



# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

---

### EXPERIMENT -1

#### Introduction to Pspice for Circuit Analysis

---

**AIM:** - To get acquainted to pspice from this experiment.

**Analysis:**

1. In modern electric circuit analysis many circuits are analyzed & verified on software before being done on hardware, Pspice is one of the common available software which is used in electrical & electronics engineering applications.
2. Pspice is one of the category belonging to spice, SPICE stands for simulation program with integrated circuit emphasis
3. The pspice software will have some common menu bars as common Microsoft word apart from this it will have different menu bars and tool bars required for circuit analysis. we will try to understand about pspice software by some visual and text aid.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

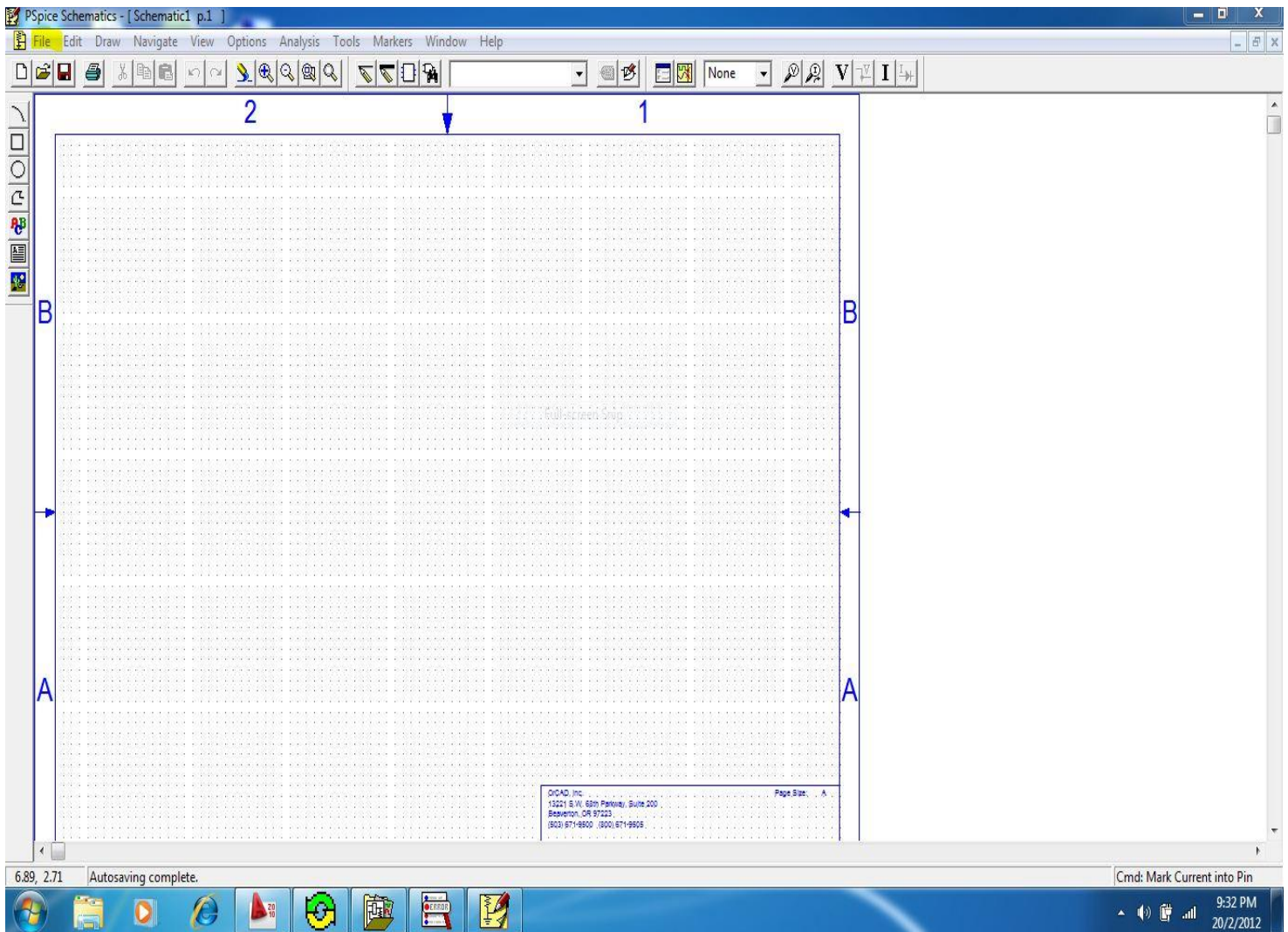


Figure-1

4. From the above Figure-1 we can see the menu bar, In menu bar you can see **file** is highlighted by yellow color, **File** and **Edit** bars in pspice have same as other general software options in it, a picture of the software working screen can be seen above.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

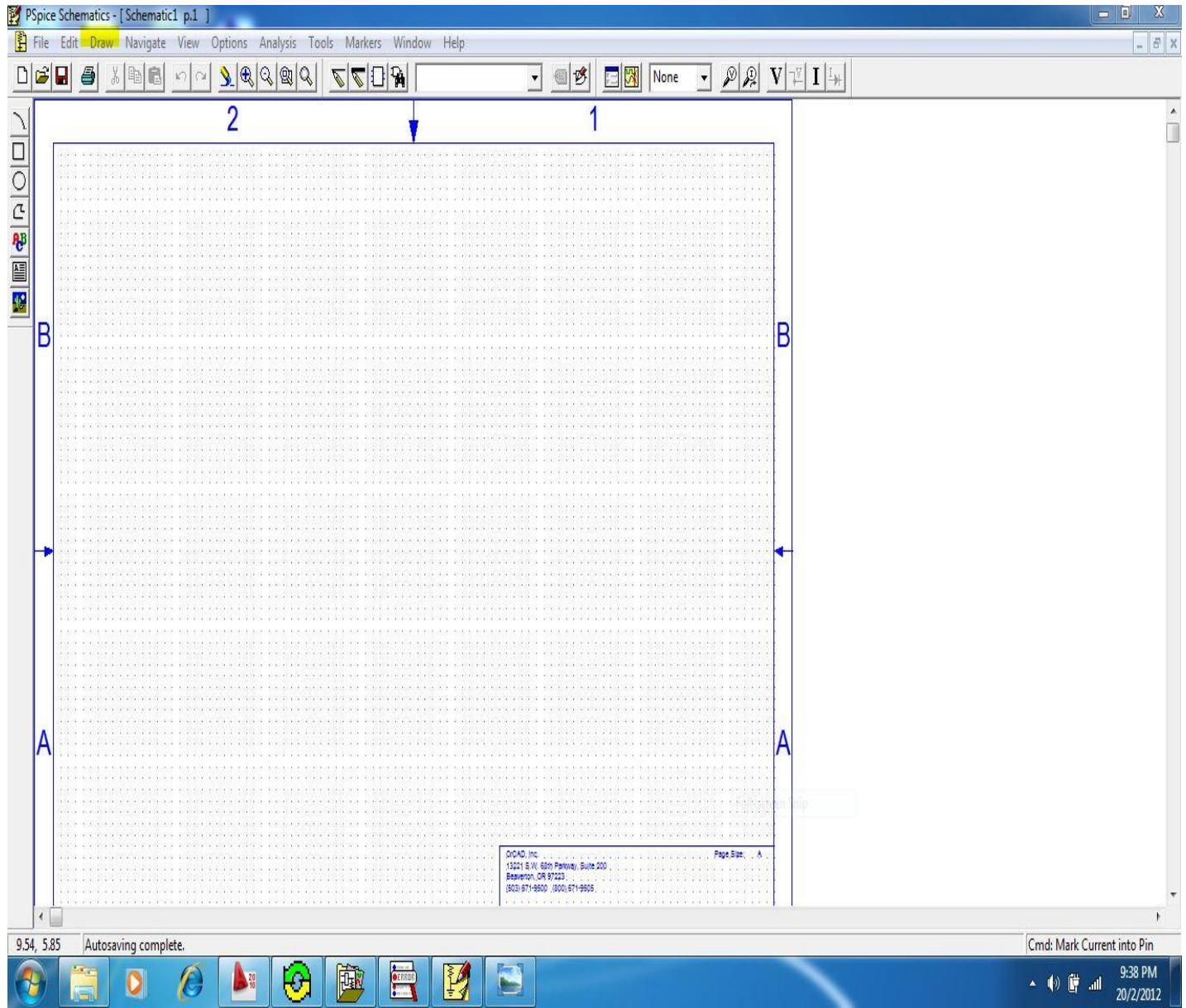


Figure-2

5. The Figure-2 shows **Draw** as highlighted by yellow color the **Draw** contains circuit drawing tools on the screen, we can select **wire** to draw a wire, **place part** to place a resistor or some other part on the screen and the **Text box** is used to write a text on screen and many other tools are available, which can be used as when required.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

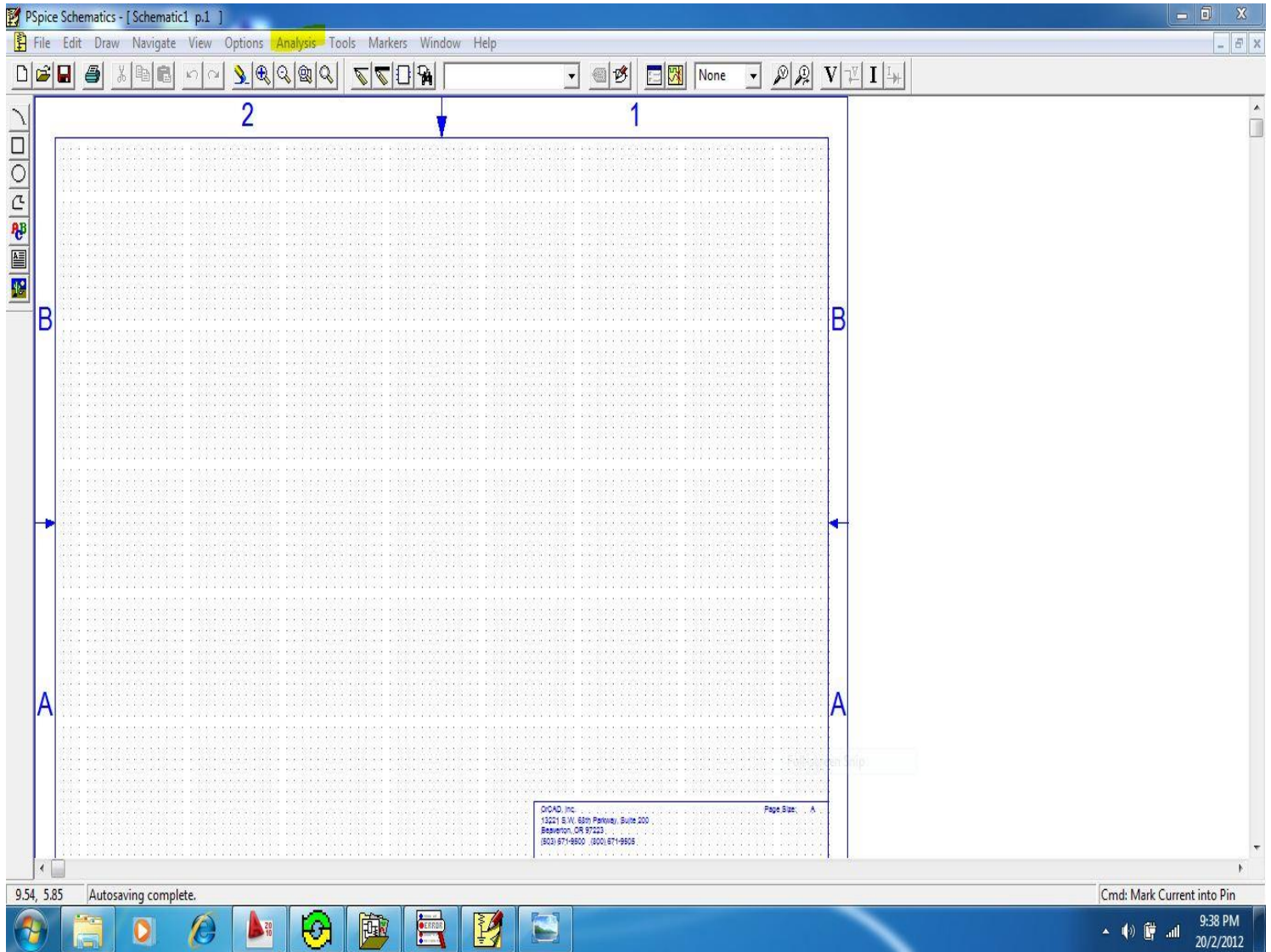


Figure-3

6. The Figure-3 highlighted the **Analysis** button by yellow color, in **Analysis** button you have
  - a. **Electrical Rule Check:** - To check the connections are right or wrong and we can connect any missing connections or correct connections.
  - b. **Simulate:** - The program is simulated you get an output text file as well as you get graphical display of voltages and currents.
  - c. **Run probe:** - This is a probe, this window acts like a oscilloscope, you can view different voltages across elements and currents flowing through elements.
  - d. **Examine Output:** - this will give you voltages and currents in text format in a notational way.
  - e. **Setup** :- this is used to setup analysis, the next step gives brief detail of this

# AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

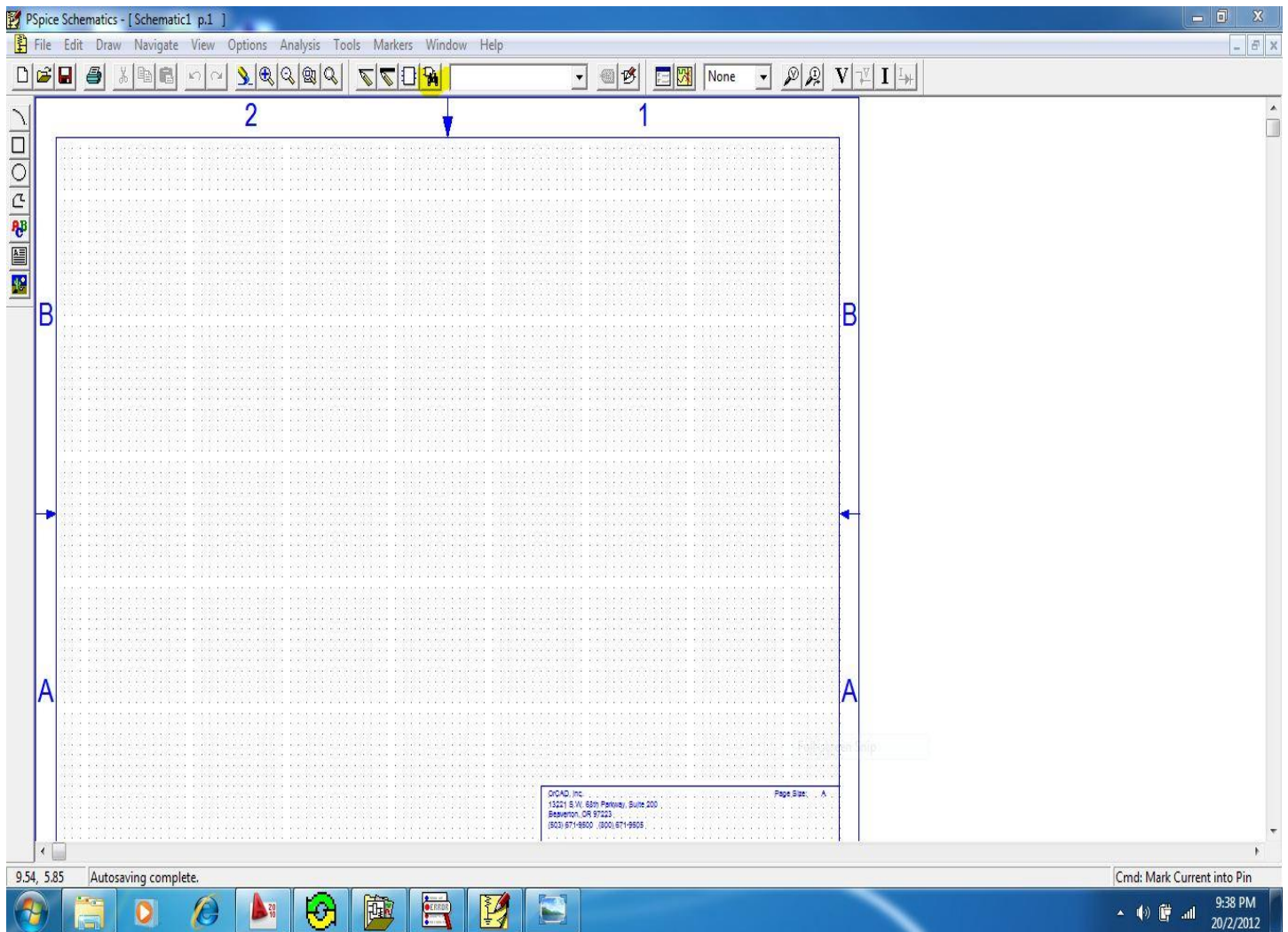


Figure-4

7. The Figure-4 shows **search a part** tool as highlighted by yellow color, by clicking that we can go to the parts list and we can look for a part and select a part and click on **Place** button, the part will be placed on the screen, later on we can move that part to any location, we can place one part after other part, finally when we finish we click on button **close**.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY ELECTRIC CIRCUITS LAB – EE 226

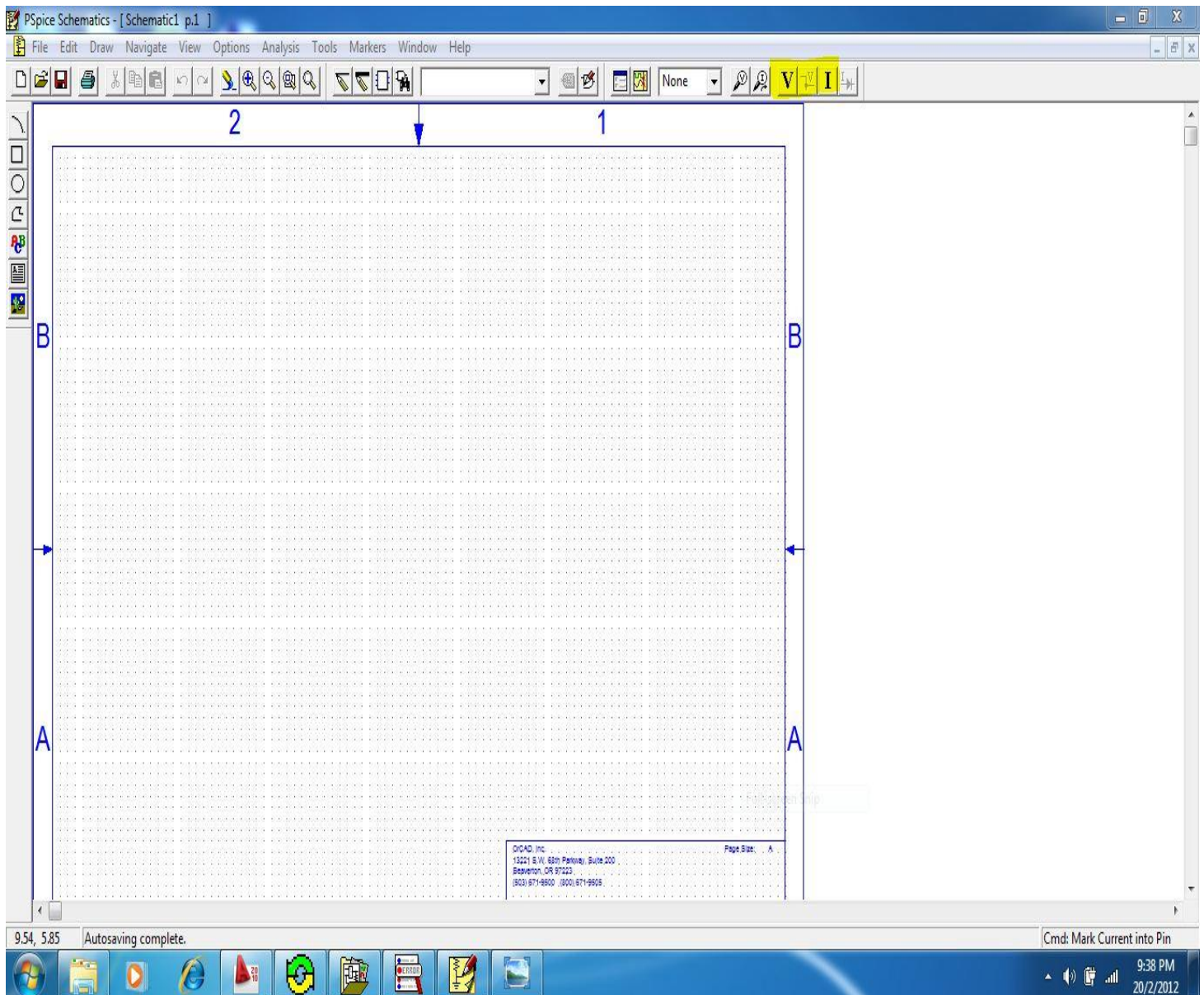


Figure-5

8. The Figure-5 highlighted **V** and **I** buttons by yellow color by clicking that we can make voltages values at nodes to appear and current values in the branches to appear directly.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

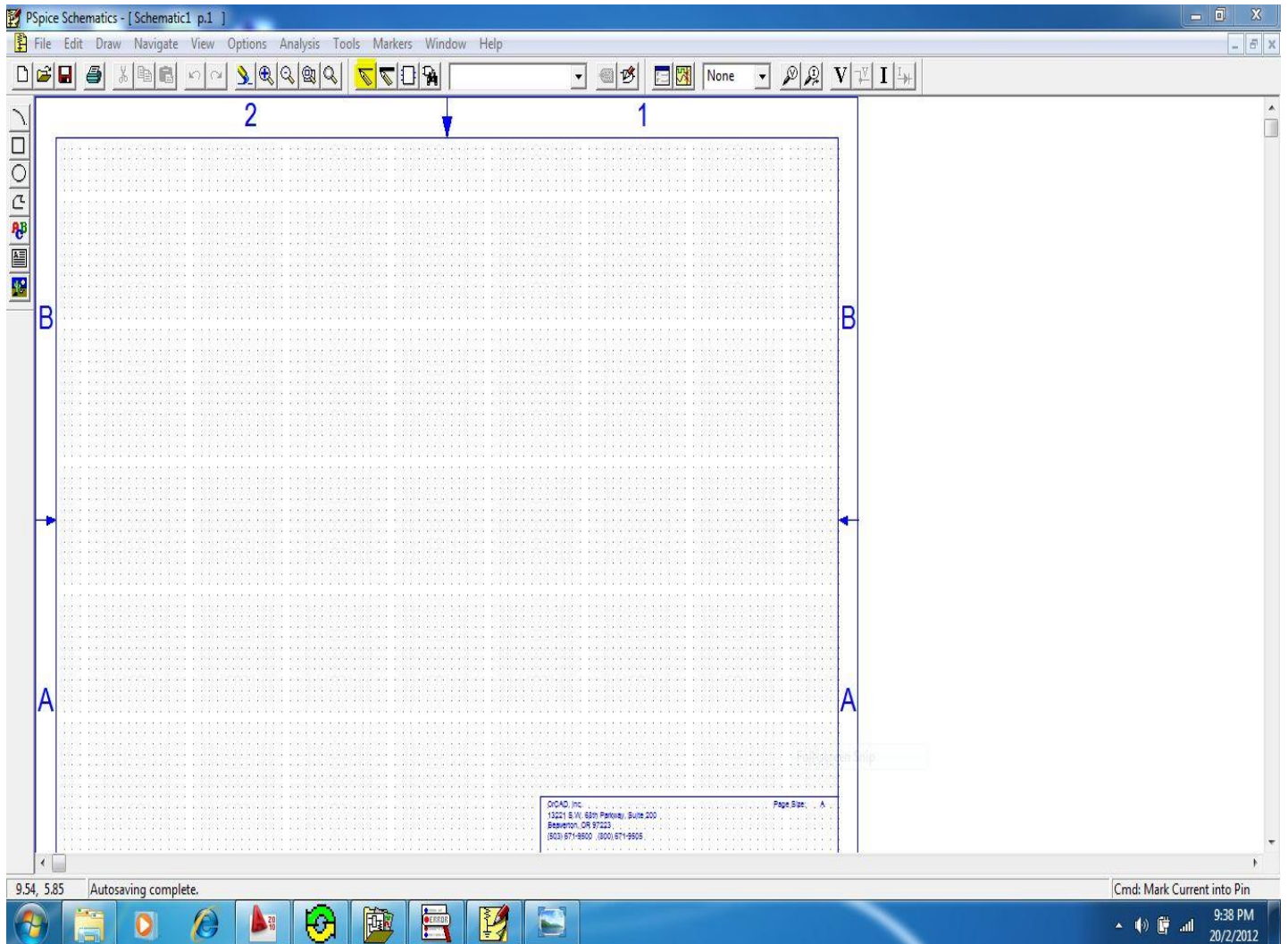


Figure-6

9. The Figure-6 highlighted **Draw wire** button by yellow color, a pencil symbol appears, by this we can draw a wire to connect between circuit elements, the wire that is drawn is either parallel or perpendicular to the x axis.

# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

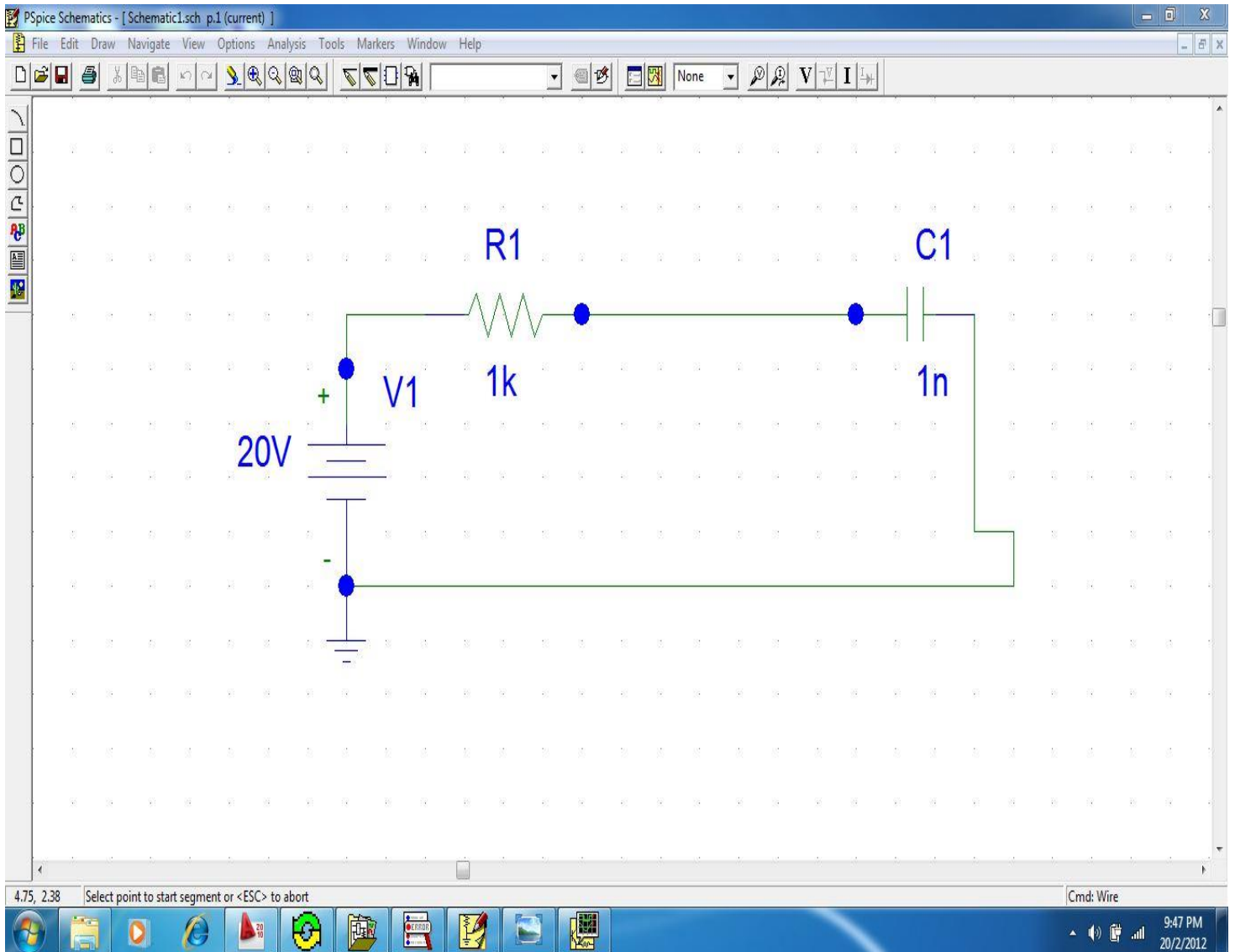


Figure-7

10. The Figure-7 shows a sample circuit, in which a DC voltage source, Resistor and a capacitor are placed, the nodes are shown by pspice by blue dots. The circuit is built by searching elements from **place a part button** and then drawing wire to connect them from **Draw wire button**.



# AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

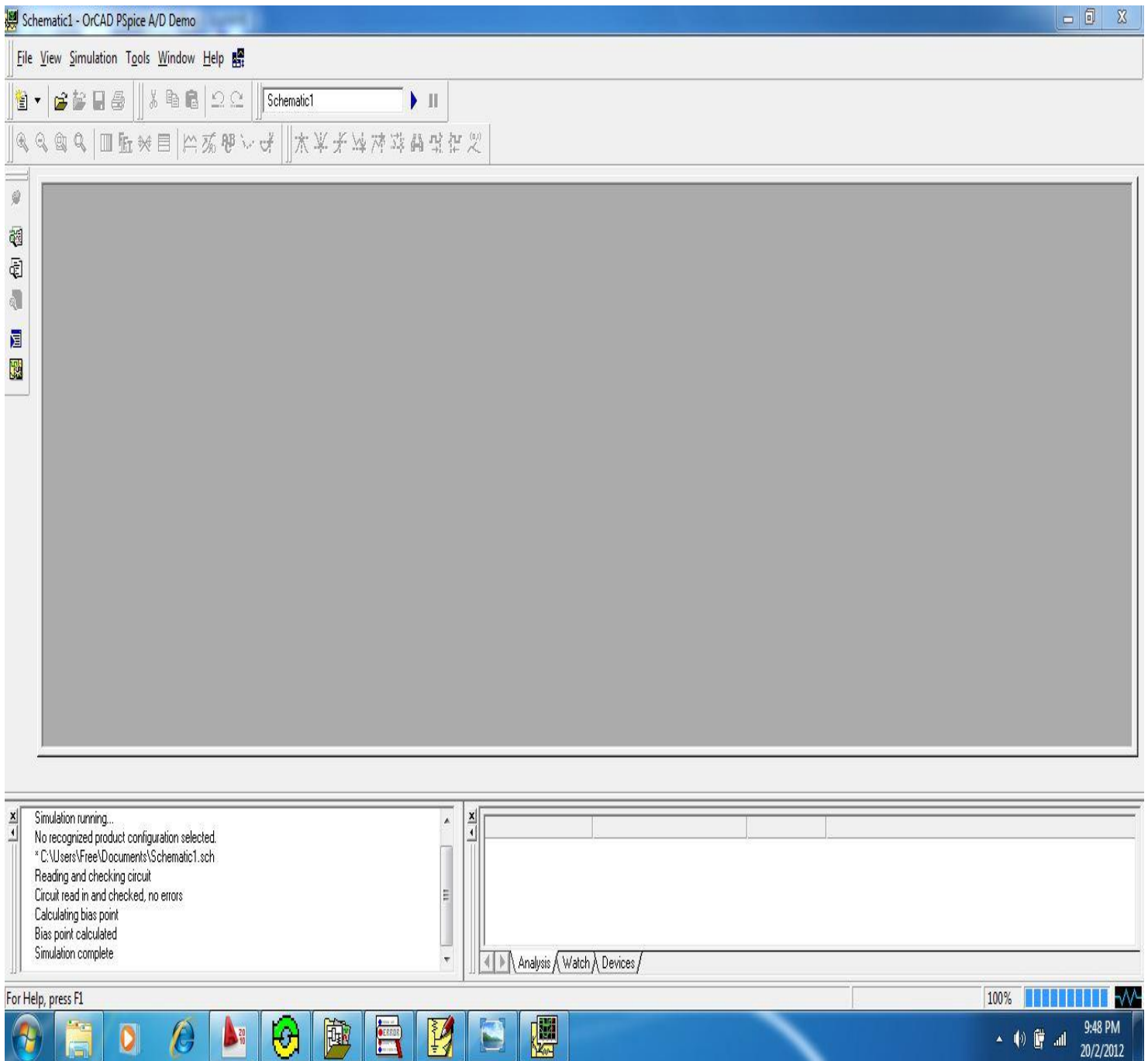


Figure-8

11. The Figure-8 above shows a simulation running where you can get idea of errors and warnings, this can be used as a further step when you get acquainted with software.

AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY  
ELECTRIC CIRCUITS LAB – EE 226

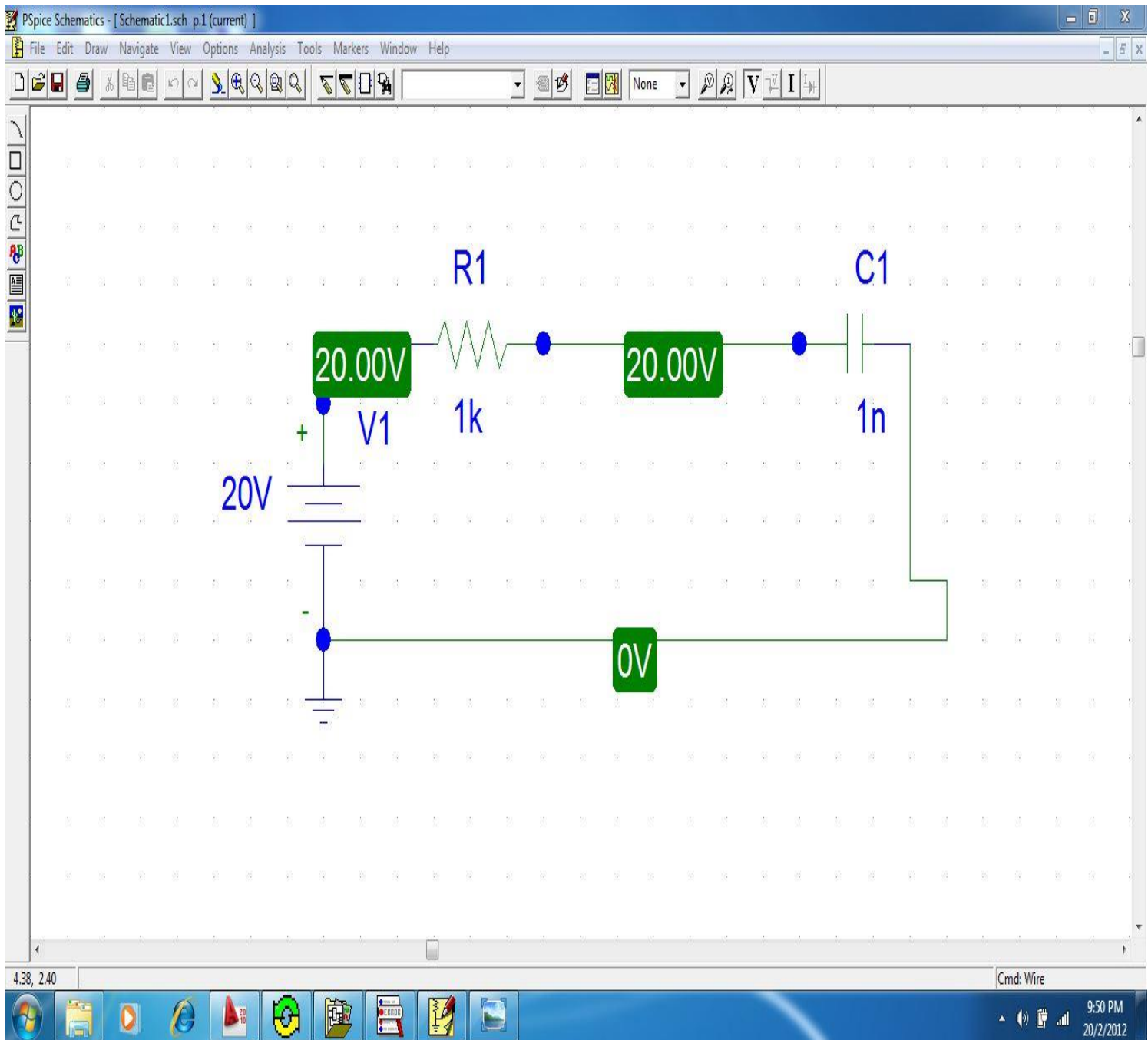


Figure-9

12. The Figure-9 shows voltages at nodes, this can be obtained by pressing the V button as shown in figure-5 step-8, the voltages appear in green boxes.

AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY  
ELECTRIC CIRCUITS LAB – EE 226

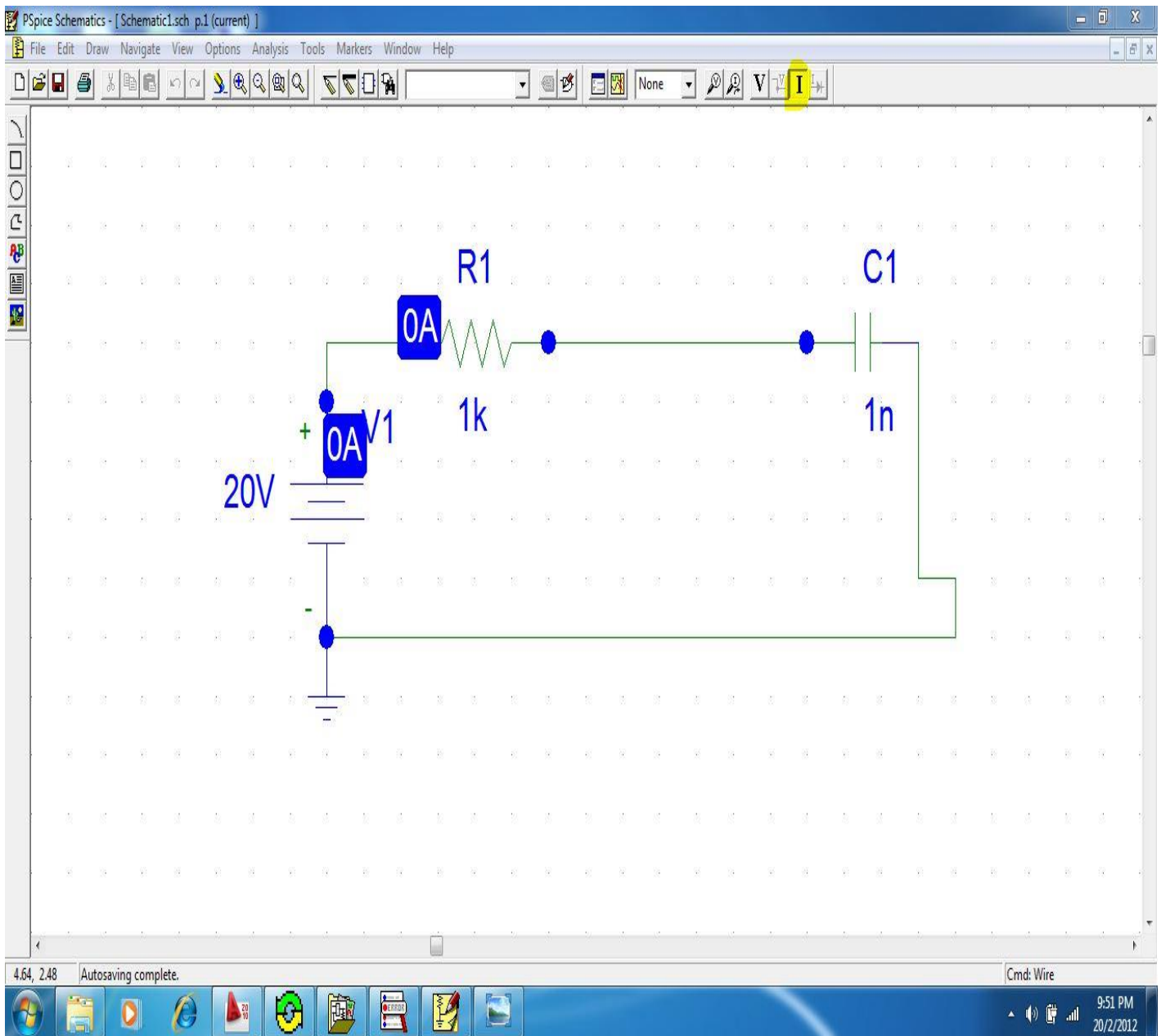


Figure-10

13. The Figure-10 shows we obtain the currents in the branches, this can be obtained by pressing the **I** button as shown in figure-5 step-8, the Currents appear in blue boxes.

# AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

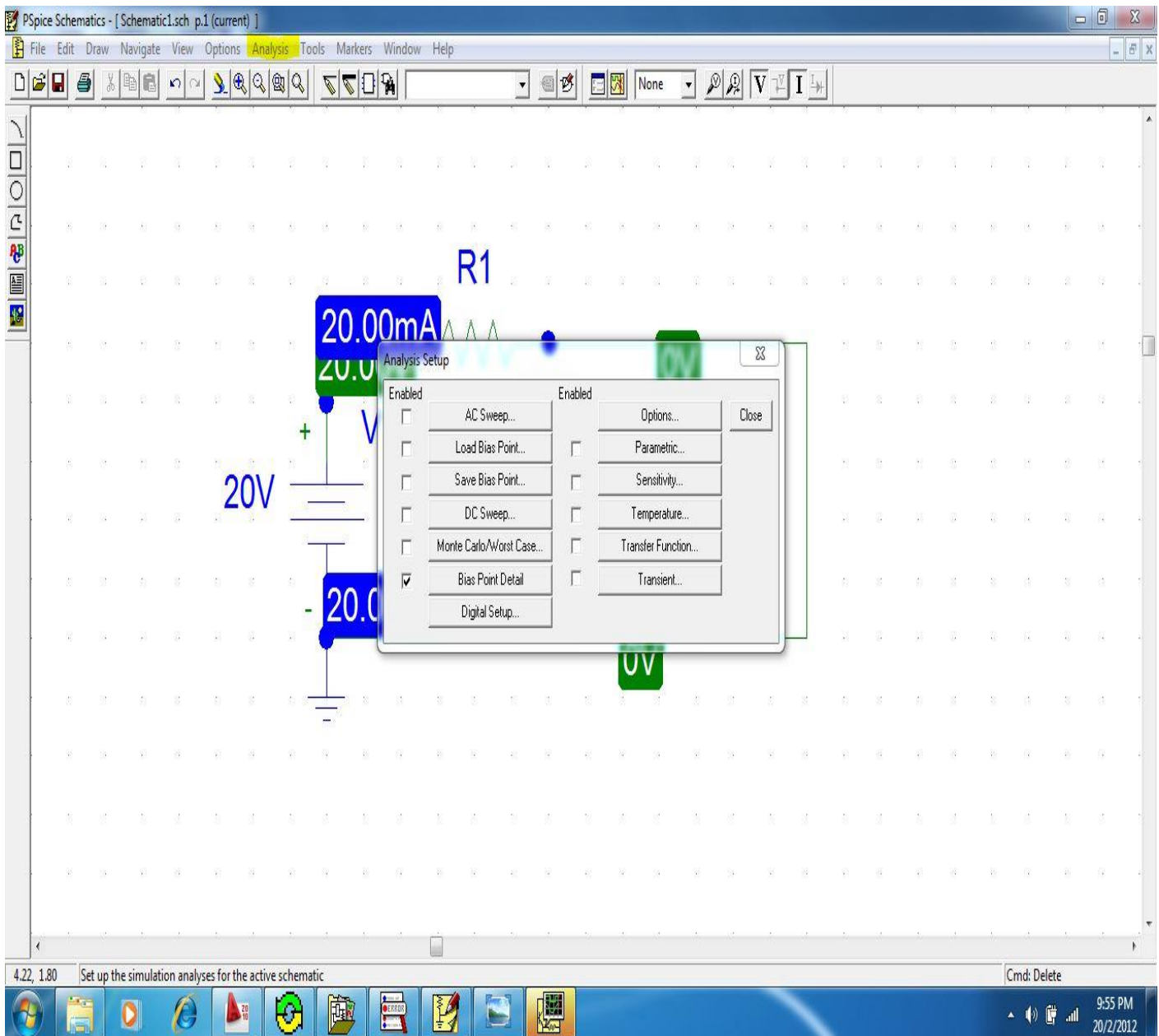


Figure-11

14. The Figure-11 shows **Analysis setup** in this you can have an option to select which analysis you want, only steady state analysis or transient Analysis or both at a time or as shown in Figure-11 can be done by selecting it.

# AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY

## ELECTRIC CIRCUITS LAB – EE 226

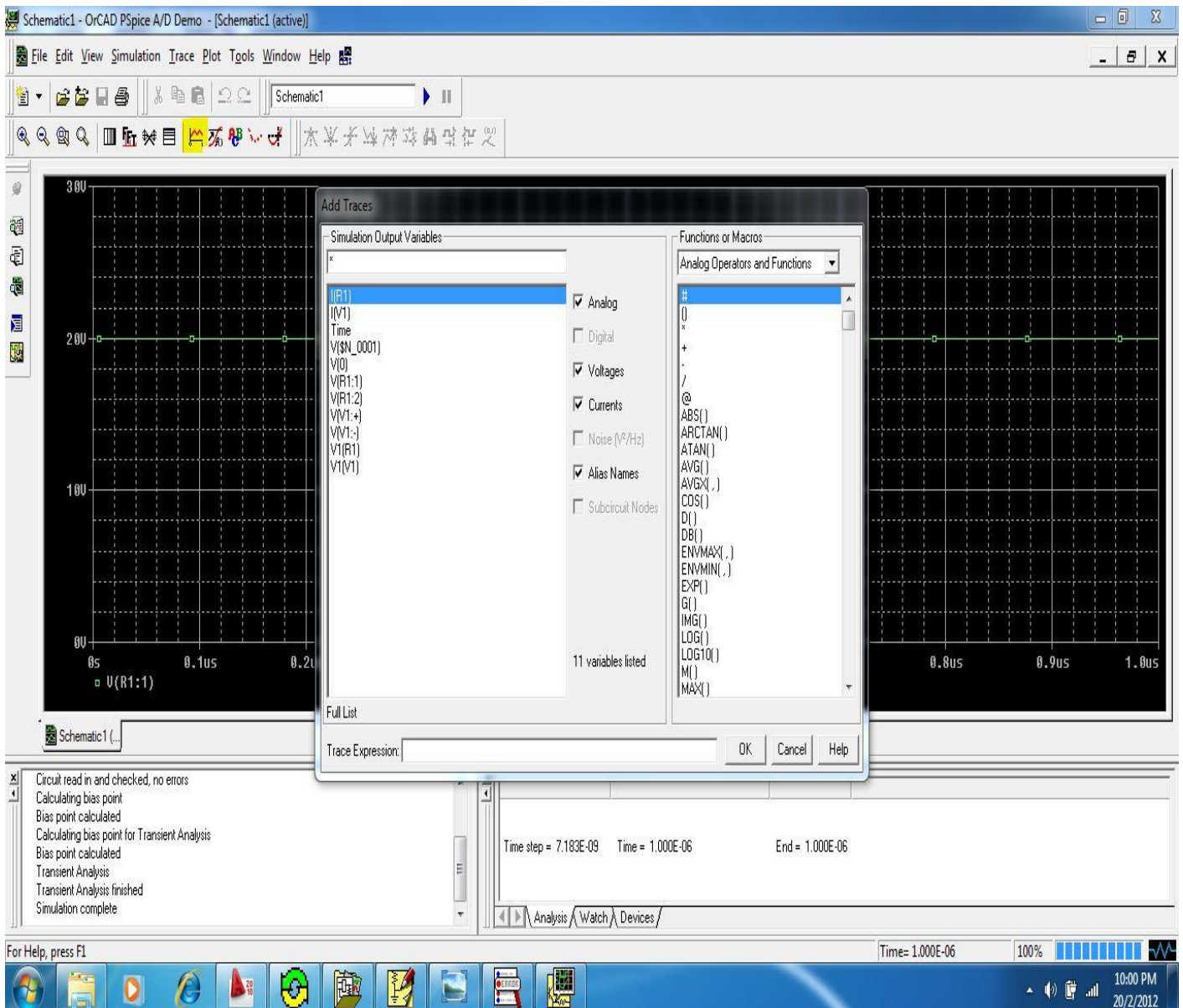


Figure-13

15. The Figure-13 shows us probe, probe can be obtained by going in Analysis button as shown in step-6, Figure-3 we will then get a window showing an empty waveform screen, The highlighted yellow color in Figure-13 shows you a button, which when pressed it we will get waveform option for a value of the circuit elements for example a waveform of current flowing through the Resistor R1 or say a voltage across some element in the circuit. Select the output variable and then press **ok**, we have all other options as you can see on the screen.

**AI-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY**  
**ELECTRIC CIRCUITS LAB – EE 226**

**Exercise:-**

- a. To Draw A RLC series circuit with DC voltage as power Source  
V=20 volts, R=100 $\Omega$  and L=0.1mH on pspice.
- b. To Draw A RLC Parallel circuit with DC voltage as power Source  
V=20 volts, R=100 $\Omega$  and C=0.1mF on pspice.

To do the above exercise follow the Steps from 4-16 as said above and construct the circuit Construct the circuit.

**Result: -** Get familiar with pspice and perform simple circuit analysis.

**Conclusion & comments:-**

**AL-IMAM MUHAMMAD IBN SAUD ISLAMIC UNIVERSITY**  
**ELECTRIC CIRCUITS LAB – EE 226**

**Questions :-**

1. In pspice VSIN & VDC means which sources?
  2. What does SPICE stands for?
  3. Explain what does simulation means?
  4. How do we see waveforms in pspice?
-