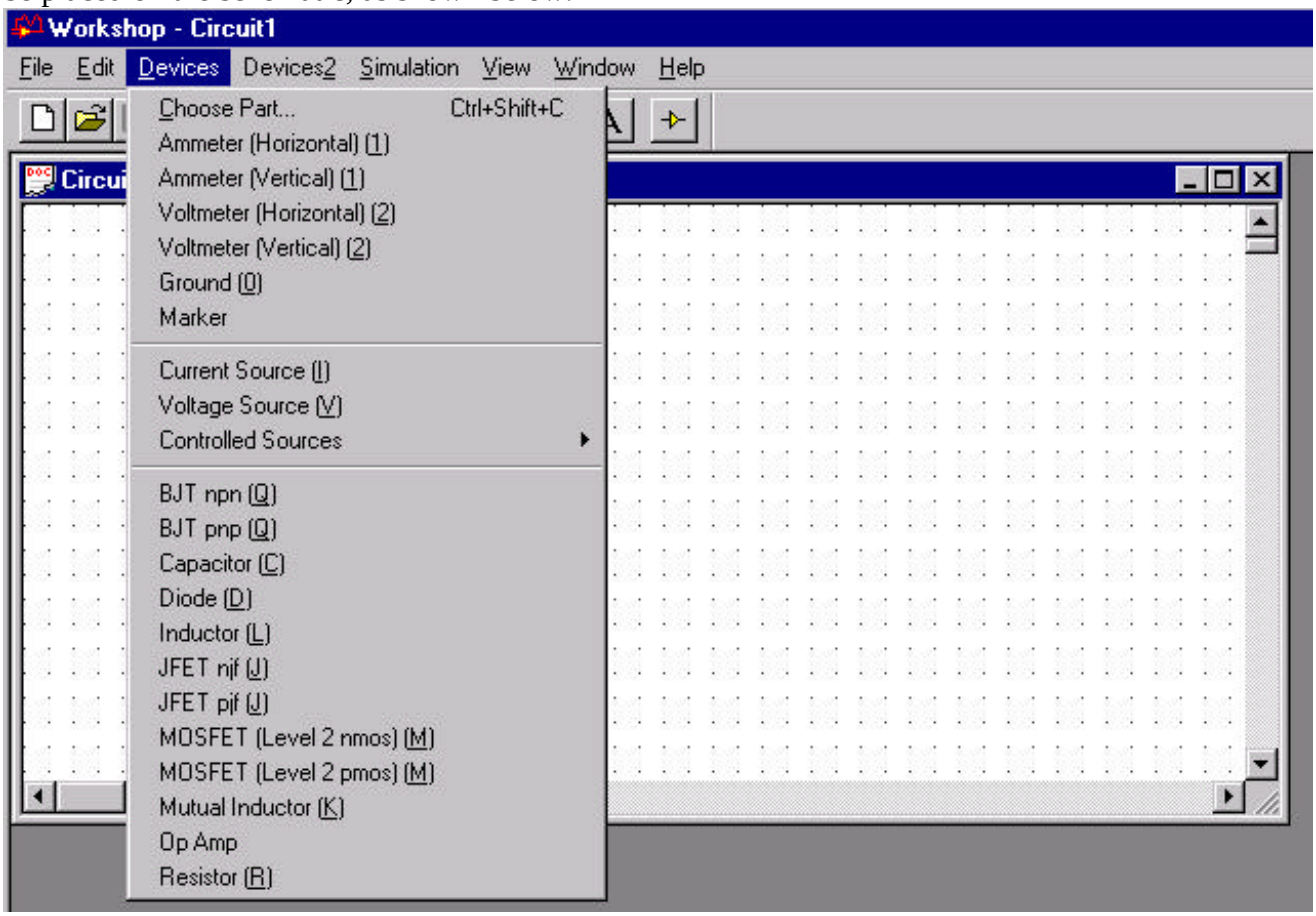
Using B2SPICE From the Microlabs

You will need an SNDS account to log onto the computers in the Civil Engineering Microlab, which is located in Room 228 of the Civil Engineering building (see http://www.uky.edu/SCS/ for locations and hours of other microlabs around the campus). If you do not have an SNDS account, you can create on from the WWW at http://www.uky.edu/userids from 7:00 am to 7:00pm. If you do not have access to the WWW, you should stop by Room 113 in McVey Hall to get an account create.

Once logged on to the SNDS account, the Windows operating system will startup. Go to the Start Menu by clicking on the "Start" button on the system taskbar. Find the folder B^2 Spice and direct the mouse over it. A sub-taskbar will appear. Move the mouse to highlight "Workshop" and click the mouse on it.

Workshop is used to edit the schematic and to simulate the circuit. The circuit is edited by dragging and dropping circuit elements onto the workbench, or schematic. If you click on the "Devices" menu on the menu bar, you will be given a list of devices that can be placed on the schematic, as shown below.



The first section of devices are voltmeters, ammeters, and ground. Voltmeters and ammeters are used to control output measurements, as well as voltages or currents for controlled sources. The second section of devices are sources. First independent current

and voltage sources, then controlled sources. For the controlled sources, there are current controlled, and voltage controlled current and voltage sources. The controlled sources will be referenced to a branch voltage or current provided by a voltmeter or ammeter, respectively, that you place on the test bench.

The final set of devices are active devices (transistors and diodes), and lumped circuit elements such as a capacitor, inductor, or resistor. An op-amp model is also included. Note that the op-amp is non-ideal, and requires DC biasing. This can be done using DC voltage sources (note that the ouput amplitude of the opamp will saturate above the DC bias).

To place an item on the schematic, select one of the circuit elements with the mouse from the "Device" menu. Then when the mouse is in the schematic window, the device will appear in a transparent manner. Clicking the mouse button at a location in the schematic will place the element on the schematic. The element can then be rotated by keying in Ctrl-r (hitting the control button and the letter r simultaneously) while the device is still highlighted. The value of the circuit element can be changed by double clicking on the element. You can then edit the value. You can also abbreviate the value using standard notation to represent engineering powers of magnitude. For example, 1K could be entered for one thousand. Or 1 MEG could be entered for 1 million. You can also enter 1 M for 0.001 (or 10^-3), 1 U for 10^-6, 1 N for 10^-9, 1 P for 10^-12. Note that the device will be assigned the appropriate units. For example, the resistors value is assumed to have the units of ohms, a capacitor value assumes the units of Farads, etc.

Once the circuit elements are placed on the schematic the device can be wired. On the toolbar you will notice an icon of a wire - or line. Click on this icon. Then, with the mouse button select a terminal of a circuit element. Then, double click on the terminal of a circuit element which need be connected by a wire. Double clicking will draw the wire and terminate that wire. Next, try clicking on the terminal end of a device, then click somewhere in the schematic. This will draw a wire, which is still active. Then double click on the terminal of another device. This will draw another segment of the wire and then terminate that wire. If you forget to double click to terminate the wire, you can hit the right-mouse-button to terminate the wire. Continue to wire up the entire circuit in this manner.

Finally, a ground must be placed at the reference node of the circuit. The ground element is found in the first section of the "Device" menu.

Before the circuit is simulated, you must do two things: 1) edit your sources by double clicking on them. You can assign them to have a DC value, a transient property, or AC. Note that these values will only be sensitive to DC, transient, or AC simulations, respectively. 2) perform a simulation setup. This is done by choosing "Set Up Simulations…" under the simulation menu. This will bring up a window. You can then select the type of simulation that you wish to perform. In this course, we will be using .AC simulations in the first half, and .TRANS simulations (transient) in the second half of the semester. Select the type of simulation you need and then specify the appropriate

properties. For example, the .AC simulation allows you to specify a start frequency (in HZ), a stop frequency, and the number of frequency steps. The frequency stepping can be based on a linear sweep in frequency (LIN), or based on a log10 frequency scale, referred to as frequency decades (DEC). You can also specify the output to produce a graph of the magnitude and/or phase of the measured quantities as well as tabulating the values.

Once the simulation is setup, choose "run simulation" under the simulation menu. This will simulate the circuit. If any errors are detected, an error window will pop up. Else, if successfully simulated, a log of the simulation, and the graphed results and tabulated results will come up in separate windows.

Note that the graph can be edited. You can either choose "Edit Graph" under the main Edit menu to edit specific properties of the graph, or you can double click on the entities in the graph's legend to edit specific properties of traces in the graph.