Tips for LTspice on Mac
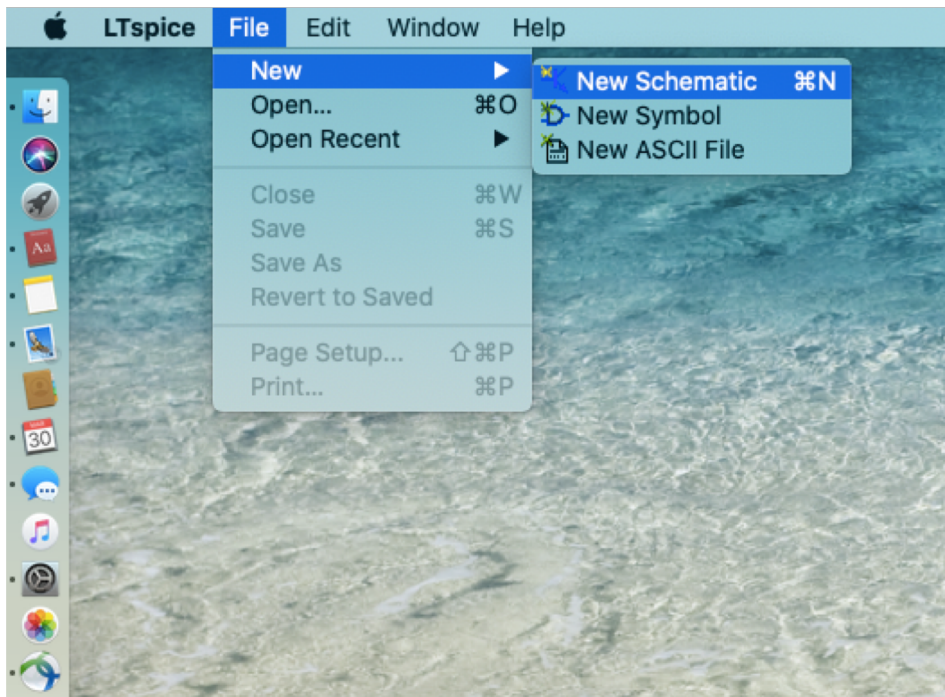
1. Download .dmg file and install it. Use the link below for download:
https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

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

Also see **LTspiceShortcuts forMac** in Canvas.

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**
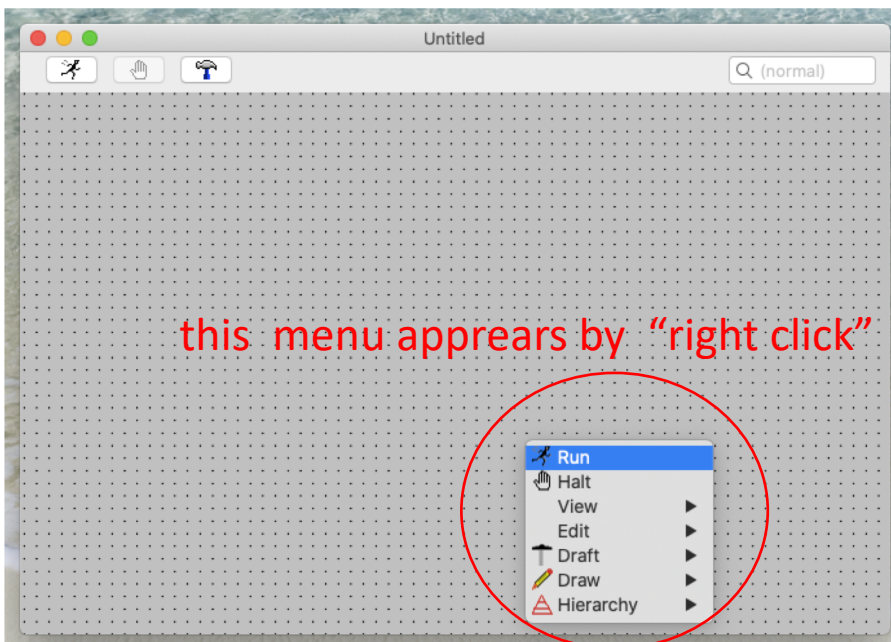
4. On Mac, LTspice uses right-click a lot! So this is a good time to check your computer setting


this menu appears only by "right click"
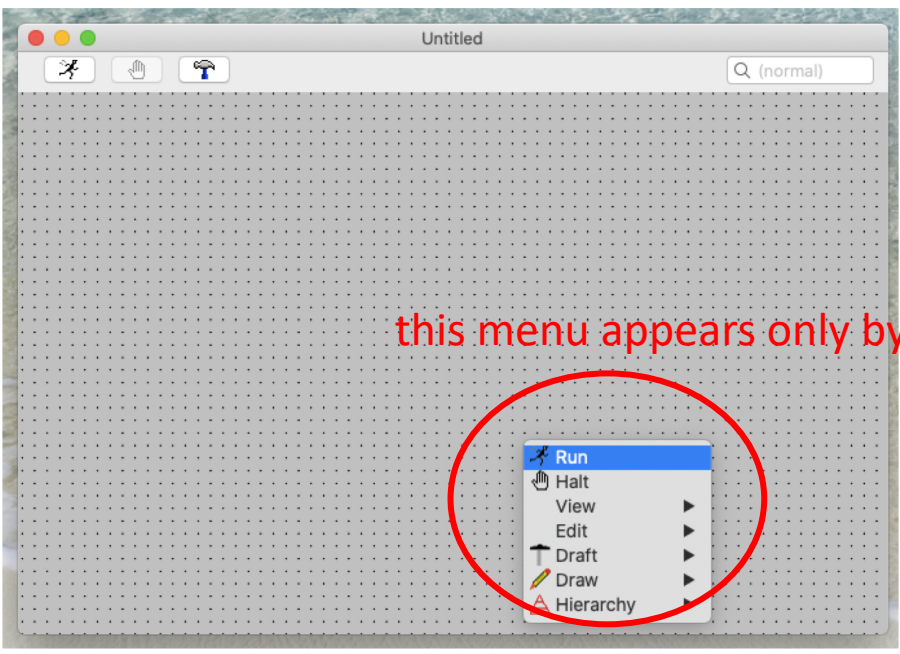
** Go to your **System Preference > mouse** or **> trackpad** and check your setting for **Secondary click is right-click for trackpad. Click with two figures.**

Check your System setting for the "Secondary click" for right click
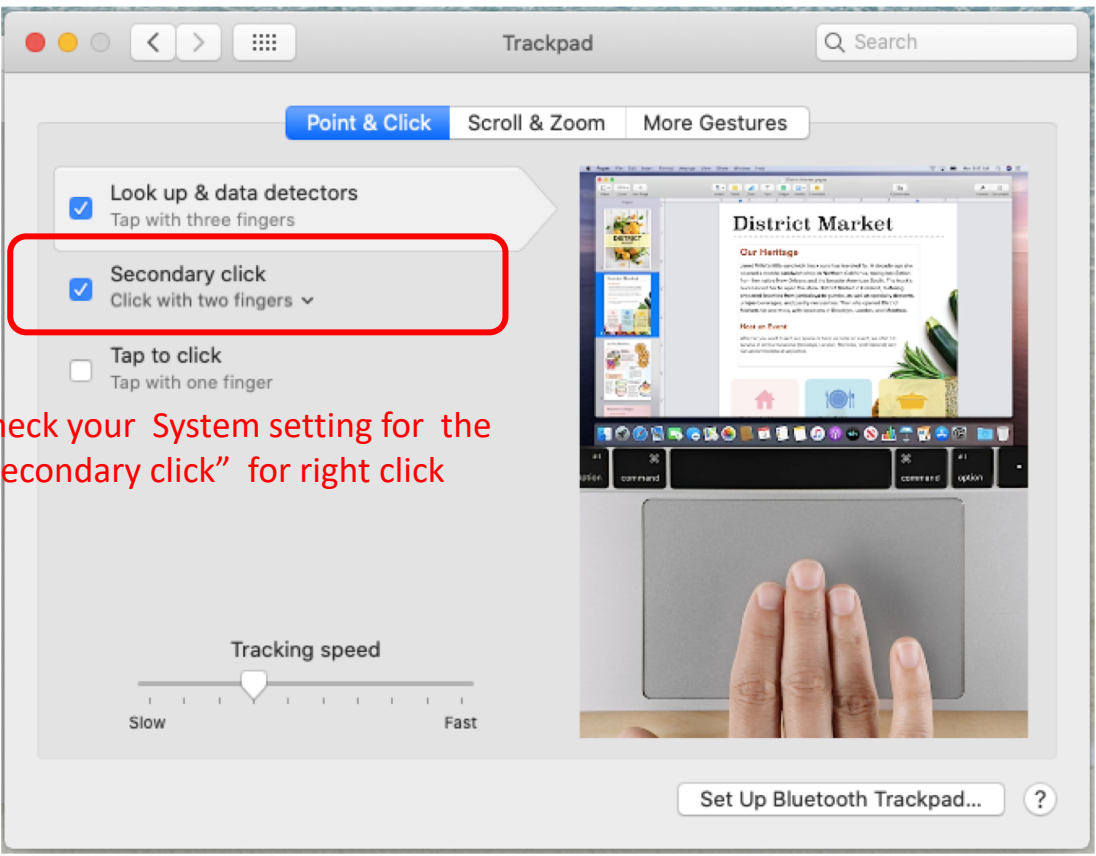
5. Now start with drawing wire by clicking **Draft> Wires:**
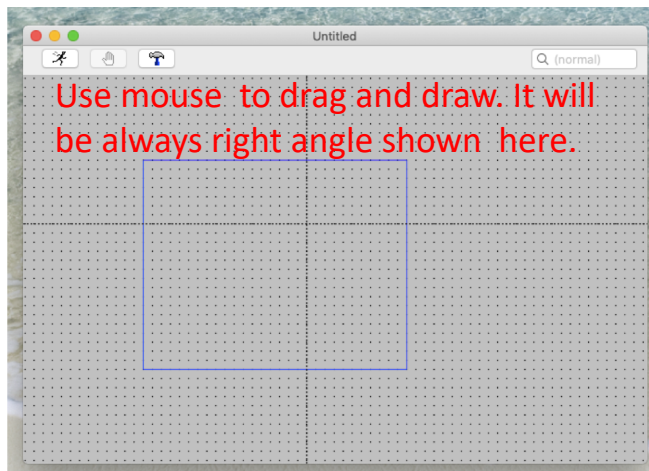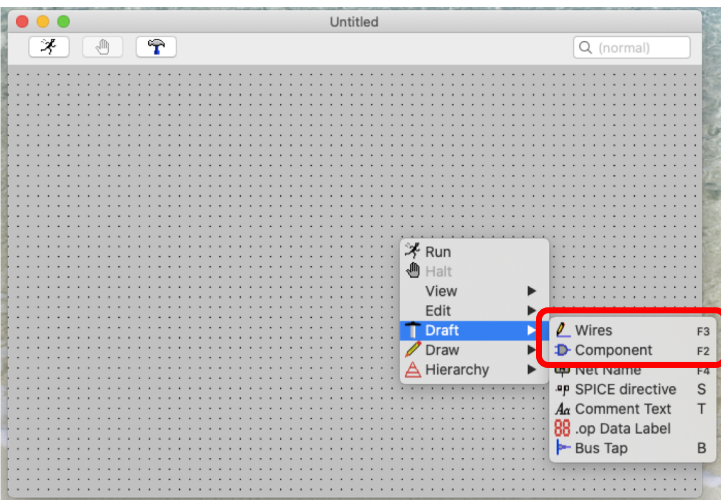
6. Now add resistor(s) (or other parts). Again go to **Draft>Component type Resistor and then click OK**
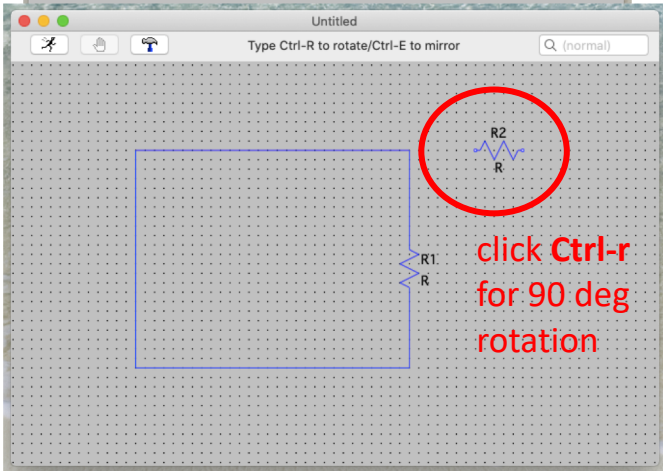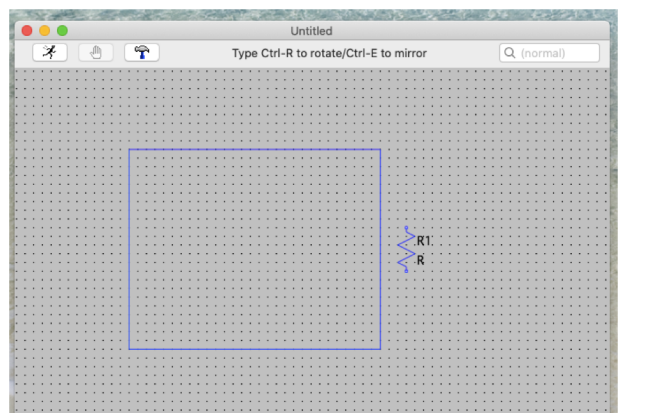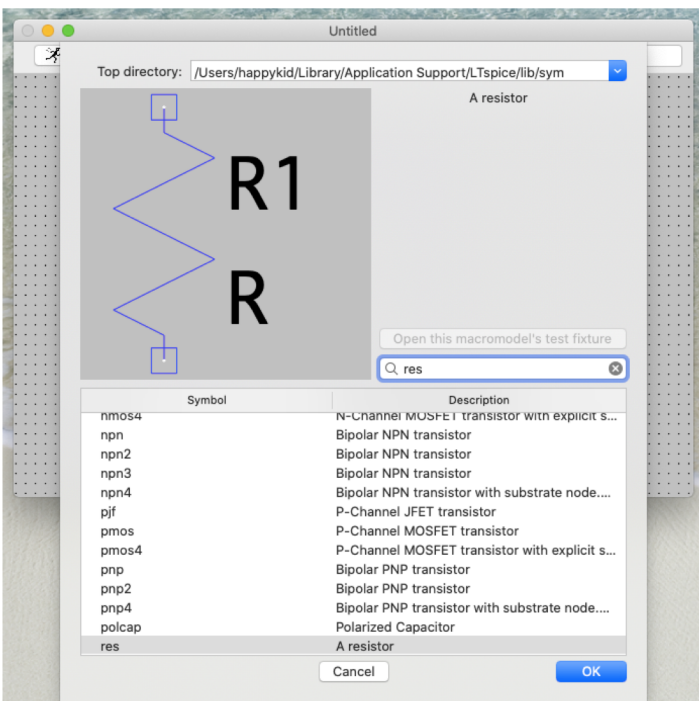

Then place it where you'd like it to be.   If you'd like to rotate, press **Ctrl-r** until the correct direction. You can escape the component cursor mode by pressing **esc** key.

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
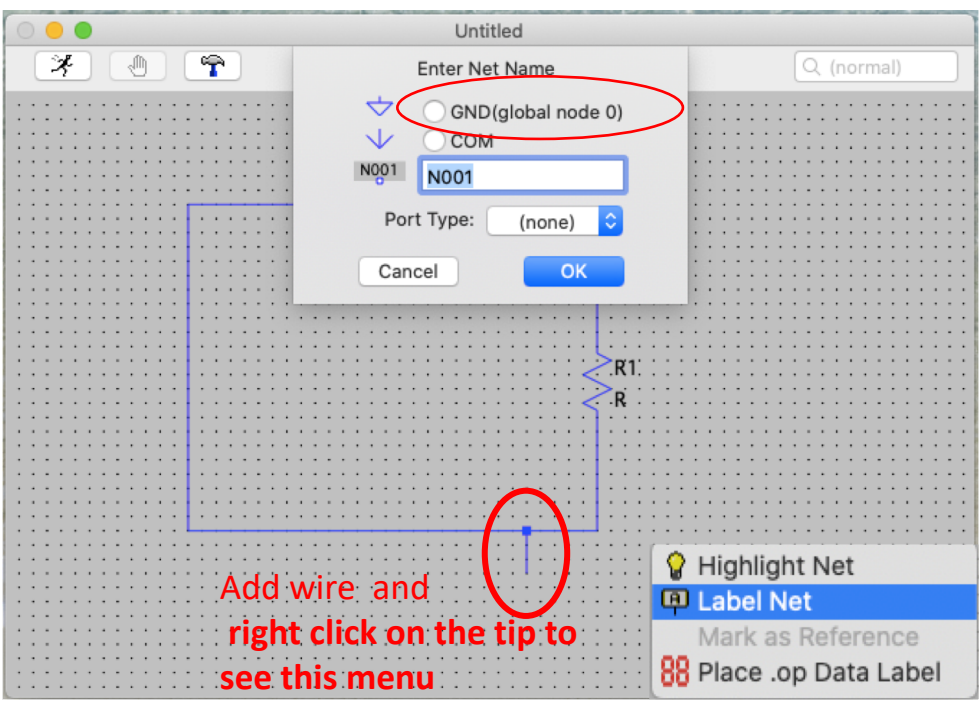

8. Now add voltage source – either DC or AC Pulse – Right click and **Draft>Component** and start typing. Then highlight the one you are looking for (here voltage source) from menu and click **OK.** You can specify the characteristics of Voltage source by **right-click.** It opens up a property menu for Voltage source (See Step 10).
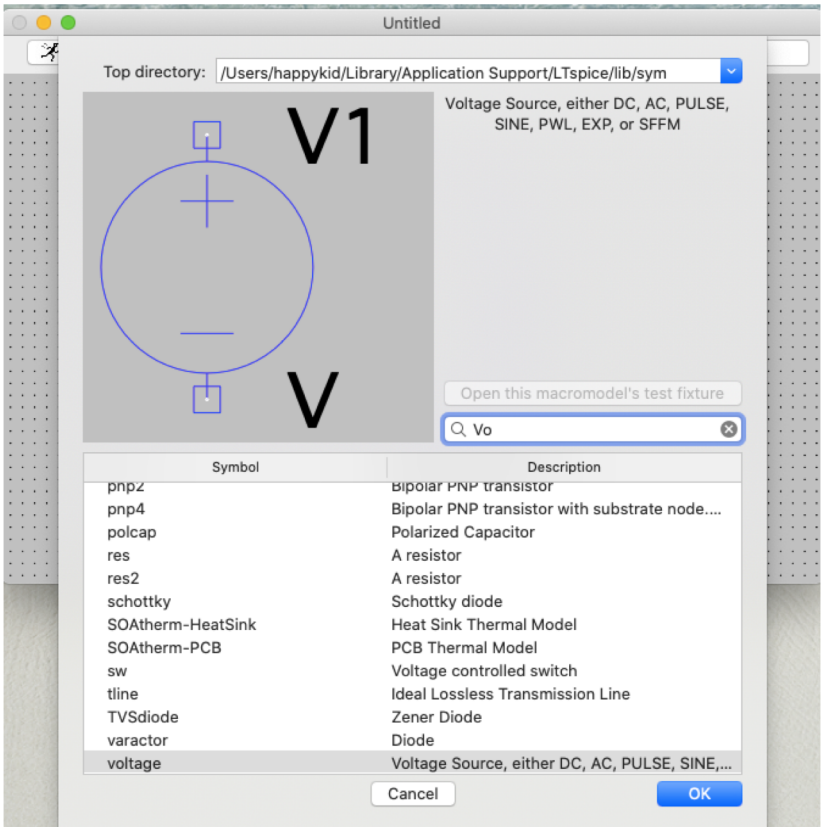
9. Add resistance values : **Right click** on the resistor (cursor changes from arrow to hand). Enter the number. "k" represent 10^3, "MEG" , 10^6, etc. Please google the numeric format.
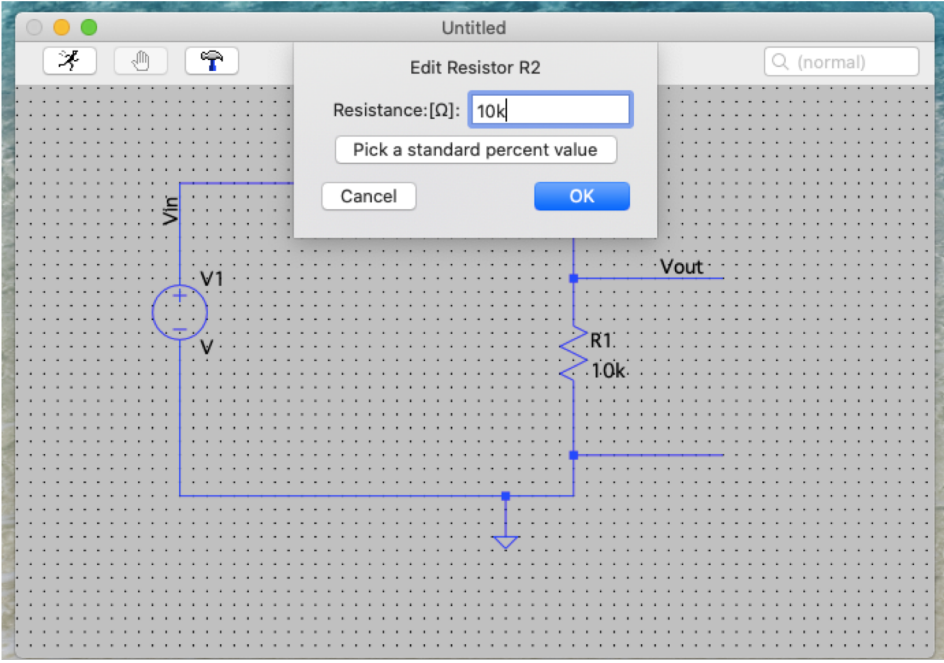
10. Specify Voltage source: Similarly **right click** on the voltage source – either DC (simply enter the value) or AC , at least give DC offset (0V), amplitude (1V) and Freq (1000 Hz)


just enter number e.g. 5 (V)

e.g. DC offset 0 V, 1V amplitude (NOT peak2peak) and f= 1000 Hz
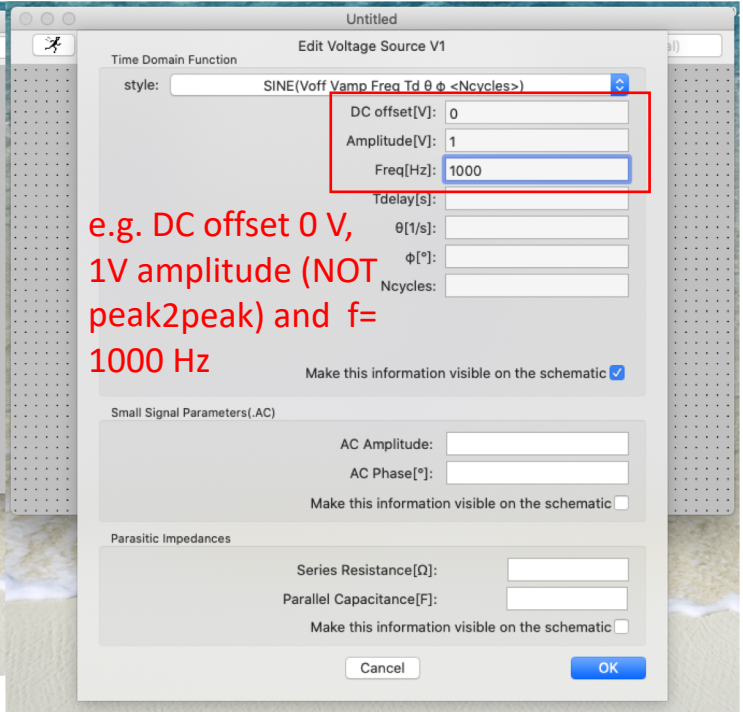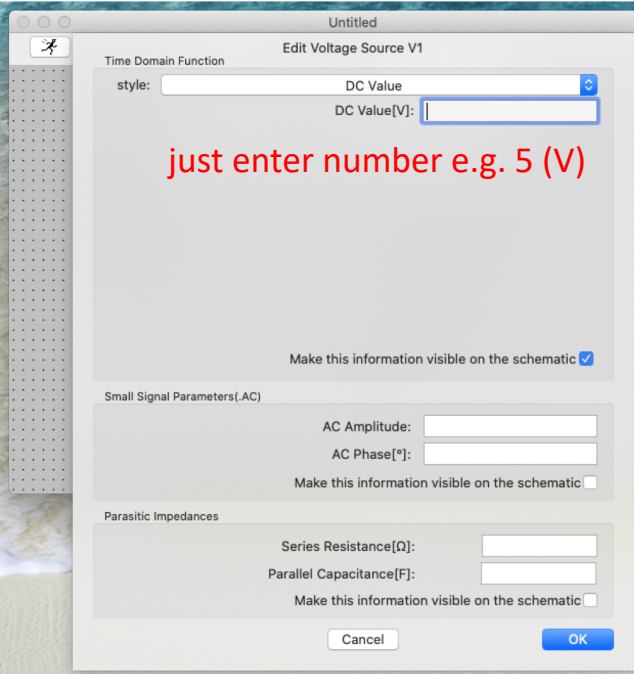
11. Finally, Each node to be measured have to be specify. Here that would be **Vin** and **Vout . In order to do this,** place the cursor for the location for Vin (or Vout) would be measured (indicated as arrow) and right click. The select **Label Net** and on third item, add name Vin or Vout
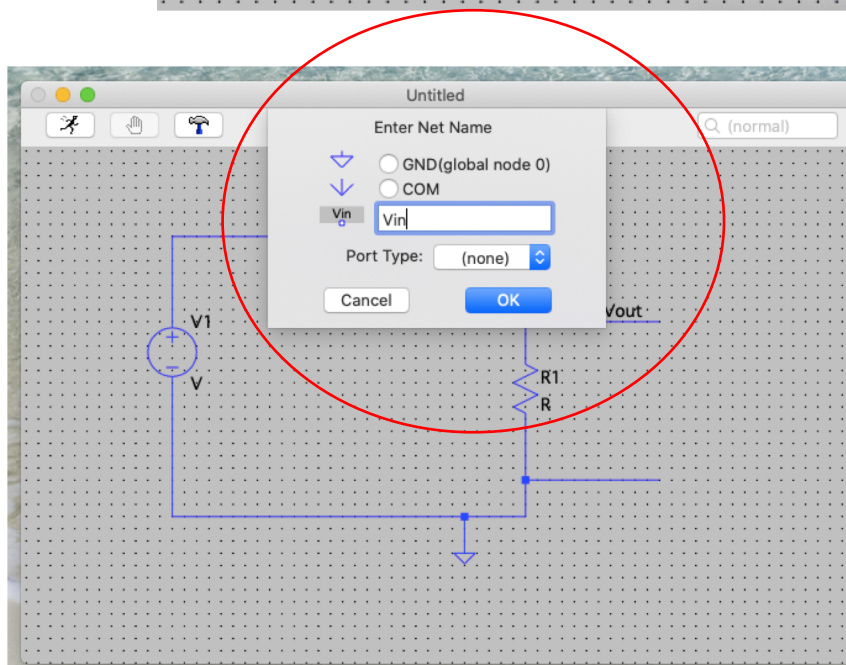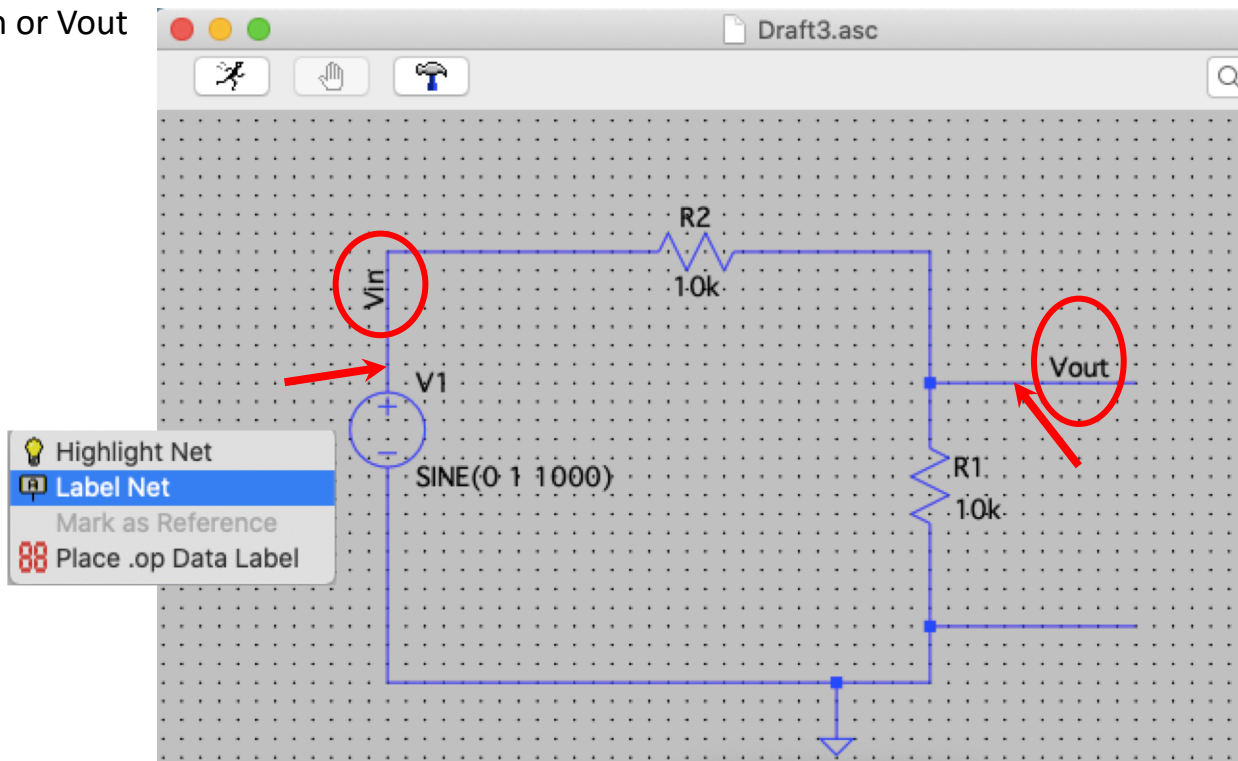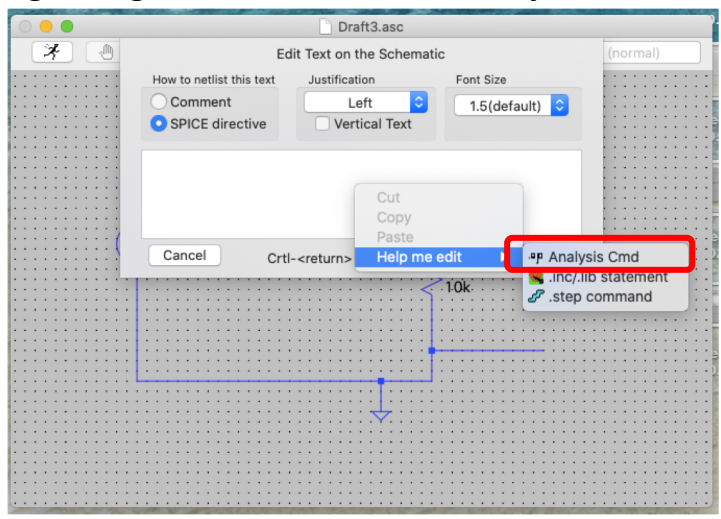

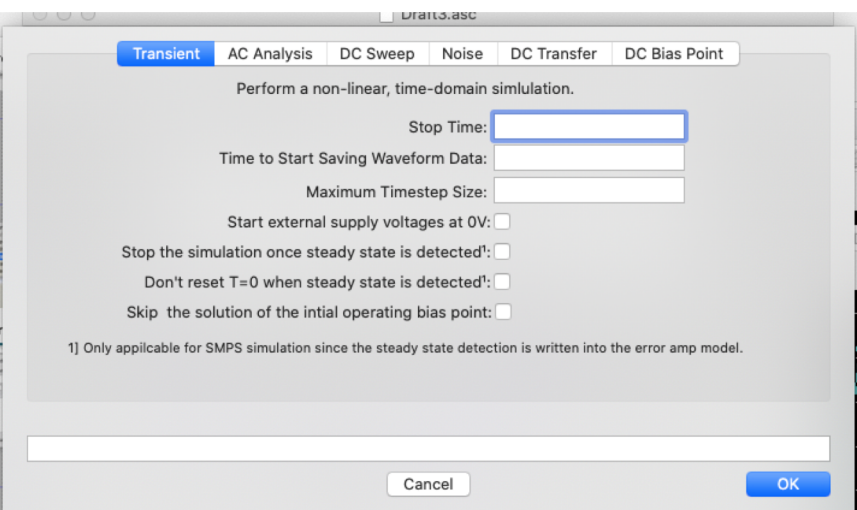** Now your circuit is completed and ready for measurements.

12. Right-Click anywhere and choose **Draft>SPICE directive .**
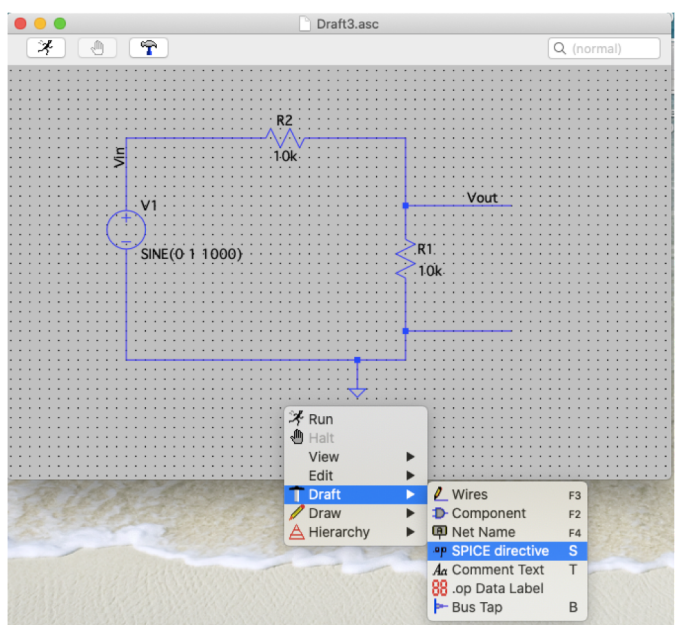
13. Edit Box appears. Don't know what to type?
Again **Right-click** and choose **Analysis Cmd**

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
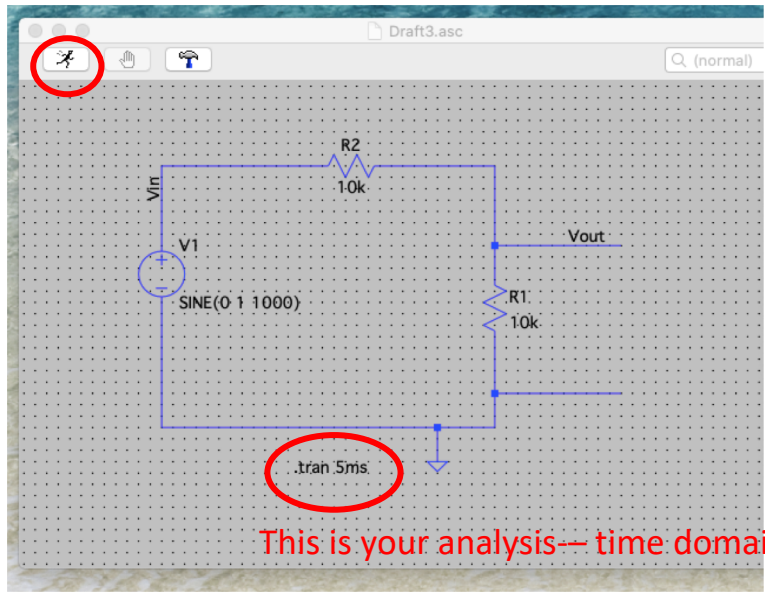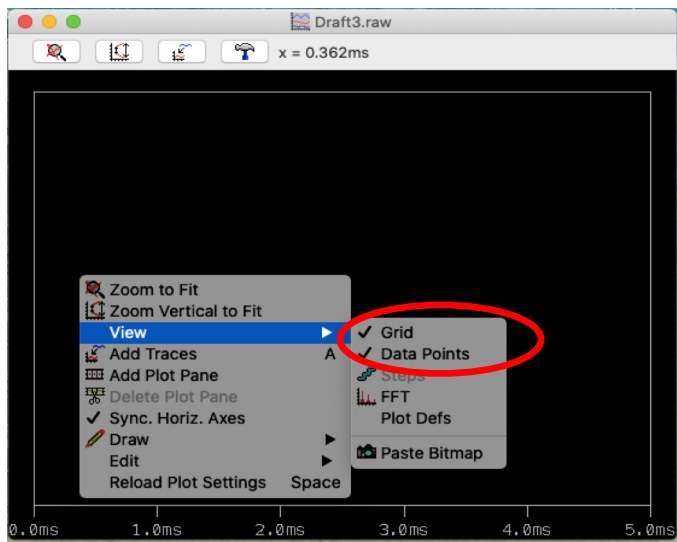
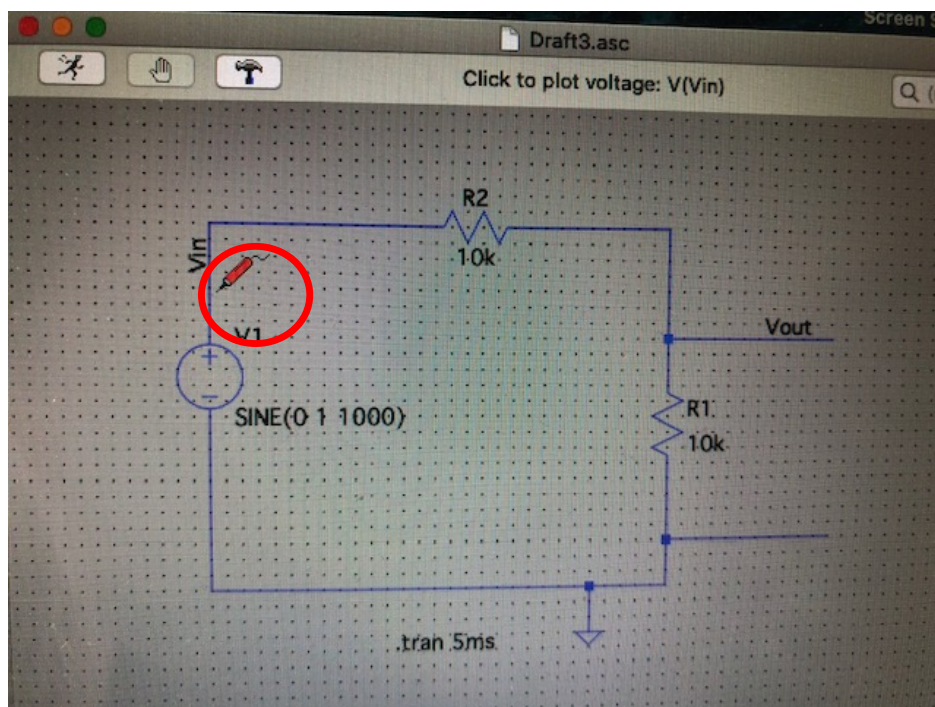This is your analysis-— time domain 5 ms

17. Then graph screen appears Note that x-axis is already marked up to 5 ms
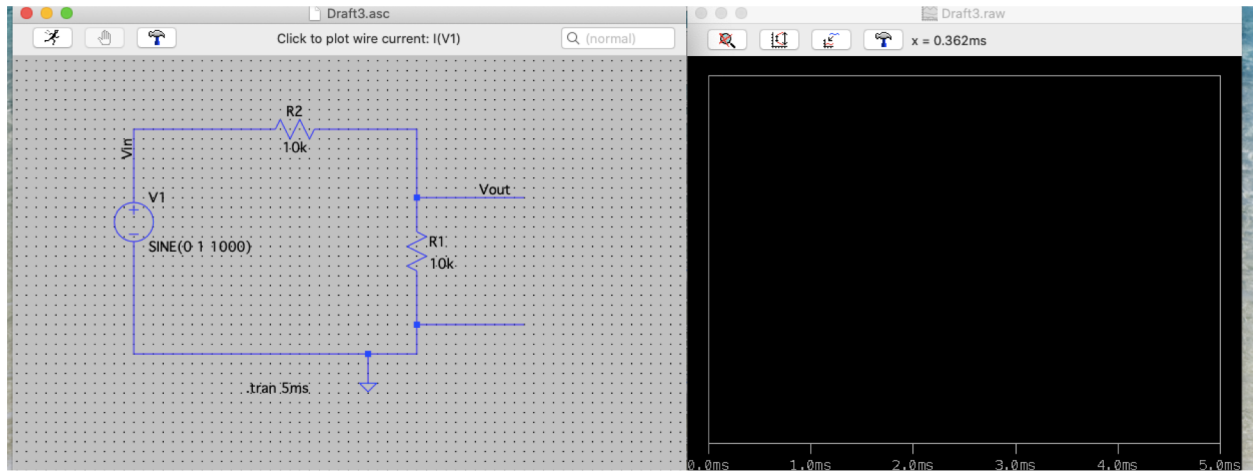
**You can right-click and specify property – like showing grid and data points.

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 Vout with the voltage probe.
It will overplot . (Below green is Vin and blue is Vout)
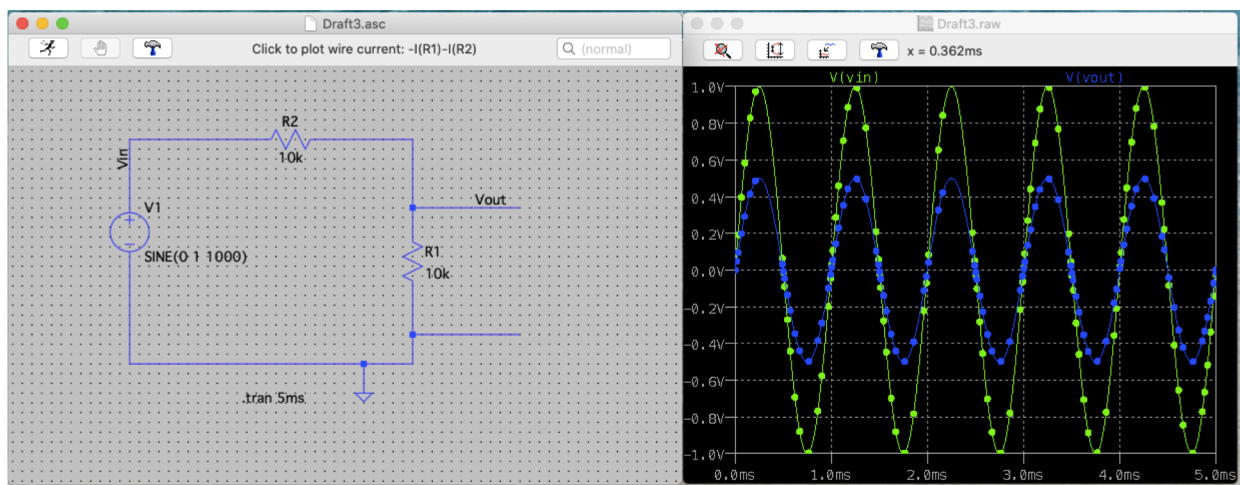
** If you click while holding down Shift-Ctrl, the voltage probe will change to the **current probe cursor** and clicking will make current vs time plot appear (on current values showns on right y-axis).

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