ABAQUS Tutorial

TF: Nanshu Lu

Email: nanshulu@fas.harvard.edu   Phone: 617-496-7107   Office: Pierce Hall 403

Schedule

Date	Action	Notes
11/05/2007 (Monday) 10am~11am	Tutorial Session 1 @ Maxwell Dworkin Basement Computer Lab (B121)	Introduction to ABAQUS, using ABAQUS CAE. **Assignment starts**
11/07/2006 (Wednesday) 10am~11am	Tutorial Session 2 @ Maxwell Dworkin Basement Computer Lab (B121)	ABAQUS Command
11/16/2006 (Friday)	**Assignment Due**	
Every Tuesday 4~5pm (starts from Nov. 6)	Office hour @ Maxwell Dworkin Basement Computer Lab (B121)	To help with homework problems and computer assignment problems

➢ **Tutorial materials**

Please download the following materials from iMechanica or our course website:
1-1 ABAQUS tutorial for ES 240l.pdf (the file you are currently looking at)
1-2 Learning ABAQUS.pdf
1-3 CAE Example.pdf

➢ **Access to ABAQUS**

I will pass around the ABAQUS v6.6 installment CD during the two tutorial sessions. Please bring your laptop or any movable disks if your computer is running with windows system. Besides, ABAQUS 6.5-1 is installed on the computers in Maxwell Dworkin B121. Your ID cards

have been activated so that you have access to MD B121 at off hours.

➢ **ABAQUS users' manuals**

I'll pass around the ABAQUS documentation CD during the two tutorial sessions as well. Please just copy and save these files to your disks and then open them in Acrobat Reader. You do not need to read them all.   Nobody does. However, take a look at the "Getting Started Manual". You may also want to look at the "Example Manuals" at some point. You will get an idea of the scope of ABAQUS, and may even get ideas for your project.

➢ **Starting ABAQUS/CAE**

Windows system: start→All programs→ABAQUS 6.5.1→ABAQUS CAE
Unix system: type abaqus cae.

➢ **Steps in running ABAQUS**

Create an input file. ABAQUS works by reading and responding to a set of commands (called KEYWORDS) in an input file. The keywords contain the information to define the mesh, the properties of the material, the boundary conditions and to control output from the program. Now ABAQUS CAE can automatically generate this input file for you.

Post processing. There are two ways to look at the results of an ABAQUS simulation. You can ask the program to print results to a file, which you can look at with a text editor. This is painful. Alternatively, you can use a program called ABAQUS/Post, which can be used to plot various quantities that may be of interest.

Please have a look at the examples in the File *1-2 Learning ABAQUS* and *1-3 CAE example* you will have a much clearer idea to start with ABAQUS.