

LTspice Tutorial

Teanette van der Spuy
u15107206@tuks.co.za



ENE 310

Wednesday 05 February 2020



UNIVERSITEIT VAN PRETORIA
UNIVERSITY OF PRETORIA
YUNIBESITHI YA PRETORIA

Denkleiers • Leading Minds • Dikgopolo tša Dihalefi

Installing LTspice

Option 1 : From ftp Server (You can do this at home!)

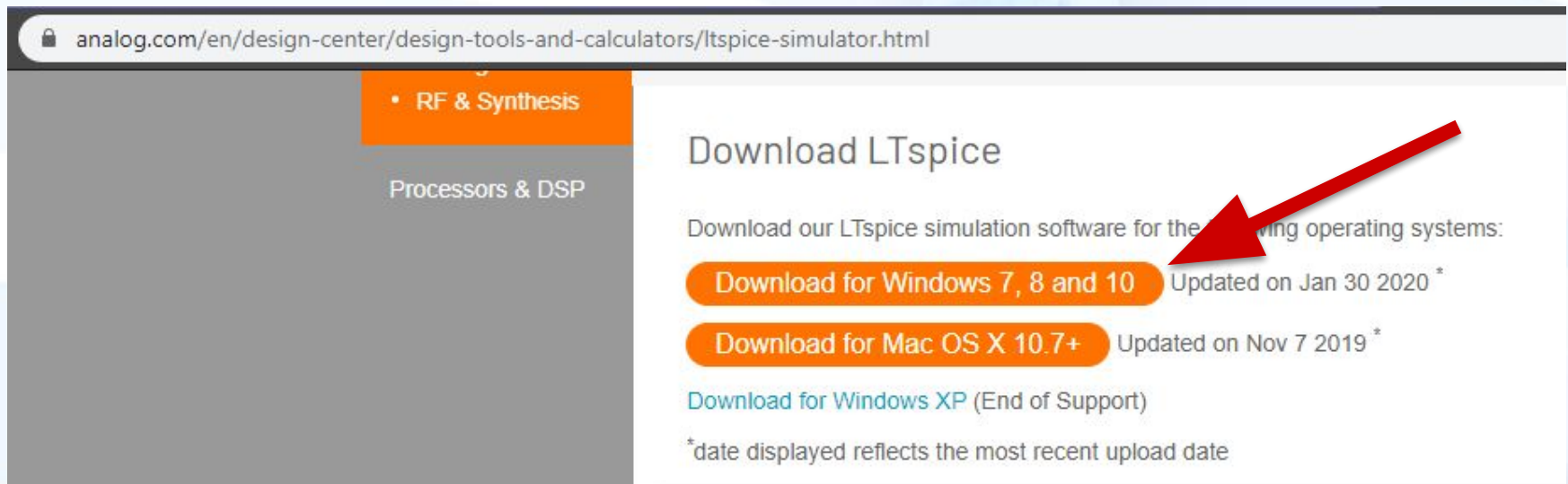
1. Go to : <http://ftp.ee.up.ac.za/pub/windows/eda/>
2. Download LTspice and install (LTspiceXVII_20191206.exe)



Installing LTspice

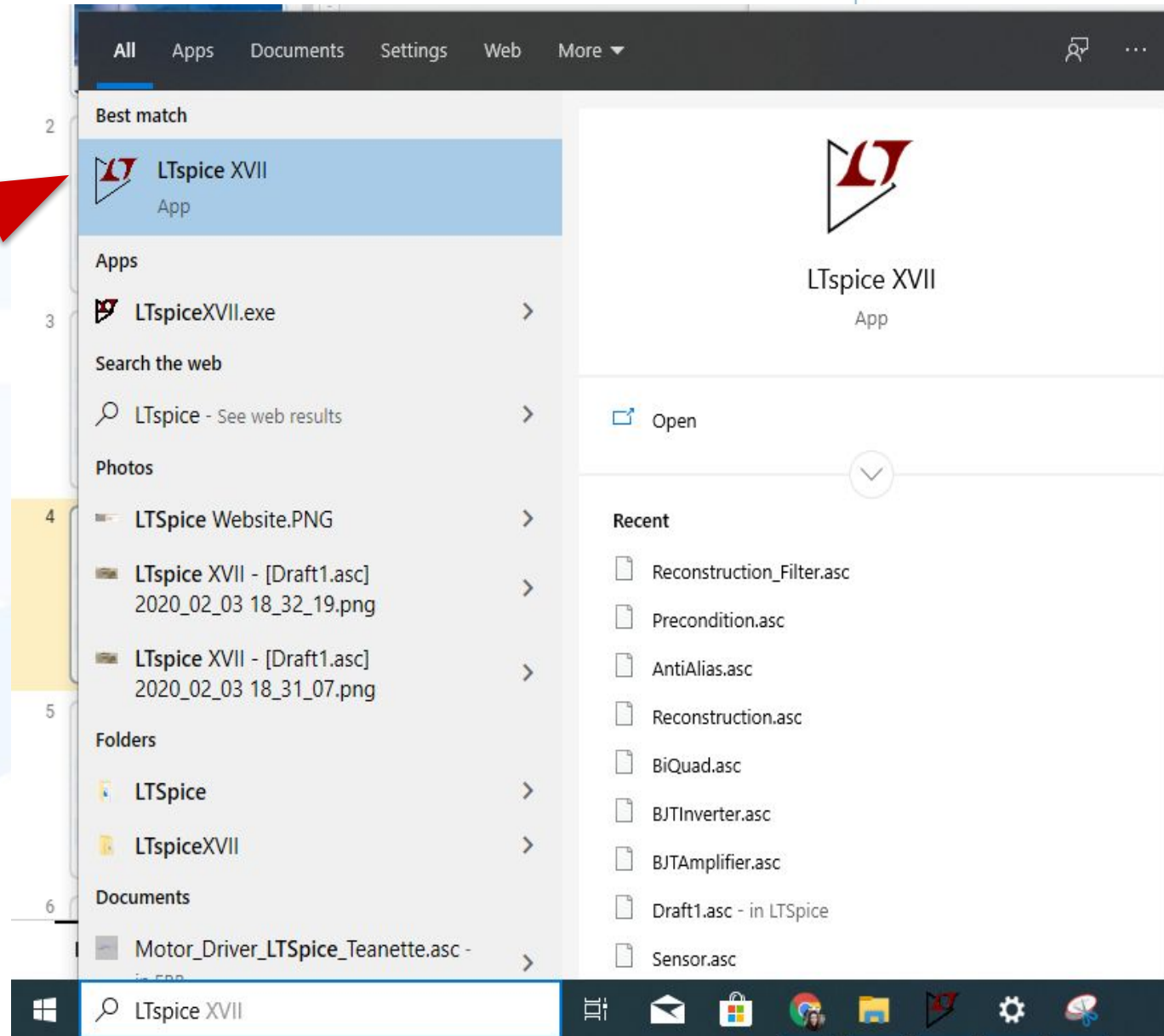
Option 2 : From Web Page (You can do this at home too!)

1. Go to LTspice Website (The link is <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>)
2. Download LTspice for Windows 7, 8 and 10 and install



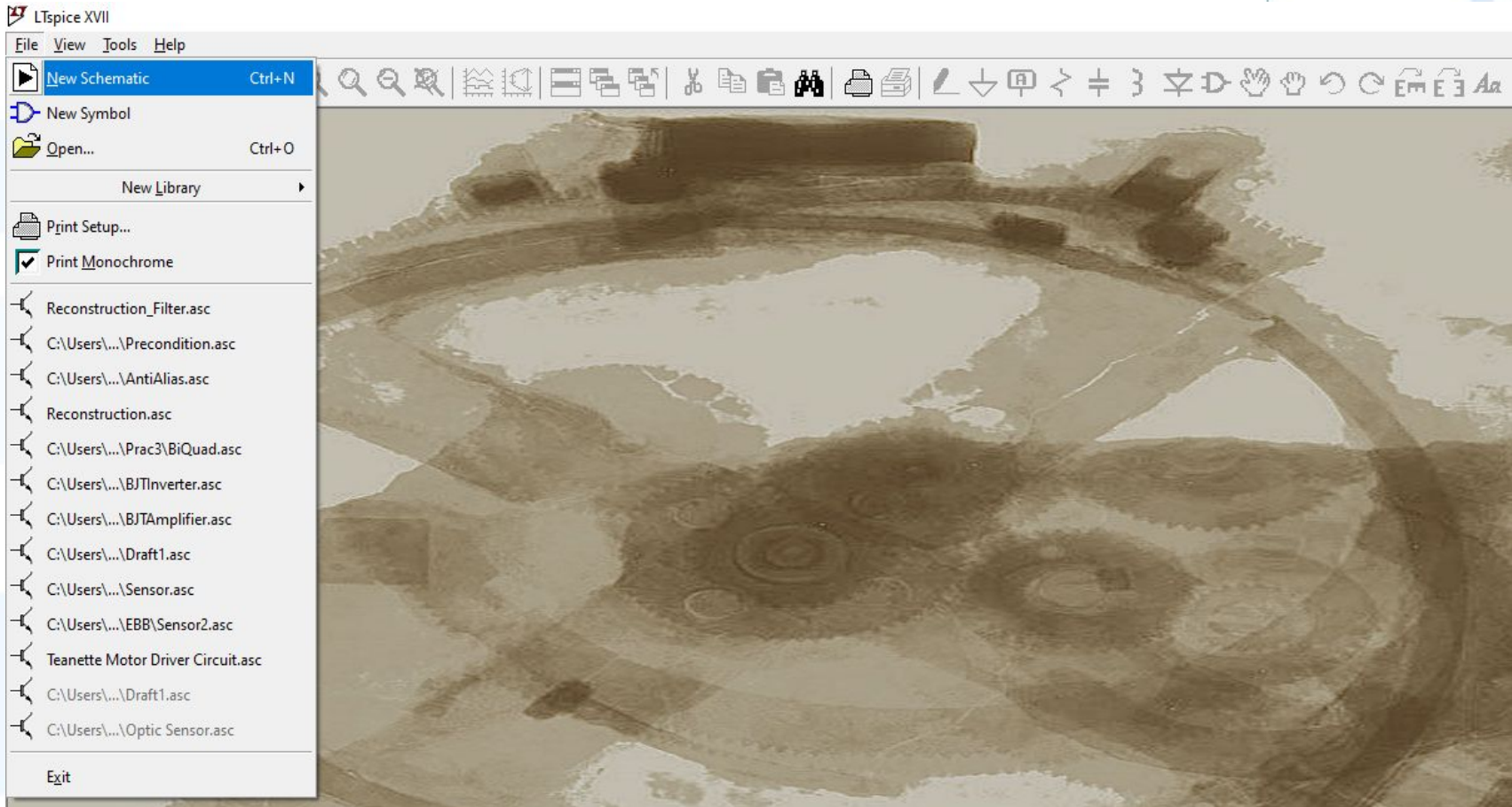
Setting up a New Project

1. Open LTspice



Setting up a New Project

2. Open a new Schematic



Some Useful Shortcuts (refer back to these)

F2 Place Part

F3 Wire Mode

F4 Place netname

F5 Delete

F6 Copy

F7 Move (This moves the part, but not the nodes it is connected to)

F8 Drag (This moves the connected nodes as well)

F9 Undo

G Gnd

R Resistor

C Capacitor

L Inductor

(You have to select the part with F7 first to use these three!)

ctrl-R Rotate

ctrl-E Mirror

ctrl-G Grid Toggle



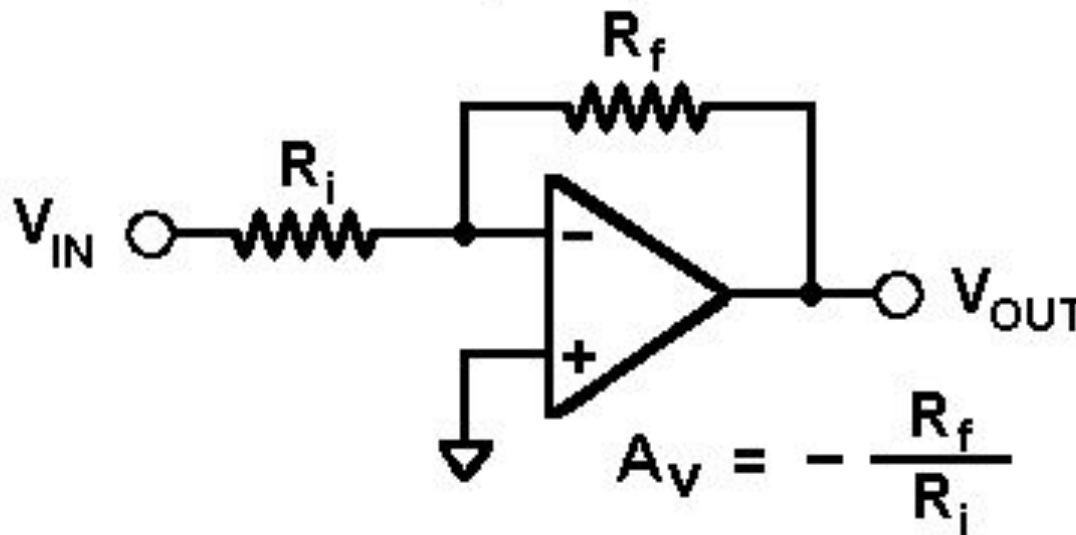
Inverting Amplifier Example



Inverting Amplifier Example

We are going to build and simulate the operation of a basic inverting amplifier with a gain of 2. The schematic and design equations are shown below.

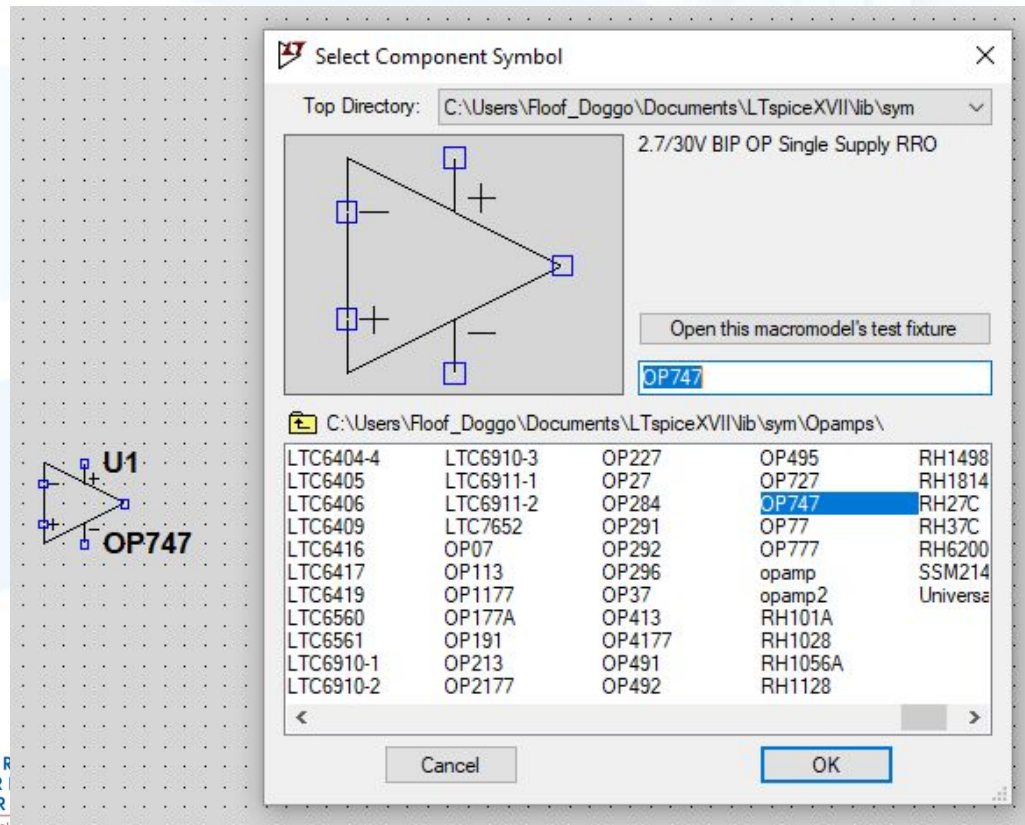
Inverting Amplifier



Inverting Amplifier Example

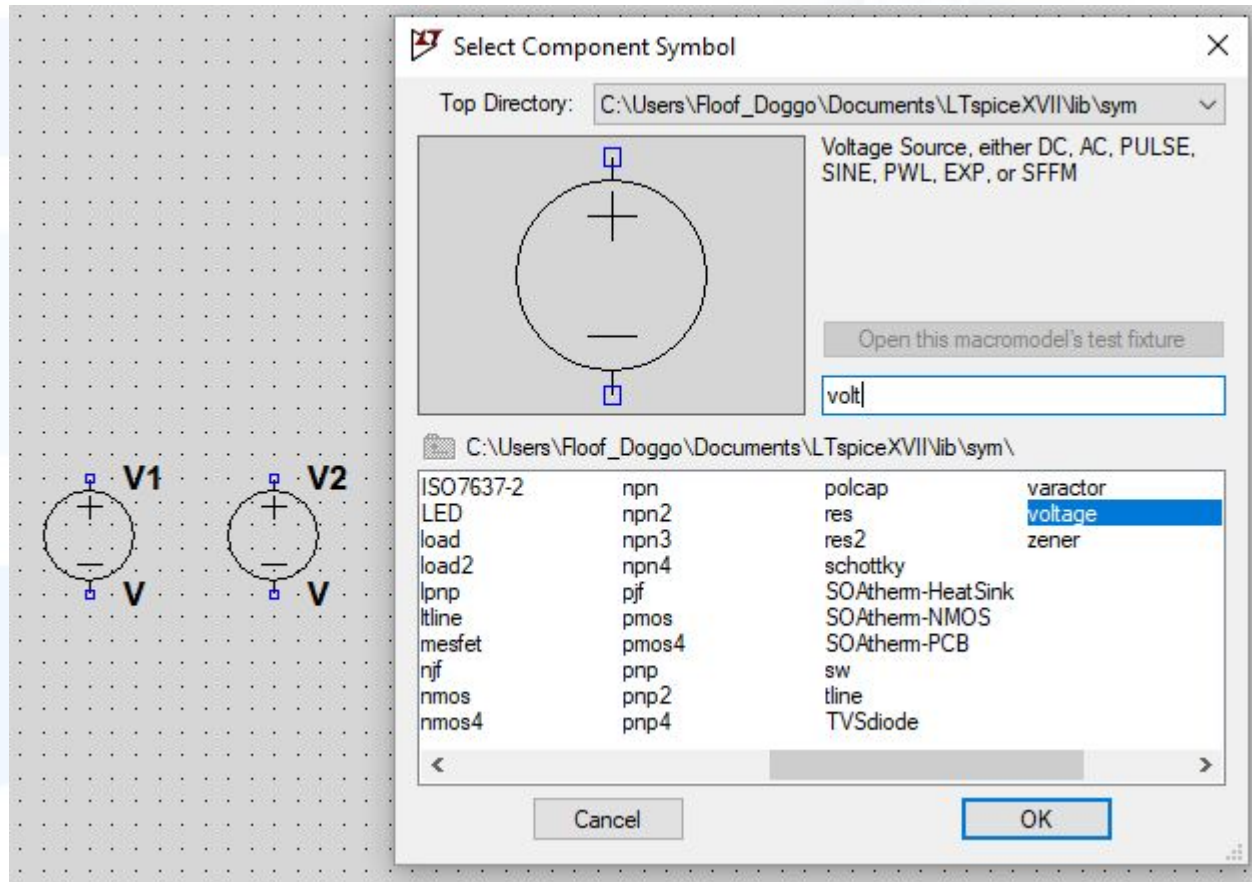
1. Press F2 to select a part
2. Select op-amp OP 747 and place the part on your schematic

(Note, OP 747 is just the op-amp model we use in this example. There are many different models on LTspice and even more that you can download. It is your responsibility to find the model that best corresponds to the op-amp you are using)



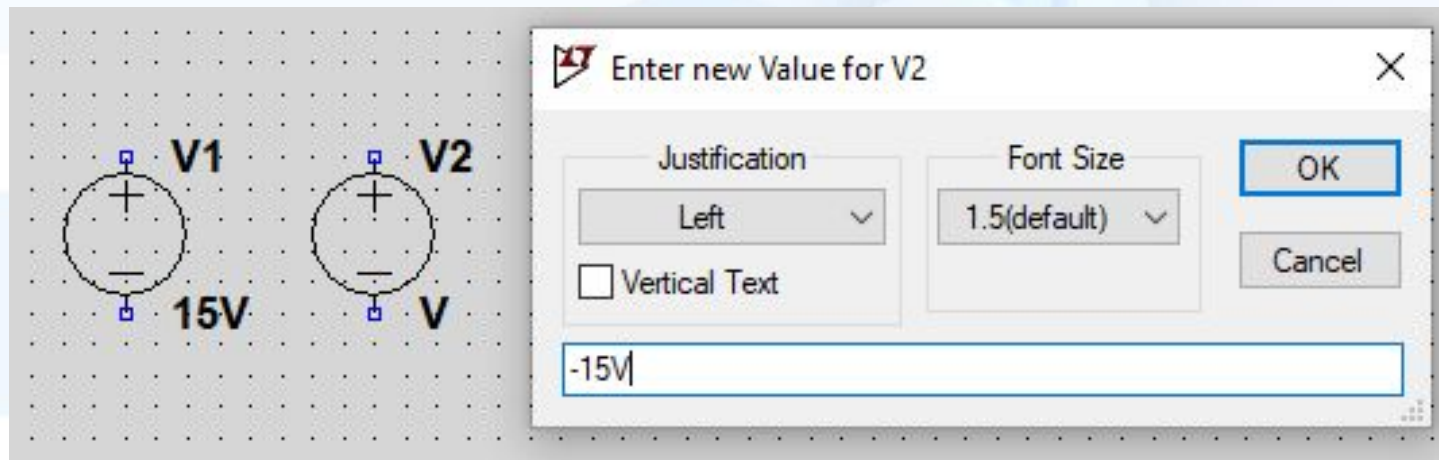
Inverting Amplifier Example

- Press F2 to select a DC voltage supply to power your op-amp (you will need a negative and a positive supply)



Inverting Amplifier Example

4. The text **V1** and **V2** at the top right of each component are the names of the components. The **V** symbol at the bottom left is the value of the component. You can edit this text by right-clicking on it.
5. Enter the value 15V for the one part and -15V for the other. It is important that the negative and positive power supplies are symmetrical.
(You can rename your voltage supplies in the same way too, but that is not important for this example)

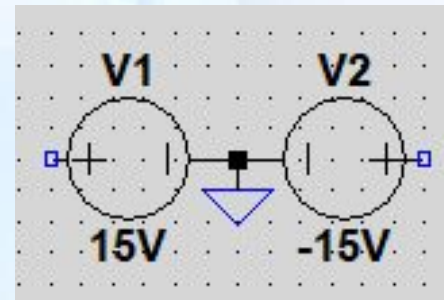
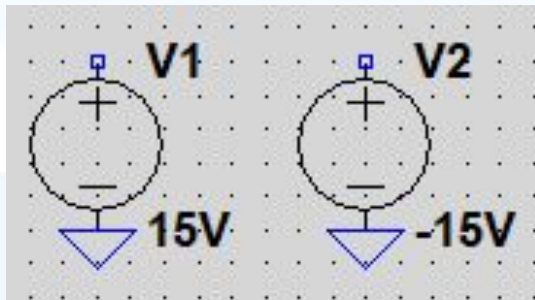


Inverting Amplifier Example

6. Don't forget to ground your power supplies!!!!!!

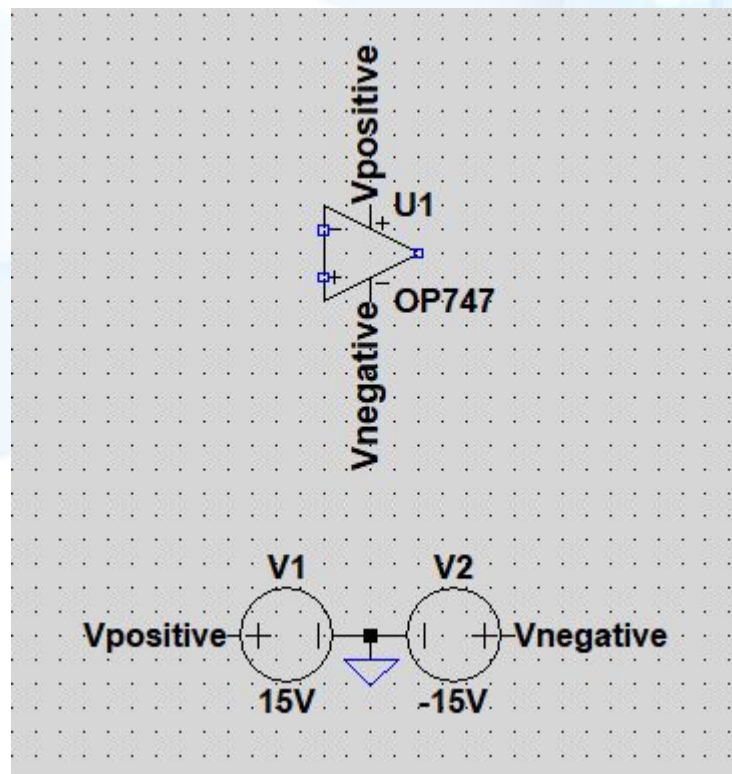
Press G for the ground symbol. You can place a ground node at each power supply, or connect both supplies to the same node - it doesn't matter! All the ground nodes on a schematic are common by default (they are all connected anyway)

Remember that the value you typed in at **V** in the previous step is the voltage at the positive node of the power supply.



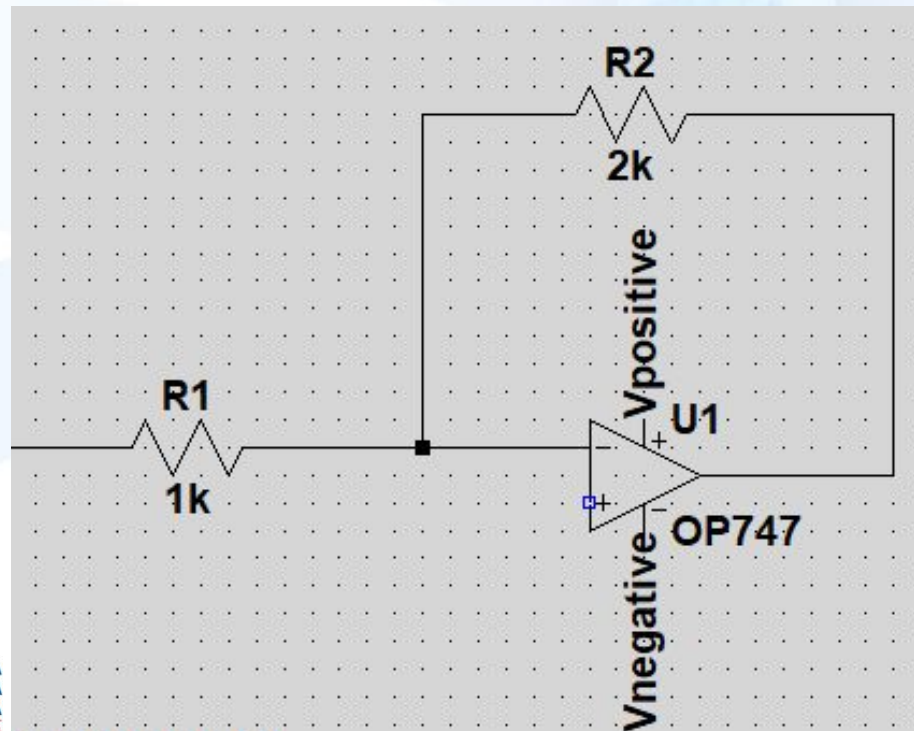
Inverting Amplifier Example

7. LTspice connects nets that have the same name (a net is like a node or wire). We can use this to our advantage to reduce the amount of wires in our schematics and make them more readable. Use F4 to name the net at the positive end of each power supply **Vpositive** and **Vnegative** respectively. Do the same for the negative and positive rails of your op-amp.



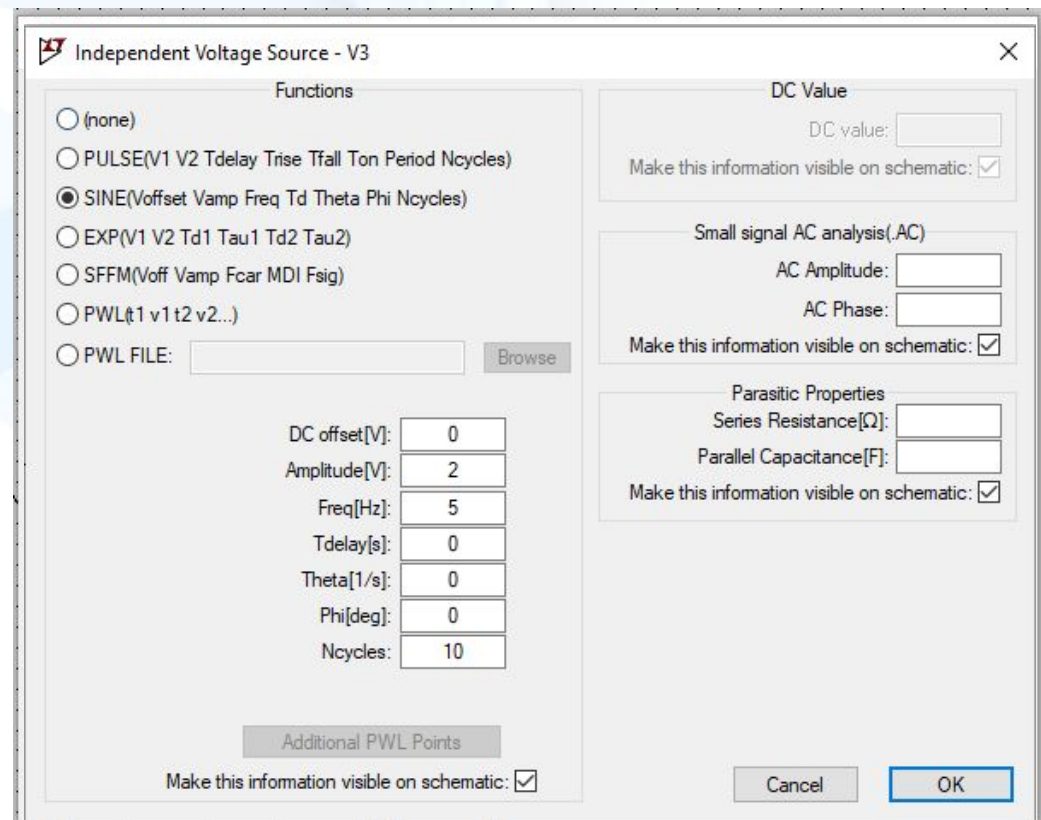
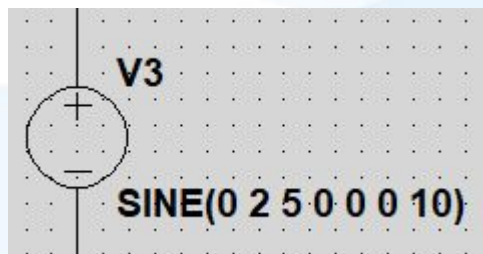
Inverting Amplifier Example

8. Press R to select a resistor. You can use ctrl-R to rotate them.
9. Place 2 resistors on the diagram, one at the negative terminal of the op-amp and one above the op-amp.
10. Right-click on the **R** of the resistor to change its value. Make the top resistor 2k ohm and the bottom resistor to 1k ohm.
11. Press F3 for the wiring tool. Connect the components as shown below.



Inverting Amplifier Example

12. We want to insert an AC voltage supply as an input to your op-amp. To do this, first press F2 and insert a DC voltage supply, as in step 3. Next right-click on the component itself. Click on the **Advanced** button.
13. Select the SINE option. Here a menu allows you to set the characteristics of your AC input (which is a sinusoidal voltage signal). Fill in the options as shown below.



Independent Voltage Source - V3

Functions

- ☐ (none)
- ☐ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- ☒ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- ☐ SFFM(Voff Vamp Fcar MDI Fsig)
- ☐ PWL(t1 v1 t2 v2...)
- ☐ PWL FILE: Browse

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis(.AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

Make this information visible on schematic: ☒

Additional PWL Points

Make this information visible on schematic: ☒

DC offset[V]:

Amplitude[V]:

Freq[Hz]:

Tdelay[s]:

Theta[1/s]:

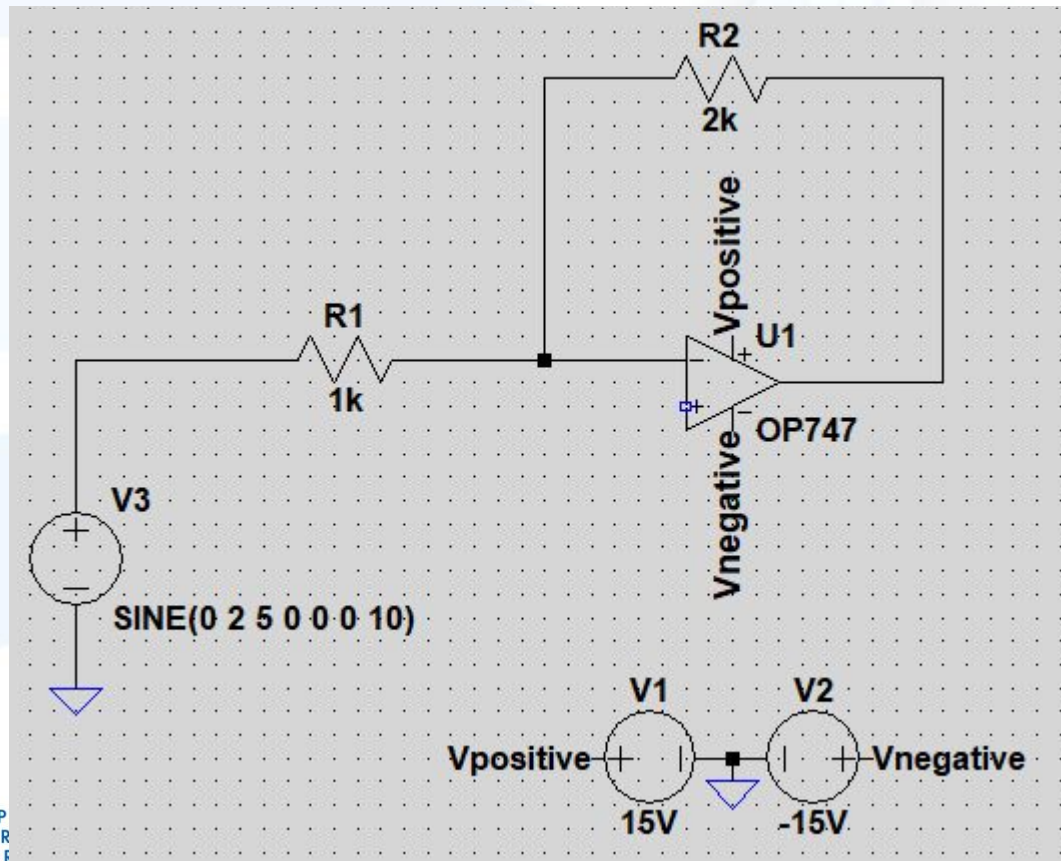
Phi[deg]:

Ncycles:

Cancel OK

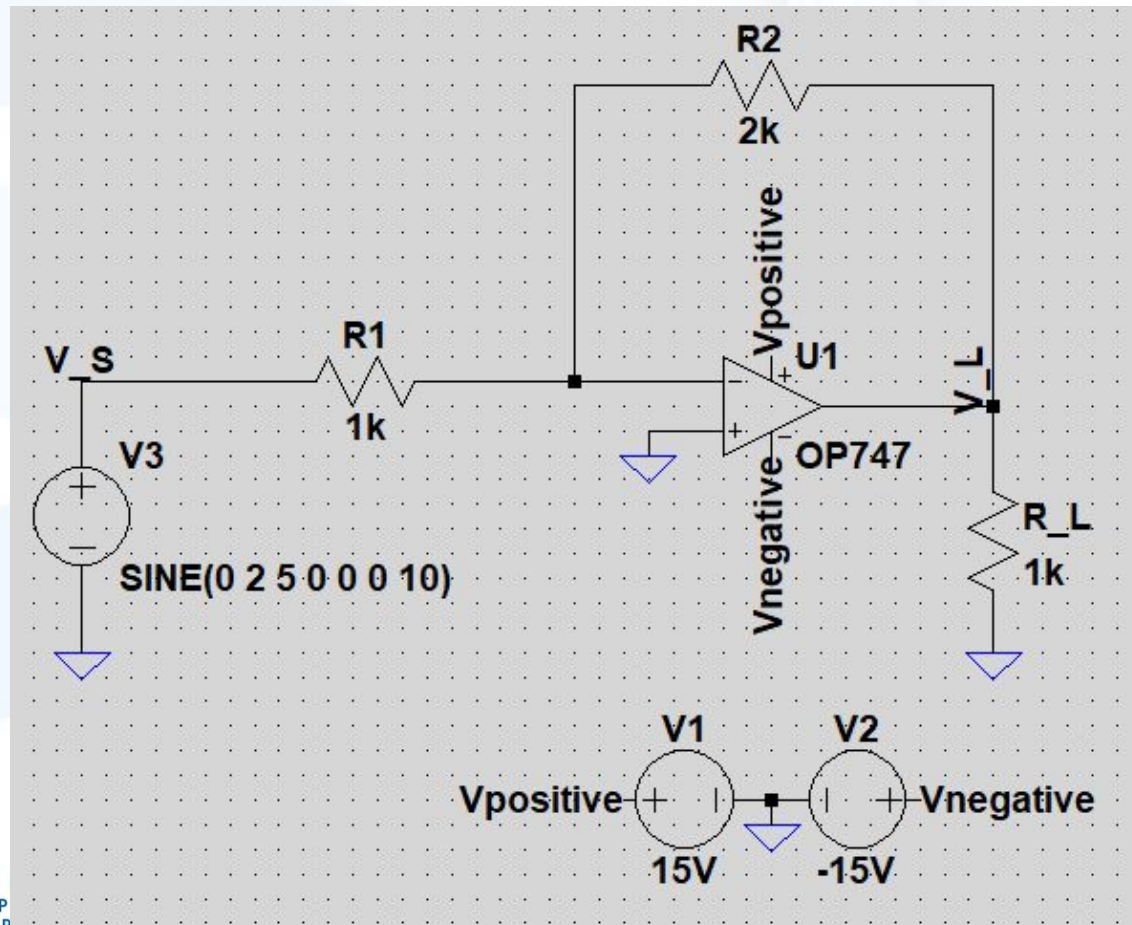
Inverting Amplifier Example

14. Use G and F3 to add a ground and connect it to the positive terminal of the op-amp.
15. Use R and F3 to add a load and connect it to the output of the op-amp. Make the value of the load 1k. Ground the load.



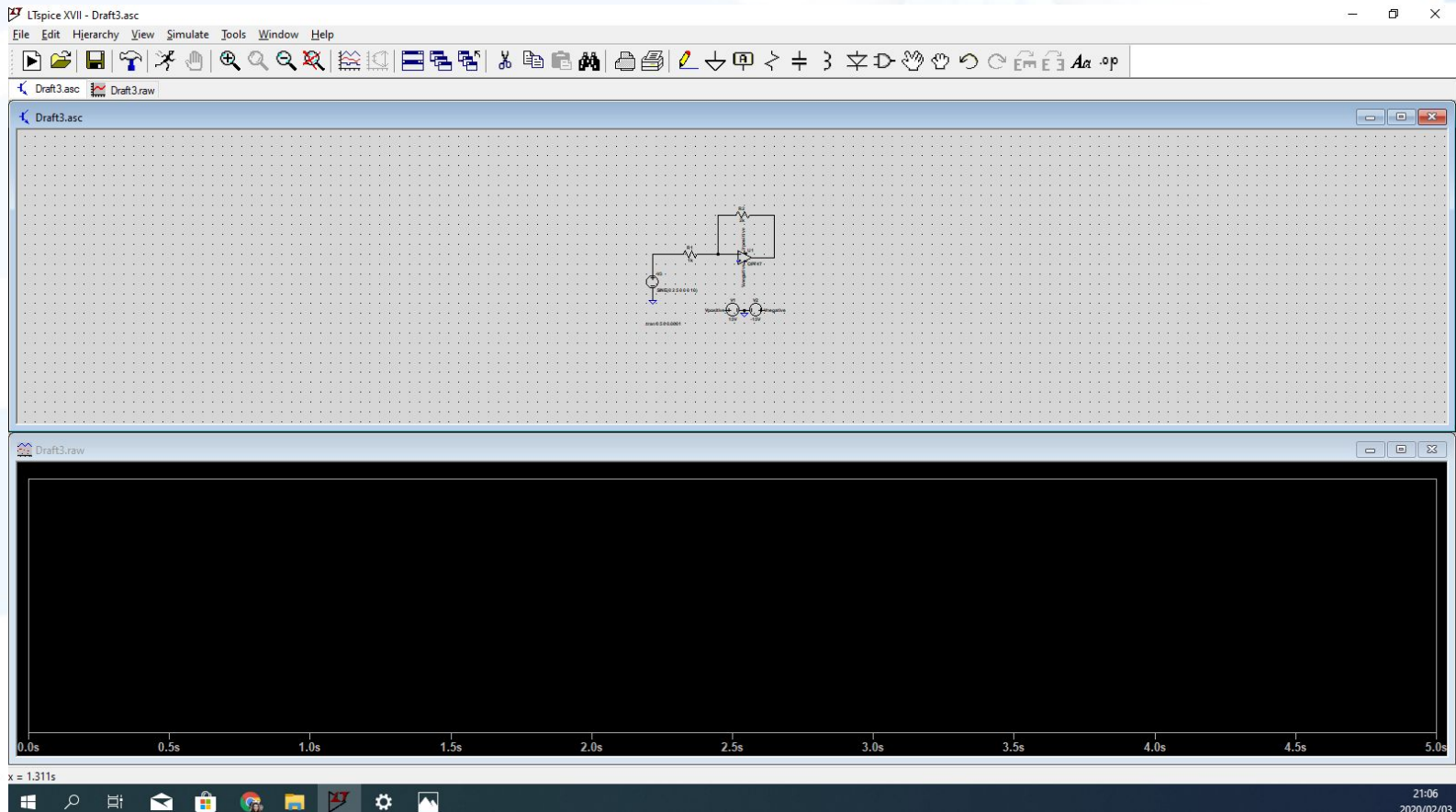
Inverting Amplifier Example

16. Right-click on the name of the load resistor and rename it R_L (R-Load).
17. Use F4 to rename the node above the load to V_L (V-Load). Do the same to the node above the AC input and name it V_S (V-Source).



Inverting Amplifier Example

18. Once you press OK on the simulation menu, your screen should be split in 2 - your schematic on top, and a blank screen on the bottom. This is where your simulation results will appear. You can maximize your simulation screen and make it a new window.



Inverting Amplifier Example

20. When you move your cursor over your schematic, it should now change shape and look like a probe.

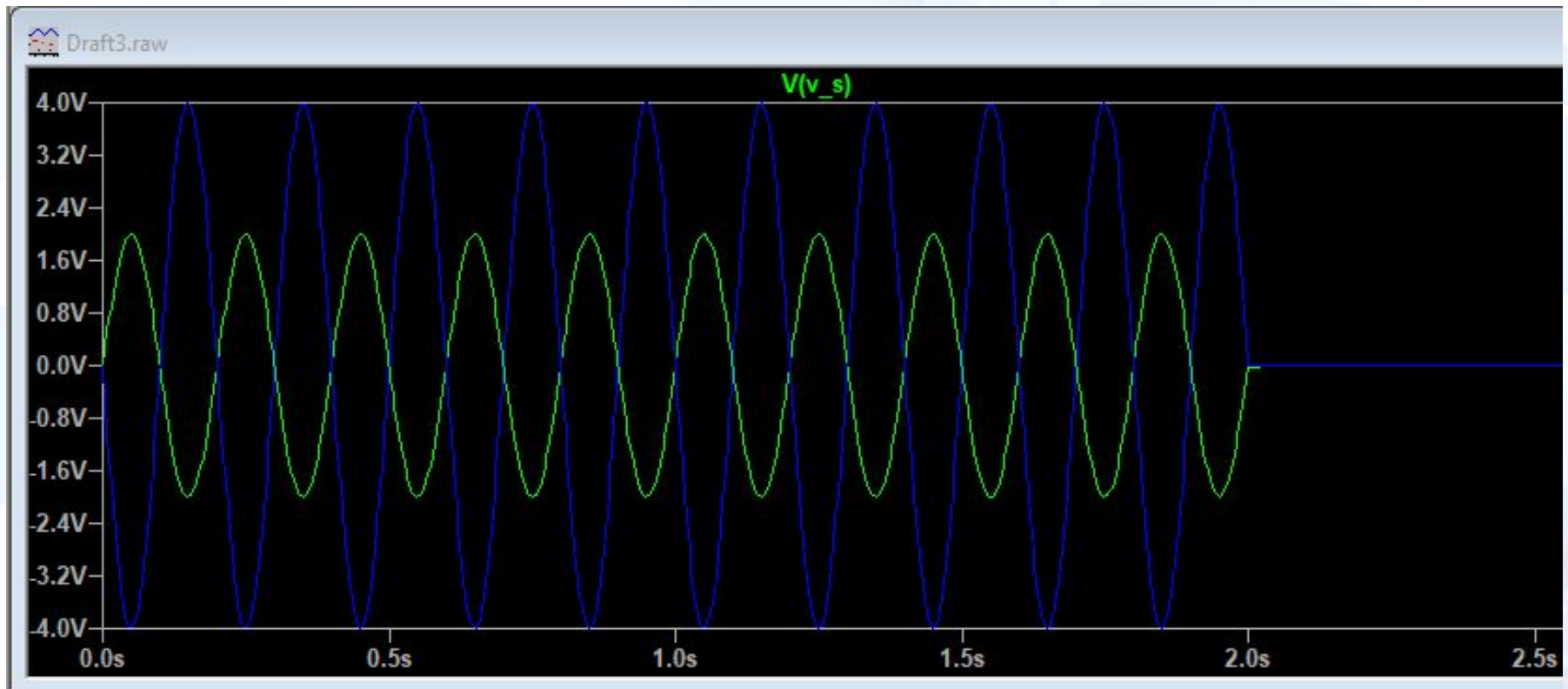


Voltage probe cursor



Current probe cursor

20. Click on the node marked V_S. Next click on the node marked V_L. Note the names, amplitudes and no. of cycles of each signal.

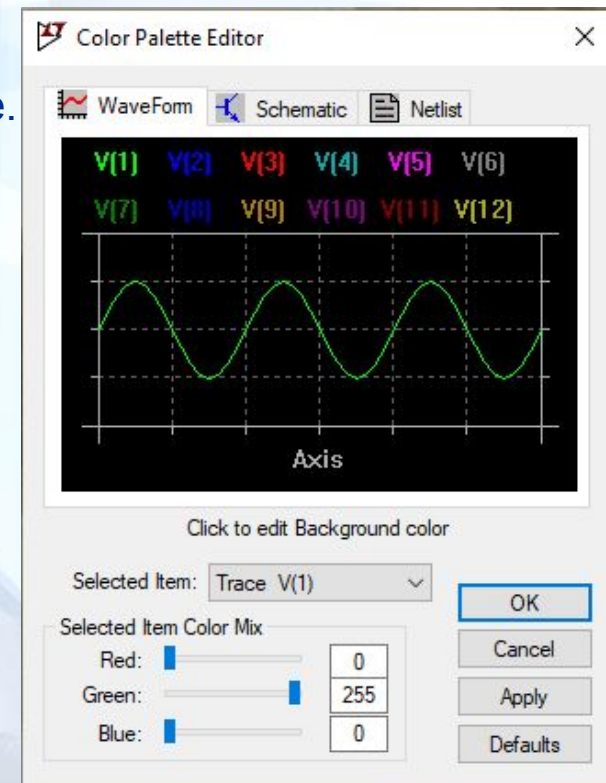


Inverting Amplifier Example

22. Do NOT put your graphs in your report with this black background!!!

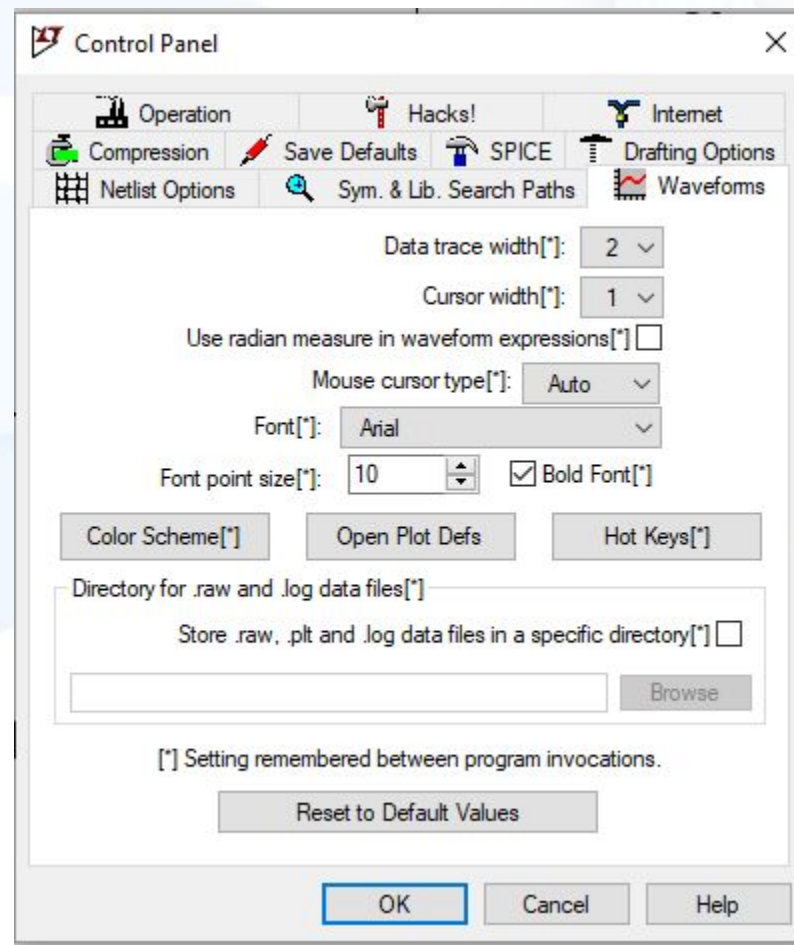
Instead, make sure your graphs are readable, neat and intuitive. Always make sure that your signals are labelled and that you refer to the signals in the text using the same label. You can change the colour of your graphs as well as your schematic under **Tools => Color Preferences**.

For your reports, make the background colour of your graphs white and make sure that the traces are visible. If you are including detailed schematics in your report (you will be) it is a good idea to edit the colours of your schematic as well **OR** to redraw your schematic using draw.io for a very professional look.



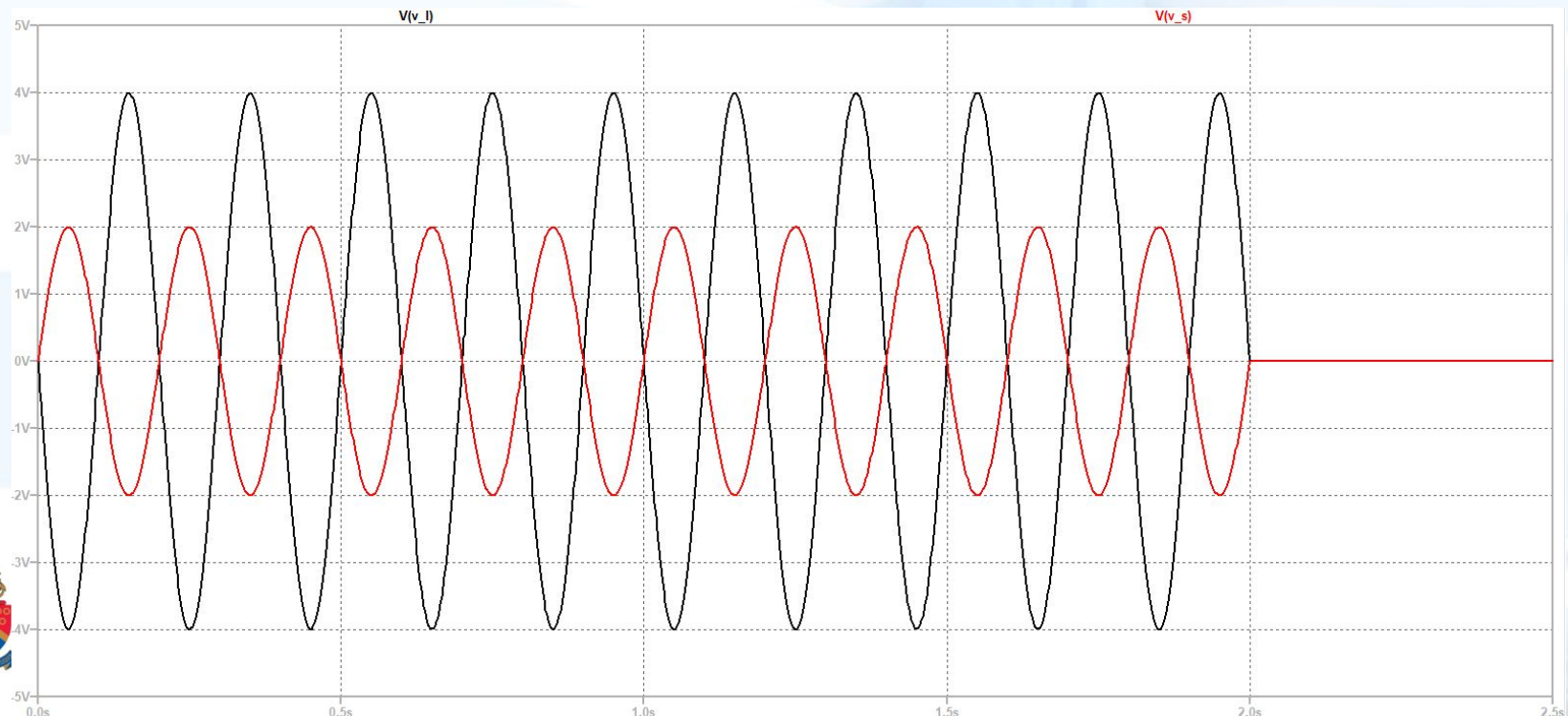
Inverting Amplifier Example

23. You can further edit your graph by going to **Simulate => Control Panel**. Select the tab **Waveforms**. Here you can change the format of the font displayed on the graph as well as other aspects, such as trace width.



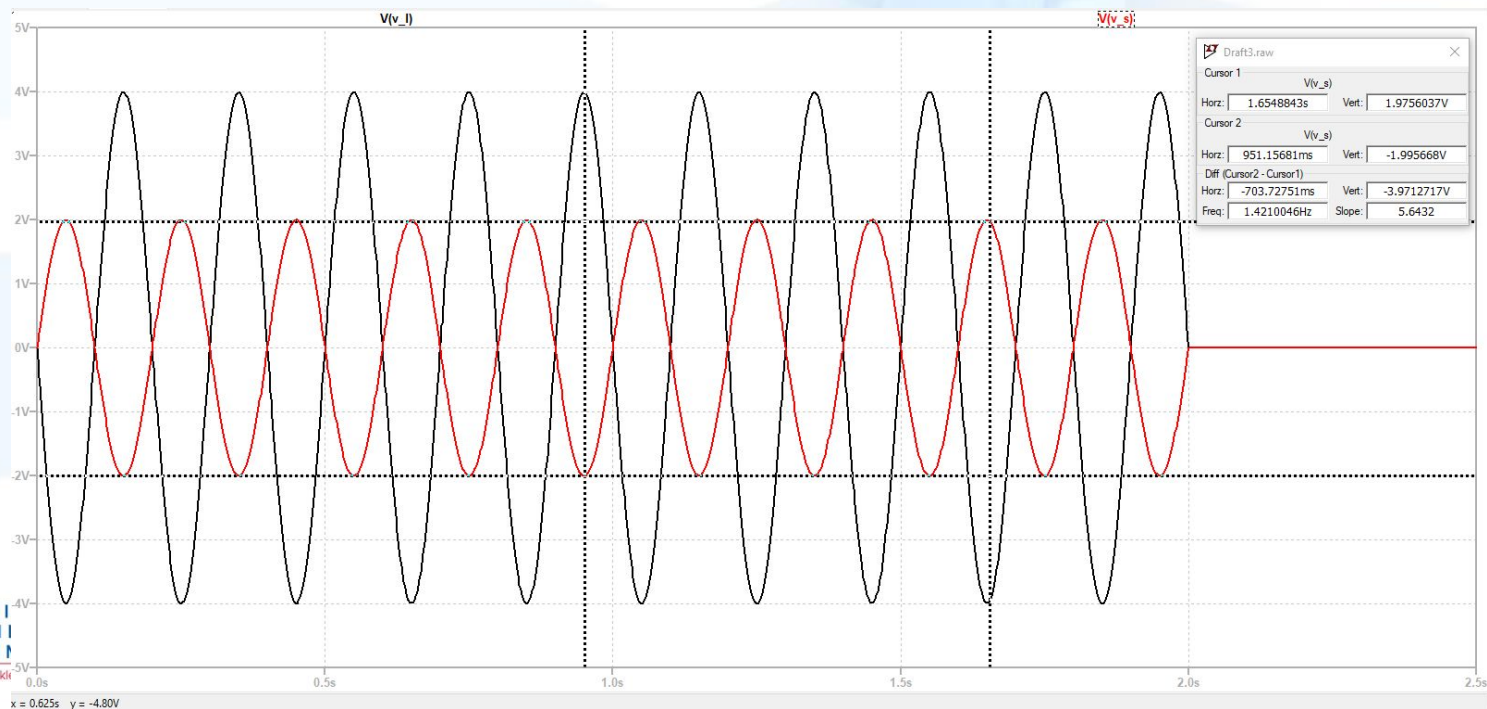
Inverting Amplifier Example

24. Change the background colour of your graph to white. Change the colour of V_L to black, and V_S to red.
25. Increase the width of your trace to 2.
26. Make sure your grid is visible (right-click on the graph => View => Grid.)
27. Right-click on the x-axis of your graphs and set your right axis limit to 2.5s. Set your y-axis limit to 5 V and -5V respectively.



Inverting Amplifier Example

28. You can add cursors to your trace by clicking on the trace name found just above the graph. One click = one cursor, while double-clicking = two cursors. A pop-up box will appear which detail their positions. Always include this box in your graph if you want to indicate a certain point using cursors. You can edit the cursor appearance from the simulation Control Panel. Add two cursors to your graph as shown.

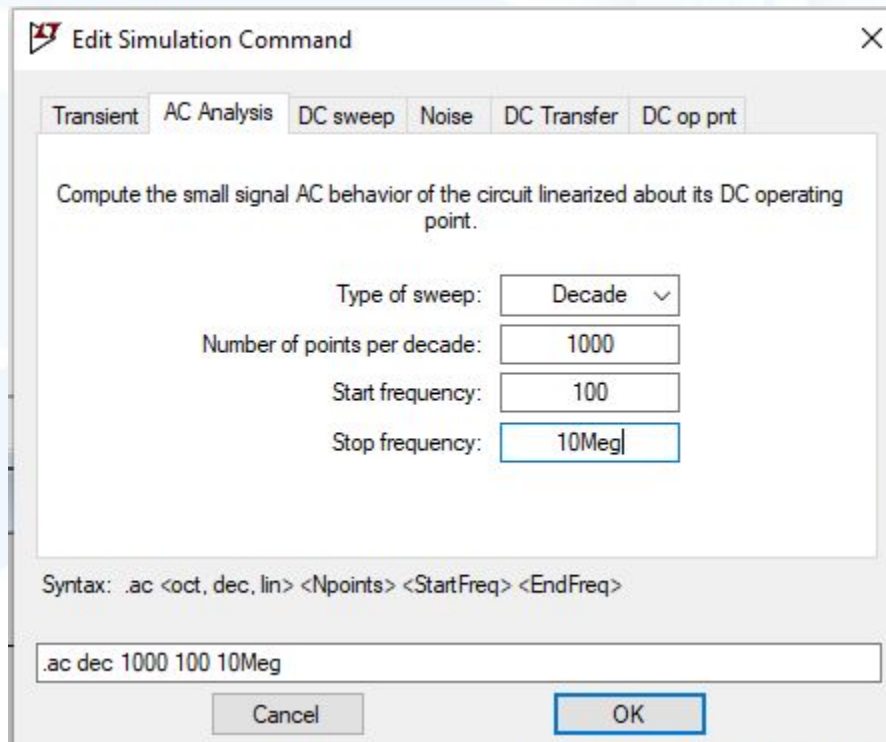


AC Analysis



AC Analysis

1. Now we want to analyse our inverting amplifier schematic in another way: AC Analysis. The first step is to edit the simulation profile. This can be done from **Simulate => Edit Simulation Cmd.**
2. Select the AC Analysis tab and fill in the profile as shown below.



Edit Simulation Command

Transient AC Analysis DC sweep Noise DC Transfer DC op pnt

Compute the small signal AC behavior of the circuit linearized about its DC operating point.

Type of sweep: Decade ▾

Number of points per decade: 1000

Start frequency: 100

Stop frequency: 10Meg

Syntax: .ac <oct, dec, lin> <Npoints> <StartFreq> <EndFreq>

.ac dec 1000 100 10Meg

Cancel OK

AC Analysis

3. Once you press OK, you will have to place your simulation command text on your schematic. You can put it anywhere.

```
.ac dec 1000 100 10Meg
```

4. You should also notice that the text of your previous simulation command has changed.

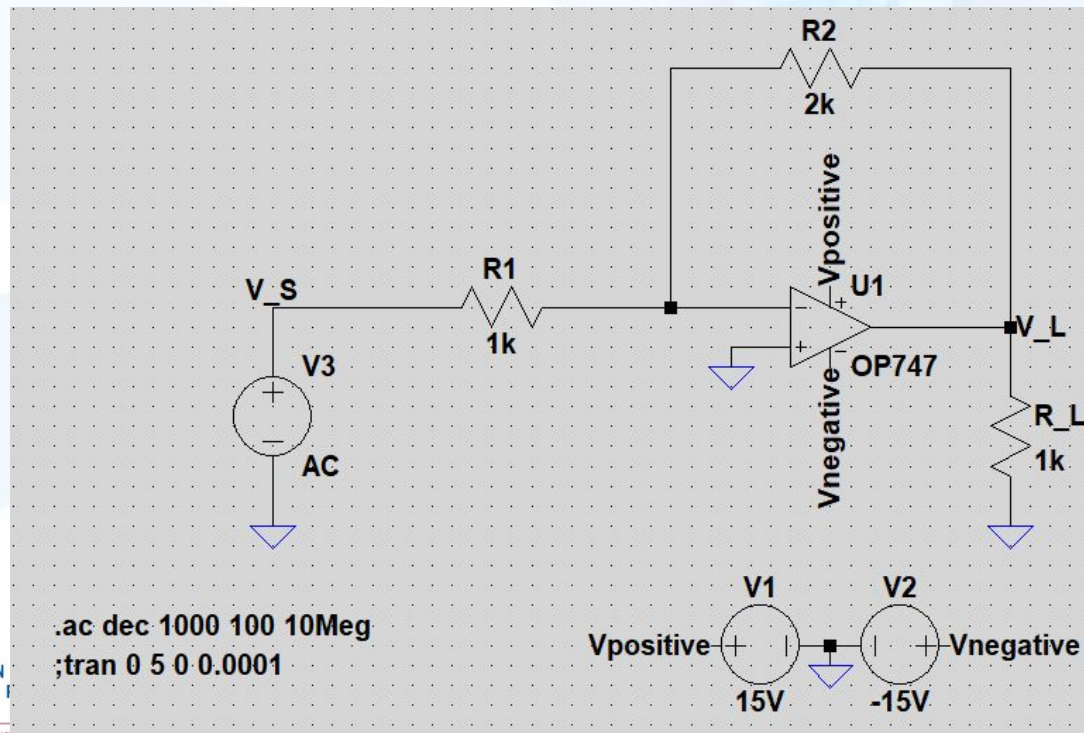
```
;tran 0 5 0 0.0001
```

Notably, where there used to be a period there is now a semicolon. This is LTspice's way of "commenting out" the simulation, or making it inactive. If you want to run the transient simulation again, simply change the semicolon back to a period, and conversely replace the period in front of your AC analysis command with a semicolon.



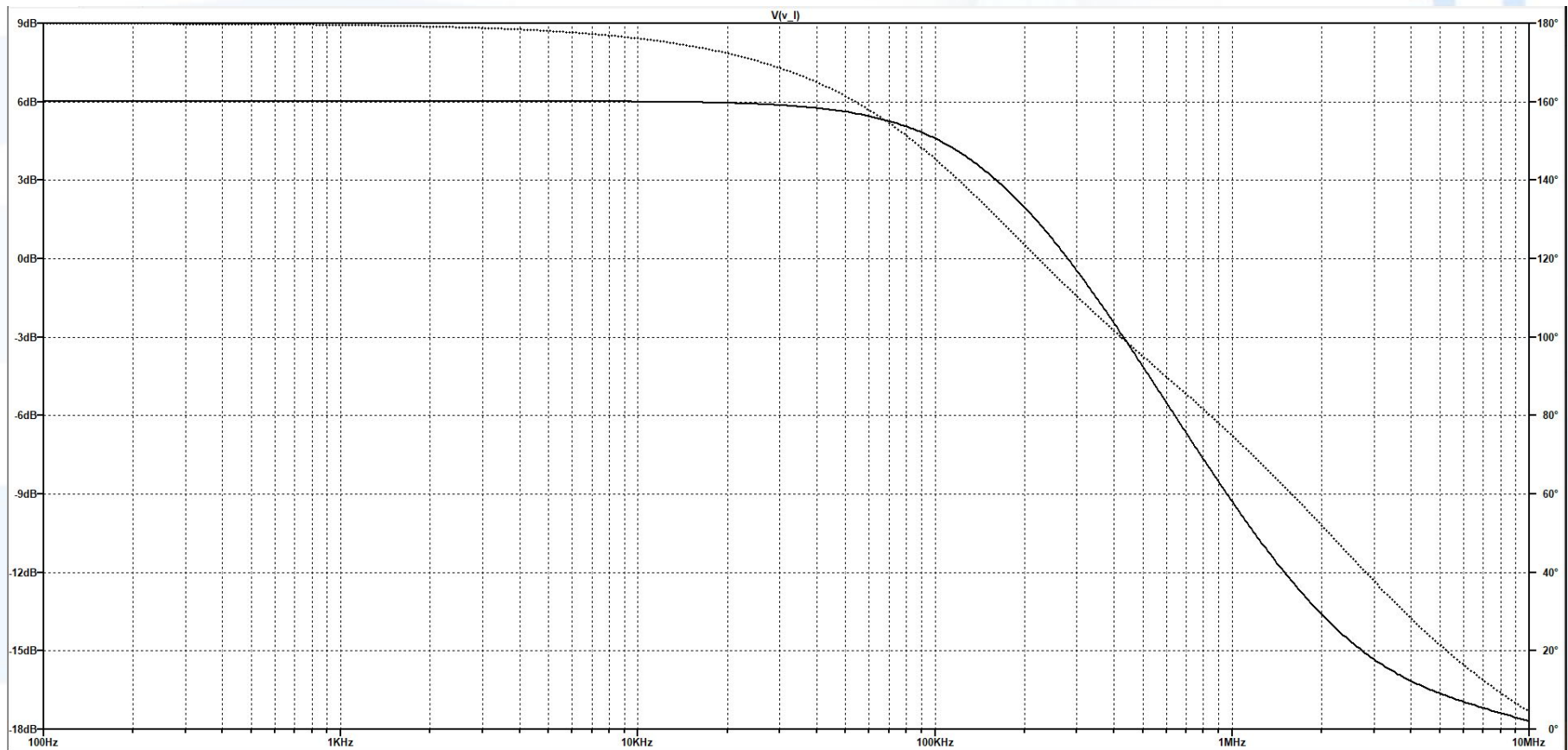
AC Analysis

- Before you can run your simulation, you need to tell it which source it can use as an AC input to the system. The simulation then sweeps that source over the specified frequencies and simulates the circuit's response to it. In our case, we want to use the voltage input V3 as our AC source. Change the value of V3 to AC.



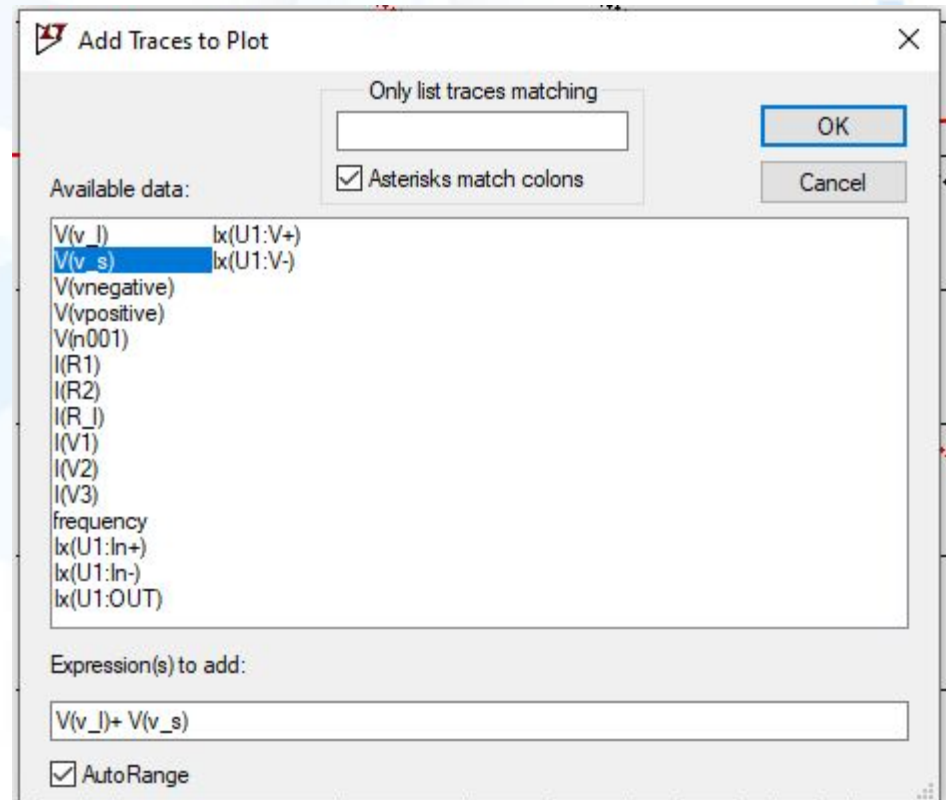
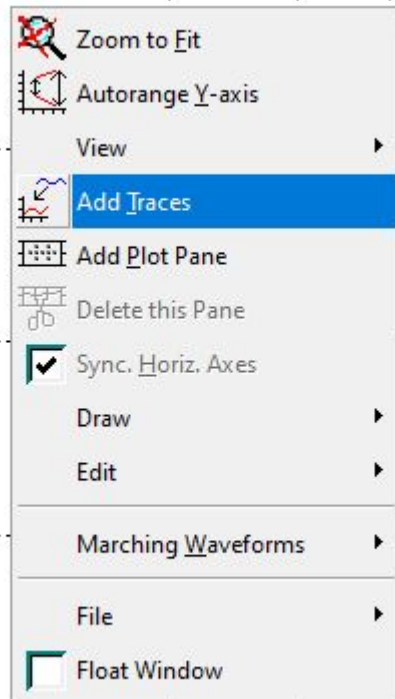
AC Analysis

- Now you can run your simulation. Follow the same procedure as described for the transient simulation. Only probe the node V_L.



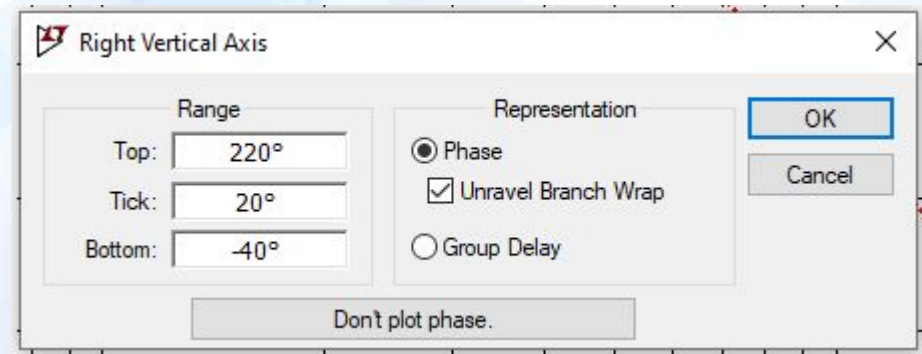
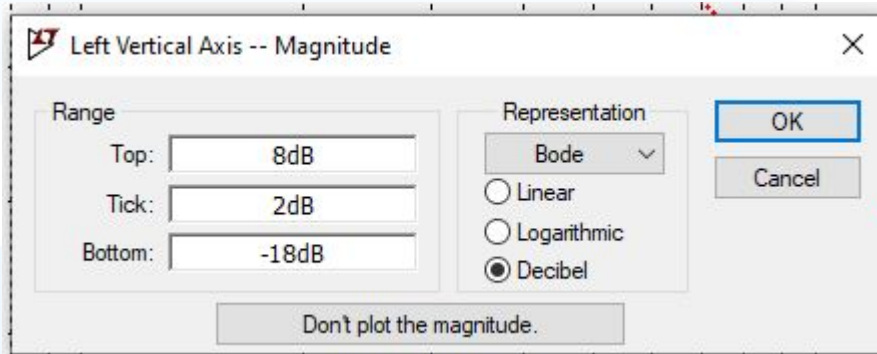
AC Analysis

7. You can also add traces by right clicking on the graph and choosing **Add Traces**. All the available traces are displayed. Mathematical expressions can also be used to create new plots.



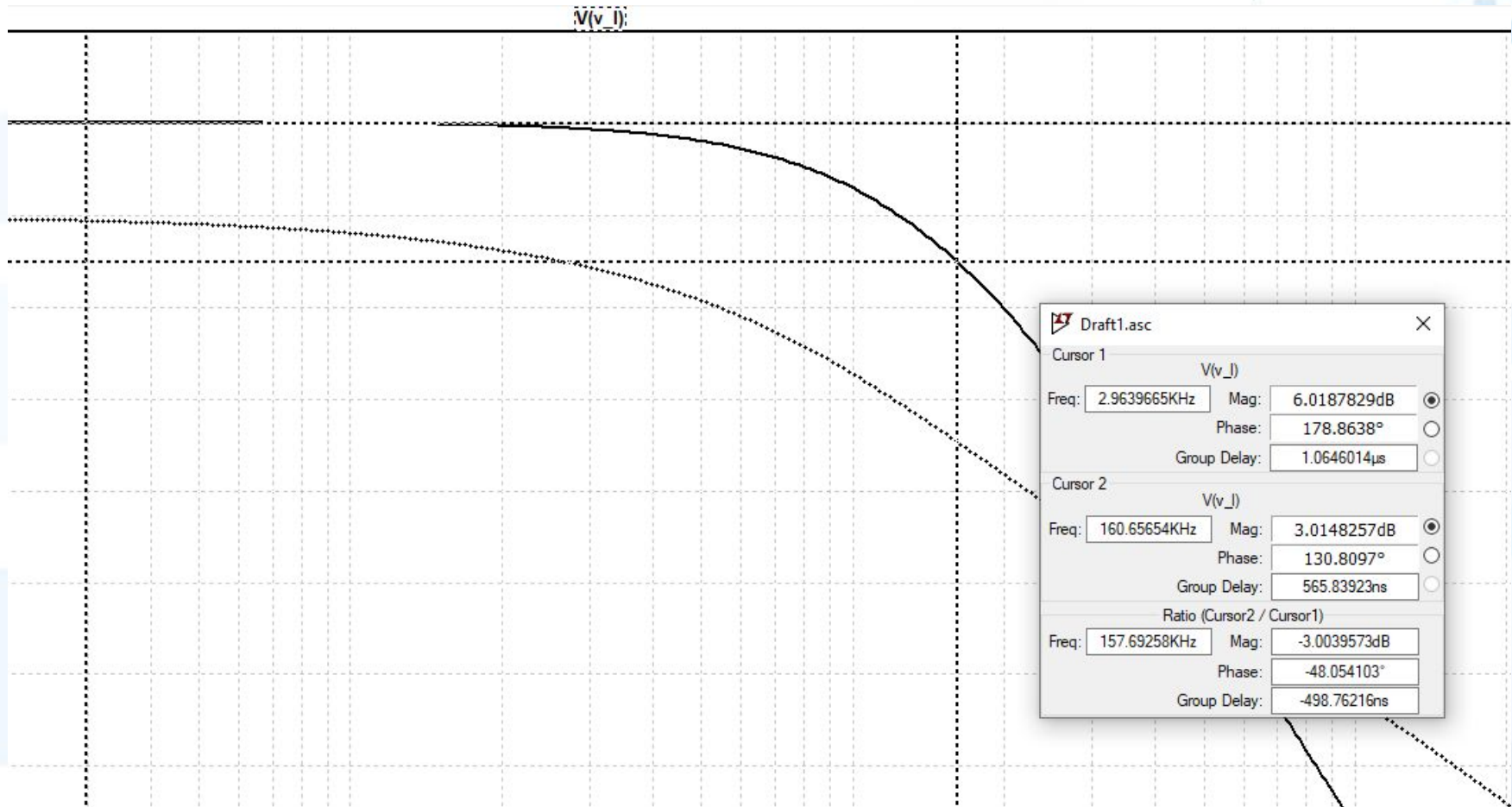
AC Analysis

8. You will notice that the y-axis scale is in dB and the right axis is in degrees. Therefore the graph is a bode plot showing both the magnitude and phase of the signal. You can change the characteristics of the left and right y-axes by right-clicking on them (if you would prefer a linear scale, for example.)



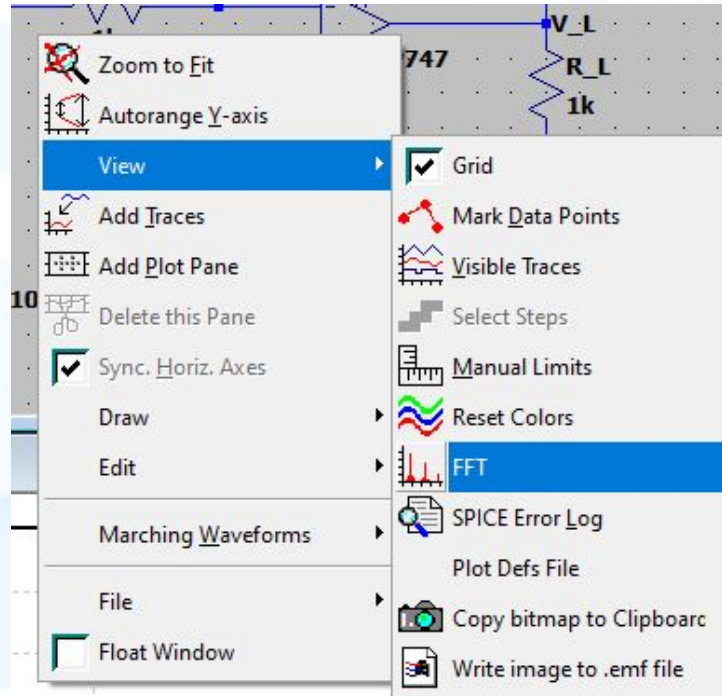
AC Analysis

9. The cursors are useful for finding the 3dB cutoff point.



FFT

1. From the AC analysis or transient simulation, you can easily generate an FFT graph. Right click on the graph, choose **View => FFT**.



The FFT data is not useful for this example, but will be important for your oscillator practical...

