

## ABAQUS Tutorial

TF: Yuhang Hu

Email: yuhanghu@fas.harvard.edu    Office: Pierce Hall 404

### Schedule

Date	Action	Notes
10/29/2008 (Wednesday)	Reading materials and Assignment #1 are posted on  imechanica	
10/31/2008 (Friday)  4:00 ~5:00 pm	Tutorial Session 1  @ Maxwell Dworkin Basement  Computer Lab (B121)	Introduction to ABAQUS,  using ABAQUS CAE.  An example
11/07/2008 (Friday)  4:00 ~5:00 pm	Tutorial Session 2  @ Maxwell Dworkin Basement  Computer Lab (B121)	ABAQUS Command  To help with computer assignment #1
11/10/2008 (Monday)	<b>Assignment #1 Due</b>	
11/14/2008 (Friday)  4:00 ~5:00 pm	Office hour @ Maxwell Dworkin Basement Computer Lab (B121)	To help with computer assignment problems
11/17/2008 (Monday)	<b>Assignment #2 Due</b>	

### ➤ Tutorial materials

Please download the following materials from iMechanica or our course website:

1 *ABAQUS tutorial for ES 240.pdf* (the file you are currently looking at)

2 *Learning ABAQUS.pdf*

3 *CAE Example.pdf*

➤ **Access to ABAQUS**

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**

ABAQUS 6.6 documentation is online. The website is <http://www.hlrs.de/organization/aw/services/struct/app/abaqus/Documentation/docs/v6.6/index.html>.

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.