ABAQUS Tutorial

TF: Nanshu Lu

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

Schedule

Date	Action	Notes
10/30/2006 (Monday)	Example & Assignment#1 available on line	Plate with circular hole
11/02/2006 (Thursday) 5pm~6pm	ABAQUS Tutorial @ Maxwell Dworkin Basement Computer Lab (B121)	ABAQUS Command & ABAQUS CAE
11/08/2006 (Wednesday)	Assignment#1 Due & #2 starts	Natural frequency problem
11/17/2006 (Friday)	Assignment#2 Due in class	

➢ **Lab sessions**

11/02/2006 (Thursday) 5pm~6pm.   Instructional Lab: Maxwell Dworkin B121.

I will do demonstrations to get you started with ABAQUS and willing to answer questions about the computer assignment.

➢ **Access to ABAQUS**

You can install ABAQUS v6.5.1 or v6.6 on your own computer through the DEAS IT office

Phone 617-495-9050
Maxwell Dworkin Rooms G107-G111
33 Oxford Street
Cambridge, MA 02138

I'll also pass around the ABAQUS v6.6 installment CD in class this week.   Besides, ABAQUS 6.5-1 is installed on the computers in Maxwell Dworkin B121.   Your ID card has been activated so that you can gain access to MD B121 at off hours.

➢ **ABAQUS users' manuals**

I'll pass around the ABAQUS documentation CD this week.   Please save these files to your hard disk and then open them. 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 example in the File *1-2 Learning ABAQUS* and you will have a much clearer idea to start with ABAQUS.