Tutorial #0: Welcome to the Elaine Cluster!

The purpose of this brief tutorial is threefold:
1) To teach you the LINUX commands required for tidy file management.
2) To make sure that you can run ANSYS from your leland account.
3) To provide an introduction to commercial FEA codes.

Introduction to LINUX

While most FEA packages are now available in PC versions, LINUX versions are still abundant, and that's what we are using here. We have ANSYS loaded on the Elaine computers at the Elaine Cluster in the Terman Engineering Building, room 104 (http://www-tecc.stanford.edu/).

Sit down at one of these computers, log in using your SUNet ID and password, and get acquainted with LINUX.

Go to **Applications>Accessories>Terminal**

Here's a list of useful LINUX commands for your reference **(try em!):**

ls	List the files in the current directory.
ls –ltr	List the files showing the file permissions, etc. It also shows the files by date, which is VERY useful.
who	Who else is connected to your computer? They may be slowing it down.
mkdir *directory*	Creates a directory.
cd *directory*	Change working directory.
cd **..**	Move back one directory.
pwd	Tells what directory you're in.
xedit *filename*	Xedit is a simple text editor.
rm *filename*	Removes a file.
rmdir *directory*	Removes a directory. rm –r also works.
more *filename*	Lists a file "read only". Use space bar to scroll through it.
ps -efl	Lists the processes that are currently running on your machine.
kill –9 *job#*	Kills a job that you have running. Get the process # from ps.
cp *filename1 filename2*	To copy filename1 to filename2.
mv *filename1 filename2*	To move filename1 to filename2 (rename).
lelandquota	Find out how much disk space and print quota you have.

Introduction to the FEA World

Now you're ready to run ANSYS on the LINUX machines. But first let's discuss some of the terminology used in the FEA world.

There are basically three steps that comprise a finite element analysis:

1. **Pre-Processing**: Create the model, and apply the loads and constraints (boundary conditions). This is the bulk of the work.
2. **Solution**: The FEA code numerically solves the problem that you have posed.
3. **Post-Processing**: This is where you view and analyze the data in the form of numbers, graphs, color contours, animations, etc.

There are many commercial FEA codes on the market and they don't, in general, do all of the steps listed above. To give you a taste for what's out there here's a non-exhaustive list of some FEA codes and related pre/post-processors that are on the market:

Pre/Post-Processors	**Solvers**
ABAQUS/CAE	ABAQUS
ALGOR	ALGOR
ANSYS	**ANSYS**
LS-PrePost	LS-DYNA
Femap	Femap
HyperMesh	General purpose
I-DEAS	I-DEAS
Mechanica (Pro-E)	Mechanica
Mentat	MARC
PATRAN	NASTRAN

Some software packages are termed "general purpose" and are compatible with many solvers. These can be very useful because you only have to learn to model in one software package which allows you to export your model into a variety of FEA solvers. The analysis capabilities vary greatly from code to code, and some codes are designed to solve specific problems; LS-DYNA, for example, is mainly used for high-speed nonlinear simulations, such as car crashes.

In ME309 we use ANSYS, which is a popular FEA code that has many analysis capabilities: Linear, nonlinear, buckling, contact, electromagnetics, etc. We will be using ANSYS to solve linear-elastic (static and dynamic) problems.

Now, let's copy an ANSYS model that has already been created (pre-processed), and solved (linear-static solution), and look at the results:

1. File Management

 Make a new directory and copy all of the ANSYS files. Linux is case sensitive, and it is very important to include all characters and spaces exactly as shown.

```
pod7:/> mkdir EXAMPLE
pod7:/> cp ~bhargav/public/example.* ~/EXAMPLE
pod7:/> cd EXAMPLE
pod7:/> EXAMPLE > ls –ltr
pod7:/> EXAMPLE > gunzip example.*
pod7:/> EXAMPLE > ls –ltr
```

2. Running ANSYS

 Start ANSYS from that directory:

```
pod7:/EXAMPLE >launcher
```

 Run ANSYS interactively:

- *In the Simulation Evironment dropdown choose ANSYS (second on the list)*
- *Change the "working directory" to EXAMPLE directory by browsing and selecting it.*
- *Change the "jobname" from file to example, and click "run".*

 Load the ANSYS database that you have in your directory

Utility Menu >> File >> Resume from
(The Utility Menu is at the top of the screen)

- *Pick example.db from EXAMPLE directory*

 If you don't see anything in the graphics window, turn on the elements:

Utility Menu >> Plot >> Elements

3. Play with the model

 Adjust the view:

Utility Menu >> PlotCtrls >> Pan, Zoom, Rotate

- *Activate Dynamic Mode. Now you can hold the right mouse button down in the graphics window and adjust the view. Zoom and fit are useful too. Select Bot. last.*
- *You can also enter the dynamic mode by holding down the control key while using the mouse. Find out what the different buttons on the mouse do to the view.*

4. Read in the solution set and post-process.

 Choose the results (you only need to do this when the results file has a different name than your "Initial jobname" that you chose)

ANSYS Main Menu >> General Postproc >> Data & File Opts..

- *Go to [FILE] and choose example.rst from your EXAMPLE directory*
- *click ok*

ANSYS Main Menu >> General Postproc >> Read Results >> First Set

- *Now you have the results loaded into ANSYS. There are many ways to view them.*

 Plot the results:

General Postproc >> Plot Results >> Contour Plot >> Nodal Solution

- *Pick Xcomponent of Stress*
- *Rotate the model around and zoom in to the high-stress areas.*

 Animate the results:

Utility Menu >> PlotCtrls >> Animate >> Deformed Shape

 Enough for now. Let's get out of ANSYS, and see what it leaves in your directory.

Utility Menu >> File >> Exit >> Save Everything

- *Close the ansys product launcher too.*

5. File Management – Again!

Look in your new directory and delete everything

pod7:/EXAMPLE> ls (here are all the files ANSYS has generated)
pod7:/EXAMPLE> rm example.* (this deletes all files at once – very dangerous, you should probably delete the files individually)
pod7:/EXAMPLE> ls

Now delete the directory:

pod7:/EXAMPLE> cd **..** (go back one directory)
pod7:/> rm -r EXAMPLE

NOTE: ANSYS generates a lot of files that take up space. It is a good idea to get into the habit of deleting the files you don't need. It is important to keep your file management tidy since disk space is limited and FEA can use it up quickly. Later in the class it will become more apparent which files should be kept. It is recommended that you create a permanent new directory (in your home directory) in which to save all your files for this class. Perhaps it would be titled "ME309".

To run any other software you must first open a new terminal window. To do so

Go to File>Open Terminal on the top left of the terminal window.

Once you have a terminal window open, you only need to type the name of the software at the command prompt. For example, try;

1. netscape: Open a new Netscape browser to surf the web,
2. xedit: Open a basic text editor,
3. Matlab: Open a new Matlab session,
4. Mathematica: Open a new Mathematica session,
5. Pine: Open an archaic email interface.

To quit the Linux session close the Terminal window .