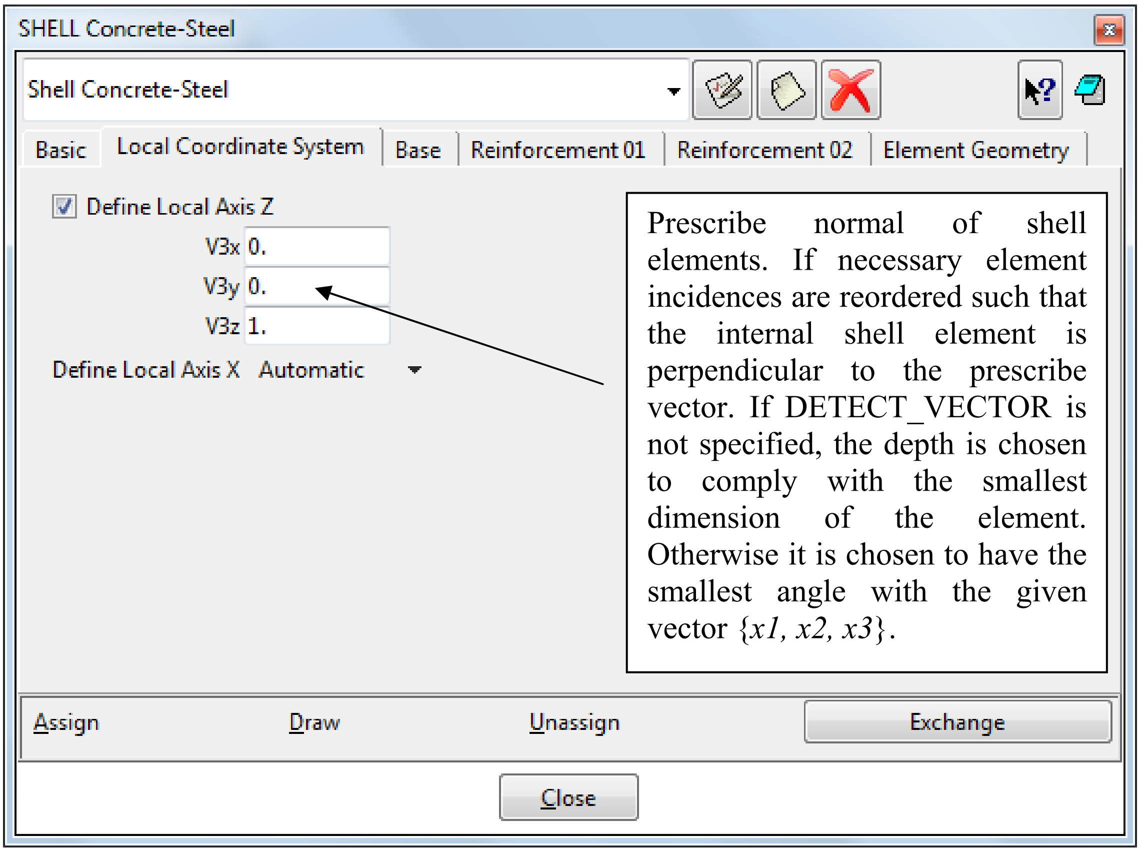


Fig. 5-48: Shell material properties – Local Coordinate System

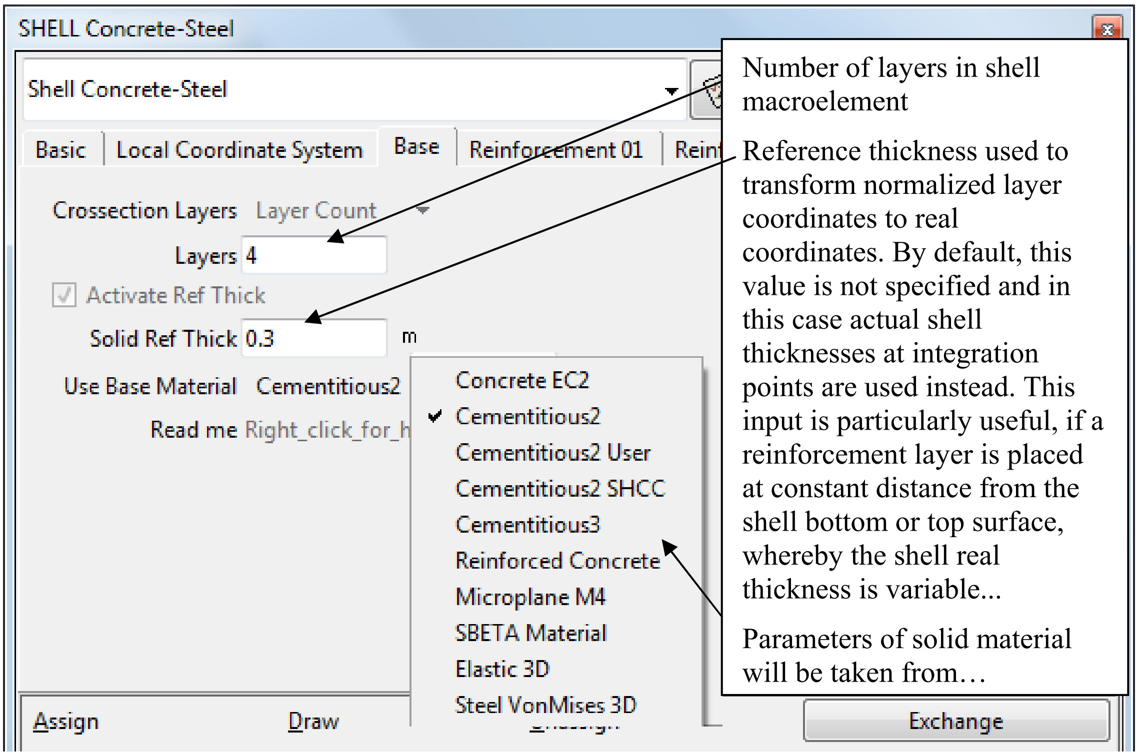


Fig. 5-49: Shell material properties - Base

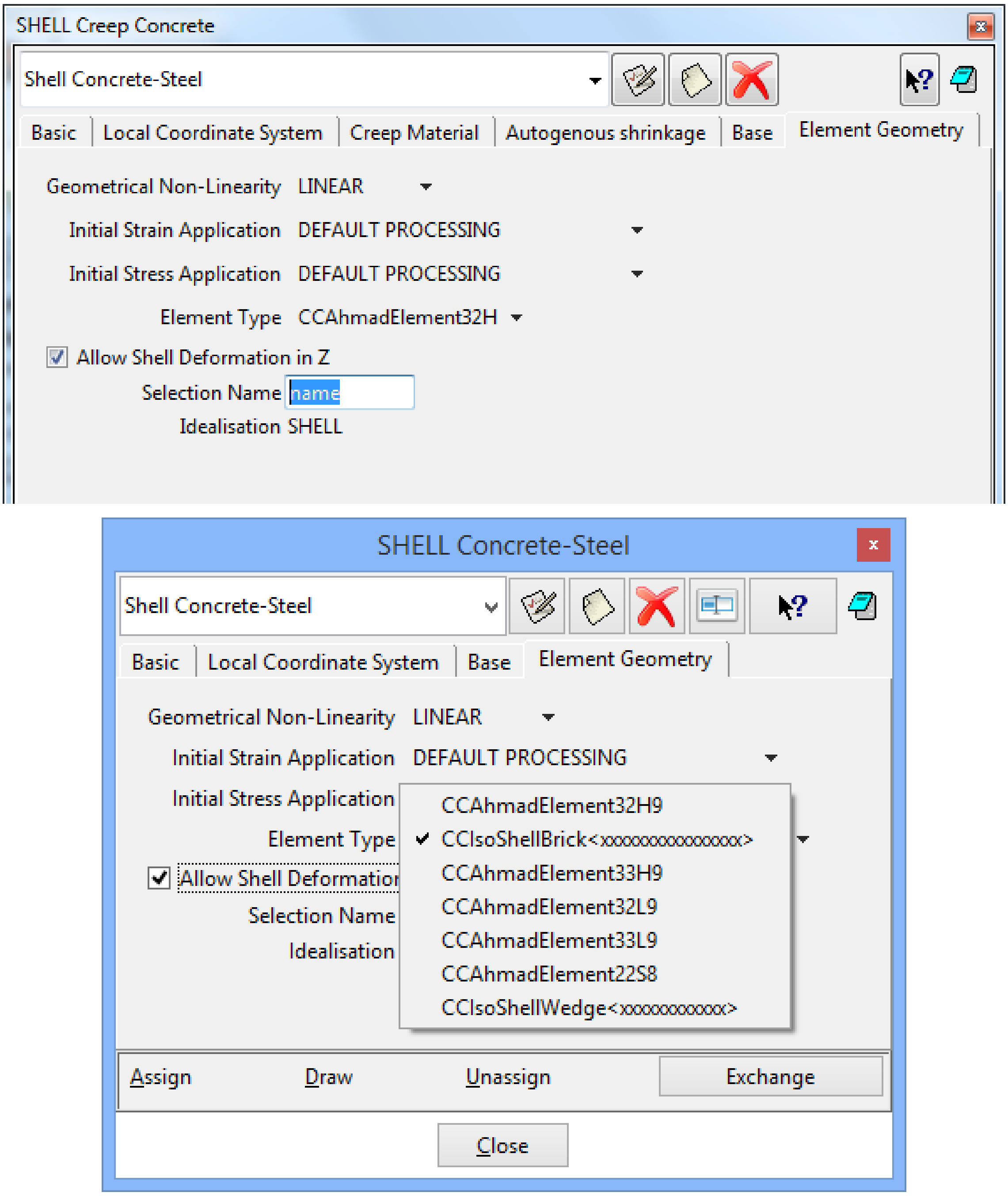


Fig. 5-50: Shell material properties – Element Geometry and Element Type

Initial Strain/Stress application: Special flag for processing initial strain/stress load for elements with embedded smeared reinforcement. By default, the load is applied to both solid and reinforcement parts of the element.

Element type: 3D shell elements. The first and the second digit in the element name specifies number of integration points for element bending and shear energy. E.g. the digit three says that the element is integrated in 3 IPs in X dir * 3 IPs in Y dir * number of layers. The last letter L,H and S stands for 9-nodes Lagrangian element, for 9 nodes Heterosis element and 8 nodes Serendipity element. See theoretical manual for more details. All the elements must use a 3D material and a LayredShell geometry! They specified by 16 nodes, 8 for top and 8 for bottom surface similar to brick elements. The top and bottom middle points for Lagrangian and Heterosis elements (for the bubble functions) are generated automatically. At each node the elements have 3 degree of freedom. As top and bottom node have altogether 6 dofs and shell theory uses only 5 dofs per shell node, the z displacement of the bottom node is automatically constrained during the execution.

Allow_Shell_Deformation_in_Z: Here the name of a selection should be specified. The selection name should be previously defined using the surface Condition "Shell-Solid Contact". Using this method it is possible to allow the normal shell deformation. It is useful when connecting the shell elements with normal solid elements, otherwise the shell elements may restrain the deformation of the surrounding solid elements.

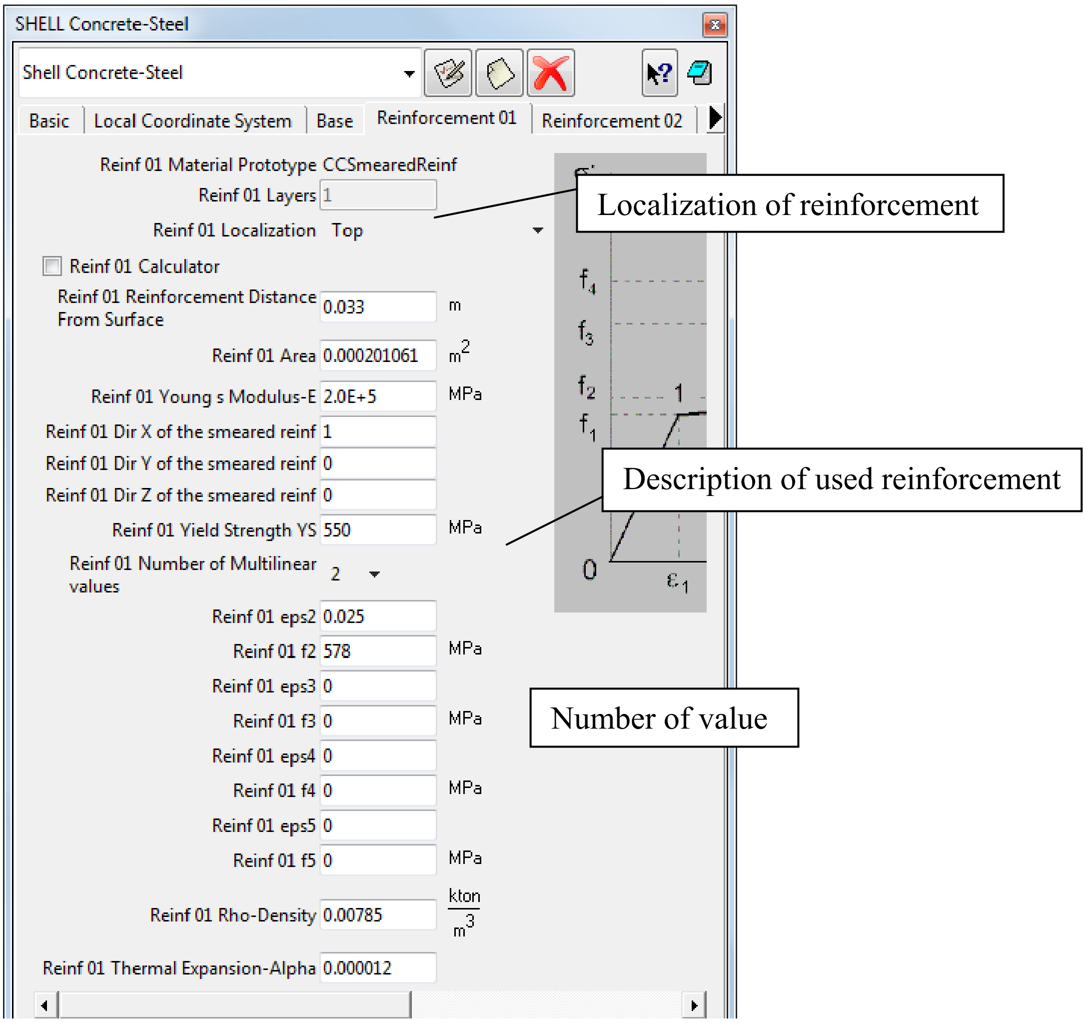


Fig. 5-51: Shell material properties – Reinforcement – detail

5.3.2.1 'Shell Solid Contact' condition

The **Ahmad** shell elements are restricted to deform in the out-of plane direction (fixed thickness). This might cause problems when they are connected to normal 3D solid elements. The neighboring solid elements will inherit this condition, which will incorrectly restrain their deformation. In this case, the surface where the shell elements are connected to normal 3D solid elements should be assigned the **Shell Solid Contact** condition. The condition's name has to be copied into the **Selection Name** box under **Allow Shell Deformation in Z** on the **Element Geometry** tab of the corresponding

Shell Material definition. This condition identifies shell-solid interfaces and allows the shell elements to deform in their out of plane direction.

It is recommended to apply this condition to all shell "side" surfaces which are attached to volume elements. It is not needed where the shell "top" or "bottom" surfaces are connected to volume elements.

For the **Iso** shell elements, this condition is not needed at all and should not be applied.

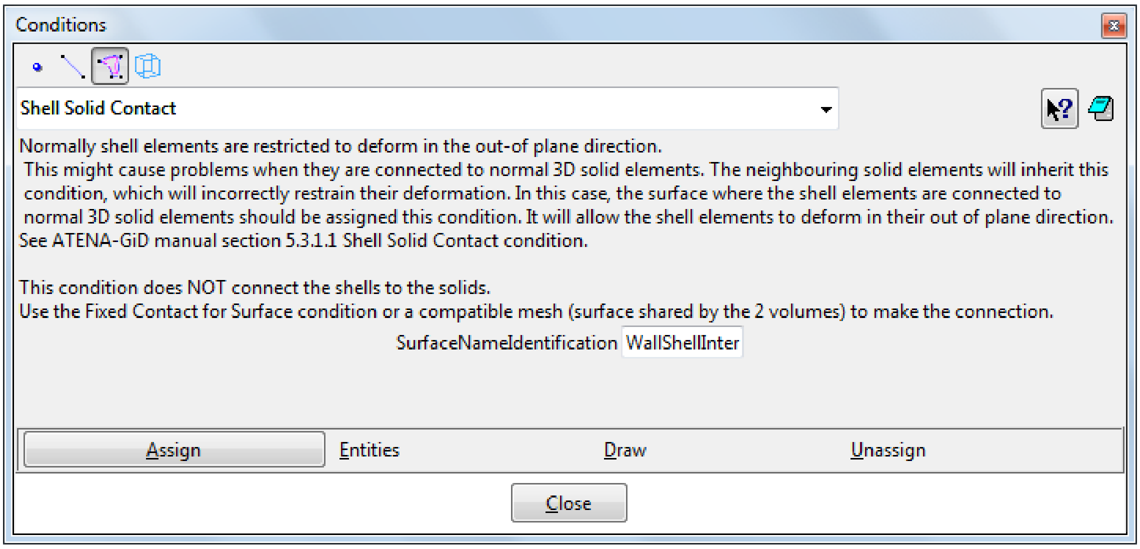


Fig. 5-52: 'Shell Solid Contact' condition for Ahmad elements

5.3.3 *Beam Material*

The fibre beam elements in **ATENA** are similar to shells, just using a similar simplification (special integration) in 2 directions (beam height and width) instead of just one (plate thickness).

The basic settings like activating smeared reinforcement (Fig. 5-53), and defining the local coordinate system (Fig. 5-54) are very similar to the shells (described in the previous section 5.3.2). The local X corresponds to the beam length direction, the local Z to its height.

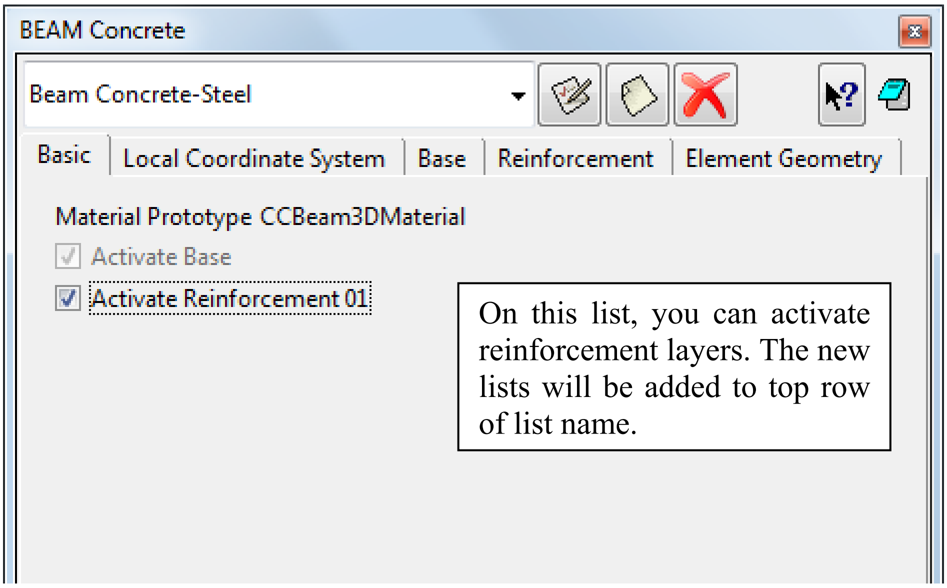


Fig. 5-53: Beam material properties – Basic

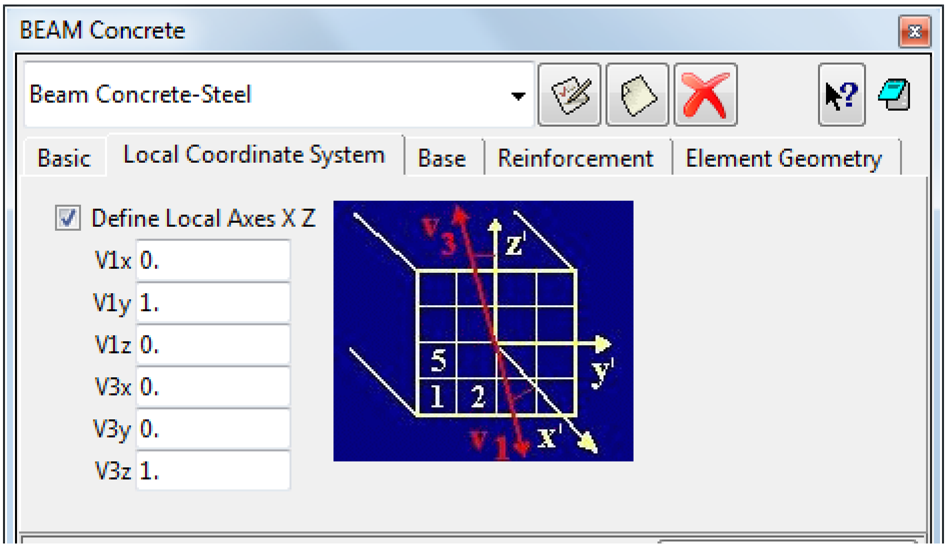


Fig. 5-54: Beam material properties – Local Coordinate System

Instead of the shell internal layers, the beam cross-section is built from rectangular cells (Fig. 5-55). Each cell can be either active (representing an area where a material is present) or inactive (void).

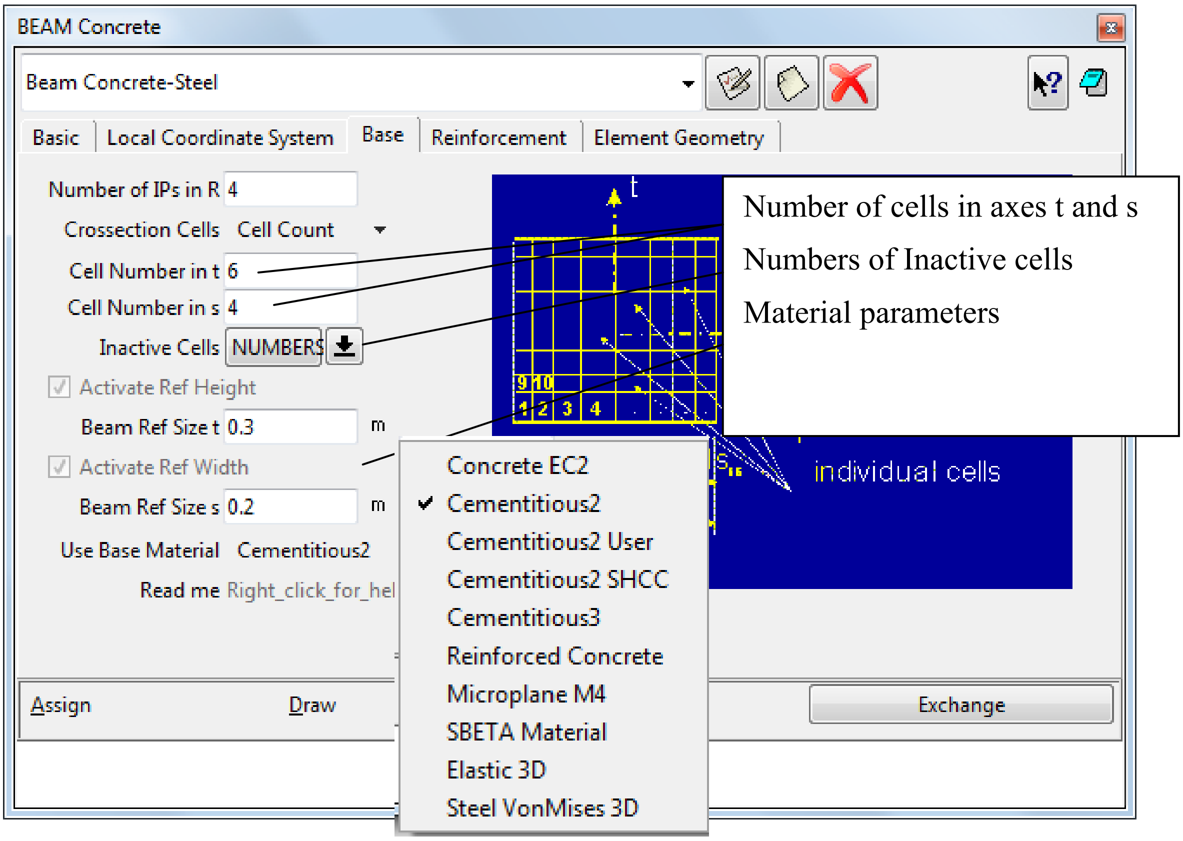


Fig. 5-55: Beam material properties – Base

The definition of the smeared reinforcement (Fig. 5-56) and the geometry properties is also very similar to the definitions in the shell elements (see Section 5.3.2).

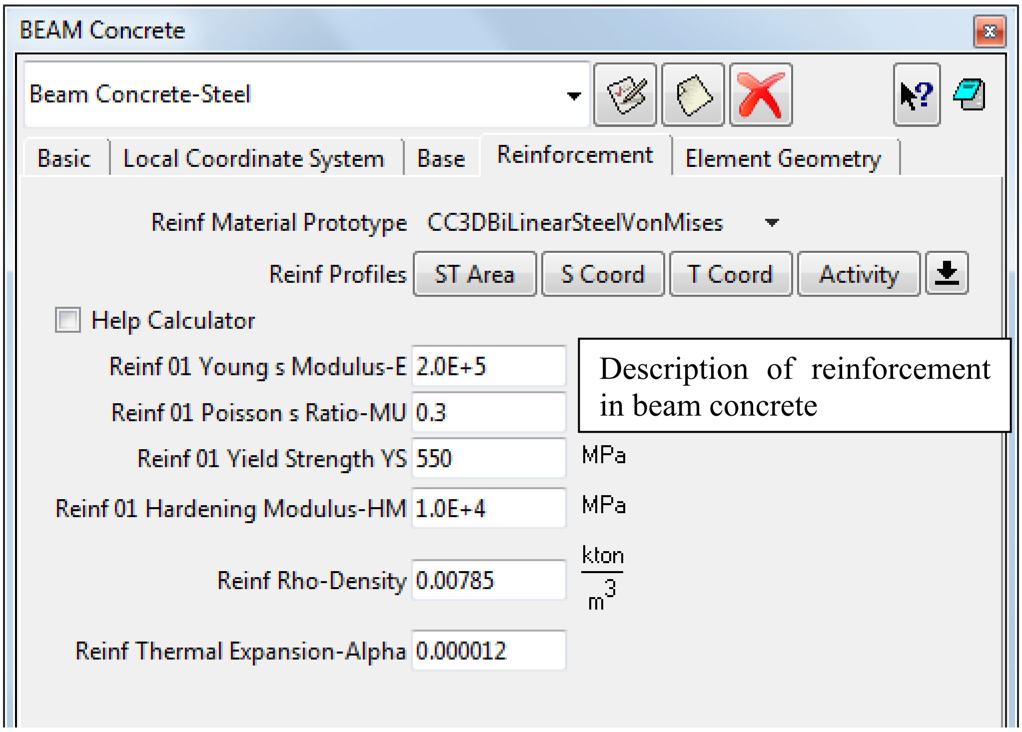


Fig. 5-56: Beam material properties – Reinforcement

5.3.4 Reinforced Concrete

The Reinforced Concrete material is used to define a composite material, consisting of a volume material (typically, Concrete) and smeared reinforcement (1D material) in one or more directions.

The basic settings like activating and defining smeared reinforcement (Fig. 5-57, Fig. 5-58) are very similar to the shells (described in section 5.3.2).

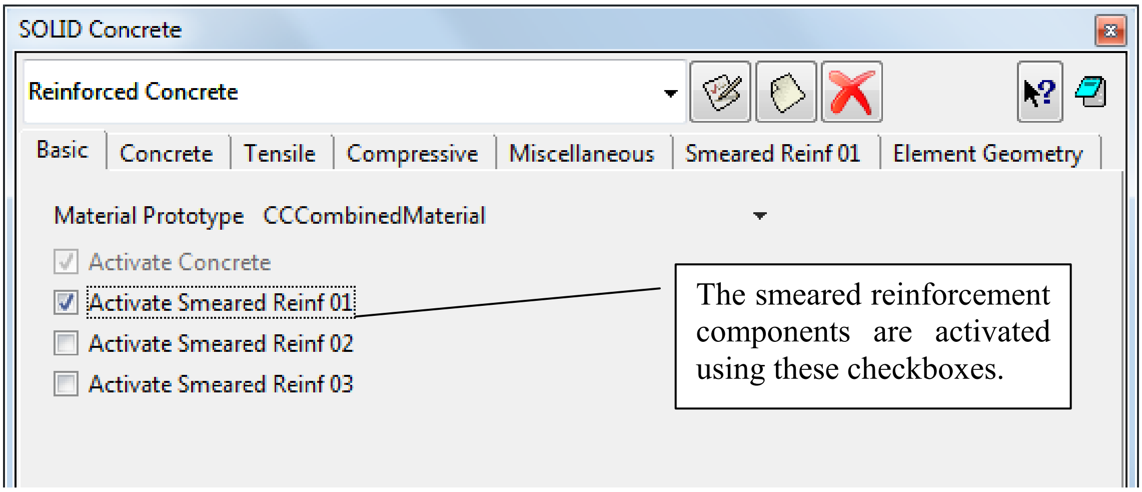


Fig. 5-57: Reinforced Concrete material properties – Basic

The tabs with the concrete properties (**basic** Fig. 5-30, **tension** Fig. 5-31, **compression** Fig. 5-32, **miscellaneous** Fig. 5-33) and geometry settings are identical to the Cementitious2 material (5.3.1.1).

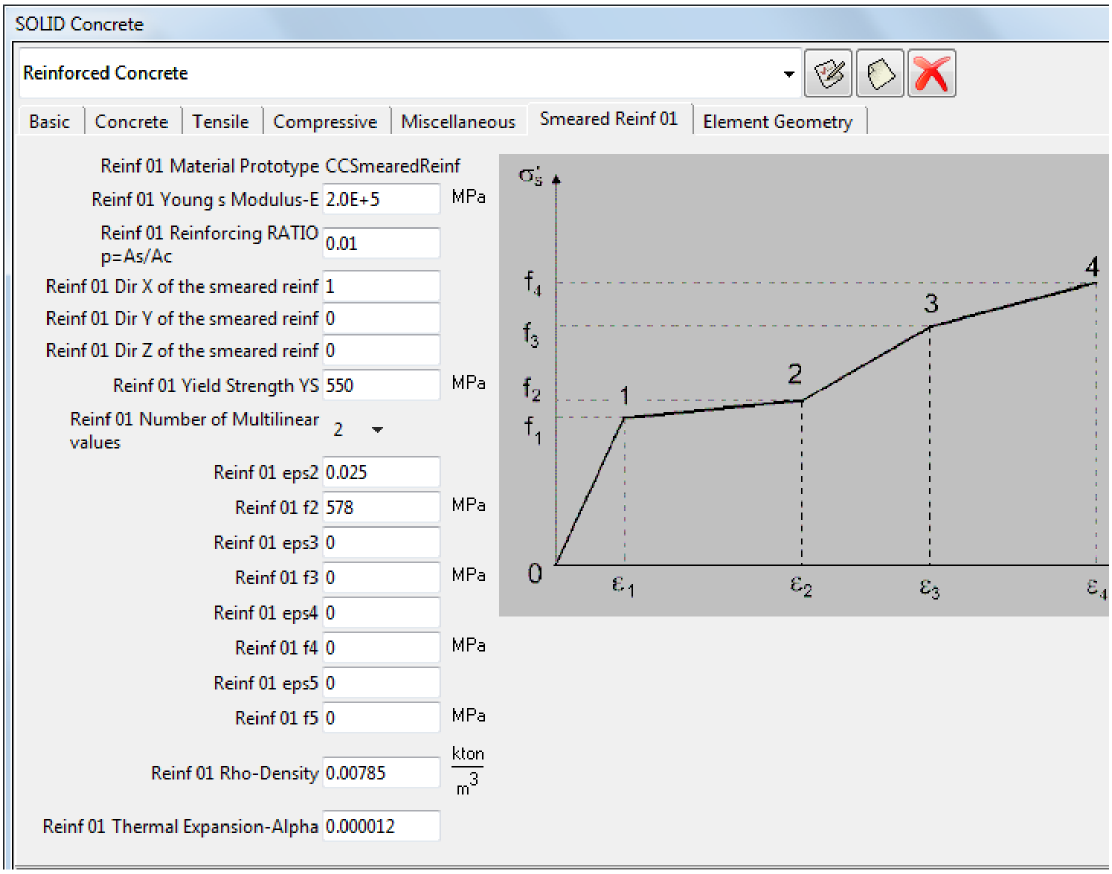


Fig. 5-58: Reinforced Concrete material properties – Smeared Reinforcement

5.3.5 1D Reinforcement Material

The basic material parameters for one-dimensional reinforcement bars are essentially the same as for the smeared reinforcement in Reinforced Concrete material (5.3.4). In the following, only the additional/different options will be explained.

There are two types of reinforcement. The **Reinforcement EC2** is used the most often. The tab EC2 can be used to define the material parameters for bars or tendons based on the reinforcement steel strength class, a few basic parameters (elastic modulus, characteristic yield strength, …) and safety format. Check the box **Generate Material** and click on the **Update Changes** icon after selecting all the parameters to generate the material.

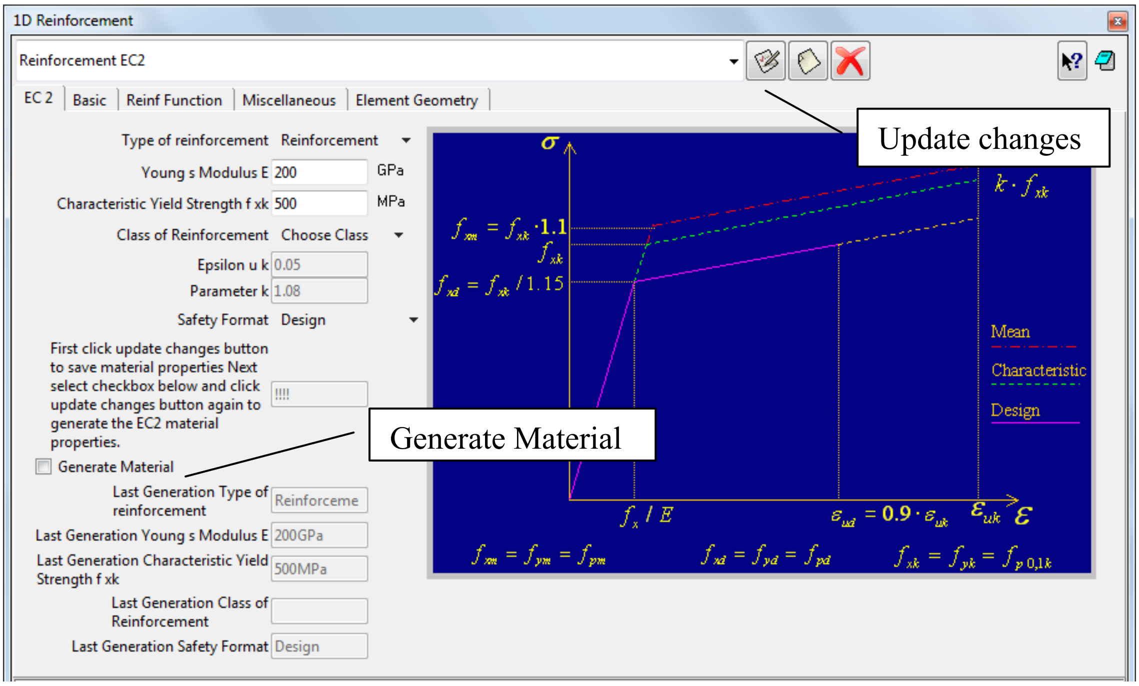


Fig. 5-59: 1D Reinforcement material properties

The second material, **Reinforcement**, has some settings different from **Reinforcement EC2**. There are four material prototypes in Basic tab. CCReinforcement and CCReinforcementWithTempDepProperties can be selected also in Reinforcement EC2. Detailed information about all material prototypes can be find in chapter 5.3 (table 1, page 24).

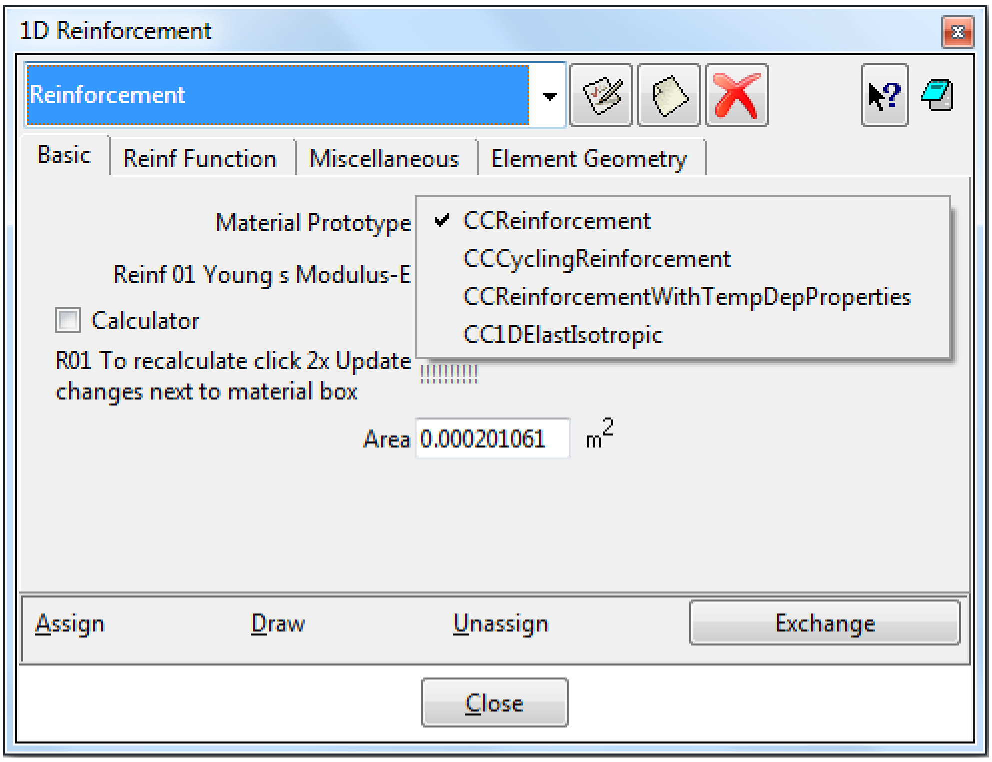


Fig. 5-57: Reinforcement material prototypes

CCReinforcementWithTemp Dep Properties - This model is used to simulate change of material properties due to current temperature. The temperature fields can be imported from a previously performed thermal analysis. Reinforcement parameters can be generated according to production method.

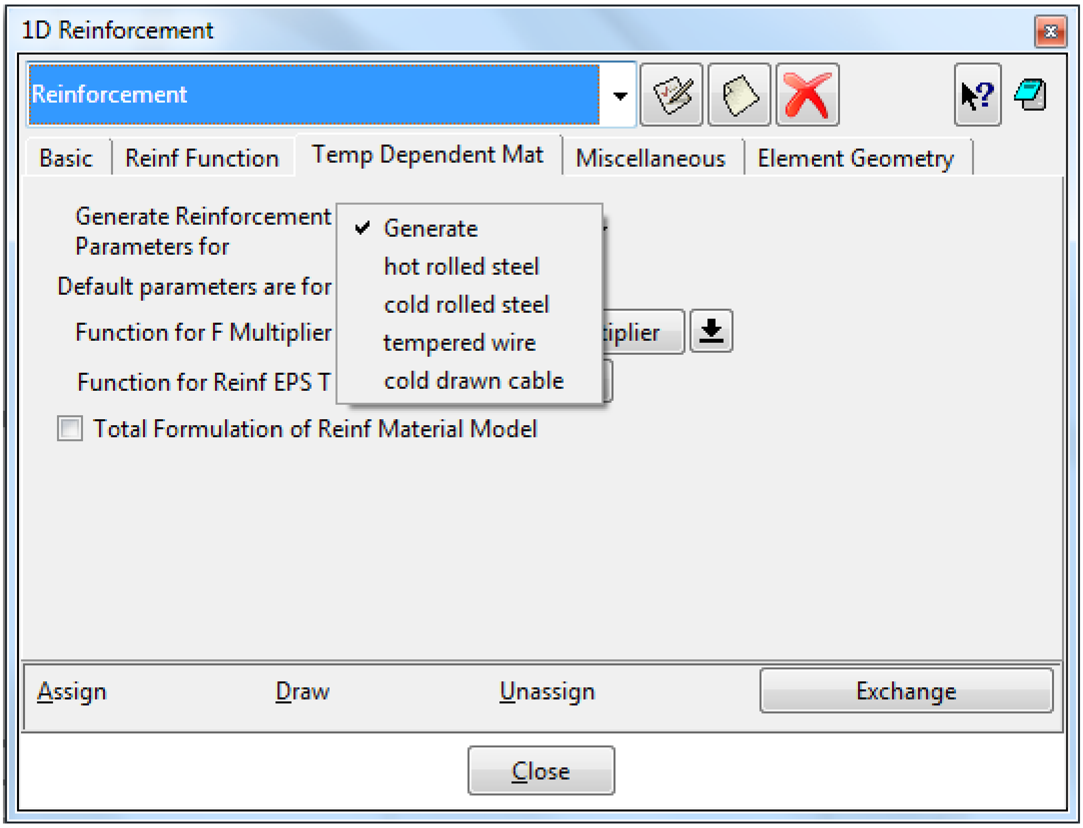


Fig. 5-58: Reinforcement material prototypes

CCCyclingReinforcement - Material for cyclic reinforcement. There is a tab Menegotto-Pinto where special parameters can be defined. Detailed information about these parameters can be find in ATENA Theory Manual [1], section 2.7.5.

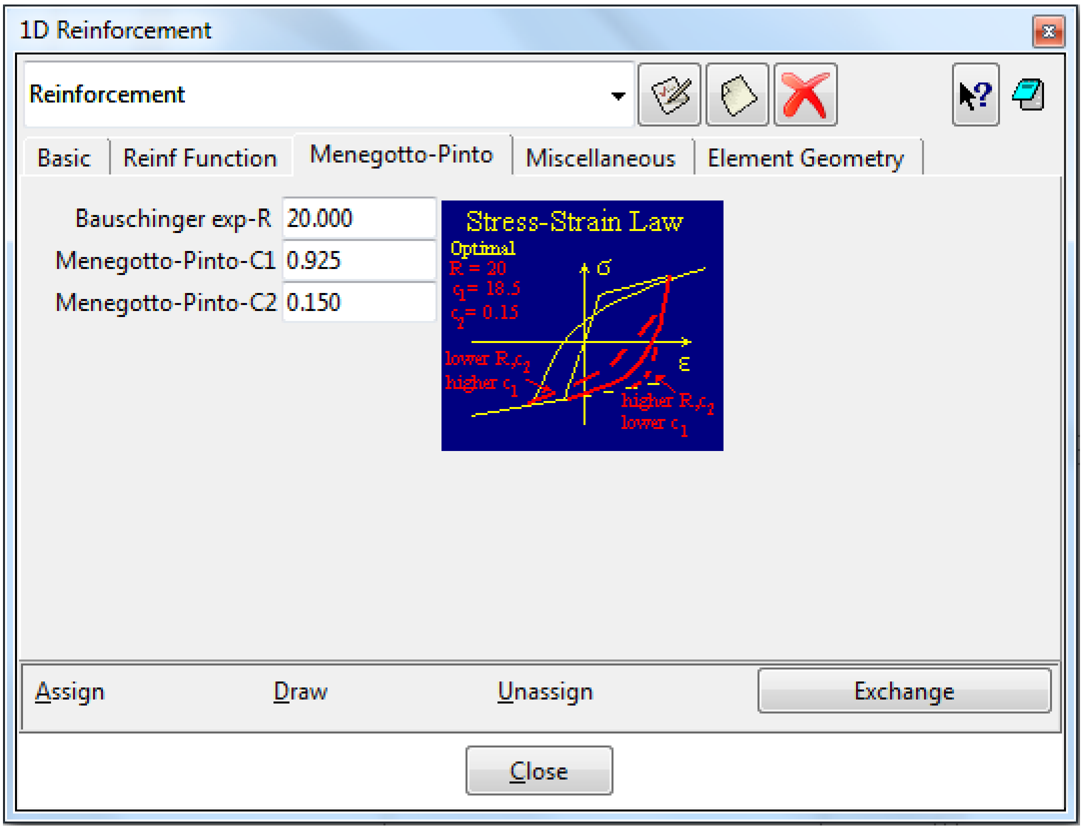


Fig. 5-59: Menegotto-Pinto

Additionally, the geometry type can be selected on the Element Geometry tab:

NORMAL – bars with perfect bond

BAR WITH BOND – bars with bond slip law

CABLE – external pre-stressing cables, only connected at anchors and deviators

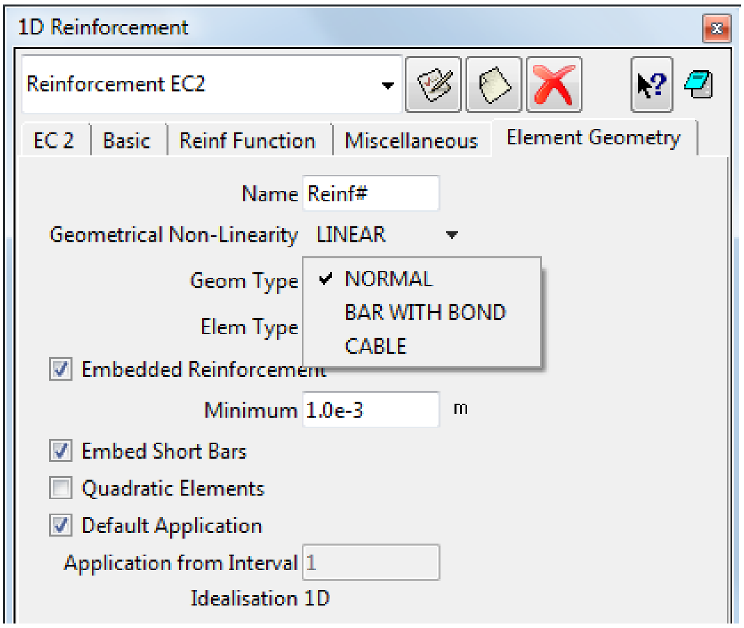


Fig. 5-60: 1D Reinforcement material properties – Element Geometry

5.3.5.1 Bond for Reinforcement

If the geometry type **BAR WITH BOND** is selected, a tab named **Bar with Bond** appears. The settings **Fixed START / END / BOTH / NONE** define where bond slip is blocked, for example due to an anchor or symmetry condition. The bar perimeter determines the steel-concrete contact area, and the function the bond slip-maximum bond stress law. Please note the stress corresponding to zero slip should be nonzero in most cases (the maximum stress the bond can transfer before the reinforcement starts to slip). See the Theory Manual [1] for details.

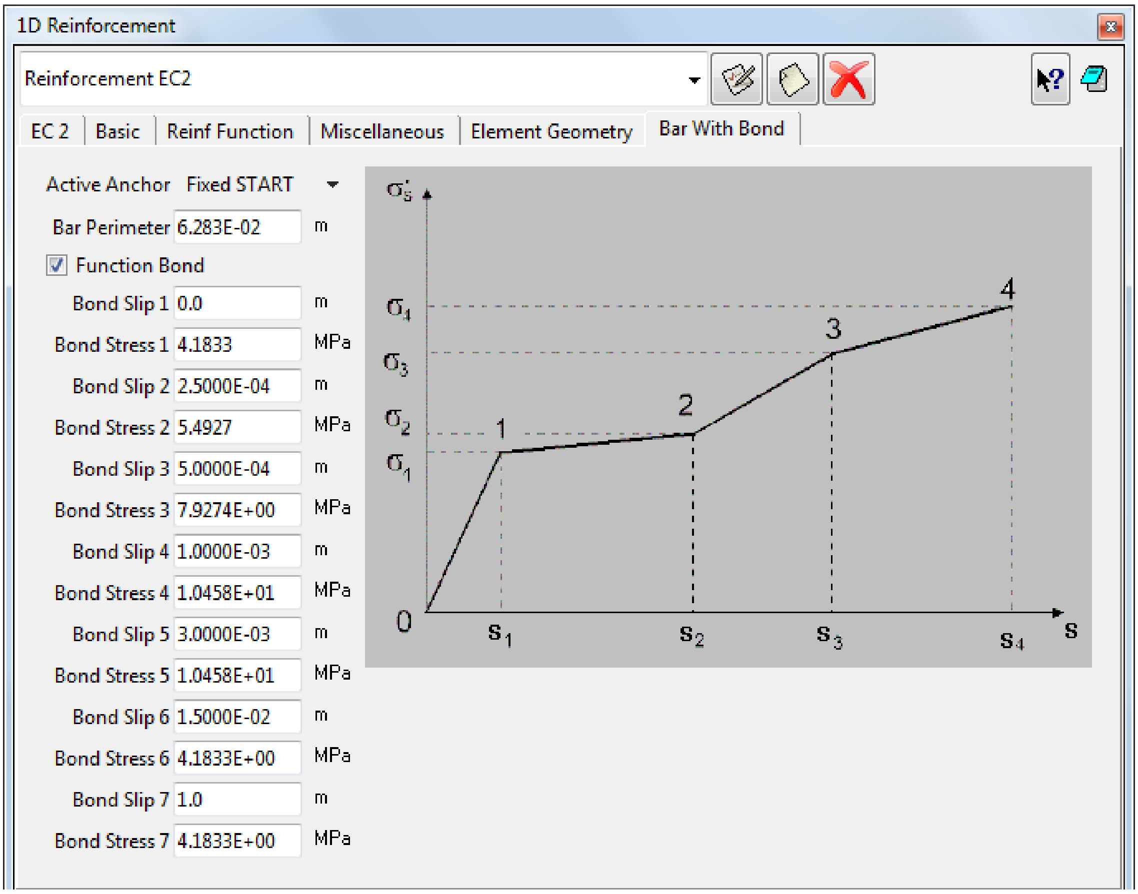


Fig. 5-61: 1D Reinforcement material properties – Bar with Bond

5.3.5.2 External Cable

If the geometry type **CABLE** is selected, the position of the active anchor (i.e., where the pre-stressing force is applied) and deviator parameters can be defined on the **Cable** tab (friction coefficient, cohesion, radius).

Friction: between the bar and the concrete

Cohesion: between the bar and the concrete, i.e., the max. stress in case of zero friction component force unit/distance unit

Radius: the radius of deviators (distance units)

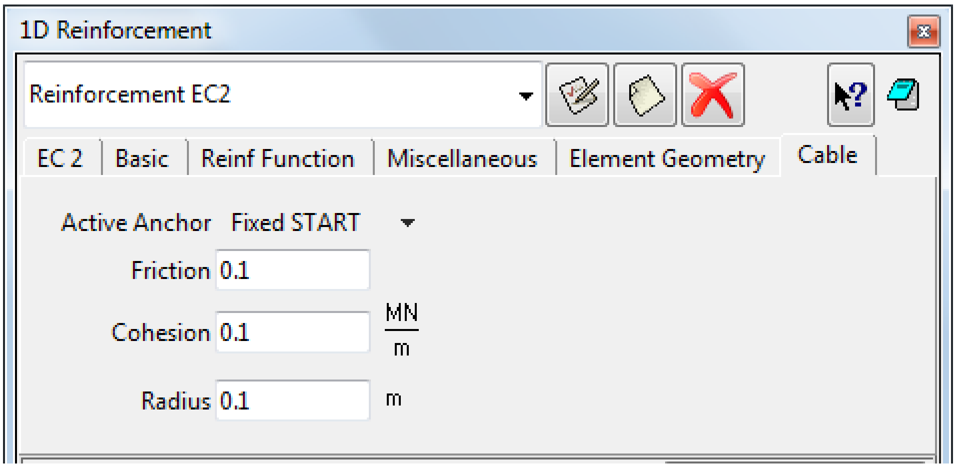


Fig. 5-62: 1D Reinforcement material properties – Cable

5.3.6 *Interface Material*

The interface material (also called GAP) has been developed to model behaviour of contacts between volumes, e.g., concrete - steel or thin layers of, e.g., mortar. This material should only be assigned to *contact volumes* (in 3D) or *contact surfaces* (in 2D). Please do not forget to choose the Material Prototype according to problem dimension (CC3DInterface or CC2DInterface) at the Basic tab.

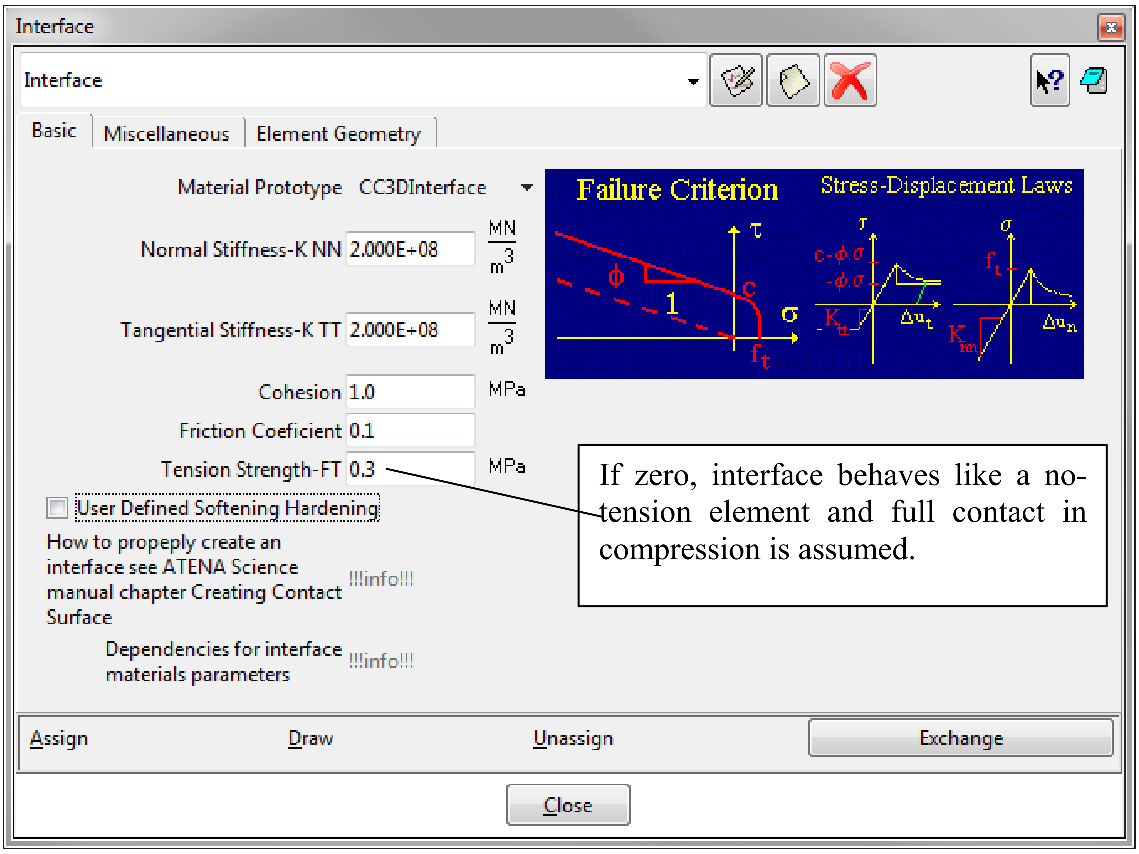


Fig. 5-63: Interface material properties – Basic

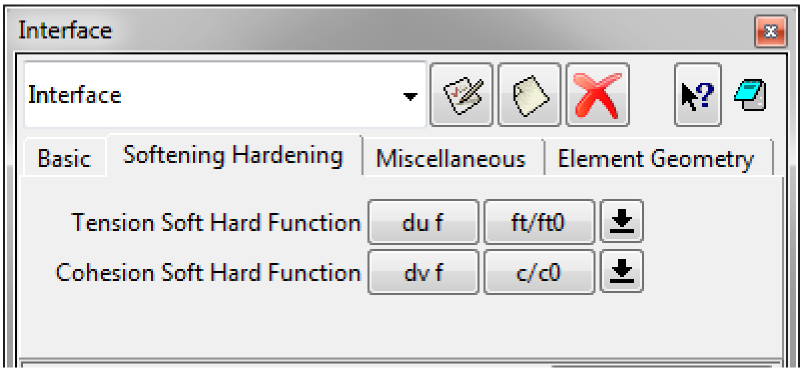


Fig. 5-64: Interface material properties – Softening/Hardening

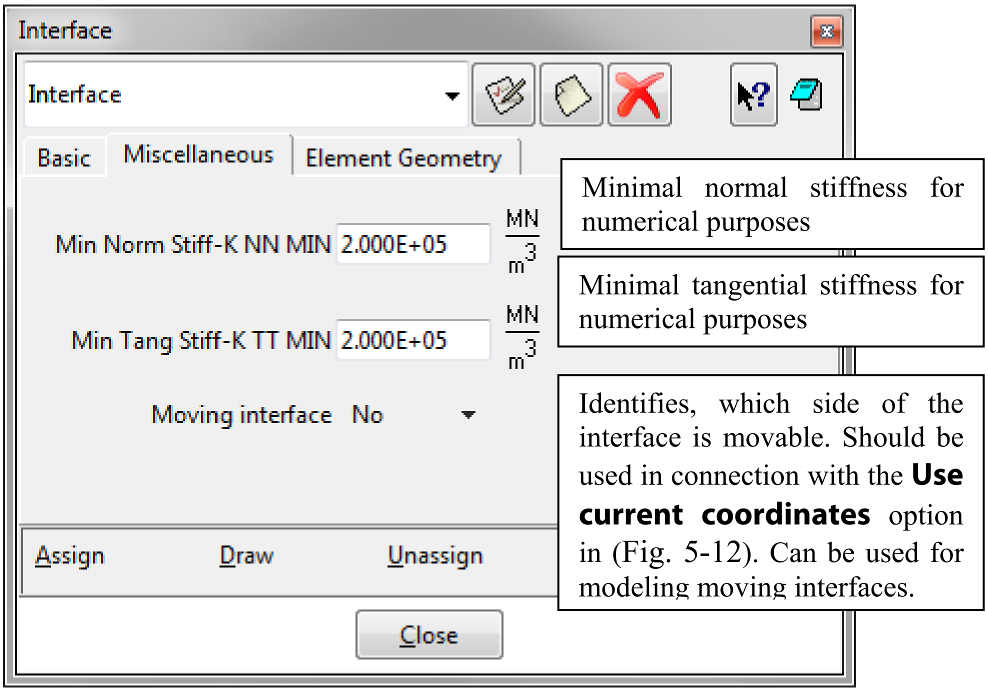


Fig. 5-65: Interface material properties – Miscellaneous

3D Interface

The normals of all surfaces have to point out of the volumes connected by the interface (i.e., both points into the contact volume). The 2 surfaces can not share any lines or points.

2D Interface

The normals of all lines have to point in the same direction (i.e., both points out of one surface and in the other surface). The 2 lines can not share any points.

In both cases, the normal directions have to be fixed before creating the contact volume/surface. An example how to create a *contact surface* is shown in section 5.3.6.4.

Refer to the *Interface Material Model* section of the ATENA Theory Manual for the explanation of the interface material parameters.

Initial_Gap_Load_for_Volume: Special type of element "load" is introduced by &ELEMENT_INITIAL_GAP_LOAD. This load is used for gaps that are initially open. Size of the opening is derived from the gap element's thickness at step INIT_STEP_ID n. See input manual: ELEMENT_LOAD description. It is not supported for 2D in **GiD** yet.

5.3.6.1 General Explanation on Ways of Connecting Neighbouring Volumes (or Surfaces in 2D)

Please understand the difference between A. compatible (shared surface) and B. incompatible (master-slave) meshes between two neighboring volumes.

In case A., all the volumes sharing surfaces build a single region from the mesh generation point of view. Basically, this means all the volumes have to be either structured or unstructured (there are ways to combine structured and semi structured and unstructured meshes, but that can only be recommended in special cases). In the FE model, the nodes on the shared surfaces belong to both volumes, and therefore there is no need for master-slave connections.

In case B., the meshes are generated independently for each volume. Master-Slave conditions (see **Fixed Contacts** in section 5.2) have to be assigned to the surfaces which should be connected (a Master-Slave connection can be even used to connect contact elements to the neighboring volume, as explained in the next section 5.3.6.2).

5.3.6.2 Contacts between Compatible Meshes

If contacts are to be introduced between a pair of neighboring volumes with compatible meshes (case A. above), the shared surface needs to be duplicated. The easiest way to do so is to move one of the volumes some distance away (such that it does not interfere with anything else in the model) and then back with the option **Duplicate entities** enabled.

The **Duplicate entities** in the Copy dialog works the following way:
If unchecked, eventual duplicate nodes, lines, surfaces are merged into one (similarly to the **Collapse** command). If checked, all are kept - nothing is merged.

For example when copying a rectangular surface just next to the original, the left line is copied over the right line. If the box is unchecked, both of them are kept and the surfaces are independent. If it is checked, the lines are merged into a single one, which is shared by the 2 surfaces.

5.3.6.3 Contacts between Incompatible Meshes

GiD only allows [prism] contact elements between surfaces of the same size and mesh settings. Therefore, if the two surfaces (lines) to be connected are of different sizes (partial contact) or with differing meshes, an extra surface (line) needs to be defined of the size of the smaller of the two, located a small distance, e.g., 0.1mm, inside the volume the bigger surface belongs to. Please keep in mind the 3 surfaces (lines) can not share any lines or points (points). The easiest way usually is to copy the smaller surface. Then, create a contact volume from the two smaller surfaces and assign the desired interface (GAP) material to it. Finally, connect the additional surface to the bigger surface using Master-Slave conditions (**Boundary conditions | surfaces | fixed contact for surface**, see the **Conditions** section (5.2) for explanation of fixed contacts).

5.3.6.4 Example - Creating a Contact Surface

The purpose of this example is to show how to create an interface between the two concrete blocks, modeled in two dimensions. The two blocks are shown in Fig. 5-66. The interface will be added at the place of the inclined line.

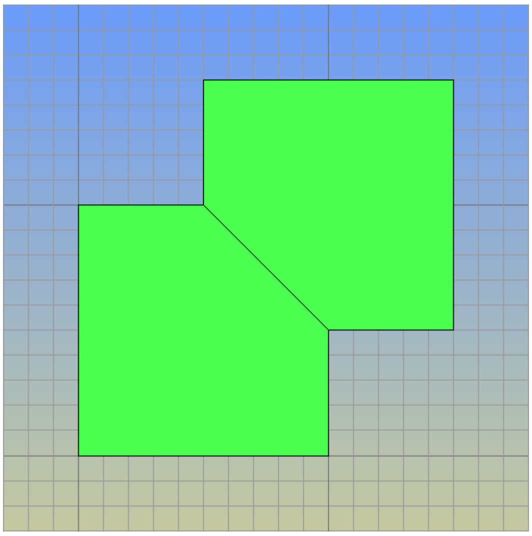


Fig. 5-66: Creating a contact surface - Introduction

The interface can be created through the following steps, illustrated in Fig. 5-67, Fig. 5-68, and Fig. 5-69:

Step 1: Create the 2 surfaces to be connected by a contact.

Step 2: Move one surface away by a small distance using **Utilities | Move**. Notice that two points (9, 10) and one line (10) is created.

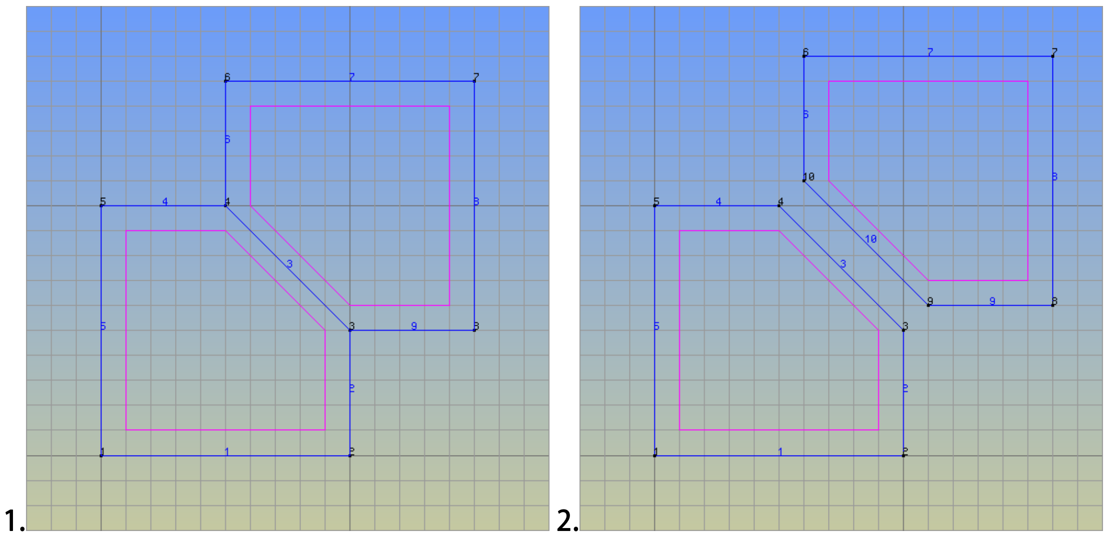


Fig. 5-67: Creating a contact surface - Steps 1-2

Step 3: Select **Utilities | Swap Normals | Lines** to check the interface line vectors. If needed, change the vector directions on the interface lines, so that both point in the same direction.

Step 4: Move the displaced surface back, with the option **duplicate entities** checked. Notice the overlapping labels of the interface lines and points.

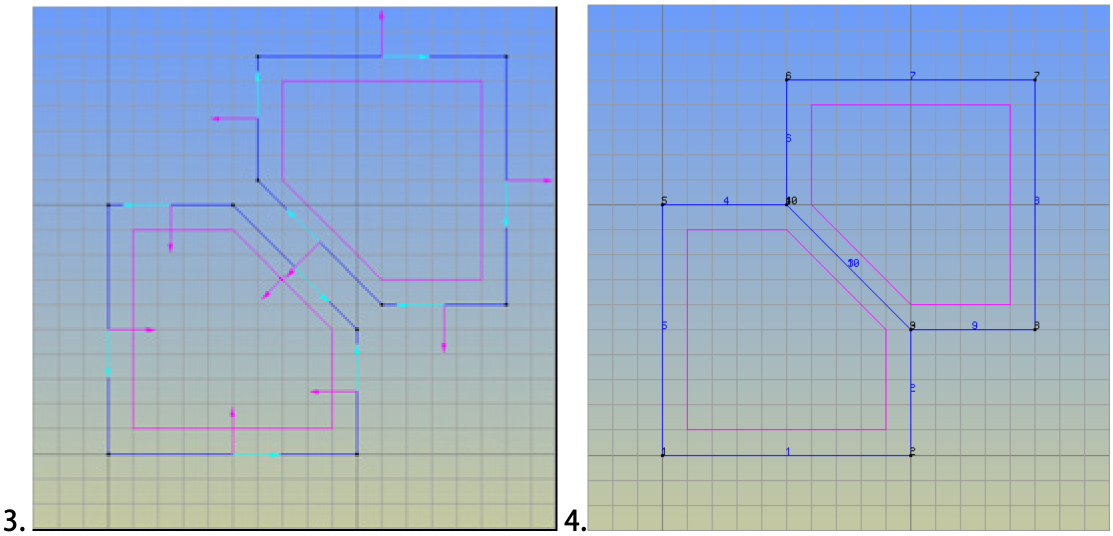


Fig. 5-68: Creating a contact surface - Steps 3-4

Step 5: Select **Geometry | Create | Contact surface** and select the two interface lines (3 and 10) in order to create the new contact surface (3). Assign the interface material to this contact surface by selecting **Data | Materials | Interface**.

Step 6: Ensure mesh compatibility for the two interface lines (3 and 10). The interface creation is now complete.

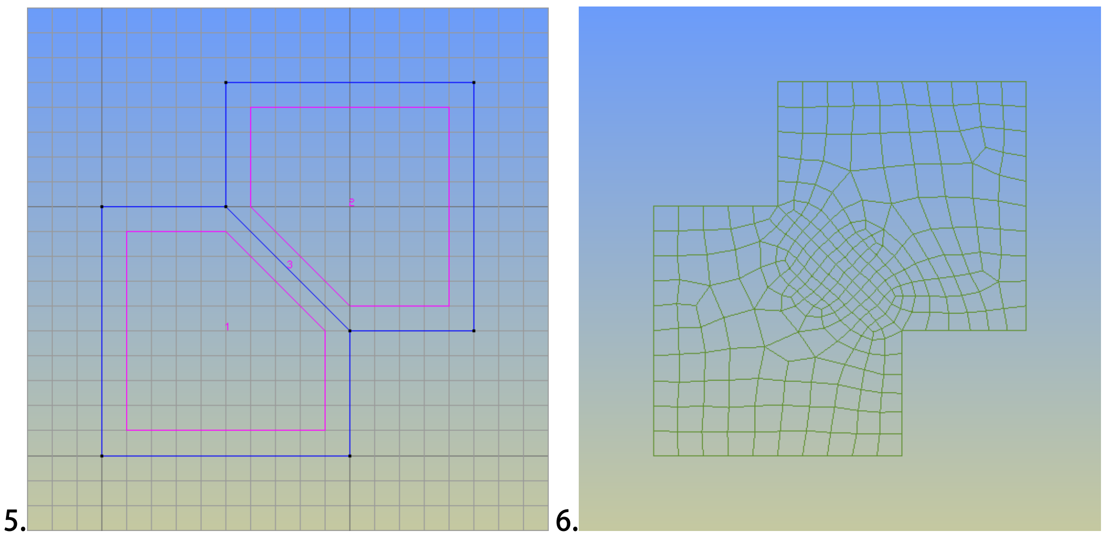


Fig. 5-69: Creating a contact surface - Steps 5-6

The procedure for a 3D interface is essentially the same, considering surfaces and volumes instead of lines and surfaces respectively, and replacing the creation command in Step 5 by **Geometry | Create | Contact | Volume**. However, the direction of the normals differs between 2D and 3D interfaces.

5.3.7 Spring material

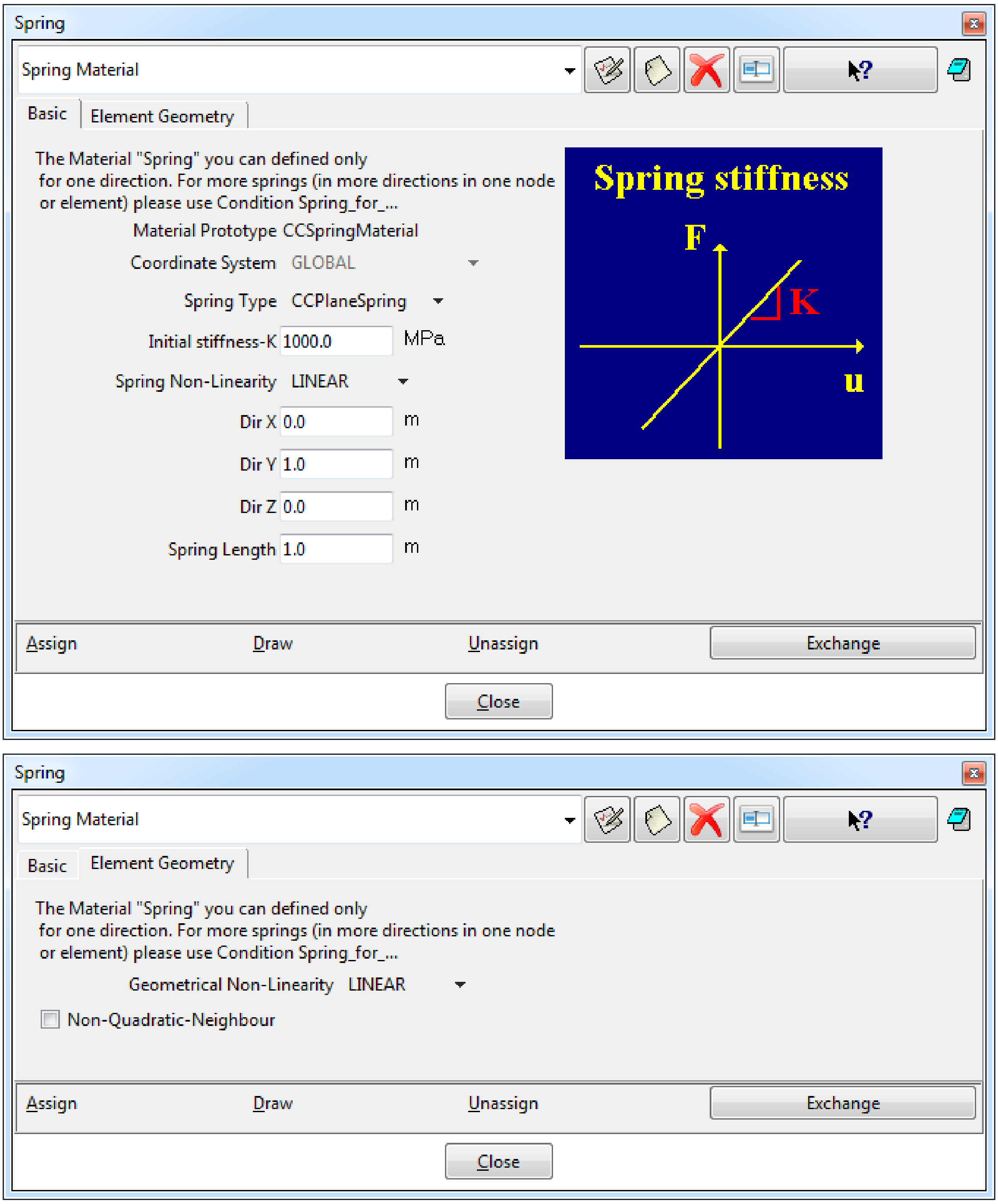


Fig. 5-70: Spring material dialog

Example to define a surface spring with 5kN/m2 response at 15mm displacement: 1. set the spring length to 1m, then 15mm displacement corresponds to relative displacement (elongation/shortening) 0.015

2. set the spring material stiffness to 0.005 [MN] / 0.015 = 0.3333333 MPa (sigma = E * epsilon)

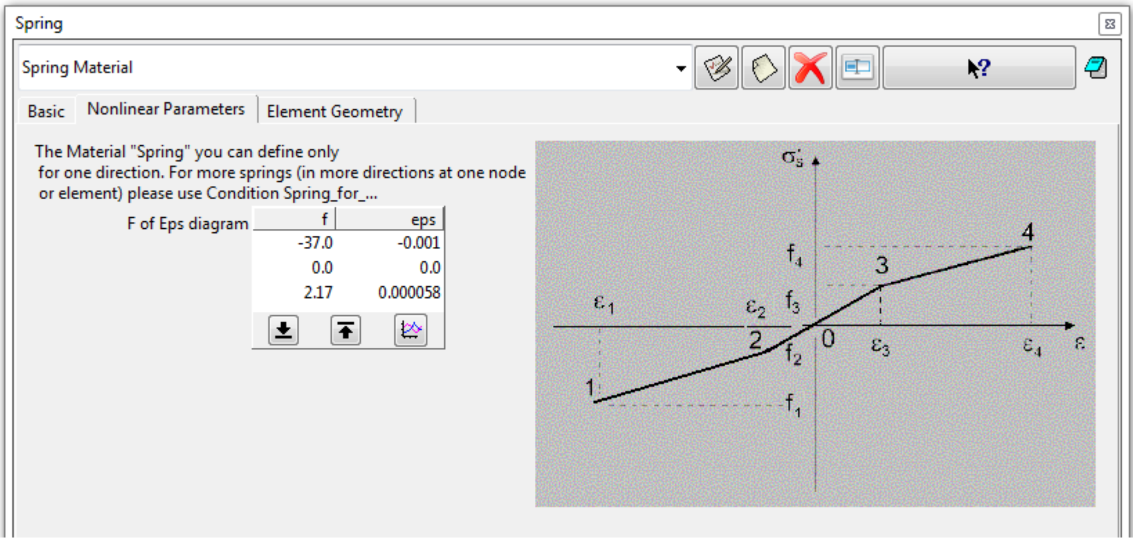


Fig. 5-71: Spring material dialog – nonlinear parameters

Imagine the spring as and elastic beam of length L (in the direction determined by the direction vector) and cross section thickness times line length.

If you have a horizontal line in a 2-D model and apply a vertical spring to it, Y is the only direction to care about.

Anyway, with nonlinear materials, you simply need to also consider geometrical nonlinearity. The switch Linear/Nonlinear geometry in ATENA only decides, if it is considered during the step iterations (nonlinear) or only the deformed shape from the end of the previous step (linear).

It is necessary to assign to the surface or line with this material to set special mesh setting. (Menu -> Mesh -> Mesh criteria -> Mesh -> (line or surface).

5.3.8 The Material Function

This material is used to easy define user function for some type of loading or material properties. You can easy import it from another GiD project. There are two ways how to define the function. The first method "USER" can be used to define x and y values in a tabular form with appropriate multipliers. The second way is to import x and y values from a file. In this case, the name of the file is to be specified. If the file does not exist, GiD will create a example file with same name, which can be edited. This example file provides the information about the necessary file format.

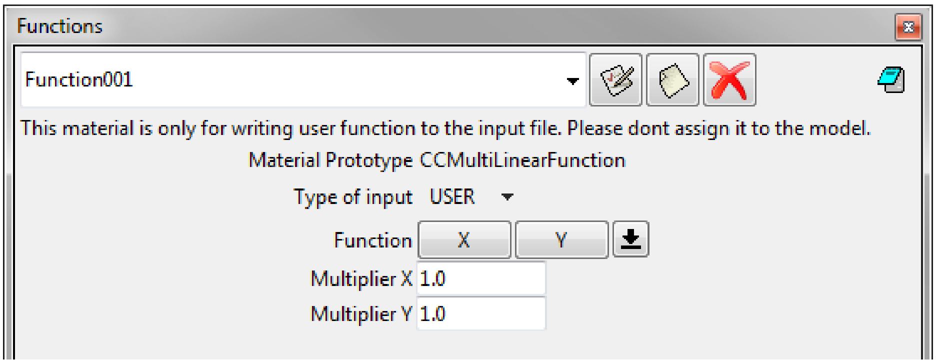


Fig. 5-72: Function material dialog

5.3.9 Material from file

This material is used to easy define user material. You just write the name of file, which contains the definition of material. If the file does not exist, GiD will create a example file with same name, which can be edited. This example file provides the information about the necessary file format.

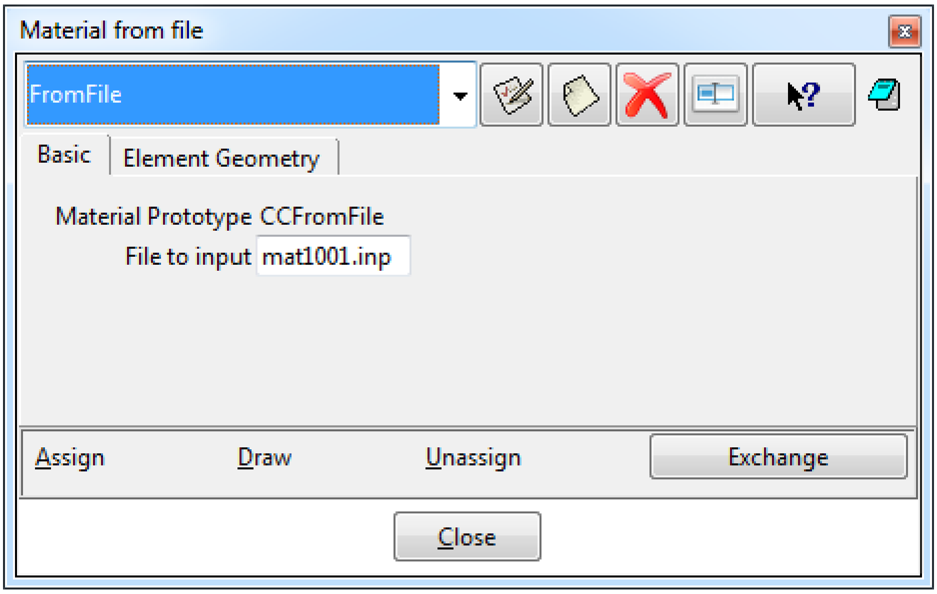


Fig. 5-73: Material from file dialog

5.4 Interval Data - Loading History

GiD recognizes 'Intervals', which approximately correspond to 'Load steps' in **ATENA**. The Interval data concept of **GiD** is used to define the loading history of the **ATENA** analysis. The load step data include the definition of loading, boundary conditions and solution methods to be used for a single analysis step. It should be noted that all conditions that are created using the command **Data | Conditions** (see Chapter 5.2) are automatically inserted into the currently active interval. By default, it is the interval number 1. Each **GiD** Interval data can be used to generate multiple **ATENA** load steps. This simplifies the model preparation if it is necessary to create many **ATENA** load steps with the same boundary and loading conditions. The user should be aware of the fact that all ***ATENA loads or boundary conditions are treated in a purely incremental fashion***. This means that a force, which is applied at certain load step, is added to the forces applied previously. If a force is to be removed, the force with the same value but opposite sign should be applied in the model.

The definition of Interval data starts by selecting the menu item **Data | Interval Data** or the icon . This command opens the dialog window as shown in Fig. 5-75, which can be used to specify the parameters for an individual interval. In this dialog, it is for instance possible to define how many **ATENA** load steps should be generated with the same conditions and parameters, or which scaling factor is to be applied to all conditions (see Chapter 5.2) in the current interval. An active Interval or a new Interval can be created using the menu **Data | Interval**. If it is necessary to create a new interval with the same conditions and properties as the current one, the best approach is to open the Interval data dialog (using the menu item **Data | Interval Data** or icon) and then using the copy button . The current interval can be change by icon .

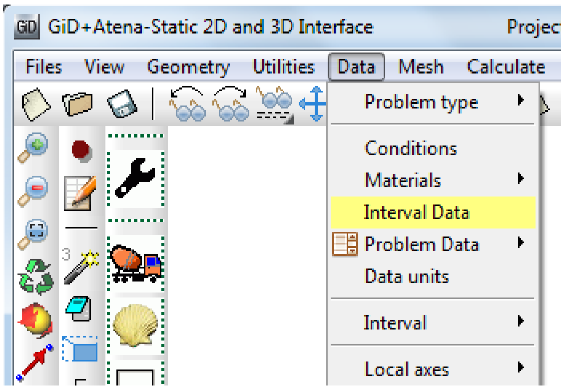


Fig. 5-74: Load steps (intervals)

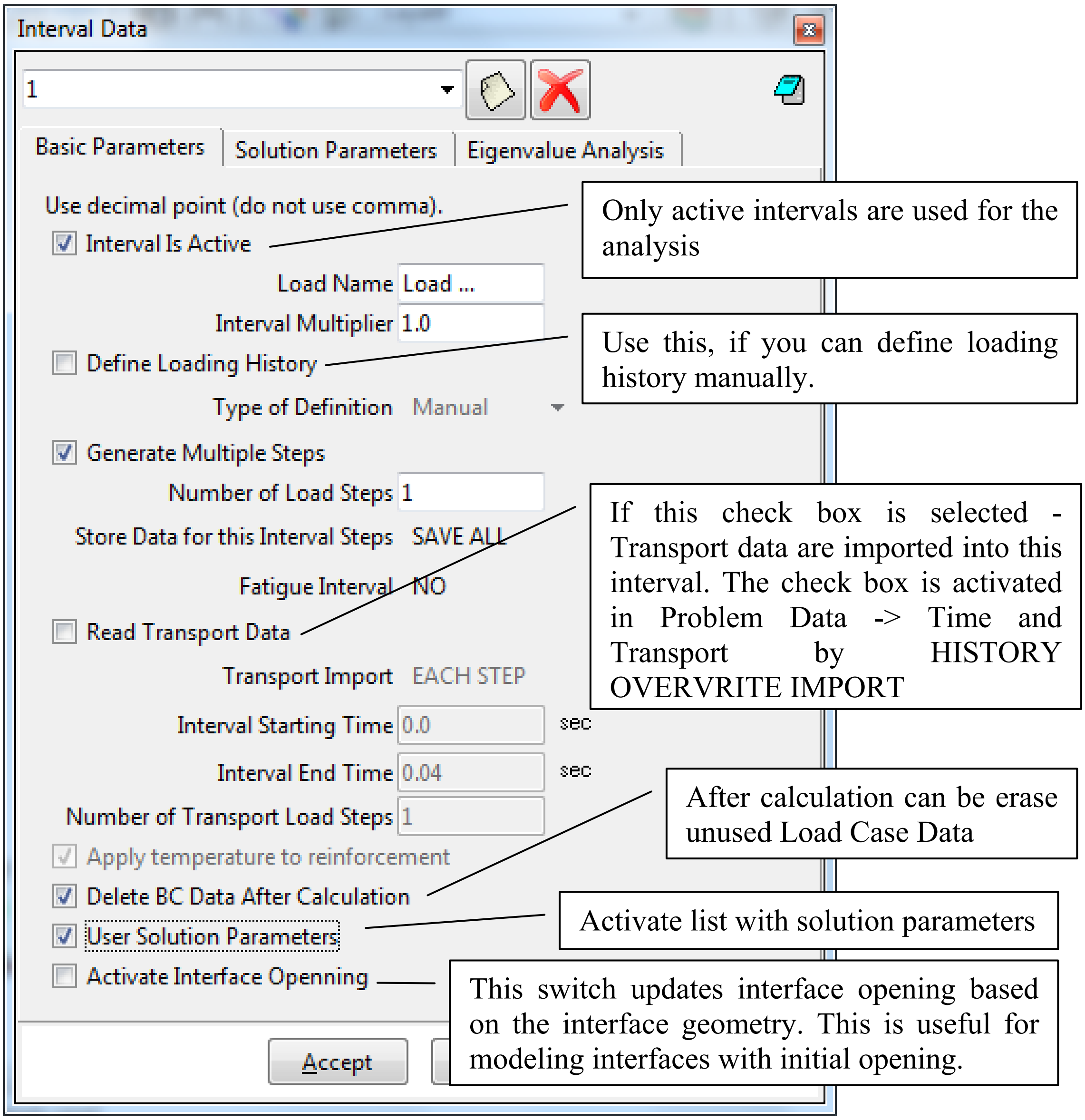


Fig. 5-75 Interval Data window - Basic parameters

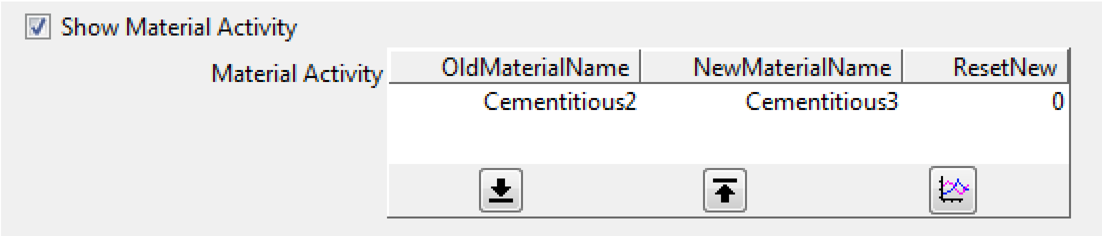


Fig. 5-76 Interval Data window – Material activity

This is new option how to set material activity for the construction process. Old Material name is name of material which is assigned to the geometry. So if you change the material second time, the Old material is still the same. ResetNew parameter set the material state to the zero.

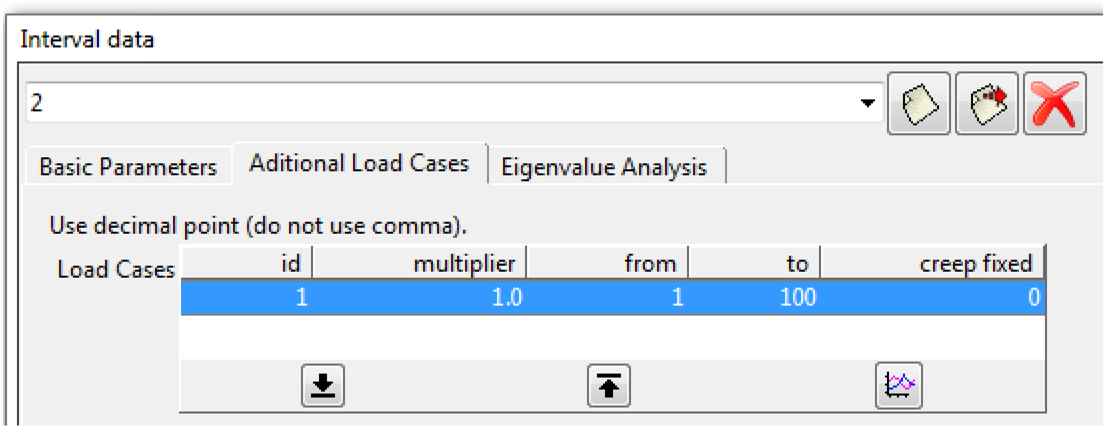


Fig. 5-77 Interval Data window – Aditional load cases

This is option to add another load case to the interval. For this case you need to disable “Delete BC Data After Calculation” in the interval, which load case you will use. The number of load case is in most cases the same as the number of interval. With this option you can add all supports to only first interval, and this load case added to each other intervals.

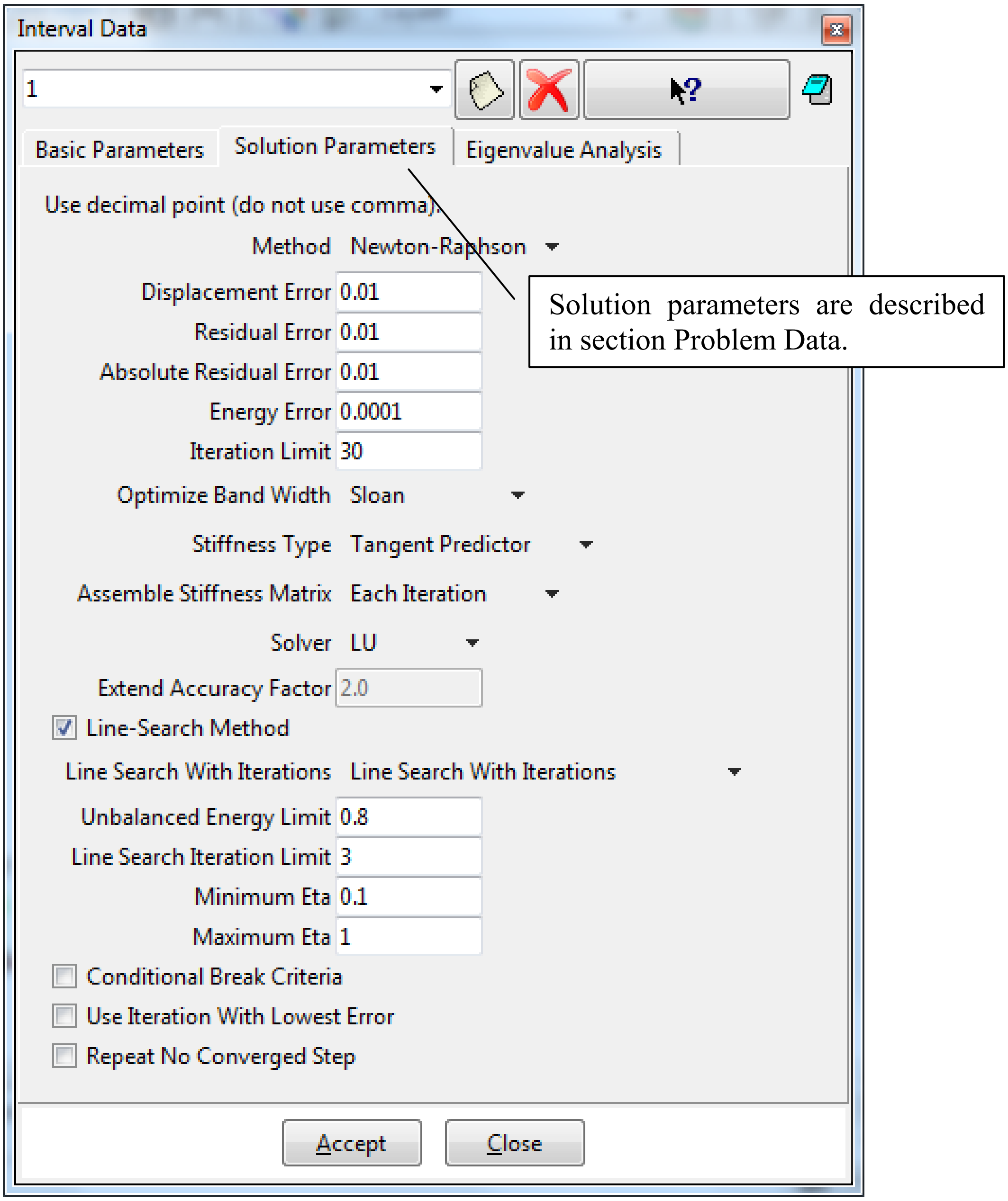


Fig. 5-78 Interval Data - Solution Parameters

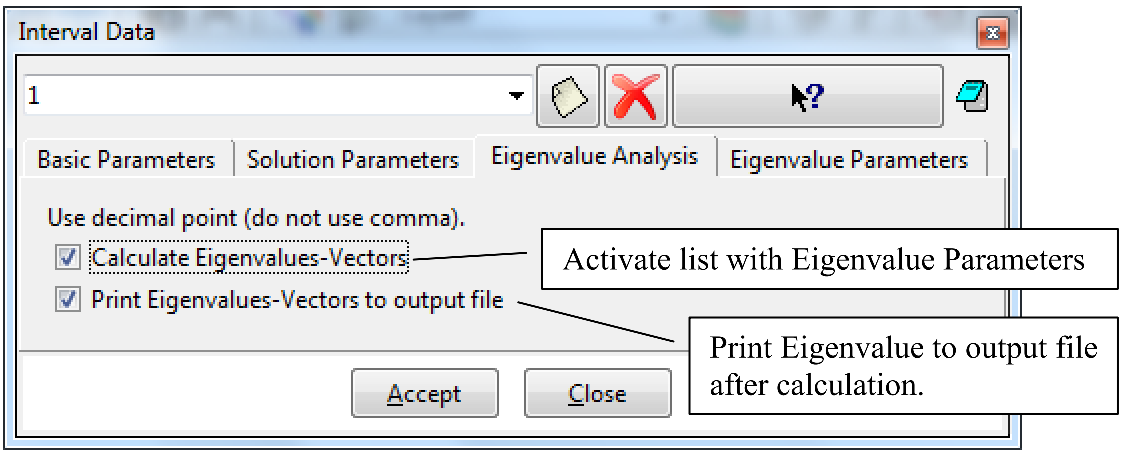


Fig. 5-79 Interval Data - Eigenvalue Analysis

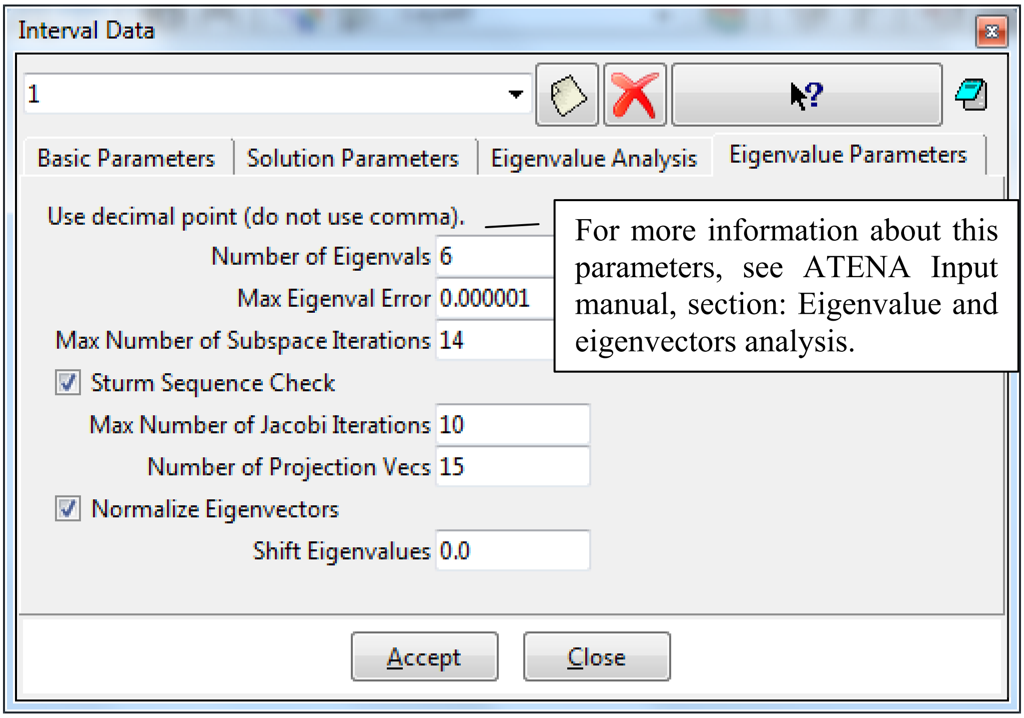


Fig. 5-80 Interval Data - Eigenvalue parameters

5.4.1 Fatigue

To consider fatigue influence of cyclic loading on the tensile properties of concrete, set the option **Fatigue Interval** to other value than the default **NO**.

Basically, **RESET AND CALCULATE** marks the interval as the cycling load, i.e., FATIGUE_TASK 3 (1 store base stress + 2 reset FATIGUE_MAX_FRACT_STRAIN) at the first load step of the interval and FATIGUE_TASK 4 (calculate fatigue damage) at the last step. The calculated damage is stored in FATIGUE_CYCLES_TO_FAILURE and FATIGUE_MAX_FRACT_STRAIN.

The option **APPLY** sets FATIGUE_TASK to 8 (apply the fatigue damage) and FATIGUE_MAX_FRACT_STRAIN_MULT to 1/num.of steps in the interval. Simply said, the previously FATIGUE_MAX_FRACT_STRAIN is added to the MAX_FRACTURING_STRAIN.

Note that all these settings only have influence when the base material prototype "CC3DNonLinCementitious2Fatigue" (described in section 5.3.1.3) is selected for at least one of the concrete materials assigned in the model. Please see the ATENA Theory Manual [1] and ATENA Input Manual [4] for more details on the fatigue model implemented in **ATENA**. Also the articles referred form the fatigue material section in ATENA Theory can be recommended.

5.4.1.1 How to consider Fatigue in ATENA

For materials (e.g., reinforcement bond) or situations (e.g., concrete in compression) with no explicit fatigue modeling support in ATENA, you can evaluate the fatigue life outside of ATENA (e.g., in a spreadsheet), based on the classical S-N (Wöhler) curves (or another approach) using the cyclic stress range (or strain range or whatever) from the ATENA analysis. For the supported materials and situations, see below.

5.4.1.1.1 Low-cycle fatigue

For low-cycle fatigue when all the load cycles are explicitly applied (i.e., every loading and unloading is applied to the model), let the option **Fatigue Interval** set to the default **NO** and use the normal "CC3DNonLinCementitious2" material prototype. Define the loading history explicitly, i.e., all loadings and unloadings.

When doing so, you should typically use the Cyclic Reinforcement material model (with Bauschinger effect/Menegotto-Pinto) for reinforcement (see also 5.3.5).

5.4.1.1.2 High-cycle fatigue with negligible redistribution

If the effects of stress redistribution are negligible during the fatigue life of the structural element being modelled, a simplified approach can be used. A typical example is a specimen cyclically subjected to direct tension loading.

Define the following intervals:

Int1. Loading up to the base (cycle bottom) level, **Fatigue Interval**: **NO**

Int2. Increasing the load from the base level to the upper (cycle top) level, **Fatigue Interval**: **RESET AND CALCULATE, Number of Fatigue Cycles**: maximum number of cycles expected or of interest (c^{max})

Int3. Introduce the fatigue damage – no load change (i.e., only apply the supports), **Fatigue Interval**: **APPLY**

To evaluate the number of cycles "survived" or "cycles to failure" c^f, note the number of the last converged analysis step S^{lc}, subtract the number of steps in previous intervals (1+2) $S^{l1} + S^{l2}$ from it, then divide by the number of steps in Interval 3, and multiply with the number of cycles defined in Interval 2:

$$c^f = (S^{lc} - (S^{l1} + S^{l2})) / S^{l3}\ c^{max}$$

One could also say each step in Int3 corresponds to c^{max} / S^{l3} cycles.

Simplified evaluation using Fatigue Cycles to Failure

Another, even simpler, option to evaluate the number of fatigue cycles is to simply take the minimum value of FATIGUE_CYCLES_TO_FAILURE. That can be done at the end of Interval 2, and Interval 3 is not needed to be defined at all.

5.4.1.1.3 High-cycle fatigue including the effects of redistribution

To consider the effects of load redistribution during the cycles, it is needed to unload and reload multiple times. One could see it as always modelling a group of cycles, then one cycle explicitly to capture the redistribution, then the next group of cycles, etc. Due to the exponential character of the process, it is efficient to combine the cycles into groups of exponentially growing numbers of cycles, e.g., 10 – 20 – 40 – 80 – 160 – 320 – 640 – 1280 – 2560 – 5120 – etc.

Intervals 1, 2, 3 are defined the same way as above (5.4.1.1.2), just the number of cycles applied corresponds to the first group of cycles (and not the expected maximum), e.g., 10. The next is unloading to the base level, followed by another fatigue calculation similar to Int2, and another damage application like in Int 3.

Int1. Loading up to the base (cycle bottom) level, **Fatigue Interval**: **NO**

Int2. Increasing the load from the base level to the upper (cycle top) level, **Fatigue Interval**: **RESET AND CALCULATE, Number of Fatigue Cycles**: number of cycles in the first cycle group (c^{G1})

Int3. Introduce the fatigue damage – no load change (i.e., only apply the supports), **Fatigue Interval**: **APPLY**

Int4. Unloading down to the base (cycle bottom) level, **Fatigue Interval**: **NO**

Int5. Increasing the load from the base level to the upper (cycle top) level, **Fatigue Interval**: **RESET AND CALCULATE, Number of Fatigue Cycles**: number of cycles in the second cycle group (c^{G2})

Int6. Introduce the fatigue damage – no load change (i.e., only apply the supports), **Fatigue Interval**: **APPLY**

Ints7-9 for the third cycle group, 10-12 for the fourth, etc.

The evaluation is based on the same formula as above (5.4.1.1.2), just used for the interval to which the last converged step belongs. We recommend preparing a spreadsheet which calculates the number of cycles from the number of the last converged step. A sample one is available upon request.

5.5 Problem Data

The solution parameters such as number of iterations, convergence criteria or the solution methods for an **ATENA** analysis are defined in the menu item **Data | Problem Data**, Fig. 5-81 or icon . The dialog window is opened and default data are offered.

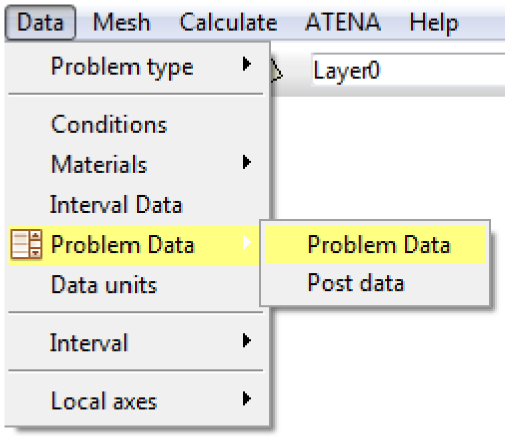


Fig. 5-81 Problem Data.

At the Global Settings **Taskname** can be any name chosen by user, and it affects the naming convention, which is used for the generated input file or other working files for the **ATENA** analysis.

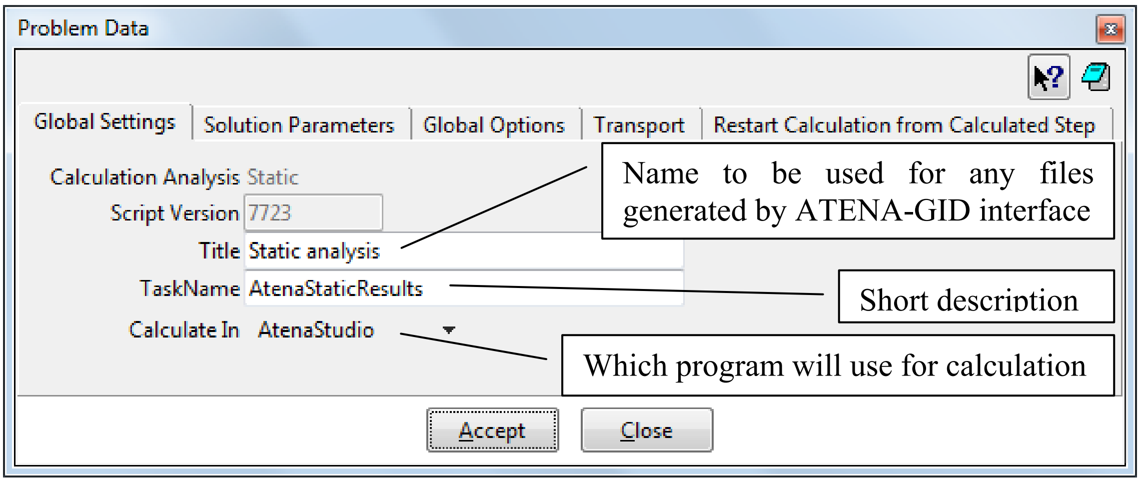


Fig. 5-82 Problem data – Global Settings.

The Solution Parameters list covers the solution parameters for non/linear methods. Their proper choice is important for a successful analysis. The meaning of solution parameters can be found in the **ATENA** documentation, Part 1 – Theory [1] and Part 2 - Users Manual [2].

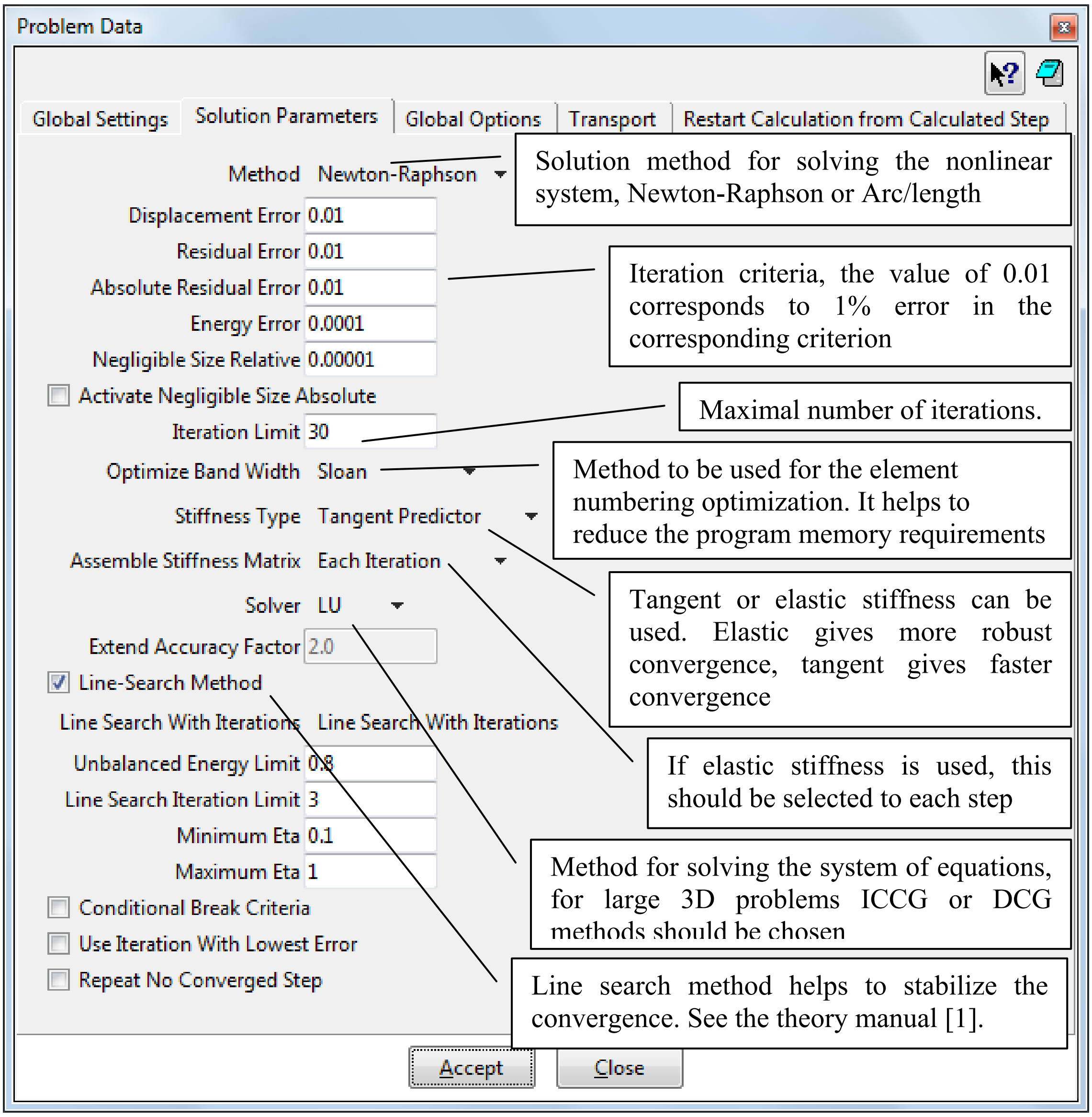


Fig. 5-83 Problem data – Solution parameters.

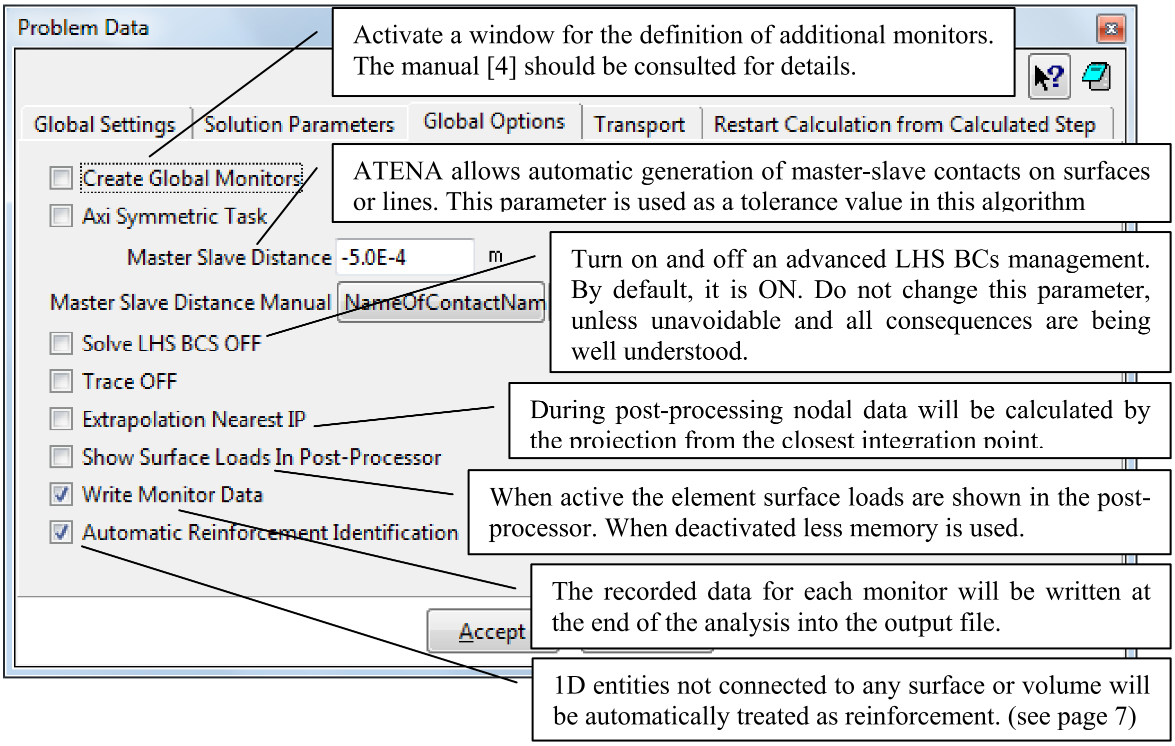


Fig. 5-84 Global Options in problem data dialog

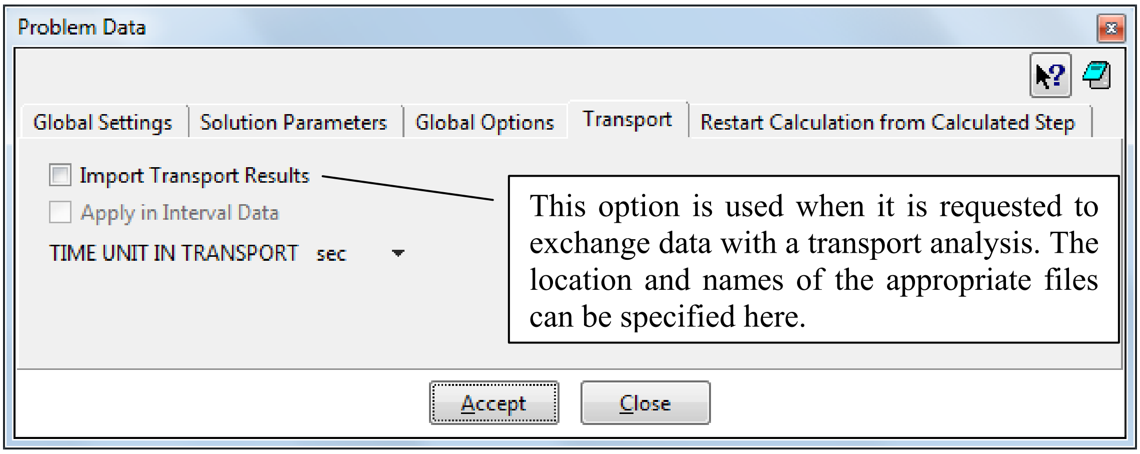


Fig. 5-85 Problem data – Solution parameters.

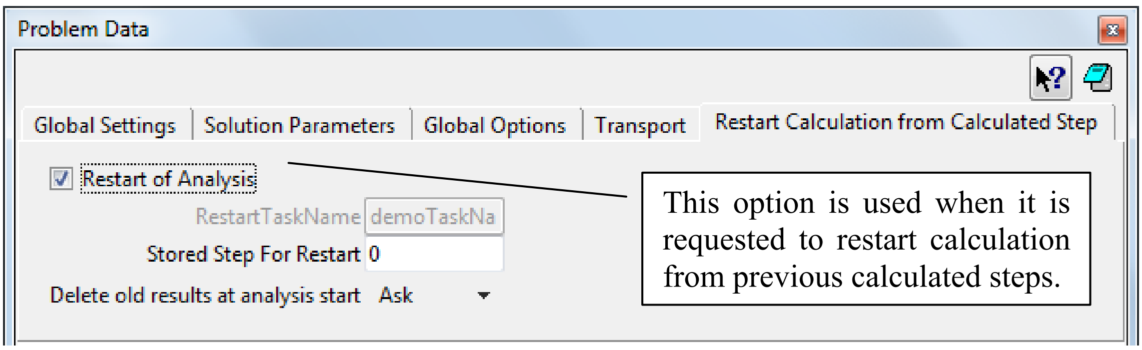


Fig. 5-86 Restart calculation options in problem data dialog

5.6 Units

Standard units in **ATENA** are SI units, which are active automatically as a default unit set, Fig. 5-87. It is also possible to define other sets of units. This can be done in the menu **Data | Data units**, where in the dialog window **data units** you can change the **Base system**. The **Model Unit** always has to be selected consistently with the **Units System**.

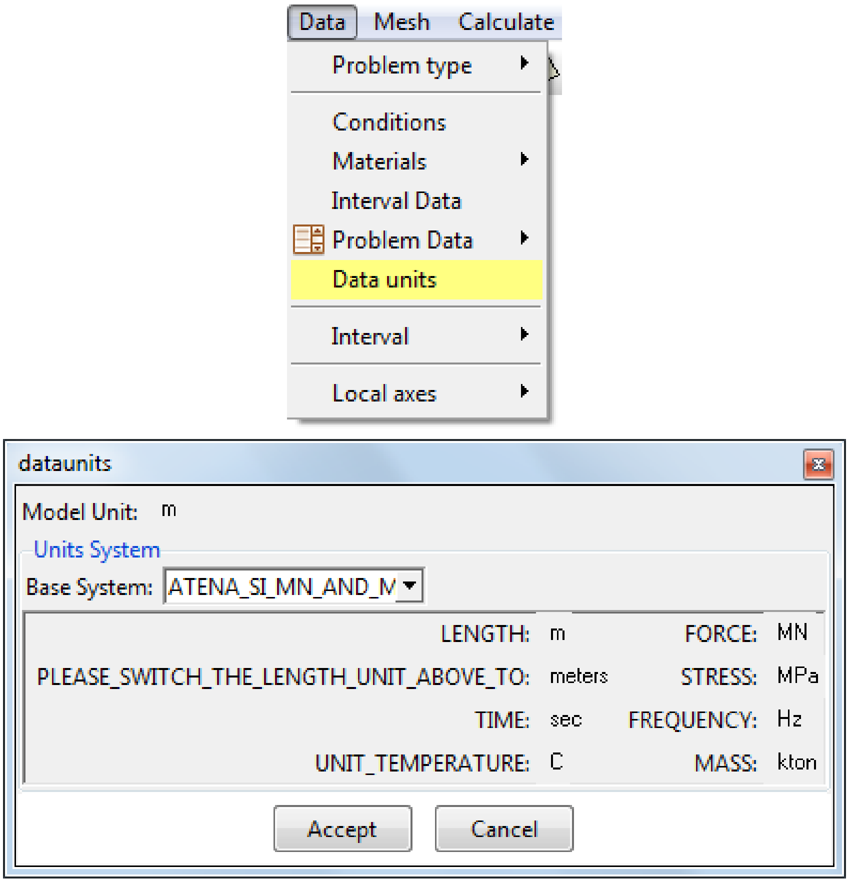


Fig. 5-87 Data units, default set.

In general the structural analysis is independent of units and can be performed in any units. The units of results are the same as those of input. In case of other units it should be realized, that the numerical values of material parameters may change. Consequently, the default material parameters in SI units offered in **GiD** cannot be used and must be modified, as it is necessary for the selected set of units.

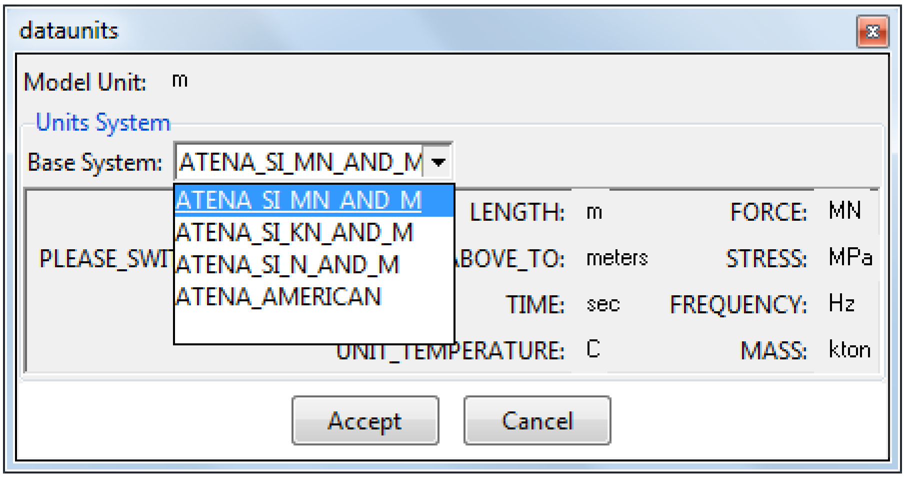


Fig. 5-88 Definition of units and possible set of alternative units.

5.7 Finite Element Mesh

The generation of a finite element mesh in **GiD** is done from the menu **Meshing**. Please, refer to **GiD** documentation for details. Here, we shall mention only meshing of reinforcing bars, which is specific for **ATENA**.

The geometrical model of a bar (discrete) reinforcement is modelled by one dimensional entities, i.e. lines. Since **GiD** does not have a capability to generate embedded bar elements, this operation is performed later at the beginning of the **ATENA** analysis. For this we need to export the geometrical forms of the bars. Since **GiD** can export only finite elements, it is always necessary to first generate some 1D truss elements along each line, which represents the reinforcement (see also page 18). It is therefore recommended to select the meshing properties of these reinforcement lines such that a single finite element is generated by **GiD**. This finite element is then used in **ATENA** to generate the embedded discrete bars depending on its intersections with the solid model. Of course, circular (or curved) bars should be meshed with more elements in order to capture the curved geometry (for example at least 8 divisions for a circle).

5.7.1 Notes on Meshing

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

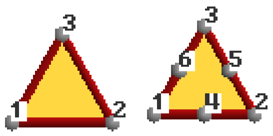
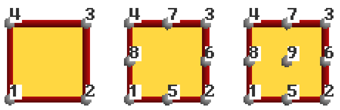
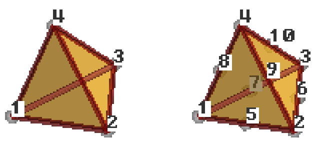
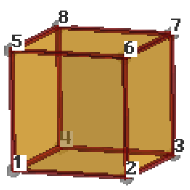
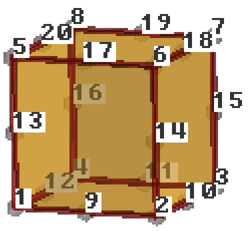
A bad mesh, like a single layer of volume elements in a region where bending plays a significant role, can produce very wrong results – see the "Mesh Study" example in the ATENA Engineering Example Manual [7]. A minimum of 4-6 elements per thickness is recommended for at least qualitative results in bending. Alternatively, shell elements may be used (see section 5.3.2).

Another frequent example of a problematic mesh are elements with extreme aspect ratios, in other words, the ratio of element edge lengths = longest to shortest edge of an element. A maximum of 3:1-4:1 is recommended for volume elements and also for surface elements in 2D models or on membranes. The higher the aspect ratio, the worse the conditioning of the system matrix, which can lead to numerical problems in the solver. For shell elements, it is no problem when the edges in the thickness direction are much shorter than the others, however, for the ratio of the two other directions (i.e., in-plane), the same condition as for normal volume elements should be fulfilled (i.e., up to 3-4:1).

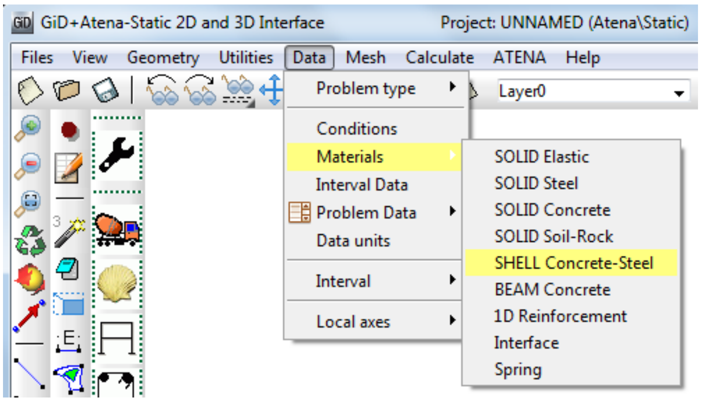
5.7.2 Finite Elements for ATENA

In each volume we must choose a type of finite element. Following types can be used in **ATENA** (in parenthesis we give also the number of nodes and a code name used in **ATENA**).

Table 3: Element library compatibility

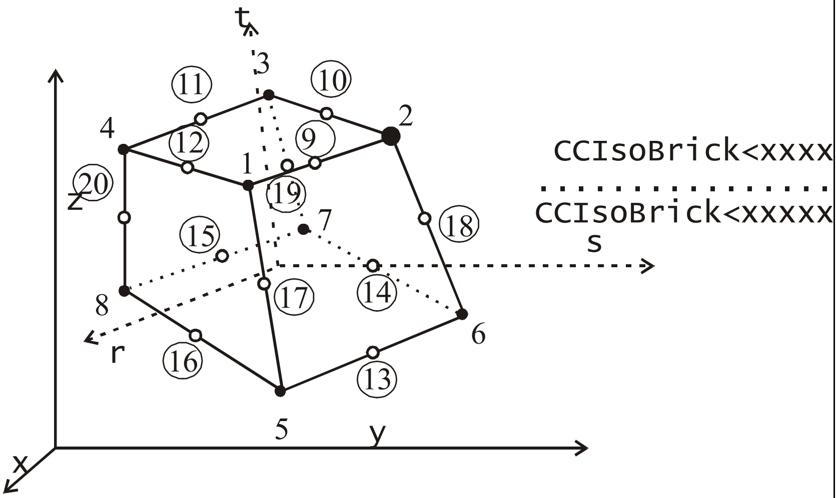
	*Linear* and *quadratic* line element 2-nodes, CCIsoTruss<xx> 3-nodes, CCIsoTruss<xxx>)
	*Linear* and *quadratic* triangular element 3-nodes, CCIsoTriangle<xxx> 6-nodes, CCIsoTriangle<xxxxxx>)
	*Linear* and *quadratic* quadrilateral elements 4-nodes, CCIsoQuad<xxxx> 8-nodes, CCIsoQuad<xxxxxxxx> 9-nodes, CCIsoQuad<xxxxxxxxx>
	*Linear* and *quadratic* tetrahedral elements 4-nodes, CCIsoTetra<xxxx> 10-nodes, CCIsoTetra<xxxxxxxxxx>
 	*Linear* and *quadratic Hexahedron* (structured mesh) 8-nodes, CCIsoBrick<xxxxxxxx> 20-nodes, CCIsoBrick<xxxxxxxxxxxxxxxxxxxx> 20-nodes, CCAhmadElement32L9 – special 3D element, which externally looks as a 20 node brick, but is internally formulated as a shell element. Good element for large scale analysis of complex structures, when large elements are needed, such as bridges, slabs etc. The shell element is activated by assigning the Shell material to 20-node brick elements.

<table>
<tr>
<td></td>
<td>

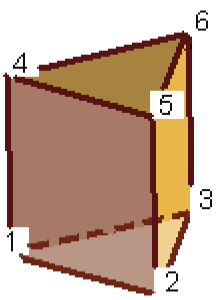


20-nodes, CCBeamNL – this is another special 3D element available in ATENA. This element on the input appears as standard 20 node element, but internally it is formulated as a fiber beam element. It is suitable for large scale analysis, when meshes with large elements are necessary.

However, ATENA is using a different nodal numbering than GiD, this means that during the export of the ATENA input file, the nodal numbering is modified to correspond with the ATENA format, as it is described in the figure below.



</td>
</tr>
<tr>
<td>



</td>
<td>

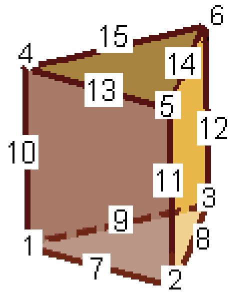
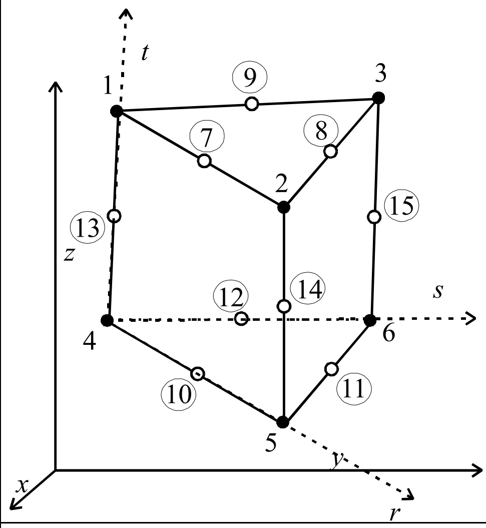
Linear and *quadratic Wedge* (structured mesh)

6-nodes, CCIsoWedge<xxxxxx>

15-nodes, CCIsoWedge<xxxxxxxxxxxxxxx>

However, ATENA is using a different nodal numbering, this means that during the export of the ATENA input file, the nodal numbering is modified to correspond with the ATENA format, as it is described in the figure below.

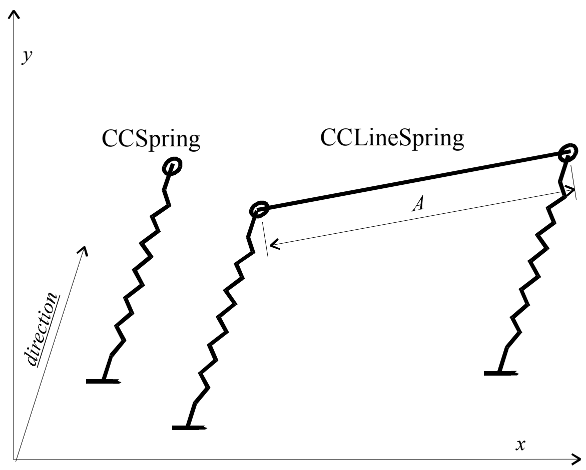
</td>
</tr>
</table>

<table>
<tr>
<td>
</td>
<td>

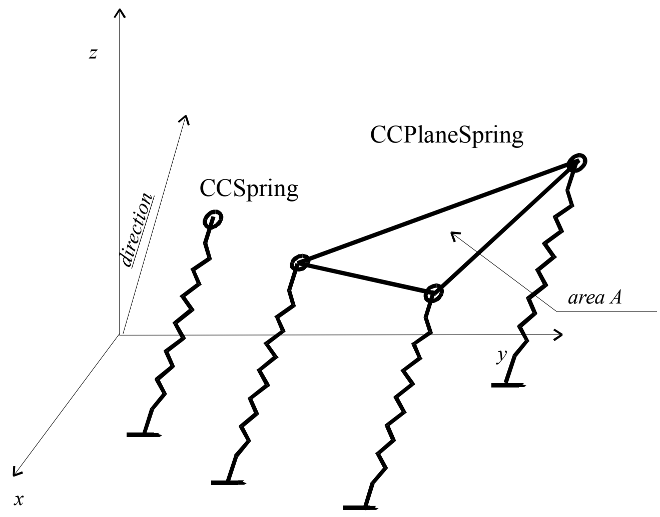
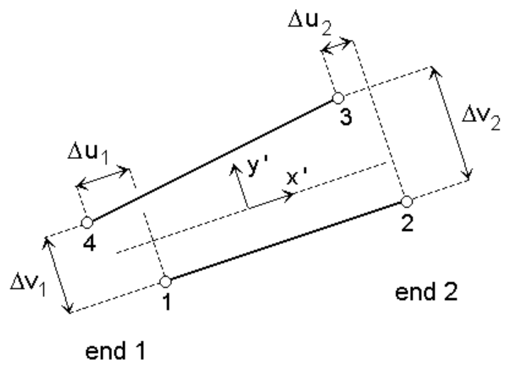
CCIsoWedge<xxxxxx>
.....
CCISoWedge<xxxxxx...x</td>
</tr>
<tr>
<td>Spring</td>
<td>In ATENA-GiD interface, it is possible to model springs in two ways. Either by generating elements along a line or surface and then by assigning them a Spring material property. Alternative approach is by prescribing springs as conditions using the Data | Conditions menu. With the second approach it is easier to define springs that are normal to a curved surface or line.

CCSpring – 2D and 3D element to model spring-like boundary conditions at a point,

CCLineSpring – 2D element to model spring-like boundary conditions along a line



CCPlaneSpring – 3D element to model spring-like boundary conditions along a triangular area.</td>
</tr>
</table>

<table>
<tr><td></td><td>
</td></tr>
<tr><td>Interface</td><td>2D line 4 node interface - CCIsoGap<xxxx>)


2D quadratic 6 node line interface – CCIsoGap<xxxxxx>
3D triangular 6 node interface - CCIsoGap<xxxxxx>
3D triangular 12 node interface – CCIsoGap<xxxxxxxxxxxx>
3D quadrilateral 8 node interface – CCIsoGap<xxxxxxxx>
3D quadrilateral 16 node interface – CCIsoGap<xxxxxxxxxxxxxxxx></td></tr>
</table>

5.8 ATENA Menu

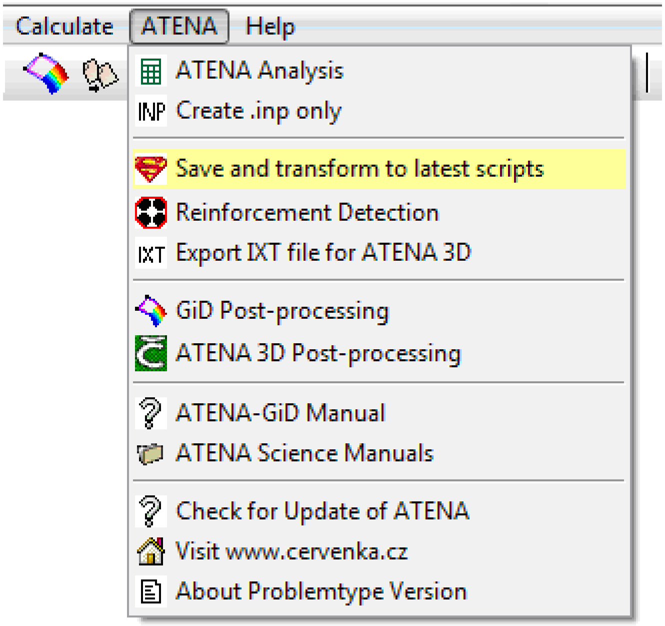


Fig. 5-89 ATENA menu in GiD

ATENA Analysis – Runs analysis
Create .inp only – Creating only .inp file in the GiD model directory
Save and transform to latest scripts – Automatic function for save and transform to latest scripts in your computer
Reinforcement Detection – Automatic function for search lines which look as reinforcements and assign special condition for reinforcements. (Reinforcement Nodes Identification and Reinforcement Elems Identification)
Export IXT file for ATENA 3D – It is also possible to export 3D mesh to an IXT format, which can be imported to ATENA 3D Pre-processor. This tool is described in section 11.1

GiD Post-processing – Toggle to GiD pre- and post-processing
ATENA 3D Post-processing – Run ATENA 3D
ATENA-GiD Manual – Open ATENA-GiD Manual
ATENA Science Manuals – Open directory with ATENA Manuals
Check for Update of ATENA – Online check if some new version of problem type is on the web.
Visit www.cervenka.cz – Go to www.cervenka.cz website
About Problem type Version – View splash screen with problem type version

6 STATIC ANALYSIS

Static analysis is activated in **GiD** by selecting an appropriate problem type **Static** (see the menu items **Data | Problem Type | Atena**). The making of model it's the same like others problem data. It's necessary to assign Conditions [5.2], for each macro element assign material properties [5.3], define the interval data [Fig. 5-75, Fig. 5-78, Fig. 6-1] and problem type properties [Fig. 5-85], meshing model [5.7] and execute the analysis by the clicking on the icon  or by the using of command **Calculate | Calculate**..

The natural frequencies of the structure and the corresponding shapes can be calculated in both dynamic and static analysis. Check the box **Calculate Eigenvalues-Vectors** at the **EigenValue Analysis** tab and the **Eigenvalue Parameters** tab appears, see Fig. 6-1.

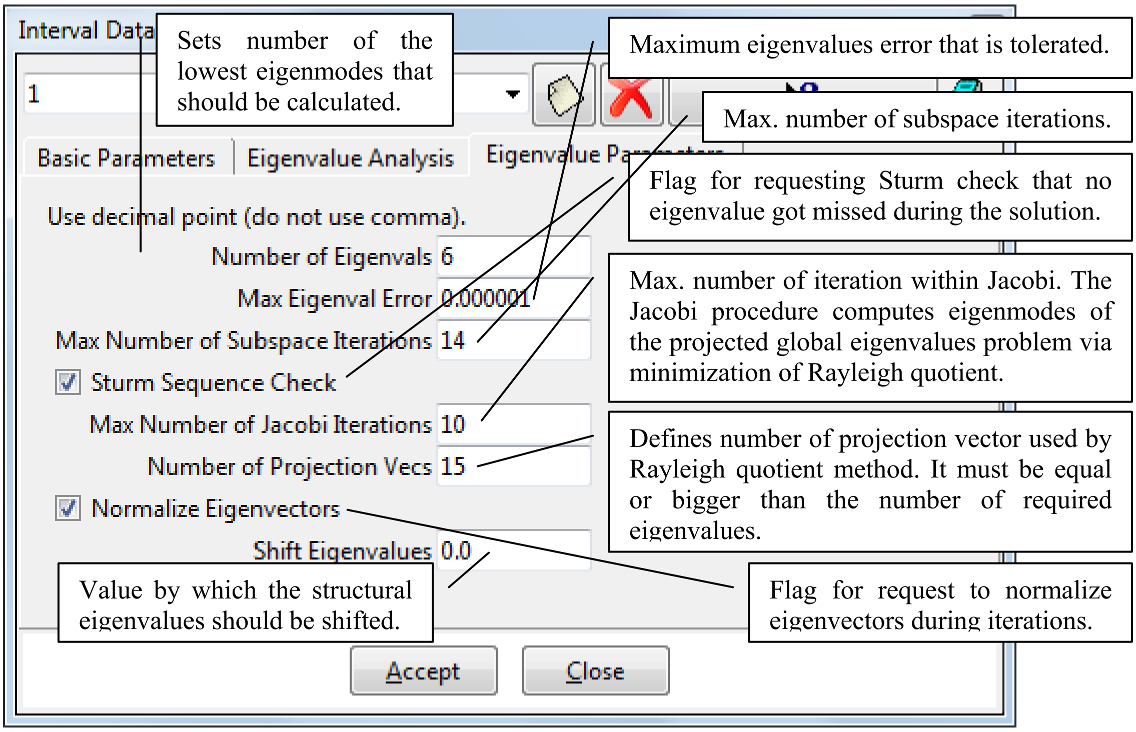


Fig. 6-1: Settings of EigenValue Parameters

Detailed example of static analysis at full length can be found in the ATENA Science example manual [8]. You can also follow the ATENA-GiD Tutorial [6] with detailed instructions to build a simple static model from scratch, run it, and post-process it.

7 CREEP ANALYSIS (AND SHRINKAGE)

This section describes use of **GiD** graphic user interface to carry out creep and shrinkage analysis within **ATENA** software. The theoretical background for such an analysis is given in ATENA Program Documentation, Part 1: Theory [1]. Here we will concentrate only on the explanation of the GUI support implemented in the **GiD** environment. For the exact meaning and deeper description of the individual input parameters the reader is referred to ATENA Program Documentation, Part 6: Input File Format Manual [4] and Part 1: Program Theory [1].

The **ATENA** software supports two kinds of creep and shrinkage analysis. The first kind involves only mechanical analysis of the structure. It is assumed that the structure has everywhere more or less similar humidity and temperature conditions and the same applies for ambient environment. The corresponding problem type for this kind of analysis is **Creep**, and it is accessible via menu item **Data | Problem type | Atena**.

The second kind of creep and shrinkage analysis is aimed for more complex situations, when the structure is subjected to significant moisture and humidity variation in time and space. In this case mechanical creep and shrinkage analysis is preceded by a transport analysis, whose aim is to compute moisture and temperature histories of the structure at each of its material (i.e. integration) point. The corresponding data type for the transport analysis is Transport. At the end of the transport analysis the calculated histories are exported into data files, from where they are later imported into the mechanical analysis. The transport analysis is described in the next section of this document.

Generally speaking, the procedure of preparing input data for creep and shrinkage analysis and its execution within **ATENA-GiD** environment is very similar to that for usual static analysis neglecting the effect of time. This process is described in the previous section of this document. Hence, in this section we will concentrate on description of the additional input commands that are specific for creep and shrinkage and we will not repeat, what is already written in the previous sections of this document (for static analysis without creep).

Clearly, the main difference between usual static and creep analysis is that the latter one carries out analysis, (integration) of structural response in time. Hence, all definitions of the analysis's steps, boundary conditions, loads etc. need additional information about time conditions. Time factor appears also in the constitutive equations, (i.e. material models). This is done by implementing models for prediction of creep and shrinkage behaviour of concrete. Such models are published in codes of practice for civil engineers and, of course, a few reputable models exist in scientific literature, too. For more information about implemented models please, have a look at the theoretical manual for ATENA [1].

There is one more thing worth mentioning here. In order to compute the structural response at a specific time, the whole history of the structure has to be analysed. It involves time integration of structural behaviour, which is done in numerical manner. Practically it means that although the structure is typically loaded only in a few steps, in order to ensure sufficient accuracy of the analysis, each step is further subdivided by the **ATENA** kernel into several sub-steps. This process of step splitting is generated automatically bearing in mind exponential character of concrete creep and shrinkage behaviour and user need not to worry about any related details. This means that in addition to the load steps, which are predefined by the user, additional sub-steps are introduced automatically during the analysis in order to accurately consider the effect of the loading history. This sub-stepping process can be adjusted through a proper selection of the parameter **Sample Times Per Decade**, see the input dialog below. It can be reached via the menu item **Data | Problem Data | Problem Data** or by pressing the icon . The parameters for the retardation time generations are specified in this dialog. The retardation times (see [1]) are also generated automatically. It is only important to set them such that time in the parameter **Retardation time for execution times** precedes the first load time of the structure and the value of the parameter **Retardation time for execution times** exceeds the last time of our interest. In addition, the number of **Retardation time per decade** should somehow correlate with the number of sample times per decade. Otherwise we would violate balance in accuracy of individual approximations involved in the creep and shrinkage analysis. The remaining data sheets of this dialog are the same as for usual static analysis.

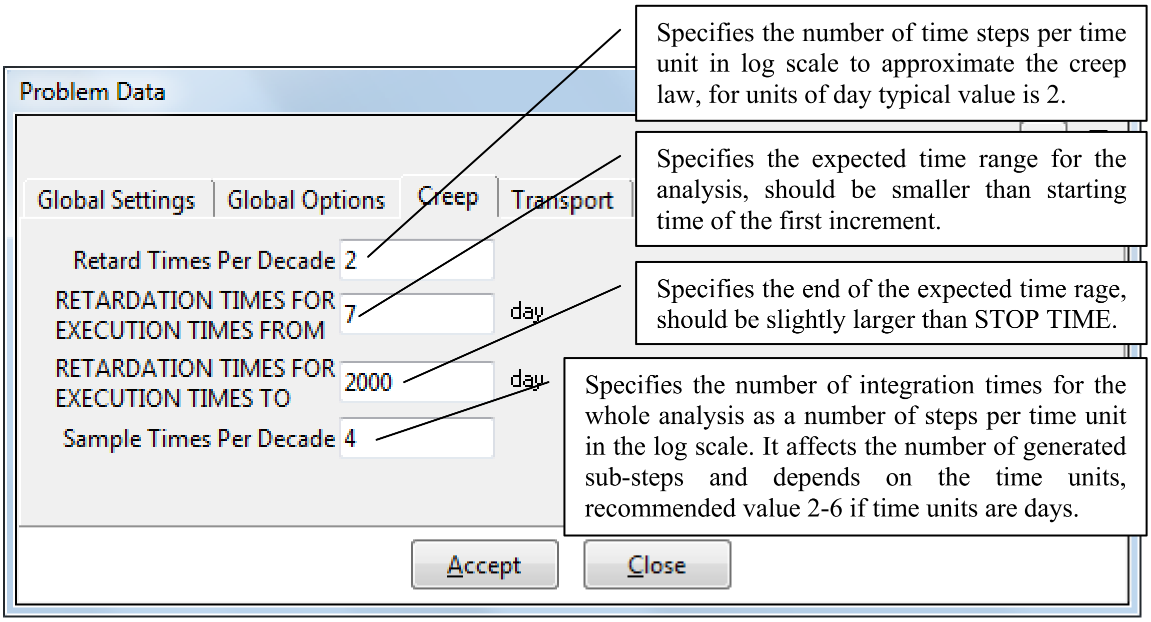
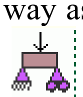


Fig. 7-1 Problem Data dialog.

7.1 Boundary Conditions and Load Cases Related Input

The essential part of any FEM analysis is to set correct boundary conditions for the analysed problem. The related input information is specified in creep and shrinkage analysis in the same way as it is in a static analysis without creep, see the dialog called by pressing the icon  from the **GiD** toolbar. However, one must be aware of the fact that the execution step, for which the user defines boundary conditions, is (automatically by **ATENA** kernel) subdivided into several sub-steps. That's why creep and shrinkage analysis must distinguish between boundary conditions that are to be applied to all internal sub-steps and boundary conditions applicable only for the first sub-step. Typically support conditions should be applied in all sub-steps, but the loading increment should be applied only in the first step. In **GiD** dialogs for the boundary conditions the two types of conditions are distinguished by the check box **Apply in Sub-increment**. If it is checked, the specified boundary conditions are assumed to be applied in all sub-increments i.e. sub-steps. In case a loading should be applied only in the first sub-step, this box should not be selected.

There are several levels, which affect the loading history definition.

Intervals – this is the main level to define the loading history for the **ATENA** analysis. Each interval consists of a set of conditions, which are defined according to the Section 5.2.

Load steps – this is the level, which is used in **ATENA**. Each interval can include multiple load steps, with the same boundary conditions.

Sub-steps – these are internal load steps, which are automatically created by **ATENA** during the creep analysis in order to properly integrate the structural time response. The number of these sub-steps is affected by the choice of the sample times per decade (see Fig. 7-1).

7.2 Specific Creep Boundary Conditions

All boundary conditions are the same as conditions for static

7.3 Material Input Data

Each creep and shrinkage material consists of two parts: a creep prediction model, (such as Bazant's B3 model) and an ordinary (short term) material model for concrete, (such CC3DNonLinCementitious2). The short term model is also called the "base" material model.

The input data in **GiD** reflect this structure. The user has to specify two sets of parameters, one for the creep prediction model, one for the base material model and each such a set is assigned a dedicated date sheet. The actual data input dialog is invoked by pressing the icon (or via menu **Data | Materials | Creep**), and it is shown in Fig. 7-2

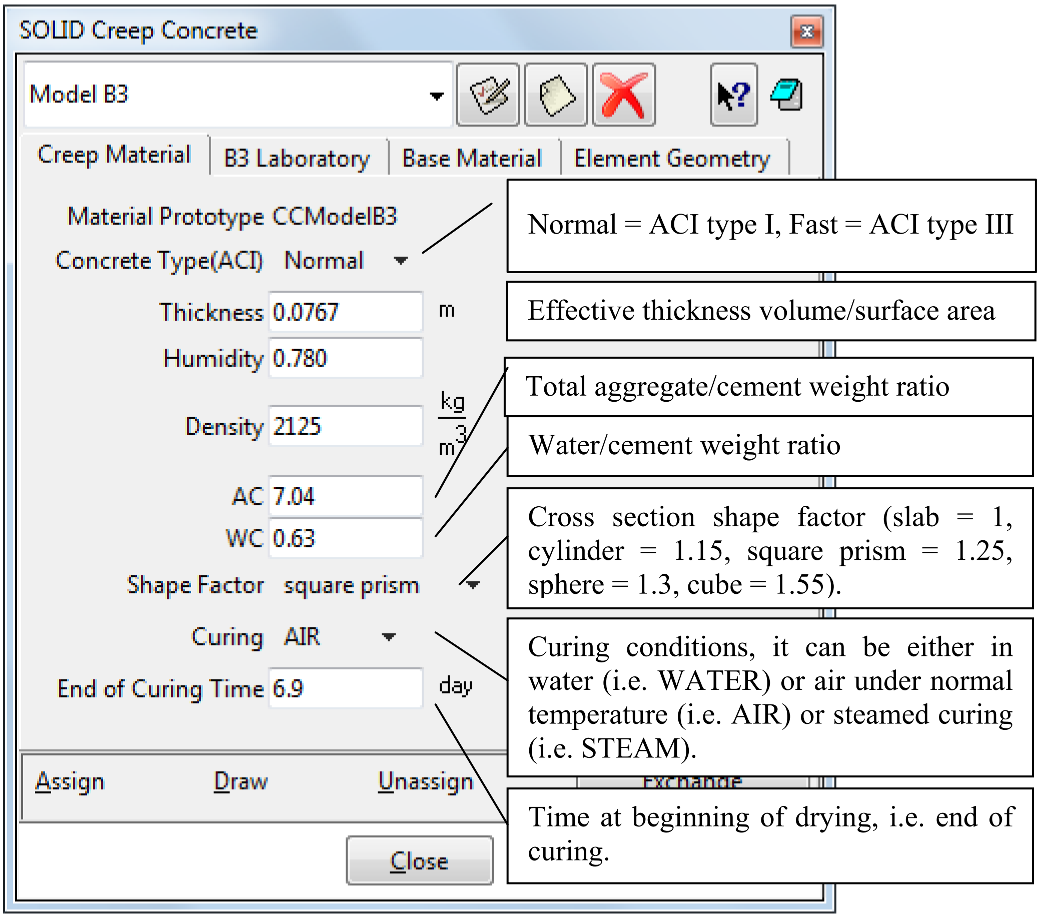
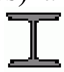


Fig. 7-2 Material input dialog

The combo box at the top of the dialog specifies a type of material model to be used and it follows a number of related input parameters. It is beyond the scope of this document to provide their description. For more information please read the ATENA Theory [1] and input data documentation [4] and/or the related literature.

The above applies for concrete structures (or for concrete structures with discrete reinforcement only). The situation is a bit more complicated in the case of concrete structures with smeared reinforcement, when a material definition (for creep and shrinkage analysis) should comprise three material models: a creep prediction model, a short term model concrete and short term model for smeared reinforcement. This type of input data in **GiD** is still in stage of development, and thus not all combinations of the material candidates (suitable for one of the three material types) are supported. The corresponding input data dialog is invoked by pressing the icon , and it pulls out the following dialog sheets:

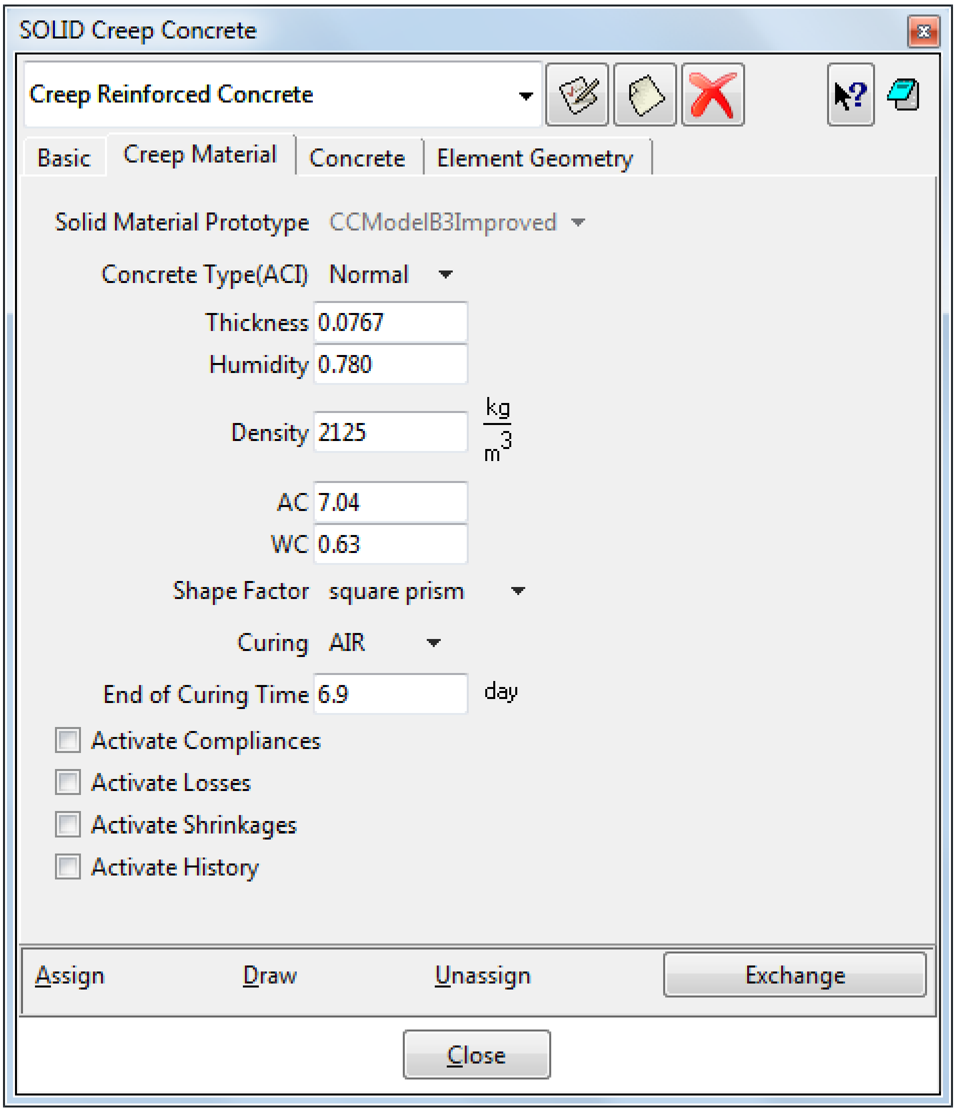
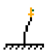
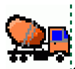


Fig. 7-3 Reinforced concrete material with smeared reinforcement

The dialog has several pages, each corresponding to a particular type of data. For example the sheet **Creep Material** serves for input data for creep prediction model (and it resembles the dialog called by pressing  . The sheet **Concrete Material** includes input data for short-term model for concrete, (similar to that invoked by , etc.)) The individual smeared reinforcement components will appear under the label **Concrete**.

Although there may be a few more differences between analyses with and without creep (and shrinkage), it is believed that most important ones have already been covered in this section. The rest should be self-explanatory and possible to being used without any further explanation.

8 TRANSPORT ANALYSIS (MOISTURE AND HEAT)

Although heat and moisture analysis can be executed as a standalone analysis, in the **ATENA-GiD** framework it is usually the first part of a static or creep/shrinkage analysis. Its goal is to calculate moisture and temperature conditions in the structure. As a result, we get histories of temperature and moisture variation at each material point of the structure, and these data are later used by a stress analysis or creep material model to better predict stress-strain relationships with the effects of temperature, creep and shrinkage.

Main use of moisture and heat transport analysis is to calculate temperature increments inside a structure. These increments are later used in the calculation of element thermal expansion and associated initial strain load in conventional static analysis. In the stress analysis by **ATENA,** it is also possible to consider the temperature dependence of material properties.

Moisture and heat transport analysis is activated in **GiD** by selecting an appropriate problem type Transport (see the menu items **Data | Problem Type | Atena**).

8.1 Material Input Data

Currently, only one material model is supported, CCTransportMaterial. Material Bazant_Xi_1994 (see section 8.1.2) is not supported since version 5.0.0 any more. The corresponding input data dialog appears by pressing the icon :

8.1.1 Material CCTransport (CERHYD)

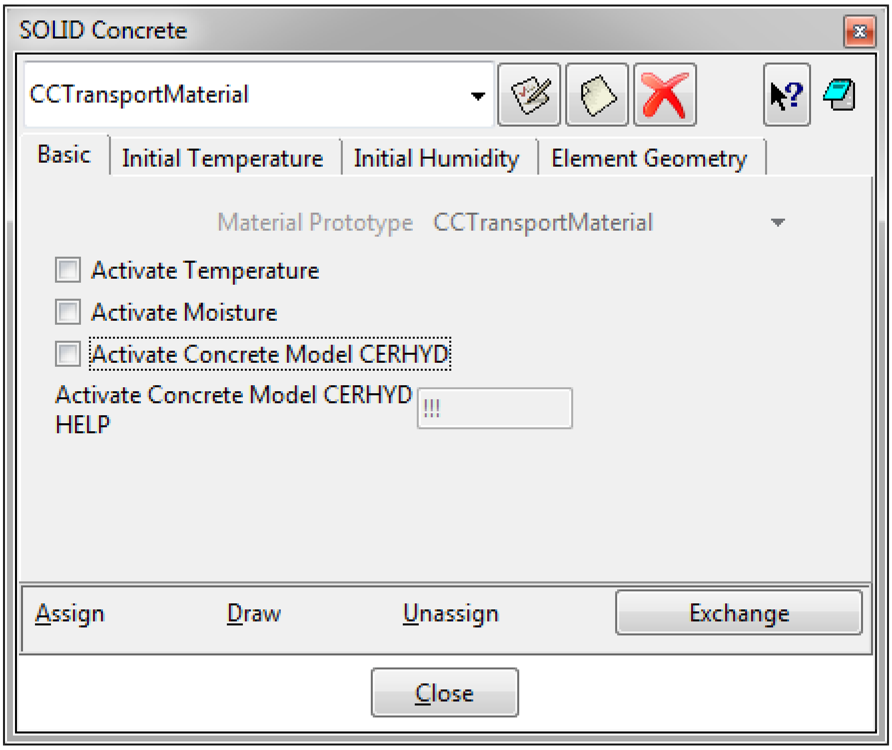


Fig. 8-1 Heat and moisture transport material model dialog

The model name is CCTransportMaterial. The Material Prototype is CCTransportMaterial or CCTransportMaterialLevel7. It depends on the check box **Activate Concrete Model CERHYD**. CCTransportMaterial is a simple constitutive law that allows users to enter laboratorial measured moisture and heat characteristics.

CCTransportMaterialLevel7 is an extension of the above CCMaterialTransport material in the way it automatically computes moisture and temperature capacity and conductivity/diffusivity incl. "sink" terms regarding hydration, (i.e. rate of hydration heat and moisture consumption during concrete hydration).

For more details about these materials see Theory manual [1], section Transport Analysis.

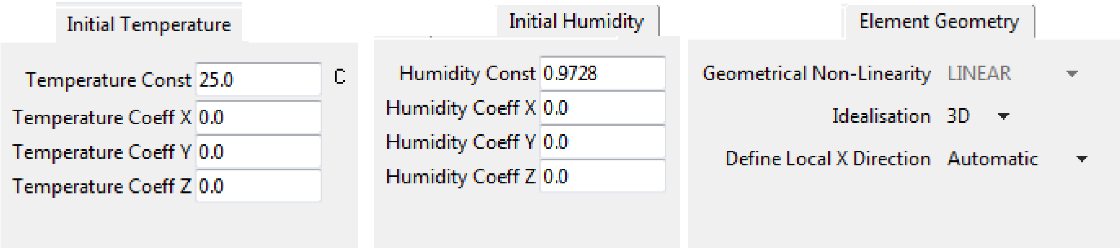


Fig. 8-2 Transport Material - Initial Temperature and Humidity Dialog

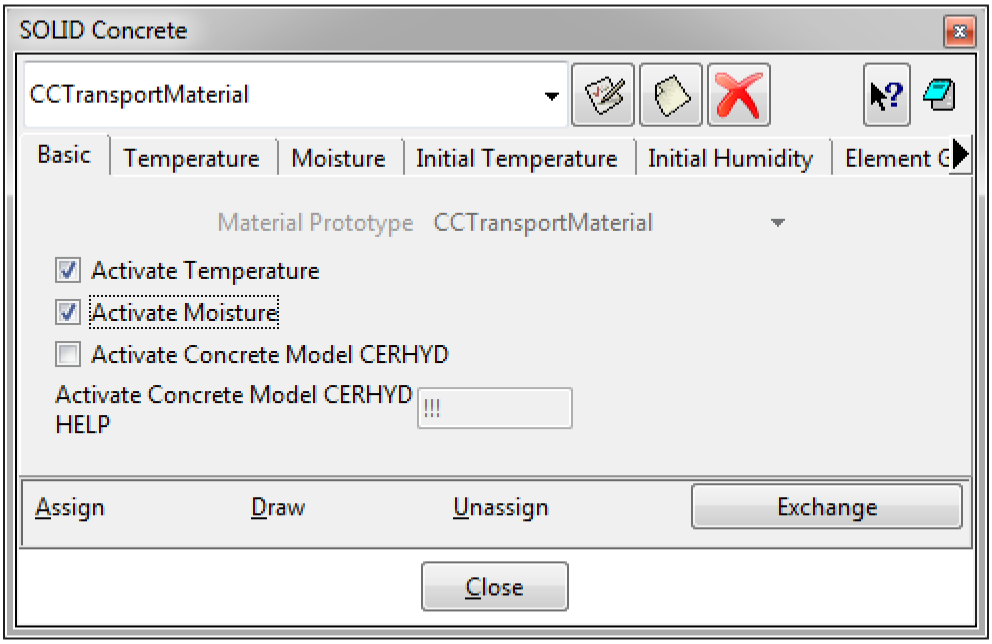


Fig. 8-3 Transport Material – Activate Options

For detailed information about all these parameters please see the ATENA Theoretical manual (7.3 Material constitutive model) [1].

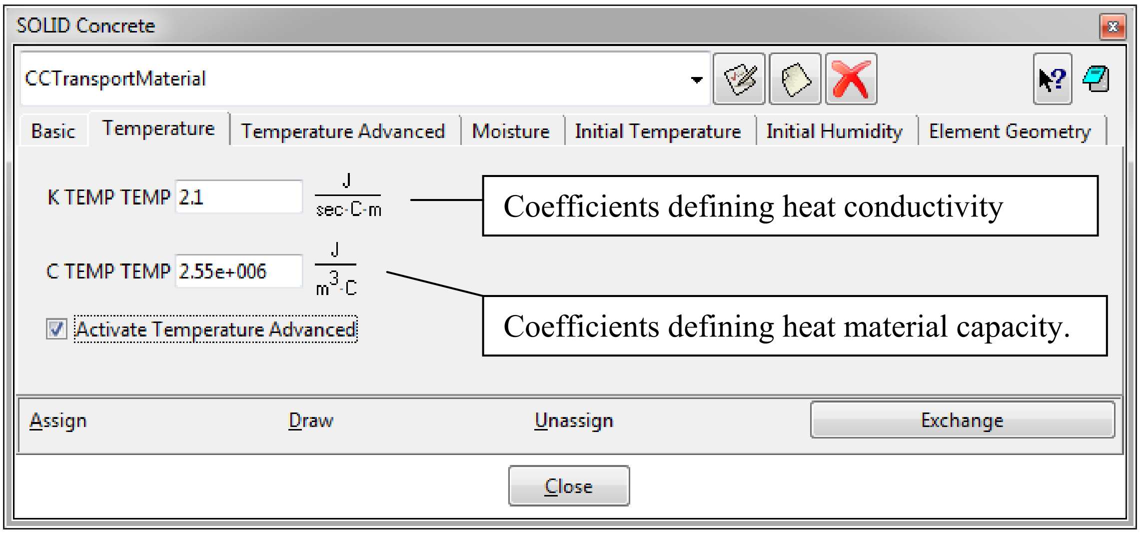


Fig. 8-4 Transport Material – Temperature

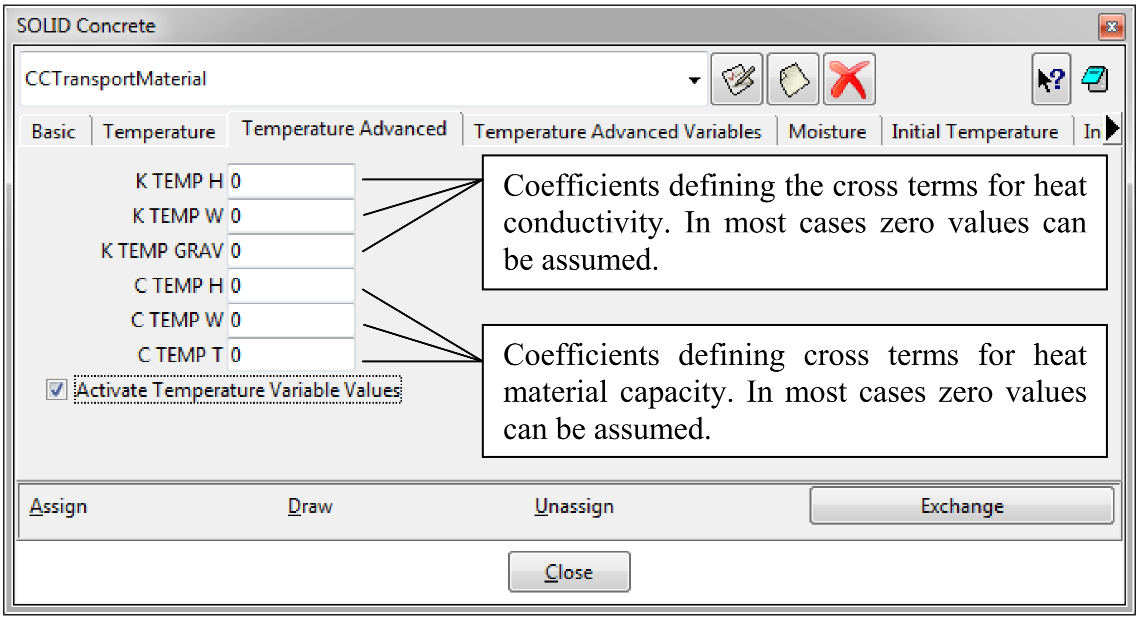


Fig. 8-5 Transport Material – Temperature Advanced options

All the above heat flux and capacity coefficients are constant with respect to state variables, i.e. humidity and temperature, but if needed a nonlinear behavior can be assumed by defining a multiplication function for each of the above parameters (see Fig. 8-6).

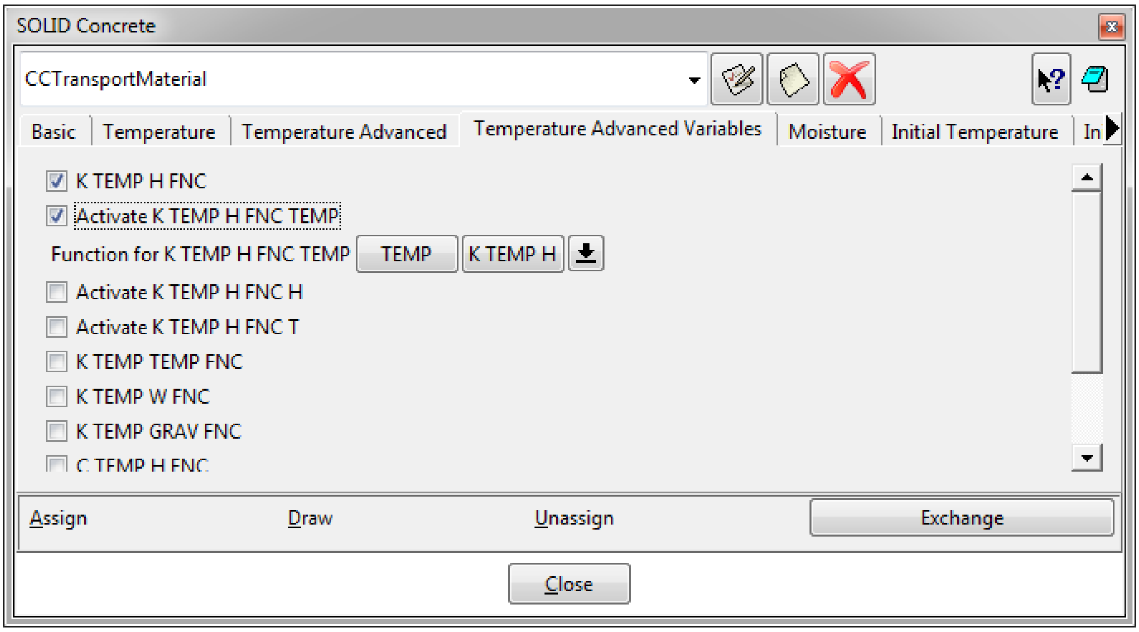


Fig. 8-6 Transport Material – Advanced variables, Activation of Nonlinear Functions

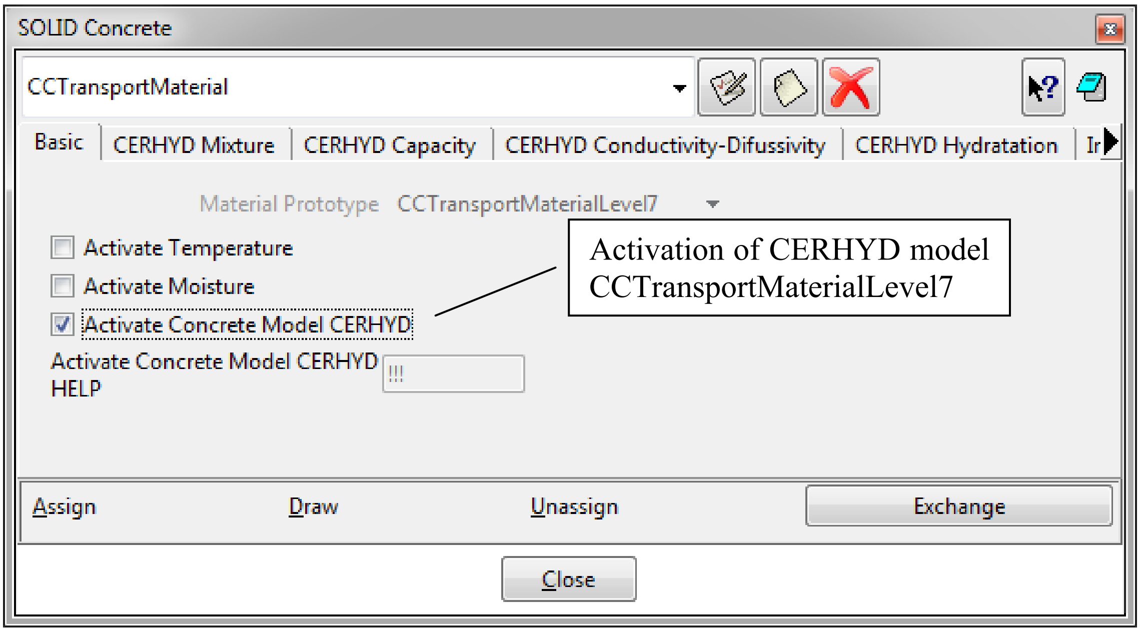


Fig. 8-7 Transport Material – CERHYD Model

Concrete model CERHYD calculates transport parameters (K_TEMP_TEMP, C_TEMP_TEMP, D_H_H and C_H_H) on the basis of concrete composition and properties of individual components. The model also includes calculation of concrete hydration based on the affinity hydration model. For more detail, the Theory Manual should be consulted [1]. If the temperature and moisture checkboxes are also activated, the calculated parameters of the concrete model CERHYD are added to the values provided in the temperature and moisture dialogs.

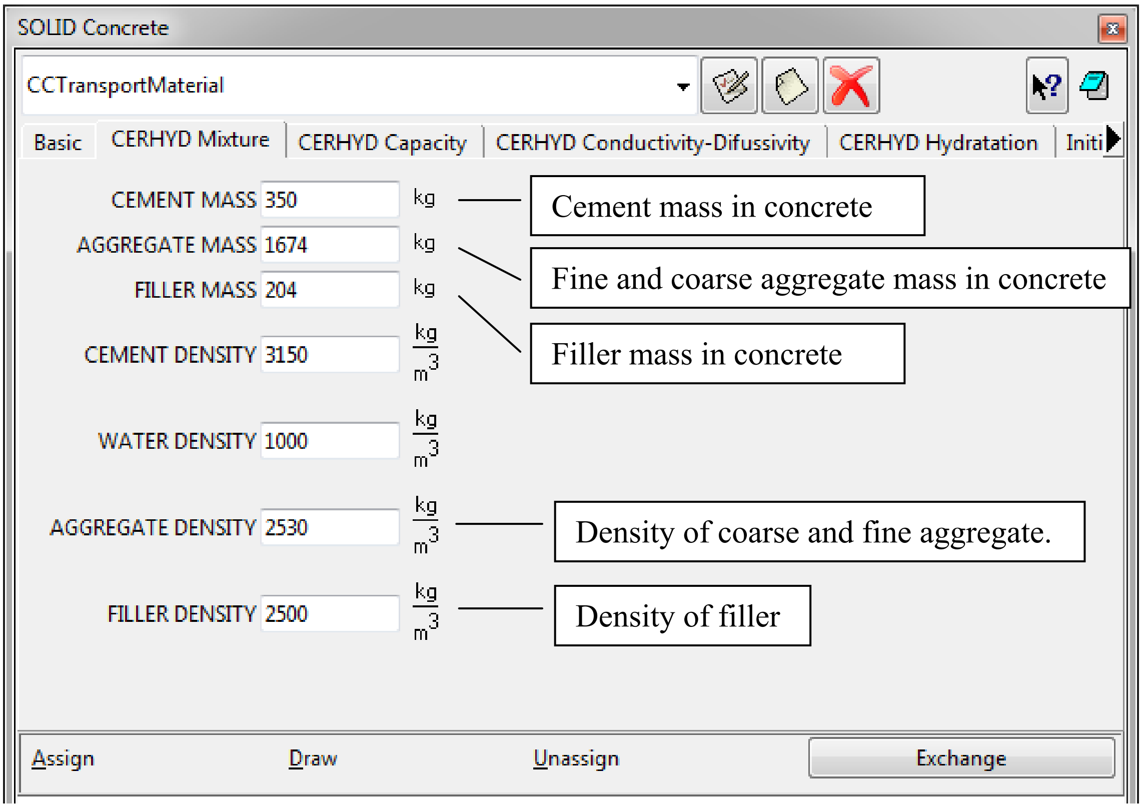


Fig. 8-8 Transport Material – CERHYD Mixture

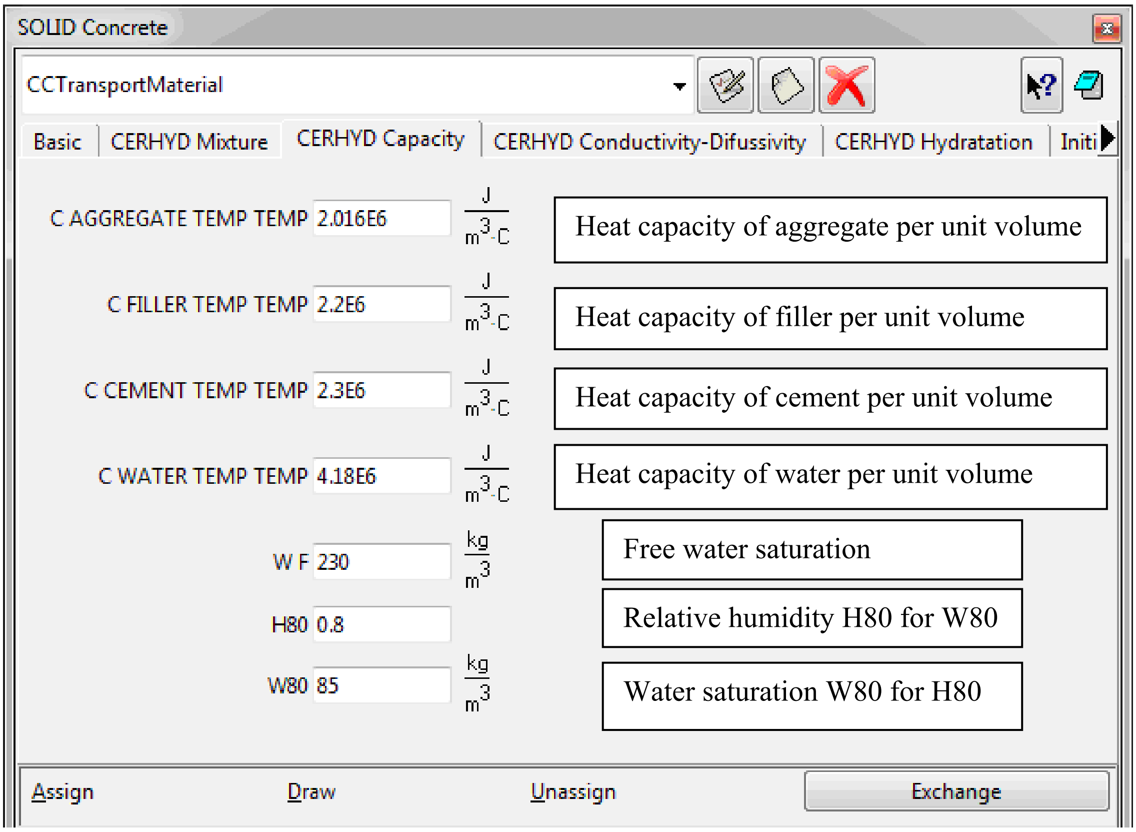


Fig. 8-9 Transport Material – CERHYD Capacity

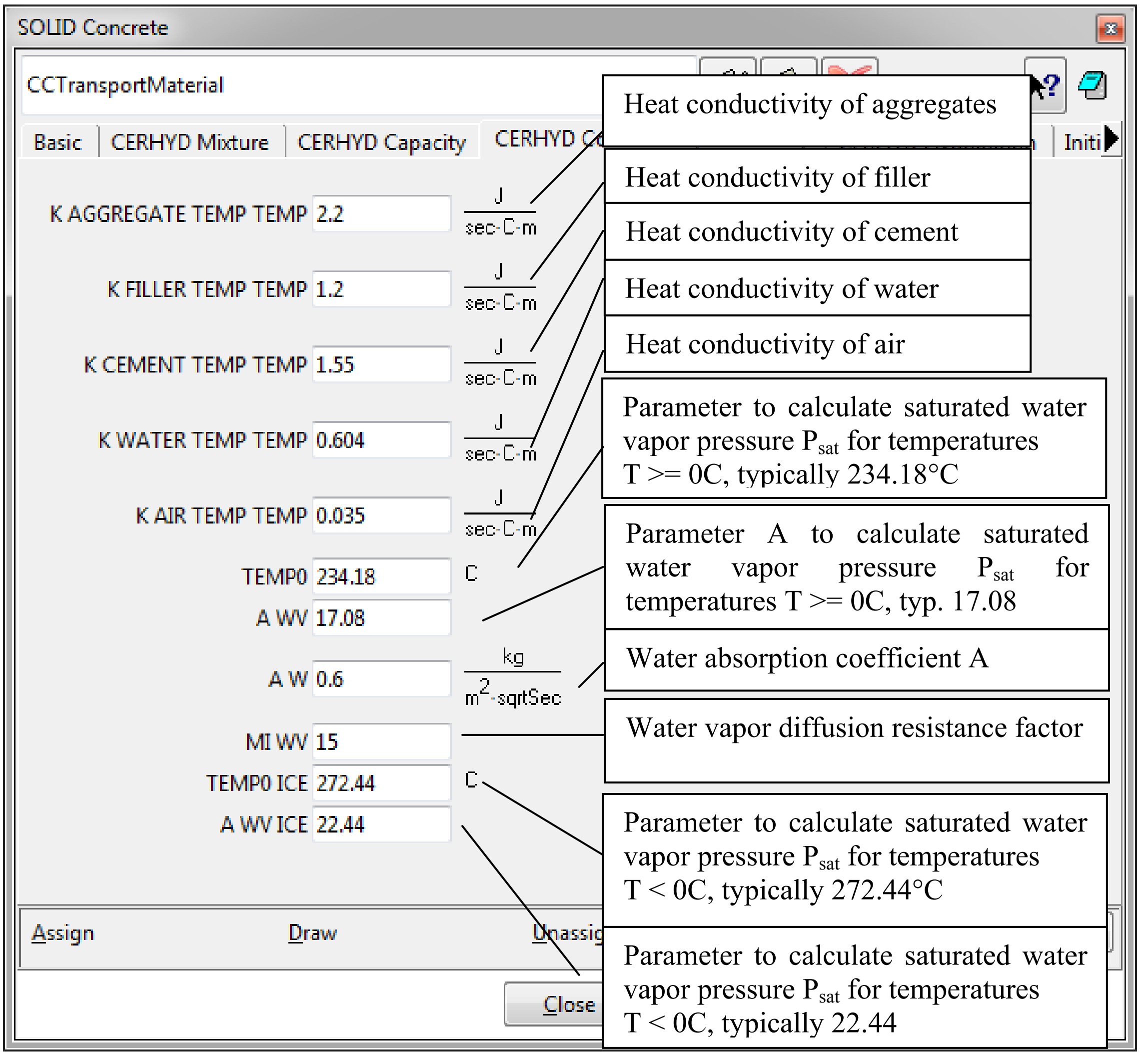


Fig. 8-10 Transport Material – CERHYD – Conductivity-Diffusivity

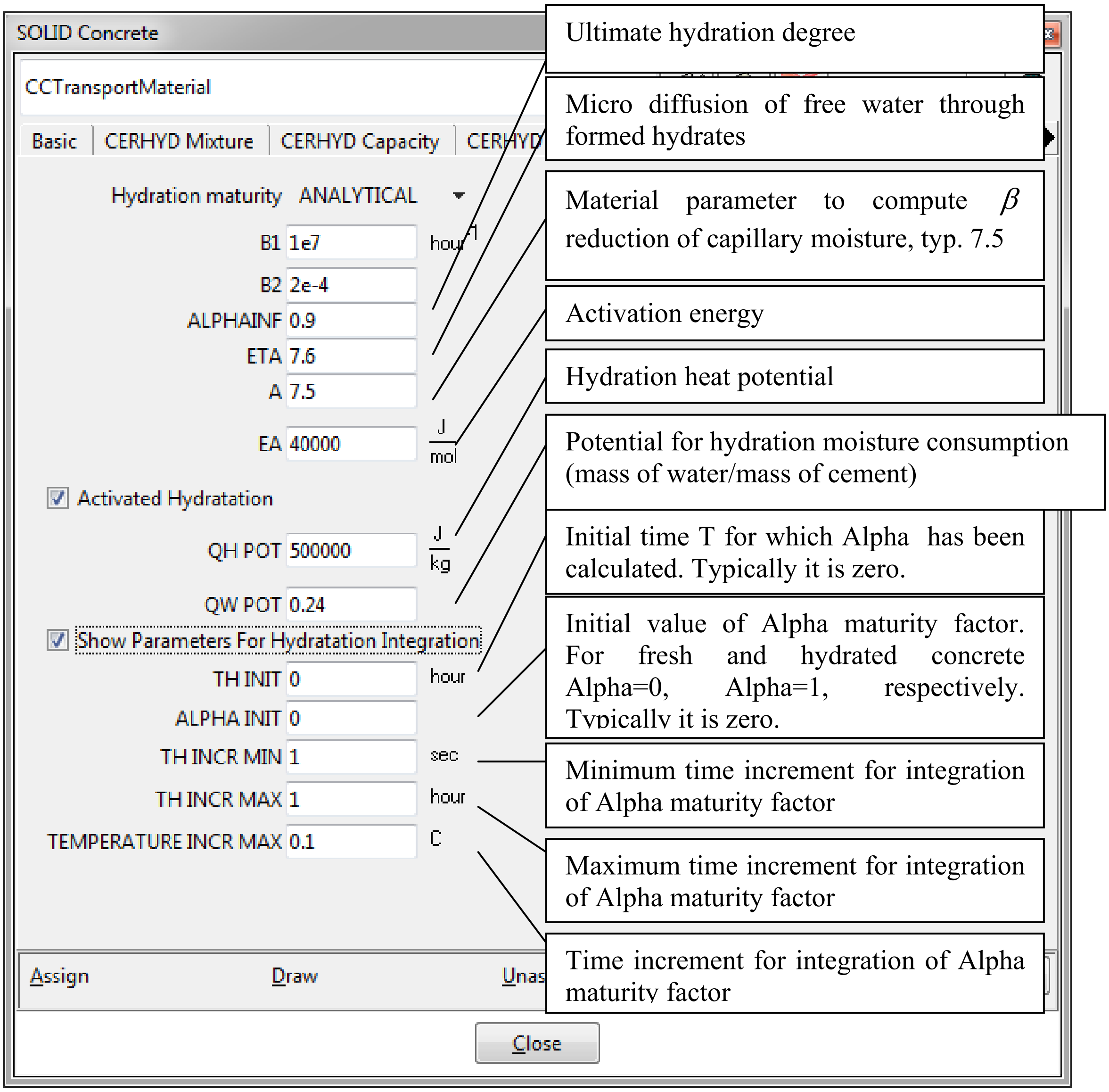


Fig. 8-11 Transport Material – CERHYD – Hydration

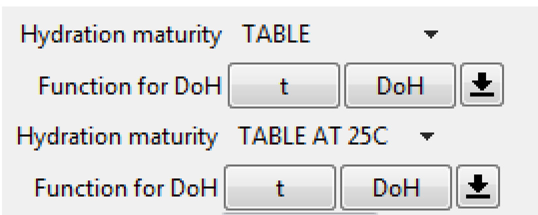


Fig. 8-12 Transport Material – CERHYD – Hydration maturity options

8.1.2 *Material Bazant_Xi_1994 (only included for backward compatibility of old models)*

Its moisture transport part is based on Bazant-Xi model (see the manual for ATENA Theory [1]) that has been developed for the modelling mortar behaviour. It accounts for water and cement paste only and hence, in case of concrete mixture it neglects the presence of aggregate. Consequently, the model can only be used, when relatively impermeable aggregate (with low absorption) is used, such as gravel etc. On the other hand, the model accounts for heat generated due to the process of hydration. The heat transport related part of the model employs linear material law.

The input dialog from Fig. 8-13 has several data sheets. The first one refers to actual material parameters, whilst the remaining sheets are used to define initial material conditions and their variation in space. Taking example of data page for humidity, it enlists parameters:

Humidity CONST ($= h_{const}$), Humidity COEFFX ($= h_x$),

Humidity COEFFY ($= h_y$), Humidity COEFFZ ($= h_z$),

The actual initial humidity in a material point is then computed as $h = h_x\, x + h_y\, y + h_z\, z + h_{const}$, where $[x, y, z]^T$ is vector of coordinates of the material point.

The same approach is used for setting initial conditions for initial temperature and moisture. Note, that moisture and humidity conditions are mutually dependent. Hence only one of these needs to be specified; the others are calculated automatically.

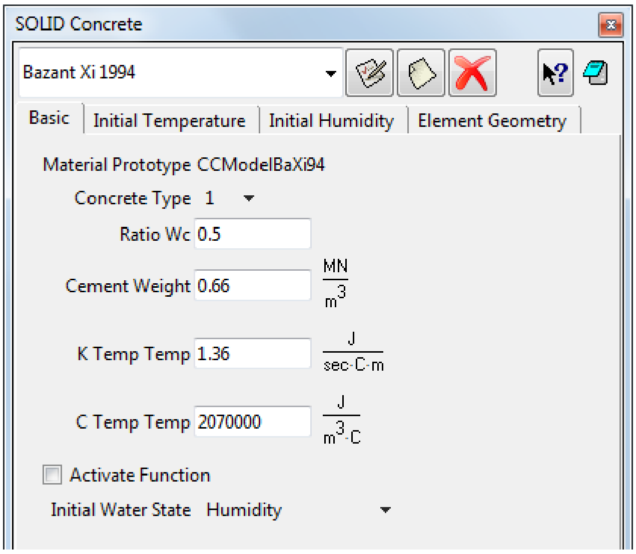


Fig. 8-13 Bazant_Xi_1994 material model dialog

8.2 Other Settings Related to Transport Analysis

Another data sheet, which is specific to the transport analysis, is described below:

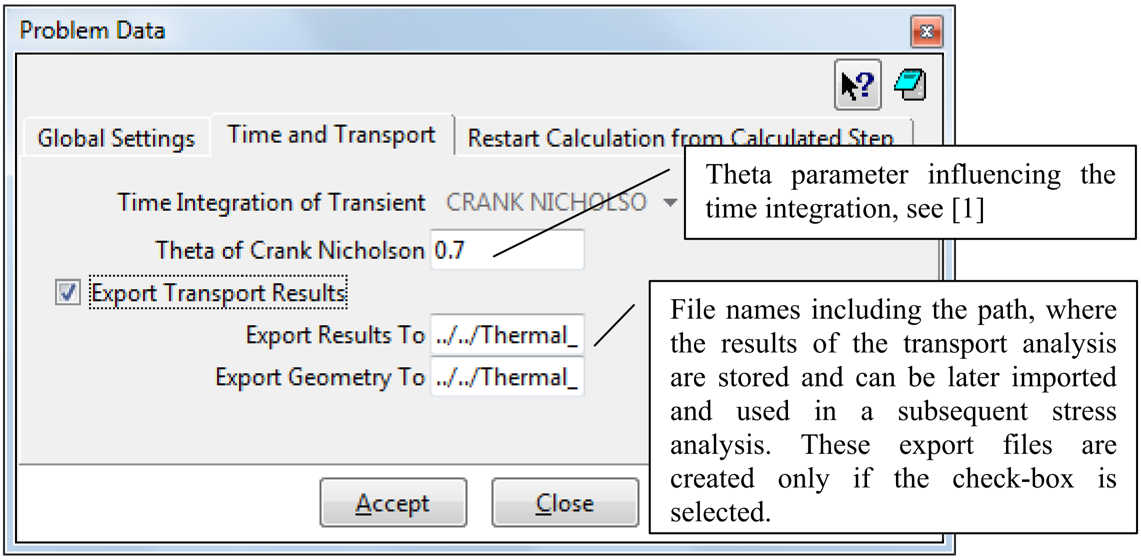


Fig. 8-14 Time and transport data sheet

This sheet is invoked by pressing the icon . In addition to other parameters (used for temporal integration) it comprises names of files, where the results of this analysis should be exported. (Note that Export Transport Result checkbox must be checked). The 1st of them contains actual humidity and temperature histories of the structure and the 2nd file keeps information about geometry of the model. The exported data are compatible with import data format of creep and shrinkage analysis, (or by element temperature load for static analysis without creep). Hence, it is very easy to transfer the histories between this analysis and any other analysis that can make use of it. This means that it is not necessary to use the same model or finite element mesh in the transport and stress analyses. During the import, the program **ATENA** automatically determines the closes nodes and makes the necessary interpolation.

The dialog in Fig. 8-15 (available by pressing ) is used to define one or multiple execution type steps. Meaning of the parameters is self-explanatory and illustrated in Fig. 8-16, but it should be noted that (unlike in creep and shrinkage analysis described in the previous section of this document) heat and transport analysis does not generate any internal sub-steps. All the steps have to be defined manually using the dialog below.

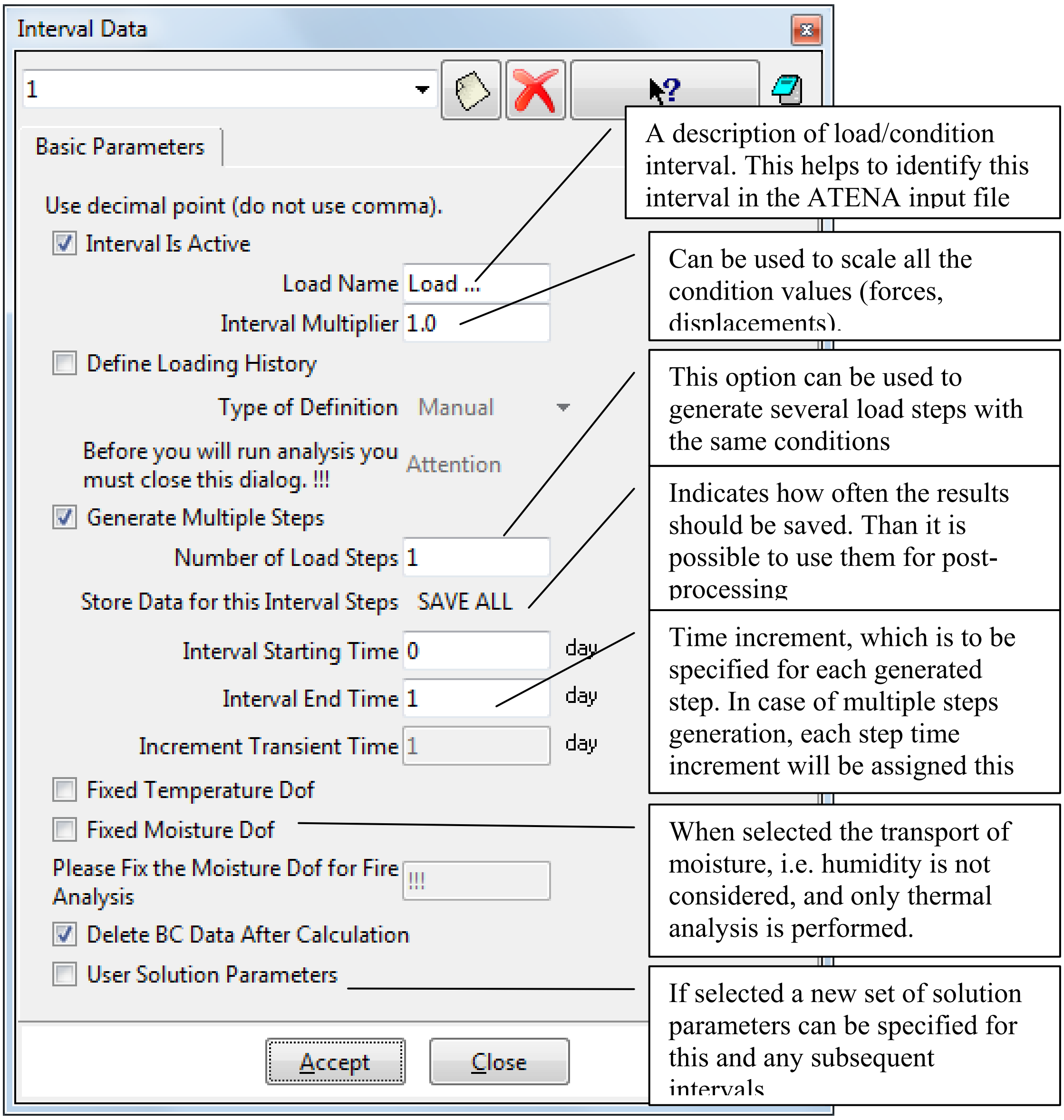


Fig. 8-15 Step data dialog

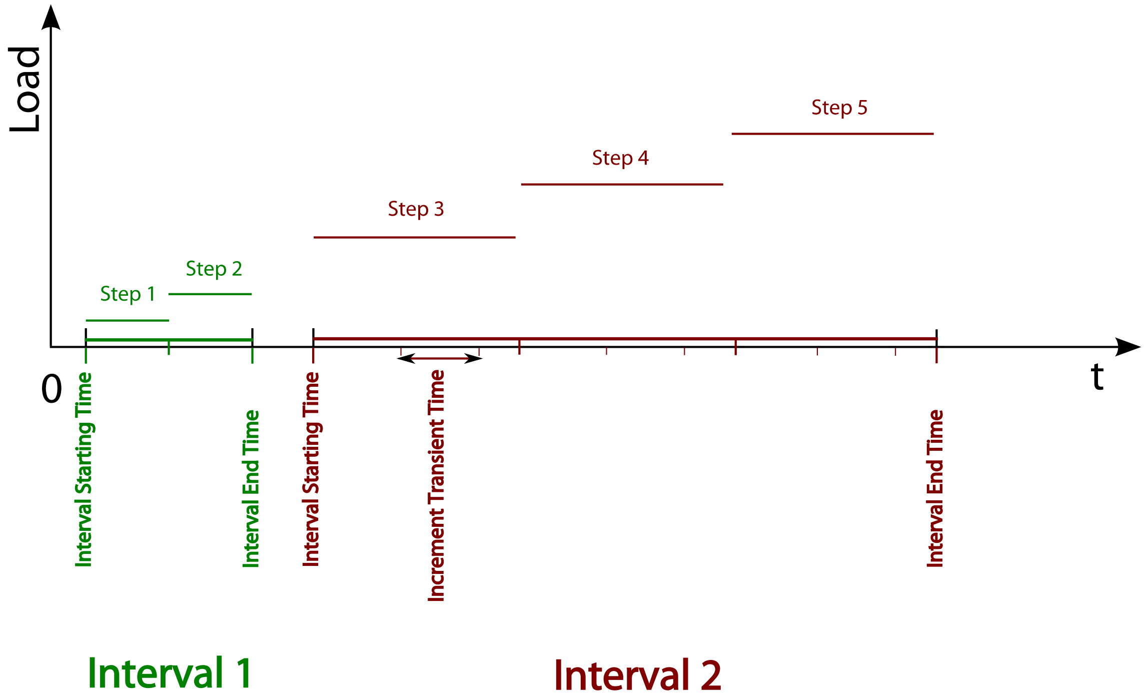


Fig. 8-16 Interval Data - time values

The remaining input data and corresponding data dialogs are similar to their form in other types of **ATENA-GiD** analysis. They were already described earlier in this document (see Section 5.4).

8.3 Specific Transport Boundary Conditions

Dirichlet temperature – Similar to the simple thermal load in static analysis (described in section 5.2). Defines a constant temperature increment for an entity.

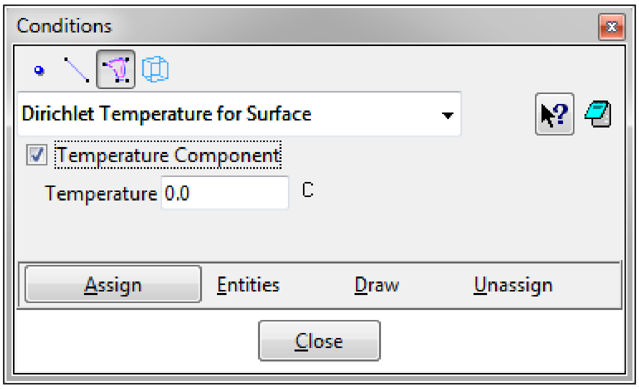


Fig. 8-17: Dirichlet temperature for …

Dirichlet humidity – Defines a constant moisture increment for an entity.

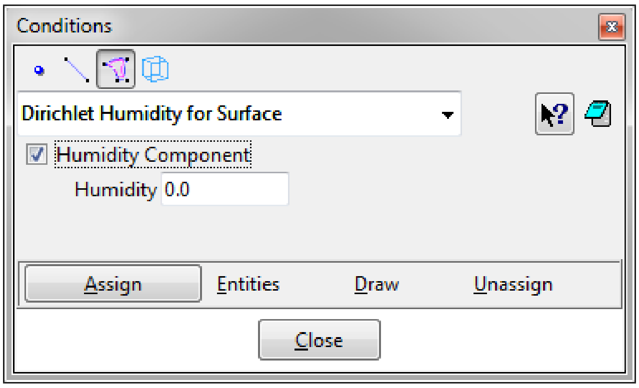


Fig. 8-18: Dirichlet humidity for …

Neumann temperature – The simplest way to prescribe a thermal flux.

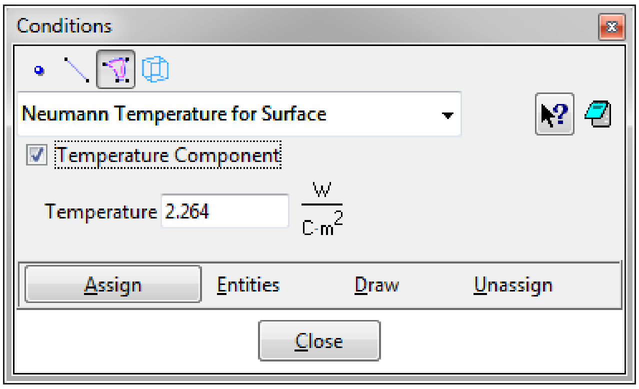


Fig. 8-19: Neumann temperature for …

Neumann humidity – The simplest way to prescribe a moisture flux.

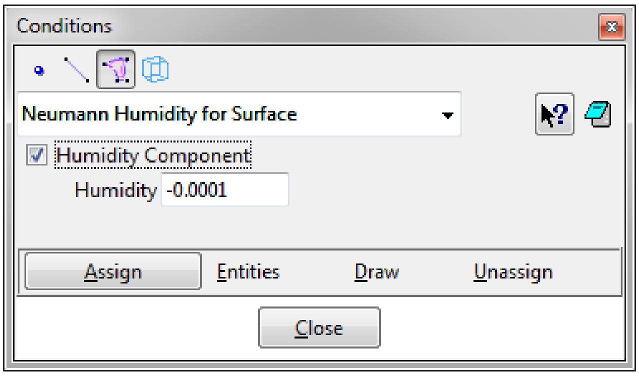


Fig. 8-20: Neumann humidity for …

Moisture Temperature boundary – A combination of heat and moisture transfer by convection, radiation and evaporation. The heat and moisture fluxes from the individual contributions are added together.

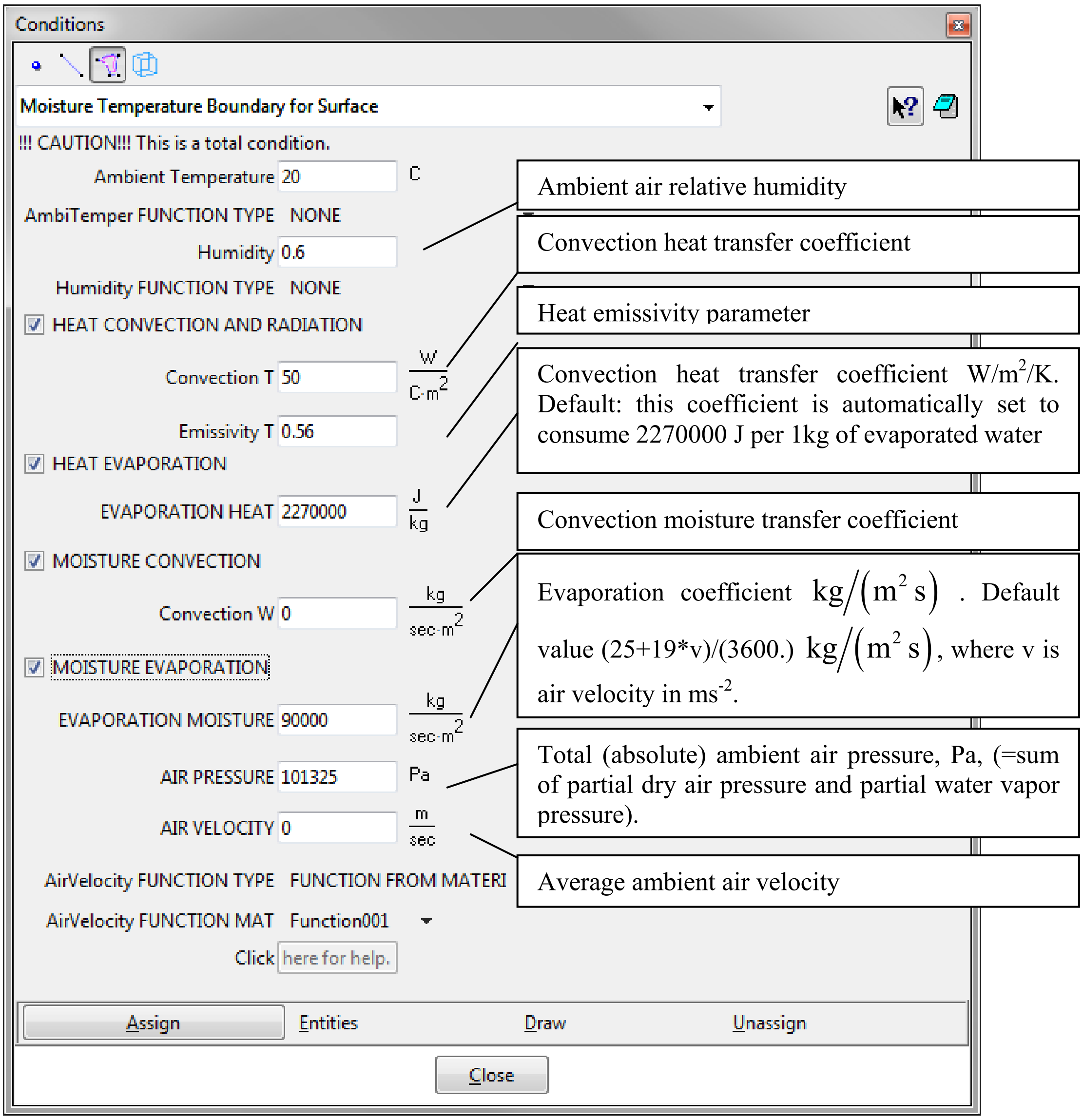


Fig. 8-21: **Moisture Temperature boundary** for …

Fire boundary – A combination of heat transfer by convection and radiation. Originally developed for modeling fire loads, but can also be used for other purposes, like sun-heated surfaces or air cooling although in this case the previous special condition should be used (see Fig. 8-21) One of the few total boundary conditions in **ATENA** (almost all other conditions act incrementally). This condition is NOT supported for quadratic mesh.

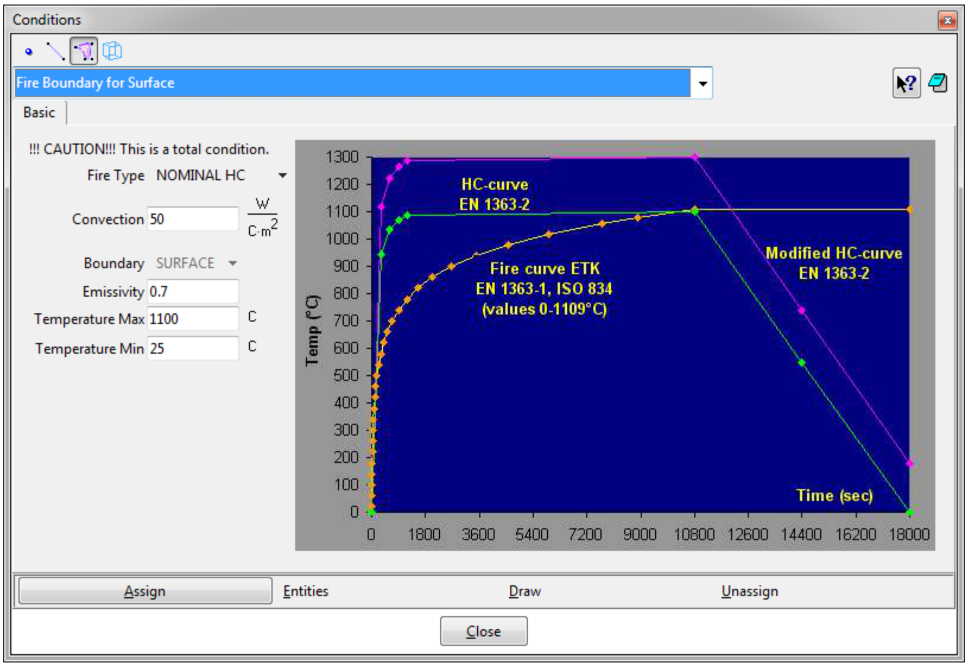


Fig. 8-22: Fire boundary for …

Internal Thermal Source – An internal heat source or sink. Volumetric generation of internal power source of Heat in 3D.

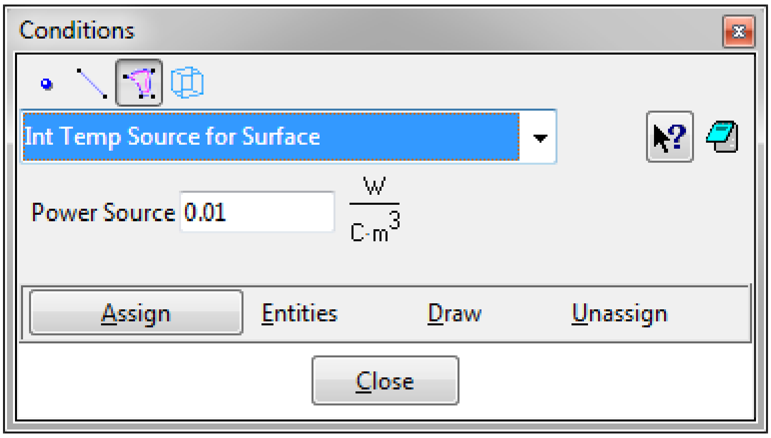


Fig. 8-23: Internal Thermal Source for …

9 DYNAMIC ANALYSIS

Dynamic analysis is activated in **GiD** by selecting an appropriate problem type Dynamic (see the menu items **Data | Problem Type | Atena**). The model preparation is the same as for the other problem types. It is necessary to assign Conditions [5.2], for each macroelement assign material properties [5.3], define the interval data [Fig. 5-75, Fig. 5-78, Fig. 6-1], and problem type properties [Fig. 5-85], meshing model [5.7], and execute the analysis.

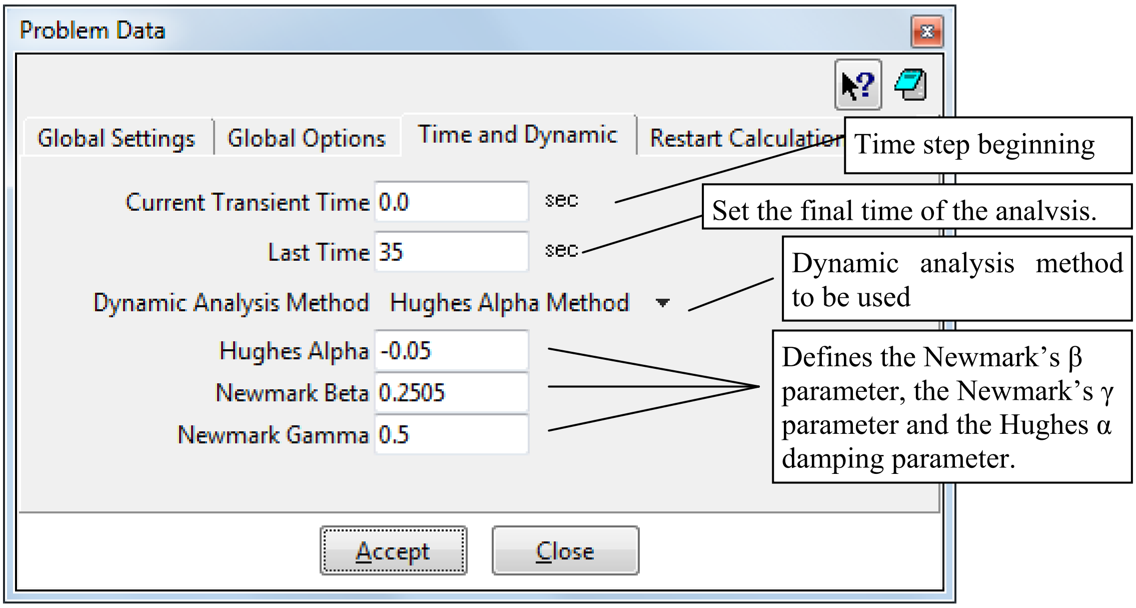


Fig. 9-1: Special dynamic "Problem data" properties

This sheet is invoked by pressing the icon . The next dialog (available by pressing ) is used to define method and parameters for dynamic analysis. The remaining input data and corresponding data dialogs (Fig. 9-2, Fig. 9-3) are similar to their form in other types of **ATENA-GiD** analysis. They were already described earlier in this document (see Sections 5.4 and 5.8).

The natural frequencies of the structure and the corresponding shapes can be calculated in both dynamic and static analysis. Check the box **Calculate Eigenvalues-Vectors** at the **Basic Parameters** tab and the **Eigenvalue Parameters** tab appears. It is identical to static analysis, see Fig. 6-1.

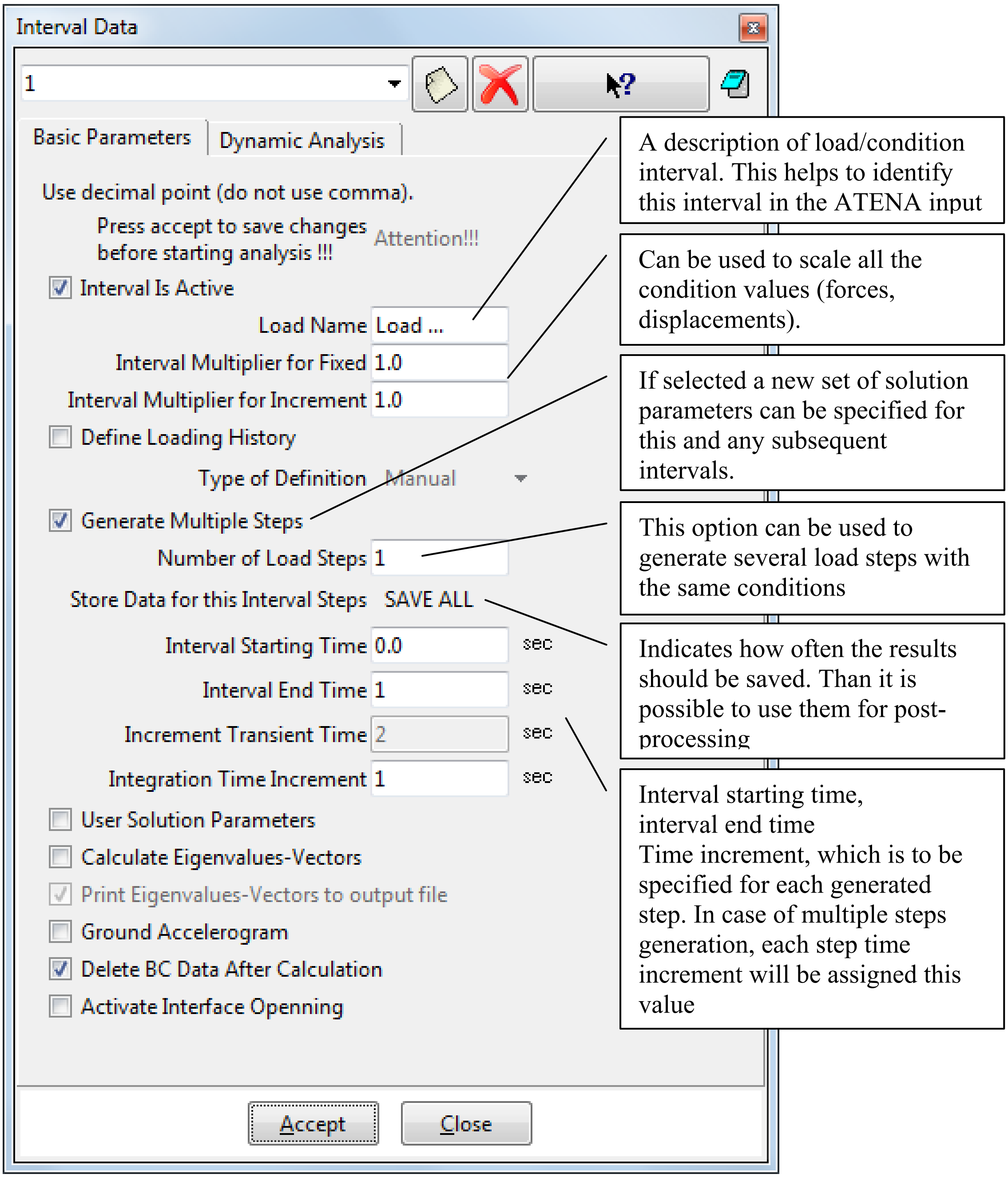


Fig. 9-2: Special dynamic "Interval data" properties

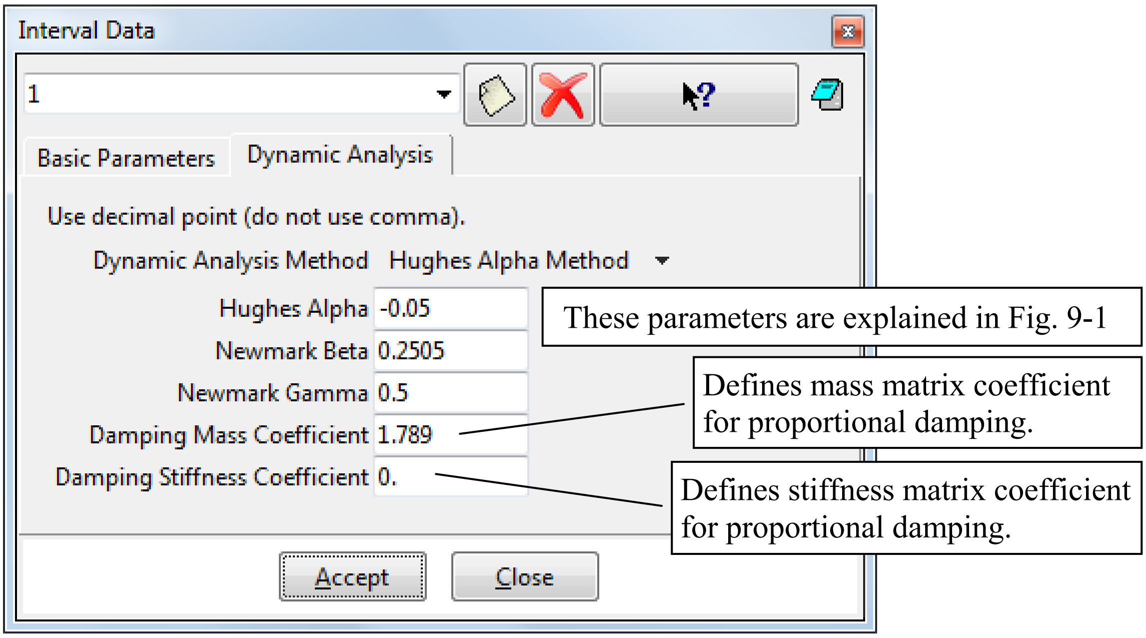


Fig. 9-3: Special dynamic "Interval data" properties

9.1 Specific Dynamic Boundary Conditions

Lumped mass for point – Inertial mass concentrated in a single point.

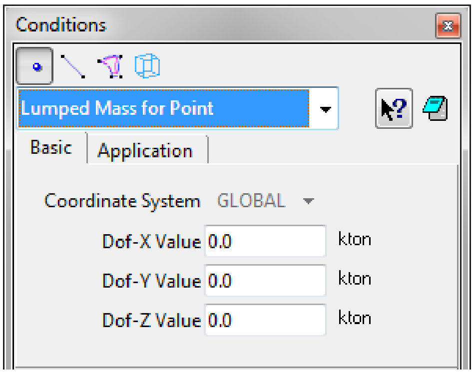


Fig. 9-4: Lumped mass for point

Velocity – Prescribe constant velocity. Typically used along with a load history defined in Interval Data (Fig. 9-2).

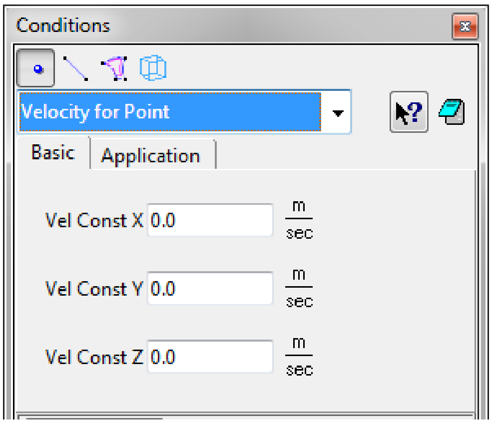


Fig. 9-5: Velocity for …

Acceleration – Prescribe constant acceleration. Typically used along with a load history defined in Interval Data (Fig. 9-2).

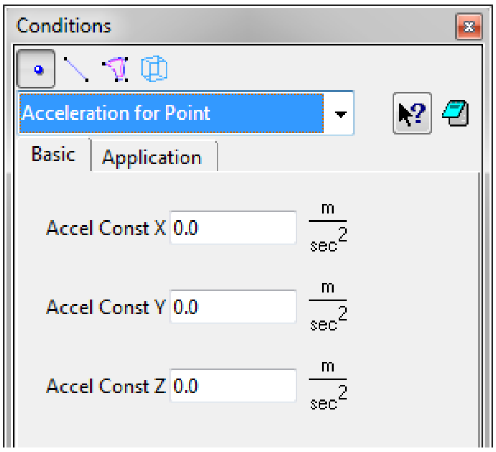


Fig. 9-6: Acceleration for …

Initial Velocity – Speed at the beginning of the analysis.

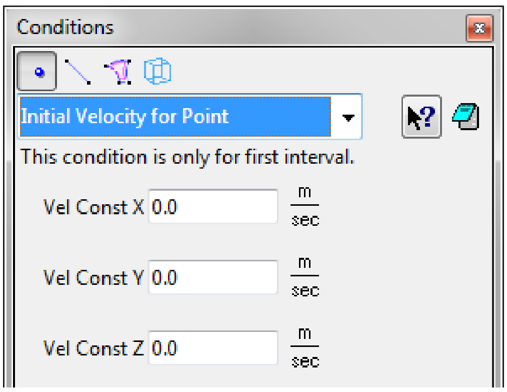


Fig. 9-7: Initial Velocity for …

Initial Acceleration – Acceleration at the beginning of the analysis.

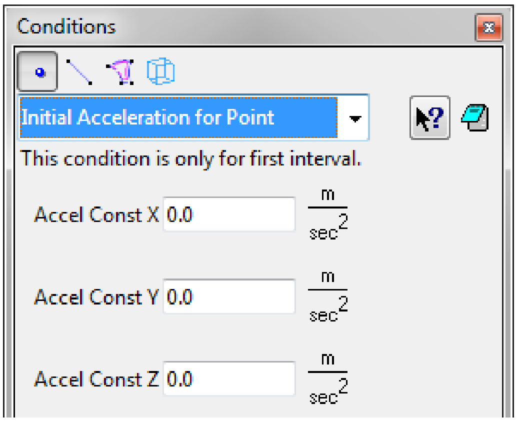


Fig. 9-8: Initial Acceleration for …

10 POST-PROCESSING IN ATENA-GID

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

Then back in the **GiD** interface the process info will appear. Through this dialog the program asks if the process of the analysed problem is finished or if the post-processing should be started. The button **Postprocess** should be selected (see Fig. 10-1).

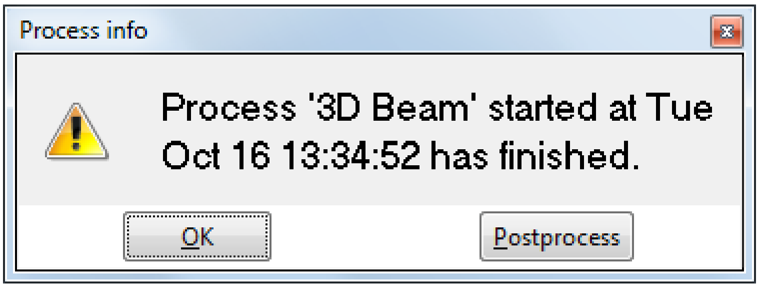


Fig. 10-1: The button Postprocess should be pressed

But before any post-processing features can be used, the results calculated in **ATENA Studio** (or **AtenaConsole**) have to be imported into **GiD**.

It is done by the clicking on the Import results from **ATENA Studio** icon  . Then the process of importing will start (see Fig. 10-3) and when it is finished the model changes its colours (see Fig. 10-4).

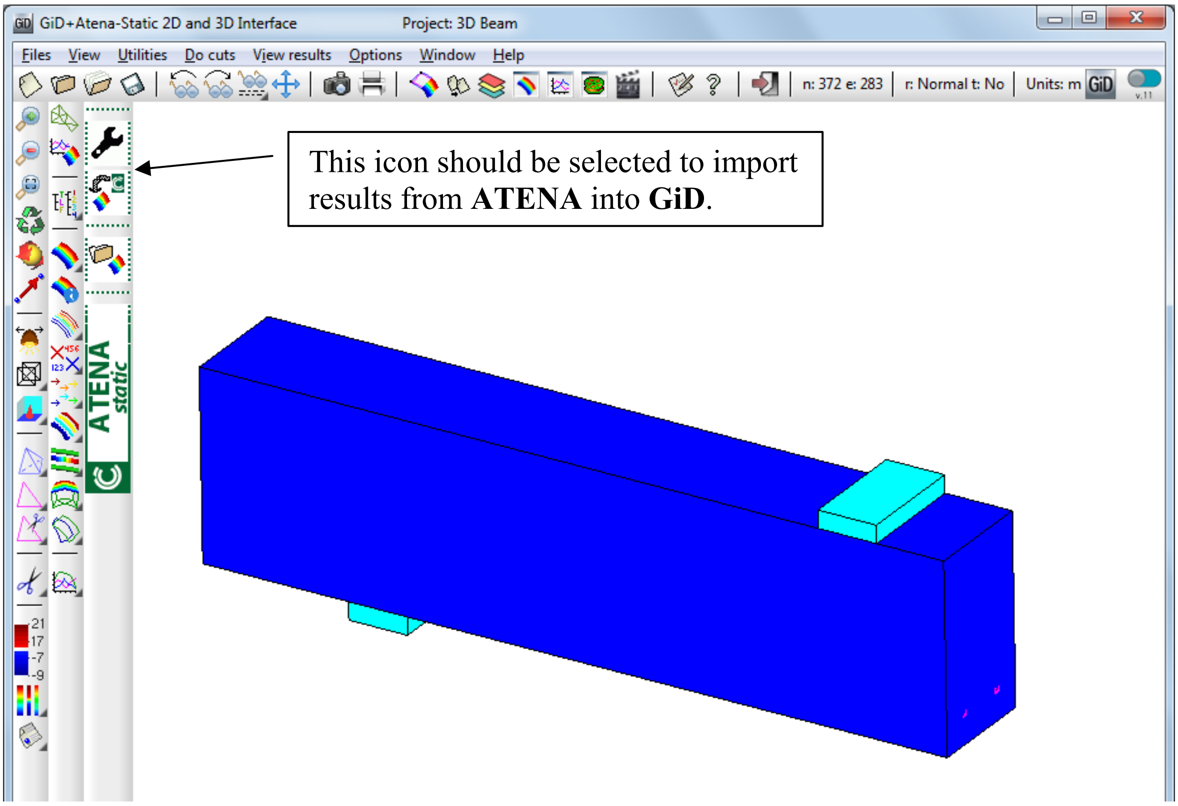


Fig. 10-2: The GiD postprocessor interface

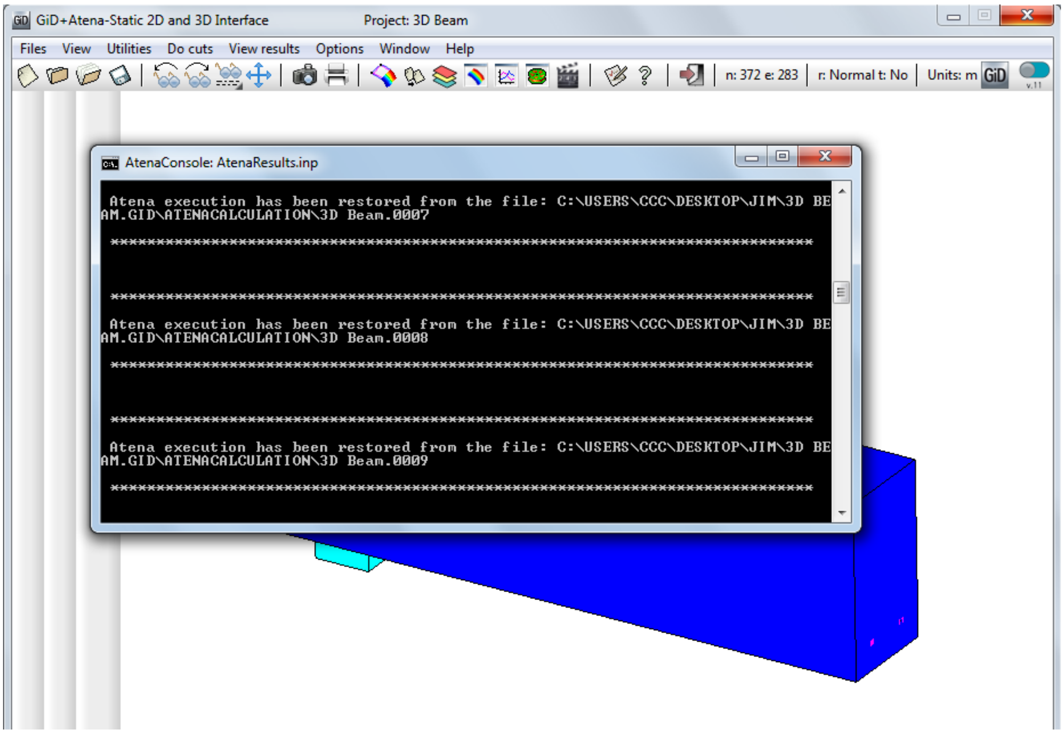


Fig. 10-3: The importing of the results from ATENA Studio into GiD

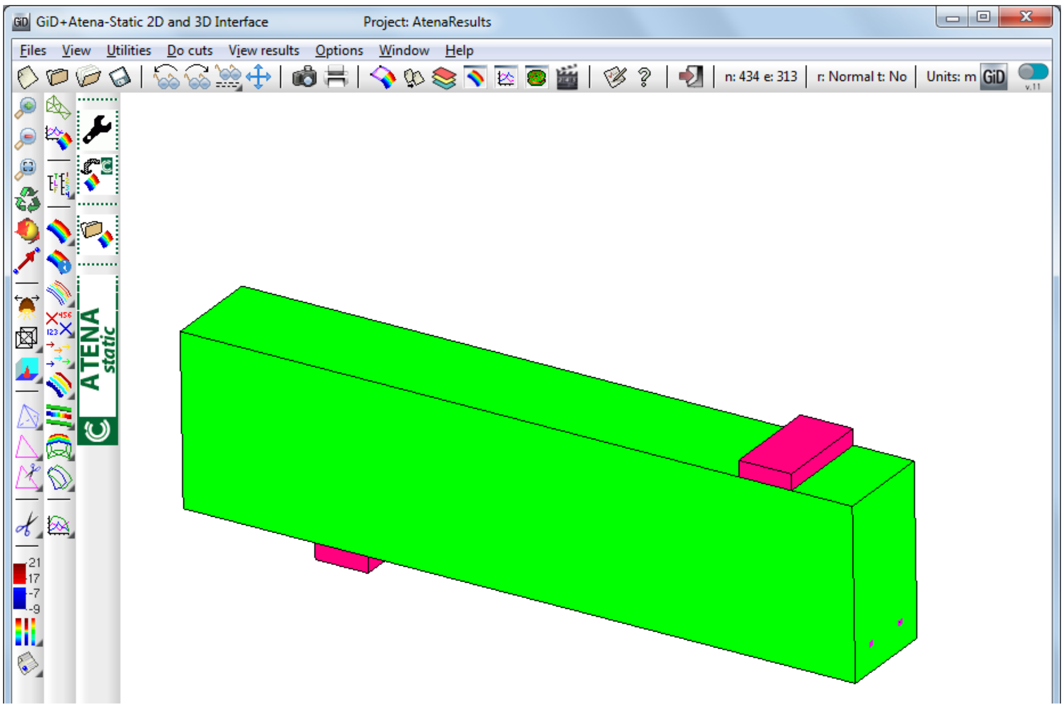


Fig. 10-4: The importing of the results from ATENA was finished

After importing data from **ATENA**, the post-processing can be started. Let's display for example cracks width.

First of all it should be checked which step will be post-processed. It is done by selecting **View Results | Default Analysis/Step | AtenaResults2GiD** in the main menu or by the Default Analysis/Step icon  for example step 35 (see Fig. 10-5).

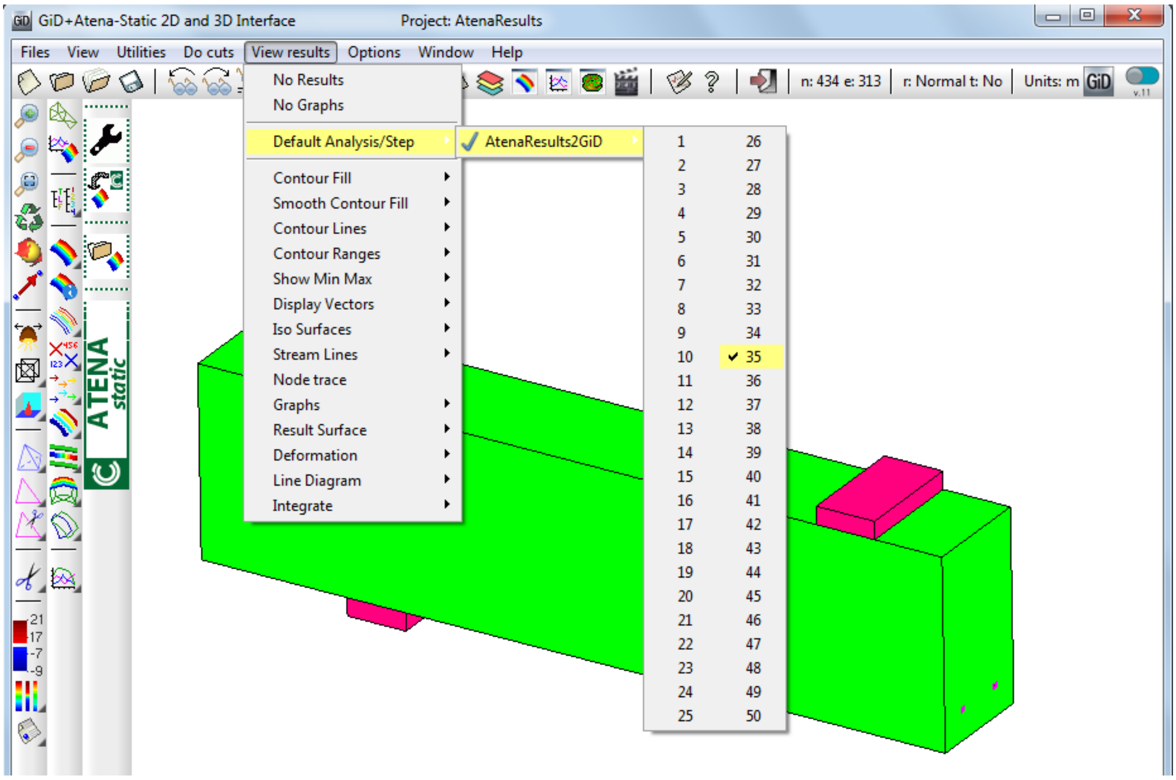


Fig. 10-5: The selection of the step which should be post-processed

By the clicking on the Contour fill icon  or by the selecting the command from main menu **View results | Contour Fill | CRACK WIDTH | COD1** crack width can be displayed (see Fig. 10-6).

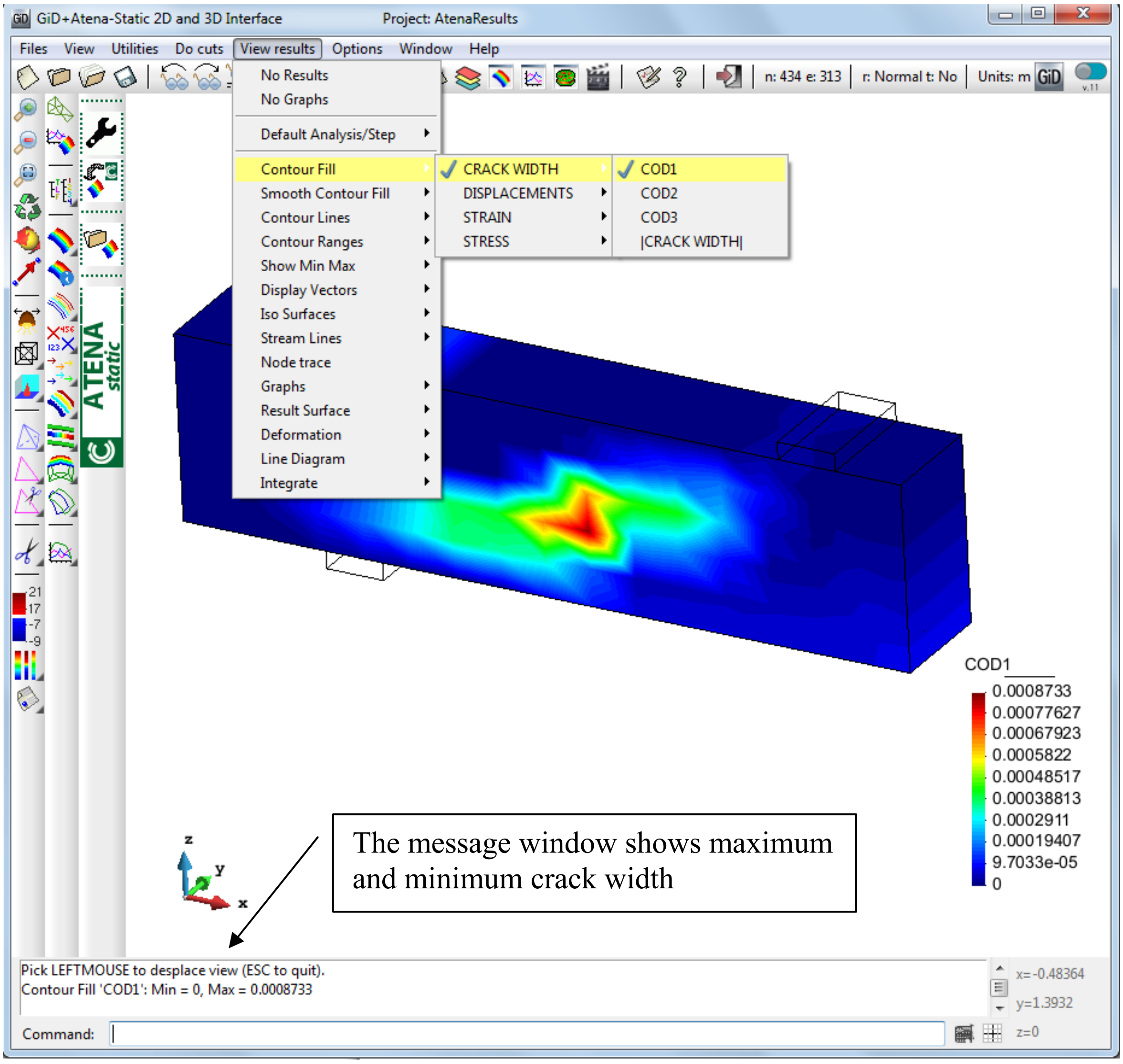


Fig. 10-6: The display of the crack width

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

It is done by selecting icon  Toggle between pre- and post-processing (see Fig. 10-7). After that a dialog window appears and the button **OK** should be pressed. The program switches into pre-processing. Then the command **Data | Problem Data | Post Data** can be selected in the main menu and a window for the definition of the post data will appear (see Fig. 10-8). This dialog you can run directly by clicking to icon  in postprocessor.

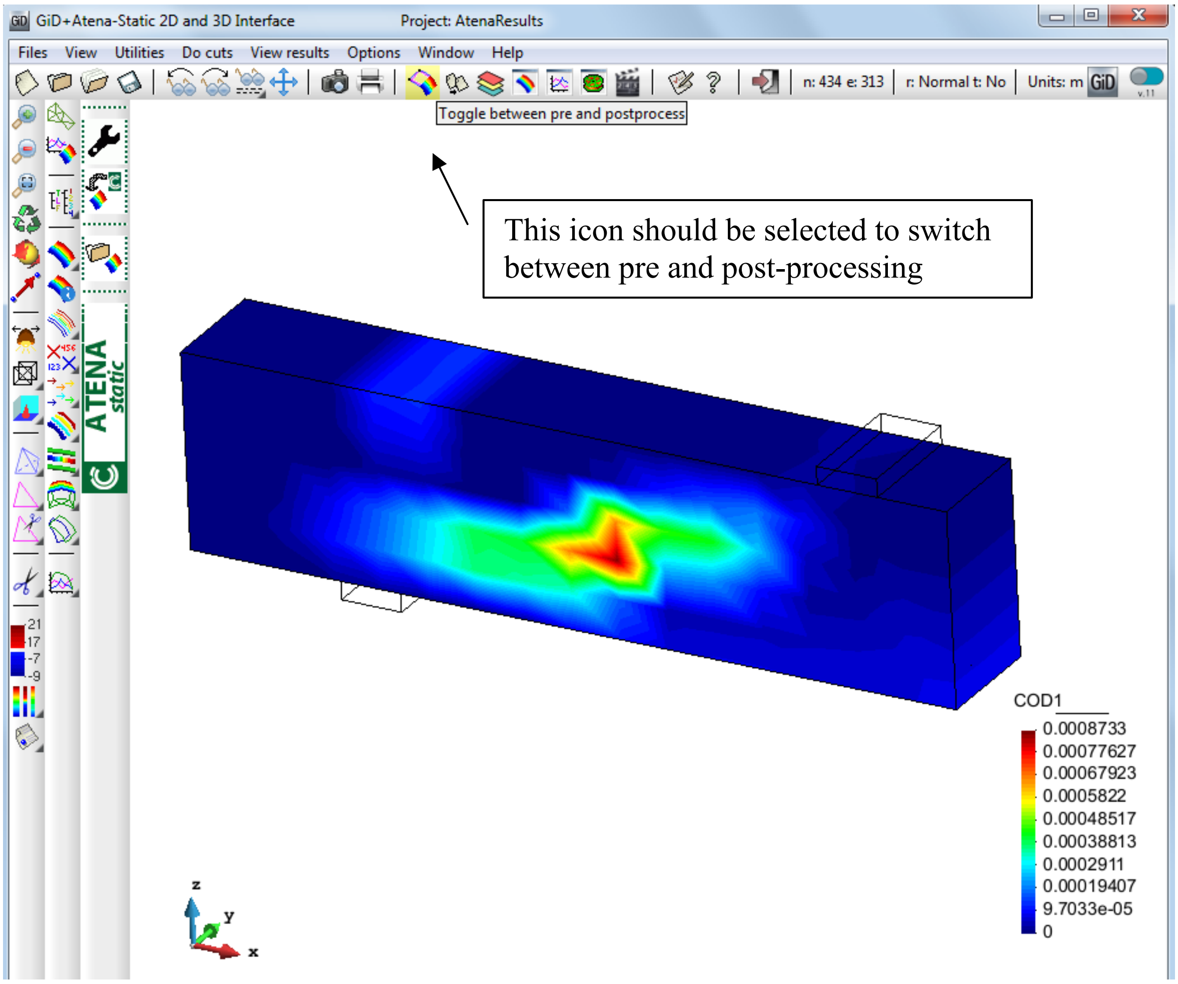


Fig. 10-7: Switch between pre and postprocessing

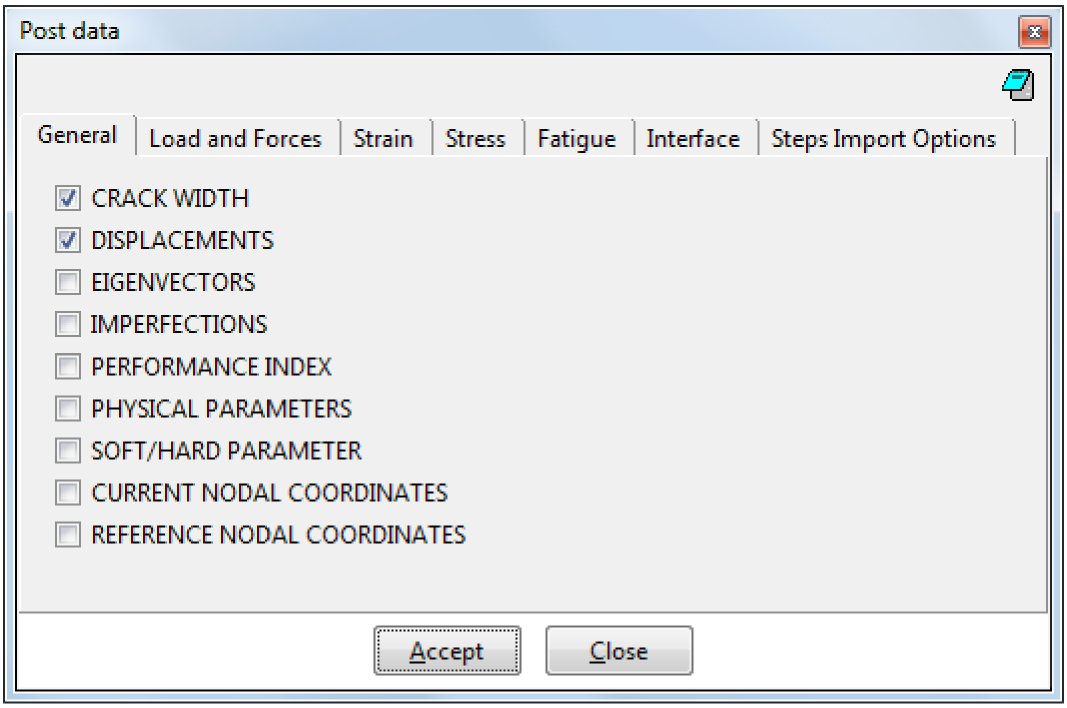


Fig. 10-8: The selection of the data which should be available for the post-processing

For example the FRACTURE STRAIN can be chosen. The definition of post data is completed by selecting **Accept** button (see Fig. 10-9). Then the button **Close** can be pressed and the **GiD** will switch to post-process automatically. But there in the post-process the data from **ATENA** has to be imported again.

It is done by the clicking on the **ATENA** icon  . Then the FRACTURE STRAIN can be found in the options for the post processing (see Fig. 10-10, to obtain this figure the 35th step has to be selected again).

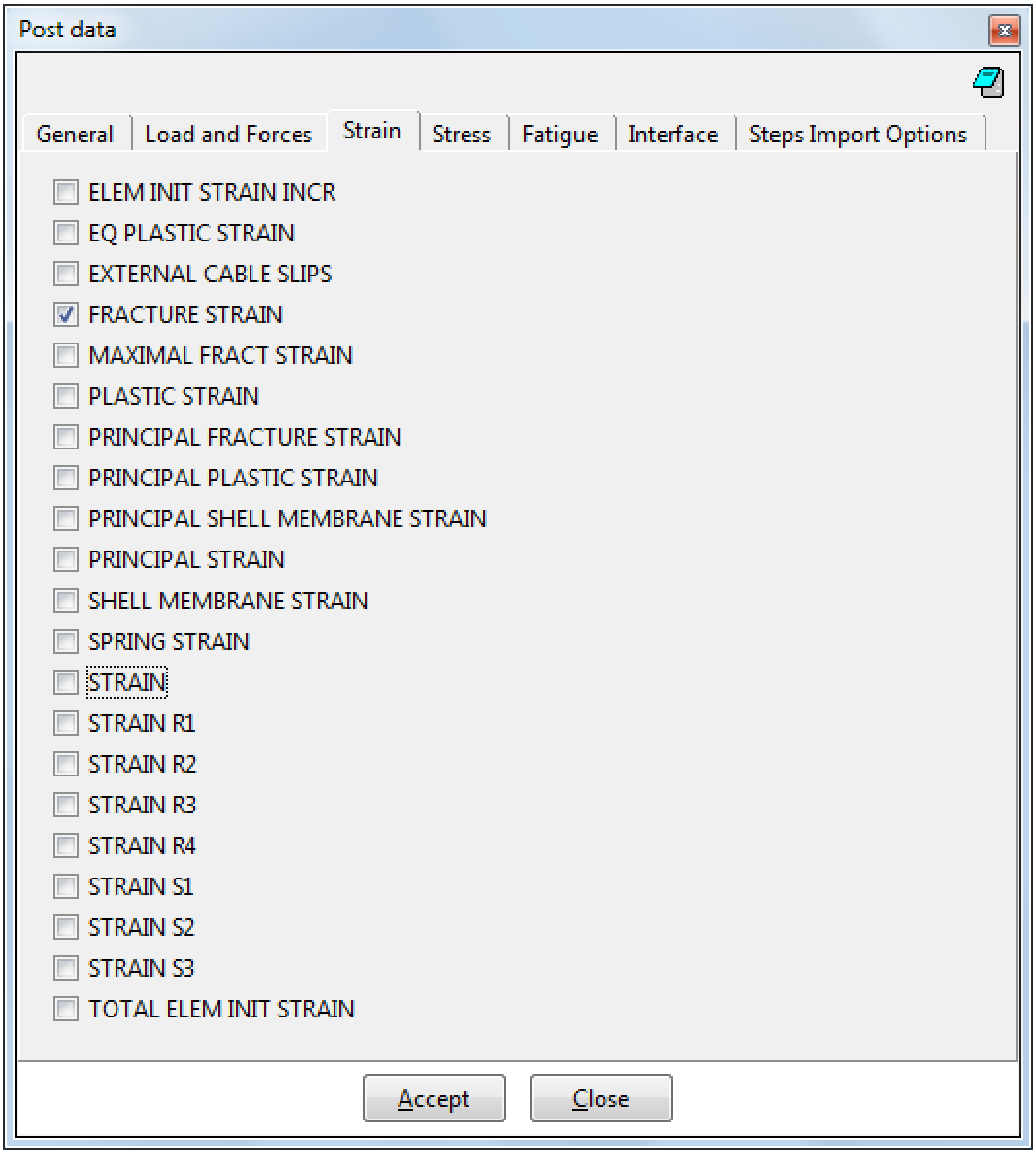


Fig. 10-9: The selection of the FRACTURE STRAIN

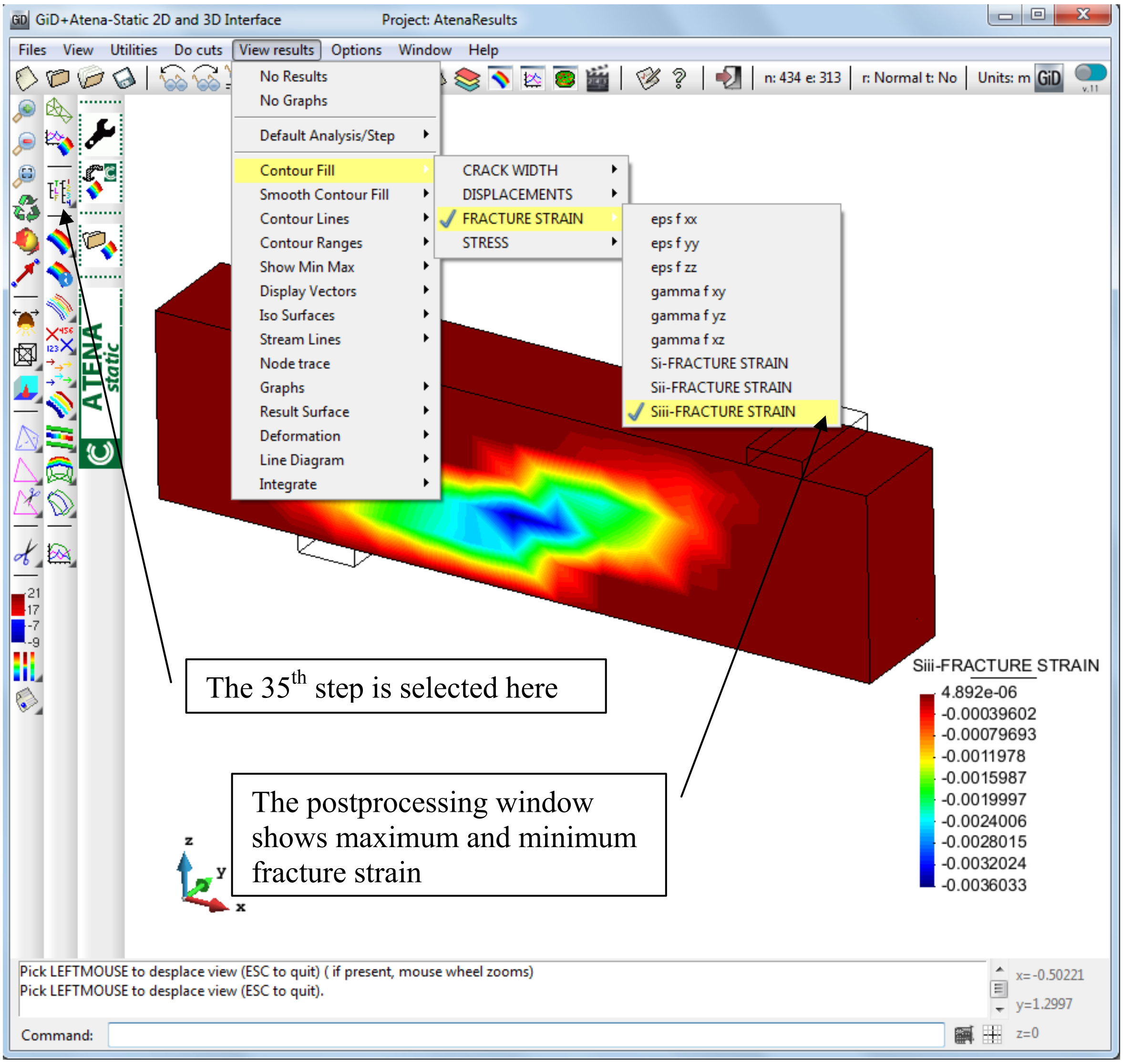


Fig. 10-10: The displayed FRACTURE STRAIN

More post-processing capabilities can be found in the Help of the **GiD**.

11 USEFUL TIPS AND TRICKS

11.1 Export IXT for ATENA 3D Pre-processor

It is also possible to export 3D mesh to an IXT format, which can be imported to **ATENA 3D** Pre-processor. This tool can be run from menu **ATENA | Export IXT file for ATENA 3D**. In this way it is possible to export meshes created by **GiD** into **ATENA 3D**. There it is possible to include ATENA specific features, such as reinforcement, materials and boundary conditions. In this approach only 3D solid finite elements will be transferred to **ATENA**. All boundary conditions, two-dimensional and one-dimensional elements will be lost as well as all material definitions. This method is useful in cases when very complex meshes for curved geometries need to be created.

12 EXAMPLE DATA FILES

Following data files of examples for **GiD** application are included in the **ATENA** installation:

Directory - Tutorial.Creep2D

BeamWithCreep.gid	Slab with creep that is modelled as a two-dimensional structure

Directory - Tutorial.Creep3D

SlabWithColumn.gid	symmetric quarter of a square 3D slab with creep modelled using shell elements
ReinforcedSlabWithSpringSupport.gid	creep experiment in Bratislava

Directory - Tutorial.Dynamic

BridgeConcreteSinusImpulsLoad.gid	Simply supported beam with sinus impulse load
BridgeConcreteSinusImpulsLoad_demo.gid	Same as above, but for demo version
BridgeElasticSinusImpulsLoad.gid	Simply supported beam with elastic material and sinus impulse load
SingleDegreeFreeVibration.gid	Single degree of freedom example with free vibration

Directory - Tutorial.Static2D

axisym.gid	Axisymmetric problem
PunchingShearFailure.gid	Axisymmetric problem of slab punching failure
InterfaceWithShear.gid	Example with an interface material model
TunnelWithConstructionProcess.gid	Two-dimensional analysis of a simple tunnel with construction process
FourPointRCBeam.gid	Only static analysis without creep of the slab specimens tested by Metrostav, Praha
FourPointRCBeam_demo.gid	Same as above, but can be analysed with ATENA demo

Directory - Tutorial.Static3D

SmallCantileverWithTorsion_DiscreteBars.gid	Example of L-shaped cantilever with discrete bars for main reinforcement as well as for stirrups.
InterfaceWithShear3D.gid	Example of interface between two concrete plates.
SlabWithColumn.gid	Slab-column connection

Tunnel3DWithConstructionProcess.gid	Three-dimensional model of a tunnel with soil and construction process
BeamWithBeamElements.gid	Example with 3D beam elements
DirectTensionFatigue.gid	Example of a notched direct tension test with fatigue material model
ShearBeam3D.gid	Example of four-point bending

Directory - Tutorial.Temperature2D

LamellaFire.gid	Example of thermal analysis with hydration of concrete
PipeBStatic.gid	Static part of a pipe analysis with thermal loading
PipeBTemp.gid	Thermal part of a pipe analysis with thermal loading

Directory - Tutorial.Temperature3D

tram014stat5_DM.gid	Static part of a 3D beam analysis with thermal loading
tram014temp5_DM.gid	Thermal part of a 3D beam analysis with thermal loading
ColumnThermal3D.gid	3D Column with temperature loading
ColumnThermal3D_demo.gid	Same as above, but for demo version
tubbing_static2-1932.gid	3D tubing with fire loading - static
tubbing_temp2-1932.gid	3D tubing with fire loading - transport
Vitek3Dfire.gid	3D four point beam with fire loading
Vitek3Dmoist.gid	3D four point beam with moisture loading
Vitek3Dstat.gid	3D four point beam with temperature loading - static
Vitek3Dtemp.gid	3D four point beam with temperature loading - transport

13 CALCULATION OF ATENA IDENTIFICATION NUMBERS

The following section describes the method that is used by **ATENA-GiD** interface to determine the numbering for various ATENA element types and element groups. The numbers of element types and element groups will not be identical to the ids in **GiD**. It is impossible to preserve the same ids in **GiD** and **ATENA**. The ATENA ids are derived based on the number of element nodes and based on the used material using the tables and formulas below.

Table 4: ATENA element type ids based on the geometric nonlinearity and number of element nodes. The element type id are calculated based on Eq. (2) and (3).

ElementType for 3D	**ElemsNnode**	Geometrical	
		LINEAR	NONLINEAR
CCIsoGap<xxxxxxxx>	8	**28**	**58**
CCIsoGap<xxxxxx>	6	**26**	**56**
CCIsoBrick<xxxxxxxxxxxxxxxxxxxx>	20	**20**	**50**
CCIsoWedge<xxxxxxxxxxxxxxx>	15	**15**	**45**
CCIsoTetra<xxxxxxxxxx>	10	**10**	**40**
CCIsoBrick<xxxxxxxx>	8	**8**	**38**
CCIsoWedge<xxxxxx>	6	**6**	**36**
CCBarWithBond	2	**5**	**35**
CCIsoTetra<xxxx>	4	**4**	**34**
CCIsoTruss<xxx>	3	**3**	**33**
CCIsoTruss<xx>	2	**2**	**32**
CCSpring/CCLineSpring/CCPlaneSpring	1	**1**	**31**

ElementType for 2D		LINEAR	NONLINEAR
CCIsoGap<xxxx>	4	**24**	**54**
CCIsoQuad<xxxxxxxx>	8	**8**	**38**
CCIsoTriangle<xxxxxx>	6	**6**	**36**
CCBarWithBond	2	**5**	**35**
CCIsoQuad<xxxx>	4	**4**	**34**
CCIsoTriangle<xxx>	3	**3**	**33**
CCIsoTruss<xx>	2	**2**	**32**
CCSpring/CCLineSpring/CCPlaneSpring	1	**1**	**31**

ELEMENT_GROUP_ID = **Mat_ID * 100** + **ELEMENT_TYPE_ID** **(1)**

3D Element:

	Increment	
AddingShellID	16	Increment if is Shell element
AddingGapElemID	20	Increment if is Gap element
AddingNonLinElemID	30	Increment if is element Geometrical Nelinearity

Formula:

ELEMENT_TYPE_ID = ElemsNnode + AddingGapElemID
+ AddingNonLinElemID + AddingShellID (2)

1D Element:

	Increment	
AddingBarWithBond	3	Increment if is element BarWithBond

Formula:

ELEMENT_TYPE_ID = ElemsNnode +
AddingBarWithBond+AddingNonLinElemID (3)

Load cases:

In Dynamic problem, there is a special load case for total conditions in each interval, numbered 510 000 + step number. Similarly, in Transport problem, load cases for Fire_Boundary_Conditions have numbers 520 000 + step number.

Function from material:l

Function ID for function from material is calculated like 25250000 + id_of_material.

REFERENCES

[1] Cervenka, V., Jendele, L, Cervenka, J., (2012), *ATENA Program Documentation, Part 1, Theory*, Cervenka Consulting, 2012

[2] Cervenka, V. and Cervenka, J., (2012), *ATENA Program Documentation, Part 2-1, User's Manual for ATENA Engineering 2D*, Cervenka Consulting, 2012

[3] Cervenka, V. and Cervenka, J., (2012), *ATENA Program Documentation, Part 2-2, User's Manual for ATENA Engineering 3D*, Cervenka Consulting, 2012

[4] Cervenka, J., and Jendele, L., (2012), *ATENA Program Documentation, Part 6, ATENA Input File Format*, Cervenka Consulting, 2012

[5] Benes, S., Mikolaskova, J., (2012), *ATENA Program Documentation, Part 12, User's manual for ATENA Studio*, Cervenka Consulting, 2012

[6] Prochazkova Z., Cervenka, J., Janda, Z., Pryl, D., (2012), *ATENA Program Documentation Part 4-6, ATENA Science – GiD Tutorial,* Cervenka Consulting, 2012

[7] Kabele, P., Cervenka, V., and Cervenka, J., (2012), *ATENA Program Documentation Part 3-1, Example Manual ATENA Engineering,* Cervenka Consulting, 2012

[8] Cervenka, V., Cervenka, J., and Janda Z., (2012), *ATENA Program Documentation Part 3-2, Example Manual ATENA Science,* Cervenka Consulting, 2012

[9] Pryl, D. and Cervenka, J., (2013), *ATENA Program Documentation Part 11, ATENA Troubleshooting,* Cervenka Consulting, 2013