



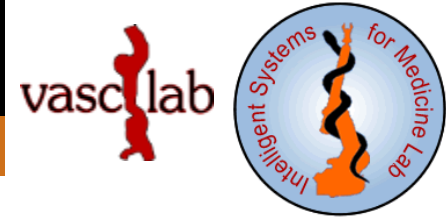
# AAA Tutorials

## 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](mailto: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](mailto:grand.joldes@uwa.edu.au)