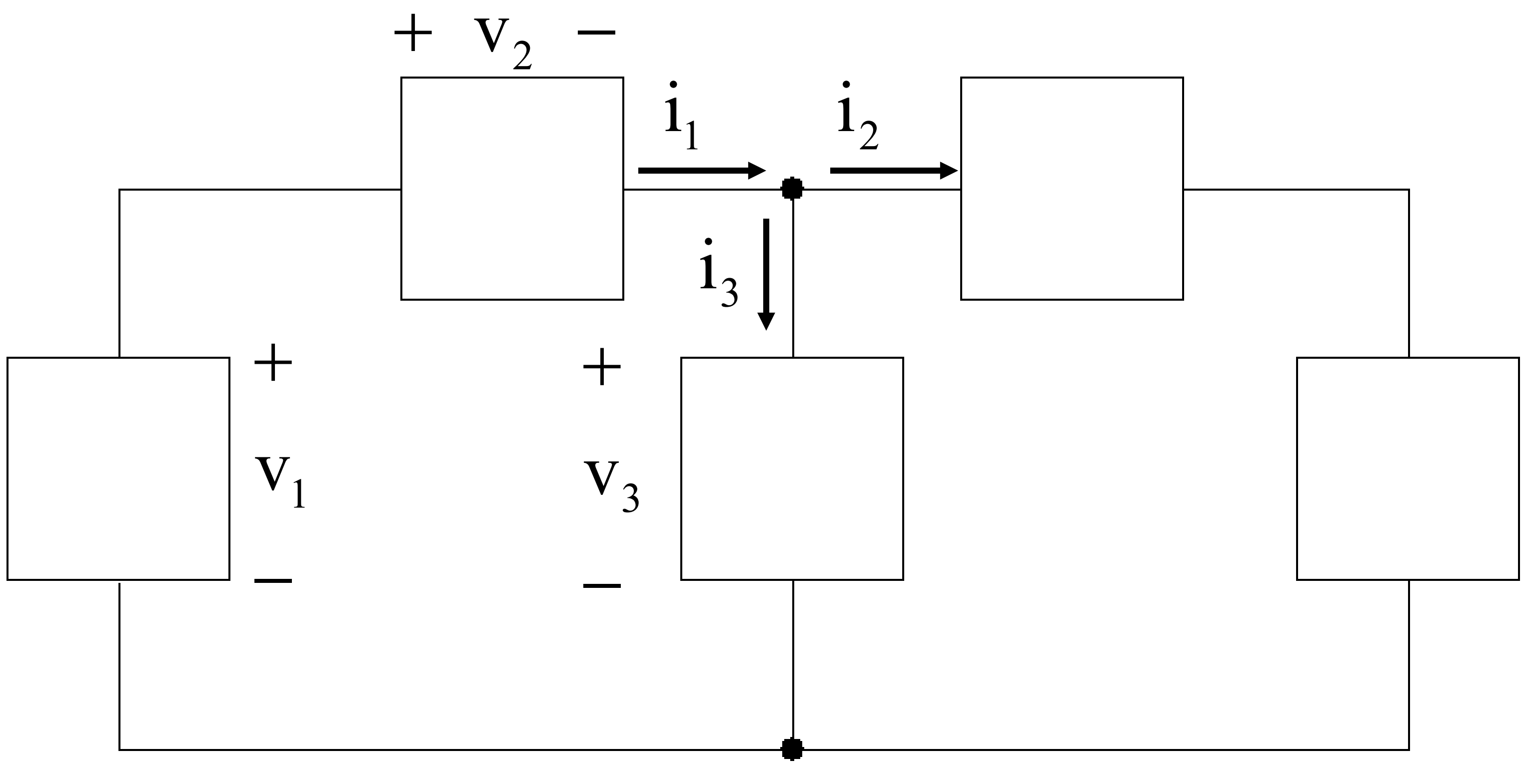
ELECTRIC CIRCUITS
ECSE-2010
Fall 2002
Class 3

REVIEW

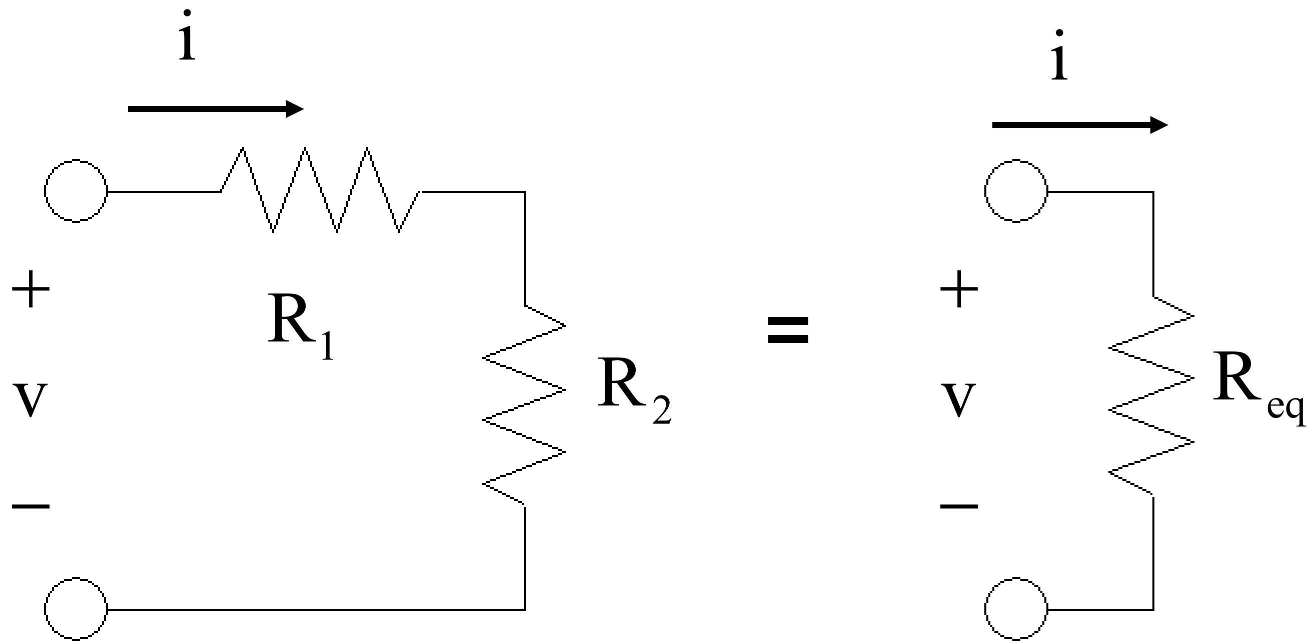
- **Kirchhoff's Laws: KCL, KVL**
  - **Current In = Current Out**
  - **Sum of Voltages Around a Closed Path = 0**
- **Series:** $R_{eq} = R_1 + R_2 + R_3 + ..$
- **Parallel:** $1/R_{eq} = 1/R_1 + 1/R_2 + 1/R_3 + ...$

KIRCHHOFF'S LAWS



KCL: $i_1 = i_2 + i_3$ KVL: $v_1 = v_2 + v_3$

RESISTORS IN SERIES



$$R_{eq} = R_1 + R_2$$

VOLTAGE DIVIDER RULE

- **$v_1 = i\ R_1 = V/R_{eq} \times R_1$**
  - **$v_1 = [R_1/(R_1 + R_2)]\ V$**

$$v_1 = \frac{R_1}{R_1 + R_2} V$$

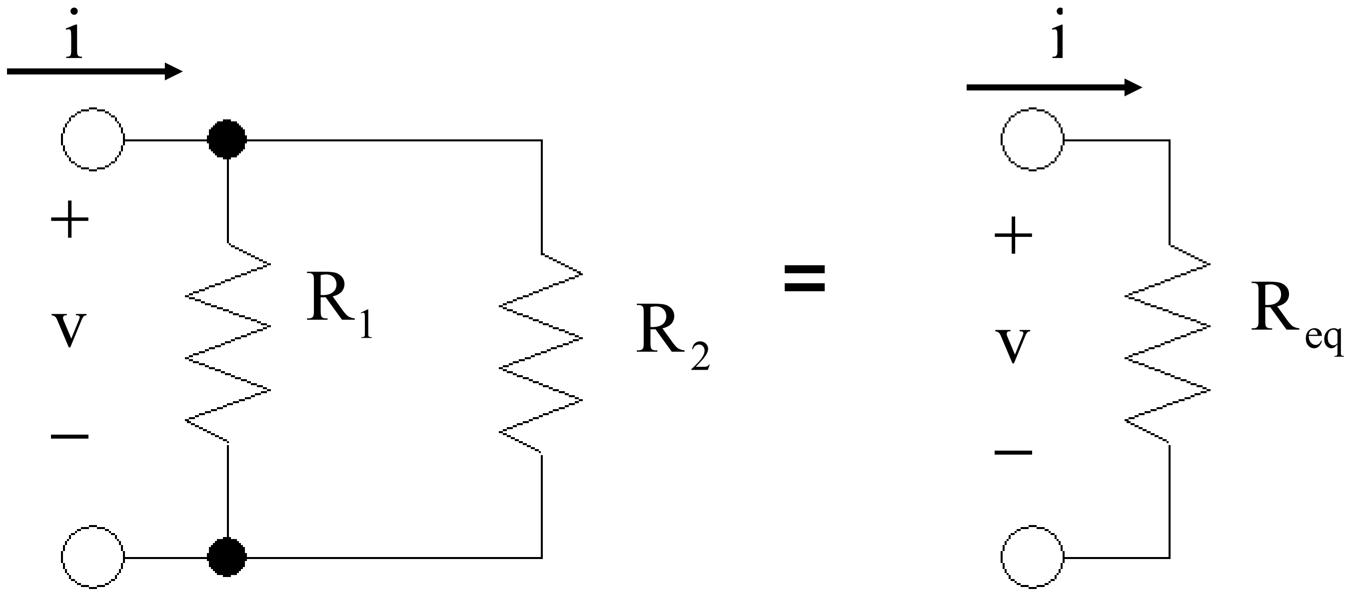
- **$v_2 = i\ R_2$**
  - **$v_2 = [R_2/(R_1 + R_2)]\ V$**

$$v_2 = \frac{R_2}{R_1 + R_2} V$$

- **Elements in Series**
  - **$v_1 \sim R_1$**
  - **$v_2 \sim R_2$**

Rensselaer

RESISTORS IN PARALLEL



$$R_{eq} = \frac{R_1 R_2}{R_1 + R_2}$$

CURRENT DIVIDER RULE

- $i_1 = [R_2/(R_1 + R_2)]\ i$

$$i_1 = \frac{R_2}{R_1 + R_2} i$$

- $i_2 = [R_1/(R_1 + R_2)]\ i$

$$i_2 = \frac{R_1}{R_1 + R_2} i$$

- **Elements in Parallel**
  - $i_1 \sim R_2$
  - $i_2 \sim R_1$

Rensselaer

PSPICE

- **Widely Used Simulation Package**
  - Industry Standard for Circuits and Electronics
- **Student Version is FREE**
  - http://cadencepcb.com/products/downloads/PSpicestudent/default/asp
  - We will use PSpice A/D and Probe primarily
  - Will also want to use Schematics; possibly Capture
- **Get a Copy and Use it Often**

PSPICE

- **PSpice has Many Rules:**
  - **Will build up slowly**
- **Use PSpice for Circuits that are Computationally Challenging:**
- **Will Often Analyze a Circuit to Learn Principles - Then "test" with PSpice:**

Rensselaer

STEPS FOR PSPICE

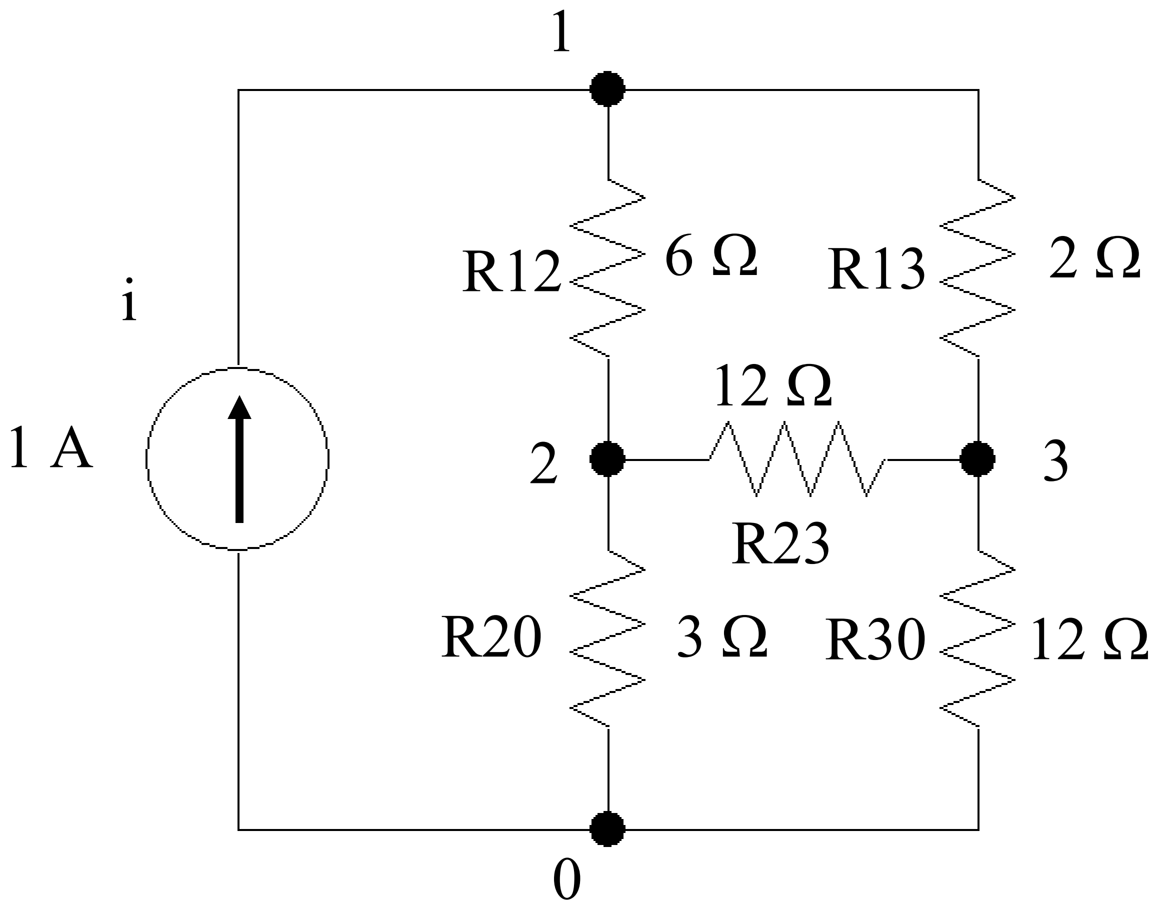
- **Create Simulation Diagram:**
  - **Original Circuit**
  - **Label Nodes, Name Elements, Identify Sources**
- **Create Circuit File:**
  - **Must save as <filename>.cir**
  - **Tells PSpice what to do**
- **Run PSpice and Observe Output**
  - **Usually using Probe**
- **OR – Use Schematics**
  - **Draw Circuit in Schematics and run PSpice**

SIMULATION DIAGRAM

- Circuit Diagram but with all Nodes and all Elements Labeled for PSpice
- Each Node must have a unique Number
- Each Element must have both a Name and a Value
  - Both Active and Passive Elements

Rensselaer

SIMULATION DIAGRAM



Simulation Diagram for PSpice

CIRCUIT FILES

- Circuit File Describes Circuit to PSpice and Tells PSpice What to Do:
- First Line Not Used by PSpice: => Name
- To Describe a Current Source:

```
i2    0    1    DC    1
```

Name of Source = i2; “i” means current (i or I)

Current flows from Node 0 to Node 1

DC Source

1 Amp

Rensselaer

CIRCUIT FILES

- To Describe a Voltage Source

v4 1 0 DC 2

Name of Source = v4; “v” means voltage

Positive terminal is Node 1

Negative terminal is Node 0

DC Source

2 Volts

CIRCUIT FILES

- **Resistors:**

R45 2 3 12

Name of Resistor = R45; "R" => Resistor

Connected between Nodes 2 and 3

Positive Terminal is Node 2

Negative Terminal is Node 3

Resistance is 12 ohms

COMMANDS

.dc lin i1 -1, 1, 0.1

Perform a DC Analysis

Do a Linear Sweep

Sweep the variable named i1

Sweep from -1 Amp to +1 Amp in steps of 0.1 Amps

COMMANDS

.probe

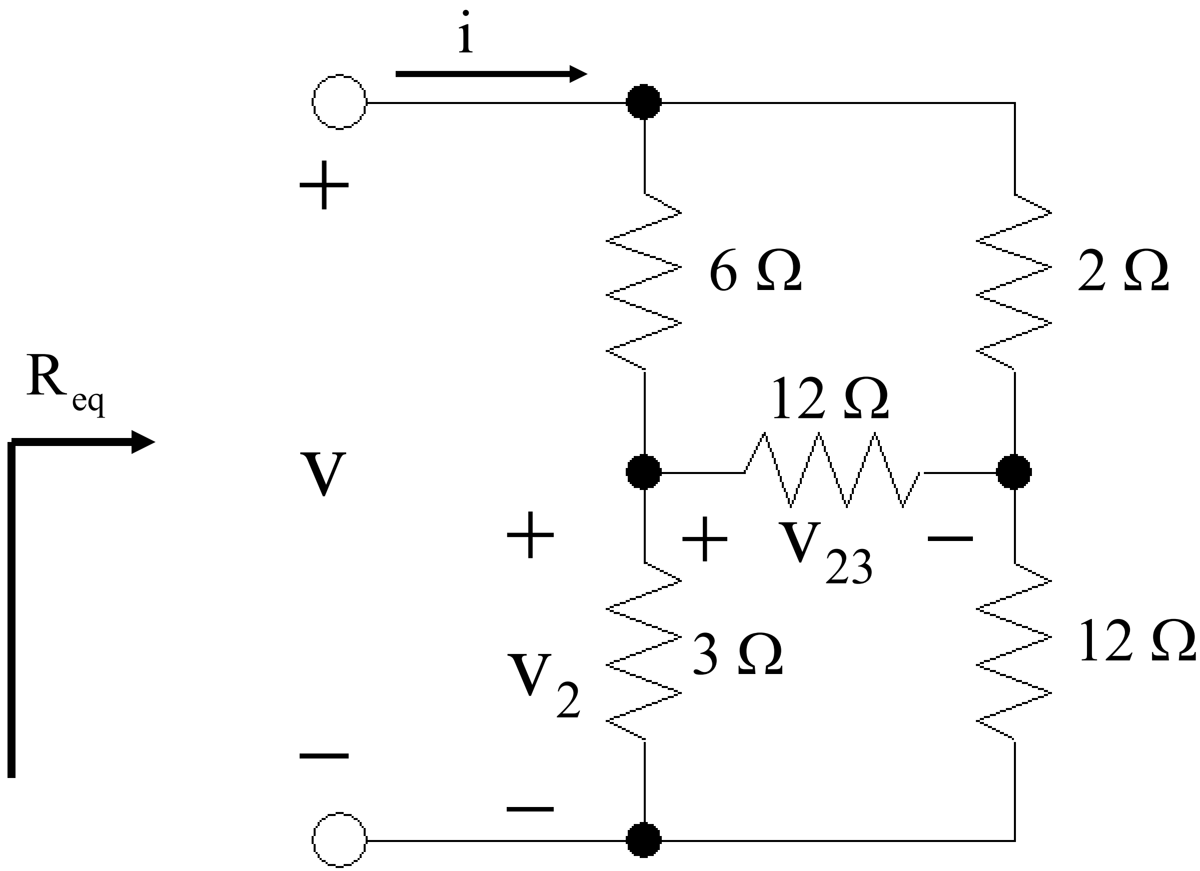
Tells PSpice to run Probe after performing the specified circuit analysis

.end

Tells PSpice that the file is at the end

Will Add More Commands Later

ACTIVITY 3-1



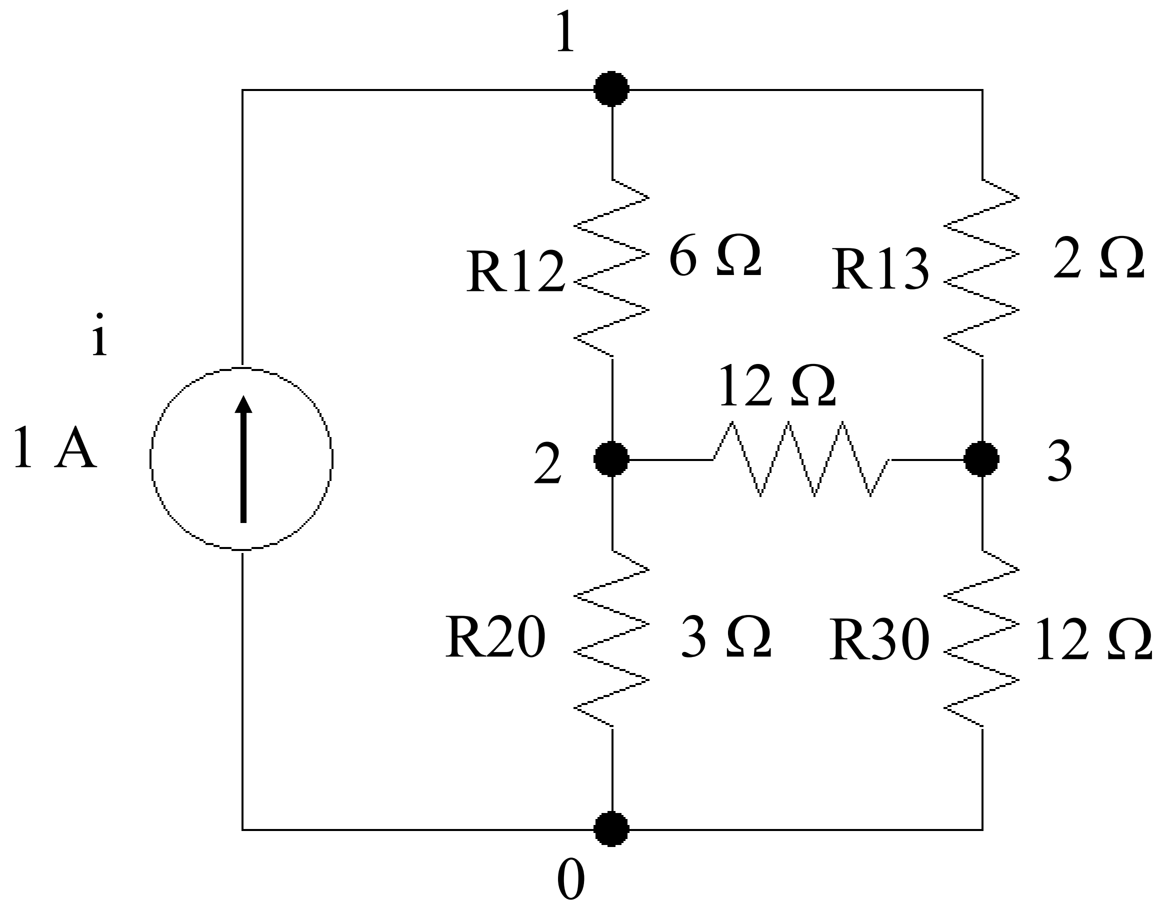
Find v_2 and v_{23}

Resistive Bridge Circuit

ACTIVITY 3-1

- **Cannot use Series/Parallel Reduction for this Circuit**
  - **Nothing is in Series or Parallel**
- **Plot i vs v (or v vs i)**
  - **Slope = 1 / R_{eq} (or Slope = R_{eq})**
- **Let's Do Using PSpice**

ACTIVITY 3-1



Simulation Diagram for PSpice

ACTIVITY 3-1

- **Circuit File**

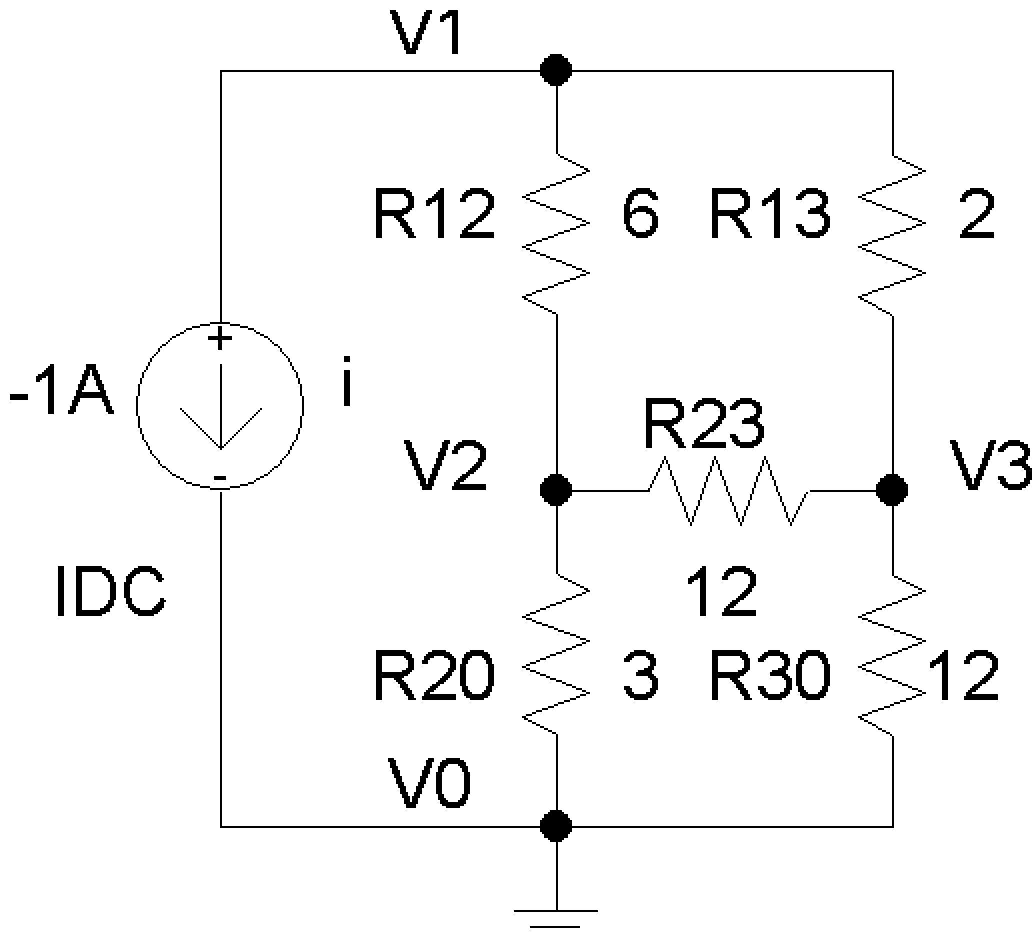
```
Activity 2-2  * PSpice does not use first line
i           0     1     dc    1
R12         1     2     6
R13         1     3     2
R23         2     3     12
R20         2     0     3
R30         3     0     12
.dc  lin    i     -1,   1,    0.1
.probe
.end
```

ACTIVITY 3-1

- Save Circuit File as <name>.cir
- Open PSpice A/D
- Open <name>.cir
- Under Simulation – Run <name>.cir
- Probe comes up automatically
- x axis = i
- Add Trace: v(1), v(2), v(2) – v(3)
- R_{eq} = 5 Ohms

Rensselaer

ACTIVITY 3-1



USING SCHEMATICS

ACTIVITY 3-1

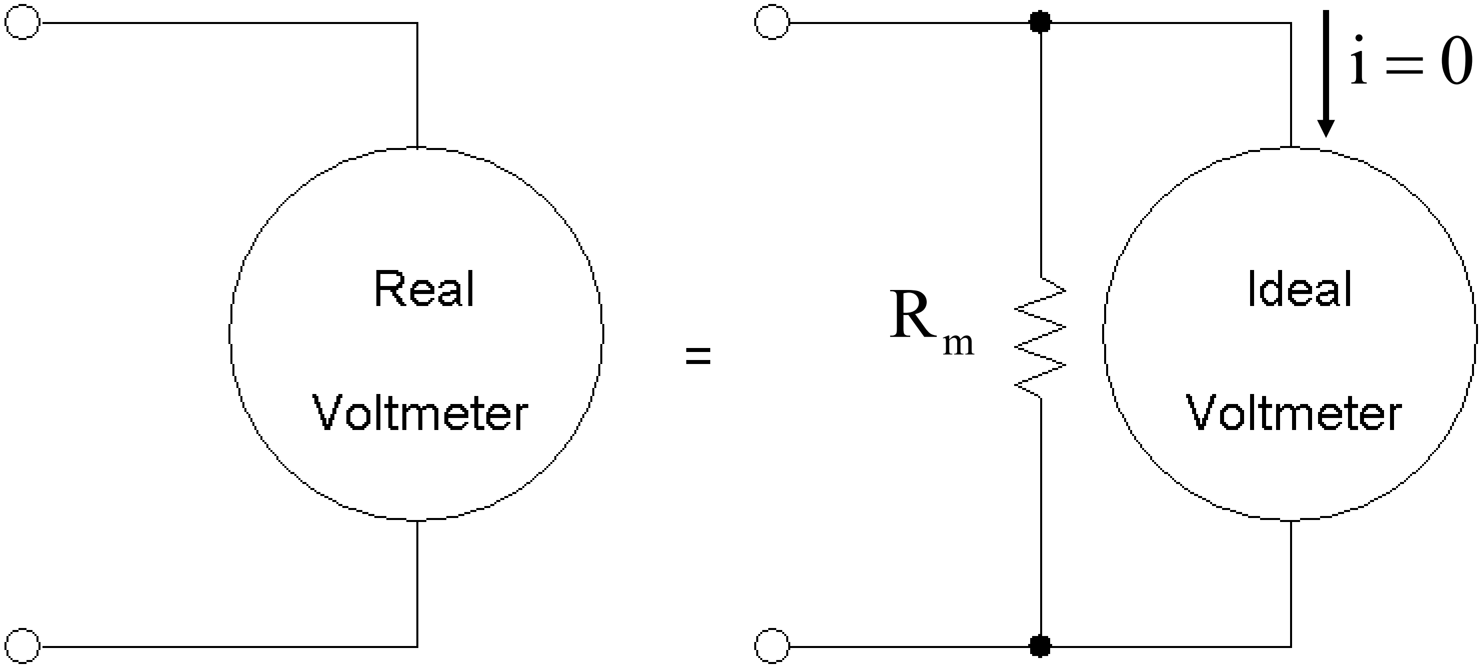
- SCHEMATICS:
  - Schematics uses Passive Convention for Sources
  - Note Current Arrow points in “wrong” direction
  - Must put in – 1 Amp Source
  - Add extra “wires” so you can label them
    - v1, v2, v3
  - Use DC Sweep
  - Sweep i from –1 to +1
  - Run Simulation
  - Change x axis variable to from i to – i
  - Add Trace; v1, v2, v2 - v3

MULTIMETER MEASUREMENTS

- **Model for Real Voltmeter:**
  - Ideal Voltmeter in Parallel with R_M
  - Ideal Voltmeter Draws No Current
  - R_M = “large”; Typically a few Mohms
- **Model for Real Ammeter:**
  - Ideal Ammeter in Series with R_m
  - Ideal Ammeter has no voltage drop
  - R_m = “small”; Typically a few ohms

Rensselaer

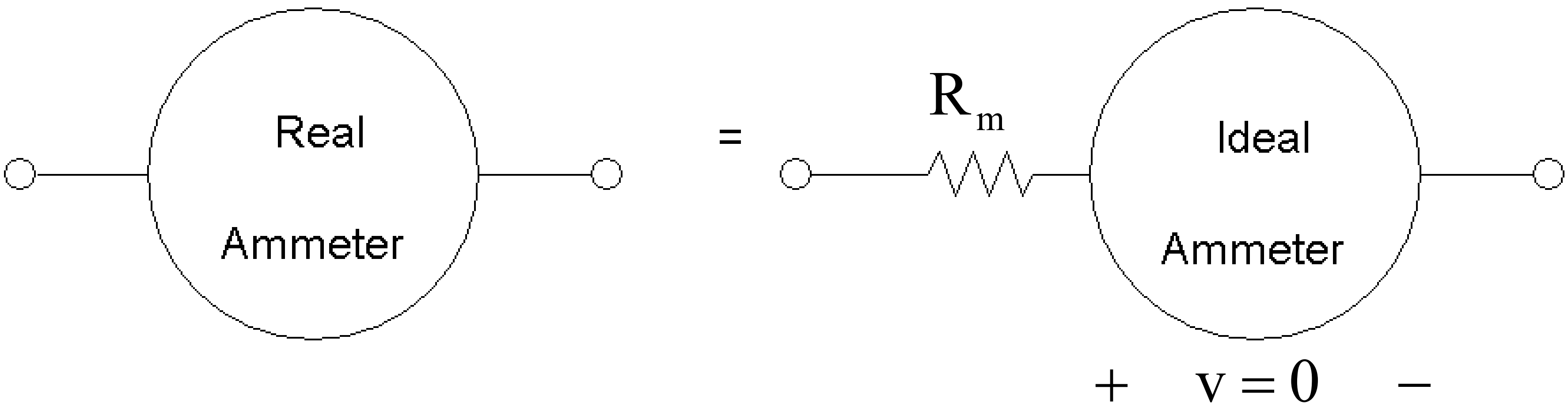
VOLTMETER



R_m = Meter Resistance

= Large (few MΩ's)

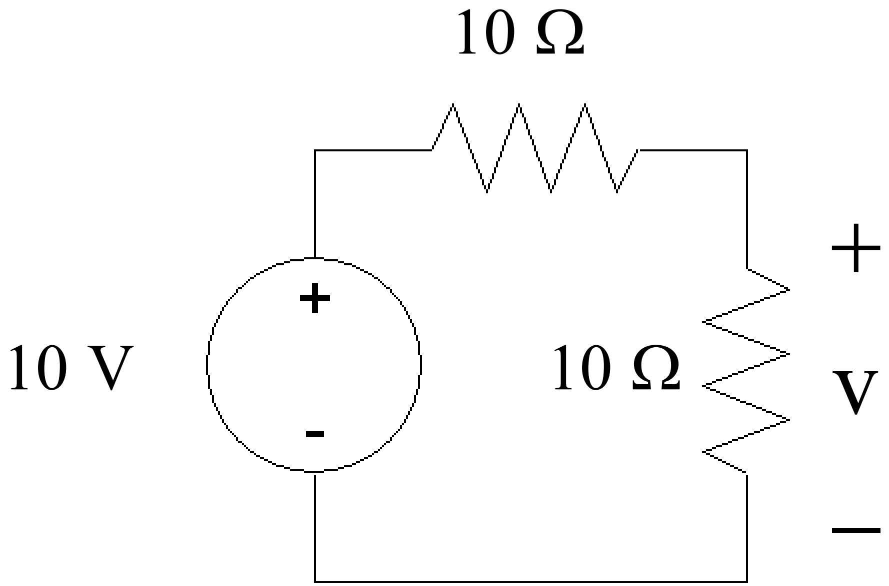
AMMETER



R_m = Meter Resistance

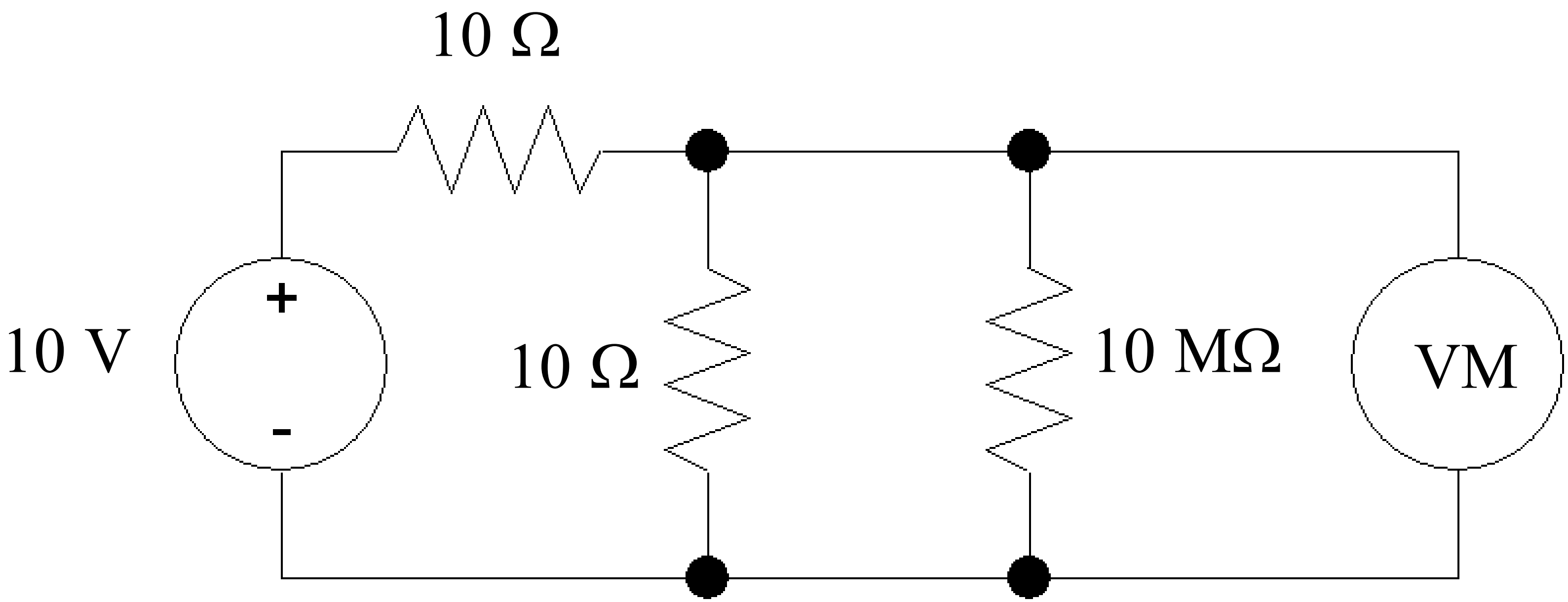
= Small (few Ω's)

EXAMPLE



$$v = \frac{10}{10+10}(10) = 5 \text{ Volts}$$

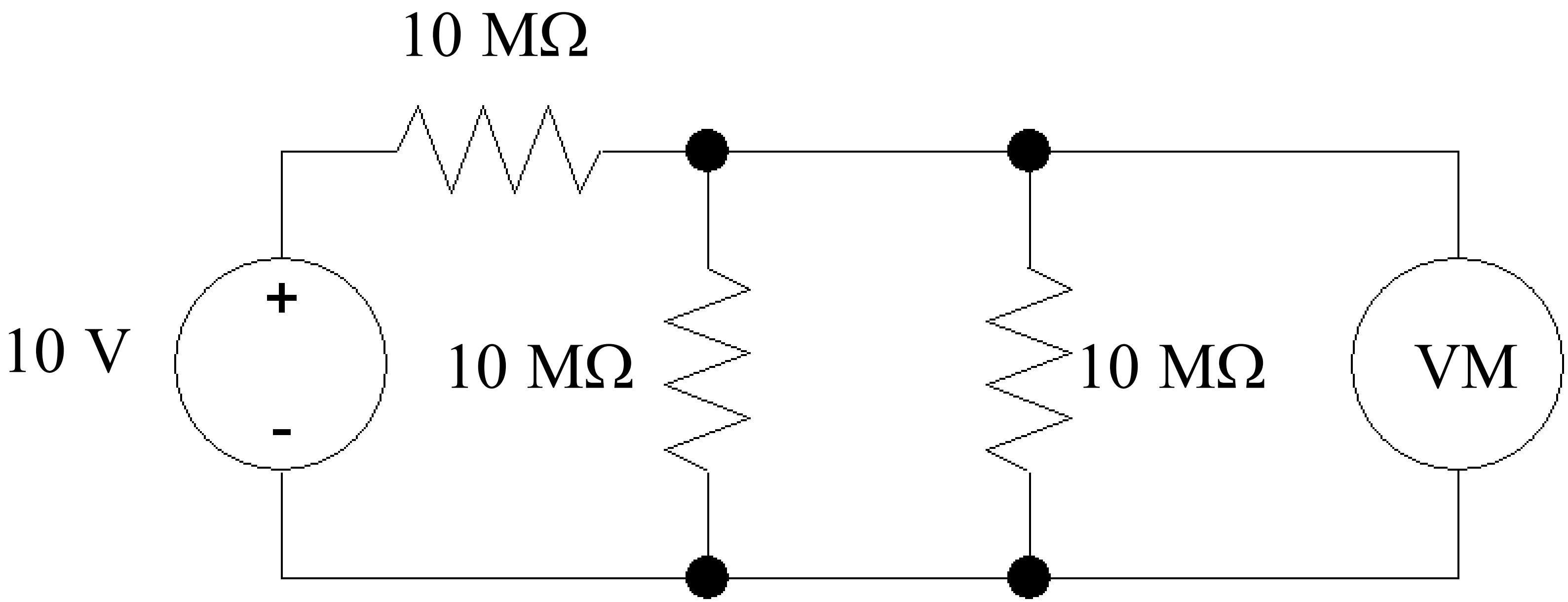
EXAMPLE



$$R_{eq} = 10 \| 10\ \text{M} \approx 10\ \Omega$$

VM reads $\approx$ 5 Volts

EXAMPLE



$$R_{eq} = 10\ \text{M} \| 10\ \text{M} = 5\ \text{M}\Omega$$

$$\text{VM reads } \frac{5}{5+10}(10) = 3.33\ \text{Volts}$$

Meter affects the measurement

EXPERIMENT 1

- **Use E3631A Voltage Source:**
  - **+ 25 V output; +, Com**
  - **Adjust output to be + 12 Volts**
- **Think About Model for Ammeter:**
  - **With Switch Open; Current flows thru Meter**
    - **Measures current as if Switch were Closed**
  - **With Switch Closed; Current flows thru Switch**
    - **Ammeter measures 0; as if Switch were Open**

EXPERIMENT 1