Abaqus2Matlab: A suitable tool for finite element post-processing

A suitable 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 are appropriately described and two problems of particular engineering significance are addressed to demonstrate its capabilities. Firstly, the software is employed to assess cleavage fracture through a novel 3-parameter Weibull probabilistic framework. Then, its potential to create and train neural networks is used to identify damage parameters through a hybrid experimental-numerical scheme, and model crack propagation in structural materials by means of a cohesive zone approach. The source code, detailed documentation and a large number of tutorials can be freely downloaded from www.abaqus2matlab.com.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, National Technical University of Athens, University of Oviedo
Contributors: Papazafeiropoulos, G., Muñiz-Calvente, M., Martínez Pañeda, E.
Pages: 9-16
Publication date: 2017
Peer-reviewed: Yes

Publication information
Journal: Advances in Engineering Software
Volume: 105
ISSN (Print): 0965-9978
Ratings:
BFI (2017): BFI-level 1
Scopus rating (2017): CiteScore 4.06 SJR 1.159 SNIP 2.367
Web of Science (2017): Impact factor 3.198
Web of Science (2017): Indexed yes
Original language: English
Keywords: Abaqus2Matlab, Post-processing, Finite Element Method, Weibull stress model, Inverse analysis
Electronic versions:
AES.pdf. Embargo ended: 26/01/2019
DOIs:
10.1016/j.advengsoft.2017.01.006
Research output: Contribution to journal › Journal article – Annual report year: 2017 › Research › peer-review