Steps to Using LTspice

1. Download LTspice IV from the below links:
Windows version of LTspice: LTspiceIV.exe
Mac os x version of LTspice: LTspiceIV.dmg
Mac os x short cuts can be found in LTspiceShortcutsForMacOSX.pdf

2. Download and open the following simple circuit file: rc_highpass.asc
Click on the little running fellow to run the simulation.

3. Try changing the simulation time, frequency, resistor and capacitor values, etc. Remember to re-run the simulation every time you make a change.

4. Download the following symbol and model files: ALD1101.asy, ALD1102.asy, IRF510.asy, IRF9510.asy, SSCSMag.lib
Put the files into a directory called "circuit_sim" (or whatever you choose to name it).

5. Download and open sample.asc.
Right click on the .lib statement in the schematic to edit the location of the SPICE model library to point at the "circuit_sim" directory you created.
Try running a simulation and plot an Id versus Vds curve of the transistor in the schematic to make sure that both the symbol and model libraries have been properly installed.

6. Right click on the .dc statement to change the voltage source to sweep to VG. Run a simulation again and try plotting an Id versus Vgs curve.

Datasheets of the transistor's whose basic LTSpice models are provided as:

ALD1101.pdf, ALD1102.pdf, IRF510.pdf, IRF9510.pdf

Quick Tips on LTspice

- Right click (not double click) on an item to edit its property.
- You can choose the type of simulation (transcient, ac analysis, dc sweep, etc) under "simulate" - "edit simulation cmd"
- After a simulation, click on a net or device terminal of interest to plot the voltage or current, respectively.
- A single voltage source can be used as different kinds of voltage sources by changing its property.
- Try to label important nets such as input and output
- You can view the spice netlist under "view" - "spice netlist".

Useful shortcuts:

F3: draw a line
F2: add a component
F7: move
F8: drag
F9: undo
F4: label net
Ctrl+E: flip sideways
Ctrl+R: rotate
ESC: cancel
Ctrl+C: copy
Ctrl+X: cut

There is a LTspice tutorial website located at http://denethor.wlu.ca/ltspice/ and http://ltwiki.org/index.php5?title=SPICE_and_LTspice_Courseware_and_Tutorials and also some LTspice tutorials available here:
scad3.pdf and LTSpiceShortGuide.pdf (it used to be called switcherCAD)

Finally, a generic spice manual can be found at
http://www.eecg.utoronto.ca/~ali/spice/spice_manual.html