Use Abaqus Scripting to create FEM simulations

Finite element method (FEM) is powerful tool to simulate and predict engineering problems. Abaqus is one of the most famous commercial software for FEM. *Abaqus CAE* is the software the whole modeling is done through it and the final model will pass to *Abaqus Solver* as it is described in figure 1.

Abaqus CAE has capability of using python scripts to perform its functions. In other word, there is possibility of using the Abaqus CAE without using its graphical user interface (GUI). When a Abaqus CAE file is saved, two files will be available in your saving directory. First file has the extension of *.cae* and the second file has *.jnl* which is the journal file. The journal file will contain the python command represents your use of GUI.

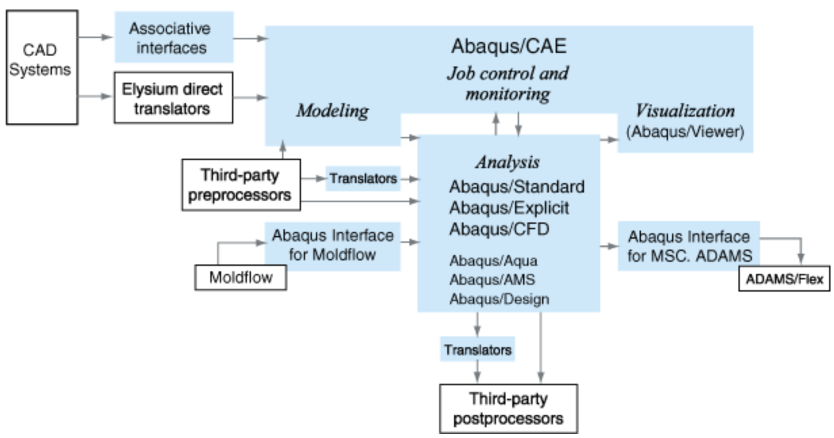

Figure 1: Abaqus products relations (Getting Started with Abaqus: Interactive edition, n.d.)

There are several ways to use scripting feature of Abaqus CAE.

1. Go to *File* -> Select on *Run Script* and choose your python file
2. Use Abaqus terminal and write you code there

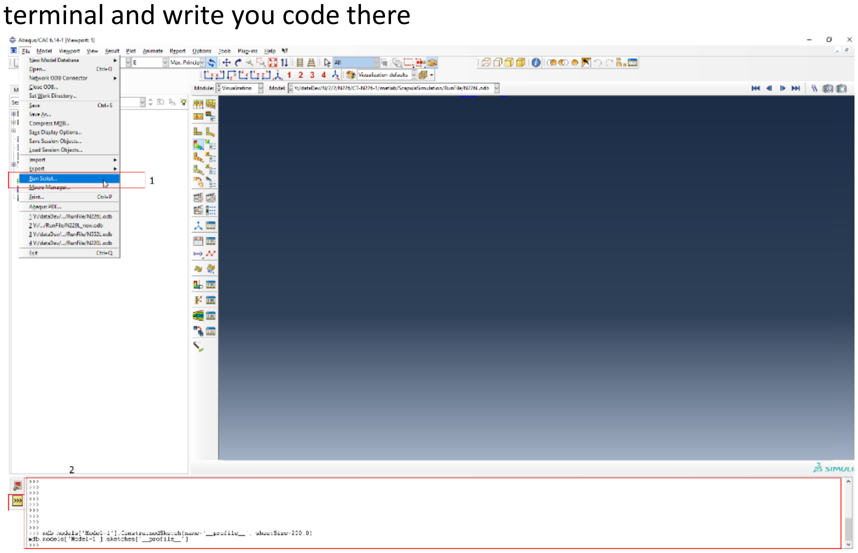

Figure 2: Execute your python code in Abaqus CAE

As an exercise in order to create a sketch and draw a line in Abaqus sketch, you can easily use the command below in abaqus terminal. There is one-hour tutorial for Abaqus scripting with detail instruction in this link.

mdb.models['Model-1'].ConstrainedSketch(name='__profile__', sheetSize=200.0)
mdb.models['Model-1'].sketches['__profile__'].Line(point1=(0.0, 42.0), point2=45.0, 42.0))

The best advantage of using scripting in Abaqus is to prevent repetitive commands or even automatize the whole process of the simulation. The best source of learning Abaqus scripting is to refer to "*Abaqus Scripting Reference Guide*" (Abaqus Scripting Reference Guide, n.d.).

The drawback of Abaqus scripting is that for small projects it may not worth investing the time to write the code. As it is mentioned before, after saving Abaqus CAE file, you will have journal file containing the python commands of your work. In that case you can do one model manually, then you can open the journal file and save it as python file and use this python file as template for modifying your model. In the python file[1] attached there is example of our project which the python script will create input files for four different variation of the model.

Bibliography

Abaqus Scripting Reference Guide. (n.d.). Retrieved from http://130.149.89.49:2080/v6.13/books/ker/default.htm

Getting Started with Abaqus: Interactive edition. (n.d.). Retrieved from http://130.149.89.49:2080/v6.11/books/gsa/default.htm?startat=ch01s01.html#gsa-abaqus-mod

[1] Please open this link with your EPFL account on google drive.