

BIRZEIT UNIVERSITY/ELECTRICAL ENGINEERING DEPARTMENT

PSpice User Manual

Student Version 9.1

PREPARED BY: Eng. AMANI ATIANI

9/23/2010

Table of Contents

Chapter1: Before You Begin	4
1.1 Welcome	5
1.2 Before You Getting Started	6
Chapter2: Getting Started	7
2.1 Overview	7
2.2 Starting a New design	8
Chapter3: Simple Example	12
Chapter4: Plotting Traces	19
4.1 Overview of DC Sweep	20
4.2 Overview of AC Sweep	25
4.3 Overview of Transient Analysis	28
Chapter5: More Examples and Exercises	32
References:	46

Chapter1

Before You Begin

1.1: WELCOME

In this manual we will explain how you can use PSpice Schematics. PSpice Schematics is one of the Orcad family products which offers a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, Orcad family products are a suite of applications built around an engineer's design flow—not just a collection of independently developed point tools.

PSpice Schematics is just one element in our total solution design flow.

PSpice Schematics is a schematic capture front-end program with a direct interface to PSpice.

PSpice Schematics is fully integrated with PSpice, giving you the flexibility to work through your circuit design in a consistent environment.

In one environment, you can do all of the following using PSpice Schematics:

- design and draw circuits
- simulate circuits using PSpice
- analyze simulation results .

1.2: Before You Getting Started

Make sure that the **PSpice Student version 9.1** package is already installed on your computer.

Getting Started

2.1:Overview:

This chapter describes Schematics: what it is, what it can do, and how you can use it.

Using Schematics:

Schematics is a schematic capture front-end program that provides a convenient system for:

- Creating and managing circuit drawings.
- Setting up and running simulations.

How to Draw a Schematic:

The following example demonstrates the basic drawing features for drawing a schematic. It shows you how to:

- Start the schematic editor and begin a new design.
- Find out which libraries are configured for Schematics.
- Place parts on a schematic.
- Connect the part using wires and buses.
- Label wires and buses.
- Change reference designators and part values.
- Move parts, wires and text.
- Use ports on a schematic.
- Place power and ground symbols on a schematic.
- Save your design.

2.2: Starting a New Design

In order to start building your design:

- **Step1 (Fig2.2.1):** Go to Start >>All Programs>> PSpice Student >>Schematics:



Fig.2.2.1: How to open a new Schematics design

✓ After you click on Schematics icon ,you can see the following:

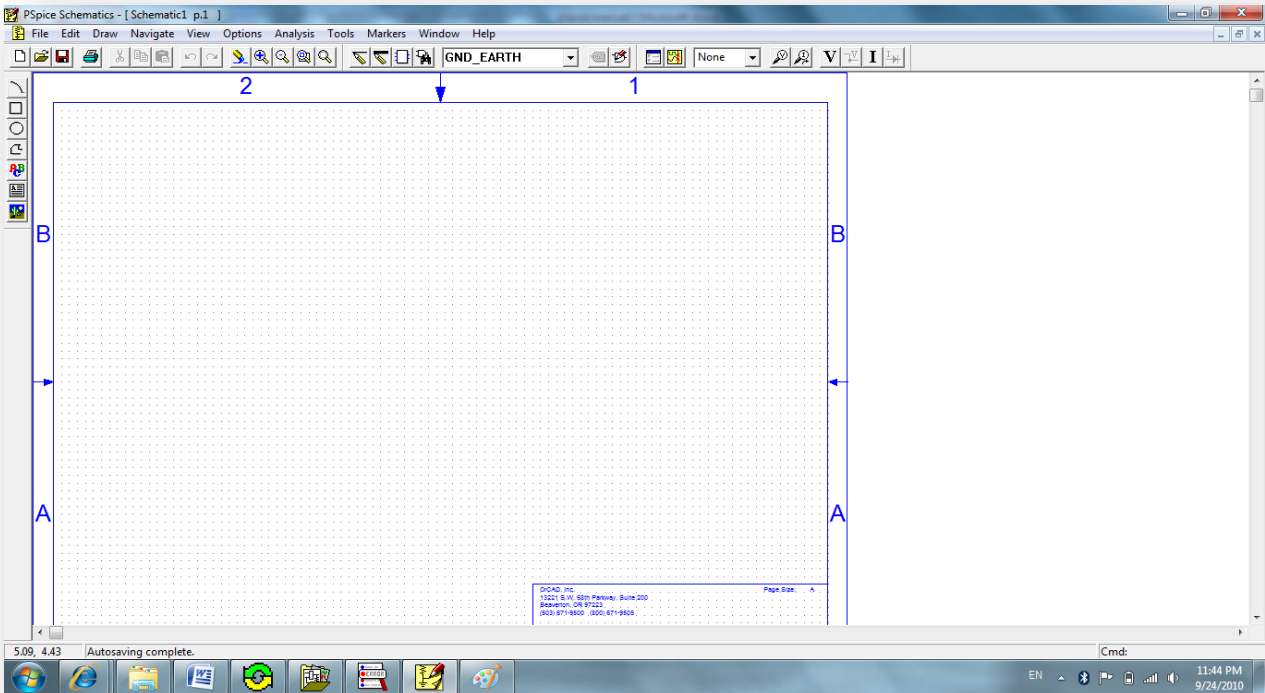



Fig.2.2.2: A new Schematics design page.

- **Step2:Selecting and Placing Parts**

From the Draw menu, select **Get New Part** to display the Part Browser dialog box, as shown in Fig2.2.3, or you can click on  directly from the menu toolbar. Then the Part Browser dialog box will appear .

Note: One of two Part Browser dialog boxes may appear: **the Part Browser Advanced** or **the Part Browser Basic**. If in the Part Browser Advanced dialog appears, click <<Basic to display the Part Browser Basic and vice versa.

Note: The Full List in the Part Browser dialog box represents all the parts in the configured symbol libraries that are available for your use.

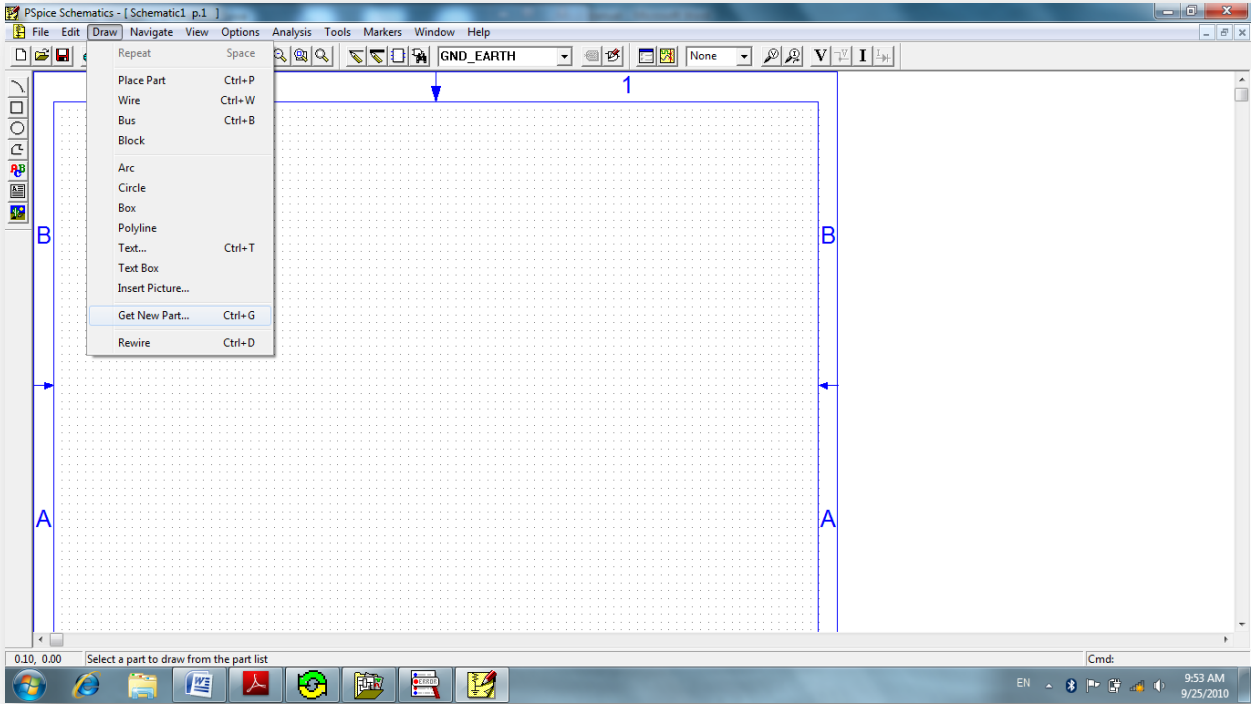


Fig2.2.3: How can you getting your design parts.

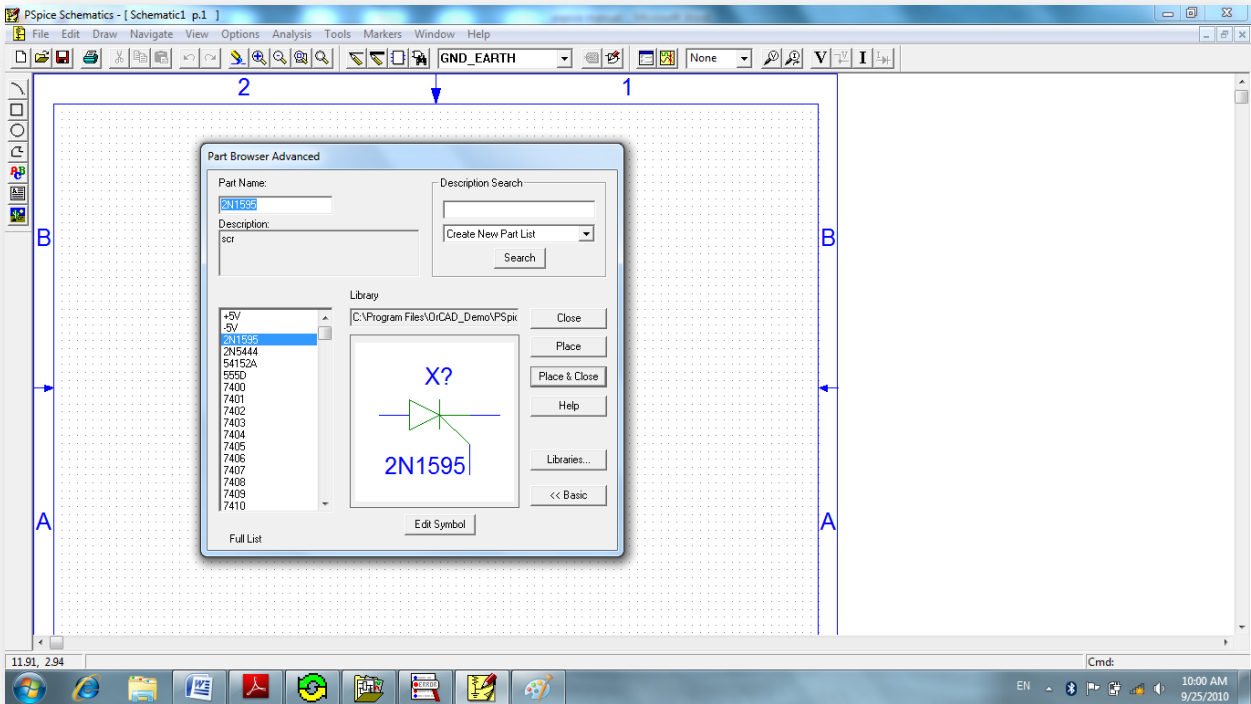


Fig2.2.4: Parts Browser Dialog Box.

- ✓ There are several ways to select a part in the Part Browser dialog box:
 - If you know the name of the part, type the name in the Part Name text box.
 - Select the part name from the Full List of part names.
 - Click Libraries to view the Library Browser dialog box, select a library, and select the part name from that library's list of parts.

Note: Another method of selecting a part is to use the Get Recent Part list box on the toolbar. You can scroll and select a previously placed part, or you can type the name of the part you want to place as shown below:



Fig.2.2.5: The Get Recent Part list box.

✓ **To Placing Parts:**

Click **Place** to place the part (with the browser remaining open) or click **Place & Close** (to place the part and close the browser).

If you leave the browser open, click **the title bar of the dialog box and drag it to a new location**. The Full List in the Part Browser dialog box represents all the parts in the configured symbol libraries that are available for your use.

Simple Example:

The following example demonstrates the basic drawing features for drawing a schematic.

Question.1 : Draw the following circuit using PSpice , then find the voltages and currents at **all nodes**.

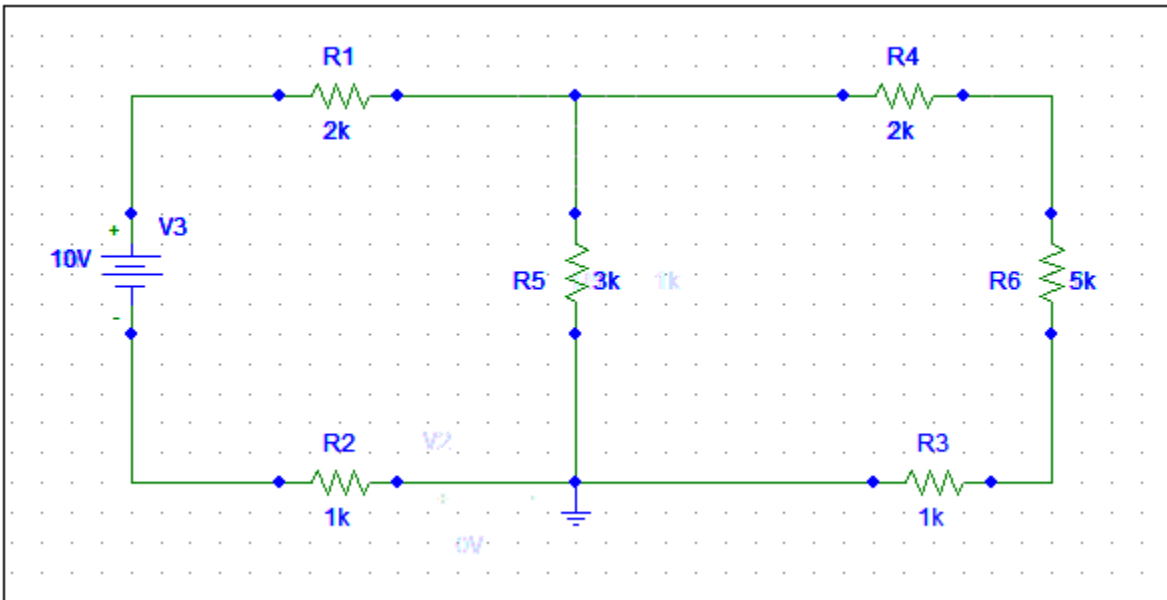





Fig3.1: Question.1

Solution:

- 1- Start the schematic editor by double-clicking on the Schematics icon  in the Orcad program group. An empty schematic page displays.
- 2- If you already have Schematics running with another displayed, click the New File icon  to start a new schematic.

- 3- From the Draw menu, select Get New Part  to display the Part Browser dialog box.
- 4- Start placing the dc voltage for example. To place Vdc type **VDC** under the part name space in the Part Browser dialog box. Then the available VDC sources will be appeared as shown below:

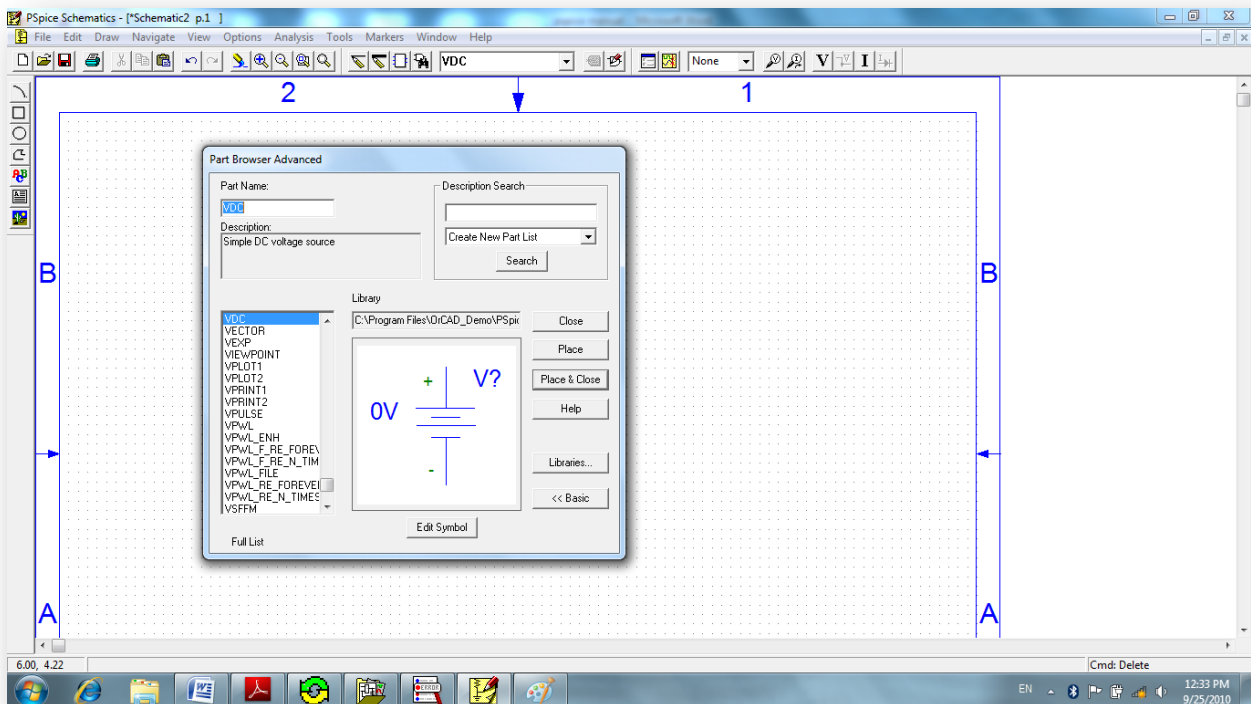


Fig 3.2: Placing the DC Voltage Source.

- 5- Do the same to place the resistors by typing R in the part name space then place how many resistors you need. you can rotate any one of the circuit components by pressing (**Ctrl+R**) you also need to determine the circuit ground, to place the ground type **GND** as shown in the following **Fig3.3**.

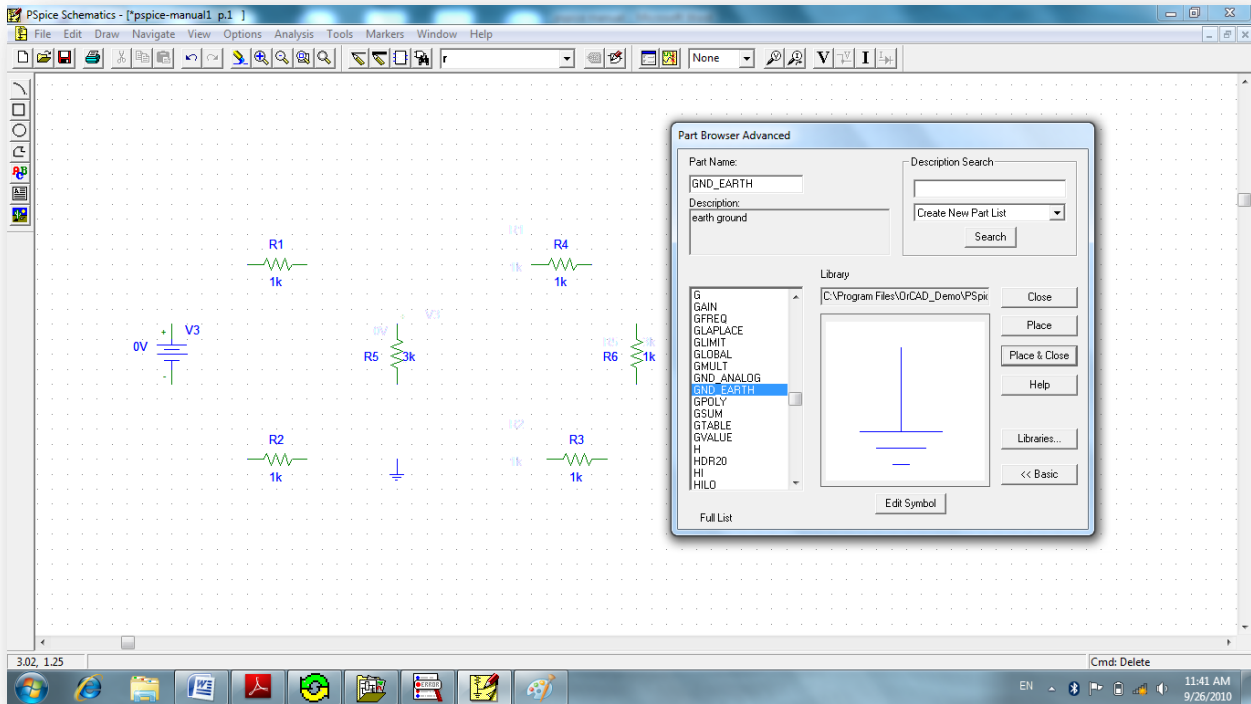




Fig.3.3: Placing the earth ground.

After this step, all of your design elements are placed you need now to connect between them by wires.

✓ **Drawing and Labeling Wires:**

Continue working on the same example (i.e: example 1), to draw the wires:

- 1- Click the Draw Wire button , The pencil pointer  indicates that you are ready to draw a wire.
- 2- Click where you want each vertex of the wire. Each click ends a wire segment and starts a new one.

Note: Notice that the wire is now ended where you clicked to place a pin. The pointer remains in the shape of a pencil and you are ready to start another wire.

- 3- Wire the rest of the schematic. Right-click to stop drawing wires.

4- To label any wire segment :

- Double-click any segment of the wire to display the Set Attribute Value dialog box.
- Type in the LABEL text box then click OK.

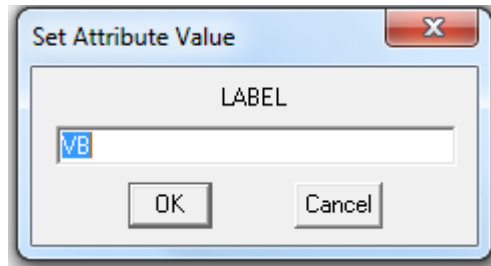


Fig.3.4: The Set Attribute Value dialog box.

For the example above, I want to label the voltage across R6 as VAB, this was done as shown in the figure below:

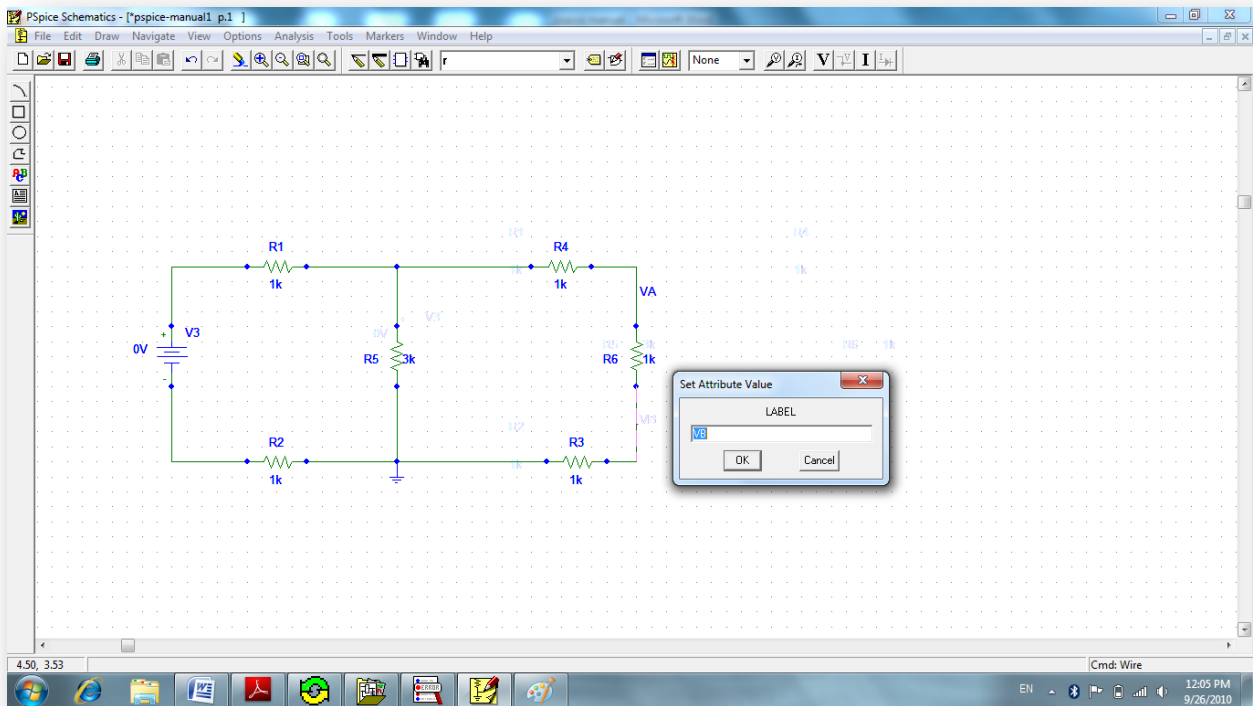


Fig.3.5: Labeling the wires.

✓ **Changing Reference Designators and Part Values:**

Change part values and reference designators by double-clicking them and typing a new value in the dialog box.

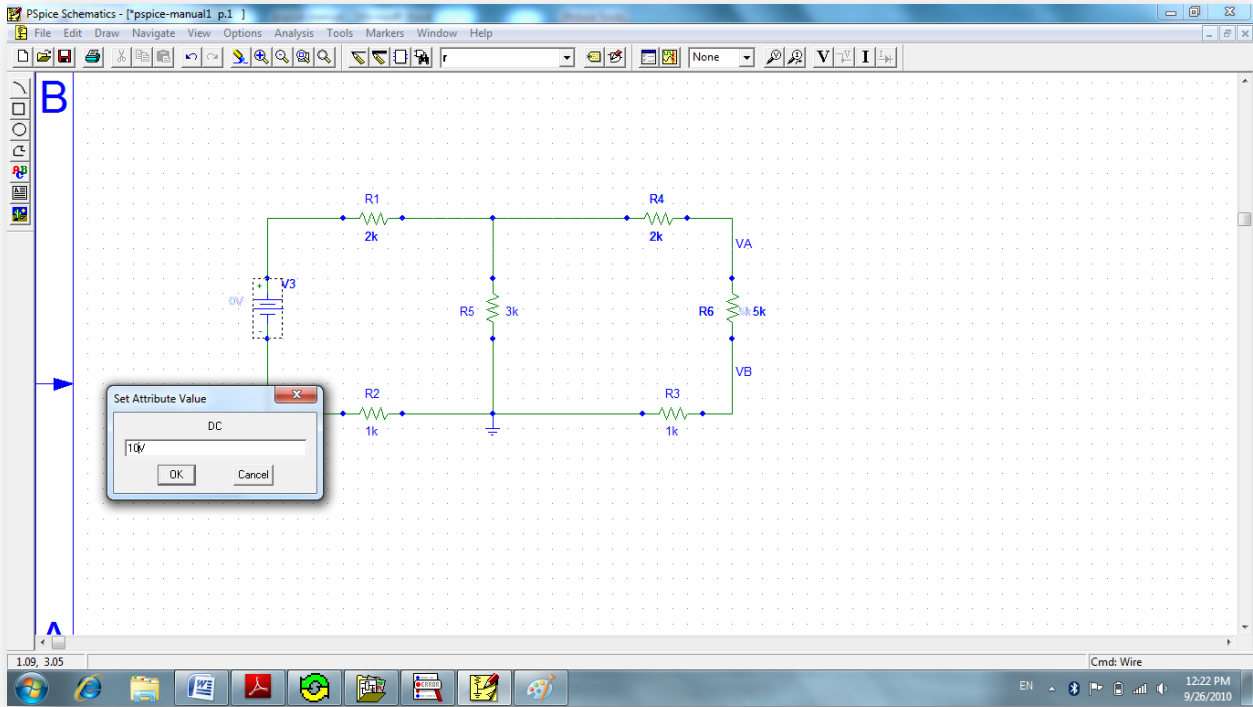


Fig.3.6:Change the part values.

✓ **Saving Your Work:**

After you changed all the values of the parts as you want in the design , you need now to save your work.

Click the **File Save** button , or select **Save (or Save As)** from the **File** menu to save the schematic.

✓ **Simulate your Circuit:**

In order to find the voltages and currents all circuit's nodes you need to simulate your circuit.

- Click on Simulate button  in the toolbar.

- To display the voltage values click on **Enable Bias Voltage Display** button **V**. (All voltages displayed are with respect to ground)
- To display the current values click on **Enable Bias Current Display** button **I**.

The values for all currents and voltages are displayed in Fig.3.7.

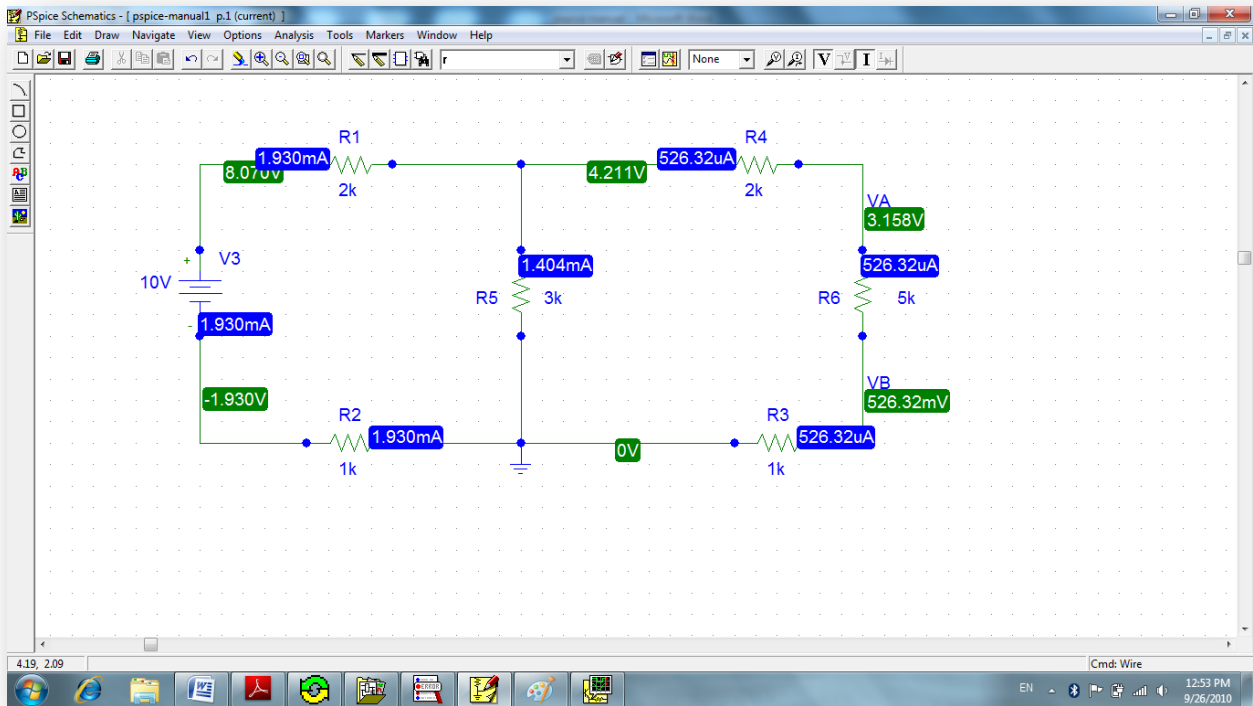


Fig.3.7: The Circuit Simulation.

Note: If you click on one of the current values you can determine its direction in the branch as explained in Fig.3.8:

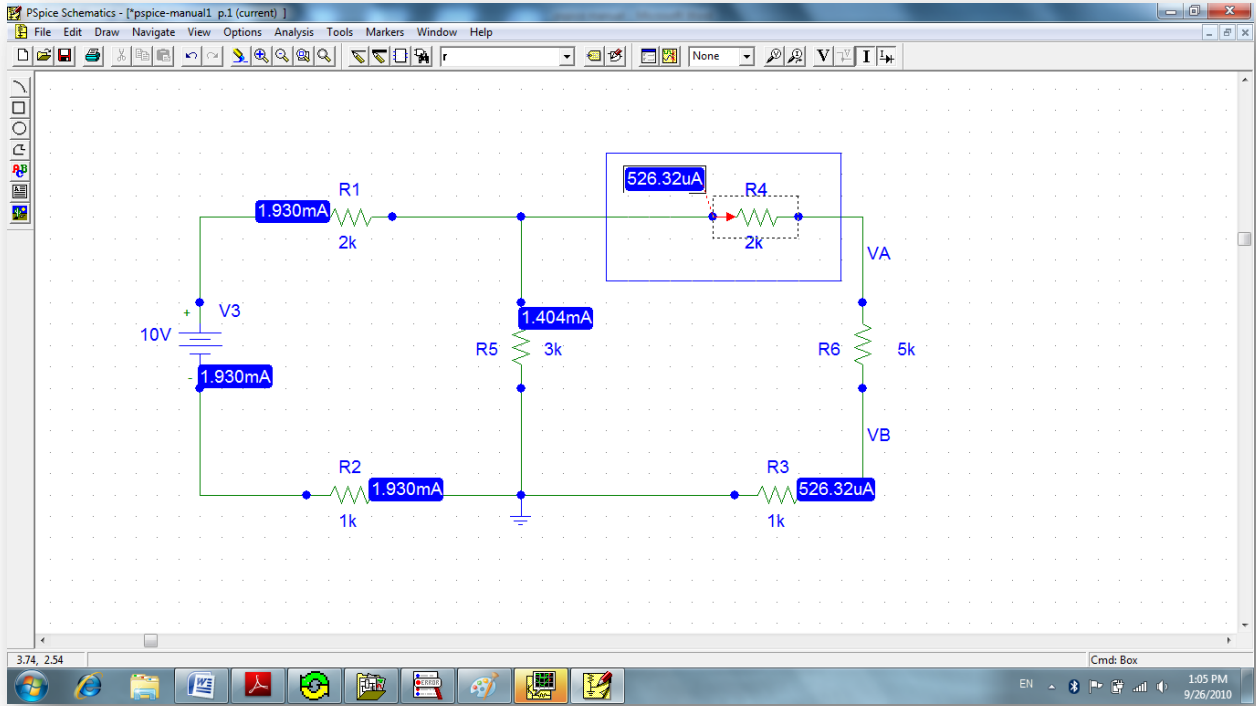


Fig.3.8: Determine the current direction.

Plotting Traces:

The PSpice Schematics has the ability to draw traces for voltages and currents in your design. There are different types of Plotting Analysis which the PSpice can deal with but in this manual we will explain only three types:


- DC sweep
- AC sweep
- Transient Analysis

4.1: Overview of DC Sweep:

✓ What is DC sweep?

The DC sweep analysis causes a DC sweep to be performed on the circuit. DC sweep allows you to sweep a source (voltage or current), a global parameter, a model parameter, or the temperature through a range of values.

Let us know how the voltage across R5 changes as the DC voltage source V3 sweeps from 0V to 10V in Example(page 12).To do this:

- 1- Click on Setup Analysis button . A small window will be opened as shown below:

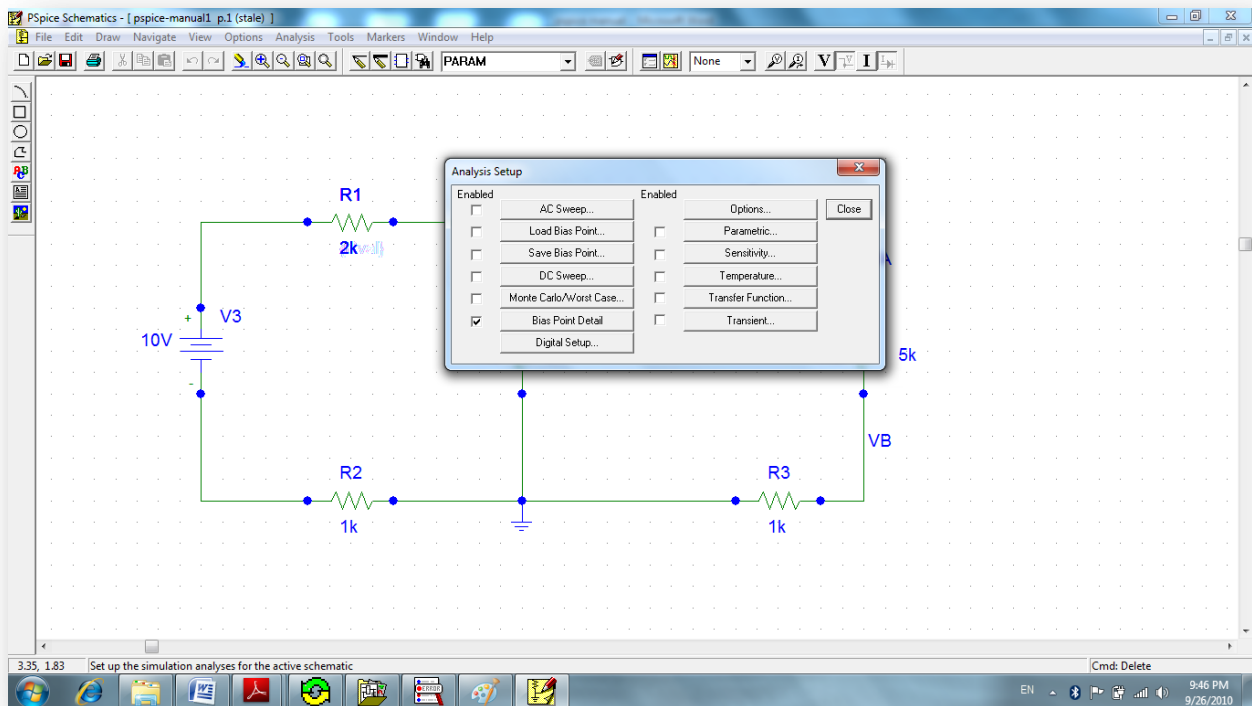
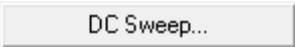


Fig.4.1.1: DC sweep Analysis.

- 2- Click on the DC sweep button  then the DC sweep window appears.

- 3- Choose **Voltage Source Type**>> **Name:V3**>> **Linear sweep** >> **Start Value=0,End Value=10,Increment =1** then click **OK** (Show Fig.4.1.2).
- 4- Checked the **DC Sweep** box keeping the **Bias Point Detail** Checked.
- 5- Click **Close** to close the window.

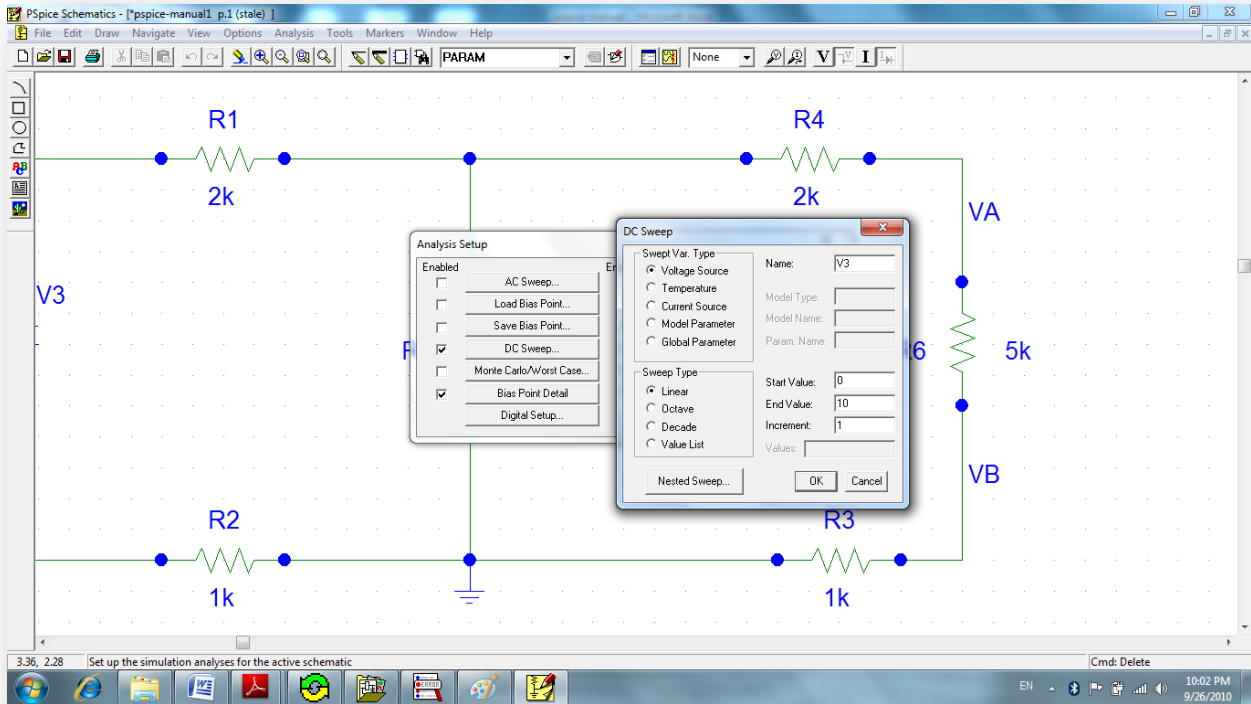







Fig.4.1.2: Set the DC Sweep Parameters.

- 6- To display the voltage across R5 click on **Volatge/Level Marker**  and place it where you want as shown in Fig.4.1.3. You can do the same thing by clicking upon **Current Maker** .
- 7- Click on **Save** again then click **Simulate**.
- 8- A black window appears plotted the voltage across R5 as the dc voltage source V3 changes from 0V to 10V. (As shown in Fig.4.1.4)
- 9- After the simulation window opened you can click on  to add a new trace on the same plot. A new window will be opened listing all the variables that can be plotted, choose for example V(VA) which plots VA.

- 10- Click on **Plot >> Add plot to window** to divide the black window to a multiple plots as you want.
- 11- By clicking on  you can determine the point on the graph then click on  to display its value.

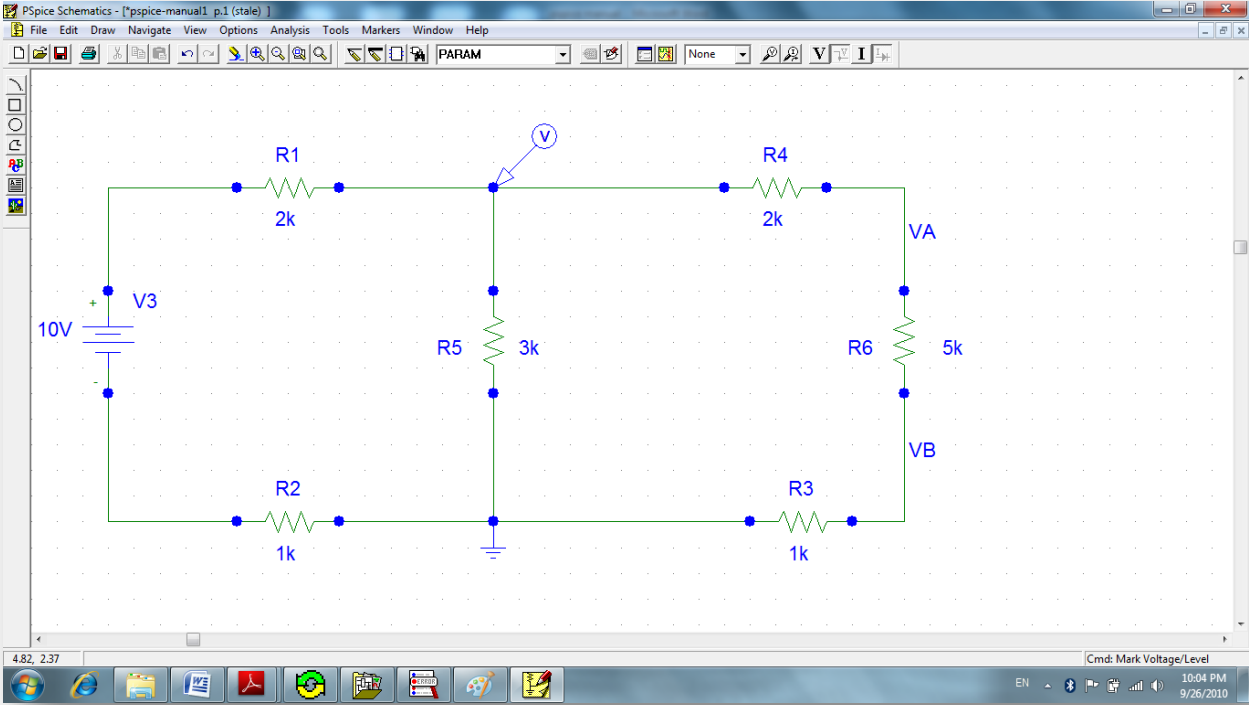


Fig.4.1.3: How to use the Voltage/Current Makers.

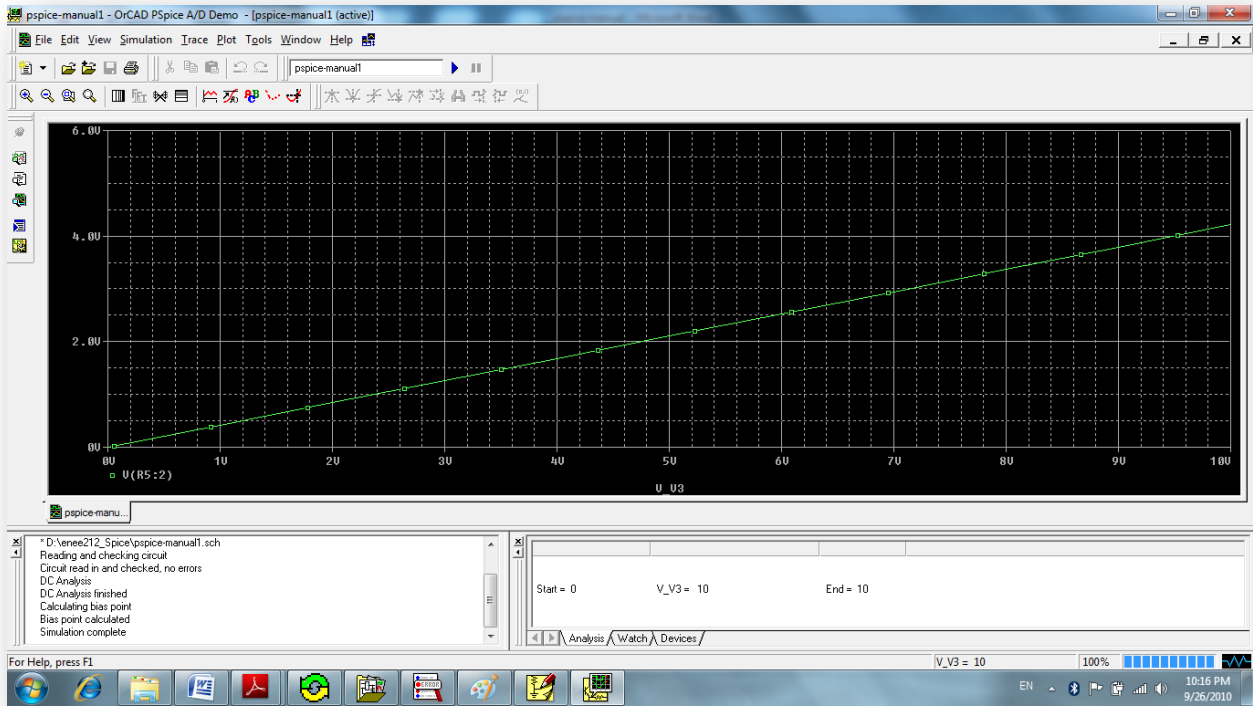


Fig.4.1.4: Simulation results Screen.

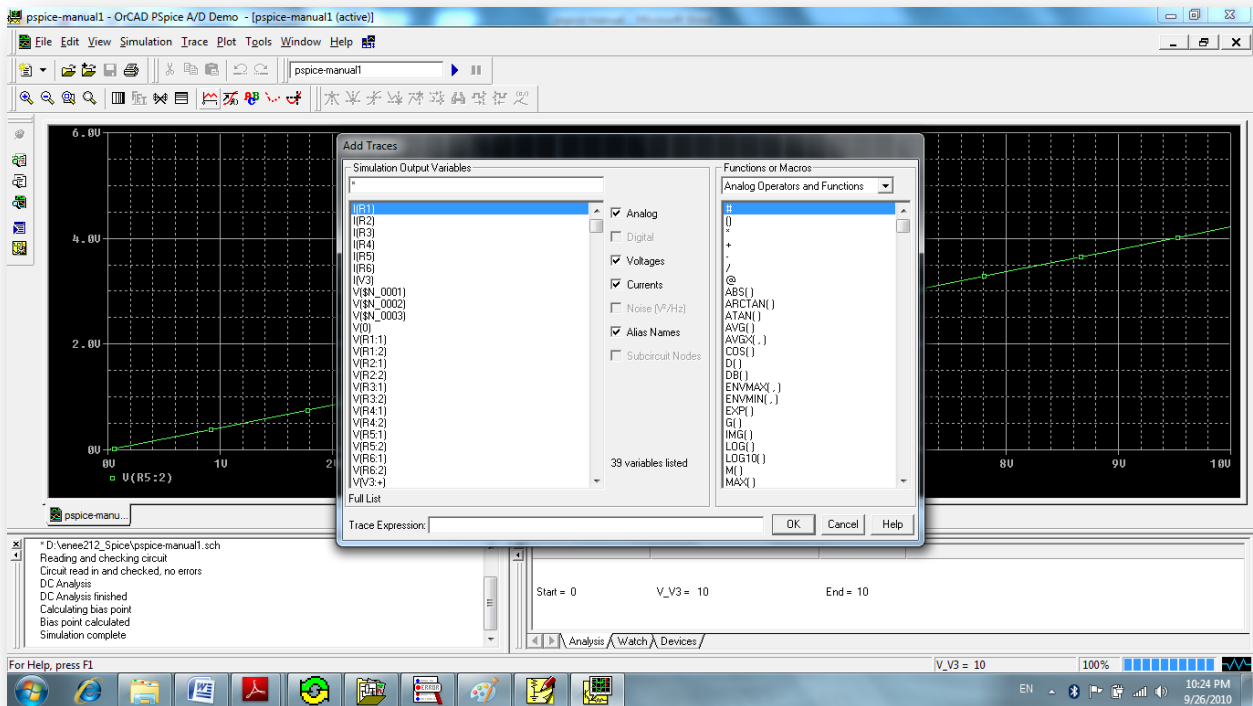


Fig.4.1.5:How can you add a Trace.

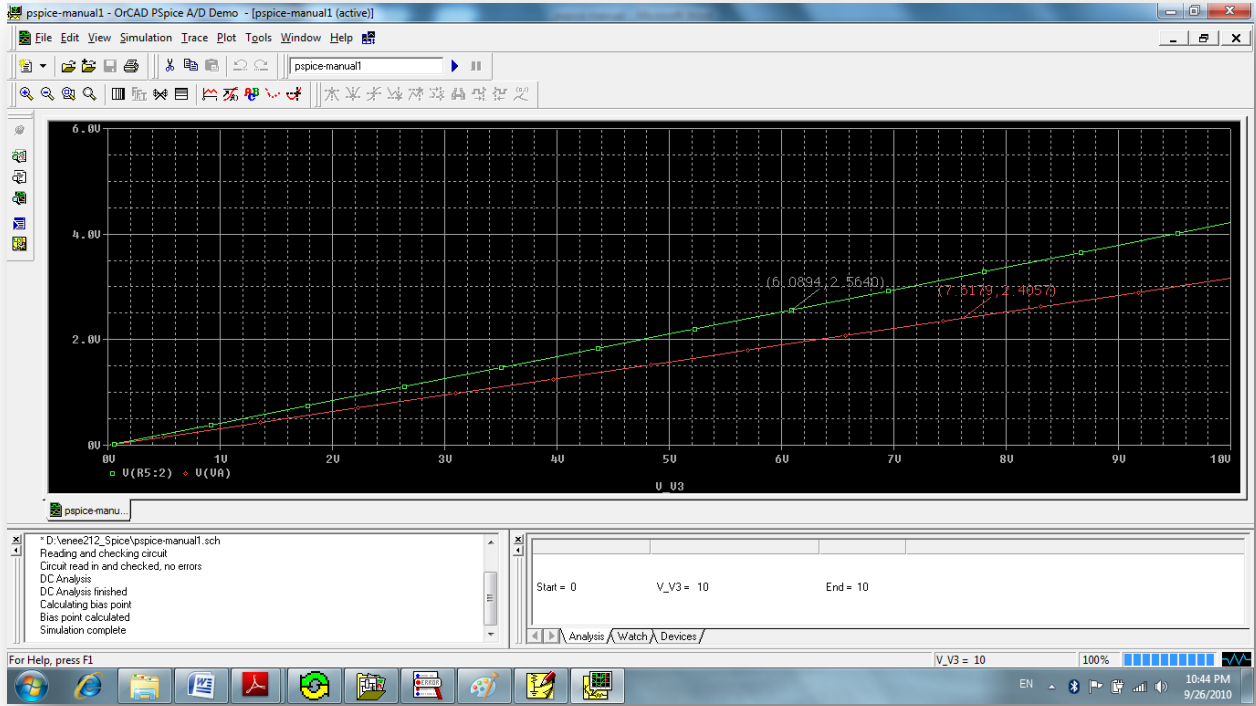


Fig.4.1.6: Plotting VA on the same plot.

3.2: Overview of AC Sweep:

✓ What is AC sweep?

AC sweep is a frequency response analysis. PSpice calculates the small-signal response of the circuit to a combination of inputs by transforming it around the bias point and treating it as a linear circuit.

✓ To set up and run an AC sweep:

- 1- Place and connect a voltage or current source with an AC input signal.
- 2- Choose AC Sweep Analysis.
- 3- Specify the required parameters for the AC sweep analysis you want to run.
- 4- Click OK to save.
- 5- Select **Simulate** to start the simulation.

AC Sweep (Frequency Response)

To study the steady state behavior of circuits often the frequency response is analyzed. One area of interest is the gain of the voltage at different frequencies; recall that voltage gain is the ratio of the output voltage with respect to the input voltage.

Let us explain the AC Sweep considering the following second order RLC circuit .

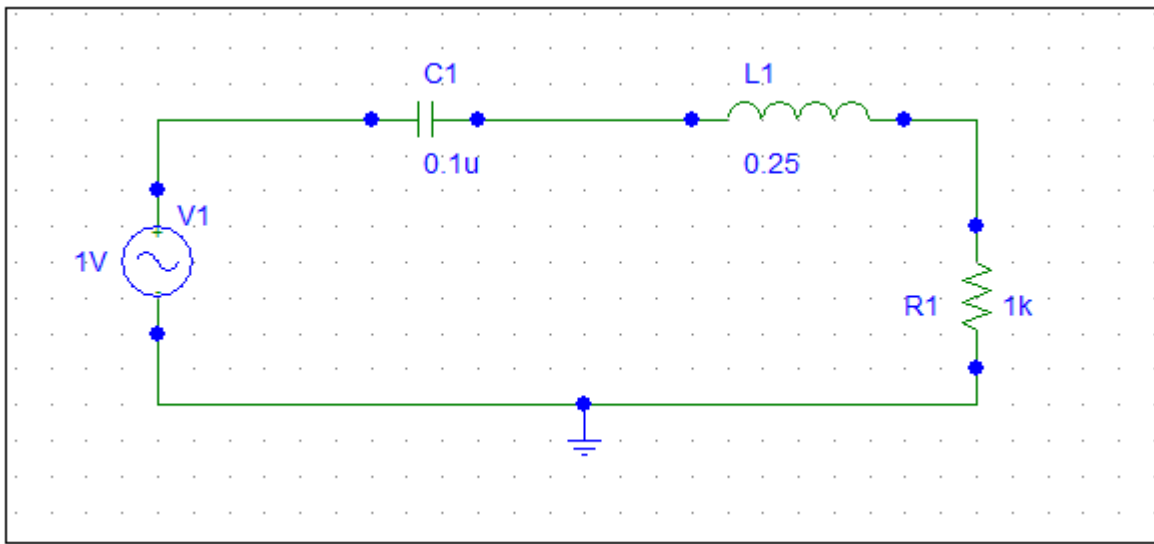


Fig.4.2.1: 2nd Order series RLC circuits (Example).

- 1- Draw the above circuit after you choose **New Schematics**, then place the R, L,C,VAC parts by clicking on **Get New Part**.
- 2- Click on **Analysis Setup>> AC Sweep**.
- 3- Set the AC Sweep parameters as shown in Fig.4.2.2.
- 4- Click on **Simulate**.

The following figures finds the voltages across R,L and C as a function of frequency (Fig.3.2.3)

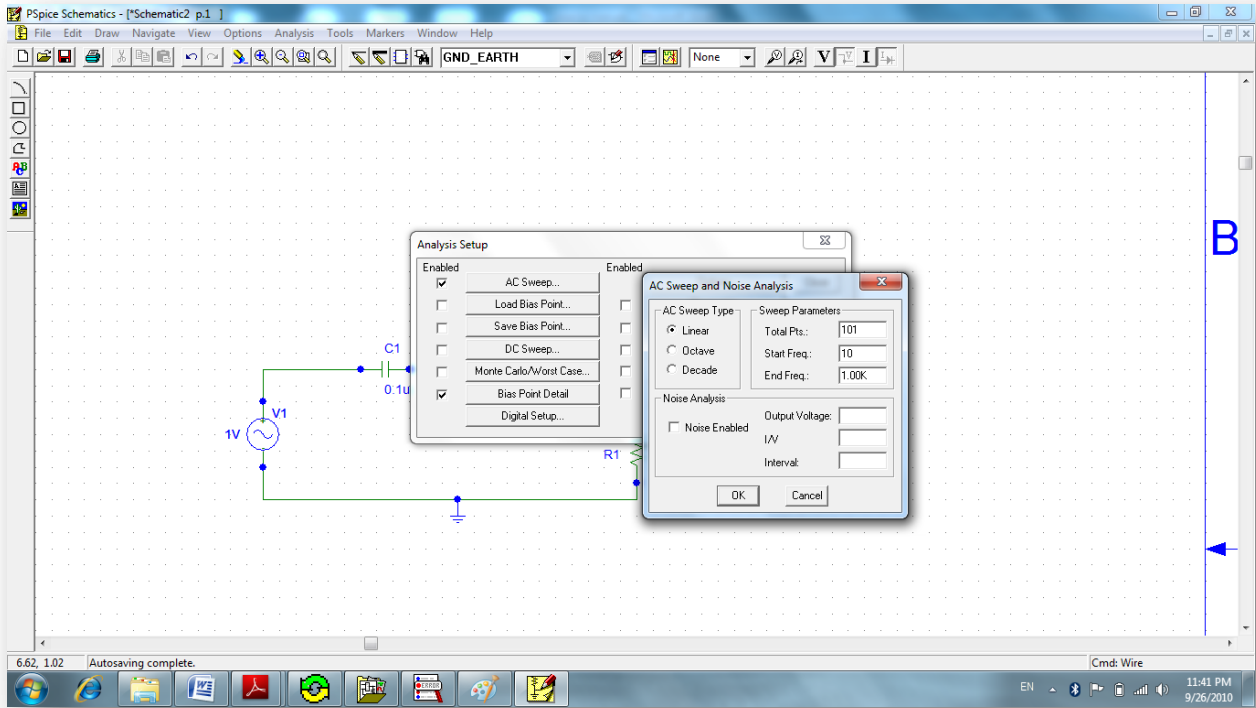


Fig.4.2.2: AC Sweep Analysis.

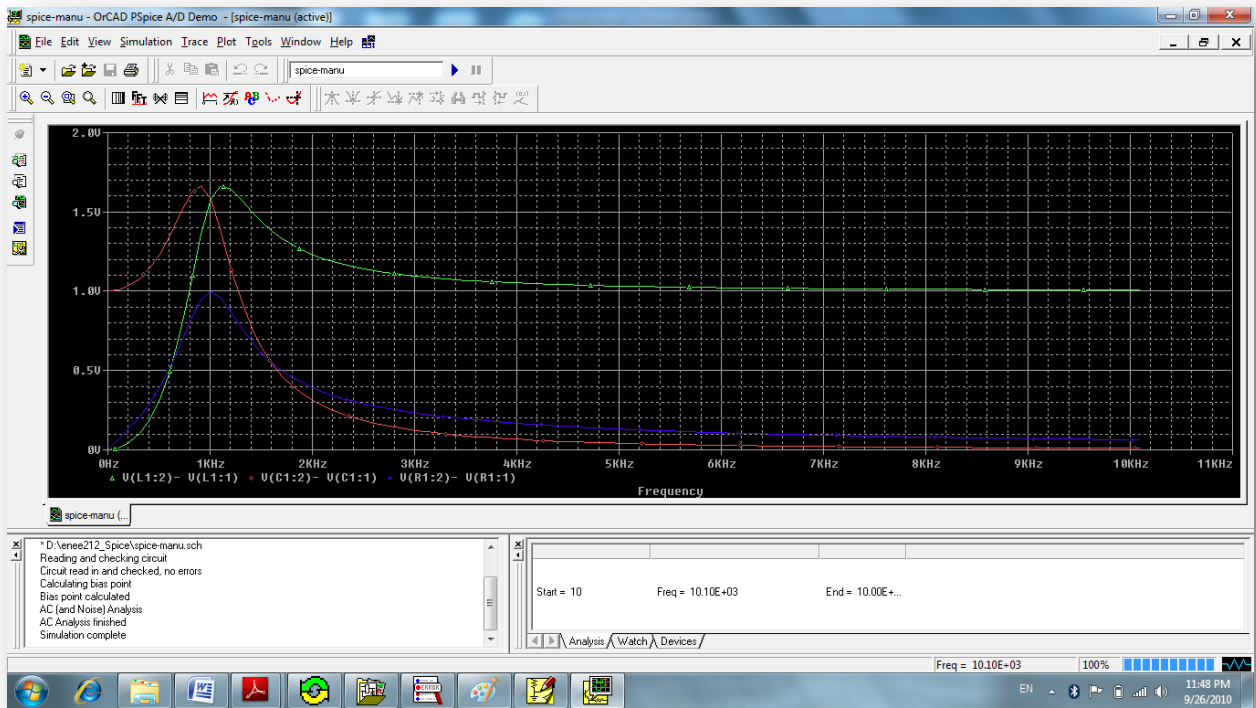


Fig.4.2.3: The Simulation Results.

3.3: Overview of Transient Analysis:

✓ What is the Transient Sweep?

Transient Sweep is a time response sweep which calculates the circuit response at a specific time interval determined by the Transient Sweep Analysis.

✓ Minimum circuit design requirements:

To do a Transient Sweep, a Circuit should contain one of the following:

- An independent source with a transient specification.
- An initial condition on a reactive element.
- A controlled source that is a function of time.

✓ To set up and run an AC sweep:

- 1- Place and connect a voltage or current source with a Transient input signal (Like VSIN, Vpulse).
- 2- Choose **Transient Sweep Analysis** from the **Setup Analysis** menu.
- 3- Specify the required parameters for the Transient sweep analysis you want to run.
- 4- Click OK to save then select **Simulate** to start the simulation.

Transient Analysis Example

Let Us apply the Transient response on the following circuit but do not forget to use a voltage source with transient specification as VPULSE.

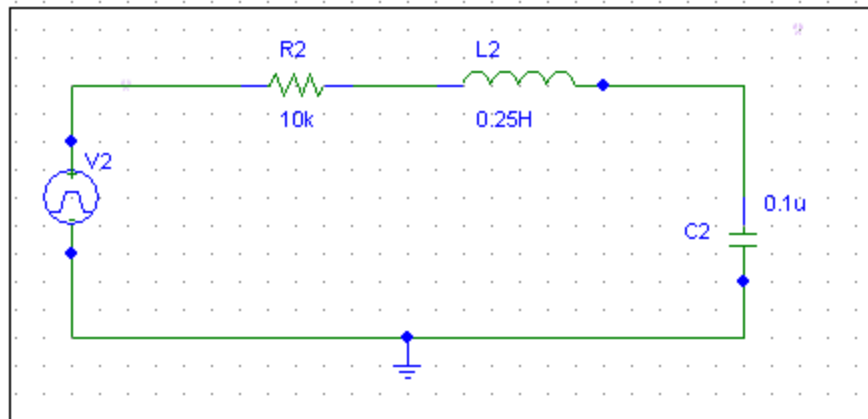


Fig.4.3.1: Plotting step response for RLC circuit (Example).

- 1- Draw the above circuit after you choose **New Schematics**, then place the R, L,C,VAC parts by clicking on **Get New Part**.
- 2- Click on **Analysis Setup>> Transient** .
- 3- Set the Transient Sweep parameters as shown in Fig.4.3.2.
- 4- Click on **Simulate**.

The following figures find the voltages across C as a function of Time (Fig.4.3.3)

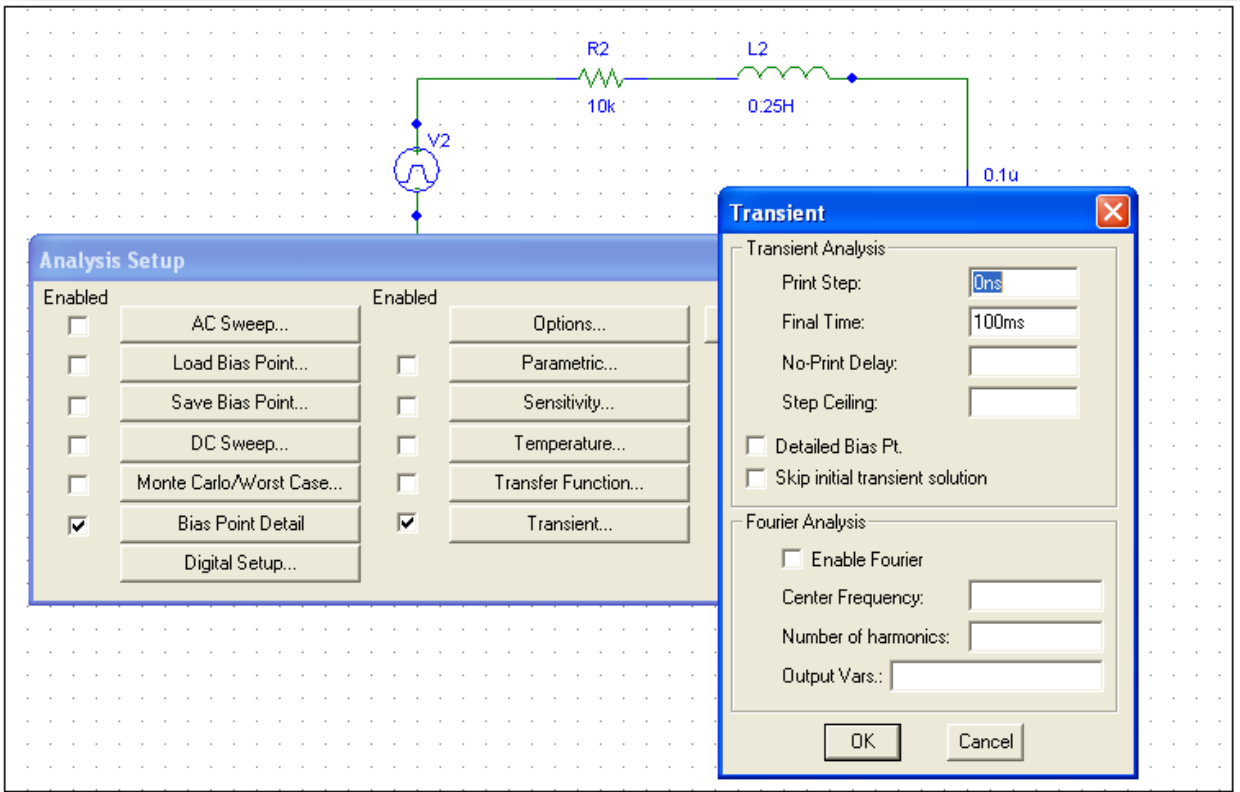


Fig.4.3.2: Setting Transient Analysis Parameters.

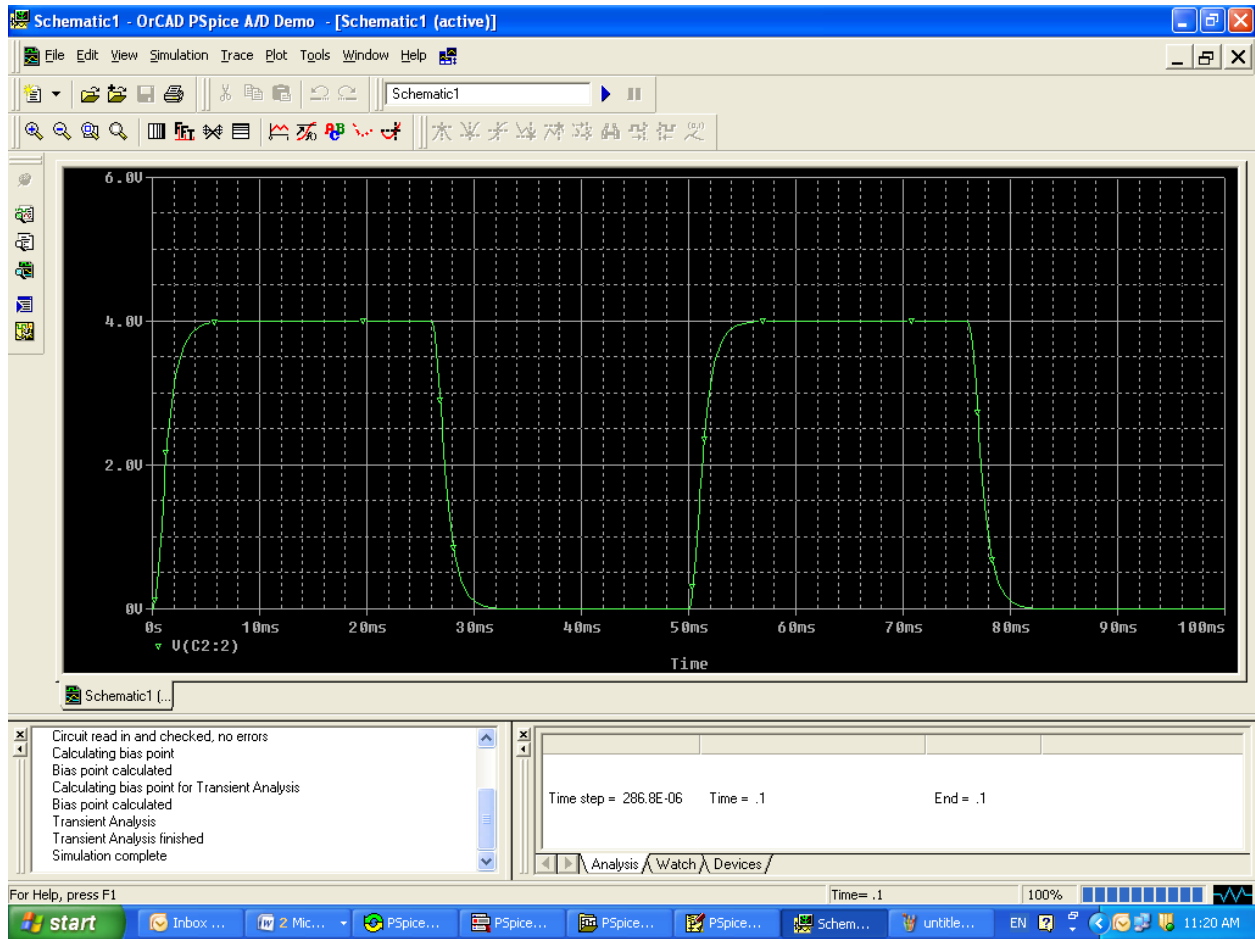


Fig.4.3.3: Transient Simulation Results.

More Examples and Exercises

Example 5.1: Construct the following RLC circuit in PSpice and display the voltage waveform (step response) across the capacitor.

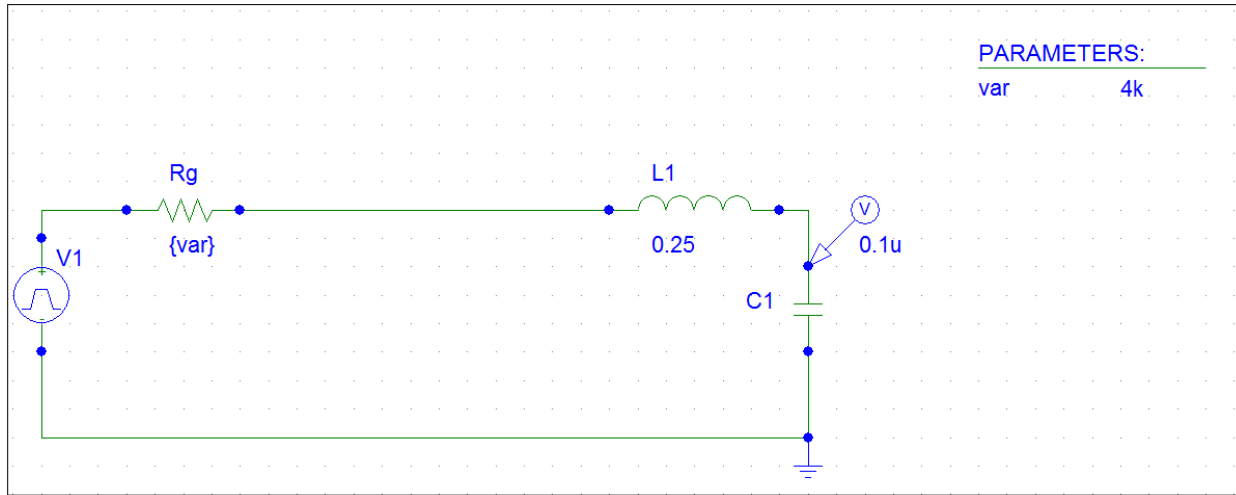



Fig 5.1: Series RLC circuit (example 5.1).

Answer: As you know the step response of this circuit varies as L or C or R varies. The step response for this series RLC circuit has three cases:

- 1- Over damping
- 2- Critical damping
- 3- Under damping

Let us discover the effect of changing R on the circuit response. To do this:

- 1- Construct the circuit which is shown in fig 5.1 using PSpice. (Initially neglect the parameters list on the right). Let $L=0.25$ H, $C=0.1\mu\text{F}$ and V_{pulse} ($V1=2$, $V2=0$, $T_f=T_r=T_d=0$, $\text{PER}=0.02$, $\text{PW}=0.01$).
- 2- Click on  to get part, then the part browser window will open, type **PARAM** (i.e.: parameters) which is used to specify predefined parameter, choose place and close.

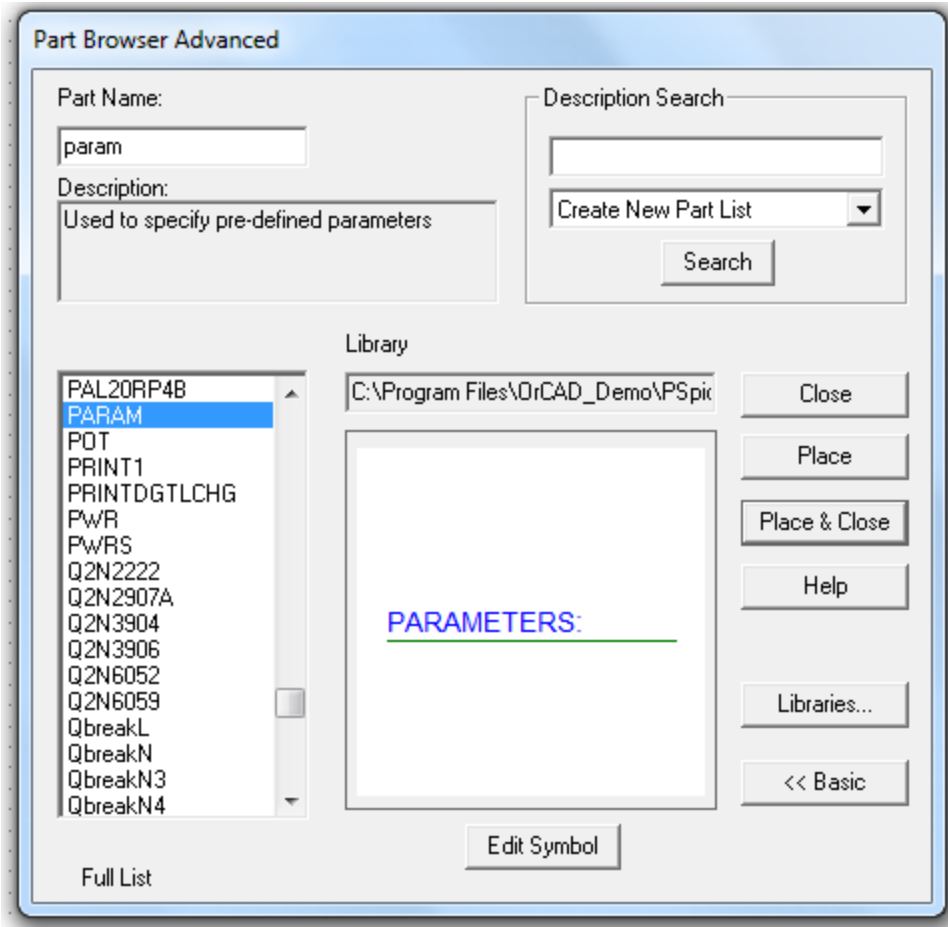




Fig 5.2: Part browser window.

- 3- Double click on parameters the following window will appear, type **var** under NAME1 and 4k for VALUE1 field (Fig 5.3), and then press OK.
- 4- Double click on the **value** of Rg and change it to **{var}** as in figure 5.4.
- 5- In order to show the simulation result, go to  (**Analysis Setup**), Check Transient, Parametric. Then fill the parametric window as shown in fig 5.5.
- 6- Finally, enter the transient setting which are shown in fig 5.6.
- 7- Go to  (Simulate) in order to run the circuit simulation, the result will be 3 traces on the same plots since the value list for R is 1k, 5k, and 10k. The result is shown in fig 5.7.

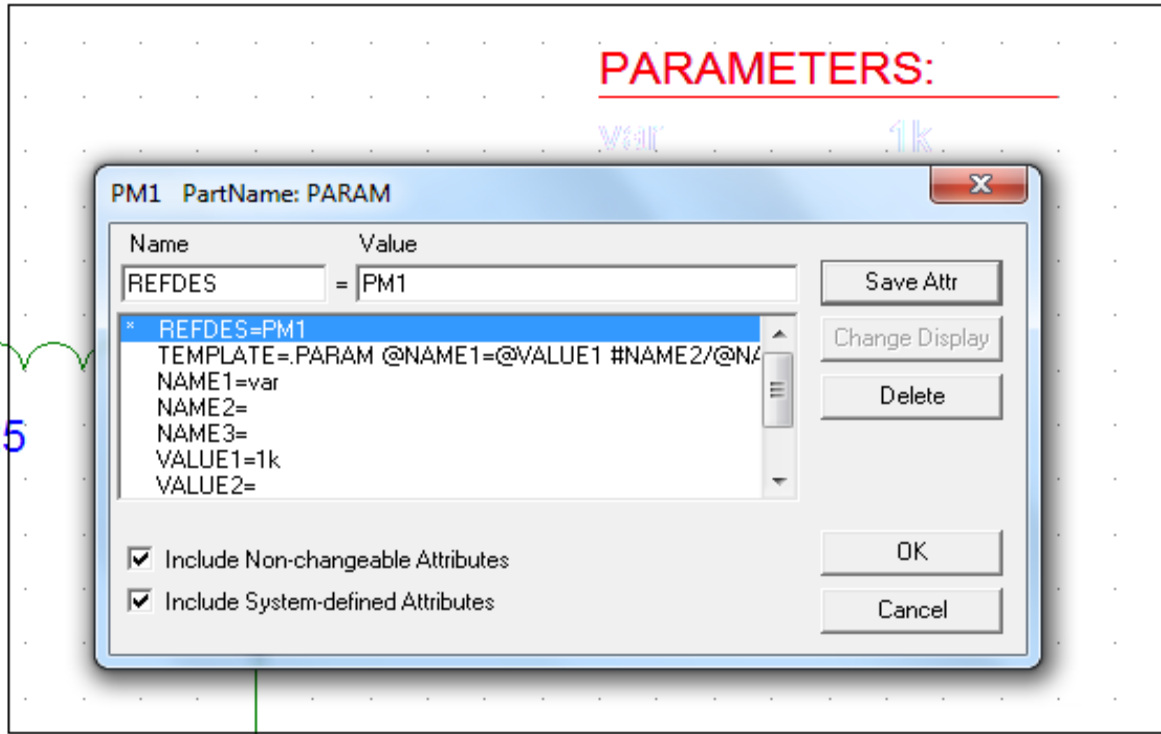


Fig 5.3: Parameters window.

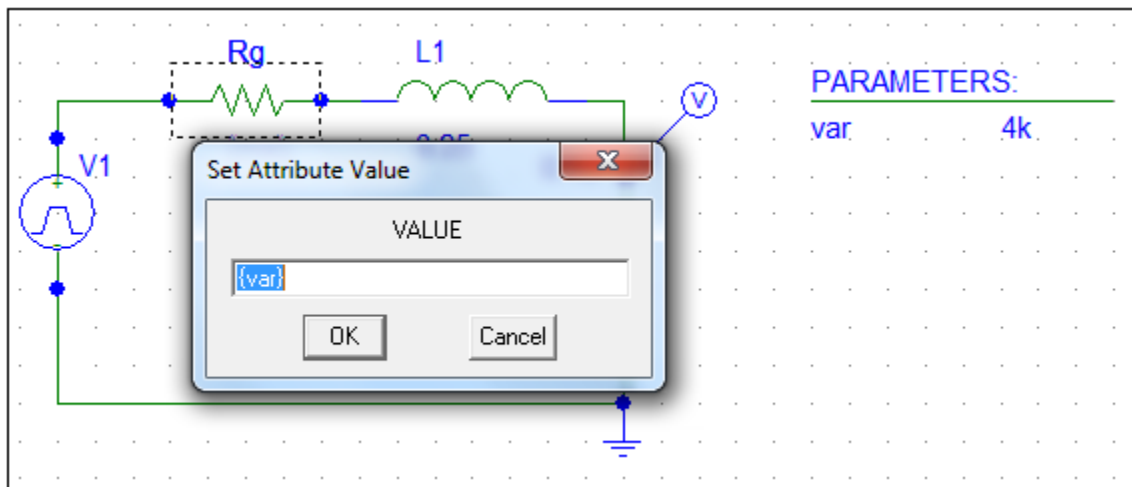


Fig 5.4: Set the attribute value for Rg.

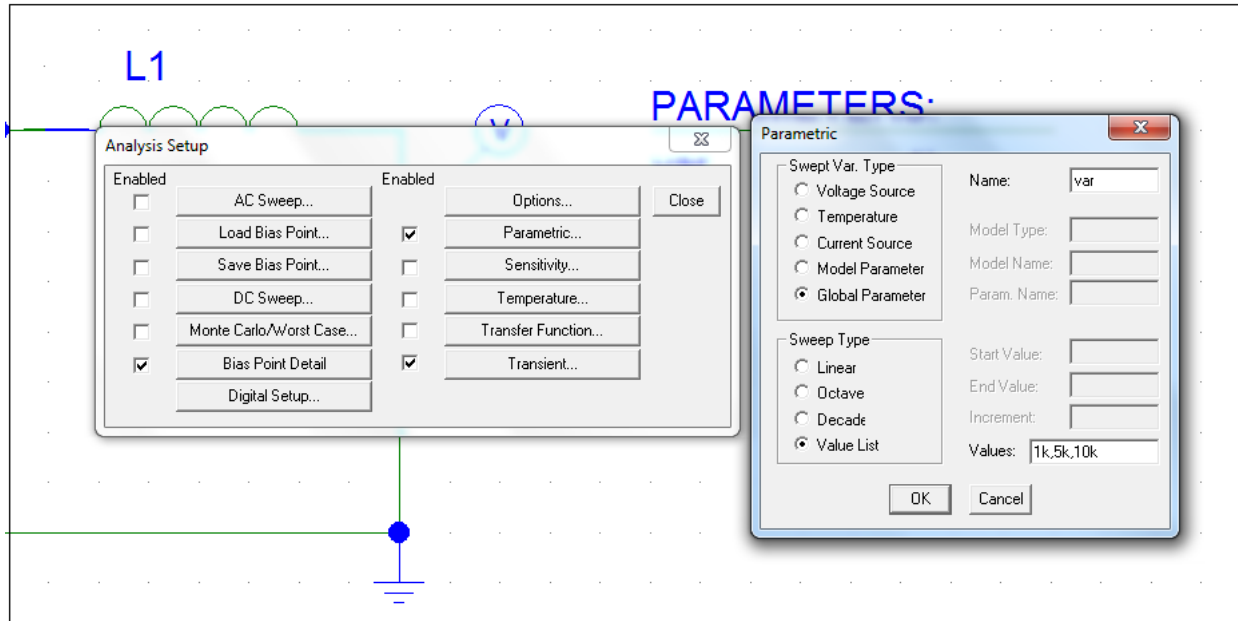


Fig 5.5: Analysis Setup.

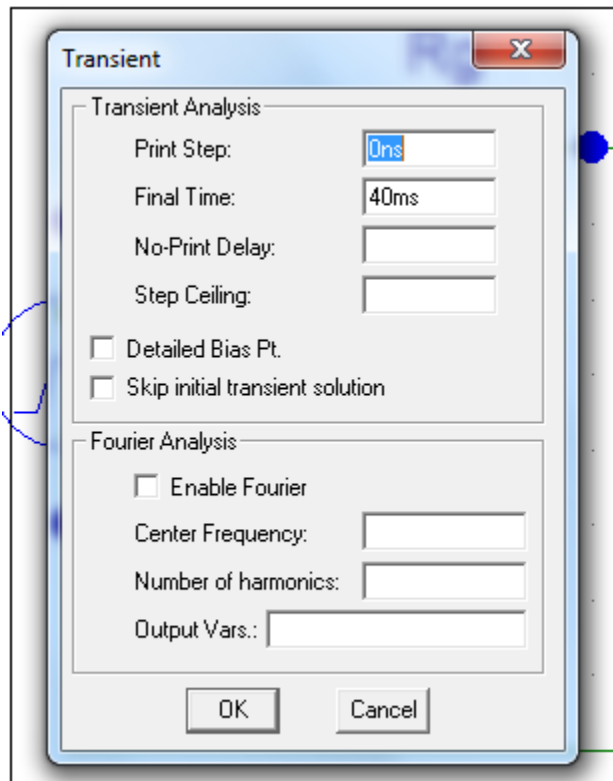


Fig 5.6: Transient Analysis window.

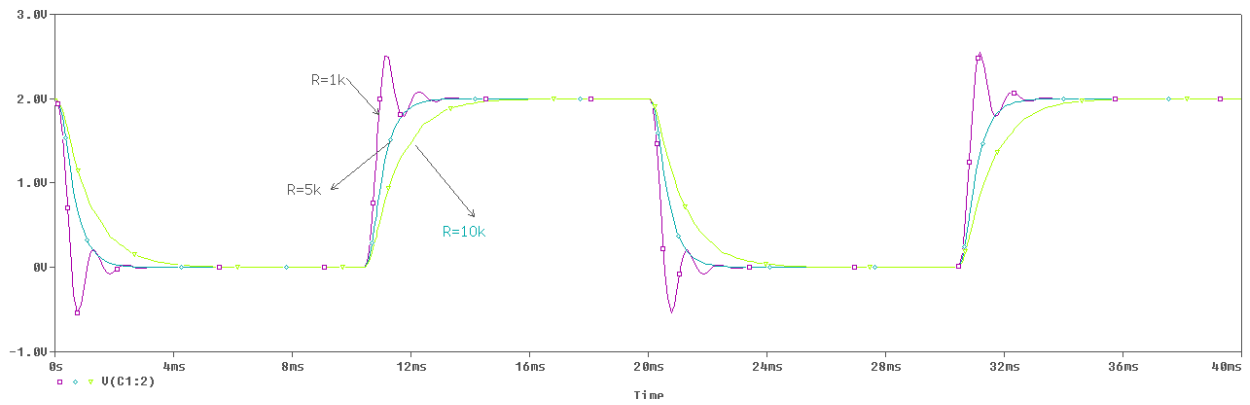


Fig 5.7: Simulation Results.

Note: from the above plot we can find that the step response for this system is **under damping** when $R=1k$ ohm, **over damping** when $R=5k$ and $10k$ ohm.

Exercise 1: Construct the following circuit using PSpice. Then run the **Bias Point Analysis** to find the current in each branch.

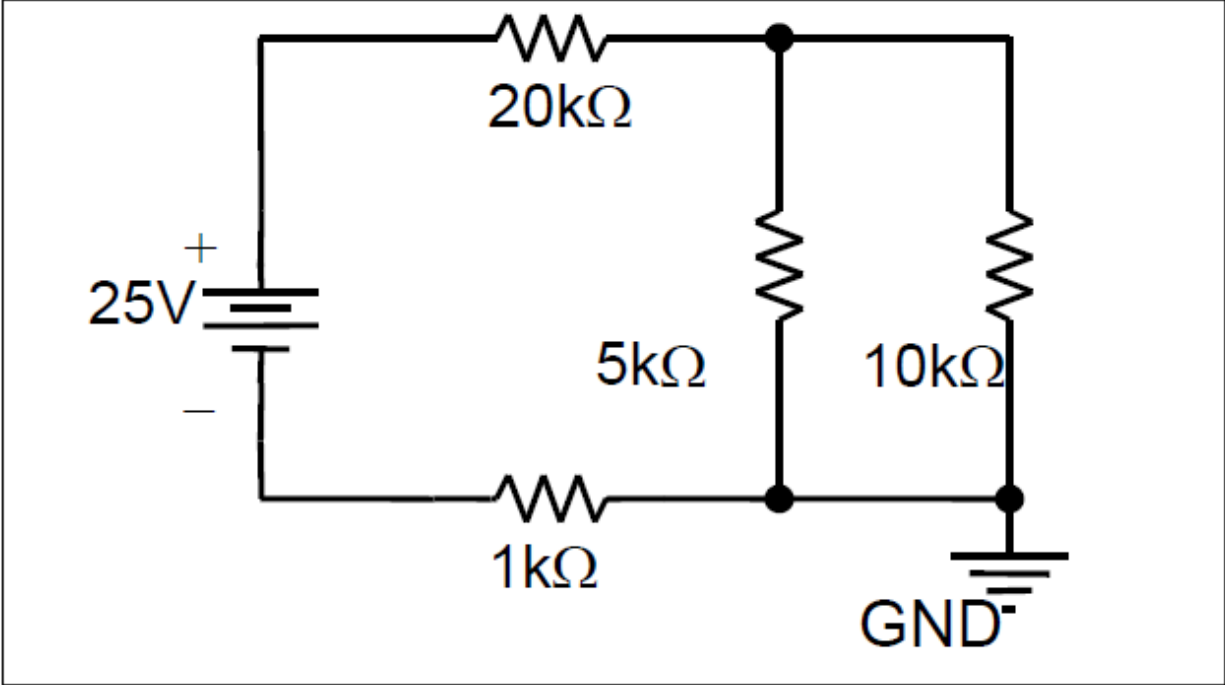


Fig 5.8: Exercise 1.

Exercise 2: Construct the following circuit using PSpice. Then run the **Bias Point Analysis** to find the voltage across 300Ω.

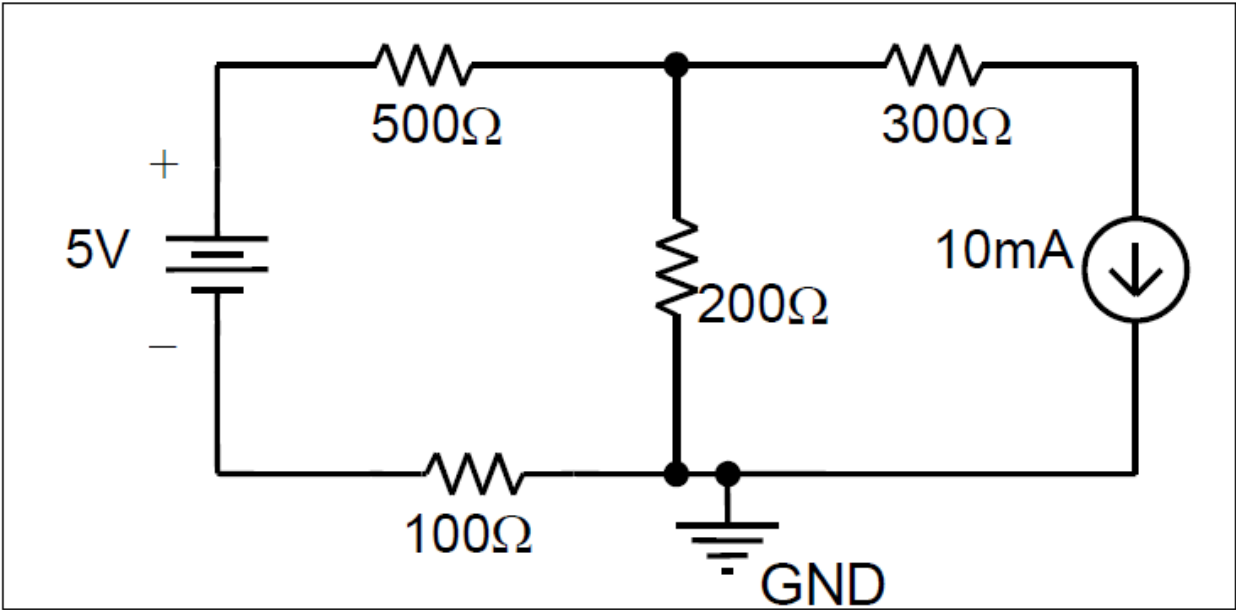


Fig 5.9: Exercise 2.

Example 5.2: Thevenin Equivalent

In PSpice you cannot calculate the Thevenin equivalent with respect to a pair of nodes directly. In order to replace a circuit with its Thevenin equivalent at a pair of nodes, one needs the open circuit voltage, V_{oc} , and the short circuit current, I_{sc} .

Construct the following circuit in PSpice.

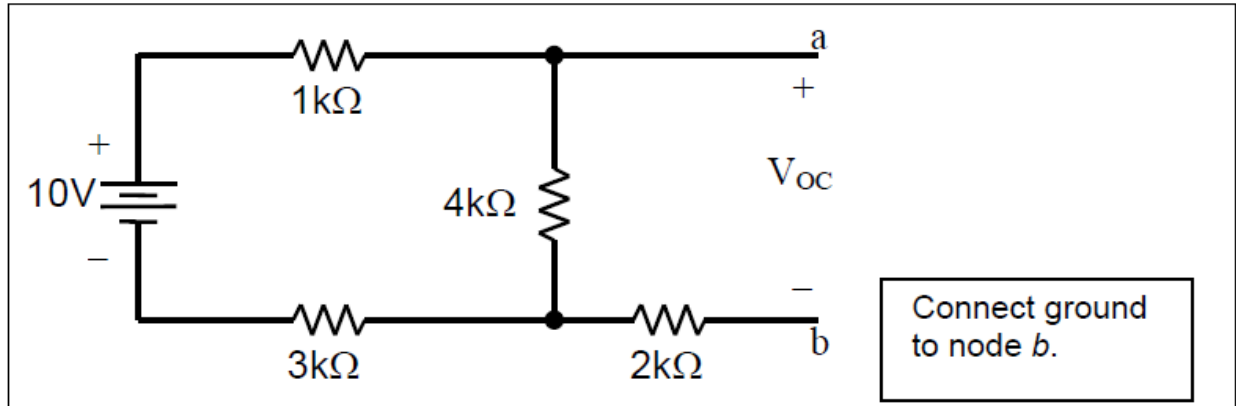


Fig 5.10: Example 5.2.

In this example to find the short circuit current and the open circuit voltage;

- Place circuit ground at node b .
- Create a short circuit between nodes a, b and run **Bias Point** analysis to find the voltage across the 2kΩ resistor. Then use Ohm's law to find the short circuit current. The short circuit current should be 1.25mA.
- Keep the circuit ground at node b .
- Create an open circuit between nodes a, b and run **Bias Point** analysis to find the open circuit voltage. This is the voltage from node a to ground (node b), and it should be 5V.

A second approach to find the Thevenin equivalent with respect to nodes (terminals) a, b is to *simulate* a short circuit and an open circuit in PSpice.

- You may need to use this approach because sometimes when you create an open circuit between nodes a and b , PSpice will not run the simulation because of the presence of open-ended circuit elements and nodes (see exercise 3).
- In other cases if you directly place a short circuit between nodes a, b and run the analysis, the process of finding the short circuit current using the results of the simulation can be complicated (see exercise4).

- For the circuit in figure 5.10, find the short circuit current by placing a resistor between nodes a and b whose resistance is much smaller than the resistance of the other resistors in the circuit.
 - In this case a $10^{-6}\Omega$ resistor between nodes a and b will be sufficient in simulating a short circuit.
 - Choose circuit ground to be at node b , this way you can obtain the voltage V_{ab} directly and then use Ohm's law to find the short circuit current.
 - Using **Bias Point** analysis you should obtain a short circuit current of 1.25mA.
- Note:** In all DC circuit analyses an inductor can be used as a short circuit.
- The voltage drop across the inductor is zero so you must use the voltage drops across the other nodes to solve for the short circuit current.
 - Replace the $10^{-6}\Omega$ resistor with an inductor of any value and confirm that the result remains 1.25mA.

To find the open circuit voltage, place a resistor between nodes a and b whose resistance is much larger than the resistance of the other resistors in the circuit.

- For the circuit in figure 5.10, a $10^{12}\Omega$ resistor between nodes a and b will be sufficient in simulating an open circuit.
- Again choose circuit ground at node b .
- You should obtain an open circuit voltage of 5V.

Note: In all DC circuit analyses a capacitor can be used as an open circuit. The voltage drop across the capacitor is the open circuit voltage, since a capacitor behaves as an open circuit in DC.

- Replace the $10^{12}\Omega$ resistor with a capacitor of any value, and confirm that the result remains 5V.
- When finding the Thevenin equivalent with respect to a pair of nodes in PSpice, you should always place circuit ground at one of those nodes. This will simplify your analysis.

With an open circuit voltage of 5V and a short circuit current of 1.25mA the Thevenin equivalent with respect to nodes a , b is a 5V DC voltage source in series with a $4k\Omega$ resistor. This is shown in figure 5.11.

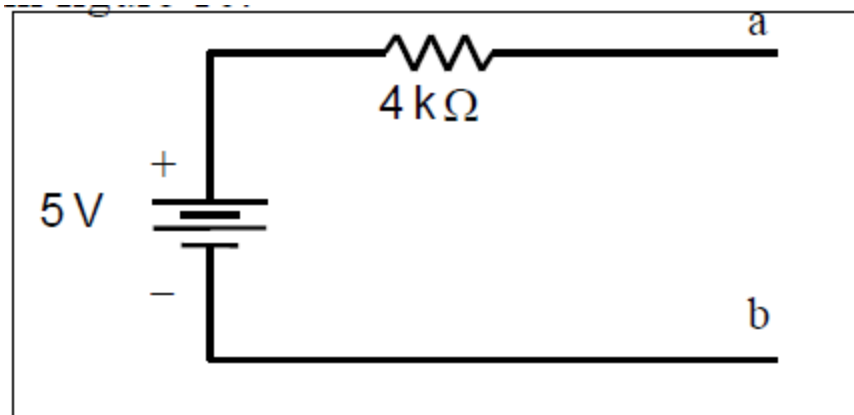


Fig 5.11: Thevenin Equivalent circuit of Example 5.2.

Exercise 3: Find the Thevenin equivalent with respect to nodes a, b of the following circuit:

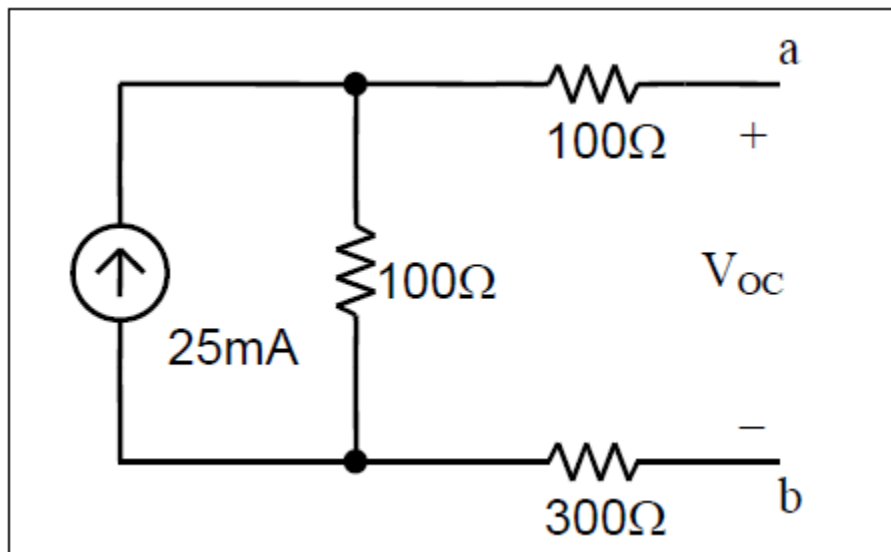


Fig 5.12: Exercise 3.

In this exercise you cannot simply create an open circuit between nodes a, b in PSpice and run the simulation. PSpice will give you a “**Less than two connections at node N_____**” error if you try to run the simulation with an open circuit. You need to place a resistor with a very large resistance between nodes a, b in order to simulate an open circuit. The correct open circuit voltage is 2.5V and the correct short circuit current is 5mA.

Exercise 4 : Find the Thevenin equivalent with respect to nodes *a*, *b* of the following circuit:

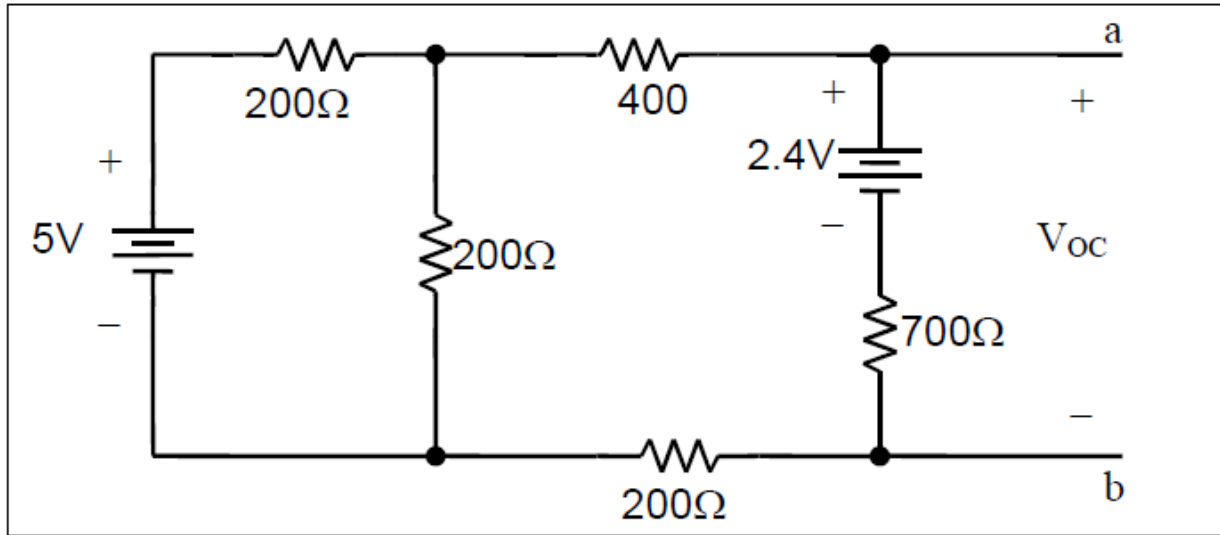


Fig 5.13: Exercise 4.

In this exercise if you place a very small resistance between nodes *a* and *b*, you will get the voltage at node *a* directly. Using Ohm's law you can then find the short circuit current. If instead you had placed a short circuit between nodes *a* and *b*, you would have to do a calculation to find the short circuit current – in this case a simple one, but still introducing an opportunity for error. You should also bear in mind that PSpice only gives you results to four significant digits, so you may lose precision if you have to go through more calculations to get to the answer.

For this circuit you should find the open circuit voltage is 2.45V, and the short circuit current is 7mA.

When finding the Thevenin equivalent you can use any of the methods described. However you should realize simulating open circuits and short circuits using very large and very small resistances will work every time, though it takes slightly longer to design a circuit.

Transient Analysis:

Exercise 5 (Step Response): Construct the following RC circuit in PSpice and display the voltage waveforms across all the circuit components.

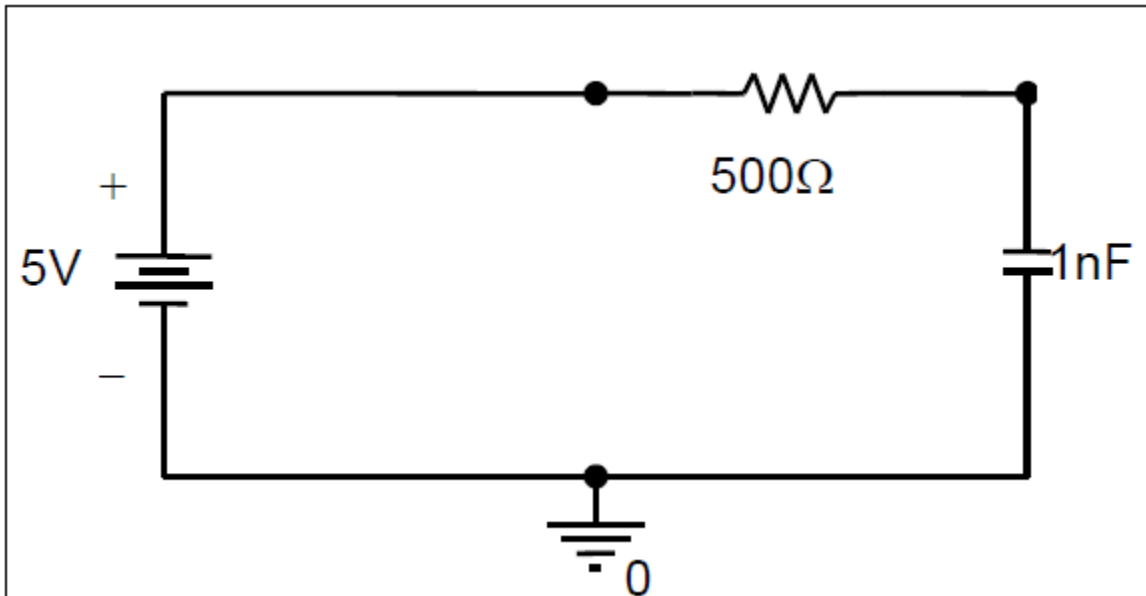


Fig 5.14: Exercise 5.

Once you constructed the circuit create a new simulation .

- In the **Simulation Settings dialog window**, under the **Analysis tab** select **Time Domain (Transient)** as the **Analysis Type**.
- In the **General Settings option**, specify start time of the simulation. This should usually be set to 0.
- Specify the end time of the simulation in the **Run to time dialog box**. This should usually be set to about 5τ to 6τ , where t is the time constant of your circuit. When analyzing RLC circuits, where the time constant is not available, you will have to play around with end time to get a good plot of the response.
- Check the **Skip the initial transient bias point calculation** box.
- Once you have made all the adjustments click **OK** and run the simulation.

The voltage across the resistor should show an exponential voltage decay starting from 5V and the voltage across the capacitor should show an exponential rise from 0V.

Exercise 6: Construct the following RLC circuit in PSpice and display the voltage waveforms across all the circuit components.

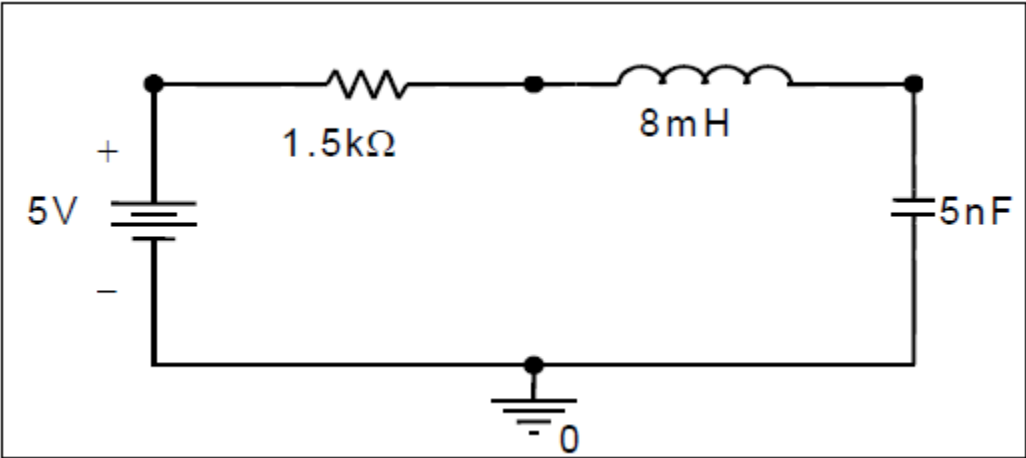


Fig 5.15: Exercise 6.

Frequency Response

Exercise 7: Construct the follow series RL circuit in PSpice and generate a Bode Plot of $H(j\omega)$ dB versus f . V_o is taken across the inductor.

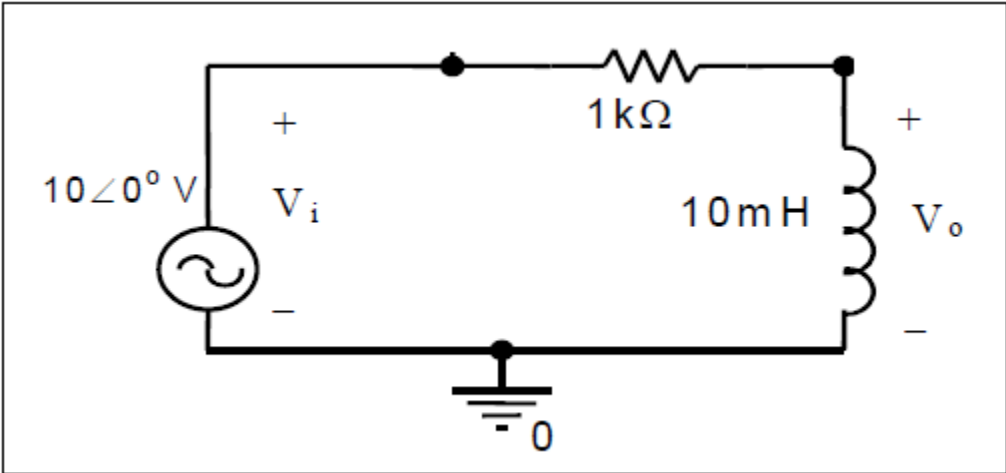


Fig 5.16: Exercise 7.

The plot of the gain should rise to 0dB. Check that the 3dB frequency is approximately 16kHz. You can do this easily with the cursor. Then Plot $\theta(\omega)$ versus f .

Exercise 8 : Construct the following RLC circuit in PSpice and then generate the gain plot and the phase angle plot. In this circuit V_o is taken across the resistor.

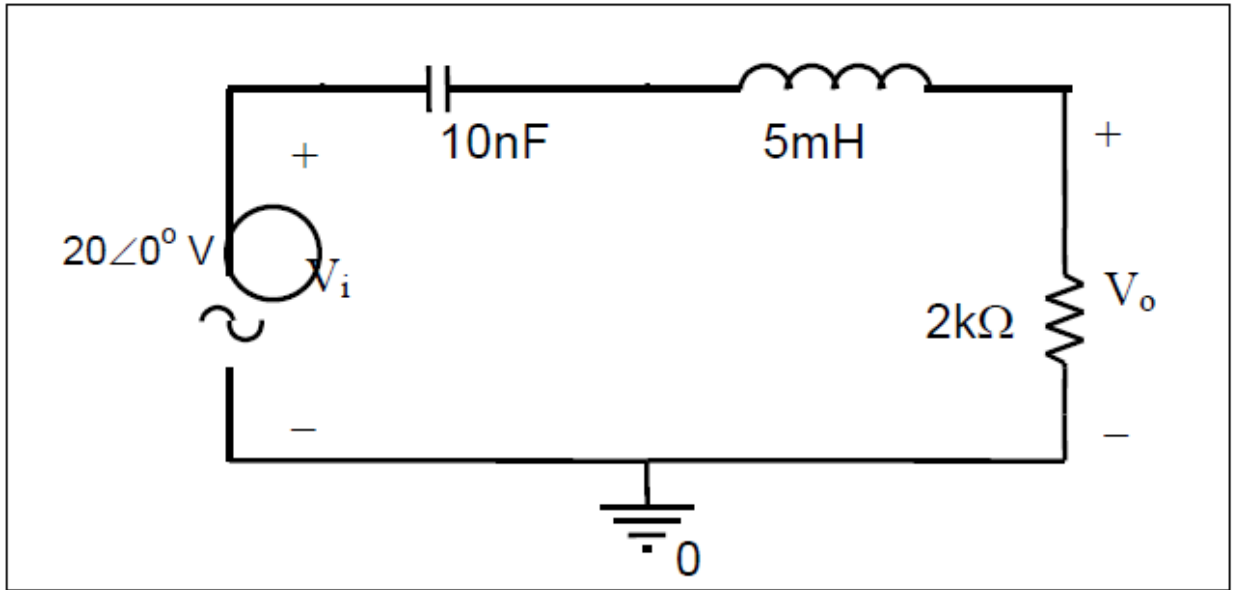


Fig 5.17: Exercise 8.

Sinusoidal Steady State Analysis : PSpice can also display the waveform of a sinusoidal signal at a fixed frequency. To perform this analysis, we need a new voltage source with a fixed frequency. Construct the following circuit in PSpice.

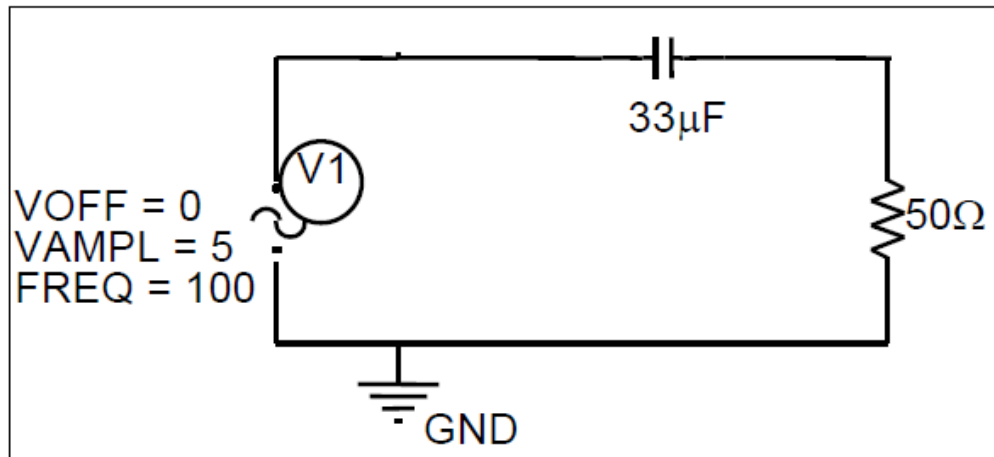


Fig 5.18: Steady State Analysis (Example 5.3).

- The voltage source used in this circuit is called **VSIN** and can be found in the **SOURCE** library. You should see three labels next to the symbol; **VOFF =**, **VAMPL =**, and **FREQ =**. These labels indicate the DC offset voltage, the amplitude of the voltage signal and the frequency (in Hertz) of the signal respectively. Double click on each of these labels to change the values.
- In the *Simulation Settings* dialog window, go to the *Analysis* tab and chose **Time Domain (Transient)** as the *Analysis type*.
- In the **General Settings** options, set the **Run to Time** text box to 0.02 seconds. Note that at a frequency of 100 Hz, the period is 0.01 seconds. We should get two full waveforms on the display. In the **Transient Options** box, set the **Maximum step size** to 1/100 of the period of the signal. In this case, this is 0.1ms.
- Click on **OK** and run the simulation.
- Plot the voltage across the capacitor then plot the voltage across the resistor.

REFERENCES:

<http://www.orcad.com>