

# PSPICE (1. DC Analysis) for Network Analysis & Lab

Dr. Charles J. Kim

Howard University

Acknowledgment: Mr. Chimaobi Mbanaso, a junior in Electrical and Computer Engineering, helped with PSpice runs and window captures.

# What is PSPICE?

- SPICE (Simulation Program for Integrated Circuits Emphasis)
  - A powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the **circuit behavior**
  - Originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975)
- Pspice
  - A **PC version** of SPICE (MicroSim Corp.)
- Hspice
  - A version that runs on workstations and larger computers

# Analysis Types

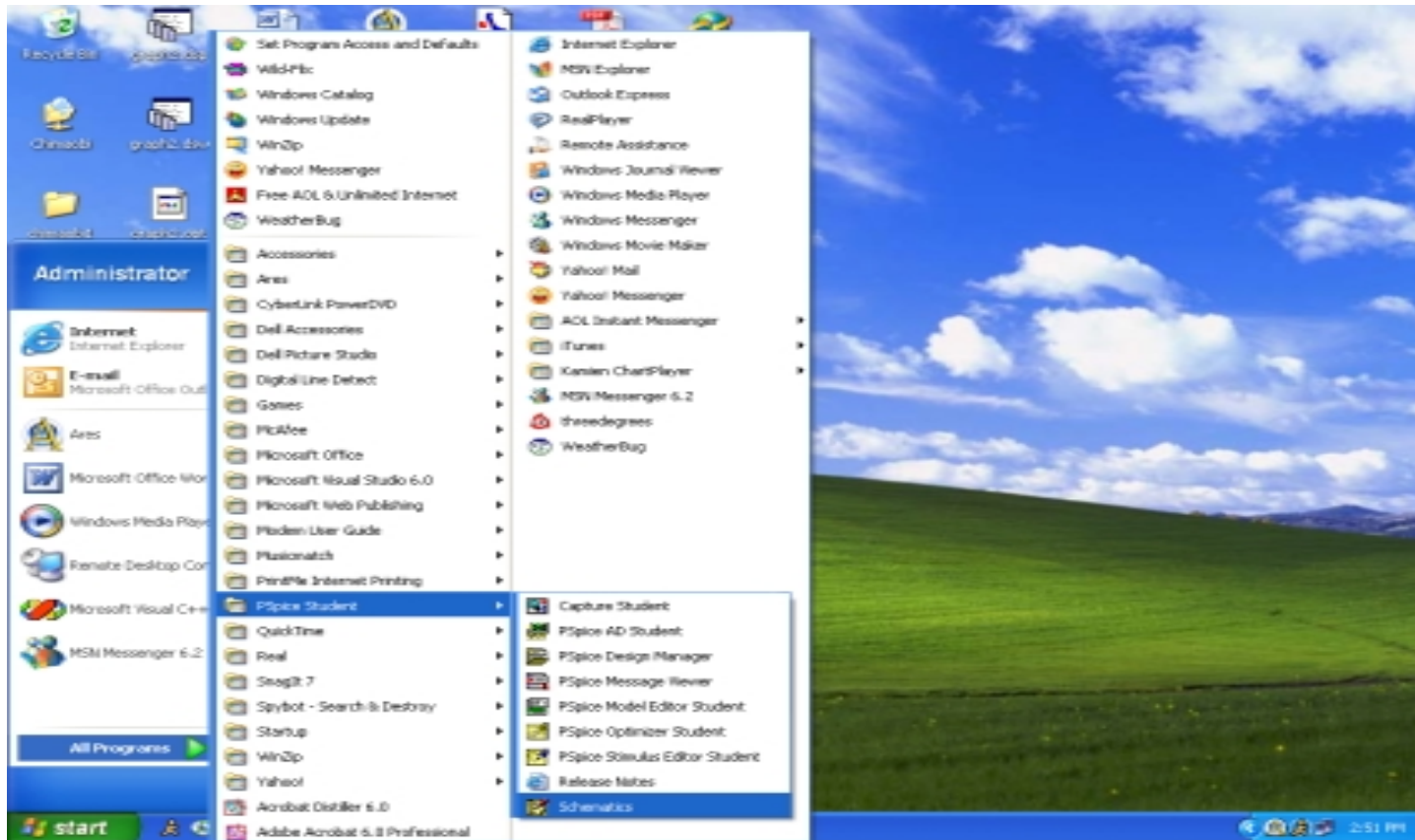
- **Non-linear DC analysis:** calculates the DC transfer curve.
- **Non-linear transient analysis:** calculates the voltage and current as a function of time when a large signal is applied.
- **Linear AC Analysis:** calculates the output as a function of frequency. A bode plot is generated.
- **Noise analysis**
- **Sensitivity analysis**
- **Distortion analysis**
- **Fourier analysis:** calculates and plots the frequency spectrum.
- **Monte Carlo Analysis**

# Components Available

- Independent and dependent voltage and current sources
- Resistors
- Capacitors
- Inductors
- Mutual inductors
- Transmission lines
- Operational amplifiers
- Switches
- Diodes
- Bipolar transistors
- MOS transistors
- JFET
- MESFET
- Digital gates

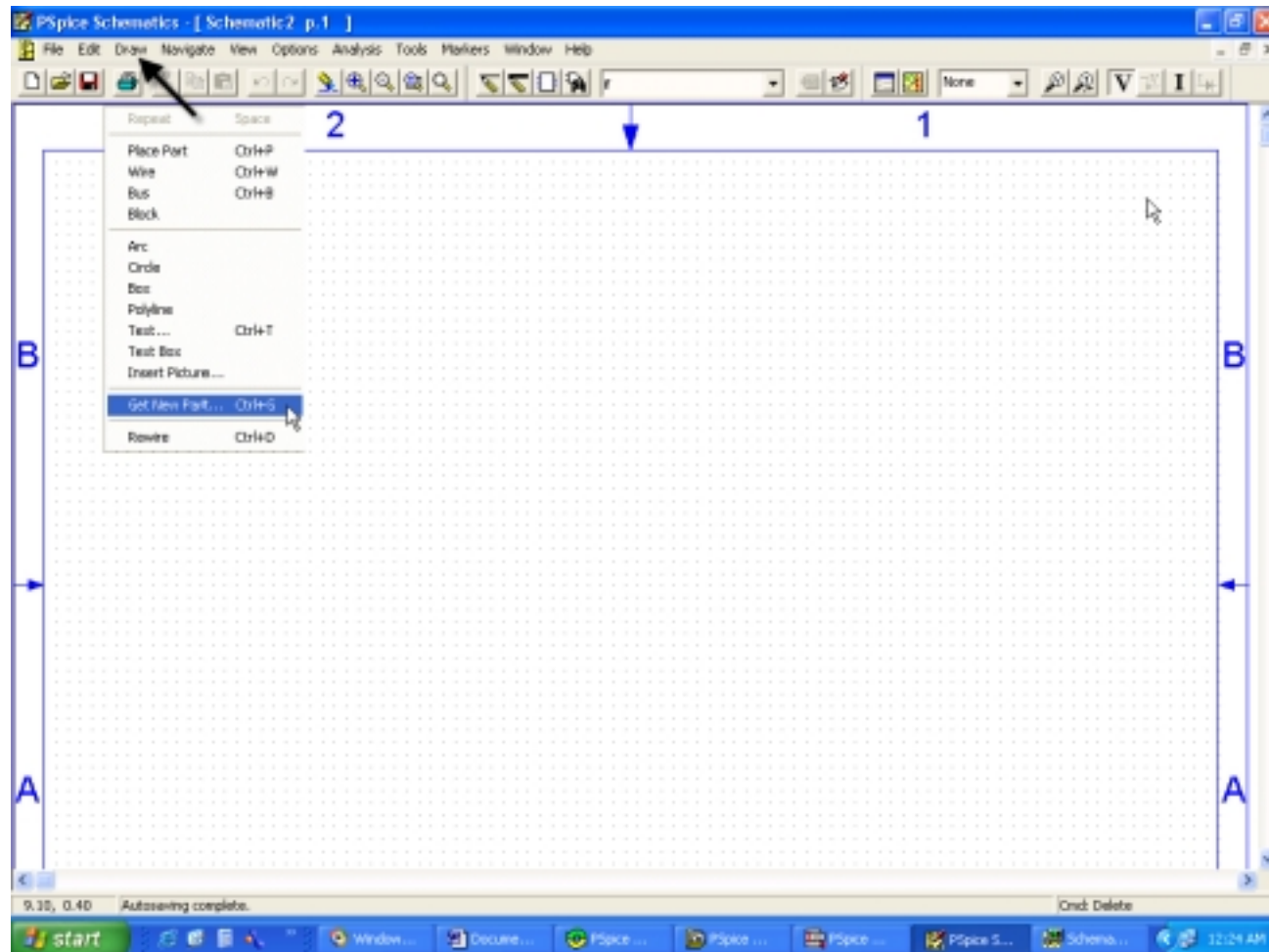
# Getting Started

- Logon to PC near you. (user/user)
- Open up the Pspice Schematics application



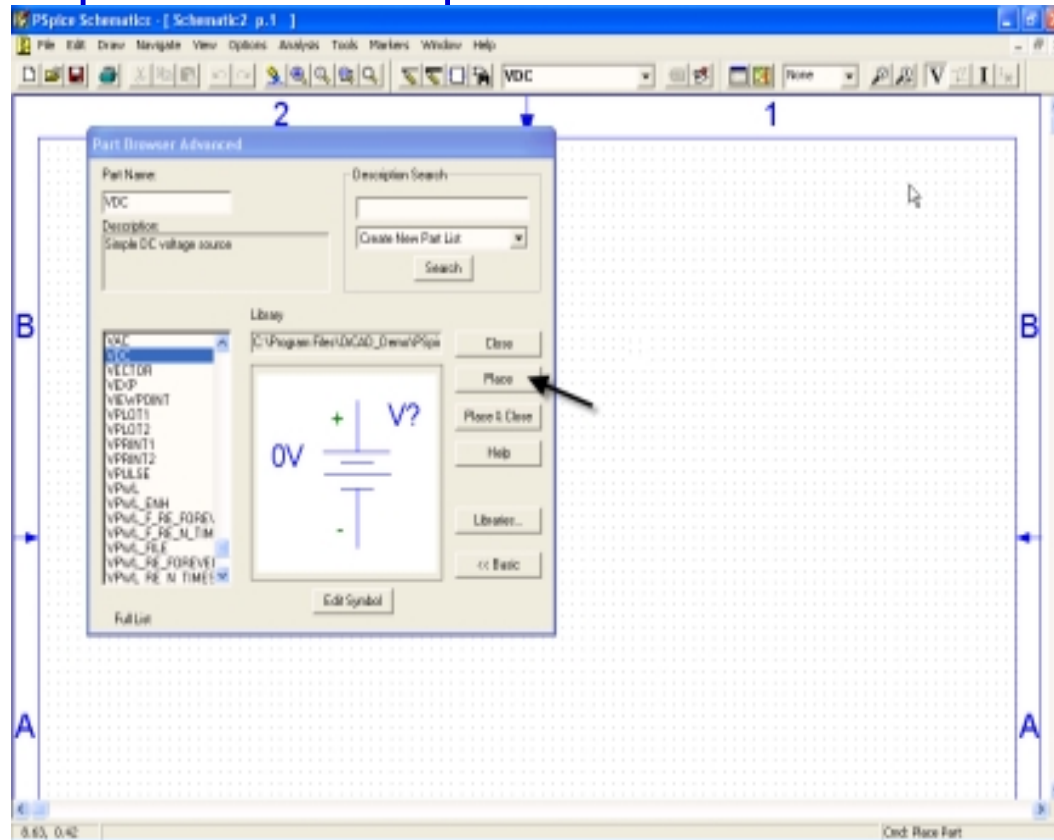
# Part Placement

- Click on **Draw** on the **Menu Bar** and then, click on the **Get New Part** option.



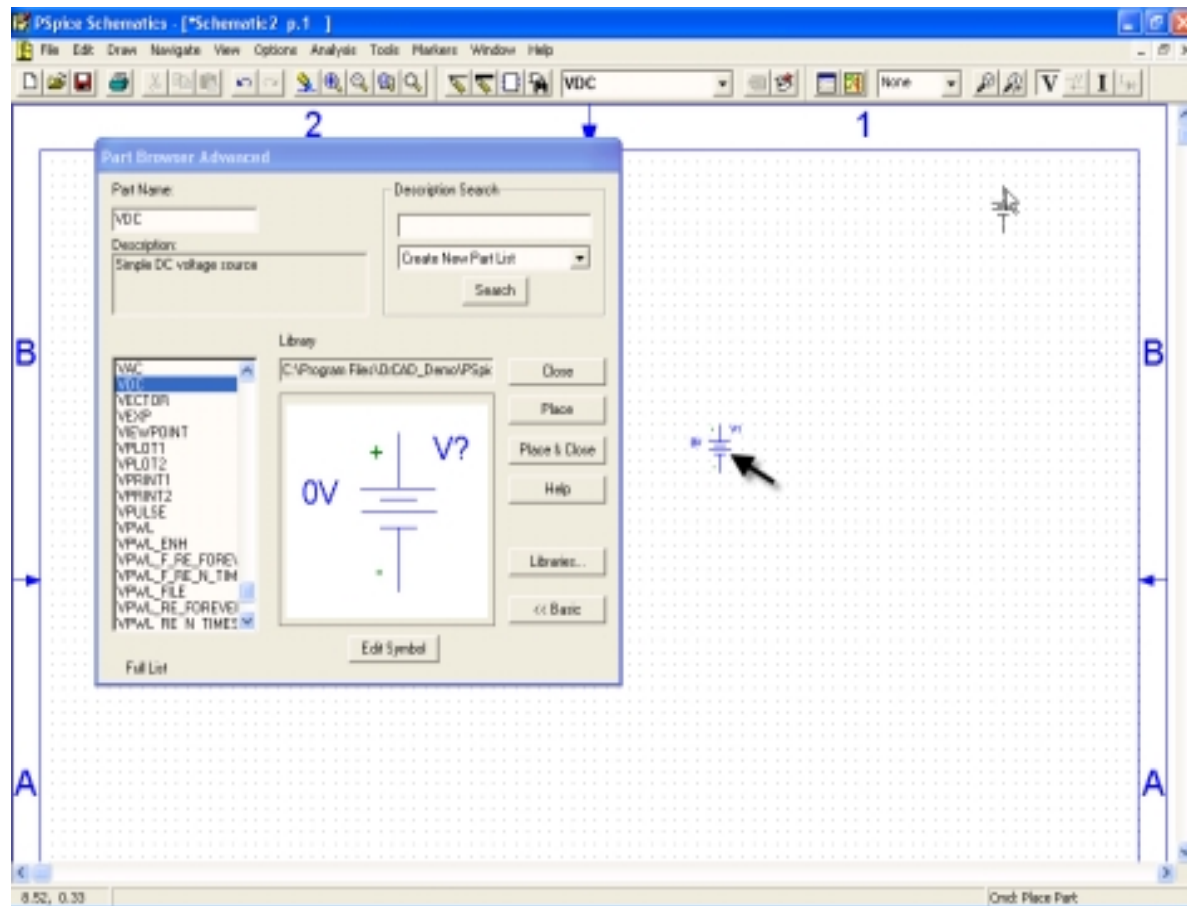
# Independent dc Voltage Source

- In the open window, click on the space under **Part Name** and type **VDC**. This is the independent voltage source . When the part shows up click on the **Place** tab.



# Placing the dc Voltage Source

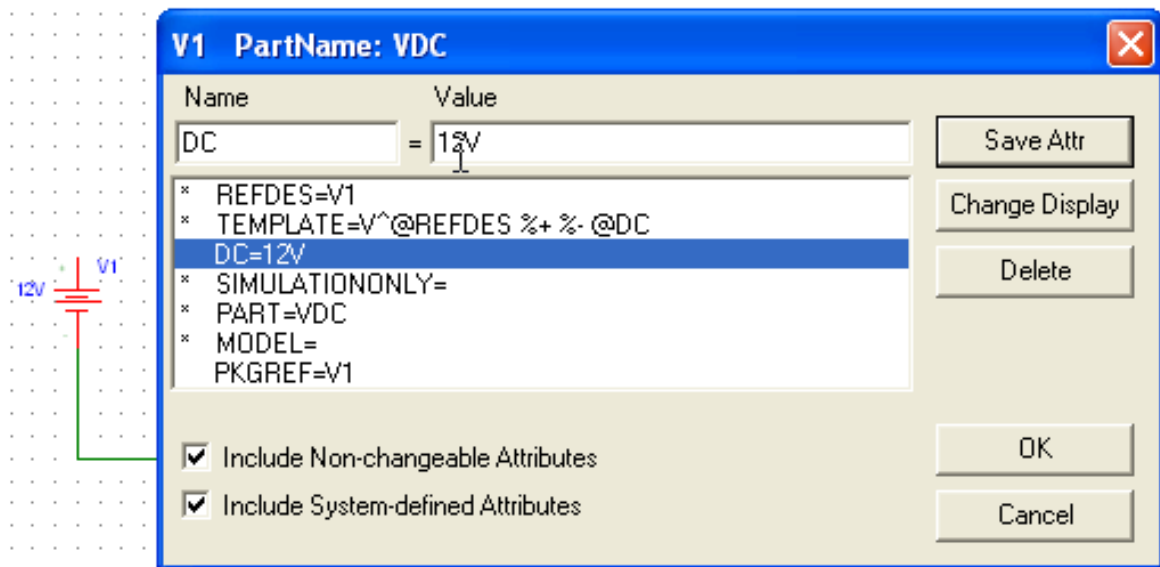
- Click **anywhere** on the schematic background to place the part and **then** right click.





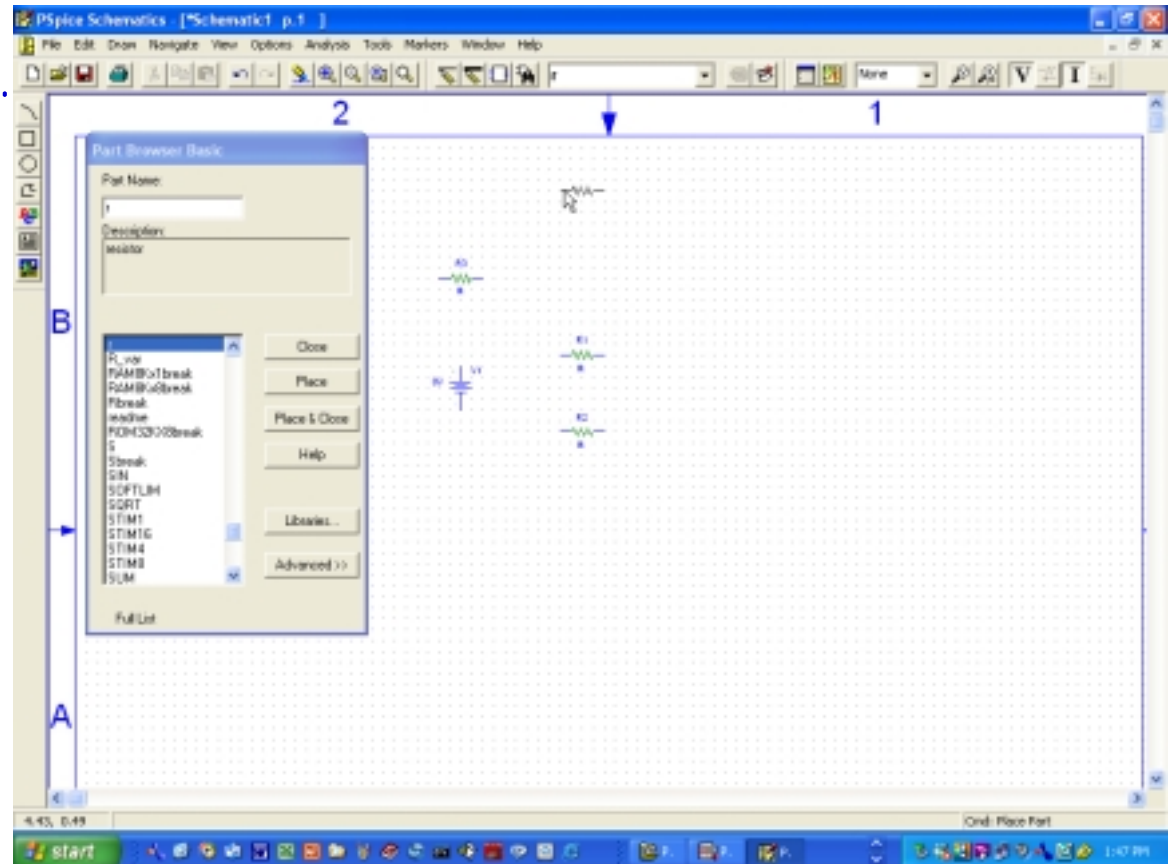
# Voltage Value Setting

- Double click the DC source
- Type Value
- Click **Save Attr** Button
- Then, OK



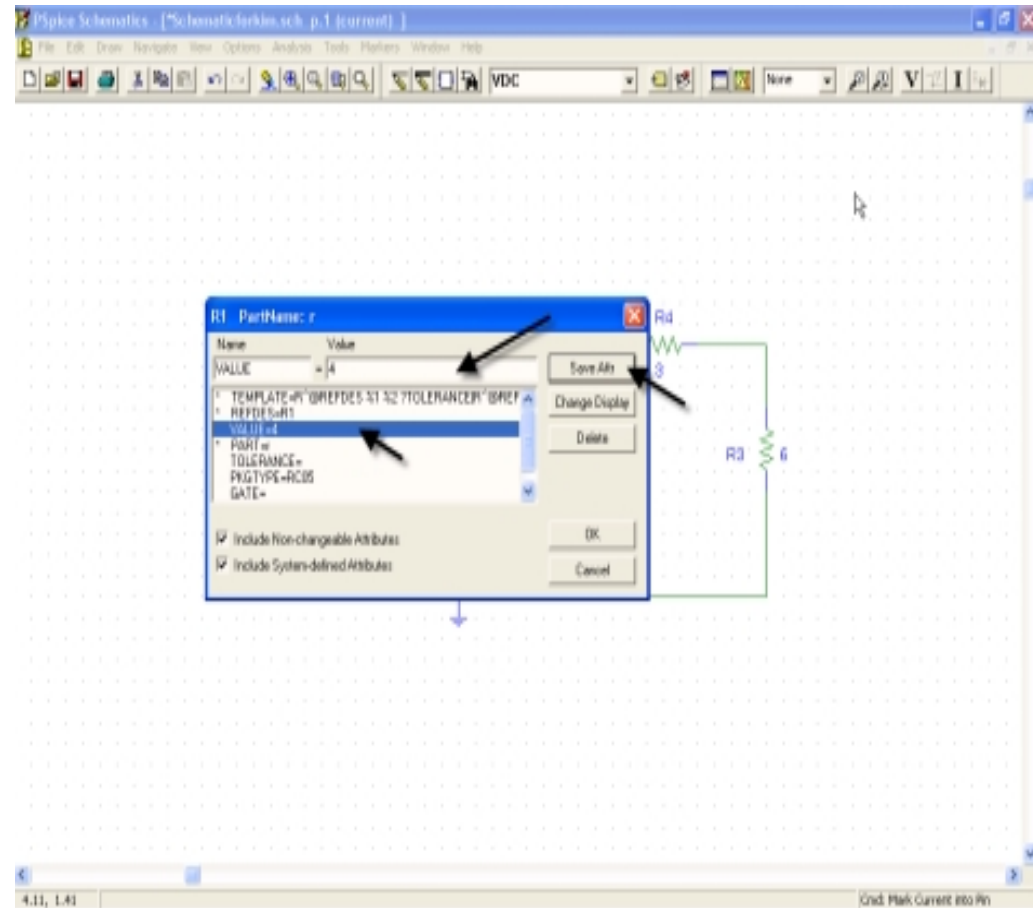
# Resistor Placement

- Go back to the **Part Brower Basic** window and in the space under **Part Name**, type **r**. Place one resistor and move the cursor on the background and **click again**. In this way you can place as many resistors as you need.
- **Right click** when done.



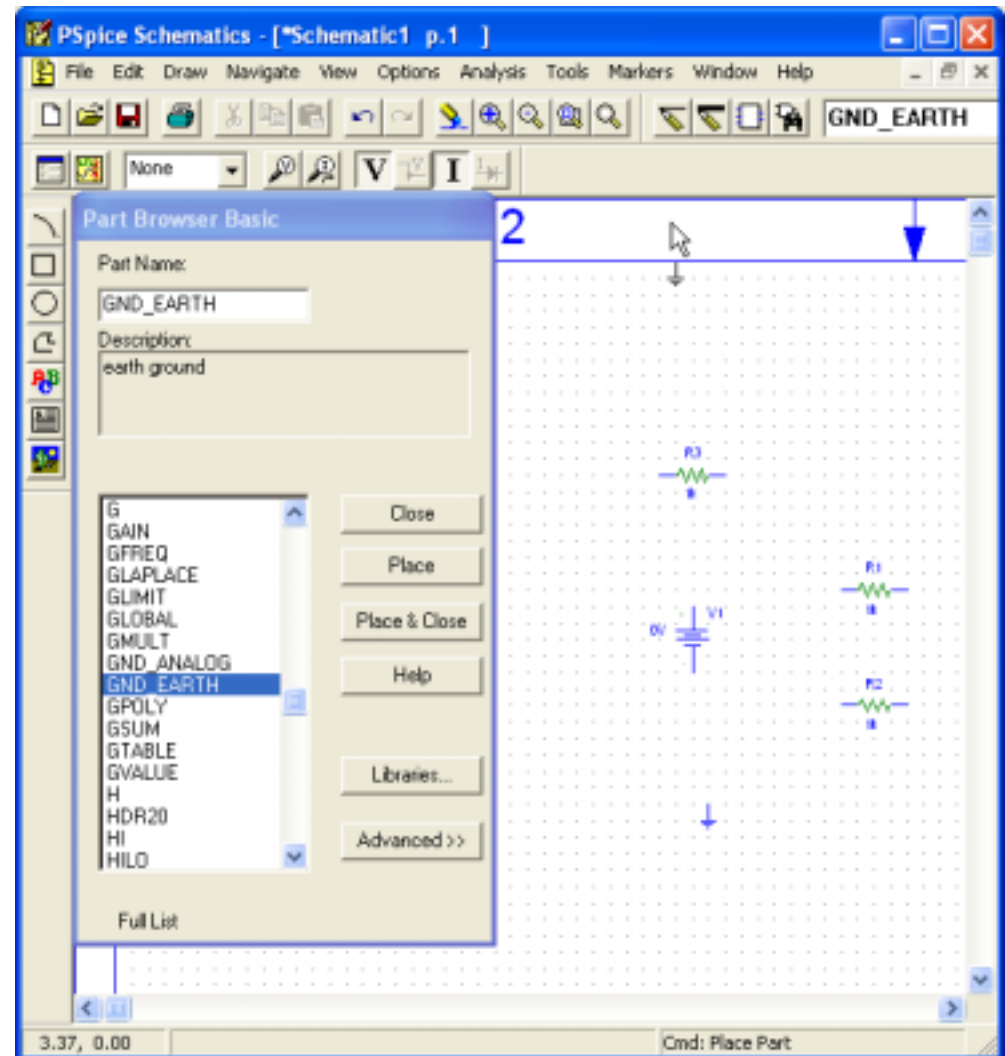
# Resistance Value Setting

- Double click on each resistor and type in the specified value into the space shown below. Click on Save Attr.



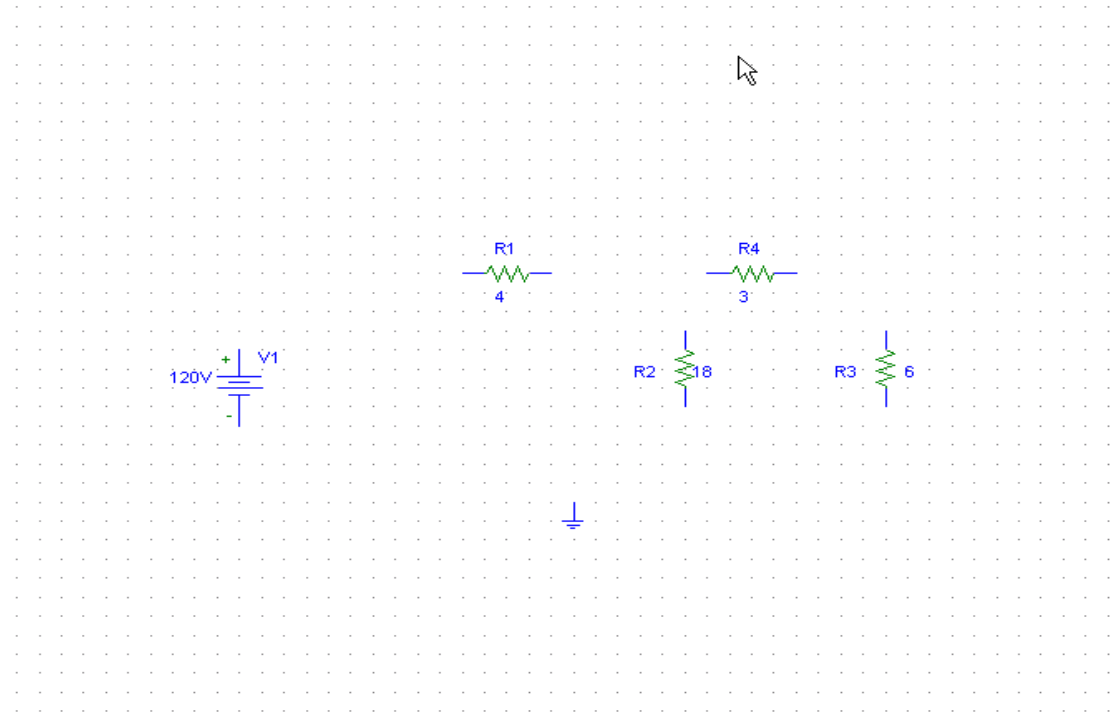
# GND placement

- Following the same procedure type **GND\_EARTH** and place the part on the background. This for the common ground.
- Close the Part window.



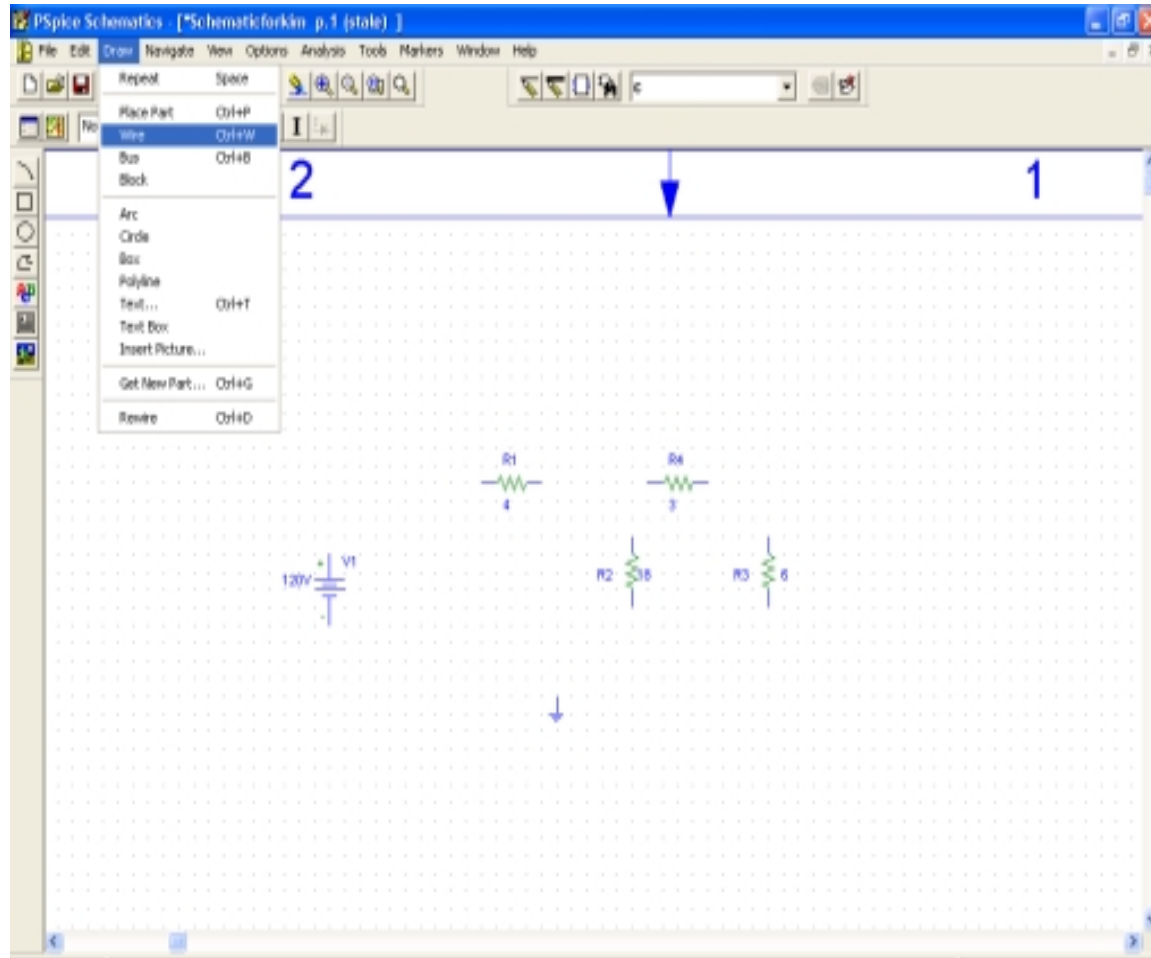
# Placement of all the Parts

- Place the parts in such a way that would ease the connection of the wires.



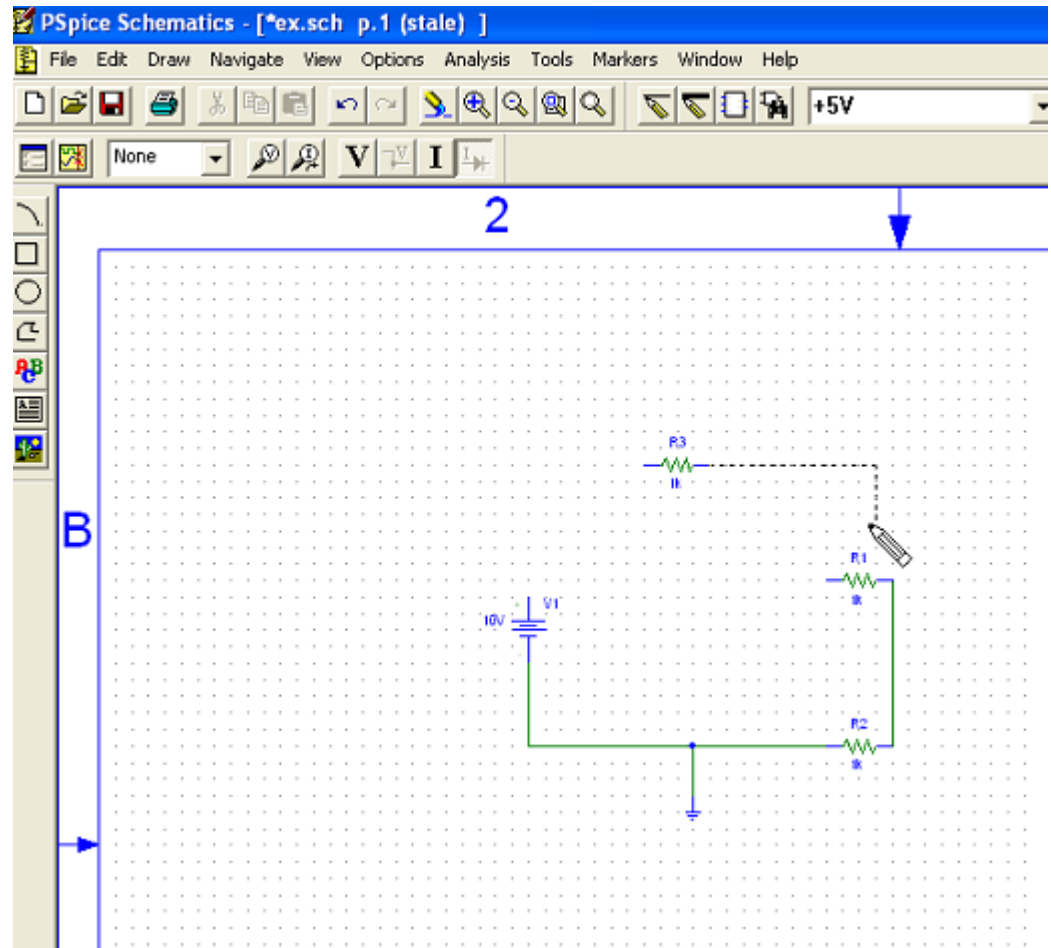
# Wring the Parts

- Click on **Draw** on the **Menu Bar** and on the drop down menu click on **wire**. ( CTRL + W is the shortcut)

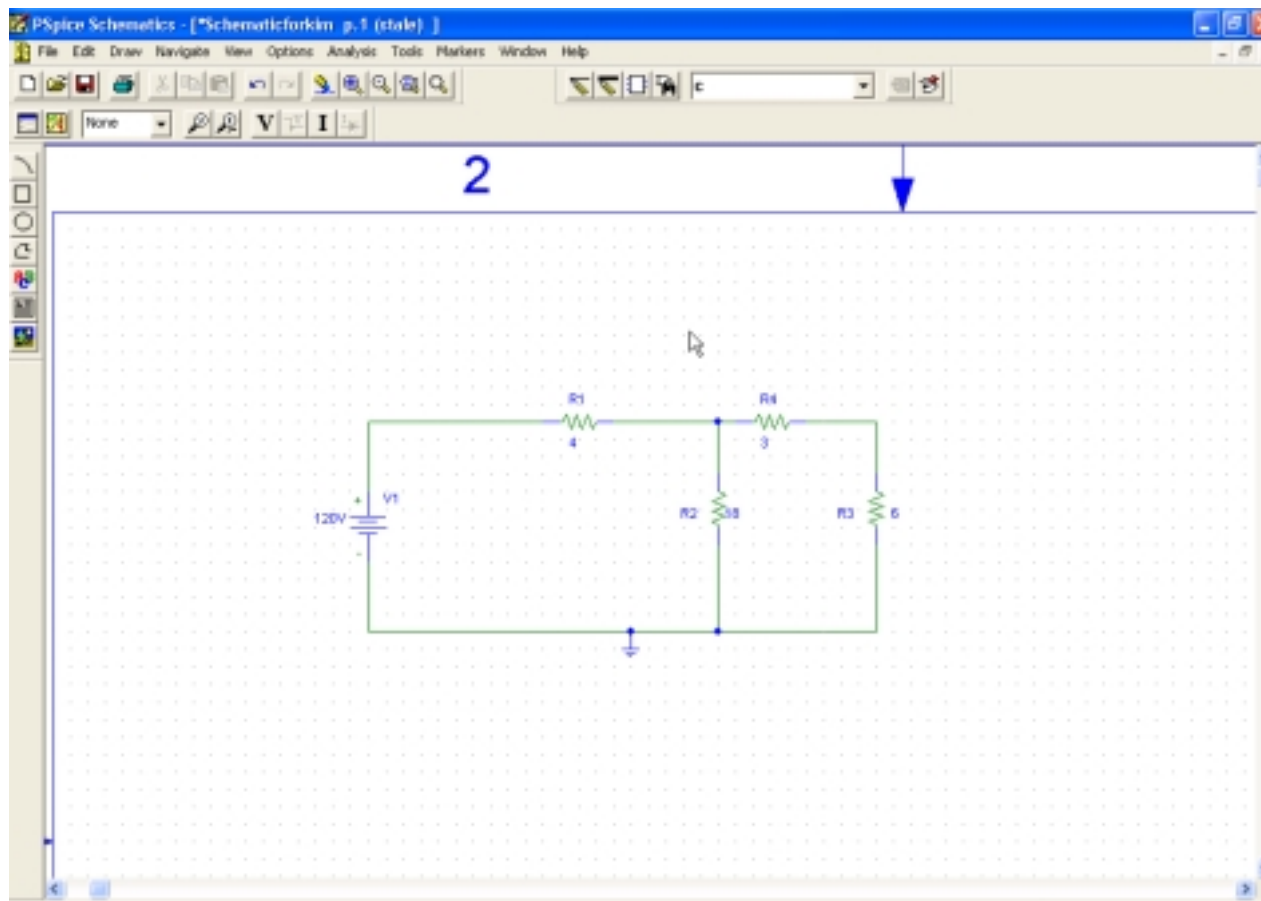


# Wiring

- First click on the **terminals/edges** of the circuit component.
- Then **release** the mouse button.
- Draw until you get to the **edge of the next component** it should be connected to.
- Then **Click Mouse Button** to finish.



# Completed Circuit

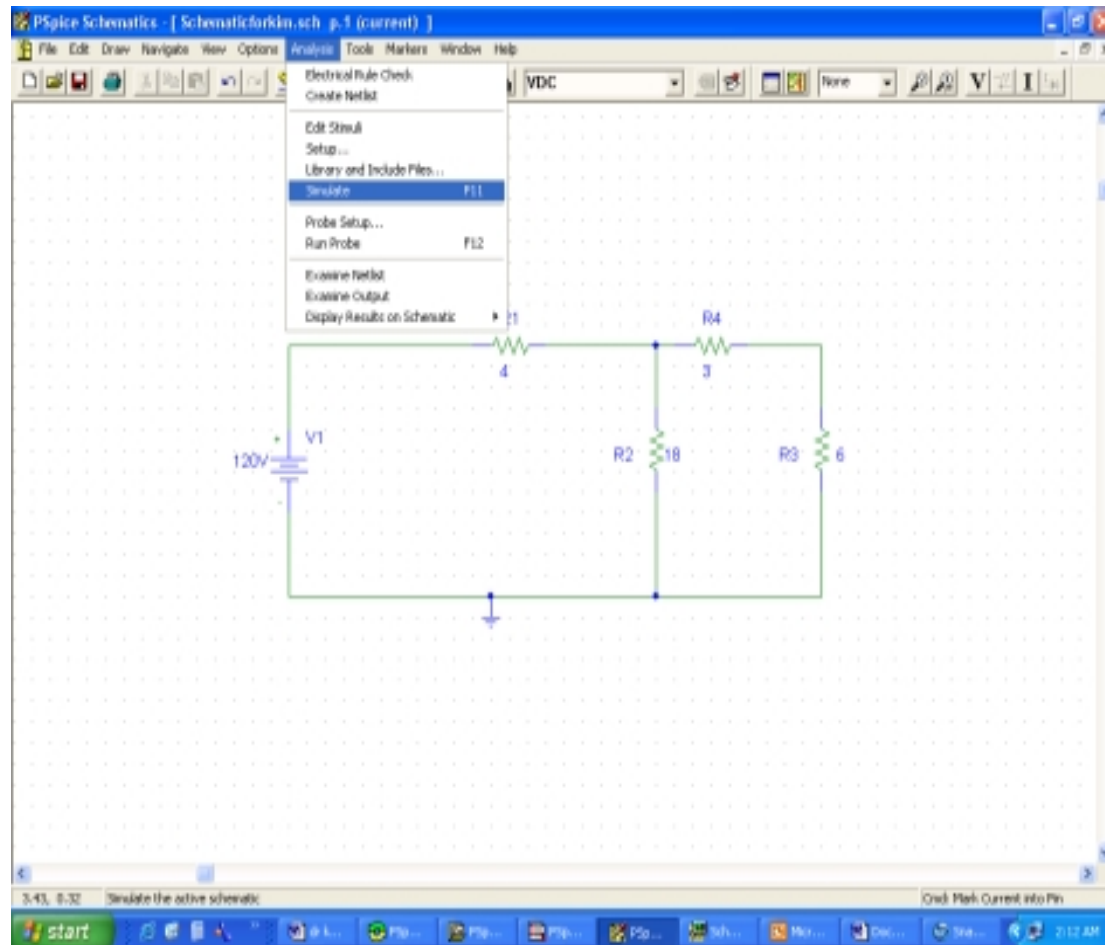


- Then, Save your schematic

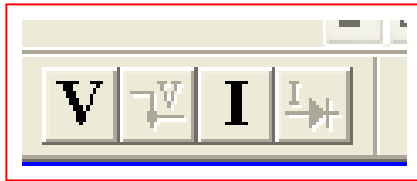


# Circuit Simulation – Analysis

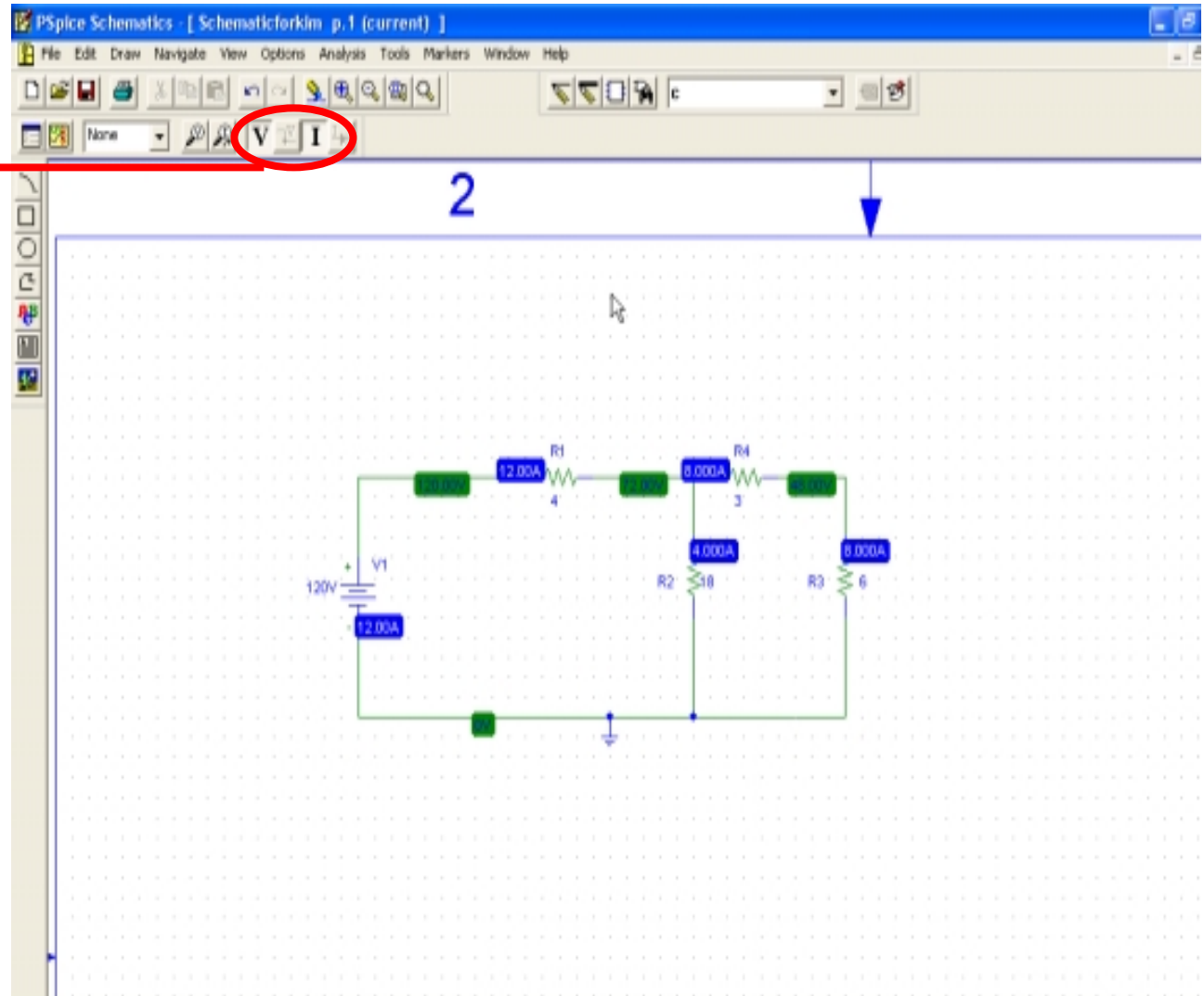
- Click on Analysis and then Simulate(F11 is the shortcut), to simulate the circuit.



# Result!

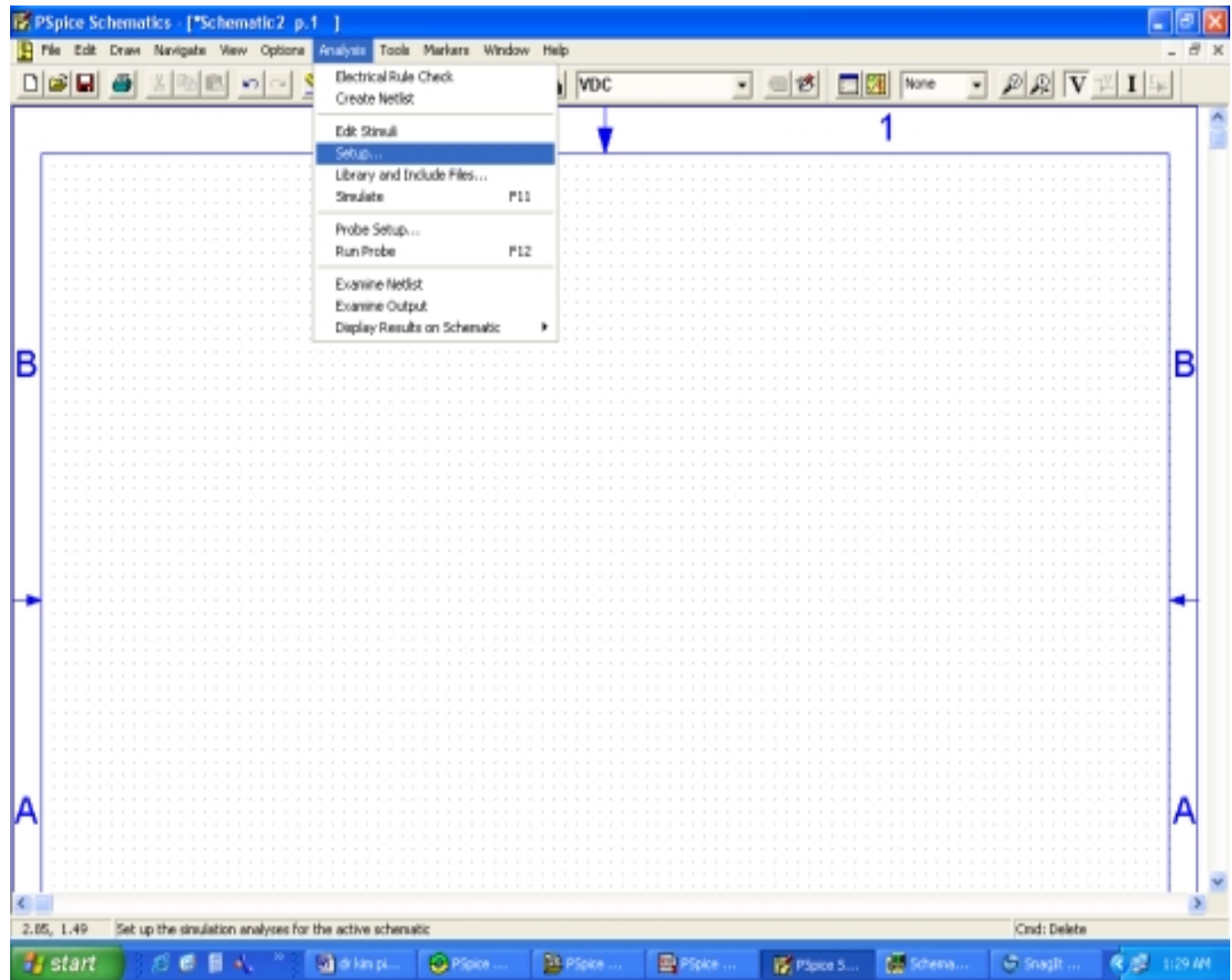


- Toggle these buttons, and see what happens



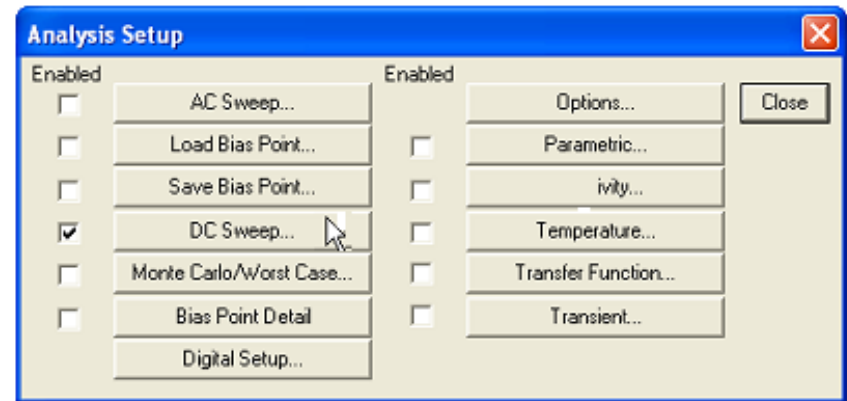
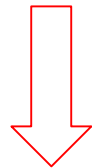
# Graphical Display of the Result

- Menu Bar
- > Display
- > Setup

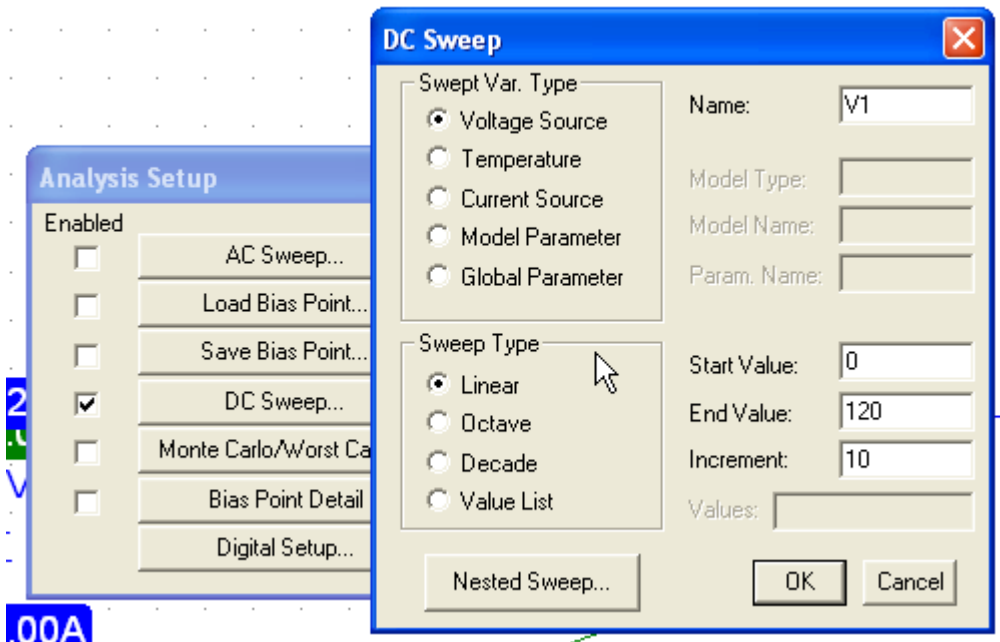


# Analysis Setup Dialog Window

- Analysis Setup Window
- Click DC Sweep Bar

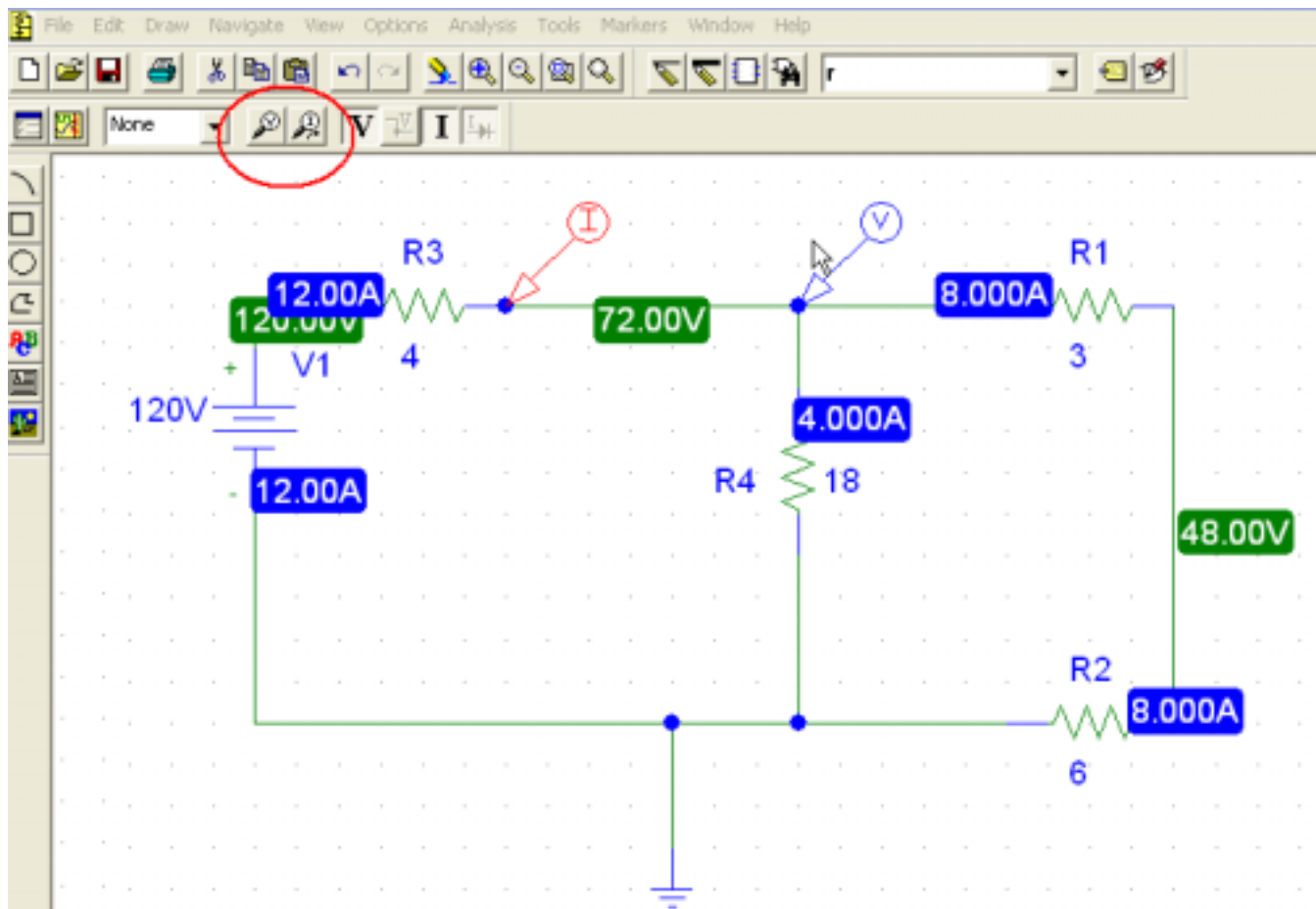


- DC SWEEP Window
  - Voltage Source
  - Sweeping
    - From 0
    - To 120
    - Increment of 10



# Measurement Probes

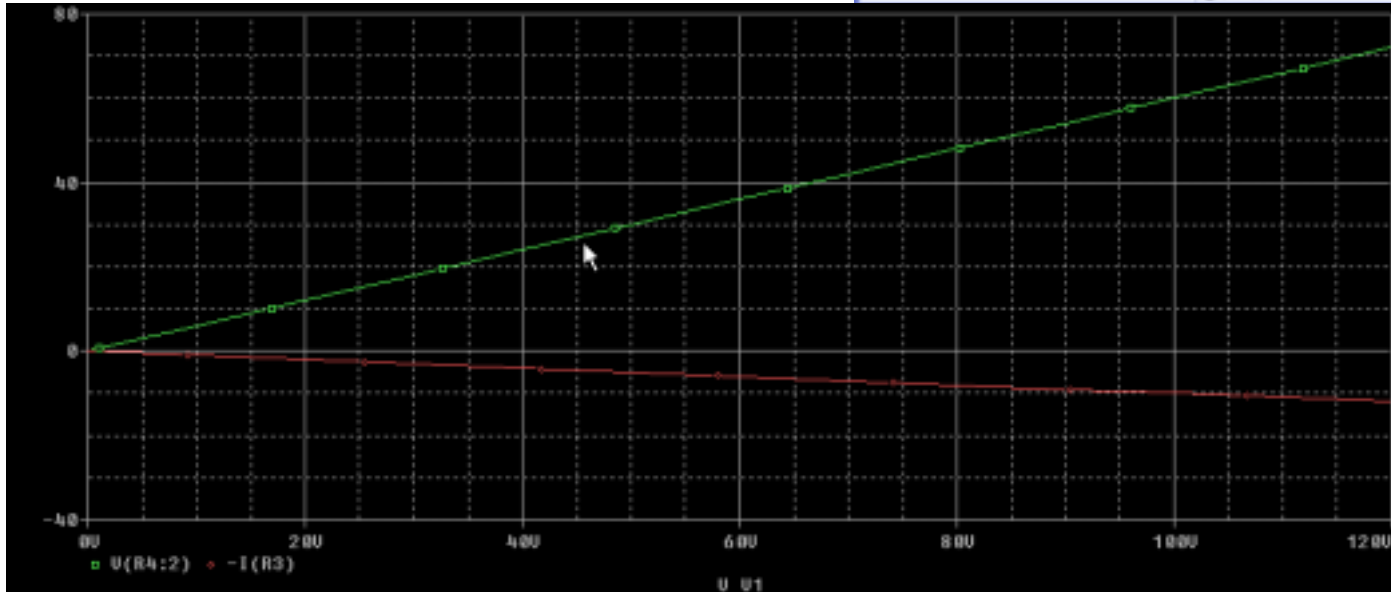
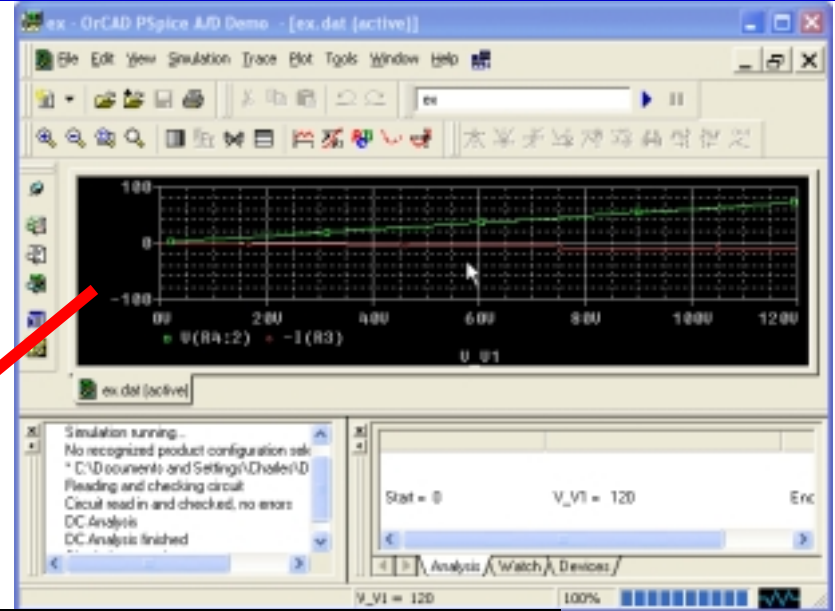
- Voltage Probe
- Current Probe



# Simulation Result

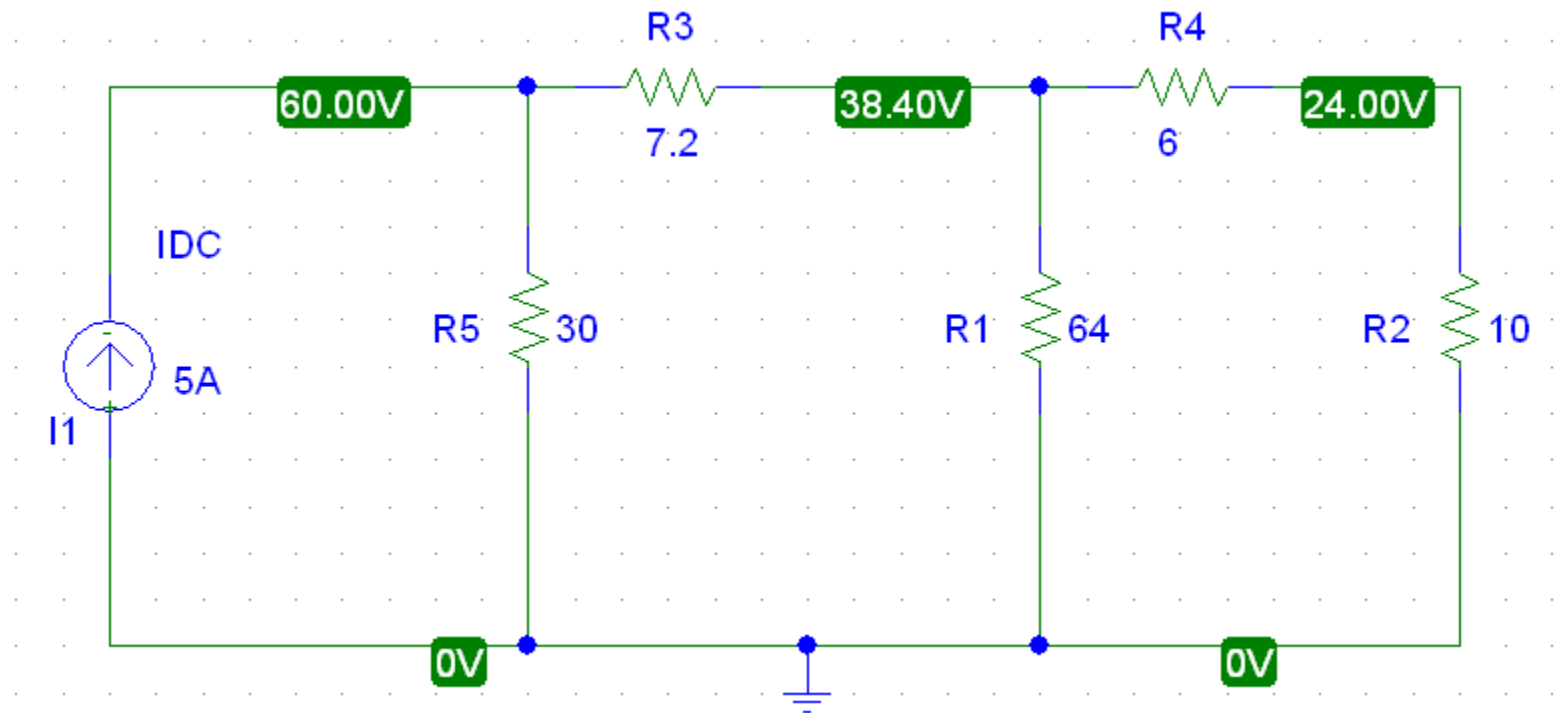
- Menu Bar
- >Analysis
- >Simulate

Detail



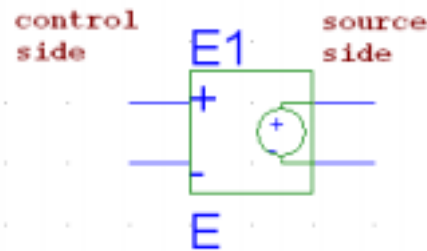
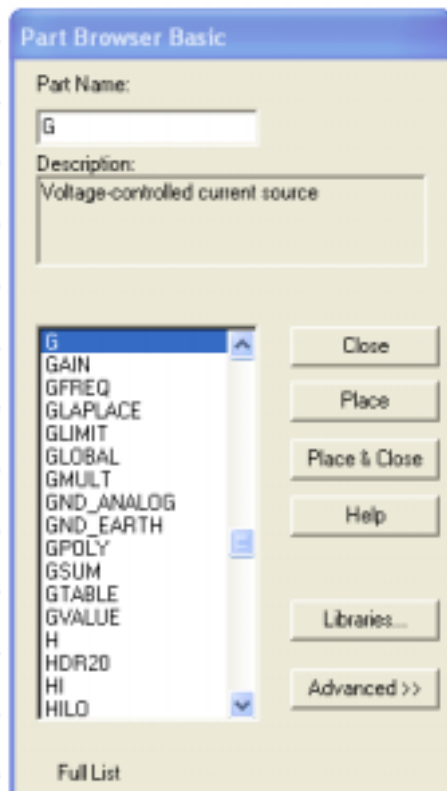
# Practice Problem

- Independent dc Current Source
- 5 Resistors
- Build the following circuit and Simulate it.

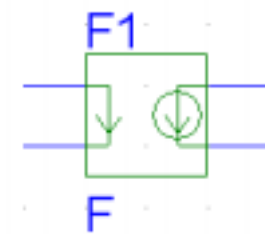


# Simulation of a simple circuit containing a Dependent source:

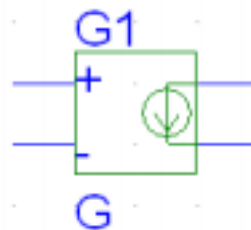
- There are Four Types of dependent sources with parts:
  - Voltage controlled voltage source = E
  - Voltage controlled current source = G
  - Current controlled current source = F
  - Current controlled voltage source = H



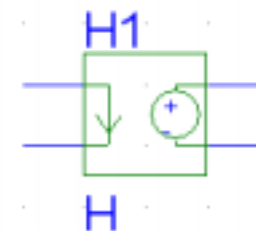
voltage controlled voltage source



current controlled current source



voltage controlled current source

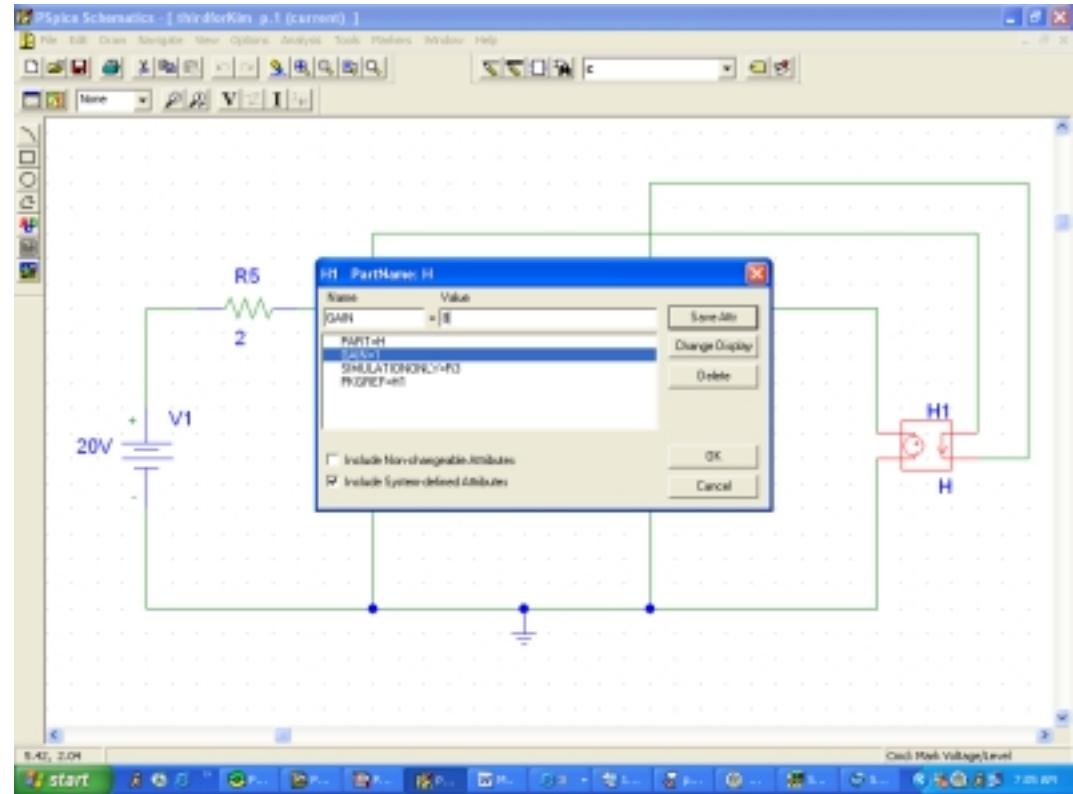


current controlled voltage source



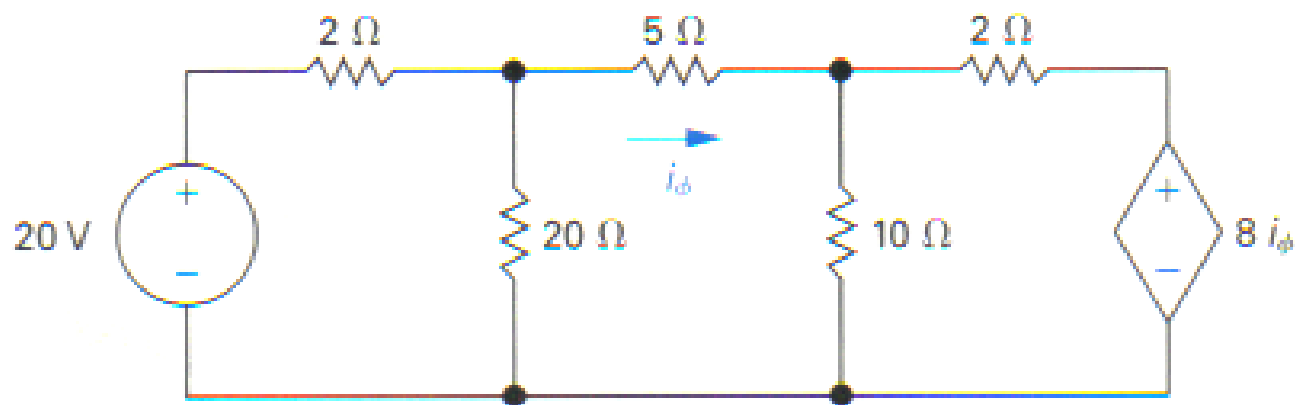
# Dependent Source Setup

- Double click on the dependent source
- Click on **gain** and change the value to 8. This gives the factor by which the current through the branch is changed for the source.

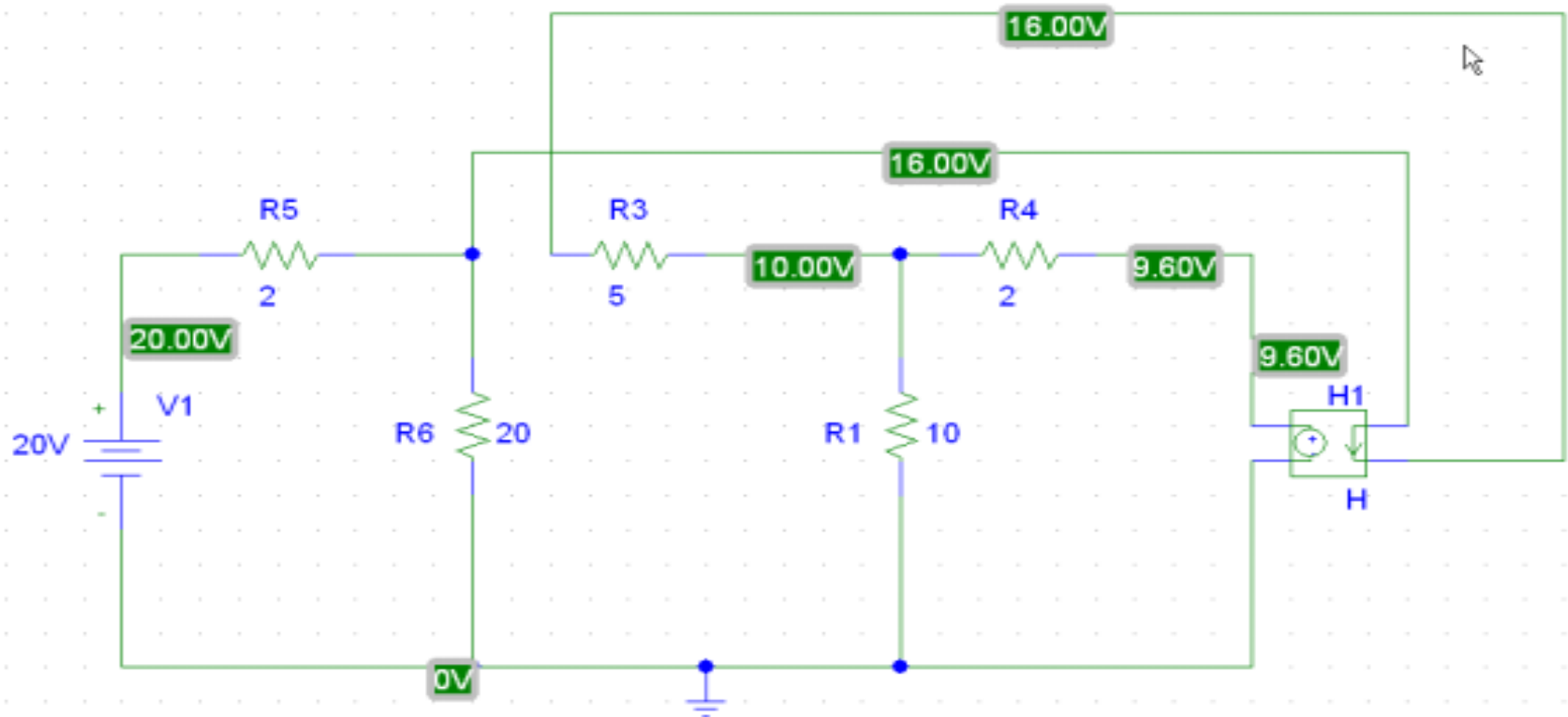


# Practice Problem with dependent source

- Simulate the following circuit



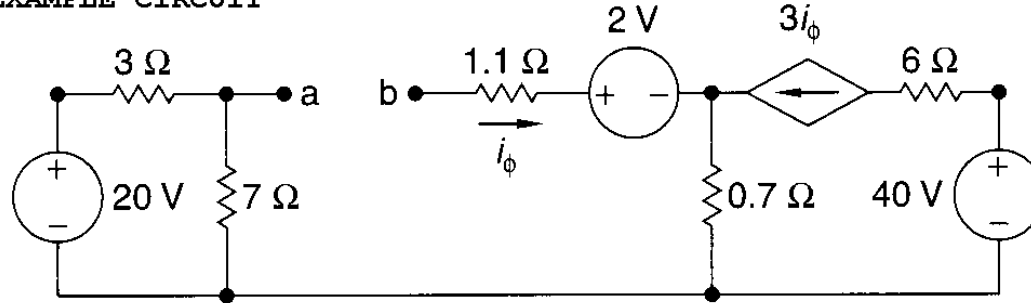
# Simulation



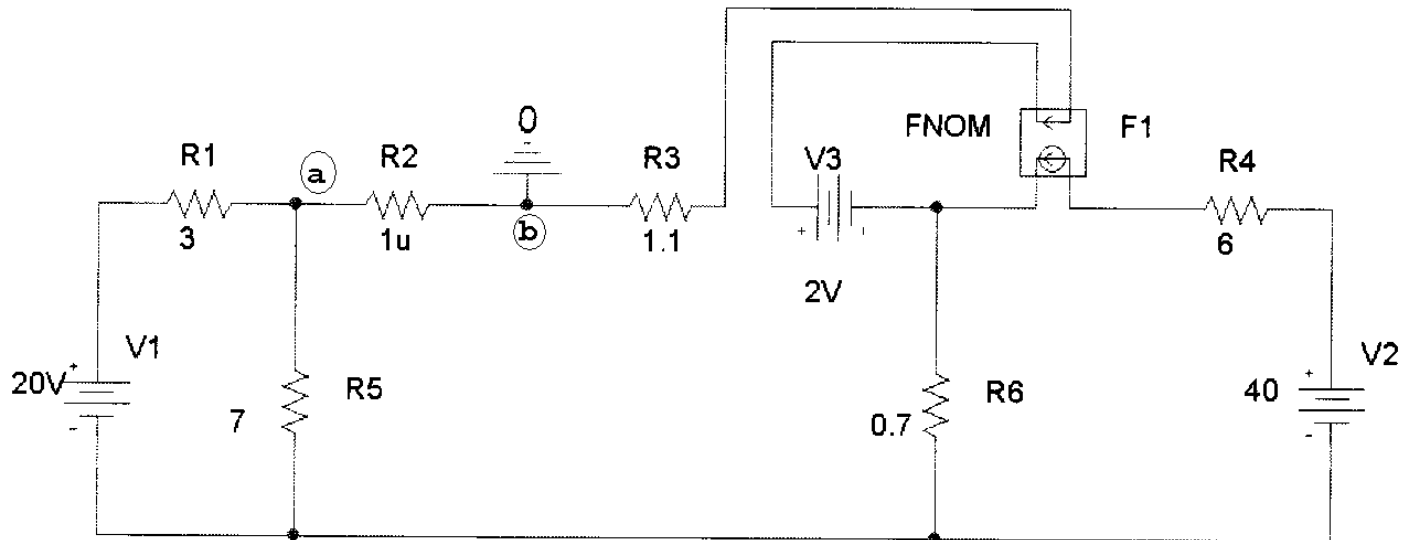
Show Your Simulation Results

# Thevenin Equivalent Circuit by PSpICE

EXAMPLE CIRCUIT

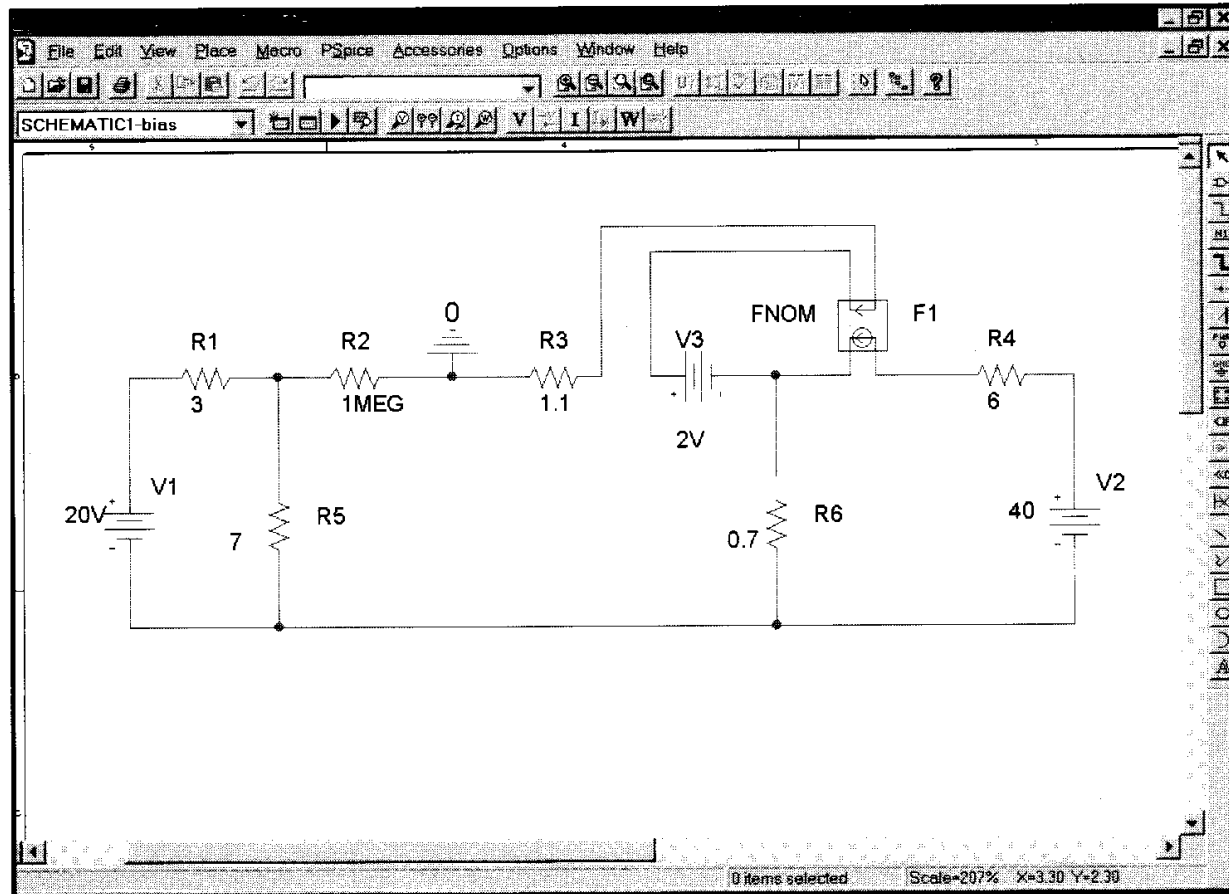


- Short Circuit Current ---very small resistance path
- Get Current ( $I_{sc}$ )



# Thevenin - continued

- Open Circuit Voltage ---very big resistance path
- Get Voltage ( $V_{th}$ )



# Trouble Shooting

- Always remember to ground your circuit with the part **GND\_EARTH**.
- If an error window pops up when you try to simulate, you need to identify which pin is “floating” i.e. not connected properly to the circuit and link it up with a wire.