

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.

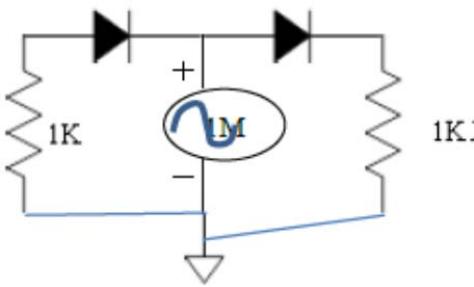
You can download LTSpice for free on your home computer:

<http://www.linear.com/designtools/software/>

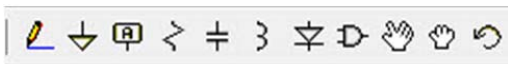
Online tutorial: <http://denethor.wlu.ca/ltspace>

**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

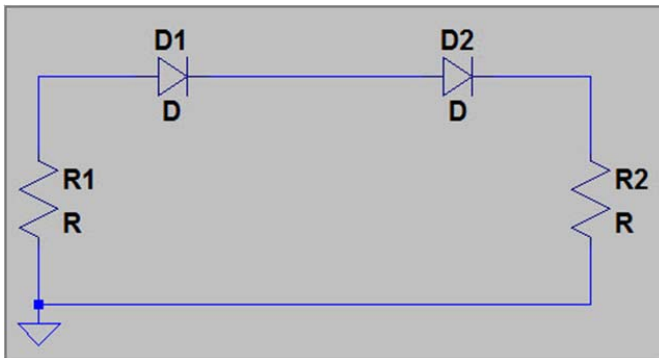
Add component

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

Resistor - R1

Manufacturer:

Part Number:

Select Resistor

Resistor Properties

Resistance[Ω]:

Tolerance[%]:

Power Rating[W]:

OK Cancel

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

Diode - D1

OK Cancel Pick New Diode

Diode Properties

Diode: D

Manufacturer:

Type:

Average Forward Current[A]:

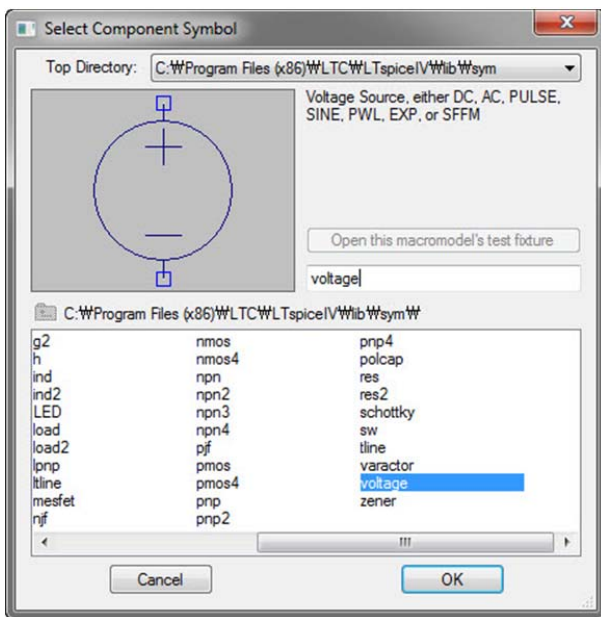
Breakdown Voltage[V]:

Select Diode

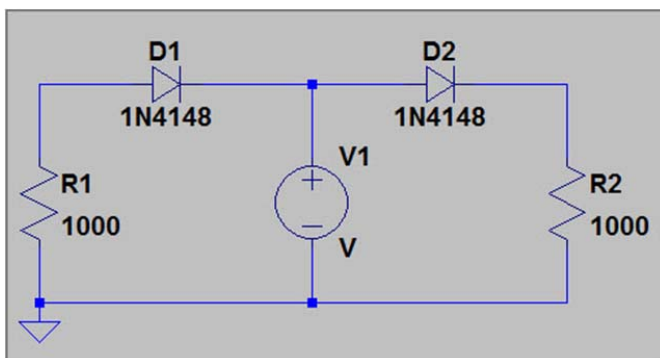
OK Cancel

Part No.	Mfg.	type	Vbrkdn[V]	Iave[A]	SPICE Model
10MQ060N	International R	Schottky	60.0	1.50	.model 10MQ060N D(Is=.1u Rs=.14 Cjo=1
1N4148	OnSemi	silicon	75.0	0.20	.model 1N4148 D(Is=2.52n Rs=.568 N=1
1N5369B	OnSemi	Zener	51.0		.model 1N5369B D(Is=1.54e-17 N=1 XTl=
1N5371B	OnSemi	Zener	60.0		.model 1N5371B D(Is=1.54e-17 N=1 XTl=
1N5373B	OnSemi	Zener	68.0		.model 1N5373B D(Is=1.54e-17 N=1 XTl=
1N5375B	OnSemi	Zener	82.0		.model 1N5375B D(Is=1.54e-17 N=1 XTl=
1N5378B	OnSemi	Zener	100.0		.model 1N5378B D(Is=1.54e-17 N=1 XTl=
1N5817	OnSemi	Schottky	20.0	1.00	.model 1N5817 D(Is=31.7u Rs=.051 N=1
1N5818	OnSemi	Schottky	30.0	1.00	.model 1N5818 D(Is=31.7u Rs=.051 N=1
1N5819	OnSemi	Schottky	40.0	1.00	.model 1N5819 D(Is=31.7u Rs=.051 N=1
1N750	OnSemi	Zener	4.7		.model 1N750 D(Is=.88f Rs=.25 Cjo=175p
1N914	OnSemi	silicon	75.0	0.20	.model 1N914 D(Is=2.52n Rs=.568 N=1.7
30BQ060	International R	Schottky	60.0	3.00	.model 30BQ060 D(Is=10u Rs=.04 N=1.4
AOT-2015	AOT	LED	5.0	0.18	.model AOT-2015 D(Is=5.961e-10 Rs=1.0
B520C	Diodes Inc.	Schottky	20.0	5.00	.model B520C D(Is=200u Rs=8m Cjo=300
B530C	Diodes Inc.	Schottky	30.0	5.00	.model B530C D(Is=200u Rs=8m Cjo=300
B540C	Diodes Inc.	Schottky	40.0	5.00	.model B540C D(Is=200u Rs=8m Cjo=300
B550C	Diodes Inc.	Schottky	50.0	5.00	.model B550C D(Is=20u Rs=25m Cjo=300
B560C	Diodes Inc.	Schottky	60.0	5.00	.model B560C D(Is=20u Rs=25m Cjo=300
BAT54	Vishay	Schottky	30.0	0.30	.model BAT54 D(Is=.1u Rs=2.2 N=1 Cjo=
BZX84C10L	OnSemi	Zener	10.0		.model BZX84C10L D(Is=.6n Rs=.5 Cjo=1
BZX84C12L	OnSemi	Zener	12.0		.model BZX84C12L D(Is=.6n Rs=.5 Cjo=1
BZX84C15L	OnSemi	Zener	15.0		.model BZX84C15L D(Is=.6n Rs=.5 Cjo=1
BZX84C6V2L	OnSemi	Zener	6.2		.model BZX84C6V2L D(Is=1.5n Rs=.5 Cjo
BZX84C8V2L	OnSemi	Zener	8.2		.model BZX84C8V2L D(Is=.8n Rs=.5 Cjo=
CMDSH2-3	Central	Schottky	30.0	0.20	.model CMDSH2-3 D(Is=210nA Rs=.5 N=

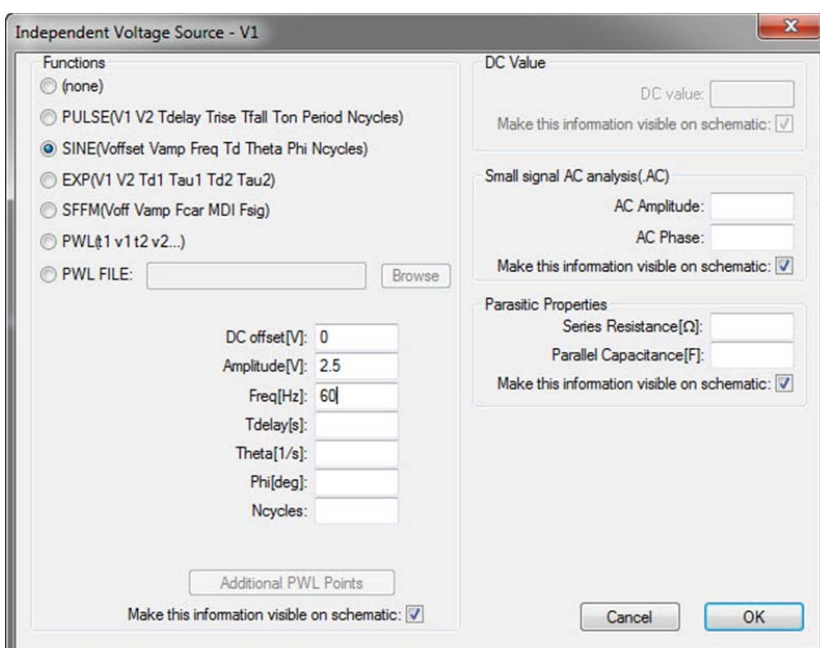
Add a voltage source (sinusoidal input). Click on add component on your toolbar:



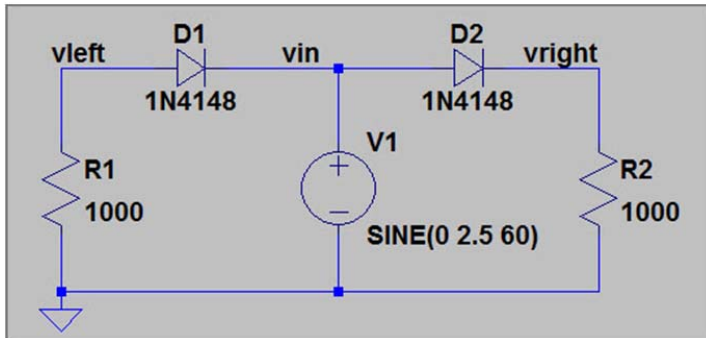
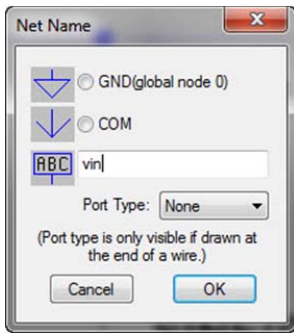
Place the voltage source, wire it, and your circuit should now look like the following:



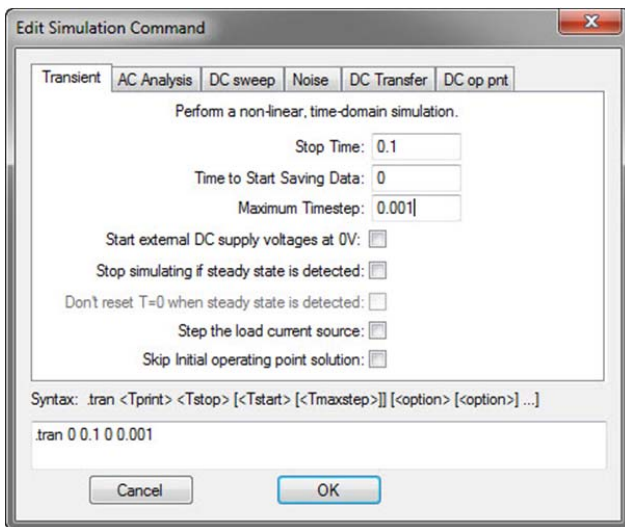
Right click on the voltage source and set it to a 60 Hz sine wave with 2.5 V amplitude.



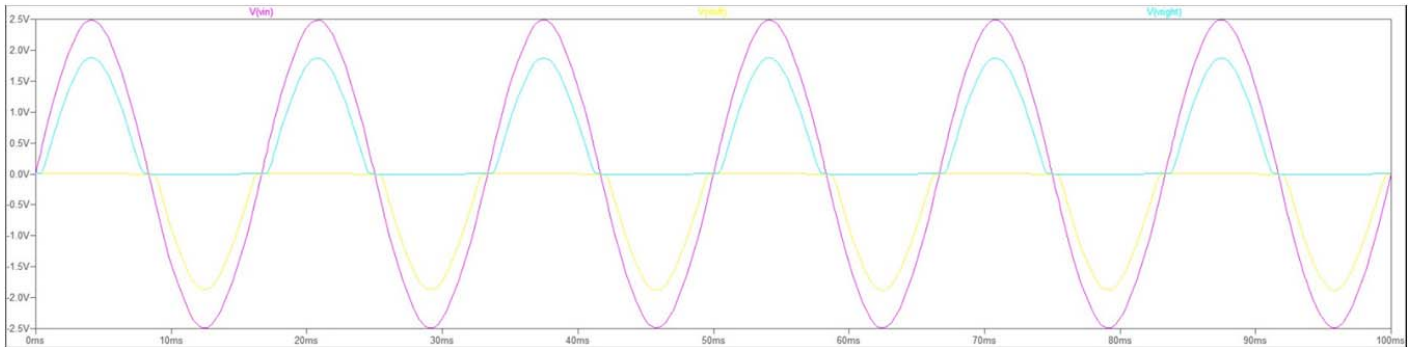
Add net names. This labels voltage nodes for later reference.



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.

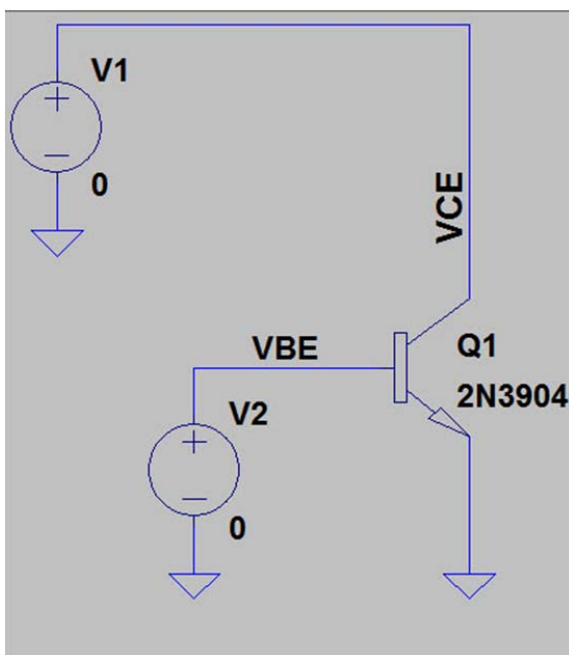


Click with the voltage probe on the wires where we want to measure the voltage and you should see the following output. Does this look familiar?

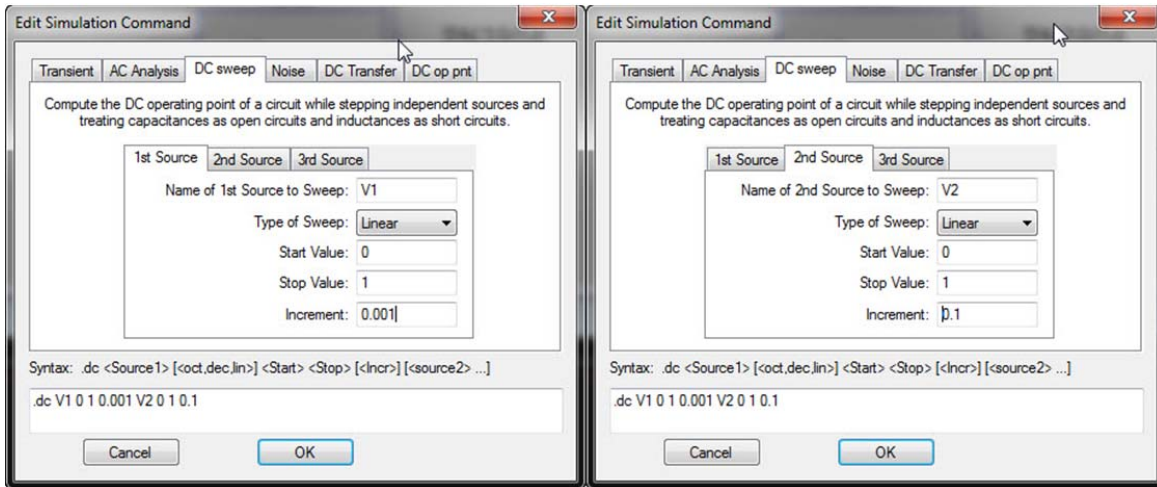


### 3. DC Bias of npn BJT

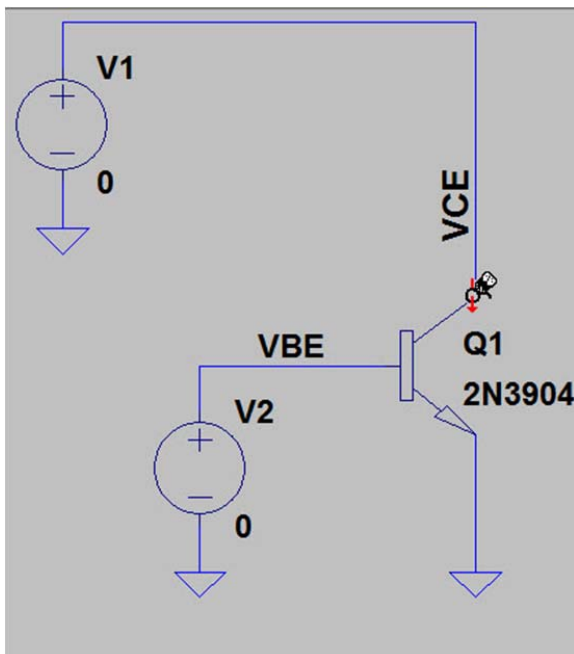
Here we will examine the BJT current with the following bias circuit. Select and place a two voltage sources, your grounds, and an npn transistor. Select the 2N3904 transistor just as you selected a specific diode in the previous example. Set the appropriate net names.



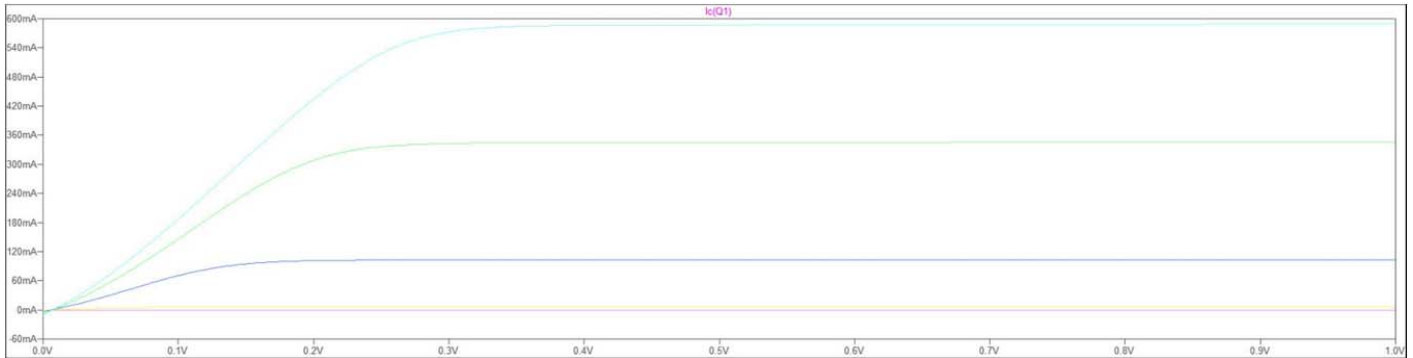
We will now do a DC sweep of the circuit. Click the simulate button, and select the DC sweep menu. Enter the appropriate values for the 1<sup>st</sup> source and 2<sup>nd</sup> source and shown below.



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.

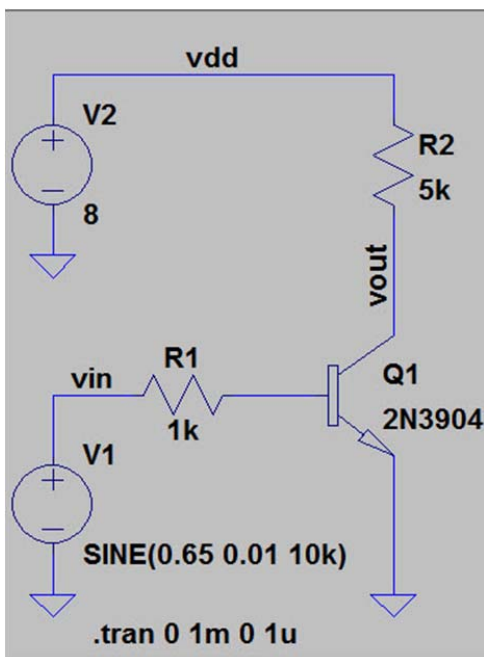


You should see a nice set of curves with the  $I_{CE}$  current plotted against  $V_{CE}$  with each curve having a different  $V_{BE}$  value. These curves will become very familiar when you begin to study the DC characteristics of the BJT.



#### 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 1 $\mu$ s 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?