UNIVERSITY OF CALIFORNIA
College of Engineering
Department of Electrical and Computer Engineering

EE 105 Lab #2: LTSpice

Name: ________________________ Partner: ____________________________

1. LTSpice Introduction

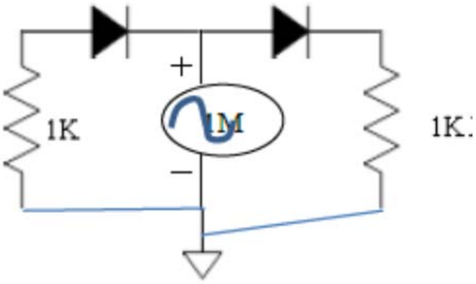
LTSpice is a graphical SPICE program, which allows you to simulate a large variety of circuits on your computer.

You can download LTSpice for free on your home computer:
http://www.linear.com/designtools/software/

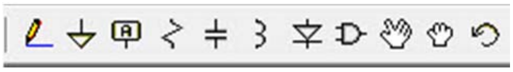
Online tutorial: http://denethor.wlu.ca/ltspice

2. Simulate Diode Rectifier

Last week we built a simple diode rectifier. You should remember the results. Now we will build the same circuit in LTSpice and see if it looks the same.



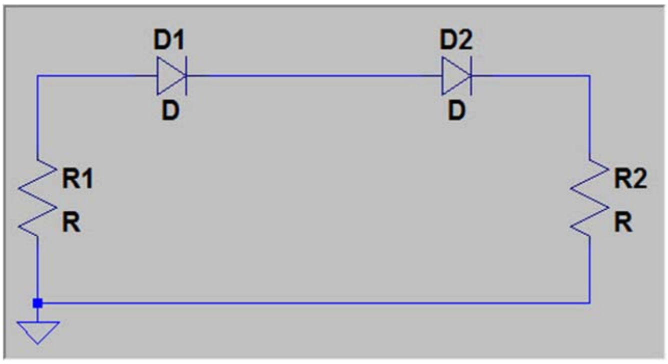
First we need to add the resistors and diodes, and wire them together and to ground.



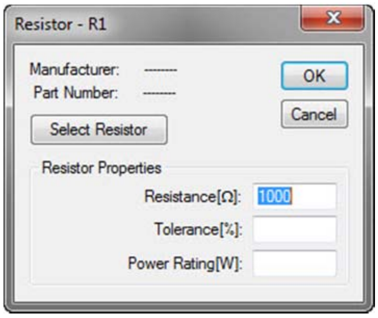
From Left to right on the LTSpice toolbar:

Add wire
Add ground
Add net name
Add resistor
Add capacitor
Add inductor
Add diode
Add component
Move
Drag
Undo

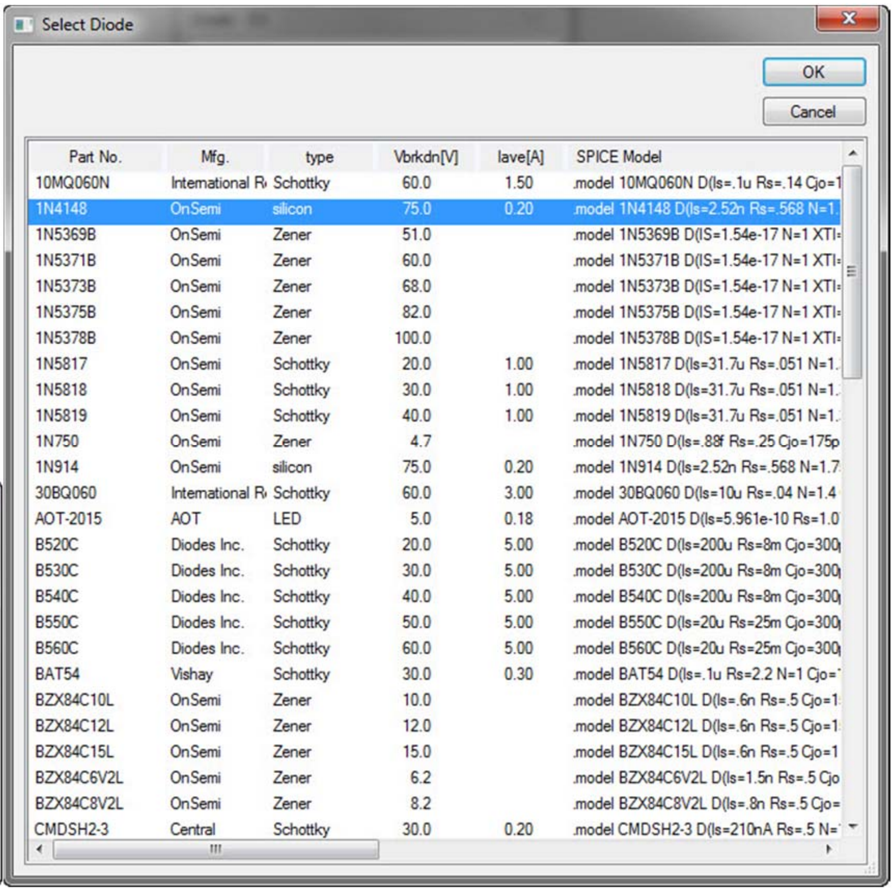
Click to select a component or wire and click again to place. Your circuit should now look like the following:

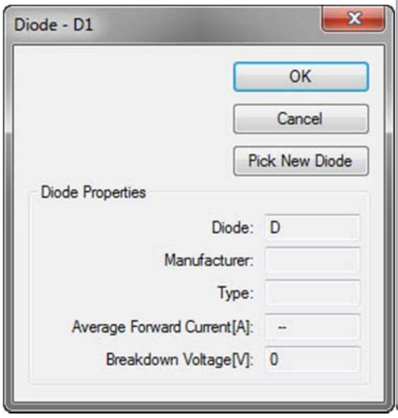


We now want to set the resistor values. Right click on the resistor and set the resistance to 1k

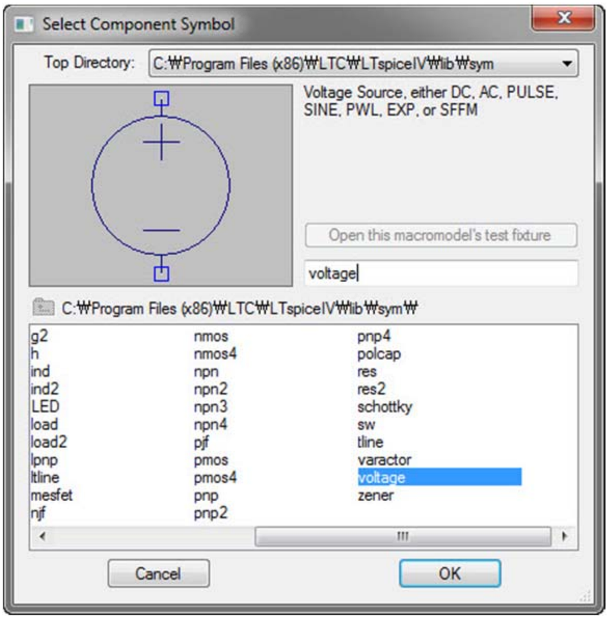


Now right click on the diodes, click 'Pick New Diode' and select the 1N4148. Note various differenct diode models can be selected. We are choosing a basic silicon diode.

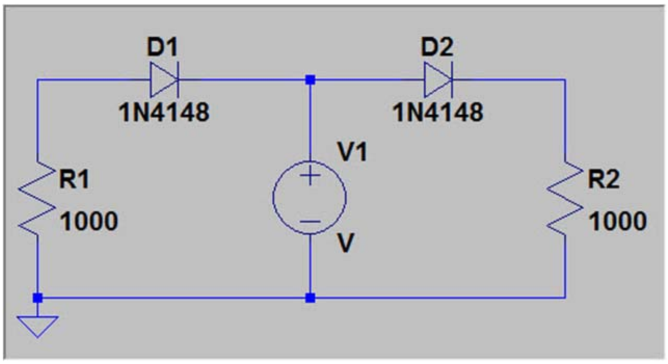




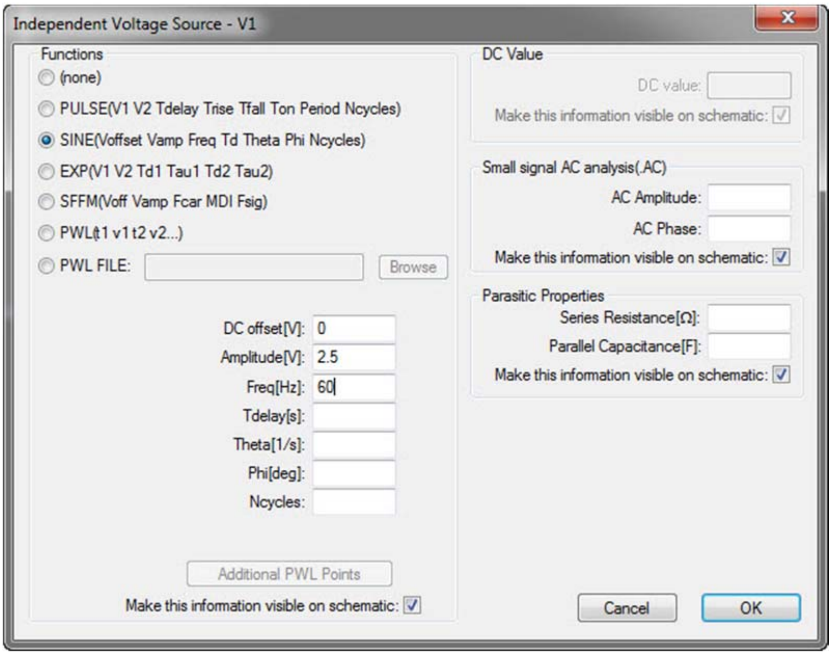
Add a voltage source (sinusoidal input). Click on add component on your toolbar:



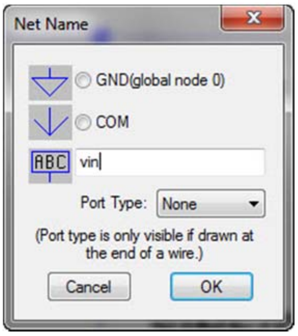
Place the voltage source, wire it, and your circuit should now look like the following:

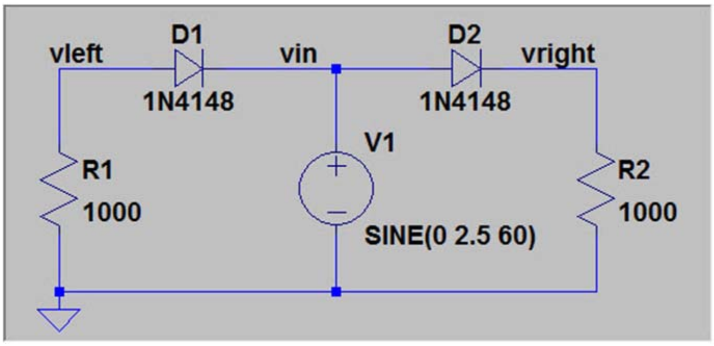


Right click on the voltage source and set it to a 60 Hz sine wave with 2.5 V amplitude.

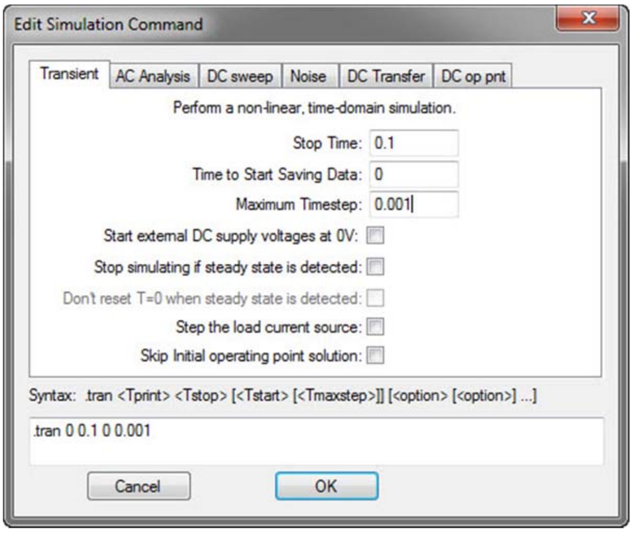


Add net names. This labels voltage nodes for later reference.

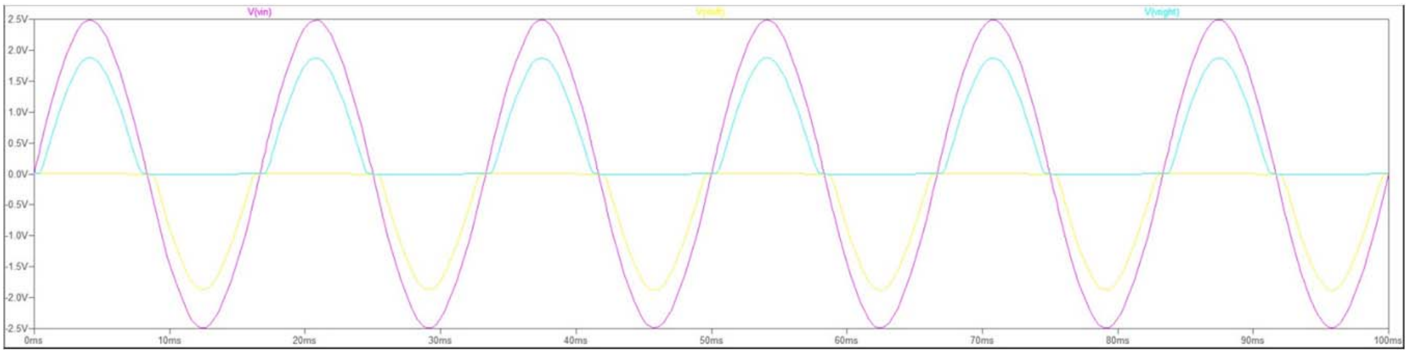




Now let's simulate the circuit using a transient analysis. Click the simulate button on the toolbar and select the Transient menu. Set the stop time to 0.1s, start time at 0s, and timestep to 0.001s.

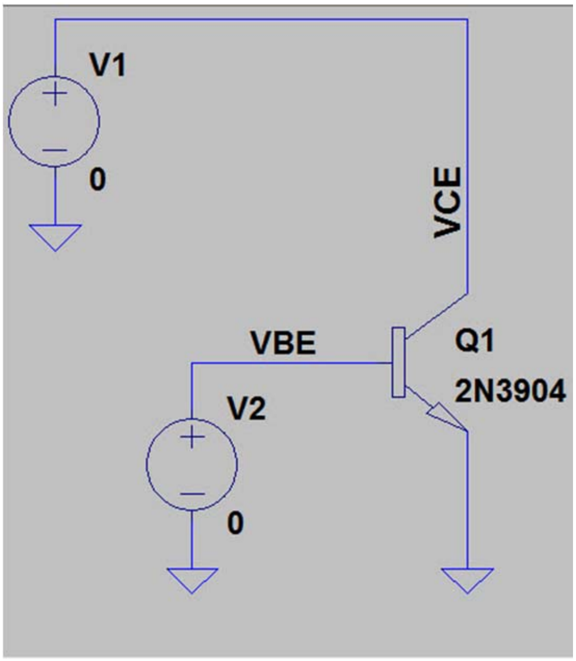


Click with the voltage probe on the wires where we want to measure the voltage and you should see the following output. Does this look familiar?

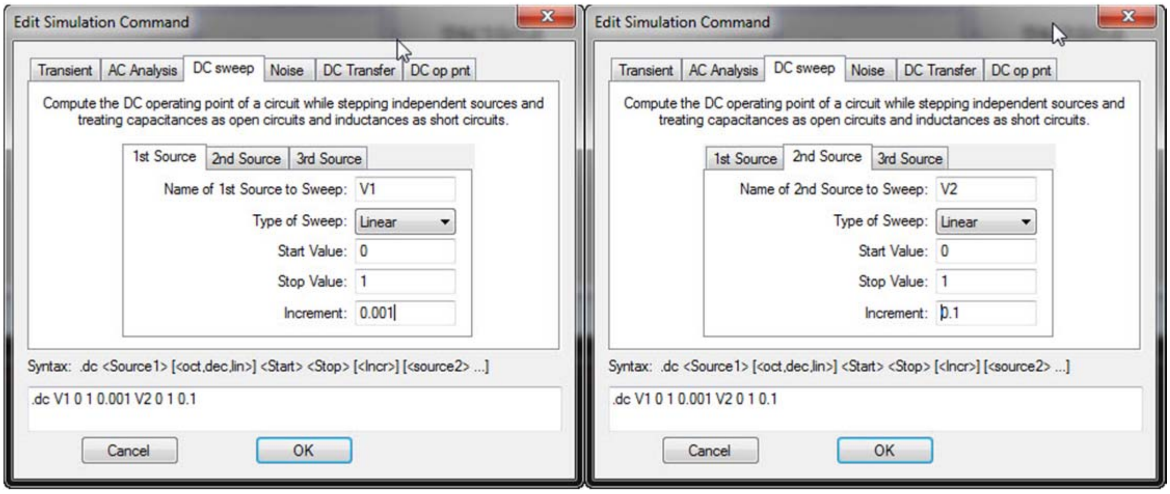


3. DC Bias of npn BJT

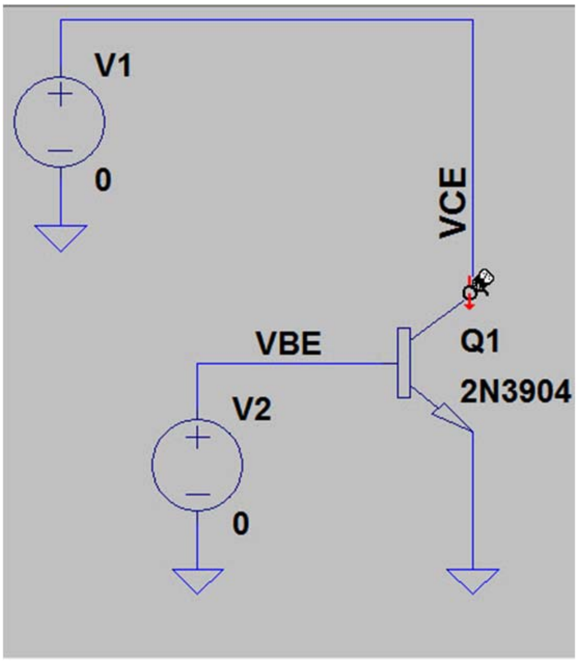
Here we will examine the BJT current with the following bias circuit. Select and place a two voltage sources, your grounds, and an npn transistor. Select the 2N3904 transistor just as you selected a specific diode in the previous example. Set the appropriate net names.



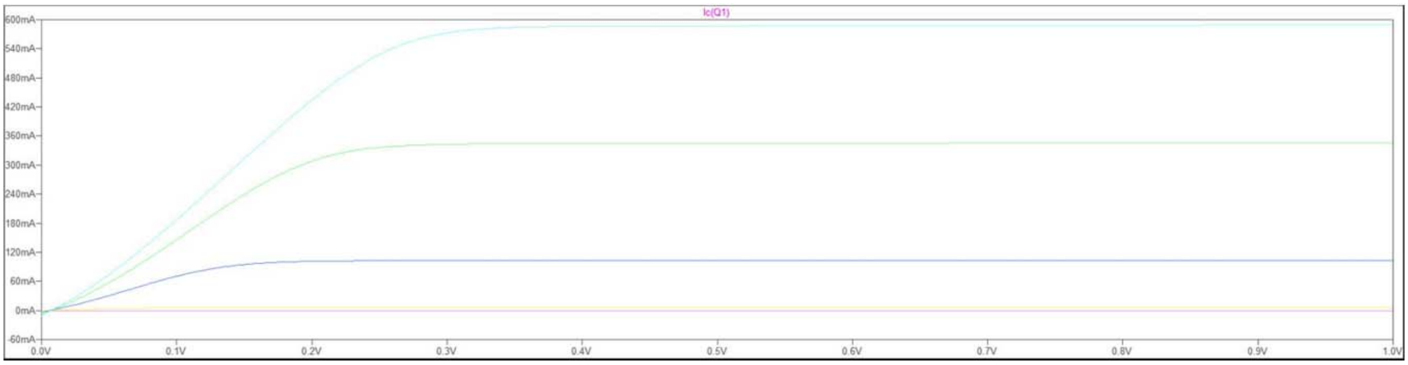
We will now do a DC sweep of the circuit. Click the simulate button, and select the DC sweep menu. Enter the appropriate values for the 1st source and 2nd source and shown below.



Now click on the wire of the BJT with the current probe. You should see the cursor change as you hover over the BJT component.

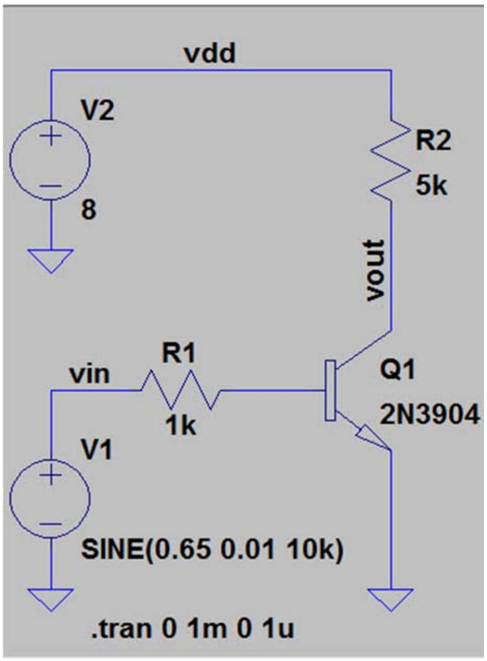


You should see a nice set of curves with the I_{CE} current plotted against V_{CE} with each curve having a different V_{BE} value. These curves will become very familiar when you begin to study the DC characteristics of the BJT.



4. Basic BJT amplifier

We will now simulate a very simple common-emitter amplifier by slightly altering the previous circuit. Using your knowledge from the previous examples, put together this circuit, with the appropriate 2N3904 npn transistor, 5k resistor, 8V voltage source at the top, and a 0.65V DC input voltage with a 10 mV 10kHz sine wave on top. Perform a transient analysis lasting about 1ms with 1µs time steps.



Plot V_{out} and V_{in} waveforms. The output waveform should be an amplified version of the input wave.

Then, build the circuit using **actual discrete components** (i.e. resistors, BJT's) on a breadboard, and see the output waveform. Does it match with the simulation result?

4-1. Extra credit: What happens if you change the DC voltage for either V_{in} or V_{dd}? What happens if you change the resistor value? What happens if you change the transistor model or change to a pnp transistor?