The Hong Kong University of Science and Technology
Department of Electronic and Computer Engineering

ELEC2400 **ELECTRONIC CIRCUITS** FALL 2021-22

LAB 2 – PSpice Circuit Simulation

1. Objective

To study PSpice simulation of circuits and verify the network theorems.

2. Component and Instrumentation

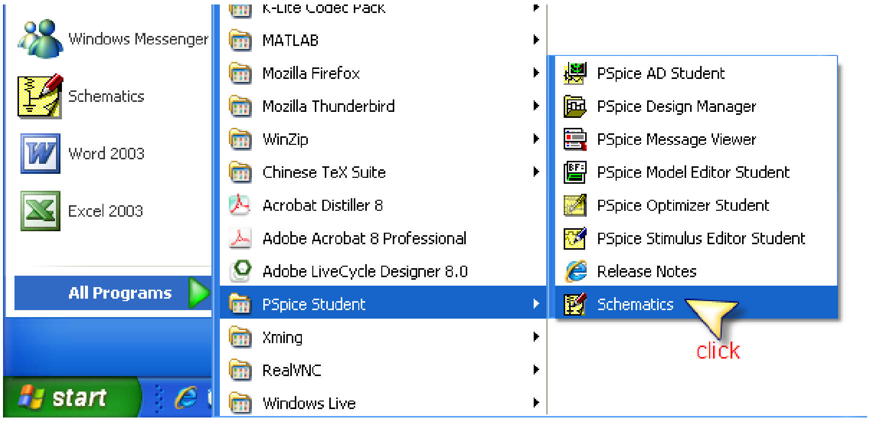
a. PSpice Schematics Version 9.1 software

3. Background Information

PSpice is a very important and powerful circuit-simulation program used by many electrical engineers involved in circuit analysis and design. It can simulate electrical circuit behavior and calculate node voltages, branch currents, power, and other parameters of a circuit. An engineer can study the behavior of circuits without having to actually build the circuits. The circuit can consist of resistors, capacitors, inductors, operational amplifiers, diodes, transistors, semiconductor devices, and other components.

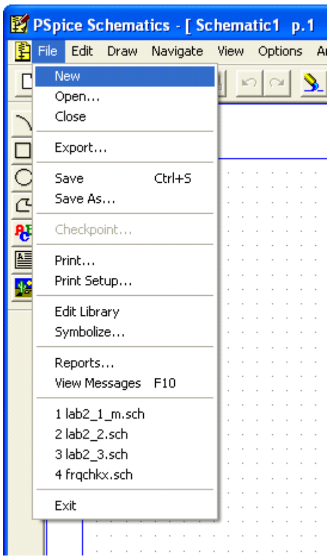
3.1 How to start the PSpice simulator?

By Start >> All Programs >> PSpice Student >>Schematics, you can directly invoke the schematics window.



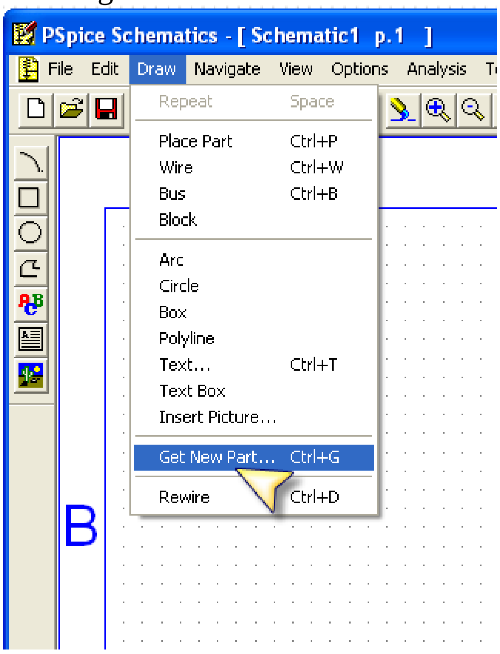
3.2 How to draw the schematics?

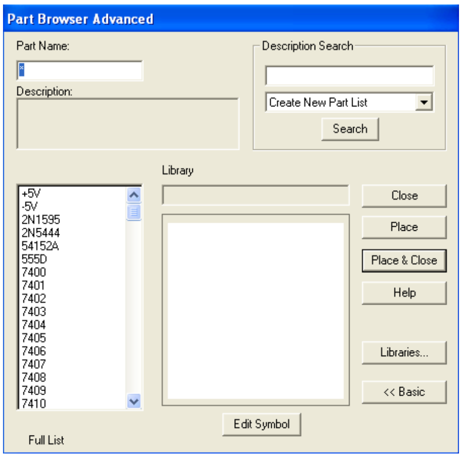
3.2.1) How to create or edit a schematic for simulation?
Select File >> New, or select File >> Open and select the schematics. Please save your schematics before you do the simulation by File >> Save shown below.



3.2.2) How to place the elements in your schematics?

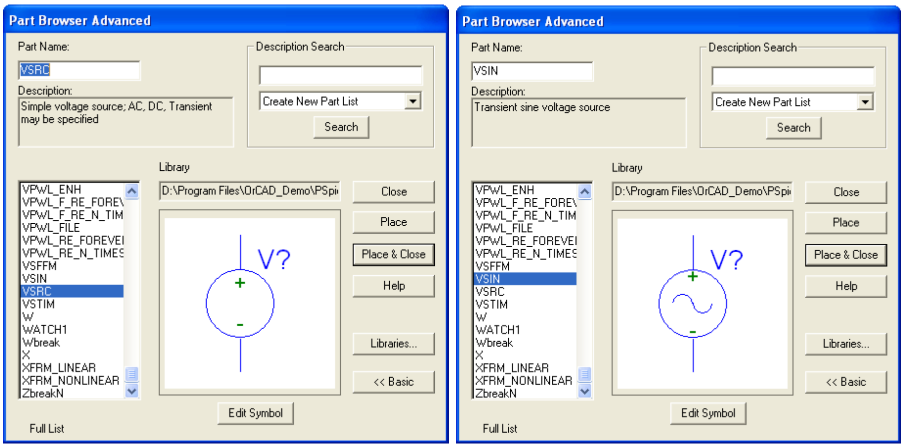
When you draw the schematics, the elements can be found by selecting ***Draw*** >> ***Get New Part…***, and a new window called "Part Browser Advanced" will be invoked. Then you could type the element names in "Part Name:" and Place them in your schematics by clicking "Place & Close".





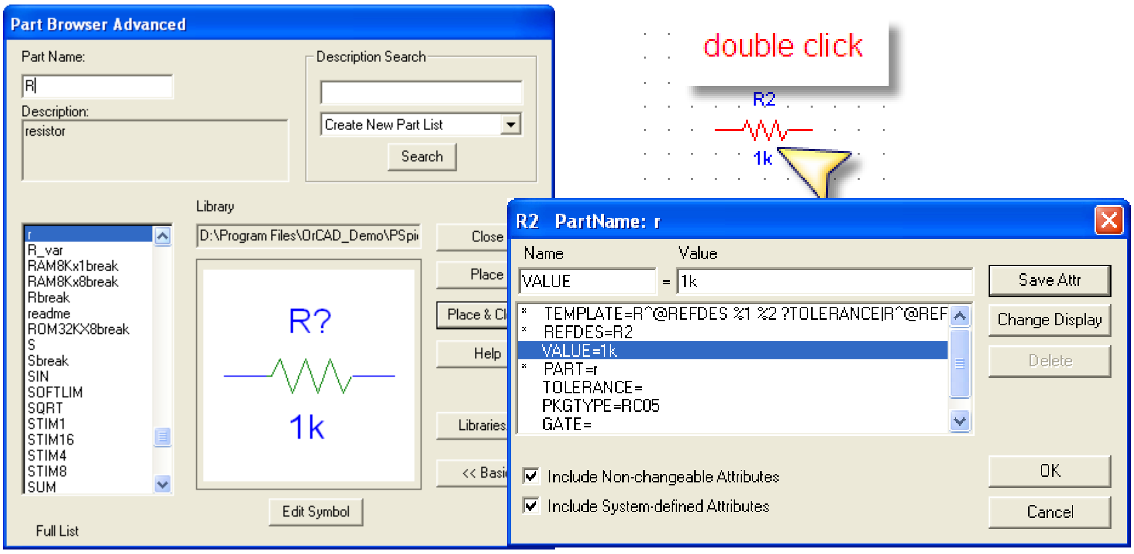
3.2.3) How to place a source?

Select **Draw** >> **Get New Part**. Type **VSRC** in the **Part name** field, click **Place&Close** and place the voltage source symbol in the Schematics. Click left key of mouse to place, repeat if necessary. To cancel the symbol, click right key and select **End Mode** (or press Esc). The source is now shown in red colour (edit mode). Double click the symbol to define the source in the **Property Editor** window After defining the source, close the **Property Editor** window. (Note that no unit is needed in the **Property Editor** window). The similar procedure is applied to a.c. voltage source, **VSIN**; current source **ISRC**; current-controlled voltage source **H**; voltage-controlled current source **G**.

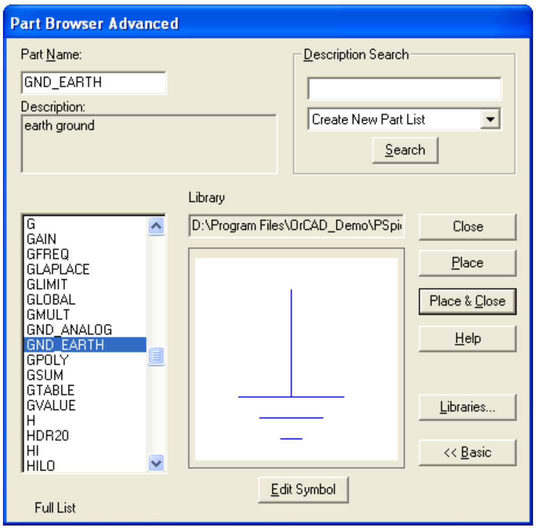


3.2.4) How to place a resistor, capacitor or inductor?

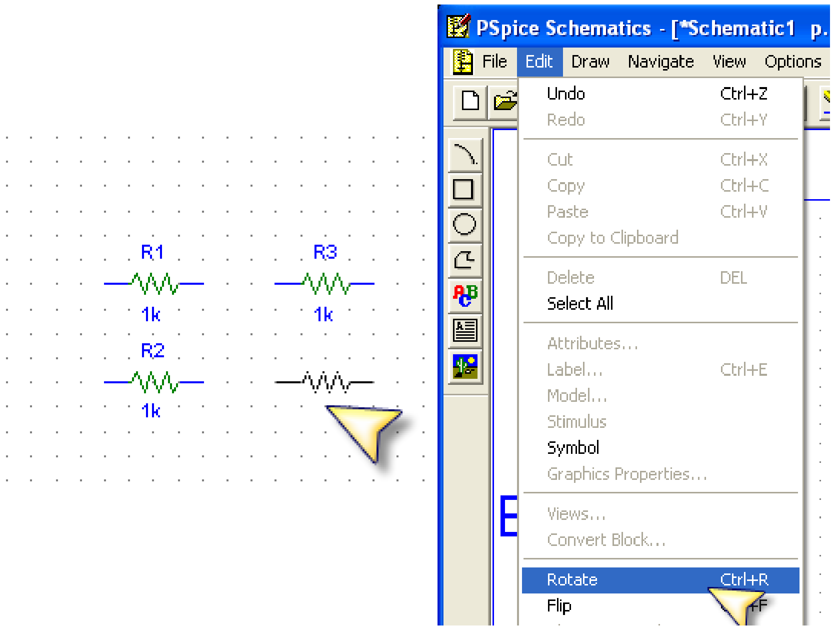
Select **Draw** >> **Get New Part**. Type **R** in the **Part name**. After you place the element in your schematics, you can change the attributes by double clicking the elements. The resistance value is defined in the **Value** field in the **Property Editor** window. For capacitor, **Part name** is **C** and the capacitance is defined in the **Value** field also. For inductor, **Part name** is **L** and the inductance is defined in the **Value** field also.



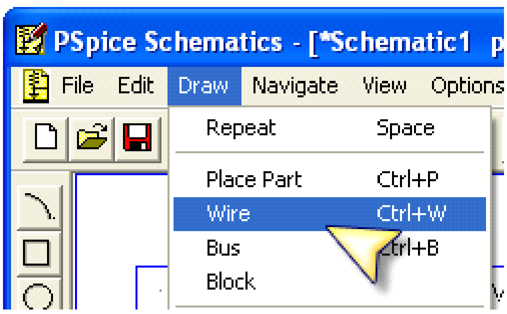
3.2.5) How to place the ground?
Select **Draw >> Get New Part**. The **Part name** is **GND_EARTH.**



3.2.6) How to rotate the elements?
To rotate a part, select it and choose **Edit >> Rotate**.



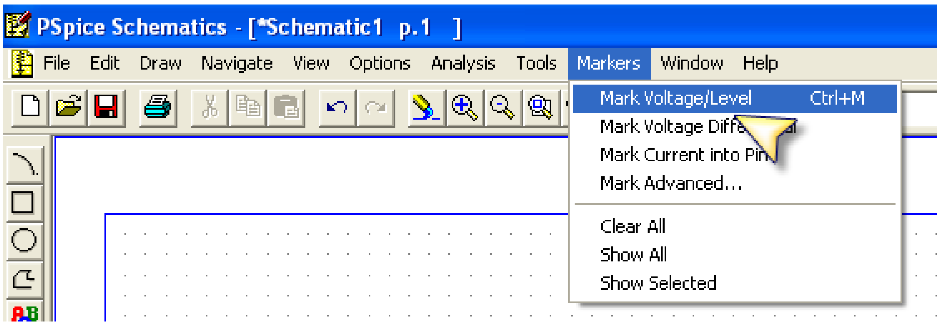
3.2.7) How to make a connection?
Select **Draw >> Wire**. The cursor is now changed to a pencil. Place it on one end of a component (on the red square dot), click once, move the cursor to end of another component, click once. When finish, right click and the cursor will be recovered .



3.3 How to start the simulations?

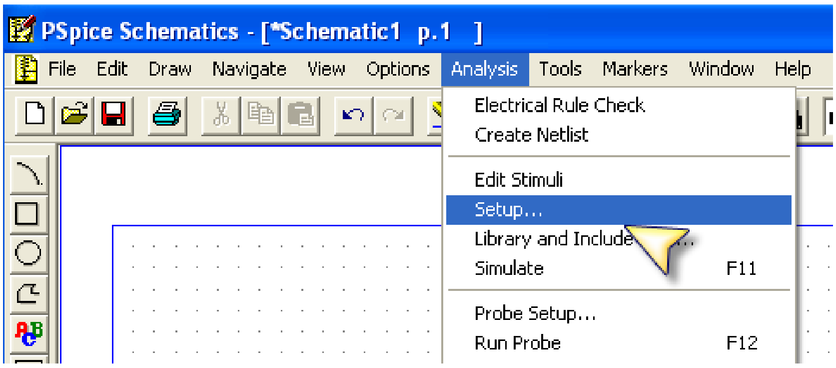
3.3.1) How to place the markers?

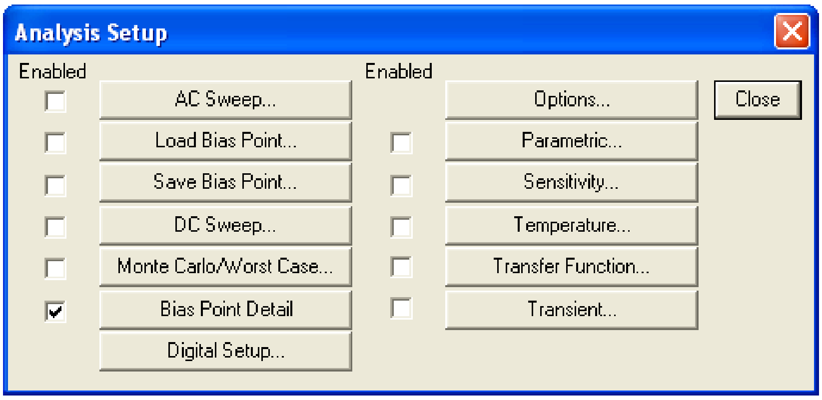
The marker is used to indicate the points for which you want to see simulation waveforms displaying in PSpice. Before placing a marker, it is necessary to complete the schematic, save it. Then select **Markers >> Mark Voltage/Level**. The resultant voltage waveform will be automatically shown after running the simulation. Also, you can select Current markers by selecting **Markers** >> **Mark Current into Pin**



3.3.2) How to set the simulation?

To simulate a new design (new schematic), you need to save the schematic first and then select **Analysis** >> **Setup**… to manage the several different types of analyses such as DC analysis, AC analysis and so on.



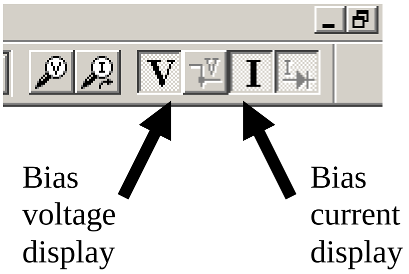


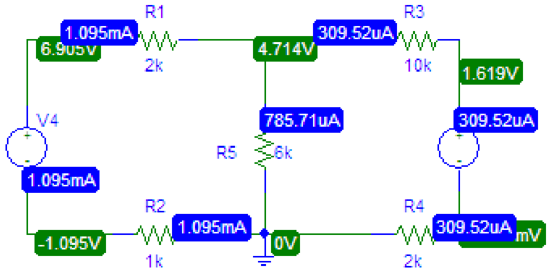
3.3.3) How to run the simulation?

Select **Analysis** >> **Simulate**. Before you simulate your circuit, please save the schematics first. For transient voltage or AC voltage plot, you should place **Voltage Level** marker in **Markers** at the node being examined in the circuit diagram.

3.4 How to check the simulation results?

3.4.1. How to check the DC voltage and DC current?
You can enable Bias-Voltage-Display and enable Bias-current-Display by clicking the short-cut icon, and the voltage and current are displayed in the circuit.





3.4.2. How to check the signal waveform?
You can examine the result in the invoked window called “OrCAD Pspice A/D Demo” after you select **Analysis** >> **Simulate**.

4. Experimental Procedures

4.1 Kirchhoff's Current Law (KCL) and Kirchhoff's Voltage Law (KVL)

4.1.1. Draw the circuit shown in Figure 1 in Schematics.
4.1.2. Enable the *Bias Point Detail* analysis in Analysis->Setup, and run the PSpice simulation.
4.1.3. Complete Table 1 according to your simulation and calculated results.

Q1. Complete Table 1 according to your simulation results.

Table 1

	PSpice Simulation Results
Voltage Va	
Current Ia	

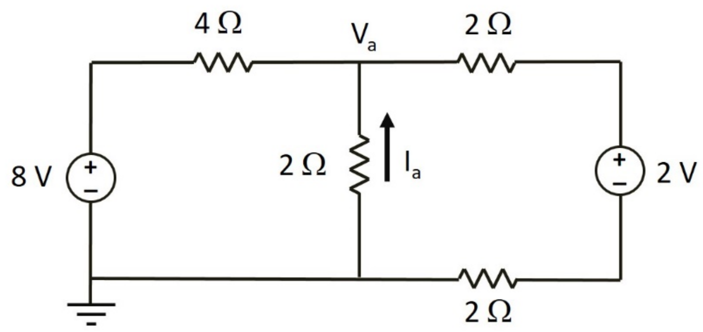


Figure 1

4.2 Thevenin`s Theorem

4.2.1. Draw the circuit as shown in Figure 2 in Schematics. Run the simulation and find the voltage across the 1Ω.

Q2. What is the simulated Vab? ________________V

4.2.2. Remove the 1Ω . Run the simulation and find the open circuit voltage V_{OC} at terminals ***ab***. (Hint: Avoid having simulation error, the 2Ω resistor should be replaced by a wire.)

Q3. What is the simulated V_{OC}? ________________V

4.2.3. Short the terminals ***ab*** with a wire. Run the simulation to find the short-circuit current **I_{SC}** at terminal ***ab***.

Q4. What is the simulated I_{SC}? ________________A

4.2.4. Calculate the Thevenin resistance R_{TH} (= V_{OC} / I_{SC}).

Q5. What is the calculated R_{TH}? ________________**Ω**

4.2.5. Calculate I_O and V_{ab} using the circuit shown in Figure 3.

Q6. What is the calculated I_O? ________________A

Q7. What is the calculated V_{ab} ? ________________V

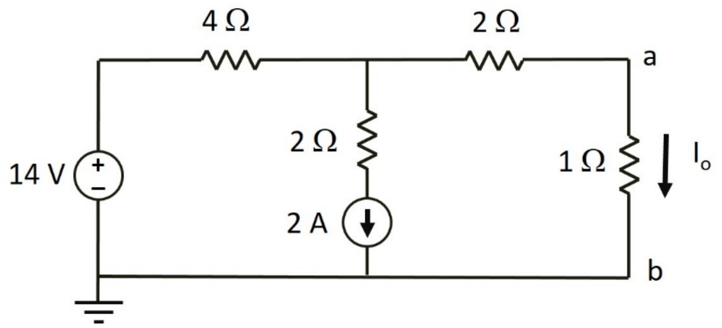


Figure 2

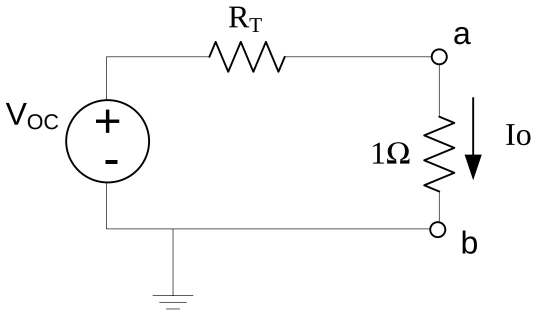


Figure 3

4.3 Circuit with dependent source

4.3.1. Draw the circuit shown in Figure 4 in Schematics.
(The part name for current-controlled voltage source in PSpice is "H" (set gain to 4).

4.3.2. With the current-controlled voltage source (H) in PSpice, the corresponding circuit is shown in Figure 5.

4.3.3. Enable the *Bias Point Detail* analysis in Analysis->Setup.

4.3.4. Run the simulation and find the simulated Va.

Q8. What is the simulated Va? ________________V

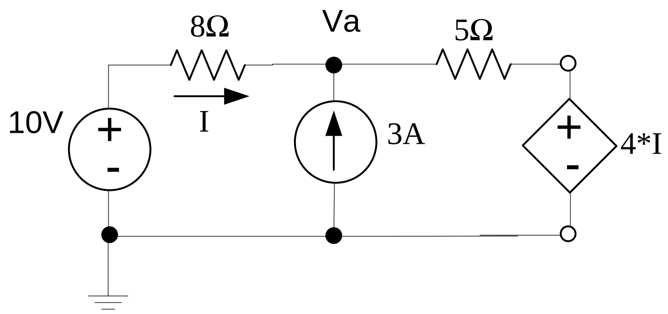


Figure 4

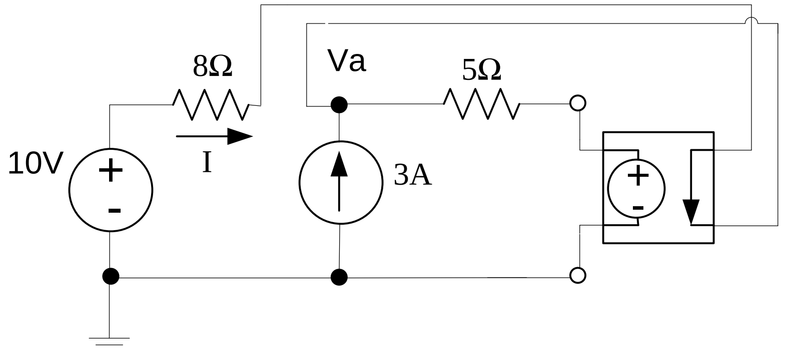


Figure 5

4.4 AC Circuit

4.4.1. Draw the circuit shown in Figure 6 in Schematics. $V_1(t) = 3 \sin(2\pi*50k*t)$ V.

4.4.2. Use VSIN as the voltage source. Set DC (dc value of V_1) to 0. Set VAMPL to 3, FREQ to 50k, VOFF to 0.

4.4.3. Enable the *Transient* analysis in Analysis->Setup.

4.4.4. In the *Transient* analysis settings, set the No-PRINT DELAY (start saving data after) to 8000u, FINAL TIME to 8050u and STEP CEILING (step size) to 0.01u.

4.4.5. Run the simulation to find $V_a(t)$

[Given that $V_a(t) = V_m \sin(2\pi*50k*t + \theta_v)$V. θ_v is the phase difference between $V_a(t)$ and $V_1(t)$. $\theta_v = (t / T)*360^o$. V_m is peak value of the signal]

Q9. What is the equation of $V_a(t)$? ____________________V

4.4.6. Enable the *AC Sweep* analysis in Analysis->Setup.

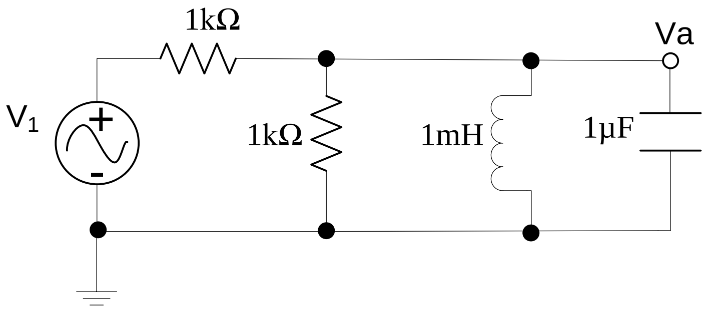
4.4.7. Set AC (AC value) to 2.121 (3 / sqrt(2) = 2.121),

4.4.8. In the *AC Sweep* analysis setting, select DECADE for log scale and set the START FREQ. (start frequency) to **1k** (1 kHz), the END FREQ. (end frequency) to **1meg** (1 MHz) and the PTS/DECADE (points per decade) to 100.

4.4.9. Run the simulation. Find the frequency (resonant frequency) when Va (the voltage at node **a**) is maximum. Find the maximum Va .

Q10. What is the maximum Va? ________________V

Q11. What is the frequency? ___________ Hz



s

Figure 6

Q12. Show all the circuits and waveforms to TA.