

## Running ANSYS in batch mode

1. Copy your input file (*filename.txt*) into the “working directory” .
2. Run the following command line:  
Preferred Command Line:  
`:-> xansys -b -p projectname -i inputfilename.txt -o outputfilename.txt`  
  
Secondary Command Line:  
`:-> xansys -b <filename.txt> & filenameoutput.txt`

## Copying files from one folder to another

1. Open the folder containing the file to transfer.
2. Run the following command line:  
`:-> cp -r filename.* /var/tmp/states [enter the path from the root directory]`

## CADE Lab UNIX machine addresses

The best way to connect to a lab machine is to use our load balancing name server, which should resolve to the machine with the lowest load and fewest users. To take advantage of this, simply connect to:

best-sun.eng.utah.edu  
for a Sun (Solaris) machine or  
best-linux.eng.utah.edu  
for a linux machine.

NOTE: If you use these best-sun or best-linux, you will be connecting to a different IP address almost every time you connect to that host. This will likely cause problems (especially with X11 forwarding) because you will get a different host public key.

If you really want to connect to a specific machine in the lab, the hostnames follow the pattern:

labM-N.eng.utah.edu

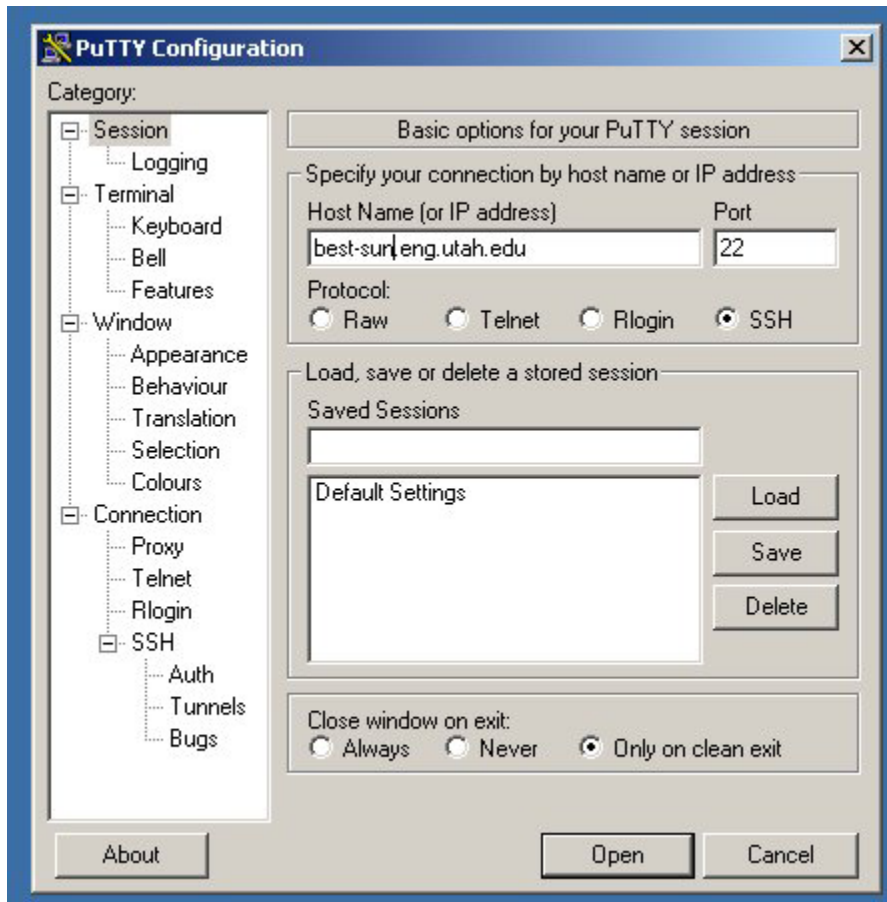
Where M is the lab number (1-4) and N is the machine number (1-48 for lab1, 1-44 for lab2, 1-25 for lab3, and 1-15 for lab4).

## Running ANSYS in batch mode remotely from a PC by using PuTTY

There are several SSH client available for MicroSoft Windows. I suggest [PuTTY](http://www.chiark.greenend.org.uk/~sgtatham/putty/download.html) from <http://www.chiark.greenend.org.uk/~sgtatham/putty/download.html>. It's simple to use and simple to download. Just download the .exe file to your preferred directory. I suggest downloading it to the Desktop.

(1)Once you are connected to the Internet, you then should start your SSH client. If you are using PuTTY, double-click the putty icon.

(2)Enter the name of the remote machine that you wish to connect to in the 'hostname' field, in our case you can chose a lab certain lab machine or just use best-sun.eng.utah.edu if you are uncertain. Make certain before you hit the "open" button (or "connect" in some cases), that you have clicked the radio button that says 'ssh'.



(3) Then enter your login name and password. You should be all set to go! If you want to run anything outside of the ssh window (i.e., xeyes, [abiword](#), etc...) You'll need to run an X server.



(4) At the command prompt, follow the instruction on the first page to run ANSYS in batch mode. Note that anything from GUI (i.e., displays, plots, etc.) will not be available in this mode. In order to run ANSYS with GUI remotely, you will need to have XWindows on your system.

*More information on remote access can be found at <http://www.cade.utah.edu>*