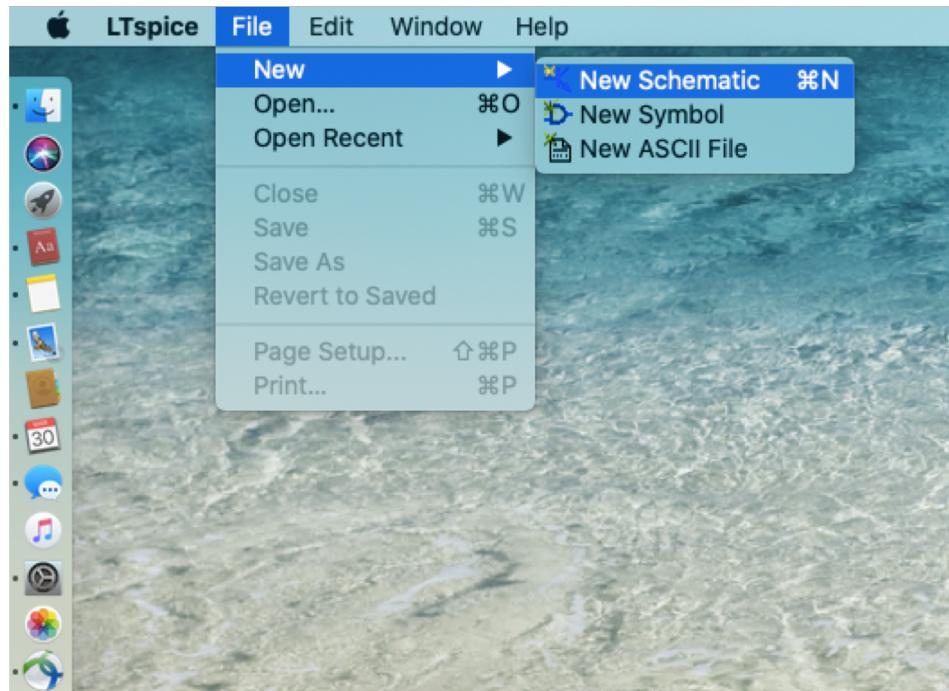


LT spice starting on a Mac

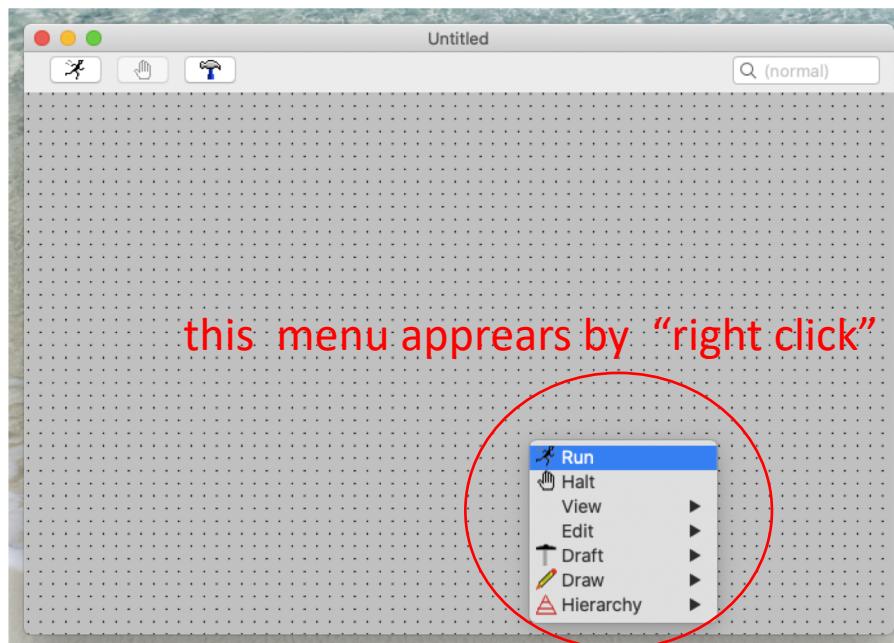
Minhyea Lee

1. Download .dmg file and install as instructed in Handout.

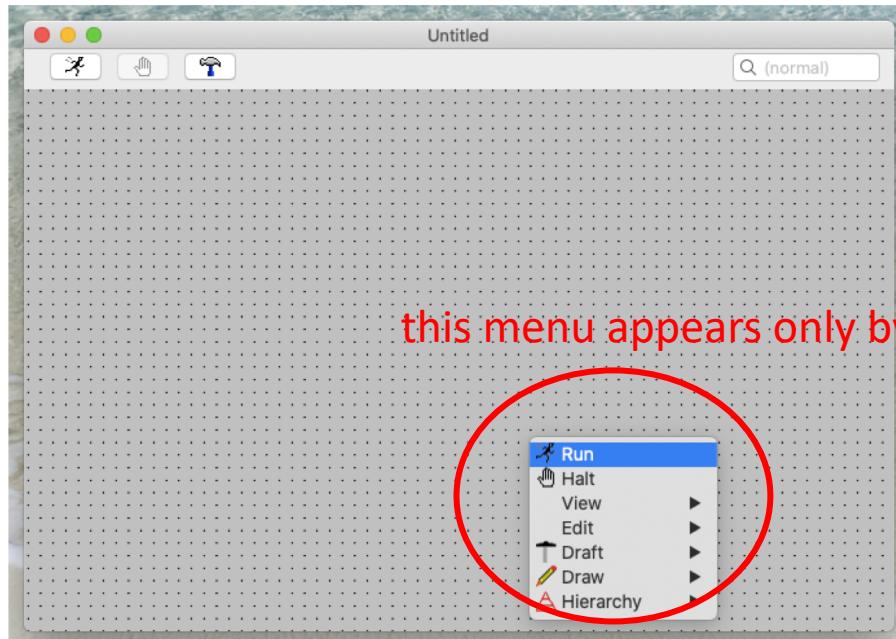
2. Starting Ltspice and open New Schematic –
this will be your **.asc** file to submit for assignment



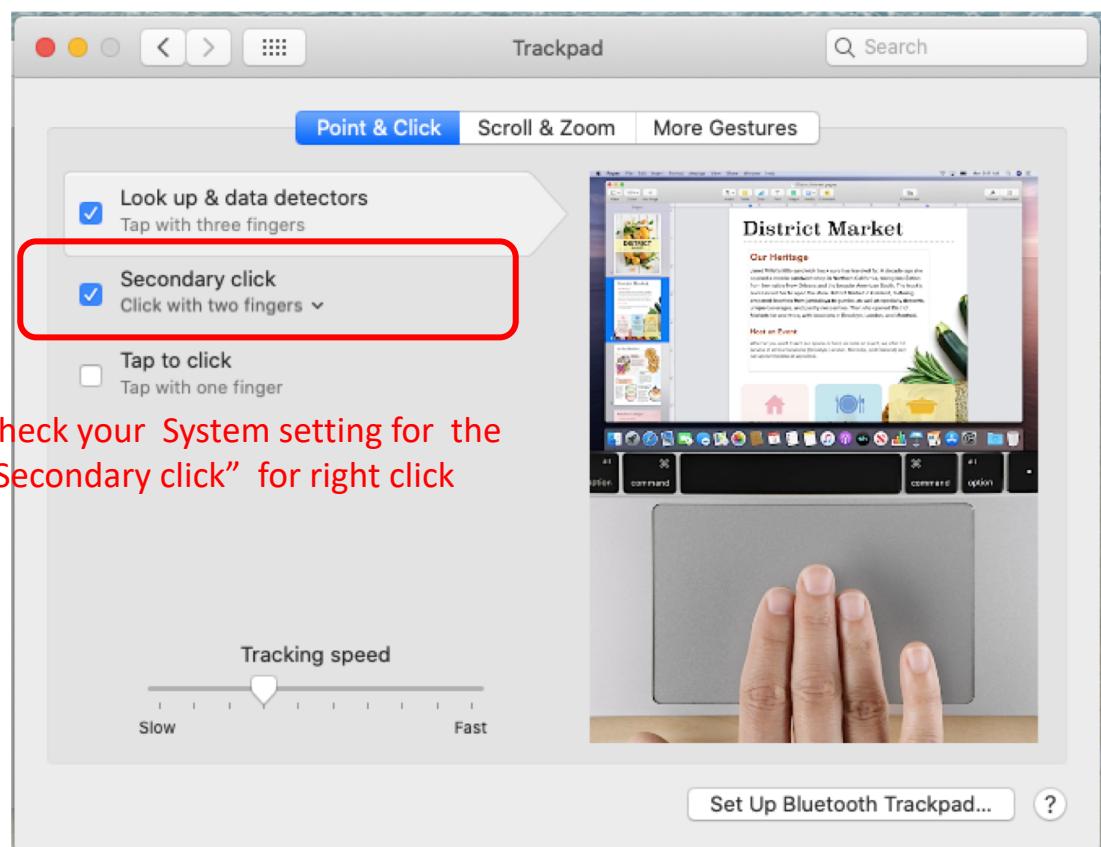
3. It will open a gray area for drawing circuits. Note that there are only 3 buttons on the menu. Dotted gray background can be selected from **View> grid**



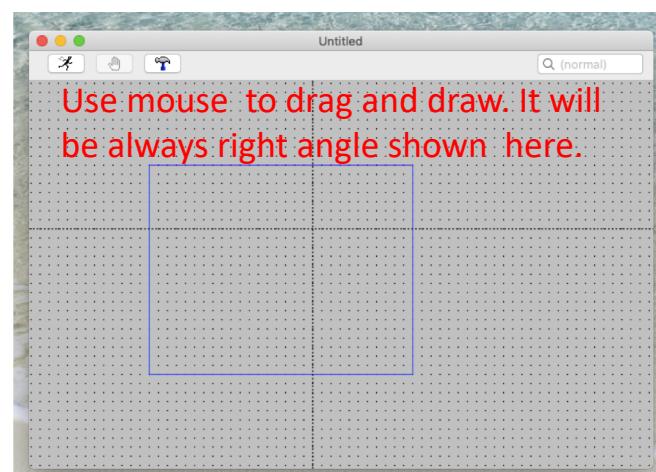
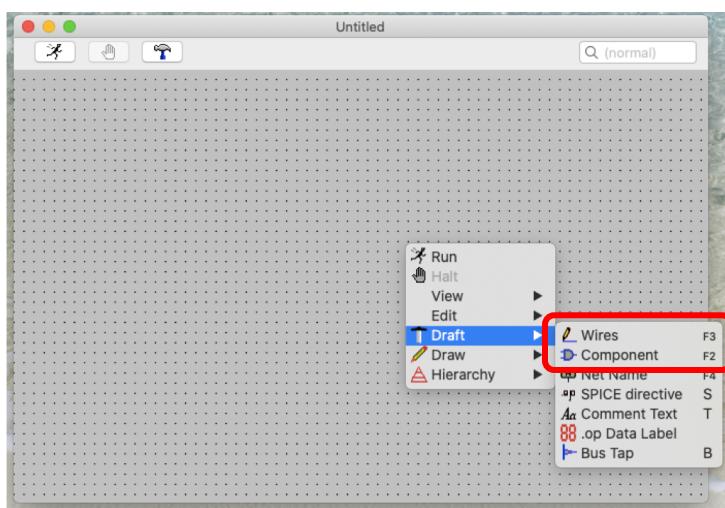
4. On Mac, LTspice uses right-click a lot! So this is a good time to check your computer how it was set .



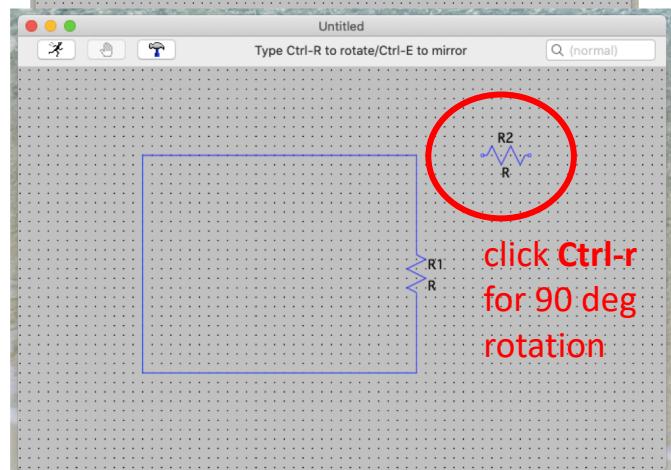
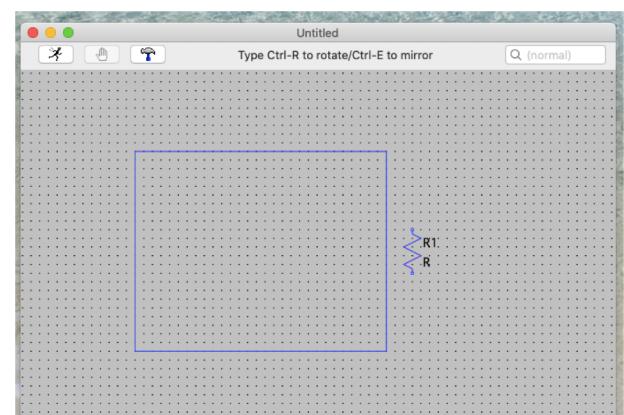
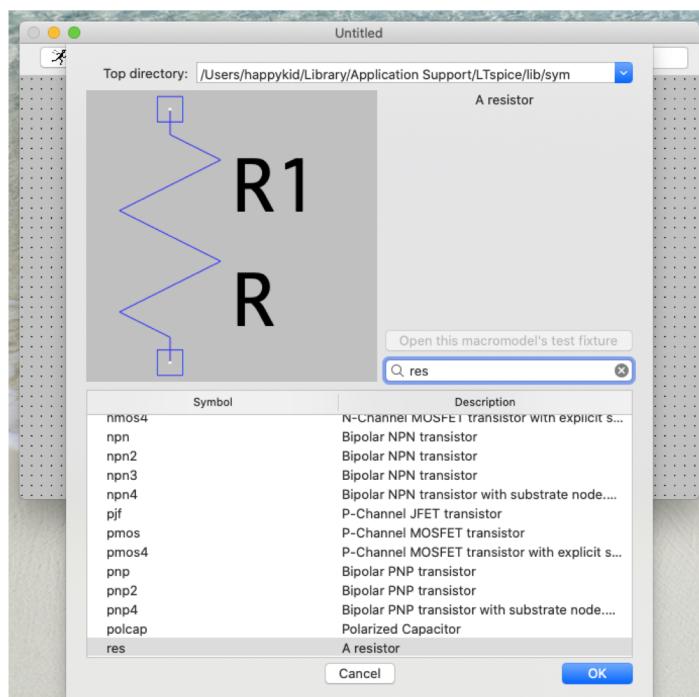
** Go to your **System Preference > mouse or > trackpad** and check your setting for **Secondary click is right-click for trackpad**.



5. Once you figure out your right click, here we can start with drawing wire by clicking **Draft>Wires**:

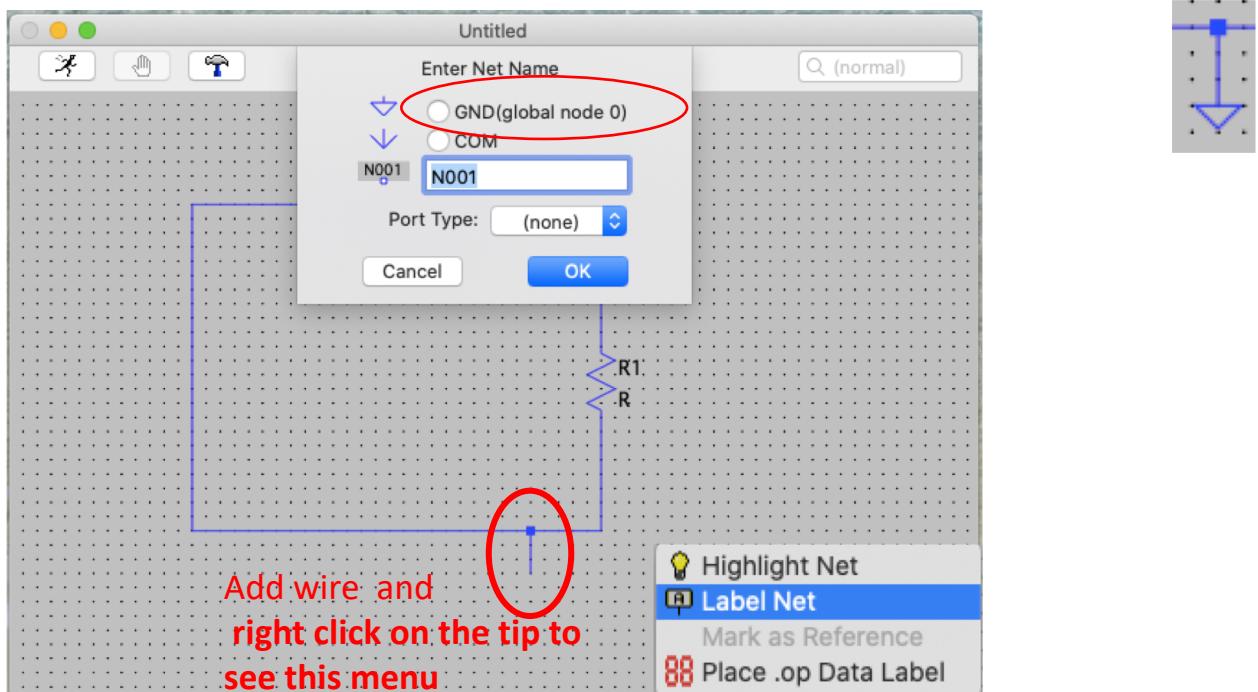


6. Here I draw a box for closed circuit. Now add resistance. Again go to **Draft>Component** type **Resistor** and then click **OK**

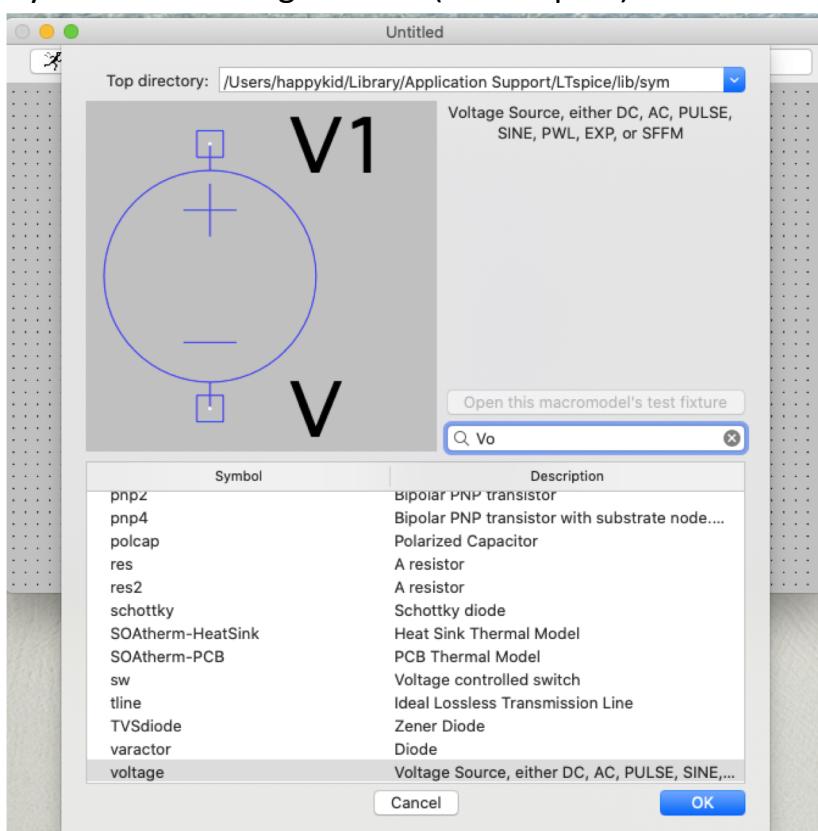


Then place it where you'd like it to be. If you'd like to rotate, press **Ctrl-r** until the correct direction. You can escape the component cursor mode by pressing **esc** key.

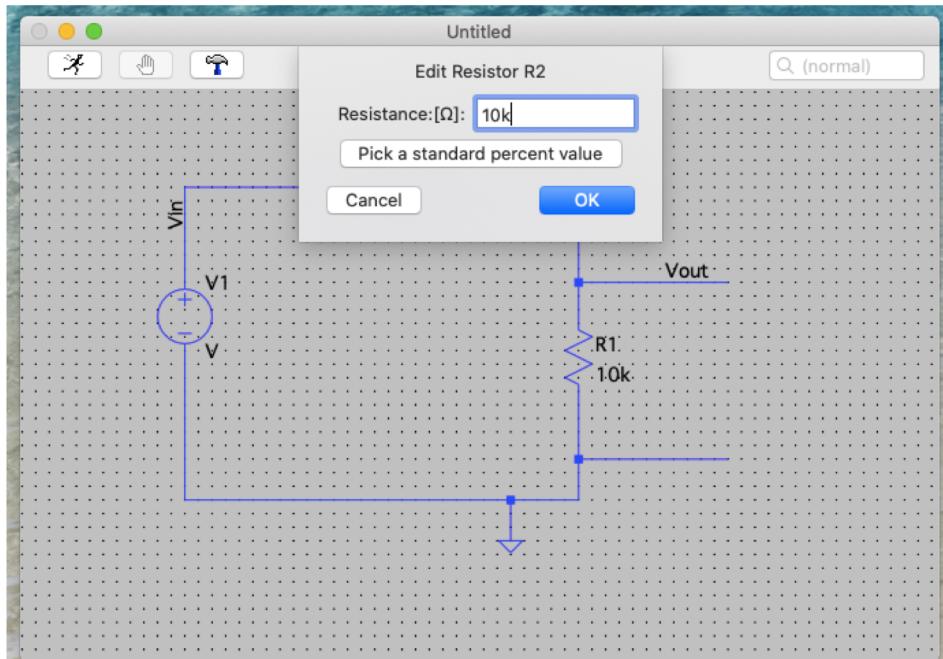
7. IMPORTANT: LTspice circuit will not work WITHOUT ground!! Add wire and right click at the end of wire . Select **Label Net** then choose **GND**, then the wire tap will change to



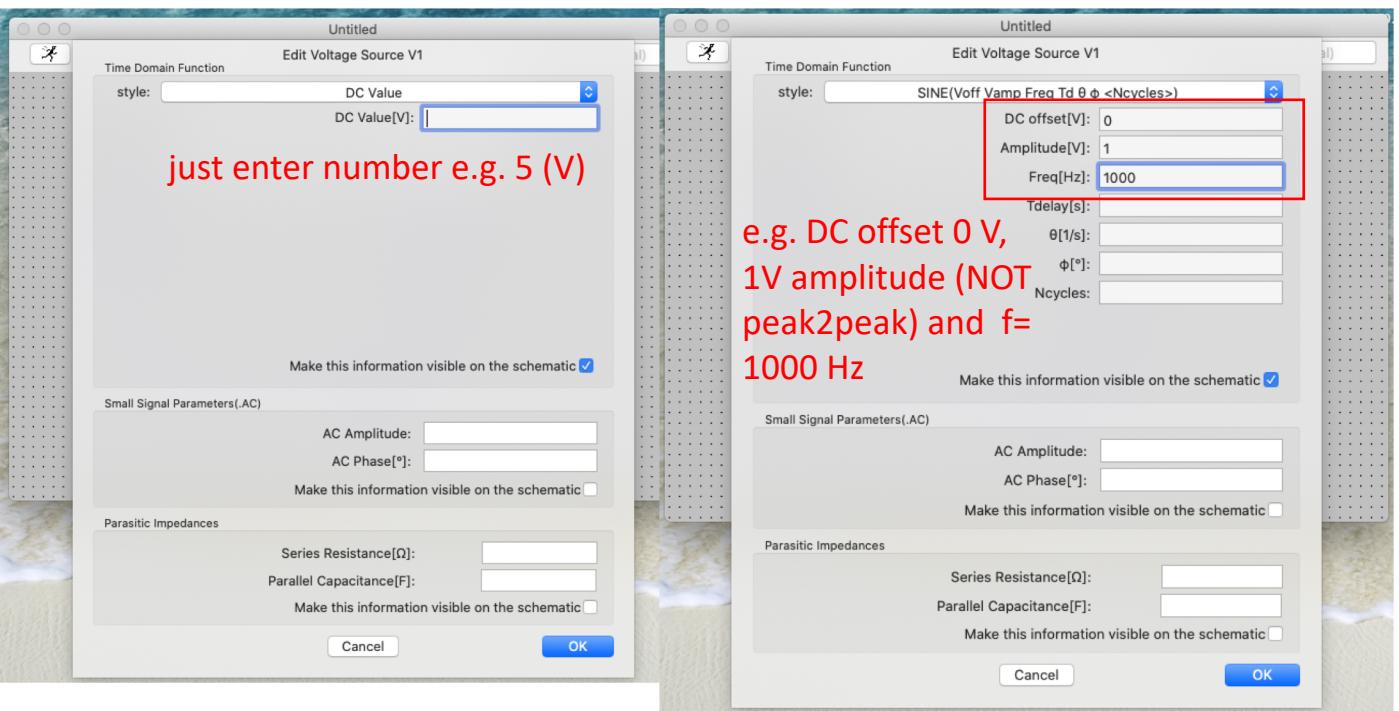
8. Now add voltage source – either DC or AC Pulse – Right click and **Draft>Component** and start typing. Then highlight the one you are looking for (here voltage source) from menu and click **OK**. You can specify the characteristics of Voltage source by **right-click**. It opens up a property menu for Voltage source (See Step 10).



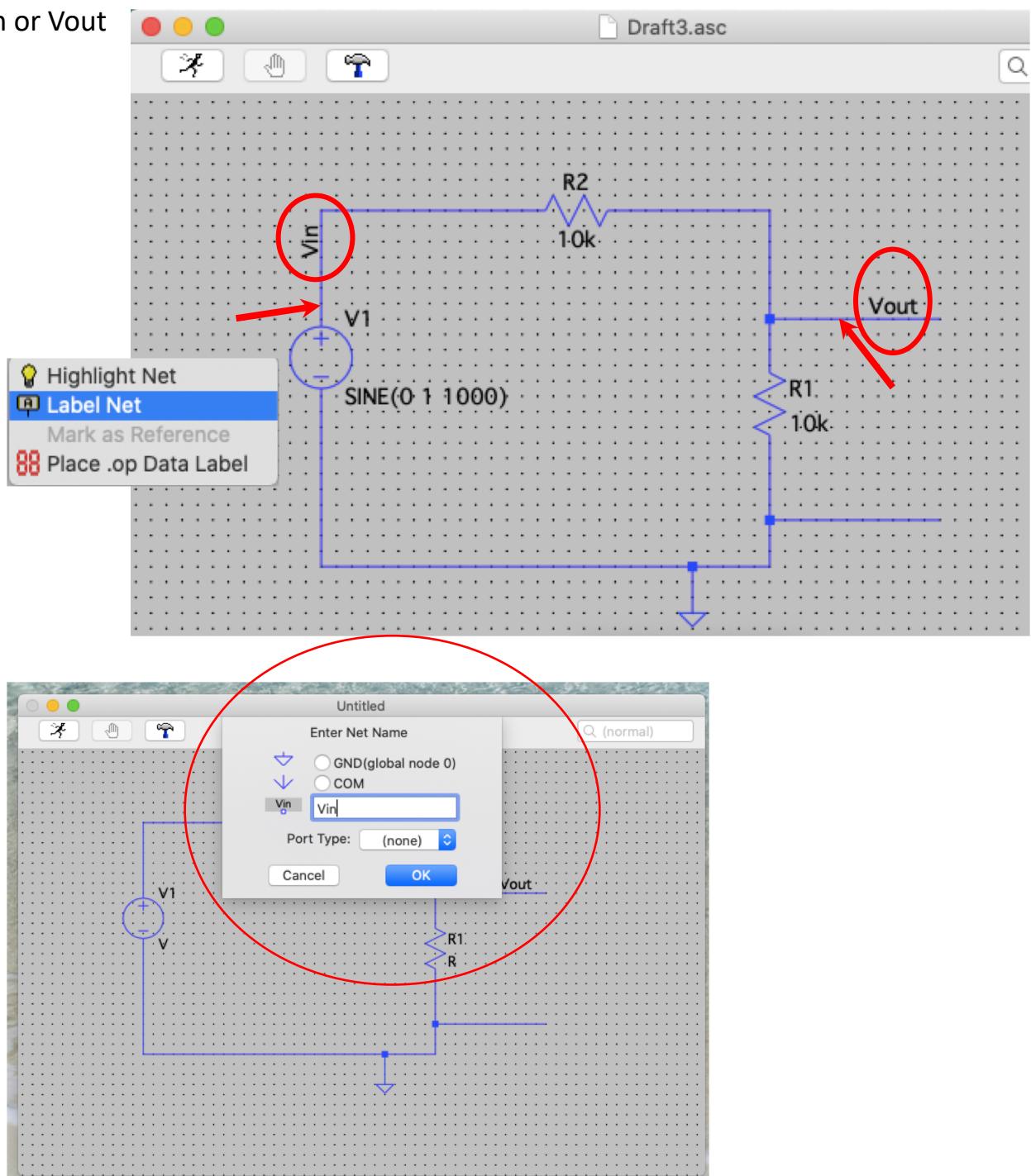
9. Add resistance values : **Right click** on the resistor (cursor changes from arrow to hand). Enter the number. "k" represent 103, "MEG", 106, etc. Please google the numeric format.



10. Specify Voltage source: Similarly **right click** on the voltage source – either DC (simply enter the value) or AC , at least give DC offset (0V), amplitude (1V) and Freq (1000 Hz)

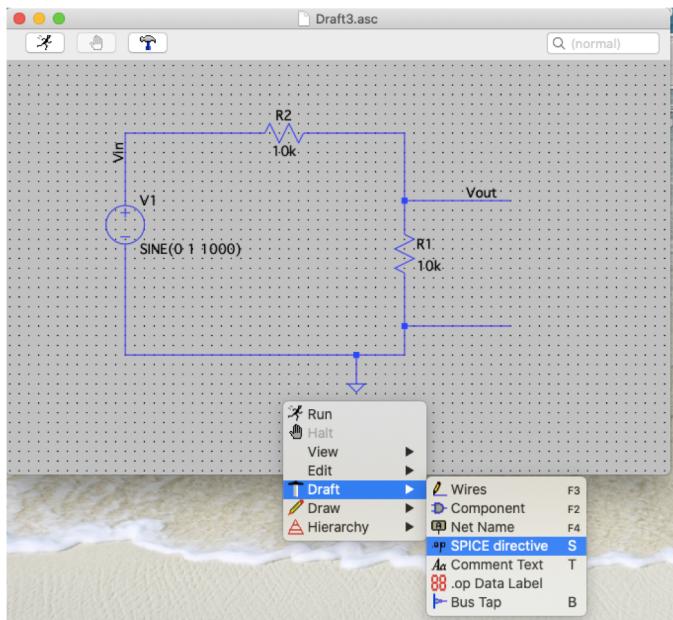


11. Finally, Each node to be measured have to be specify. Here that would be **Vin** and **Vout**. In order to do this, place the cursor for the location for Vin (or Vout) would be measured (indicated as arrow) and right click. The select **Label Net** and on third item, add name Vin or Vout



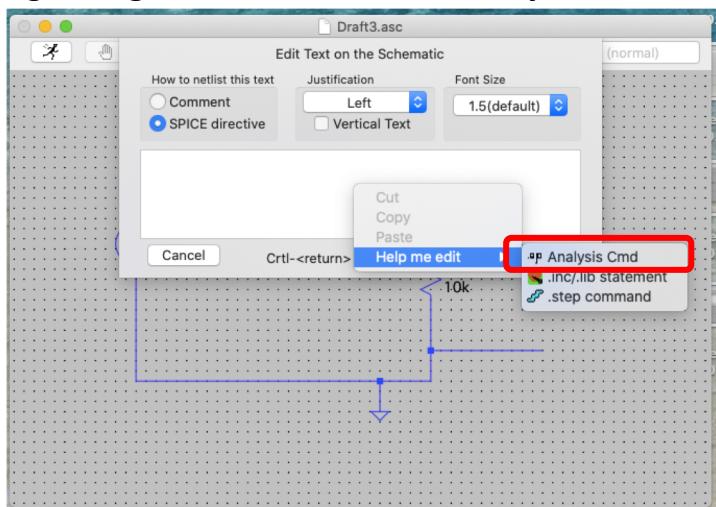
** Now your circuit is completed and need to measure What Vin and Vout looks like

12. Right-Click anywhere and choose **Draft>SPICE directive** .

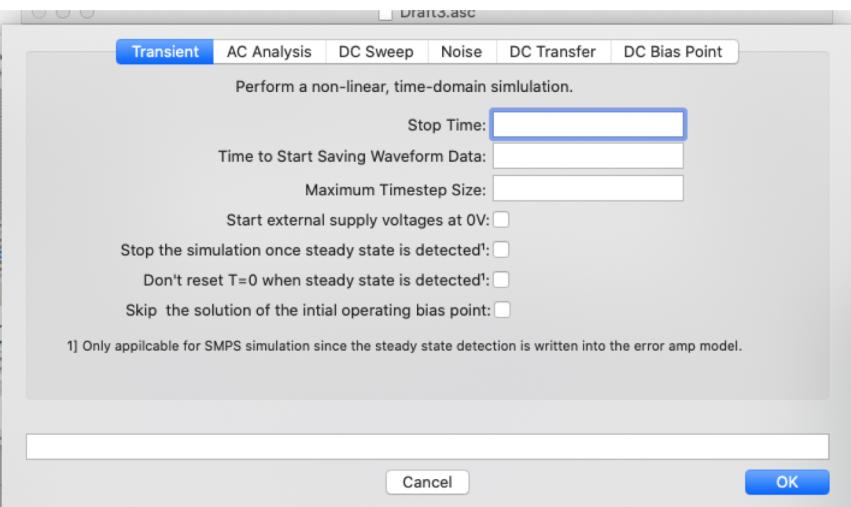


13. Edit Box appears. Don't know what to type?

Again Right-click and choose **Analysis Cmd**

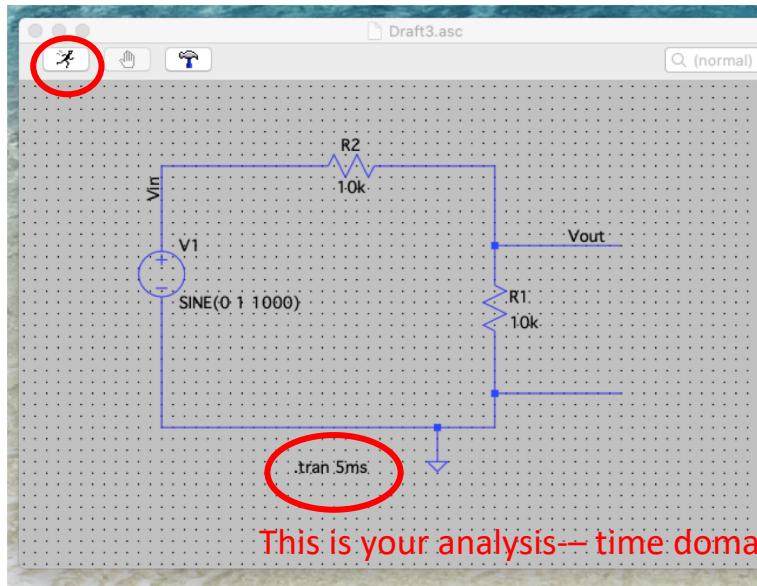


14. We will mostly use **transient** and AC analysis. **Transient** is for time-domain (i.e. x-axis is time as oscilloscope screen).

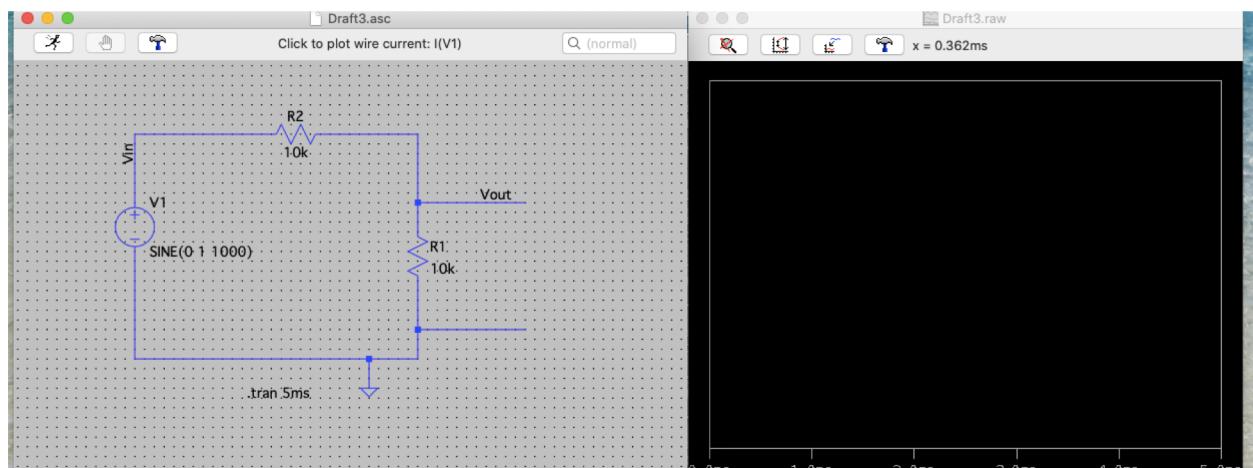


15. Specify **Stop time** -- consider your frequency. 5 periods. e.g. for 1kHz, it would be 5 ms. For now, just specify the **Stop Time** and leave others blank (which is always good thing to do , if you don't know what to do with it.) Then click **ok**

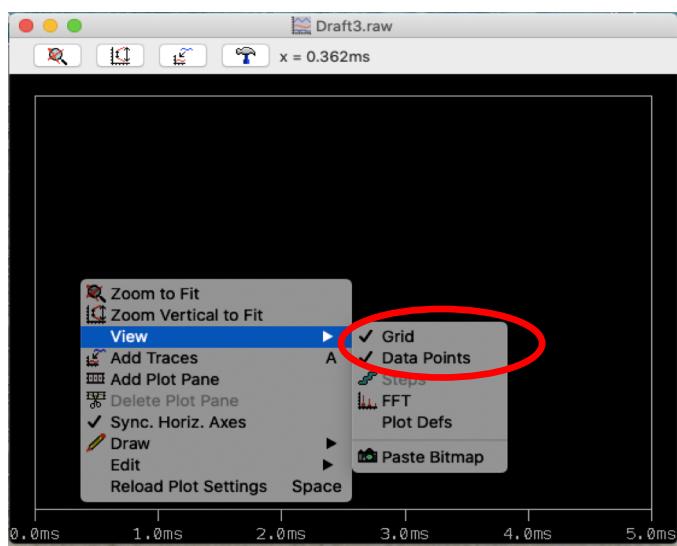
16. Now your circuit simulation is ready to run by clicking (obvioualy) run button



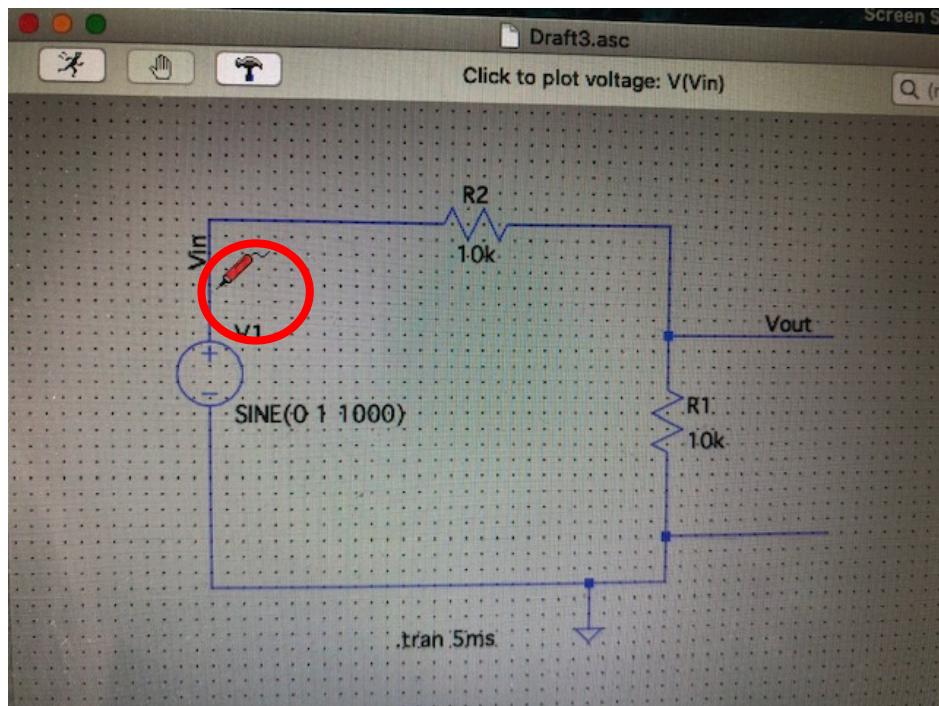
17. Then graph screen appears Note that x-axis is already marked up to 5 ms



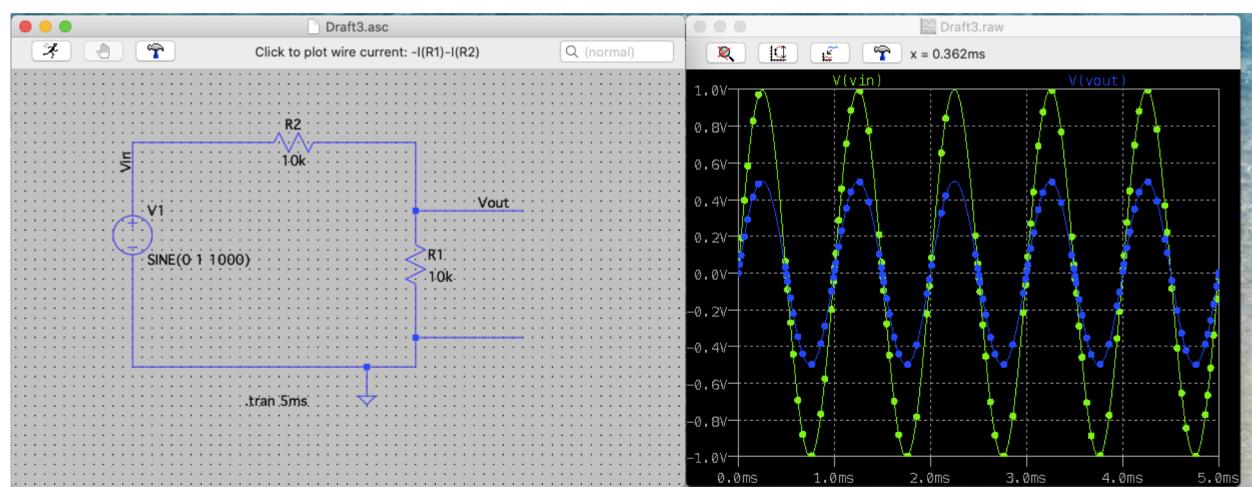
**You can right-click and specify property – like showing grid and data points.



18. Bring the cursor near Vin. The arrow will change to voltage probe and clicking while "Voltage probe" shape cursor, it will plot the Vin.



19. Then click V_{out} with the voltage probe.
It will overplot . (Below green is V_{in} and blue is V_{out})



** If you click while holding down Shift-Ctrl, the voltage probe will change to the **current probe cursor** and clicking will make current vs time plot appear (on current values shown on right y-axis).

Ok .. this is just a bare-bone instruction and now go and play around !!!
END