# Using LTspice – a Short Intro with Examples

LTspice, also called SwitcherCAD, is a powerful and easy to use schematic capture program and SPICE engine, which is a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analysis. LTspice authored by Mike Engelhardt can be downloaded for free at http://www.linear.com/designtools/software/

On the left side there are 4 downloads
- the program itself, you must run the LTspiceIV.exe program to install the software
- user guide
- getting started
- demo circuits (not useful for this course, download instead the example files posted on the course's website or, alternatively open files in File->examples->Educational directory)

1. Example of a simple DC analysis – file LTspice1.asc
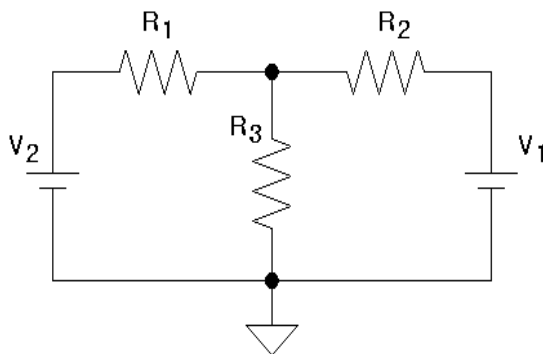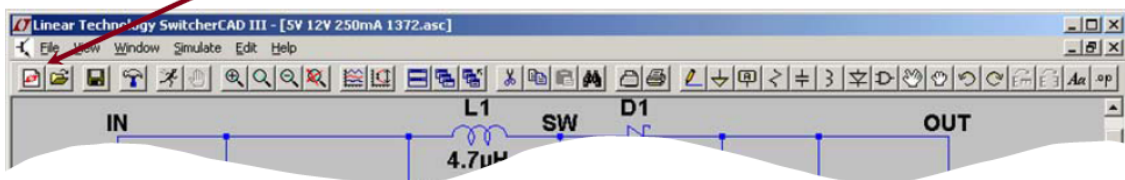

The values of components are:
V_1= 5V
V_2=10V
R_1=10Ω
R_2=20Ω
R_3= 5Ω

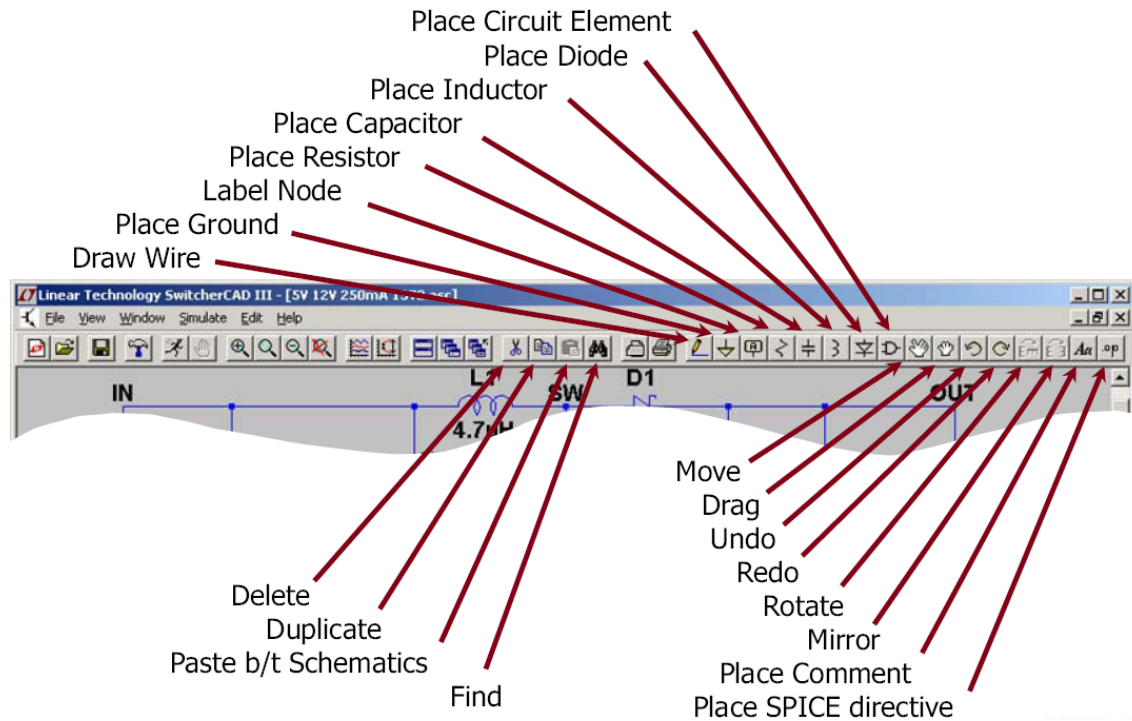*Goal: determine the current flowing through each resistor and voltages of nodes.*

Step 1. Draw the circuit.

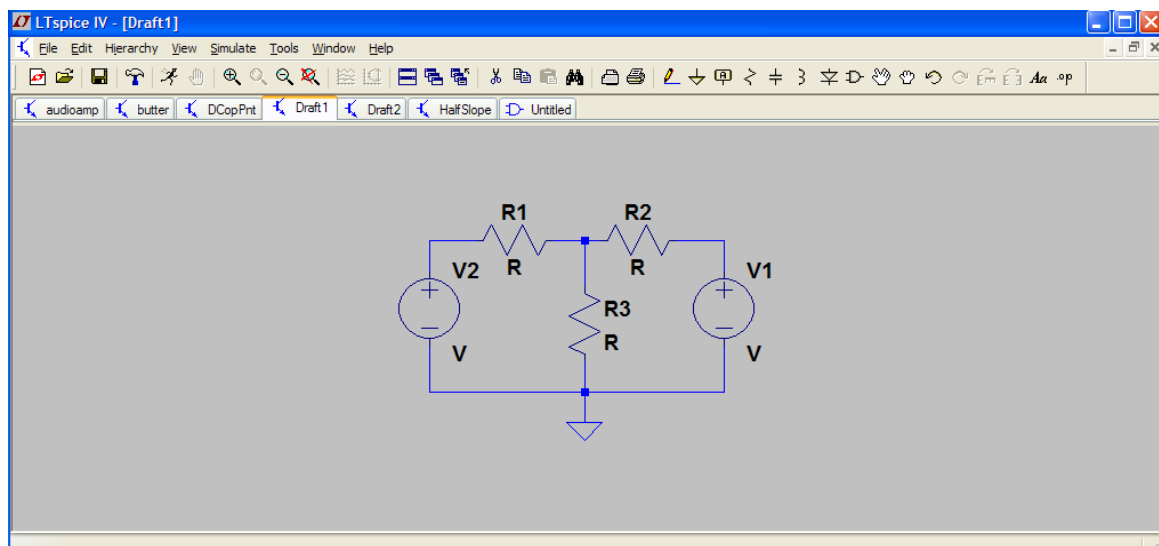Go to File ->New schematic to create a new circuit.

Select the components from the Schematic Editor Toolbar. In this example, you'll need three resistors [image], two DC voltages (select Component [image], type voltage and hit ok), a ground [image], and wires connecting the components [image]. By default, components are placed vertically, so you may want to rotate two resistors.

Place Circuit Element
Place Diode
Place Inductor
Place Capacitor
Place Resistor
Label Node
Place Ground
Draw Wire

Linear Technology SwitcherCAD III - [5V 12V 250mA ...]
File View Window Simulate Edit Help

IN   L1   SW   D1   OUT
            4.7µH

Move
Drag
Undo
Redo
Rotate
Mirror
Place Comment
Place SPICE directive

Delete
Duplicate
Paste b/t Schematics
Find

Once you are done your circuit should look like this. Note that you can zoom in/out by scrolling.

LTspice IV - [Draft1]
File Edit Hierarchy View Simulate Tools Window Help

audioamp | butter | DCopPnt | Draft1 | Draft2 | HalfSlope | Untitled

R1   R2
V2   R   R   V1
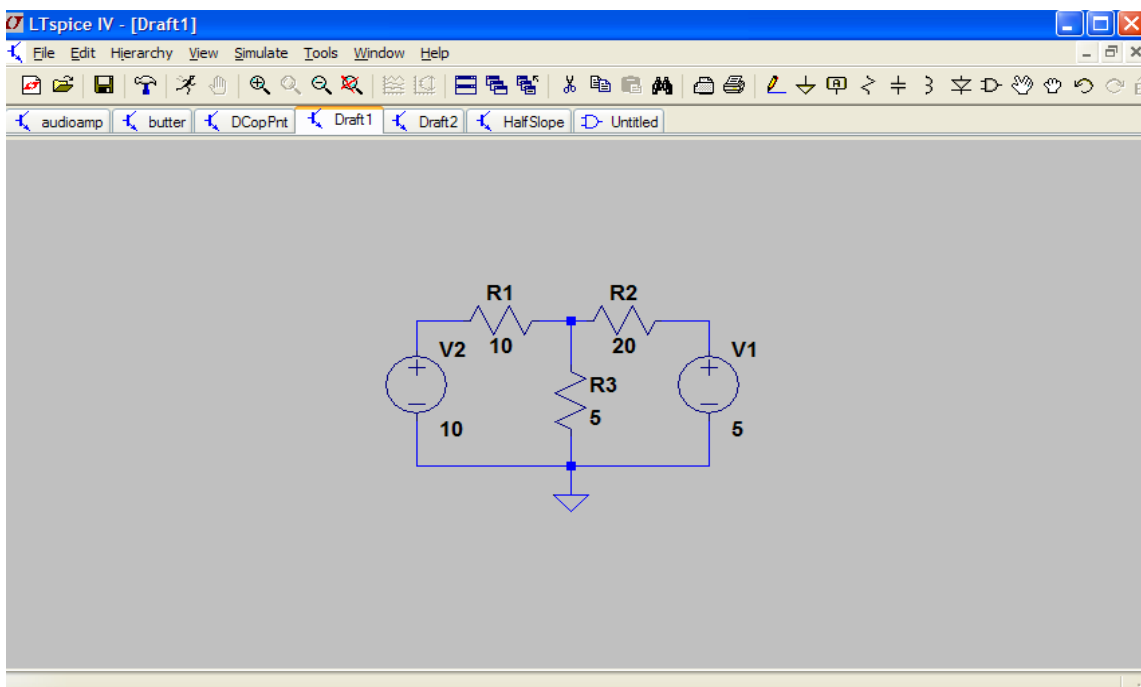   R3
V   R   V

Step 2. Enter the values.

Right click on each component. A window will pop up like this. Enter the value and hit ok.
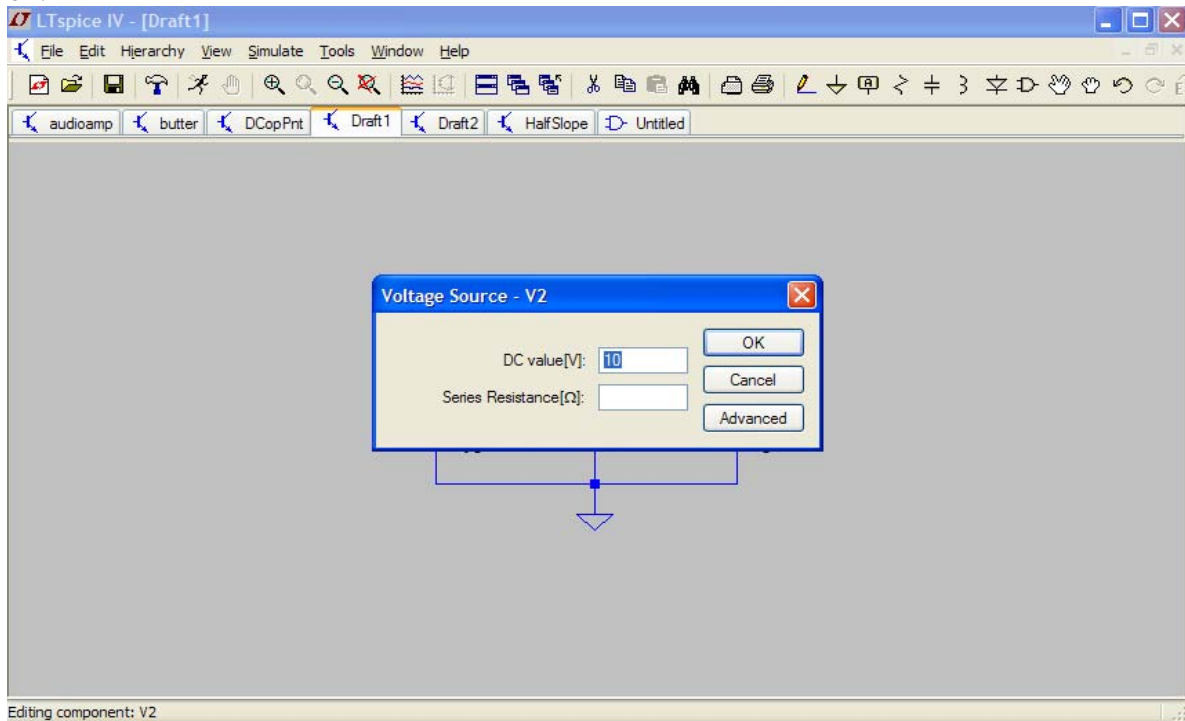

Finally, your circuit diagram should look like this.

Result will be shown in a pop-up window.
```
        --- Operating Point ---

V(n002):        3.57143         voltage
V(n001):        10              voltage
V(n003):        5               voltage
I(R3):          0.714286        device_current
I(R2):          0.0714286       device_current
I(R1):          -0.642857       device_current
I(V2):          -0.642857       device_current
I(V1):          -0.0714286      device_current
```

2. Example of a transient analysis – files LTspice2.asc and LTspice2-step.asc
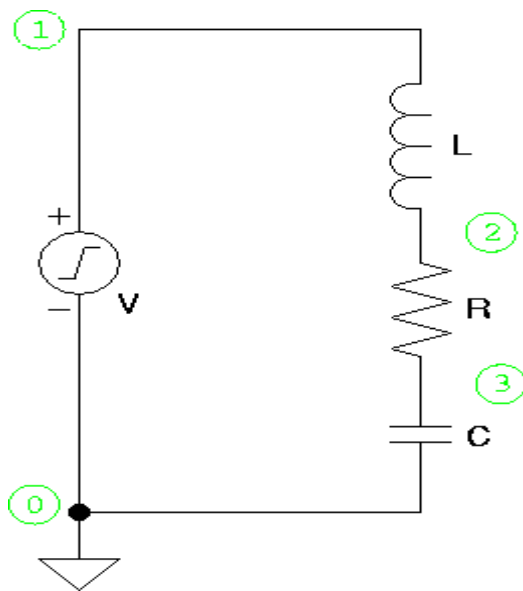
Components:
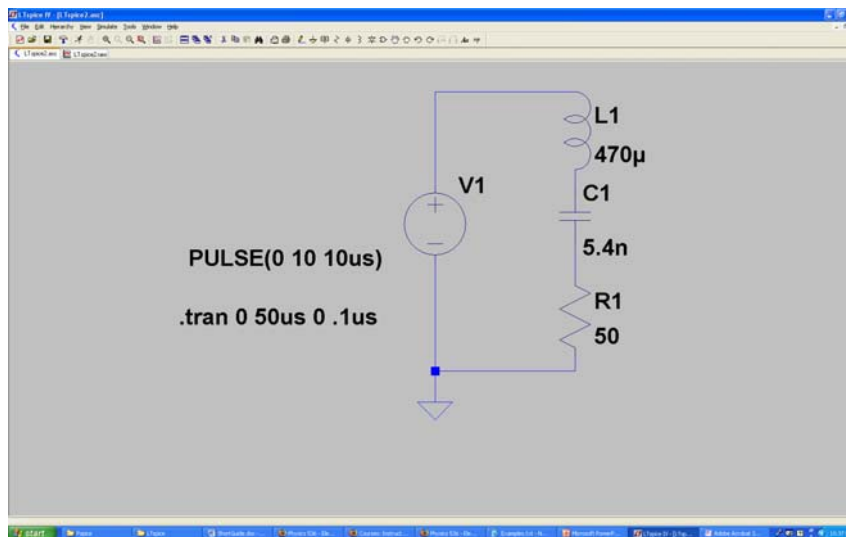
V=0 for t<0 and 10V for t>0
L=470uH
C=5.4nF
R=to be determined

Goal: determine the range or resistance that produces oscillations. From the analysis of the resulting differential equation $R^2/(4L^2)<1/(LC)$
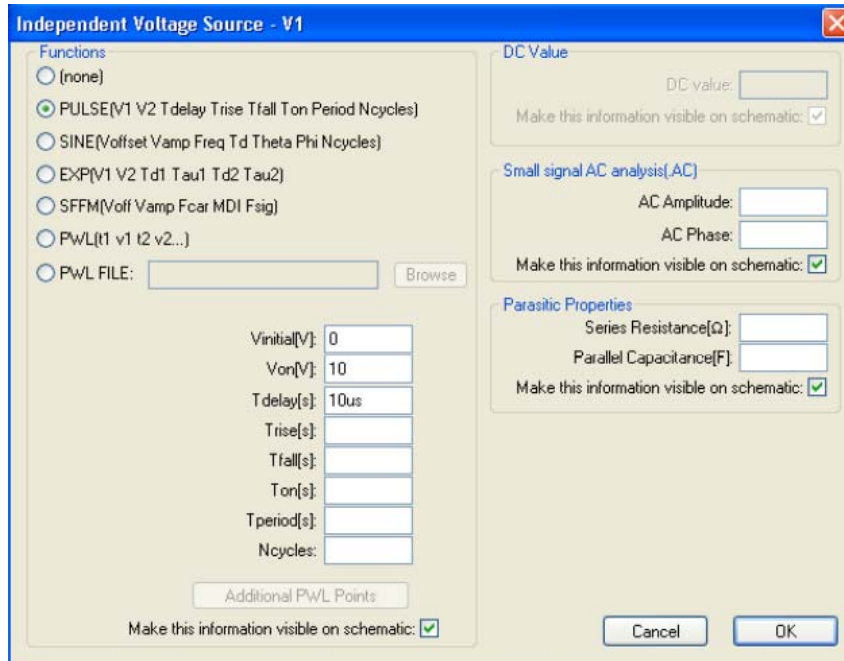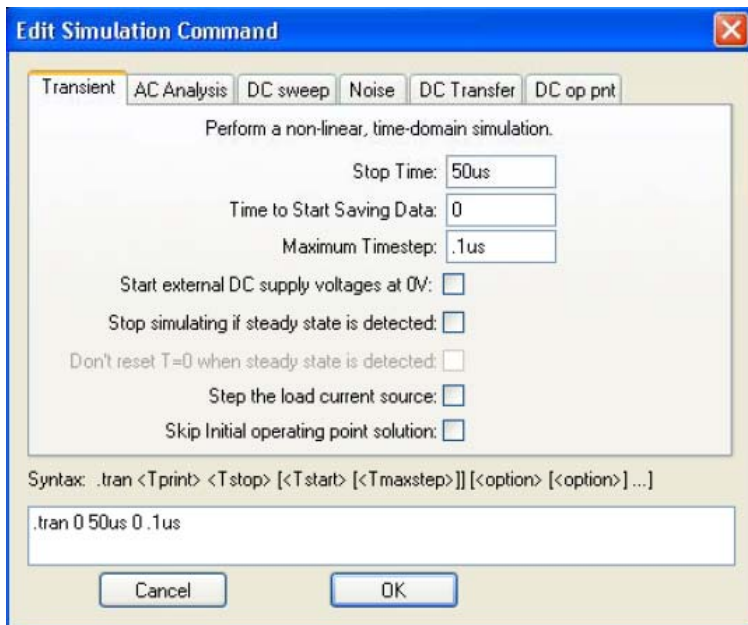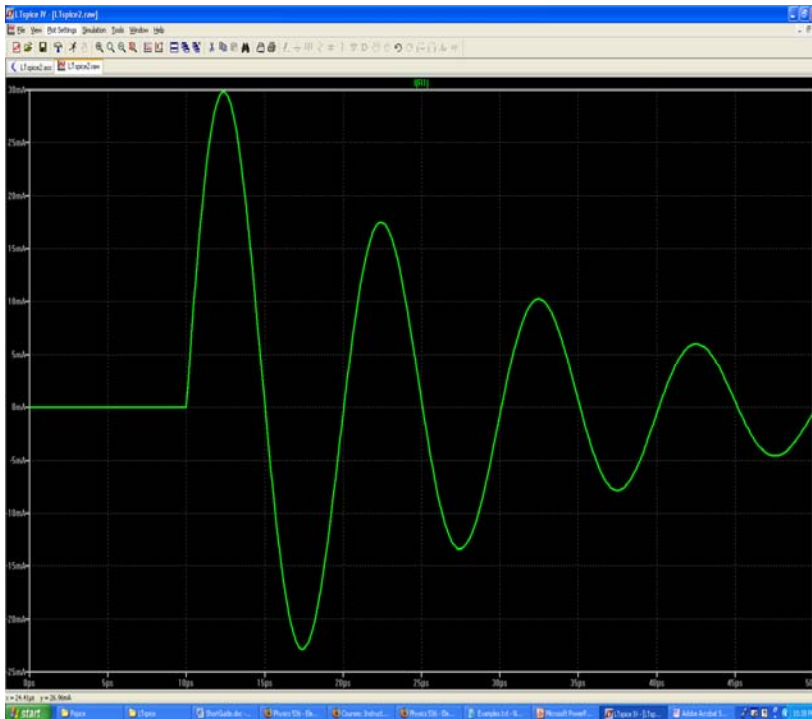
Set the values of the passive elements and set the voltage source to Pulse with Vini=0V and Von=10V. One can also pick a delay of 10µs.

Check the appearance of the circuit.

Step 2. Select from menu
Simulate ->
**Edit SimulationCmd ->**
Transient.

Step 3. Run simulation **then place probe onto a wire to plot voltage or onto a terminal of a device to plot current into that terminal. The result for the current through the circuit is shown on the left.**
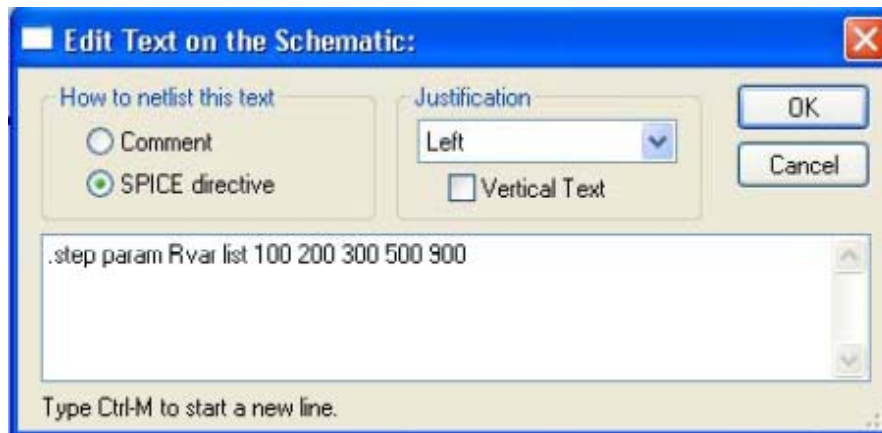
You might save ASCII data in a file: File -> Export
Then run simulation for other values of the resistance.

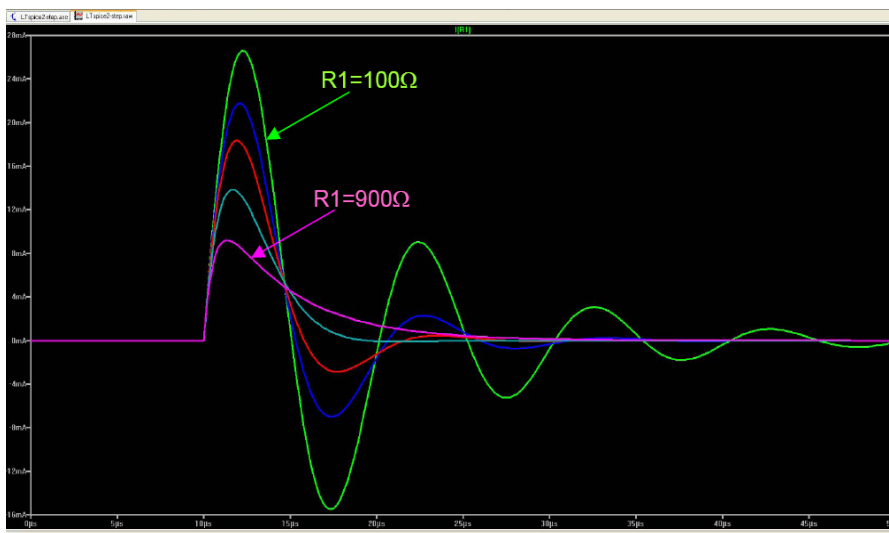# Alternative *Goal: Run simulations for a list of resistances and overlay the results.*
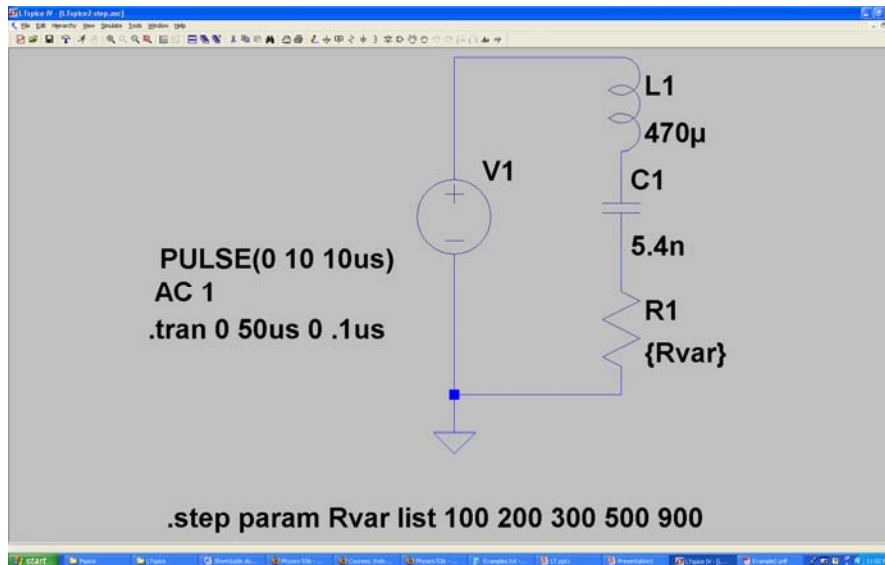
Change the numerical value of the resistance to {symbol}.

Edit-> Text Then fill out box as seen on the left, i.e. enter .step directive and list of resistors, check SPICE directive bullet.
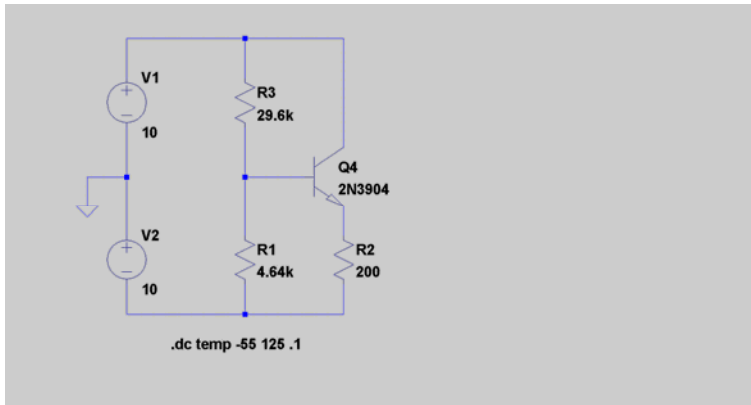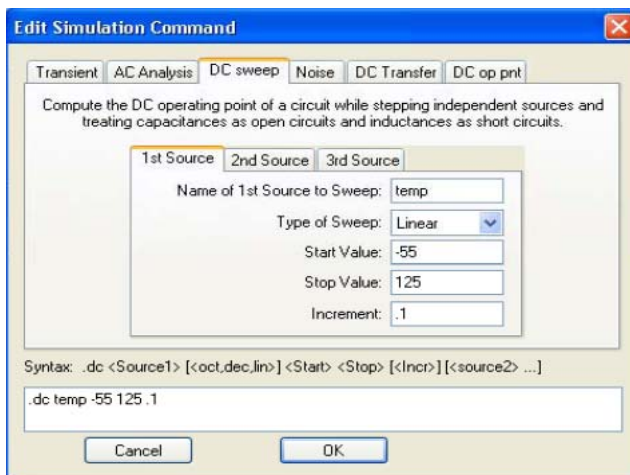

Run simulation

and place probe onto a wire to plot a voltage or onto a terminal to plot a current flowing into a device.

The result for the current through the circuit is seen on the left.

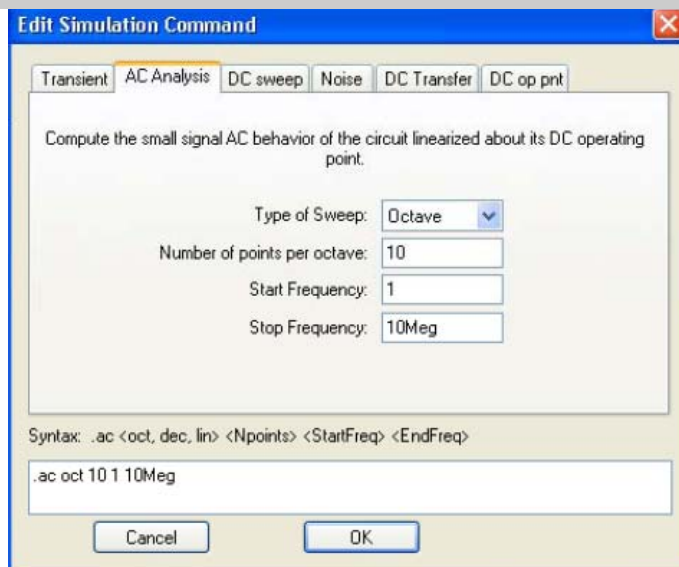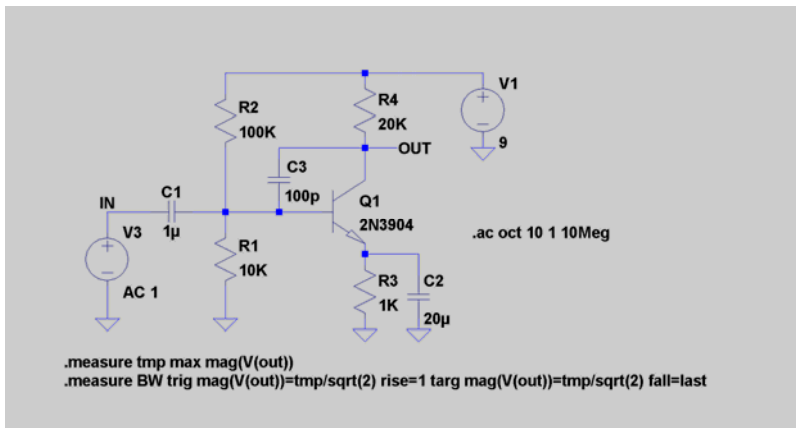# 3. Example of a temperature dependence of a BJT amplifier – file LTspice3.asc

*Goal: Find temperature dependence of collector current*

Once circuit is completed, pick temperature to be swept in the DC sweep tab of the Edit Simulation Command, which is found in the Simulate menu.

Run simulation.

4. Example for measuring bandwidth – file LTspice4.asc


```
.measure tmp max mag(V(out))
.measure BW trig mag(V(out))=tmp/sqrt(2) rise=1 targ mag(V(out))=tmp/sqrt(2) fall=last
```


Once circuit is completed, pick the AC signal analysis tab of the Edit Simulation Command, which is found in the Simulate menu.

Run simulation.
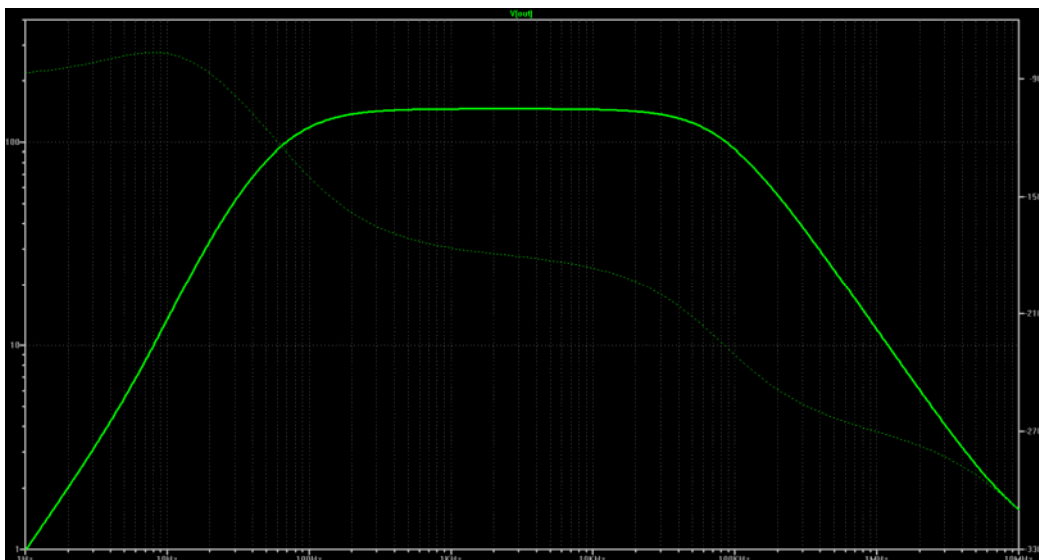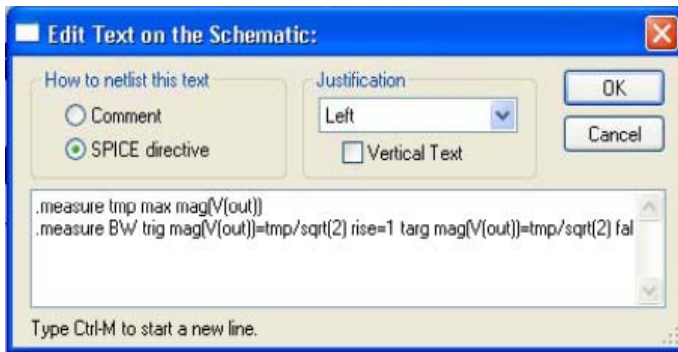
In addition one can measure the maximum value of the voltage output as well as the bandwidth by choosing Edit-> Text. Then fill out box as seen on the left, i.e. enter .measure directive and the commands seen and check the SPICE directive bullet.
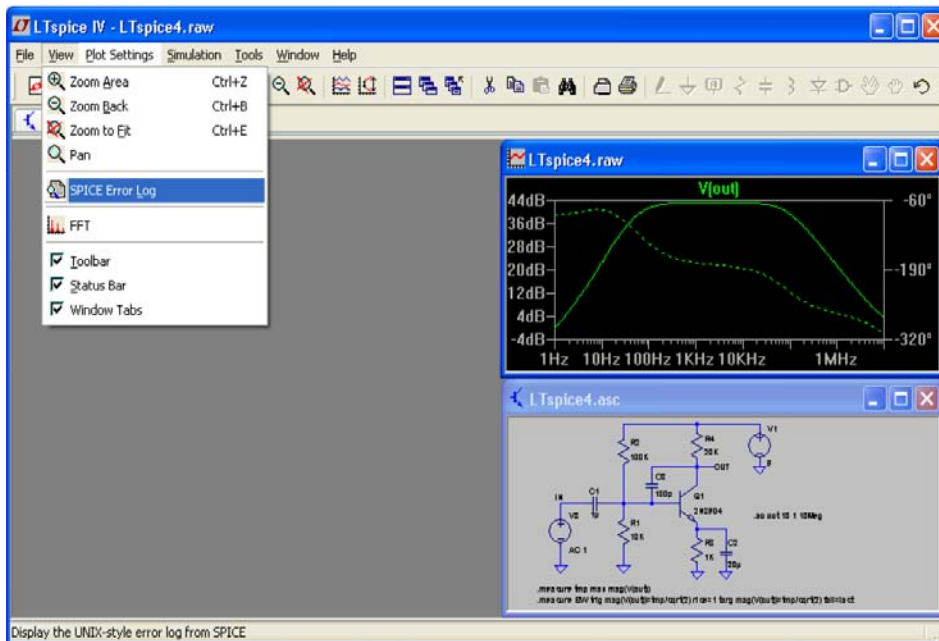
For the results look in the SPICE Error Log file within the View menu.

5. Example for a curve tracer – file LTspice5.asc

.dc V1 0 5 10m I1 0 100u 20u
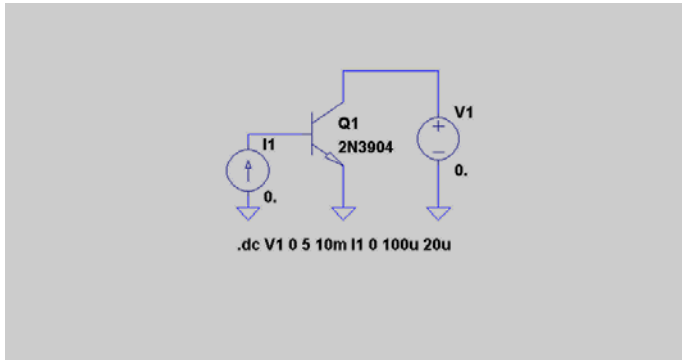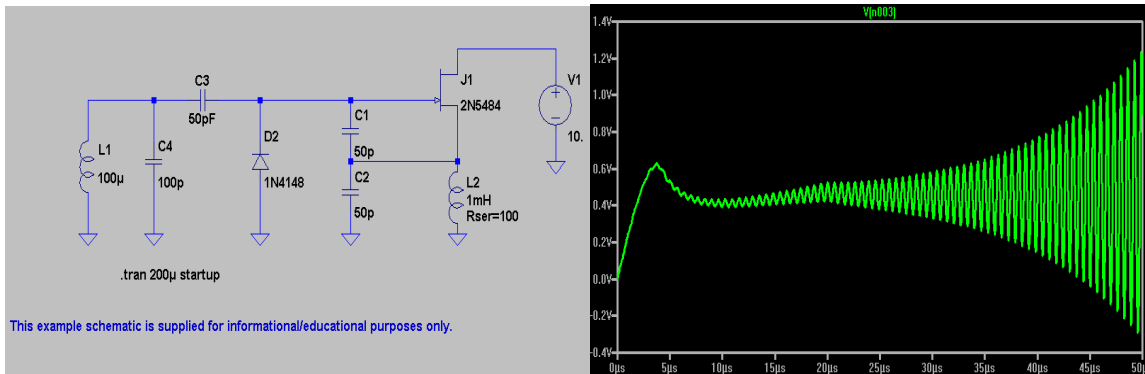
Goal: Find the characteristics of a pnp bipolar transistor

Set up two DC sweeps – one for collector voltage, the other for base current

Then Run simulation
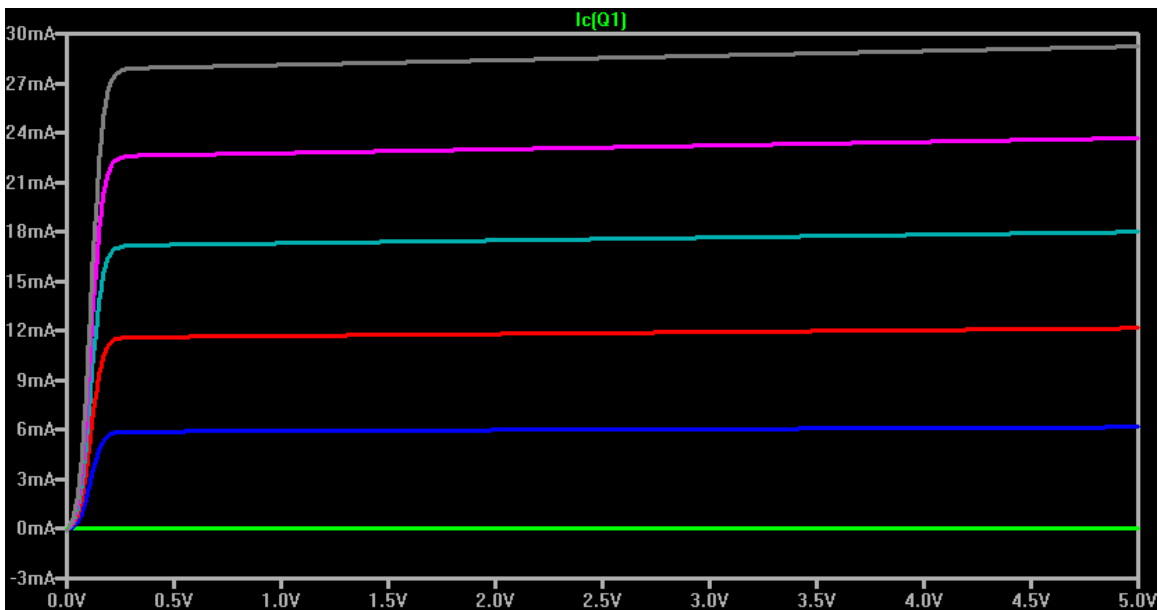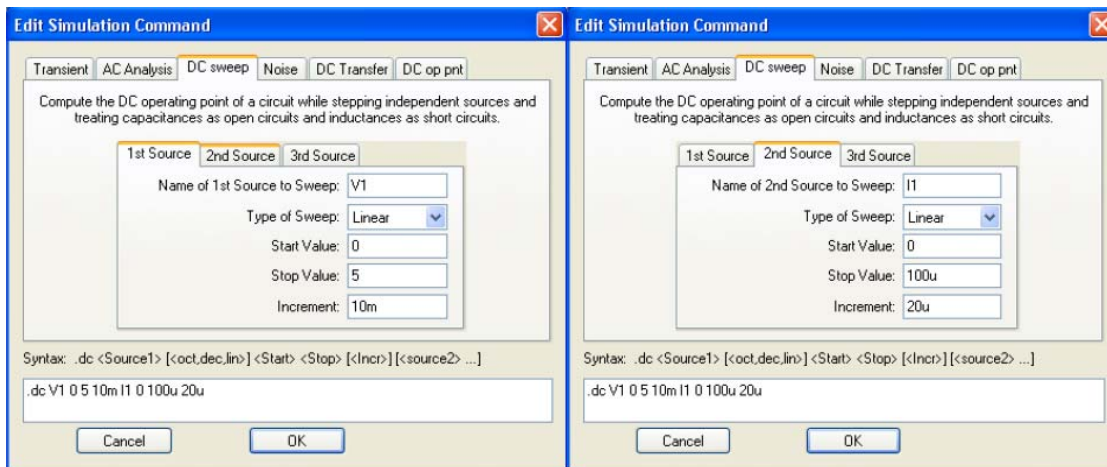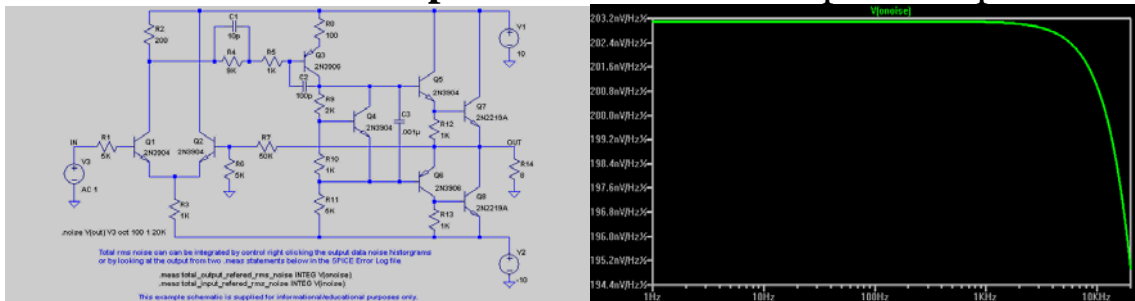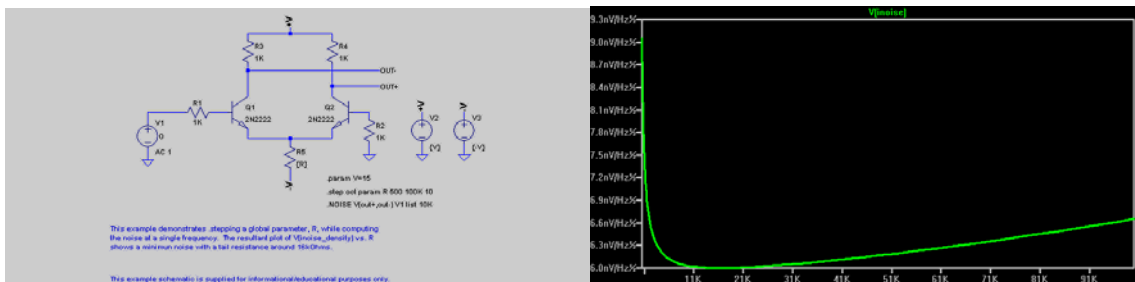
6. Example of an oscillator – file LTspice6.asc

7. Example of a noise vs frequency– file LTspice7.asc
and noise vs resistance paramenter – file LTspice7-step.asc

You must click on the Output pin on the schematics to see the output refererred

You must click on the Visible Traces icon ⎍ Visible Traces then select V(inoise) to plot the input referred noise.