



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

ATENA Program Documentation Part 8

User's Manual for ATENA-GiD Interface

Written by

**Vladimír Červenka, Jan Červenka,
Zdeněk Janda, and Dobromil Pryl**

Prague, September 3rd, 2021



Acknowledgements:

The software was developed with partial support of Eurostars funding program.

*The following models were developed with the financial support of TA ČR:
CC3DNonLinCementitious2HPRFC, CC3DNonLinCementitious2FRC,
CC3DNonLinCementitious2SHCC, CCMoDelGeneral, CCTransportMaterial*



Trademarks:

ATENA is registered trademark of Vladimir Cervenka.

GiD is registered trademark of CIMNE of Barcelona, Spain.

Microsoft and Microsoft Windows are registered trademarks of Microsoft Corporation.

Other names may be trademarks of their respective owners.

Copyright © 2000-2021 Červenka Consulting, s.r.o.

CONTENTS

| | |
|--|----------|
| CONTENTS | 1 |
| 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..... | 24 |
| 5.3.1 <i>Solid Concrete Material</i> | 29 |
| 5.3.2 <i>Shell Material</i> | 42 |
| 5.3.3 <i>Beam Material</i> | 50 |
| 5.3.4 <i>Reinforced Concrete</i> | 55 |
| 5.3.5 <i>1D Reinforcement Material</i> | 56 |
| 5.3.6 <i>Interface Material</i> | 62 |
| 5.3.7 <i>Spring material</i> | 68 |
| 5.3.8 <i>The Material Function</i> | 70 |
| 5.3.9 <i>Material from file</i> | 70 |
| 5.3.10 <i>Dummy Material</i> | 71 |
| 5.3.11 <i>Steel von Mises 3D Material</i> | 71 |
| 5.4 Interval Data - Loading History..... | 73 |
| 5.4.1 <i>Fatigue</i> | 78 |
| 5.5 Problem Data | 82 |
| 5.6 Concrete 3D Printing..... | 86 |

| | | |
|-----------|---|------------|
| 5.7 | Units..... | 90 |
| 5.8 | Finite Element Mesh | 91 |
| 5.8.1 | <i>Notes on Meshing</i> | 91 |
| 5.8.2 | <i>Finite Elements for ATENA</i> | 92 |
| 5.9 | ATENA Menu..... | 96 |
| 6 | STATIC ANALYSIS..... | 97 |
| 7 | CREEP ANALYSIS (AND SHRINKAGE)..... | 99 |
| 7.1 | Boundary Conditions and Load Cases Related Input..... | 101 |
| 7.2 | Specific Creep Boundary Conditions..... | 102 |
| 7.3 | Material Input Data | 102 |
| 8 | TRANSPORT ANALYSIS (MOISTURE AND HEAT) | 105 |
| 8.1 | Material Input Data | 105 |
| 8.1.1 | <i>Material CCTransport (CERHYD)</i> | 105 |
| 8.1.2 | <i>Generating material characteristics by ConMix</i> | 112 |
| 8.1.3 | <i>Material Bazant_Xi_1994 (only included for backward compatibility of old models)</i> | 116 |
| | <i>Other Settings Related to Transport Analysis</i> | 117 |
| 8.2 | Specific Transport Boundary Conditions | 120 |
| 9 | DYNAMIC ANALYSIS | 123 |
| 9.1 | Other Settings Related to Dynamic Analysis..... | 123 |
| 9.2 | Specific Dynamic Boundary Conditions..... | 128 |
| 10 | POST-PROCESSING IN ATENA-GiD..... | 131 |
| 11 | USEFUL TIPS AND TRICKS | 139 |
| 11.1 | Export IXT for ATENA 3D Pre-processor | 139 |
| 12 | EXAMPLE DATA FILES..... | 141 |
| 13 | CALCULATION OF ATENA IDENTIFICATION NUMBERS..... | 147 |
| | REFERENCES | 149 |

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 also be done 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 axisymmetrical 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 on 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 descriptions 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 sometimes be useful, and it allows the user to exploit all the features of the **ATENA** software. The 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-day 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 the 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 computer¹. 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.

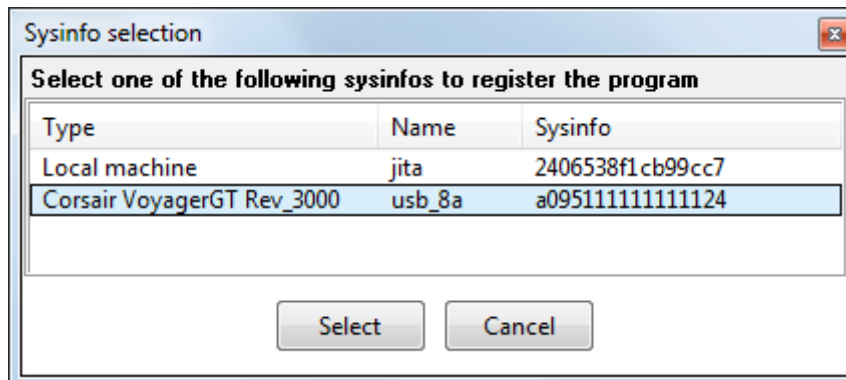


Fig. 3-1: Choice of USB or PC protection

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

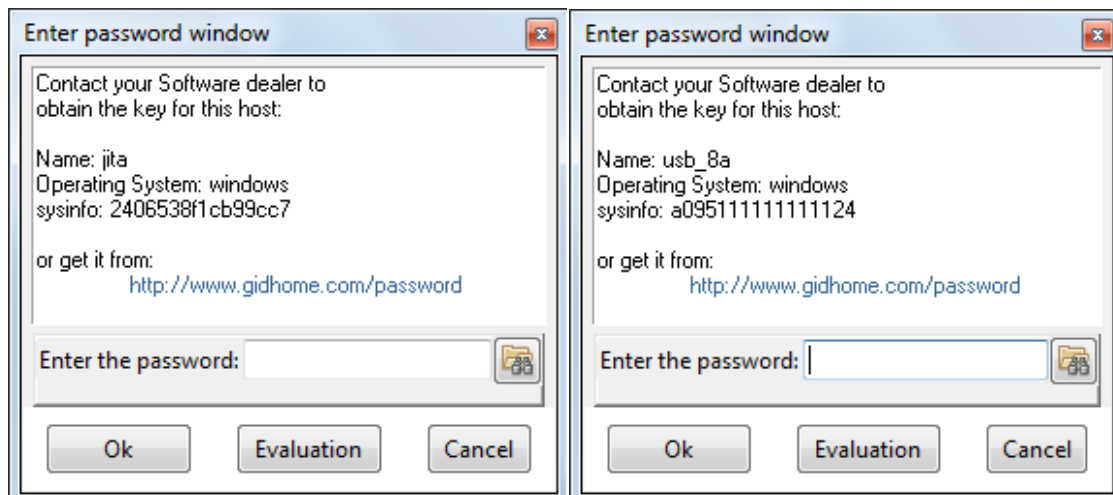



Fig. 3-2: GiD register window (PC protection left, USB protection right)

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 a license server. Follow the instructions from the **GiD** web <http://www.gidhome.com/documents/passserver/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 (do not forget to expand the **ATENA Science** branch at the selection page of the installer).

4.1 Manual Installation of the ATENA-GiD Scripts

Alternatively, the **ATENA-GiD** interface can also be 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 a 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. 5-3. 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*.

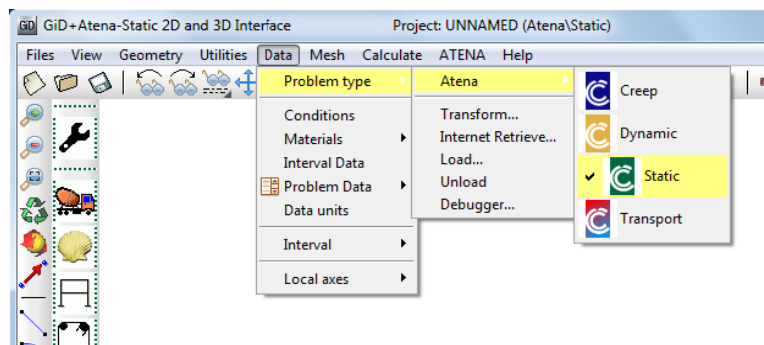



Fig. 5-3 Problem type menu.

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. 5-4, left or by

icon . You can view all currently defined conditions in the 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 the 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 geometrical 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. 5-4. For each condition, the appropriate parameters can be defined as shown in Fig. 5-4, right.

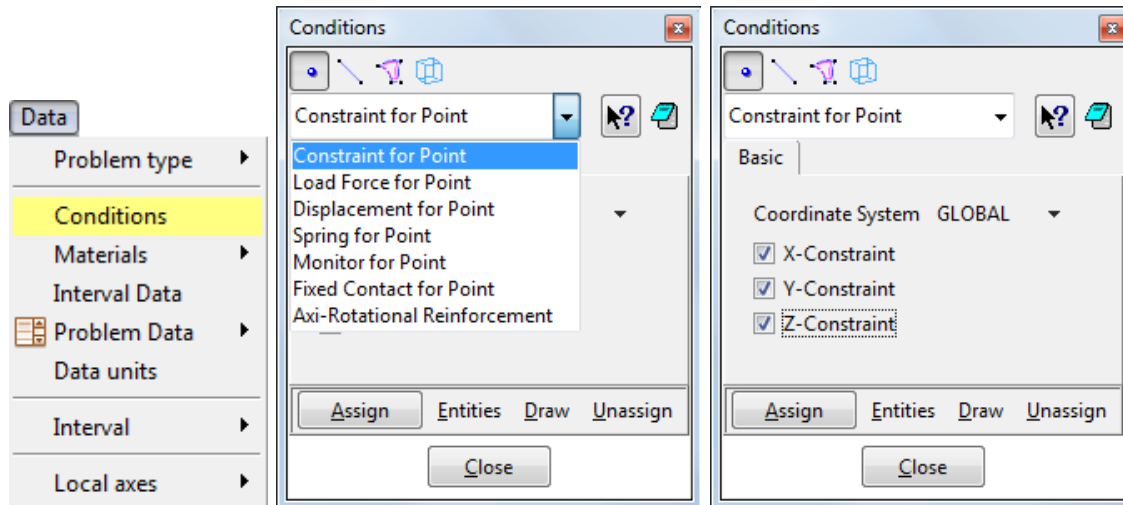


Fig. 5-4 Conditions: menu, list at Point, applied at Point.

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

Assign - The target of the 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 the command line. There is always a hint what kind of action is required from the user.

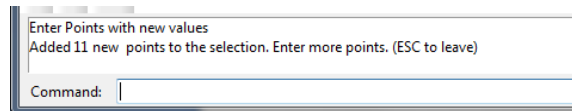


Fig. 5-5 Hints at the Command Line at the bottom of GID Window

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 the existing assignment of the selected condition type (for selected or all entities).

If it is necessary to modify the parameters of a 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.

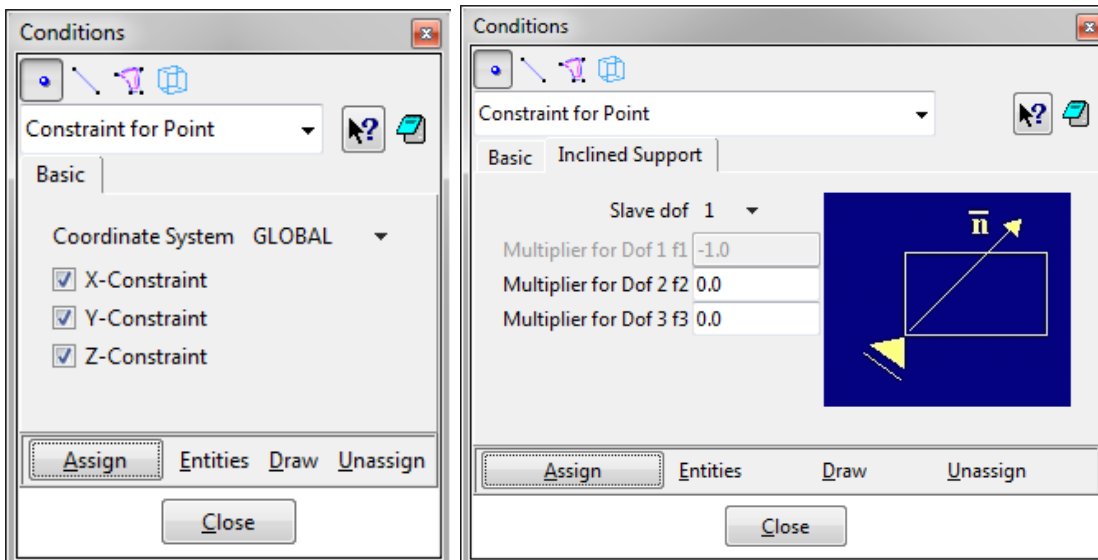


Fig. 5-6 Conditions: Constraint for ...

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. 5-7). Similarly, condition for point needs to be applied to each point where different conditions for the line of the same type are intersecting.

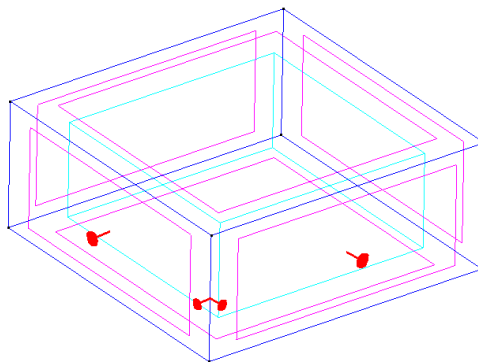


Fig. 5-7: Proper Support Assignment at the Edge of Two Surfaces

Rotation Constraint - This is a boundary condition similar to Constraint condition for modeling rotation supports and can be defined for point, line, and surface of 1D Beam or 2D Shell. This condition you can apply on elements that have more than 3 degrees of freedom – DoFs (1D beam, 2D Shell, ...). The simplest way how to set the condition is to choose the global coordinate system and select the directions to be fixed. The inclined coordinate system enables rotated support conditions.

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 using 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.

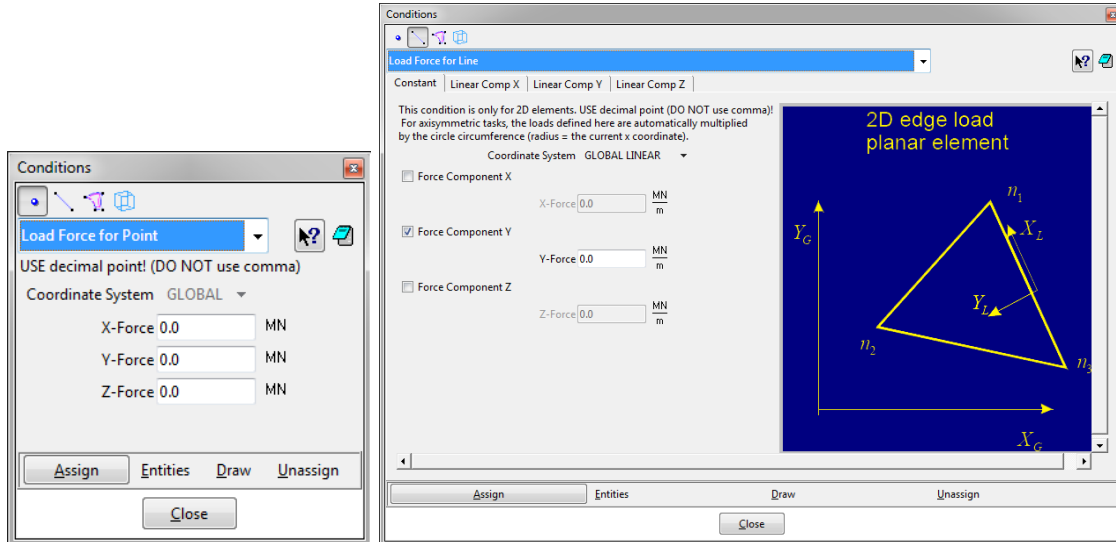


Fig. 5-8 Conditions: Load Force for ...

Load universally for line and surface - Load can be applied to a part of a line or surface. When entering the force magnitudes for each component, it is possible to select suitable units. You can apply this condition to any line or surface in the model. If you don't want to mesh this entity, and you want to use the line or surface only to apply the load, you can assign a material from book Dummy to this entity. The number A DIV and B DIV define into how many parts the element will be split for the forces application.

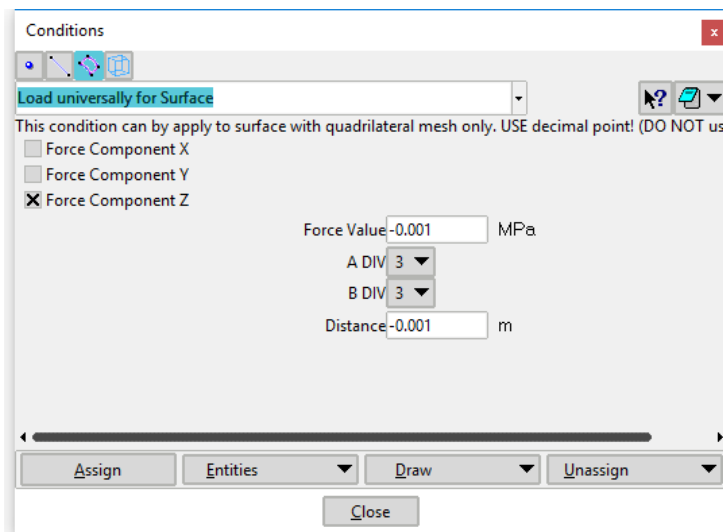


Fig. 5-9 Conditions: Displacement for ...

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.

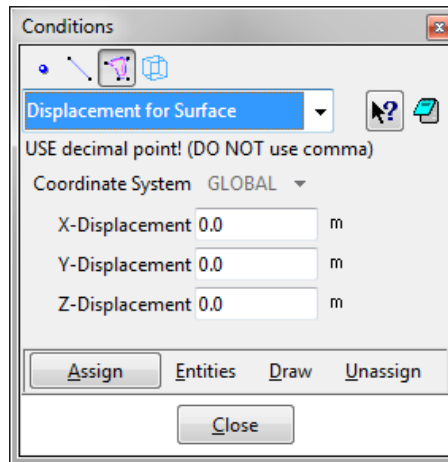


Fig. 5-10 Conditions: Displacement for ...

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.

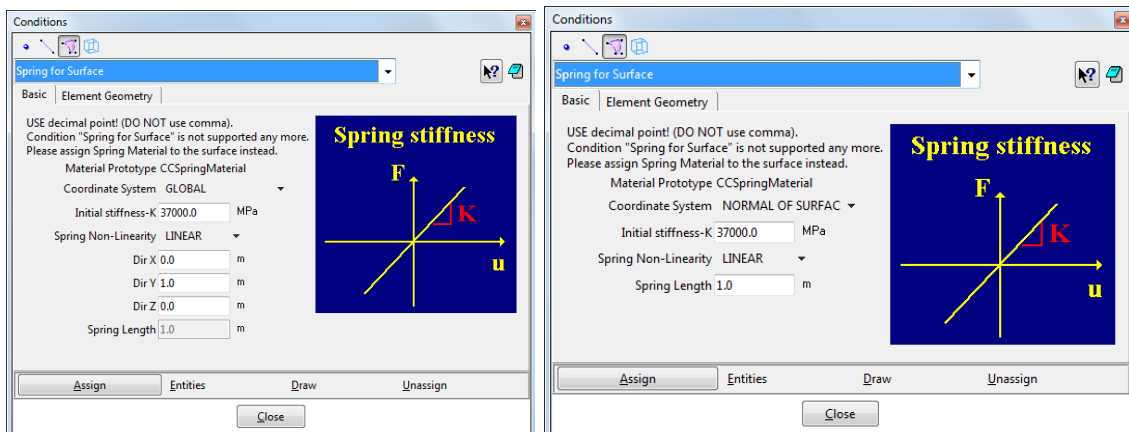


Fig. 5-11 Conditions: Spring for ...

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

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).

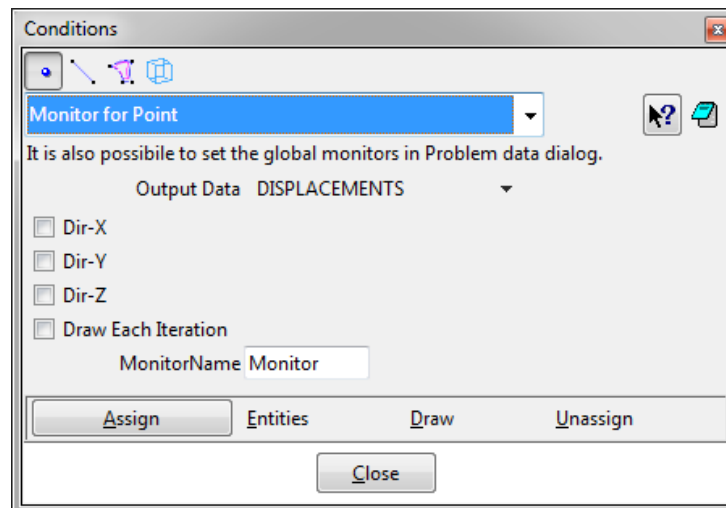


Fig. 5-12 Conditions: Monitor for ...

Monitor for Point Nearest Node IP - This condition can be used for monitoring data closest to a given location. It is possible to select at which entity/location (NODE, IP, GNODE) the quantity shall be monitored. This condition can be assigned to either existing modelling point or to a point specifically created for this purpose. In **GiD** 9.0.6 and older, it is also necessary to mark the point with the command **Mesh | No skip | Point** if it does not belong to any meshed entity.

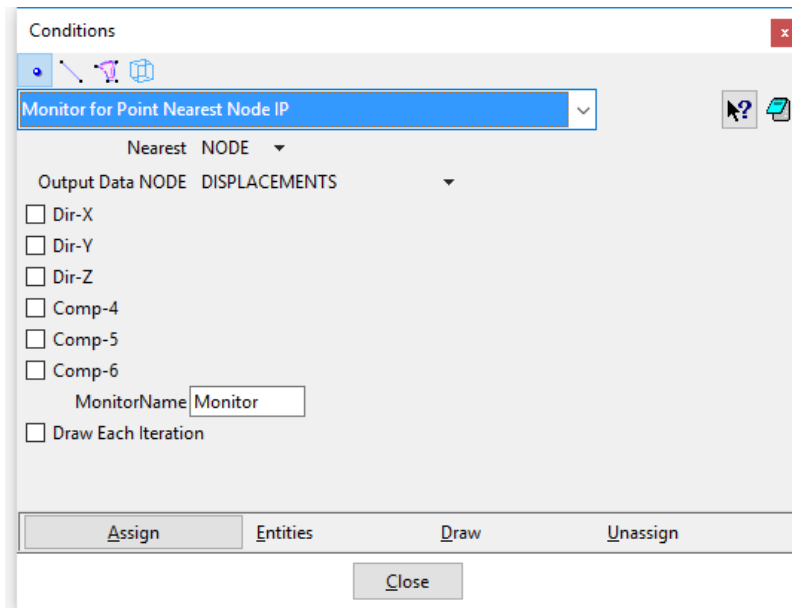


Fig. 5-13 Conditions: Monitor for Point Nearest Node IP

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.

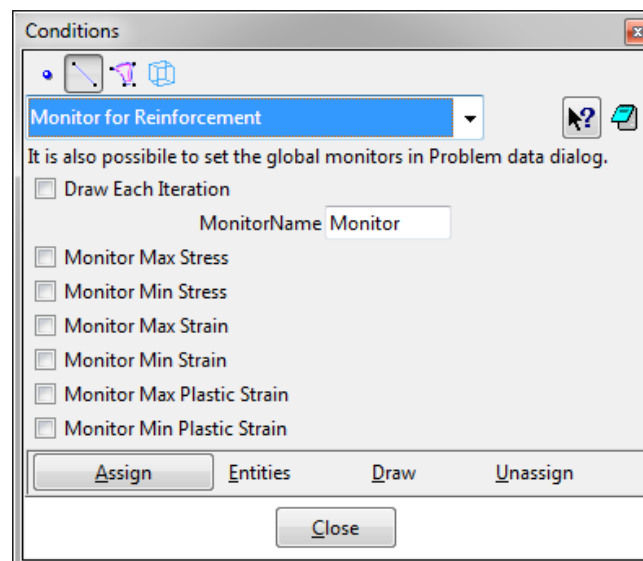


Fig. 5-14 Conditions: Monitor for Reinforcements

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.

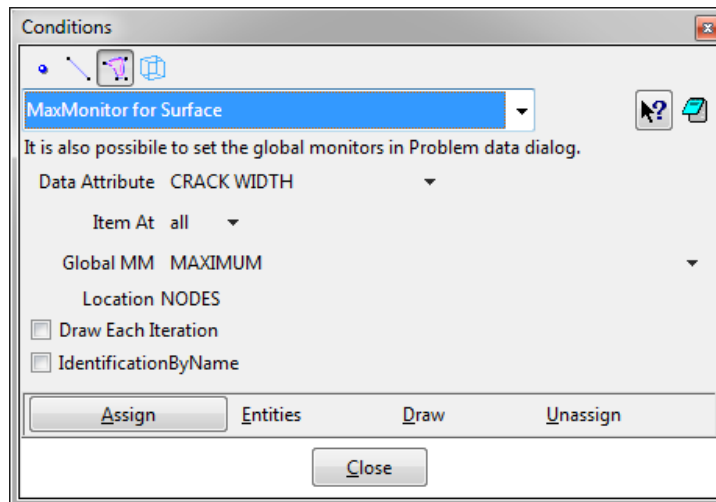


Fig. 5-15 Conditions: Max Monitor for ...

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 5.3.5.3 for information on Interface (GAP) elements. Please also understand this condition can only be used for connections between 1D Beams and 2D Shells if all rotational DoFs are to be connected, i.e, you need the conditions **Fixed_1D_Beam_to_Solid_for...** and **Fixed_2D_Shell_to_Solid_for_...** (see below) when some of the rotations should not get connected (pinned connections).

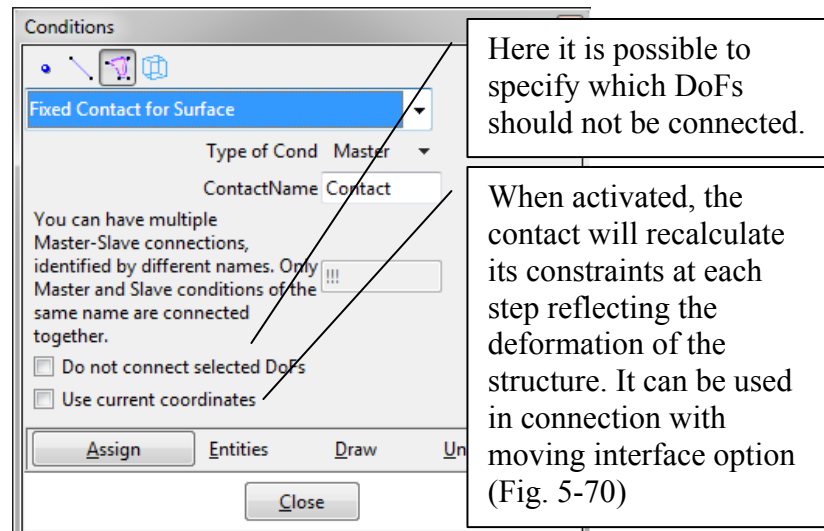


Fig. 5-16 Conditions: Fixed Contact for ...

Fixed_1D_Beam_to_Solid_for... and **Fixed_2D_Shell_to_Solid_for_...** – These conditions are similar to Fixed contact, but they are intended for connecting 1D beam and 2D shell to other parts of the structure. Assign the parameter **Type of Cond** as **1D_BEAM** or **2D_SHELL** to one side (the 1D Beam or the 2D Shell) of the intended connection and **Type of Cond** as **SOLID** to the second part of the connection. If you connect 1D Beam to 2D Shell, assign **1D_BEAM** to the beam side and **SOLID** to the 2D Shell side. This connection can be defined for points, lines, and surface depending on the type common of the boundary between the two objects.

You can have multiple connections identified by different names. Only 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 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 modeling 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.

The **ROTATION_ARMS** n parameter specified how many points are used to fix its CS's rotation. By default, $n=1$, i.e. 1 node above and 1 node below the mid-plane. For $n=2$ we use nodes at $-h$, $-h/2$, $h/2$, h , where h is CS's height.

FIX_ROTATION_xx specifies the range $\langle x;x \rangle$ of the cross-section, on which shell's / beam's edge nodes are connected. The "x" can be **B**, **M**, **T**, **L** and **R** for the bottom, middle, top, left and right, respectively. For example, in case of a 1D beam, if options **FIX ROTATIONS BT** and **FIX ROTATION MR** are defined, the nodes from bottom to top and from middle to right of the beam's cross-section will be connected (i.e. one half of the cross-section).

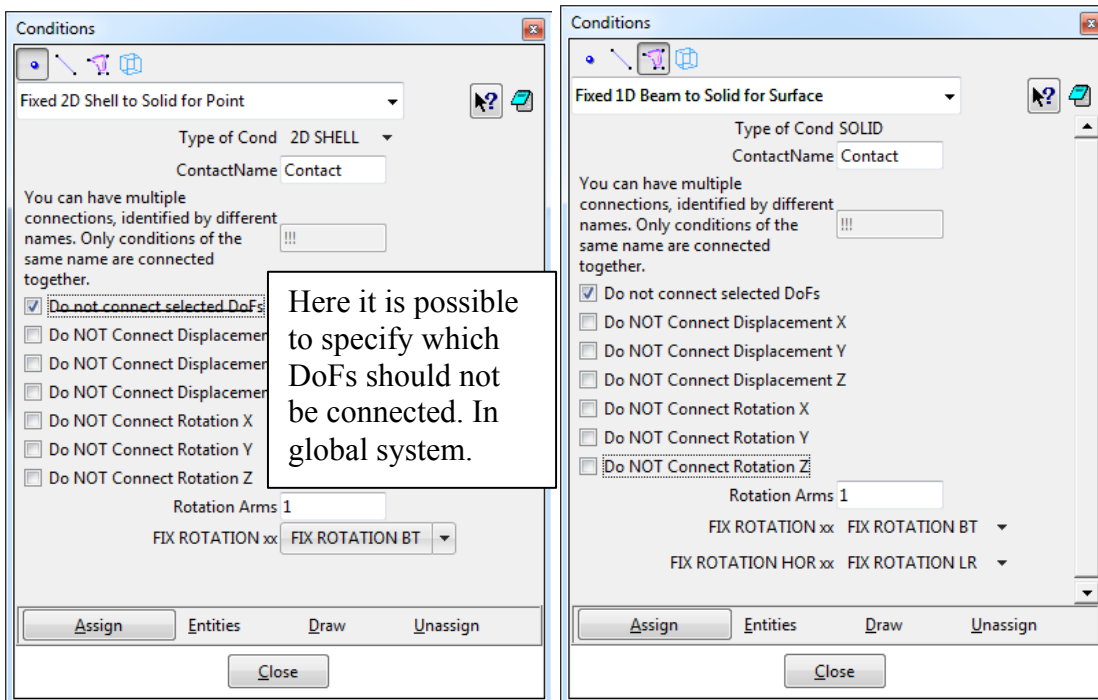


Fig. 5-17 Conditions: Fixed 1D Beam and 2D Shell to Solid for ...

If the option **Do not connect selected DoFs** is chosen, the user can define which degrees of freedom will not be connected. This option is useful for creating a pinned connection when the rotation DoF can be released. The DoF separation is defined in global coordinates for displacement DoFs, but in local for the rotational ones.

When connecting two 1-D Beams in a point, please do NOT use the condition **Fixed_1D_Beam_to_Solid_for_Point**, as the moments are then not transferred correctly. In that case, please use a shared node, or a Master-Slave connection (**Fixed Contact for Point**).

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.

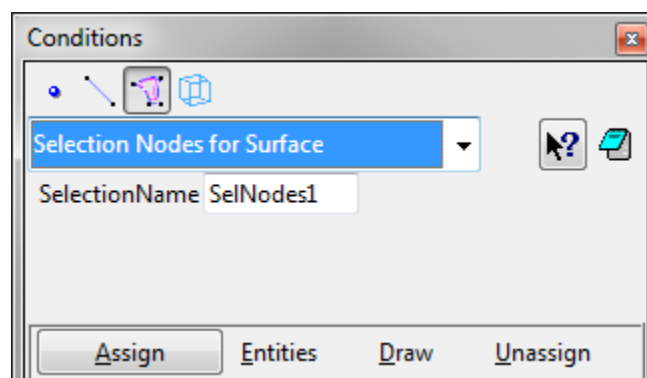


Fig. 5-18 Conditions: Selection Nodes for ...

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.

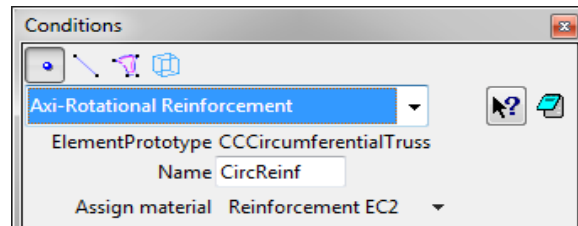


Fig. 5-19 Conditions: Axi-Rotational Reinforcement

Weight - The weight can be defined for reinforcement line, 2D elements surface, and volume. Typically, it is used to consider the dead weight load, because the dead load is not considered automatically in **ATENA**. For weight loading of 1D beams there is a special condition, Weight for 1D Beam.

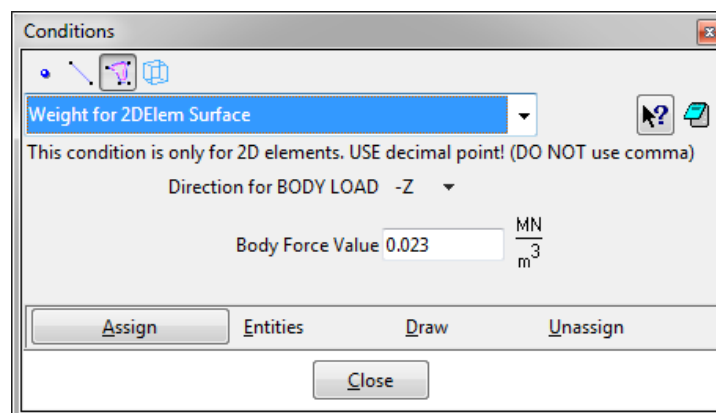


Fig. 5-20 Conditions: Weight for ...

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).

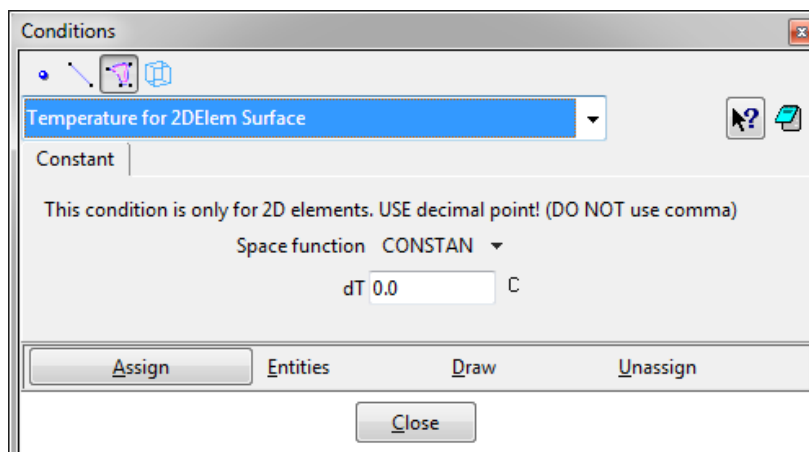


Fig. 5-21 Conditions: Temperature for ...

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).

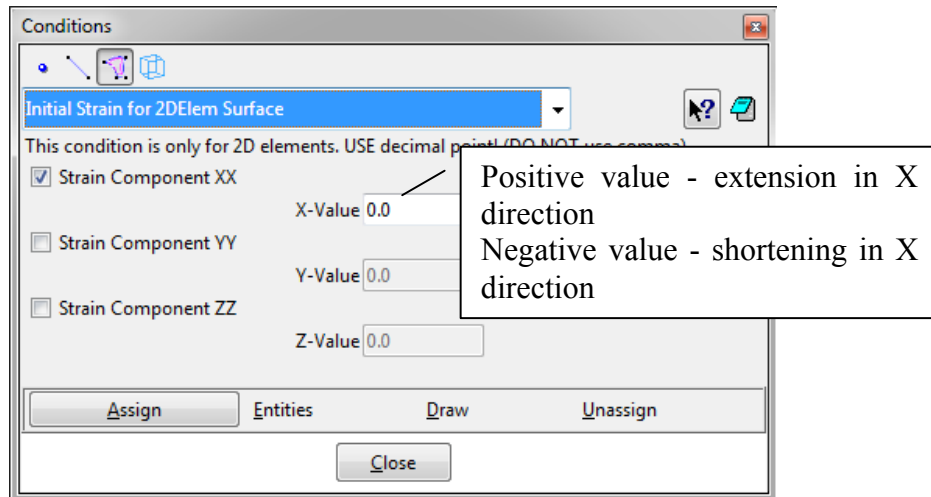


Fig. 5-22 Conditions: Initial Strain for ...

Initial Stress² – 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.

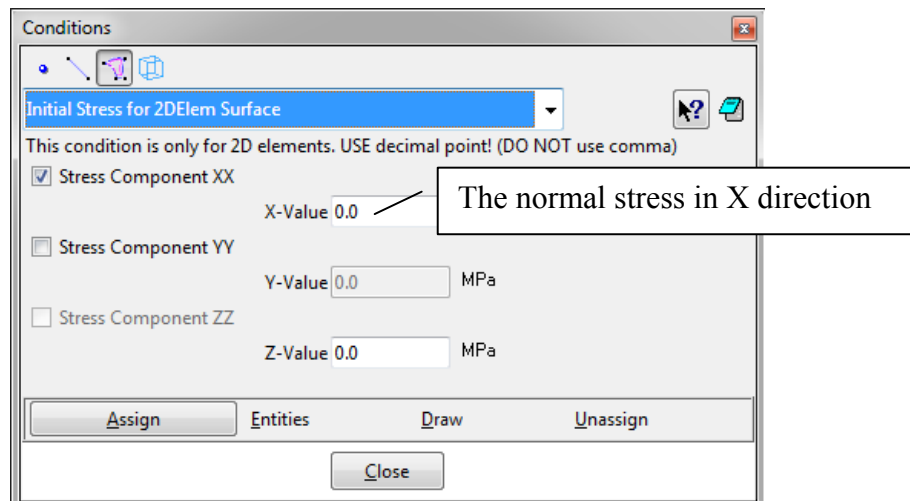


Fig. 5-23 Conditions: Initial Stress for ...

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. This condition is only needed for Shell with CCAhmadElements element type. For CCiso Shells, it is not necessary.

² Initial Stress for Reinf Line only available in version 5.1.3b and older, replaced with Prestressing for Reinf Line. Still available for 2 Elem Surface.

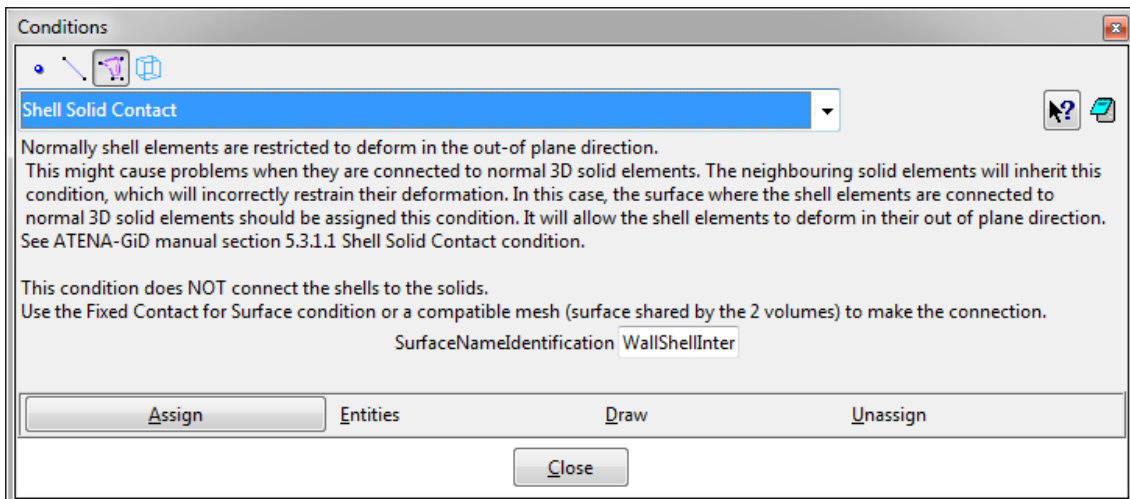


Fig. 5-24 Conditions: Shell Solid Contact for Surface

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 behavior 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 for ensuring that the mesh along each line is compatible with the rest of the model.

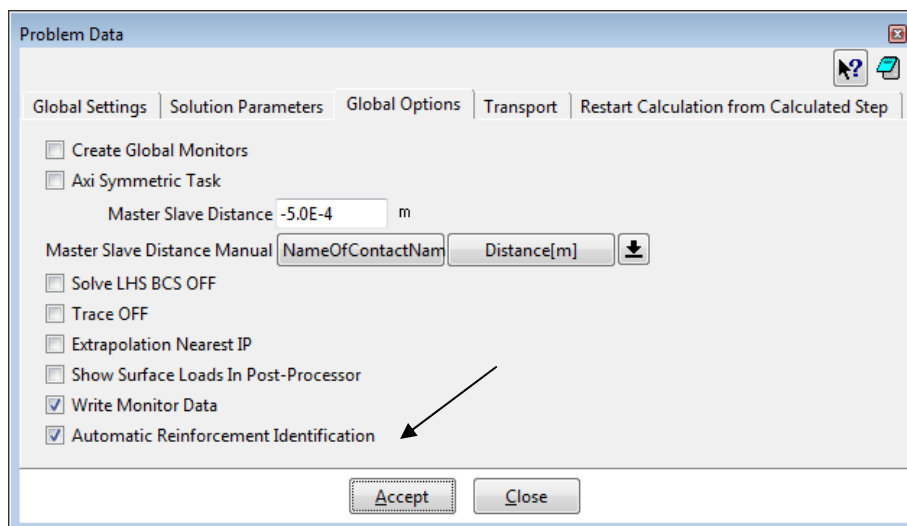


Fig. 5-25: Automatic reinforcement identification in the Problem Data dialog

Initial Gap Load for Volume – This load is used for gaps that are initially open. See material Interface, Section 5.3.5.3.

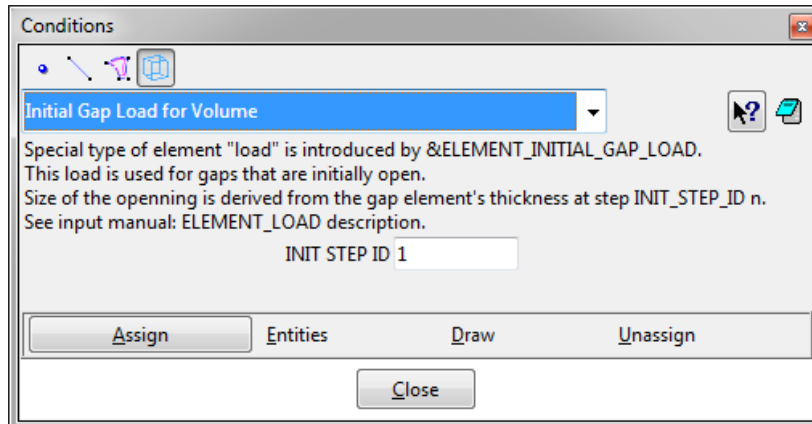


Fig. 5-26: Initial Gap Load for Volume

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

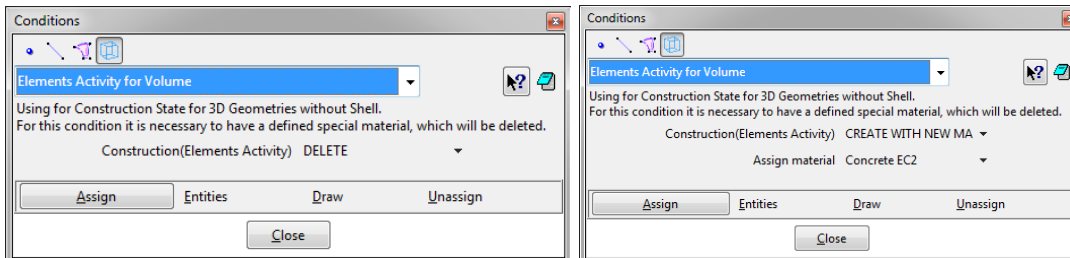


Fig. 5-27: Elements activity for ...

Reinforcement Inactivity – By this condition, you can inactivate and activate reinforcement. When applying the ACTIVE mode you can change the material of reinforcement.

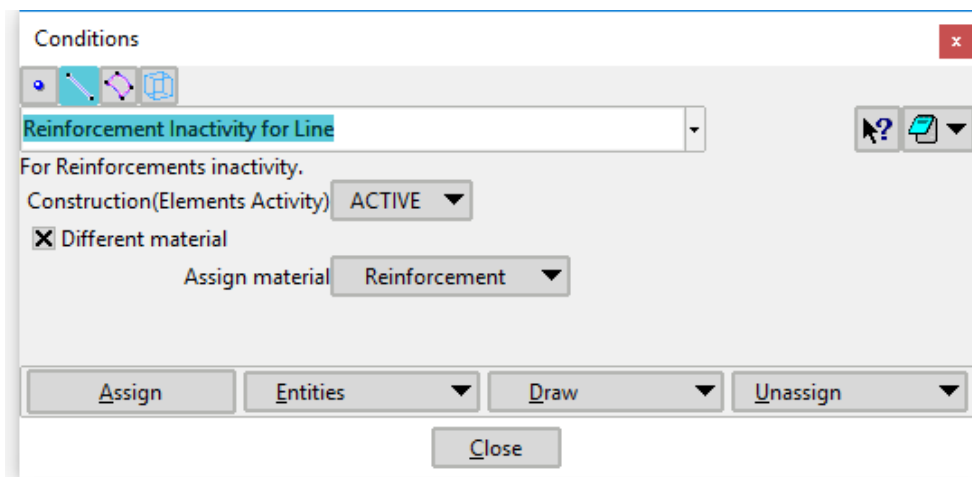


Fig. 5-28: Reinforcement Inactivity for line

Reinforcement Prestressing – By this condition, you can define the prestressing of the reinforcement, see also ATENA Troubleshooting [9], 2.2.7 How can I model prestressing losses.

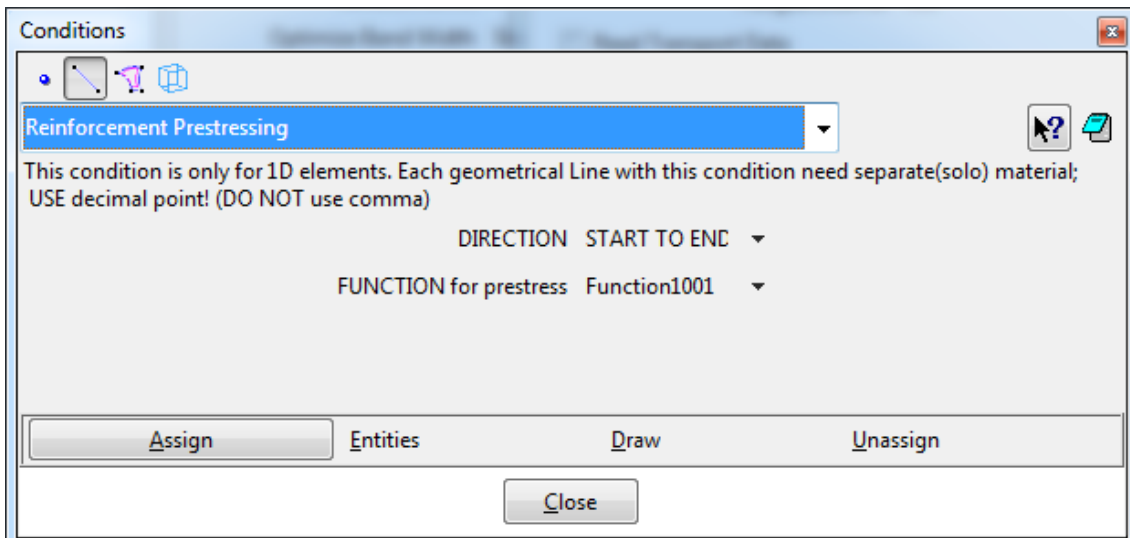


Fig. 5-29: Reinforcement Prestressing

Boundary Reactions for ... – Support for the new Austrian tunneling 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.

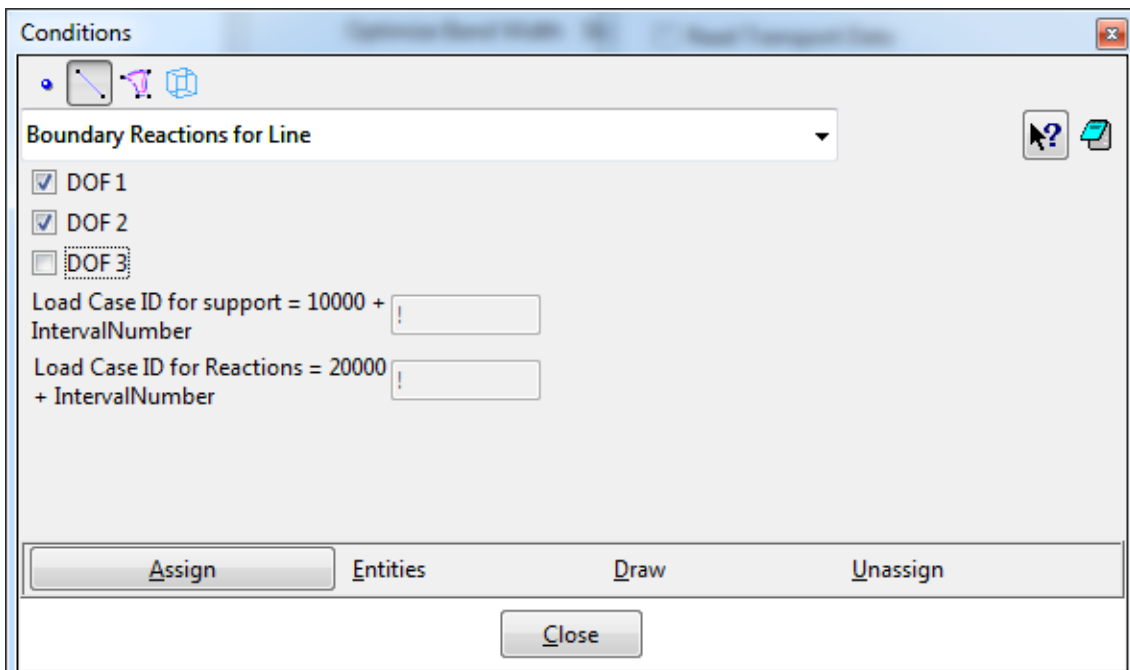







Fig. 5-30: Boundary reactions for ...

5.3 Materials

The materials are first defined and then assigned to the model. The latter 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 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 , , ,  or .

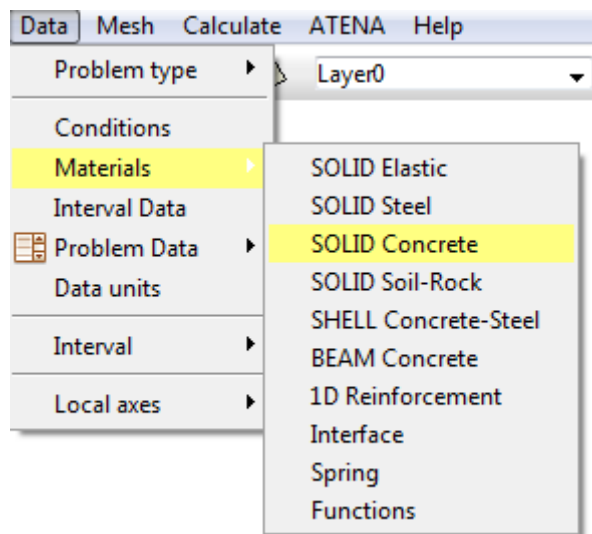


Fig. 5-31 Example of available material categories for static analysis.

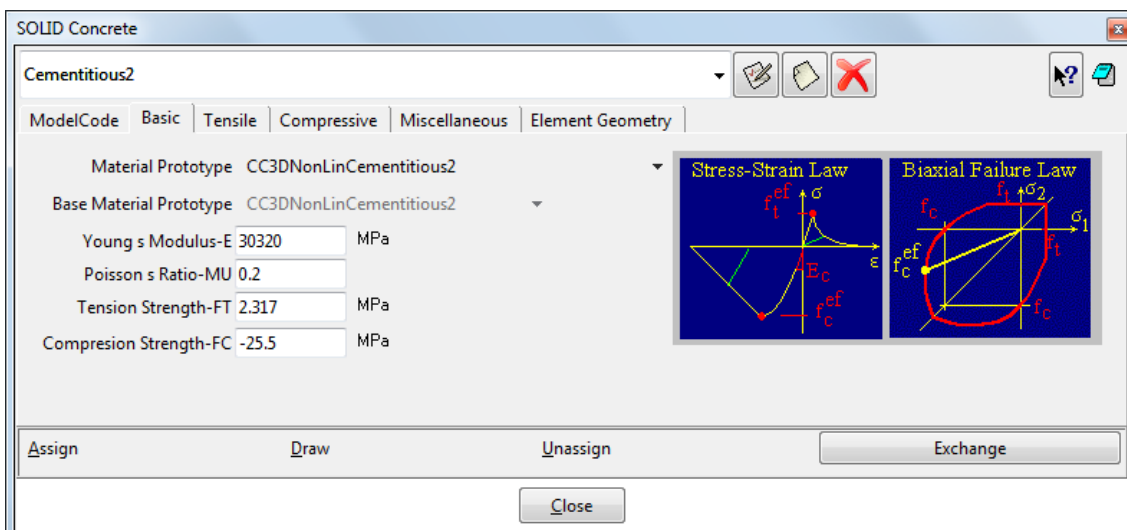


Fig. 5-32 Example of menu window for the material concrete.

Each material can be defined in a special dialog window. An example of such a window for concrete material is shown in Fig. 5-32. Each material offers default parameters. They can be changed to any desired values. After the definition of material parameters,

the material can be assigned to the numerical model. Operations for the material assignment are done with the buttons at the bottom of the dialog.



Assign - The target of the 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.



Unassign – 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 a new project to another project (exchange).

Table 1: Materials supported by GiD interface to ATENA

| <i>GiD name</i> | <i>ATENA name (INP command)</i> | <i>Description</i> |
|---|--|---|
| 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 | CC3DBiLinearVonMisesWithTempDepProperties | 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 (5.3.1.5). |
| 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 | <p>Shell geometry with support Ahmad elements, described in section 5.3.2.</p> <p>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.</p> <p>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.</p> <p>Shell material can be used only on 3D quadratic brick elements (5.8.2).</p> |
| BEAM Concrete  | | |
| Beam Concrete | CCBeam3DMaterial | <p>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).</p> <p>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.</p> <p>Beam material can be used only on 3D quadratic brick elements (5.8.2).</p> |

| 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 5.3.5.3 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 |
| CCSmearReinf | 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. 5-33). The generated values are displayed in a window (Fig. 5-34). 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**.

Generation is according the Model Code 2010:

$$\begin{aligned} \Gamma_{design} &= 1.5, \Gamma_{characteristic} = 1.0, \\ \Gamma_{mean} &= 1.0, \Gamma_{en1992-2} = 1.18577 \end{aligned}$$

$$F_{cs} = F_{ck} / \Gamma_{design}, G_f = 73 * F_{cs}^{0.18}$$

$$F_{cm} = F_{ck} + 8, E = 21500 * \left(\frac{F_{cm}}{10}\right)^{\frac{1}{3}}$$

For $F_{ck} \leq 50\text{MPa}$

$$\begin{aligned} F_{ctm} &= 0.3 * F_{ck}^{2/3} \\ \alpha_{cc} &= 0.52 \end{aligned}$$

For $F_{ck} > 50\text{MPa}$

$$\begin{aligned} F_{ctm} &= 2.12 * \ln(1 + 0.1 * F_{cm}) \\ \alpha_{cc} &= 0.51 \\ F_{ctk} &= 0.7 * F_{ctmk} \end{aligned}$$

For SafetyFormat = Mean

$$\begin{aligned} F_t &= F_{ctm} / \Gamma_{design} \\ F_{cs} &= F_{cm} / \Gamma_{design} \end{aligned}$$

For others

$$\begin{aligned} F_t &= F_{ctk} / \Gamma_{design} \\ F_{cs} &= F_{ck} / \Gamma_{design} \end{aligned}$$

$$F_{t0} = -2.1 * F_t$$

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 a 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.

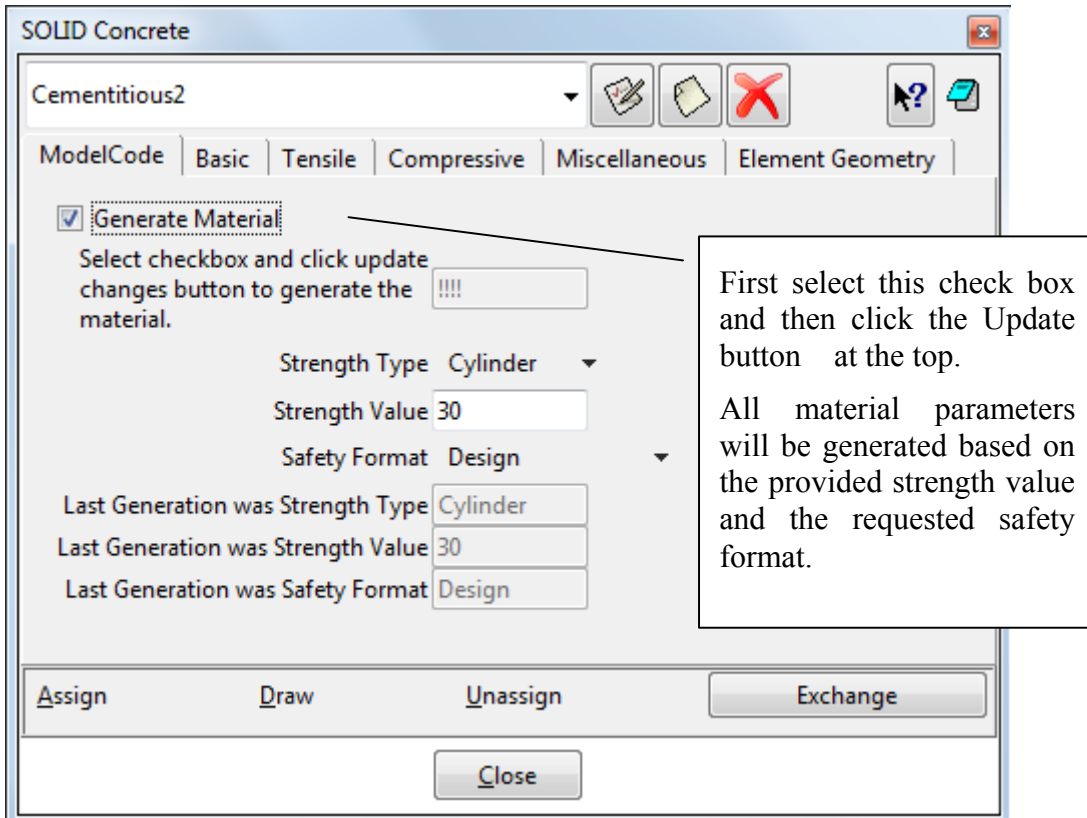


Fig. 5-33: Cementitious2 – Model Code

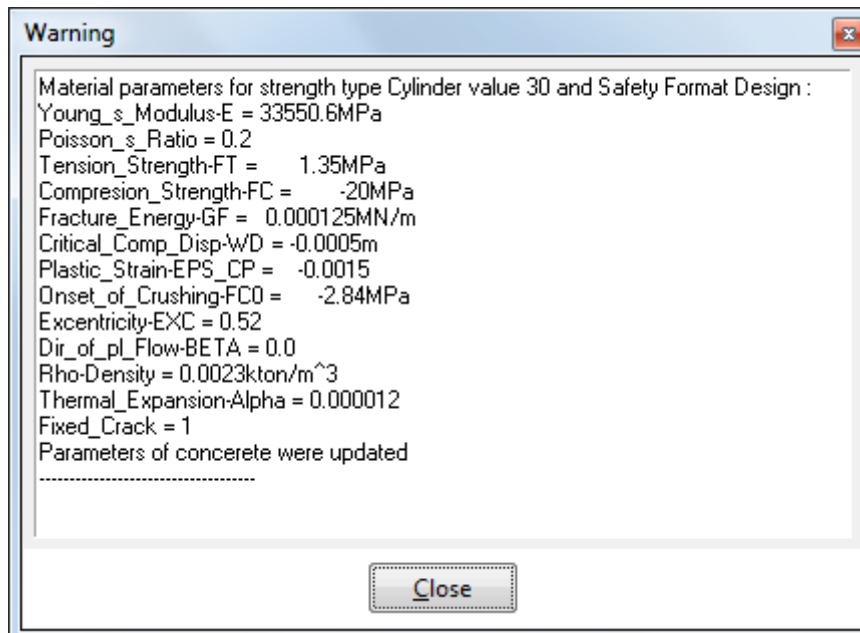


Fig. 5-34: Concrete EC2 – Generated values

The material prototype list box from the **Basic** tab (Fig. 5-35) 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 F_t , and the cylinder compressive strength F_c .

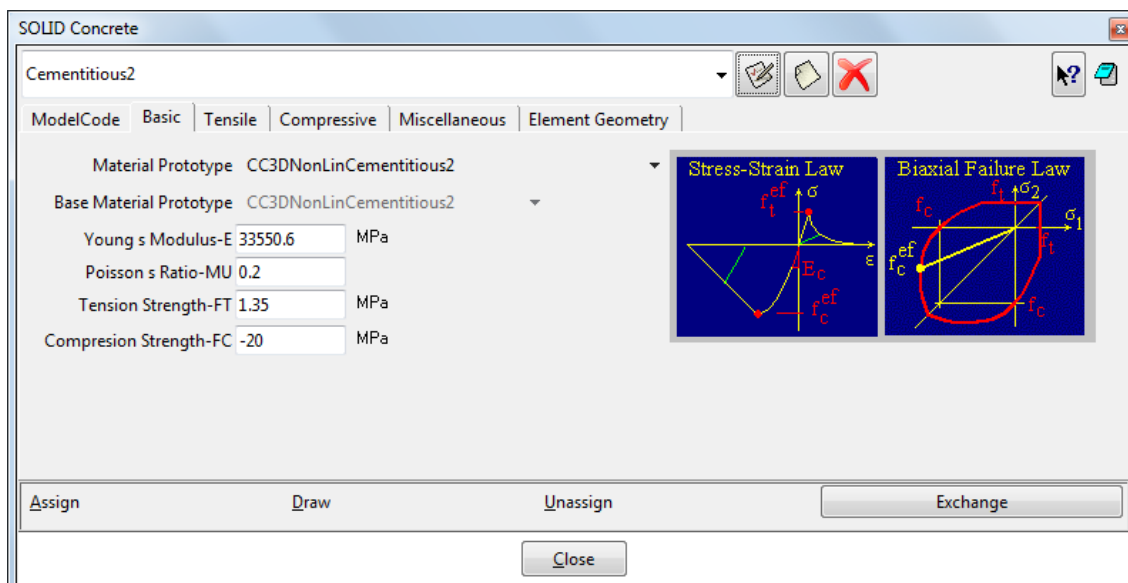


Fig. 5-35: Cementitious2 – Basic

The advanced parameters related to tension are defined at the **Tensile** tab (Fig. 5-36): Fracture energy G_f , Fixed Crack coefficient (0 = rotated, 1 = fixed, more details you can find in ATENA Theory in section"2.1.6 Two Models of Smearred Cracks"), Crack Spacing, Tension Stiffening, Aggregate Interlock, manual definition of Shear Factor,

and Unloading Factor (0 = the default unloading to the 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].

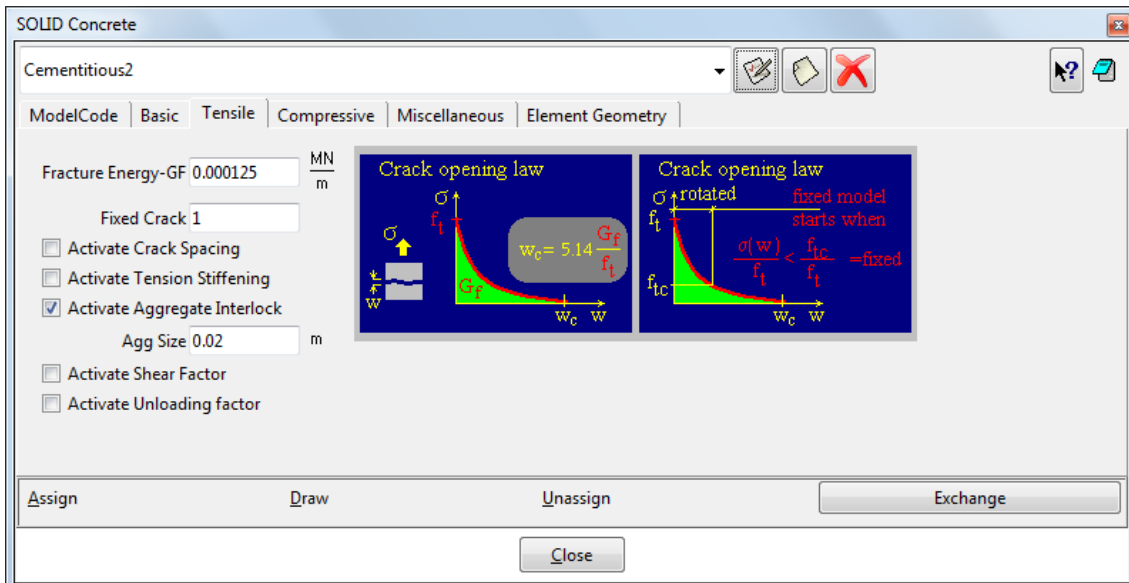


Fig. 5-36: Cementitious2 – Tensile

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., f_t 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. 5-37):

Plastic Strain at peak load `eps_cp` (ϵ_c^p) – this corresponds to the total strain at F_c from which the elastic part should be subtracted.

Onset of Crushing `Fc0` represents the linearity limit.

Critical Compressive Displacement w_d is the crushing compressive displacement after which the compressive stress drops to zero.

The relative limit for reduction of compressive strength due to cracking **Fc Reduction** controls the limit of f_c reduction due to cracking.

The checkbox **Activate Crush Band Min** is used to specify the minimal value of the crush band size for compressive concrete crushing. Normally crush band size is calculated as a projection of the finite element size in the direction of the highest compressive stress. If this value is defined, the crush band size is limited to the provided value, i.e., if the calculated crush band size is smaller than the limiting value, the minimal values are used. It is recommended to define it as the smallest dimension of the compressive element. See ATENA Theory manual [1] chapter on Cementitious2 material for more details.

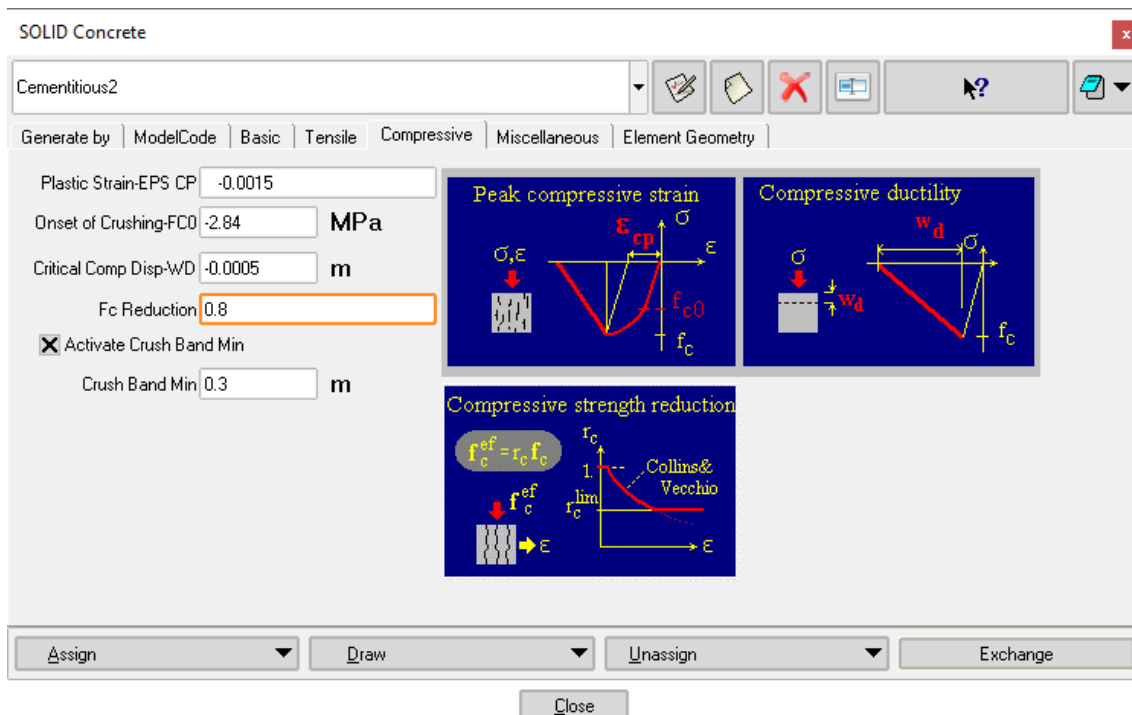


Fig. 5-37: Cementitious2 – Compressive

The **Miscellaneous** tab (Fig. 5-38) 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 ($\text{Beta} < 0$) or expansion ($\text{Beta} > 0$) during crushing, i.e., plasticization, and two general parameters: **Density Rho** (only used in dynamic analysis) and the coefficient of **Thermal Expansion Alpha** (only used when the thermal load is applied).

The settings at the **Element Geometry** tab (Fig. 5-39) 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**). **Idealization** has to be set corresponding to the type of the analysis (3-dimensional, 2-dimensional plane stress or plane strain, rotational

symmetry). For rotational symmetry you need to check `Axi_symmetric_task`. (see 5.5 Problem Data) 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.

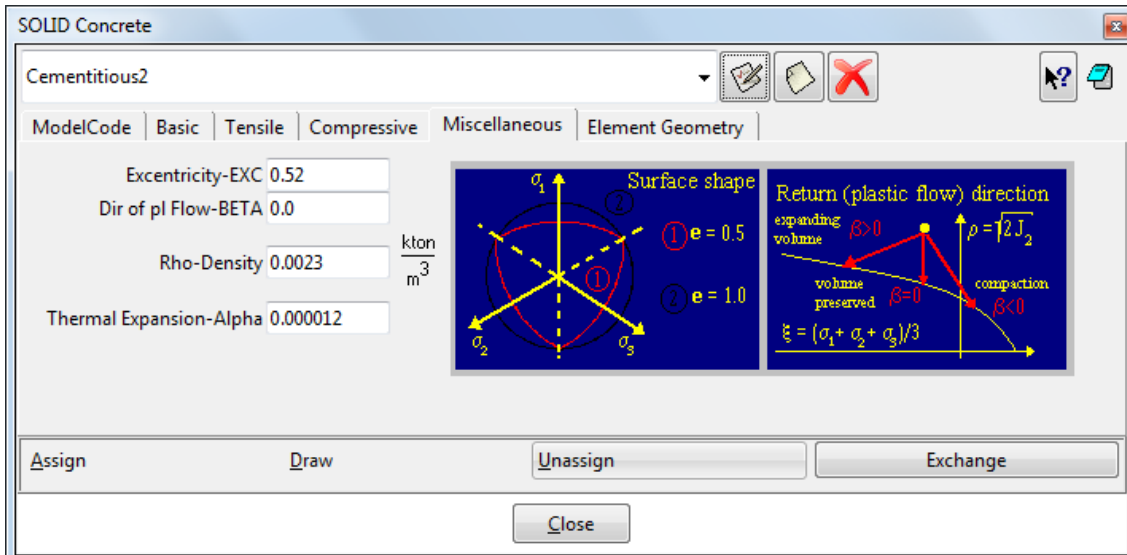


Fig. 5-38: Cementitious2 – Miscellaneous

Rho-density is used in dynamic and eigenvalue analyses. Not used for self-weight. For self-weight, use **Weight** in Conditions definition.

IGNORE_NEGATIVE_JACOBIAN: By default, if an element at an IP has negative jacobian, (i.e., there is something wrong with its geometry), then the appropriate exception is thrown and the execution is terminated. If this keyword is specified, the negative value of the jacobian is ignored and the execution continues (although with disputable results).

IGNORE_ELEMENT_TYPE_EXCEPTION: By default, if an exception occurs within element calculation, then the appropriate exception is thrown and the execution is terminated. If this keyword is specified, execution of the offending element is skipped and remaining elements get proceeded.

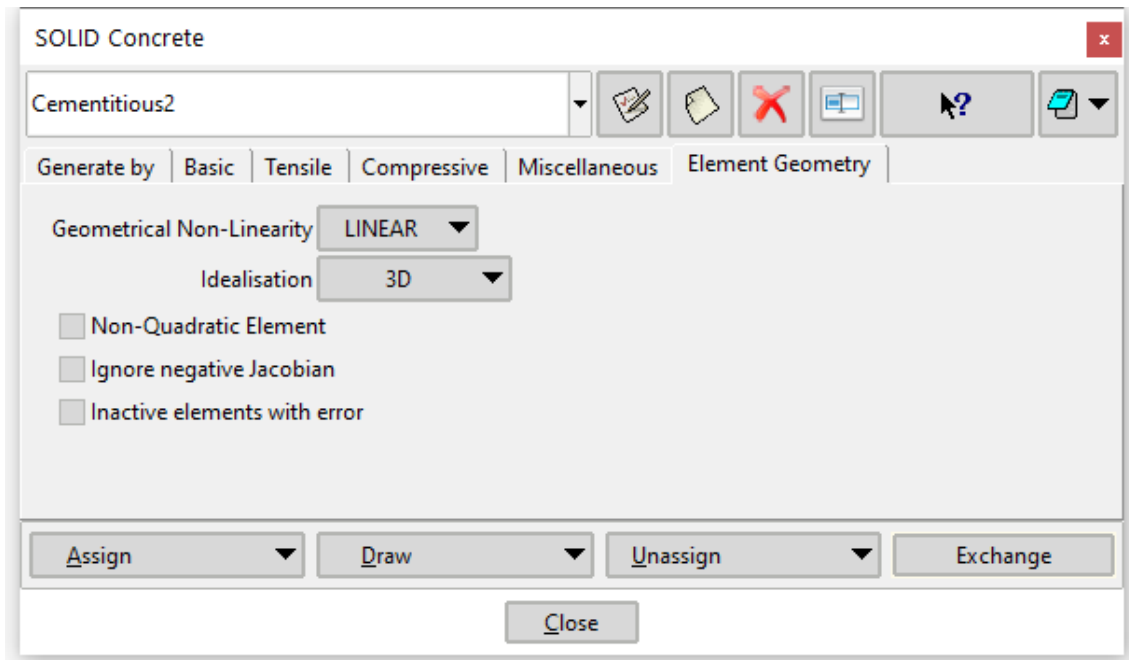



Fig. 5-39: Cementitious2 – Element Geometry

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. 5-40). The generated values are displayed in a window (Fig. 5-41). 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.

Generation is according the EuroCode2. See 3.1 chapter Material – Concrete.

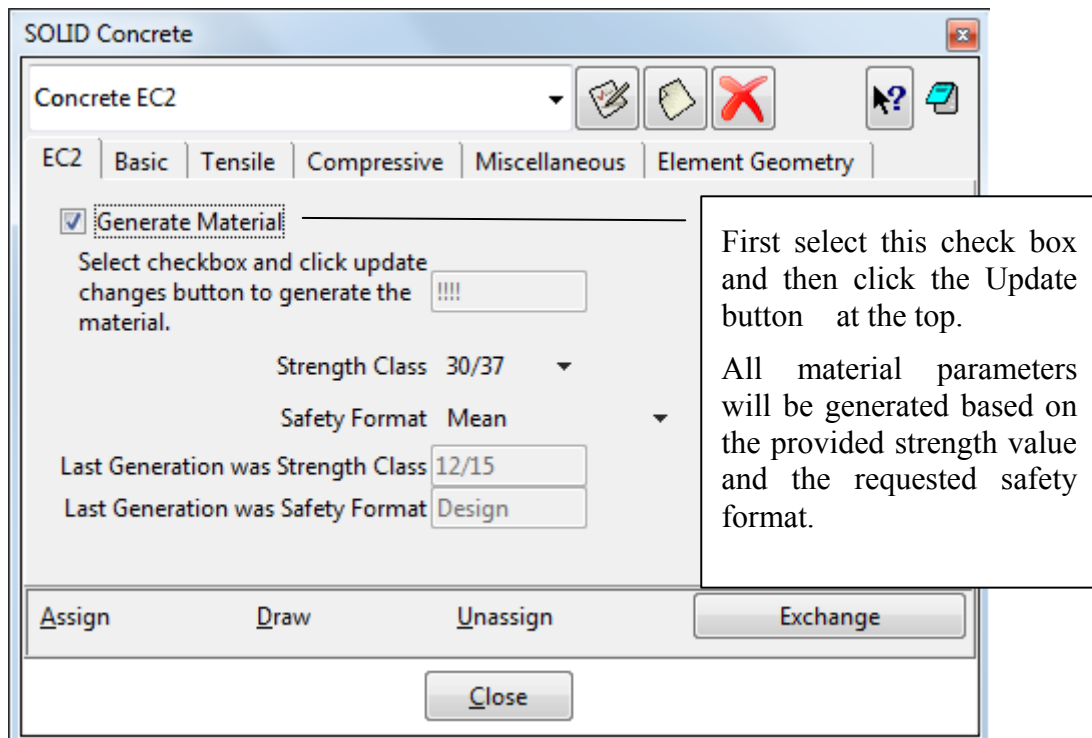


Fig. 5-40: Concrete EC2 – Generation parameters

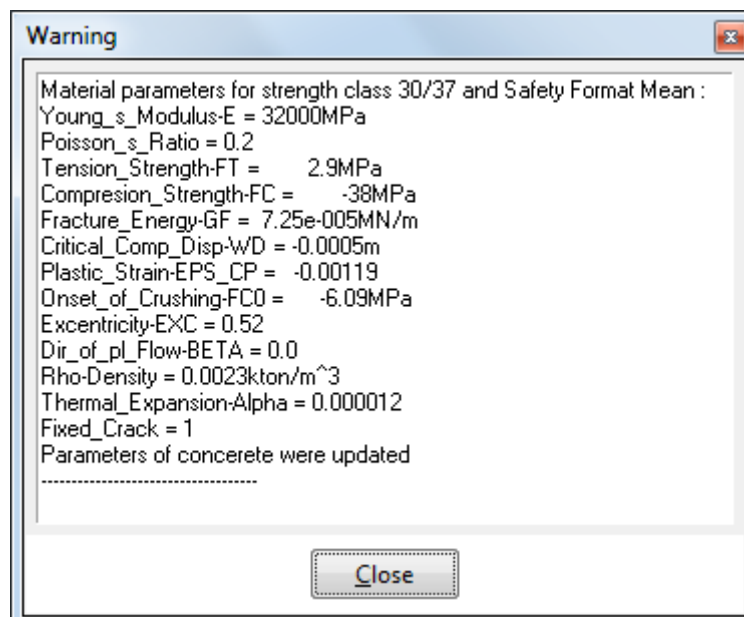


Fig. 5-41: Concrete EC2 – Generated values

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. 5-42):

- 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.

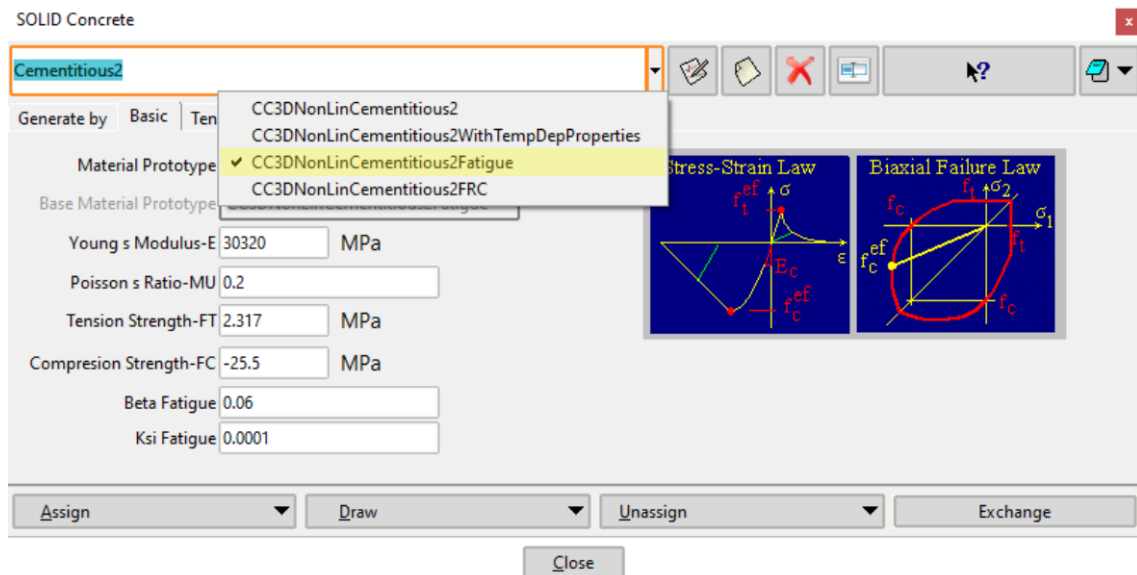


Fig. 5-42: Cementitious2 Fatigue – Basic

5.3.1.4 CC3DNonLinCementitious2FRC

Another material prototype available is CC3DNonLinCementitious2FRC. This material is based on CC3DNonLinCementitious2 and allows modification of the tensile softening law by adding additional fracture energy to the basic concrete material and thus simulating the increased residual strength and ductility typical for fiber reinforced mixtures. A more detailed description of the model can be found in the ATENA Theory manual.

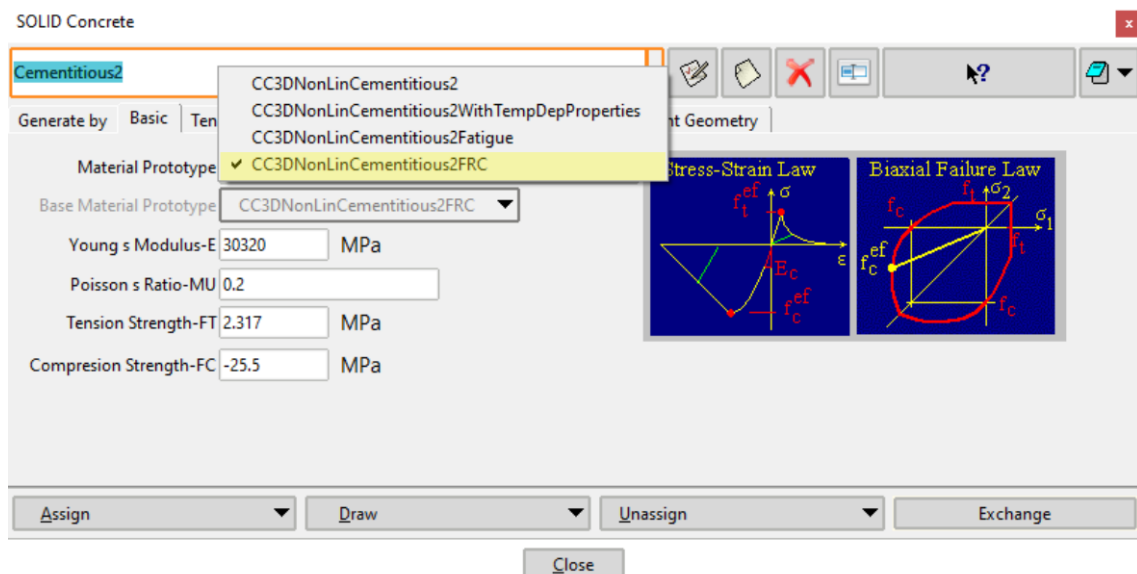


Fig. 5-43: Cementitious2 FRC – Basic

The material prototype is selected in the **Basic** tab, as shown in Fig. 5-43. Once CC3DNonLinCementitious2FRC is selected, a new tab called **Tensile FRC** appears. The added fracture energy is specified based on two parameters; the value of the added fracture energy and the maximum crack opening width of the FRC. A tool is available for the automatic generation of the parameters for typical concretes and fiber types.

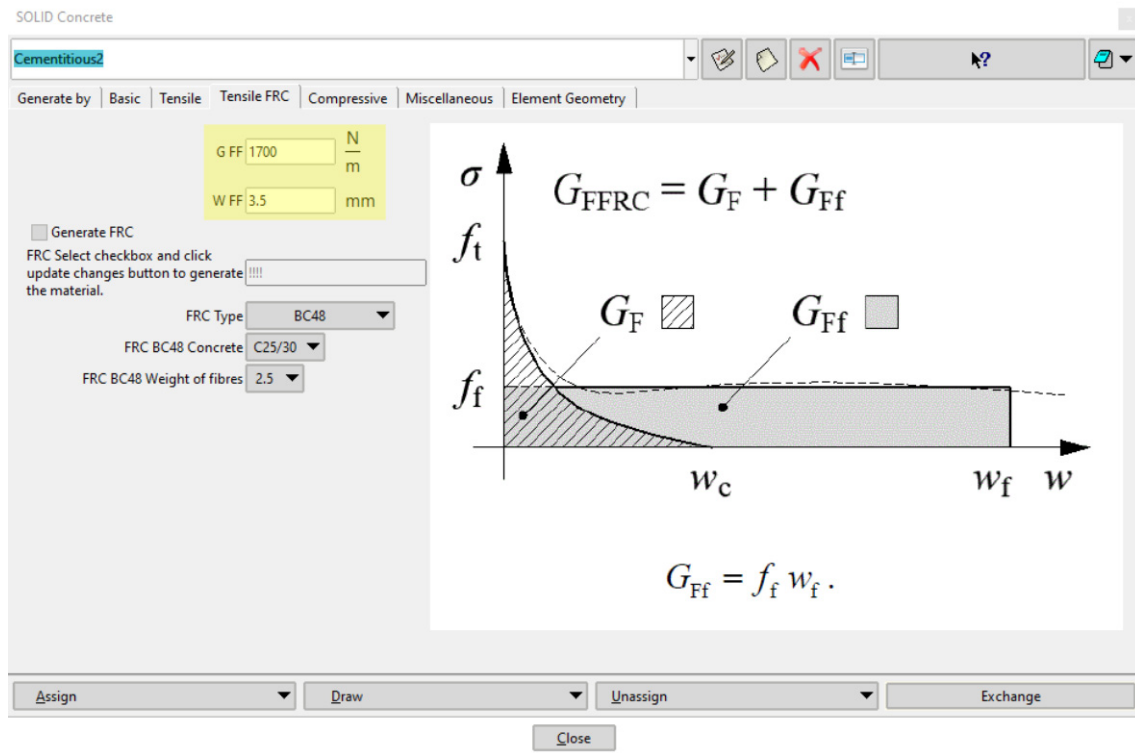


Fig. 5-44: Cementitious2 FRC – Tensile softening law

5.3.1.5 Cementitious2 User (FRC)

Cementitious material with a user-defined response functions. This material is especially useful for modeling fiber reinforced concrete materials (FRC). Unlike for CC3DNonLinCementitious2FRC, more tailor behavior can be achieved for Cementitious2 User material by adjusting multiple tensile characteristics and concrete softening behavior. 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. 5-45), **Compressive** (Fig. 5-46), **Shear** (Fig. 5-47), and **Tension-Compressive** (Fig. 5-48), the corresponding user functions and the localization parameters are to be defined. For instructions on 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.

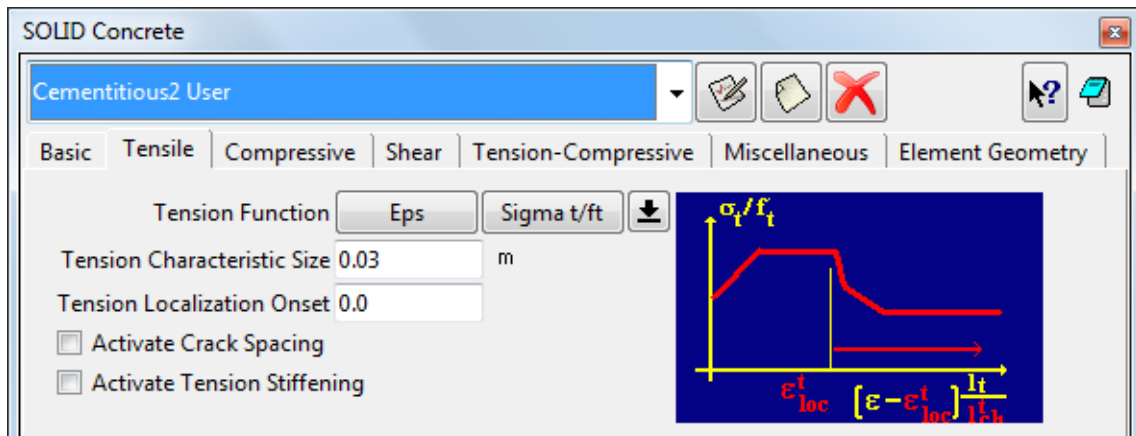


Fig. 5-45: Cementitious2 User – Tensile

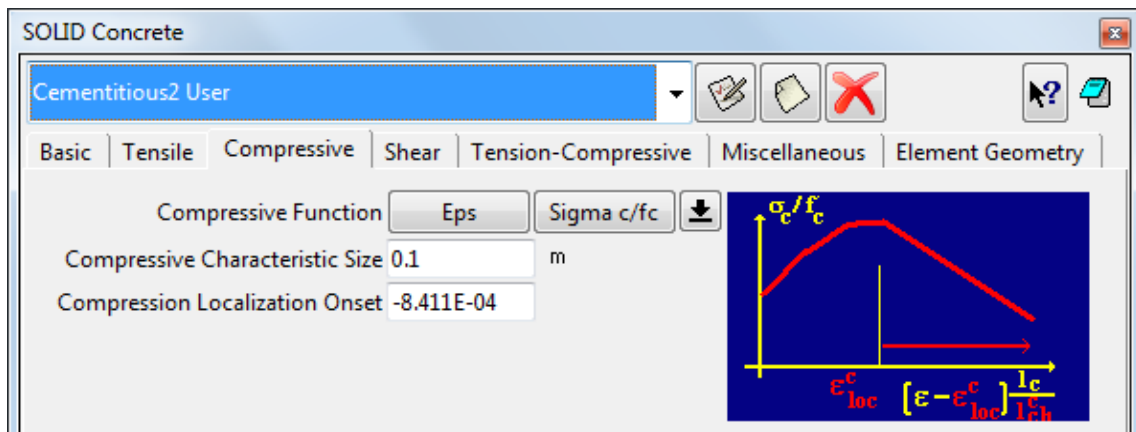


Fig. 5-46: Cementitious2 User – Compressive

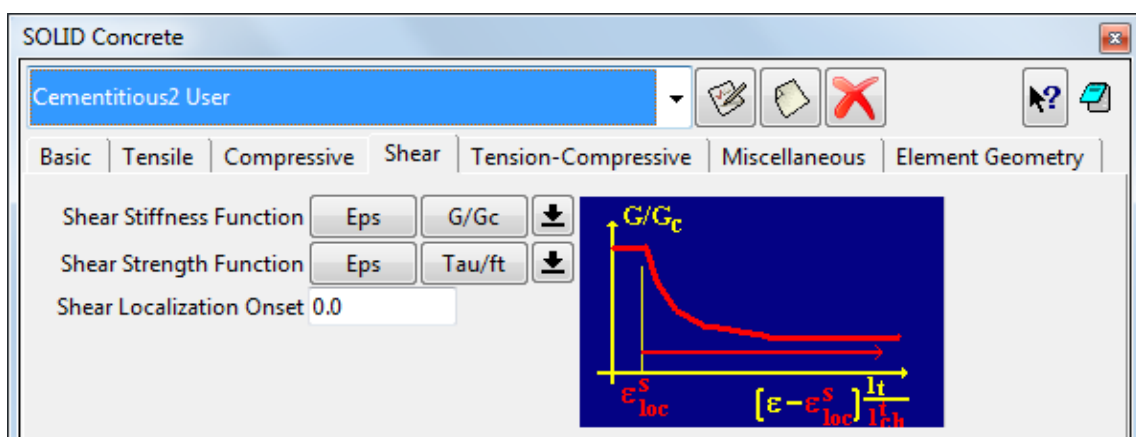


Fig. 5-47: Cementitious2 User – Shear

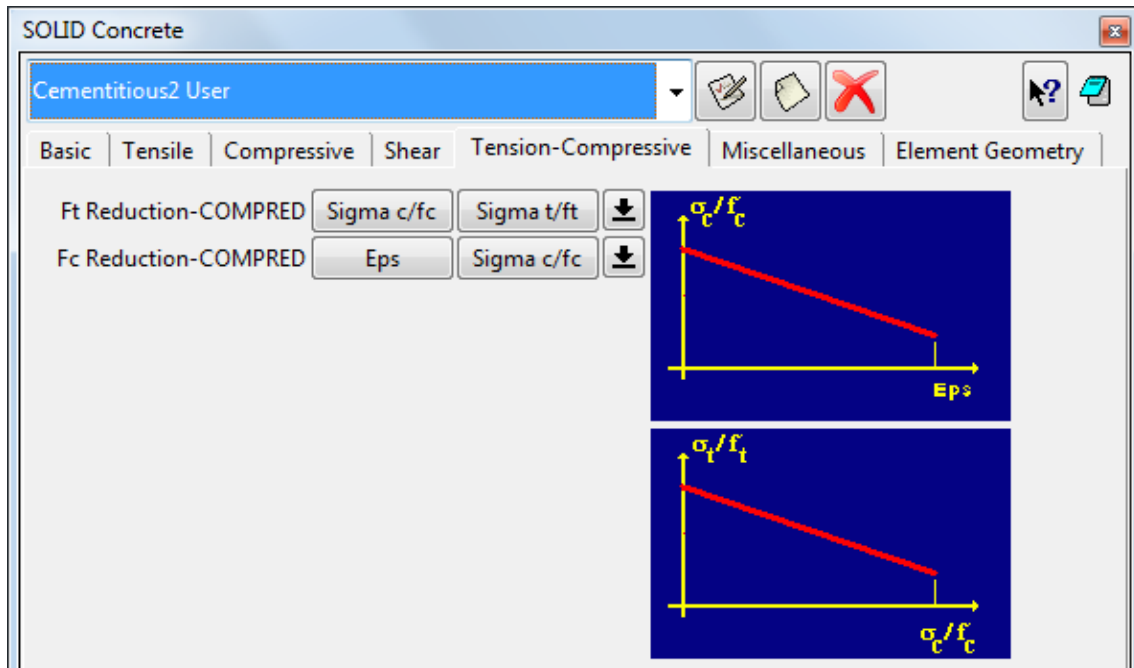


Fig. 5-48: Cementitious2 User – Tension-Compressive

5.3.1.6 Cementitious2 SHCC (HPFRC)

Cementitious2 SHCC is a special material for high-performance strain-hardening cementitious composites (e.g., special mixtures with addition of plastic fibers). The only difference from **Cementitious2 User** (5.3.1.5) is the **Fibre Reinforcement** tab replacing the **Shear** tab (Fig. 5-49). 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.

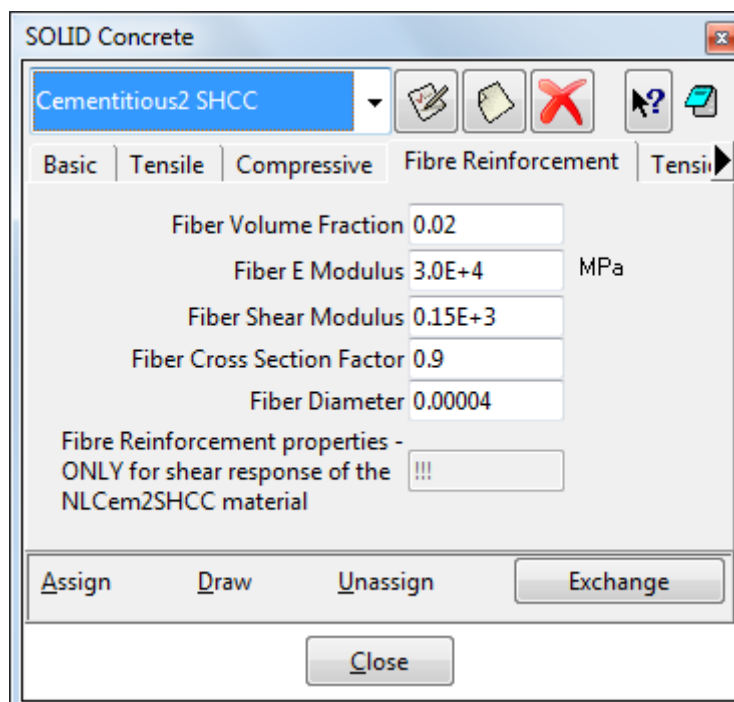


Fig. 5-49: Cementitious2 SHCC – Fibre Reinforcement

5.3.2 Shell Material

In this section, the shell material is described. ATENA supports modelling with either 2D or 3D shell elements, which are assigned to surfaces and volumes, respectively³. The quad and triangle geometry is supported for 2D shell elements, and brick and wedge geometry is supported in case of the 3D shells. The general modelling approach and material definition are the same for both types; however, the difference lays in the way in which the element thickness is defined. For the case of the 2D shells, the surface represents the center plane of the element and the thickness is specified as the part of the material definition (**Solid Ref Thick** at the **Base** tab). The thickness of the 3D shell elements is given by the volume they are assigned to. To improve the precision of the model, it is not recommended to place multiple layers of shell elements on top of each other (meaning in the direction of element thickness) but instead to increase the number of internal layers. To take the bending behavior correctly into account, at least 4 layers are recommended. Please note that it is crucial to correctly define the local z-axis in both cases (described in more details below).

The 3D shells have 3 translation DoFs and 3 corresponding nodal forces in each node while some DoFs are not free due to the introduction of kinematic constraints, ensuring shell displacement model. The 2D shells have 3 translation and 2 rotation DoFs with corresponding nodal forces and bending moment (along in-plane axes) in each node.

3D shell material has geometry which supports **Ahmad** elements (**CCA AhmadElement**) 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.

2D shell material supports **CCIsoShellQuad** and **CCIsoShellWedge** elements.

All Shell elements require a quadratic mesh (**Mesh | Quadratic Type | Quadratic**). If linear elements are to be used for other parts of your model, please see 5.3.2.3.

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 the 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. 5-50, Fig. 5-51). It is also recommended to set the local X direction such that the in-plane directions are continuous over neighboring elements. See Fig. 5-55 showing the corresponding dialog.

It is recommended to check the orientation of the elements through displaying the **Element Orientation** (in Element Nodes) when you run the analysis (or in post-processing).

³ In **ATENA-GiD**, the shell material has to be assigned to volumes or surfaces where shell (plate) elements are to be used (unlike **ATENA Engineering 3D**, where one switches between volume and shell elements in Macroelement definition).

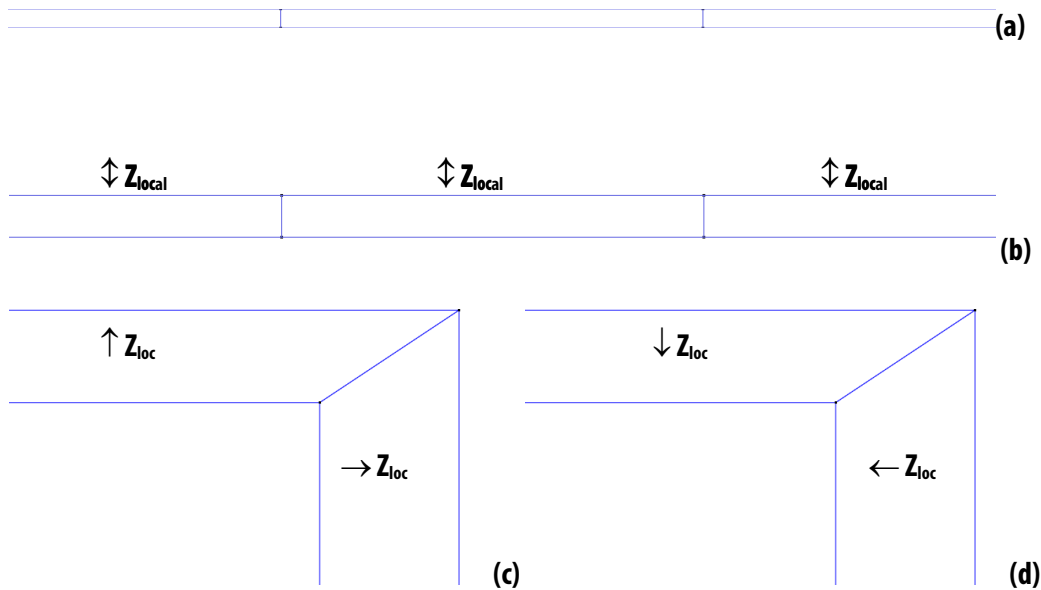


Fig. 5-50: Shell - recommended local Z orientation

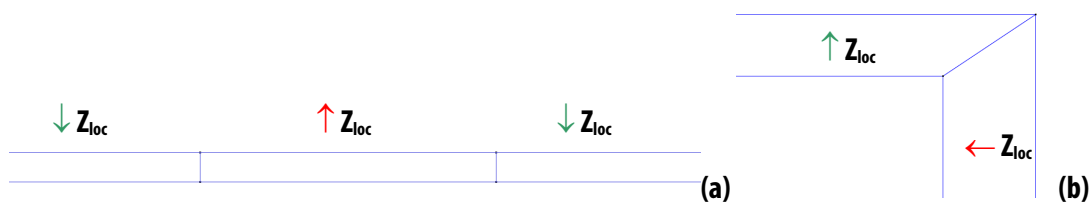


Fig. 5-51: Shell – problematic local Z orientation with orientation jumps

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 constraints ensuring shell displacement model. For more details, see Theory Manual [1].

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. 5-52 (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. 5-52 (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 behavior. Connecting like in Fig. 5-53 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. 5-53. However, connections from Fig. 5-52 can also be used.

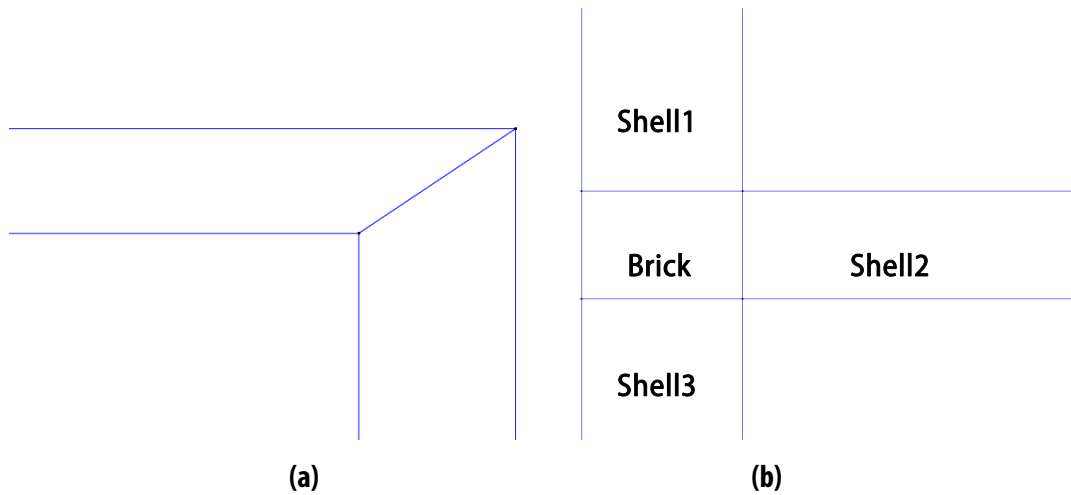


Fig. 5-52: Shell - recommended connection (a) 2 shells (b) 3 shells

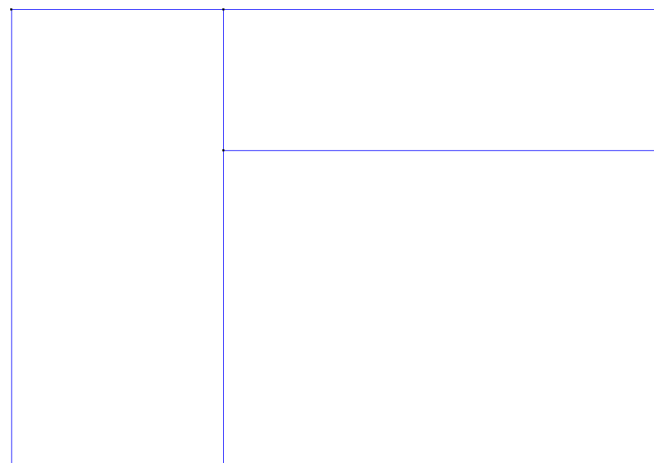


Fig. 5-53: Shell - recommended connection for Iso, not recommended for Ahmad

The **ATENA** implementation of the **Ahmad, IsoBrick/Wedge** and **IsoQuad/Triangle** 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.

When defining the distance of the reinforcement from the top or bottom surface, please note the distance is translated into relative position along the shell thickness based on the **Solid Ref Thick** value.

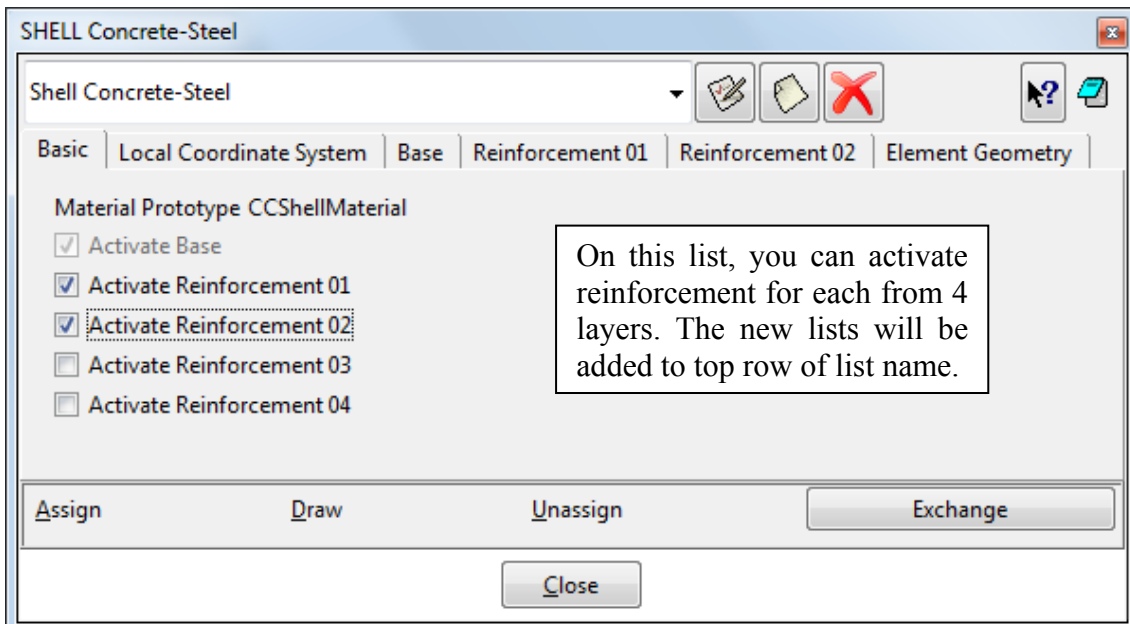


Fig. 5-54: Shell material properties - Basic

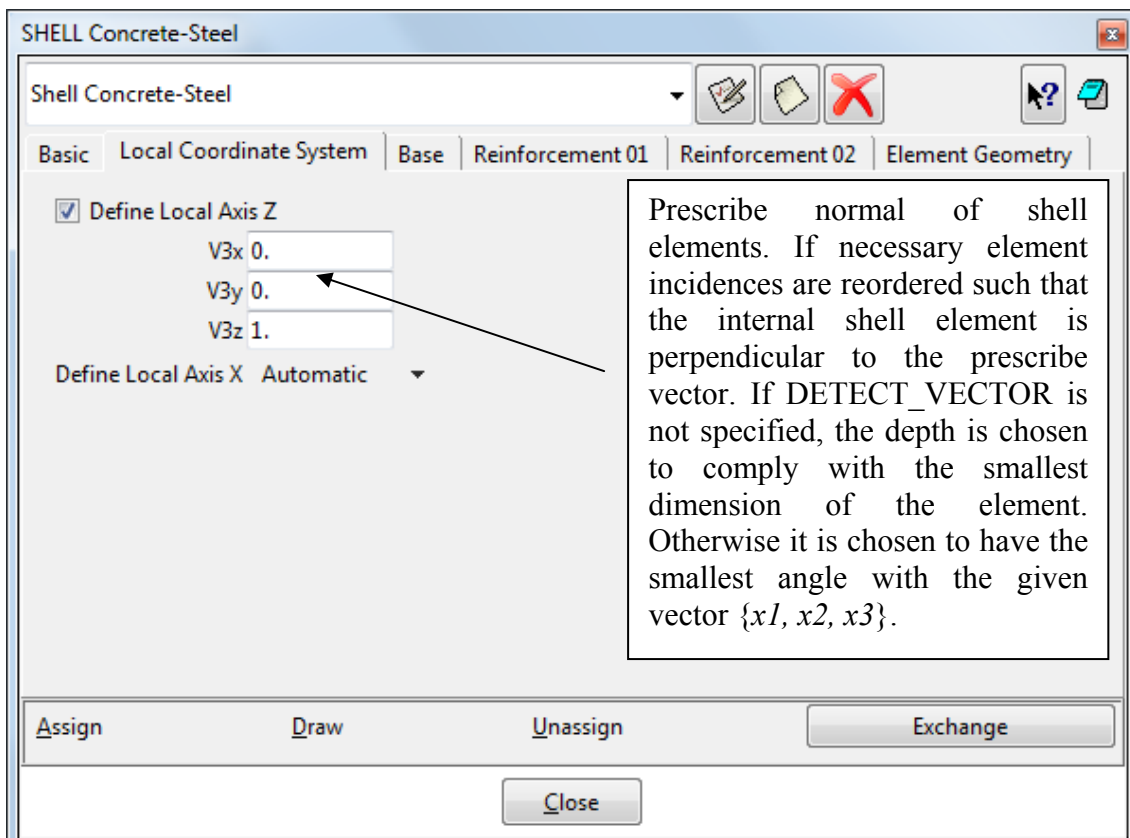


Fig. 5-55: Shell material properties – Local Coordinate System

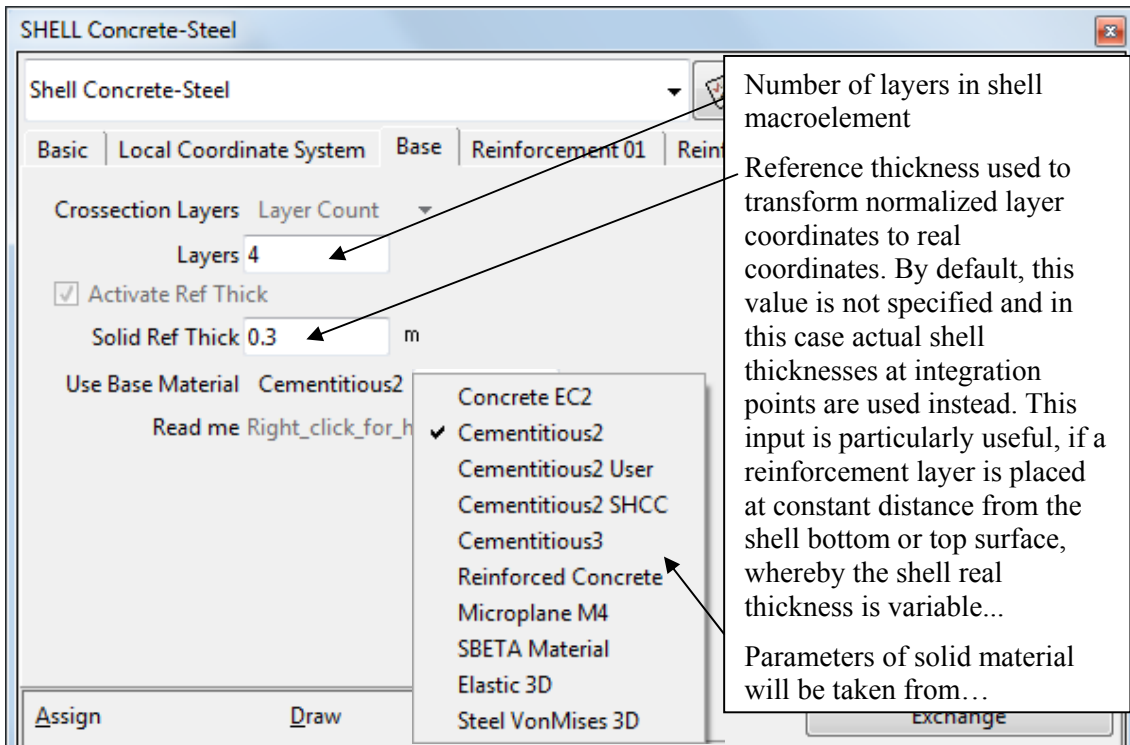


Fig. 5-56: Shell material properties - Base

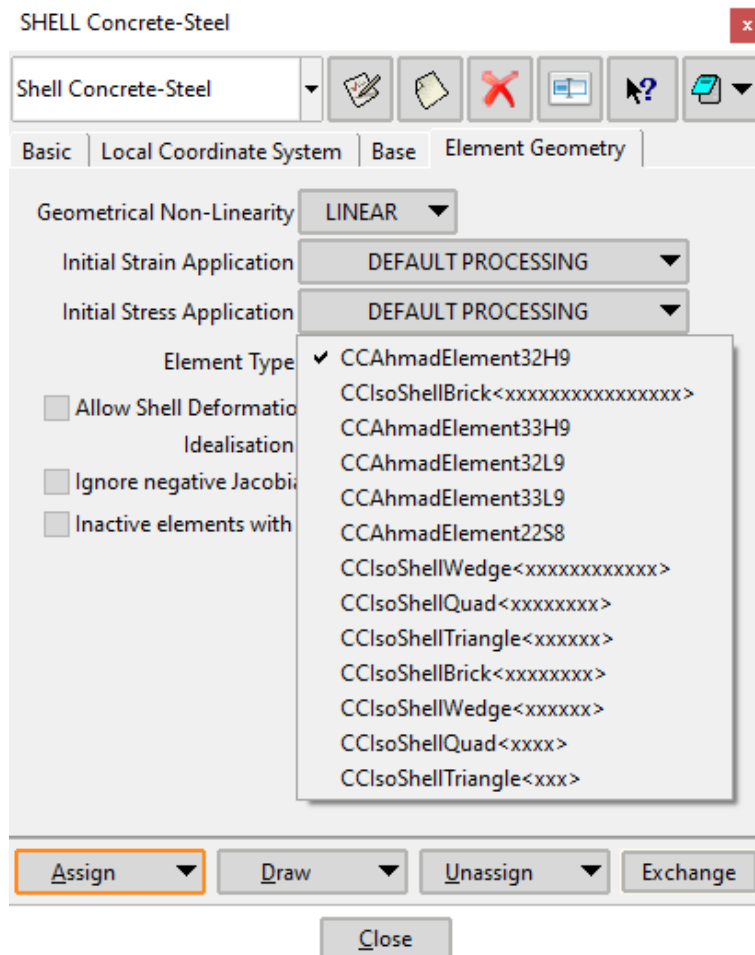
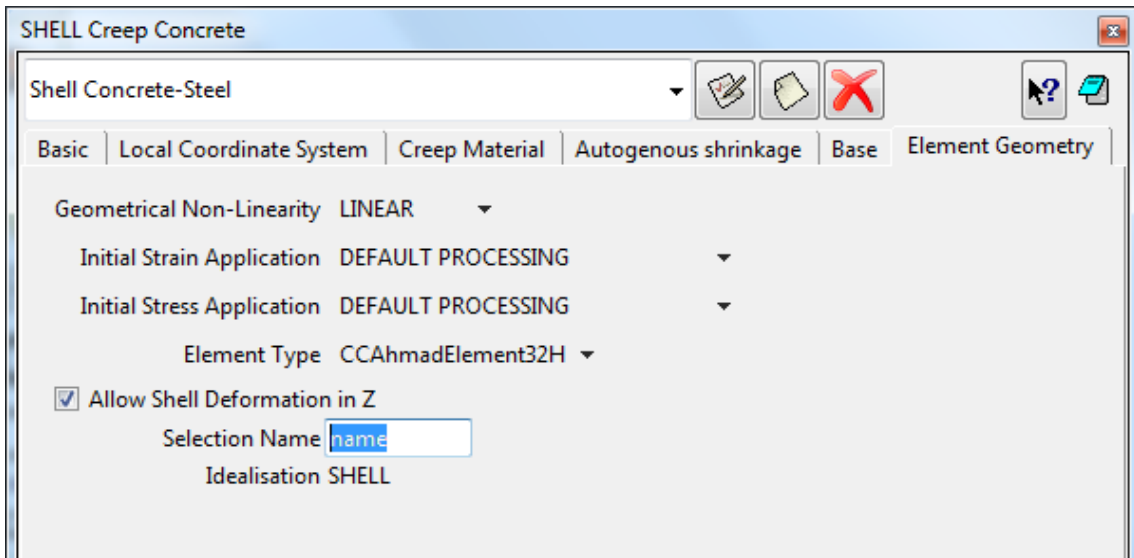


Fig. 5-57: Shell material properties – Element Geometry and Element Type

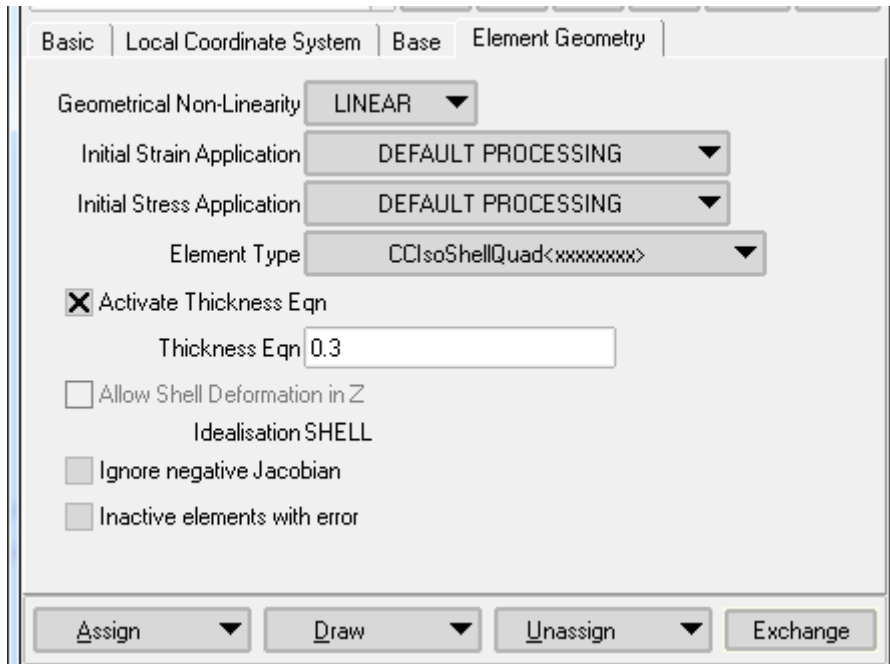


Fig. 5-58: Shell material properties – Thickness Specification in Case of 2D Shells

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: For 3D **Ahmad** shell elements, the first and the second digit in the element name specifies the 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 LayeredShell 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 degrees 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. For 2D elements is necessary to set the thickness of elements. If Activate_thickness_Eqn is unchecked, Solid_Ref_thick is used. If you want to add user-defined equations, you can by checking this checkbox.

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 **Ahmad** shell elements with normal solid elements, otherwise, the shell elements may restrain the deformation of the surrounding solid elements.

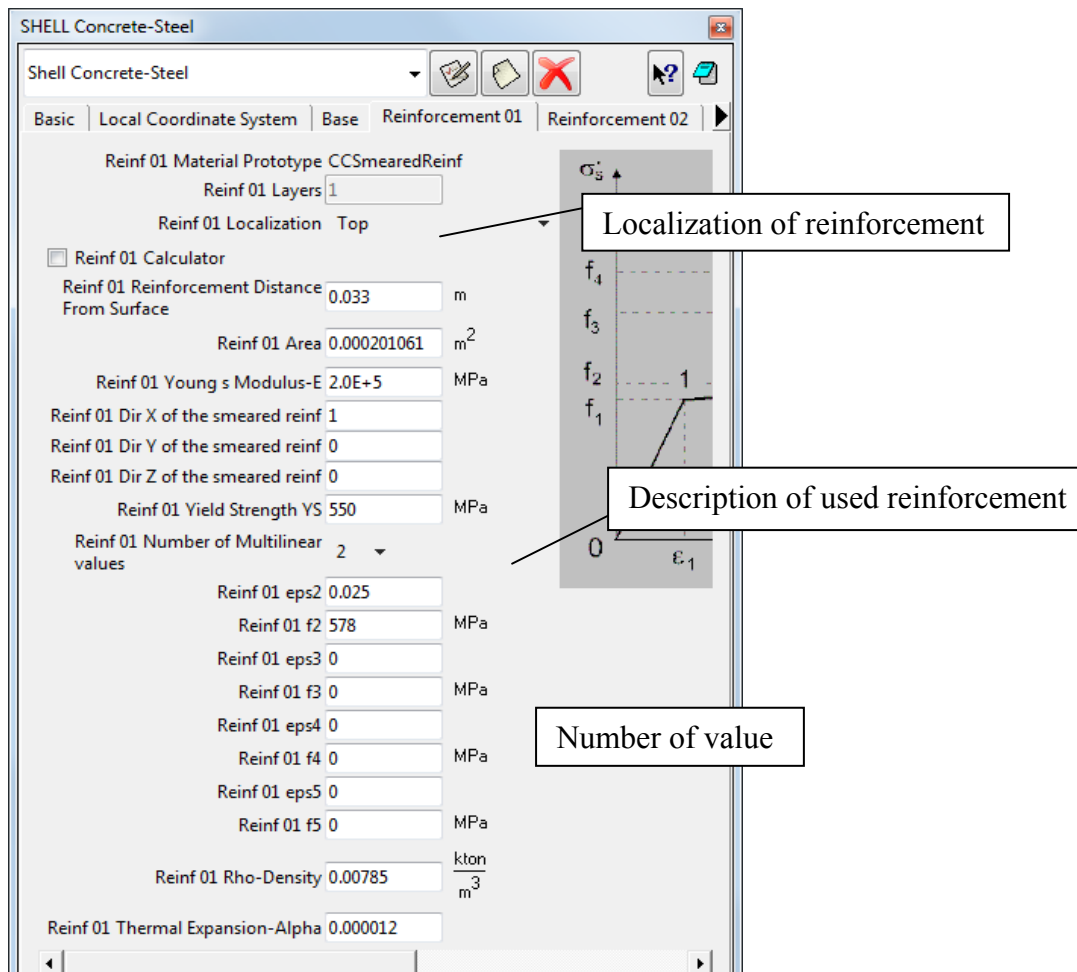


Fig. 5-59: Shell material properties – Reinforcement – detail

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. This condition is only needed for Shell with CCAhmadElements element type. For CCIso Shells, it is not necessary.

It is recommended to apply this condition to all shell "side" surfaces that 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.

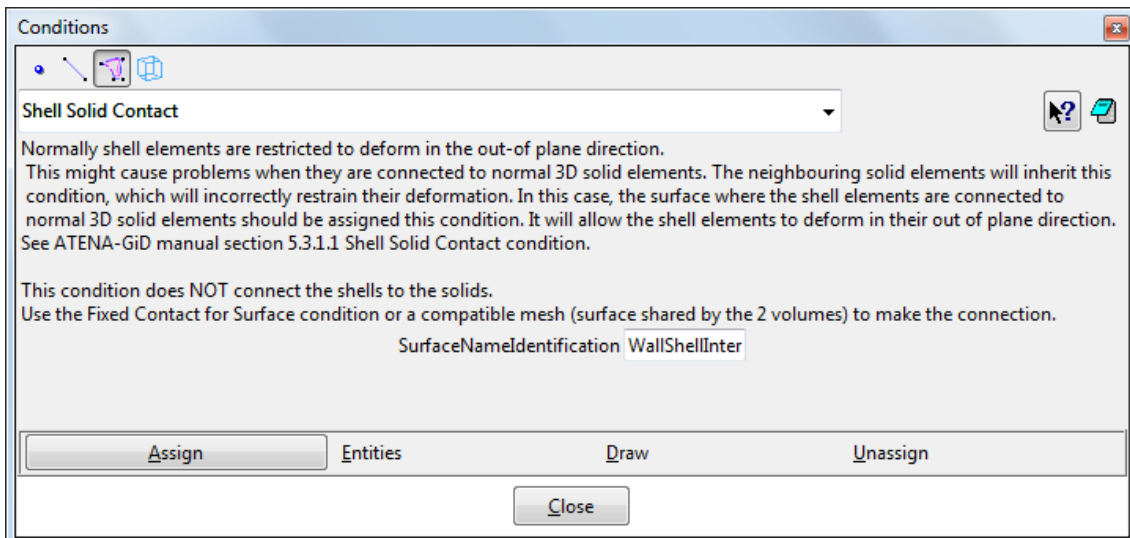


Fig. 5-60: 'Shell Solid Contact' condition for Ahmad elements

5.3.2.2 Connecting shells to other parts of the model

In the case of 3D shells, the elements can be connected using the master-slave conditions (**Fixed Contact for...**) since both 3D shells and 3D solids have only translation DoFs in the element nodes. The same approach will work for connecting 2D shells to 3D solids for a fixed connection. However, if a 2D shell needs to be connected to the surface of a 3D solid with a pinned connection, the user should define the connection through **Fixed 2D Shell to Solid for Surface** while releasing the desired rotation DoF. In case that the connection between the 2D surface and the 3D solid is in a single point or along a line, the proper variation of the **Fixed 2D Shell to Solid for ...** should be chosen.

For more details about how to connect 2D shells to other components of the model, please refer to chapter 5.2 and also check the examples for modeling in with 2D shells in the GiD Example Manual [8]. See also the description of connecting beam elements in section 5.3.3.1.

5.3.2.3 'Non-Quadratic Element' option

When combining "normal" (volume) elements with shells or beams in a model, it is often advantageous to only use quadratic geometry and approximation for the shells (using it for all elements in the model typically brings an increase in the memory and time consumption, which is not proportional to the improvement in result quality). Therefore, **ATENA-GiD** offers the option to make some elements linear using the checkbox **Non-Quadratic Element** at the **Element Geometry** tab of the **Material** dialog, see the paragraph about **Element Geometry** in 5.3.1.1.1 for more information. See also 5.8 for general information about meshing and element types.

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). Analogically to the shell elements, there are 1D elements, which can be assigned to lines, and 3D beam elements, which can be assigned to the

volumes. In the case of 1D beam elements, which are assigned to lines, the line represents the centerline of the 1D beam element, and the cross-section dimensions are defined by the values of **Beam Ref Size t** (height) and **Beam Ref Size s** (depth) at the **Base** tab. In case that the 3D beam elements are assigned to a volume, the geometry of the volume defines the overall dimensions of the structural member. For both cases, the orientations of the x-axis and z-axis need to be specified in the **Local Coordinate System** window, as shown in Fig. 5-62. The x-axis should run along the length of the modeled member, which in the case of a 1D beam, is the orientation of the line and, in case of 3D beams, the principal direction of the member. The z-axis should correspond to the height of the member.

The basic settings like activating smeared reinforcement (Fig. 5-61), and defining the local coordinate system (Fig. 5-62) are very similar to the shells (described in the previous section 5.3.2).

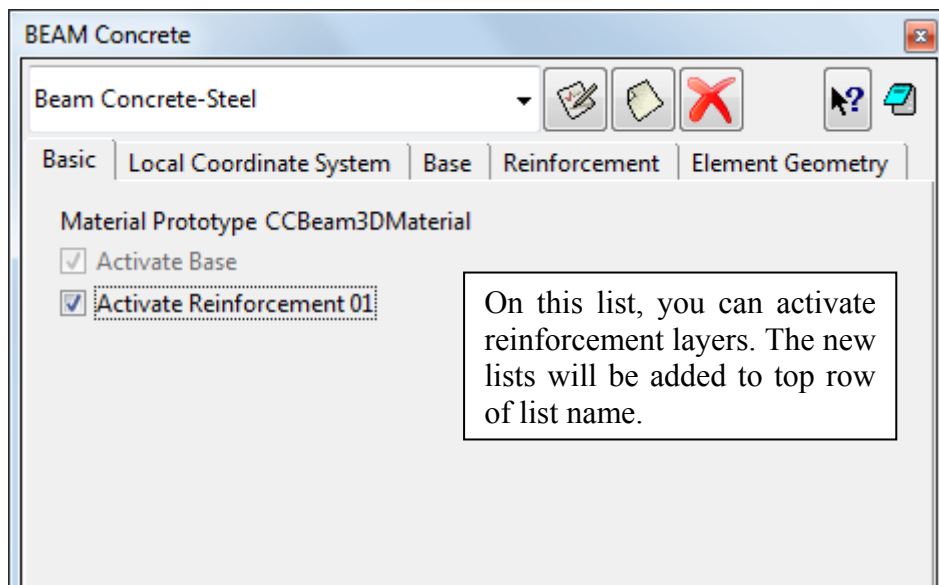


Fig. 5-61: Beam material properties – Basic

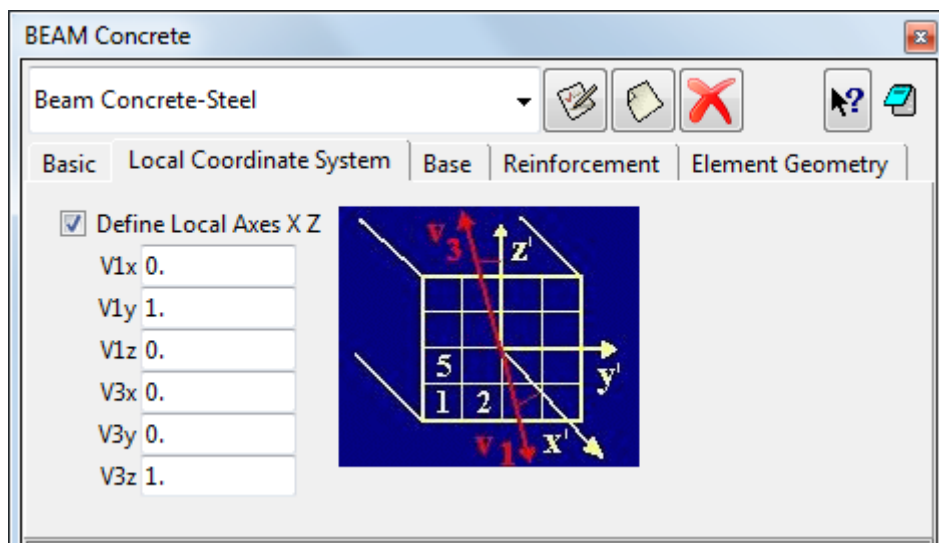


Fig. 5-62: Beam material properties – Local Coordinate System

The numbers of material layers over the height and depth of the cross-section are specified at the Base tab. It can also be set, which of these layers represent active materials (where the material is present), and which represent voids (Fig. 5-63). In this way, various cross-sections can be modeled, including common I-shaped, H-shaped, and U-shaped profiles.

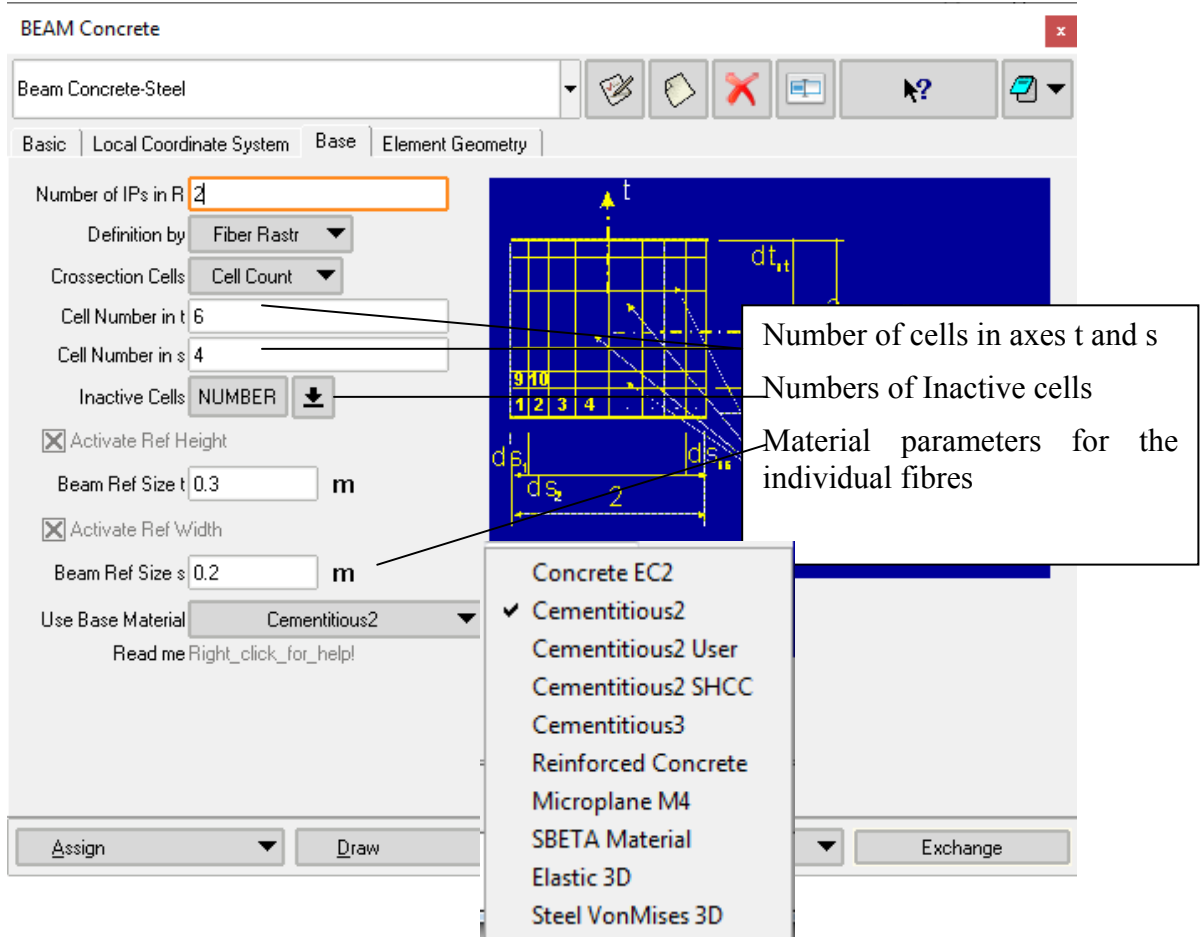


Fig. 5-63: Beam material properties – Base – Fiber Rastr

The definition of the smeared reinforcement (Fig. 5-65) and the geometry properties is also very similar to the definitions in the shell elements (see Section 5.3.2). It should be noted that this element also enables the definition of integration order along the beam length. It is however, recommended to use 2 integration points, i.e., the parameter **Number of Ips in R** is recommended to be 2. For lower integration points, zero energy modes can be obtained. On the contrary for more integration points than 2 oscillatory shear stresses and forces can be observed along the beam length.

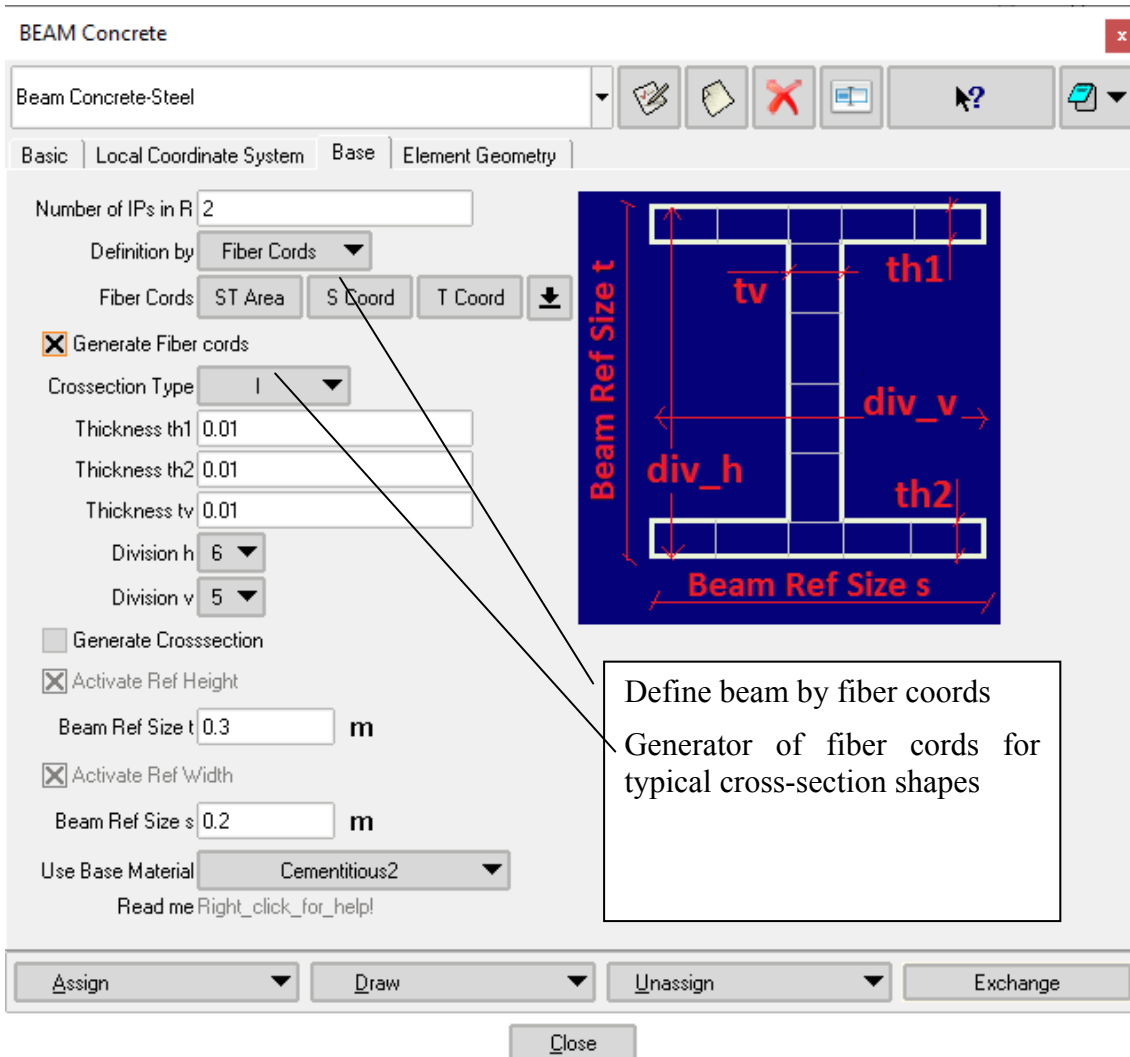


Fig. 5-64: Beam material properties – Base – Fiber Raster

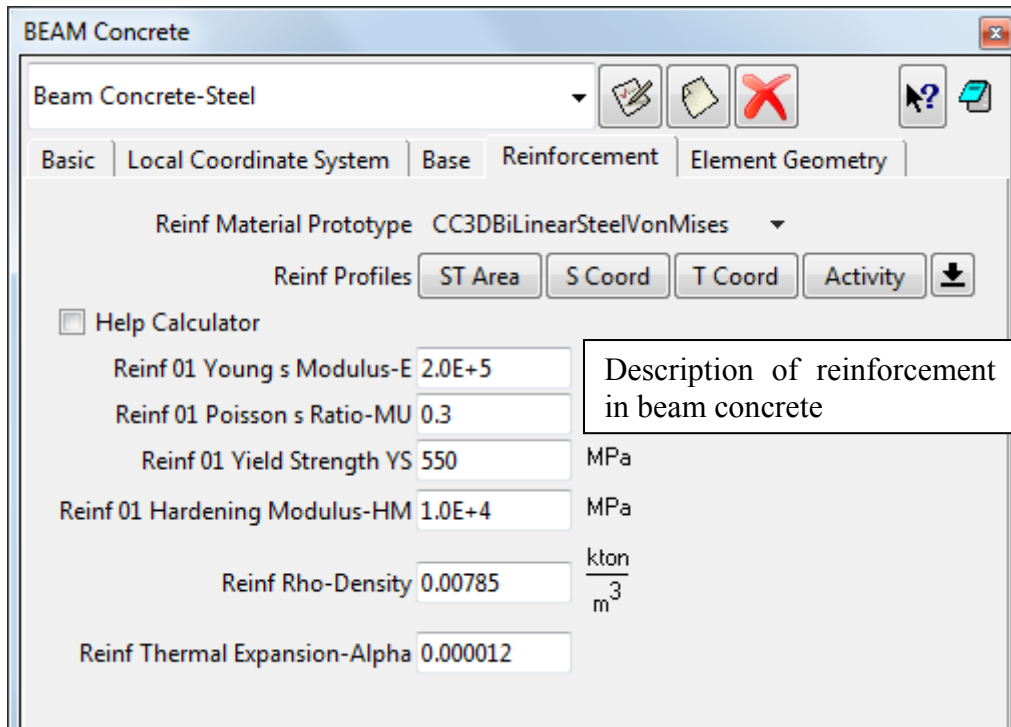


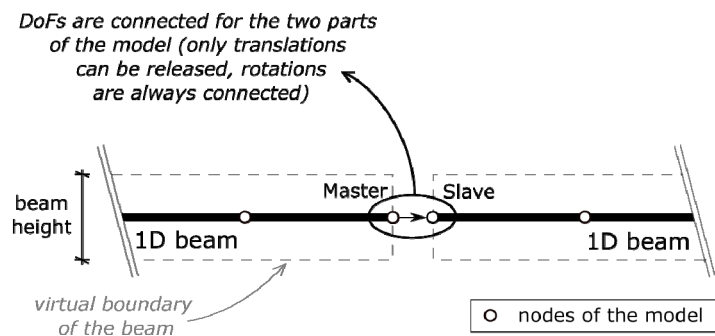
Fig. 5-65: Beam material properties – Reinforcement

5.3.3.1 Connecting beams to other parts of the model

Similar to the methods for connecting 2D shells (as described in section 5.3.2.2), there are two methods for connecting a 1D beam element to other parts of the model. Fixed connections or connections, where only translation DoF needs to be released, should be done through applying **Fixed Contact for Point** condition. In case of a pinned connection, which can be either between two 1D beams (i.e., two separate lines without a common point) or for 1D beam element connected to the surface of a volume, the rotational DoFs can be released if **Fixed 1D Beam to Solid for Point** condition is used. **Fixed 1D Beam to Solid for...** condition creates additional connection points between the two parts of the model (Fig. 5-66) and the user can define that only a part of the cross-sectional area (e.g., only one quarter or one half) is connected.

Please see section 5.2 for more details, how to specify the conditions and also the examples in our GiD example manual [8].

CONNECTING USING "Fixed Contact for Point"



CONNECTING USING "Fixed 1D Beam to Solid for Point"

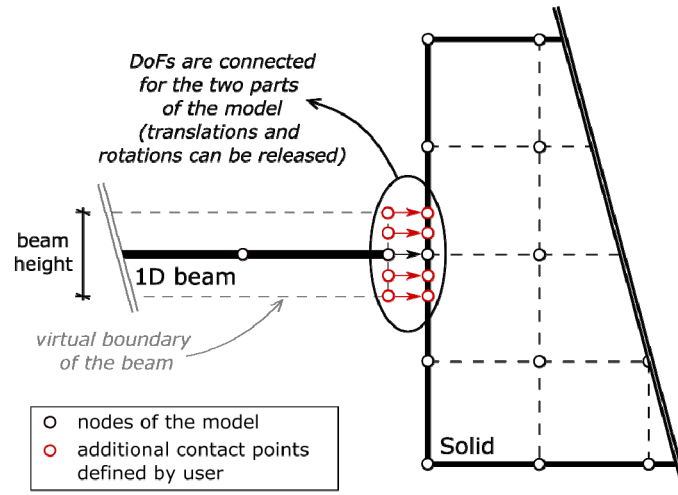


Fig. 5-66: Beam material properties – methods for connecting 1D beam elements

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. 5-67, Fig. 5-68) are very similar to the shells (described in section 5.3.2).

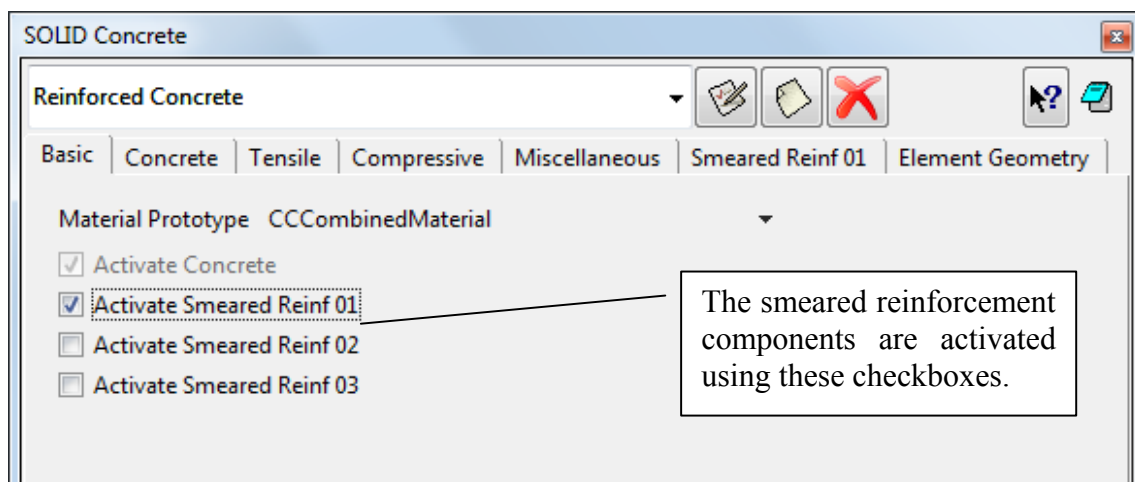


Fig. 5-67: Reinforced Concrete material properties – Basic

The tabs with the concrete properties (**basic** Fig. 5-35, **tension** Fig. 5-36, **compression** Fig. 5-37, **miscellaneous** Fig. 5-38) and geometry settings are identical to the Cementitious2 material (5.3.1.1).

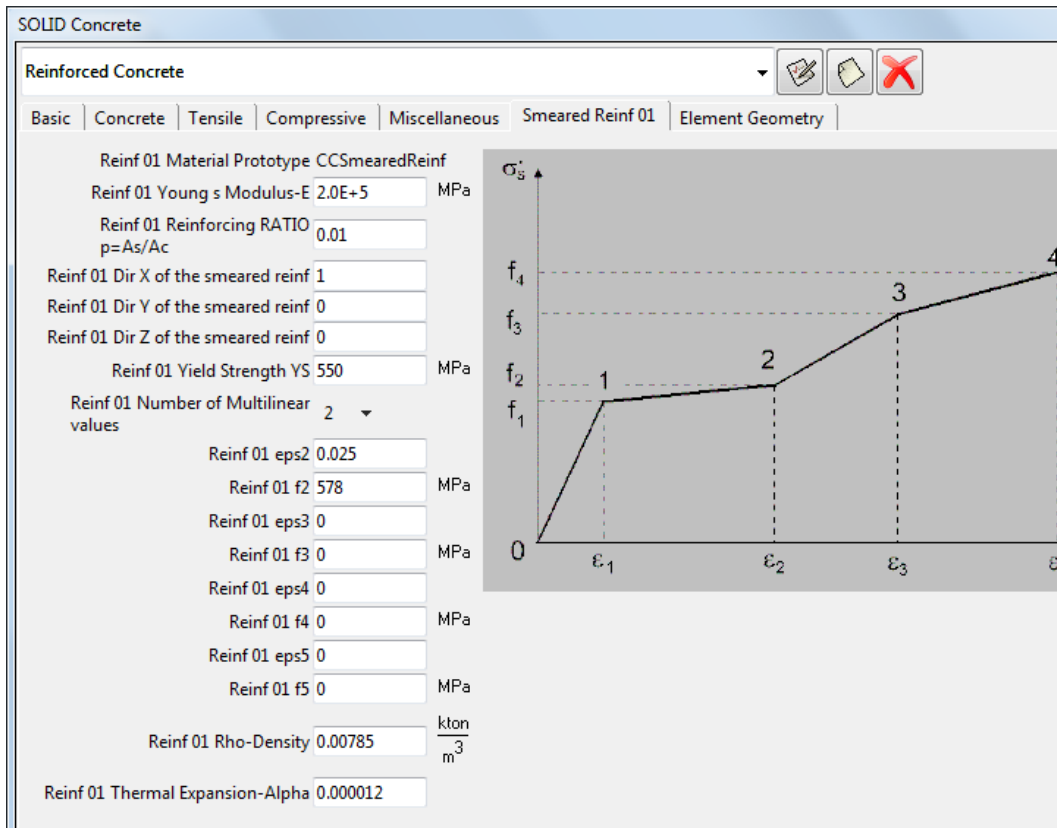


Fig. 5-68: Reinforced Concrete material properties – Smeared Reinforcement

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.

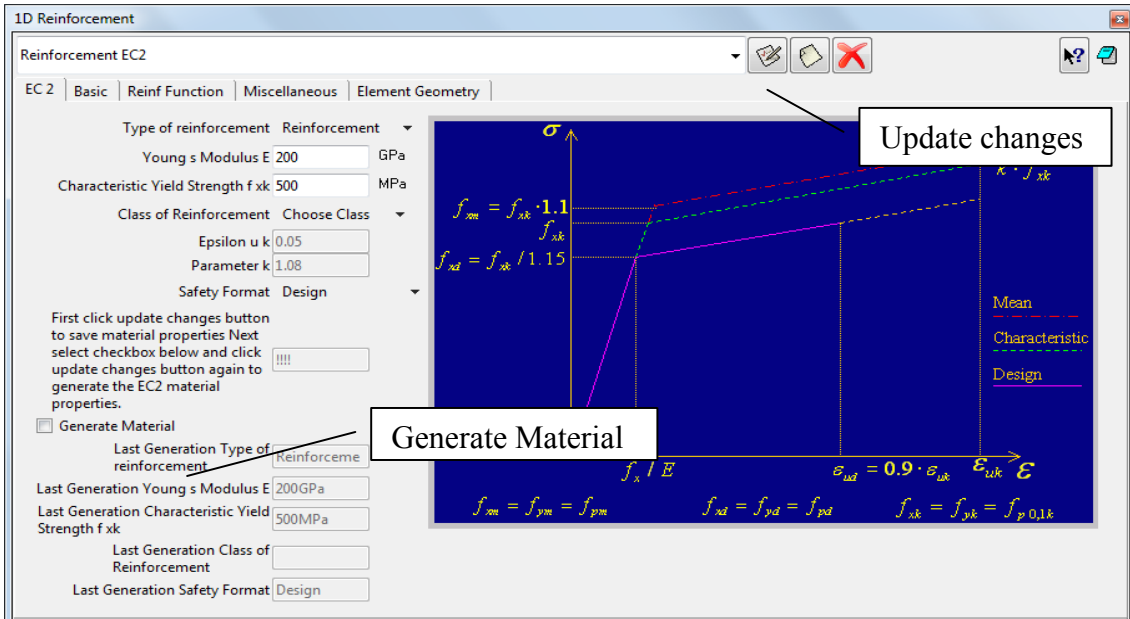


Fig. 5-69: 1D Reinforcement material properties

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

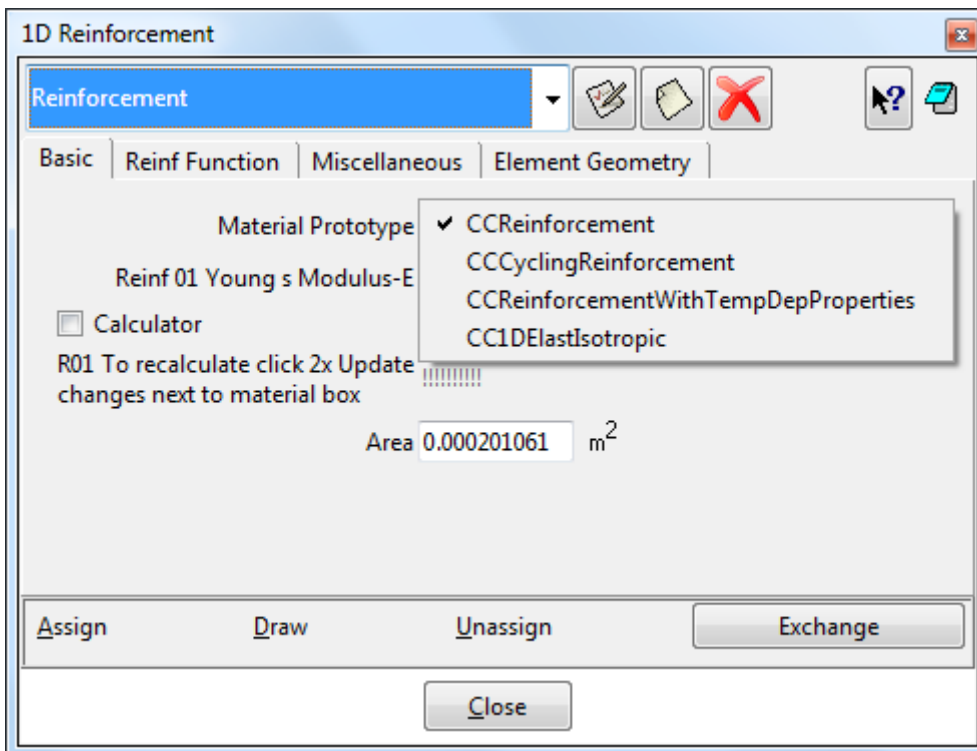


Fig. 5-57: Reinforcement material prototypes

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 the production method.

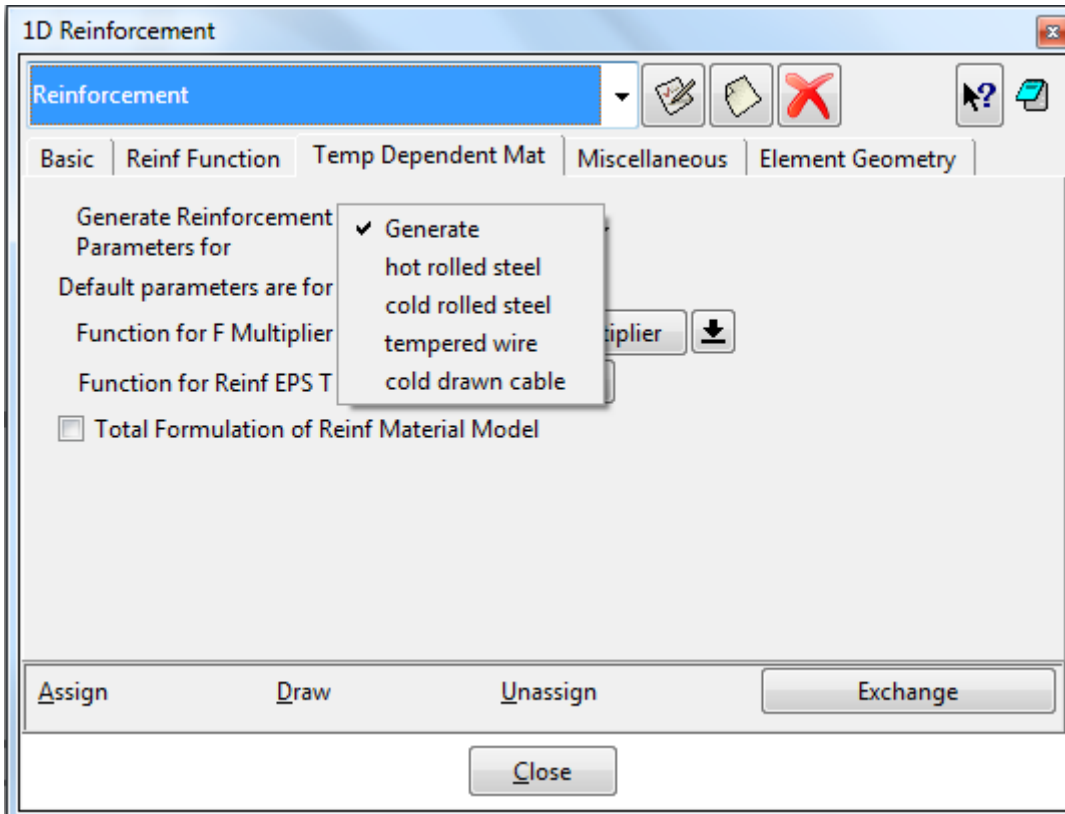


Fig. 5-58: Reinforcement material prototypes

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

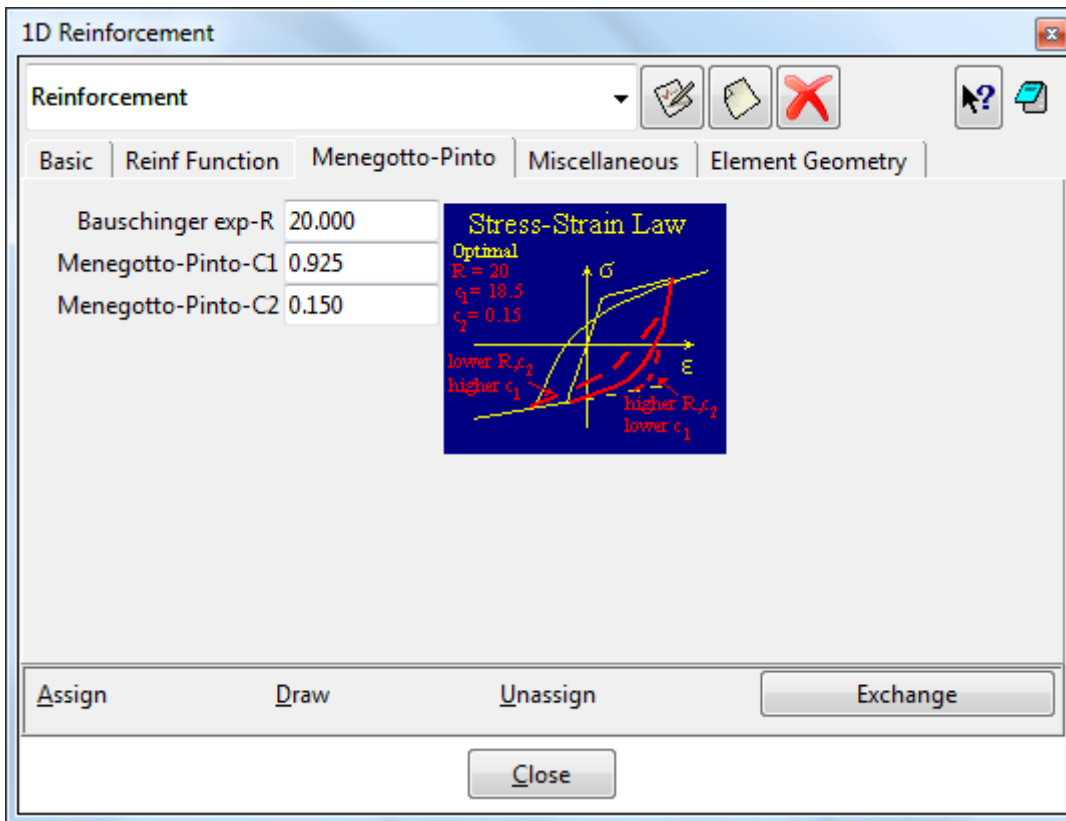


Fig. 5-59: Menegotto-Pinto

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

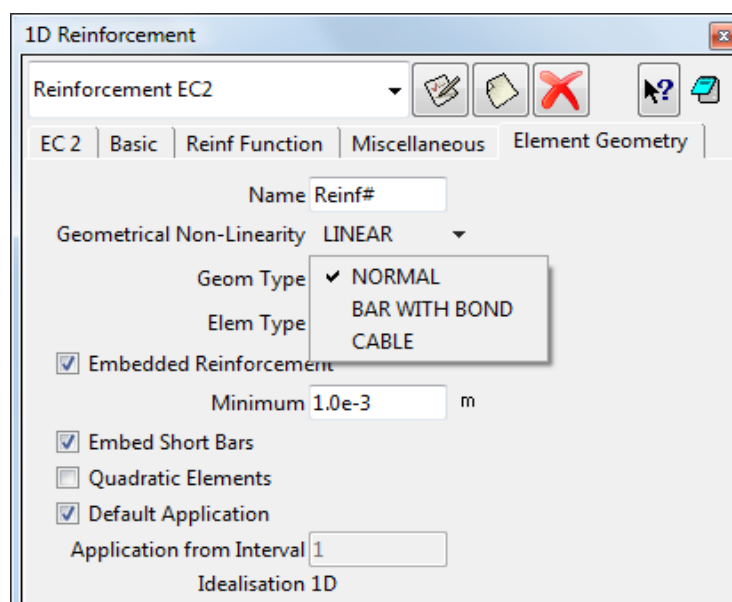


Fig. 5-70: 1D Reinforcement material properties – Element Geometry

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.

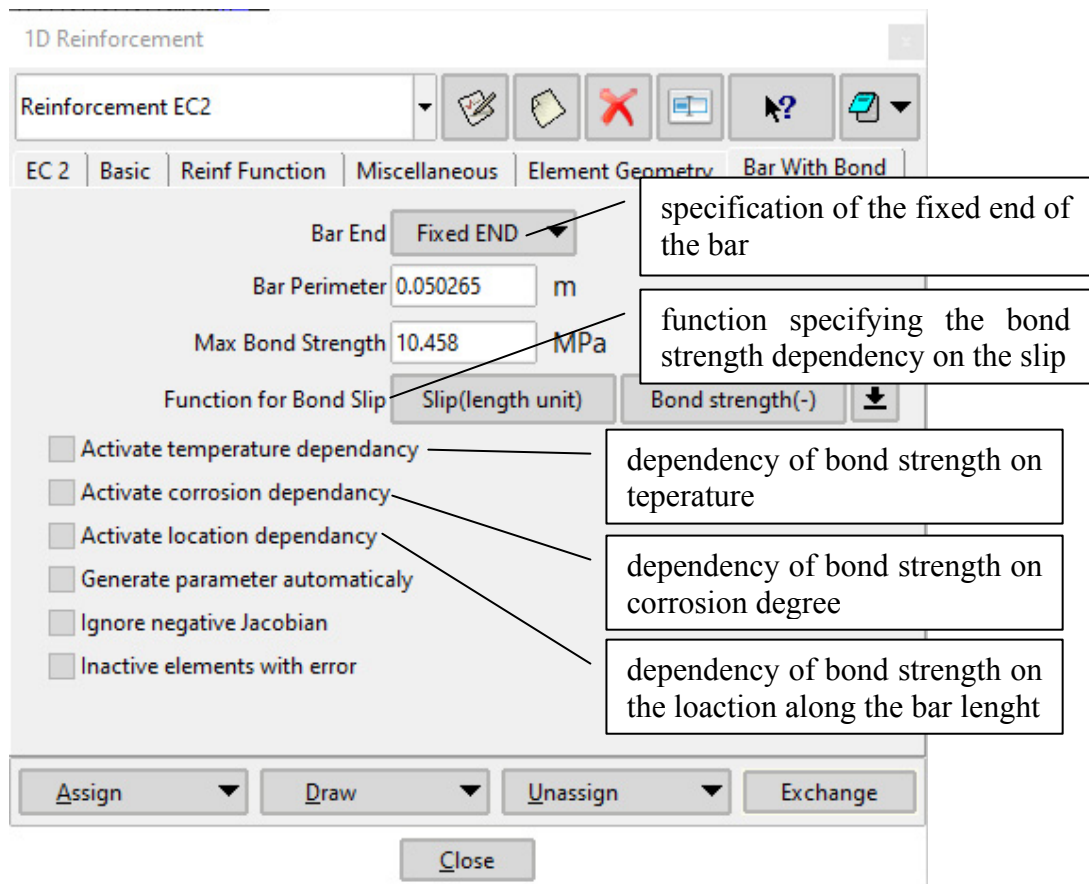


Fig. 5-71: 1D Reinforcement material properties – Bar with Bond

The bond slip law can be generated automatically based on the Fib code (1990 or 2010) or the model proposed by Bigaj (Bigaj, 1999). Furthermore, a dependency of the bond strength on the corrosion of the reinforcement, temperature, or length of the reinforcement can be defined through **Activate temperature dependency**, **Activate corrosion dependency**, and **Activate location dependency** options, respectively.

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)

If you want to model external cable with deviators, you need to model it as a Polyline (**Geometry | Create | Polyline**) and set mesh settings (**Mesh | Structured | Lines**) to 1 for the cable. Next, you need to edit the polyline (**Geometry | Edit | Polyline | Edit polyline**) and set **Use Points**.

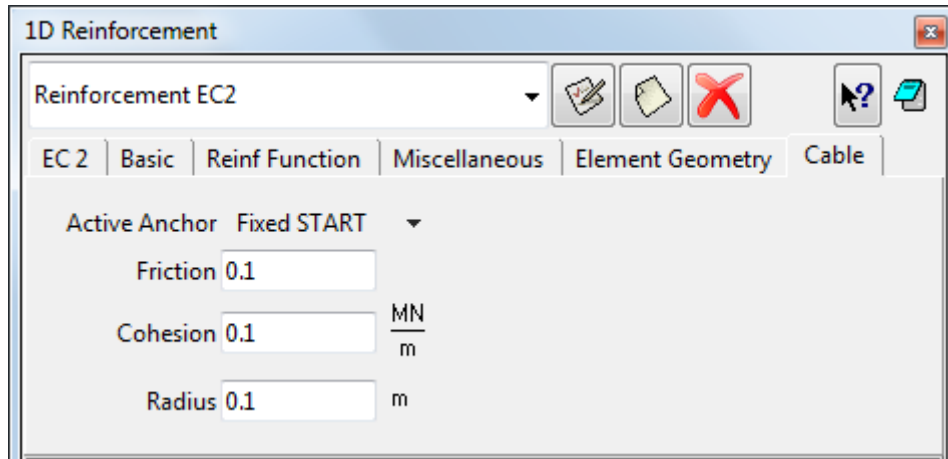


Fig. 5-72: 1D Reinforcement material properties – Cable

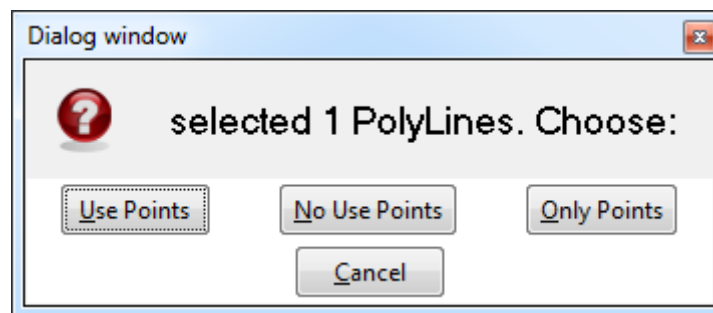


Fig. 5-73: Dialog for editing polylines

5.3.5.3 1D Reinforcement with Bending and Shear Stiffness

In case we need to have reinforcement with shear or bending stiffness, we can use the **1D beam** material (5.3.3)⁴. In the material, we can use cross-section generation and generate circle shape with required diameter. For base material, we can use the **Steel Von Mises** material. Use this material instead of reinforcement one. Set mesh size for this line like the element size around the reinforcement. Next, we need to connect these **1D beam** elements to solid concrete. It will be done by boundary condition **Fixed contact for Line** (slave) for reinforcement and **Fixed contact for Volume** (master) for solid elements.

⁴ The 1D beam elements can only be connected to the surrounding volume in ATENA version 5.7.0 and later (although the elements themselves have been already available in older versions)

A detailed example can be found in the ATENA installation: Tutorial.Static3D\ShearBeam3D_1dbeam.gid

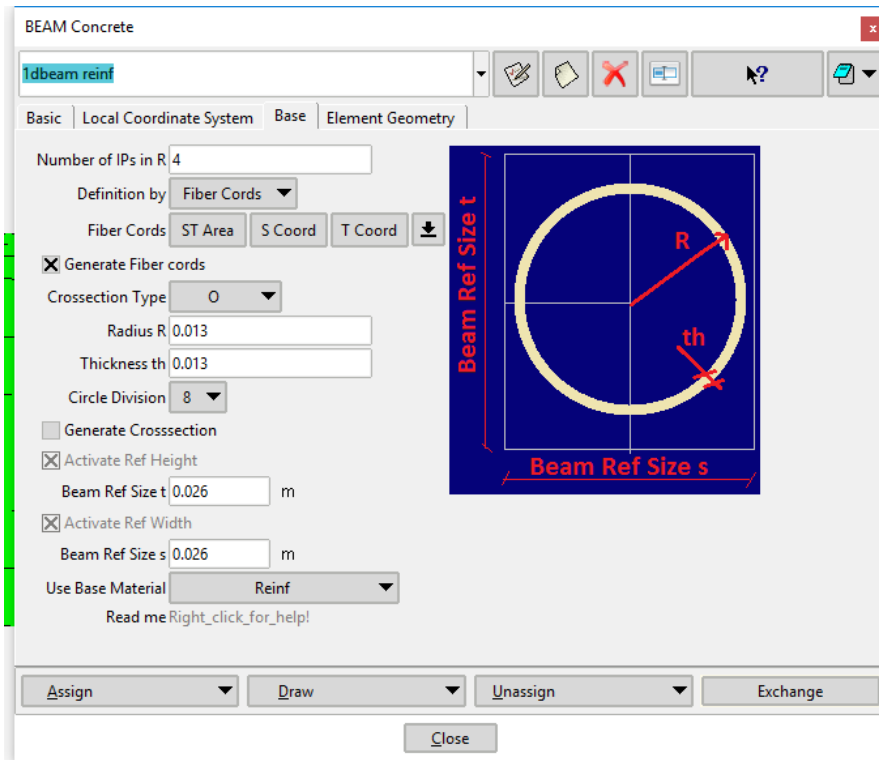


Fig. 5-74: Dialog with circle shape generator

5.3.6 Interface Material

The interface material (also called GAP) has been developed to model behavior 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 and axisymmetry). Please do not forget to choose the Material Prototype according to the problem dimension (**CC3DInterface** or **CC2DInterface**) at the Basic tab.

Element thickness (Element geometry tab) is important for 2D elements and for 2D problemtype. In axisymmetric task, please set 1.0m. See 5.3.1.1.1 Adjusting generated values, figure Cementitious2 – Element Geometry.

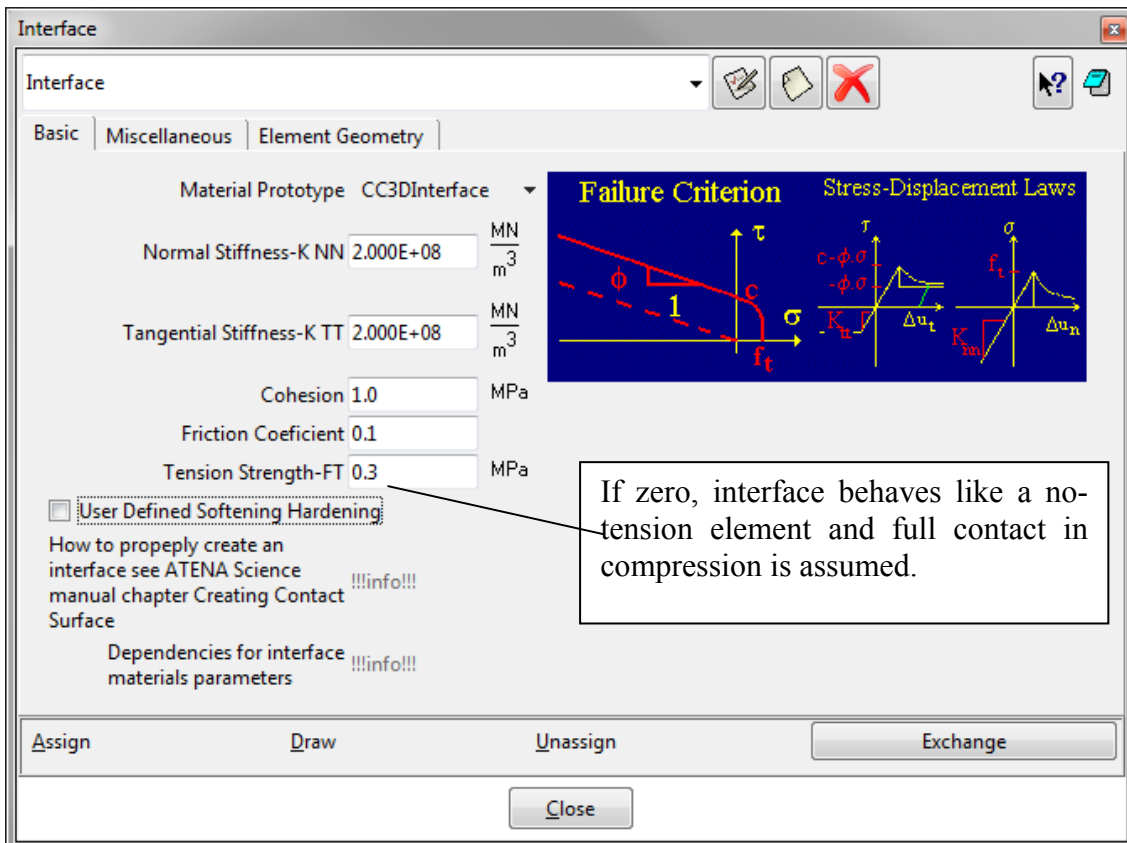


Fig. 5-75: Interface material properties – Basic

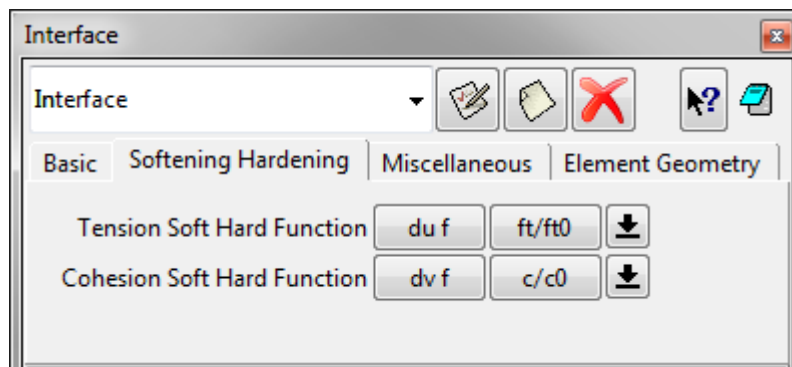


Fig. 5-76: Interface material properties – Softening/Hardening

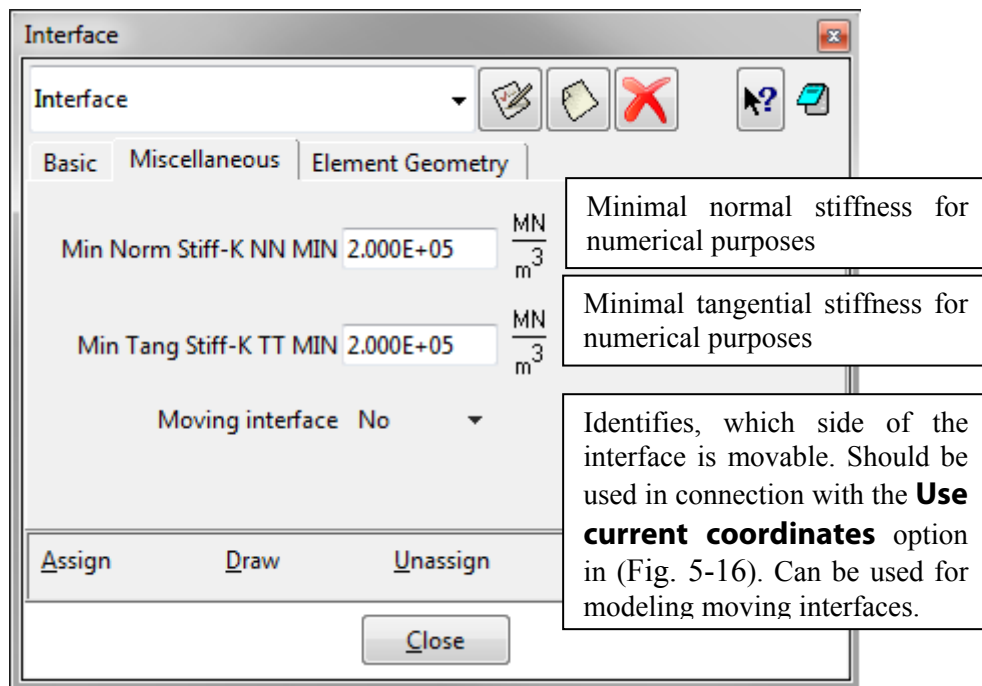


Fig. 5-77: Interface material properties – Miscellaneous

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 (e.g., using the command **Utilities - Swap normals - Surfaces - Select**) 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.

Open Gaps

Special Open Gap elements can be used to represent interfaces with initial opening, i.e., when the 2 surfaces come into contact later, when the initial empty space between them closes.

For this type of gap elements, the special geometry Separated contact volume (**Geometry - Create - Contact - Separated volume**) must be used. It is also necessary to activate it through applying the condition **Initial Gap Load for Volume** and activating the **Activate Interface Opening** checkbox in the **Interval Data** dialog (see also 5.4) for the interval in which the gap is to be set (typically, this is going to be the first Interval).

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 the Input manual [4]: ELEMENT_LOAD description. The 2D variant of this option (Separated contact surface) is not supported 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).

Apart from **Fixed Contacts** condition, there are **Fixed 1D Beam to Solid for...** and **Fixed 2D Shell to Solid for...** conditions, which are suitable for creating pinned connections between 1D beam and 2D shell elements with other parts of the model. Please refer to section 5.2. of this manual for a more detailed description.

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 **Collapse** disabled (**Duplicate entities** enabled in GiD 11 and older).

The box **Collapse**⁵ in the Copy dialog works the following way: If checked, eventual duplicate nodes, lines, surfaces are merged into one (similarly to the **Collapse** command). If unchecked, 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

⁵ In GiD 11 and older, the box name is **Duplicate entities** and its meaning the opposite

the size of the smaller of the two, located at the same place (or, if it helps to keep track of the entities, 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 the 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. 5-78. The interface will be added at the place of the inclined line.

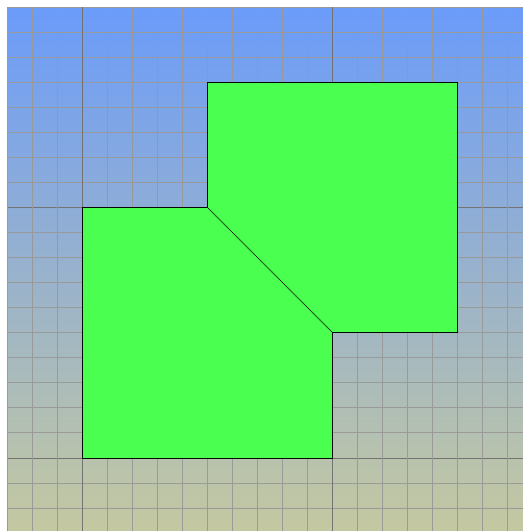


Fig. 5-78: Creating a contact surface – Introduction

The interface can be created through the following steps, illustrated in Fig. 5-79, Fig. 5-80, and Fig. 5-81:

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.

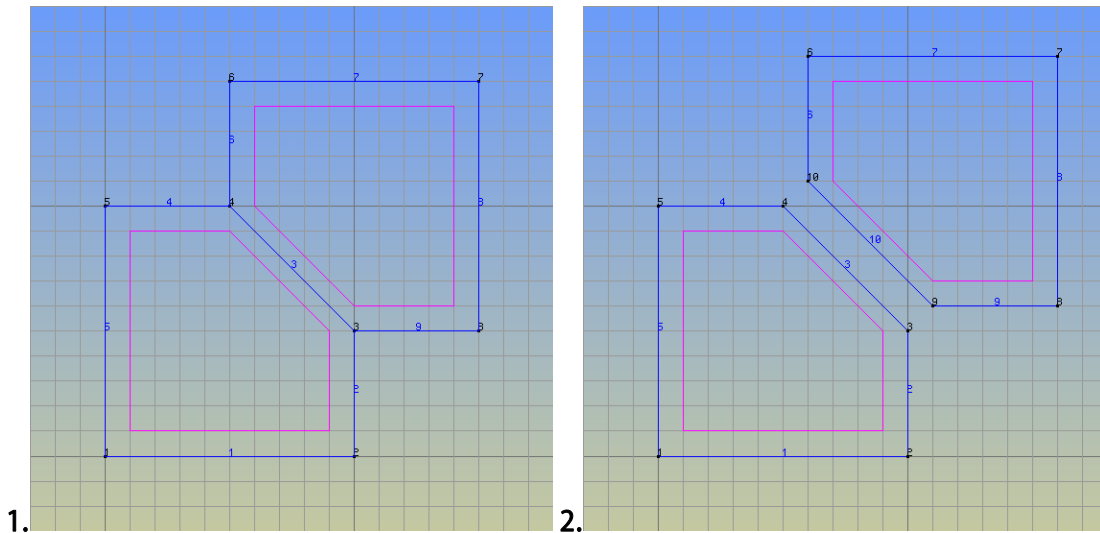


Fig. 5-79: Creating a contact surface - Steps 1-2

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.

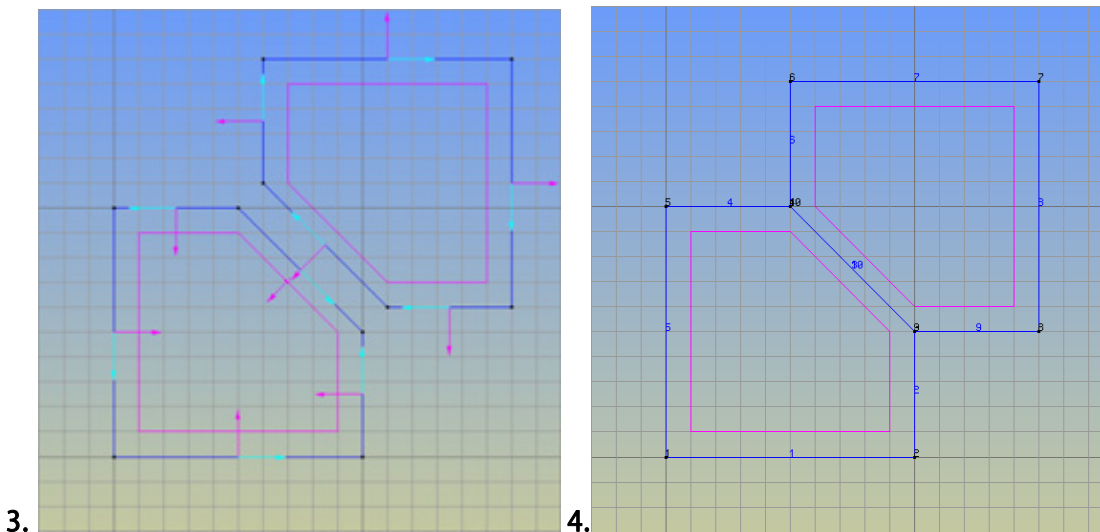


Fig. 5-80: Creating a contact surface - Steps 3-4

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.

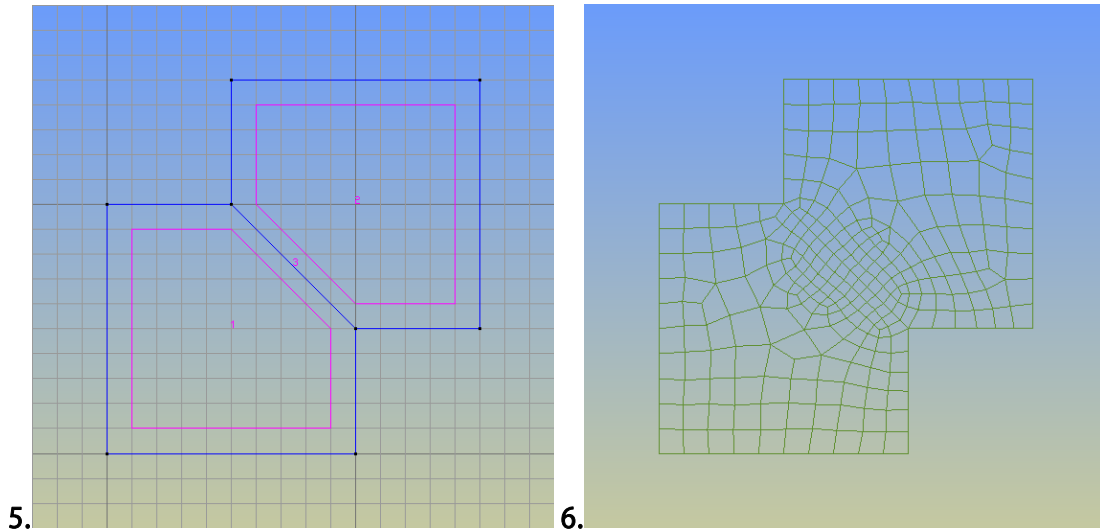
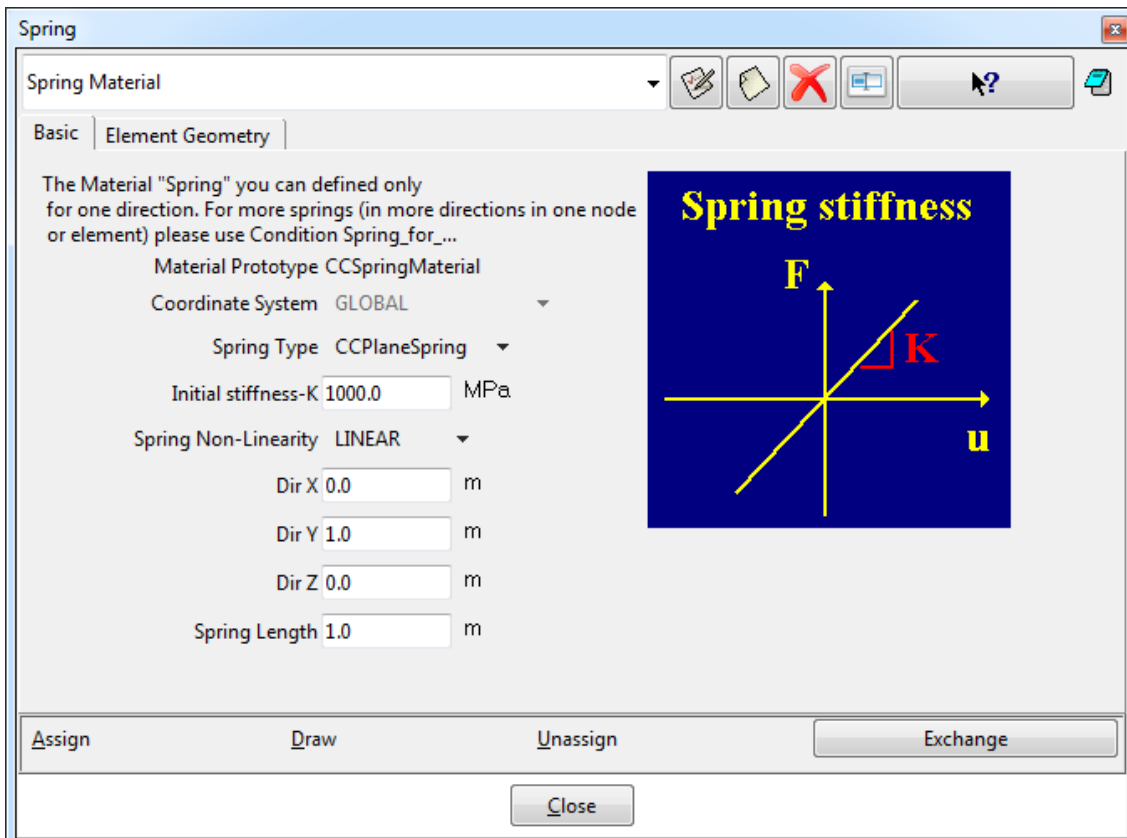


Fig. 5-81: Creating a contact surface - Steps 5-6

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

Spring material can be used for surface and line springs. For point springs, see the condition Spring for Point in 5.2.



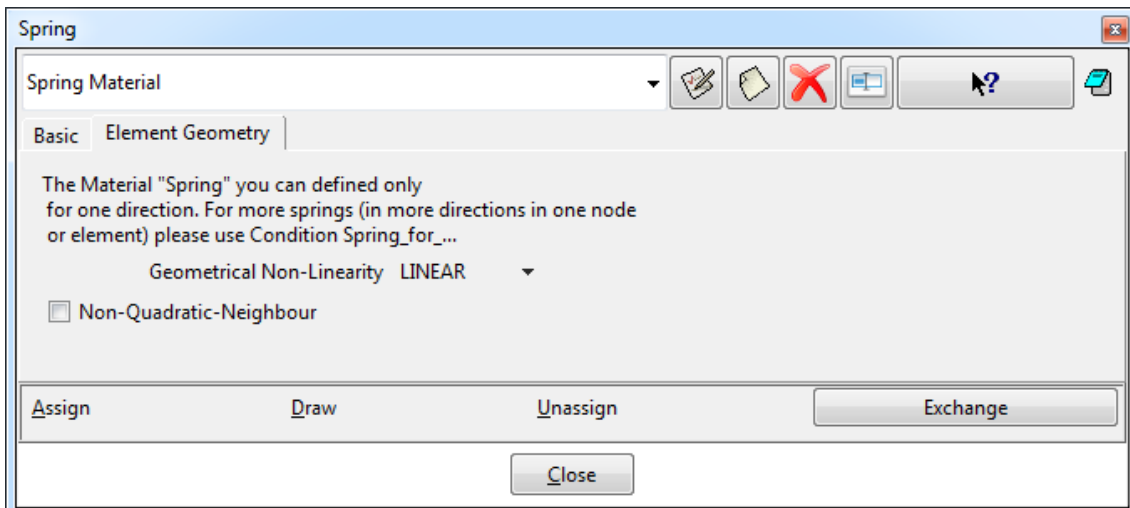


Fig. 5-82: Spring material dialog

5.3.7.1 Surface Spring Definition 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 $0.005 \text{ [MN]} / 0.015 = 0.3333333 \text{ MPa}$ ($\sigma = E * \epsilon$)

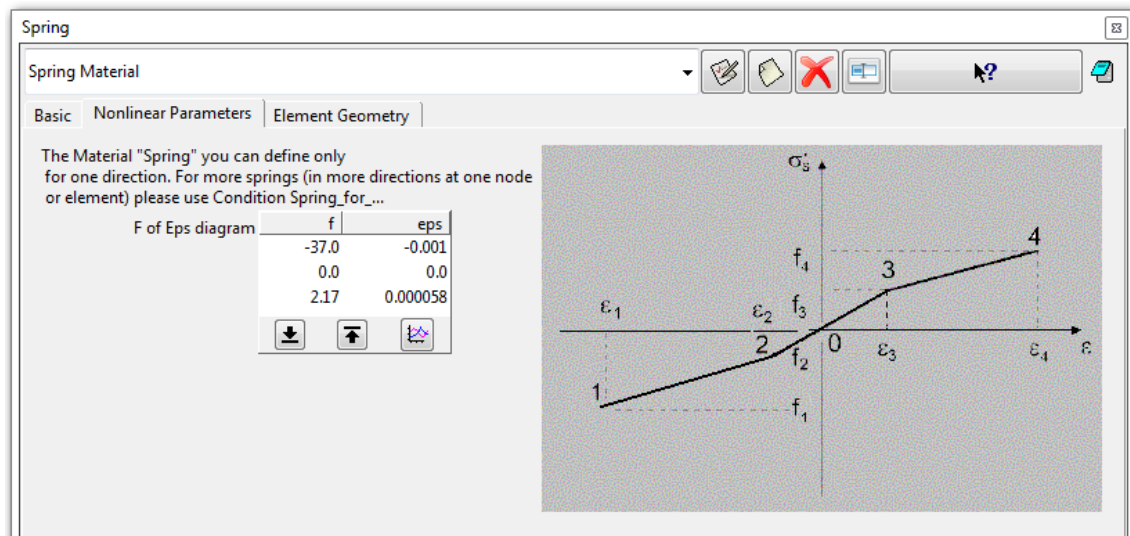


Fig. 5-83: Spring material dialog – nonlinear parameters

5.3.7.2 Line Spring Dimensions

Imagine the spring as an 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.

5.3.7.3 Nonlinear Geometry

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).

5.3.7.4 Mesh Settings

It is necessary to assign special mesh settings to the surface or line with this material using the command **Menu | Mesh | Mesh criteria | Mesh** (line or surface).

GiD automatically creates mesh for entities not belonging to any higher entity, i.e., lines not belonging to any surface and surfaces not being part of any volume, but springs are typically assigned to surfaces/lines which are part of a volume/surface. For additional information, see also ATENA Troubleshooting [9], 4.2.5 GiD claims: You don't have assigned Material 0 at Element xxxx For the first number of unassigned elements, see *.dat file.

5.3.8 The Material Function

This material is used to easily define user function for some type of loading or material properties. You can easily 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 an example file with the same name, which can be edited. This example file provides information about the necessary file format.

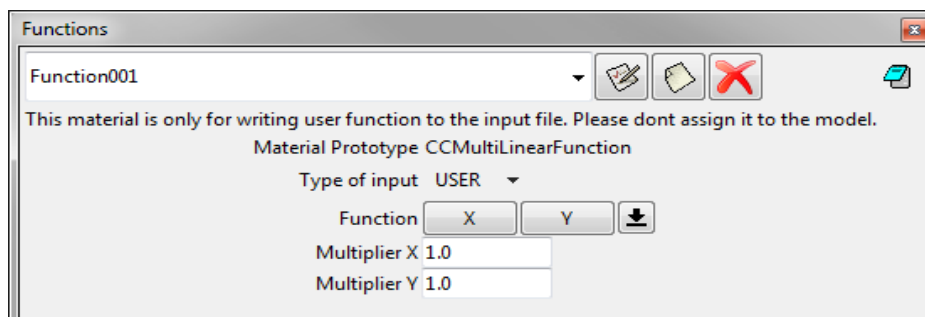


Fig. 5-84: Function material dialog

5.3.9 Material from file

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

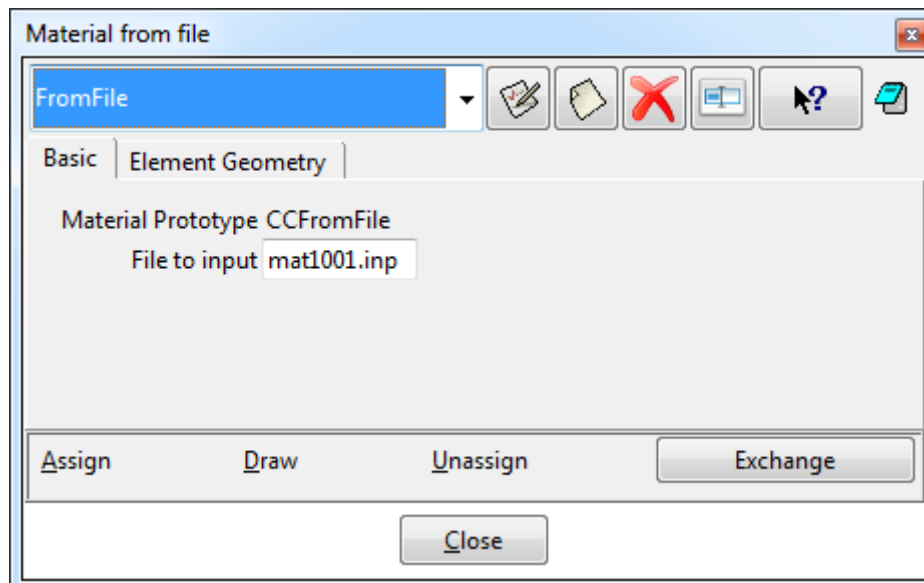


Fig. 5-85: Material from file dialog

For example, for creep material CCMoelGeneral please find your GiD script directory and copy file “Atena\Materials\CCMoelGeneral\default.inp” to your project directory. Set the filename to the dialog and use like normal material.

5.3.10 Dummy Material

This material is used to easily define some special kinds of conditions, for example, Load Universally. If you assign this material to some entities, these entities are not included in the ATENA calculation. You can use conditions on it, but the elements are not written into the ATENA Input File. In other words, it only makes sense to use it with the special conditions where the "Dummy" material book is mentioned.

5.3.11 Steel von Mises 3D Material

The plastic steel material model offers 2 prototypes, the "normal" **CC3DBilinearSteelVonMises** and a variant supporting thermal degradation **CC3DBilinearVonMisesWithTempDepProperties**. It is also possible to define cyclic properties through enabling the option **Activate Cyclic Params**.

This material model is targeted for volume members made from steel and other similar (ductile) materials, i.e., elements which can not be well represented with the 1D reinforcement (truss) elements. Note the hardening is not limited, which means it has to be checked in post-processing if the ultimate strain has been exceeded.

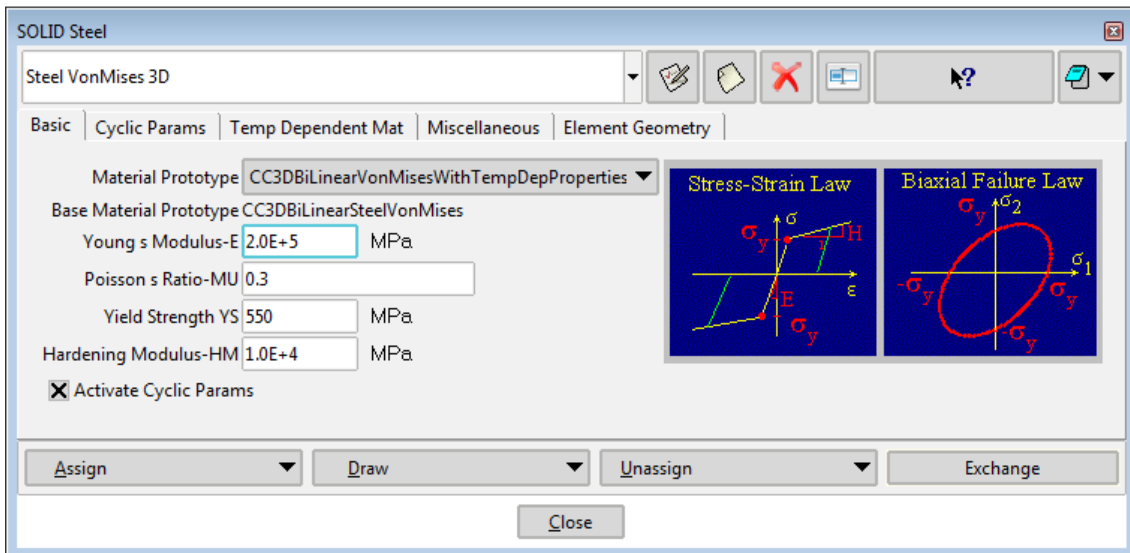


Fig. 5-86: Steel von Mises material dialog – Basic parameters

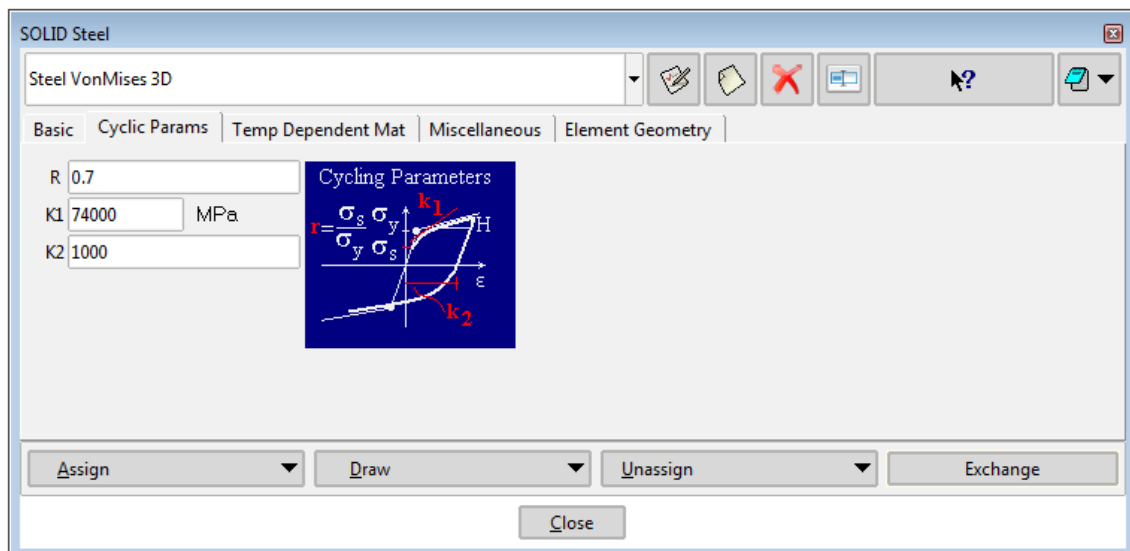


Fig. 5-87: Steel von Mises material dialog – Cyclic parameters

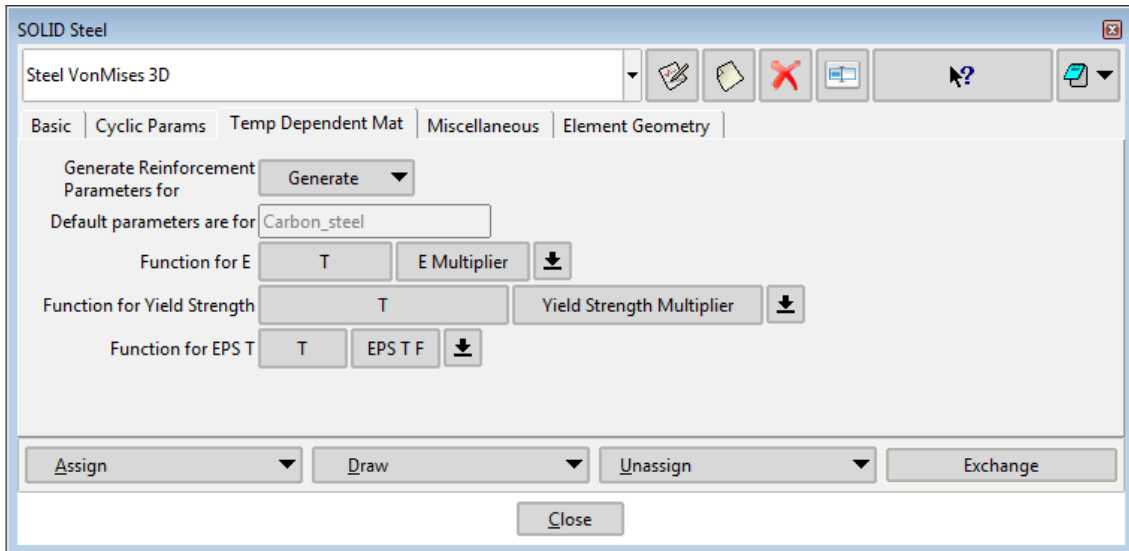






Fig. 5-88: Steel von Mises material dialog – Temp Dependent Mat parameters

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. 5-90, 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 changed by icon .

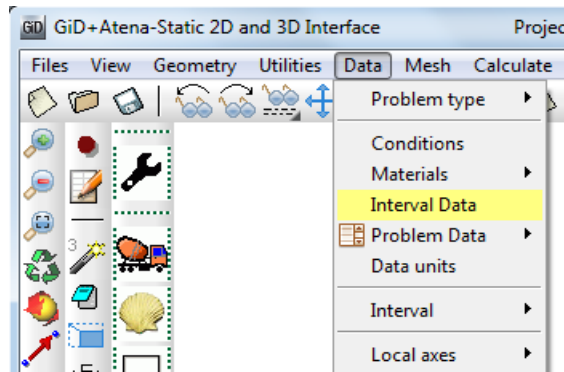


Fig. 5-89: Load steps (intervals)

 A screenshot of the 'Interval data' dialog box. It has two tabs: 'Basic Parameters' and 'Eigenvalue Analysis'. The 'Basic Parameters' tab is active. The dialog contains several settings:

- Interval: 1
- Use decimal point (do not use comma): checked
- Interval Is Active: checked
- Load Name: Load ...
- Define Loading History: unchecked
- Type of Definition: Manual
- Generate Multiple Steps: checked
- Activate Step multiplier: checked
- Step Multiplier: 1.0
- Interval Multiplier: 5
- Number of Load Steps: 5
- Store Data for this Interval Steps: SAVE LAST ONE
- Fatigue Interval: NO
- Read Transport Data: unchecked
- Delete BC Data After Calculation: checked
- User Solution Parameters: unchecked
- Activate Interface Opening: unchecked
- Add SubInp Before Steps: unchecked
- Add Additional Load Cases: unchecked
- Set Reference Configuration: unchecked
- Show Material Activity: checked
- Material Activity Step: OldMaterialName, NewMat
- Digicon settings: unchecked

 Several callout boxes provide additional information:

- Only active intervals are used for the analysis (points to the Interval field).
- Use this, if you can define loading history manually. (points to the Define Loading History checkbox).
- You can use Step or Interval multiplier for the loading. (points to the Step Multiplier and Interval Multiplier fields).
- If this check box is selected - Transport data (Temperature) are imported into this interval. The check box have to be activated in Problem Data -> Transport by IMPORT TRANSPORT RESULTS (points to the Activate Step multiplier checkbox).
- After calculation unused Load Case Data can be erased (points to the Store Data for this Interval Steps dropdown).
- Activate tab with solution parameters. The there defined User settings remain valid until new values are defined in a later Interval (points to the Eigenvalue Analysis tab).
- This switch updates interface opening based on the interface geometry. This is useful for modeling interfaces with initial opening. See 5.3.6 Interface material section Open Gaps (points to the Activate Interface Opening checkbox).

Fig. 5-90: Interval Data window - Basic parameters

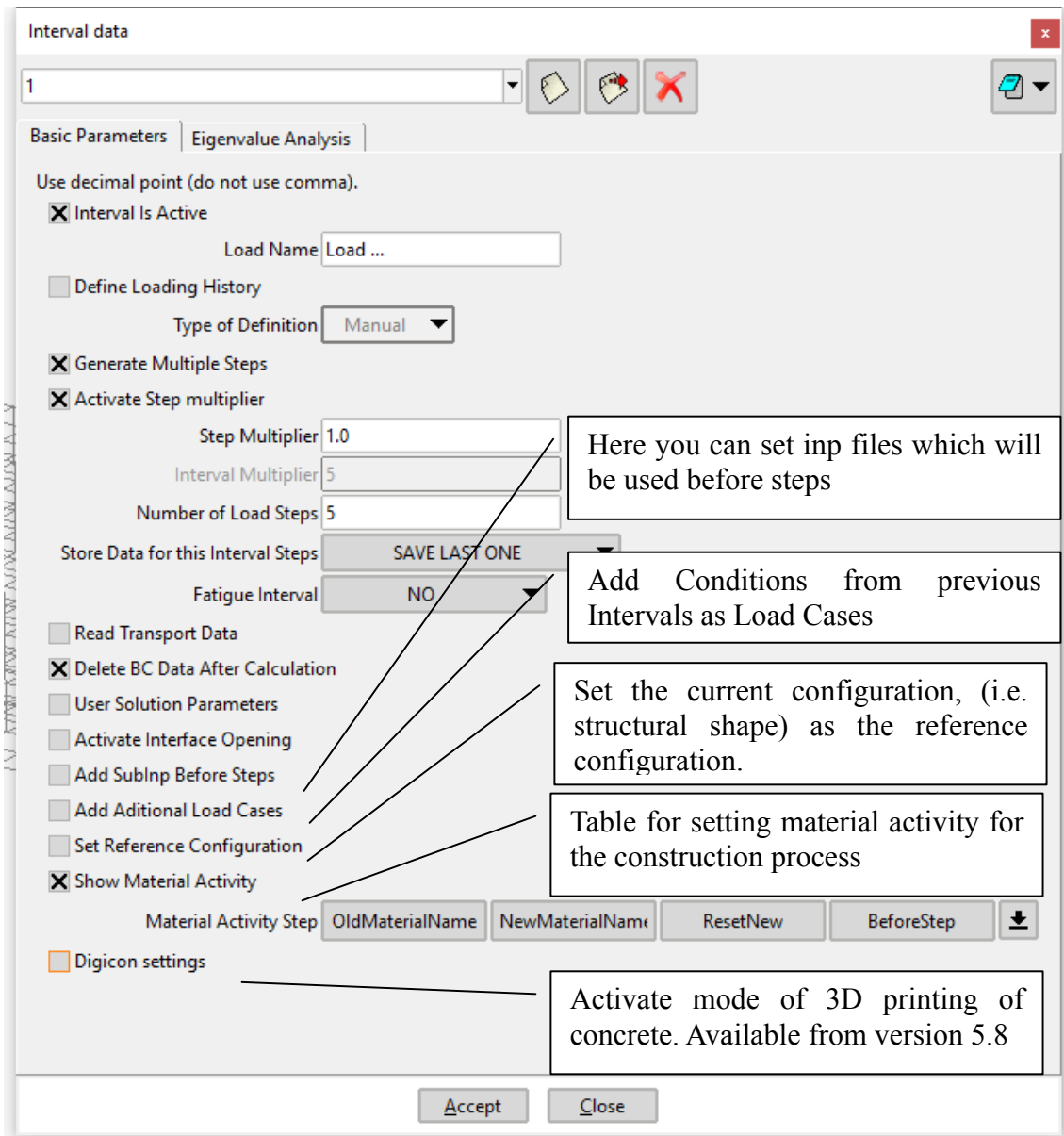


Fig. 5-91 Interval Data window – Basic parameters 2

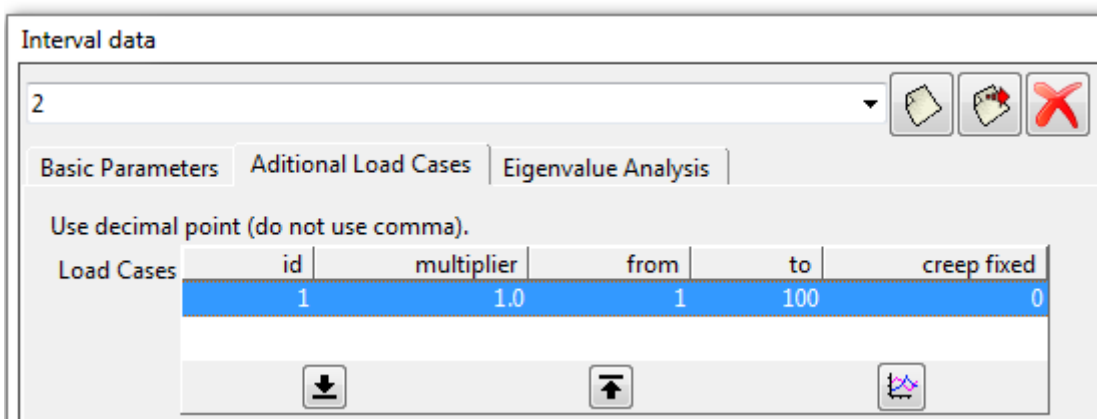


Fig. 5-92 Interval Data window – Additional Load Cases

The option **Add Additional Load Cases** to add the Conditions from a previous Interval to this interval as a load case. For this case, you need to disable the box **Delete BC Data**

After Calculation in the interval, which you wish to use as a load case. The load case number (**id**) is, in most cases, the same as the interval number (Interval 1 = Load case 1, Int 2 = LC2). If you disable interval 2, then Int 3 = LC2. You can, for example, define all supports only in the first interval, and add them as a load case to all other (subsequent) intervals, i.e., you don't need to define all these supports again in each interval. The main advantage compared to copying all the Conditions when creating the subsequent Intervals is that when you find a mistake in the supports, you only need to correct it once (instead of correcting separately in each of the Intervals with the copied Conditions).

The most typical use of this option is to define all supports and fixed contact conditions only in the first interval, and set the number of steps to 0 for this interval (i.e., there is no need to calculate a step with no load but just the supports to be used later), and use it in all other intervals.

You can use the first interval for body load as well, but if you want to use supports in subsequent intervals, you need to set the **multiplier** to 0 (otherwise, the body load will be applied again). See example: GiDExamples\Tutorial.Static2D\ TunnelWith ConstructionProcessNew.gid.

The additional Load case application can be limited to only some of the steps of the Interval by defining the step range using the **from** and **to** columns (step numbering inside the interval, i.e., 1 means the first step of the Interval; most typically, the default 1-9999 is used, adding the LC to all steps of the Interval). The option **Creep fixed** is only included in Creep analysis, determining if the Load case should be used as FIXED or INCREMENT (see also section 9).

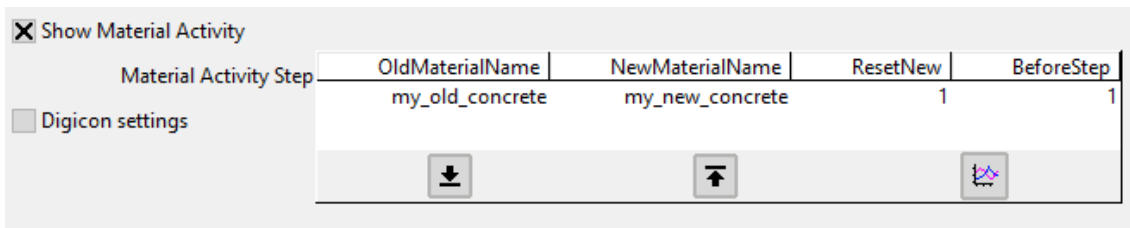


Fig. 5-93 Interval Data window – Material Activity

Material Activity is a new option how to exchange materials to represent the construction process. **OldMaterialName** is the name of the material which has been assigned to the geometry. So, even if you change the material for the second time, the Old material is still the same = the originally assigned one. The parameter **ResetNew** sets the material state to zero (fresh undamaged state). **BeforeStep** defines when the change is to take place - 1 means before the first step of the Interval.

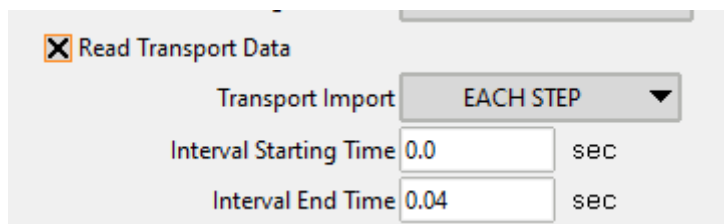


Fig. 5-94 Interval Data window – Read Transport Data

Transport import has two options how to import the temperature:

INTERVAL_BEGINNING - Temperature data are only read once at the interval start and the resulting initial strain is subdivided for each step;

EACH_STEP - Temperature data are read for each step at generated times based on the starting time, time increment and number of load steps. For complex temperature histories not well approximated by a linear interpolation throughout the interval (e.g., fire analysis), this should be changed to Each Step. Then, the thermal results are imported in each load step and the temperature fields are interpolated from the 2 nearest transport analysis steps.

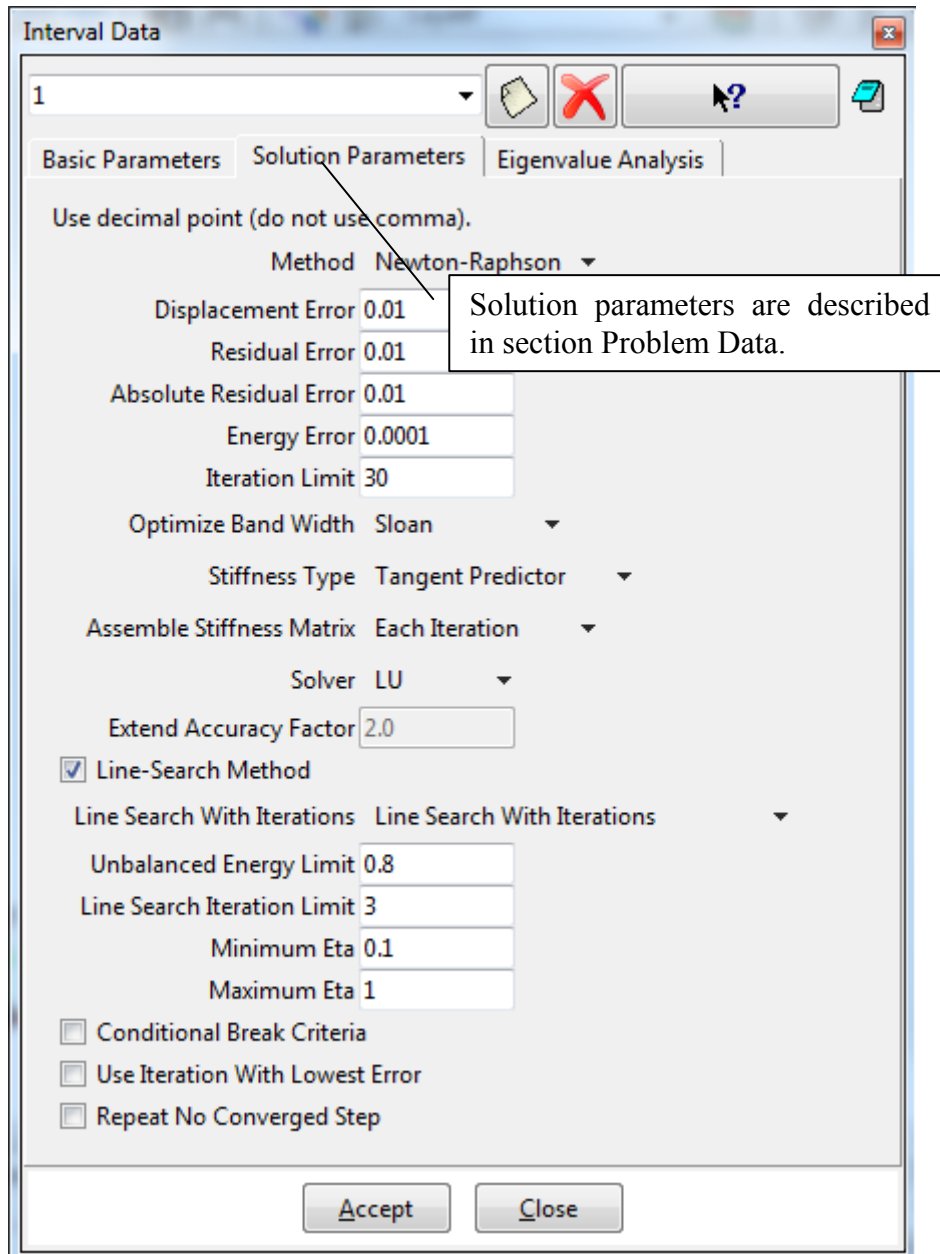


Fig. 5-95 Interval Data - Solution Parameters

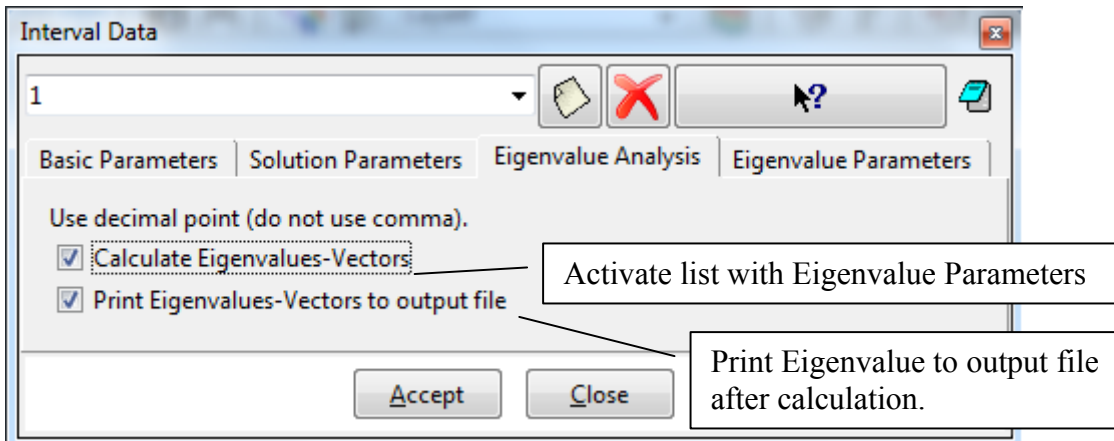


Fig. 5-96 Interval Data - Eigenvalue Analysis

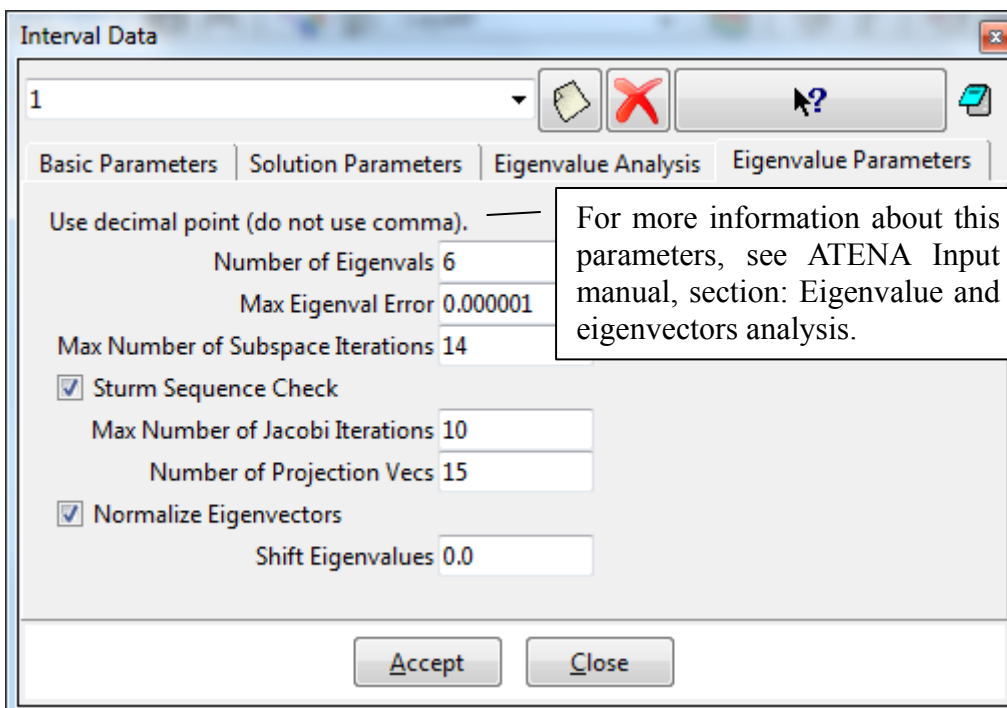


Fig. 5-97 Interval Data - Eigenvalue parameters.

5.4.1 Fatigue

To consider the 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 from 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 modeled, 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**

Evaluating the number of cycles using predefined Monitor

When conducting fatigue analysis in ATENA 5.1.3 or 5.3.2 or later, please enable the checkbox **Write Fatigue Applied Cycles Monitor** at the **Global Options** tab of the **Problem Data** dialog. Then, look at the Monitor named **Ncycles** while running the analysis or during post-processing. The component **N_appl** shows the number of cycles which have been calculated and the corresponding damage applied to the material, which is typically the most interesting. The components **N_curr** and **N_calc** display the cycle currently being calculated and the number of cycles for which the damage has been calculated (but not necessarily applied into the material), respectively.

Manual evaluation (ATENA versions older than 5.1.3/5.3.2)

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.

Starting from version 5.1.3 and 5.3.2, the above calculation can be done by the program – please see the paragraph Evaluating the number of cycles using predefined Monitor above.

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 modeling 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.

Evaluating the number of cycles

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. The easiest way is to use the predefined monitor (see also above; when using versions older than 5.1.3 or 5.3.2, 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 the number of iterations, convergence criteria, or the solution methods for an **ATENA** analysis are defined in the menu item **Data | Problem**

Data, Fig. 5-98 or icon . The dialog window is opened and default data are offered.

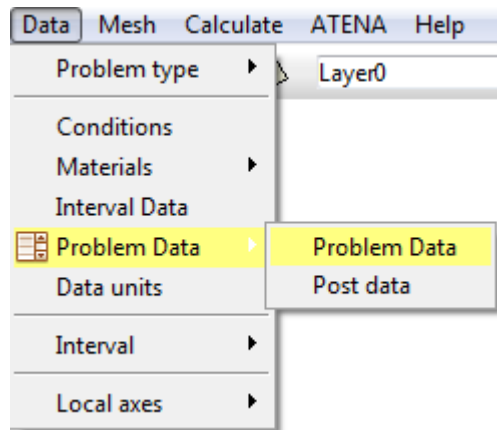


Fig. 5-98 Problem Data

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.

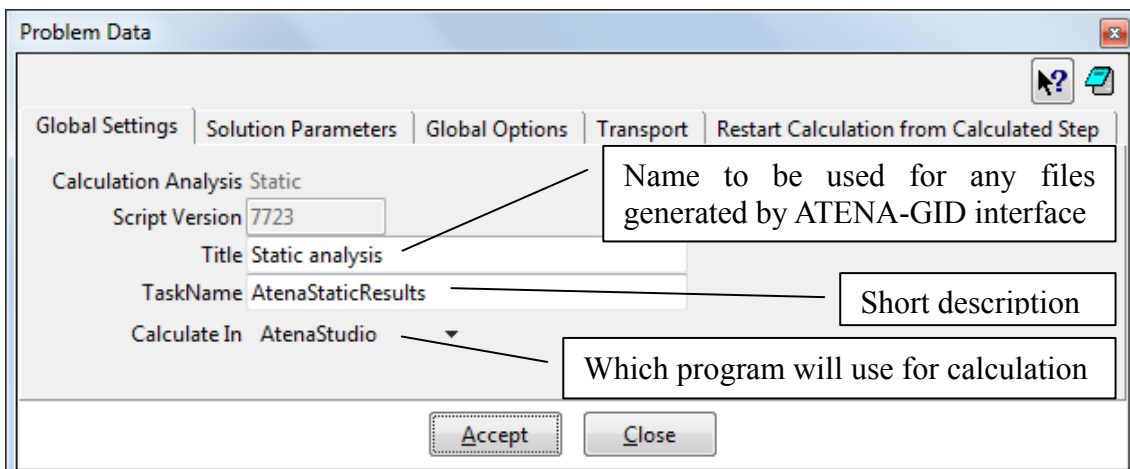


Fig. 5-99 Problem data – Global Settings

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].

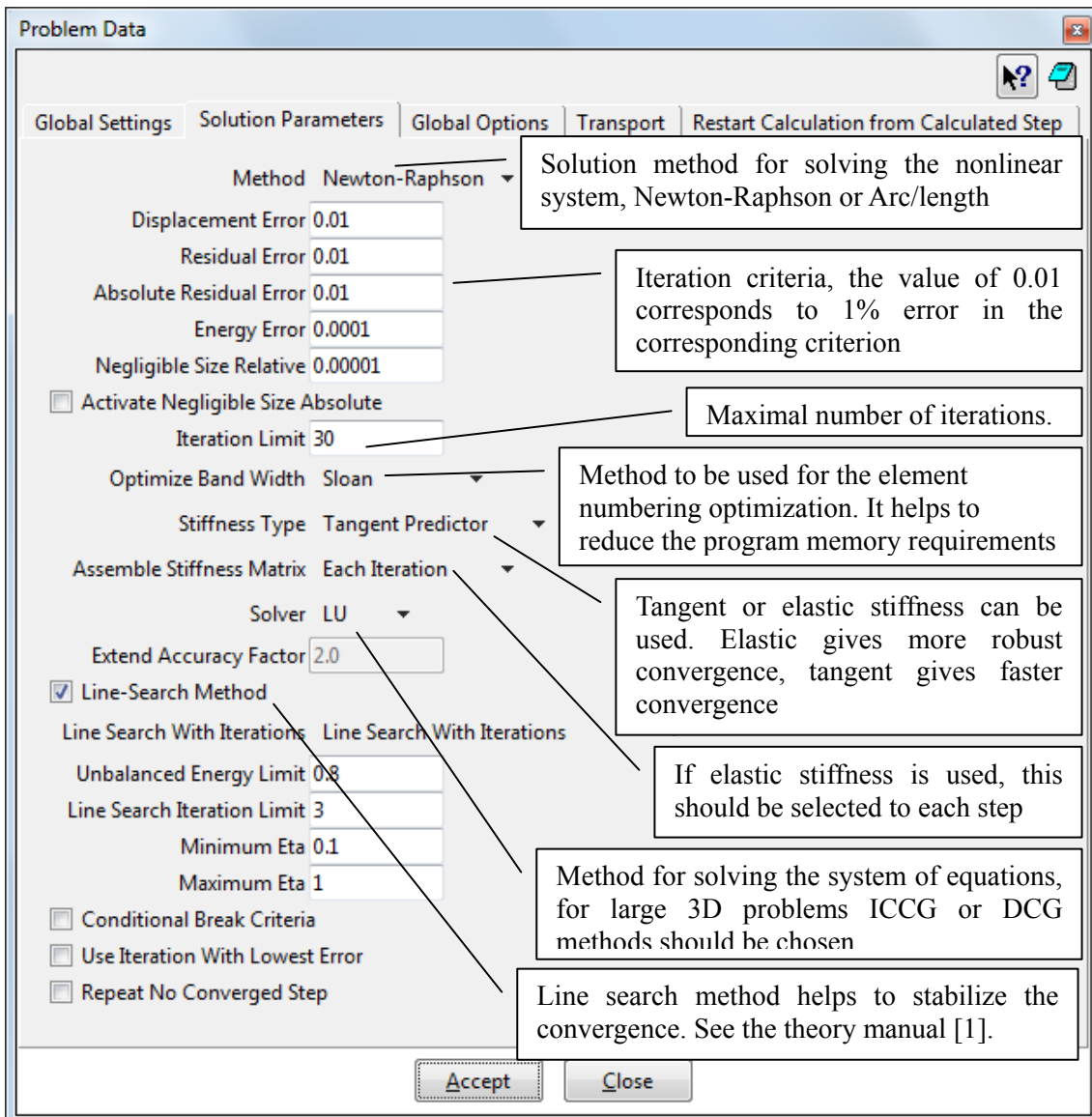


Fig. 5-100 Problem data – Solution parameters.

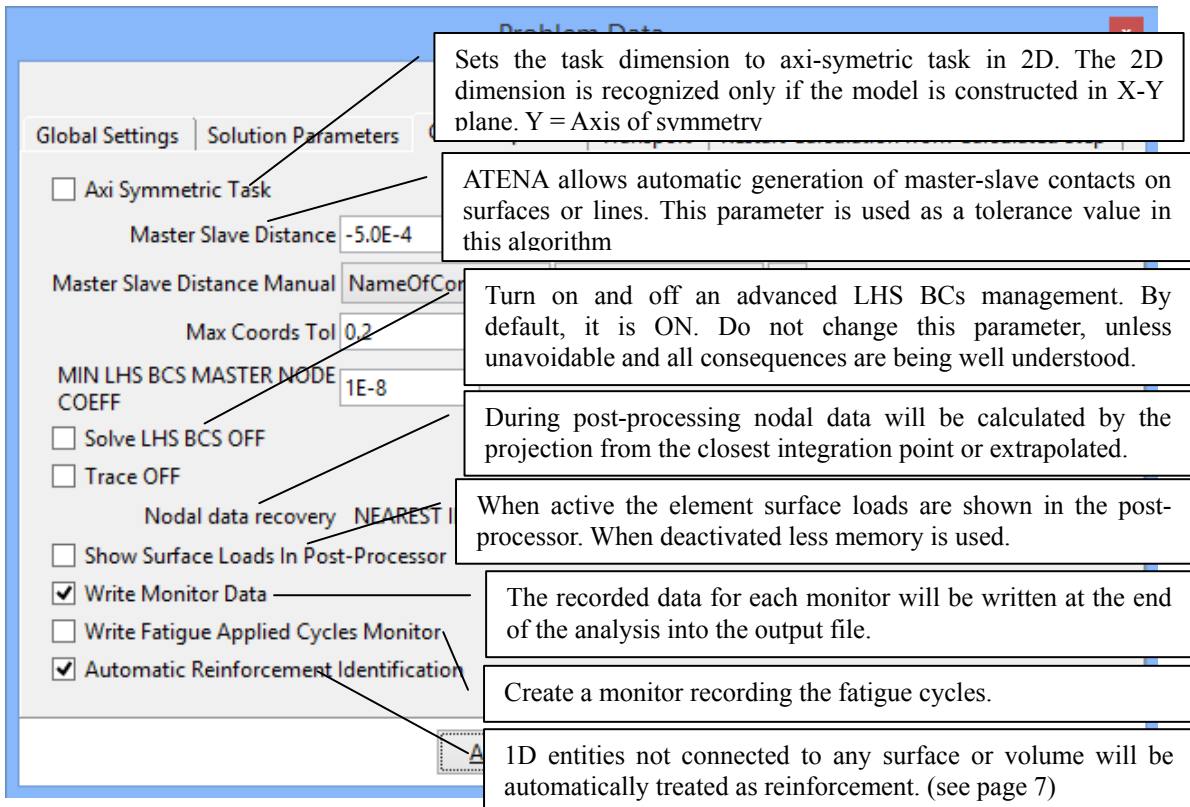


Fig. 5-101 Global Options in problem data dialog.

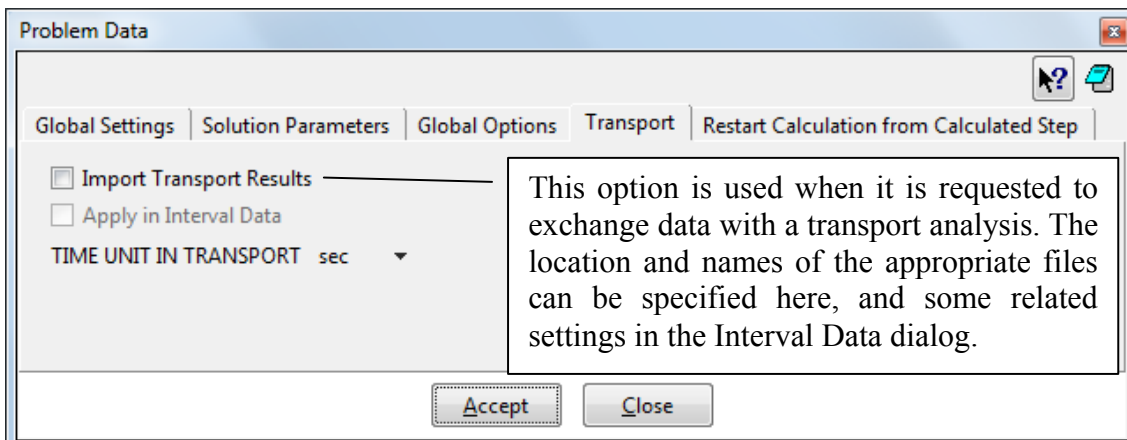


Fig. 5-102 Problem data – Transport

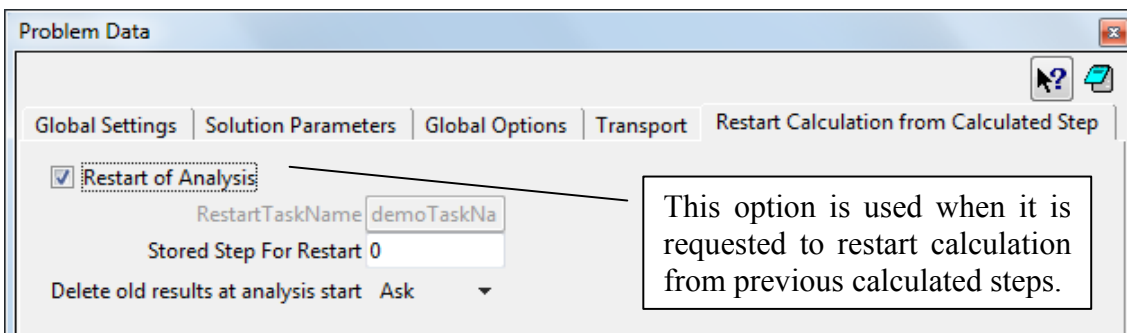


Fig. 5-103 Restart calculation options in problem data dialog

5.6 Concrete 3D Printing⁶

Printing of concrete structures is an emerging manufacturing process in the field of construction engineering. The automatized technology increases the construction speed, reduces the labor cost while allowing for the construction of complex shapes. The main concept behind the analysis in **ATENA** lies in a gradual activation of the finite elements along the specified printing head trajectory and in the application of the material with time-dependent material properties. This chapter summarizes the key aspects of the modeling process in GiD pre-processor.

To activate the analysis of a concrete printing process, this option has to be first activated in **Data | Problem data | Problem Data** settings by selecting **Use Digicon** option in **Global Options** tab. This dialog is shown in Fig. 5-104. If this option is not activated, the pre-processor will ignore all commands related to the 3D printing.

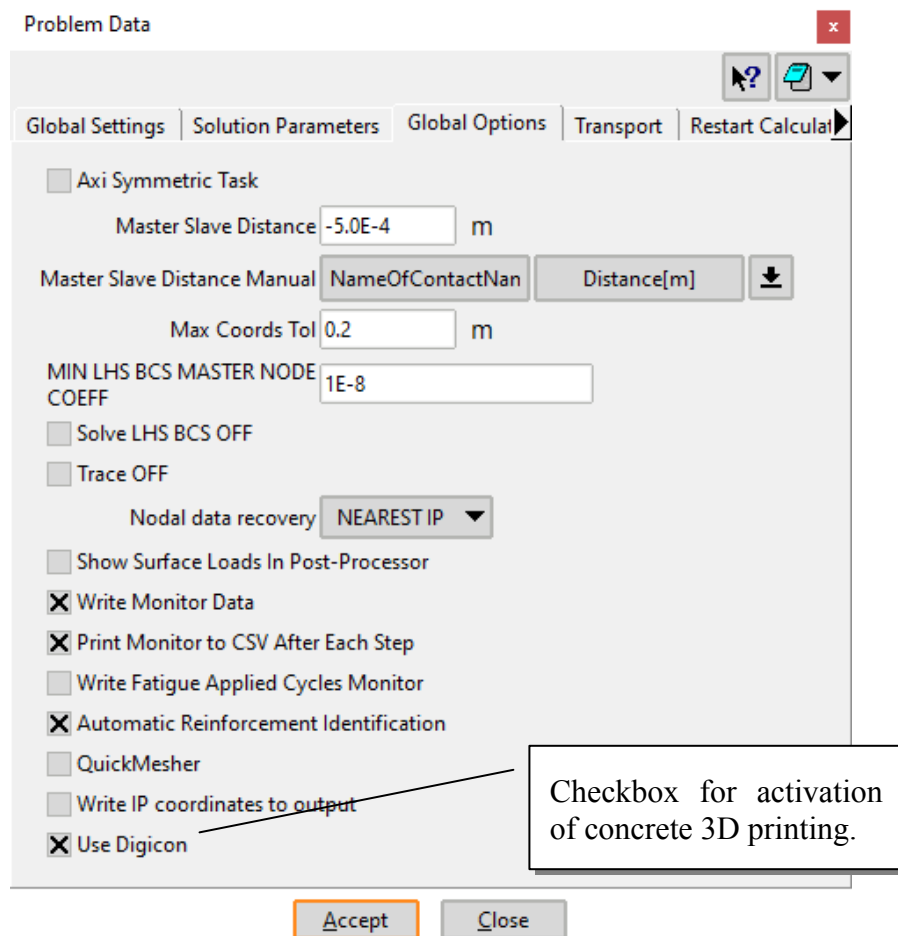


Fig. 5-104 Activation of concrete 3D printing.

A crucial step for the analysis is the definition of the printing head trajectory, which is used for computing the construction time of each element. This setting is open through **Data | Materials | DigiCon Tracks**. The details should be specified upon selecting **3D**

⁶ Available from ATENA version 5.9.0

Printing Parameters in the tab with the same name as shown in Fig. 5-110. The inputs are assigned to the elements based on the specified material name.

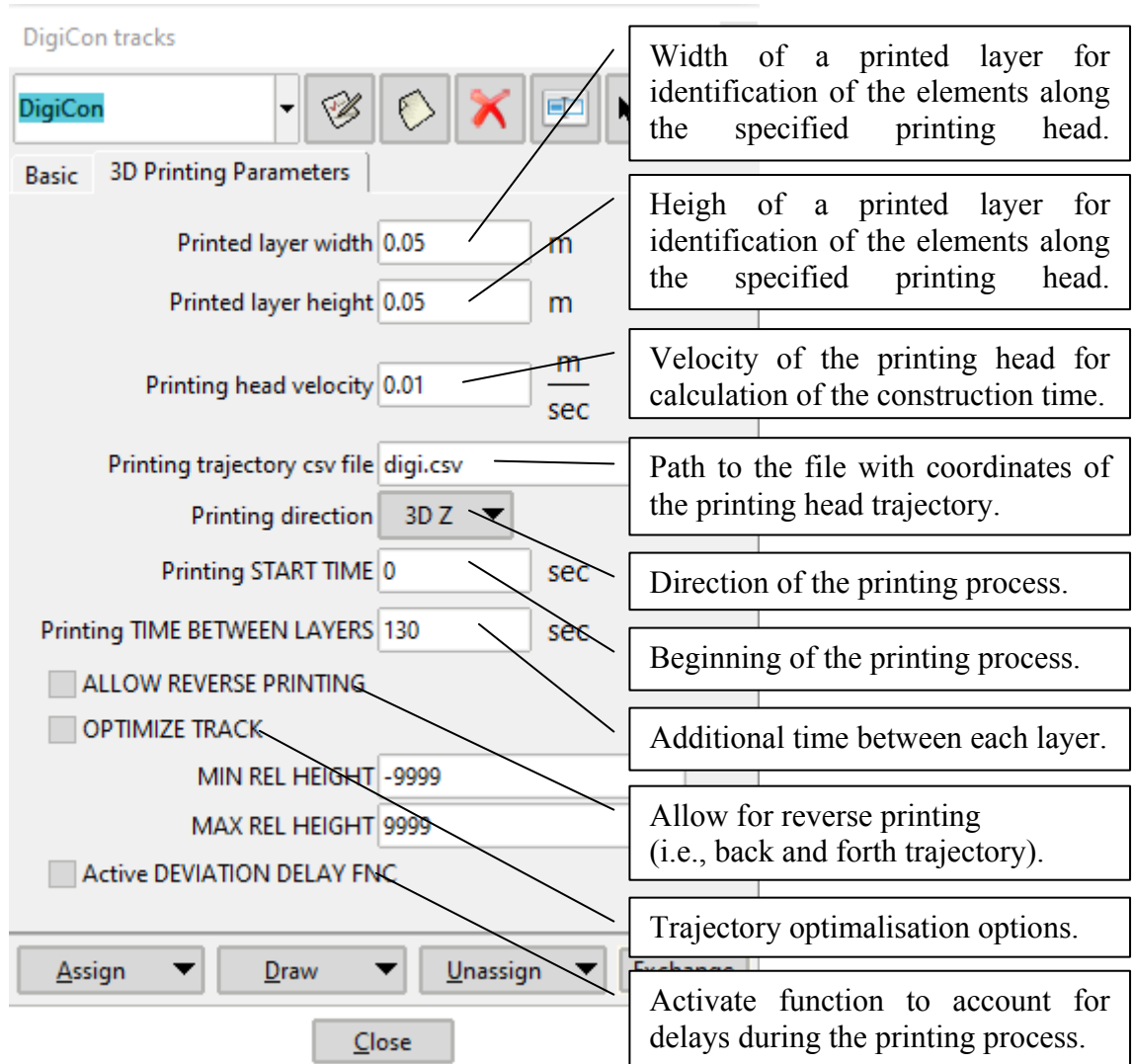


Fig. 5-105 Inputs for computation of elements' construction time

The printing head trajectory is given through a separate data file. It is important to specify this trajectory using the coordinates of the nodes at the **right bottom edge** of the elements in the bottom layer of the model as shown in Fig. 5-106. The nodes' coordinates should be separated by a semicolon and, by default, the file is expected to be placed inside of the folder with the GiD model.

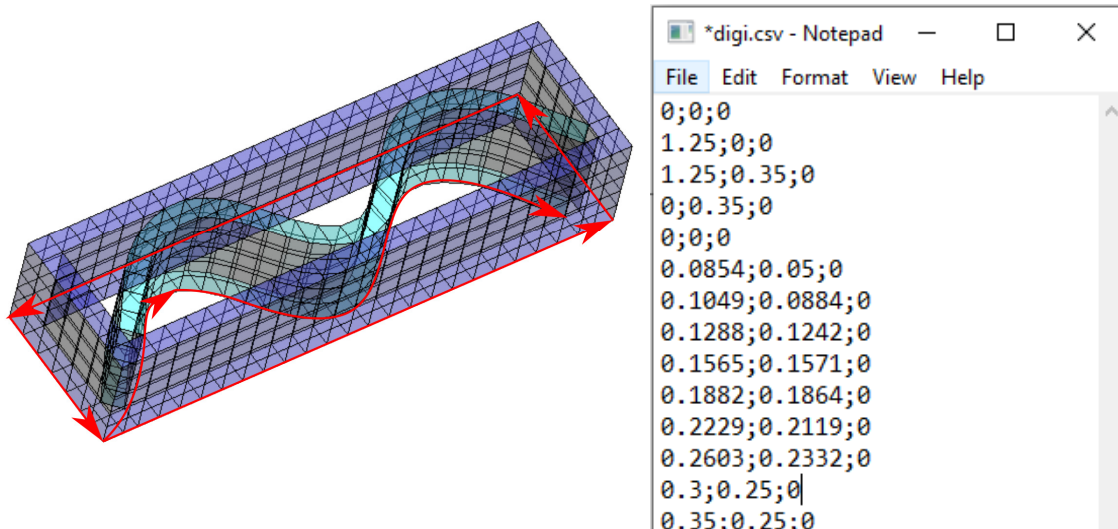


Fig. 5-106 Input of the printing head trajectory

It should be noted that if the element is not located on the printing track (i.e, the printing trajectory does not pass around the element or is not within the printed layer width), the element will not be activated.

If a certain part of the model should be activated at a given, user-defined time regardless of the settings of the printing head, the **Construction time** option can be used. This is useful for example for specifying various elements, which support the newly-printed structures, or other structural parts of the model such as lintels or steel reinforcement.

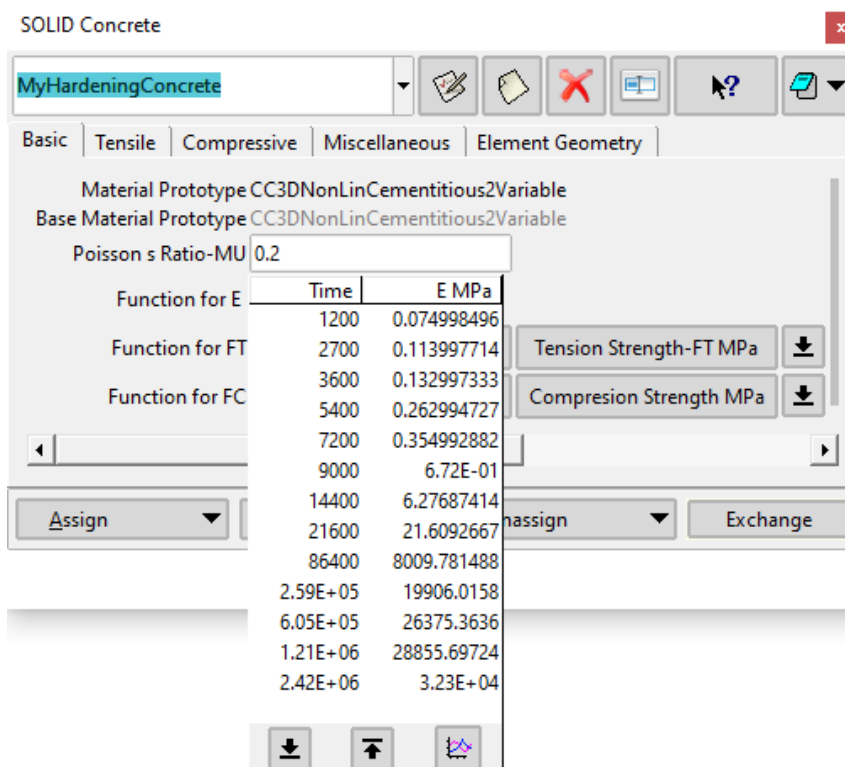


Fig. 5-107 Specification of the time development of elastic modulus

When it comes to material definition, **Cementitious2Variable** material model should be used as it allows the definition of development of material properties with time. The time-dependent development can be set for the elastic modulus, tensile and compression strength, fracture energy, plastic strain at compression peak load, compression ductility, and onset of concrete crushing. An example of such a definition is given in Fig. 5-107. For the details about his model, please refer to our Theory manual [1]. It is worth mentioning that time 0 seconds in the material model corresponds to the moment when the element appears (i.e., is activated) in the model.

The relationship between the printing time and computational step is part of the interval definition, which is open through **Data | Interval data**. Upon activation of the option **Digicon settings** in the interval setting, a new tab labeled **Digicon** appears, where the time duration of each computational step is defined as shown in Fig. 5-108.

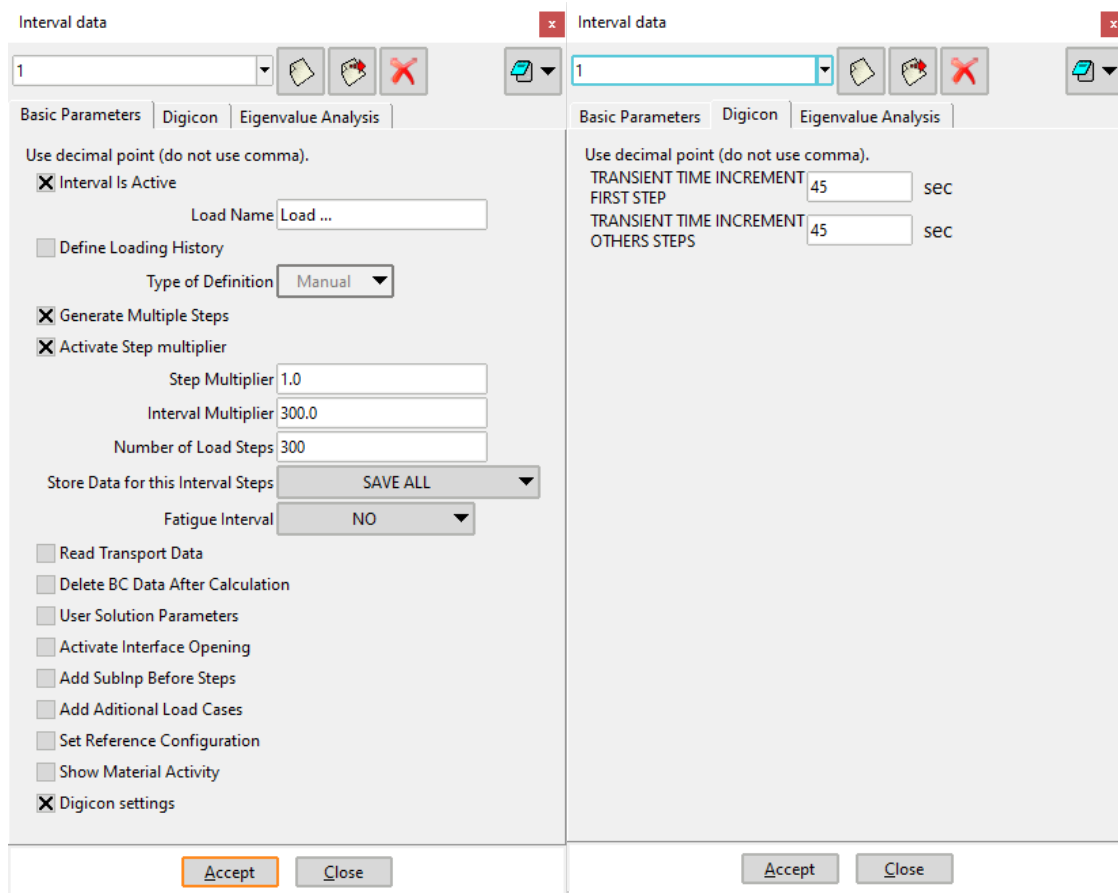


Fig. 5-108 Interval settings for 3D printing

5.7 Units

Standard units in **ATENA** are SI units, which are activated automatically as a default unit set, Fig. 5-109. It is also possible to select 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 consistent with the **Units System**.

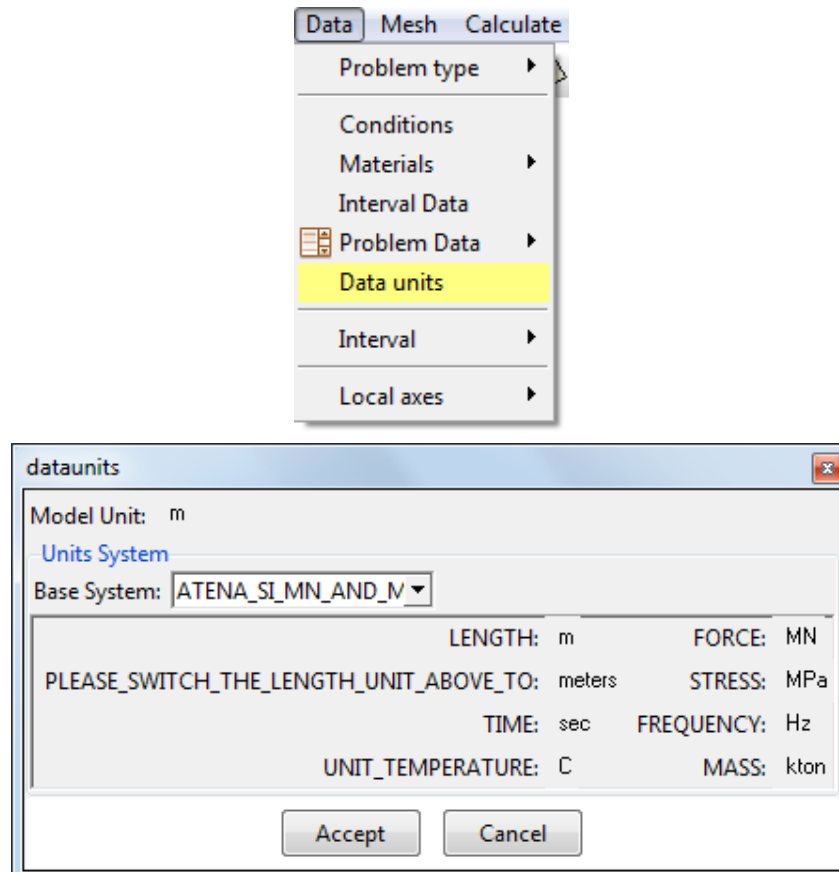


Fig. 5-109 Data units, default set.

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 the 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.

Basically, anywhere you see a unit behind a value in **GiD**, you can click on the unit and choose another one, and the value gets converted to the newly selected unit.

When the model is passed from **GiD** to **ATENA** for analysis, all values are converted to the units from the Base System selected in **Data - Data units**. Therefore, even if your analysis runs in metric system, you can define some material or other values using for example **lbf** (however in post-processing, you then have everything in metric units - and vice versa, if you set **GiD** to American, you can define some values in **MN**, but see everything in American units in post-processing).

The default units in all materials and other dialogs are defined in metric units even if another unit system is selected in **Data - Data units**. This means when another unit set is selected, the default values will be converted, which may result in some rounding errors or numbers with many decimal points.

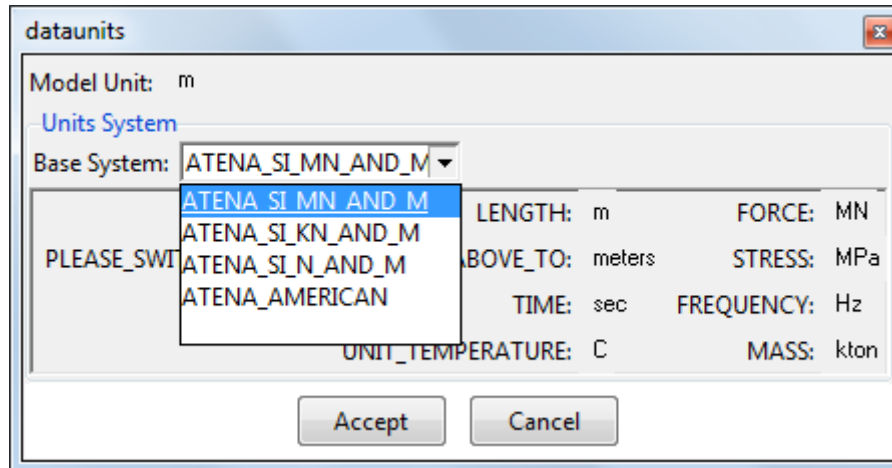


Fig. 5-110 Definition of units and possible set of alternative units.

5.8 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 modeled by one-dimensional entities, i.e., lines. Since **GiD** does not have the 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 21). 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 their 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).

5.8.1 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).


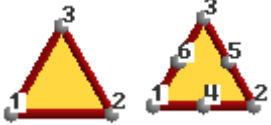
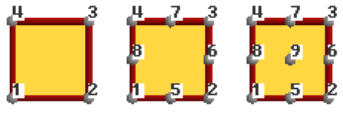

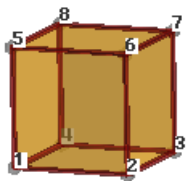
Another frequent example of a problematic mesh are elements with extreme aspect ratios, in other words, the ratio of element edge lengths = the longest to the shortest edge of an element. A maximum of 3:1-4:1 is recommended for volume elements and also for surface elements in 2D models or on membranes. 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).

5.8.2 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**).

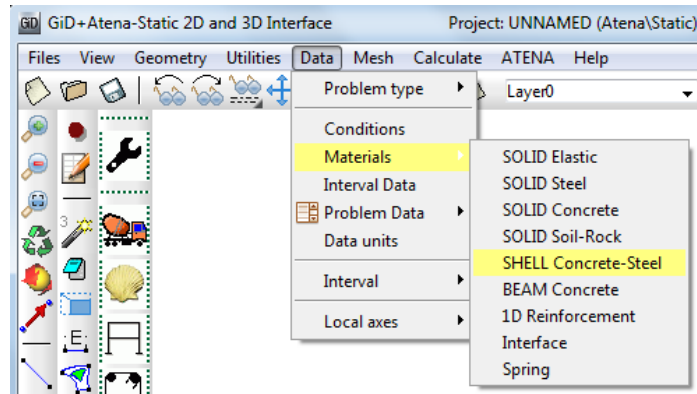
When combining linear and quadratic elements in a model using the option from 5.3.2.2, please note you can not have a shared surface (or a shared line for 2-dimensional elements) to get mesh compatibility between neighboring elements of different degrees. Instead, use Master-Slave connections (Fixed Contact, see also 5.2).

Table 3: Element library compatibility

| | |
|---|---|
|  | <p><i>Linear and quadratic</i> line element 2-nodes, CCIsoTruss<xx> 3-nodes, CCIsoTruss<xxx>)</p> |
|  | <p><i>Linear and quadratic</i> triangular element 3-nodes, CCIsoTriangle<xxx> 6-nodes, CCIsoTriangle<xxxxxxx>)</p> |
|  | <p><i>Linear and quadratic</i> quadrilateral elements 4-nodes, CCIsoQuad<xxxx> 8-nodes, CCIsoQuad<xxxxxxxx> 9-nodes, CCIsoQuad<xxxxxxxxx>)</p> |
|  | <p><i>Linear and quadratic</i> tetrahedral elements 4-nodes, CCIsoTetra<xxxx> 10-nodes, CCIsoTetra<xxxxxxxxxx>)</p> |
|  | <p><i>Linear and quadratic Hexahedron</i> (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</p> |

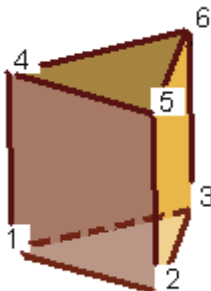
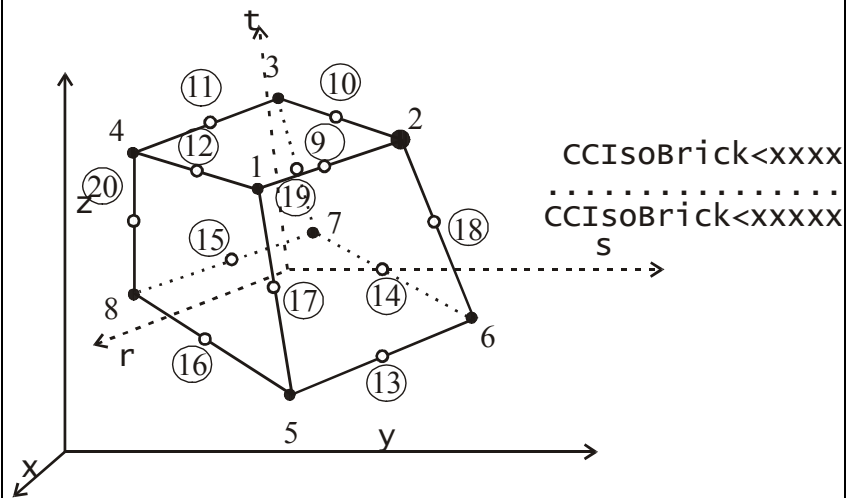


complex structures, when large elements are needed, such as bridges, slabs, etc. The shell element is activated by assigning the Shell material to 20-node brick elements.



20-nodes, CCBBeamNL – 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 analyses, 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.

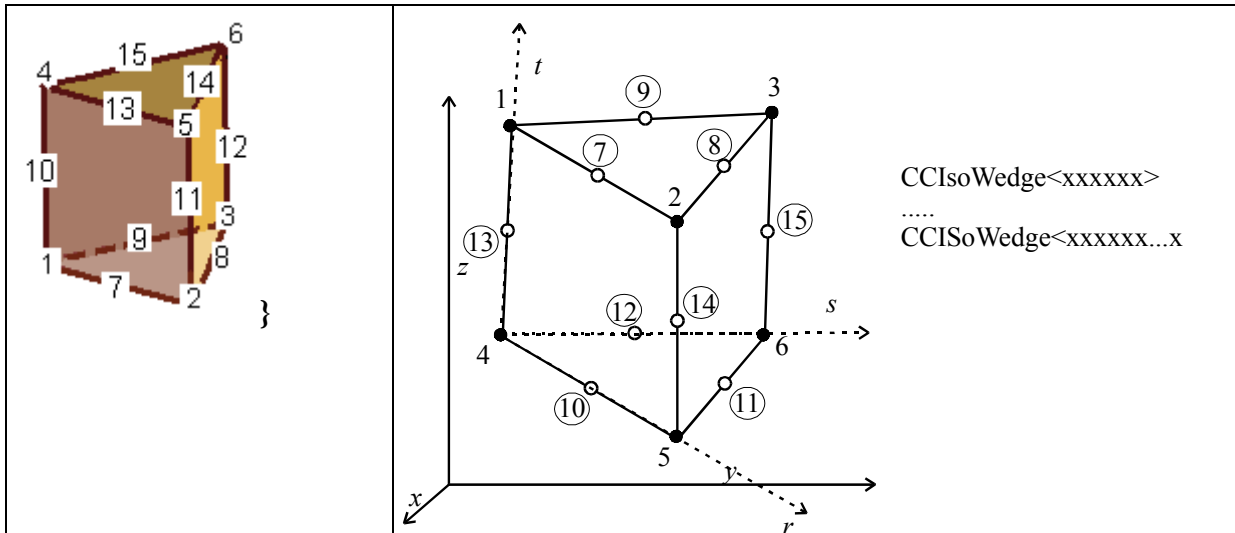


Linear and quadratic Wedge (structured mesh)

6-nodes, CCIsoWedge<xxxxxx>

15-nodes, CCIsoWedge<xxxxxxxxxxxxxxxx>

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.

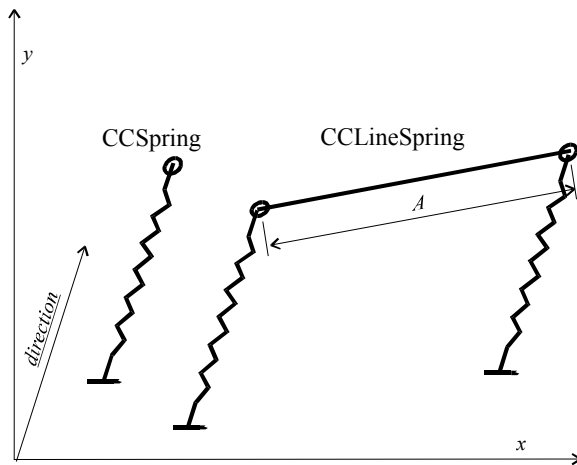


Spring

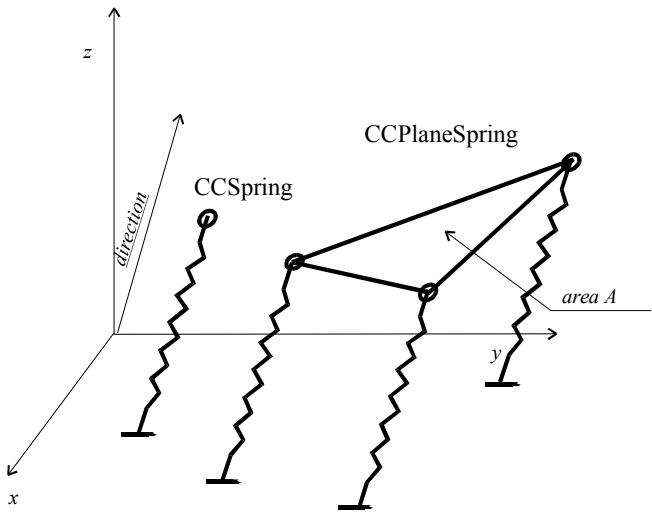
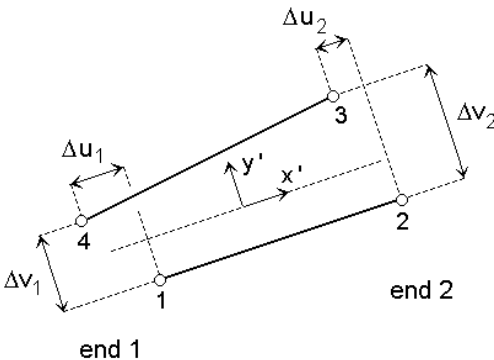
In ATENA-GiD interface, it is possible to model springs in two ways. Either by generating elements along a line or surface and then by assigning them a Spring material property. Alternative approach is by prescribing springs as conditions using the Data | Conditions menu. With the second approach it 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.

| | |
|------------------|---|
| |  |
| <p>Interface</p> | <p>2D line 4 node interface - CCIsoGap<xxxx>)</p>  <p>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></p> |

5.9 ATENA Menu

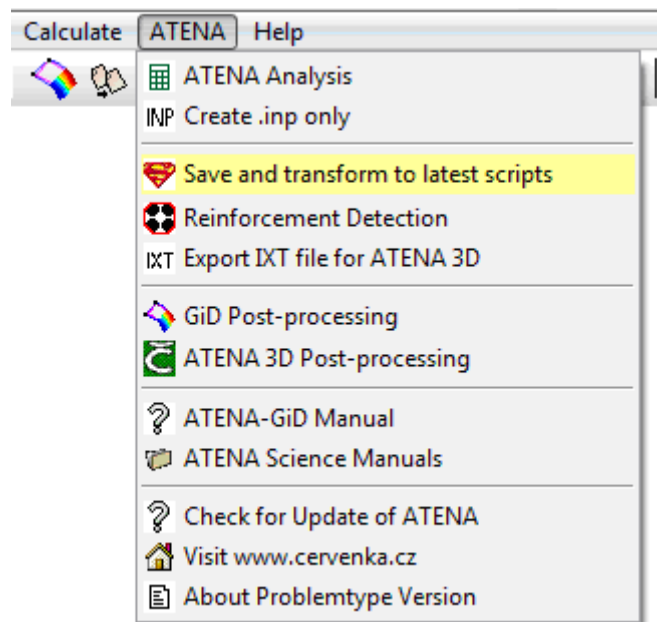


Fig. 5-111 ATENA menu in GiD

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 Elms 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 the model it's the same as others problem data. It's necessary to assign Conditions [5.2], for each macro element assign material properties [5.3], define the interval data [5.4] and problem type properties [Fig. 5-102], meshing model [5.8] and execute the analysis by 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. 6-112.

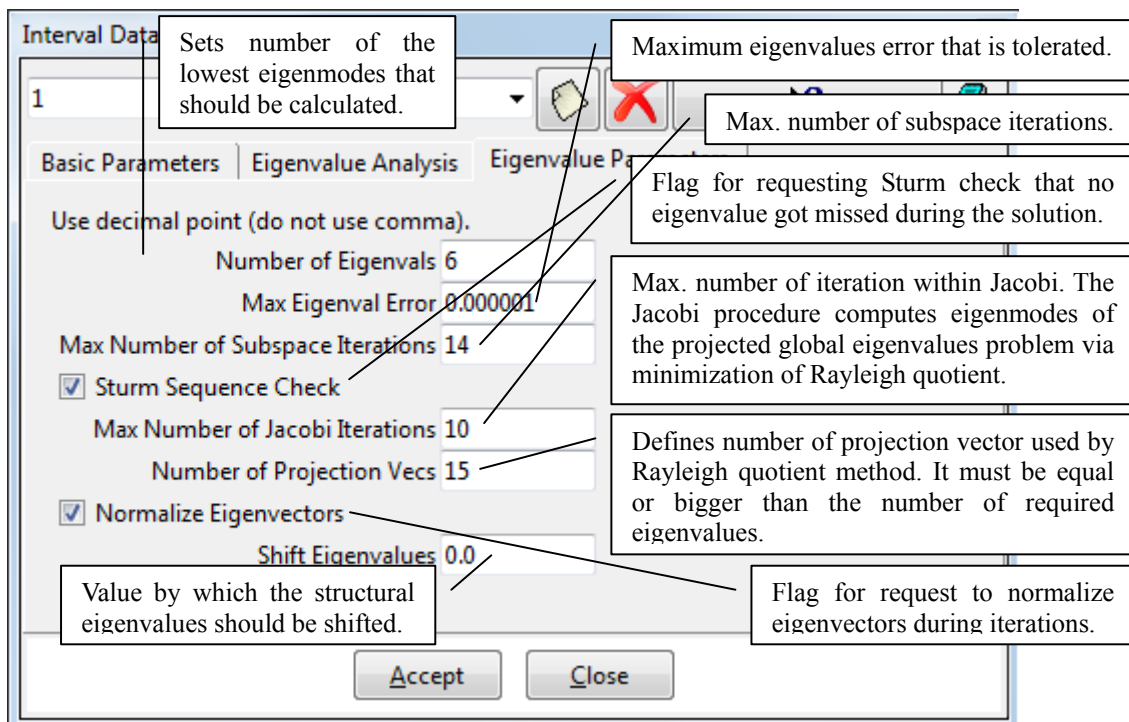


Fig. 6-112: Settings of EigenValue Parameters

A 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 the 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 the 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 at 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) points. 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 the 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. The time factor also appears in the constitutive equations (i.e., material models). This is done by implementing models for the prediction of creep and shrinkage behavior 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 analyzed. It involves time integration of structural behavior, which is done in a 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 the exponential character of concrete creep and shrinkage behavior, and the 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 the accuracy of individual approximations involved in the creep and shrinkage analysis. The remaining data sheets of this dialog are the same as for the usual static analysis.

In order to decrease the memory requirements, the average values for humidity/temperature can be used for multiple elements with same/similar history. The user needs to specify the number of different humidity/temperature groups in the **Transport** window. If the number is equal 1, this means that all elements in the structure will share the same (humidity/temperature) history. If the number is for instance 10, it means the elements in the model based on their histories will be divided into 10 groups sharing the same average history.

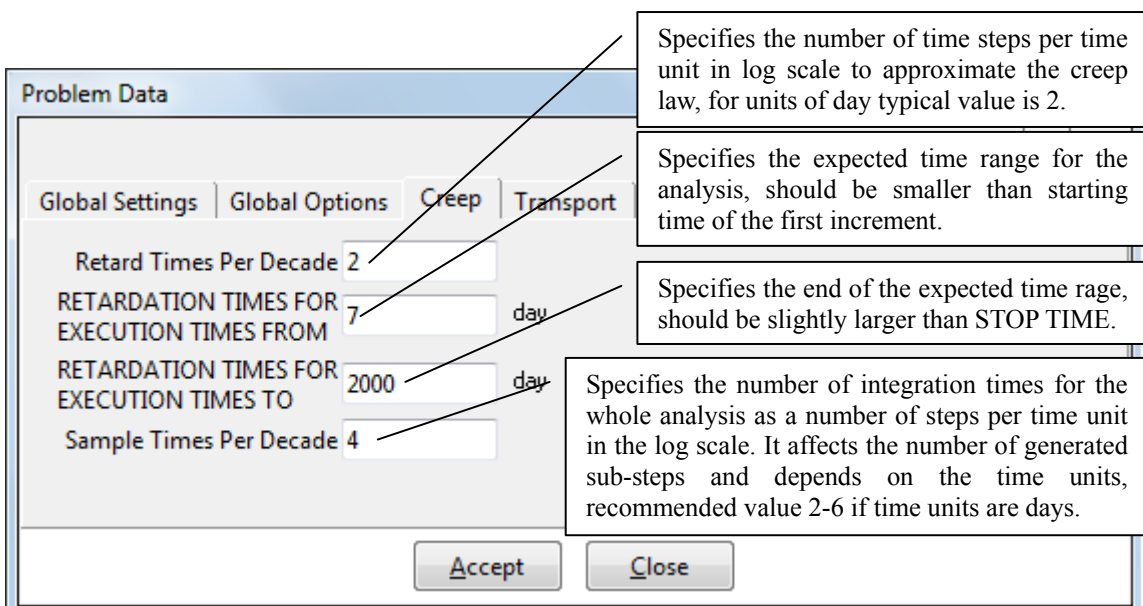


Fig. 7-113 Problem Data dialog.

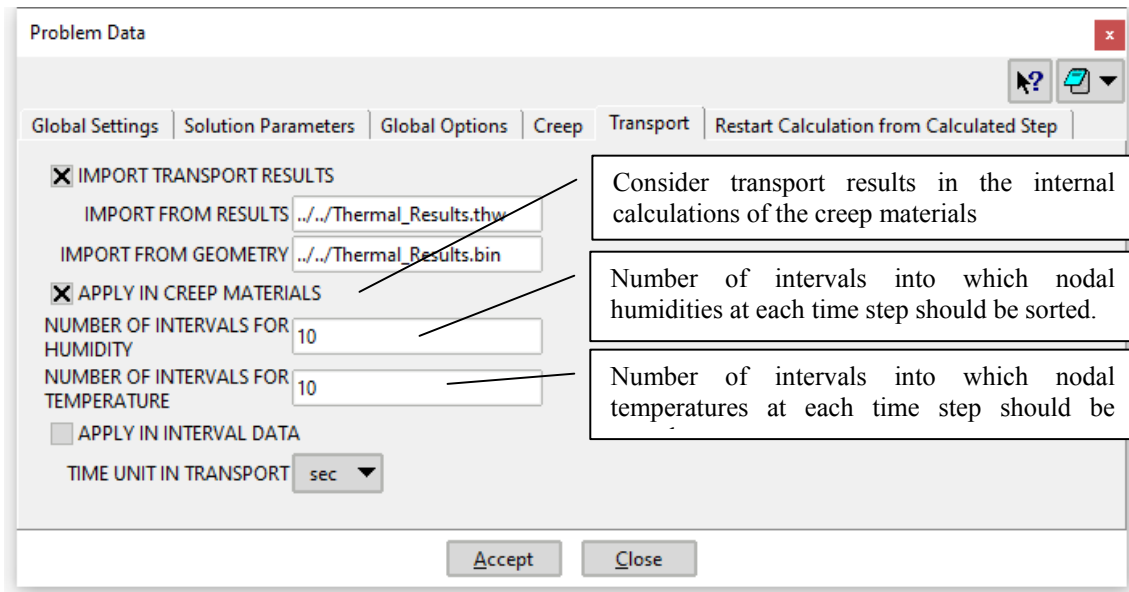



Fig. 7-114 Problem Data dialog - Transport.

7.1 Boundary Conditions and Load Cases Related Input

The essential part of any FEM analysis is to set correct boundary conditions for the analyzed 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. 7-113).

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 as 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 set is assigned a dedicated data sheet. The actual data input dialog is invoked

by pressing the icon  (or via menu **Data | Materials | Creep**), and it is shown in Fig. 7-115

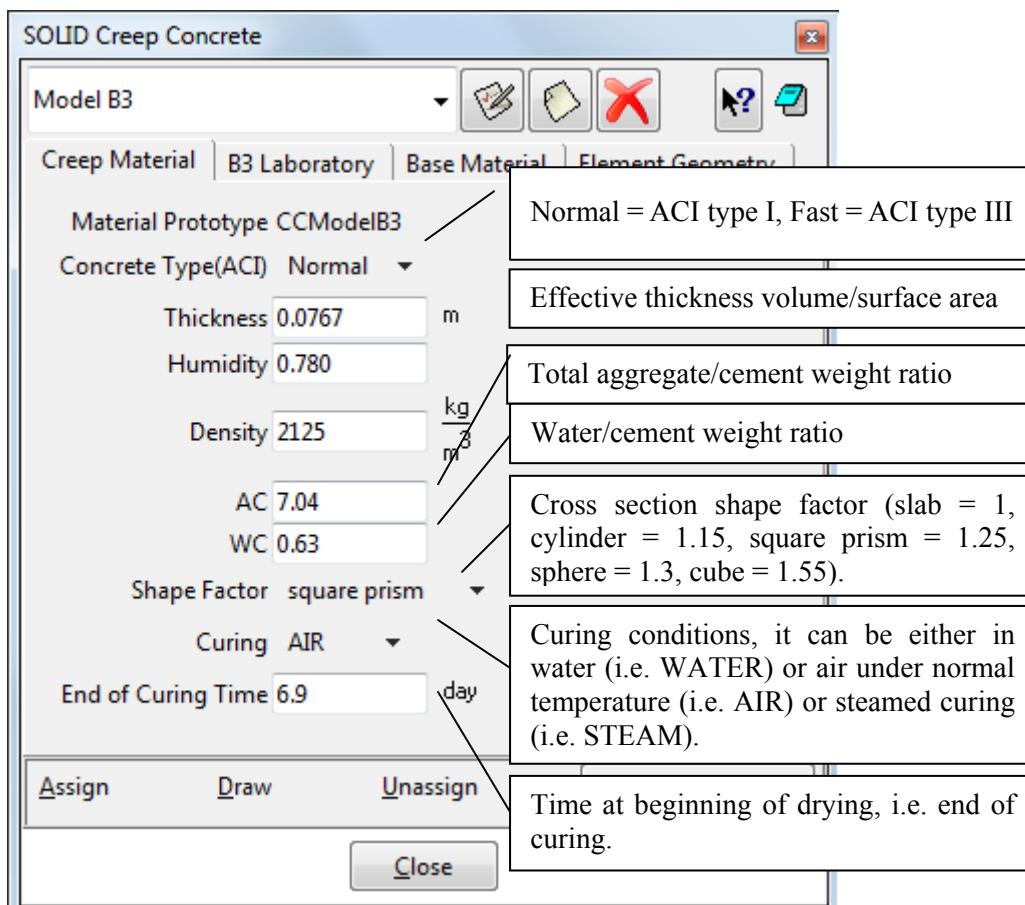



Fig. 7-115 Material input dialog

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 a short-term model for smeared reinforcement. This type of input data in **GiD** is still in the 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:

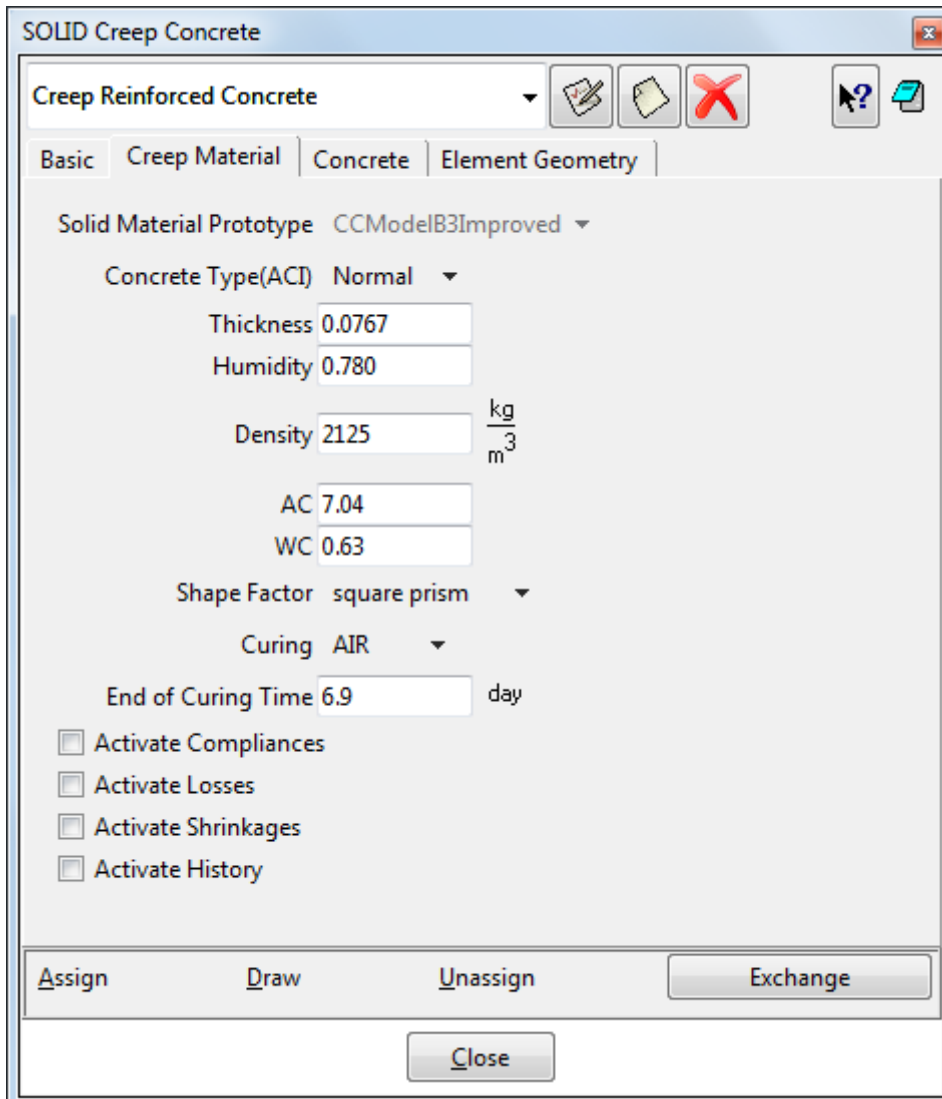




Fig. 7-116 Reinforced concrete material with smeared reinforcement

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 the 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.

For using material CCMoelGeneral (creep model for direct input of material compliance, strength and shrinkage at times typically measured in a laboratory), please see 5.3.9 Material from file.

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.

The 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 anymore. The

corresponding input data dialog appears by pressing the icon :

8.1.1 Material **CCTransport** (**CERHYD**)

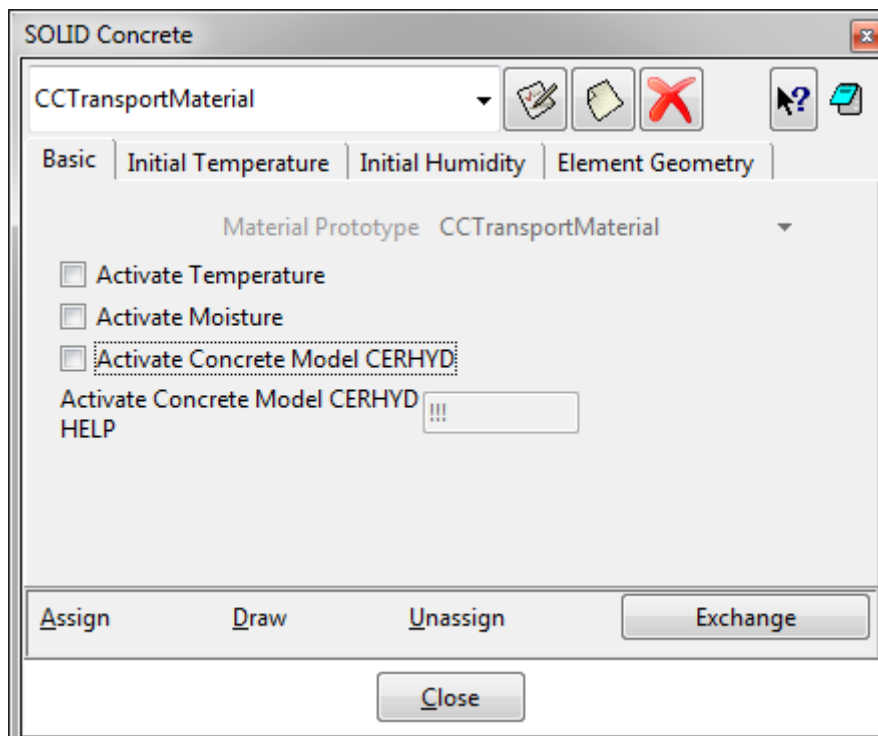


Fig. 8-117 Heat and moisture transport material model dialog

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 laboratory-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.

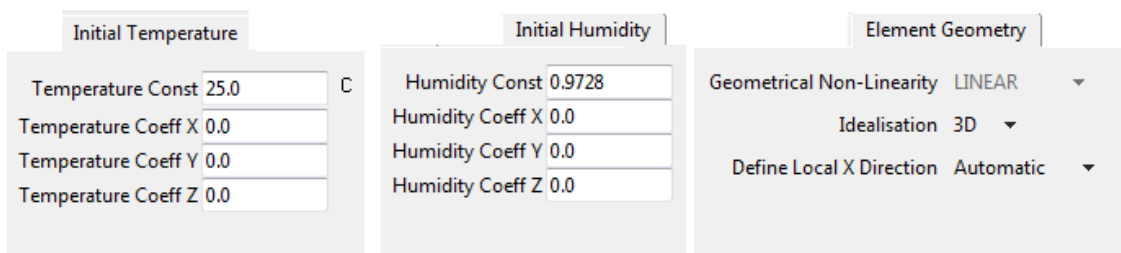


Fig. 8-118 Transport Material - Initial Temperature and Humidity Dialog

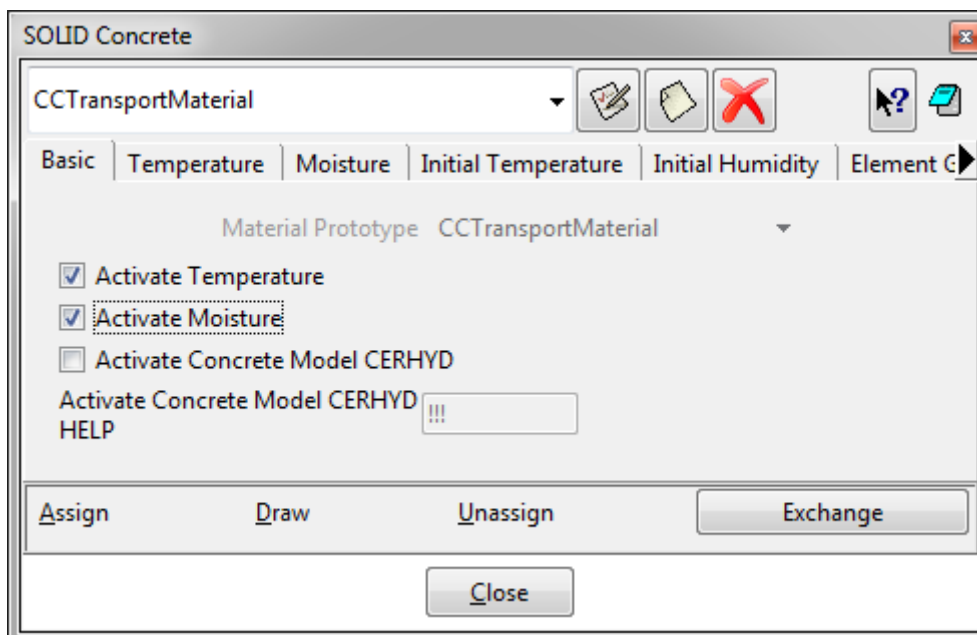


Fig. 8-119 Transport Material – Activate Options

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

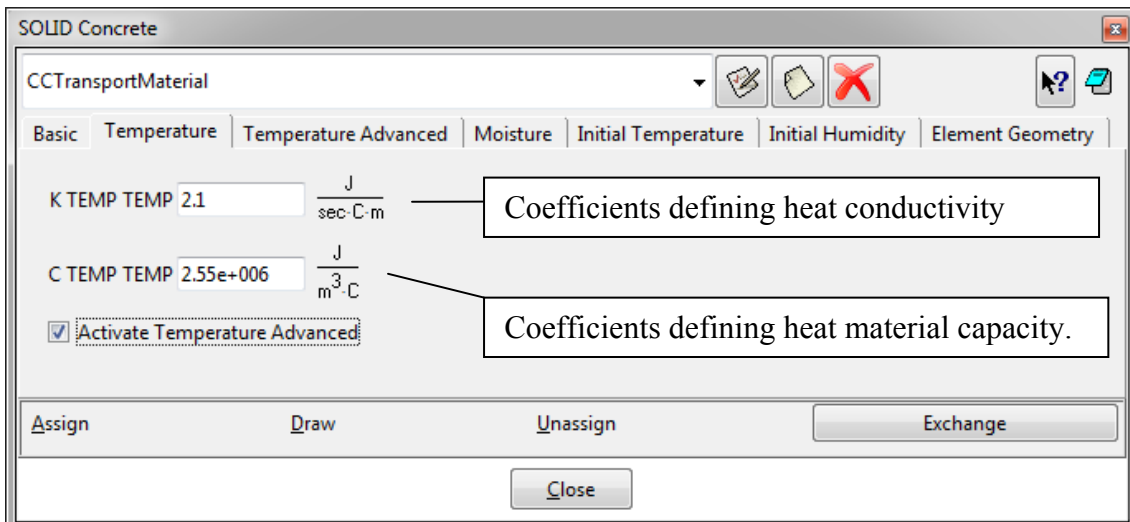


Fig. 8-120 Transport Material – Temperature

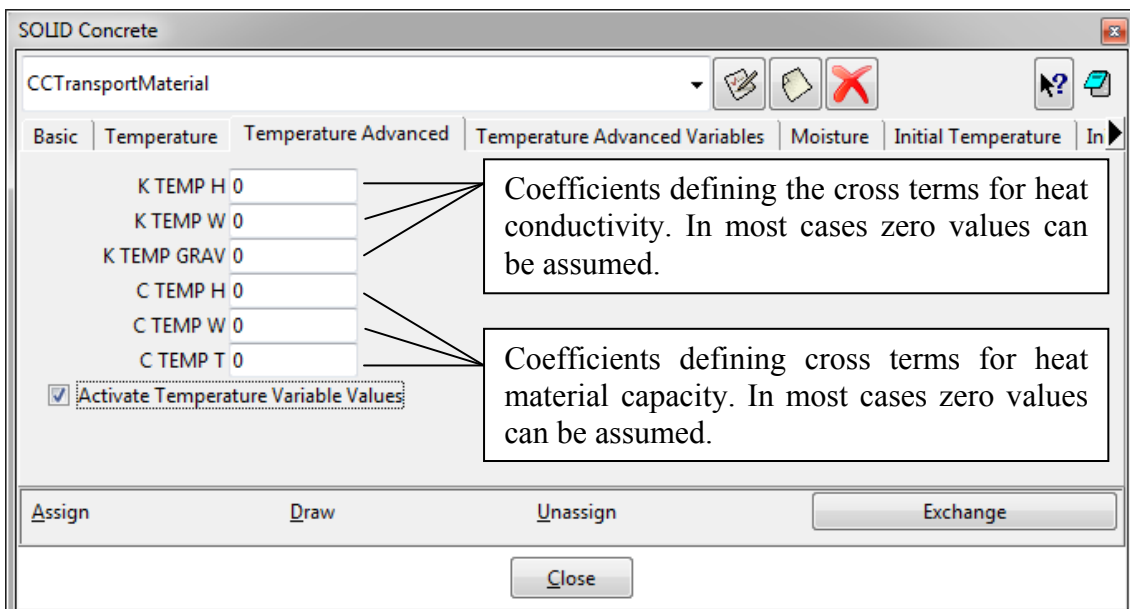


Fig. 8-121 Transport Material – Temperature Advanced options

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. 8-122).

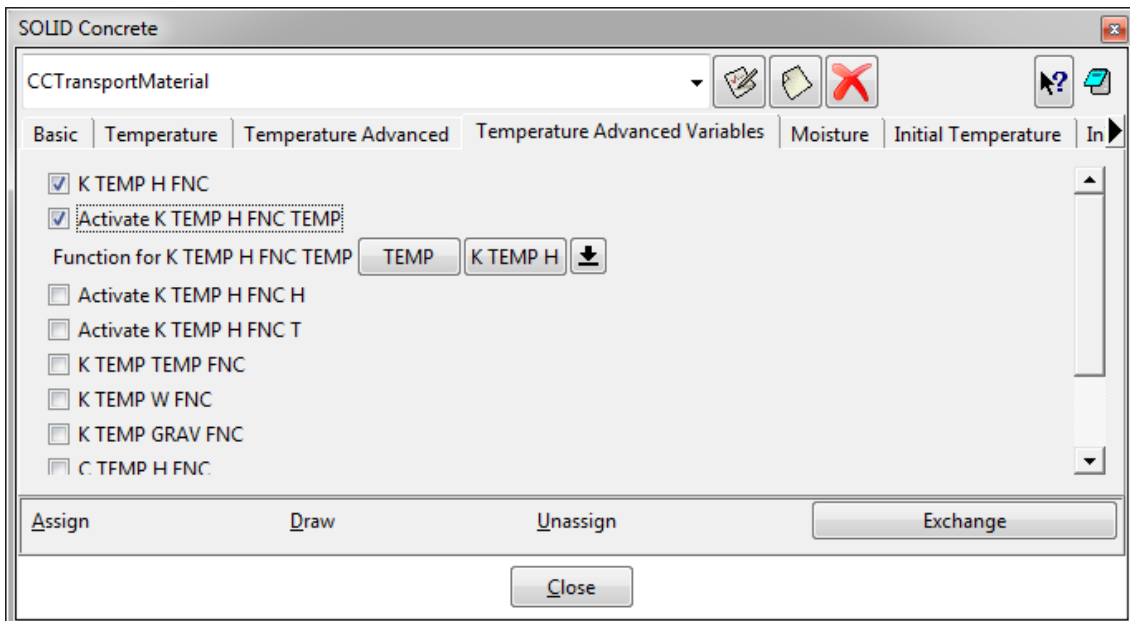


Fig. 8-122 Transport Material – Advanced variables, Activation of Nonlinear Functions

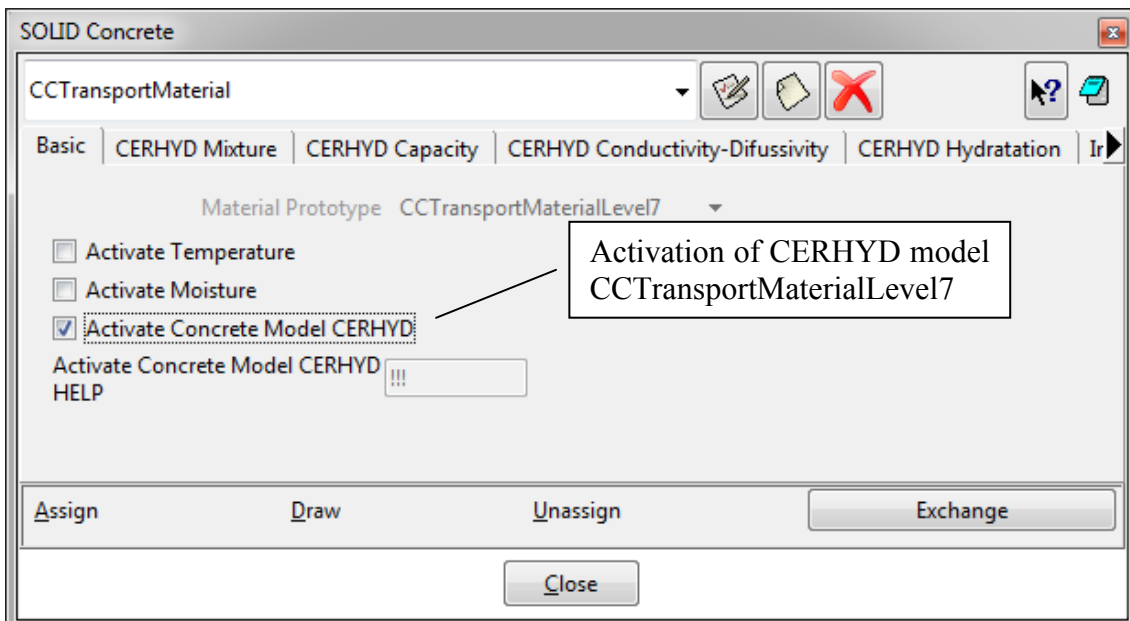


Fig. 8-123 Transport Material – CERHYD Model

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 the 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.

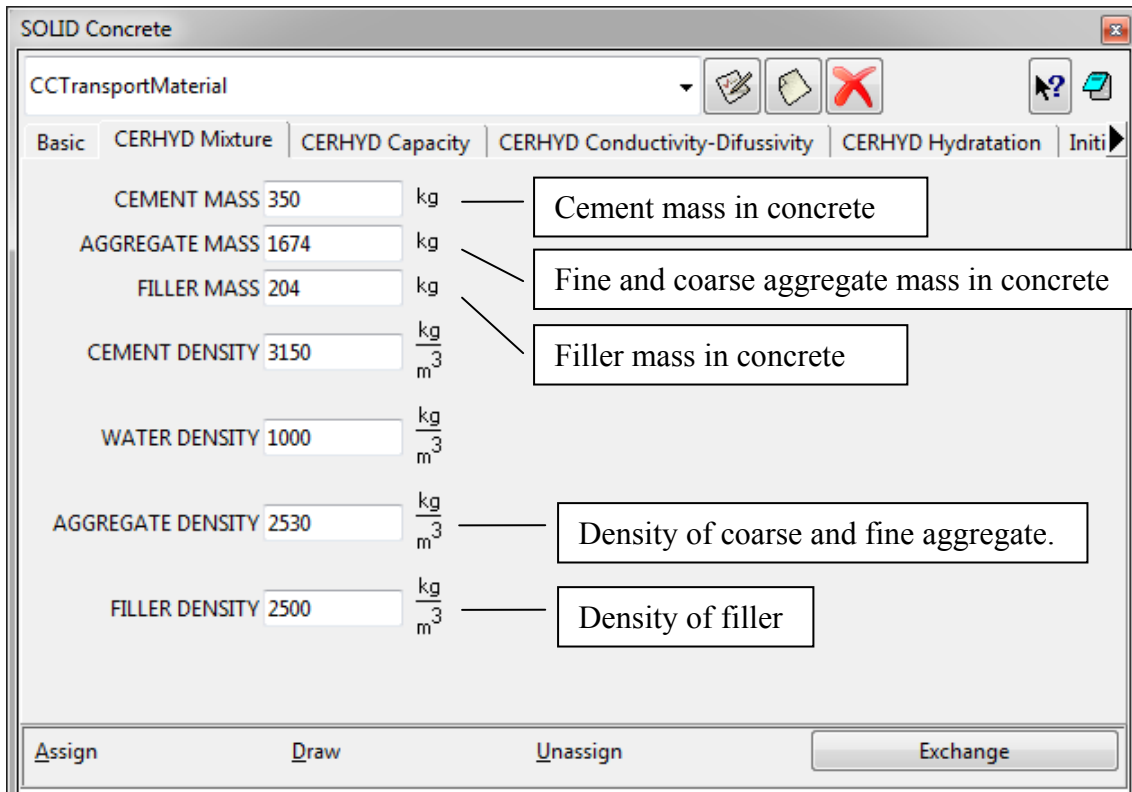


Fig. 8-124 Transport Material – CERHYD Mixture

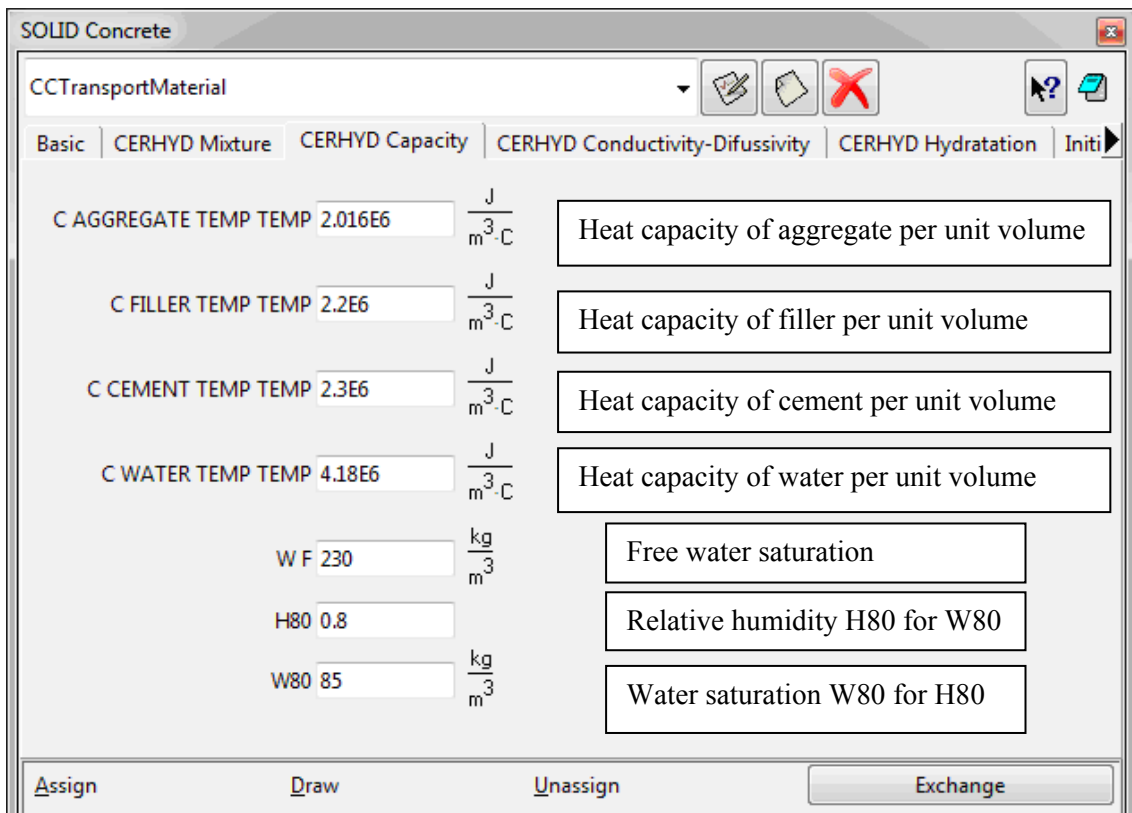


Fig. 8-125 Transport Material – CERHYD Capacity

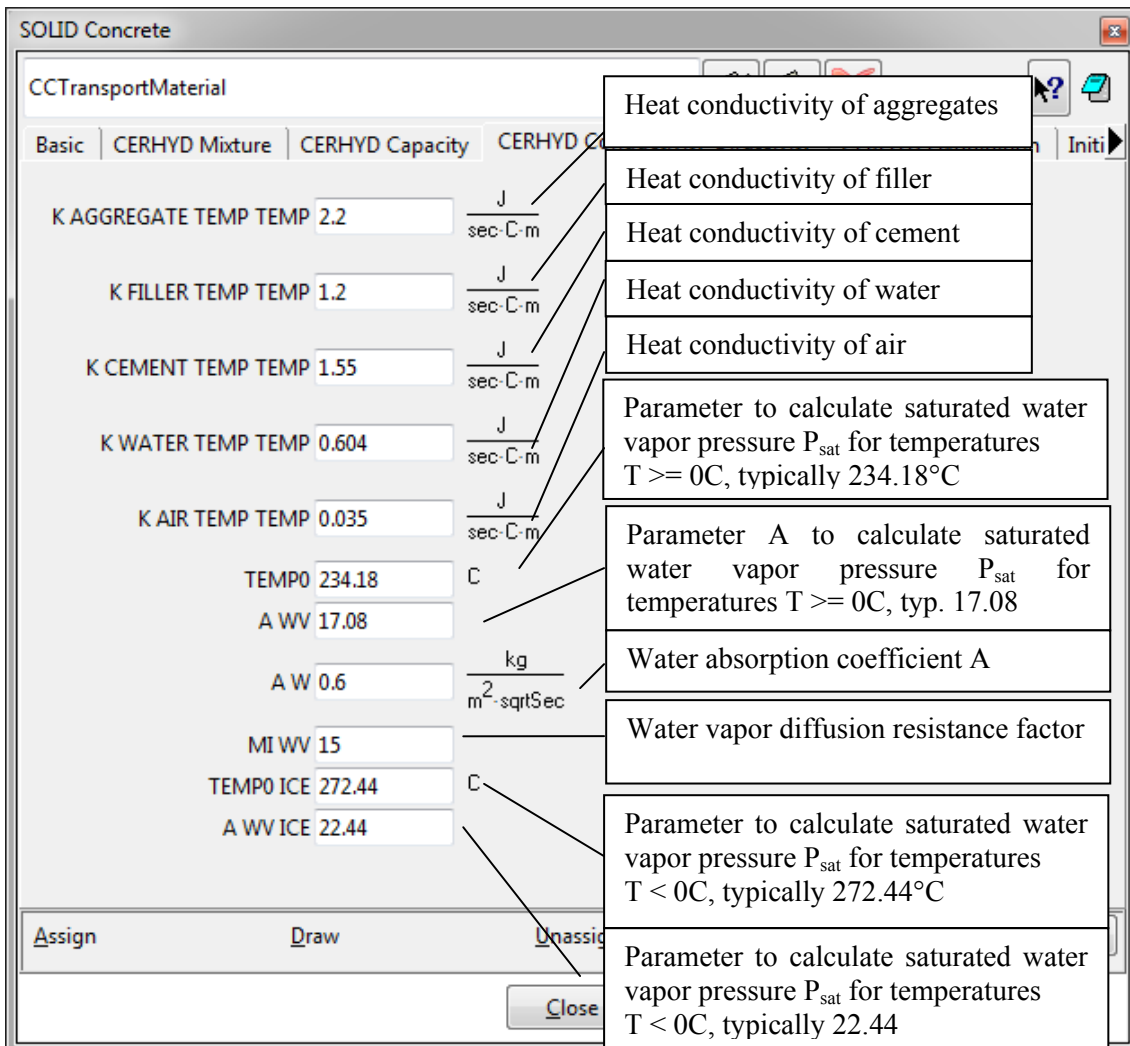


Fig. 8-126 Transport Material – CERHYD – Conductivity-Diffusivity

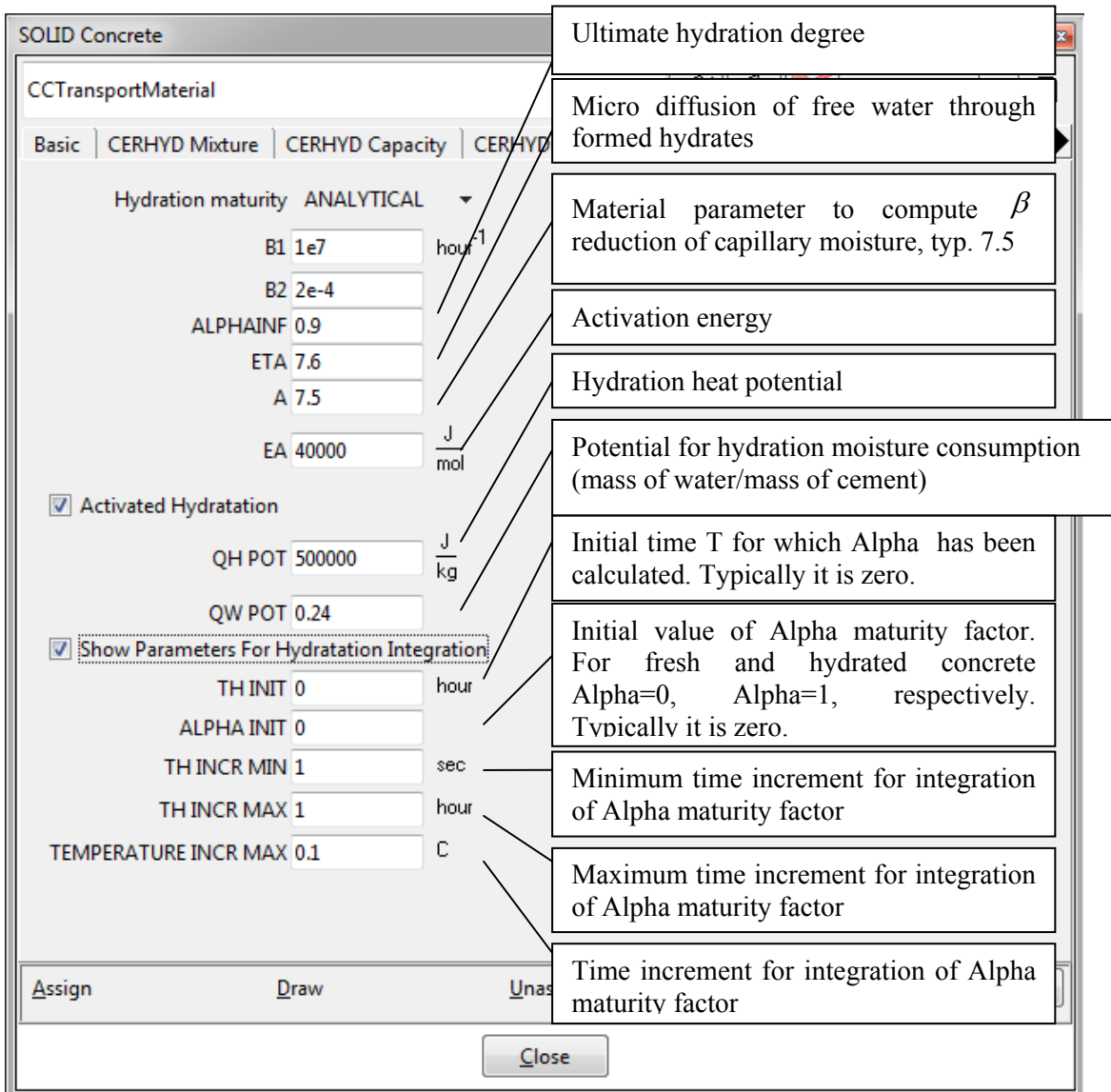


Fig. 8-127 Transport Material – CERHYD – Hydration

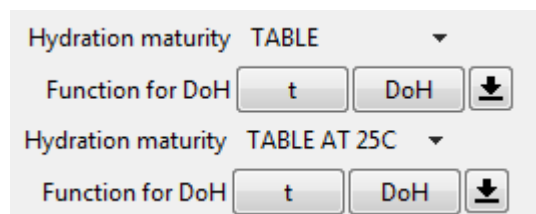


Fig. 8-128 Transport Material – CERHYD – Hydration maturity options

8.1.2 Generating material characteristics by ConMix

For different concrete mixtures, the inputs parameters for CERHYD can be automatically generated using the ConMix tool. It should be noted that ConMix can be used separately outside of the GiD environment to generate concrete mixture characteristics for various applications.

From GiD, the ConMix tool can be launched from the material definition, which is opened through **Data | Materials | SOLID Material**. Then, in the **Basic** tab, the type of CERHYD definition should be chosen as **FILE**, and the checkbox Run ConMix for CERHYD should be selected as shown in Fig. 8-129.

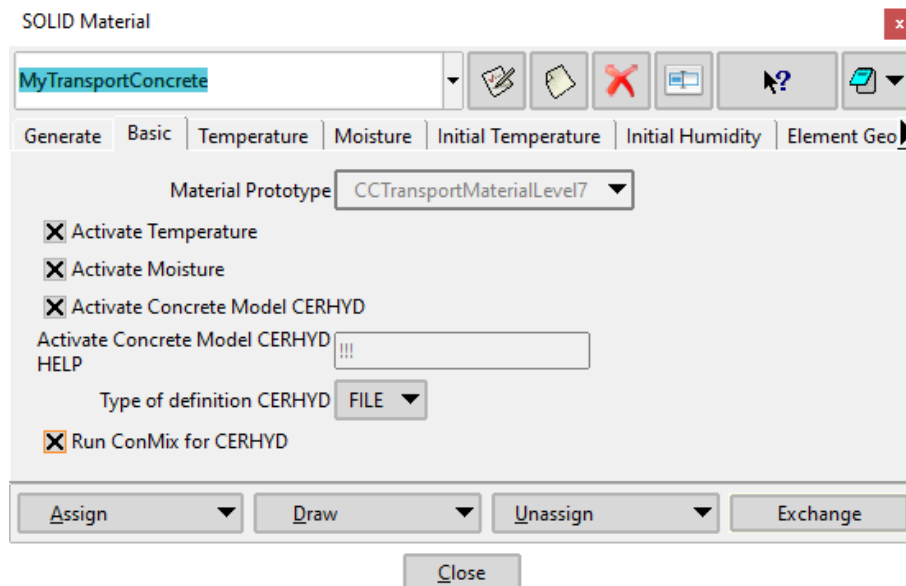



Fig. 8-129 Launching of ConMix

Upon clicking the update changes button , ConMix is automatically launched. The ConMix tool has the following structure:

- **Units** – unit system for ConMix,
- **Concrete** – concrete class and mixture proportions,
- **Cement** – characteristics of the cement and its hydration function,
- **Params** – specification of the fundamental material characteristics,
- **Output** – summary of the generated parameters.

It is important to note that the units specified in ConMix must match the unit system chosen in GiD (given in **Data | Data Units**). In the **Concrete** tab, the first approximation of the mixture proportion can be made based on the concrete strength class. The mixture proportions, such as water-to-cement ratio or cement and aggregate contents can be further adjusted. Furthermore, the cement replacement by supplementary cementitious materials can be specified. After any adjustment, **Normalize** button should be clicked to recalculate the volumes of each component in the unit volumed based on the specified masses and densities. Details of the **Concrete** tab are shown in Fig. 8-130.

| | Mass [kTon] | Dens. [kTon/m3] | Volume [m3] |
|--------------------------|-------------|-----------------|-------------|
| Cement (clinker+gypsum): | 0.00014 | 0.00322 | 0.04348 |
| Cement (SCM): | 6e-05 | 0.0024 | 0.025 |
| Water: | 0.00011 | 0.001 | 0.11 |
| Aggregate: | 0.002161 | 0.0028 | 0.7718 |
| Filler: | 0 | 0.0027 | 0 |
| Air: | | | 0.05 |
| Total: | 0.002471 | 0.002471 | 1 |

Fig. 8-130 Mixture proportions specification in ConMix

The development of the cement hydration (α) is defined in the **Cement** tab. The user can either choose from the database containing several cement types or specify the hydration curve by himself. The shape of the function is affected by parameters B1, B2, Eta, and the ultimate degree of hydration is given by α_{∞} . After any change of these parameters, the shape of the function should be recalculated by clicking on **Calculate cement hydration** button. It should be noted, that the hydration curve is specified under isothermal temperature conditions of 25 °C. If the temperature differs during the calculation, the rate of hydration is automatically adjusted based on the specified Activation energy by applying the Arrhenius law. The total heat released due to the hydration process is given by the Potential heat parameter. The details of the **Cement** tab are shown in Fig. 8-131.

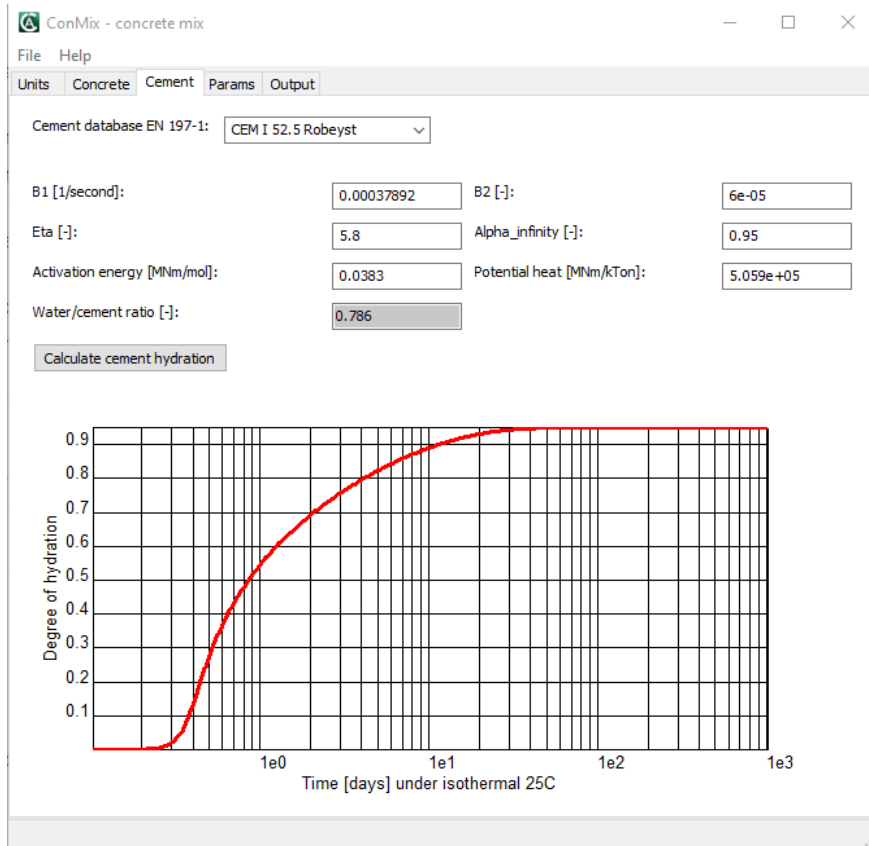


Fig. 8-131 Specification of cement hydration in ConMix

The transport properties, such as thermal conductivity and specific heat capacity of each component of the mixture are given in the **Params** tab as shown in Fig. 8-132. These characteristics can be either generated automatically based on the aggregate and supplementary cementitious types. Finally, the water content at 80 % relative humidity (w_{80}) and water absorption coefficient (A_w) are specified to define the water transport properties of the mixture.

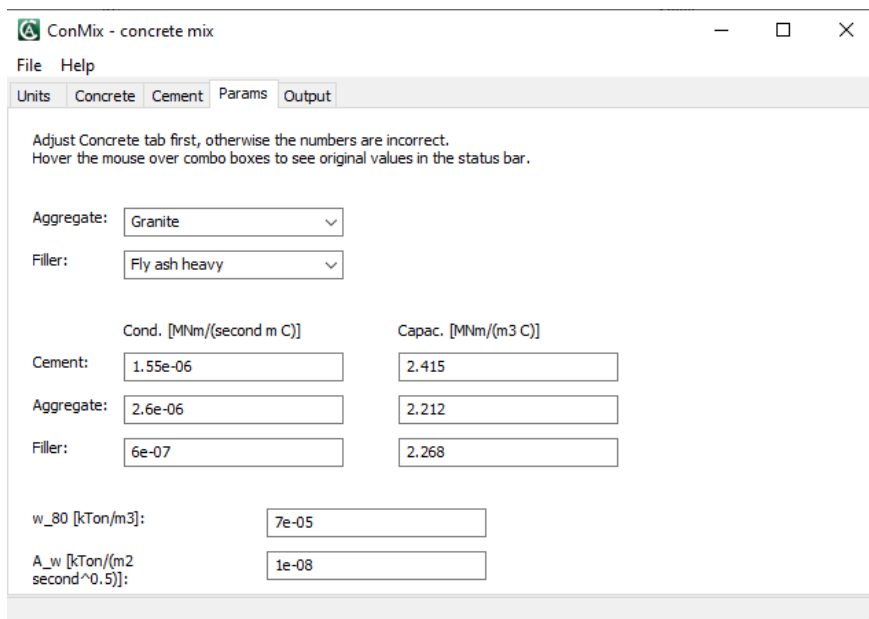


Fig. 8-132 Specification of basic transport properties in ConMix

In the last tab labeled **Output**, the results from ConMix can be reviewed in a form of ATENA input file. After closing the ConMix tool the outputs are stored in two *.inp files in the AtenaCalculation folder. Once the solution is initiated from GiD, the information commands from ConMix are automatically copied in the ATENA input file. The second *.inp file from ConMix (with suffix “_2”) contains the hydration curve of the cement as a list of values for time and hydration degree (Alpha).

8.1.3 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 modeling of mortar behavior. It accounts for water and cement paste only, and hence, in the 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. 8-133 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.

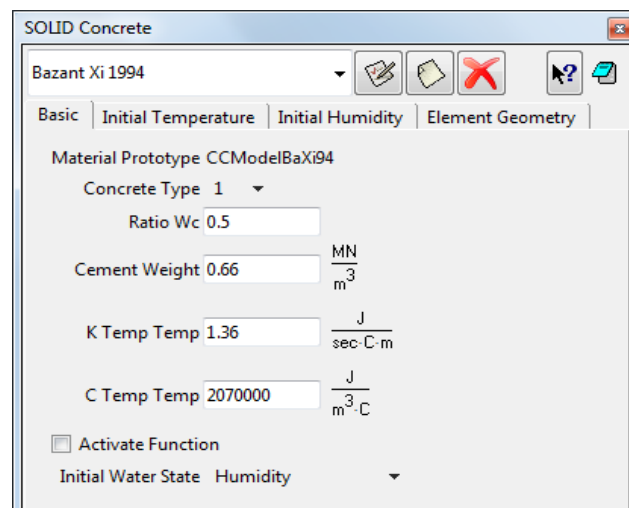


Fig. 8-133 Bazant_Xi_1994 material model dialog

Other Settings Related to Transport Analysis

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

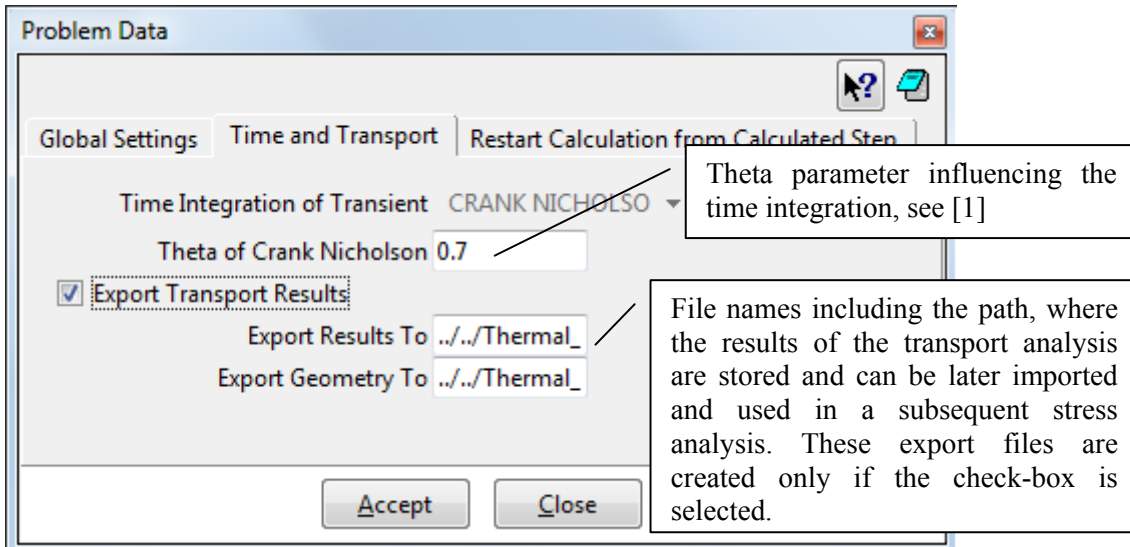




Fig. 8-134 Time and transport data sheet

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 the input 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 closed nodes and makes the necessary interpolation.

The dialog in Fig. 8-135 (available by pressing ) is used to define one or multiple execution type steps. The meaning of the parameters is self-explanatory and illustrated in Fig. 8-136, 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.

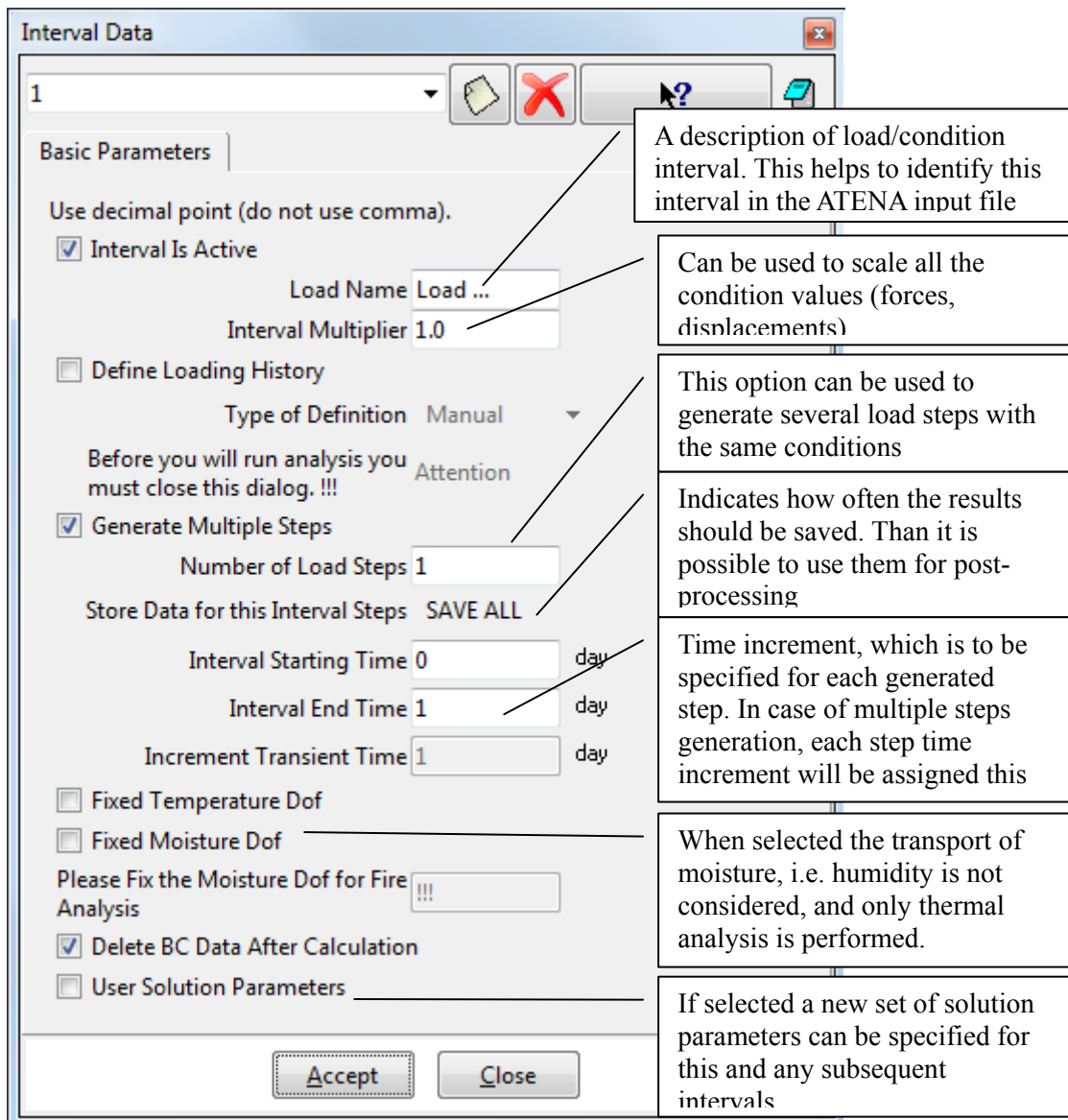


Fig. 8-135 Step data dialog

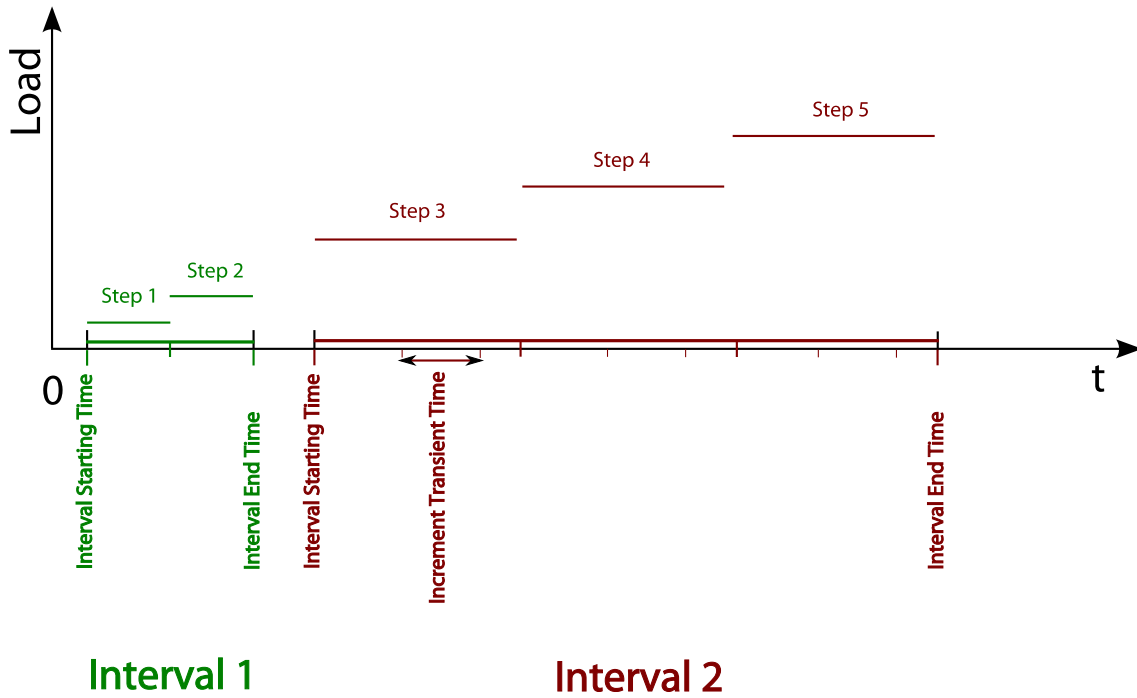


Fig. 8-136 Interval Data - time values

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).

8.2 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.

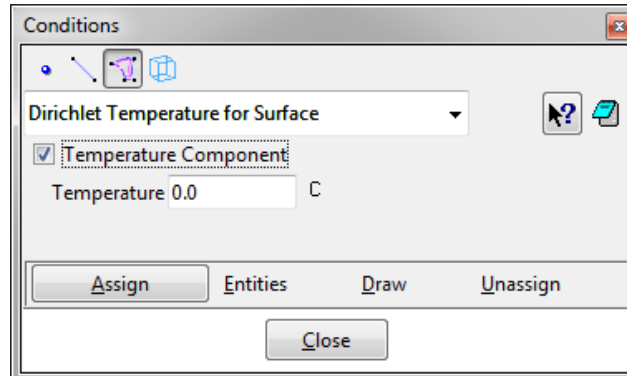


Fig. 8-137: Dirichlet temperature for ...

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

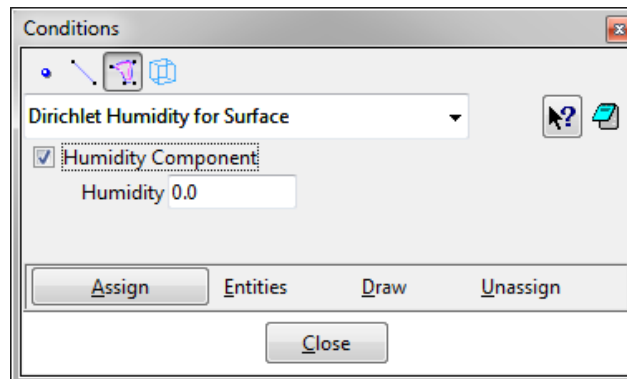


Fig. 8-138: Dirichlet humidity for ...

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

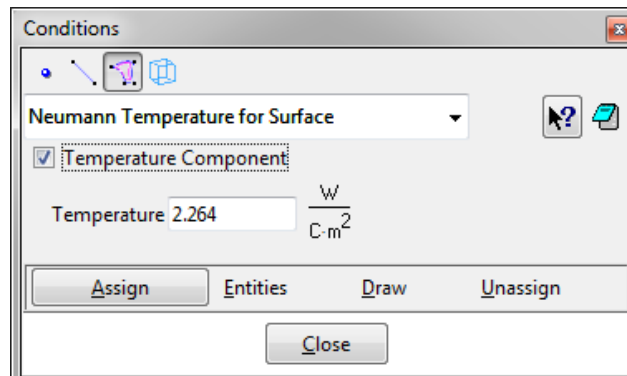


Fig. 8-139: Neumann temperature for ...

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

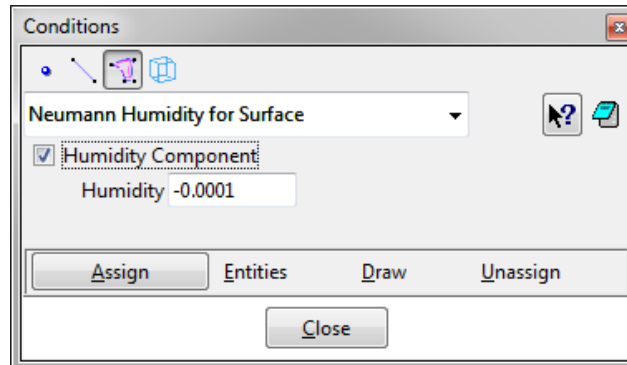


Fig. 8-140: Neumann humidity for ...

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.

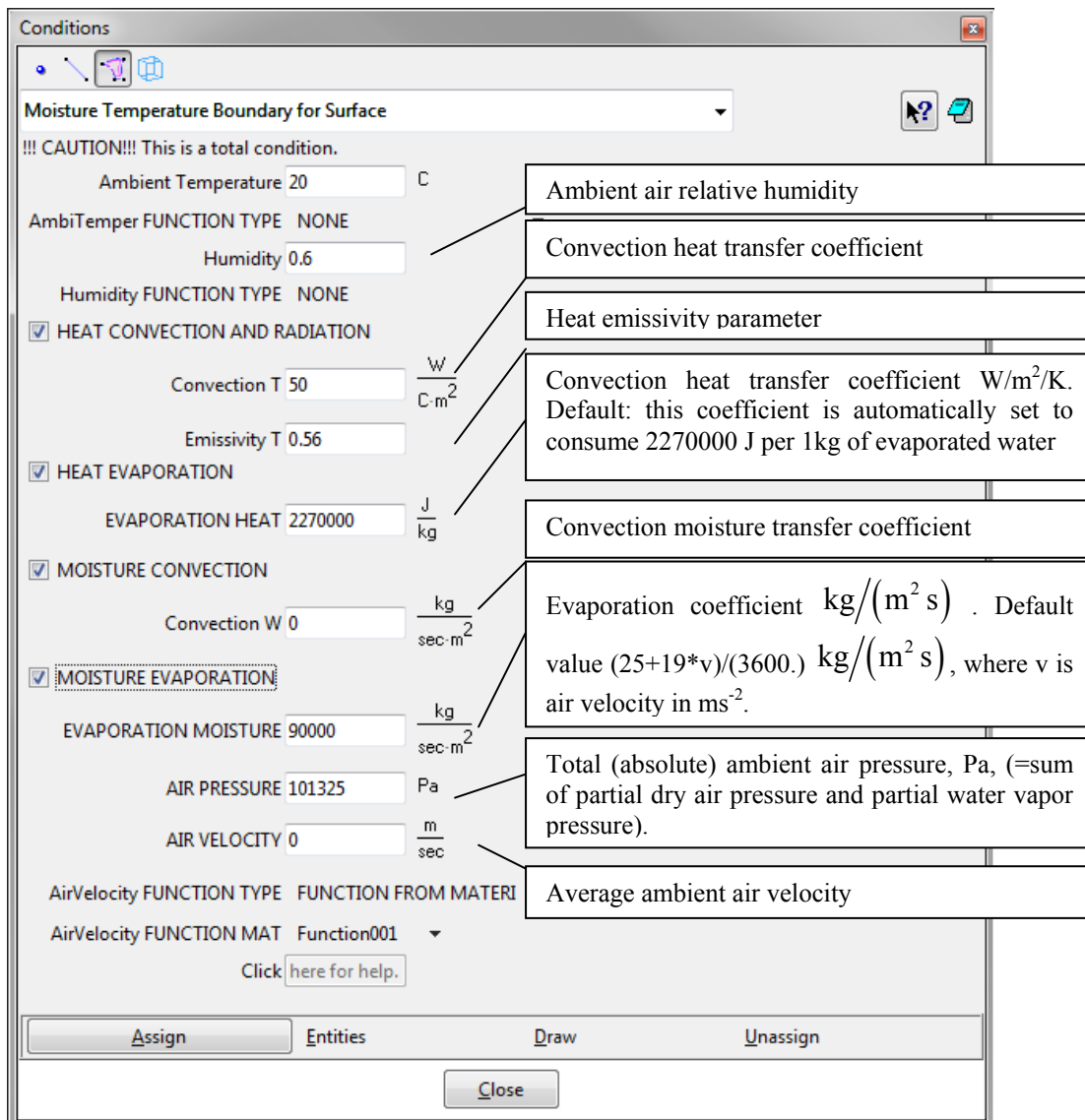


Fig. 8-141: Moisture Temperature boundary for ...

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. 8-141) One of the few total boundary conditions in **ATENA** (almost all other conditions act incrementally). This condition is NOT supported for quadratic mesh.

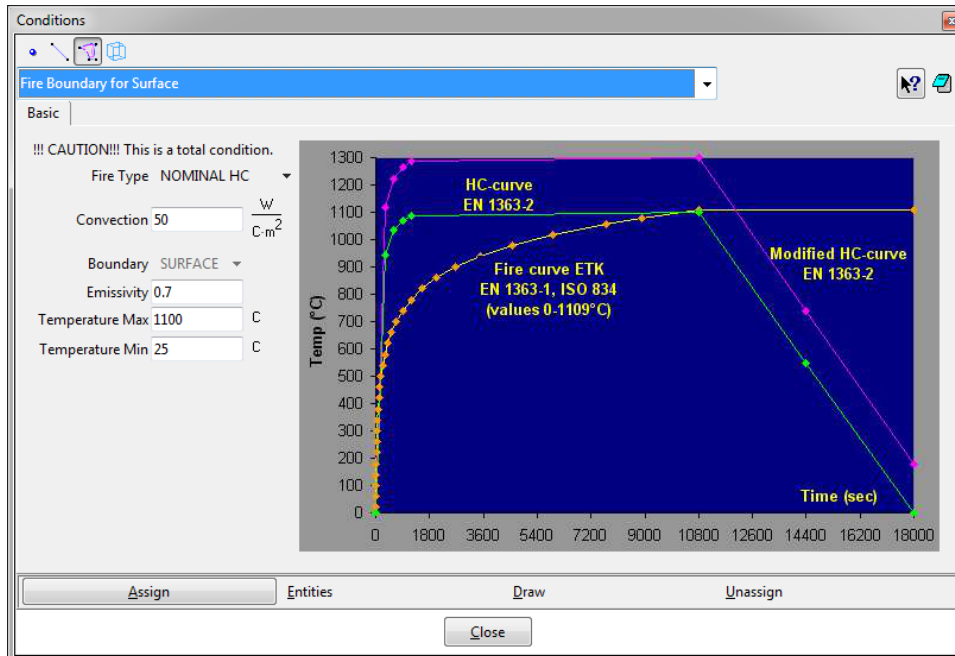


Fig. 8-142: Fire boundary for ...

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

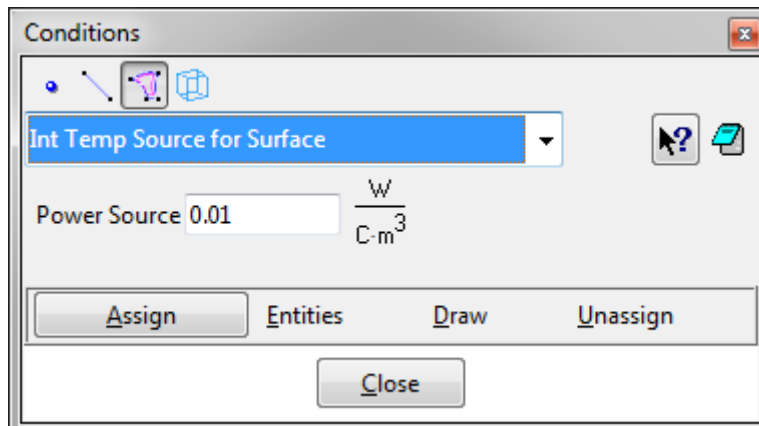


Fig. 8-143: Internal Thermal Source for ...


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 [5.4], set solution parameters [5.5], meshing model [5.8], and execute the analysis.

The ATENA software supports implicit dynamic analysis. ATENA provides Rayleigh damping. The Rayleigh damping coefficients can be automatically computed or defined by the user. See “Dynamic analysis” section of the ATENA Theory Manual, for more details on dynamic analysis.

9.1 Other Settings Related to Dynamic Analysis

In this section, the settings specific to the dynamic analysis are described. Dynamic calculation parameters are defined in the menu item **Data | Problem Data | Problem**

Data or by pressing the icon . At the Time and Dynamic tab (Fig. 9-144), it is necessary to set the start and end time of the analysis, dynamic analysis method and global damping for an entire model. ATENA allows to specify global damping factors using the **Rayleigh damping coefficients**. The Rayleigh damping coefficients can be defined directly by the user or automatically generated from specified damping ratios $Ksi [-]$, eigenfrequencies $Omega [s^{-1}]$ and associated weight factor $Weight [-]$ (Fig. 9-145). In the case of automatic generation by the program, the user must define the values for at least two eigenmodes. Damping may also be specified as part of a material definition. Material damping can be defined at the Miscellaneous tab of the material dialog.

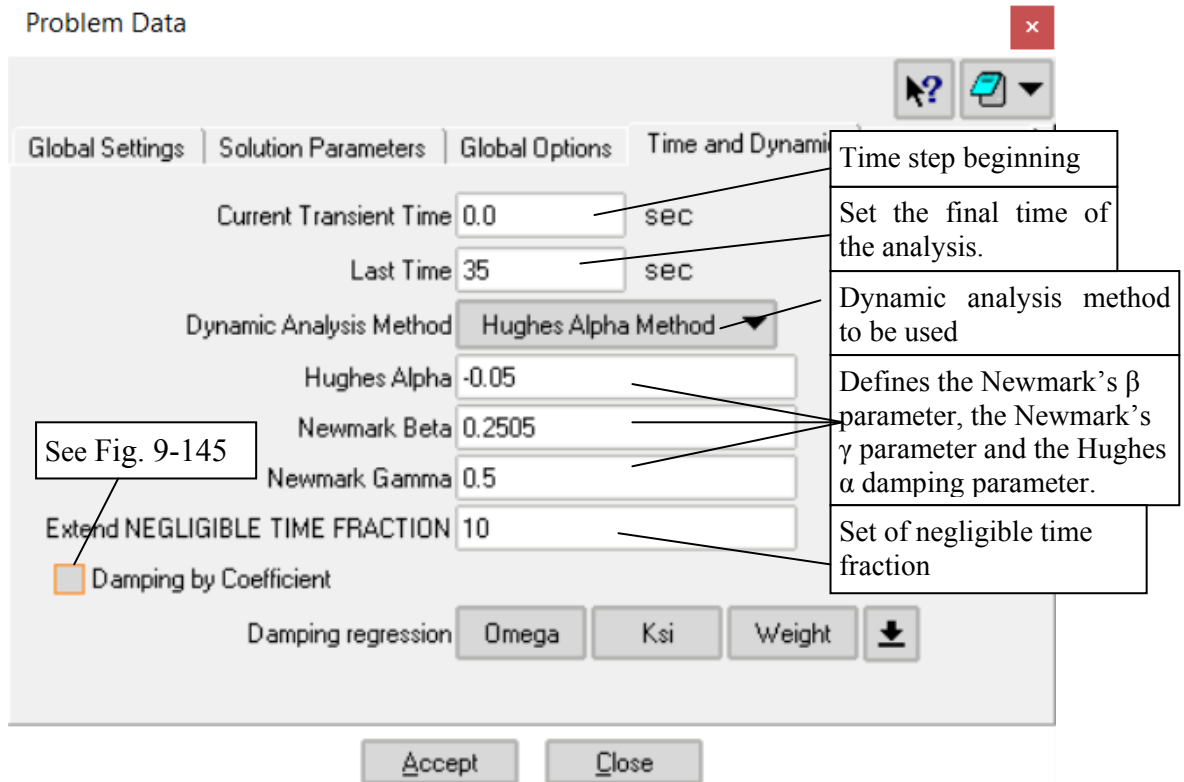


Fig. 9-144: Problem data - Time and Dynamic sheet

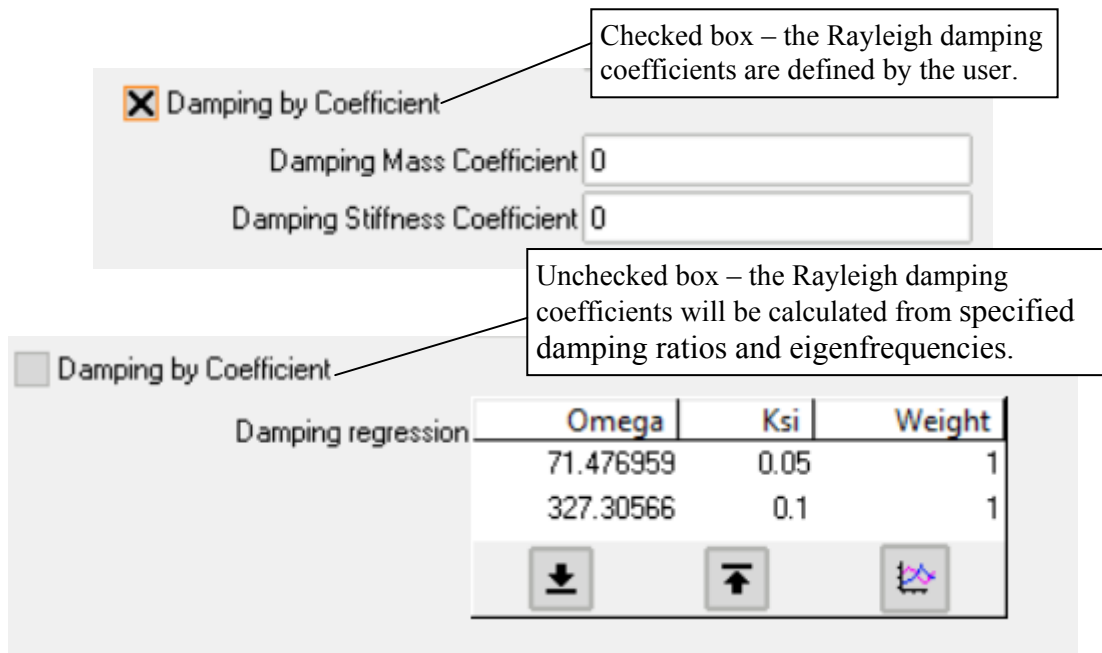


Fig. 9-145: Problem data - Damping

In Dynamic analysis, there are two options for the definition of boundary conditions. Each boundary condition can be applied as **FIXED** or **INCREMENT**. Typically loads are specified as of **INCREMENT** type and support conditions as of **FIXED** type. The type of condition is set using the **Apply in Subincrement** checkbox at the **Application** tab of the conditions dialog (see Fig. 9-146). In ATENA, the response of the structure in the dynamic analysis is calculated using implicit time integration. The load step can be subdivided into several sub-steps (depending on the definition of Integration Time Increment in the Interval Data dialog). Therefore, it must be distinguished between boundary conditions that are to be applied to all internal sub-steps and boundary conditions applicable only for the first sub-step. Support conditions should be applied in all sub-steps, but the loading increment should be applied only in the first step.

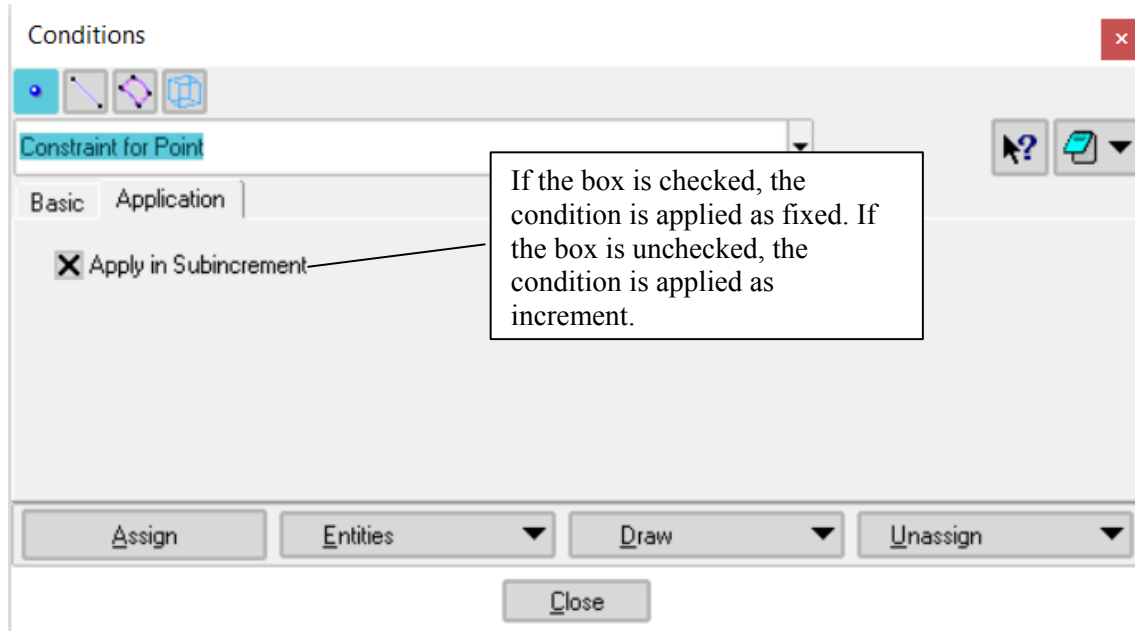


Fig. 9-146: Definition of fixed or increment type of load

The **Interval Data** dialog in dynamic analysis is shown in Fig. 9-147. In this section, the definition of interval data related only to dynamic analysis is described. The remaining input data that are also used for static analysis are described in the previous section 5.4 of this document.

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. 6-112.

The **Ground Accelerogram** representing the earthquake can be entered by choosing the Ground Accelerogram box in the Interval data dialog. By means of the accelerogram (also seismogram) the time history of the base acceleration is specified. ATENA offers to import the accelerogram from an external data file and choose the direction of the movement of the incoming wave. The external file with a user-defined accelerogram must be in a .txt file. The text file should contain two columns of data. The first column is time and the second column represents the acceleration:

Time [s];Acceleration [in units which you using for the model]

The example of the text file you can find in:

AtenaExamples\Science\GiD\Tutorial.Dynamic\SeismicStoneHenge.gid\ground.txt

Please note that the step multiplier for fixed conditions must be 1 when the accelerogram is applied. (or in the case of using the interval multiplier, it is necessary to set same value for the interval multiplier and number of step).

Dynamic analysis is used to determine the response of the structure to time-dependent loads. Therefore, it is necessary to define the time length of each load interval. The user must define the starting and ending times of the interval. After confirming the Accept button, the length of **Increment transient time** will be determined automatically from the specified number of steps and the duration of the interval. The user can adjust the length of the **Integration time increment**. The integration time increment must be shorter or equal to the length of the time step (increment transient time). The correct determination of the integration time step is important for convergence and accuracy of the results.

Interval data

1

Basic Parameters

Use decimal point (do not use comma).
Press accept to save changes before starting analysis !!! Attention!!!

Interval Is Active

Load Name

Define Loading History

Generate Multiple Steps

Activate Step multiplier for Fixed

Activate Step multiplier for Increment

Interval Multiplier for Fixed

Interval Multiplier for Increment

Number of Load Steps

Store Data for this Interval Steps

Interval Starting Time sec

Interval End Time sec

Increment Transient Time sec

Integration Time Increment sec

User Solution Parameters

User Dynamic Parameters

Calculate Eigenvalues-Vectors

Print Eigenvalues-Vectors to output file

Ground Accelerogram

Delete BC Data After Calculation

Activate Interface Opening

Add Additional Load Cases

Set Reference Configuration

Show Material Activity

You can use Step or Interval multiplier for loading. A different value of load multiplier can be used for fixed and increment load.

Interval starting time, Interval end time
Increment Transient Time = automatically determined value from the number of steps and duration (update using Accept button)
Integration Time Increment = must be shorter or equal to Increment Transient time

Active tab with Dynamic parameters. New settings of dynamic calculation parameters can be specified for this and any subsequent intervals.

Upload the accelerogram from an external data file.

Fig. 9-147: Interval data – dynamic analysis

9.2 Specific Dynamic Boundary Conditions

For mass in dynamic analyses, the **Rho-density** parameter in the material definition is used, and the Condition **Lumped mass for point**.

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

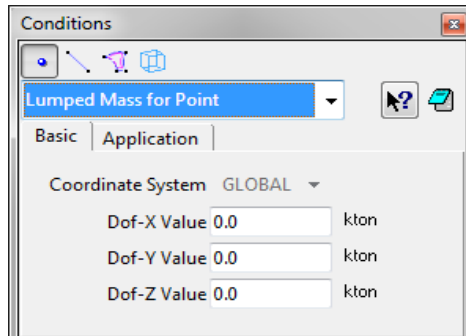


Fig. 9-148: Lumped mass for point

Velocity – Prescribe constant velocity. Typically used along with a load history defined in Interval Data (Fig. 9-147).

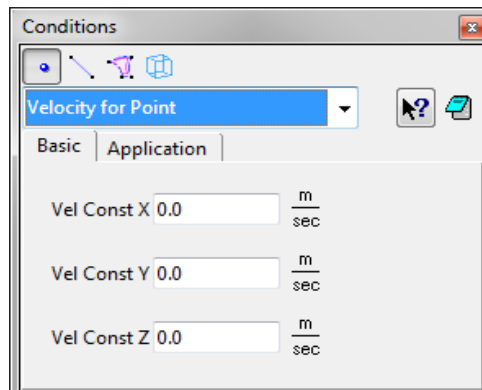


Fig. 9-149: Velocity for ...

Acceleration – Prescribe constant acceleration. Typically used along with a load history defined in Interval Data (Fig. 9-147).

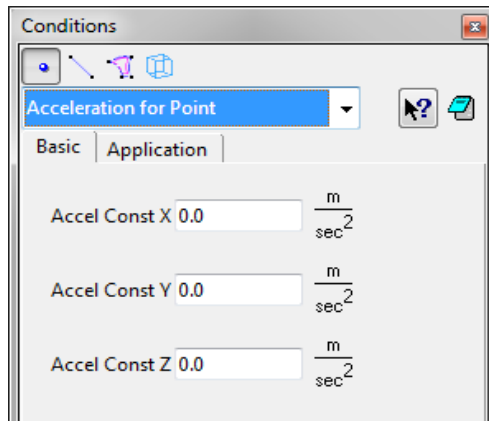


Fig. 9-150: Acceleration for ...

Initial Velocity – Speed at the beginning of the analysis.

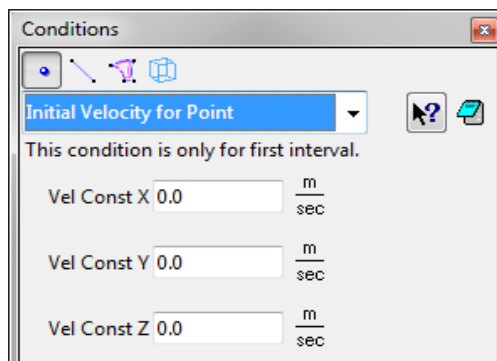


Fig. 9-151: Initial Velocity for ...

Initial Acceleration – Acceleration at the beginning of the analysis.

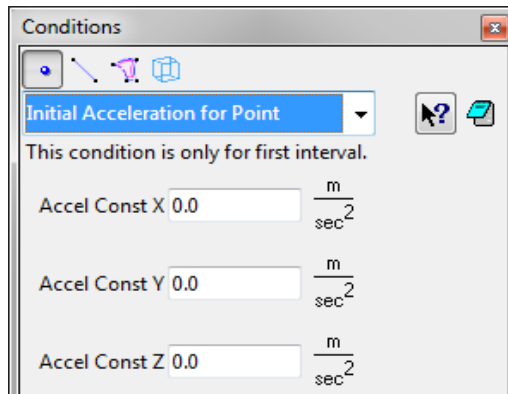


Fig. 9-152: Initial Acceleration for ...

10 POST-PROCESSING IN ATENA-GiD

The created model can be post-process in the **ATENA Studio** or in the **GiD**⁷. 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 analyzed problem is finished or if the post-processing should be started. The button **Postprocess** should be selected (see Fig. 10-153).

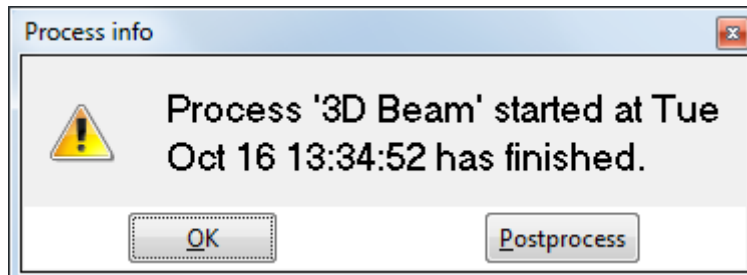



Fig. 10-153: The button Postprocess should be pressed

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 clicking on the Import results from **ATENA Studio** icon . Then the process of importing will start (see Fig. 10-155) and when it is finished the model changes its colors (see Fig. 10-156).

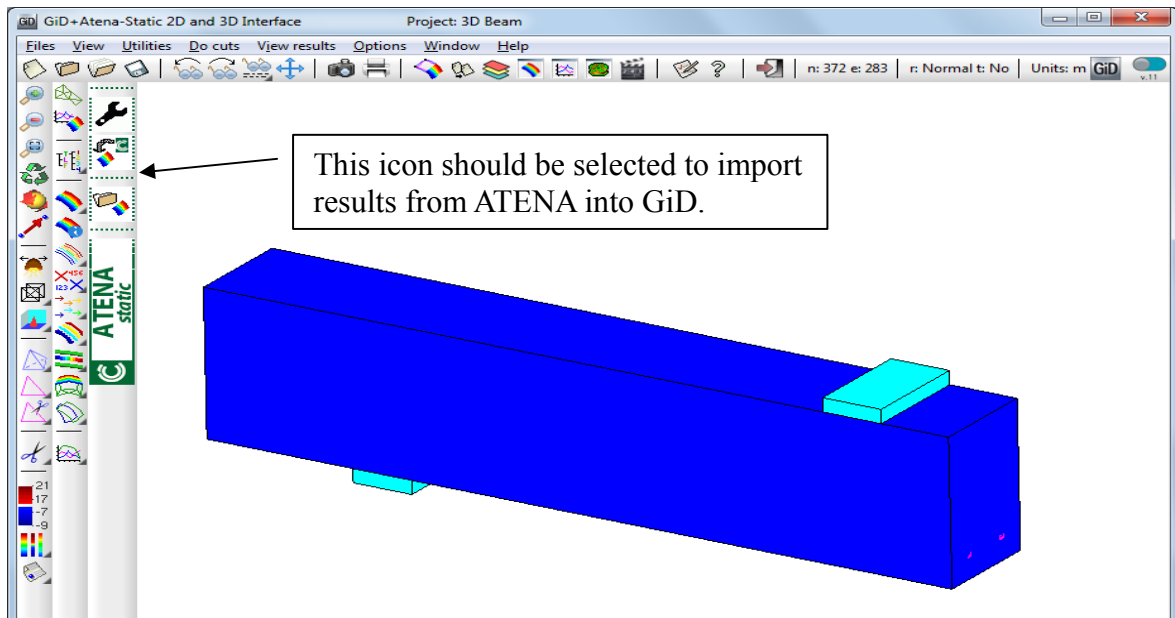


Fig. 10-154: The GiD postprocessor interface

⁷ In ATENA version 5.3 and later, we recommend postprocessing in **ATENA Studio**.

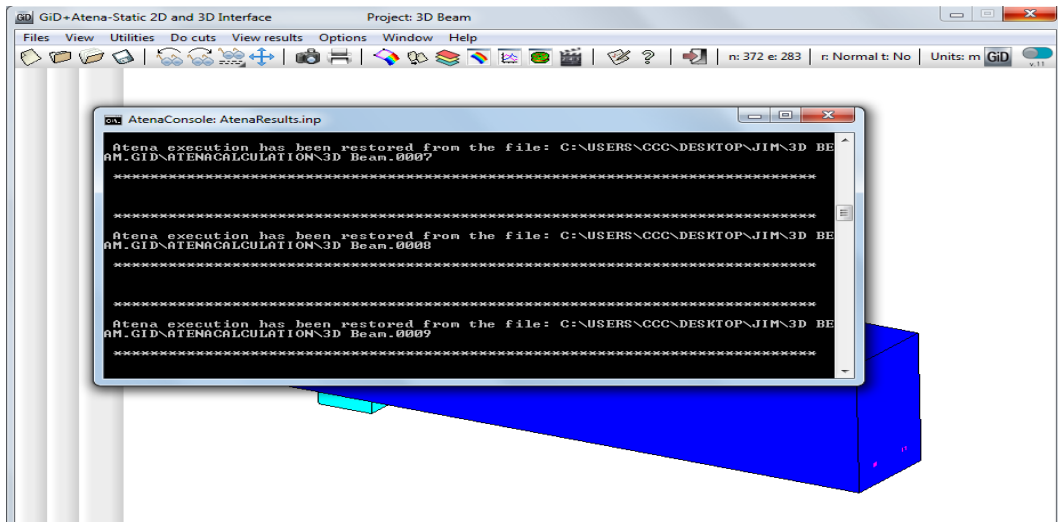


Fig. 10-155: The importing of the results from ATENA Studio into GiD

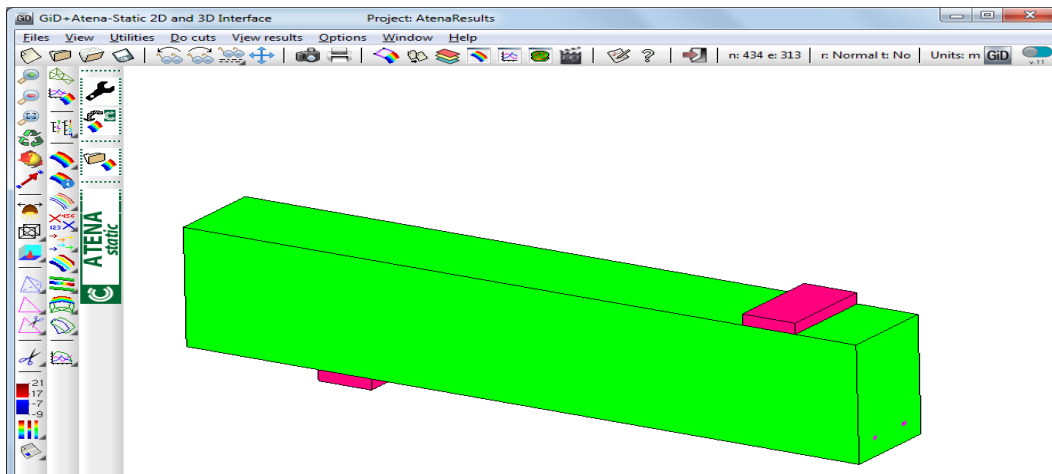



Fig. 10-156: The importing of the results from ATENA was finished

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. 10-157).

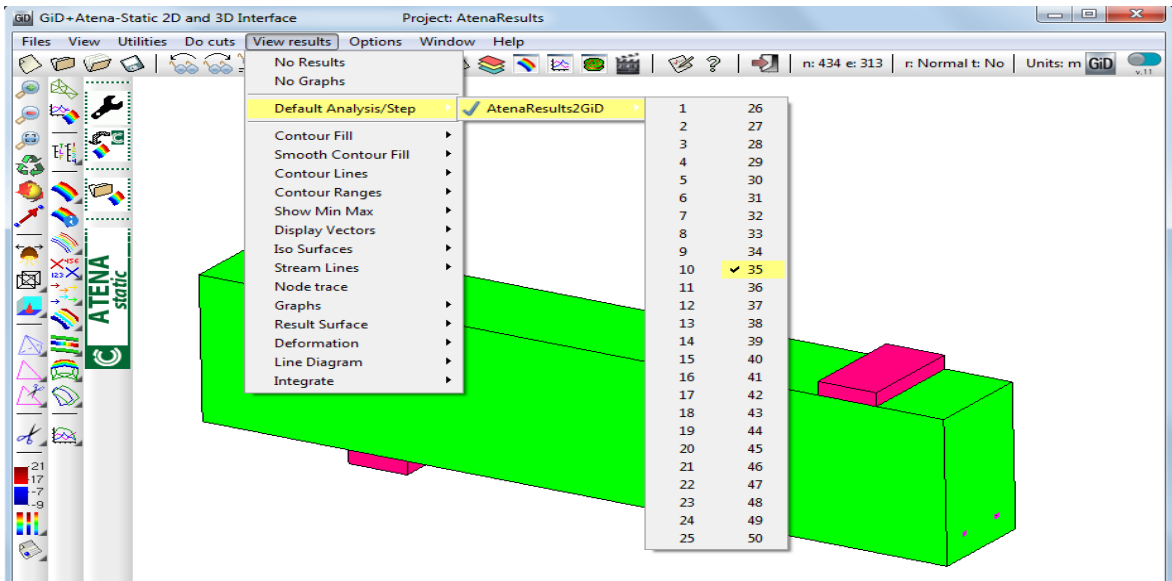


Fig. 10-157: The selection of the step which should be post-processed

By clicking on the Contour fill icon  or by selecting the command from the main menu **View results | Contour Fill | CRACK WIDTH | COD1** crack width can be displayed (see Fig. 10-158).

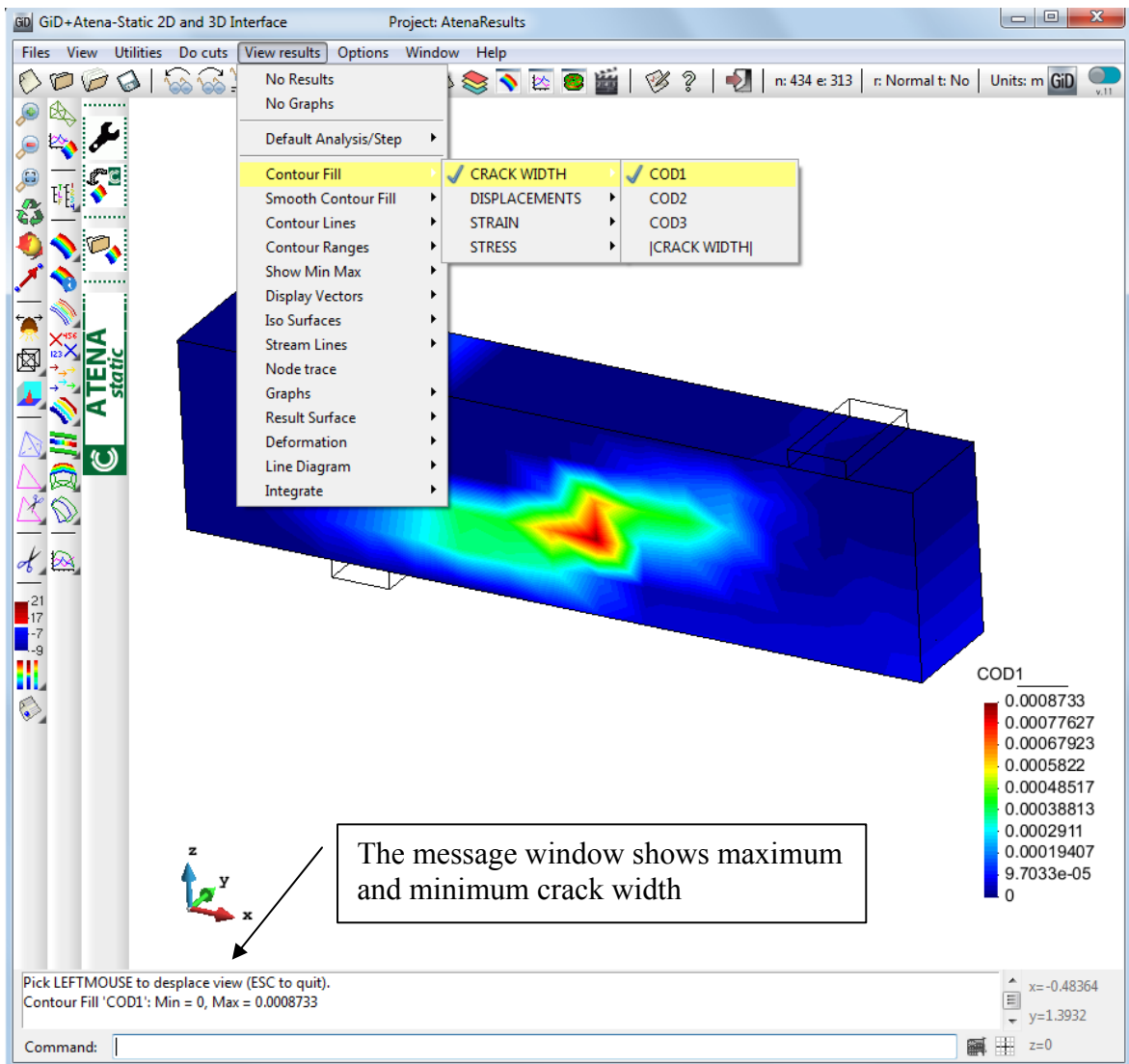




Fig. 10-158: The display of the crack width

In the command Contour Fill, the drop-down menu offers options that 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. 10-159). 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. 10-160). This dialog you can run directly by clicking to icon  in post-processor.

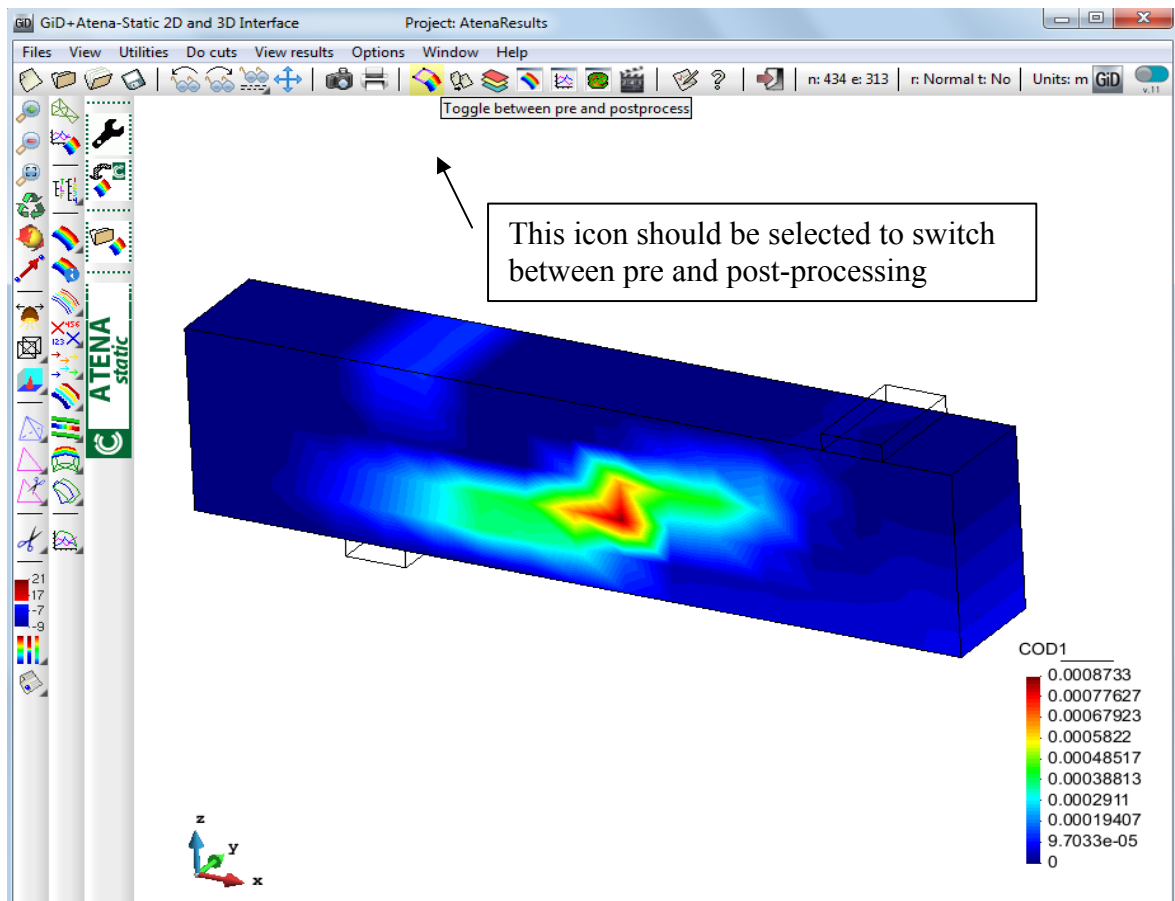


Fig. 10-159: Switch between pre and postprocessing

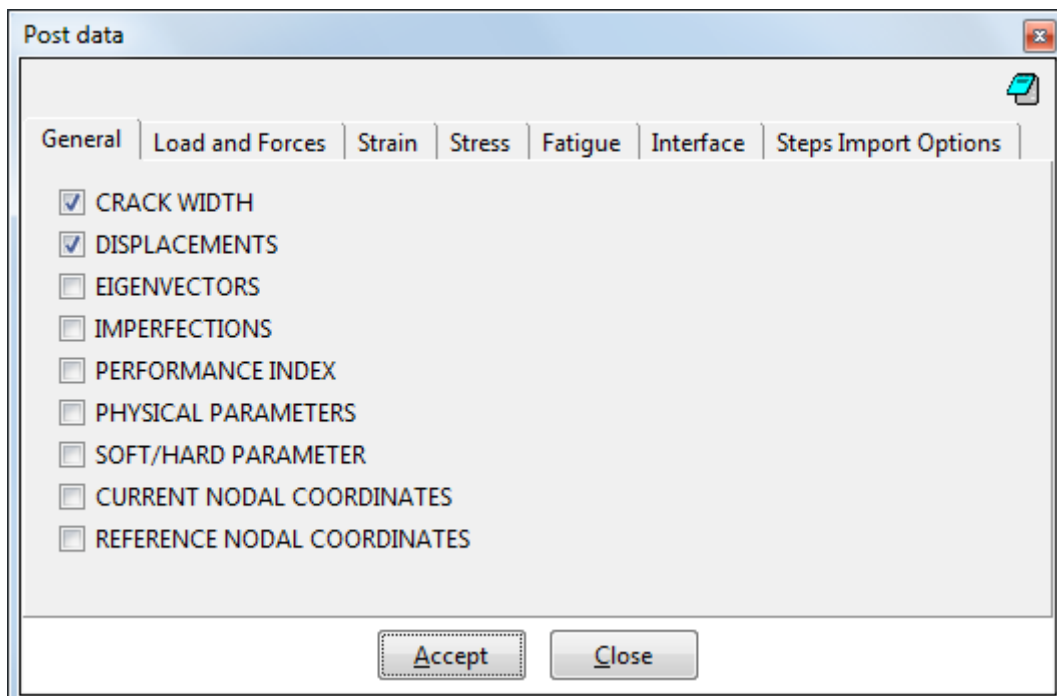



Fig. 10-160: The selection of the data which should be available for the post-processing

For example, the FRACTURE STRAIN can be chosen. The definition of post data is completed by selecting **Accept** button (see Fig. 10-161). 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 clicking on the **ATENA** icon . Then the FRACTURE STRAIN can be found in the options for the post-processing (see Fig. 10-162, to obtain this figure the 35th step has to be selected again).

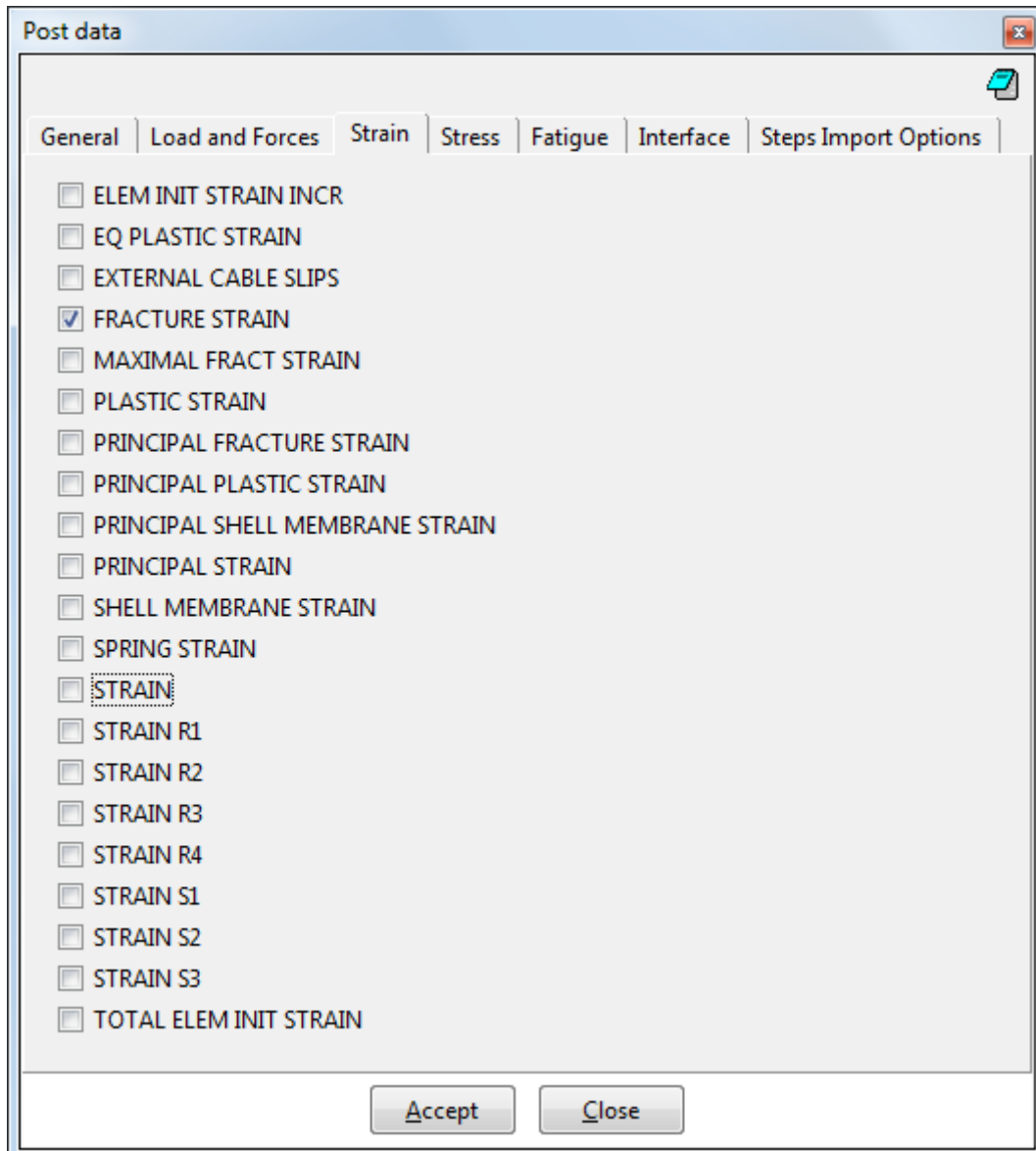


Fig. 10-161: The selection of the FRACTURE STRAIN

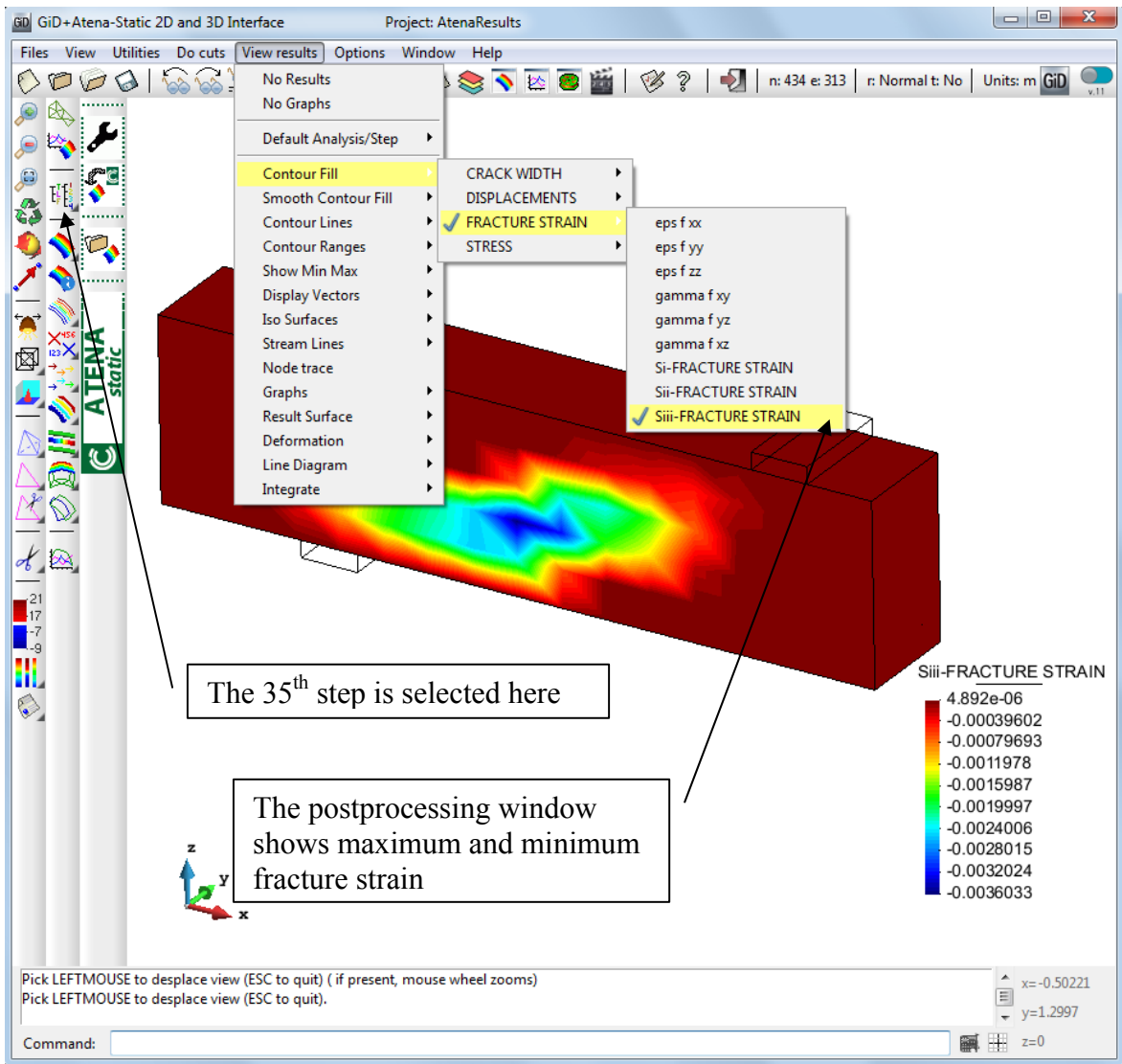


Fig. 10-162: The displayed FRACTURE STRAIN

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 (examples listed in **bold font** are described in details in ATENA Science Example Manual [8]):

Directory - Tutorial.Creep2D

| | |
|--------------------------------------|--|
| BeamWithCreep.gid | Slab with creep that is modelled as a two-dimensional structure |
| BeamWithCreep_demo.gid | Same as above, but for demo version |
| TunnelWithConstructionProcessNew.gid | Advanced example of support for the new Austrian tunnelling method |

Directory - Tutorial.Creep3D

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

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 |
| SeismicStoneHenge.gid | Dynamic analysis of Stonehenge-inspired structure |
| SingleDegreeFreeVibration.gid | Single degree of freedom example with free vibration |

Directory - Tutorial.FRC

(examples in this folder are covered in ATENA Science – GiD FRC Tutorial [10])

| | |
|--------------------------------|---|
| _FRC_creep | Model of 3-point bending beam for the FRC in creep |
| /3PBT_creep_constant_load.gid | |
| /3PBT_static_ultimate_load.gid | |
| _FRC_fire | Model of 3-point bending beam for the FRC in transport (fire) |
| /static_01.gid | |
| /transport_01.gid | |
| FRC_moisture_transport | Model of 3-point bending beam for the FRC in transport |

| | |
|---------------------------------------|--|
| /FRC_moisture_transport.gid | |
| _FRC_shrinkage | Model of 3-point bending beam for the FRC with shrinkage |
| /FRC_shrinkage.gid | |
| /FRC_shrinkage_moisture_transport.gid | |
| FRC_4PBT_2D_initial model.gid | Initial model of 4-point bending beam for the FRC tutorial |
| FRC_4PBT_2D_modification_1.gid | Modified model of FRC beam(1) |
| FRC_4PBT_2D_modification_2.gid | Modified model of FRC beam(2) |
| FRC_4PBT_2D_modification_3.gid | Modified model of FRC beam(3) |
| FRC_4PBT_2D_modification_4.gid | Modified model of FRC beam(4) |

Directory - Tutorial.Static2D

| | |
|---|---|
| axisym.gid | Axisymmetric problem |
| CyclicColumn.gid | Column subjected to cyclic loading |
| 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 |
| FRC_4PBT_2D_force.gid | Model of 4-point bending test of FRC beam using material with implemented the added fracture energy approach. |
| PunchingShearFailure.gid | Axisymmetric problem of slab punching failure |
| InterfaceWithShear.gid | Example with an interface material model |
| Slab_Strip_Toronto.gid | Model of a shear beam test |
| TunnelWithConstructionProcess.gid | Two-dimensional analysis of a simple tunnel with construction process |
| TunnelWithConstructionProcess_Interface.gid | Support for the new Austrian tunnelling method with interface contact. |
| TunnelWithConstructionProcessNew.gid | Support for the new Austrian tunnelling method. |

Directory - Tutorial.Static3D

| | |
|---------------------------------|---|
| 1Dbeam_middleforced.gid | Frame modelled using 1D beams with joints modelled with 3D solids |
| 8beams_loading_test.gid | Examples of beams modelled with various approaches |
| BeamWithBeamElements.gid | Example with 3D beam elements |
| BeamWithCarbonation_30years.gid | Durability analysis example of a bending beam subjected to carbonation for 30 years |

| | |
|---|--|
| BeamWithCarbonation_60years.gid | Durability analysis example of a bending beam subjected to carbonation for 60 years |
| BeamWithChlorides_30years.gid | Durability analysis example of a bending beam subjected to chloride attack for 30 years |
| BeamWithChlorides_60years.gid | Durability analysis example of a bending beam subjected to chloride attack for 60 years |
| BondCabel_Injected_Both.gid | Example of a pre-stressed beam including calculation of pre-stressing losses due to cable friction |
| BondCabel_Injected_Left.gid | Same as above but with pre-stressing applied from left side of the beam only |
| BondCabel_Injected_Right.gid | Same as above but with pre-stressing applied from right side of the beam only |
| Bridge_ConstructionProcess_3D.gid | Example of the construction process of a pre-stressed girder box from ATENA tutorial [12] |
| Building_2DShell_1DBeam.gid | Example of a building structure modelled using 2D shell and 1D beam elements |
| DirectTensionFatigue.gid | Example of a notched direct tension test with fatigue material model |
| Frame1DBeamsFixedHinge.gid | Example of a frame with fixed and hinge connections modelled using 1D beam |
| FrameSolidsAndBeam.gid | Example of a simple frame with fixed connections modelled using 1D beam and 3D solids |
| FrameSolidsAndShell.gid | Example of a simple frame with fixed connections modelled using 2D shell and 3D solids |
| FrameWithHingesSolidsAndBeams1.gid | Example of a simple frame with hinge connections modelled using 1D beam and 3D solids |
| FrameWithHingesSolidsAndShell.gid | Example of a simple frame with hinge connections modelled using 2D shell and 3D solids |
| FrameWithHingesSolidsAndShell2.gid | Same as above but with different placement of the beam |
| InterfaceWithShear3D.gid | Example of interface between two concrete plates |
| InterfaceWithShear3D-OpenGap.gid | Same as above but with initial opening between the plates |
| ShearBeam3D.gid | Example of four-point bending with shear failure |
| ShearBeam3D_1dbeam.gid | Example of four-point bending with bending behaviour in reinforcement |
| ShearBeam3D_asr.gid | Example of a bending beam with ASR |
| ShearWallCASH.gid | Example of a shear wall model |
| SlabWithColumn.gid | Example of connection of 3D shells-3D solids |

| | |
|---|--|
| SlabWithColumn_1Beam+2Dshell.gid | Example of connection of 1D beam-2D shells |
| SlabWithColumn_beam.gid | Example of connection of 3D shells-3D beam |
| SmallCantileverWithTorsion.gid | Example of L-shaped cantilever with discrete bars for main reinforcement and smeared for stirrups. |
| SmallCantileverWithTorsion_DiscreteBars.gid | Example of L-shaped cantilever with discrete bars for main reinforcement as well as for stirrups. |

Strengthening (folder)

(examples in this folder are explained in the ATENA Science – GiD Strengthening of concrete structures [11])

| | |
|--|---|
| /3DBeam-strengthening.gid | Reference for strengthening examples. |
| /3DBeam-strengthening-A-6 lamellas.gid | Example of a beam strengthened using 6 lamellas |
| /3DBeam-strengthening-A-8 lamellas.gid | Example of a beam strengthened using 8 lamellas |
| /3DBeam-strengthening-B-fabric-2Dcomposite.gid | Example of a beam strengthened using 3D composite elements |
| /3DBeam-strengthening-B-fabric-2Dplasticity.gid | Example of a beam strengthened using 3D composite elements with plasticity |
| /3DBeam-strengthening-B-fabric-2Dshell.gid | Example of a beam with strengthening modelled as 2D shells |
| /3DBeam-strengthening-B-fabric-3Dshell.gid | Example of a beam with strengthening modelled as 3D shells |
| Truck passing bridge_moving contact.gid | Example of a bridge loaded by a passing truck |
| Tunnel3DWithConstructionProcess.gid | Three-dimensional model of a tunnel with soil and construction process (using delete and create elements) |
| Tunnel3DWithConstructionProcessNew.gid | Three-dimensional model of a tunnel with soil and construction process (using show material activity) |

Directory - Tutorial.Temperature2D

| | |
|---------------------------|--|
| BridgePierHydratation.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

| | |
|--------------------------|---|
| ColumnThermal3D.gid | 3D Column with temperature loading |
| ColumnThermal3D_demo.gid | Same as above, but for demo version |
| 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 |
| Vitek3Dfire.gid | 3D four point beam with fire loading - transport |
| Vitek3Dmoist.gid | 3D four point beam with moisture loading - transport |
| 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. and .

| <i>ElementType for 3D</i> | ElemsNnode | Geometrical | |
|--------------------------------------|-------------------|-------------|-----------|
| | | LINEAR | NONLINEAR |
| CCIsoGap<xxxxxxxx> | 8 | 28 | 58 |
| CCIsoGap<xxxxxx> | 6 | 26 | 56 |
| CCIsoBrick<xxxxxxxxxxxxxxxxxxxxxxxx> | 20 | 20 | 50 |
| CCIsoWedge<xxxxxxxxxxxxxxxxxxxx> | 15 | 15 | 45 |
| CCIsoTetra<xxxxxxxxxx> | 10 | 10 | 40 |
| CCIsoBrick<xxxxxxxx> | 8 | 8 | 38 |
| CCIsoWedge<xxxxxxx> | 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<xxxxxxx> | 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 |

$$\mathbf{ELEMENT_GROUP_ID} = \mathbf{Mat_ID} * 100 + \mathbf{ELEMENT_TYPE_ID}$$

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:

$$\mathbf{ELEMENT_TYPE_ID} = \mathbf{ElmsNnode} + \mathbf{AddingGapElemID} + \mathbf{AddingNonLinElemID} + \mathbf{AddingShellID}$$

1D Element:

| | Increment | |
|-------------------|-----------|-------------------------------------|
| AddingBarWithBond | 3 | Increment if is element BarWithBond |

Formula:

$$\mathbf{ELEMENT_TYPE_ID} = \mathbf{ElmsNnode} + \mathbf{AddingBarWithBond} + \mathbf{AddingNonLinElemID}$$

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:

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

REFERENCES

- [1] Cervenka, V., Jendele, L., Cervenka, J., (2015), *ATENA Program Documentation, Part 1, Theory*, Cervenka Consulting, 2, 2015
- [2] Cervenka, V. and Cervenka, J., (2015), *ATENA Program Documentation, Part 2-1, User's Manual for ATENA Engineering 2D*, Cervenka Consulting, 2015
- [3] Cervenka, V. and Cervenka, J., (2015), *ATENA Program Documentation, Part 2-2, User's Manual for ATENA Engineering 3D*, Cervenka Consulting, 2015
- [4] Cervenka, J., and Jendele, L., (2015), *ATENA Program Documentation, Part 6, ATENA Input File Format*, Cervenka Consulting, 2015
- [5] Benes, S., Mikolaskova, J., Altman, T. (2015), *ATENA Program Documentation, Part 12, User's manual for ATENA Studio*, Cervenka Consulting, 2015
- [6] Prochazkova Z., Cervenka, J., Janda, Z., Pryl, D., (2016), *ATENA Program Documentation Part 4-6, ATENA Science – GiD Tutorial*, Cervenka Consulting, 2016
- [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., (2016), *ATENA Program Documentation Part 3-2, Example Manual ATENA Science*, Cervenka Consulting, 2016
- [9] Pryl, D. and Cervenka, J., (2016), *ATENA Program Documentation Part 11, ATENA Troubleshooting*, Cervenka Consulting, 2016
- [10] Sajdlová T., (2016), *ATENA Program Documentation Part 4-7, ATENA Science – GiD FRC Tutorial*, Cervenka Consulting, 2016
- [11] Sajdlová T., (2016), *ATENA Program Documentation Part 4-9, ATENA Science – GiD Strengthening of concrete structures*, Cervenka Consulting, 2016
- [12] Sajdlová T.(2016), *ATENA Program Documentation Part 4-8 ATENA Science – GiD Construction Process Tutorial*, Cervenka Consulting, 2016