# LTSpice Tutorial

Satoshi Suzuki

# Table of Contents

# 1. Revision History

| Version | Date | Comments |
|---|---|---|
| 1.1 | 1/19/2009 | Added Waveform Characteristics in Section 6.1 |
| 1.0 | 12/19/2008 | Initial Creation |

# 2. Introduction

LTSpice is a simulation tool designed by Liner Technology. It's free and it runs on Windows. Now, LTSpice IV is available. If you a UNIX user and you want to run LTSpice on UNIX, you can use "Wine" to run this software.

You can download the software from here.
http://www.linear.com/designtools/software/
Also, user guide, start-up guide, and demos are available there, too.

This software is very easy to get familiar with and it is well designed. It would be nicer if shortcut keys are the same as other SPICE software such as PSPICE.

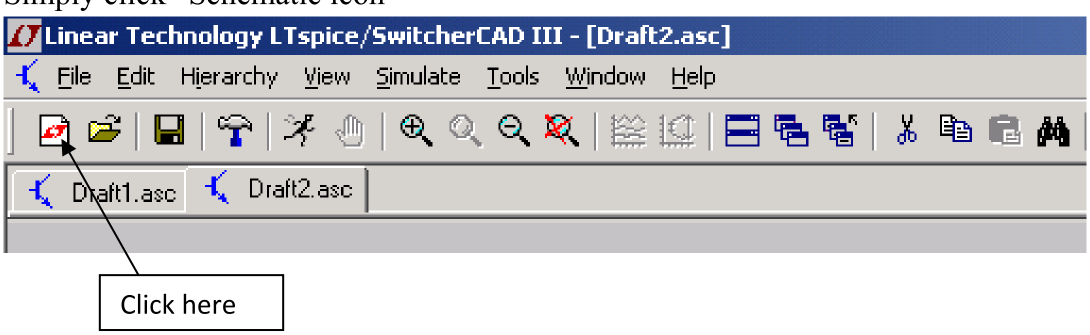
# 3. Create a schematic

The first thing you need to do is to create a schematic.

Go to "File" on the menu bar once you open your LTSpice.
Click "New Schematic"

Or

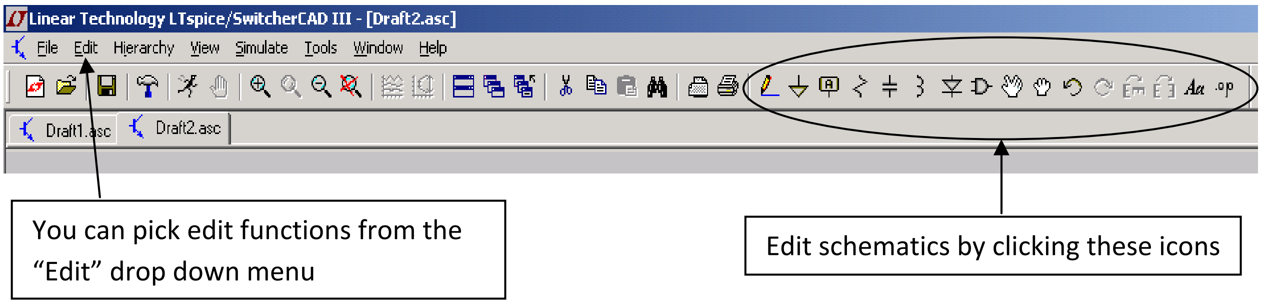
Simply click "Schematic icon"




A new schematic shows up and now you can start editing.

# 4. Editing and Shortcut keys for Schematics

Once you create a new schematic, you can go ahead and start building your circuit. The way to do it is shown below. It’s faster to just place components first and then connect the components through wires.




If you think moving a mouse to choose edit options is tedious, Table 2.1 shows the shortcut keys for editing schematics.

**Table 2.1 Shortcut Keys used when designing schematics**

| Function | Key | Description |
|---|---|---|
| | | Editing schematics |
| Draw wire | F3 | Connect components |
| Label Net | F4 | Name a node/wire and helpful when you simulate and plot graphs |
| Delete | Delete or F5 | Delete components or wires |
| Duplicate | F6 | Helpful when you want to copy and paste |
| Move | F7 | Move components, and also move multiple components by highlighting |
| Drag | F8 | Highlight components and wires, can shorten or widen wires |
| | | Placing components |
| Resistor | Press ‘ and r | Place a resistor |
| Capacitor | Press ‘ and c | Place a capacitor |
| Inductor | Press ‘ and l | Place an inductor |
| Diode | Press ‘ and d | Place a diode |
| Component | F2 | A window pops up and you can choose components to place |
| Ground | Press ‘ and g | Place a ground |
| | | SPICE Analysis and comments |
| Directive | Press ‘ and s | Place a SPICE command for simulation |
| Text | Press ‘ and t | Place a text as comments |

Voltage and current sources can be found from Component Window.

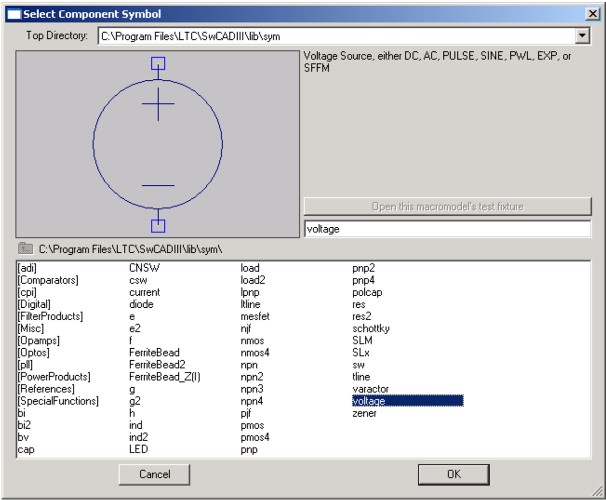



Figure 2.2 Voltage source

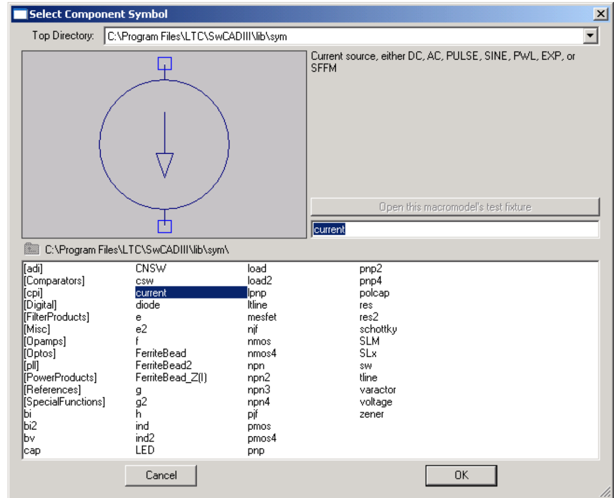



Figure 2.3 Current Source

If you want to make AC voltage or current source, once you place the default voltage or current source, right click the component and go to “Advanced.”

Then, you can choose your voltage or current source to produce pulse, sine, piece-wise liner and etc. Figure 2.4 and 2.5 below show the settings for a current source.

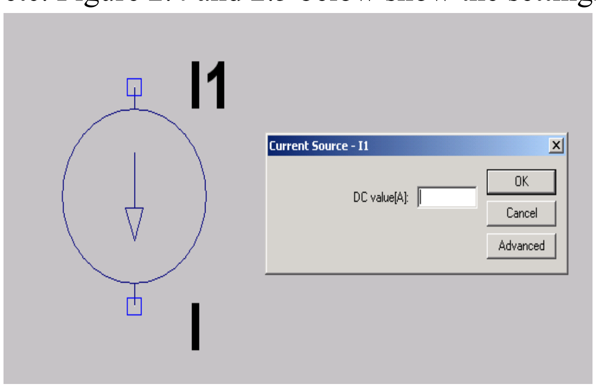



Figure 2.4 Current Source Option

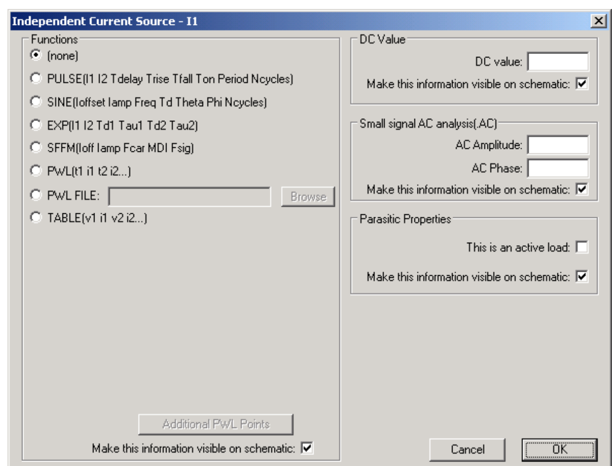



Figure 2.5 Advanced setting

# 5. Simulation

Once you finish designing your schematic, now it is time to simulate your circuit.

First, you need to determine what type of simulation you want to do.

Use SPICE directive (Type ‘ and s at the same time) or “Edit Simulation Cmd” from “Simulate” menu tab.

## 5.1 Simulation with “Edit Simulation Command”

For those who use SPICE for the first time, it’s better to use “Edit Simulation Cmd” until you get to know what commands you need to type for SPICE simulation.

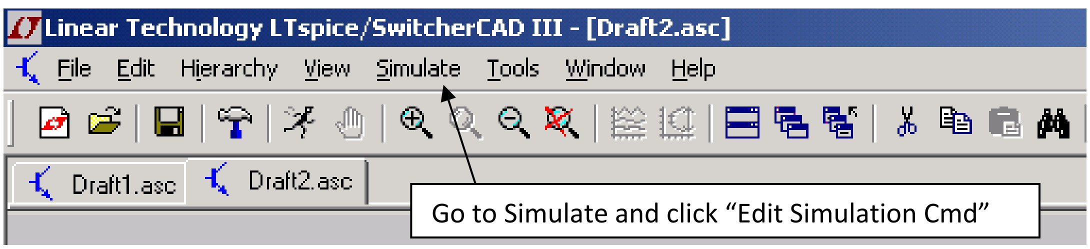



Figure 5.1.1 Simulation Command in LTSpice

Then, a window (Figure 3.2) pops up.

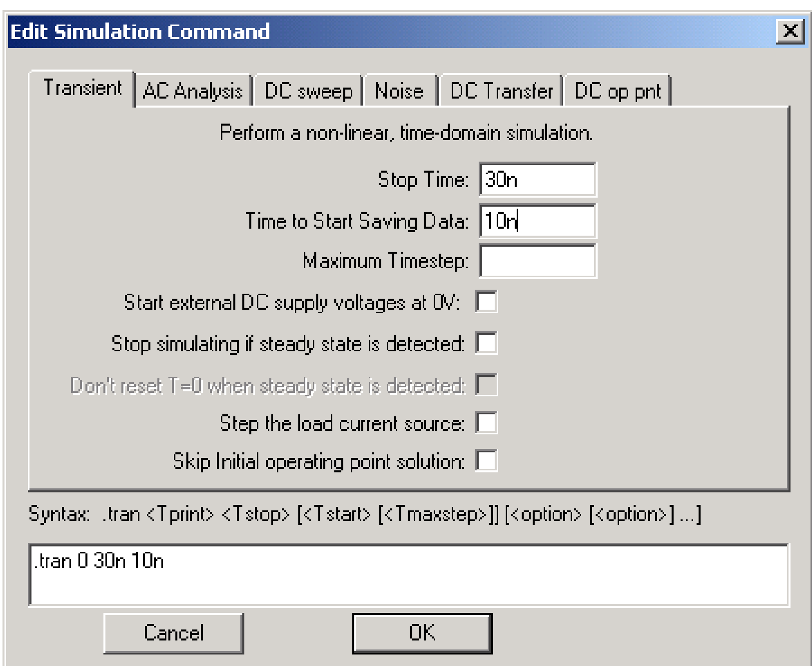



Figure 5.1.2 Transient Analysis from “Edit Simulation Command Window”

Once you finish the setting, click OK and a box shows up. This box needs to be placed somewhere in your schematic.

Then click “Run” icon shown in the next figure.


Figure 5.1.3 “Run” Icon from Menu bar

## 5.2 Simulation with SPICE Directive

If you press ‘ and t at the same time or choose “SPICE Directive” from “Edit” menu bar, the figure below shows up. You can start typing SPICE commands. The following shows examples of different types of SPICE commands.

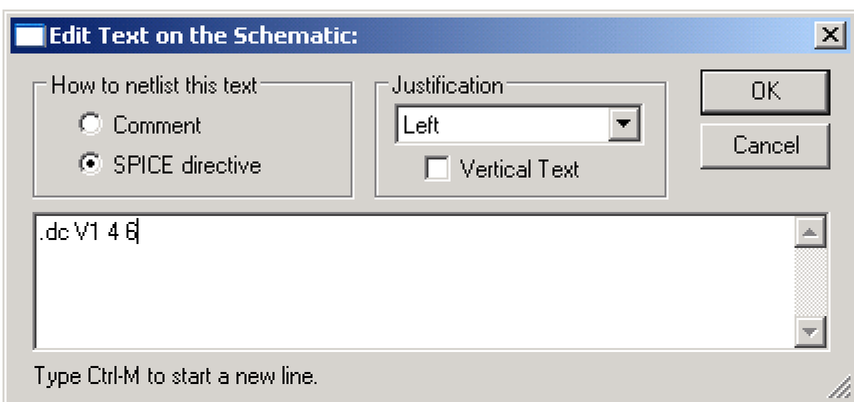



Figure 5.2.1 SPICE directive

Table 5.2.2 SPICE Command Examples

| Type | Commands | Comments |
|---|---|---|
| Transient | .tran 0 30n 10n | Sweep time till 30nsec and start collecting and plotting data from 10nsec |
| DC Sweep | .dc V1 4 6 0.2 | Sweep a voltage source called “V1” from 4V to 6V with increment = 0.2V. |
| AC Sweep | .ac dec 250 100 10Mega | Sweep frequency from 100Hz to 10MHz taking 250 pts over decade (20dB/dec) |

| | | |
|---|---|---|
| Temperature Sweep | .op<br>.step temp -55 125 | When you sweeping parameters, you will need ".op" SPICE command. |
| Parameter Sweep | .step param R 1k 2k 100 | Sweep a resistor whose parameter is called "R" from 1kΩ to 2kΩ in steps of 100Ω<br>You need to change the resistance value to {R} shown in Figure 3.2.2 below. |
| Calculate values | .meas ac H1 find V(s11) when freq = 1e6 | Find the again and phase of the voltage at node "s11" when frequency is 1MHz with AC response. The result is shown in Fig 3.2.4. |

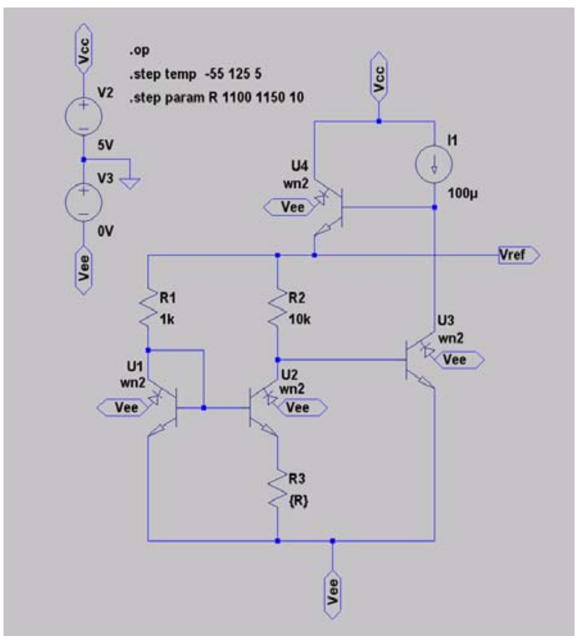


Figure 5.2.2 Parameter Sweep Example

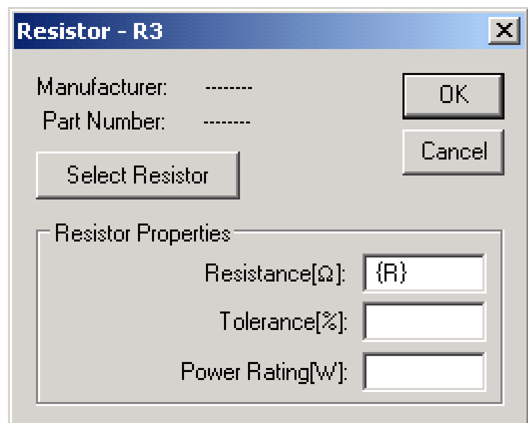


Figure 5.2.3 R3 Parameter Sweep

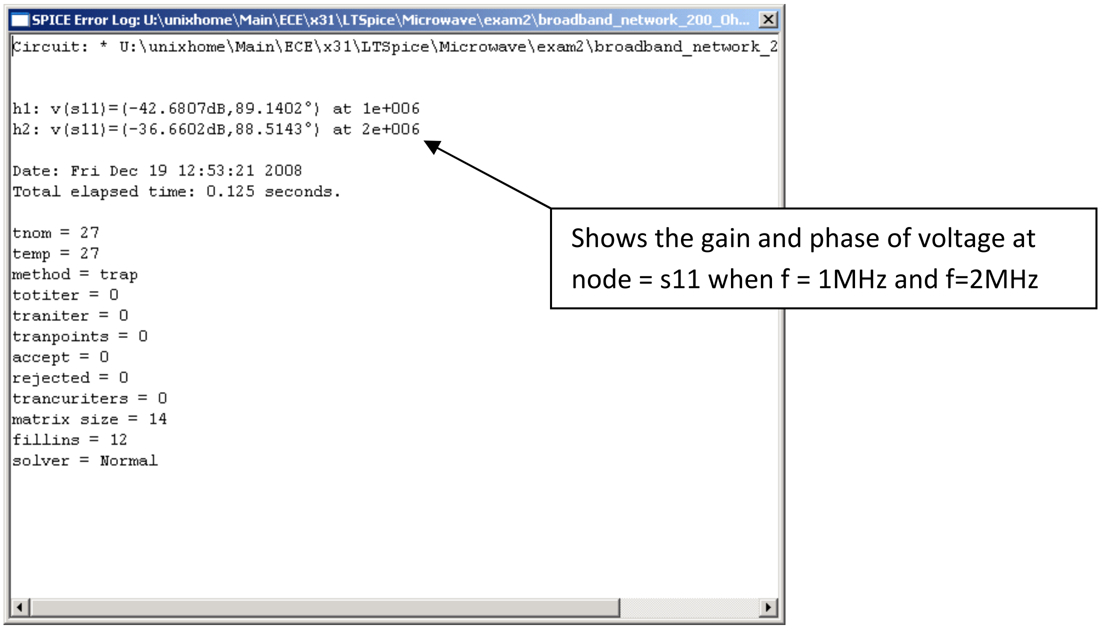



Figure 5.2.4 Gain and Phase shown in Error log from "View"

# 6. Plot Settings

## 6.1 Basic Plot Settings

Once you click "run," the program starts simulating your circuit and a plot plane shows up.

With the plot window selected, you need to click the node that you want to measure. You should see your graph as follows.

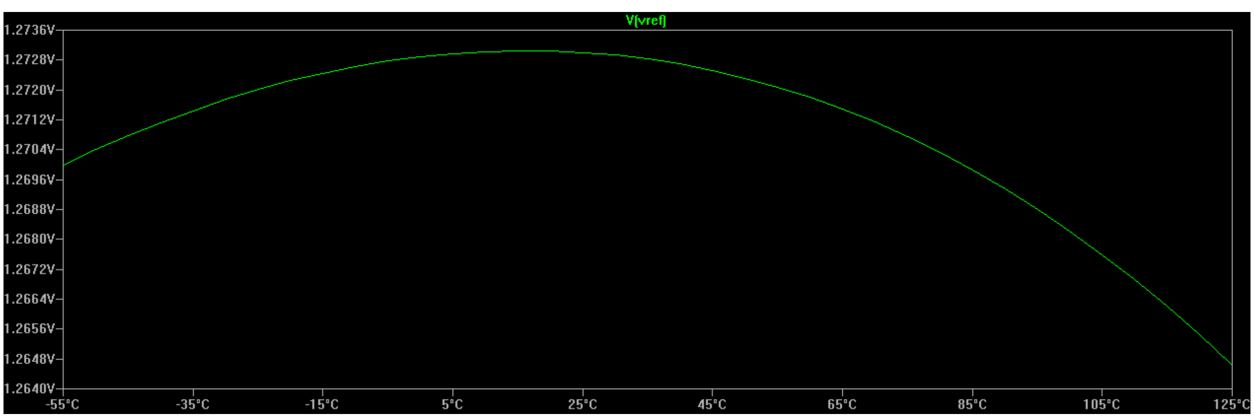



Figure 6.1.1 Example Plot in LTSpice

If you want to change the graph to the way you want to be such as current and resistor, there is a way to plot these. Figure 6.1.2 shows how to change graphs.

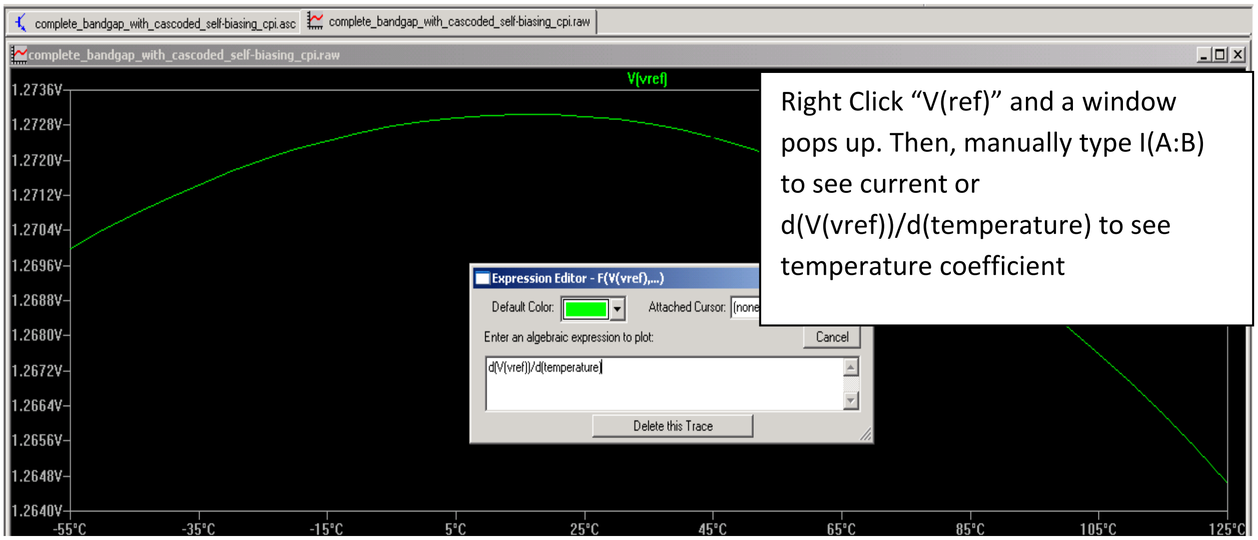


Figure 6.1.2 Changing Plots

You can also show a grid on the plane and add another plot plane. Moreover when you want to show x and y coordinates on the plot, then place your cursor where you want to measure and go to "Notes and Annotations" under "Plot Settings" and choose "Label Curs. Pos."

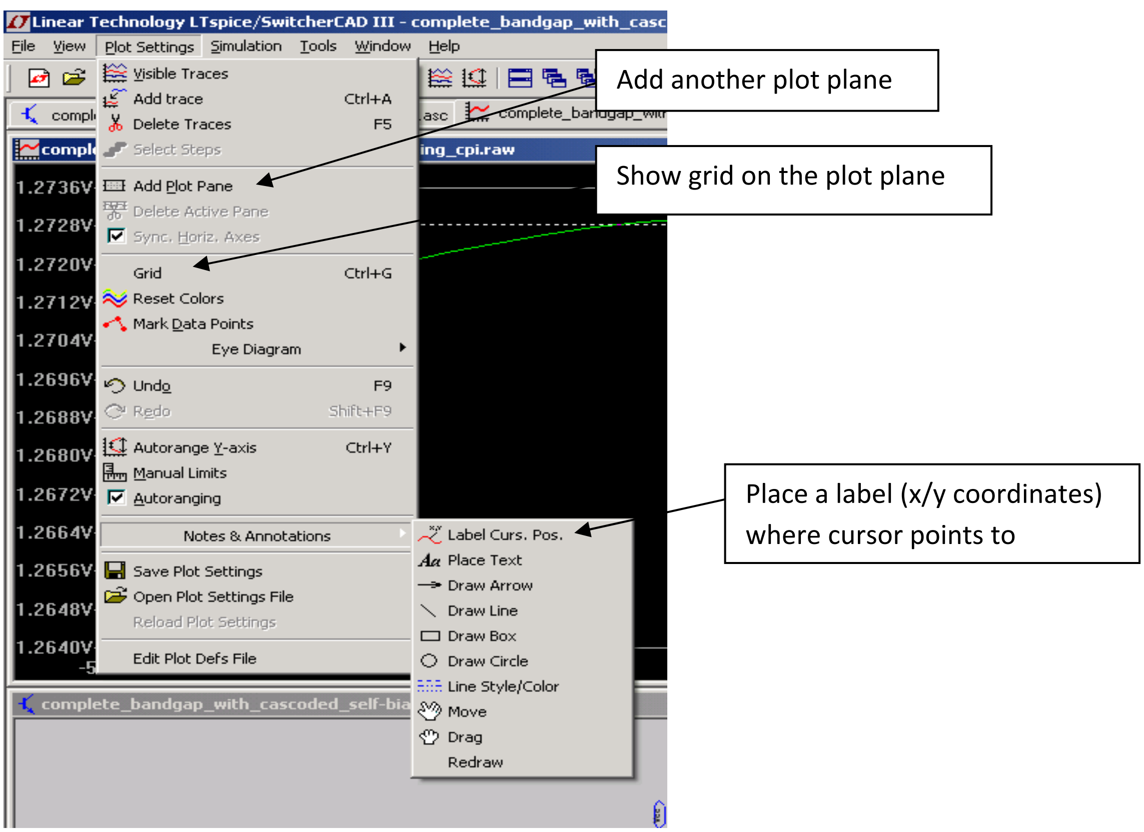


Figure 6.1.3 Plot Settings Commands

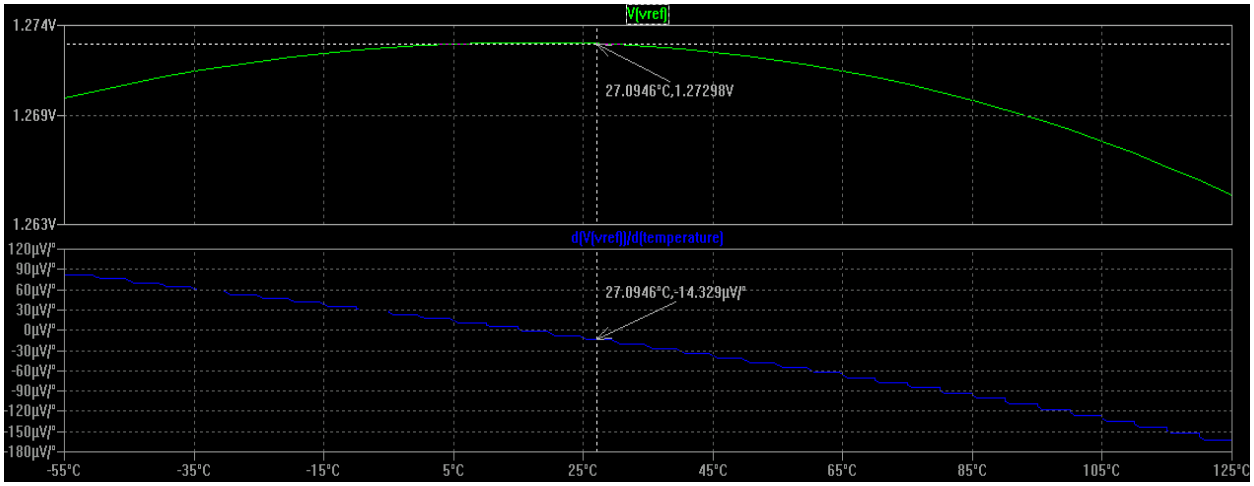


Figure 6.1.4 Plots After Plot Settings Changes

If you want to change the colors of anything, just right click where you want to change and choose the color you like.

Also, if you want to see rms value for transient analysis or bandwidth of your filter design for ac analysis, if you press ctrl key and right-click your mouse, you will see rms value or bandwidth of your circuit. One example is shown below.

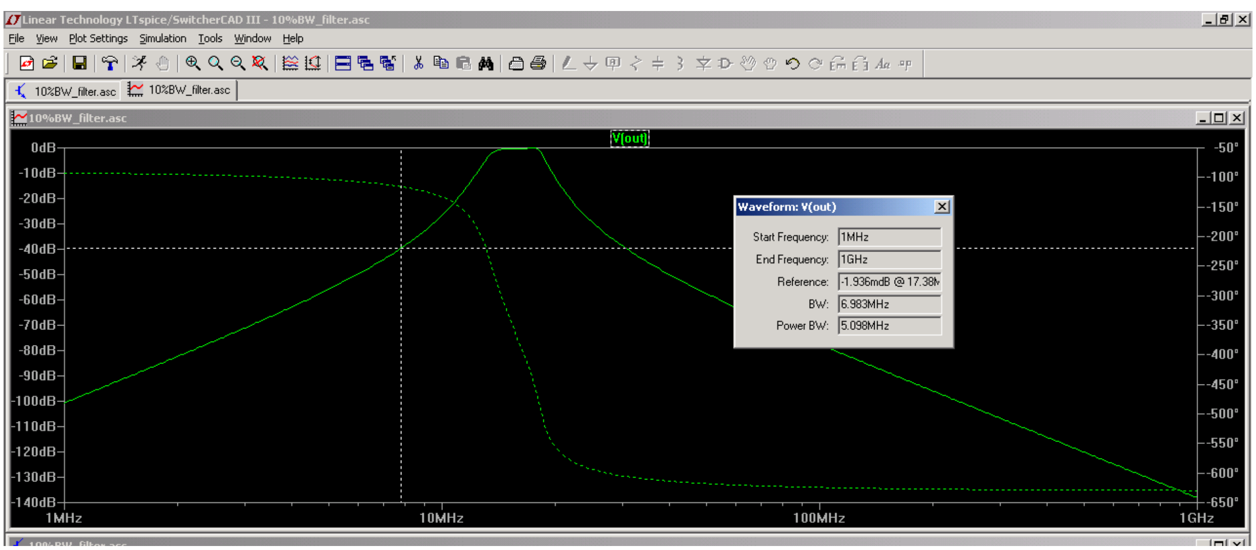


Figure 6.1.5 Waveform Characteristics

# 6.2 Create a wmf file for Schematics and Plots

You can capture schematics and plots by a "Print Screen" key or choosing "Copy bitmap to Clipboard" from Tools (you can find it from Figure 6.2.1)

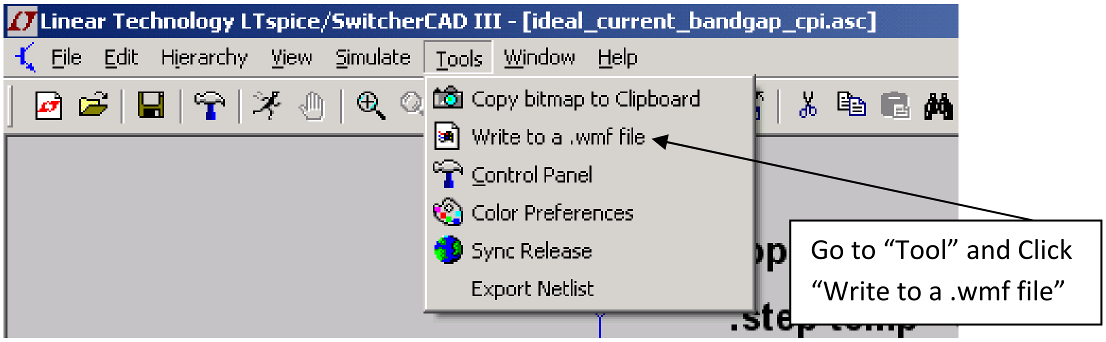


Figure 6.2.1 Create a wmf file

You can place wmf files on MS Word by just dragging the icons of wmf files to a word document like below.

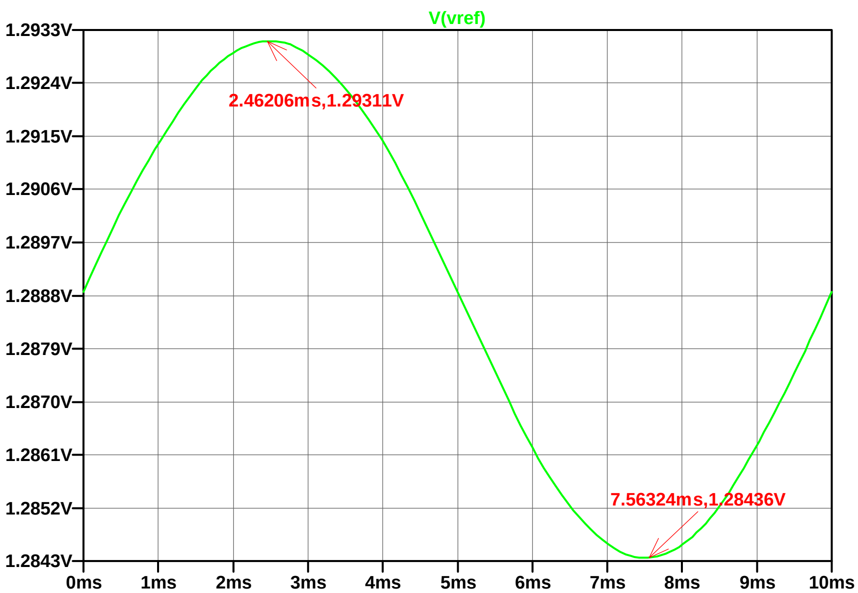


Figure 6.2.2 Example of a wmf file

# 6.3 Make Plot Curves/Lines and Letters Bigger

After you convert plots to wmf files by default and look at the graphs, you will notice that x/y labels are too small to see. In this case, you can make all texts and plot lines bigger.

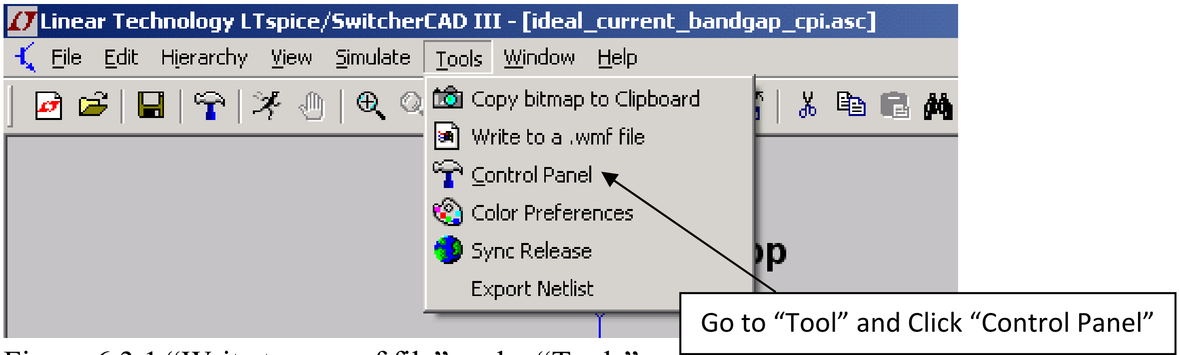



Figure 6.3.1 "Write to a .wmf file" under "Tools"

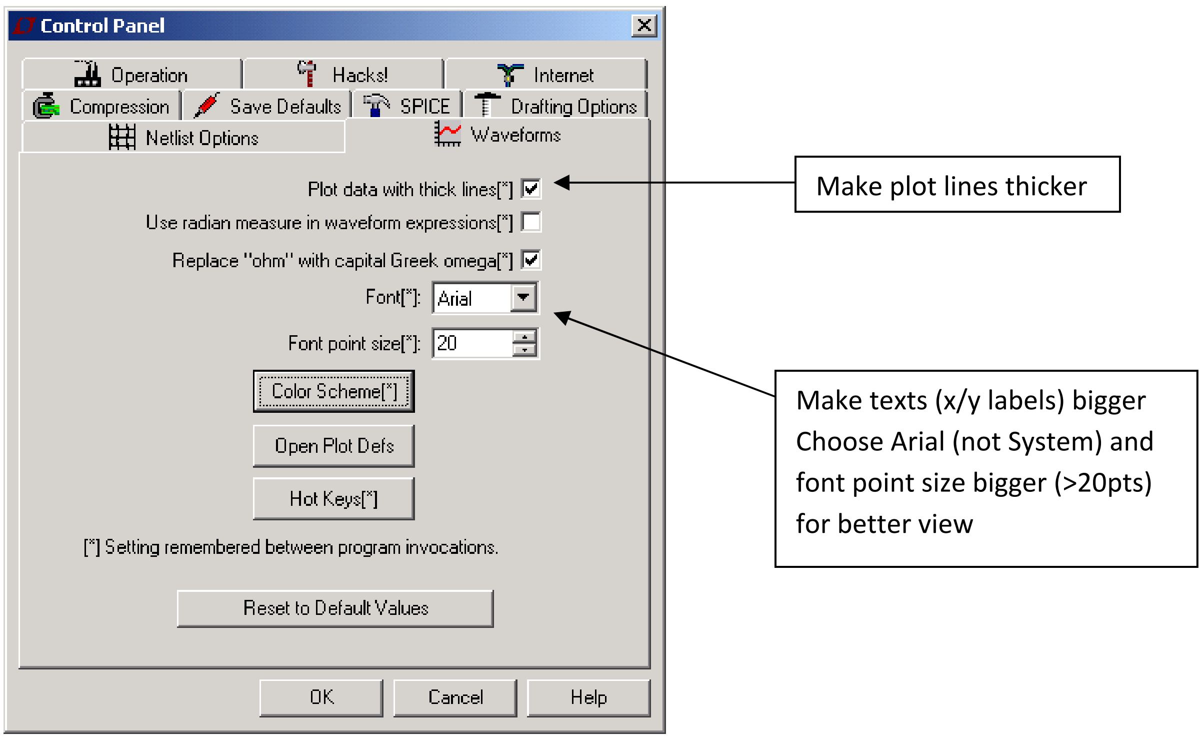
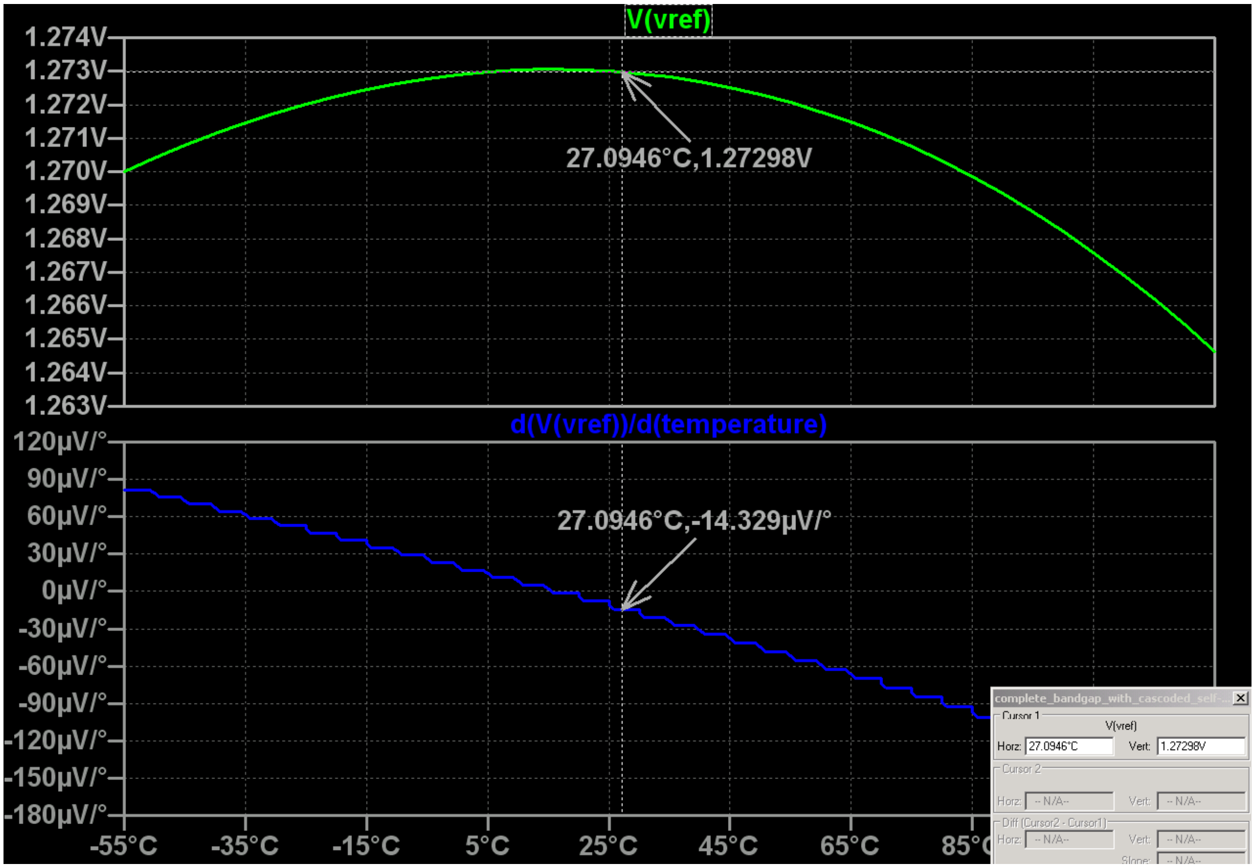



Figure 6.3.2 "Waveforms" Properties under "Control Panel"




# 7. Subcircuits

In LTSpice, if you need to design complex circuits and make some of the components in schematics as a block, you can build a subcircuit to make your schematic nicer.

1. Create a schematic that will be in your subcircuit and save the file.

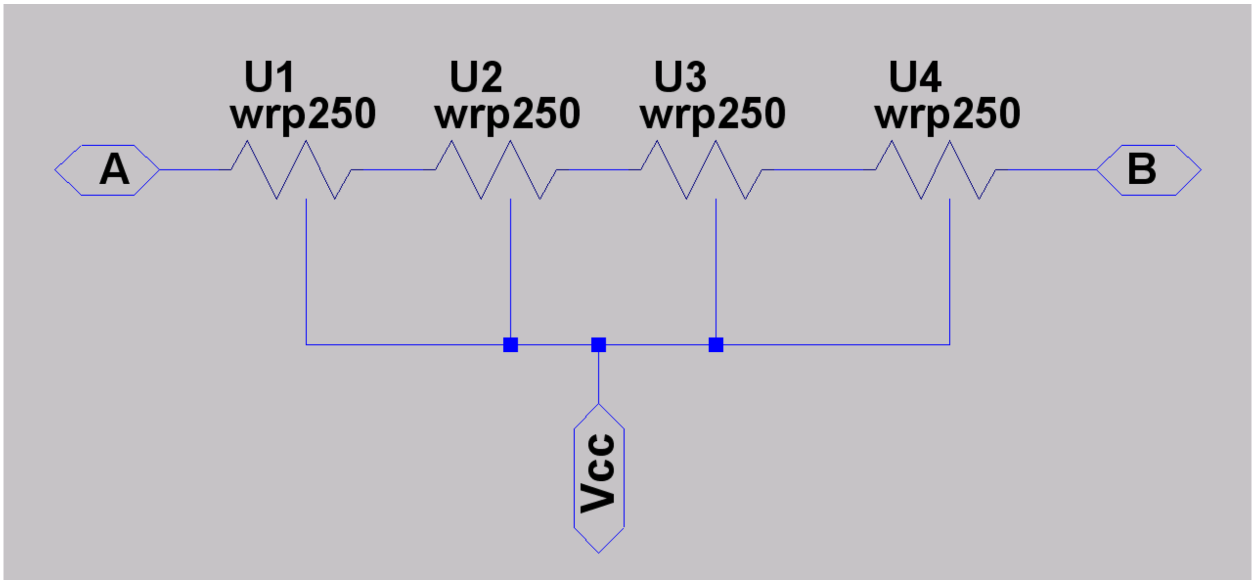


Figure 7.1 Subcircuit Schematic

2. Create a symbol and save it as .asy file. Make sure the schematic and symbol files are under the same directory.

a. Create a text from “Text” under “Draw” menu
b. Create a rectangle from “Rect” under “Draw” menu or simply type ‘ and r
- better to cover the text with the rectangle
c. Create a port by typing p
- make sure the pin name matches with the schematic created previously
d. Draw a line from “Line” under “Draw” menu or simply type ‘ and l
- connect the rectangle box to all ports
e. Save the file as .asy file under the same directory as the schematic

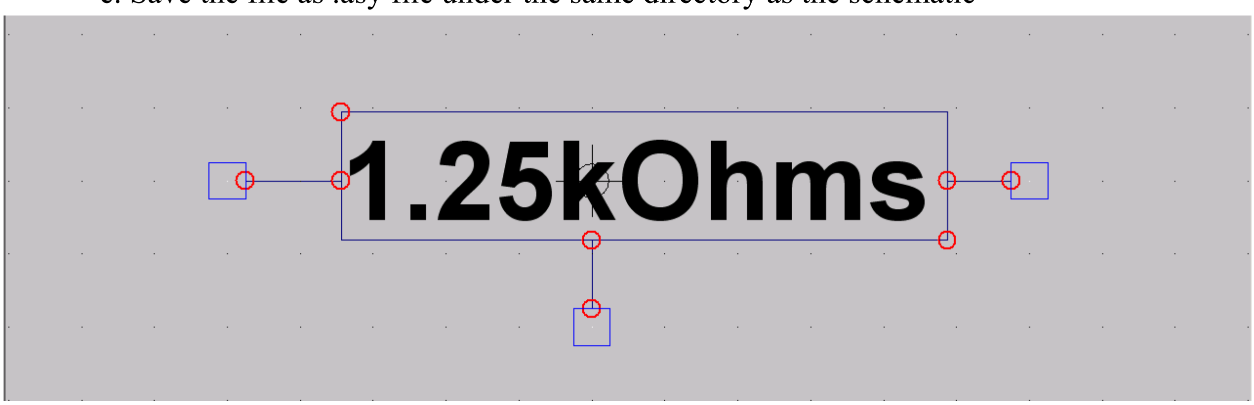
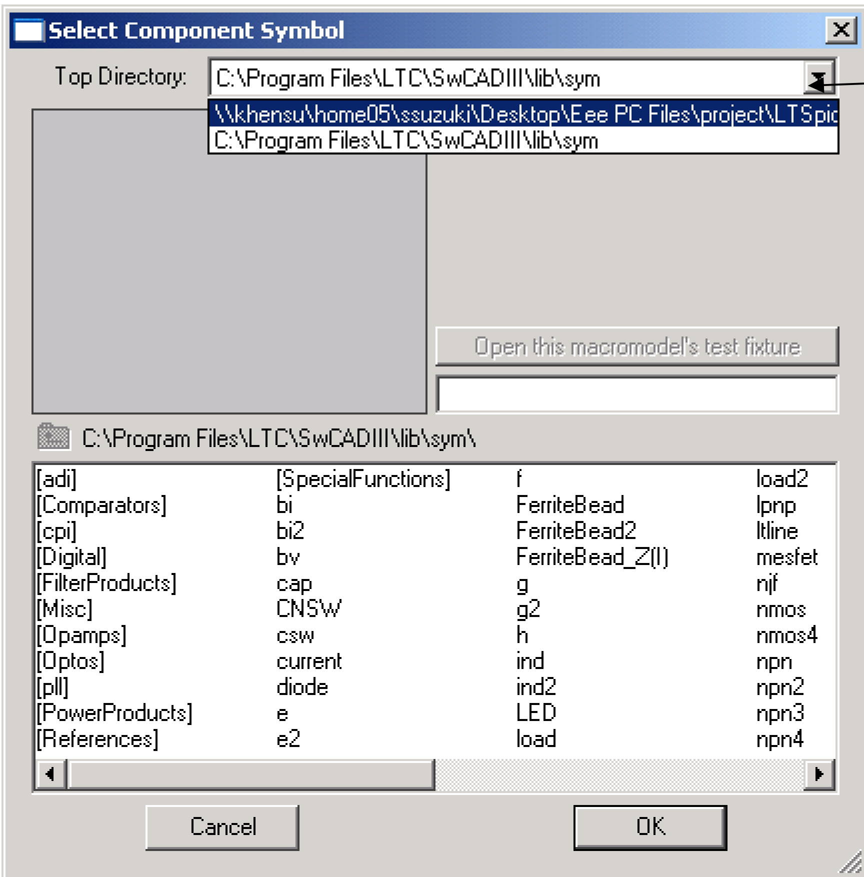


Figure 7.2 Subcircuit Symbol

3. Now, open a schematic that you want to add your subcircuit to.

a. add component by clicking F2

b. choose the directory where you saved your symbol file.




Choose the folder where you saved your symbol and schematic files from here.

Figure 7.3 Selecting Subcircuit Directory

c. Now you can pick your subcircuit.

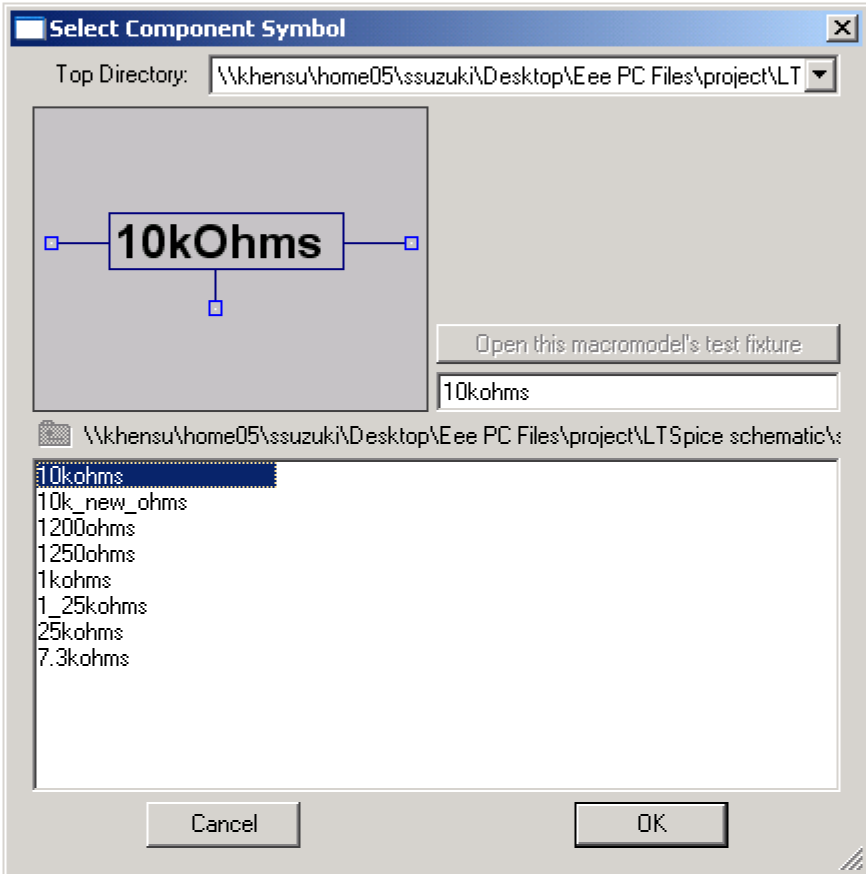


Figure 7.4 Selecting Subcircuit Symbol

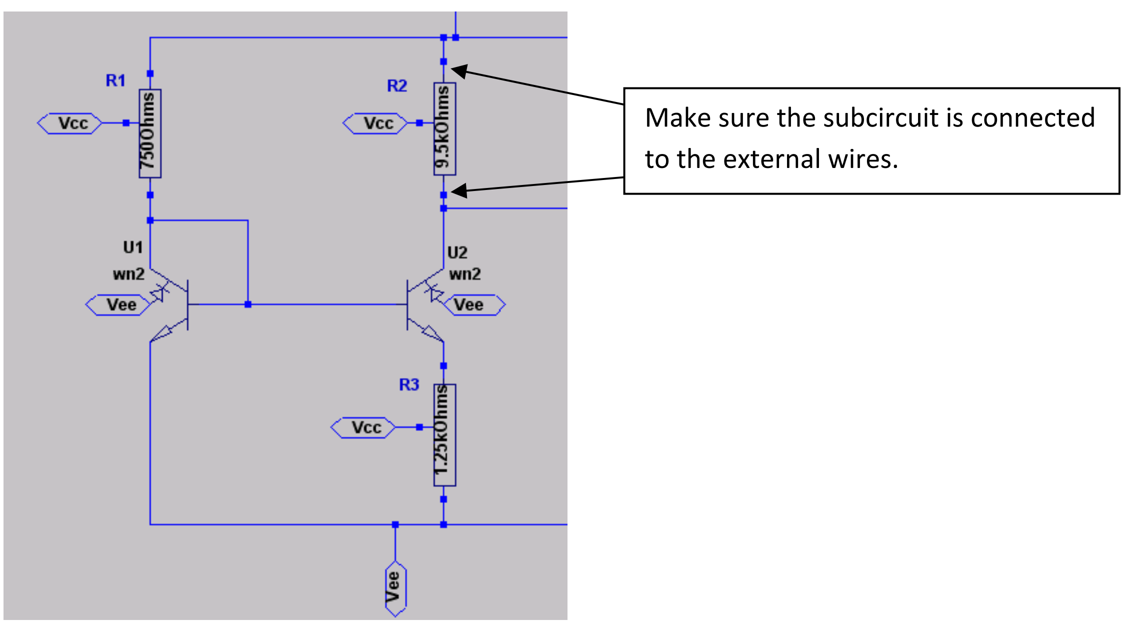


Figure 7.5 Subcircuits on a Main Schematic

# 8. Conclusion

LTSpice is very easy to learn and also a powerful tool. This report shows some of the LTSpice features. If I find more interesting features or useful tools, I would like to add more to this report.