# GID-ATENA INTERFACE – ADVANCED SIMULATION OF REINFORCED CONCRETE STRUCTURES

**Jan Cervenka*, Jiri Niewald***

* Cervenka Consulting
Predvoje 22, 162 00 Praha 6
e-mail: cervenka@cervenka.cz , web page: http://www.cervenka.cz

**Key words:** Nonlinear Analysis, Reinforced Concrete, Fracture Mechanics, Thermal Analysis, Creep Analysis, Dynamic Analysis

**Abstract.** *This paper describes the implementation and features of the GID interface to program ATENA. The interface supports several analysis types available in ATENA such as: static, creep, transport and dynamic. Special attention is devoted to the reinforcement modelling and implementation in the GID-ATENA interface.*

## 1 INTRODUCTION

ATENA[i] software is a finite element analysis system for advanced simulation of real behavior of reinforced concrete structures. It includes many unique features specific to the reinforced concrete analysis such as: robust and tested material models for concrete [ii], reinforcement, steel and soil structure interaction. The system supports advanced analysis of statics, dynamics, transport as well as creep problems or extreme loading conditions such as for instance the structural damage and degradation due to fire.

The GID interface (Figure 1) for the ATENA software supports all analysis types available in ATENA, i.e. static, transport, creep and dynamics. Special treatment was necessary to facilitate the reinforcement modeling, which is an essential part of software targeted for the simulation of reinforced concrete structures. ATENA supports several methods of reinforcement modeling. The most comprehensive way of reinforcement modeling is based on the so-called embedded element approach (Figure 2). This approach is not directly supported by the GID pre-processing environment and special method had to be developed to facilitate a user-friendly as well as realistic reinforcement definition. The other supported method are smeared reinforcement, layered reinforcement in shell elements or reinforcement fibers in beam elements.

The reinforcement modeling in GID-ATENA interface is schematically depicted in Figure 3. In the embedded reinforcement approach, the reinforcement crossing a mesh with three-dimensional elements is subdivided into truss elements, such that each truss element is defined by the intersection of the reinforcement bar with the individual solid elements, as it is shown in Figure 2. The displacements of the nodes of these reinforcement trusses are then constrained to the deformation of the surrounding solid element. This subdivision of the reinforcement bars into the truss elements is not supported by GID, and it is automatically handled by ATENA software at the beginning of the analysis.

The whole process is shown in Figure 3. The reinforcement is modeled as a line or poly-line, which in general case might be a curved entity. After the mesh generation, all one-dimensional entities that are not connected to any surface or region are automatically subdivided into 1D finite elements. These elements are then passed into ATENA software to describe the reinforcement geometry.

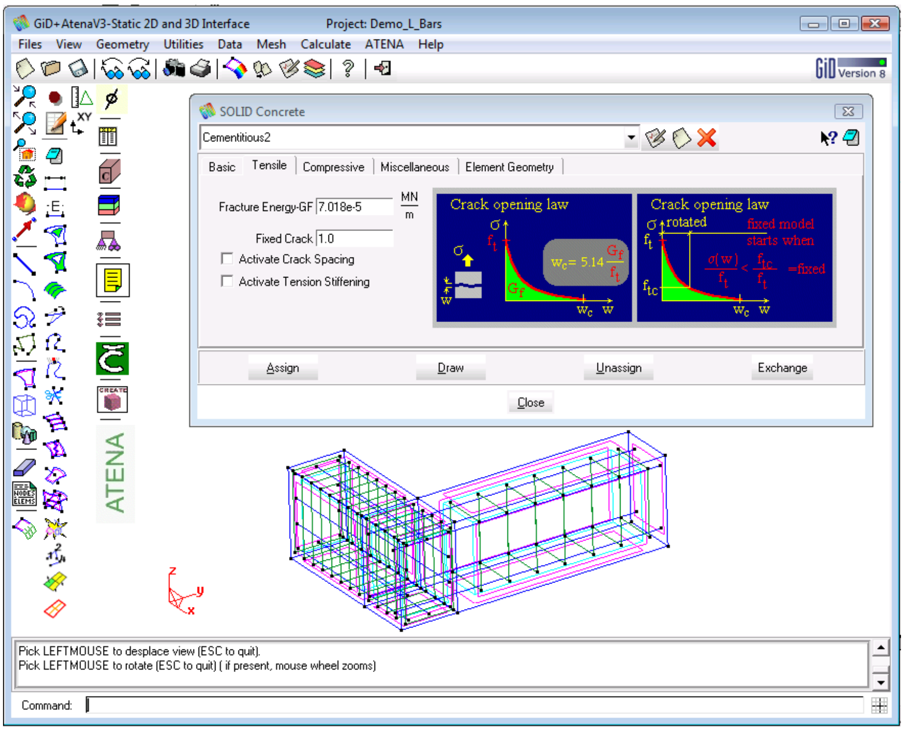


Figure 1: GID-ATENA Interface – Reinforced Concrete

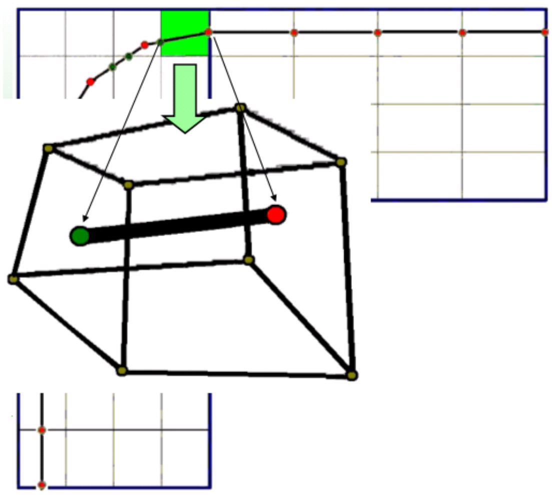

Figure 2: Modelling Reinforcement – Embedded Element Approach

Based on this information, ATENA determines the intersections with the basic three-dimensional solid model, and new truss elements and corresponding nodes are added to the structural numerical model. The newly created nodes are constrained to the existing nodes of the solid elements using the appropriate weight coefficients[iii].

The presented approach has been successfully applied for many practical engineering problems. As an example of application, a nonlinear analysis of pre-stressed nuclear containment is selected. Figure 4 shows the resulting temperature distribution for an accident scenario from ATENA transport analysis. Figure 5 depicts the three-dimensional model created in GID of the containment including the internal steel liner. The failure crack pattern and stresses in the complex arrangement of pre-stressing tendons is described in Figure 6.

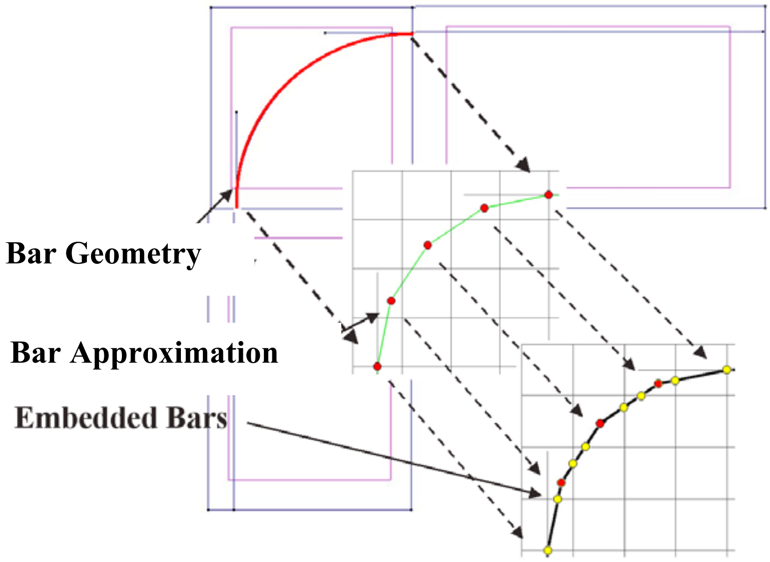


Figure 3: Modelling Reinforcement – Embedded Element Approach

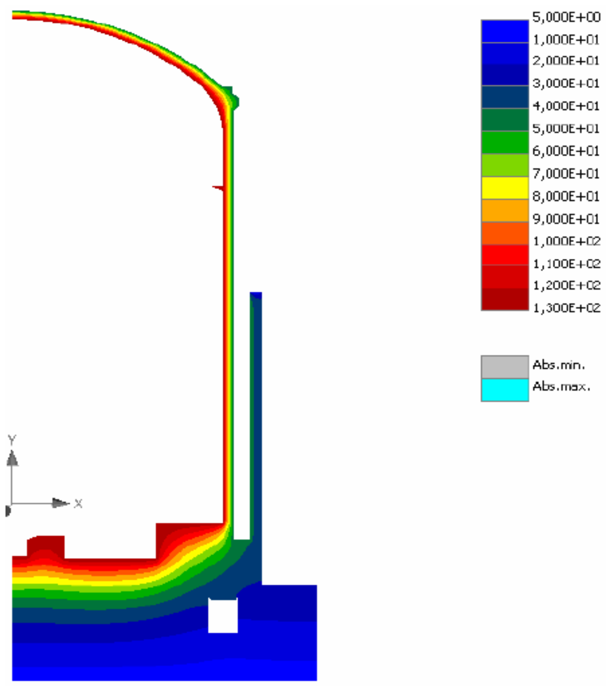


Figure 4: Thermal Analysis of Temperature Distribution at a Nuclear Containment during an Accident

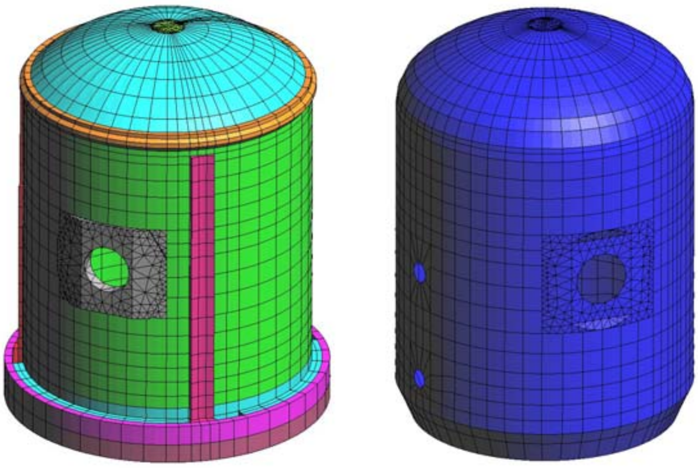

Figure 5: Numerical Model of Pre-stressed Containment (a) containment (b) steel liner

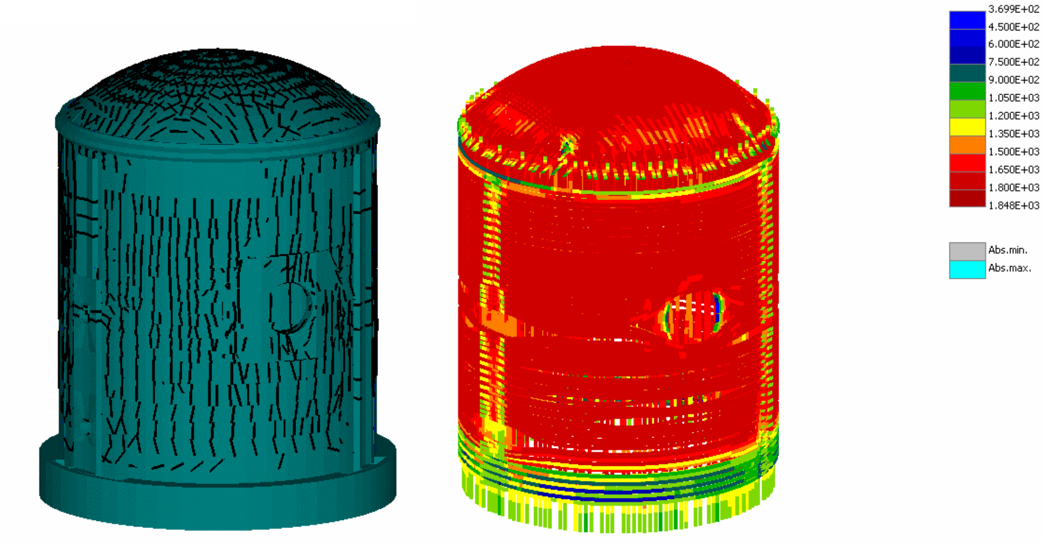


Figure 6: Crack Pattern at Failure and Yielding of Prestressing Tendons

## 2 CONCLUSIONS

The authors acknowledge the financial support of GACR project 103/07/1660 for the development of the transport interface and the European project I-SSB NMP2-CT-2006-026661 for the dynamic interface.

REFERENCES


[i] Červenka Consulting, (2007) *ATENA program documentation*, Part 1-8, www.cervenka.cz

[ii] Červenka, J., and Papanikolaou, V.K. (2008) “Three dimensional combined fracture-plastic material for concrete”, International Journal of Plasticity, in press, doi:10.1016/j.ijplas.2008.01.004

[iii] Jendele L., Cervenka J., (2007), On the Solution of Multi-Point Constraints - Application to FE Analysis of Reinforced Concrete Structures, submitted to Computers and Structures