AAA Tutorials

9. Abaqus analysis and stress extraction

Grand Roman Joldes
Senior Research Fellow

Intelligent Systems for Medicine Lab. (ISML)
Vascular Engineering
Crawley, WA, 6009, AUSTRALIA

Phone: + (61) 8 6488 3125
Email: grand.joldes@uwa.edu.au

Software installation

Have Abaqus installed and available via command line.
Copy the files AAA_RunAbaqus.bat, AAA_ExtractStress.bat, ExtractResults.py and abaqus_v6.env in the folder where AAA.inp (which you want to analyze) is located.

Software configuration

You can modify abaqus_v6.env to change configuration of Abaqus, especially the scratch folder (where temporary files will be placed).

Expected inputs

The AAA.inp (which includes Wall.inp and, depending on case, ILT.inp).

The configuration of the Abaqus model is done in AAA.inp and can be modified.

Running the software

Run AAA_RunAbaqus.bat and then AAA_ExtractStress.bat.

Another option is to use the Abaqus CAE to import AAA.inp, review the configuration, create a job, perform the analysis, then run the python script ExtractResults.py to extract stress information.

Expected outputs

An output database with the analysis results, AAA.odb.
A file with the AAA wall geometry including info about von-Mises and max. principal stresses at the nodes, stress.vtk.

Thank You !

grand.joldes@uwa.edu.au