17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9 – 12 May 2011

EASY GENERATION OF MODELS/MESHES USING AN OPEN-SOURCE SOFTWARE SALOME

P. Hlaváček*, V. Šmilauer*, B. Patzák*

Abstract: *Computer-aided engineering (CAE) represents the broad usage of computers for engineering problems. It includes computer-aided design (CAD) or computer-aided analysis (CAA) to prepare representation of physical object, to discretize the object, to run analysis and to plot the results. Commercial software have often integrated modeler/mesher/solver/post-processor into one big package. These include ATENA, ABAQUS, ANSYS, SCIA, MARC to mention a few. The open-source CAE has been dropping behind in terms of user friendliness and efficiency. The situation is changing dramatically now since commercial companies support development of open-source. Salome presents a nice platform for pre/post-processing, released under the terms of the GNU LGPL license. This paper will demonstrate application of Salome modeler/mesher for various engineering analyses: steel buckling and multiscale heat analysis. Generation of appropriate mesh is easy and very efficient and can be exported to several computational codes, such as an object-oriented OOFEM. Post-processing can be carried out via VTK data format, for example in Paraview open-source post-processor. Python language can easily glue all components into one package, using advantages of control/modification/verification stages in each data transfer process.*

Keywords: ***CAE, CAD, Salome, OOFEM.***

1. Introduction

Computer Aided Engineering (CAE) represents a broad computer software support in engineering tasks, covering, e.g., Computer Aided Design (CAD) or Computer Aided Analysis (CAA). This paper elucidates CAE design in integration with finite element package used for civil engineering purposes.

The open-source platform Salome is a pre/post-processing environment for numerical simulations. In the presented paper the pre-processing, which includes creation of model, meshing and input data converting, for analysis in in-house package OOFEM is carried out. The OOFEM (Object Oriented Finite Element Solver) is an in-house open-source package for solving of multiphysics problems (Patzák, 2011). The python-based unv2oofem converter efficiently integrates pre-processing software with computationally-based OOFEM. For example, after the modeling and meshing in Salome platform, the mesh could be exported as UNV data-file. The unv2oofem converter then exports the data to OOFEM-readable input file. Unv2oofem reads the mesh geometry from UNV data set assembles material properties and node/boundary conditions defined in extra file and writes the OOFEM input file in native format.

2. Selected open-source tools for CAE

The Salome is an open-source platform for numerical simulations (Open CASCADE, 2011). It is developed by Open Cascade (France) and LGPL licensed, meaning the source could be modified and redistributed. The Salome is primarily designed as a platform for integration of external codes. The main feature of Salome represents the link between CAD modeling and computational software in terms of CAA. The Salome, as distributed, allows a simple and fast creation of various geometry models and sophisticated meshing.

* Ing. Petr Hlaváček, doc. Ing. Vít Šmilauer, Ph.D. and prof. Dr. Ing. Bořek Patzák: Faculty of Civil Engineering, Czech Technical University in Prague; Thákurova 7; 166 29, Prague; CZ, e-mails: petr.hlavacek@fsv.cvt.cz, vit.smilauer@fsv.cvut.cz, borek.patzak@fsv.cvut.cz

The OOFEM is an open-source finite element solver with object oriented architecture developed at the Faculty of Civil Engineering in Prague by B. Patzák *et al.* (Patzák & Bittnar, 2001). The OOFEM is released under the GNU General Public License (GPL) and could be used for solving of multiphysics tasks as mechanical, transport or fluid mechanics problems (Patzák, 2011). The OOFEM input file has an open ASCII format and generally can be written/modified in any text software. The output from OOFEM can be exported to VTK data set, for example.

The ParaView is an open-source data visualization application, which allows to build fast visualizations for data analysis using qualitative and quantitative techniques (Kitware, 2011). The data exploration can be done interactively in 3D. The ParaView supports input file in the VTK format.

3. Application of CAE/CAD tools to engineering problems

Two selected engineering problems solved by above mentioned software are presented in this section. The mechanical problem of steel structure buckling and the heat transport problem during concrete casting of a highway bridge are described.

3.1. Steel plasticity with buckling

This example demonstrates analysis of a structural detail, where a girder beam crosses a column. A welded pair of U profiles 300 mm creates both cross-sections of the beam and the column. The geometry was meshed into 10 383 nodes and 30 786 linear tetrahedral elements.

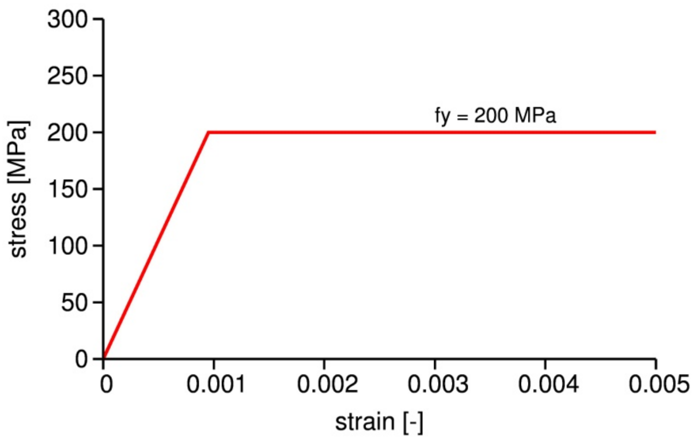

Fig. 1: Stress–strain curve for ideal elasto-plastic material.

Perfectly plastic material is assigned to steel with the yield strength of 200 MPa (Fig.1), which roughly corresponds to the steel grade S235. Updated Lagrangian formulation ensured updating nodal position after each time increment thus facilitating second-order theory and equilibrium on deformed shapes assuming small strains in constitutive laws (Eq. 1) (Jirásek & Bažant, 2002).

$$\boldsymbol{\sigma} = \mathbf{D} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_{pl}) \tag{1}$$

Fig. 2 shows the mesh created in Salome package; also the stresses and the displacement vector immediately before collapse are depicted. The maximum axial force 1.58 MN applied on the vertical column leads to collapse in plastic/buckling mode. The computation took 11.5 min in eight loading steps. The whole analysis, including construction of geometry, meshing, computation, post-processing and interpretation took less than one hour of work.

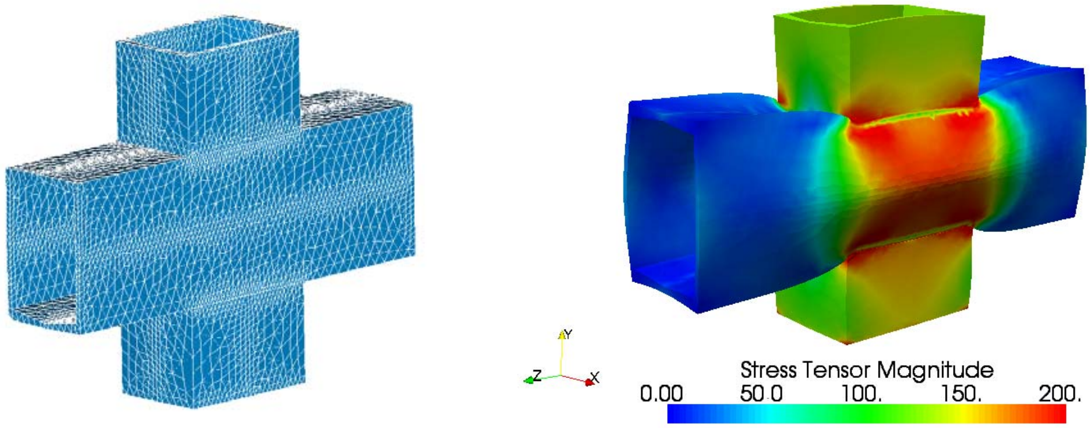

Fig. 2: Detail of a joint with the mesh and computed stress magnitude.

3.2. Multiscale heat transport analysis

The second example shows the simulation of heat transport during hardening of concrete in Oparno highway bridge. Hydrating concrete produces significant amount of hydration heat, which causes several problems in massive concrete elements. Multiscale simulation helped to find an optimal position of cooling pipes and cooling regime on the bridge arch. The well-known non-stationary heat conduction equation, mentioned for example by Bao-Lin Wang *et al.* (Bao-Lin Wang & Yiu-Wing Mai, 2005), reads:

$$\frac{-\partial q(\mathbf{x},t)}{\partial x} + \overline{Q}(\mathbf{x},t) = \rho(\mathbf{x},t)c(\mathbf{x},t)\frac{\partial T(\mathbf{x},t)}{\partial t} \tag{2}$$

The simulation runs on the left symmetric part of the arch cross-section. The model and mesh were created in Salome platform, the 2D triangular finite elements were oriented in directions of supposed temperature gradients and refined in the region of cooling pipes, see Fig. 3.

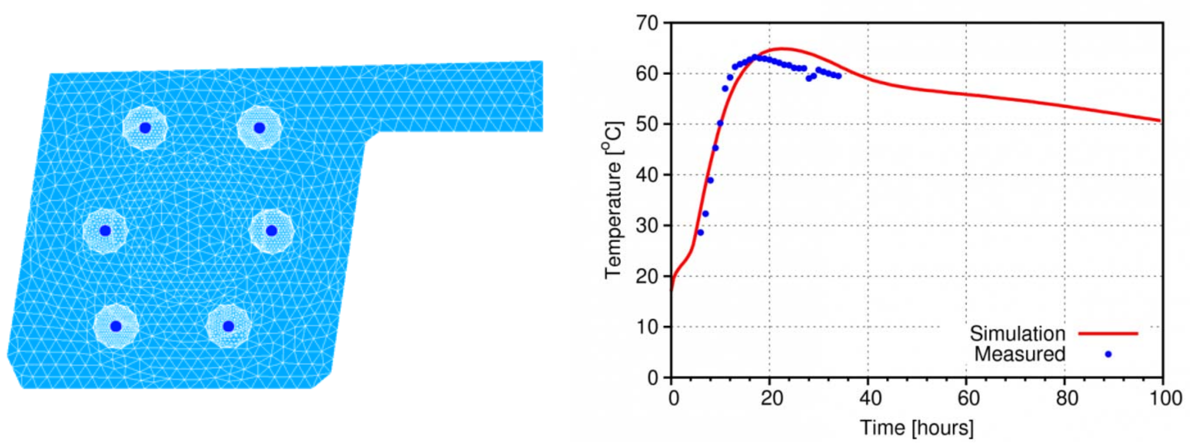

Fig. 3: Left: Mesh of Oparno bridge cross-section, Right: Validation of simulation with measured temperatures in the core of beam.

Fig. 4 shows the temperature evolution during concrete hardening and induced out-of-plane stress when considering B3 model for concrete creep. Optimal position of cooling pipes is apparent. Note that the cooling turns off after several hours which detaches natural Dirichlet's boundary conditions and changes number of equations. The flat bottom subfigure shows the 2D triangular mesh and the assignment of hydration models to groups of finite elements on the cross-section. Fig. 2 also validates the multiscale simulation with the temperature in the core of the beam. Temperature remained below 65°C during summer casting, which was found acceptable.

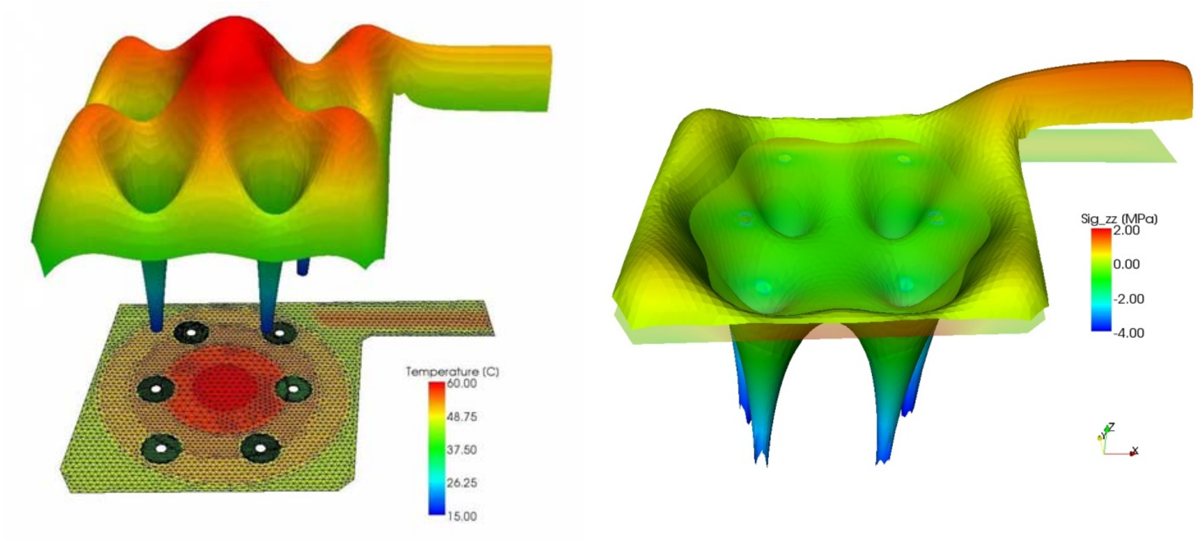

Fig. 4: Temperature and stress field during concrete hardening.

4. Conclusions

With the development of open-source software, CAE/CAD analysis can be easily integrated with other computational software. This facilitates necessary step for the creation of a user-friendly platform, which is required in the majority of modeling steps. The Salome platform as a pre-processor in the combination with the OOFEM as a solver and with the ParaView as a post-processor present an open-source alternative to the commercial computational programs. The assessment of a steel joint and the design of cooling pipes proved the excellent features of the presented software.

Acknowledgement

Financial support from the Grant Agency of the Czech Technical University in Prague, grant No. SGS10/135/OHK1/2T/11 and from the Czech Science Foundation GA ČR103/09/H078 are gratefully acknowledged.

References

Kitware (2011) ParaView application home page, http://www.paraview.org.

Open CASCADE (2011) Salome platform home page, http://www.salome-platform.org.

Patzák B. (2011) OOFEM project home page, http://www.oofem.org.

Patzák B. & Bittnar Z. (2001) Design of object oriented finite element code. Advances in Engineering Software, 32(10-11):759--767.

Šmilauer, V. et al. (2011) Multiscale simulation of fracture of braided composites via repetitive unit cells. Engineering Fracture Mechanics, in press.

Jirásek M. & Bažant Z. P. (2002) Inelastic Analysis of Structures, Book, John Willey & Sons, Ltd, ISBN 0-471-98716-6

Bao-Lin Wang & Yiu-Wing Mai (2005) Transient one-dimensional heat conduction problems solved by finite element. International Journal of Mechanical Sciences, 47(2):303-317.