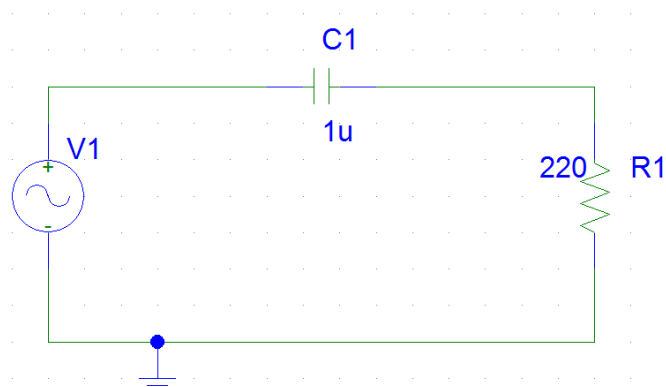
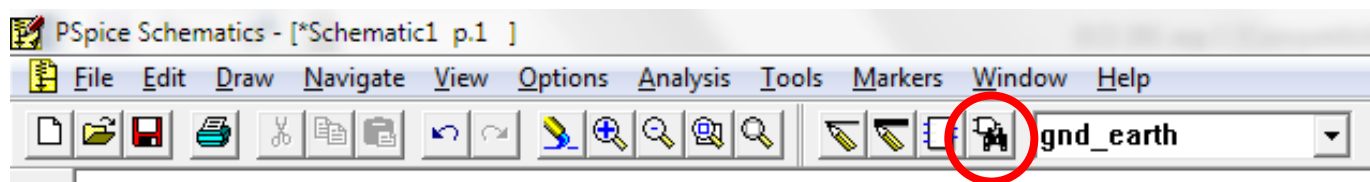


Pspice Task 1- Familiarization with AC Waves

1. Go to the Start menu and locate the application “Schematics”
2. Now, set up the following circuit following the instructions given below-



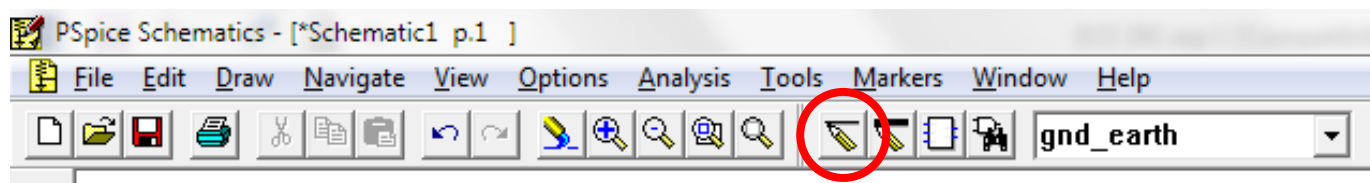
3. Get the following parts by clicking the “Get new Part” button from the toolbar above, OR press Ctrl+G.



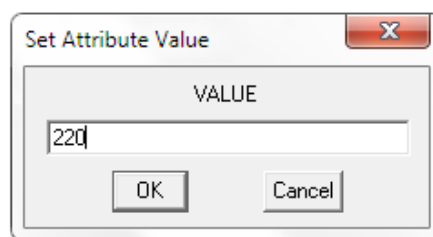
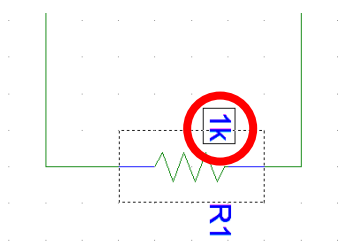
In the box “Part Name” that appears, type in-

- a) **Vsin** (for AC Voltage source)
- b) **R** (Resistor)
- c) **C** (Capacitor)
- d) **Gnd_earth**- (for ground- i.e. a reference potential from which all voltages are measured)

4. Place the parts as shown in the first figure. To rotate a part, select the part and press Ctrl+R. After that, connect each part using wires. To select wire, use the button from the toolbar above.



5. To change the parameter values (resistance for resistors, capacitance for capacitors, etc) simply double click on the default value that is shown alongside the part, and enter the desired value in the window that appears.



CSE250 LABORATORY

Pspice Task 1- Familiarization with AC Waves

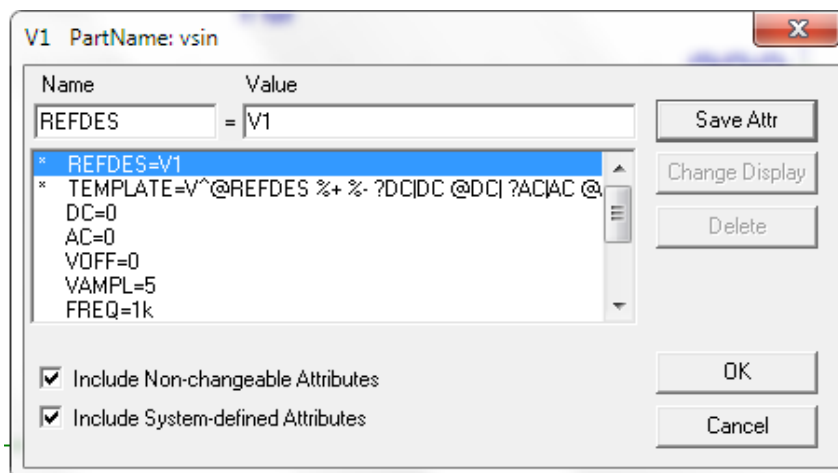
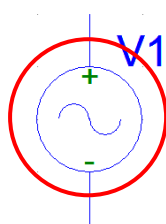
Pspice Schematics understands-

n for nano
u for micro
k for kilo

and so on.

So, to enter the capacitance value of 1 micro farad, enter "**1u**".

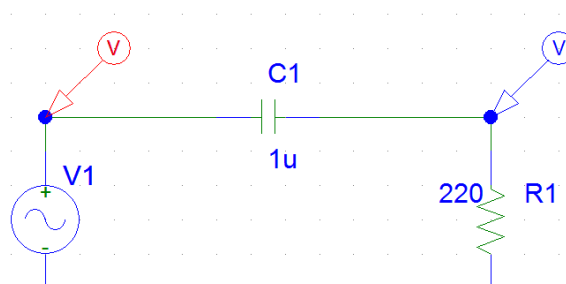
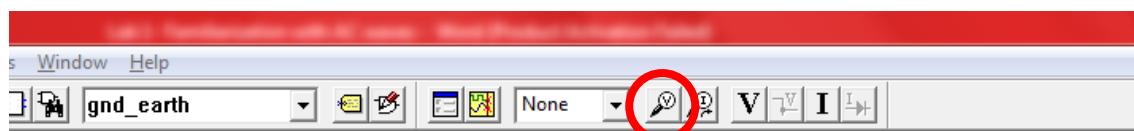
6. Set up the AC voltage source by double clicking on it. In the window that appears, enter the values as shown-



DC=0
AC=0
VOFF=0
VAMPL=5 (if you want 10V peak to peak)
FREQ=1k

There is no need to change any other values.

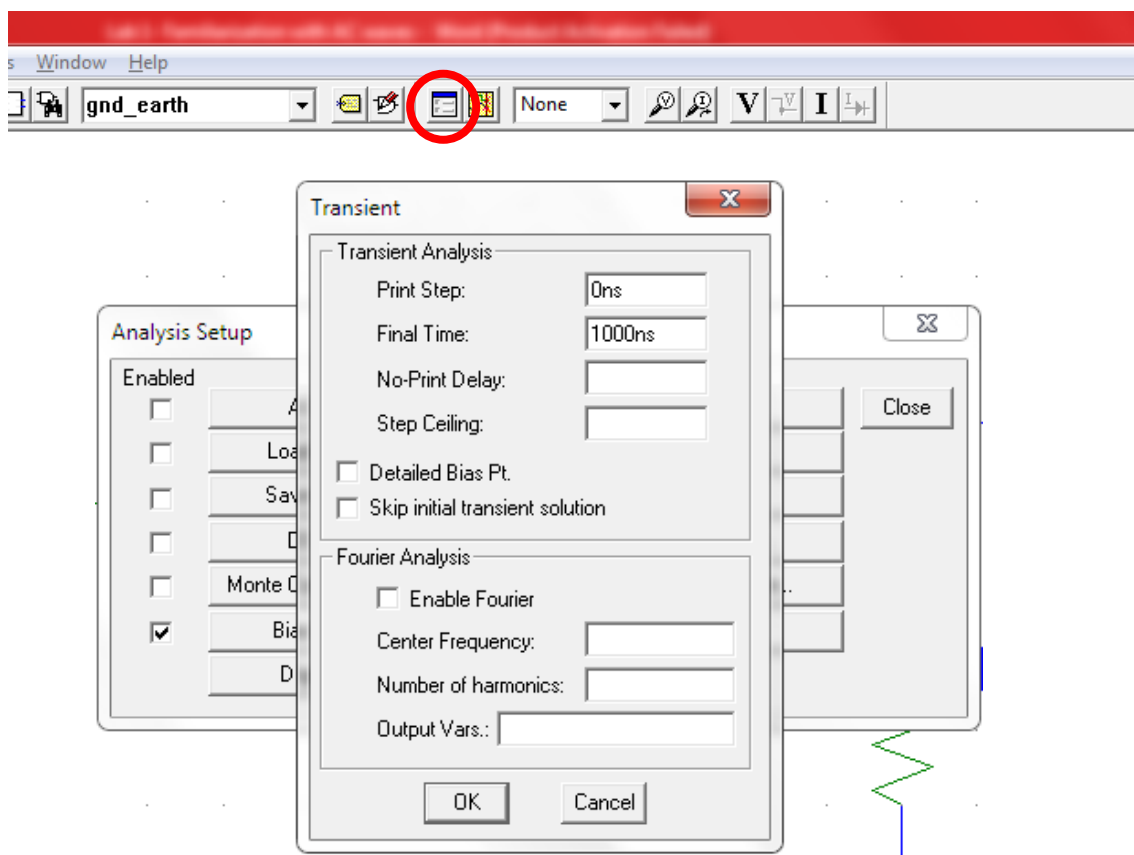
7. Get the voltage markers using the button from the toolbar above, and place the markers above the resistor (load) and the voltage source (Vsin).



CSE250 LABORATORY

Pspice Task 1- Familiarization with AC Waves

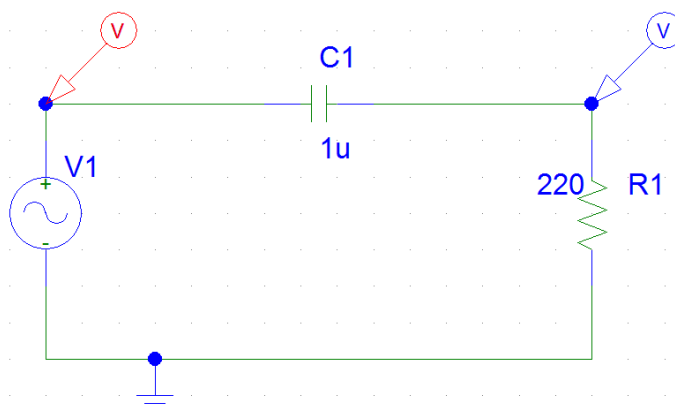
8. Next, from the toolbar again, press on the 'Setup Analysis' button. In the window that appears, click on the button 'Transient'.



For '**Print Step**', type in a very small value, e.g.- **1ns**. (ns= nano seconds)

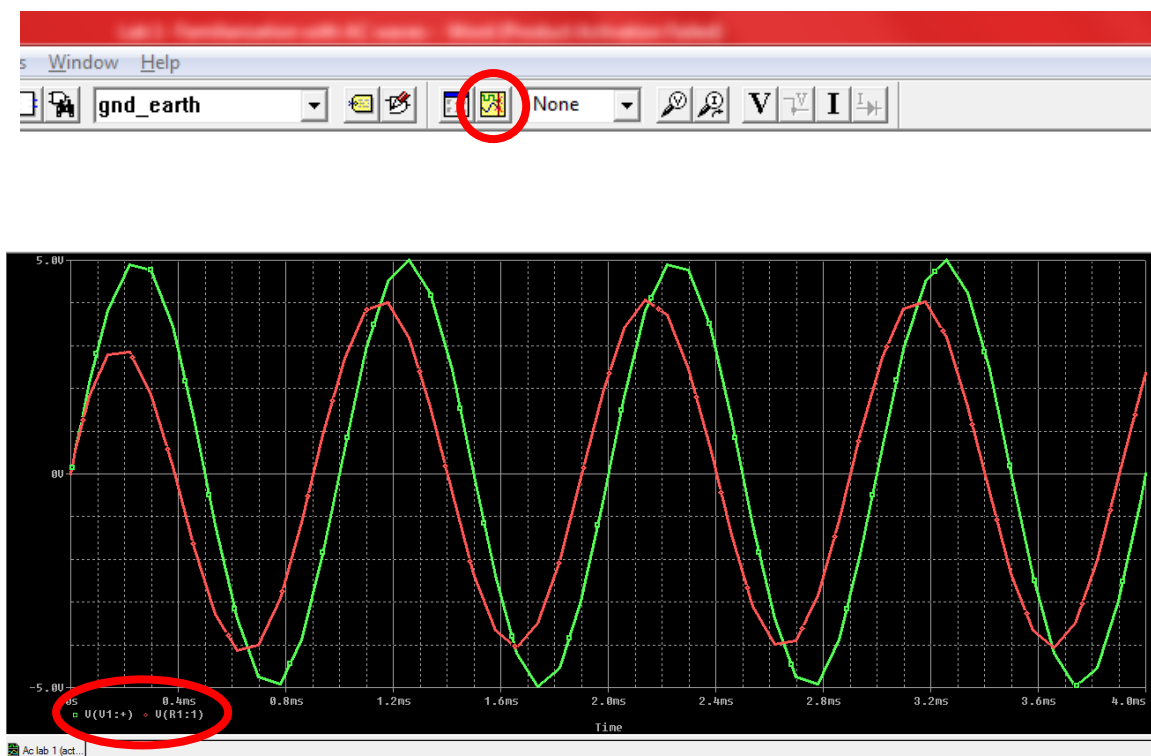
For '**Final Time**', type in 4 ms (ms=millisecond). The final time tells the software when to stop the simulation. We want to see the input and output waveshapes for 4 complete cycles of the input sinusoid; the frequency of our sine wave source was kept at 1 khz, which gives the corresponding time period to be= 1ms. Thus, to see 4 complete cycles, we need to enter (4x1=) 4 ms.

Press 'OK', and then press 'Close' to get back to the schematic design.



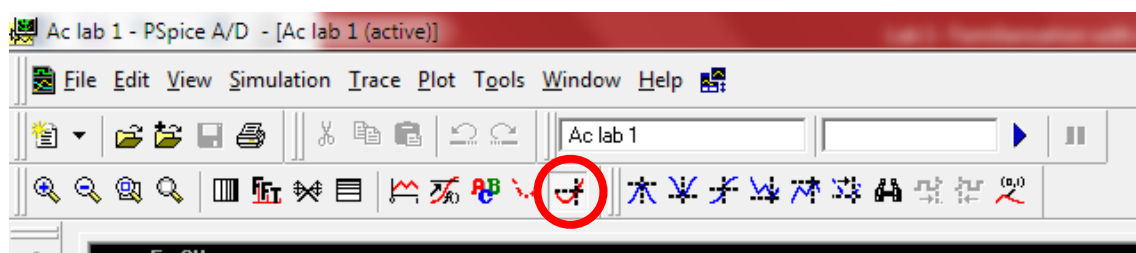
CSE250 LABORATORY
Pspice Task 1- Familiarization with AC Waves

9. Go ahead and save the schematic in your desired directory, and press the button 'Simulate' from the toolbar above-

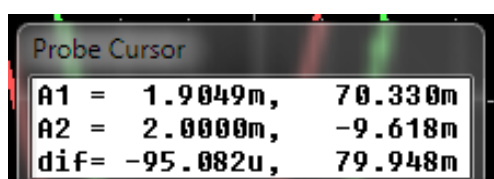


In a new window, you will be shown the simulation results as seen in the figure above. The red circle at the bottom shows where you can find which waveshape is which. In this case, the green one is the input waveshape, and the red one is appearing across the load, i.e. the resistor.

10. In the same window on the top ribbon, find and press the 'Toggle cursor' button.



Notice that as you click on the waveshapes, a cursor runs along the curve. Left clicking selects one waveshape and its corresponding cursor, and right clicking selects the other. A small window with the values of the cursors appear on the window which can be dragged around.



Pspice Task 1- Familiarization with AC Waves

Position the two cursors, such that they both are at the adjacent zero crossings of the two waveshapes, so as to be able to measure the time difference between them. The difference in the x axis appears on the left hand column of the “Probe Cursor” window, beside ‘dif’.

Using this information and the formula $2\pi f t$, find the phase difference between the two waveshapes, where, ‘t’ is the time difference, ‘f’ is the source frequency, and ‘pi’ is equal to 3.142.

Change the source (from Vsin) frequency to 2 kHz, and then to 500 Hz, and for both the frequencies obtain the output and input waveshapes together. Using the cursors, determine the phase difference as well.

Include the schematic circuit design, the waveshapes for all the three frequencies and the calculated phase differences in a new document and submit the hard copy. (See the file- “How to submit Pspice Assignment” for submission guidelines).