DTU Library

Abaqus2MatLab: A Novel Tool for Finite Element Post-Processing

Martínez Pañeda, Emilio; Papazafeiropoulos, George; Muñiz-Calvente, Miguel

Link back to DTU Orbit

ABAQUS2MATLAB: A NOVEL TOOL FOR FINITE ELEMENT POST-PROCESSING

Emilio Martínez-Pañeda[1(*)]**, George Papazafeiropoulos**[2]**, Miguel Muñíz-Calvente**[3]

[1]Department of Mechanical Engineering, Technical University of Denmark, Kgs. Lyngby, Denmark
[2]Department of Structural Engineering, National Technical University of Athens. Athens, Greece
[3]Department of Construction and Manufacturing Engineering, University of Oviedo, Gijón, Spain
[(*)]*Email:* mail@empaneda.com

ABSTRACT

A novel piece of software is presented to connect Abaqus, a sophisticated finite element package, with Matlab, the most comprehensive program for mathematical analysis. This interface between these well-known codes not only benefits from the image processing and the integrated graph-plotting features of Matlab but also opens up new opportunities in results post-processing, statistical analysis and mathematical optimization, among many other possibilities. The software architecture and usage will be appropriately described and two problems of particular engineering significance addressed to demonstrate its capabilities. The source code, detailed documentation and a large number of tutorials can be freely downloaded from www.abaqus2matlab.com.

Keywords: Abaqus2Matlab, post-processing, finite element method.

INTRODUCTION

Despite the popularity of Abaqus, a leading commercial finite element package, and Matlab, one of the most powerful mathematical analysis tools, a connection between them is still lacking. To fill this gap, a novel software tool is here proposed: *Abaqus2Matlab*, which allows to run Abaqus directly from Matlab and to post-process the results, providing a link between the two well-known packages in a non-intrusive and versatile manner. *Abaqus2Matlab* is distributed as source code with the aim of facilitating research.

RESULTS AND CONCLUSIONS

The range of applications of Abaqus2Matlab is enormous, as it provides a non-intrusive connection between a sophisticated finite element package and the most comprehensive mathematical analysis tool. For demonstration purposes, two problems of particular interest from the scientific and engineering perspective have been addressed.

First, the toolbox proposed is used to estimate cleavage fracture in metals, where a probabilistic approach is needed due to the statistical nature of the micromechanisms involved. Since the seminal work by Beremin (Beremin, 1983), cleavage fracture toughness estimations are based on Weibull statistics and the weakest link model. Grounded on this approach, a novel probabilistic framework is proposed that takes advantage of the advanced statistical tools of MATLAB to estimate all Weibull-related parameters without any *a priori* assumptions. Thus, the threshold stress for crack growth σ_{th}, the scaling parameter σ_u and the

Weibull modulus *m* are obtained by means of an iterative procedure involving least squares estimates of the cumulative distribution functions. Therefore, taking advantage of Matlab capabilities, Weibull parameters are calculated by finding the distribution whose cumulative function best approximates the empirical cumulative distribution function of the experimental data. The capabilities of *Abaqus2Matlab* to model cleavage fracture are benchmarked with an extensive experimental data set developed within the Euro toughness project (Heerens and Hellmann, 2002). Weibull parameters and failure probabilities are computed, and a *hazard map* constructed (see Figure 1).
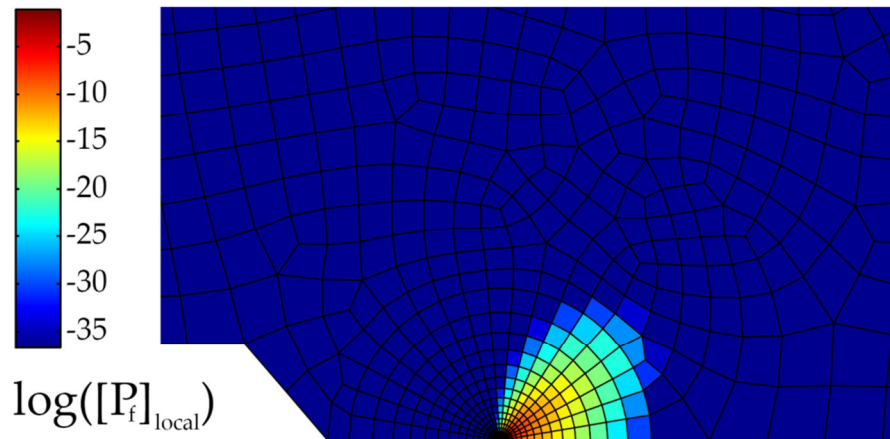

Fig. 1 - Hazard map. The legend shows the local probability of failure

The software capabilities are also employed to perform an advanced inverse-optimization methodology so as to obtain the parameters governing the traction-separation law that describes deformation and fracture (see Papazafeiropoulos et al., 2017). Hence, quantitative insight into the initiation and subsequent propagation of damage is obtained through neural network optimization and a hybrid experimental-numerical strategy. Thus, *Abaqus2Matlab* largely facilitates structural integrity assessment by taking advantage of advanced damage models available in Abaqus and modern optimization capabilities of Matlab.

Diverse examples (including the ones outlined here), comprehensive documentation and the source code can be downloaded from www.abaqus2matlab.com.

REFERENCES

[1]-Beremin FM. A local criterion for cleavage fracture of a nuclear pressure vessel steel. Metallurgical and Materials Transactions A, 1983, 14, p. 2277-2287.

[2]-Heerens J, Hellmann D. Development of the Euro fracture toughness dataset. Engineering Fracture Mechanics, 2002, 69, p. 421-449.

[3]-Papazafeiropoulos G, Muñiz-Calvente M., Martínez-Pañeda E. Abaqus2Matlab: a suitable tool for finite element post-processing (*submitted*).