

**ELECTRIC CIRCUITS**  
**ECSE-2010**  
**Fall 2002**  
**Class 3**

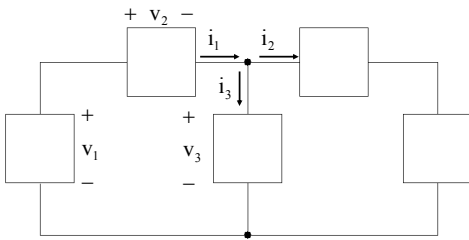


**REVIEW**

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

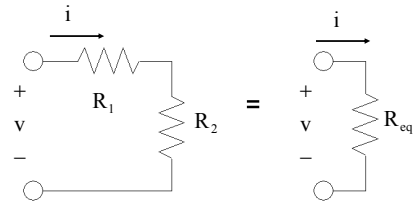


**KIRCHHOFF'S LAWS**



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**

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

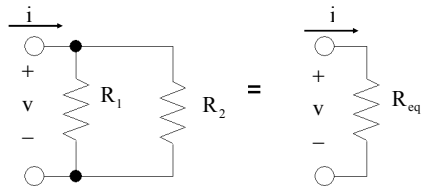
•  $v_2 = i R_2$   
 •  $v_2 = [R_2/(R_1 + R_2)] V$        $v_2 = \frac{R_2}{R_1 + R_2} V$

• **Elements in Series**

- $v_1 \sim R_1$
- $v_2 \sim R_2$



**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
  - $i_1 \sim R_2$
  - $i_2 \sim R_1$



## 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:



## 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

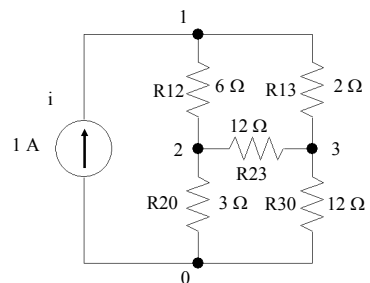


## SIMULATION DIAGRAM

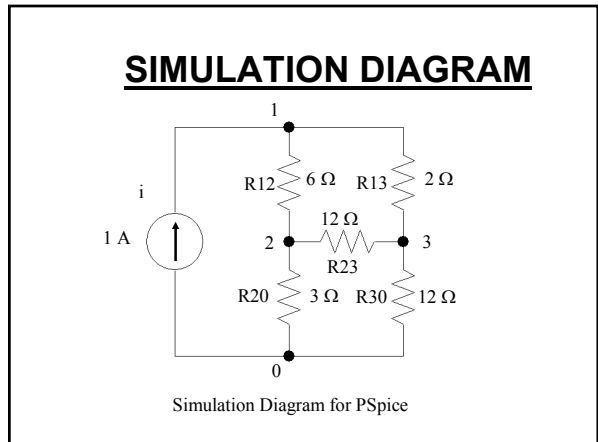
- Circuit Diagram but with all Nodes and all Elements Labeled for PSpice
- Each Node must have a unique Number
- Each Element must have both a Name and a Value
  - Both Active and Passive Elements



## 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



## 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

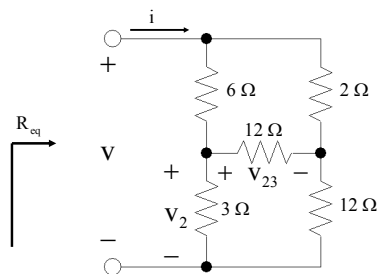
```
.end
```

Tells PSpice that the file is at the end

Will Add More Commands Later



## ACTIVITY 3-1



Find  $v_2$  and  $v_{23}$

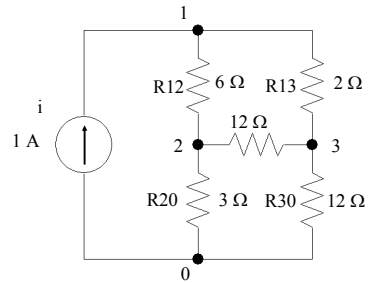
Resistive Bridge Circuit

## 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
```

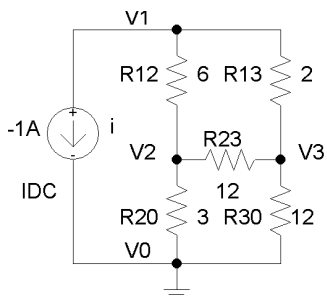


## ACTIVITY 3-1

- 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



## ACTIVITY 3-1



**USING SCHEMATICS**

## ACTIVITY 3-1

### • SCHEMATICS:

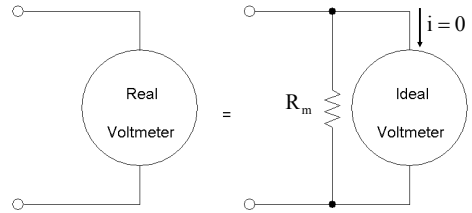
- Schematics uses Passive Convention for Sources
- Note Current Arrow points in “wrong” direction
- Must put in – 1 Amp Source
- Add extra “wires” so you can label them
  - $v1$ ,  $v2$ ,  $v3$
- Use DC Sweep
- Sweep  $i$  from  $-1$  to  $+1$
- Run Simulation
- Change x axis variable to from  $i$  to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



## MULTIMETER MEASUREMENTS

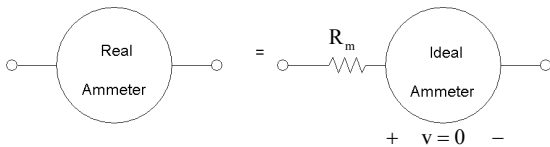
- **Model for Real Voltmeter:**
  - Ideal Voltmeter in Parallel with  $R_m$
  - Ideal Voltmeter Draws No Current
  - $R_m$  = "large"; Typically a few Mohms
- **Model for Real Ammeter:**
  - Ideal Ammeter in Series with  $R_m$
  - Ideal Ammeter has no voltage drop
  - $R_m$  = "small"; Typically a few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

## VOLTMETER



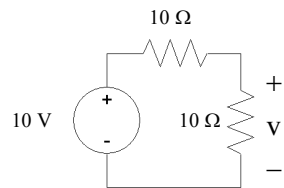
$R_m$  = Meter Resistance  
= Large (few  $M\Omega$ 's)

## AMMETER



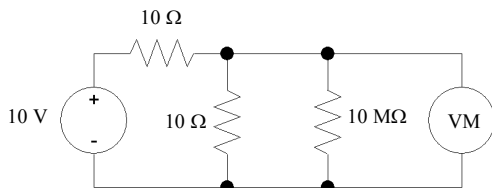
$R_m$  = Meter Resistance  
= Small (few  $\Omega$ 's)

## EXAMPLE



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

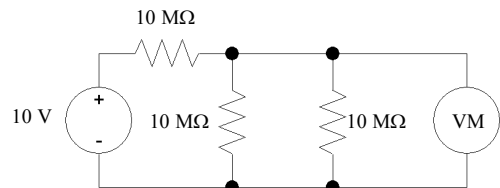
## EXAMPLE



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

VM reads  $\approx 5$  Volts

## EXAMPLE



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

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

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

