LTSpice Quickstart Guide

Download LTSpice:

The 6.012 staff strongly encourages you to use LTSpice for all simulations you will complete in this class.

Linear Technology Corporation provides a widely used, free graphical SPICE simulator known as LTSpice. Download and install LTSpice IV from Stellar or the Linear Technology website, http://www.linear.com. LTSpice is only currently available for Windows. If you do not own a Windows machine, you may want to consider downloading the 30 day free trial of Crossover, a Windows compatibility layer for Mac (Intel processor) or Linux, at http://www.codeweavers.com/products/ . Alternatively, there are Windows clusters in:

37-312 (faster, newer machines)
Student Center

If you install LTSpice in a Windows cluster, make sure to install it in your winathena directory, not the computer's local directory.

Quick Tips:

- In schematic editing, you can place any device from the current library, or any that you define yourself right into the schematic. LTSpice will automatically generate a netlist based on how you wire your components.

- To exit any placement mode hit ESC.

- Once you wire your devices and provide the necessary sources, visit View → SPICE netlist to see the netlist LTSPICE has generated. This will be useful should you need to debug your circuit.

- You can create a symbol of your schematic to instantiate in other schematics. When you have a schematic open, visit Hierarchy → Open this Sheet's Symbol. This allows you to create hierarchical schematics that are much easier to read and manipulate. For example, you can create an inverter schematic and generate a symbol for it. In the top level schematic you can instantiate that inverter several times such that when you manipulate the inverter schematic, it will change the schematic for all inverters you instantiate at the top level.

- LTSpice (and most commonly used simulators) is generally much faster and friendlier to use if you get to know keyboard shortcuts well. Visit Tools → Control Panel → Drafting Options → Hot Keys to either get to know default keyboard shortcuts or set up a set of shortcuts that are intuitive to you.

- Any SPICE directive you place on the schematic will be incorporated into the netlist.

- LTSpice documentation is available in its help menu F1.

Device Models:

For your convenience, the staff has created subcircuits to model the four terminal MOSFET. You must provide the model with the gate length (lg) and gate width (wg). Backgates should be explicitly tied to GND for NFETs, and V_{DD} for PFETs.

When instantiating a MOSFET, identify the MOSFET as a subcircuit in the component attribute editor by using the 'X' prefix, and type the subcircuit name into the first Value line. Parameter values for lg and wg must be entered in the Value2 line.

Save the following as a .sub file, or download the model file directly from Stellar, and call it in your schematic with the SPICE directive:
.inc [filename].sub

```
.subckt NFET D G S GND
.model NCH NMOS LEVEL=1 VTO=0.5 TOX=1.5E-8 U0=220
+ LAMBDA='(1.5u/lg)*1.0E-1' CJ=1.0E-4
+ CJSW=5.0E-10 PB=0.95 GAMMA=0.6
M1 D G S GND NCH l='lg' w='wg' ps='12u+wg'
+ pd='12u+wg' as='6u*wg' ad='6u*wg'
.ends NFET

.subckt PFET D G S VDD
.model PCH PMOS LEVEL=1 VTO=-0.5 TOX=1.5E-8 U0=110
+ LAMBDA='(1.5u/lg)*1.0E-1' CJ=3.0E-4
+ CJSW=3.5E-10 PB=0.9 GAMMA=0.6
M1 D G S VDD PCH l='lg' w='wg' ps='12u+wg'
+ pd='12u+wg' as='6u*wg' ad='6u*wg'
.ends PFET
```

6.012 Microelectronic Devices and Circuits
Spring 2009