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

Dr. Charles Kim

Howard University

WWW.MWFTR.COM

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

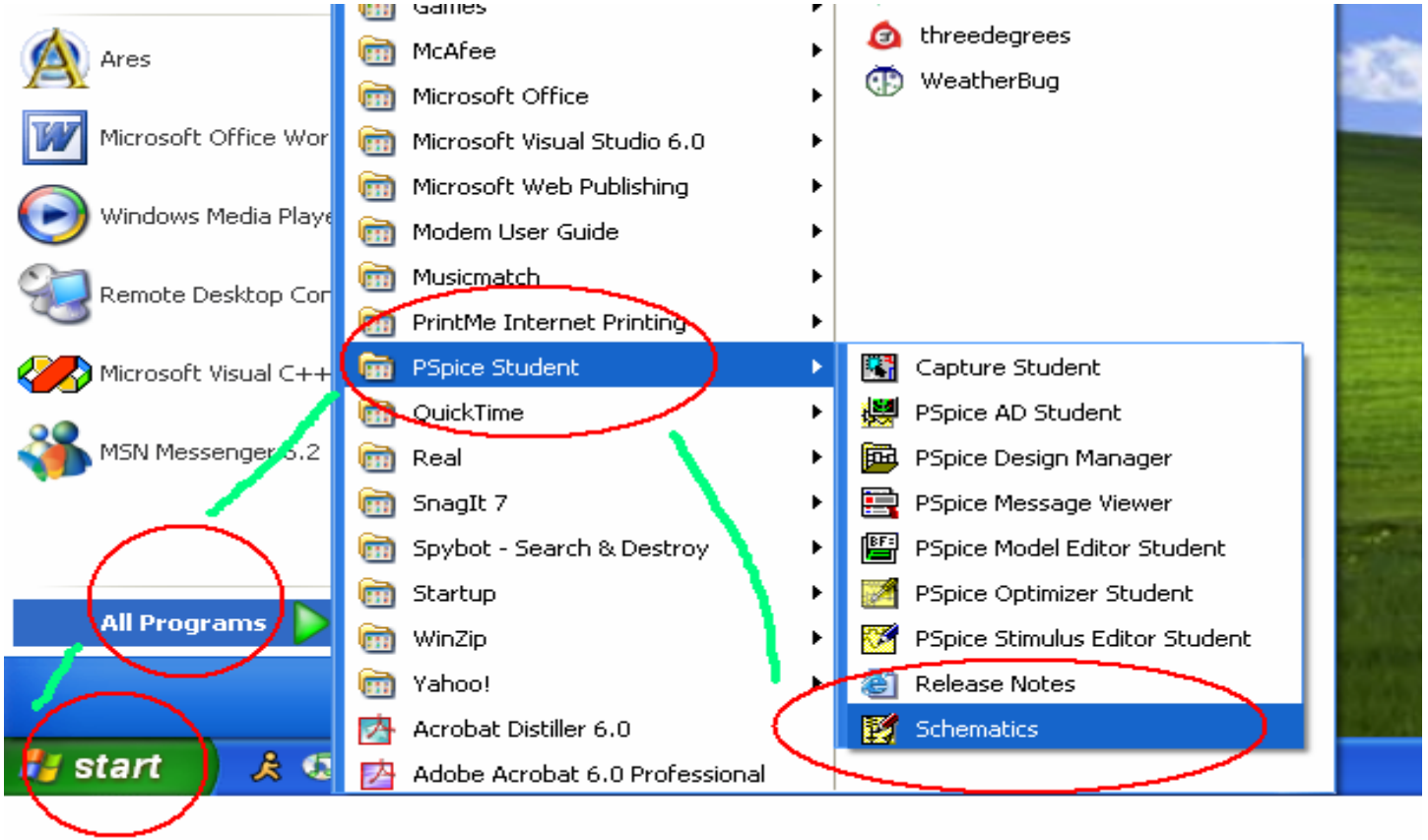
- Non-linear DC analysis: calculates the DC transfer curve.
- Non-linear transient analysis: calculates the voltage and current as a function of time when a large signal is applied.
- Linear AC Analysis: calculates the output as a function of frequency. A bode plot is generated.
- Noise analysis
- Sensitivity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

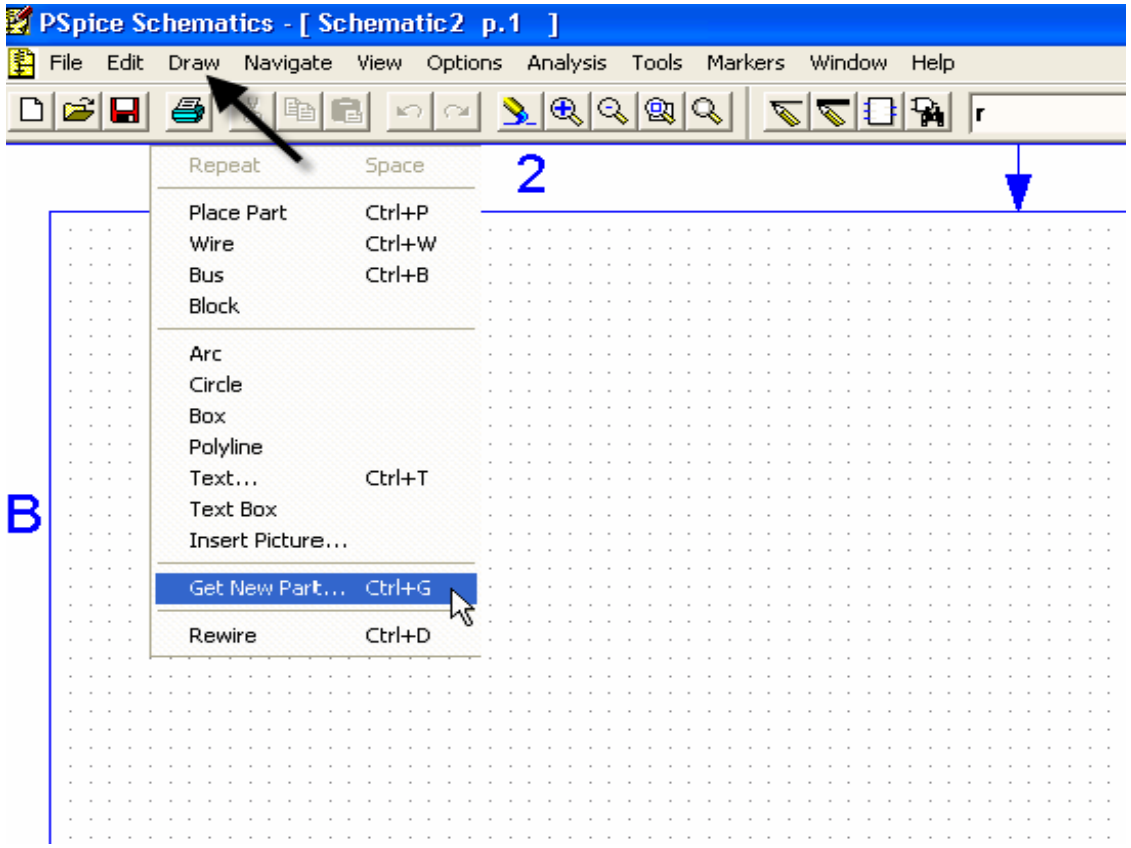
Getting Started

- Logon to a Tablet or Desktop near you.
- Open up the “Pspice Schematics” application



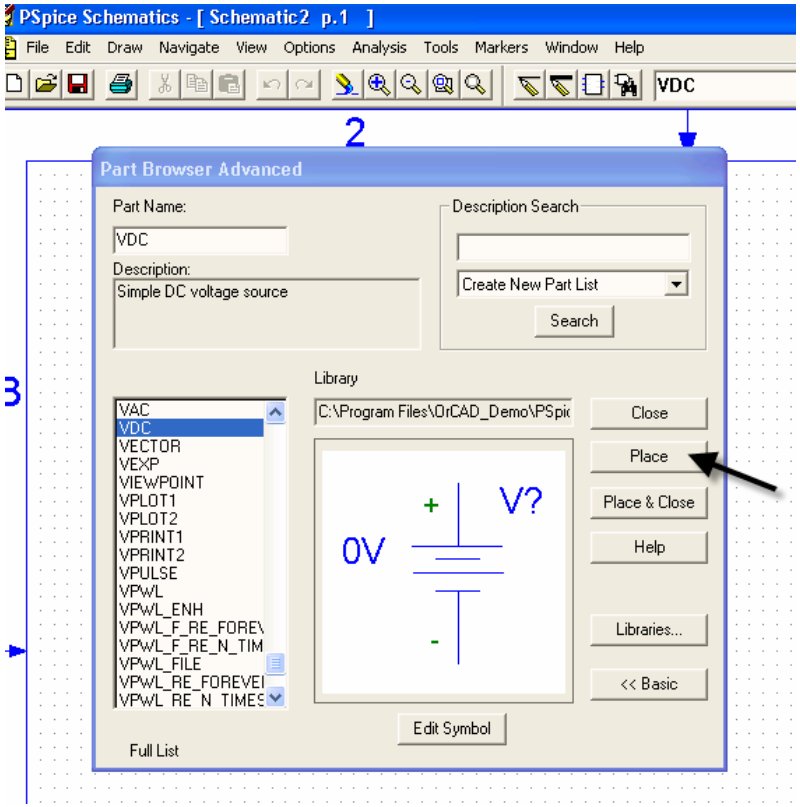
Part Placement

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



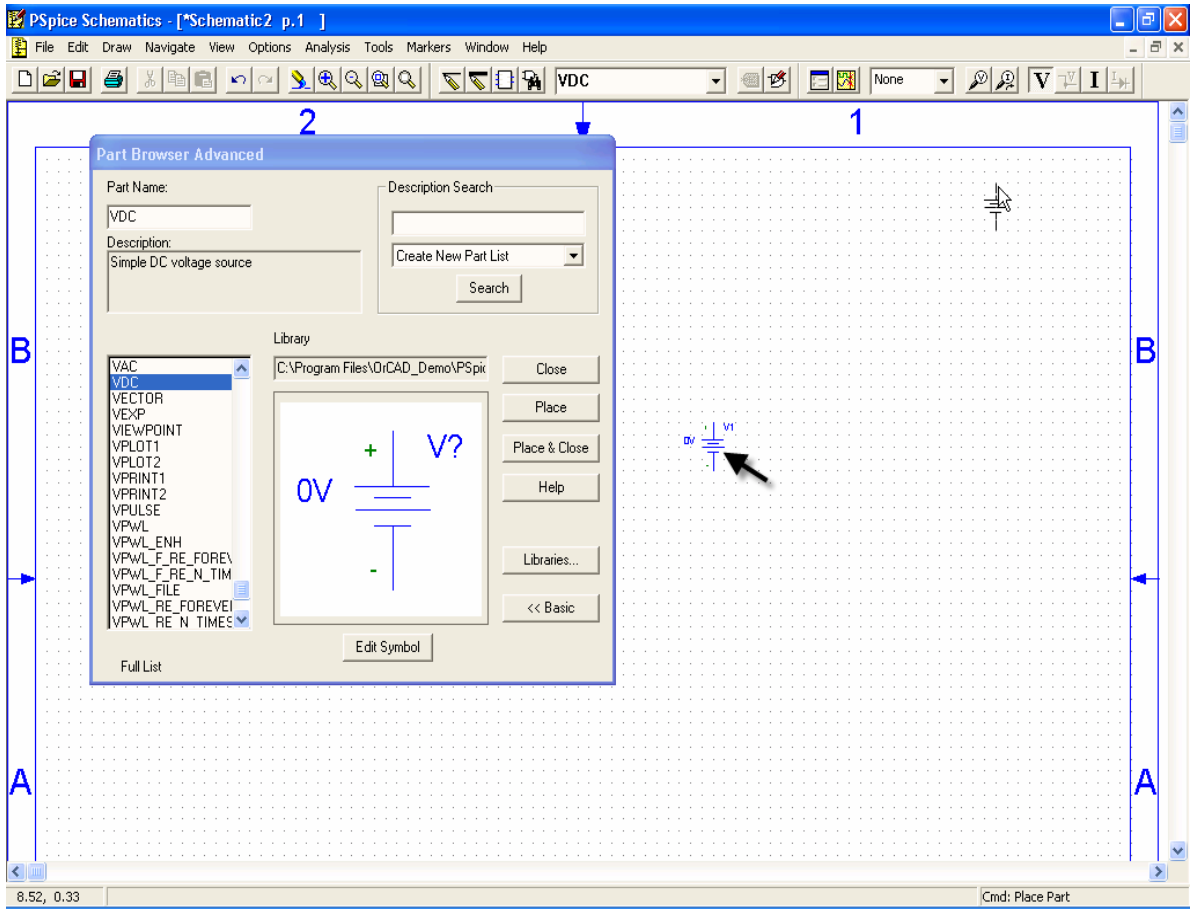
Independent dc Voltage Source

- 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.



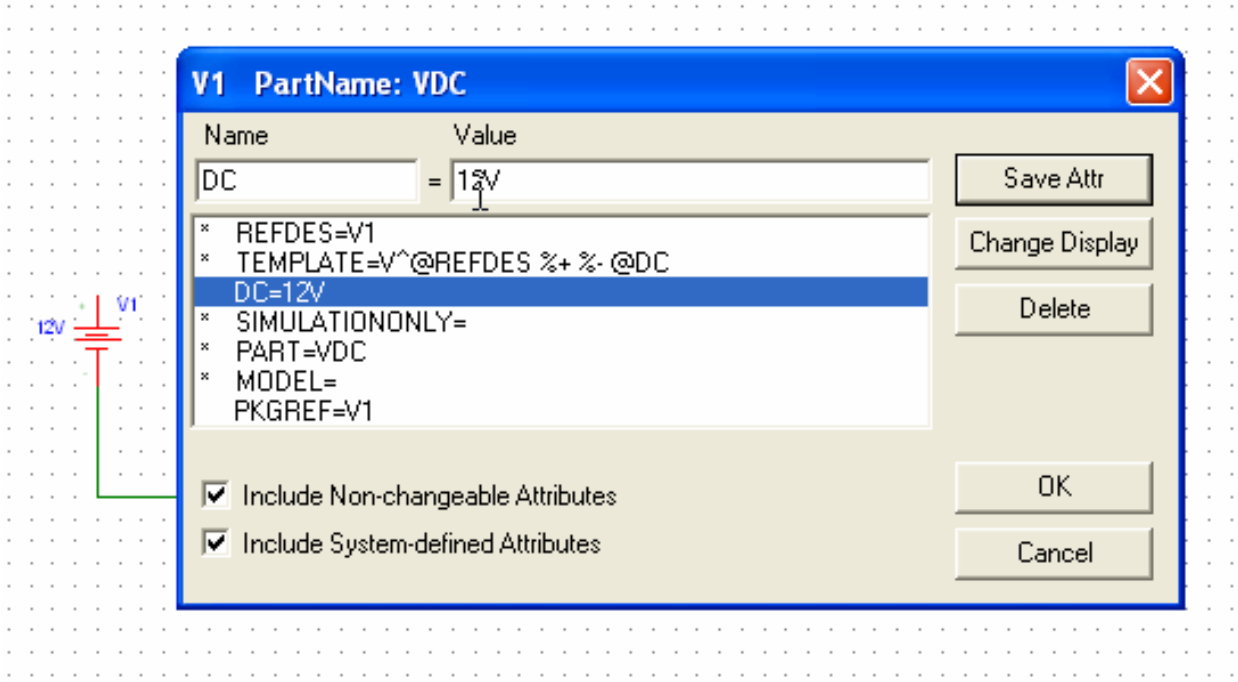
Placing the dc Voltage Source

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



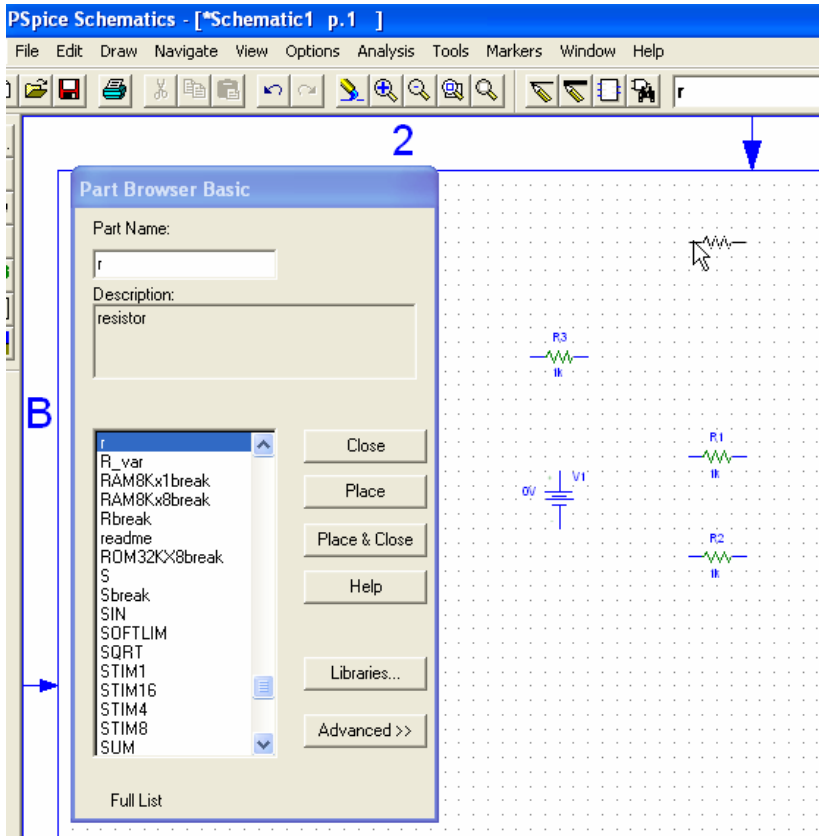
Voltage Value Setting

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



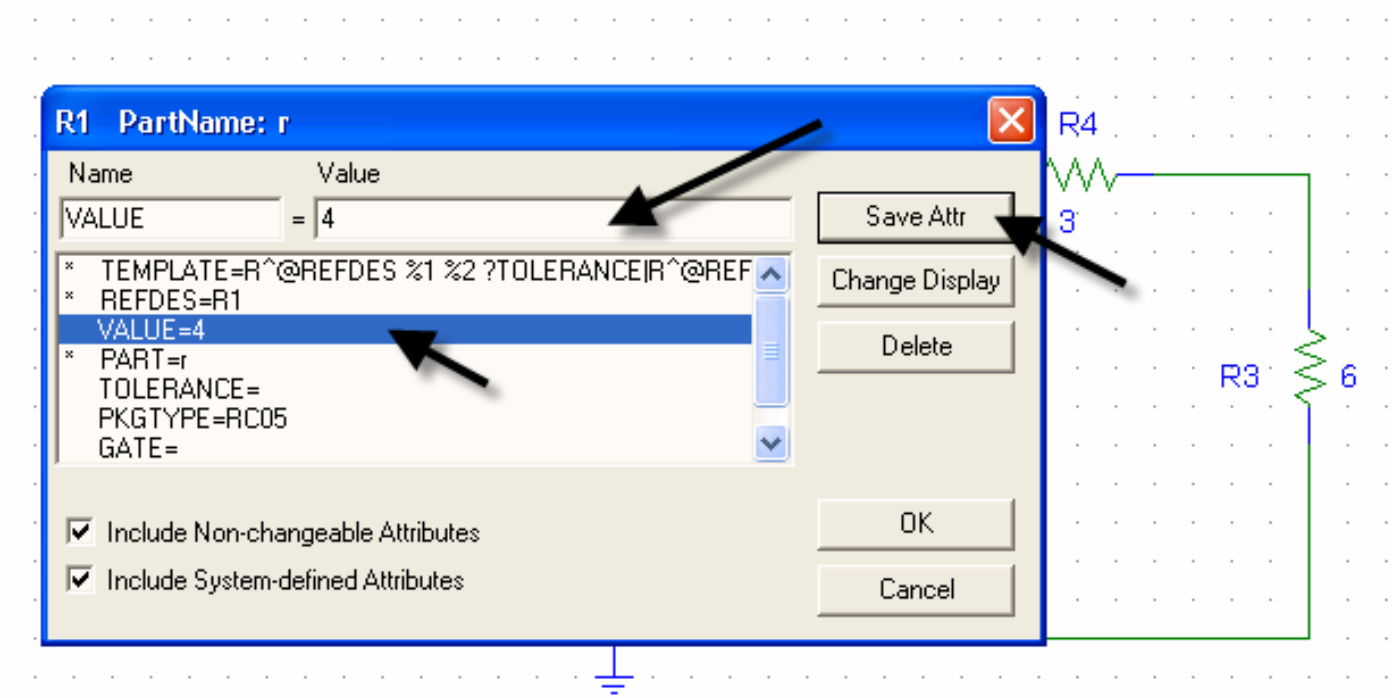
Resistor Placement

- Go back to the Part Draw 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.
- **Rotation of elements**:
  - Select the element
  - CTRL_R turns the element



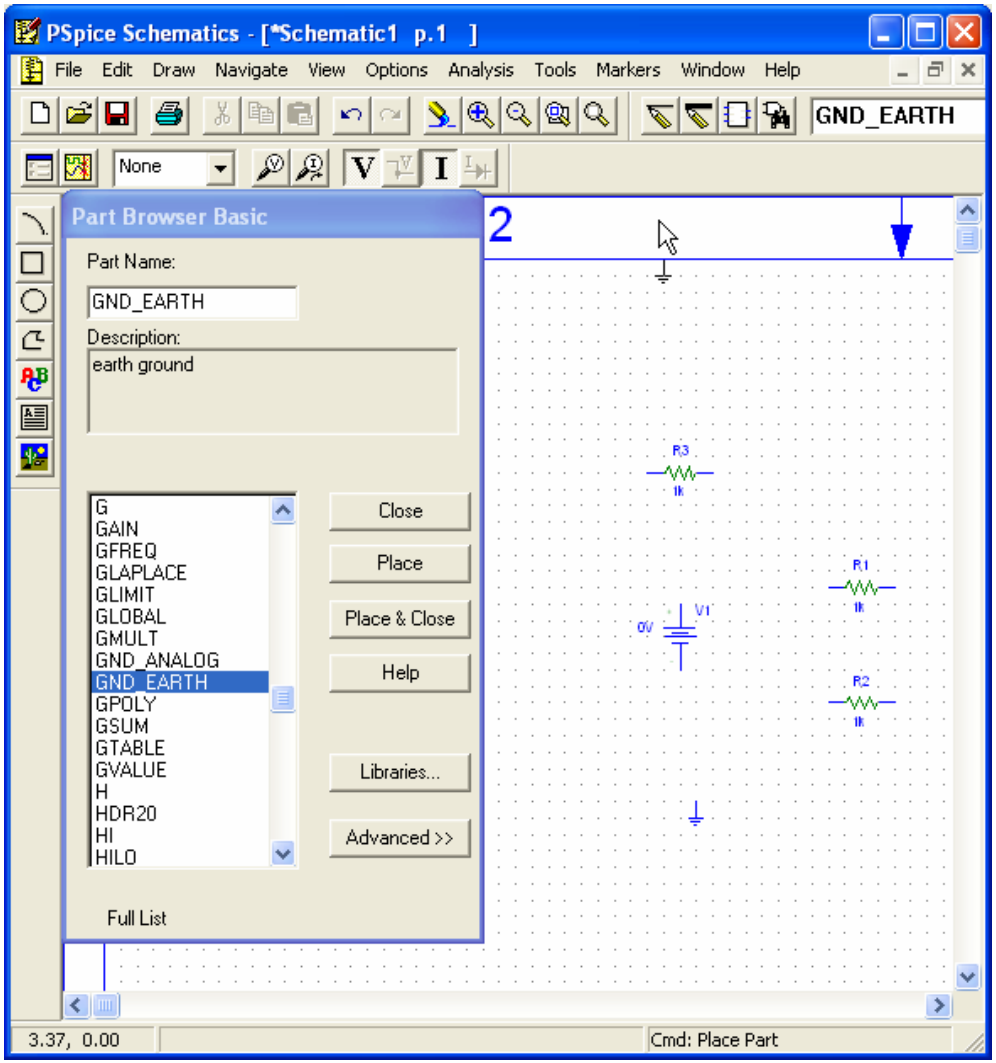
Resistance Value Setting

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



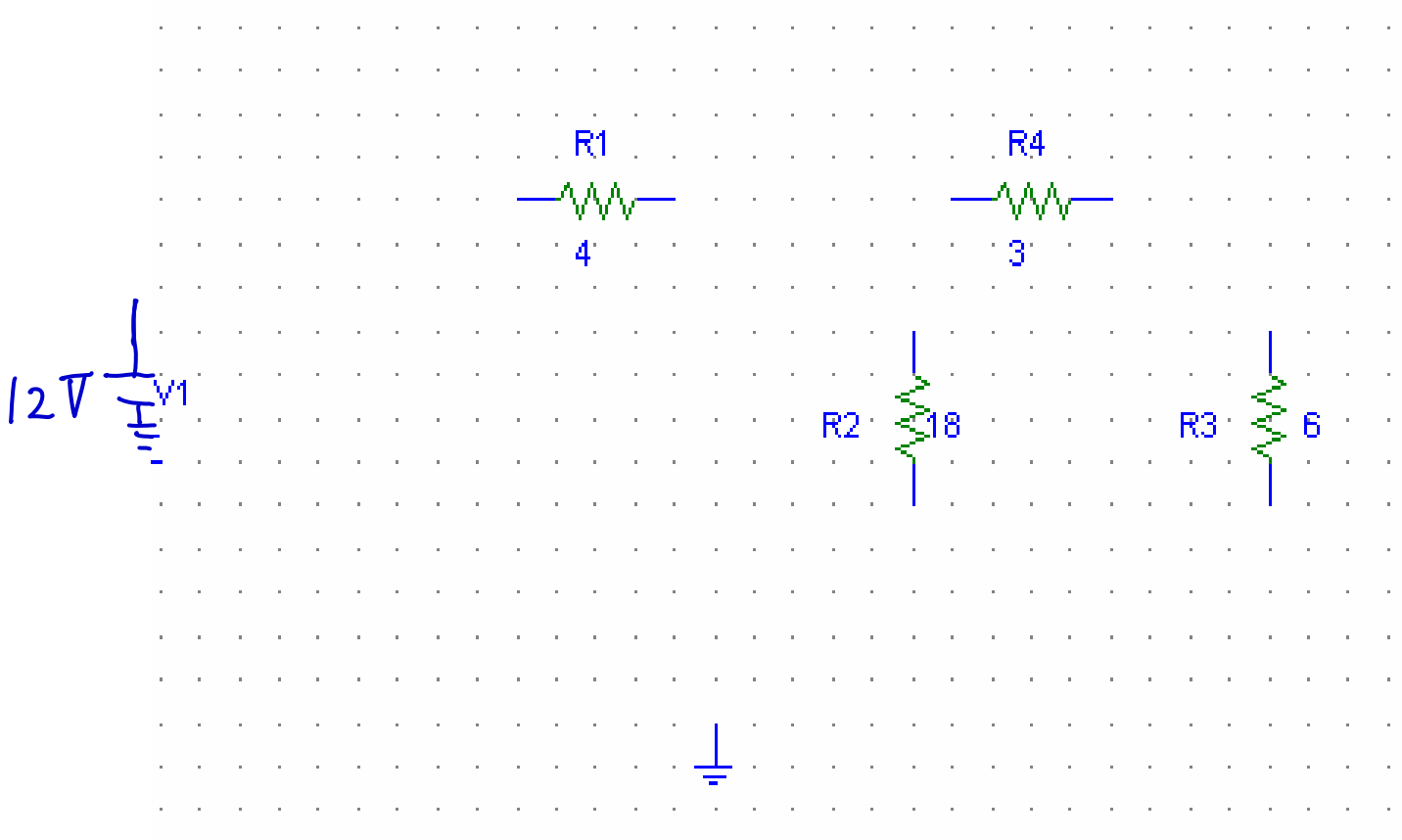
GND placement

- Following the same procedure, type GND_EARTH and place the part on the background. This for the common ground.
- Close the Part window.
- GND_EARTH must be connected to a circuit; otherwise, error would occur.



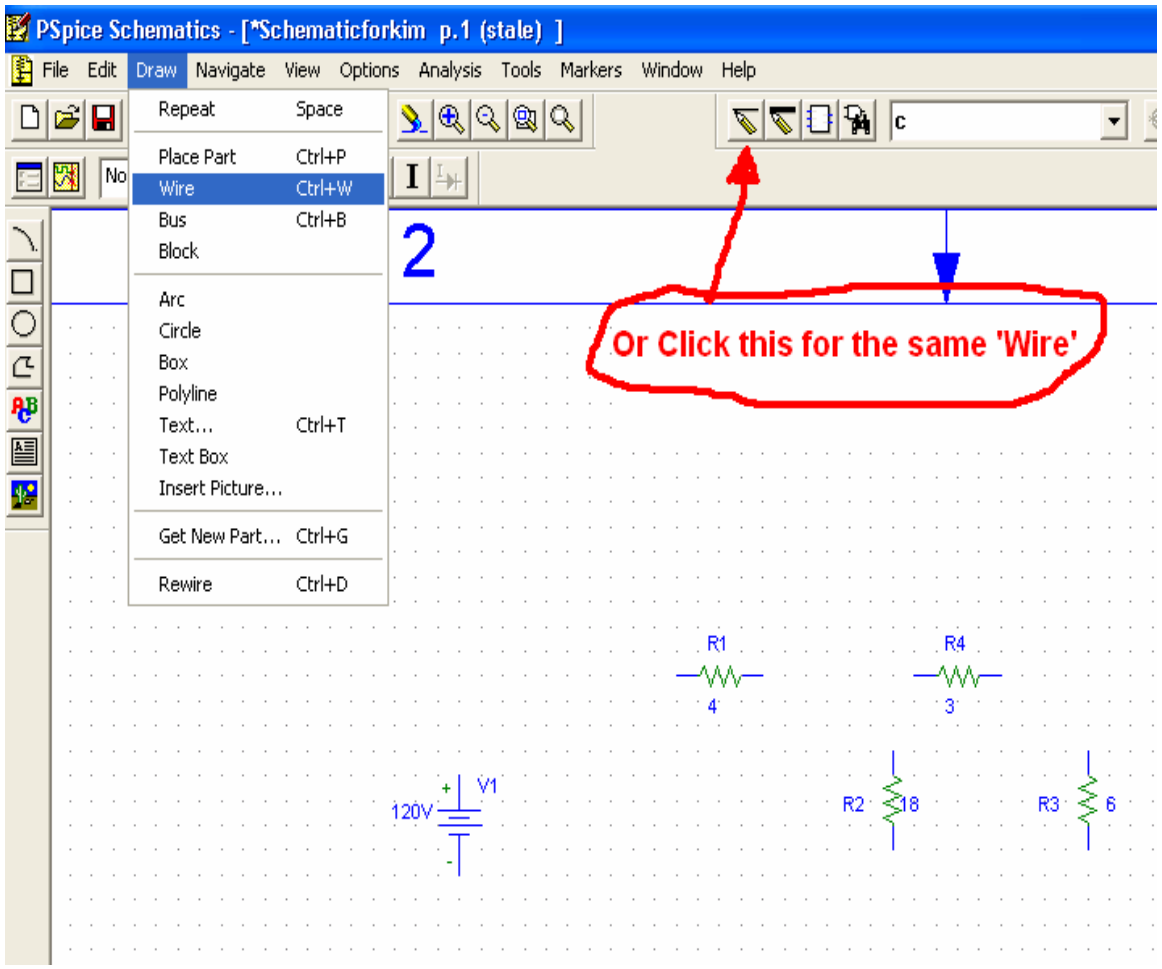
Placement of all the Parts

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



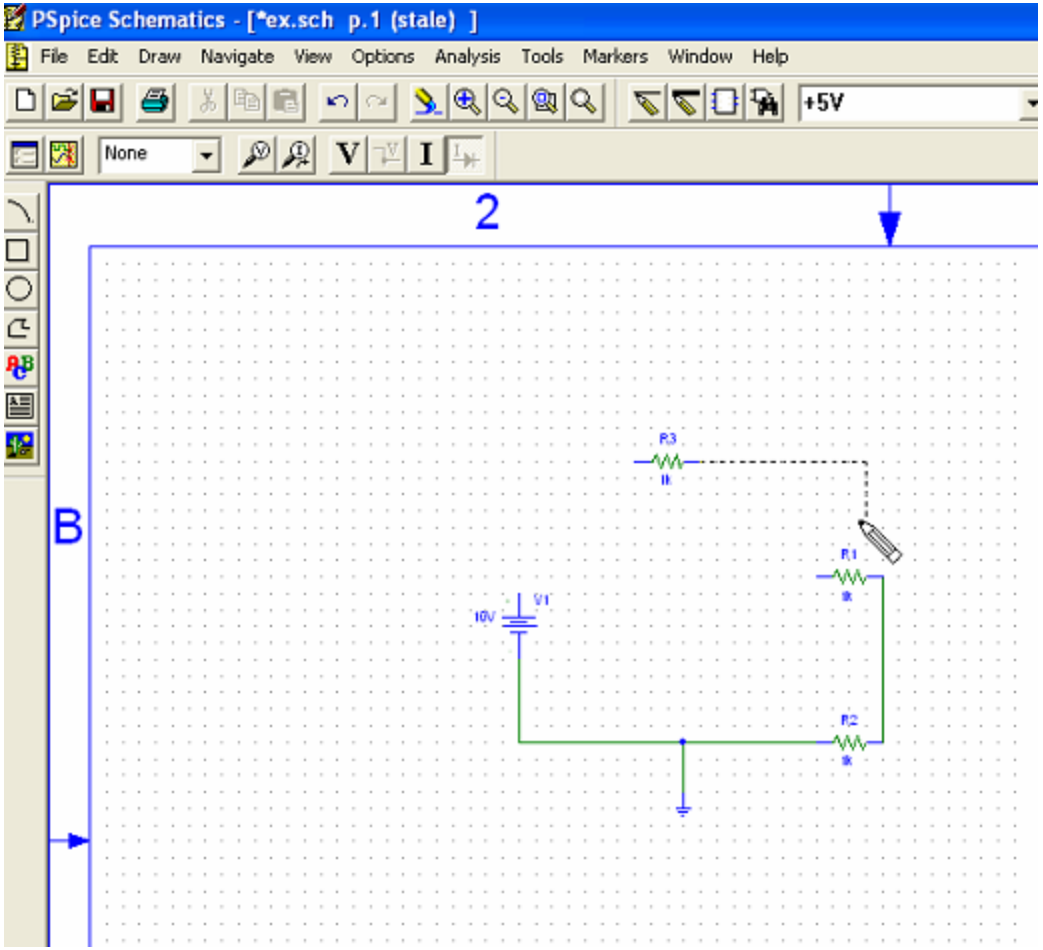
Wring the Parts

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

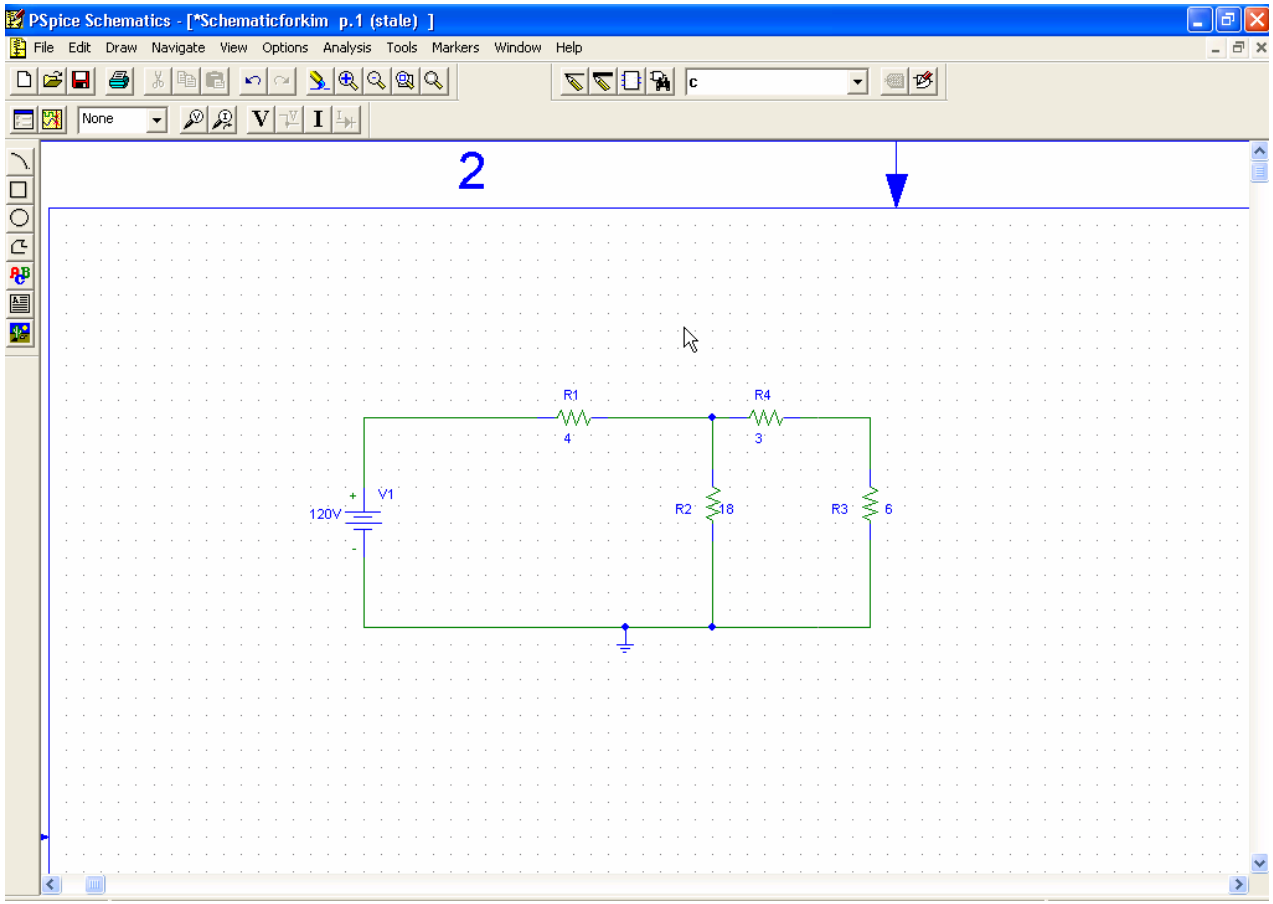


Wiring

- 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.



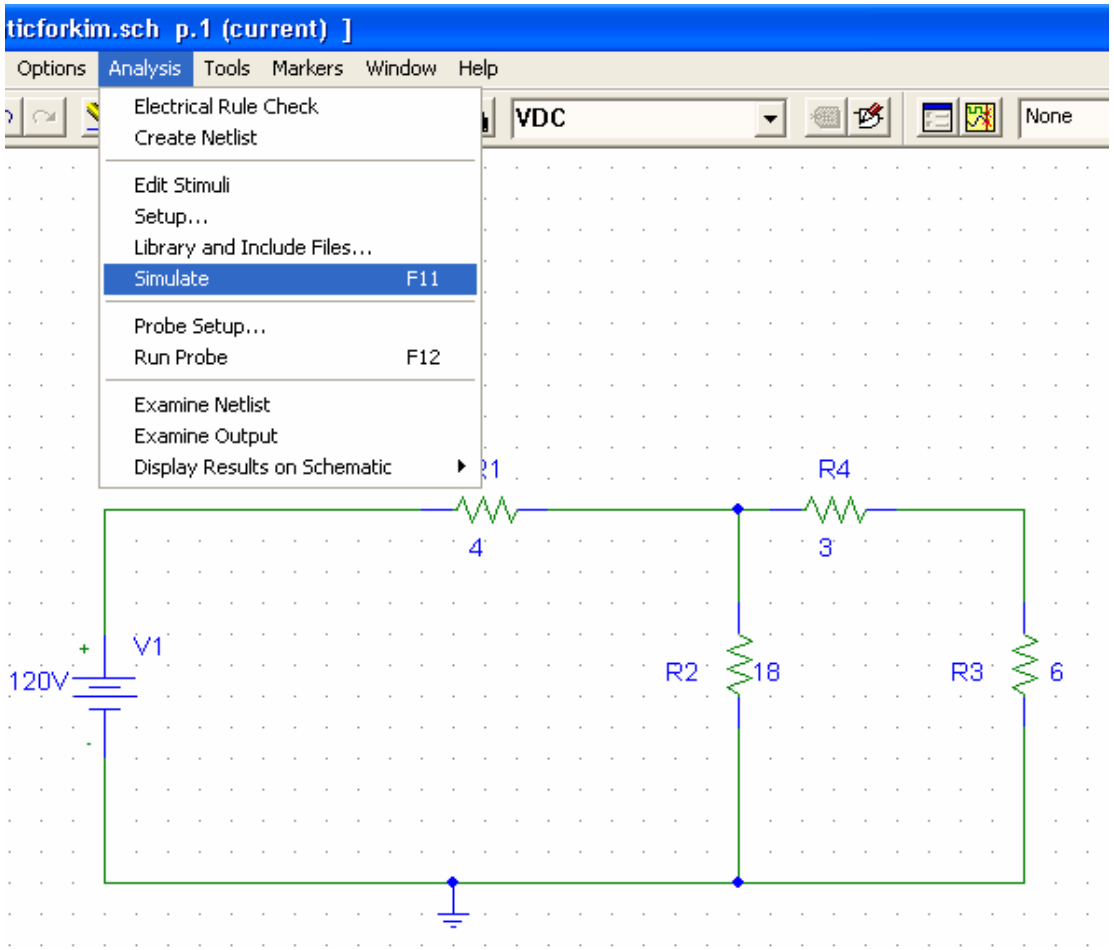
Completed Circuit



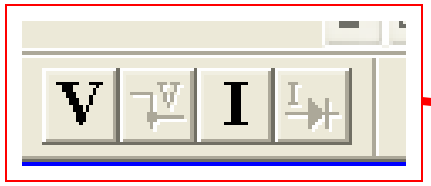
- **Then, Save your schematic**

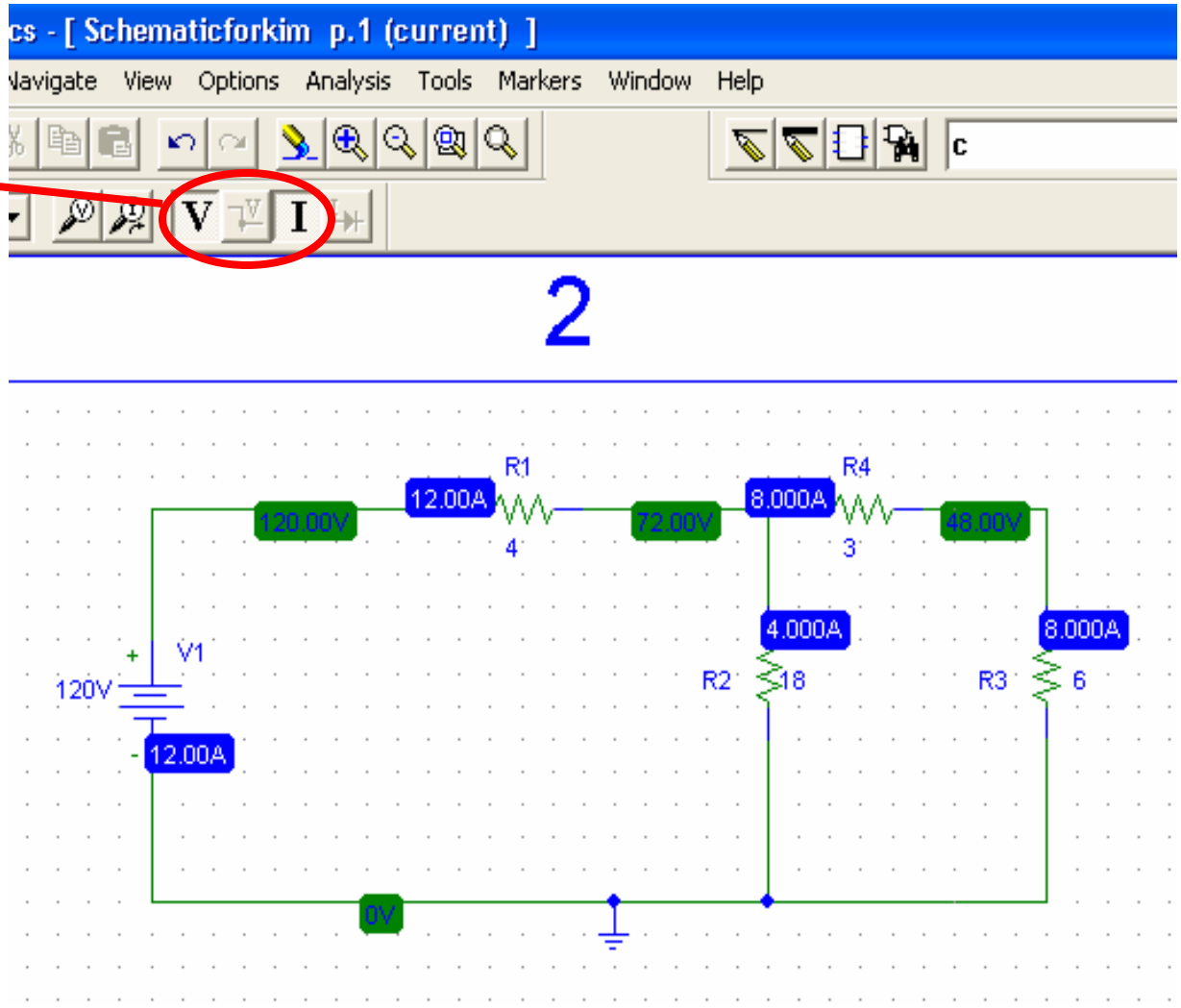
Circuit Simulation – Analysis

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



Result!

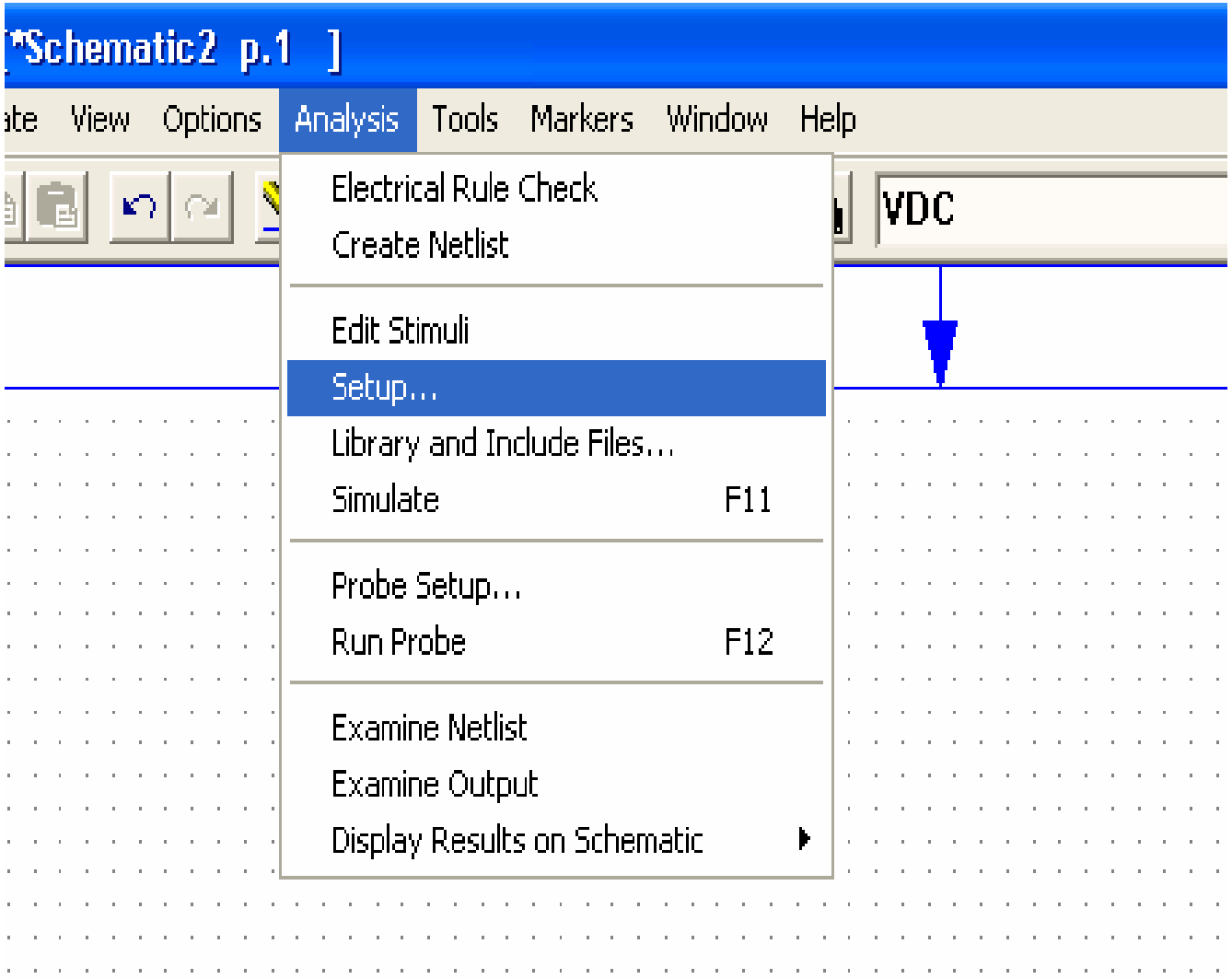




- Toggle these buttons, and see what happens

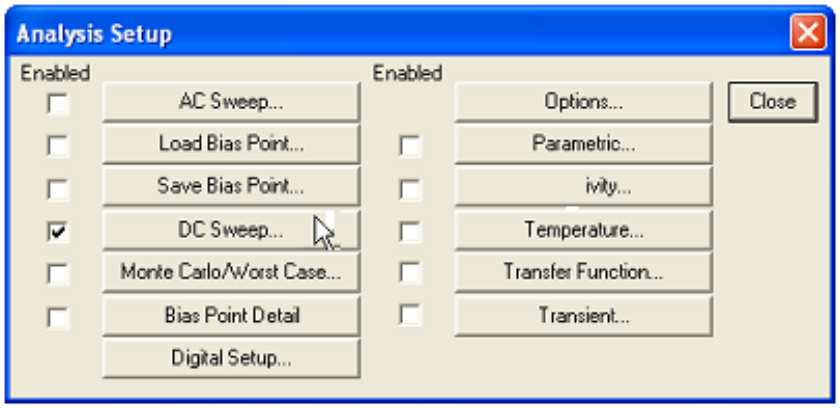
Graphical Display of the Result

- Menu Bar
- >Display
- >Setup

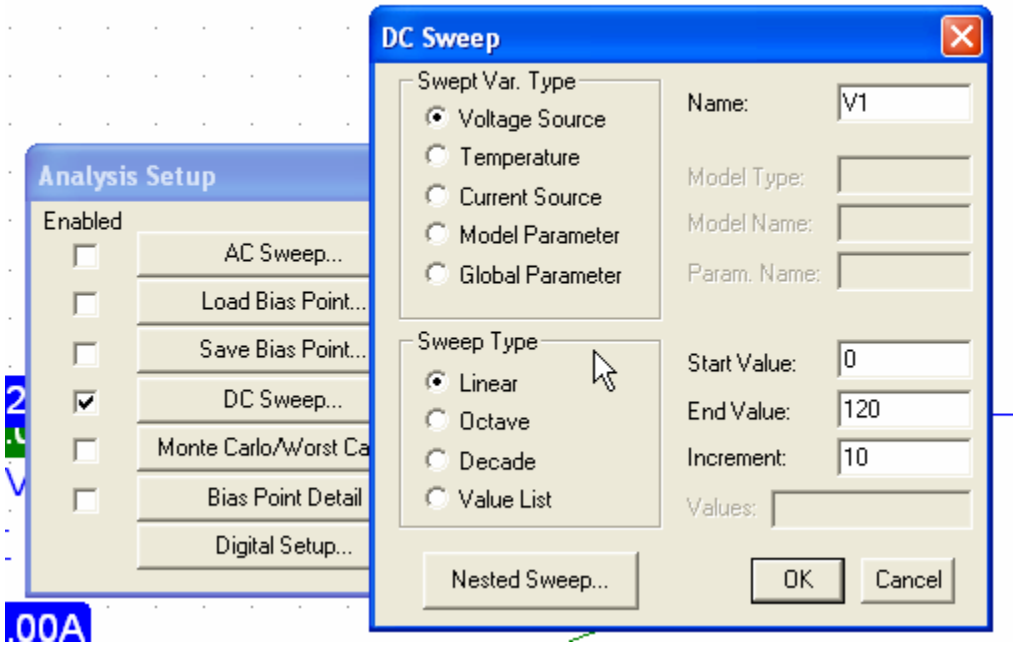


Analysis Setup Dialog Window

- Analysis Setup Window
- Click DC Sweep Bar

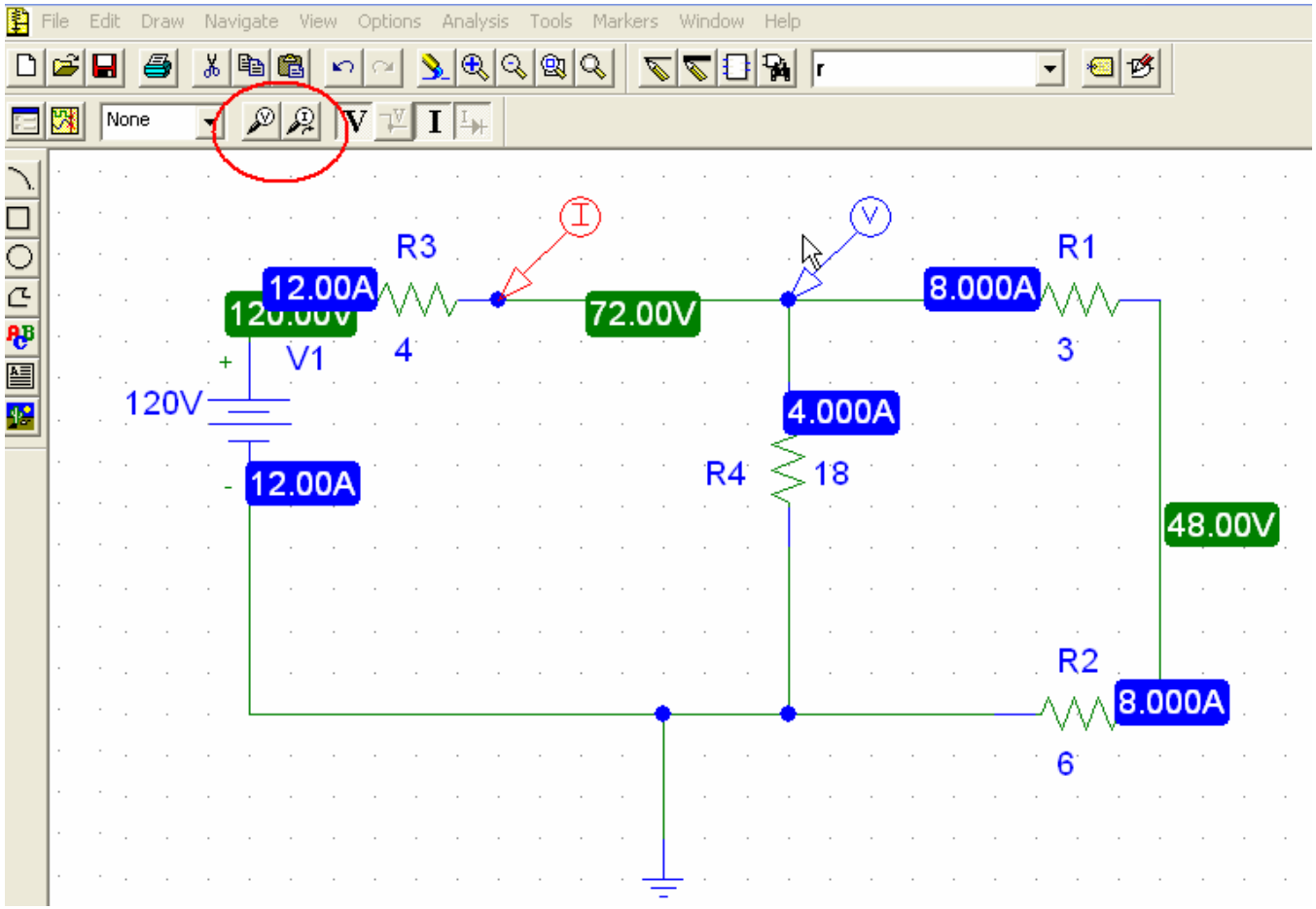


- DC SWEEP Window
  - Voltage Source
  - Sweeping
    - From 0
    - To 120
    - Increment of 10



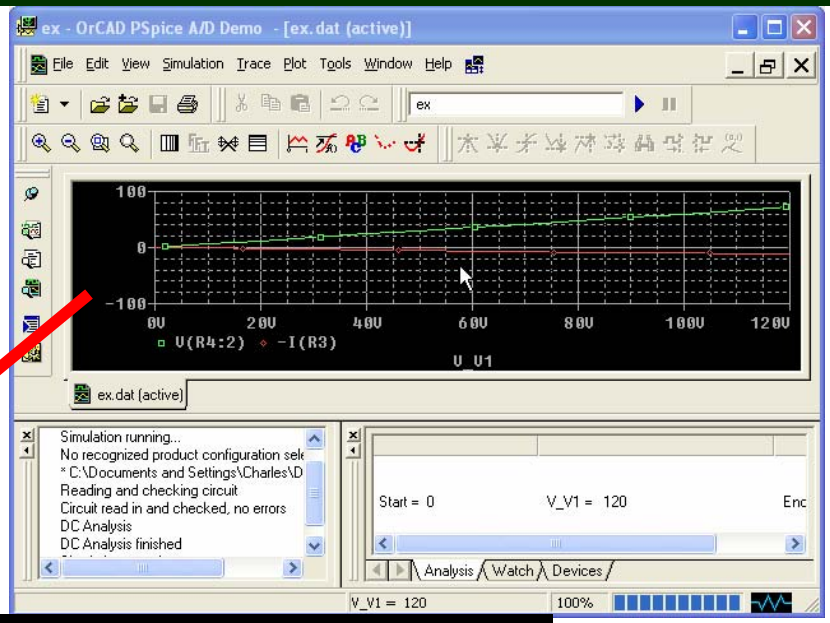
Measurement Probes

- Voltage Probe
- Current Probe

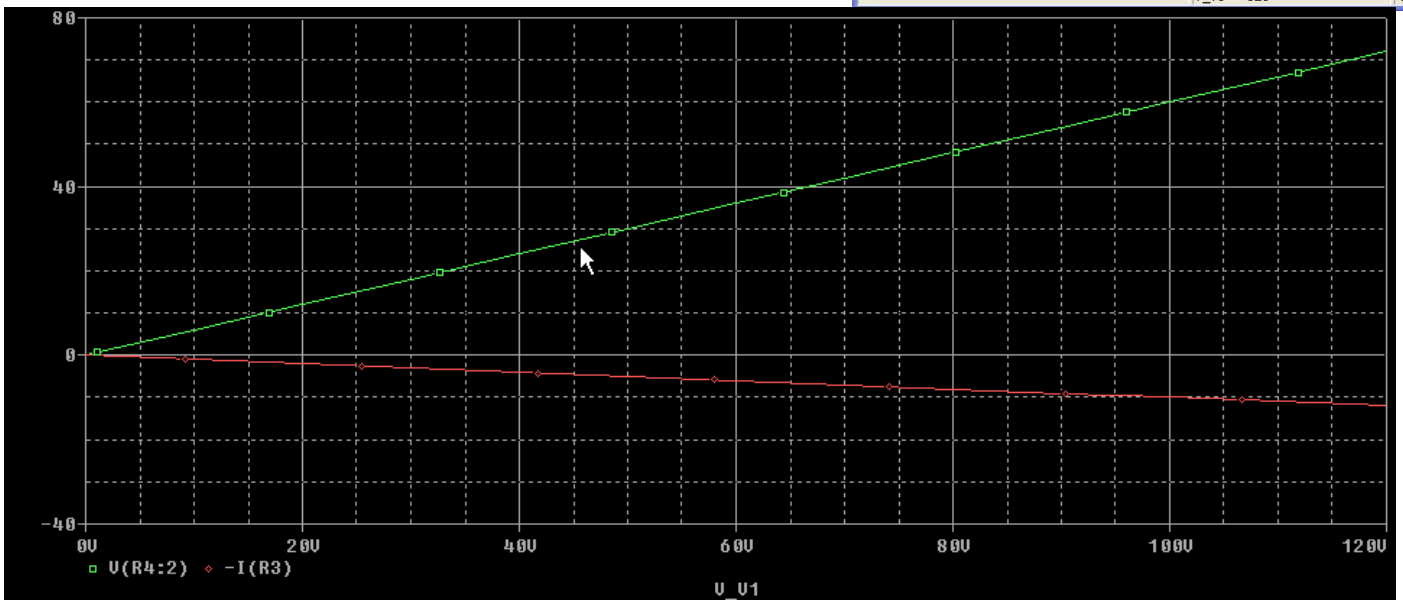


Simulation Result

- Menu Bar
- >Analysis
- >Simulate

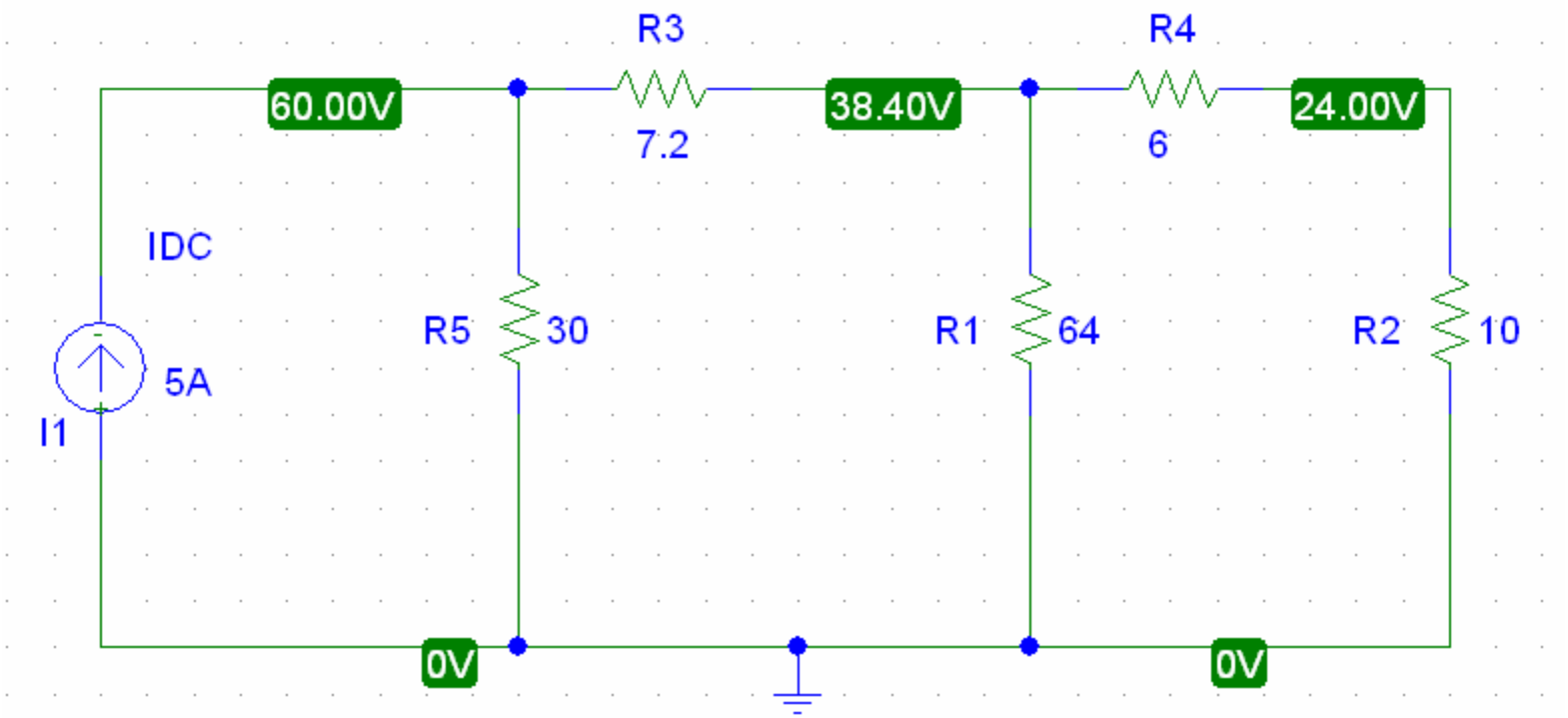


Detail

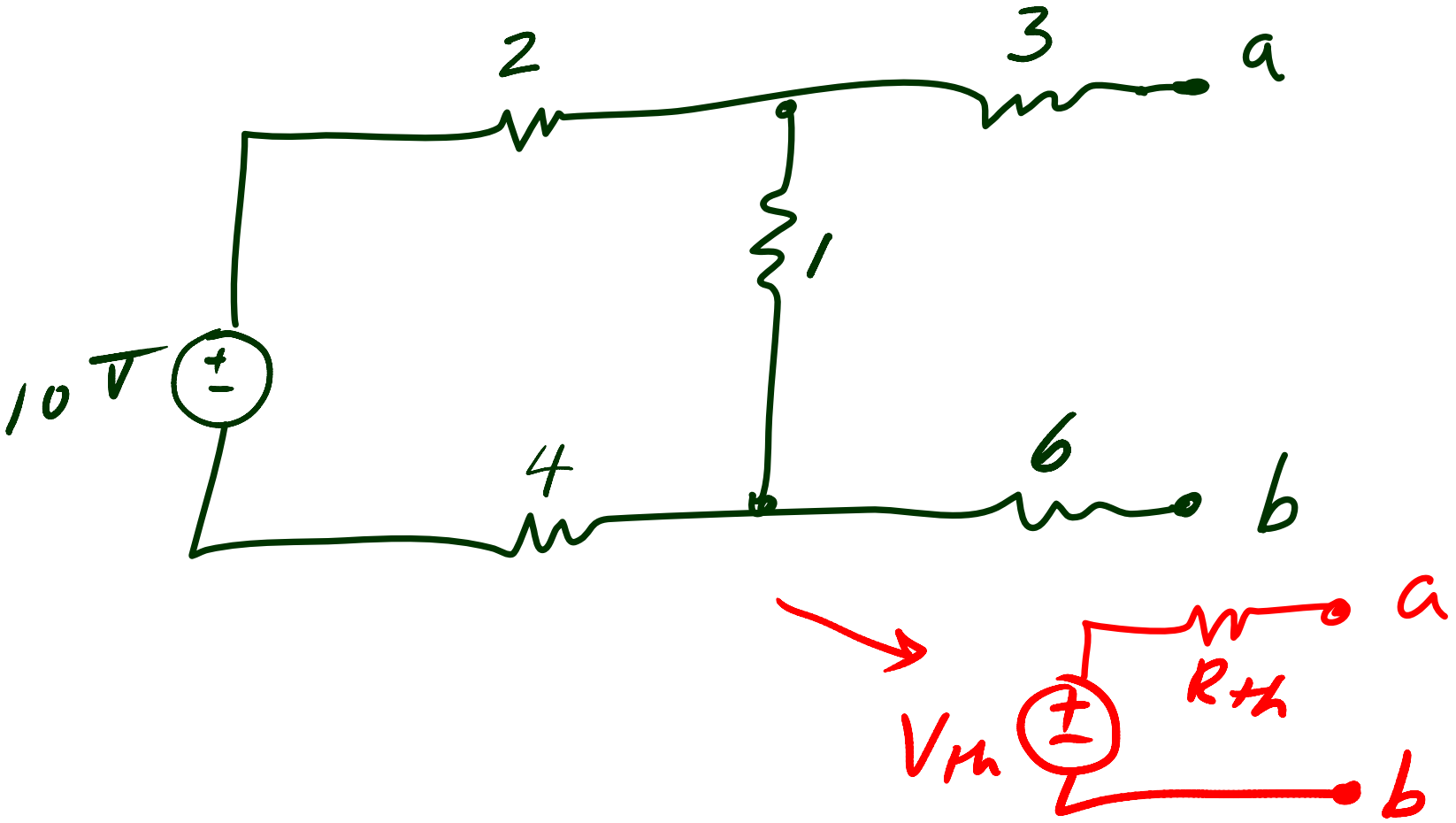


Practice Problem

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

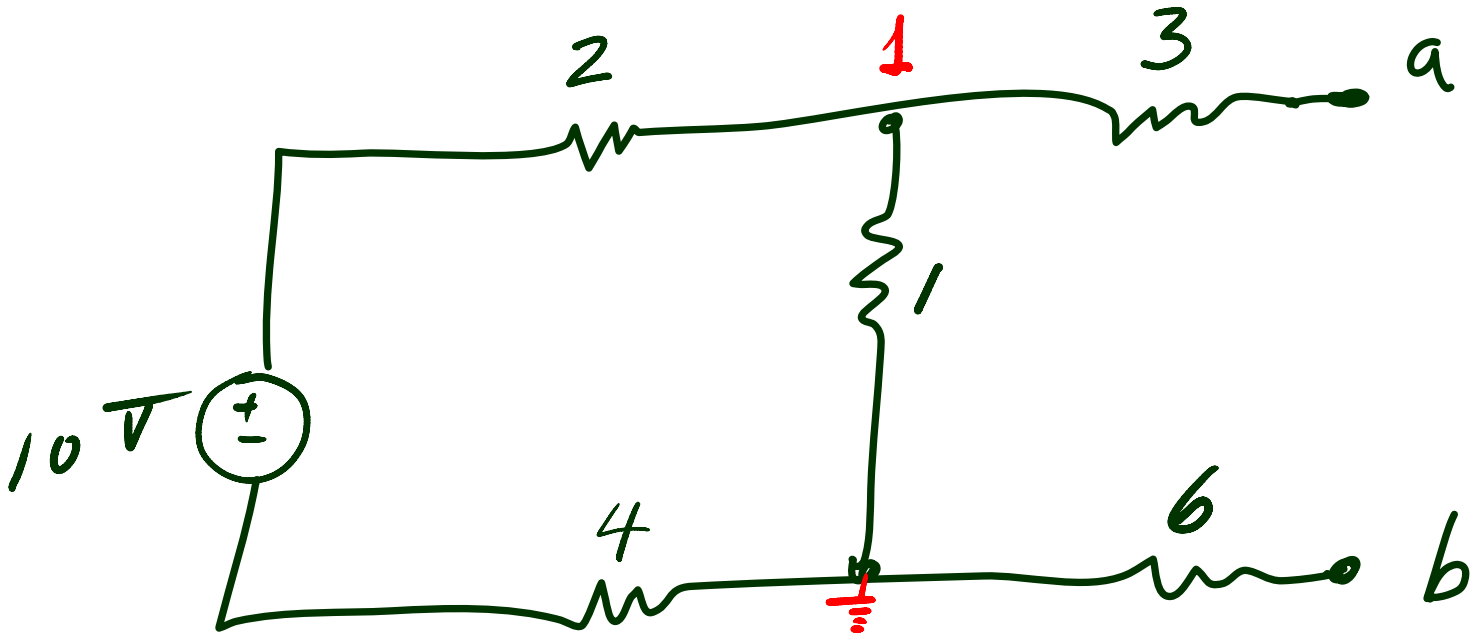


Solving for Thevenin Equivalent Circuit



Solving for Thevenin Equivalent Circuit

By Calculation:

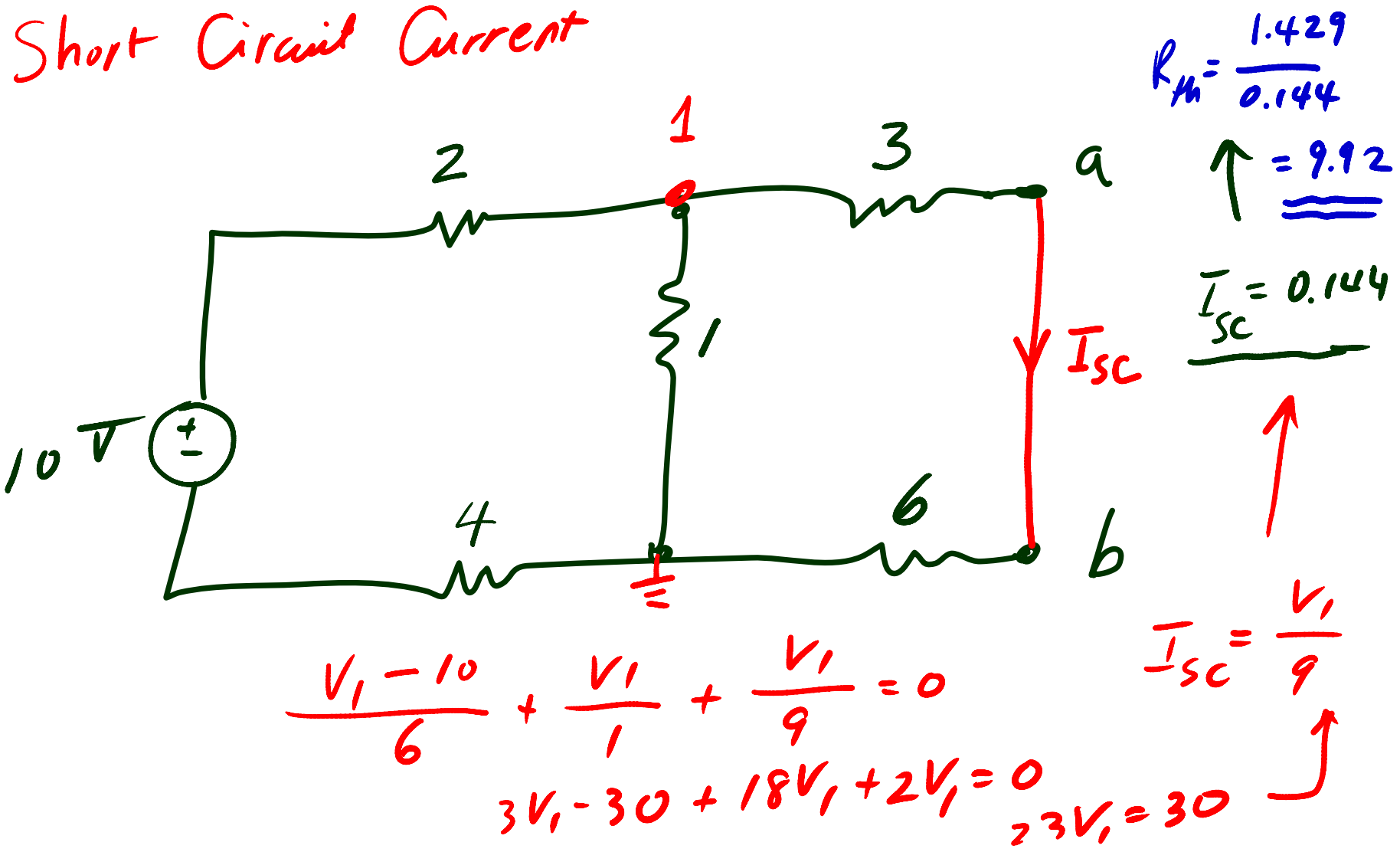


Open Circuit Voltage: $V_{oc} = V_{th} = V_{ab}$

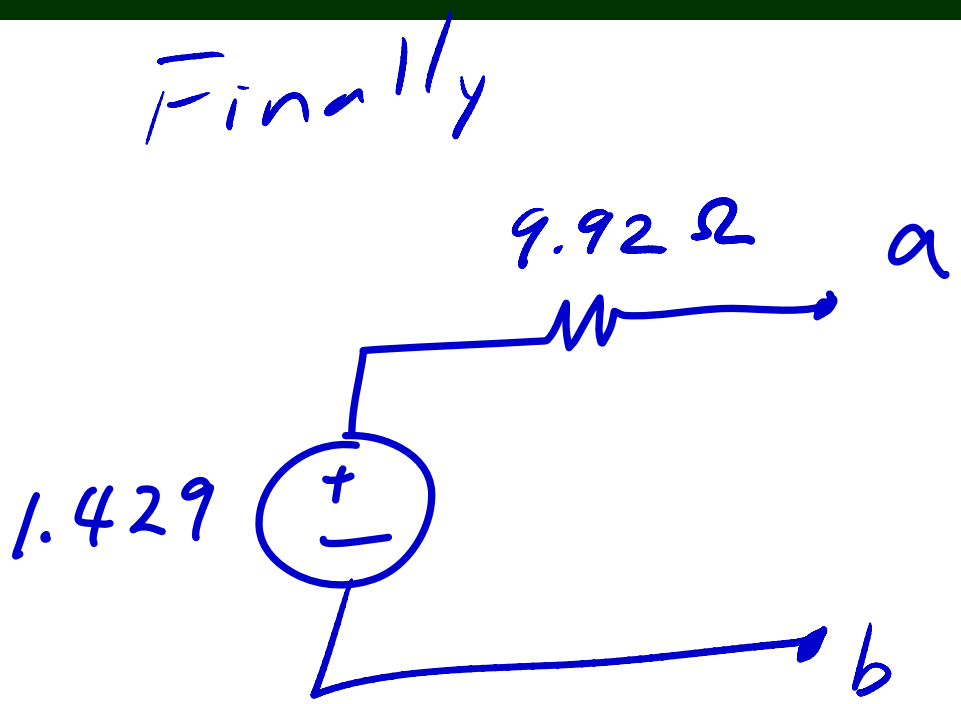
$V_1 = V_{ab}$, $V_1 = 10 \cdot \frac{1}{2+1+4} = 1.429\,[V]$

$= \underline{\underline{V_{th}}}$

Solving for Thevenin Equivalent Circuit



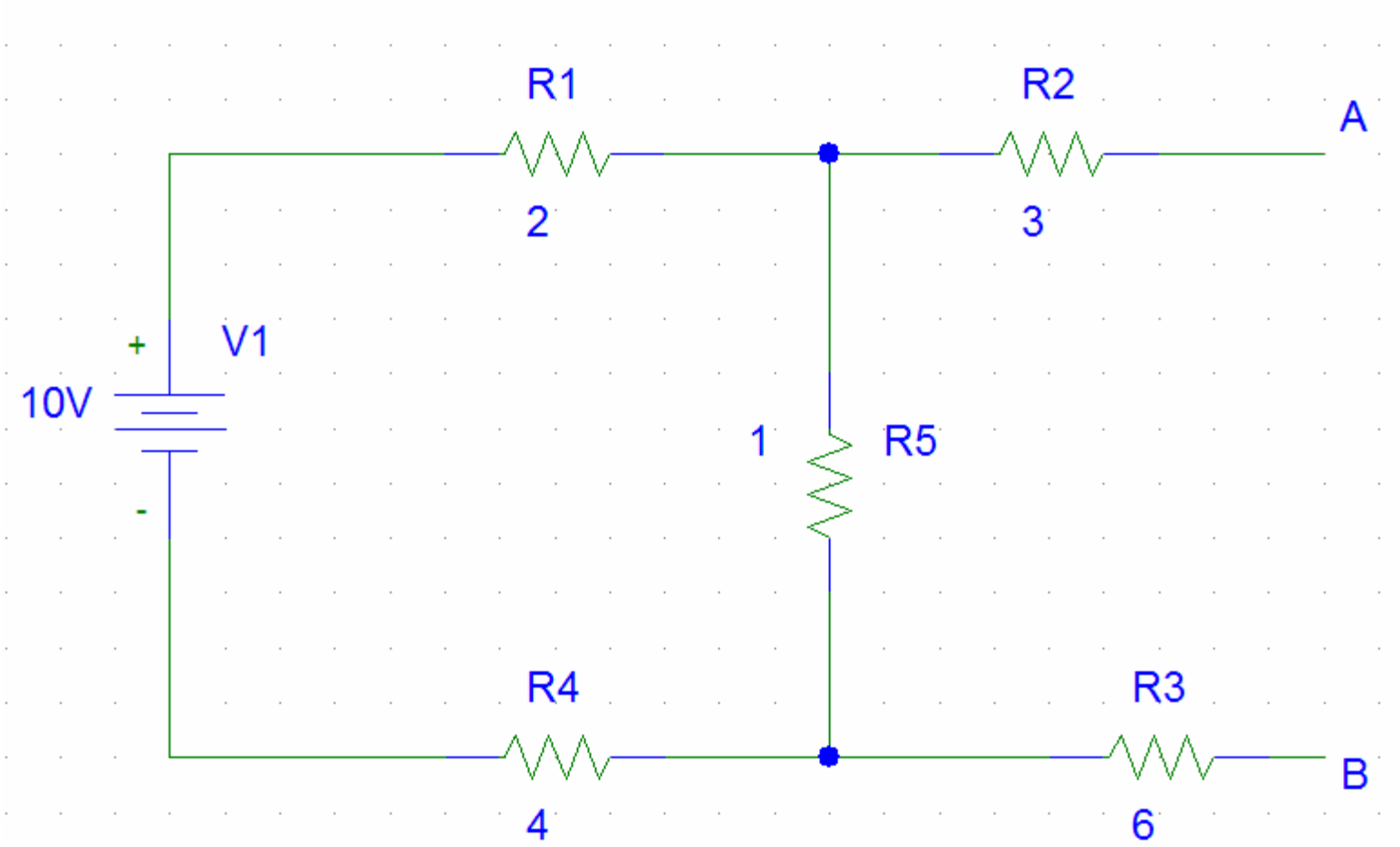
Final Thevenin Circuit



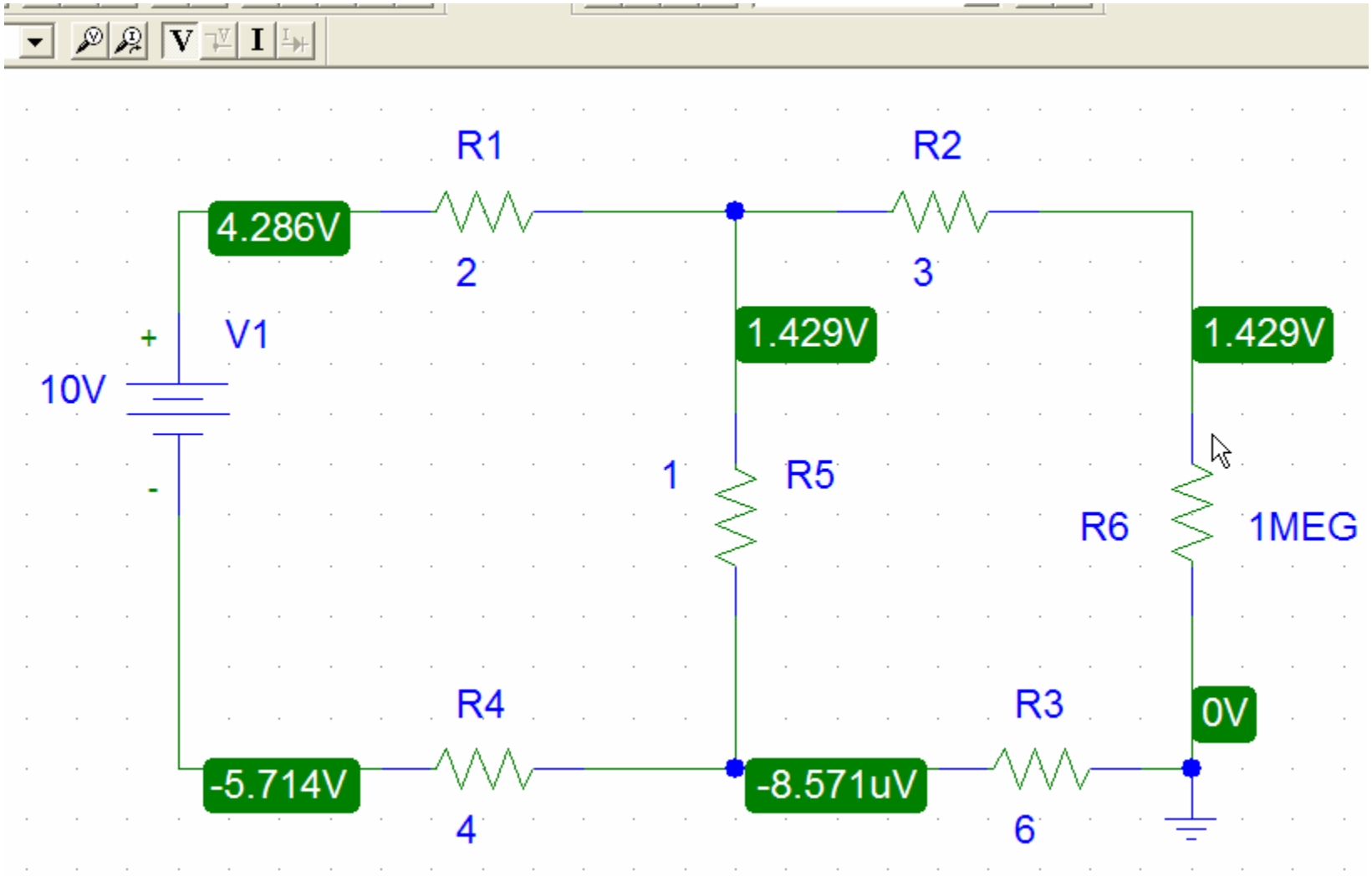
Thevenin Circuit using PSpice

- Find Vth by Open Circuit Voltage
- Apply “Short Circuit Method” for Rth
- Always GND at one of the terminals
- PSpice Tips for Thevenin Circuit
  - Open Circuit
    - Insert a resistor of big value, like 1 MEG
    - Find the node voltage at the terminal → Vth
  - Short Circuit
    - Insert a resistor of very little, like 1 u [micro]
    - Find the current through the little resistor
    - Vth / Isc → Rth

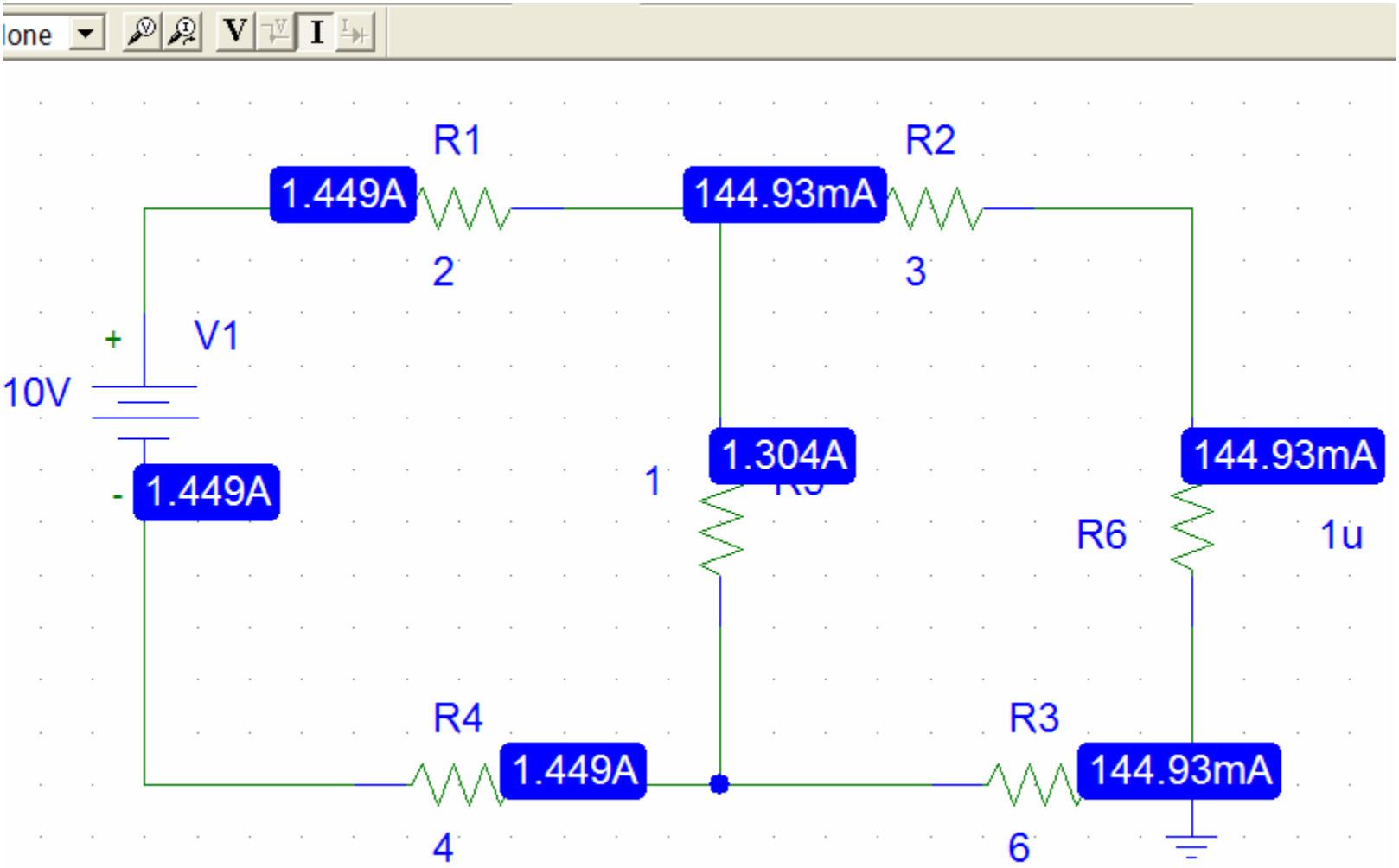
Thevenin Circuit Practice



Open Circuit Voltage

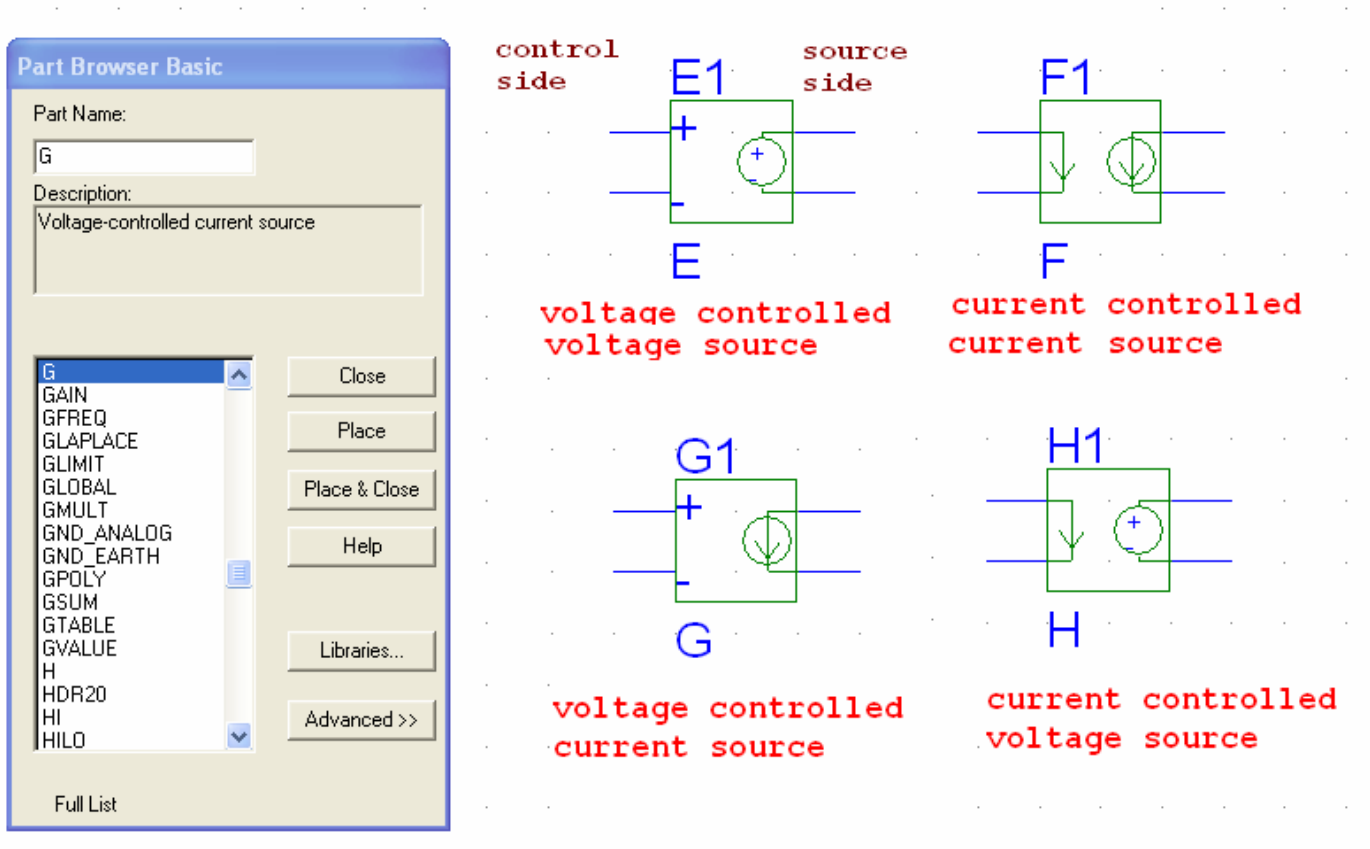


Short Circuit Current



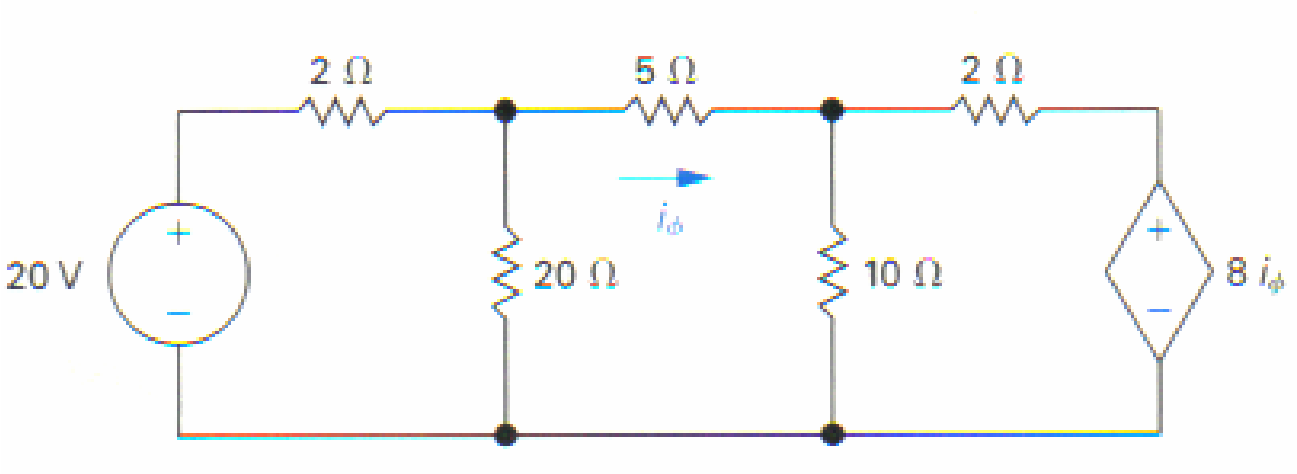
Simulation of a simple circuit containing a Dependent source:

- 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



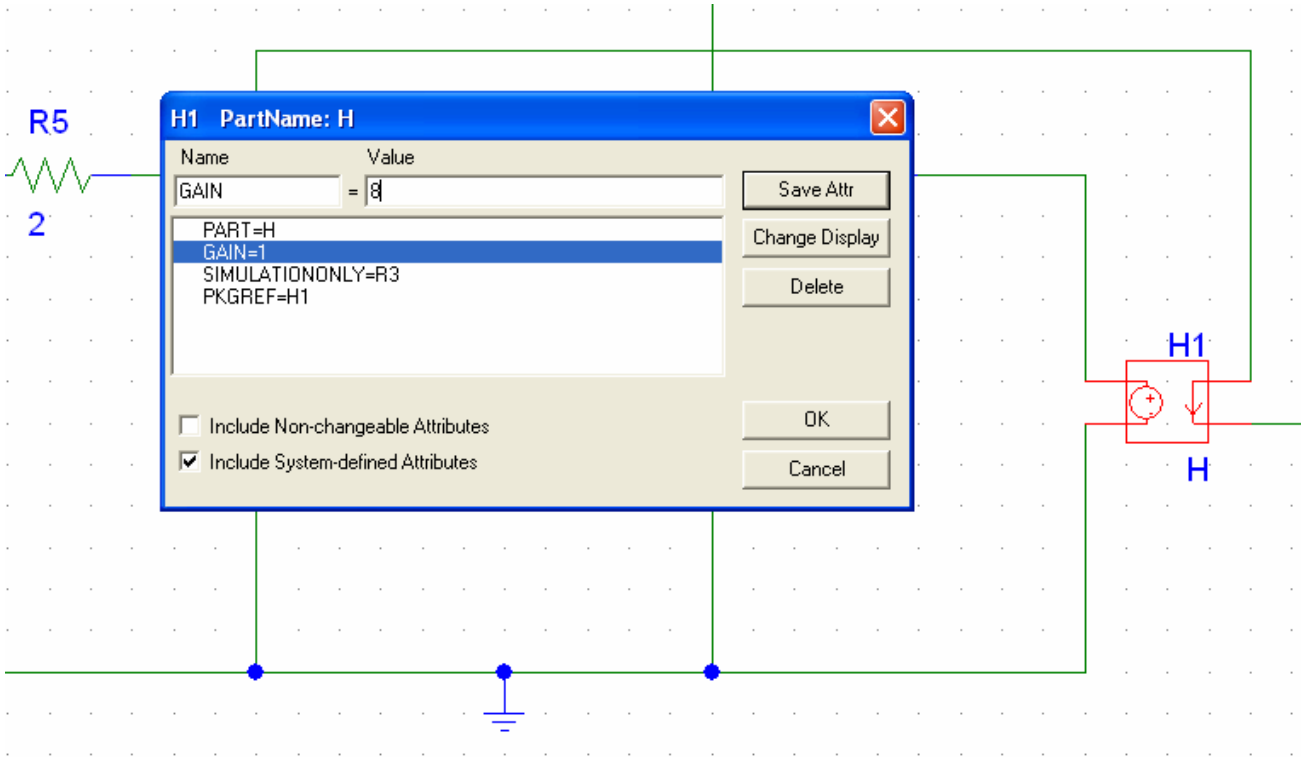
Practice Problem with dependent source

- Simulate the following circuit

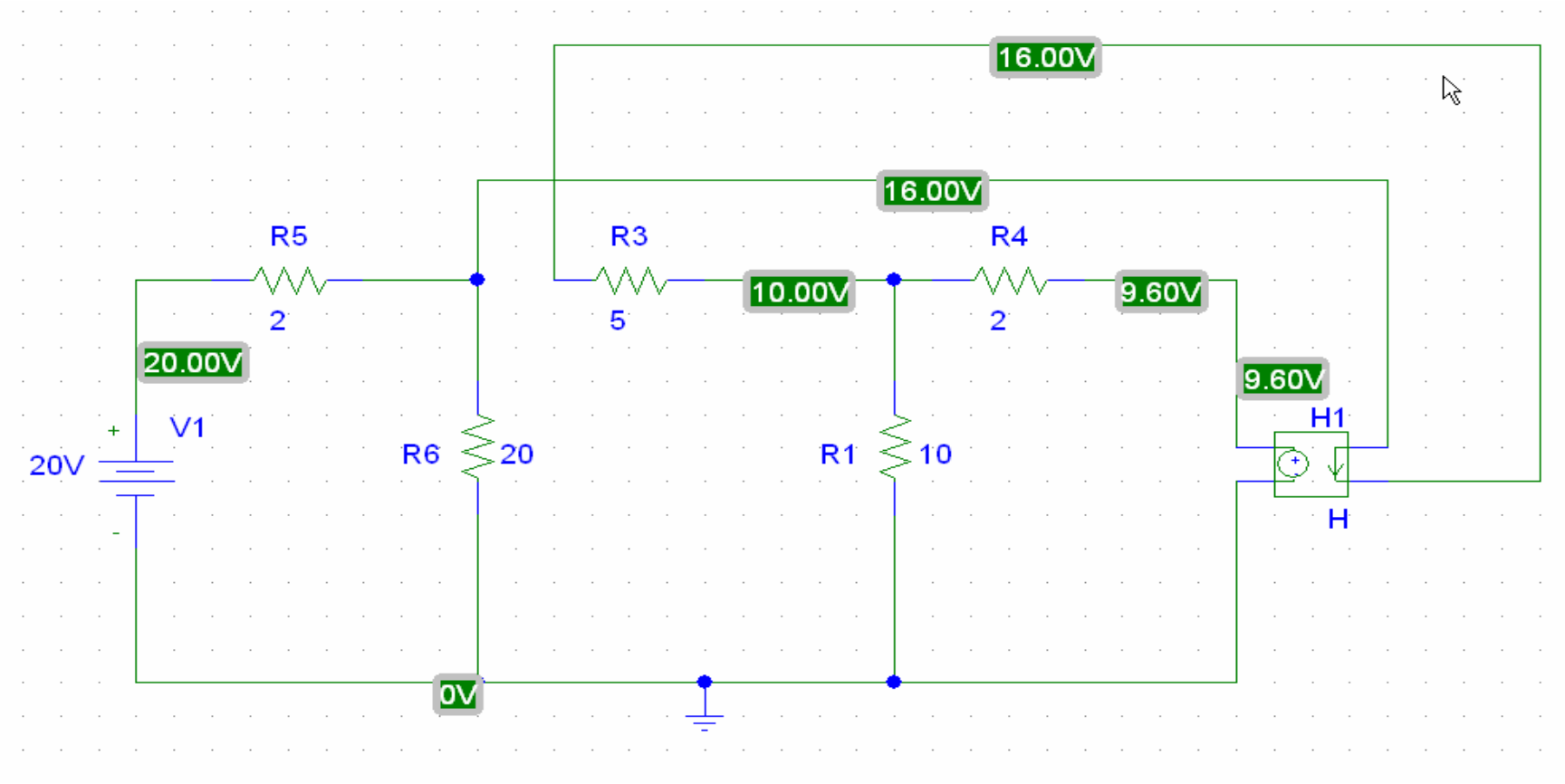


Dependent Source Setup

- Place all the elements including current-controlled-voltage-source, H.
- 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.

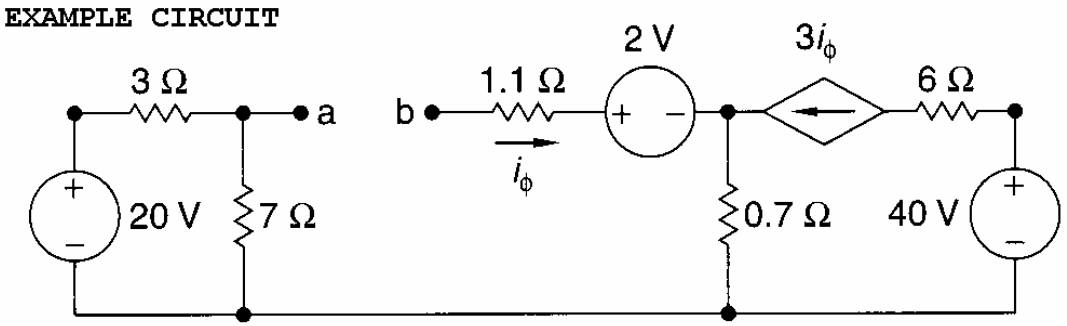


Simulation

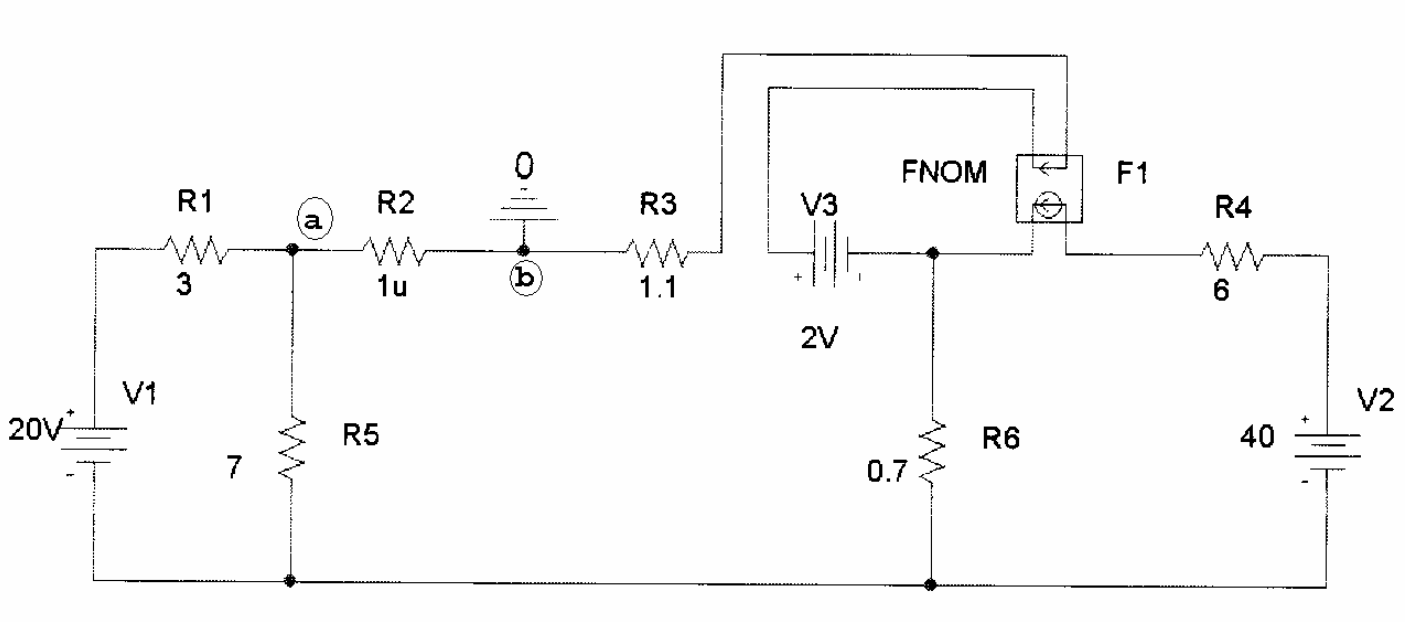


Show Your Simulation Results

Thevenin Equivalent Circuit by PSPICE

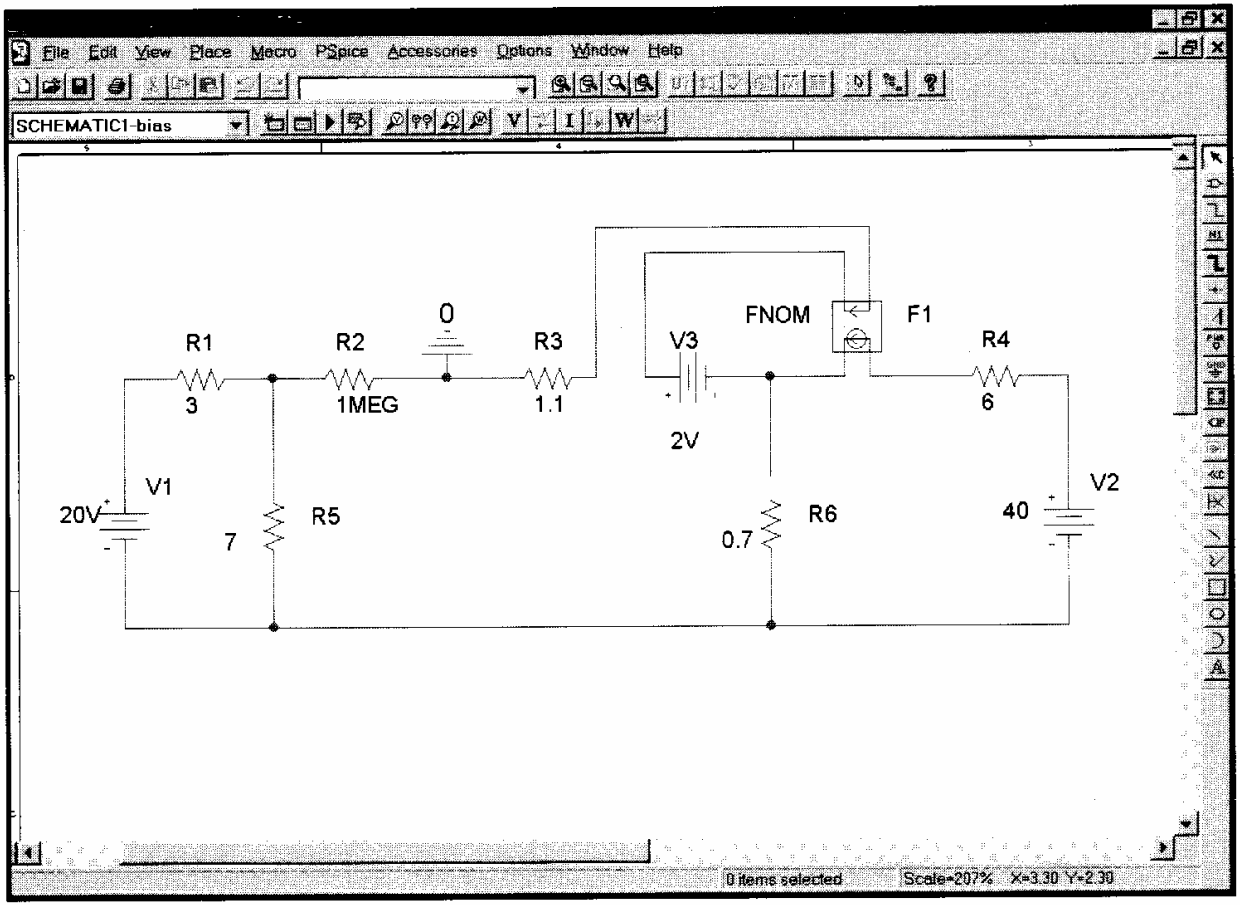


- Short Circuit Current ---very small resistance path
- Get Current (I_{sc})



Thevenin - continued

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



Trouble Shooting

- Always remember to ground your circuit with a 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.
- Common Problem:
  - When every connections seem OK but there is error message consistently →the best way is to erase all the wires, while keeping the elements, then re-wire. This is the surefire way to solve the problem.