ČERVENKA CONSULTING

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

ATENA Program Documentation
Part 8

User's Manual for ATENA-GiD Interface

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

Prague, 26. 5. 2009

CONTENTS

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 8 or 9 is recommended):

- ATENAV4/Static, - static 2D and 3D analysis
- ATENAV4/Creep, - creep 2D and 3D analysis
- ATENAV4/Temperature, - transport 2D and 3D analysis
- ATENAV4/Dynamic - dynamic 2D and 3D analysis

They make it possible to define a finite element model within GiD including specific data needed for ATENA and export it to AtenaWin [5], where a non-linear analysis can be performed. Visualization of ATENA results is also possible in GiD, but can be done also in the Pre/Post-processor ATENA3D, which is a powerful ATENA postprocessor. This option is available only if ATENA Engineering is installed on your computer. Alternatively results can be presented directly in AtenaWin [5].

The problem types with the label ATENAV4 can be used with ATENA version newer than 4.0.0. These problem types support ATENA analysis with two- and three-dimensional models. In addition it is possible to perform stress, creep, thermal (i.e. transport) and dynamic analyses.

A demo version of GiD limited to 3000 elements (or 1010 nodes) 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 can be found in the GiD documentation.

2 OVERVIEW

2.1 Working with GiD

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

- Select one of the problem types in ATENAV4.
- 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 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 AtenaWin.

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 of the later features.

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

The practical aspects of the GiD use can be exercised on the examples described Chapter 5.

2.2 Limitations of ATENA-GiD interface

It should be noted that ATENA-GID interface supports the most common features of the ATENA software. However, the direct modification of the ATENA input file may be sometimes useful, and it allows the user to exploit all the features of the ATENA software. Detailed syntax of all ATENA commands is described in the ATENA documentation [4]. This ATENA command file is typically 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 GiD developer at http://www.gidhome.com/,

In order to get the better of GiD 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 valid license number it is necessary to obtain a password for the computer, on which the GiD will be operated. This process is activated by starting GiD and proceeding to the menu Help | Register. It should 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. Then it is possible to operate GiD on every computer, to which the registered flash disk is attached. The license price for USB protection is slightly different then the one for PC protection, so it is important to choose this option during the program purchase. If the USB protection is wanted, it is necessary to attach the USB flash disk to the computer. Then the item Help | Register should be selected. If the 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.

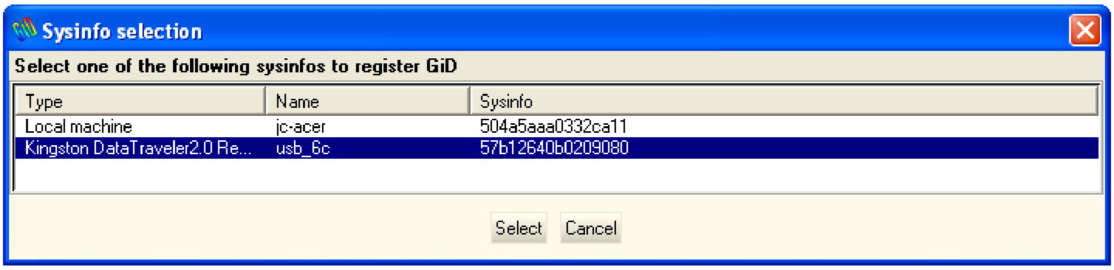


Fig. 3-1: Choice of USB or PC protection

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

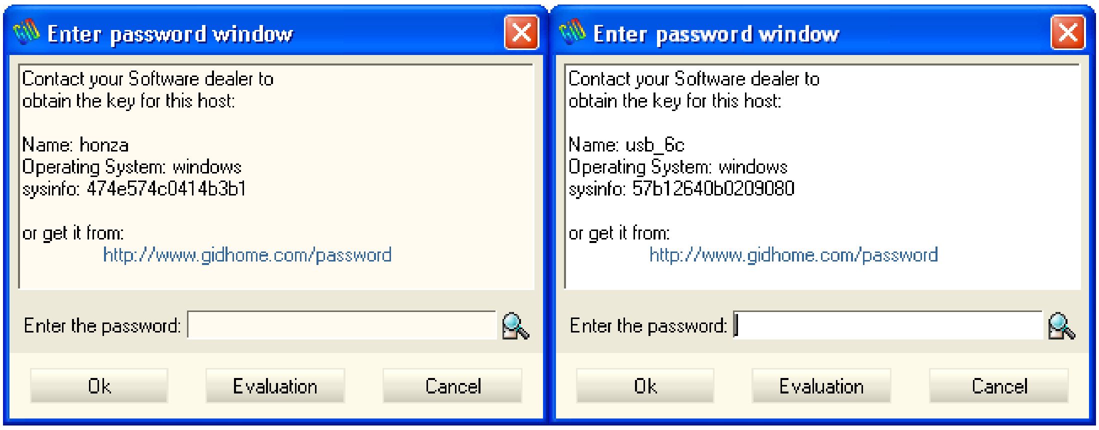


Fig. 3-2: GiD register window (PC protection left, USB protection right)

If GiD have been registered previously (a 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. The user then should follow the instructions of the GiD server to obtain the password, which should be typed or copied into the bottom line in the above dialog.

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

4 ATENA-GiD Installation

The installation of ATENA-GiD interface can be also performed during ATENA installation. During this process, a user will need to confirm the location of GiD directory. Alternatively, the ATENA-GiD interface can be also installed manually as it is described in the following paragraphs.

After installing ATENA on your computer, there should be a subdirectory GiD in the directory where ATENA is installed. Please note, that this subdirectory is installed only if the ATENA-GiD interface is selected during the installation. If the subdirectory GiD does not appear in the ATENA directory, the ATENA setup should be started again and ATENA-GiD interface should be selected for installation.

The next step is to copy all the subdirectories of the directory

...ProgramFiles\CervenkaConsulting\AtenaV4\GiD

into appropriate subdirectories in the GiD installation directory. On most computers the GiD is installed in the directory:

C:\Program Files\GiD\GiDx.x

The following two subdirectories (with its contents) should be copied into the GiD directory:

problemtypes – This directory contains the definition of special problem types for ATENA.

After that Atena becomes one of the problem types, which are available under the GiD menu ‘Data | Problem type’.

5 ATENA - SPECIFIC COMMANDS

5.1 Problem type

The program GiD is a general-purpose pre- and post-processing tool for variety of numerical problems (and analysis software). In this menu we can define a problem type, which in our case is ATENA analysis. This is done by selecting for example the menu item 'Data | Problem type | AtenaV4 | Static as shown in Fig. 5-1. By this command GiD is configured to create data for analyses, which are compatible with ATENA input format (units, materials, conditions, etc.). The data resulting from the GiD modelling will be later transferred to ATENA by the via an input file usually called *name.inp*.

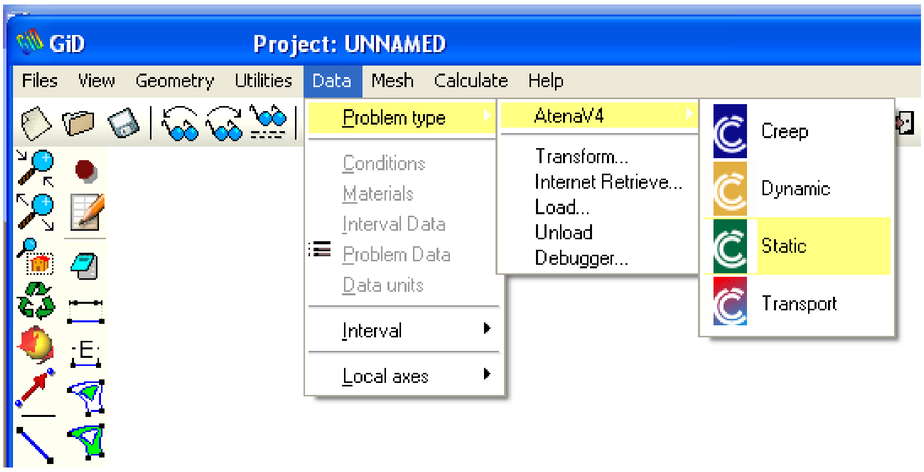
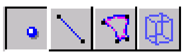
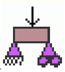
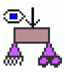


Fig. 5-1 Problem type menu.

The problem type definition must be done before starting input of any data. Executing this command later may cause losing any existing data.

5.2 Conditions

The supports and loading conditions for ATENA can be defined in a way, which is compatible with ATENA through the menu 'Data | Conditions', Fig. 5-2, left. 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 included into the current Interval data.

The conditions can be assigned to four kinds of geometrical objects: nodal points (finite element nodes), lines, surfaces and volumes. The object dimension is selected by choosing one of the buttons . For each geometric entity an appropriate list of possible conditions can be unfolded and a required type of condition can be selected. Example of the list for conditions in a point is shown in Fig. 5-2, middle. Applied conditions are then selected by filling the appropriate boxes, Fig. 5-2, right or by icon . You can view all currently defined conditions in current interval by clicking to icon 

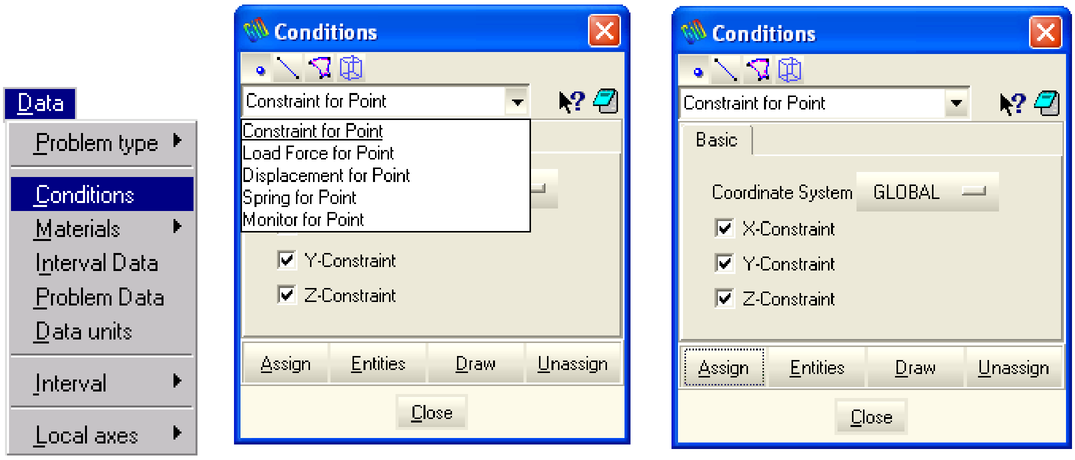


Fig. 5-2 Conditions: menu, list at Point, applied at Point.

Operations for condition assignment are done with the following buttons in the bottom of the dialog.

'Assign' - The target of assignment command depends on the display type. In case that geometry is displayed, then geometrical objects (point, line, surface) can be selected and condition can be assigned to geometry. In case that elements are displayed the condition can be assigned to element nodes.

'Entities' – Shows a list of entities for assigned conditions.

'Draw' – Display of nodes assigned for conditions is shown.

'Unassign' – Reverse operation. It cancels the current assignment of selected objects.

There are certain conditions in the dialog in Fig. 5-2, which are strongly ATENA specific.

Monitors - It is for instance the condition Monitor. This 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 intervals other than the first one are ignored.

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. The meshes on the contact entities do not need to be compatible. ATENA creates special master/slave conditions that introduce the compatibility of displacements.

Reinforcement identification – This condition is used to identify that certain line entities should be treated as ATENA discrete reinforcement bars. The truss elements, which will be generated along these entities, will be embedded into the ATENA model as discrete reinforcement bars. This means that they will be further subdivided depending on their intersections with the solid finite elements. By default, the GiD program automatically detects lines, which are not connected to any volume or surface and treats these lines as reinforcement. This default behaviour can be controlled by the corresponding check box in Problem data dialog. If this check box is deactivated, it is necessary to manually assign the conditions reinforcement nodes and elements identification to the corresponding line entities. The lines which are not identified as reinforcement are treated as standard truss elements (see Section 10.1).

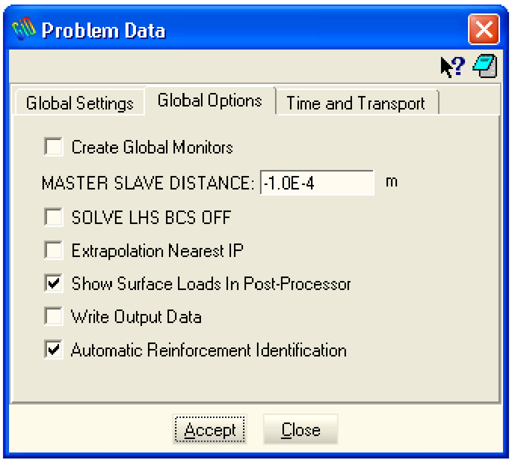


Fig. 5-3: Automatic reinforcement identification in the Problem Data dialog

BC-Springs – Springs created using boundary conditions (Spring_for_Point, Spring_for_Line, Spring_for_Surface) cannot be modified. It is neccessarry to delete them, and create again.

5.3 Materials

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

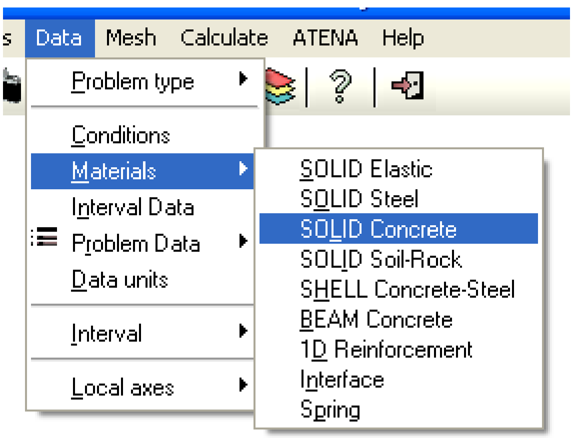


Fig. 5-4 Example of choice material definition.

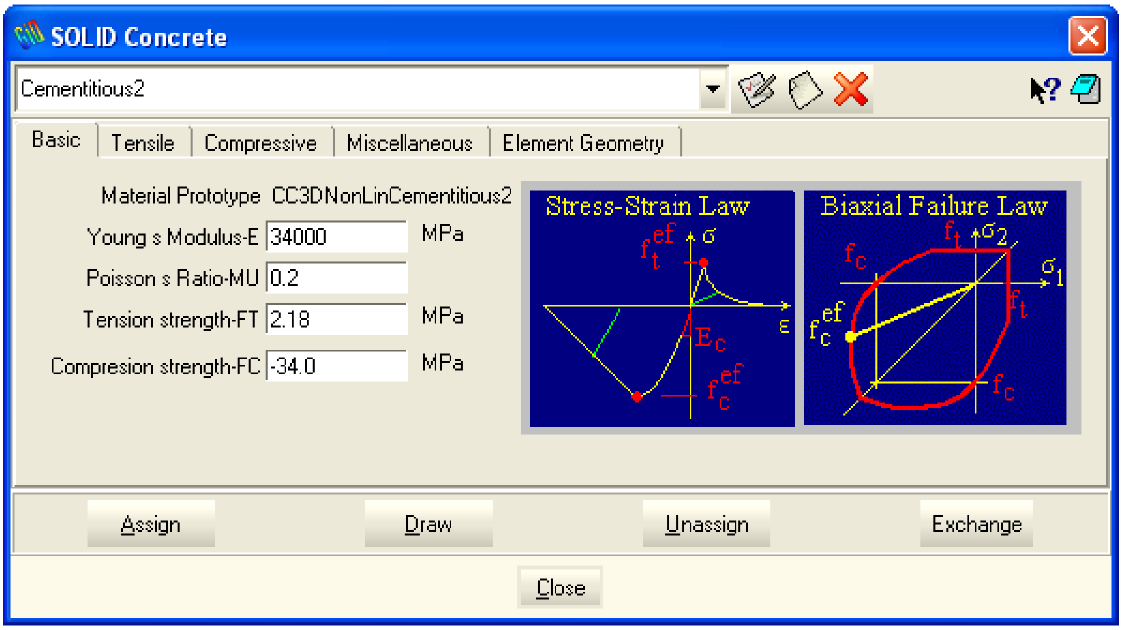


Fig. 5-5 Example of menu window for the material concrete.

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

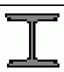
'Assign' - The target of assignment command depends on the display type. In case that geometry is displayed, then geometry type is to be selected (line for reinforcement, volume for concrete), and material can be assigned to the geometric entities. In case that the finite elements are displayed, the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

'Draw' – displays the material assignment to volumes or elements.

'Unassisgn' – Reverse operation. It deletes the material assignment.

'Import/Export' – Read/ write material parameters from/to a text file.

Table 1: Materials supported by GiD interface to ATENA

GiD name	*ATENA name (INP command)*	*Description*
SOLID Elastic		
Elastic 3D	CC3DElastIsotropic	Linear elastic isotropic materials for 3D
SOLID Steel 		
Steel Von Mises 3D	CC3DBiLinearSteelVonMises	Plastic materials with Von-Mises yield condition, e.g. suitable for steel.

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 User	CC3DNonLinCementitious2User	Materials suitable for rock or concrete like materials. This material is identical to 3DNONLINCEMENTITIOUS2 except that selected material laws can be defined by user curves.
Cementitious2 SHCC	CC3DNonLinCementitious2SHCC	Strain Hardening Cementitious Composite material. Material suitable for fibre reinforced concrete, such as SHCC and HPFRCC materials.
Cementitious3	CC3DNonLinCementitious3	Materials suitable for rock or concrete like materials. This material is an advanced version of 3DNONLINCEMENTITIOUS2 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.
Microplane M4	CCMicroplane4	Bazant Microplane material models for concrete
Bazant_Xi_1994	CCModelBaXi94	Material for transport analysis (fTransport3D PROBLEMTYPE)
SOLID_Creep_Concrete (only for Creep PROBLEMTYPE)		
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.

M4RC	CCM4RC	Extension of the CCM4R material model that also accounts for the effect of material creep and shrinkage.
SOLID Soil-Rock		
Drucker Prager	CC3DDruckerPragerPlasticity	Plastic materials with Drucker-Prager yield condition.
SHELL Concrete-Steel		
Shell Concrete-Steel	CCShellMaterial	Shell geometry with support Ahmad elements 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-axes points out of the shell. Other two axes, X and Y, must be in the shell plane. Such orientation must be ensured by user. In each shell node there are 3 displacement degrees of freedom and corresponding nodal forces. However, some DOFs are not free due to introduction of kinematic constrains ensuring shell displacement model. For more details see Theory Manual. **Shell material can be used only on 3D quadratic brick elements. (10.6)**
BEAM Concrete		
Beam Concrete	CCBeam3DMaterial	Special material, which activates the usage of special fiber beam element suitable for large scale analysis of complex structures with large elements 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 the it can be curvilinear, twisted, with variable dimensions of the cross-sections. Moreover, beam's cross-sections can be of any shape, optionally even with holes. The element belongs to the group of isoparametric elements with Gauss integration along its axis and trapezoidal (Newton-Cotes) quadrature within the cross-section. The integration (or material) points are placed in a way similar to the layered concept applied to shell elements, however, the "layers" are located in both "*s,t*" directions. **Beam material can be used only on 3D quadratic brick elements. (10.6)**

1D Reinforcement		
Reinforcement EC2	CCReinforcement	Material is like “Reinforcement”. You can generate material properties according the EC2
Reinforcement	CCReinforcement	Material for discrete reinforcement.
Interface		
Interface	CC3DInterface	Interface material for 2D and 3D analysis.
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 GiD-ATENA 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:	Static	Creep	Transport	Dynamic
CC3DElastIsotropic	X	X		X
CC3DBiLinearSteelVonMises	X	X		X

CC2DBiLinearSteelVonMises	X	X		X
CC3DBiLinearVonMisesWithTempDepProperties	X			
CC3DCementitious				
CC3DNonLinCementitious				
CC3DNonLinCementitious2	X	X		X
CC3DNonLinCementitious2User	X			
CC3DNonLinCementitious2SHCC	X			
CC3DNonLinCementitious2WithTempDepProperties	X			
CC3DNonLinCementitious3	X			
CCCombinedMaterial	X	X		X
CCCombinedMaterialWithTempDepProperties	X			
CCMicroplane4	X	X		X
CC3DInterface	X	X		X
CC2DInterface				
CCPlaneStressElastIsotropic	X			
CCPlaneStrainElastIsotropic				
CCPlaneStressSteel	X			
CCSBETAMaterial				
CC1DElastIsotropic	X	X		X
CCReinforcement	X	X		X
CCReinforcementWithTempDepProperties	X			
CCSmearedReinf	X	X		X
CCCyclingReinforcement	X	X		X
CCM4RC		X		
CC3DDruckerPragerPlasticity	X	X		X
CCSpringMaterial	X	X		
CCShellMaterial	X	X		X
CCBeam3DMaterial	X	X		X
CCModelB3		X		
CCModelB3Improved		X		
CCModelBP_KX		X		
CCModelCEB_FIP78		X		
CCModelCSN731201		X		
CCModelBP1		X		
CCModelBP2		X		
CCModelACI78		X		
CCModelBaXi94			X	

Materials with difficulty parameters used in more problem types are described below.

5.3.1 Shell Material

In this section is described shell material. Shell material has geometry with support Ahmad elements. 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-axes points out of the shell. Other two axes, X and Y, must be in the shell plane. Such orientation must be ensured by user. In this local system are all reinforcement and all outputs form post-processor.

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

Shell material can be used only on 3D quadratic brick elements.

With 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. 5-6 (a). Another option is to use a volume brick element at the corner, which is the only feasible way when more than two shells are connected, see Fig. 5-6 (b). The 'Shell Solid Contact' condition has to be assigned on the shell surface connected to the volume element for correct behavior. Connecting like in Fig. 5-7 is not recommended, as the master-slave relations induced by the fixed thickness of the shell may cause numerical problems.

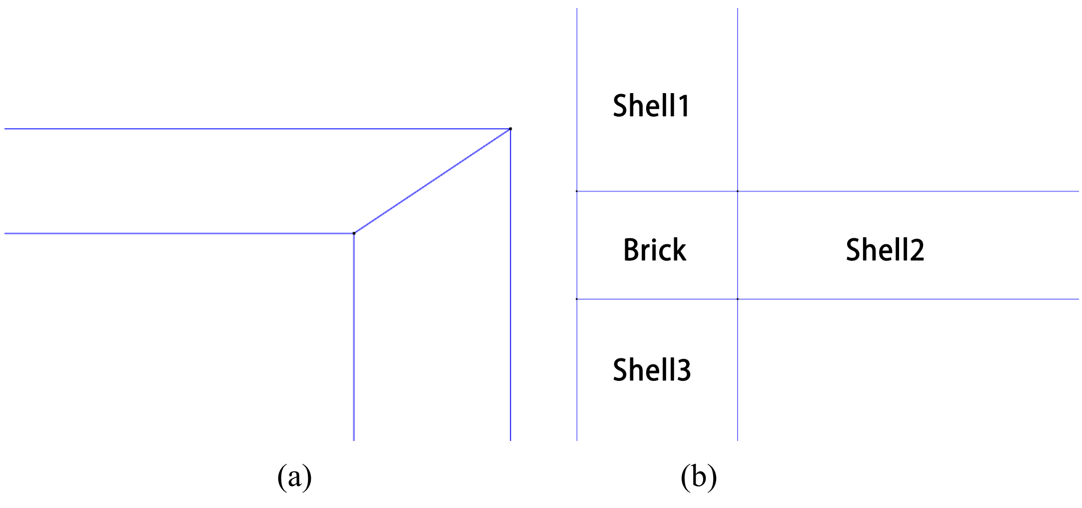


Fig. 5-6: Shell - recommended connection (a) 2 shells (b) 3 shells

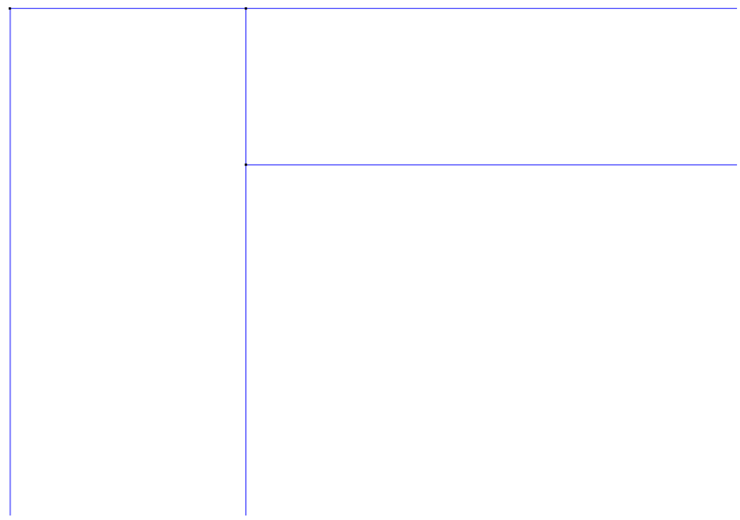


Fig. 5-7: Shell - not recommended connection

The ATENA implementation of the Ahmad shell element 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 CCSmeardReinforcement material law, which makes possible to account for the original reinforcement bars' direction.

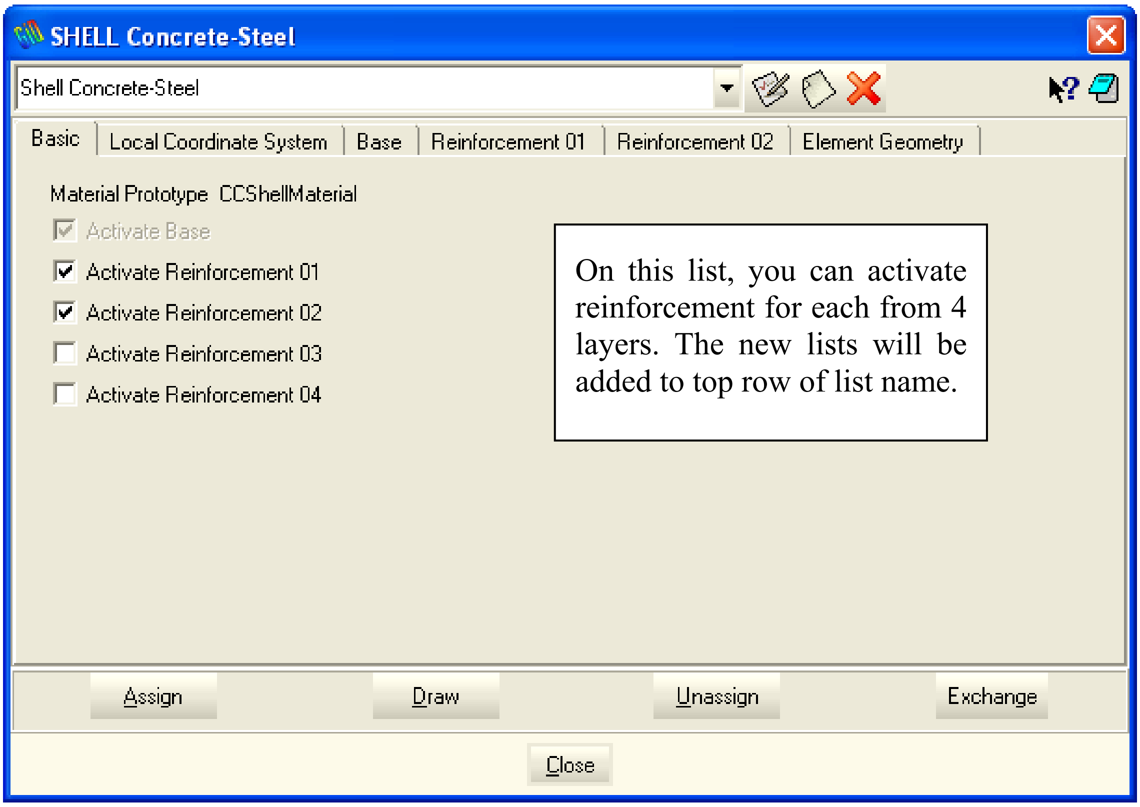


Fig. 5-8: Shell material properties - Basic

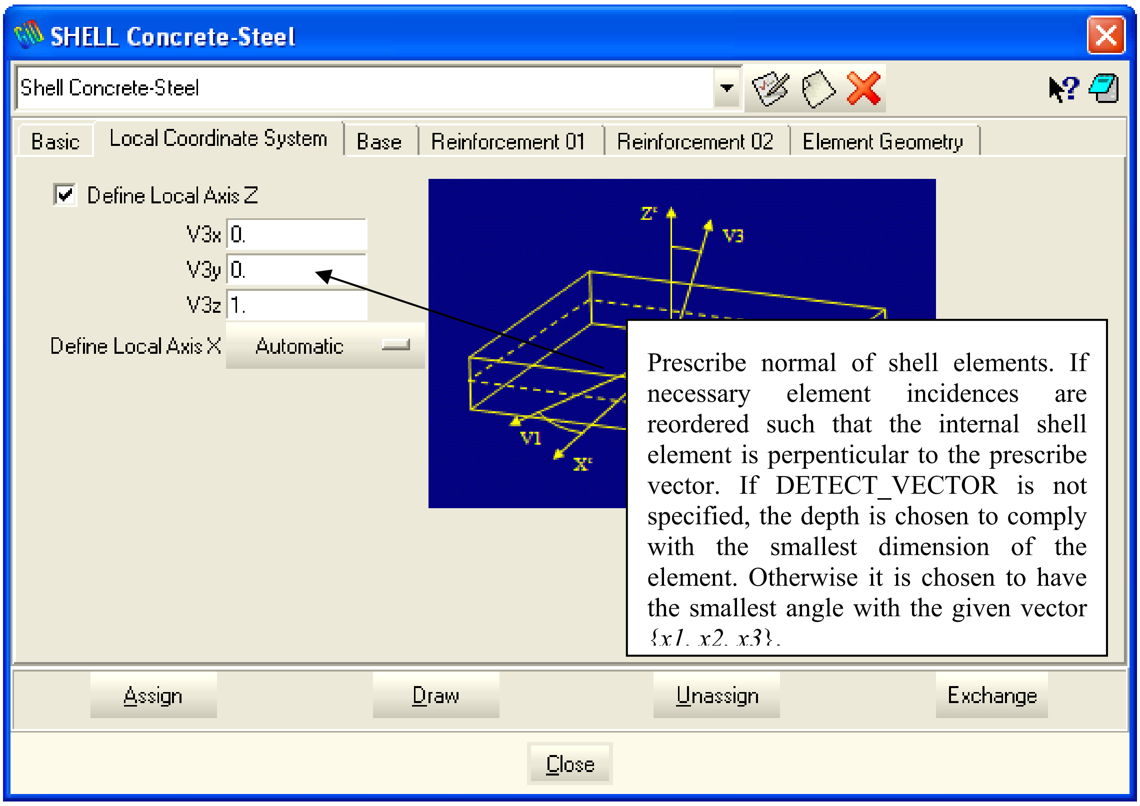


Fig. 5-9: Shell material properties – Local Coordinate System

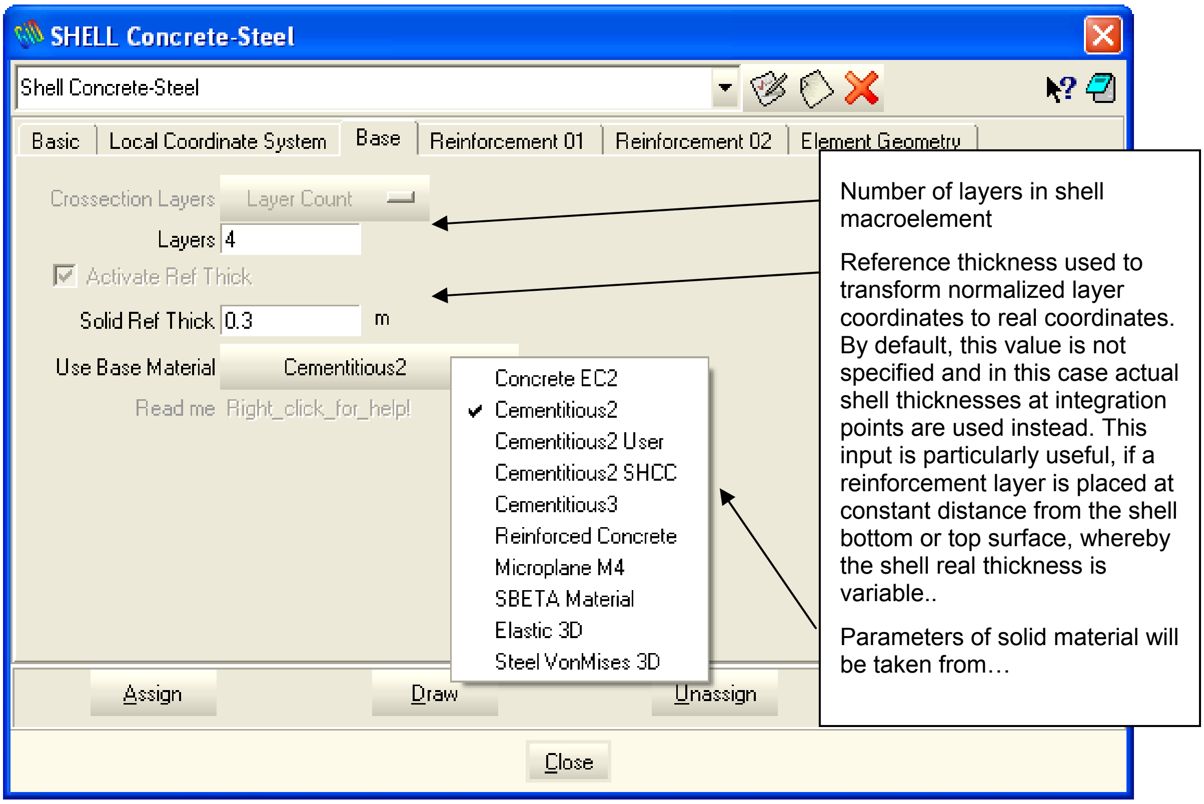


Fig. 5-10: Shell material properties - Bae

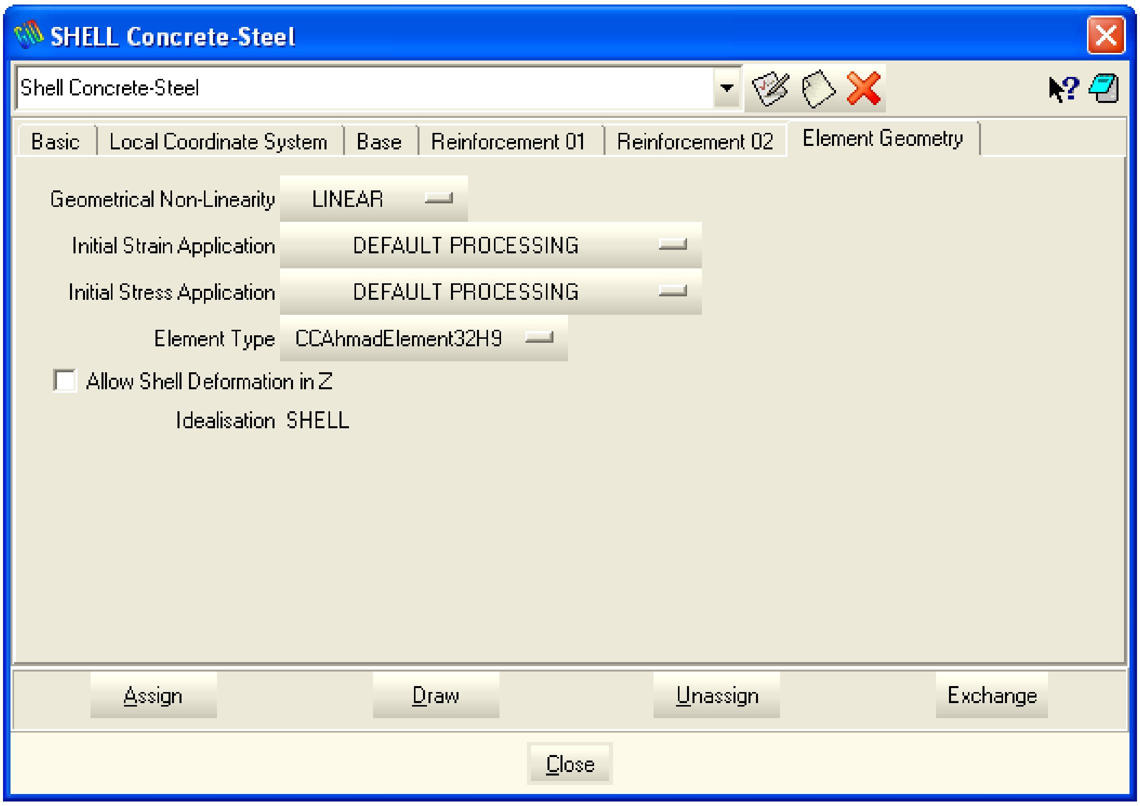


Fig. 5-11: Shell material properties – Element Geometry

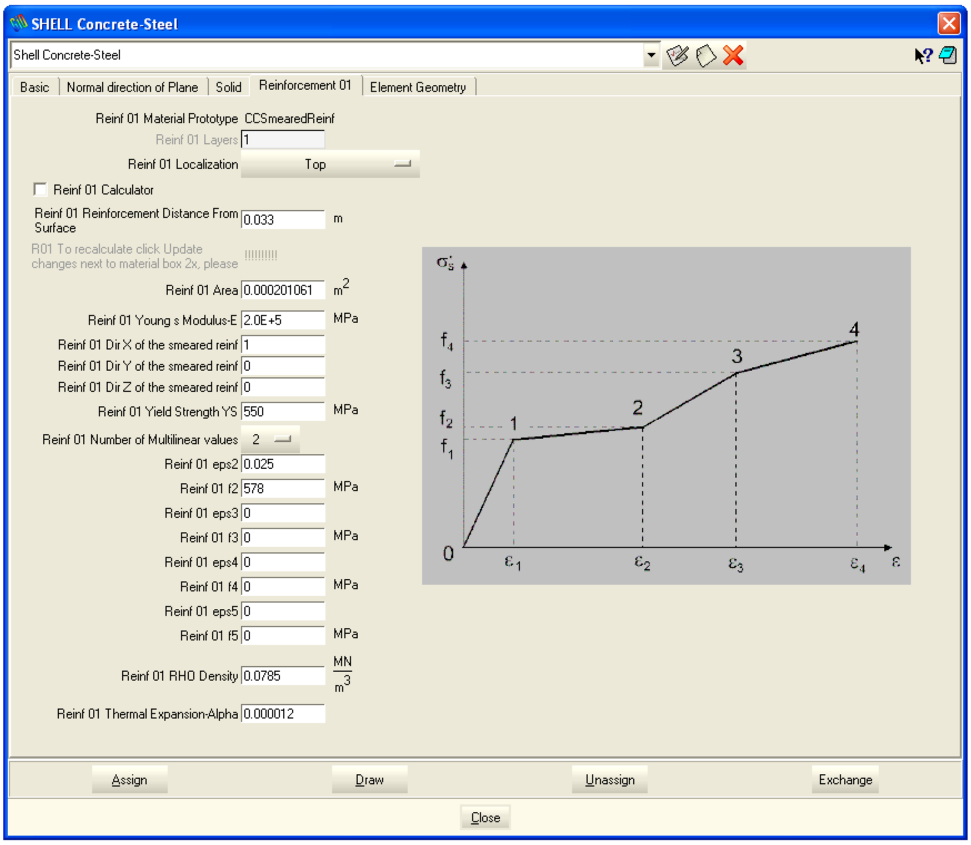


Fig. 5-12: Shell material properties - Reinforcement

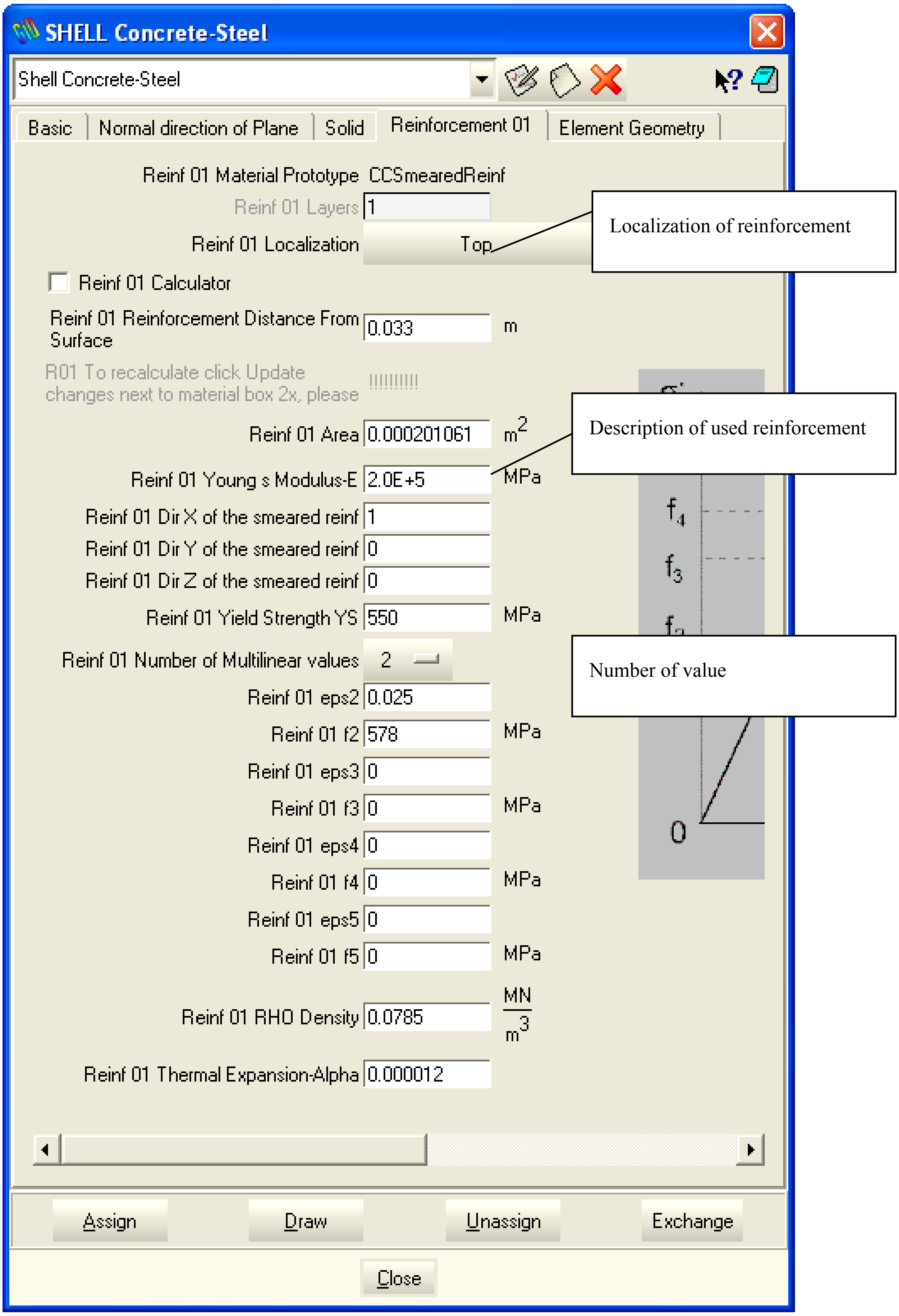


Fig. 5-13: Shell material properties – Reinforcement – detail

5.3.2 *Beam Material*

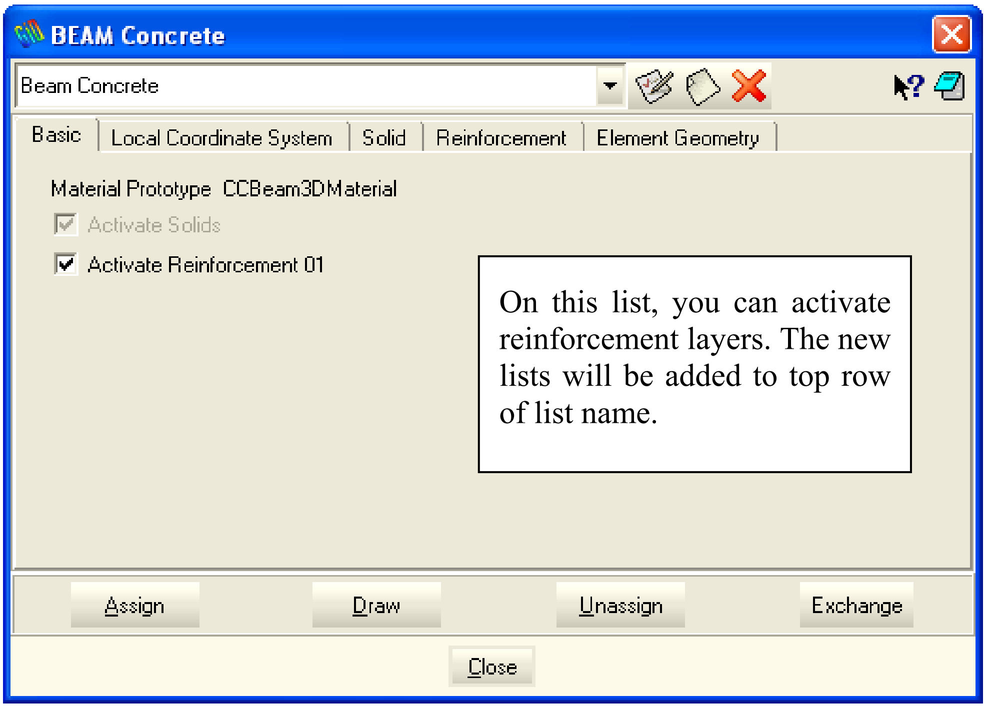


Fig. 5-14: Beam material properties – Basic

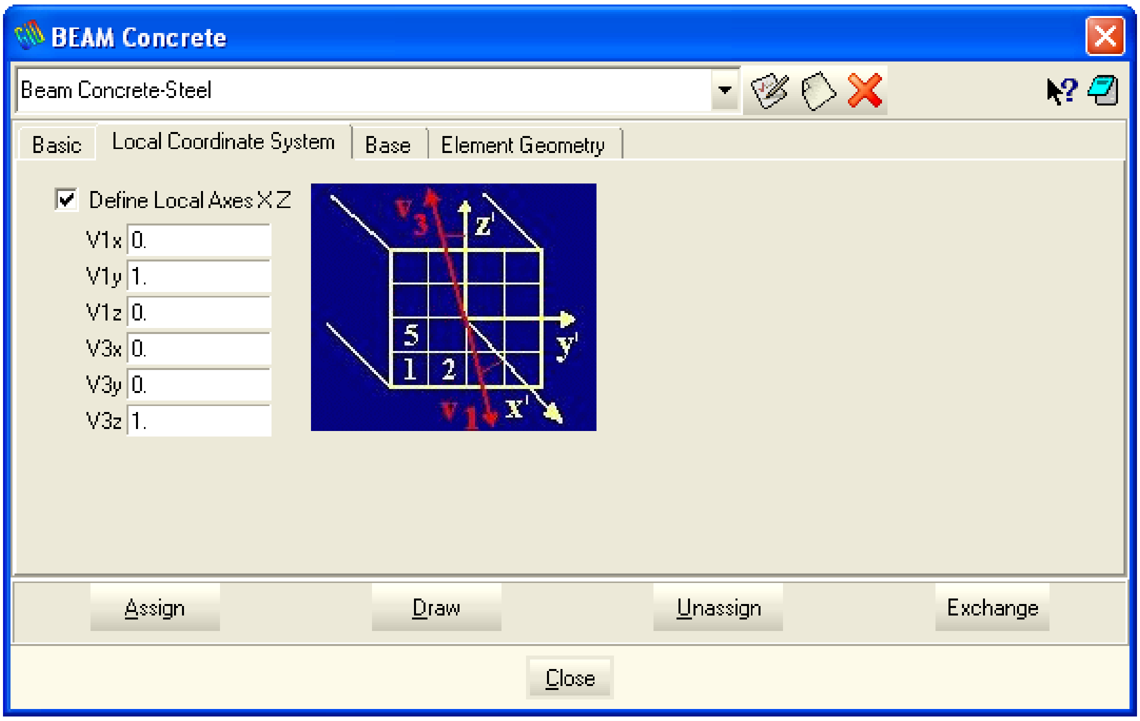


Fig. 5-15: Beam material properties – Local Coordinate System

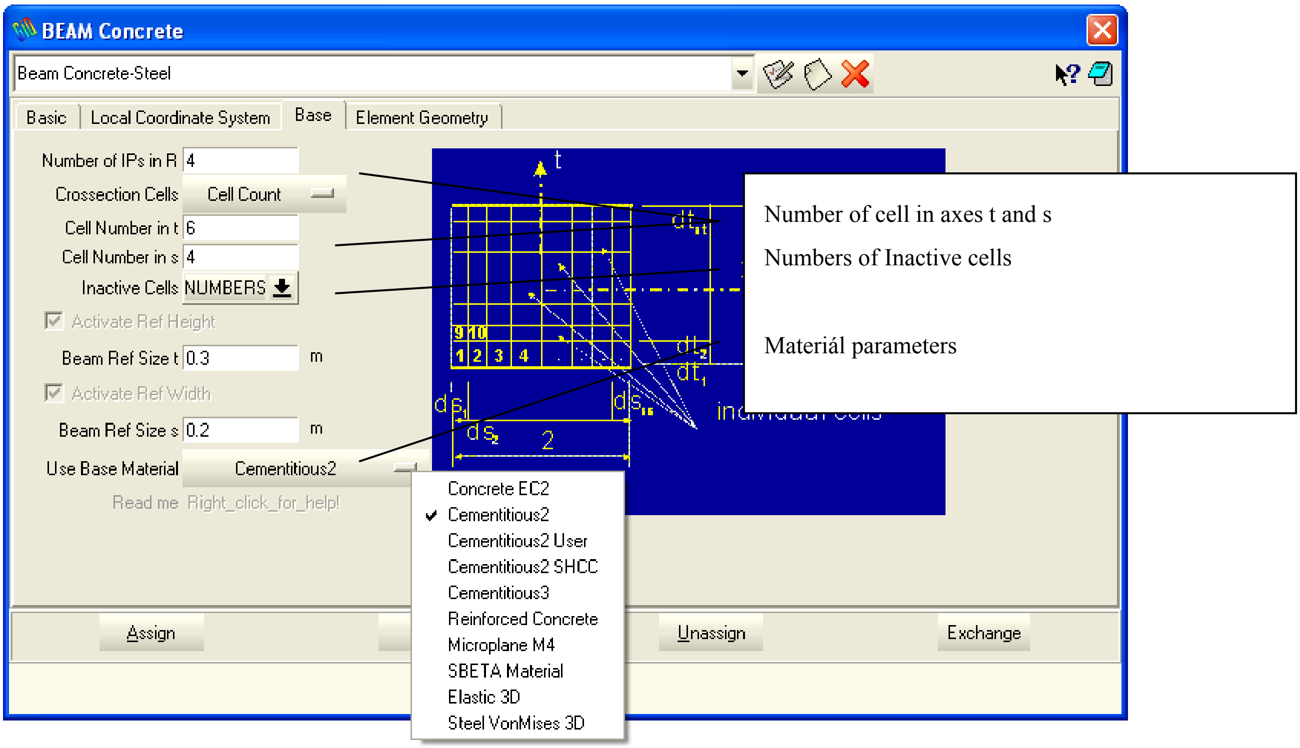


Fig. 5-16: Beam material properties – Base

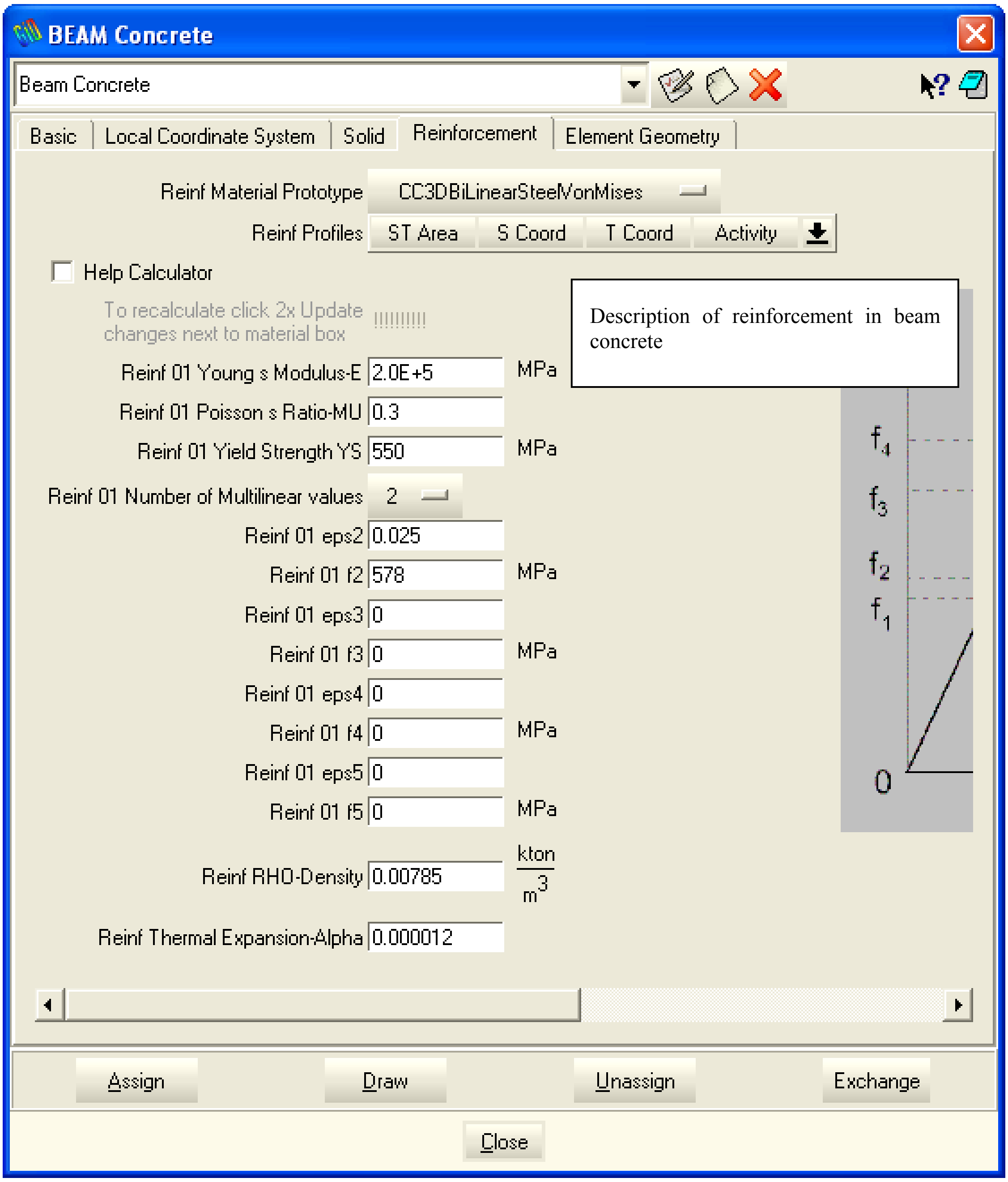


Fig. 5-17: Beam material properties – Reinforcement

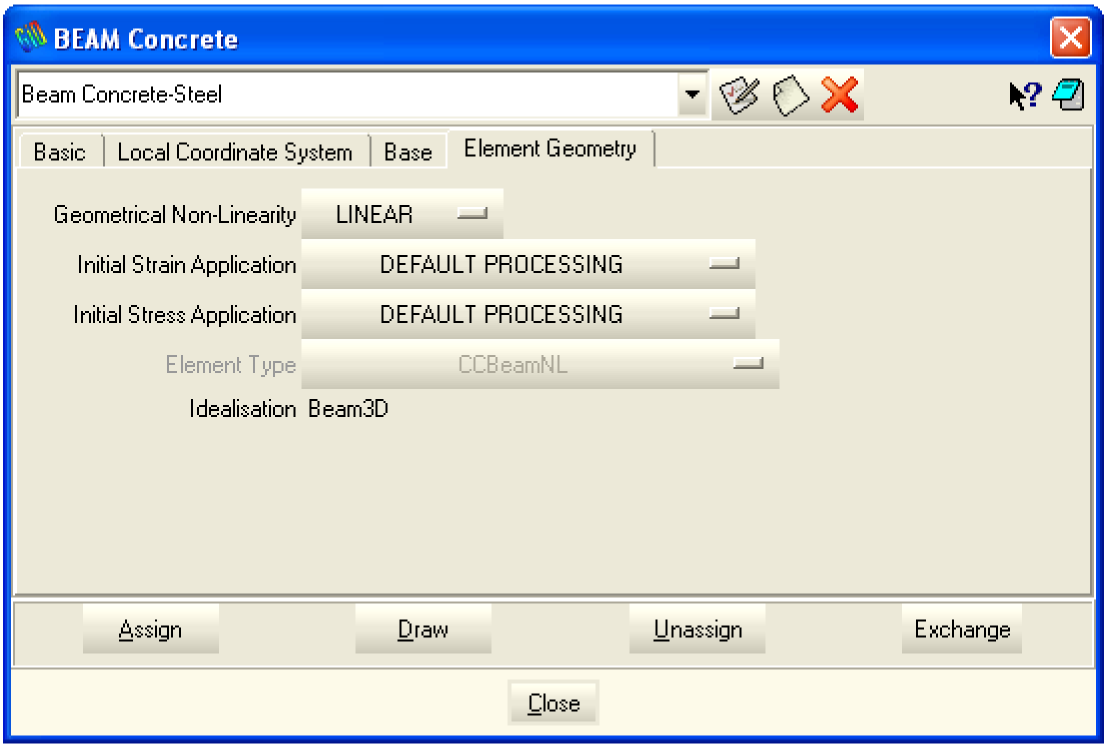


Fig. 5-18: Beam material properties – Element Geometry

5.3.3 Reinforced concrete

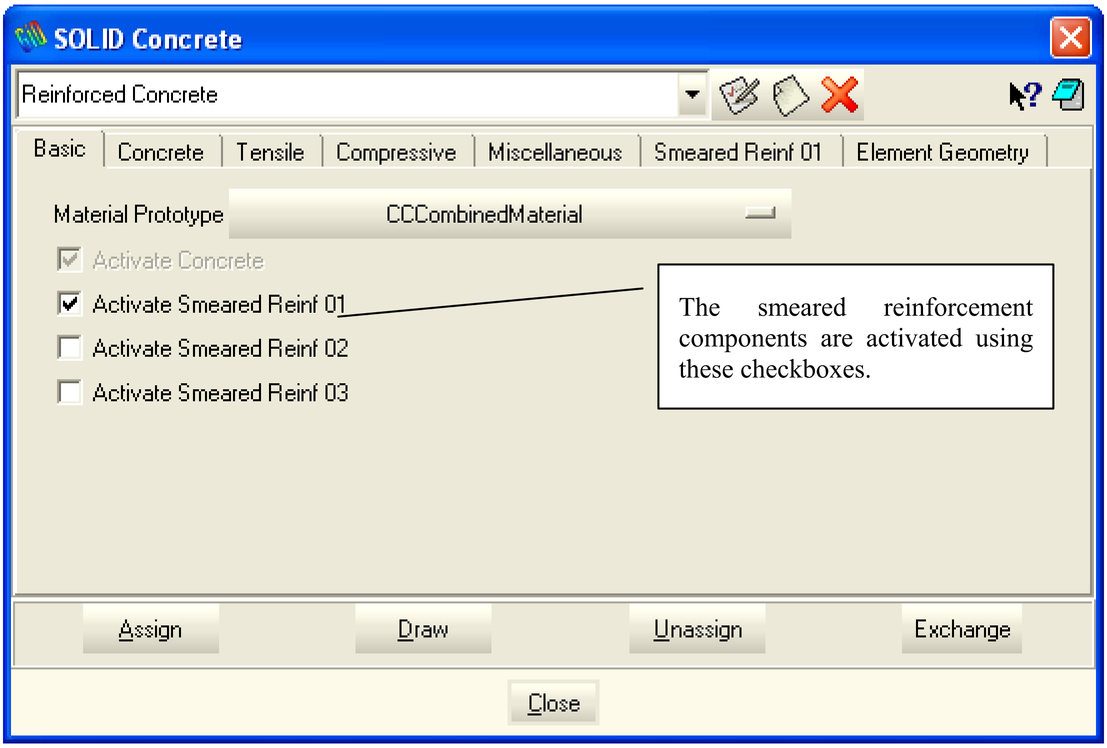


Fig. 5-19: Reinforced Concrete material properties – Basic

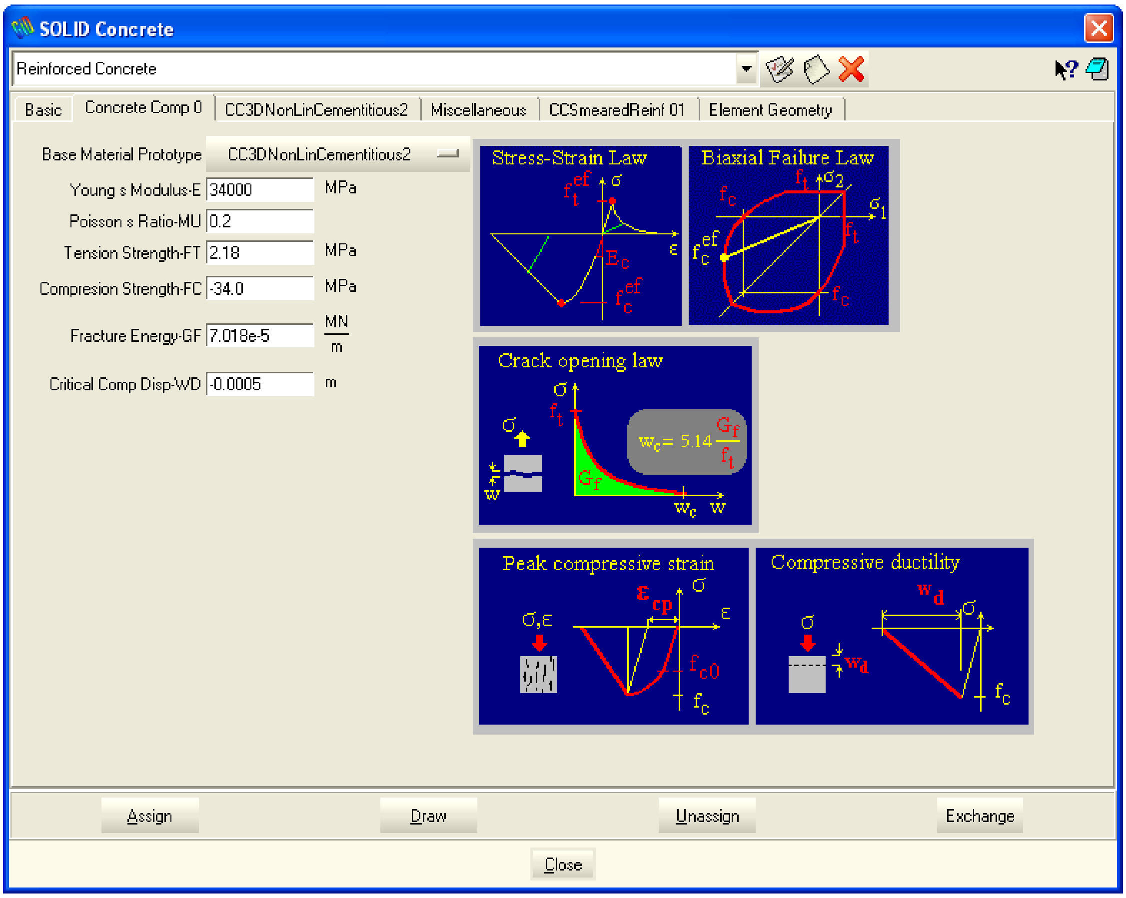


Fig. 5-20: Reinforced Concrete material properties – Concrete compressive

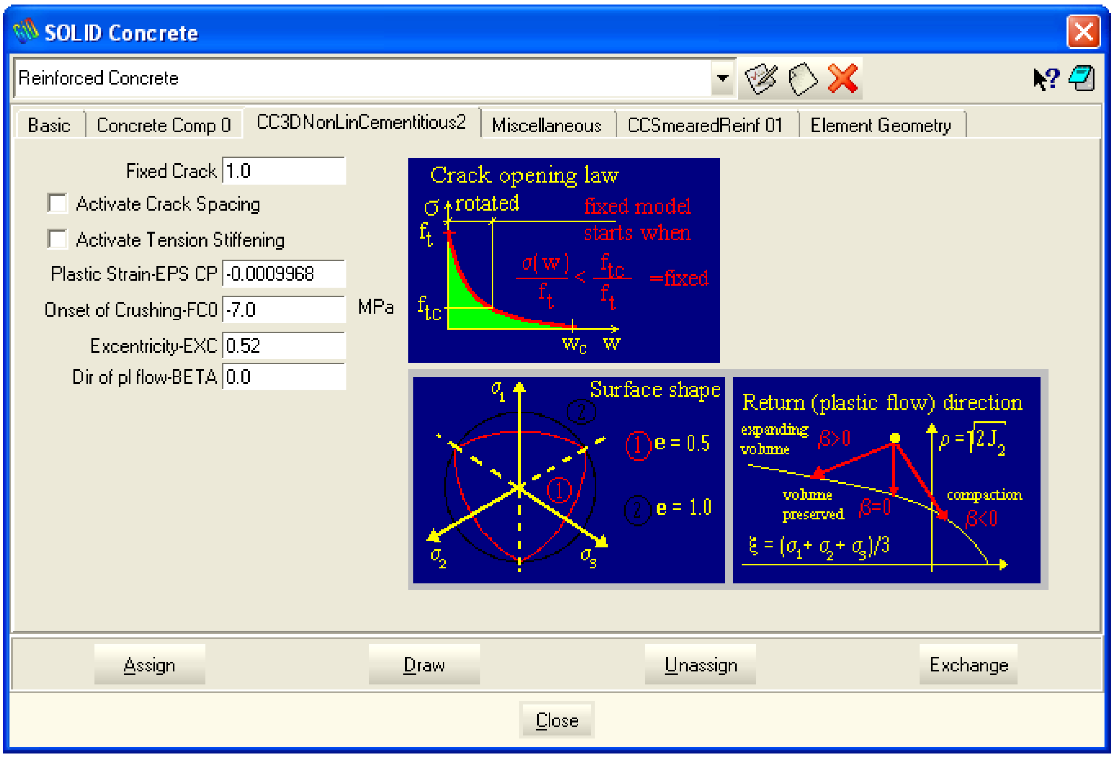


Fig. 5-21: Reinforced Concrete material properties – CC3DnonLinCementitious2

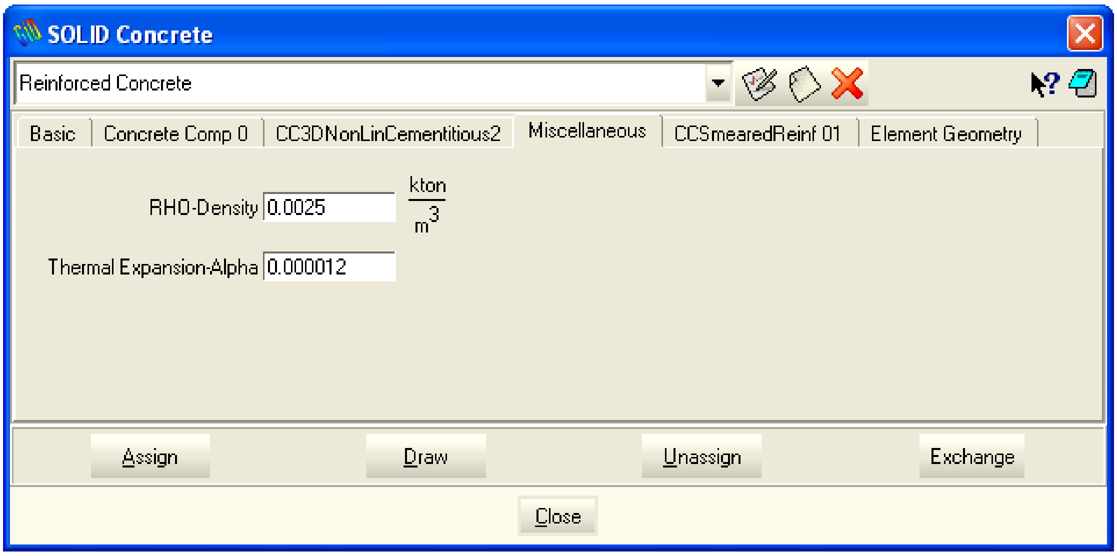


Fig. 5-22: Reinforced Concrete material properties – Miscellaneous

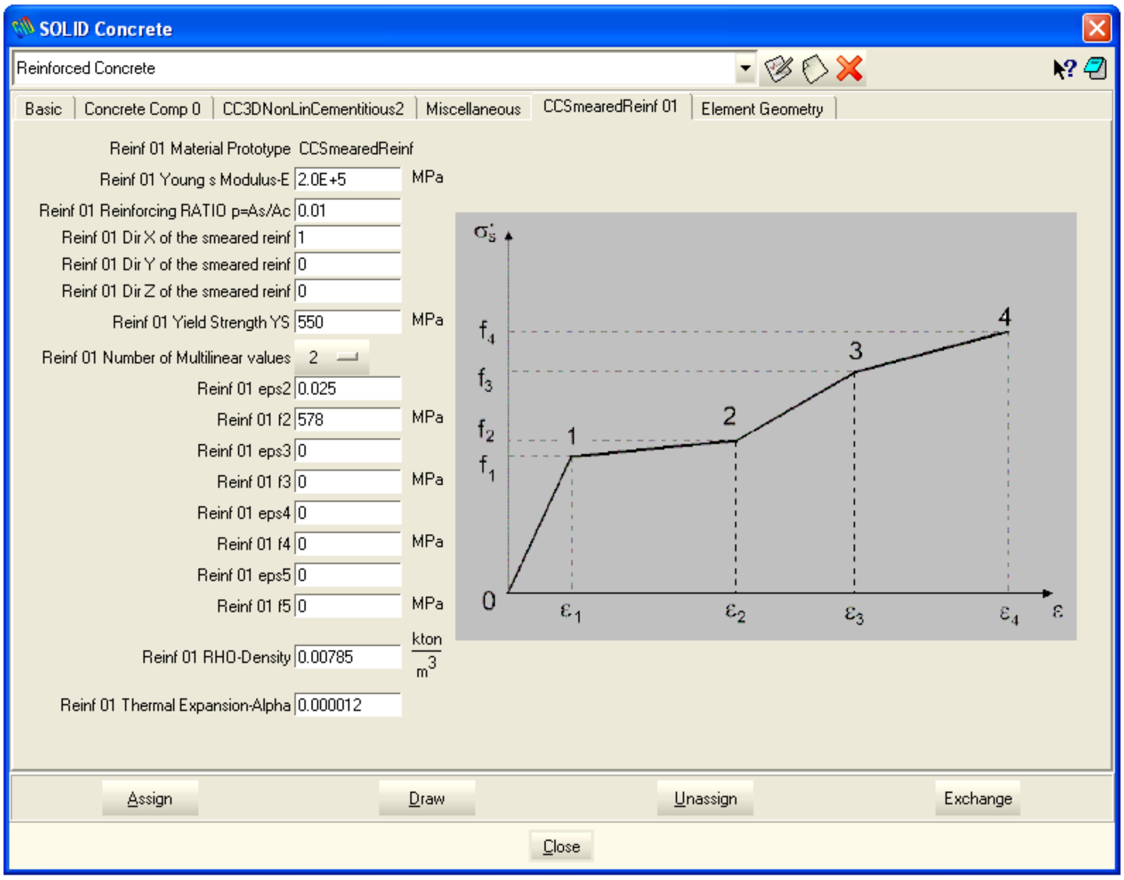


Fig. 5-23: Reinforced Concrete material properties – CCSmearedReinforcement

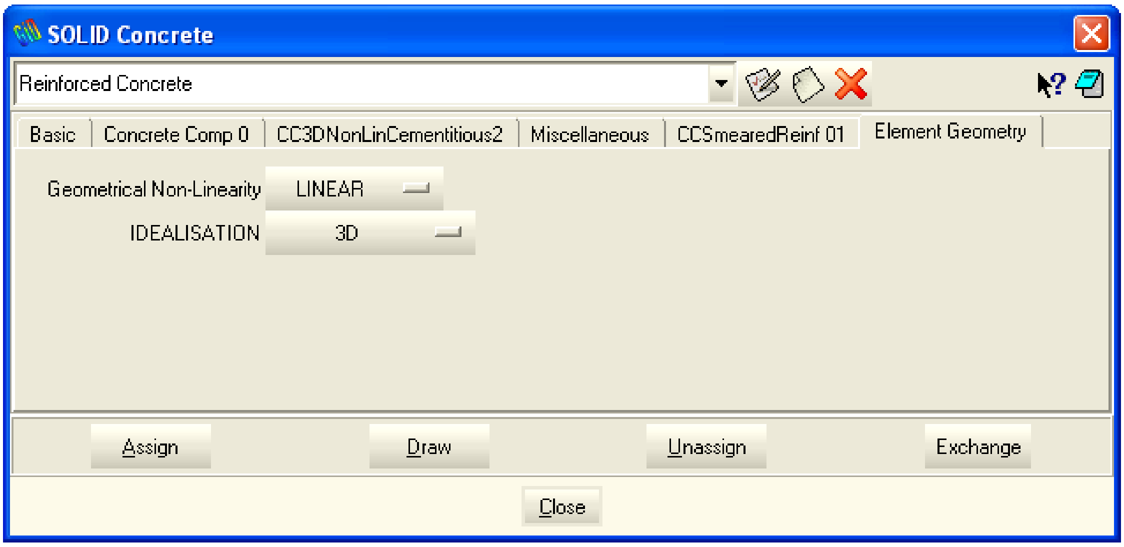


Fig. 5-24: Reinforced Concrete material properties – Element Geometry

5.3.4 *Interface Material*

The interface material (also called GAP) has been developed to model behaviour of contacts between volumes, e.g., concrete - steel or thin layers of, e.g., mortar. This material should only be assigned to contact volumes.

GiD only allows [prism] contact elements between surfaces of the same size and mesh settings. Therefore, if the two surfaces to be connected are of different sizes (partial contact) or with differing meshes, an extra surface needs to be defined of the size of the smaller of the two, located a small distance, e.g., 0.1mm, inside the volume the bigger surface belongs to. The easiest way usually is to copy the smaller surface. Then, create a contact volume from the two smaller surfaces and assign the desired interface (GAP) material to it. Finally, connect the additional surface to the bigger surface using Master-Slave conditions (*Boundary conditions - surfaces - fixed contact for surface*, see the *Conditions* section (5.2) for explanation of fixed contacts).

The normals of all surfaces have to point out of the volumes connected by the interface. The normal directions have to be fixed before creating the contact volume.

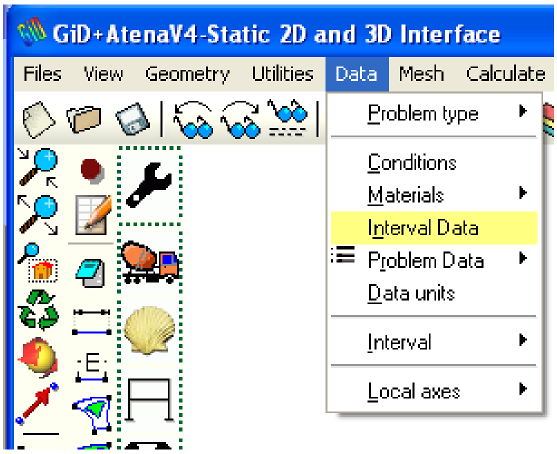
Refer to the *Interface Material Model* section of the ATENA Theory Manual for the explanation of the interface material parameters.

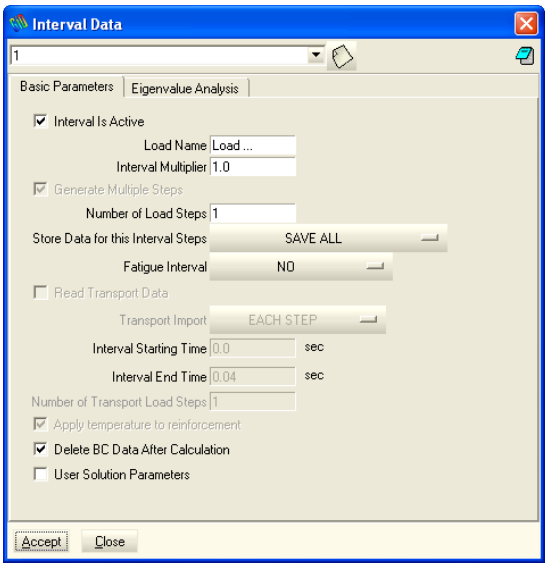
5.4 Interval data - Loading history

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. 5-25, which can be used to specify the parameters for an individual interval. In this dialog it 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 .





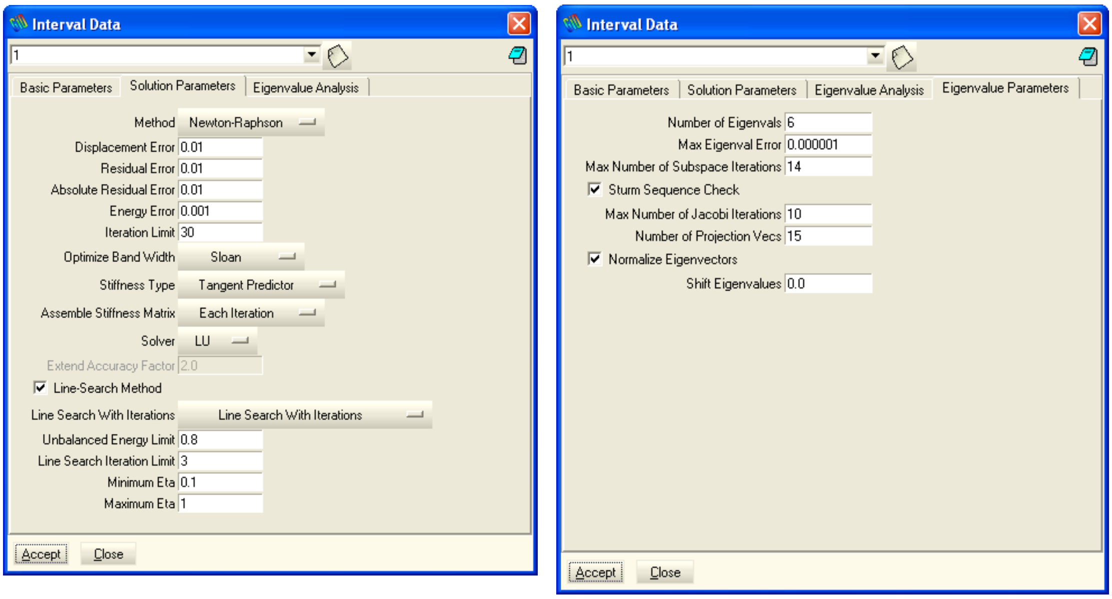


Fig. 5-25: Load steps (intervals)

5.5 Problem Data

The solution parameters such as number of iterations, convergence criteria or the solution methods for an ATENA analysis are defined in the menu item 'Data | Problem Data', Fig. 5-26 or icon . The dialog window is opened and default data are offered. At the top section 'Task name' 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.

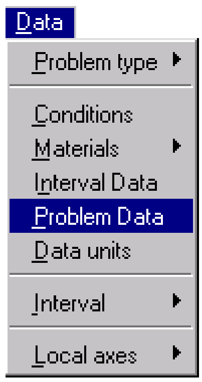


Fig. 5-26 Problem Data.

The middle section 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].

The last section in bottom of this window makes it possible to generate a load history of identical load steps. In this case the box in front of 'Automatic gen. Load Step' is checked and number of load steps and step multiplier can be entered. It takes the last interval (load step) defined in GiD and repeats it. The series of intervals defined in GiD is extended by indicated number of intervals. This generation is done after finishing GiD modelling and before ATENA analysis.

The box in the bottom enables to activate writing data from all load steps to files. The format of file name is *TaskName.iii*, where *TaskName* is the name given the most top box and *iii* is the load step number.

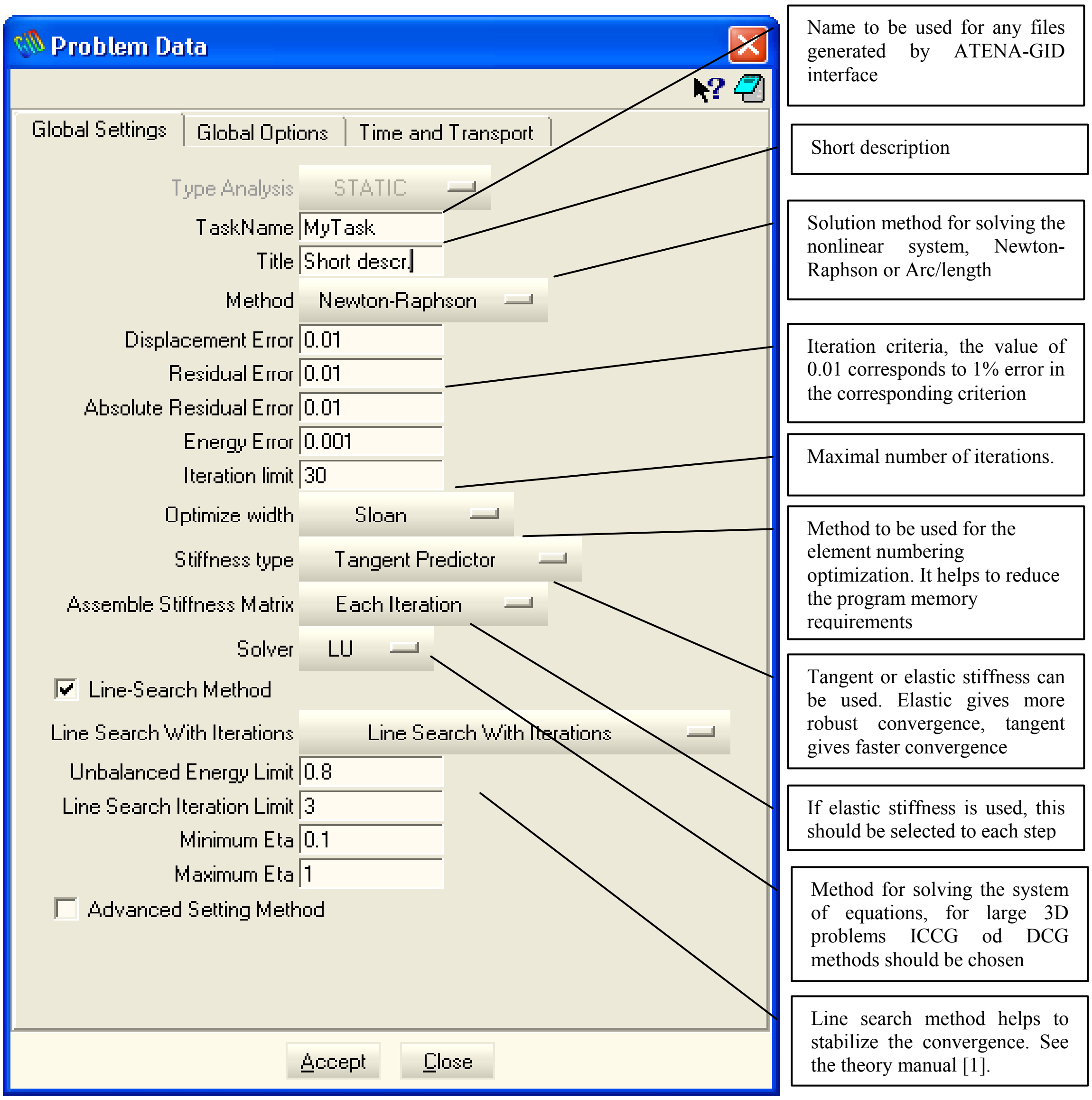


Fig. 5-27 Problem data – Solution parameters.

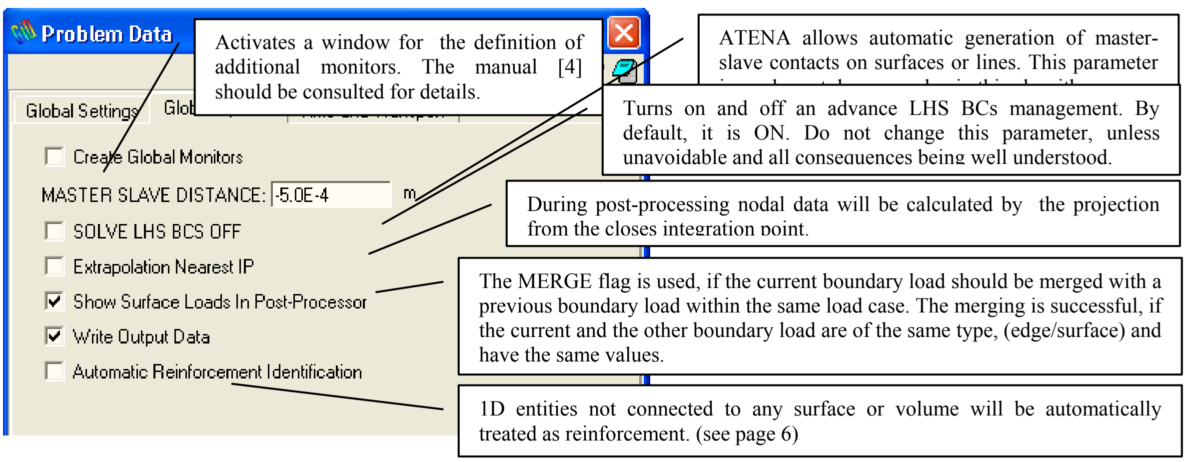


Fig. 5-28 Global options in problem data dialog

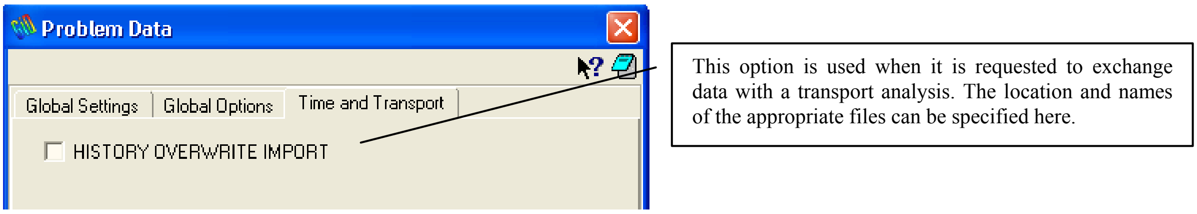


Fig. 5-29 Time and Transport options in problem data dialog

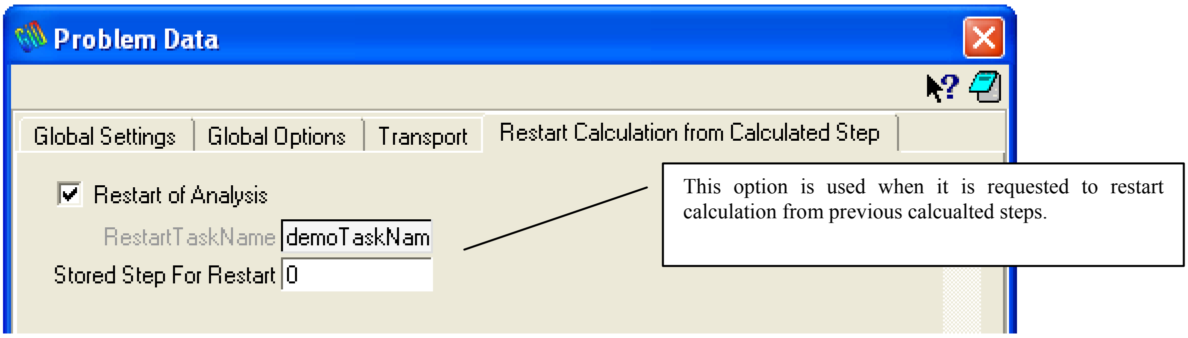


Fig. 5-30 Restart calculation options in problem data dialog

5.6 Units

Standard units in ATENA are SI units, which are active automatically as a default unit set, Fig. 5-31. It is also possible to define other sets of units. This can be done in the menu ‘Data | Data units’, where in the dialog window ‘Problem units’ you can change the “Base system”.

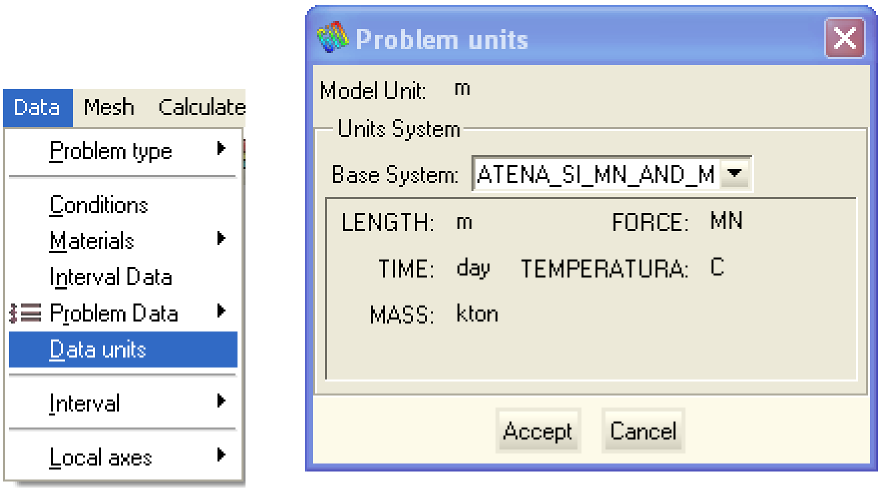


Fig. 5-31 Data units, default set.

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

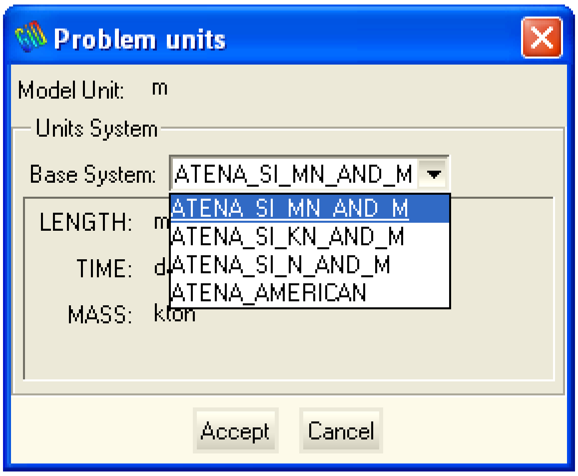


Fig. 5-32 Definition of units and possible set of alternative units.

5.7 Finite Element Mesh

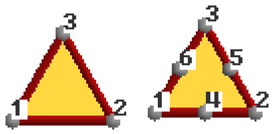
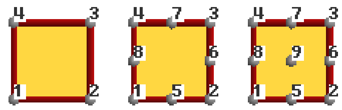
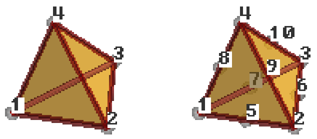
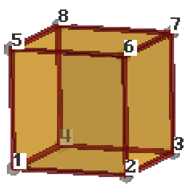
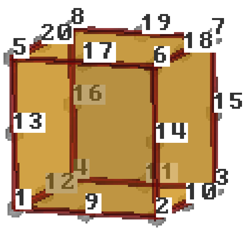
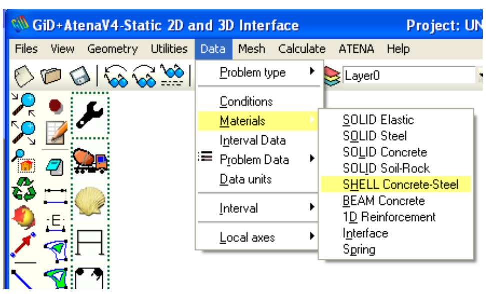
The generation of a finite element mesh in GiD is done from the menu 'Meshing'. Please, refer to GiD documentation for details. Here, we shall mention only meshing of reinforcing bars, which is specific for ATENA.

The geometrical model of a bar (discrete) reinforcement is modelled by one dimensional entities, i.e. lines. Since GiD does not have a capability to generate embedded bar elements, this operation is performed later at the beginning of the ATENA analysis. For this we need to export the geometrical forms of the bars. Since GiD can export only finite elements, it is always necessary to first generate some 1D truss elements along each line, which represents the reinforcement (see also page 7). It is therefore recommended to select the meshing properties of these reinforcement lines such that a single finite element is generated by GiD. This finite element is then used in ATENA to generate the embedded discrete bars depending on its intersections with the solid model. Of coarse, 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 Finite Elements for ATENA

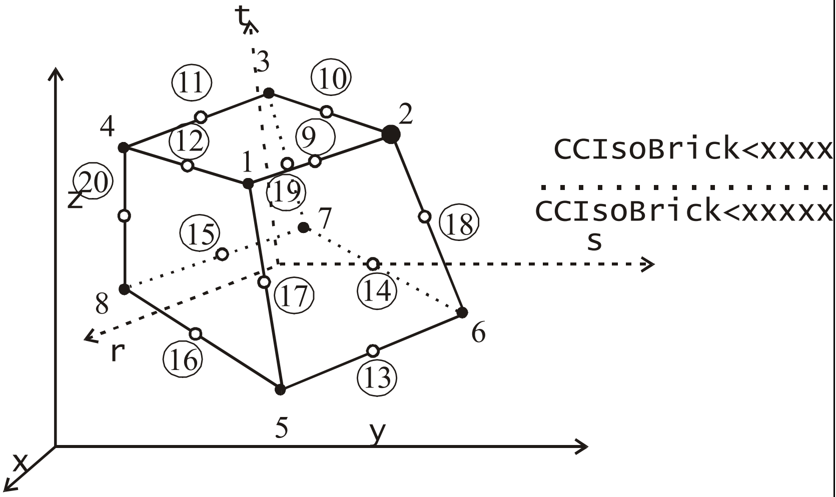
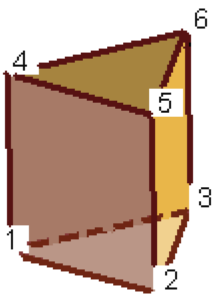
In each volume we must choose a type of finite element. Following types can be used in ATENA (in parenthesis we give also the number of nodes and a code name used in ATENA).

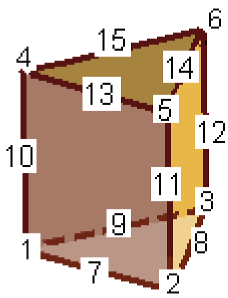
Table 3: Element library compatibility

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

20-nodes, CCBeamNL – this is another special 3D element available in ATENA. This element on the input appears as standard 20 node element, but internally it is formulated as a</td></tr>
</table>

<table>
<tr>
<td></td>
<td>fiber beam element. It is suitable for large scale analysis, when meshes with large elements are necessary.

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

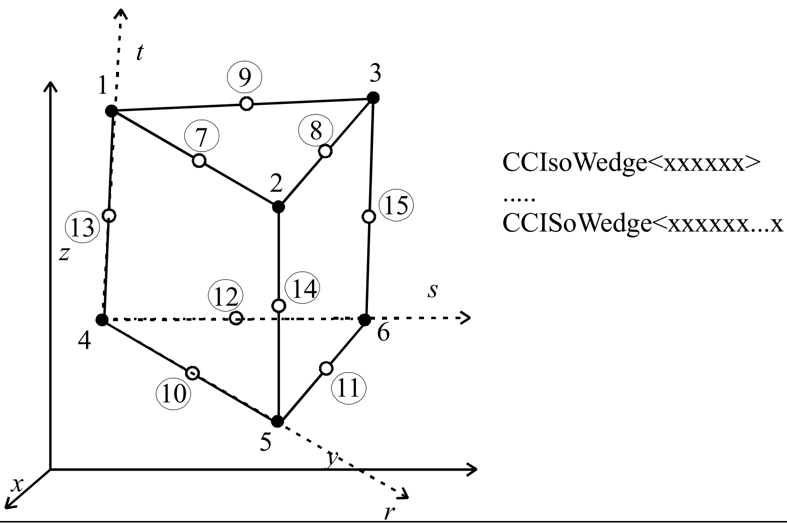

</td>
</tr>
<tr>
<td>

</td>
<td>Linear and quadratic Wedge (structured mesh)

6-nodes, CCIsoWedge<xxxxxx>

15-nodes, CCIsoWedge<xxxxxxxxxxxxxxx>

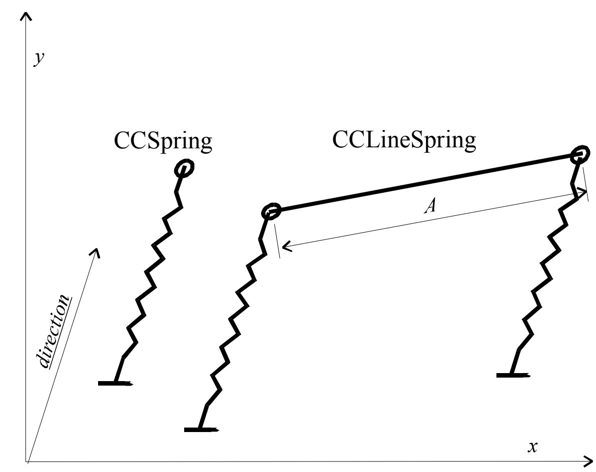
However, ATENA is using a different nodal numbering, this means that during the export of the ATENA input file, the nodal numbering is modified to correspond with the ATENA format, as it is described in the figure below.


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

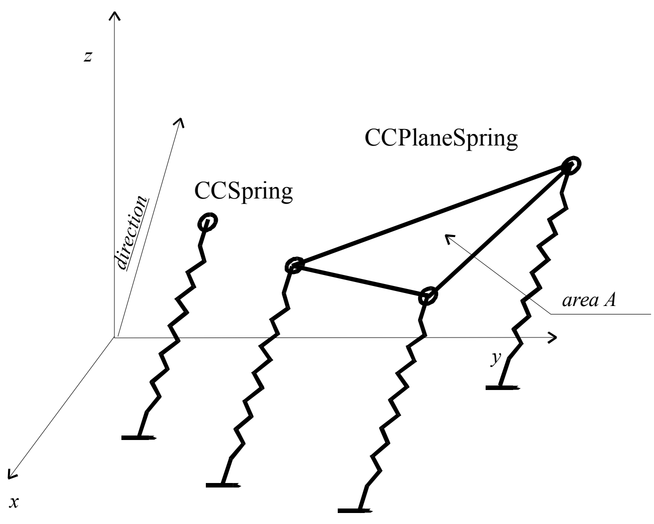
<table>
<tr>
<td></td>
<td>define springs that are normal to a curved surface or line.

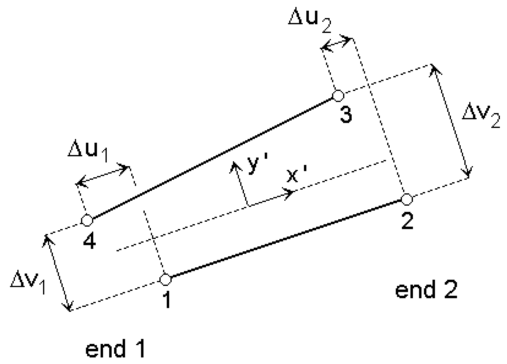
CCSpring – 2D and 3D element to model spring-like boundary conditions at a point,

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



CCPlaneSpring – 3D element to model spring-like boundary conditions along a triangular area.


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

	 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>

6 STATIC ANALYSIS

Static analysis is activated in GiD by selecting an appropriate problem type Static (see the menu items Data | Problem Type | AtenaV4). The making of model it's a same like others problem data. Its neccessarry to assign Conditions [5.2], for each macroelement assign material properties [5.3, 10.4], define the interval data [5.4, Fig. 5-25, Fig. 6-1] and problemtype properties [Fig. 5-27, 10.2], meshing model [5.7, 10.6] and execute program.

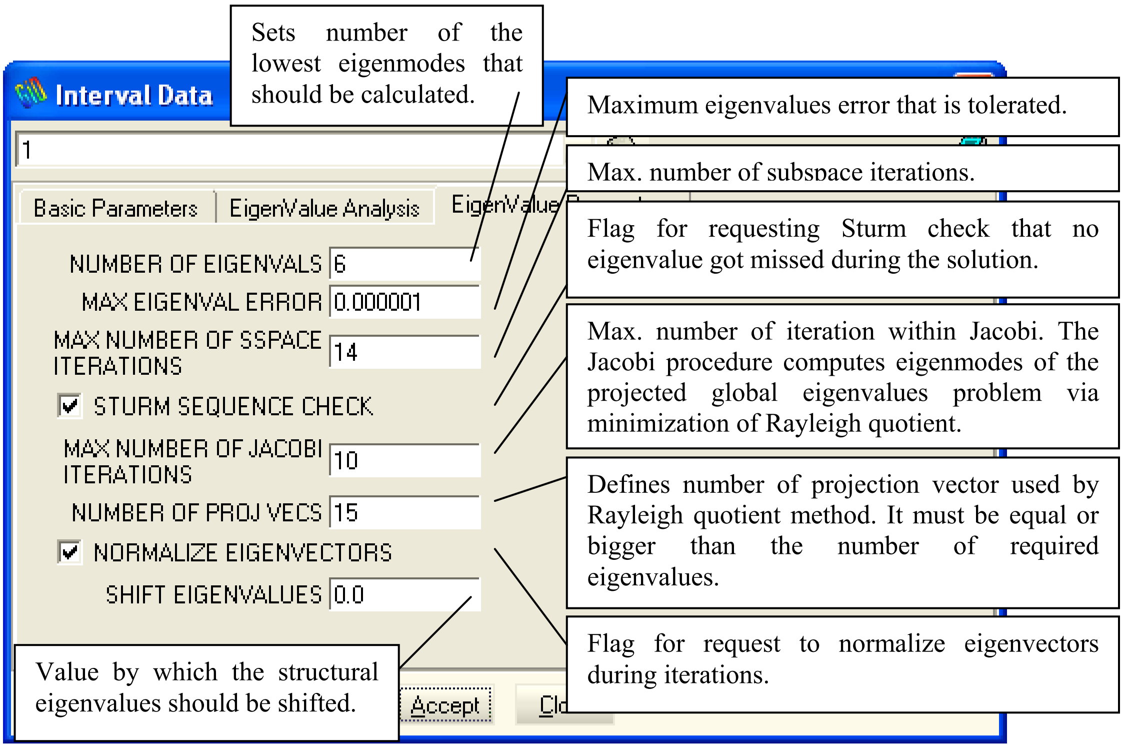


Fig. 6-1: Settings of EigenValue Analysis

Detailed example of static analysis is at full length in section [10].

7 CREEP AND SHRINKAGE ANALYSIS

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

The ATENA software supports two kinds of creep and shrinkage analysis. The first kind involves only mechanical analysis of the structure. It is assumed that the structure has everywhere more or less similar humidity and temperature conditions and the same applies for ambient environment. The corresponding problem type for this kind of analysis is Creep, and it is accessible via menu item Menu | Data | Problem type | AtenaV4.

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

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

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

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

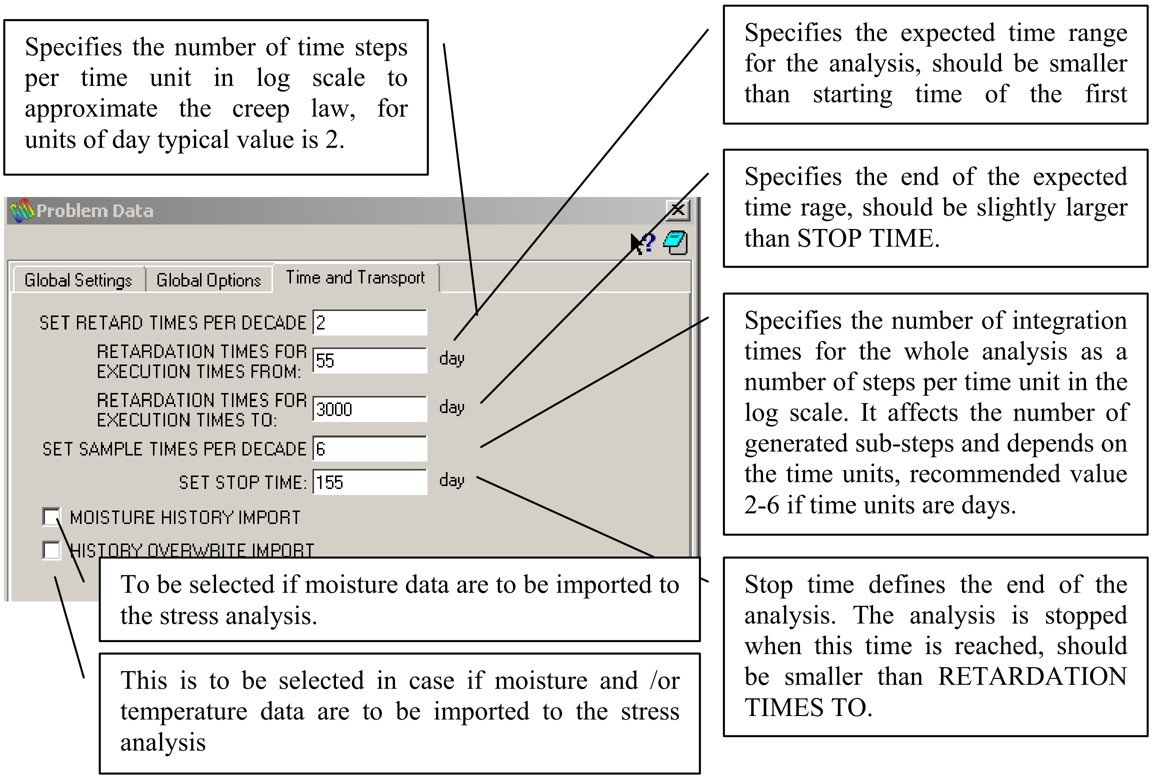
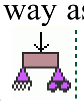


Fig. 7-1 Problem data dialog.

7.1 Boundary conditions and load cases related input

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

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

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

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

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

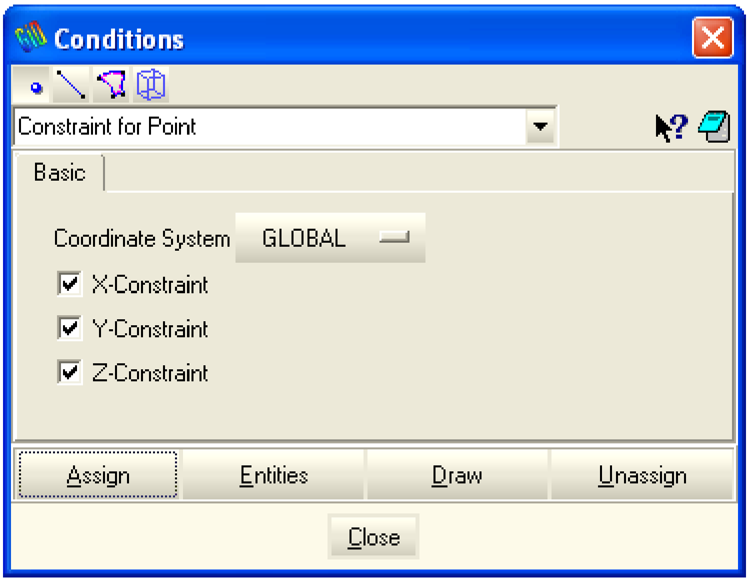


Fig. 7-2 Boundary conditions dialog in creep analysis

7.2 Material input data

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

The input data in GiD reflect this structure. The user has to specify two sets of parameters, one for the creep prediction model, one for the base material model and each such a set is assigned a dedicated date sheet. The actual data input dialog is invoked by pressing the icon (or via menu Data | Materials | Creep), and it is shown in Fig. 7-3

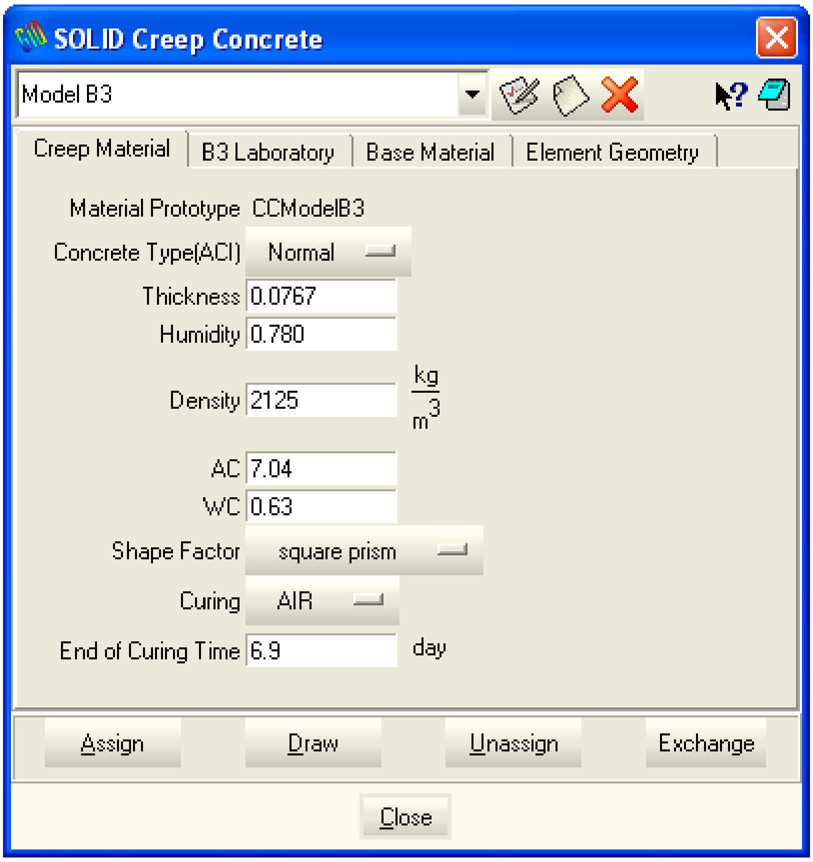


Fig. 7-3 Material input dialog

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 literature that is referred to.

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

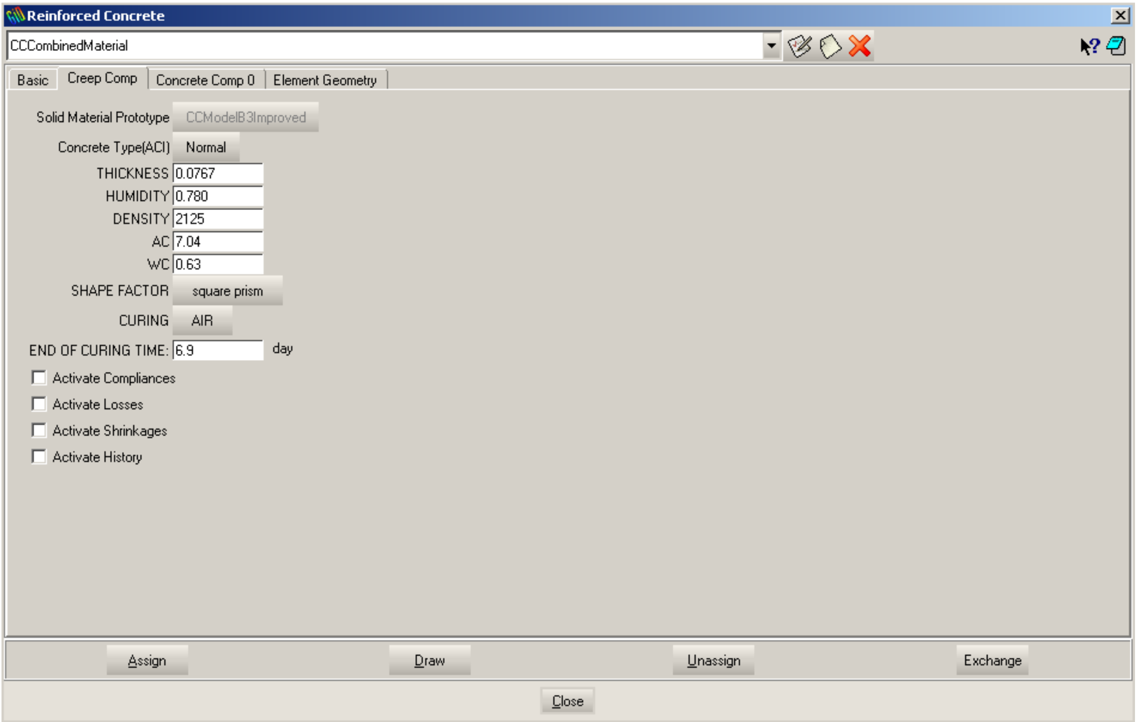
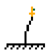


Fig. 7-4 Reinforced concrete material with smeared reinforcement

The dialog has several pages, each corresponding to a particular type of data. For example the sheet “Creep Comp” serves for input data for creep prediction model (and it resembles the dialog called by pressing . The sheet “Concrete Comp” includes input data for short-term model for concrete, (similar to that invoked by , etc. The individual smeared reinforcement components will appear under the label Concrete Comp 0 – 3.

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

8 ANALYSIS OF MOISTURE AND HEAT TRANSPORT

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 and shrinkage analysis. Its goal is to calculate moisture and temperature conditions in the structure. As a result, we get histories of temperature and moisture variation at each material point of the structure, and these data are later used by a stress analysis or creep material model to better predict stress-strain relationships with the effects of temperature, creep and shrinkage.

Main use of moisture and heat transport analysis is to calculate temperature increments inside a structure. These increments are later used in calculation of elements' thermal expansion and associated initial strains 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 Temperature (see the menu items Data | Problem Type | AtenaV4). Currently, only one material model is supported. The corresponding input data dialog appears by pressing the icon  :

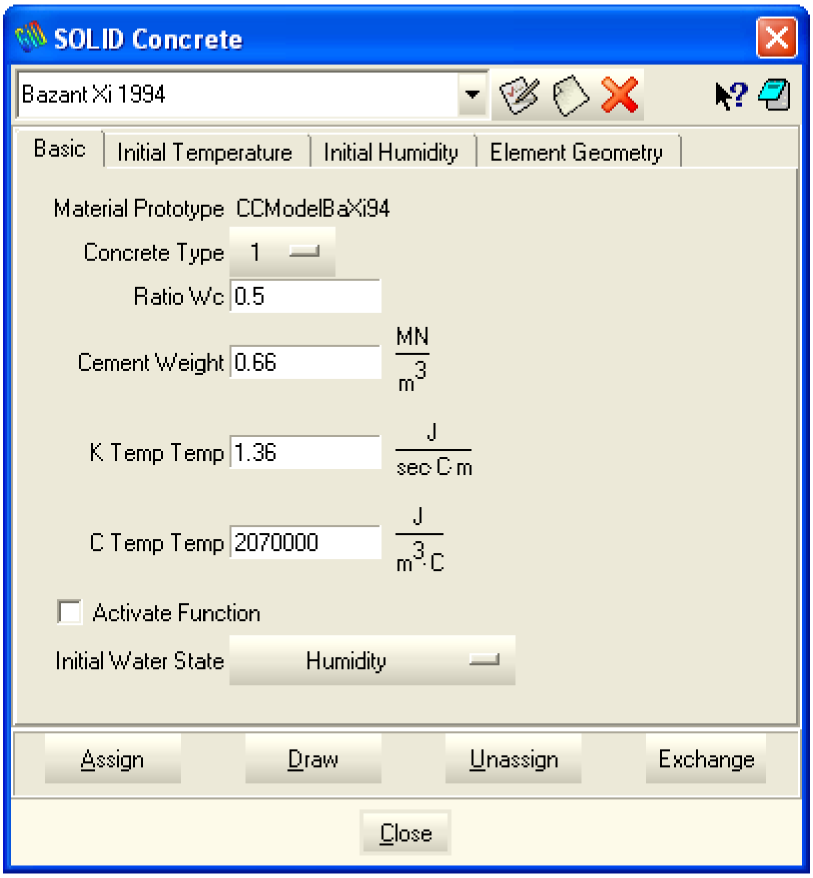


Fig. 8-1 Heat and moisture transport material model dialog

The model name is CCModelBaXi94. Its moisture transport part is based on Bazant-Xi model (see the manual for Atena theory [1]) that has been developed for the modelling mortar behaviour. It accounts for water and cement paste only and hence, in case of concrete mixture it neglects the presence of aggregate. Consequently, the model can be used only, 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. 8-1 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.

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

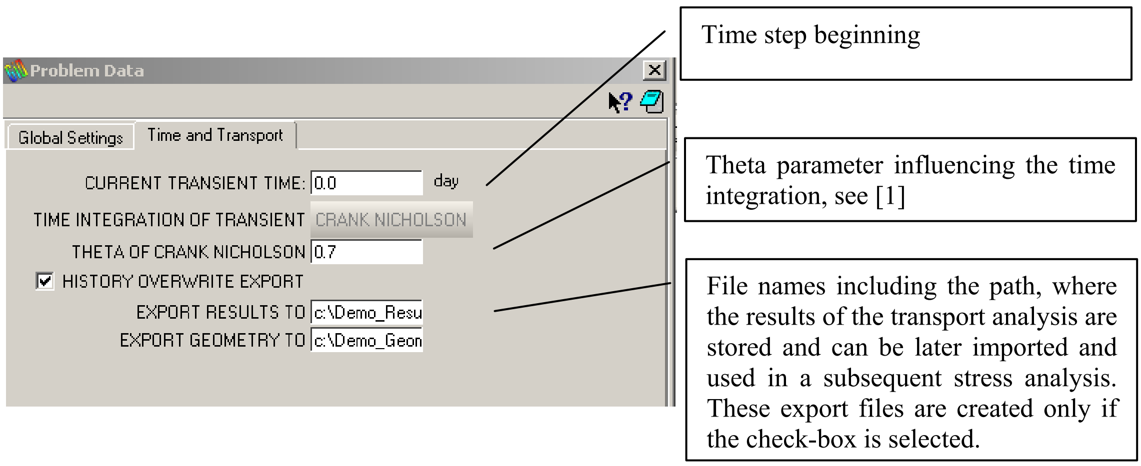


Fig. 8-2 Time and transport data sheet

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 History Overwrite Export checkbox must be checked). The 1st of them contains actual humidity and temperature histories of the structure and the 2nd file keeps information about geometry of the model. The exported data are compatible with import data format of creep and shrinkage analysis, (or by element temperature load for static analysis without creep). Hence, it is very easy to transfer the histories between this analysis and any other analysis that can make use of it. This means that it is not necessary to use the same model or finite element mesh in the transport and stress analyses. During the import, the program ATENA automatically determines the closes nodes and makes the necessary interpolation.

The dialog in Fig. 8-3 (available by pressing ) is used to define one or multiple execution type steps. Meaning of the parameters is self-explanatory 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.

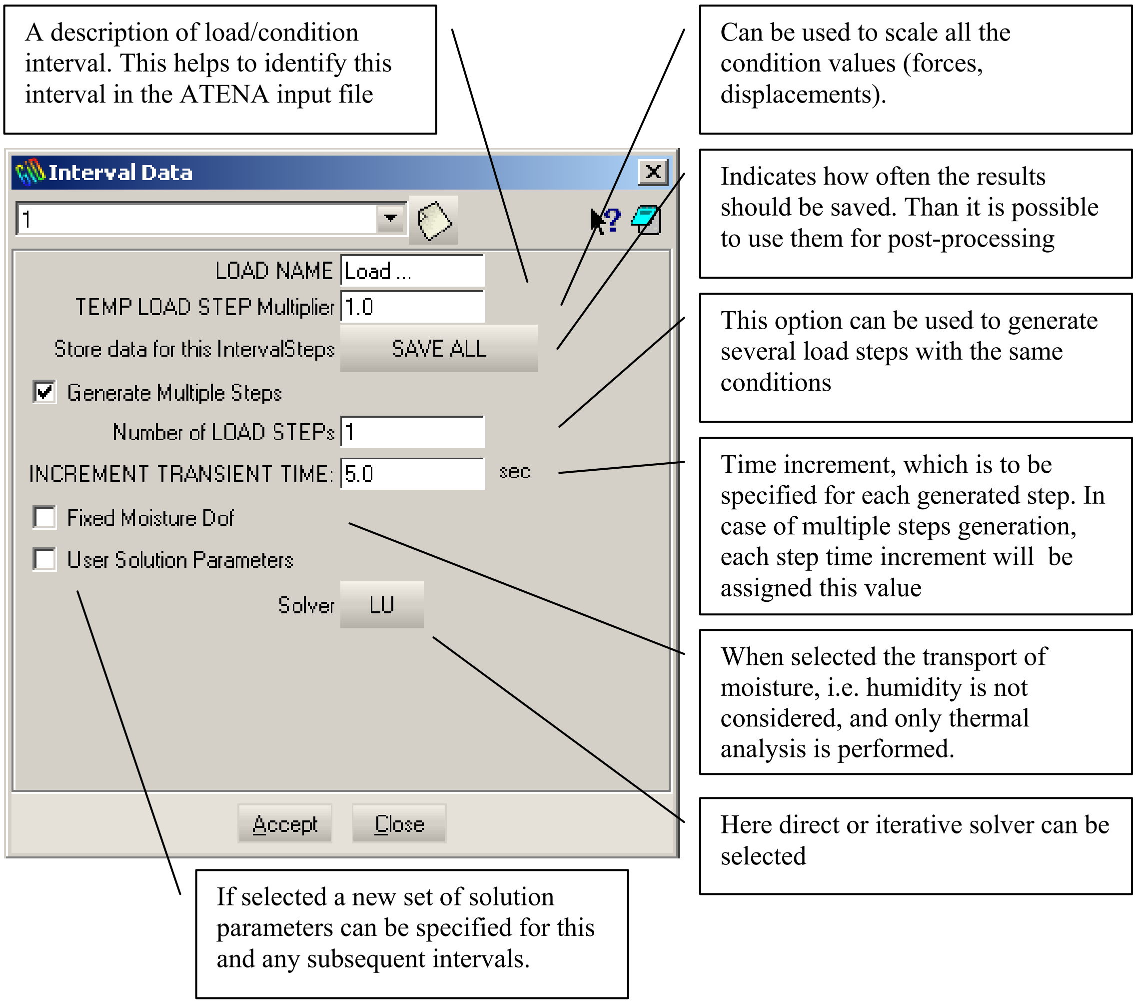


Fig. 8-3 Step data dialog

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).

9 DYNAMIC ANALYSIS

Dynamic analysis is activated in GiD by selecting an appropriate problem type Dynamic (see the menu items Data | Problem Type | AtenaV4). The making of model it's a same like others problem data. It is necessary to assign Conditions [5.2], for each macroelement assign material properties [5.3], define the interval data [5.4] and problemtype properties [Fig 9-1], meshing model [5.7] and execute program.

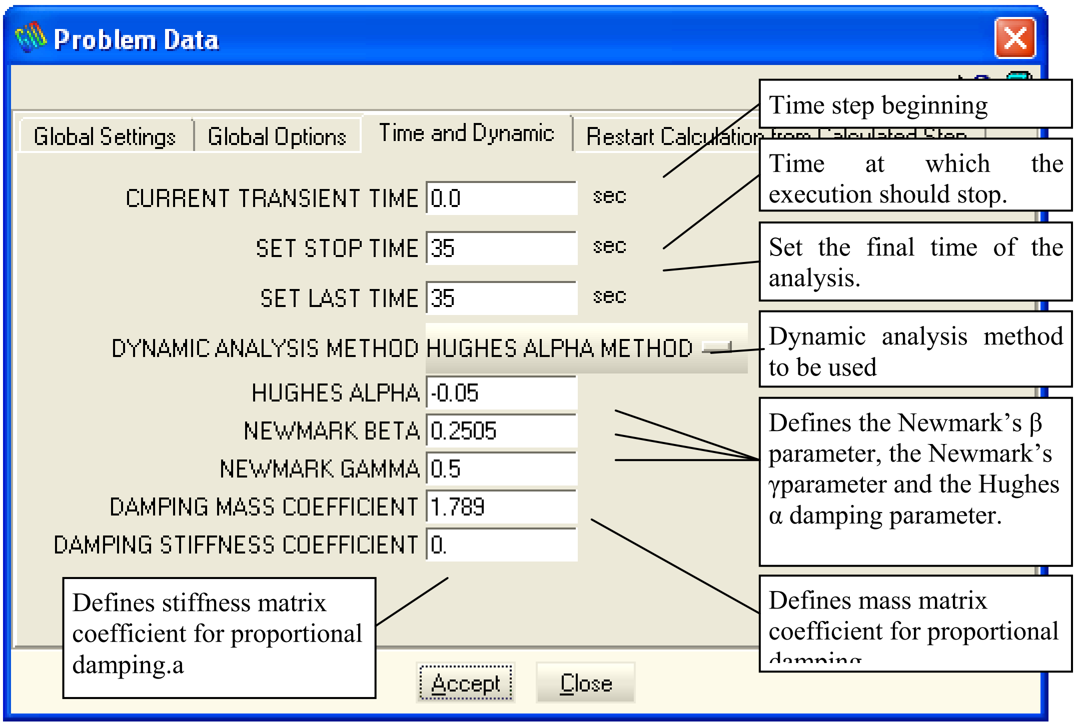


Fig. 9-1: Special dynamic "Problem data" properties

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

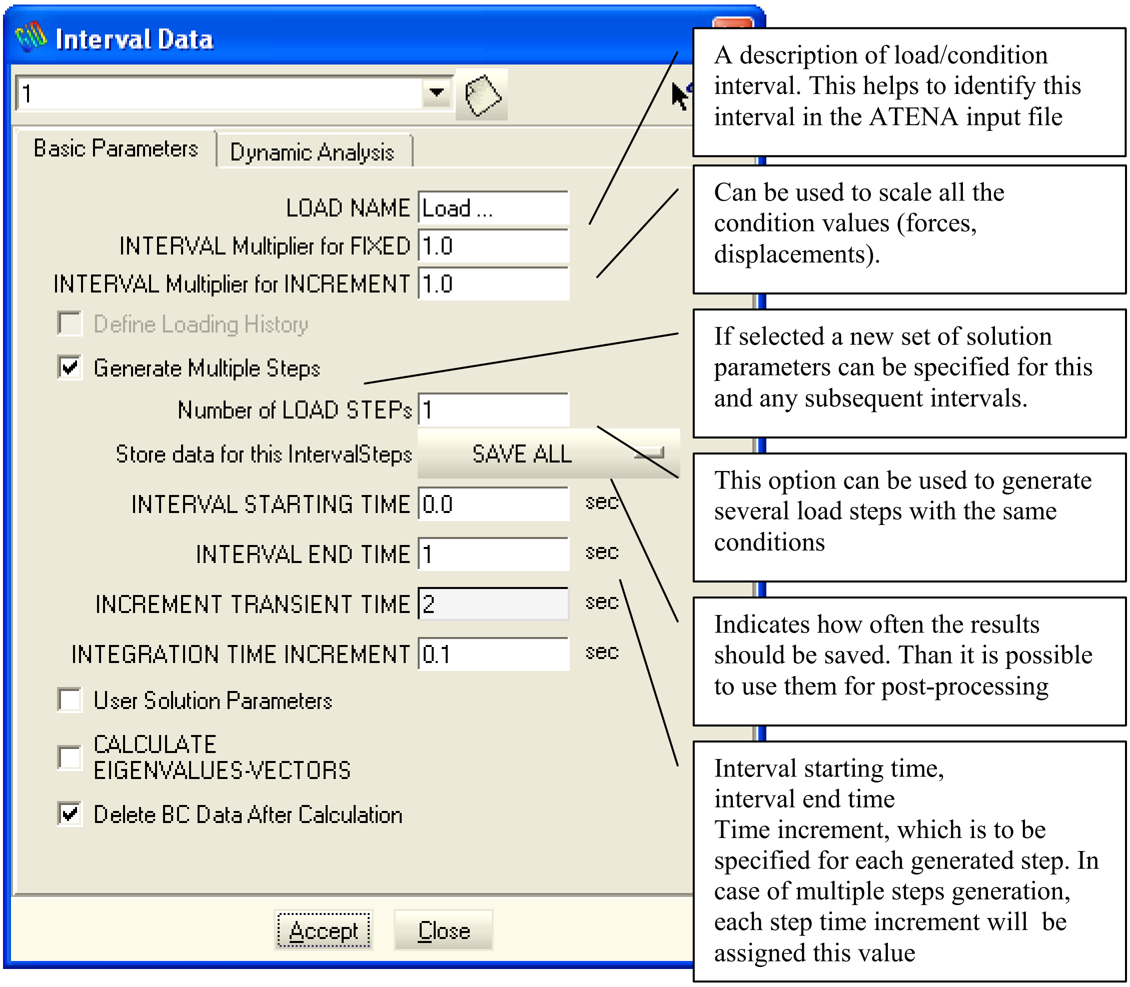


Fig. 9-2: Special dynamic "Interval data" properties

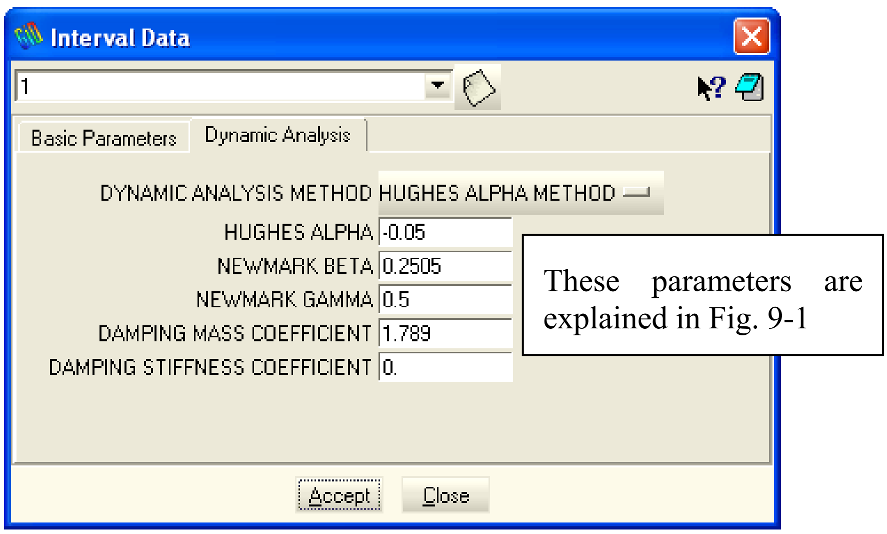


Fig. 9-3: Special dynamic "Interval data" properties

10 EXAMPLE OF A STATIC ANALYSIS WITH REINFORCEMENT

In this example we demonstrate the usage of GiD for data generation of a simple structure. The structure is a reinforced concrete L-shaped cantilever. It has fixed supports on one end and is loaded by vertical force near the free end. See Fig. 10-1. The first beam adjacent to the fixed end is subjected to a simultaneous action of bending and torsion while the second beam, is only under bending. A complex three-dimensional behaviour can be well analysed by ATENA, and for this purpose, the input data can be prepared in GiD.

10.1 Reinforcement modelling

The longitudinal reinforcement is by bars 4Ø28 that are located long the edges, and by stirrups Ø12 with spacing 100mm in the first beam, (section A) and with spacing 200mm in the second beam (section B).

Since there are different possibilities to model reinforced concrete we make first a decision about the modelling approach. Concrete shall be modelled by 3D brick elements. For this we chose the hexahedra elements. The longitudinal reinforcement shall be modelled by discrete bars. The stirrups shall be modelled as a smeared reinforcement within the reinforced concrete composite material. This is a simplified method, by which we avoid an input of detail geometry of stirrups. In smeared model the exact position of individual stirrups is not captured, and only their average effect is taken into account.

The resulting model is shown in Fig. 10-2. The colours of elements show two types of materials used: the composite material named Cantilever1 in the short beam and Cantilever2 in the longer beam. The discrete bars are modelled by linear elements as shown in Fig. 10-3. In the following, we shall treat the generation of the model in more details. A data file with this example can be found in the ATENA installation under the name *Demo_L_RC.gid* in the subdirectory \Examples\Atena-GiD\Tutorial.Static3D.

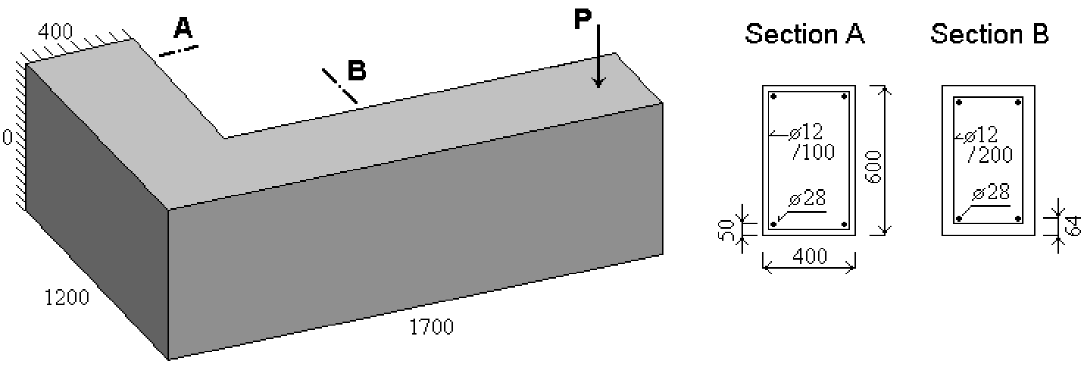


Fig. 10-1 L-shaped cantilever beam. Dimensions in mm.

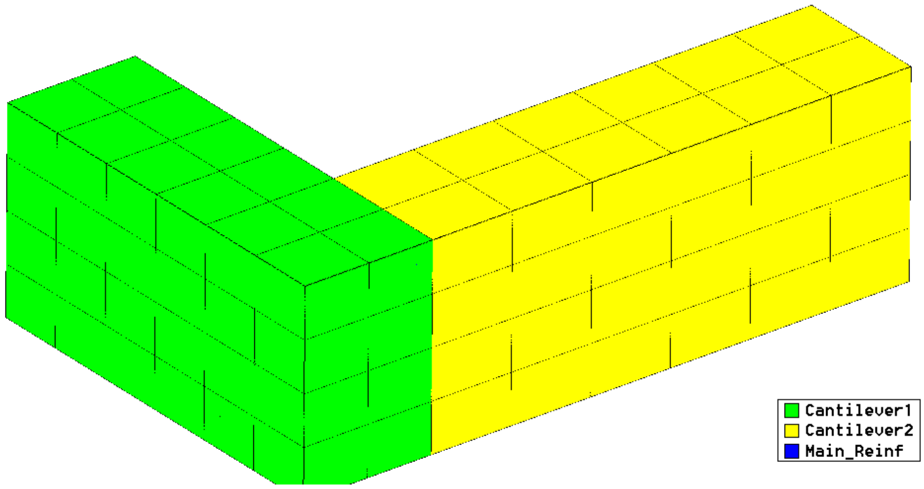


Fig. 10-2 The model with two composite materials: Cantilever 1 and Cantilever 2.

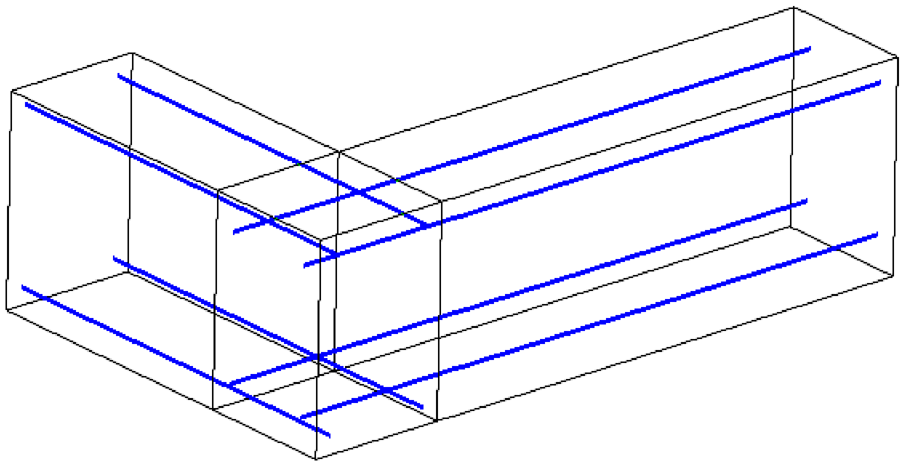


Fig. 10-3 The model of the discrete bars.

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

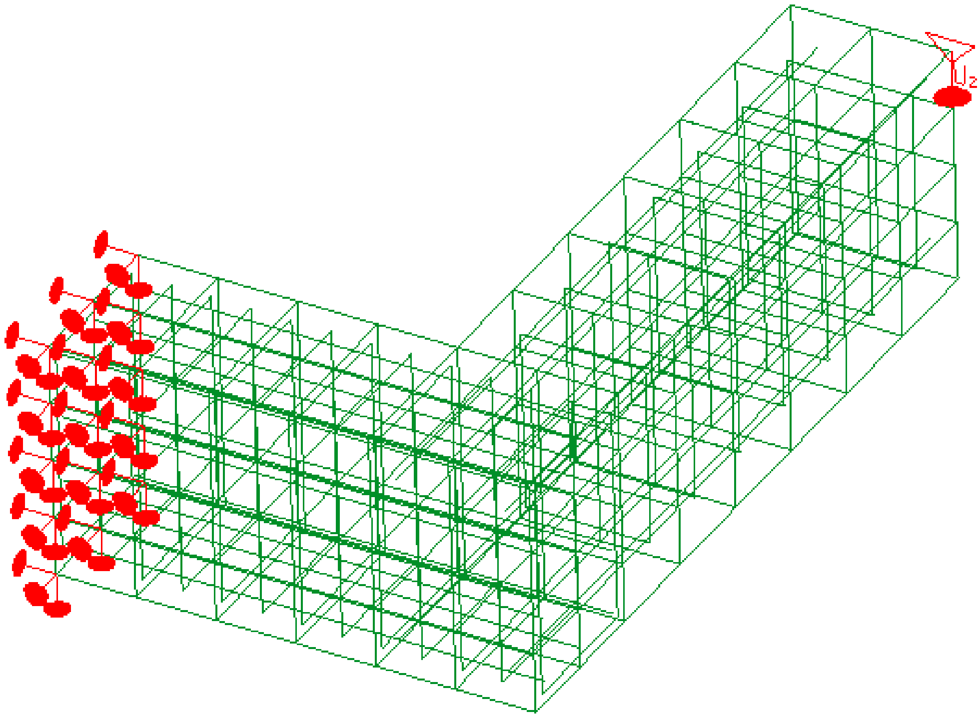


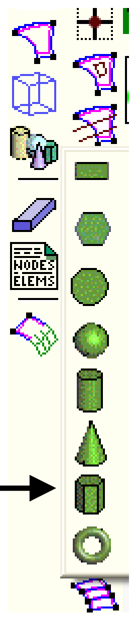
Fig. 10-4 Final finite element model with supports and loading.

10.2 Problem type and data

Typically, the problem definition starts by choosing an appropriate problem type by selecting the menu item 'Data | Problem type | Atena V4 | Static' and then the general solution data in 'Data | Problem Data'. Both steps were already described in Chapter 4. However, the parameters of 'Problem Data' can be also changed later.

10.3 Geometry

The geometry is created by using the GiD graphical tools from elementary objects sequentially, starting from points, lines and finally surfaces and volumes. We start with the definition of points. Points are connected to lines. From lines we can form surfaces and from surfaces we can form volumes (solid objects). Details of this input shall be skipped, since it belongs to standard GiD functions. The final geometrical model is shown in Fig. 10-5. Note that it contains two types of objects: volumes for concrete (and reinforced concrete) and lines for the discrete reinforcement.



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

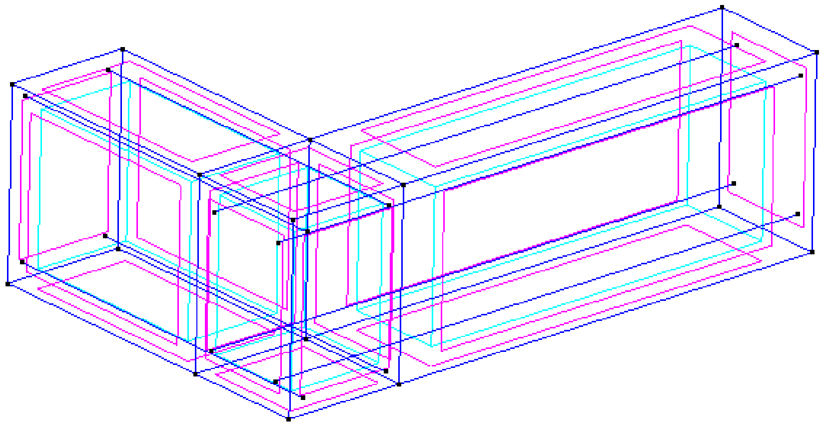
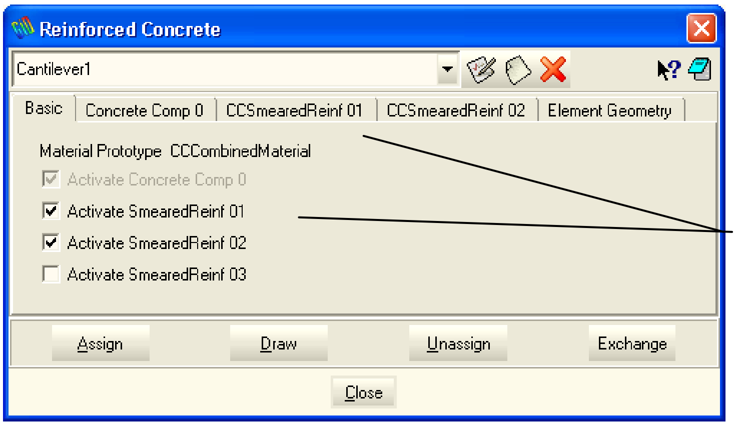


Fig. 10-5 Geometrical model.

10.4 Materials

The materials can be defined and assigned to the geometry using the menu item ‘Data | Materials‘, see

Fig. 5-4. Recommended procedure is to keep the default material unchanged for later reference and create any number of user-defined materials. Since we intend to model the vertical stirrups by smeared reinforcement we shall use the material type ‘Reinforced concrete’. CCCombinedMaterial is a default material and Cantilever1, Cantilever2 are user-defined composite materials that are created from the default material by pressing the button . This command creates a new material of the same type, which can be assigned a suitable user-defined name (see Fig. 10-6).



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

Fig. 10-6 Reinforced concrete material. Two composite materials created.

10.4.1 Reinforced concrete as composite material

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

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

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

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

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

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

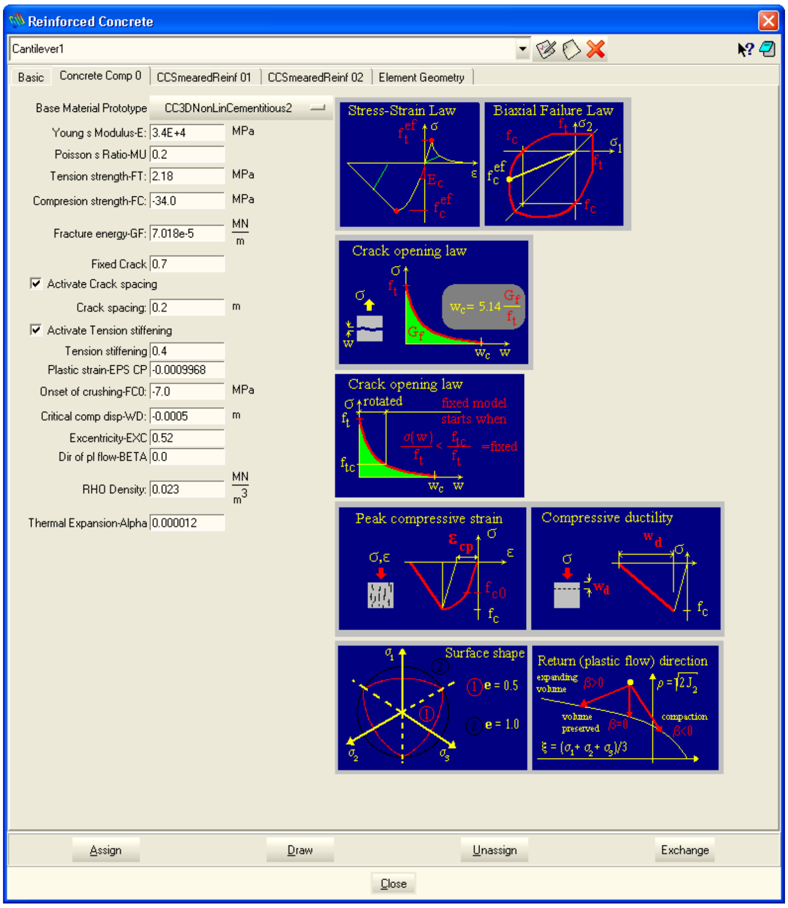


Fig. 10-7 Concrete component in the ‘Reinforced concrete’ material

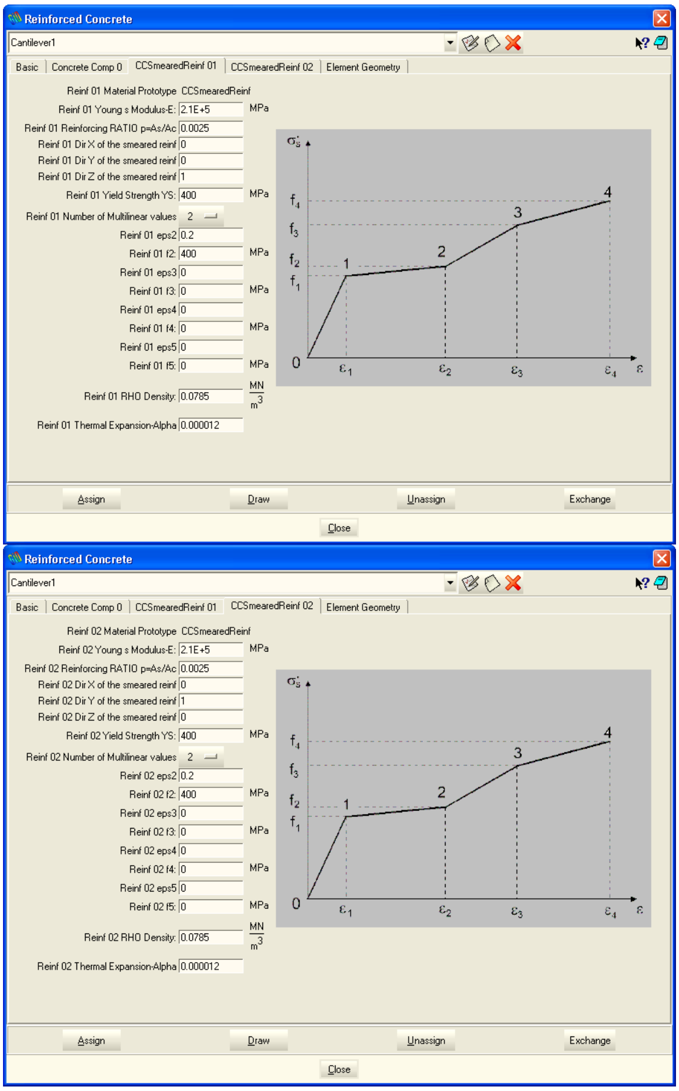


Fig. 10-8 Components of smeared reinforcement in the composite material

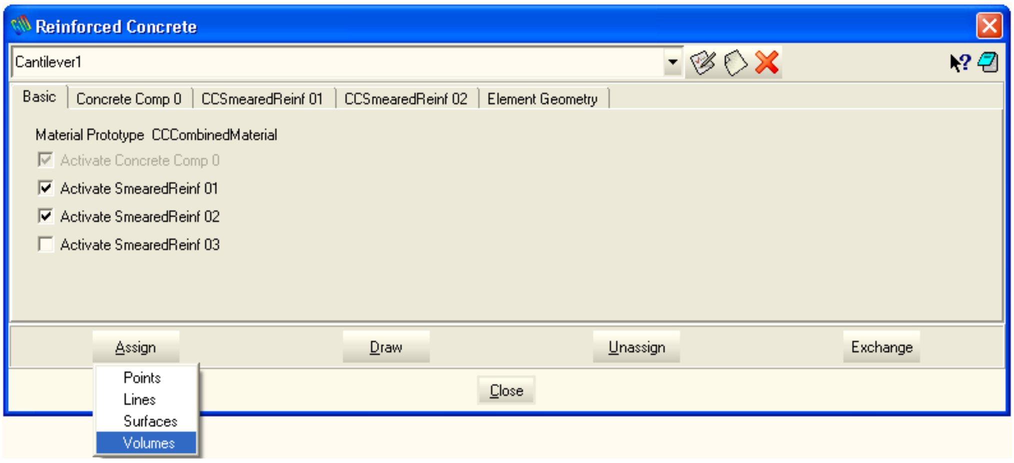


Fig. 10-9 Menu item ‘Assign | Volumes’.

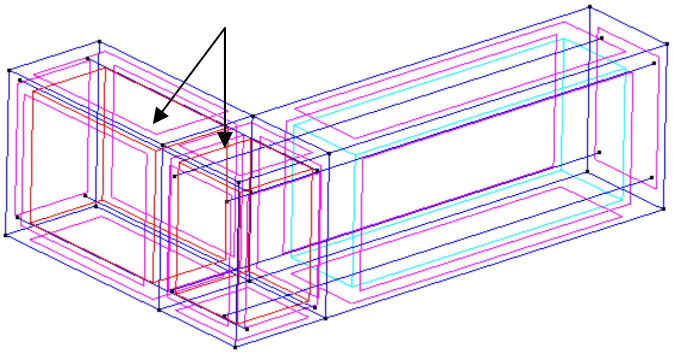


Fig. 10-10 Selected volumes are highlighted by red colour.

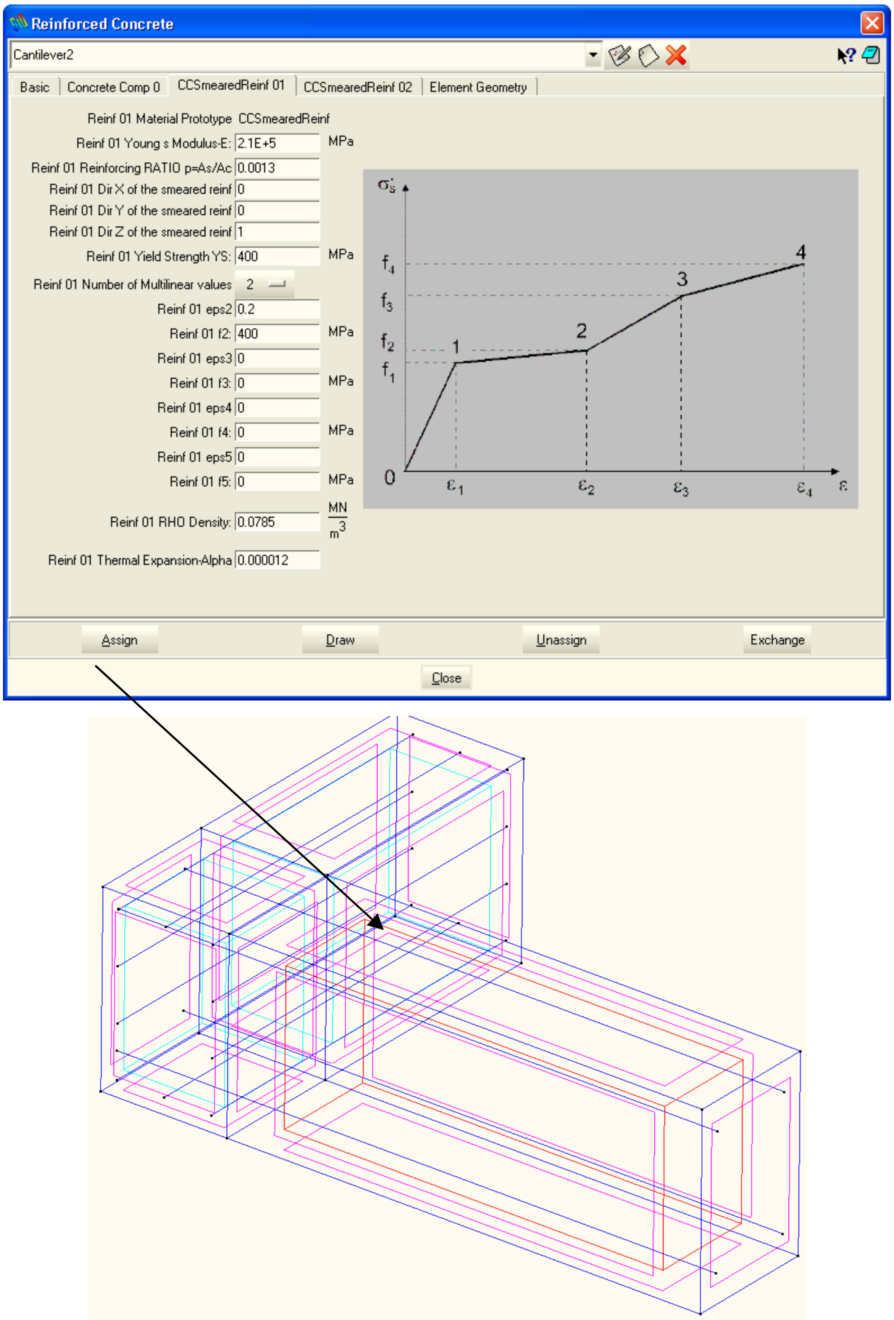


Fig. 10-11 Assignment of the material 'Cantilever2'

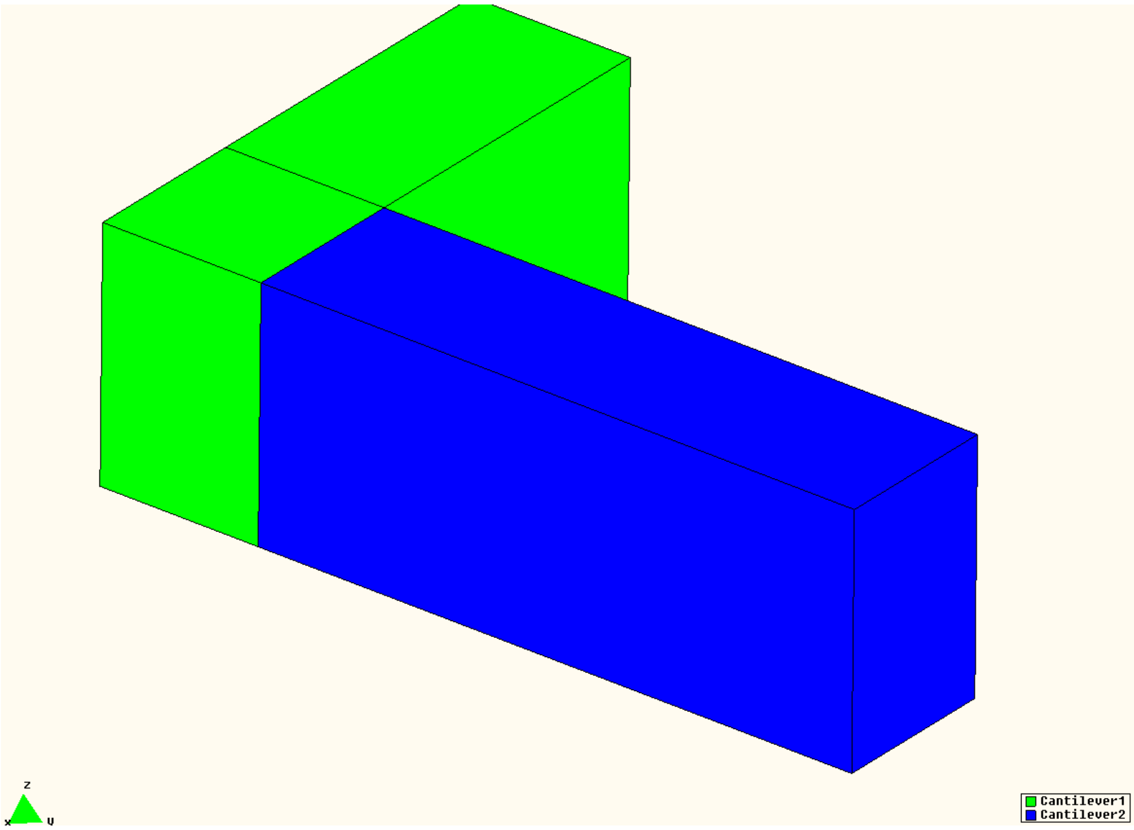


Fig. 10-12 Display of the assigned material groups

A composite material for the second part of the structure (named as Cantilever2) can be defined in a similar way, where the only difference is in the value of reinforcement ratio, Fig. 10-11.

10.4.2 Bar reinforcement

From the menu 'Data | Materials' we select the material 'Reinforcement', which is designated for discrete bars. There we choose from the list the ATENA-model 'CCReinforcement' and then click on the button 'New reinforcement' and enter the name for the reinforcement material, Fig. 6-12.

After confirmation by OK a dialog for material parameters appears. The parameters include initial elastic modulus, yield strength and optionally points on the stress-strain curve. The last parameter is the bar cross-sectional area (see Fig. 10-14).

The material is then assigned to the geometry by pressing the button 'Assign' and selecting 'line' geometric entities by the mouse. The selected bars are marked by red color, Fig. 10-15. Applying the command 'Draw' at the bottom of reinforcement material dialog (see Fig. 10-16) can check a correct assignment, which shows the geometry (in this case lines) with the currently assigned material.

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

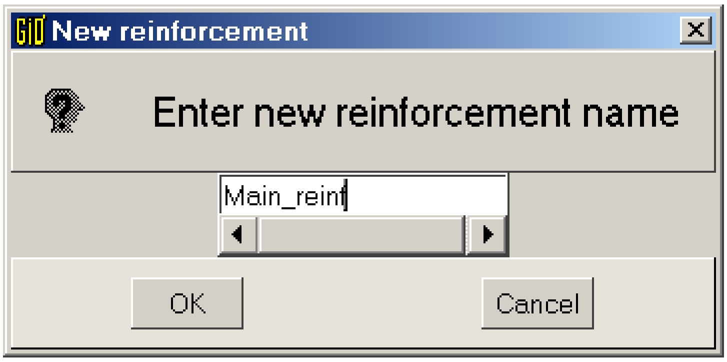


Fig. 10-13 New material for bar reinforcement.

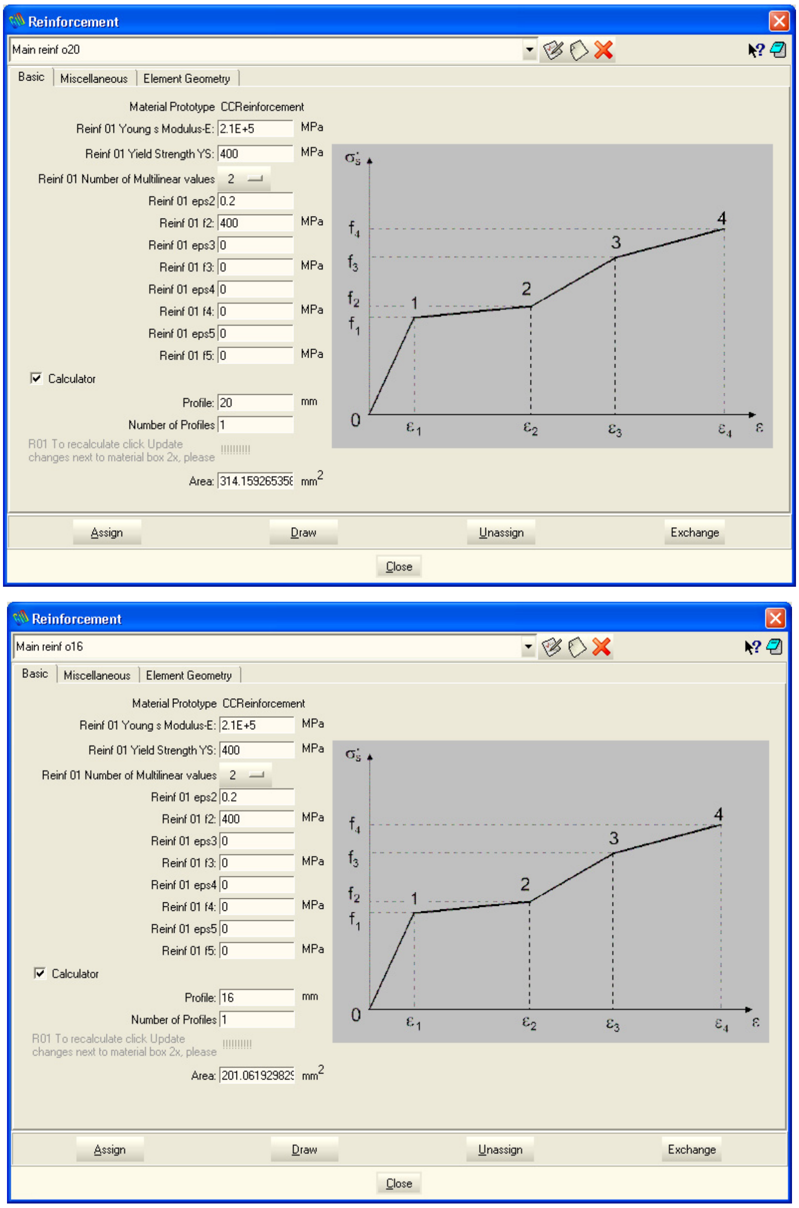


Fig. 10-14 Material parameters for the ‚Reinforcement' model

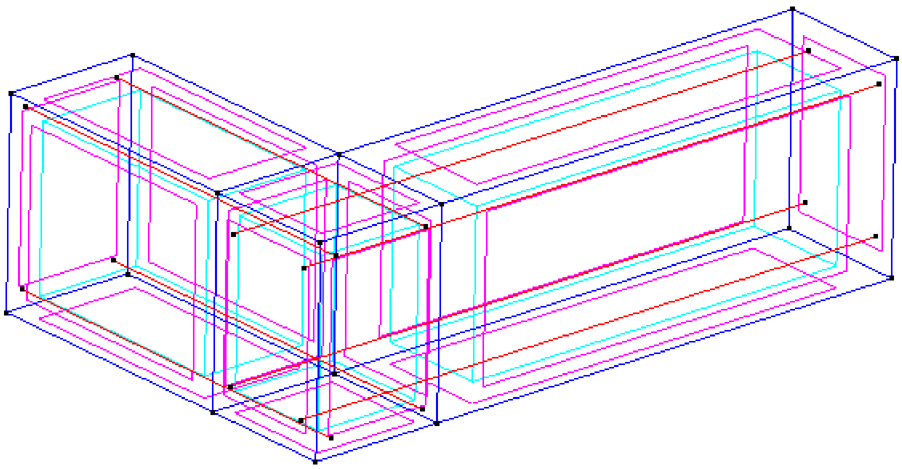


Fig. 10-15 Assigning material to the geometry of bars.

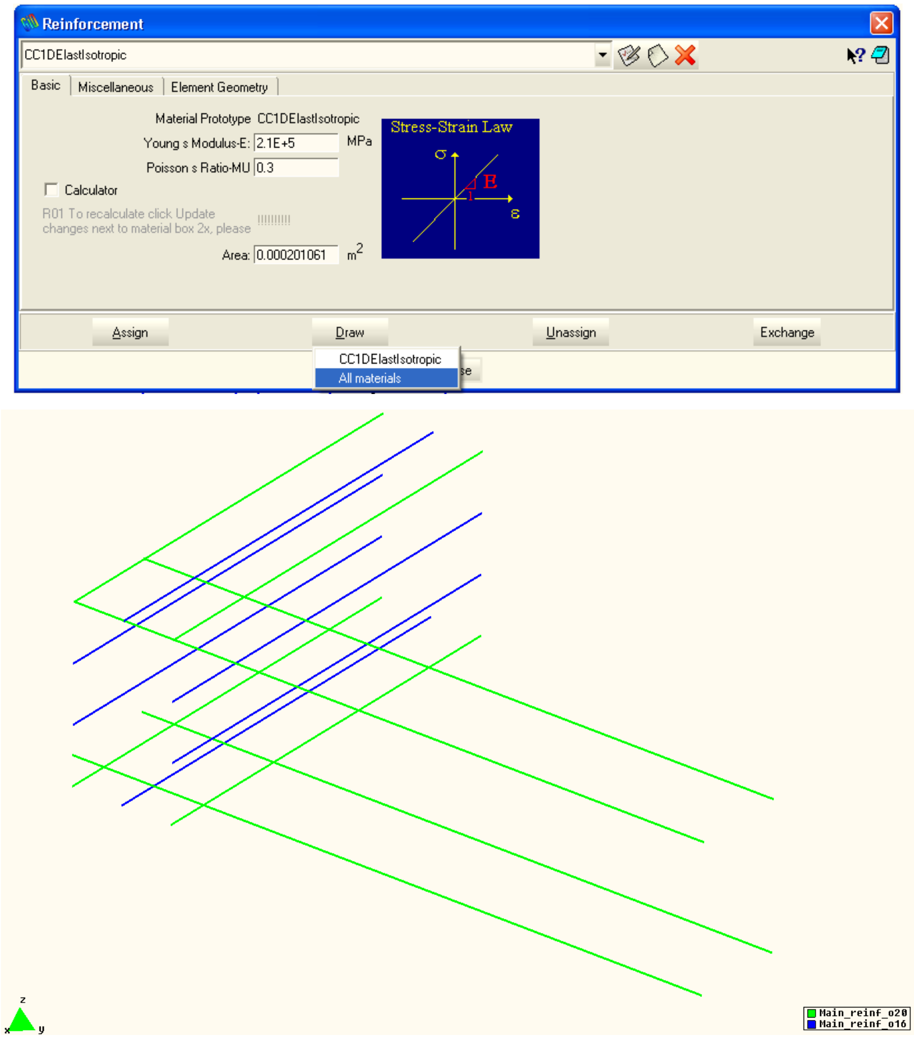


Fig. 10-16 Display of the reinforcement material assignment

10.5 Supports and loading

The supports and loading can be a specified using the menu 'Data | Conditions' described in Fig. 5-2. We define the fixed nodes by checking X-, Y-, Z-Constrains and the type of geometry 'Surface'. Using the command 'Assign' we selects the end face of the cantilever and finish the assignment of support conditions.

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

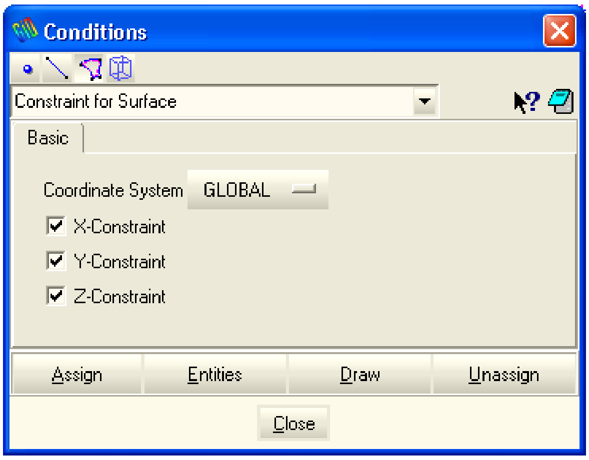


Fig. 10-17 Definition of the surface support in all directions

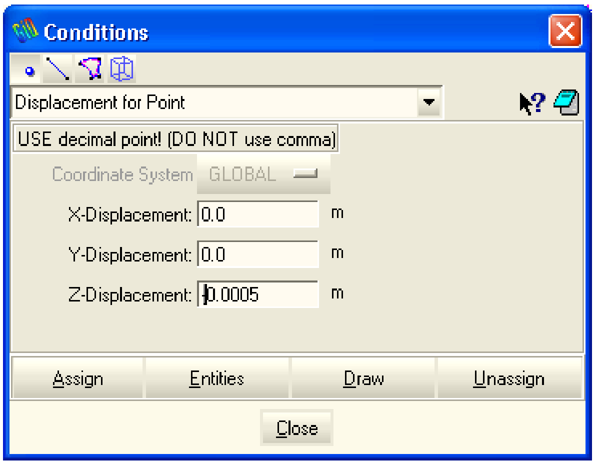


Fig. 10-18 Definition of prescribed displacement in vertical direction

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

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

The definition process of the above conditions and monitors is described in Fig. 10-20. The resulting assignment of the boundary conditions can be checked using the command Draw | All Conditions | Exclude local axis, which can be located at the bottom of the Conditions dialog. It should be noted that it is also possible to apply these conditions directly on the generated finite element model, but then the applied conditions are lost every time the mesh is regenerated.

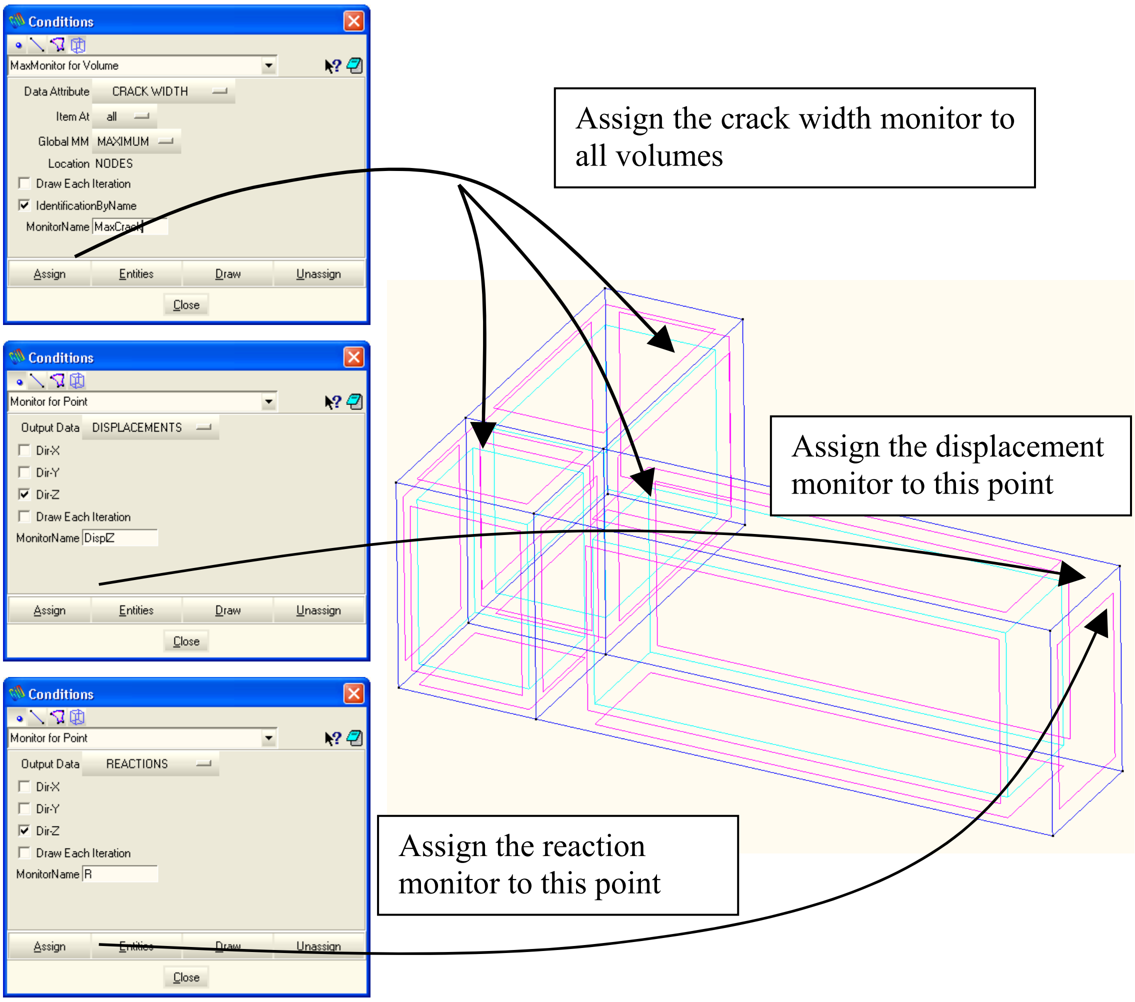


Fig. 10-19 Definition of the ATENA monitors

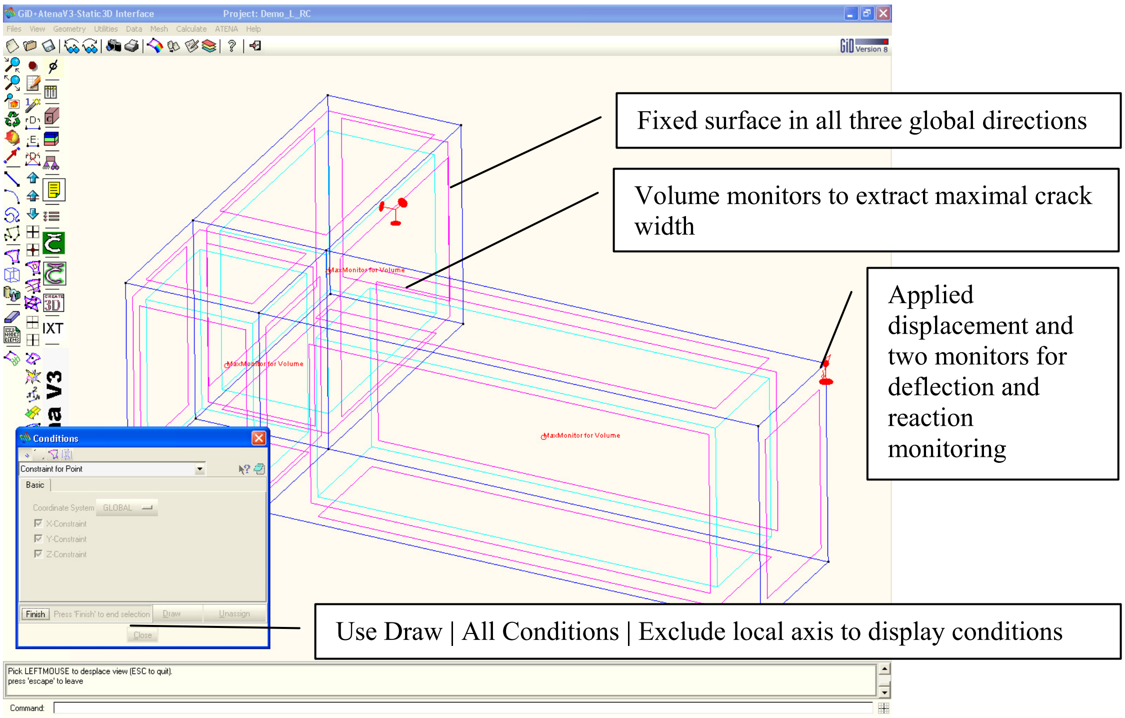


Fig. 10-20 Display of assigned conditions

In certain cases it may be advisable to manually identify which line entities represent reinforcement. By default the GiD-ATENA interface attempts to treat all lines that are not connected to any surface or volume as reinforcement. This default behavior is activated by the corresponding check box in the Problem Data dialog, see Fig. 5-3. In certain cases the automatic identification does not work properly. In this case it is advisable to deactivate this default behavior, un-assign all reinforcement node and element identification and then assign it again manually.

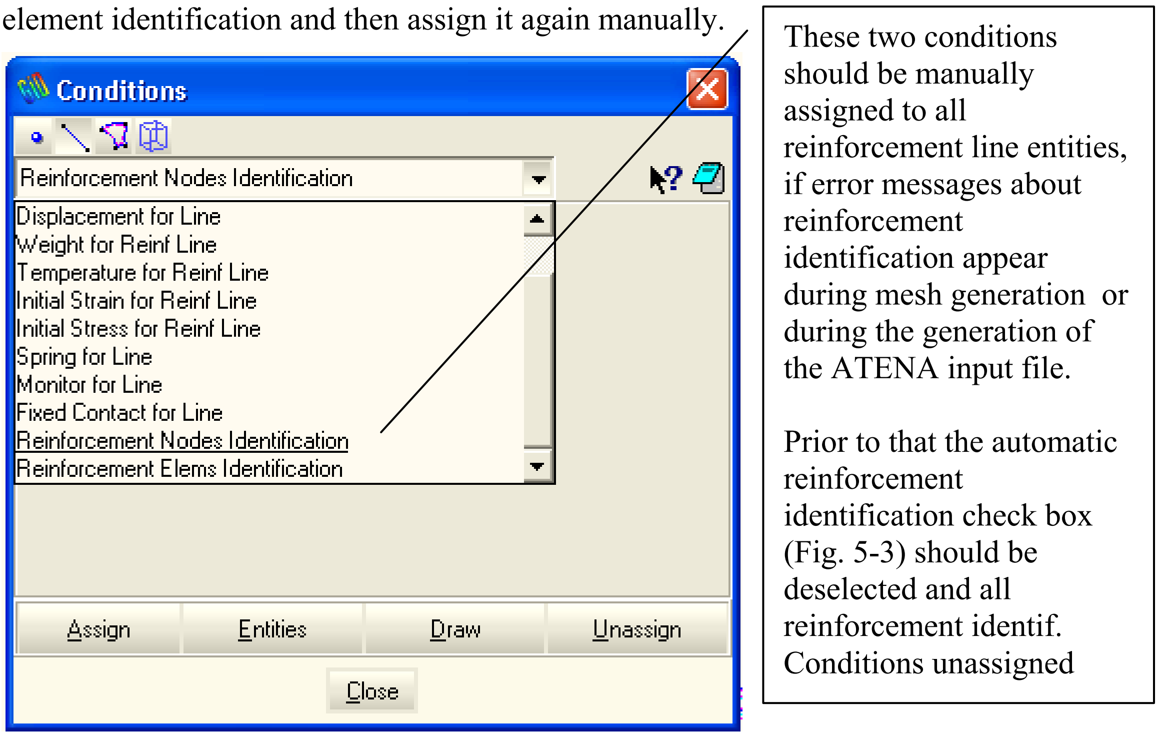


Fig. 10-21 Manual identification of reinforcement nodes and elements

10.6 Meshing

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

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

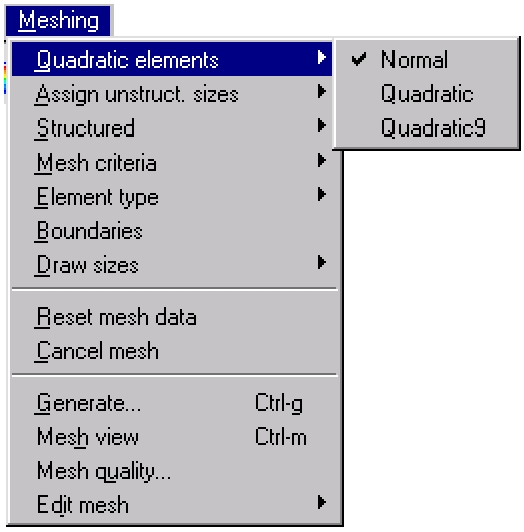


Fig. 10-22 Meshing menu.

10.6.1 Mesh definition for volumes (concrete)

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

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

10.6.2 Mesh definition for lines (reinforcement)

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

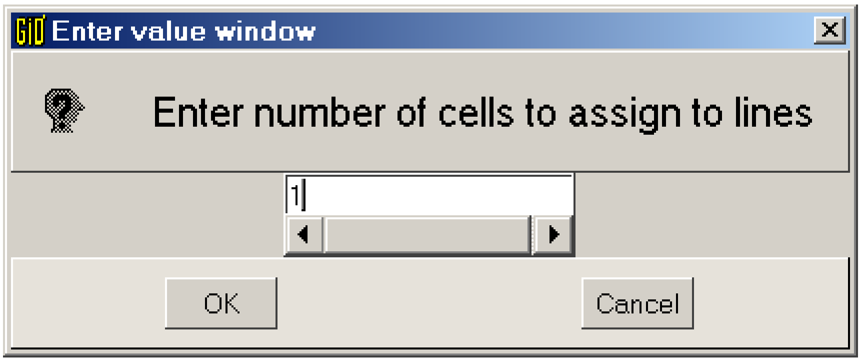


Fig. 10-23 One division in lines of reinforcement.

- In the item ‘Quadratic elements’ we define low order elements by checking ‘Normal’.
- In ‘Structured’ define 1 division on lines, Fig. 10-23
- In ‘Mesh criteria’ select lines.

- In ‘Element types’ select ‘Linear | Lines ’.

10.6.3 Mesh generation

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

10.6.4 Assign conditions to mesh nodes

Now, if needed it is possible to assign additional conditions or materials directly to finite elements of nodes. Select ‘Data | Conditions’ as shown in Fig. 10-24. For this we must select by mouse the node, where condition should be applied. It is however, recommended to assign the material properties and boundary conditions on the geometric entities rather then on the mesh, otherwise it is necessary to reassign such properties every time the mesh is regenerated.

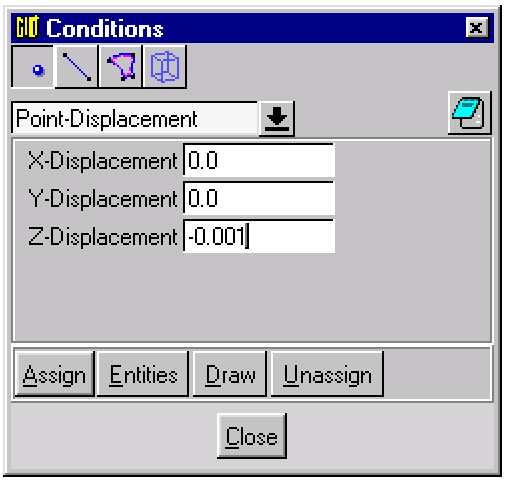


Fig. 10-24 Assigning condition of point-displacement to a mesh node.

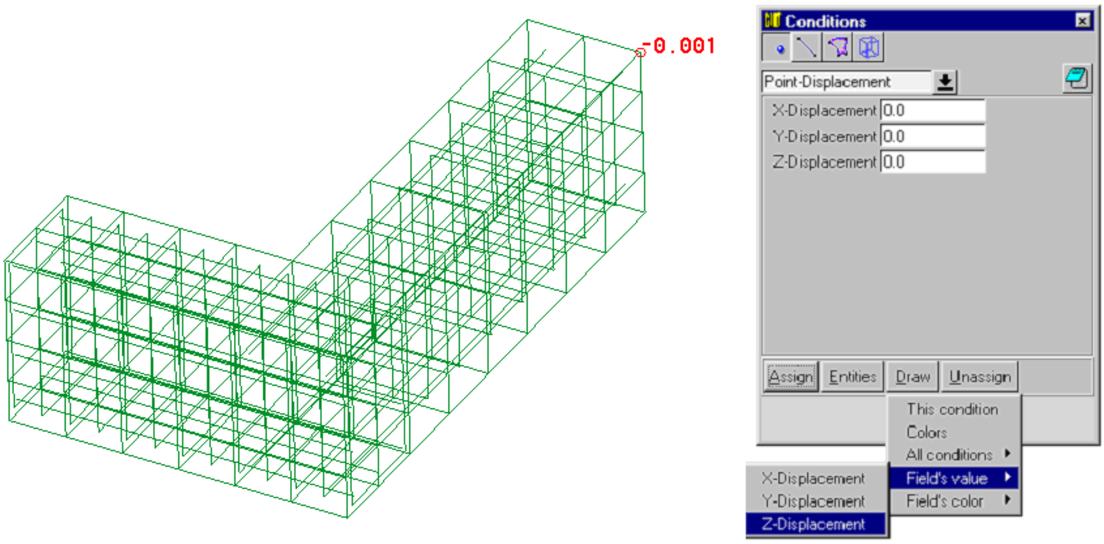


Fig. 10-25 View and inspect a condition in a mesh node.

If we want to inspect the assigned values, we can do it by clicking on the button 'Draw' and select 'Field value | Z-Displacement'. Then the assigned condition value appears at the concerned node. See Fig. 10-25.

10.7 Monitoring points

Analogically to the Section 10.5 it is also possible to specify the monitoring points directly on the finite element mesh. The monitoring points are tools to record a structural response, for example a load-displacement diagram. In GiD we can for instance specify only force and displacement monitoring at a mesh node. This is done also in 'Conditions'. For applied force we select 'Force-Monitor', for reaction force 'Reaction-Monitor', for nodal displacement 'Displacement-Monitor'. Displacement component is selected by checking the appropriate box.

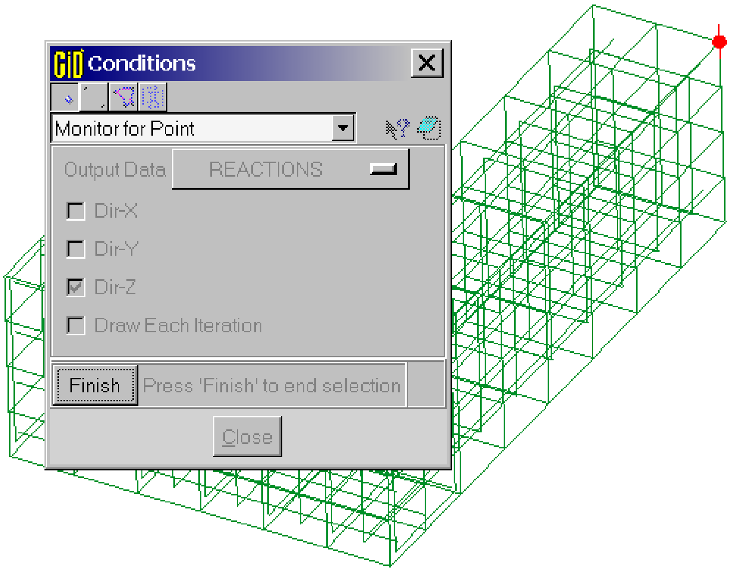


Fig. 10-26 Definition of a monitor for reaction at node.

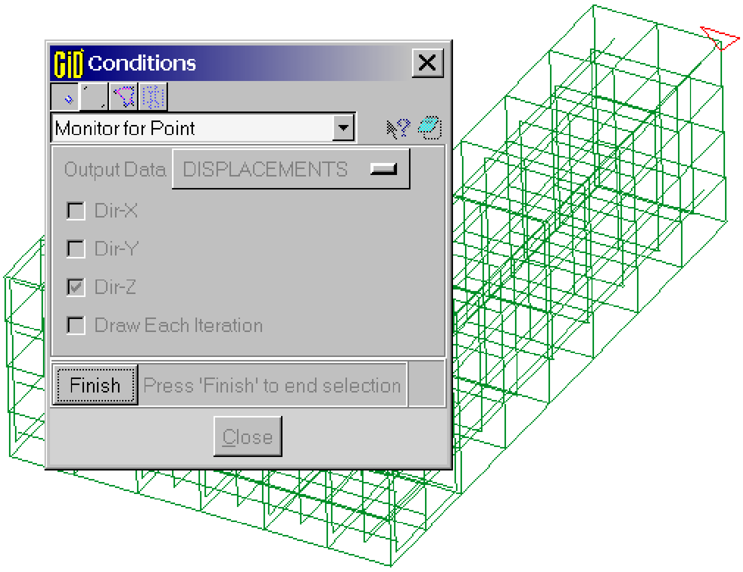


Fig. 10-27 Definition of monitor for displacement at node.

Fig. 10-26 and Fig. 10-27 show definition of force (reaction) and displacement monitors at a node. An inspection of monitors can be done by the command ‘Draw’ in the same manner as in other conditions.

The monitoring points must be included within ‘Conditions’ of the first load interval in GiD. Monitors included in other intervals will not be active in ATENA analysis.

10.8 Load history

For analysis in ATENA a load history as a sequence of load steps must be defined. The load steps can be proportional or non-proportional. In this example the load history is simple. We define first interval, which includes a set of conditions for supports at the fixed end and point-displacement. This can be checked and changed in the menu item ‘Data | Interval Data’, Fig. 5-25. Next load steps can be done in two ways. The simplest way is to enter the number of repeated load steps and multipliers in the window of ‘Interval Data’, Fig. 10-28, which is a proportional load history. In case of a non-proportional history, for example, first a vertical load followed by a horizontal load, we can use ‘Data | Interval Data’. Default settings of calculation method and global settings are in ‘Data | Problem Data’.

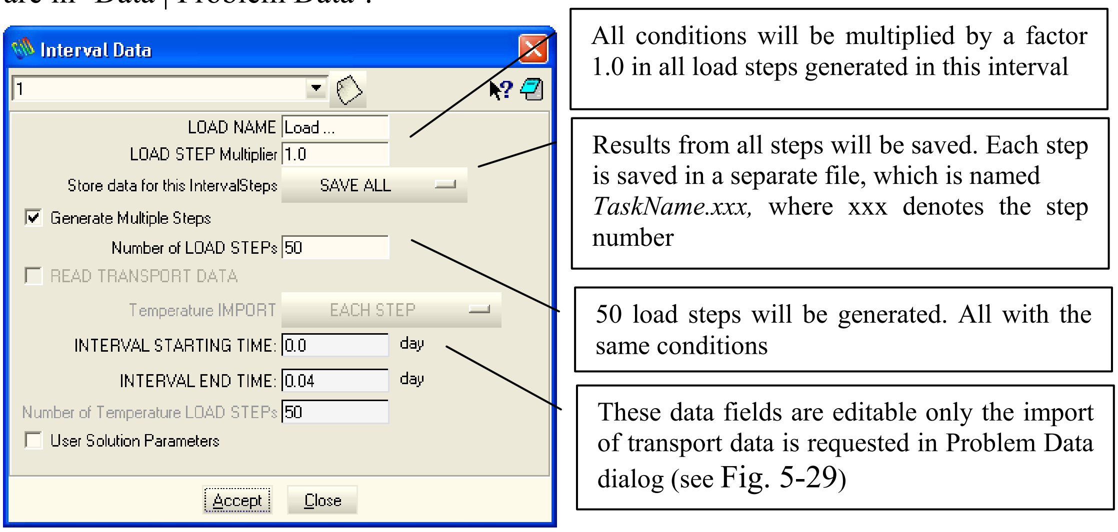
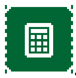


Fig. 10-28 Interval data definition

10.9 Analysis and post-processing

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

During the execution of AtenaWin variety of intermediate results can be viewed and inspected. The results of analysis can be presented in the program Atena3D. The Post-processing in ATENA 3D is started via menu ATENA | ATENA 3D post-processing. Then it is necessary to import the binary result files (TaskName.xxx) from the required load steps into ATENA 3D. This is accomplished through the ATENA 3D menu File | Open other | Results by step.

For operation of AtenaWin, Atena3D or any other details of ATENA software see the Atena Documentation volumes 1 to 7, see [2], [3] and [5].

11 Tutorial for Construction Process

11.1 Introduction

The objective of this tutorial is to show how the graphical environment of GID can be used to model the construction process. The finite element solution core of ATENA supports the possibility to add or remove groups of finite elements. This feature can be used to model the construction process in GID. The ATENA-GiD extension of the GID graphical environment includes direct support for this feature. This feature can be modeled using the conditions for surface and it will be demonstrated in this manual on the example of a tunnel (see Figure 1).

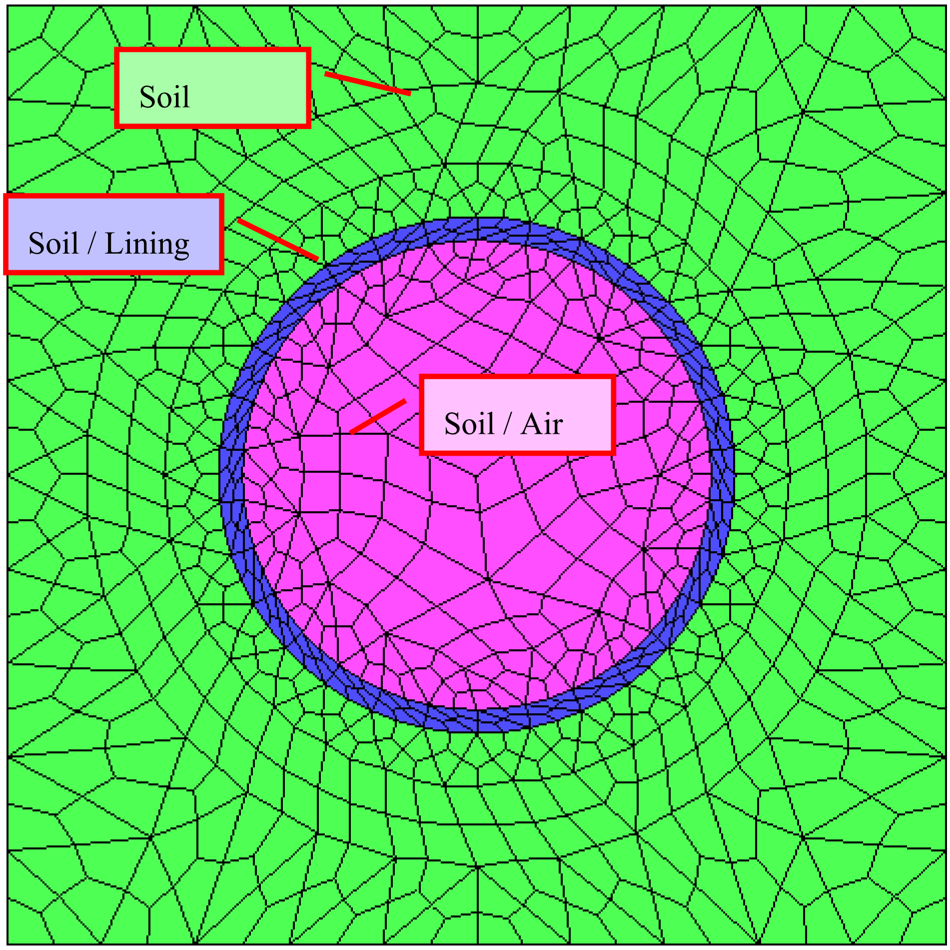


Figure 1: Model with three macro-elements.

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

11.2 Geometry, boundary conditions and load

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

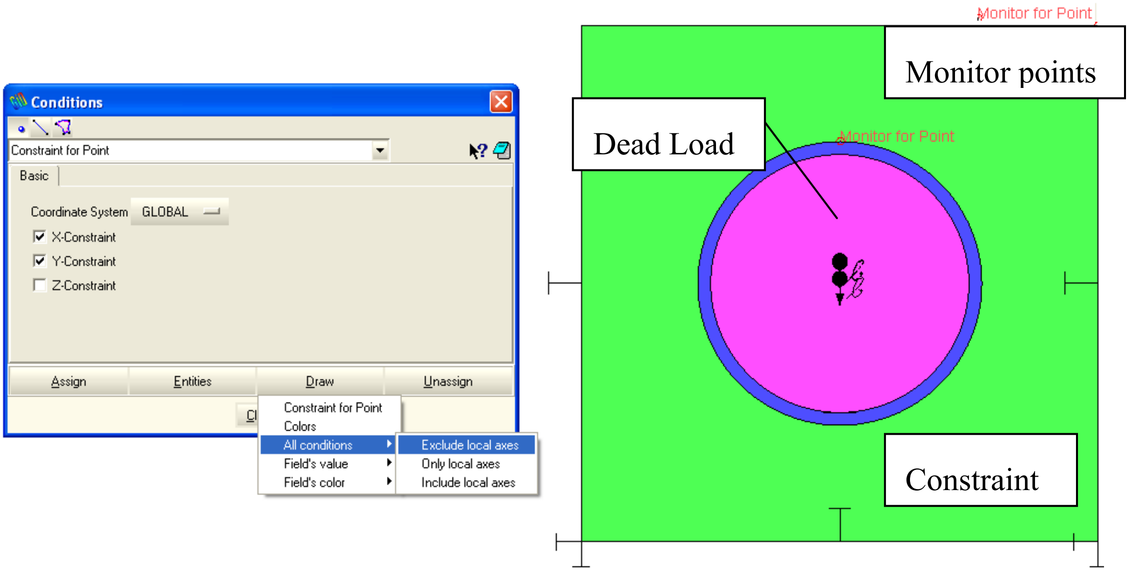


Figure 2: Draw all conditons on model

The construction should proceed as follows:

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

First, it is necessary to construct the model of the whole structure. Three separate macro-elements will be created for all four intervals.

Interval 1: this interval is used to define the basic boundary conditions to support the model from the bottom and both sides.

Interval 2: this interval is used for excavation of a circular hole in the soil by deleting two centered macroelements.

Interval 3: this interval is used to add lininig (ring shape) with concrete material characteric around the hole.

Interval 4: this interval is used to add load to top face of the model.

At the beginning, the whole area consists of soil, however, we must define separate macroelements for future changes (soil, lining, air). We assign the soil material to all these macroelements for the first interval (Figure 3). The additional intervals will be needed for the subsequent phases of the construction process.

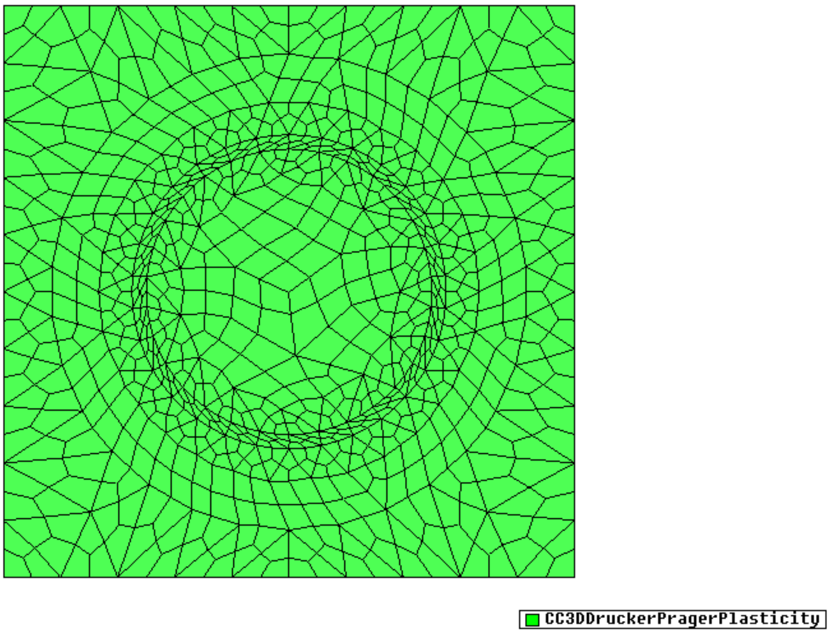


Figure 3: Material for interval 1

In next step (excavation) we need to remove both circles from the center. It can be made using conditions for surface. In menu "**Data > Interval > Current**" we switch to interval No.2 which we want to edit (Figure 4). In menu "**Data > Conditions >** ***Conditions for surface***" we choose "**Elements Activity for Surface**" and select „**Construction (Elements Activity) : DELETE**" (Figure 5). Next we can "**Assign"** areas which we want to excavate (Figure 6). We can draw all macroelements which have assigned some conditions by choosing "**Draw > Colors**" (Figure 6).

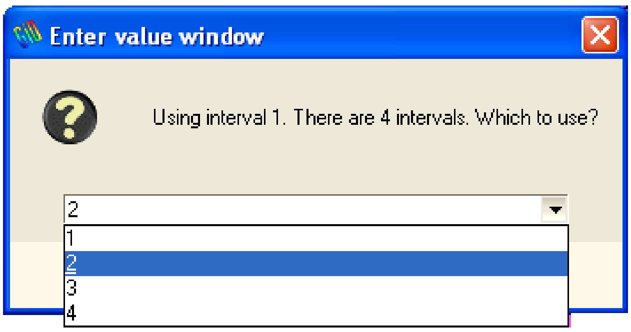


Figure 4: Switching current interval.

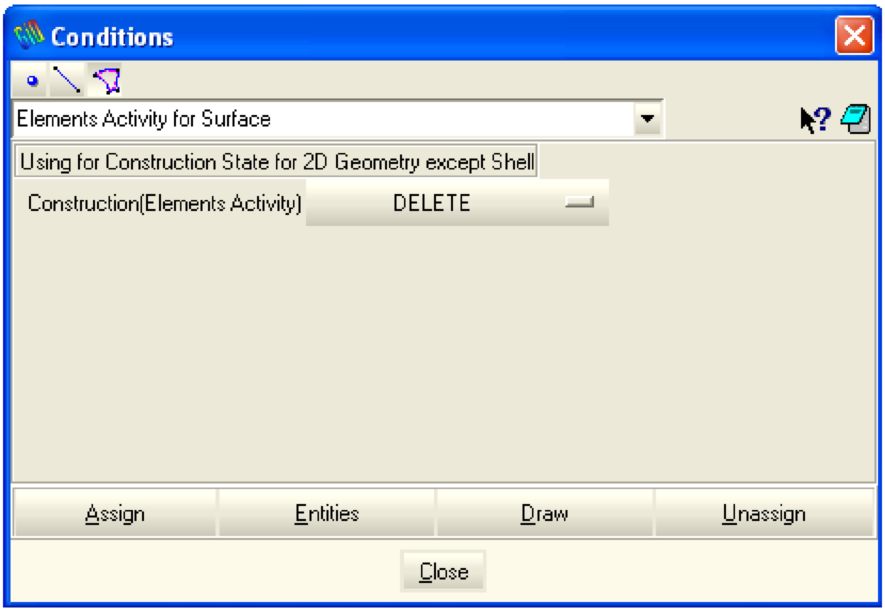


Figure 5: Conditions for surfaces

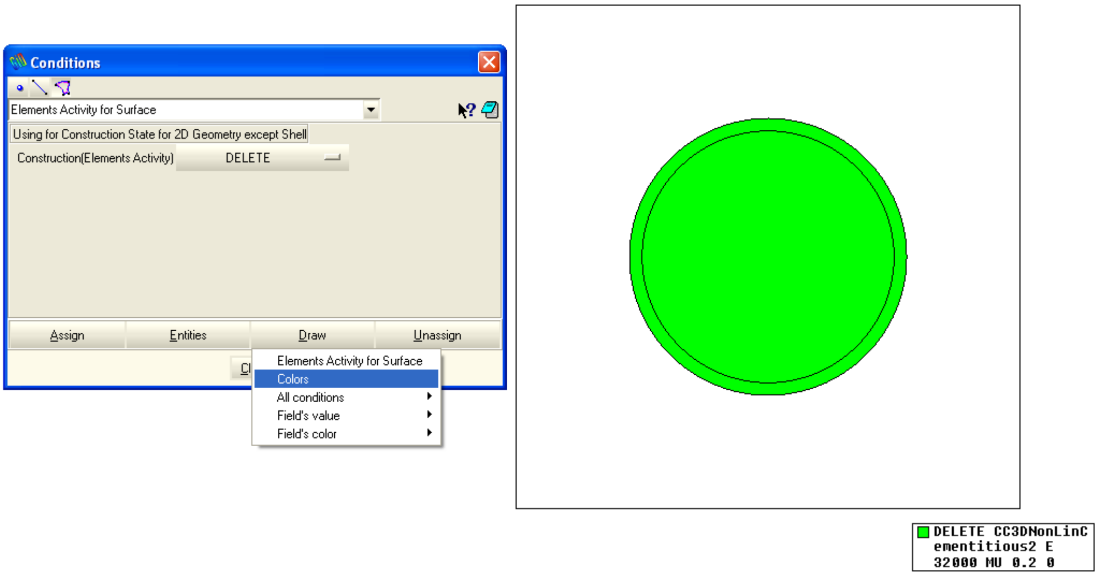


Figure 6: Deleting materials in interval 2

In the next step, we need to create the lining with non-linear concrete material. We switch the current interval to No.3. In menu "**Data > Conditions >** ***Conditions for surface***" we choose "**Elements Activity for Surface**" and select „**Construction (Elements Activity): CREATE WITH NEW MAT**" (Figure 7), and choose the CC3DNonLinCementitious2 material. We can create specific material for this case and assign to surfaces, which we want to create (Figure 8).

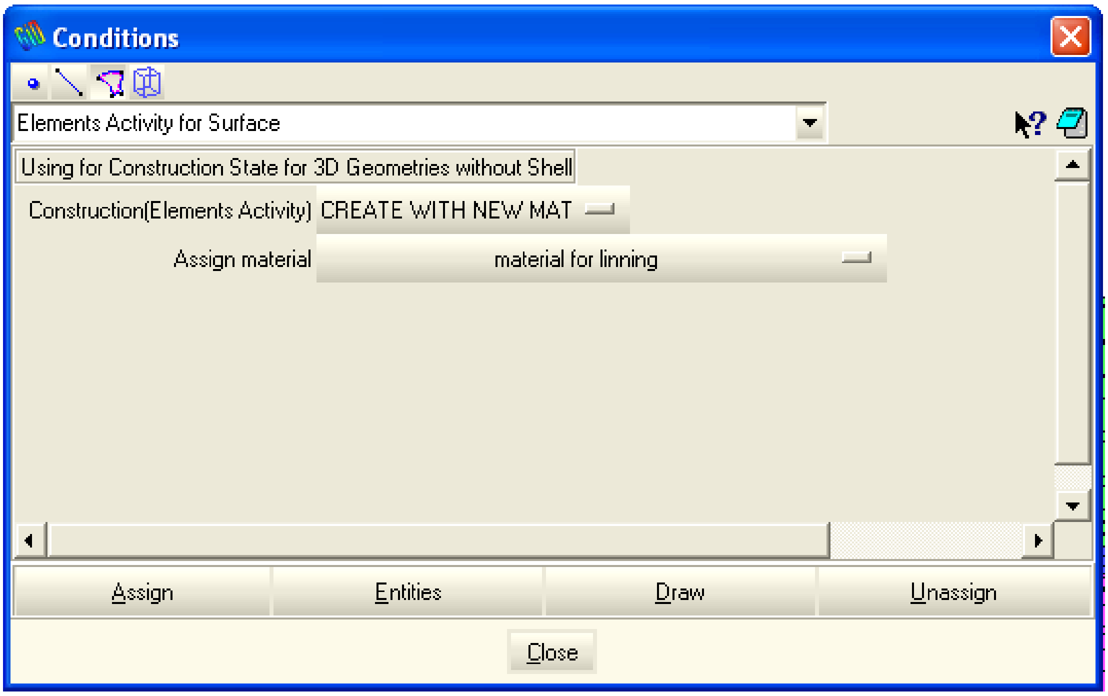


Figure 7: Condition for surface, create new material

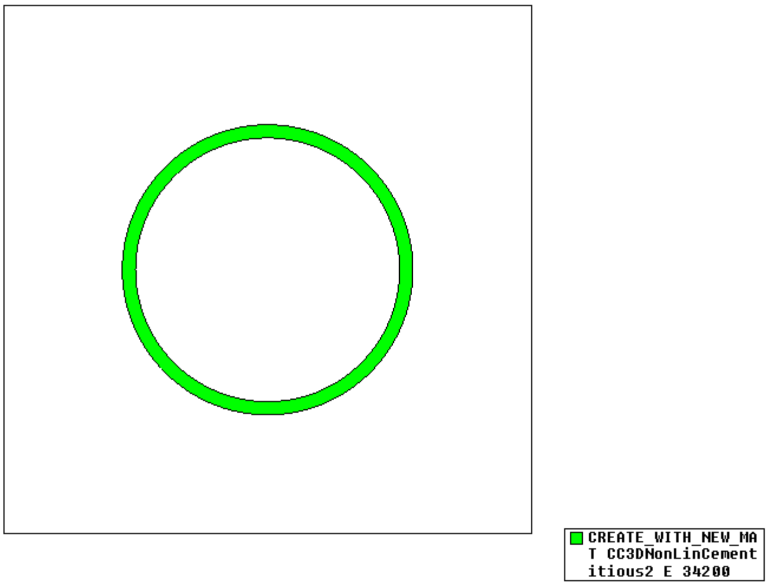
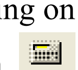


Figure 8: Creating concrete lining.

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

11.3 Running analysis

Analysis can be run by selecting button in menu "**Calculate > Calculate**" or clicking on the button  "**Atena Calculate**" and in Atena Win by clicking on button  "**Execute**".

12 USEFUL TIPS AND TRICKS

12.1 Export IXT for Atena3D pre-processor

It is also possible to export 3D mesh to an IXT format, which can be imported to Atena3D 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.

13 EXAMPLE DATA FILES

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

Directory - Tutorial.Creep2D

Vitek2D.gid	Slab with creep that is modelled as a two-dimensional structure

Directory - Tutorial.Creep3D

Shell_Slab02.gid	symmetric quarter of a square 3D slab with creep modelled using shell elements
SlovDeska3D.gid	creep experiment in Bratislava
Vitek3D.gid	Slab modelled as a 3D structure that was tested in Metrostav, Prague

Directory - Tutorial.Static2D

axisym.gid	Axisymmetric problem
ngap7e.gid	Example with an interface material model
Tunel2D_deep.gid	Two-dimensional analysis of a deep tunnel with construction process
Tunel2D_shallow.gid	Two-dimensional tunnel analysis with smaller model of the surrounding soil, including construction process
Vitek2DStatic.gid	Only static analysis without creep of the slab specimens tested by Metrostav, Praha

Directory - Tutorial.Static3D

Demo_L_Bars.gid	Example of L-shaped cantilever with discrete bars for main reinforcement as well as for stirrups.
Demo_L_RC.gid	Example of L-shaped cantilever with discrete bars and smeared stirrups in composite material. It is used in this manual.

ngap7d.gid	Example of interface between two concrete plates.
Shell_slab02.gid	Slab-column connection
Tunel3D_shallow.gid	Three-dimensional model of a tunnel with soil and construction process

Directory - Tutorial.Temperature2D

PipeBStatic.gid	Static part of a pipe analysis with thermal loading
PipeBTemp.gid	Thermal part of a pipe analysis with thermal loading

Directory - Tutorial.Temperature3D

tram014stat5_DM.gid	Static part of a 3D beam analysis with thermal loading
tram014temp5_DM.gid	Thermal part of a 3D beam analysis with thermal loading

14 CALCULATION OF ATENA IDENTIFICATION NUMBERS

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

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

ElementType for 3D	**ElemsNnode**	14.1.1.1.1.1.1 Geometrical	
		LINEAR	NONLINEAR
CCIsoGap<xxxxxxxx>	8	**28**	**58**
CCIsoGap<xxxxxx>	6	**26**	**56**
CCIsoBrick<xxxxxxxxxxxxxxxxxxxx>	20	**20**	**50**
CCIsoWedge<xxxxxxxxxxxxxxx>	15	**15**	**45**
CCIsoTetra<xxxxxxxxxx>	10	**10**	**40**
CCIsoBrick<xxxxxxxx>	8	**8**	**38**
CCIsoWedge<xxxxxx>	6	**6**	**36**
CCBarWithBond	2	**5**	**35**
CCIsoTetra<xxxx>	4	**4**	**34**
CCIsoTruss<xxx>	3	**3**	**33**
CCIsoTruss<xx>	2	**2**	**32**
CCSpring/CCLineSpring/CCPlaneSpring	1	**1**	**31**

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

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

3D Element:

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

Formula:

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

1D Element:

	Increment	
AddingBarWithBond	3	Increment if is element BarWithBond

Formula:

ELEMENT_TYPE_ID = ElemsNnode +
AddingBarWithBond+AddingNonLinElemID (3)

Load cases:

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

REFERENCES

[1] Cervenka, V., Jendele, L, Cervenka, J., (2007), *Atena Program Documentation, Part 1, Theory*, Cervenka Consulting, 2007

[2] Cervenka, V., and Cervenka, J., (2007), *Atena Program Documentation, Part 2-1, User's Manual for Atena 2D*, Cervenka Consulting, 2007

[3] Cervenka, V., and Cervenka, J., (2007), *Atena Program Documentation, Part 2-2, User's Manual for Atena 3D*, Cervenka Consulting, 2007

[4] Cervenka, J., and Jendele, L, (2007), *Atena Program Documentation, Part 6, Atena Input File Format*, Cervenka Consulting, 2007

[5] Jendele, L., (2007), *Atena Program Documentation, Part 7, AtenaWin Description*, Cervenka Consulting, 2007