

March 10, 2019

1(1)

Aalto Students

**Ref:** Implementation of a material model in Abaqus

Dear students,

During the course "Continuum Mechanics and Material Modelling" some students told to me that they are interested in implementation of a material model in the Abaqus finite element code. I have arranged a course, equipment and agreement with Simulia for those ones who are ready to implement. Please, notice the following:

It takes at least 60 active working hours to prepare the code. Probably you need more. Please, be sure that you can invest so much time for the implementation. The dead-line is end of June 2019.

The programming language of the code is Fortran. Abaqus accepts programming language C as well, but I cannot provide any help for C programming.

You will get a very limited amount of support. I will give to you a template for coding and a 16 pages long instruction how to work with Abaqus for this particular case.

You have to share a computer which is in the office 206 in the K3 building. I try to create a system which allows you to make a reservation to use the computer.

You can enter the K3 building only during the office hours.

I will create a private account for you. So send your user name and password to me. When you log in first time, you can change your password.

I do not have any backup system. So use a private memory stick for your backup.

Save your files into your personal folder to avoid someone to copy your work.

Best Regards,

Kari Santaoja

Material Model implementation 2019 03 10.wpd/Santaoja