Application of GID for nonlinear analisis of reinforcement concrete structures with ATENA

Section: Interface of GID with comercial code (Atena)

Authors: Jiri Niewald, Jan Cervenka, both from Cervenka Consulting from Czech Republic

Keywords: non-linear analysis, fracture mechanics, plasticity, failure, pre-stressed concrete

1 INTRODUCTION

Non-linear analysis of concrete structures became a novel design tool. It employs the power of computer simulation to support and enforce the creativity of structural engineers. Although it is frequently used in research and development its potential for engineering practice has not been yet fully discovered.

In the new design standards for concrete structures, such as Eurocode 2 (1992), the non-linear behaviour of concrete is described by a uni-axial stress-strain diagram. However, this is only a fraction of the available knowledge of material models and theories discovered and validated by research. The finite-element-based failure analysis can take advantage of rational and objective theories such as fracture mechanics, plasticity and damage mechanics. It makes possible a "virtual testing" of building structures under designed loading and environmental conditions. The implementation of these advanced methods and techniques introduces new possibilities for the engineer, not only in research and development but also in practical design. Computer simulation is a modern tool for optimisation of structures, which can contribute to better economy in design.

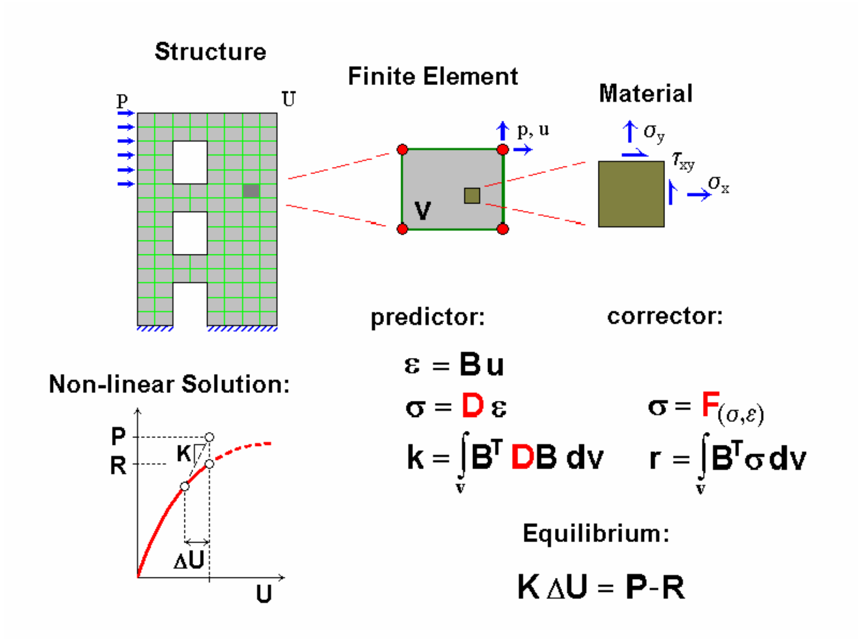

Fig. 1 Scheme of the non-linear finite element algorithm.

2 GID AND ATENA

ATENA is a simulation package for modeling the behavior of structures with the emphasis on reinforced concrete. The product is based on finite element method and modern concepts in material modeling. The system is fully programmed in C++ using object oriented techniques as well as some modern concepts of template metaprogramming. The program allows engineers to virtually test their structures either in the design phase or during the service life. The ATENA system contains also a transport module for thermal, seepage analysis or for pollution transport in partially saturated rock and soil.

The Fig. 2 shows the GID problem type AtenaV2, which is used to interface GID with ATENA. This interface currently supports sub-problem types for static stress analysis and for creep analysis both in 2D as well as 3D. The sub-problem type “Static3D” supports the following materialsr:

- plane concrete (3D),
- reinforced concrete (3D),
- steel (3D),
- discrete reinforcement (1D),
- rock or soil (3D),
- contact material (2D, 3D) and
- classic elastic (1D,2D,3D).

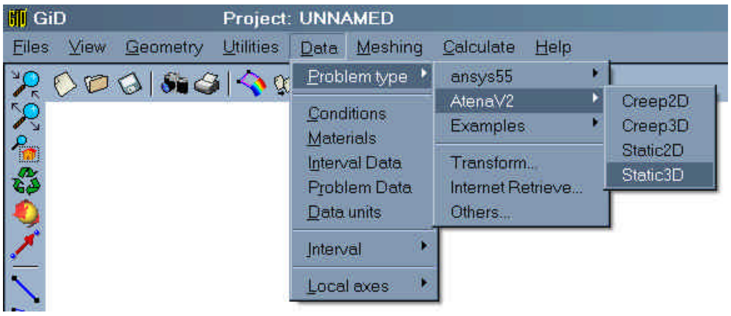

Fig. 2 Problem type “AtenaV2” in the GID program

3 TYPE OF FINITE ELEMENTS

The Fig. 3 depicts schematically the main elements ATENA for the modeling of reinforced concrete:

- Contact-interface,
- Smeared reinforcement,
- Classic bar reinforcement,
- Bar reinf. with bond
- Other classic 3D elems.

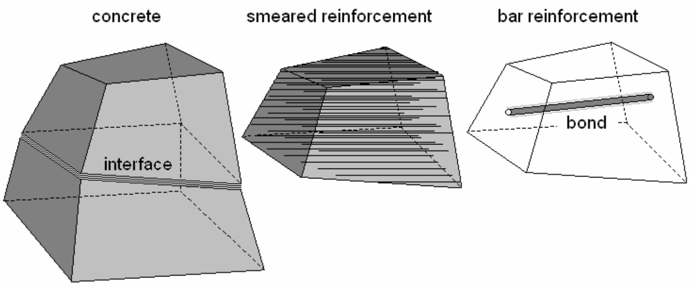

Fig. 3 Three-dimensional elements of reinforced concrete.

ATENA supports two models for reinforcement: smeared and discrete. Smeared reinforcement assumes the reinforced concrete to be modeled by a composite material model. This case is shown at Fig. 4. The visible bookmark shows the input dialog for the nonlinear concrete material and the next bookmarks are for the smeared reinforcement (Fig. 5). This approach generally does not represent a problem, when connecting ATENA to general purpose pre-processing packages.

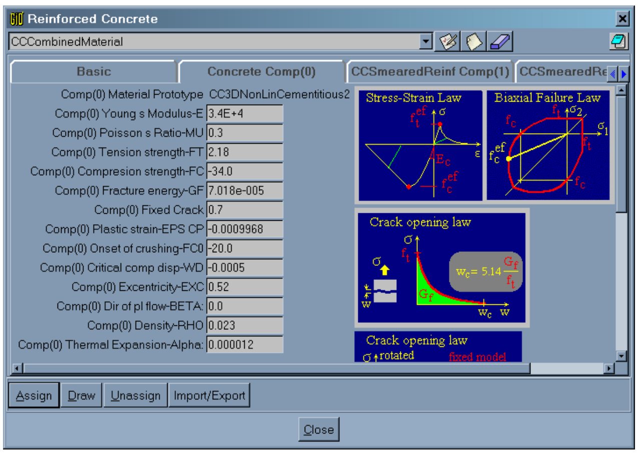

Fig. 4 Composite material model in GID.

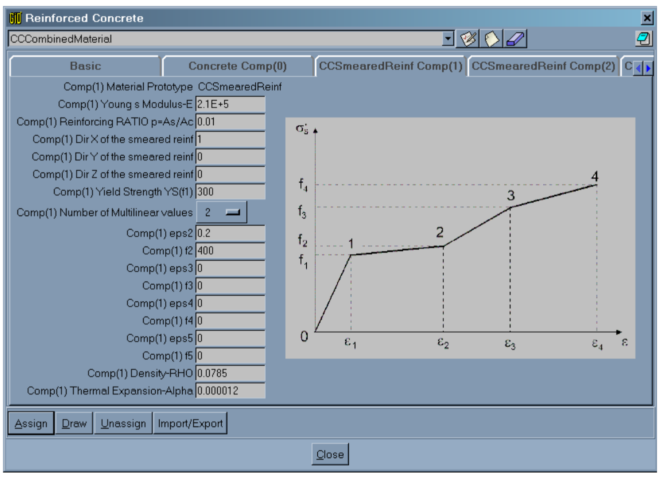

Fig. 5 Multi-Linear Material model of Reinforcement in GID.

The discrete reinforcement, however, is modeled using the embedded bar approach, in which the reinforcement is defined totally independently to the solid mesh. The program ATENA automaticaly decomposes the reinforcement into individual segments, which are then embedded into the solid 3D mesh using special constraints(Master/Slave). Such an approach is usually not well supported by general purpose FE modellers. The discrete reinforcement is modeled using GID curves that are discretized into trusses totally independently to the existing 2D or 3D meshes. These trusses are passed into ATENA as special reinforcement macro-elements. These macro-elements are then decomposed into ATENA embedded bar elements. The nodes of these embedded bar elements are connected to the finite element nodes of the 2D/3D mesh via special master-slave conditions.

4 Simple, complex supports and master-slave boundary conditions.

Simple support and complex support boundary conditions represent boundary conditions of Dirichlet types, i.e. boundary conditions that prescribe displacements. On the other hand, Simple load boundary conditions is an example of von Neumann type boundary conditions, when forces are prescribed.

Let $\mathbf{K}$ is structural stiffness matrix, $\underline{u}$ is vector of nodal displacements and $\underline{R}$ is a vector of nodal forces. Futher let $\underline{u}$ is subdivided into vector of free degrees of freedom $\underline{u}_N$ (with von Neumann boundary conditions) and constrained degrees of freedom $\underline{u}_D$ (with Dirichlet boundary conditions):

$$\underline{u} = \begin{bmatrix} \underline{u}_N \\ \underline{u}_D \end{bmatrix} \qquad (0.1)$$

The problem governing equations can then be written:

$$\begin{bmatrix} \mathbf{K}_{NN} & \mathbf{K}_{ND} \\ \mathbf{K}_{DN} & \mathbf{K}_{DD} \end{bmatrix} \begin{bmatrix} \underline{u}_N \\ \underline{u}_D \end{bmatrix} = \begin{bmatrix} \underline{R}_N \\ \underline{R}_D \end{bmatrix} \qquad (0.2)$$

ATENA software supports that any constrained degree of freedom can be a linear combination of other degrees of freedom plus some constant term:

$$\underline{u}_D^i = \underline{u}_D^{i,0} + \sum_k a_k \underline{u}_N^k \qquad (0.3)$$

where $\underline{u}_D^{i,0}$ is the constant term and a_k are coefficients of the linear combination. Of course, the equation (0.3) can include also the term $\sum_l a_l \underline{u}_D^l$, however it is transformed into the constant term.

The free degree of freedom are then solved by

$$\underline{u}_N = \left(\mathbf{K}_{NN}\right)^{-1}\left(\underline{R}_N - \mathbf{K}_{ND}\underline{R}_D\right) \quad (0.4)$$

and the dependent $\underline{R}_D$ are solved by

$$\underline{R}_D = \mathbf{K}_{DN}\underline{u}_N + \mathbf{K}_{DD}\underline{u}_D \qquad (0.5)$$

The ATENA **simple support** boundary conditions mean that the boundary conditions use only constant terms are $\underline{u}_D^{i,0}$, (i.e. $a_k = 0$). The **complex support** boundary conditions use the full form of (0.3).

The boundary conditions as described above allow to specify for one degree of freedom either Dirichlet, or von Neumann boundary condition, but not both of them the same time. It comes from the nature of finite element method. However, ATENA can deal also this case of more complex boundary conditions by introducing Lagrange multipliers. The derivation of theory behind this kind of boundary conditions is beyond the scope of this manual. Details can be found elsewhere, e.g. in (2). To apply this type of boundary conditions in ATENA, specify for those degree of freedom both simple load and complex support boundary condition, the latter one with the keyword “RELAX” keyword in its definition.

Nice feature about ATENA is that at any time it stores in RAM only $\mathbf{K}_{NN}$ and all the elimination with the remaining blocks of **K** is done at element level at the process of assembling the structural stiffness matrix.

A special type of complex boundary conditions of Dirichlet type are so-called master-slave boundary conditions. Such a boundary condition specifies that all (available) degrees of one finite node, (i.e. slave node) are equal to degrees of freedom of another node (i.e. master node). If more master nodes are specified, then these master nodes are assumed to form a finite element and degrees of freedom of the slave node is assumed to be a node within that element. Its (slave) degrees of freedom are approximated by element nodal (i.e. master) degrees of freedom in the same way as displacements approximation within a finite element. The coefficients a_k in (0.3) are thus calculated automatically. This type of boundary conditions is used for example for fixing discrete reinforcement bars to the sorounding solid element .

5 Conclusion

The proposed solution allows user friendly definition of reinforcement for numerical analysis of reinforced concrete structures. In the simulation of reinforced concrete structural elements, it is of utmost importance to correctly model the location of each reinforcement bar, since the location and orientation of reinforcement bars is the main parameter affecting the structural load carrying capacity or the resulting crack width. The ATENA-GID interface allows engineers to define the reinforcement bars independently of the 2D/3D mesh, which greatly simplifies the definition of the numerical model, while preserving its accuracy. This approach allows engineers to concentrate on the analyzed structure rather then on the fine and tedious details of the numerical model.

6 Acknowledgement

The authors would like to thank the Czech National Grant Agency for supporting this development work under the research project „Bond Modelling in Reinforced Concrete by Finite Element Method“, 103/04/2083.

7 References

- BATHE, K.J. (1982), Finite Element Procedures In Engineering Analysis, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, ISBN 0-13-317305-4.
- CERVENKA, J., KEATING, S.C., AND FELIPPA, C.A. (1993), “Comparison of strain recovery techniques for the mixed iterative method”, Communications in Numerical Methods in Engineering, Vol. 9, 925-932.
- ZIENKIEWICZ, O.C., TAYLOR, R.L., (1989), The Finite Element Method, Volume 1, McGraw-Hill Book Company, ISBN 0-07-084174-8.
- Vladimír Cervenka, Libor Jendele and Jan Cervenka, ATENA Program Documentation Part 1 Theory