Exercise 0: „How to start and use ANSYS“

Learn…

… how to start a remote session from your local Linux machine
… how to start the FE program ANSYS on a remote host
… about the classical ANSYS GUI and how to use it

A. Getting started

A.1 Login to your local Linux PC

Enter your account name and password to login to the local Linux PC.

A.2 Create a new directory

Start a file manager (e.g. "Konqueror") from the desktop main menu (Fig 1) with a mouse click on the Konqueror button and create a new directory with the name **"`FE-Lab`"** inside your home directory. Alternatively, enter "**`mkdir ./FE-Lab`**" in a terminal window.

Hint: For all PCs operated by the computer center (KIZ) the home file system is identical, it therefore doesn't matter which machine you use. Your home directory is always located at **`/users/student1/<your login>`** (alias: "**`~`**"; issue "**`echo ~`**" at the command line to print the path to your home directory).

A.3 Open a Terminal

Start a terminal (e.g. "Konsole") from the desktop main menu:

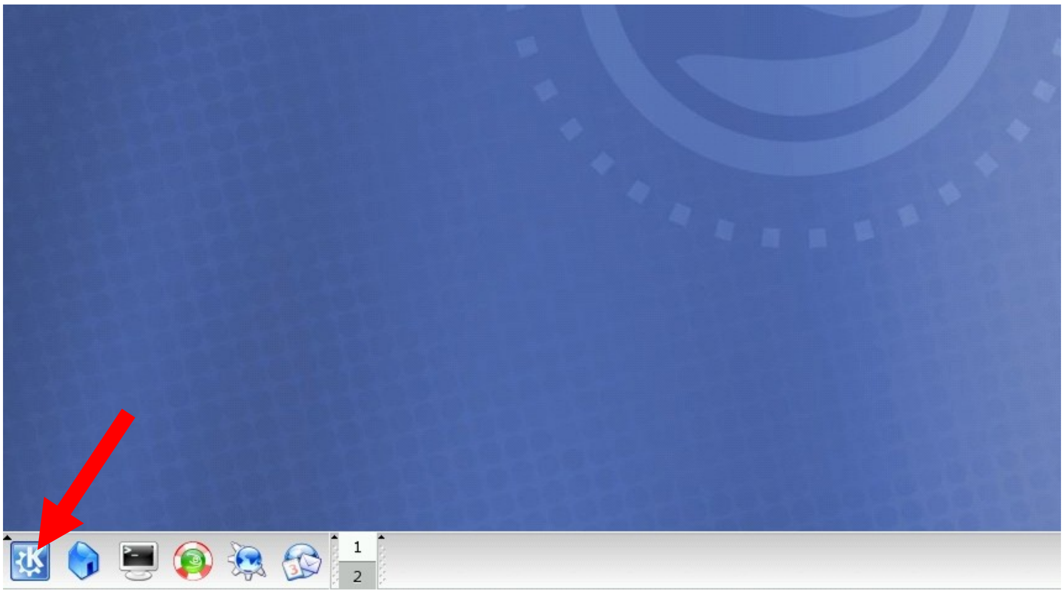

Figure 1: Open a terminal and a file manager from a desktop menu.

A.4 Start a remote session

Start a remote session (SSH = secure shell) by entering the command

ssh -X <yourname>@zeus.rz.uni-ulm.de

or

ssh -X <yourname>@andromeda.rz.uni-ulm.de

in the terminal. Please use your account name as "**<yourname>**" and enter your password when being asked. The password should be the same as the one you used to login locally to your Linux machine.

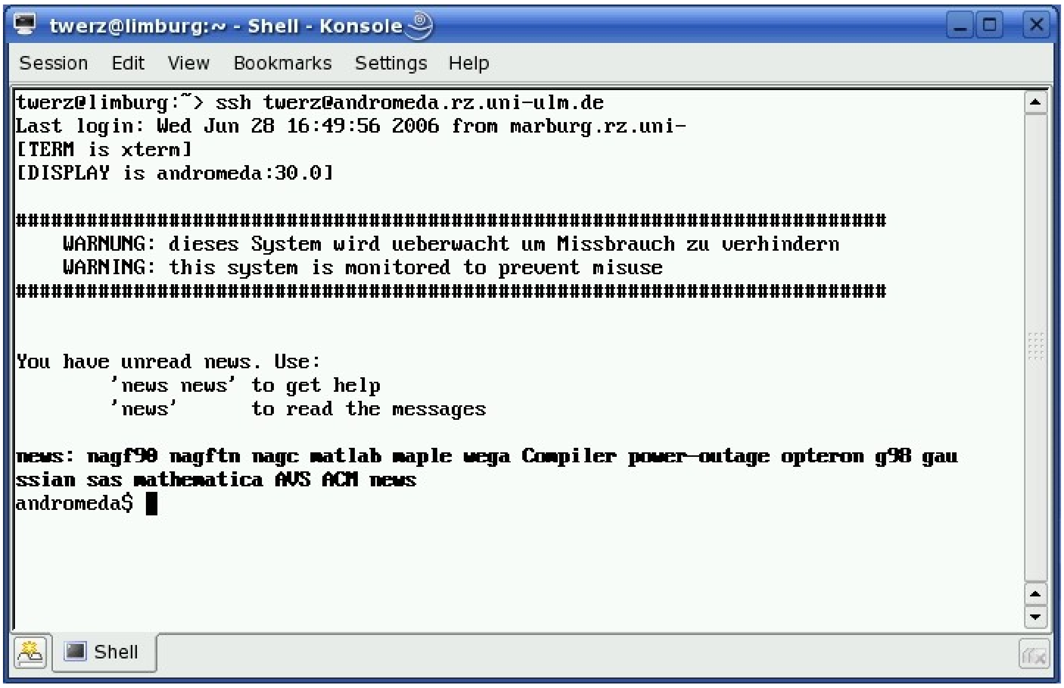

Figure 2: Remote login via ssh command.

For testing purposes, try to run a new **xterm** session on the remote host by issuing the command

xterm &

(*Note:* The ampersand detaches the new xterm process from the SSH console session so you can enter further commands without needing to close the xterm window.)
This is also a first check whether the display settings are appropriate to let the host computer (hera/zeus/andromeda) export whole X11 windows (and not only text output) to the local screen.

A.5 Start the FE software ANSYS

Before being able to use a software package on the computer centers' Unix machines you have to choose an appropriate "**module**" command to set necessary environment variables. A list of all available software is displayed if you simply enter the command “**`module avail`**” without any further parameters.

To start the launcher of the FE software ANSYS Version 12.0 (www.ansys.com) first enter

```
module load cae/ansys/13.0
```

and then

```
launcher130 &
```

either into the xterm or directly into your SSH terminal.

Important: Please make sure to choose the right license type

“ANSYS Academic Teaching Advanced”

Do **not** choose “ANSYS Academic Research”!

Enter your newly created directory as the **working directory**

```
/users/student1/<yourname>/FE-Lab
```

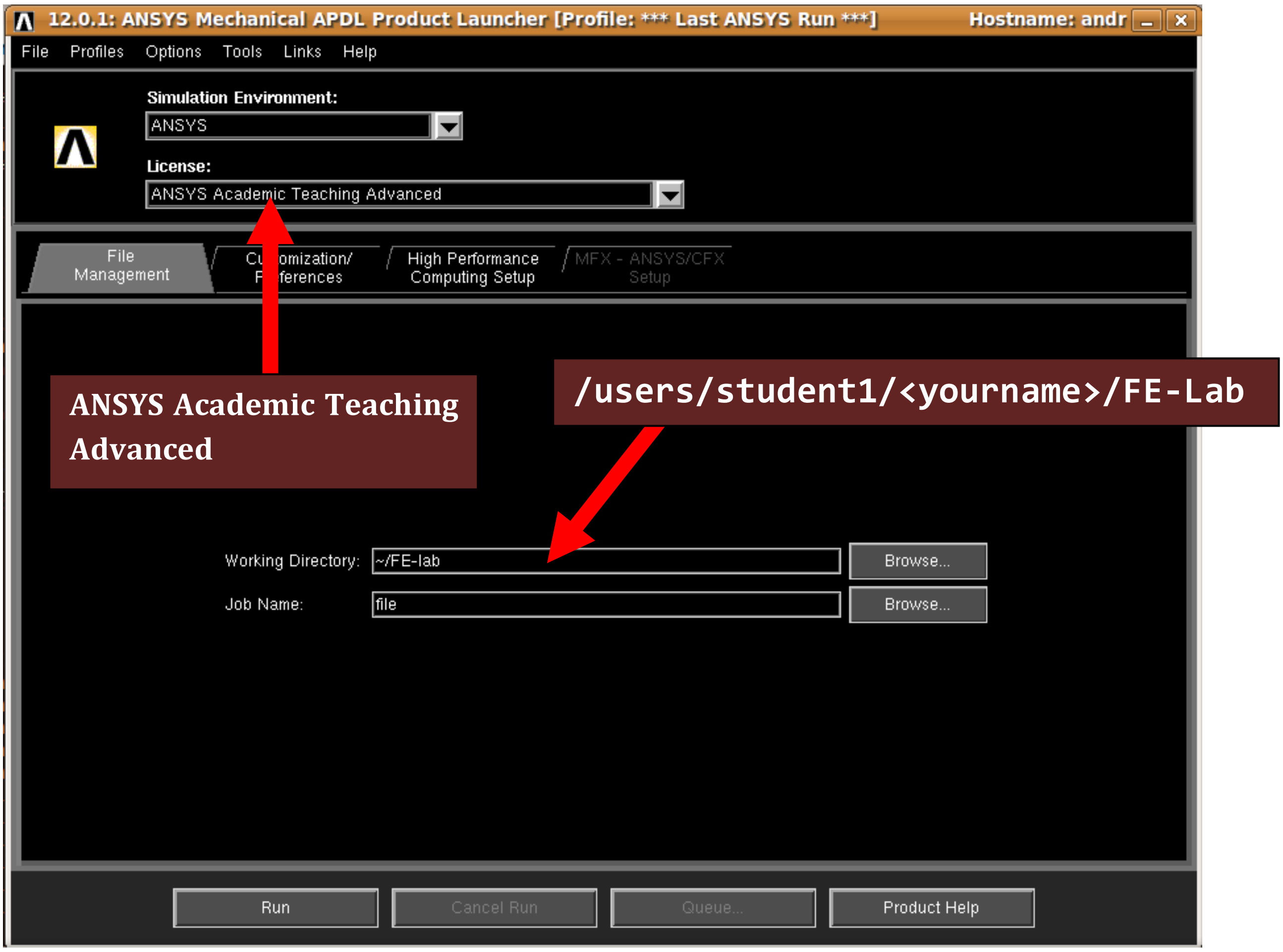

Figure 3: Choose product “Teaching Advanced” and set the working directory.

Press the **[Run]** button to finally start ANSYS.

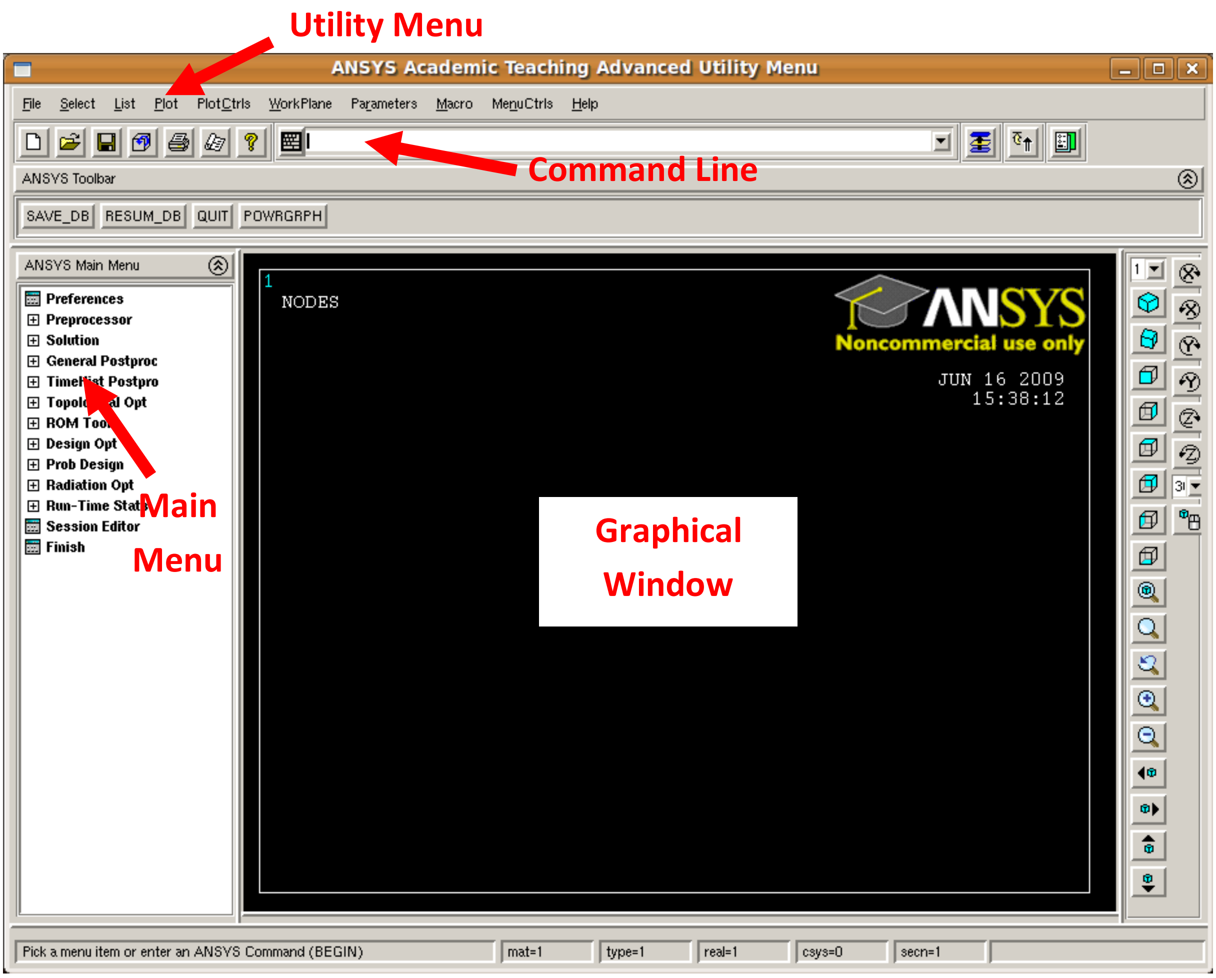

Figure 4: Elements of the classical ANSYS GUI.