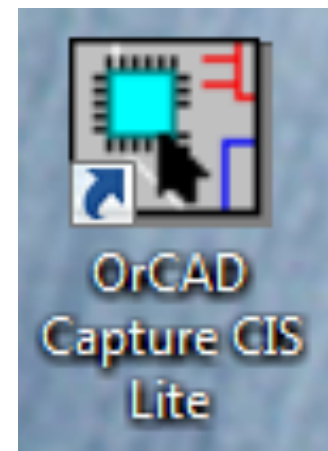# Introduction to PSPICE

OrCAD Capture CIS Lite

Reference: (see course web site)

- *Sample PSPICE Report*
- PSPICE Example: *DC Circuit – First Example*
- PSPICE Example: *Analyzing Circuits with Dependent Sources*
- PSPICE Assignment #1
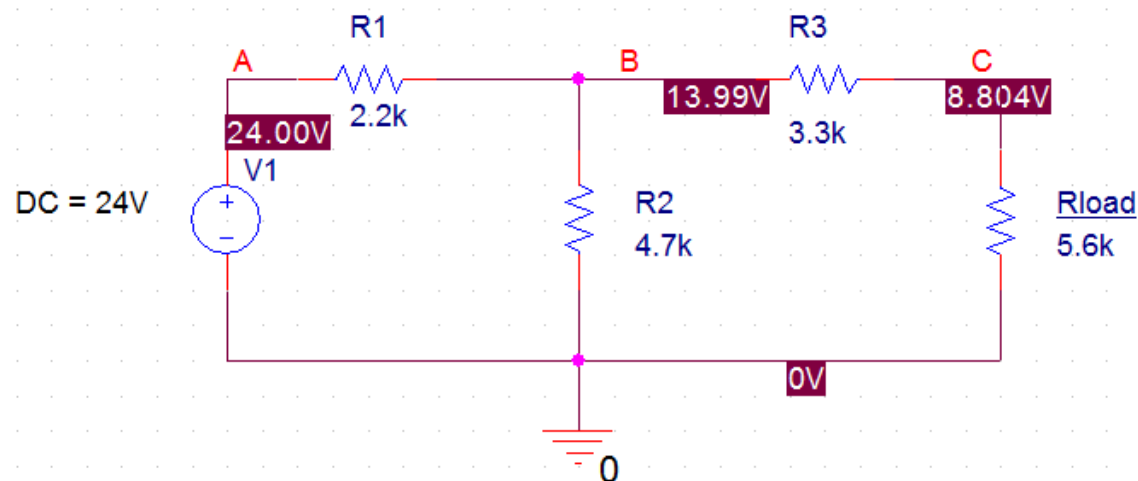
Topics to be presented:

- Introduction to PSPICE
- Creating your first project
- Bias Point Analysis
- Using PSPICE to analyze circuits with dependent sources

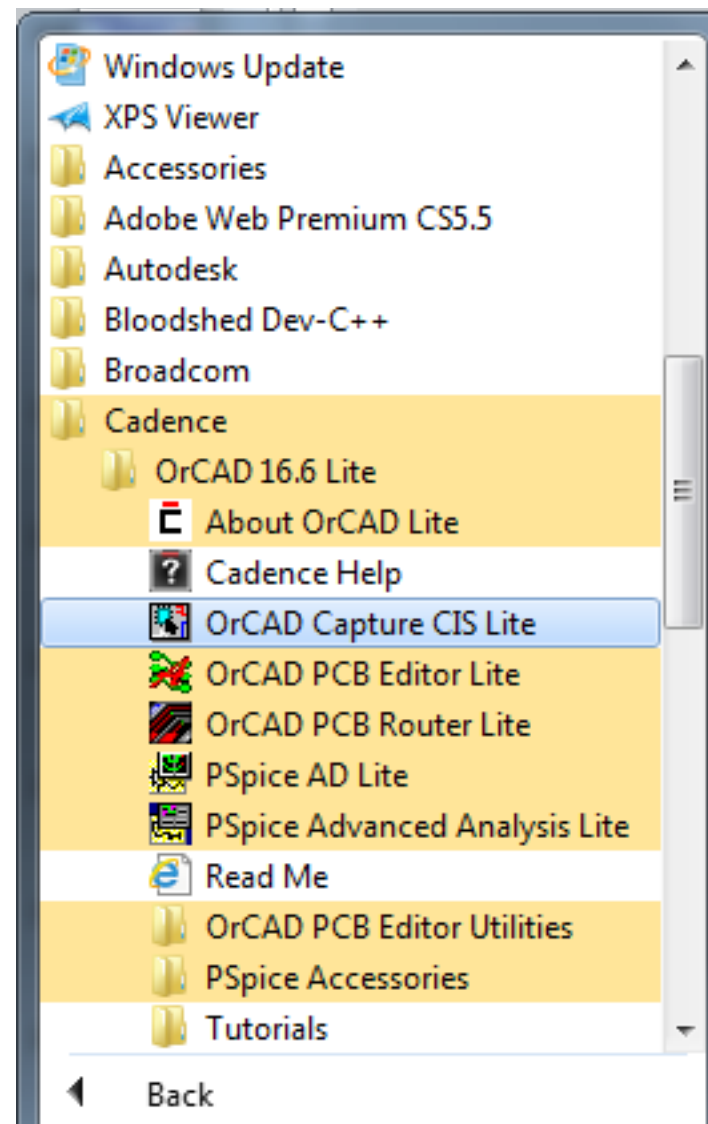EGR 271 – Circuit Theory I
PSPICE Assignment #1
Problem 3-15
Purpose: Determine the voltage across resistor R1

R1 2.2k
R3 3.3k
24.00V
V1
DC = 24V
13.99V
8.804V
R2 4.7k
Rload 5.6k
0V
0

ORCAD Capture (PSPICE)

- The software is referred to by the generic name PSPICE. It was originally owned by MicroSim and then by ORCAD. It is currently owned by Cadence, but has kept the ORCAD name.

- See examples on the course web site. The examples contain numerous comments. A classroom demonstration will provide further examples.

- The software is free to download from the Cadence website. Search for ***Cadence ORCAD download*** or use the following link:

www.cadence.com/products/orcad/pages/downloads.aspx

- Run the software by selecting **Start – All Programs – Cadence – Cadence 16.6 Lite - ORCAD Capture CIS Lite**

Windows Update
XPS Viewer
Accessories
Adobe Web Premium CS5.5
Autodesk
Bloodshed Dev-C++
Broadcom
Cadence
    OrCAD 16.6 Lite
        About OrCAD Lite
        Cadence Help
        OrCAD Capture CIS Lite
        OrCAD PCB Editor Lite
        OrCAD PCB Router Lite
        PSpice AD Lite
        PSpice Advanced Analysis Lite
        Read Me
        OrCAD PCB Editor Utilities
        PSpice Accessories
        Tutorials
Back

Comparison - Evaluation Version versus Full Version

Evaluation Version

- Limit of 64 nodes
- Limit of 1 page schematic
- Standard libraries for sources and basic components are available
- EVAL library (eval.slb) contains a couple hundred models of components, such as transistors, diodes, 7400 series IC's, etc.
- Cost: Free
- Some advanced features not included (such as printed circuit board layout)
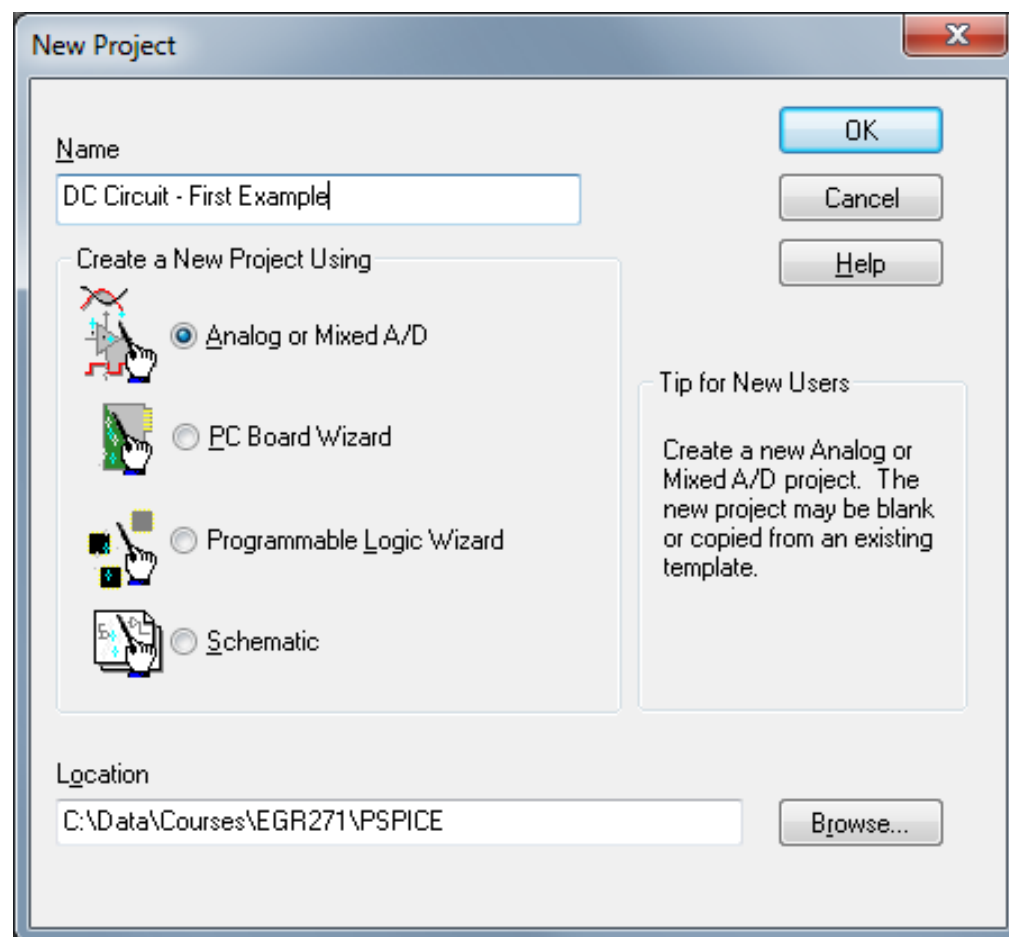
Full Version

- Unlimited number of nodes
- Unlimited number of pages
- Standard libraries for sources and basic components are available
- Libraries for most commercially-available components available (tens of thousands of models).
- Cost: Thousands of dollars (depends on the platform and the types of features needed)
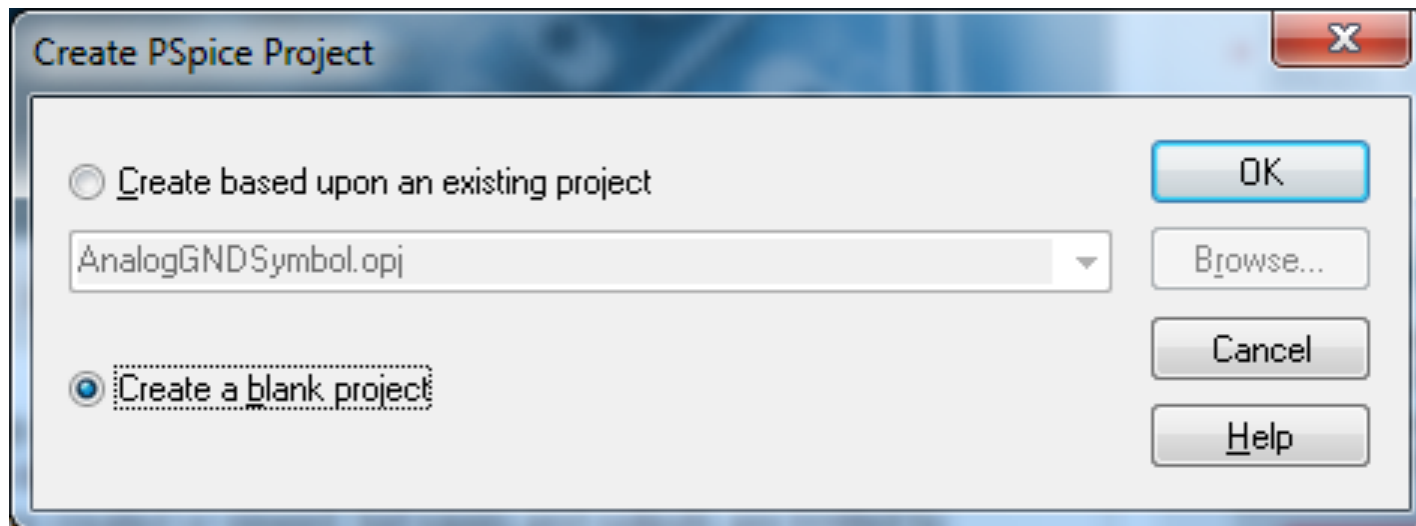
Basic steps in creating and analyzing a circuit using PSPICE

(See examples on class web site for more details and illustrations):

1) Create a Project

A) Select **File – New – Project** from the main menu

B) When the New Project box appears, specify the following:

- Assign the project a **Name** (any name is fine)

- Create project using **Analog or Mixed Analog/Digital**

- Pick the **Location** (select Browse and then select Create Dir to make a new folder for your project)

New Project

Name
DC Circuit - First Example

OK
Cancel
Help

Create a New Project Using
○ Analog or Mixed A/D
○ PC Board Wizard
○ Programmable Logic Wizard
○ Schematic

Tip for New Users

Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.

Location
C:\Data\Courses\EGR271\PSPICE   Browse...

<u>Basic steps in creating and analyzing a circuit using PSPICE</u>

(See examples on class web site for details and illustrations):

   1) <u>Create a Project</u>
        C)   If asked to *create based on an existing project* or *create a blank project*, select **<u>Create a blank project</u>.**

Basic steps in creating and analyzing a circuit using PSPICE
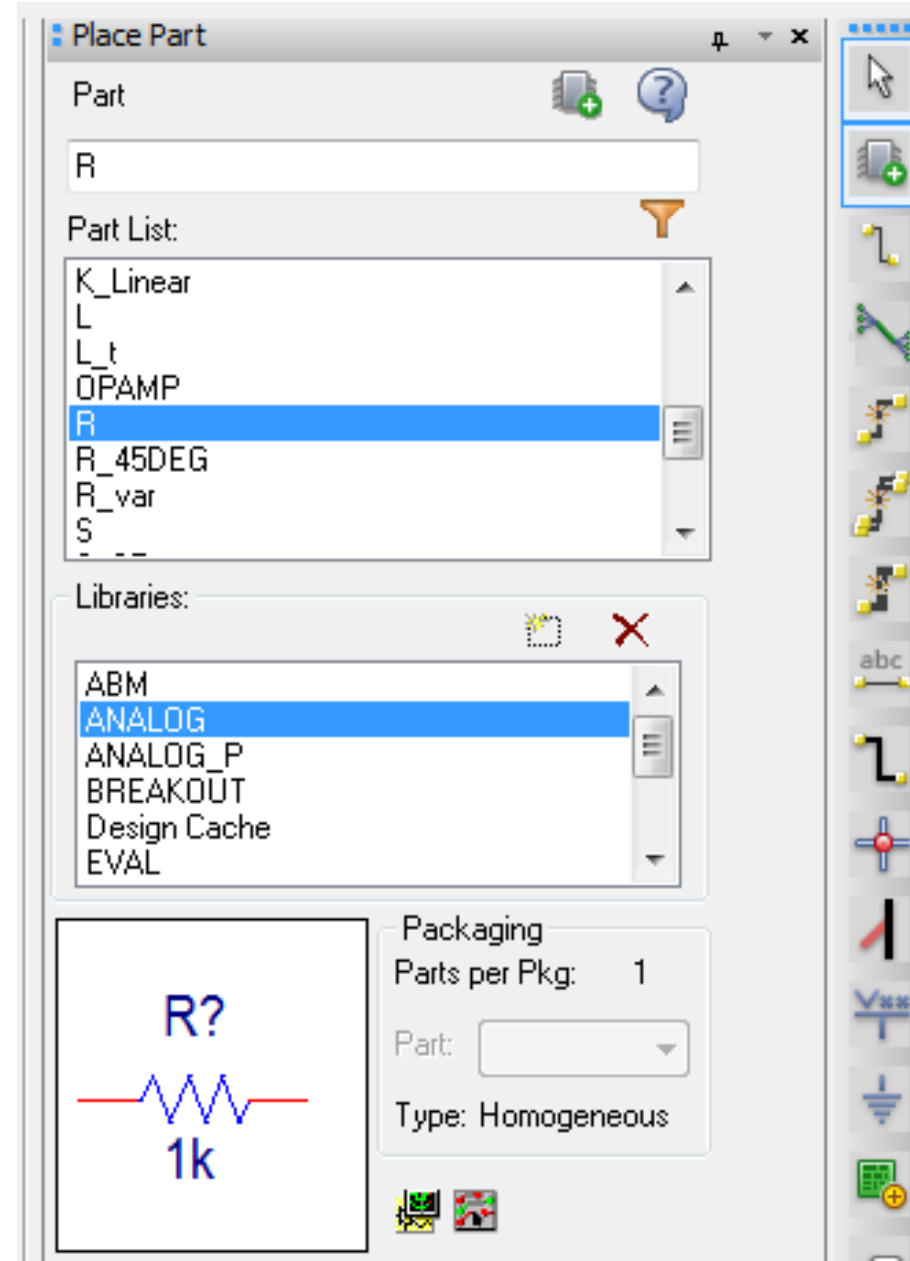
2) Draw the schematic

  A. Select **Place - Part** from the main menu. A list of libraries and parts should appear.

  B. If no libraries appear, select **Add Libraries** and select all libraries listed in the PSPICE folder as shown below. Select **Open**.
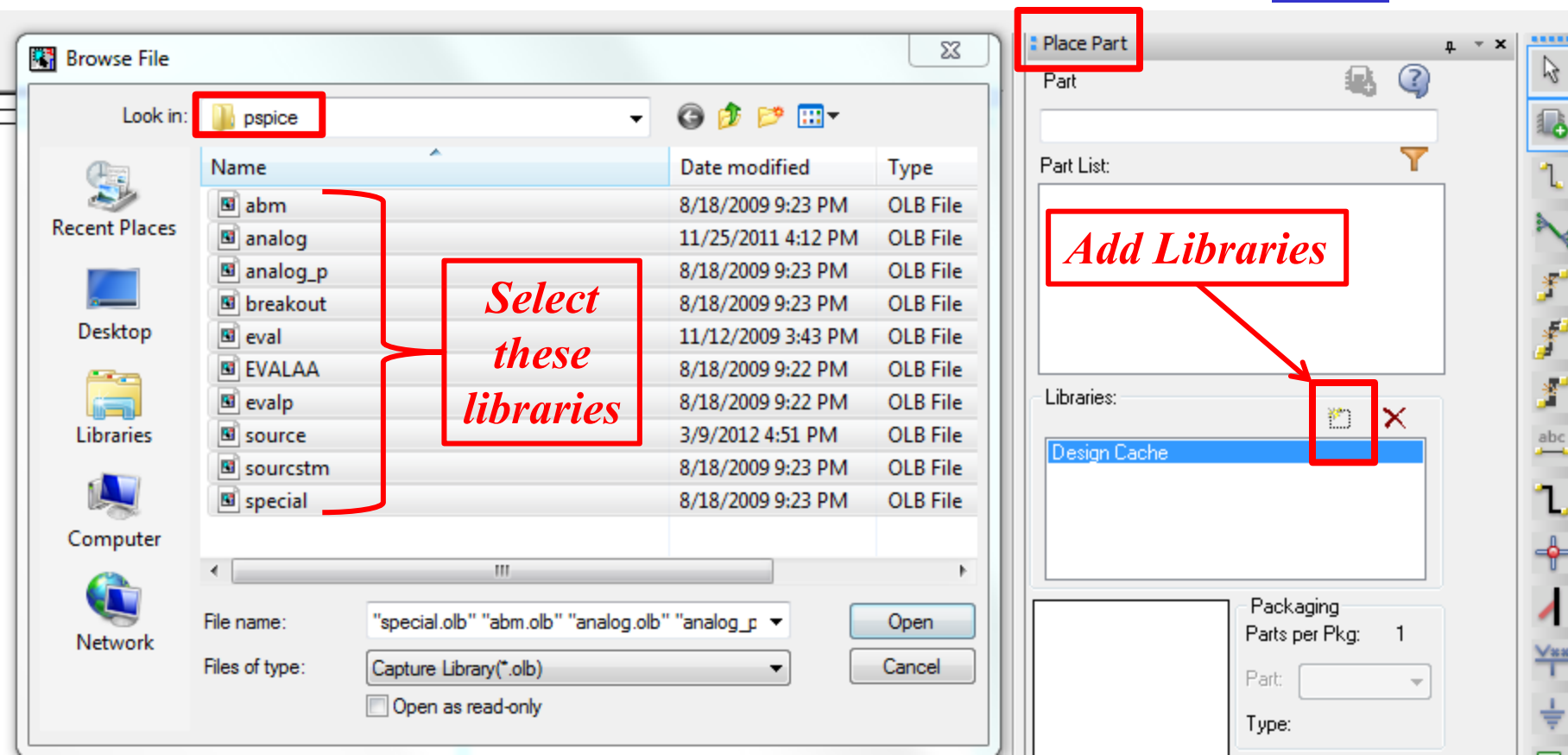
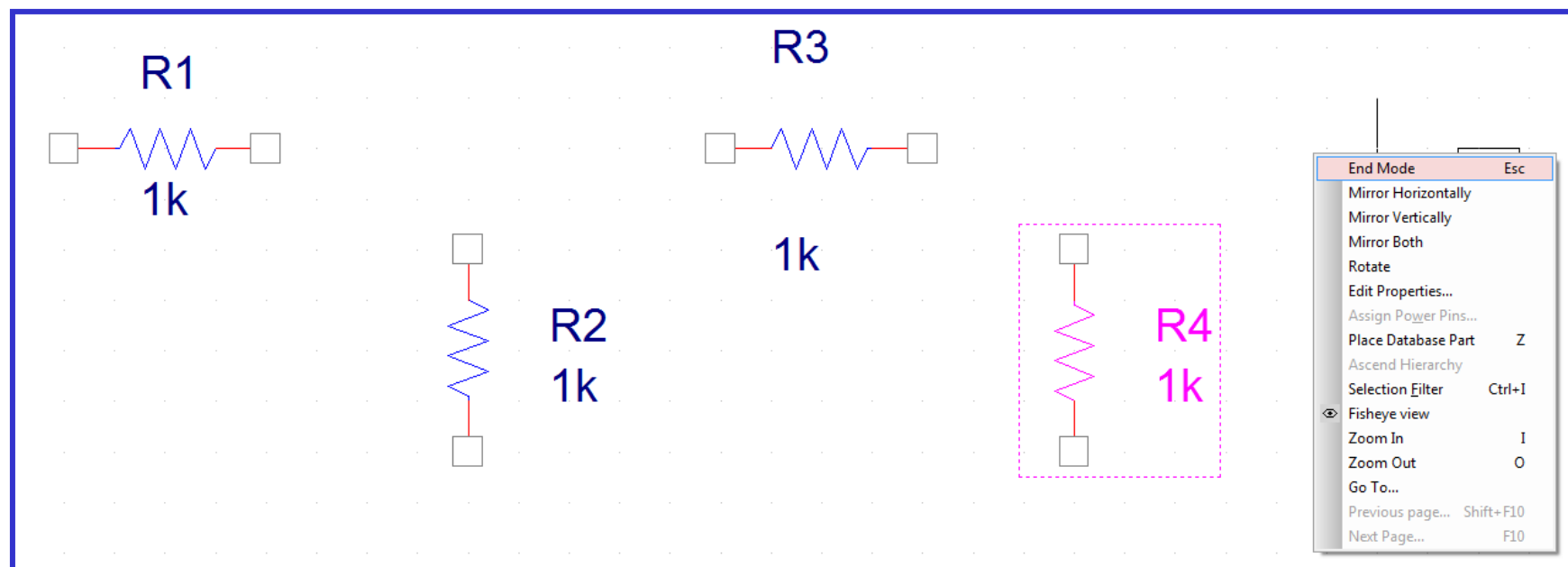## Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

   C. Select the desired part (such as R in the **Analog** library as shown to the right).

Basic steps in creating and analyzing a circuit using PSPICE
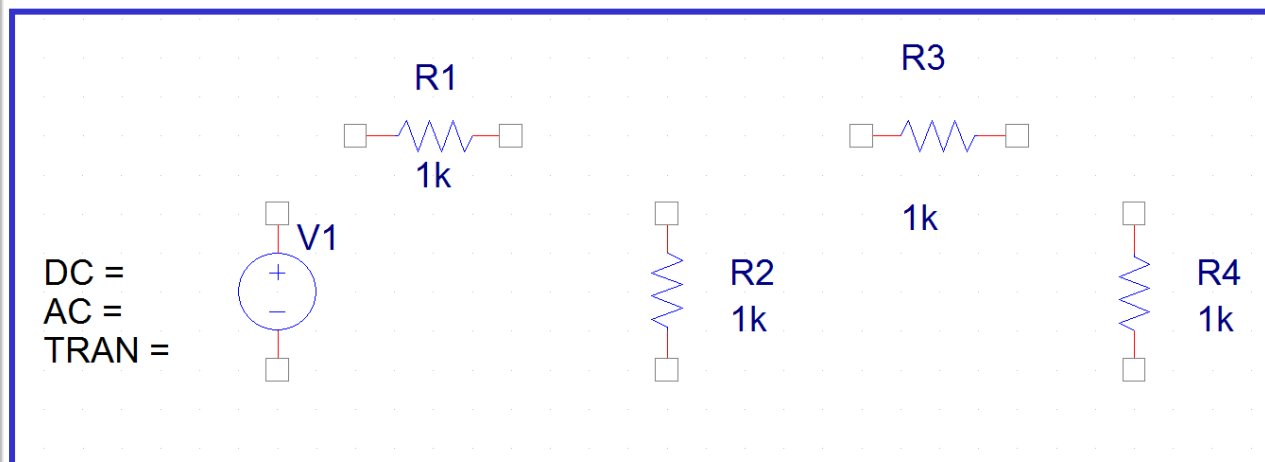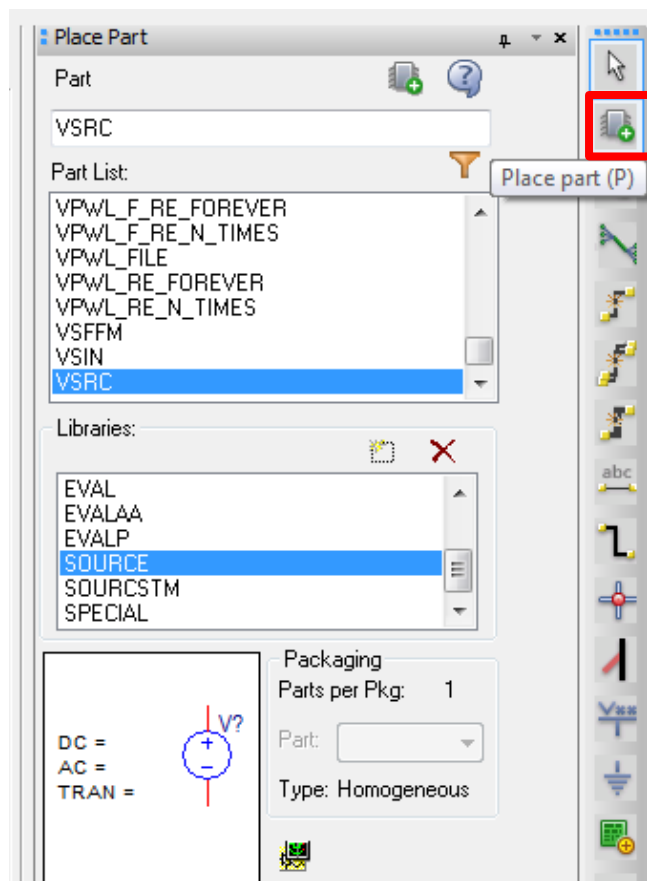
2) Draw the schematic

   D. Click where you would like to place the part on the schematic.

   E. Use **R** to rotate the part as you place it on the schematic. Once it has been placed, it can be rotated, flipped, or mirrored by right-clicking on the part.

   F. Continue clicking on the schematic to place multiple parts (such as several resistor) and then right click and select **End Mode** to stop.

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

    G. Similarly, add a voltage source – part **VSRC** from the **Source** library. You can use Place – Part from the main menu or use the toolbar on the right of the screen. Note that the voltage source has been added to the schematic below.
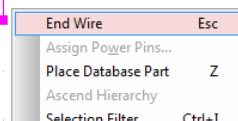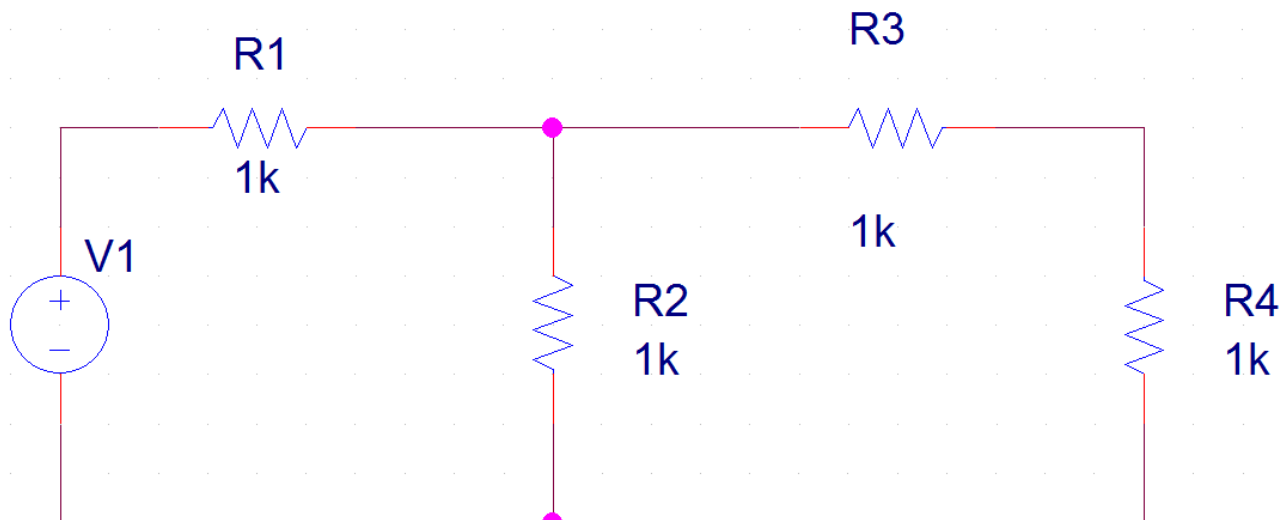

Place Part

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

H. Select **Place - Wire** (or use the toolbar) and add wires to connect the parts. Right-click and select End Wire (or hit the Esc key) to exit the Place Wire tool.

Place Wire

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

I. Analysis of circuits in PSPICE is based on nodal analysis, so each circuit must contain a ground symbol. The ground symbol named 0 must be used. If this is omitted, a "floating nodes" error will occur during analysis. The ground is added using **Place Ground**. If the 0 ground is not available, select **Add Library** and you should find it in the PSPICE – SOURCE library.
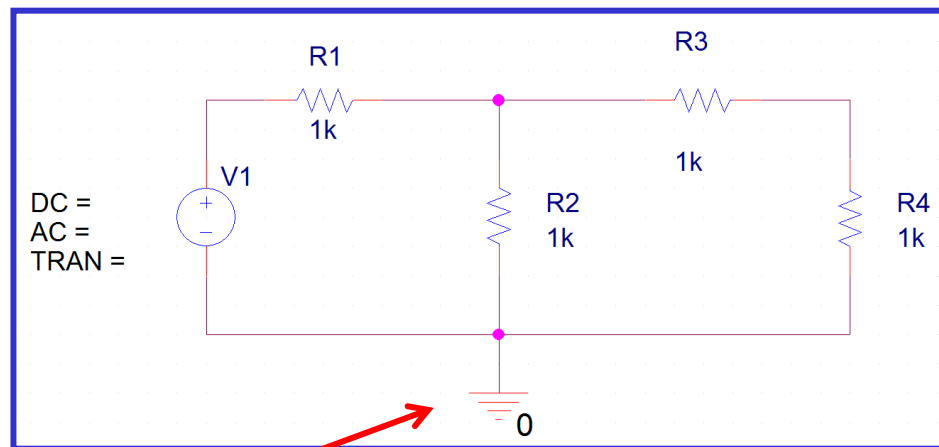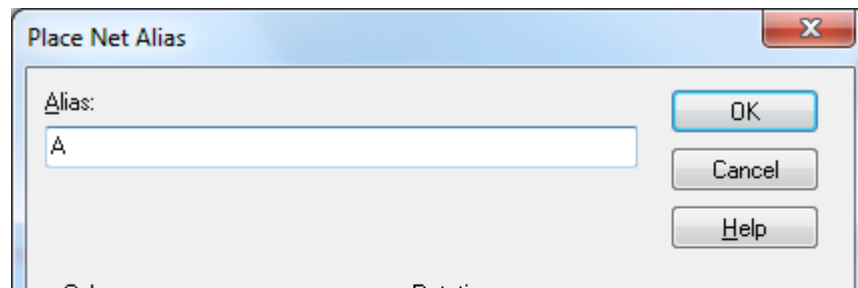

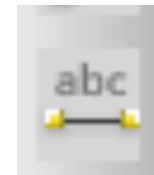*Ground added to the schematic*
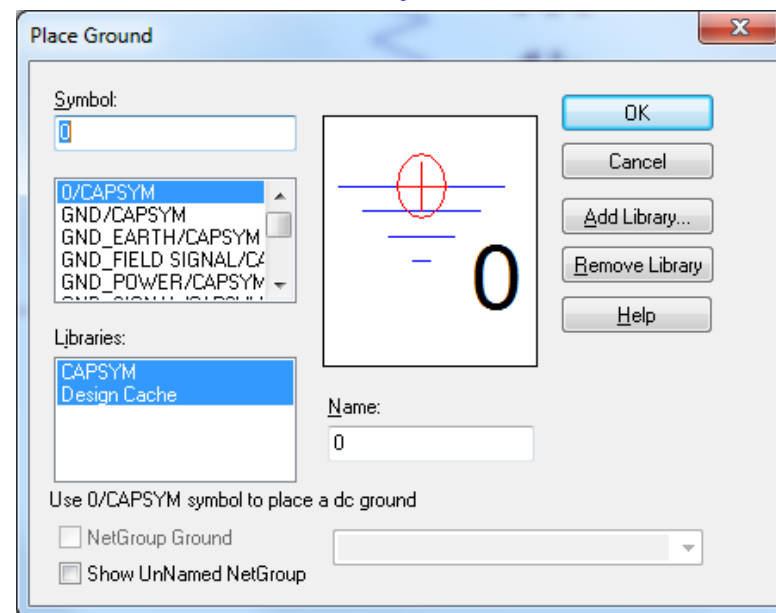
Place Ground

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

    J.   Use **Place - Net Alias** (or the tool shown to the right) to label each node (N1 button in version 10.5 or abc button in version 16.5 as shown to the right).

Net Alias

Place Net Alias

Alias:

A

OK

Cancel

Help

Nodes A, B, C, labeled using Net Alias tool. Note that the ground node is already labeled (0).

R1
1k

R3

A

B

C

V1

DC =
AC =
TRAN =

R2
1k

1k

R4
1k

0
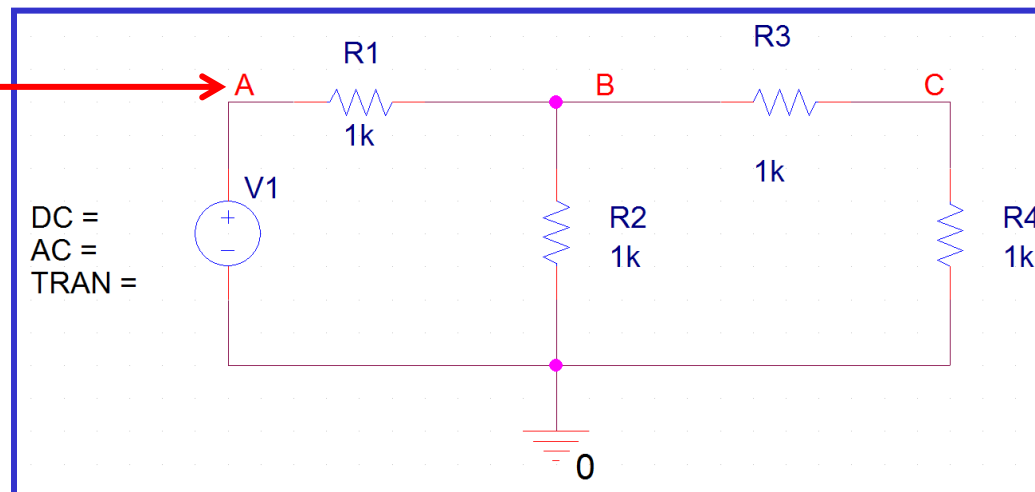
Text

Note: There is a similar looking button on the toolbar used to place text. Do not use this tool to label nodes.

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

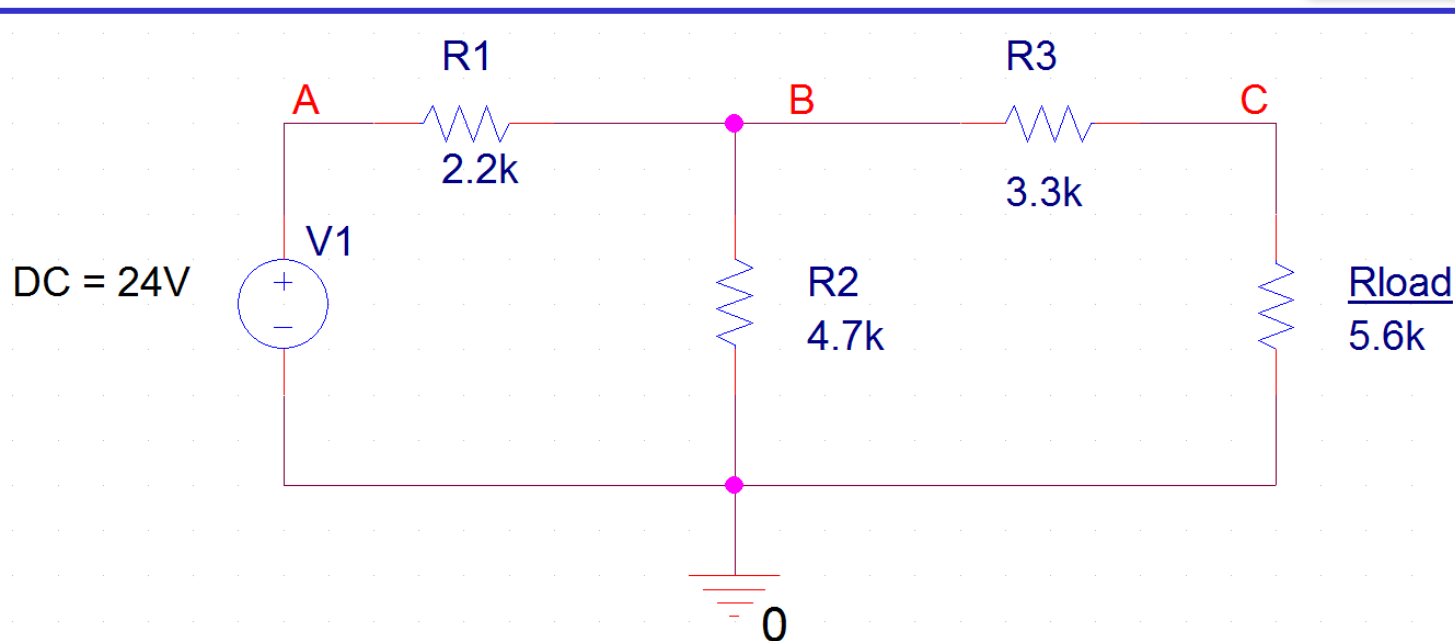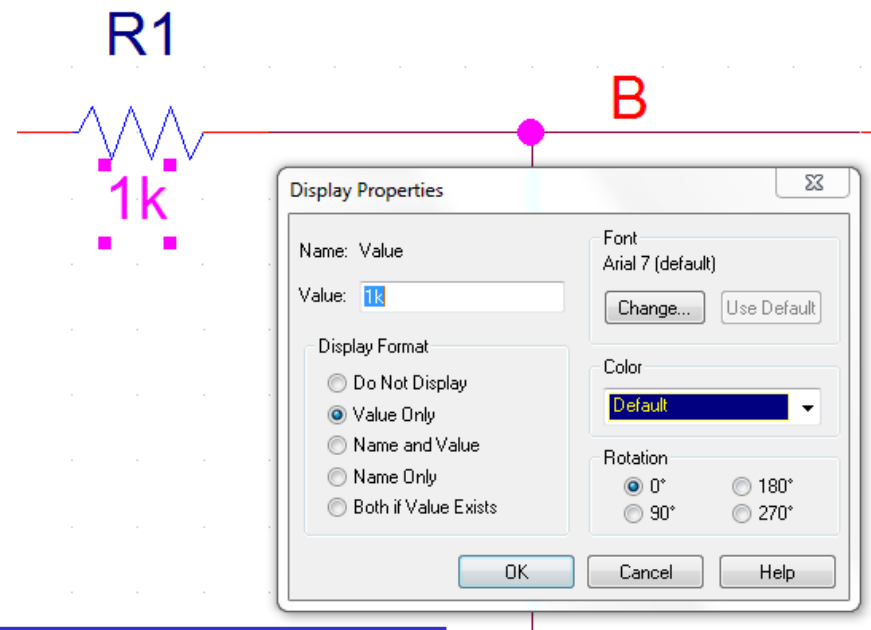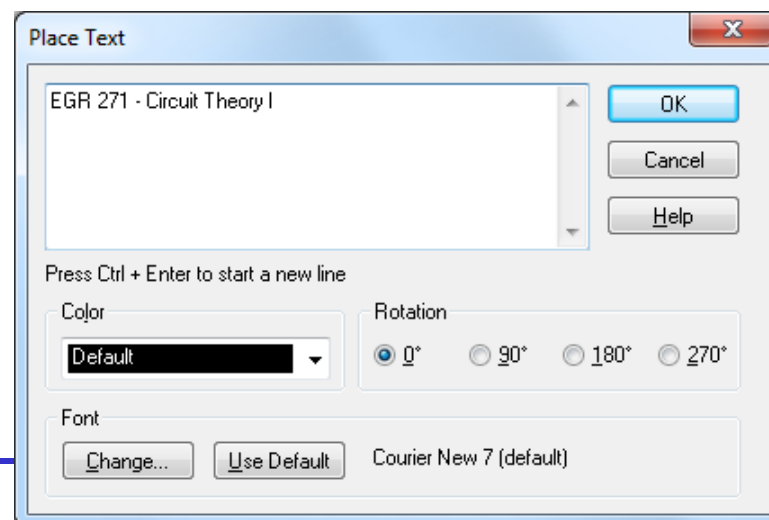K. Double-click on the values next to each component to change their value. Similarly, the names of the components can be changed. R4 was changed to Rload below.

Basic steps in creating and analyzing a circuit using PSPICE

2) Draw the schematic

    L. Add text to the schematic using the **Place Text** tool. Text can be placed anywhere on the schematic.

Place Text dialog box:

EGR 271 - Circuit Theory I

[OK]
[Cancel]
[Help]

Press Ctrl + Enter to start a new line

Color: Default

Rotation: ◉ 0°   ○ 90°   ○ 180°   ○ 270°

Font: [Change...] [Use Default]   Courier New 7 (default)

```
EGR 271 - Circuit Theory I
PSPICE Assignment #1
Problem 3-15
Purpose:  Determine the voltage across resistor R1
```

R1   2.2k
R3   3.3k
A   B   C
V1
DC = 24V
R2   4.7k
Rload   5.6k
0
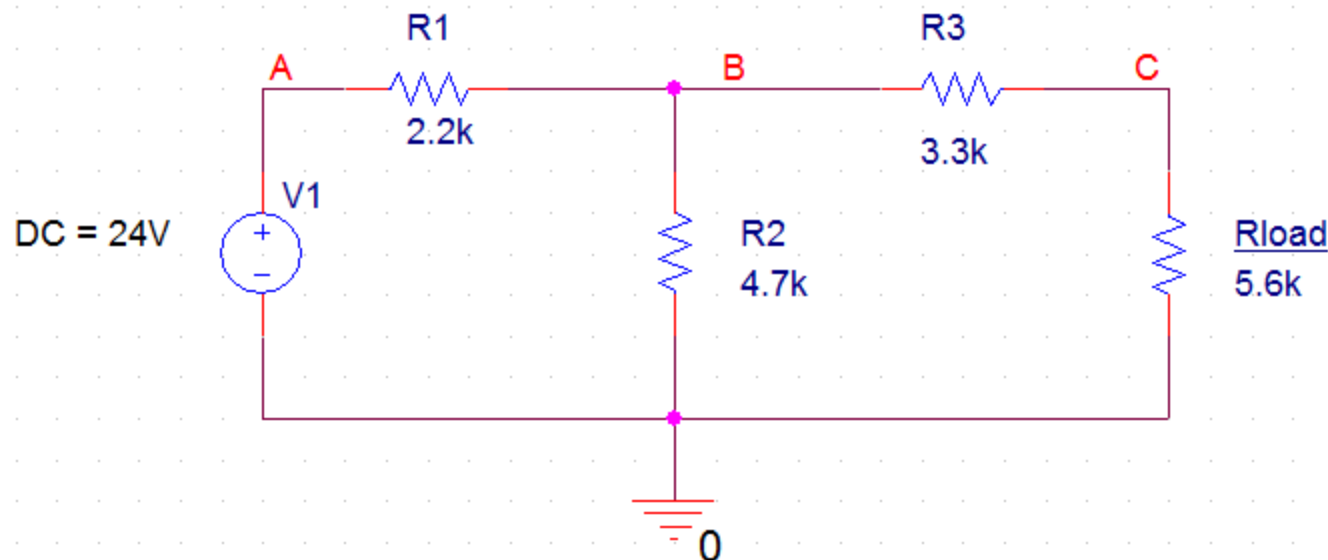
Place Text → abc

<u>Basic steps in creating and analyzing a circuit using PSPICE</u>

3) Create a simulation profile (select **PSPICE – New Simulation Profile** from the main menu)

A. A)   If the PSPICE menu doesn't appear, you may not have selected **<u>Schematic</u>** instead of **<u>Analog or Mixed Analog/Digital</u>** when you created the project, so start over and create a new project. (You can copy your schematic and paste it into the new project.)

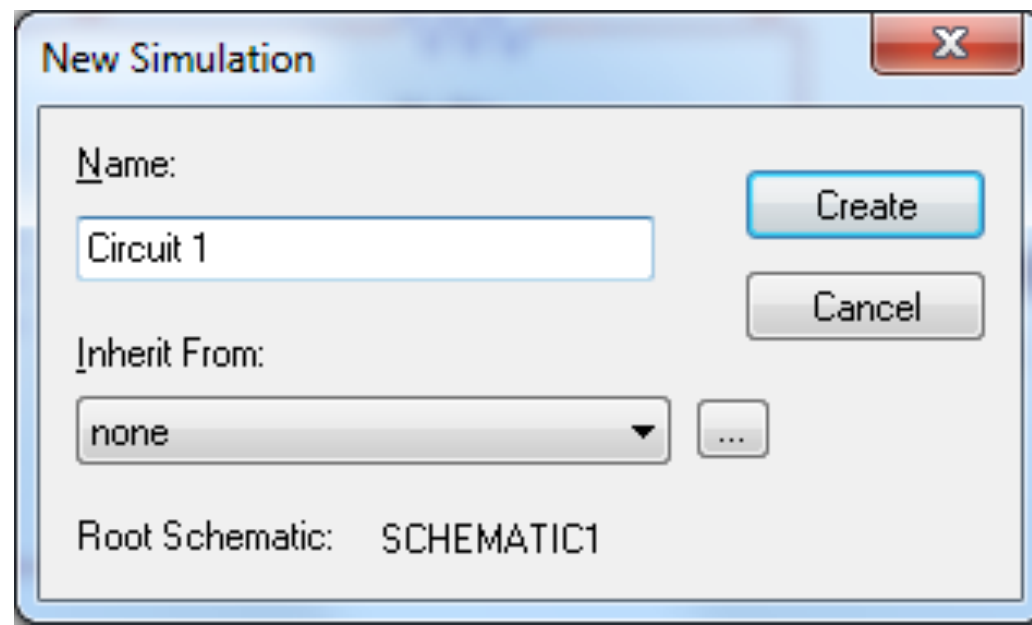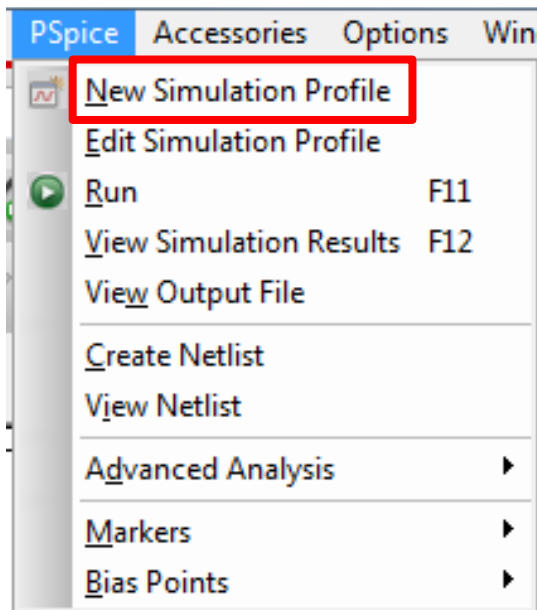B. Enter a **<u>Name</u>** for the Simulation Profile. Any name is OK.
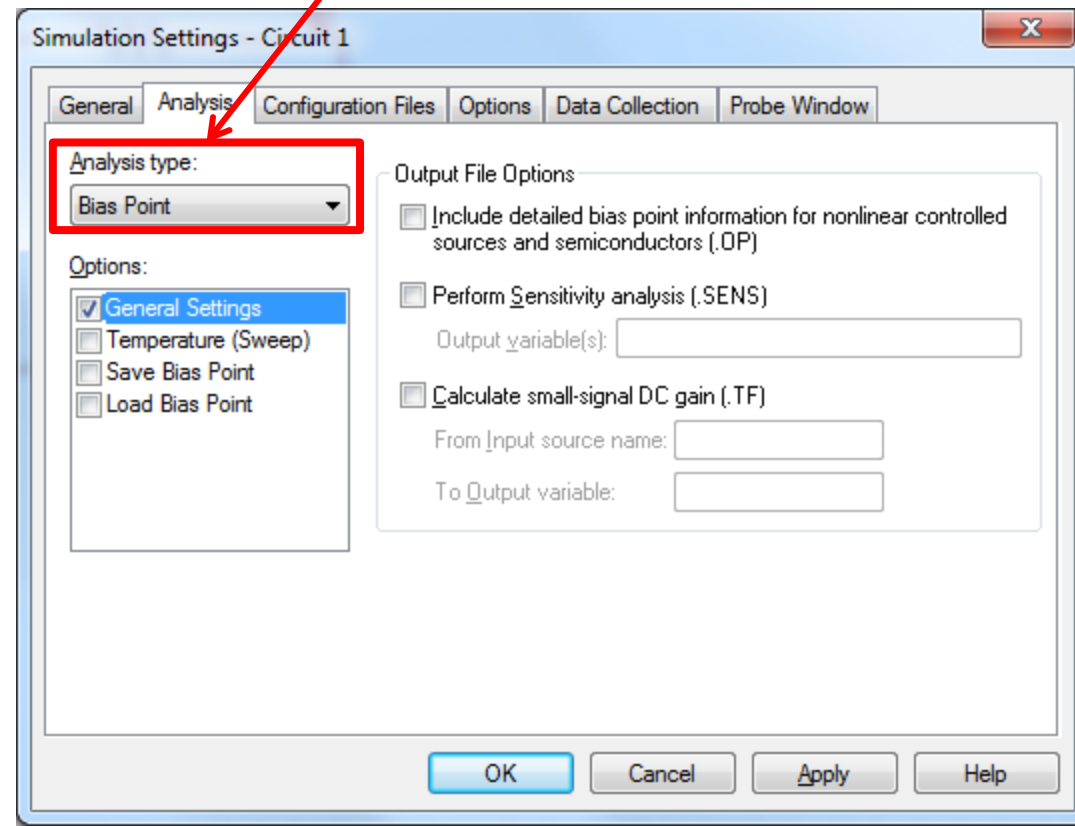
C. Select **<u>Create</u>**.

Basic steps in creating and analyzing a circuit using PSPICE

3) Create a simulation profile (select **PSPICE – New Simulation Profile** from the main menu)

   D. Select the type of analysis to be performed (**Bias Point**) and select OK. There are 4 types of analysis in PSPICE. This presentation will only focus on Bias Point analysis.

   - **Bias Point** (DC analysis where you place voltages, currents, and power on the schematic)

   - **DC Sweep** (vary a source or component)

   - **AC Sweep** (vary frequency)

   - **Transient** (vary time)

Change to Bias Point

Simulation Settings - Circuit 1

General | Analysis | Configuration Files | Options | Data Collection | Probe Window

Analysis type:
Bias Point

Options:
☑ General Settings
☐ Temperature (Sweep)
☐ Save Bias Point
☐ Load Bias Point

Output File Options
☐ Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)

☐ Perform Sensitivity analysis (.SENS)
   Output variable(s):

☐ Calculate small-signal DC gain (.TF)
   From Input source name:
   To Output variable:

OK   Cancel   Apply   Help

Basic steps in creating and analyzing a circuit using PSPICE

4) Analyze the circuit (select **PSPICE – Run** from the main menu or use the RUN button)

RUN button
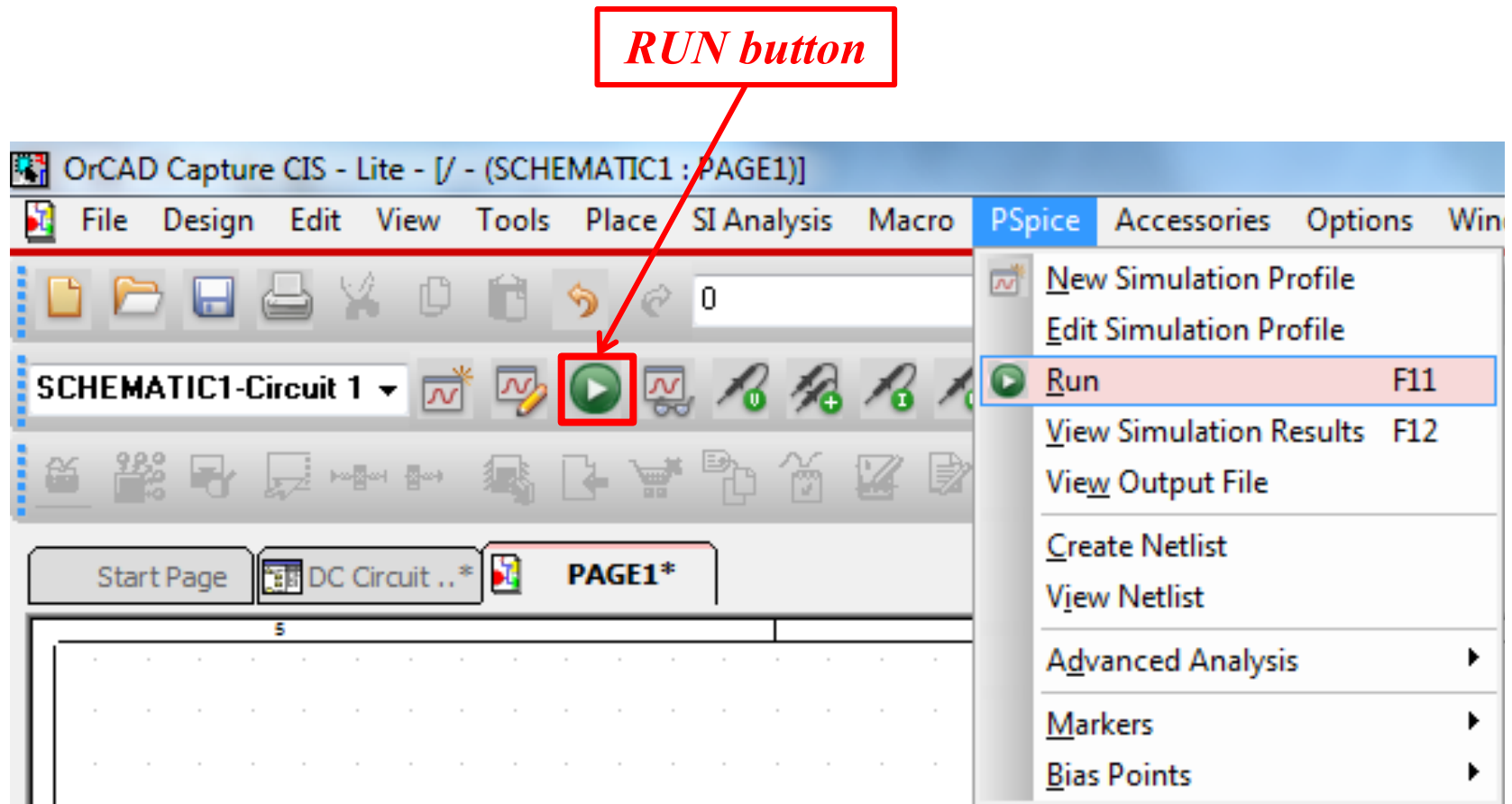
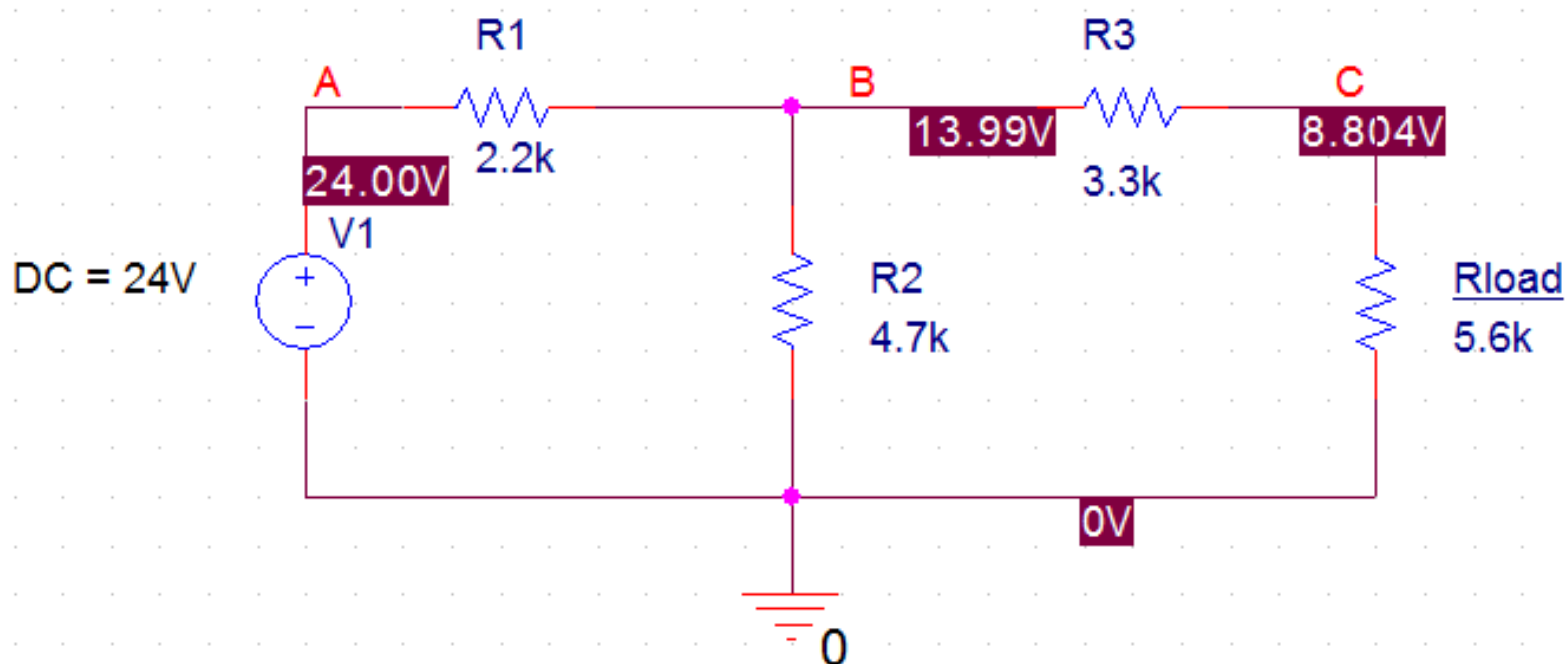## Basic steps in creating and analyzing a circuit using PSPICE

5) View the results. Note that some values may have appeared automatically on your schematic. Note voltages are displayed in the example below. We will see how to display other values shortly.
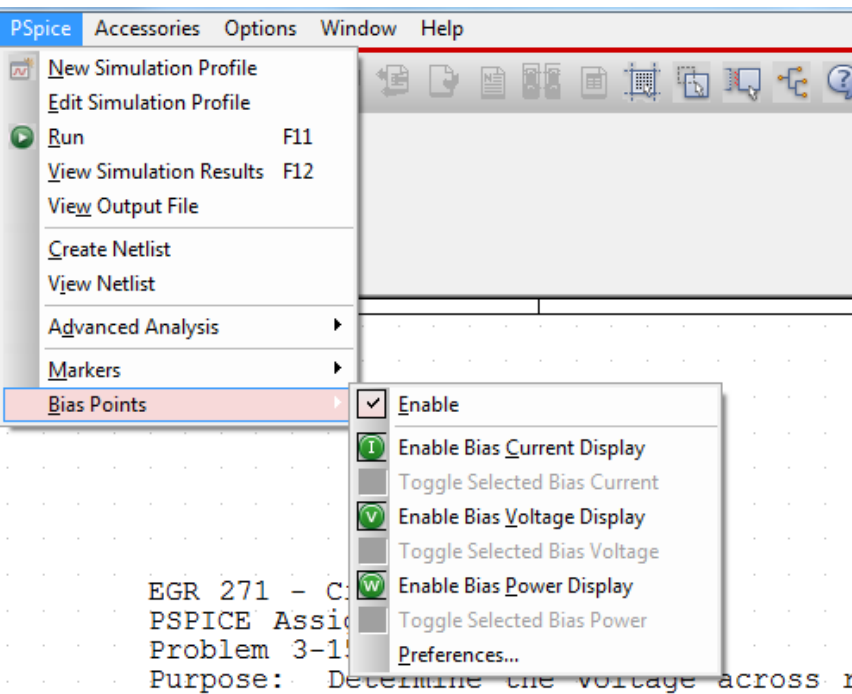
```
EGR 271 - Circuit Theory I
PSPICE Assignment #1
Problem 3-15
Purpose:  Determine the voltage across resistor R1
```
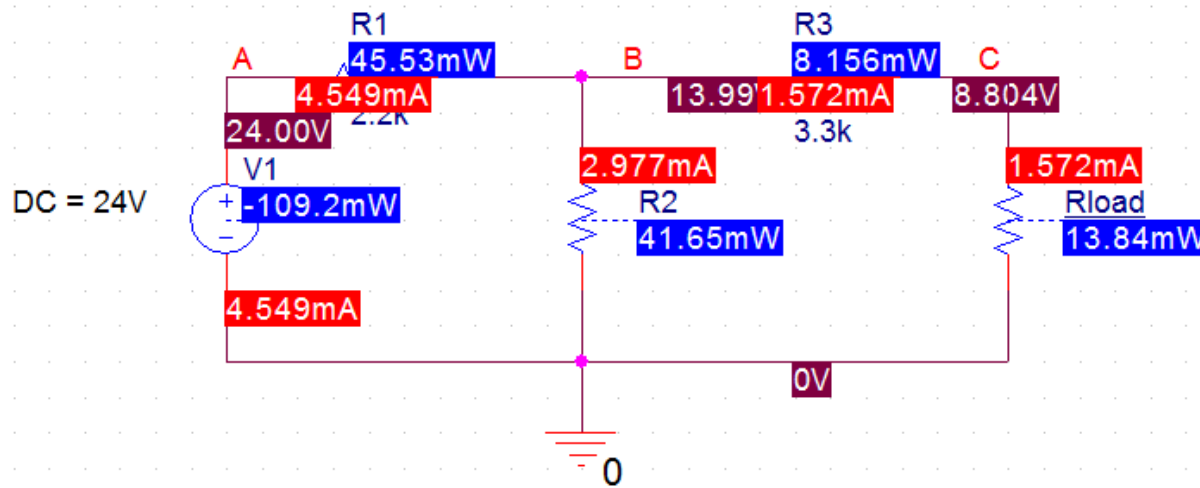
# Basic steps in creating and analyzing a circuit using PSPICE

A. Select **PSPICE – Bias Points** to select which values you would like to see on the schematic (node voltages, currents, and/or power absorbed). The schematic can get crowded with all values displayed as shown below.

PSpice   Accessories   Options   Window   Help

New Simulation Profile
Edit Simulation Profile
Run   F11
View Simulation Results   F12
View Output File

Create Netlist
View Netlist

Advanced Analysis   ▶
Markers   ▶
Bias Points   ▶

✓ Enable
ⓘ Enable Bias Current Display
   Toggle Selected Bias Current
Ⓥ Enable Bias Voltage Display
   Toggle Selected Bias Voltage
Ⓦ Enable Bias Power Display
   Toggle Selected Bias Power
   Preferences...

EGR 271 – C
PSPICE Assi
Problem 3-1
Purpose: Determine the voltage across resistor R1

Notes on ***bias point values*** displayed in PSPICE:

- **<u>Only node voltages can be displayed</u>**. Node voltages can be used to calculate component voltages. For example, the voltage across R1 (+ on left) is:

$V_{R1} = V_A - V_B = 24.00 - 13.99 = 10.01$ V

- **<u>PSPICE only shows positive current values</u>**. A dotted line shows which end of the component the positive current enters. As examples, the current through R1 is to the right, the current through R2 is down, the current through the 24V source is up, etc.

- **<u>PSPICE only shows power absorbed</u>**. Note that R2 absorbs 41.65 mW of power while the 24V source absorbs -109.2 mW (or delivers +109.2 mW).
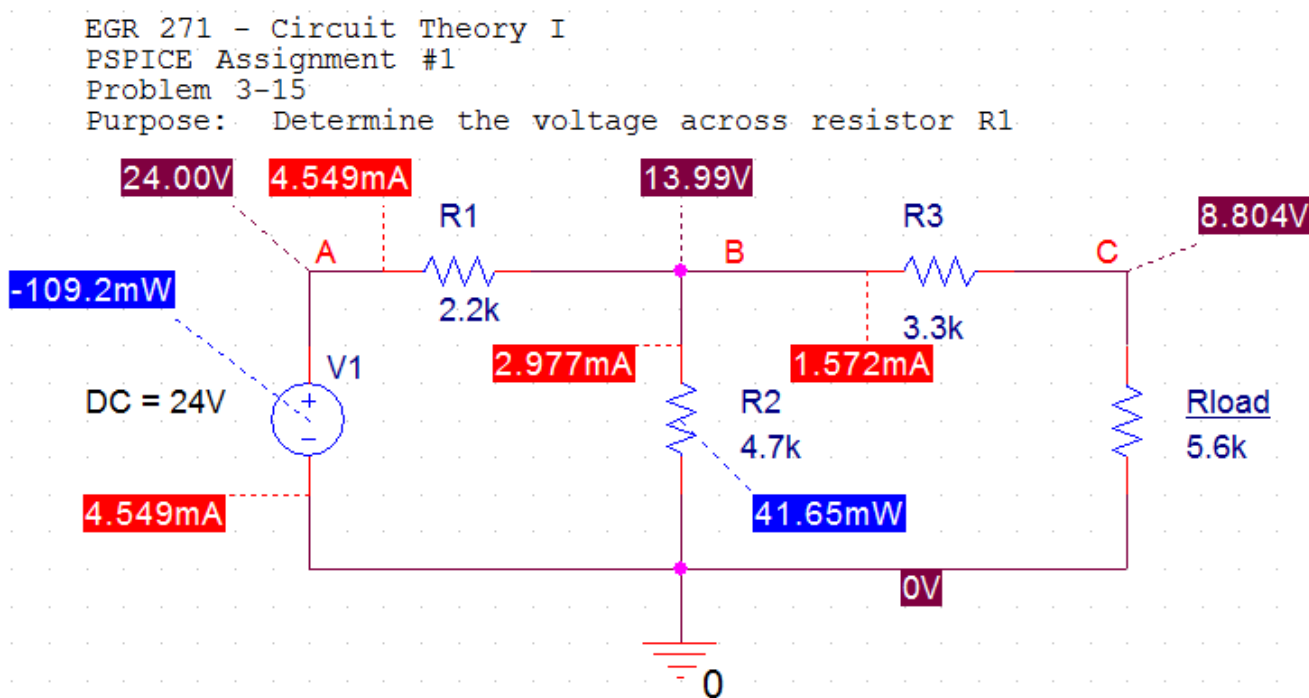
Component Values in PSPICE

- PSPICE is not case-sensitive

- Allowable prefixes are shown in the table below

Multiplier	SI Prefix	PSPICE notation
10^{+9}	giga, G	G (or g)
10^{+6}	mega, M	MEG (or meg) *
10^{+3}	kilo, k	k (or K)
10^{-3}	milli, m	m (or M)
10^{-6}	micro, μ	u (or U)
10^{-9}	nano, n	n (or N)
10^{-12}	pico, p	p (or P)
10^{-15}	femto, f	f (or F)

* Note that m or M are used for milli. Use MEG for mega, not M.

Component Values in PSPICE

When PSPICE reads a component value:

- No space is allowed between the value and the prefix or unit name

- PSPICE essentially only looks at the first letter after the value to see if it is a valid prefix. All other letters are ignored, so unit names can be entered in a variety of ways. Several examples of specifying a 100 kΩ resistor are shown below:
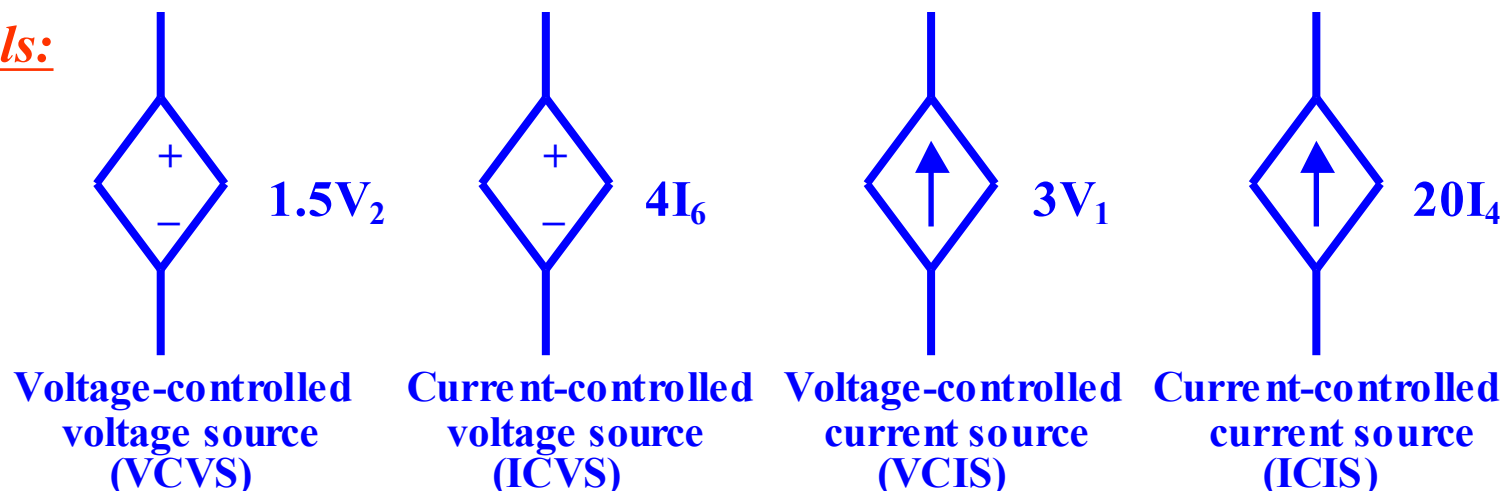
R2   R2   R2   R2   R2

1k   1K   1kOhm   1kiloOhm   1 k
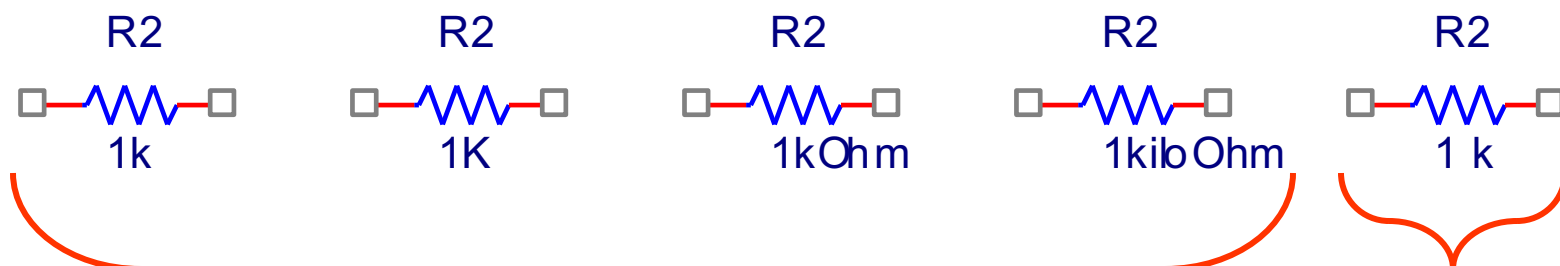
Valid resistor values

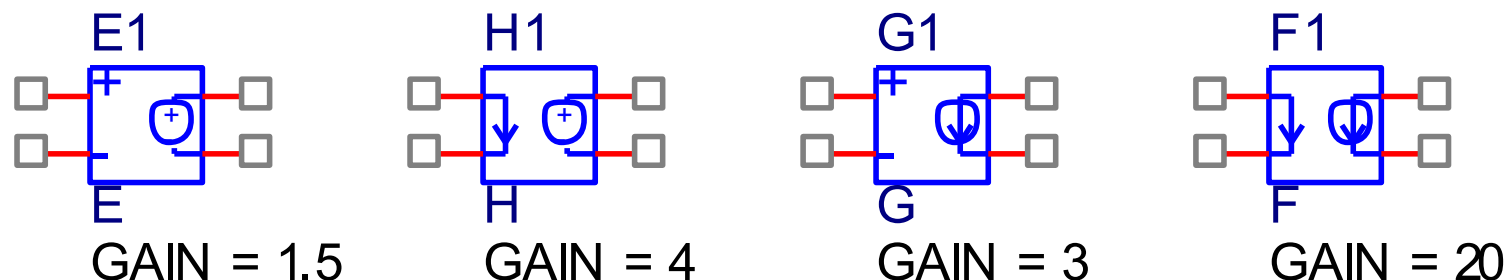*Invalid resistor value (no space allowed)*

Dependent sources in PSPICE

- Located in the analog library (**analog.olb**)
- Use E, F, G, and H (not EPOLY, FPOLY, GPOLY, and HPOLY)
- Each source has a ***round symbol that can be thought of as the source***. The other terminals are for the control variable.
- Double click on the source to set the GAIN property (default value = 1). Be sure to change the property settings so that the GAIN is displayed.
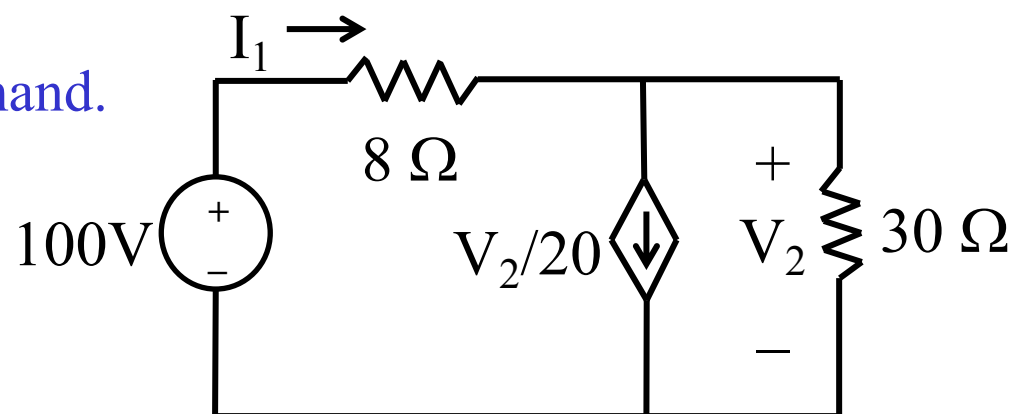
Textbook symbols:

$1.5V_2$	$4I_6$	$3V_1$	$20I_4$
Voltage-controlled voltage source (VCVS)	Current-controlled voltage source (ICVS)	Voltage-controlled current source (VCIS)	Current-controlled current source (ICIS)

PSPICE symbols:

E1	H1	G1	F1
E	H	G	F
GAIN = 1.5	GAIN = 4	GAIN = 3	GAIN = 20

Example using dependent sources
Analyze the circuit shown below by hand.
Verify the results using PSPICE.

$\underline{\text{KVL, outer loop}}: \; -100 + 8I_1 + V_2 = 0$

$\underline{\text{KCL, node A (out} = +)}: \; -I_1 + V_2/20 + V_2/30 = 0$

$$-60I_1 + 3V_2 + 2V_2 = 0$$

$$5V_2 = 60I_1$$

$$V_2 = 12I_1$$

Substitute the last equation into the KVL equation :
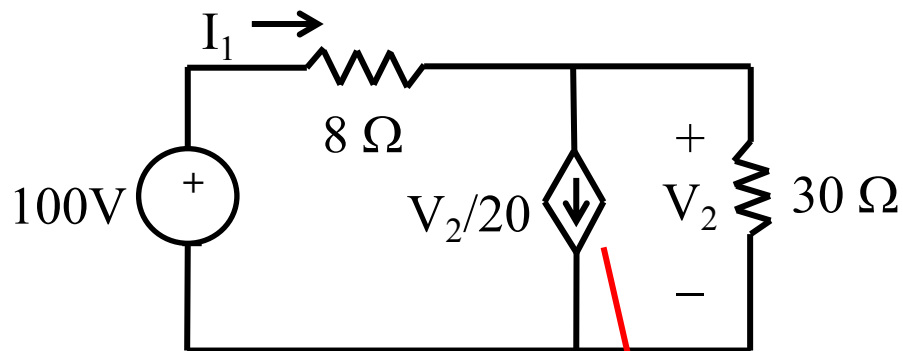
$100 + 8I_1 + 12I_1 = 0$

$20I_1 = 100$
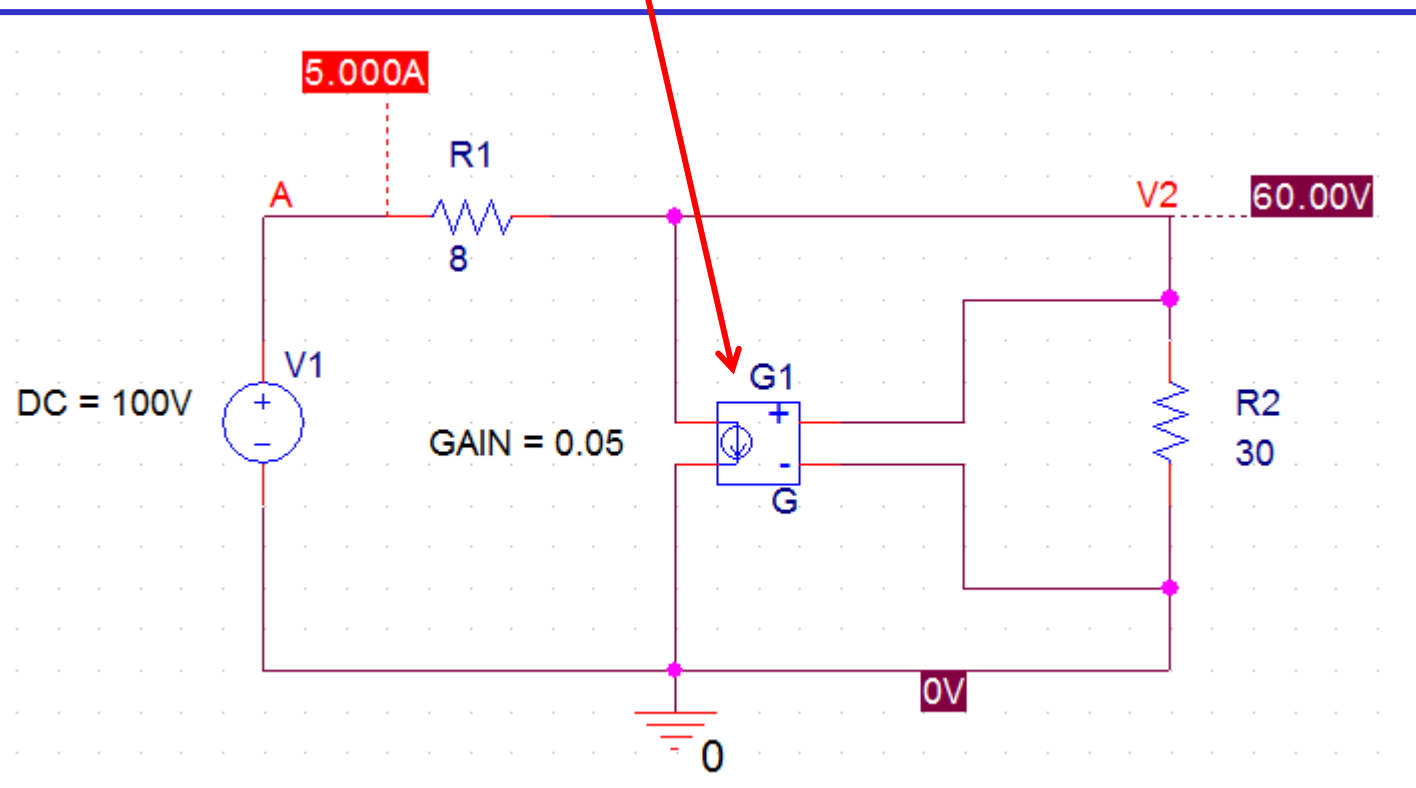
$\boxed{I_1 = 5A}$

and $V_2 = 12I_1$, so

$\boxed{V_2 = 60V}$

Example using dependent sources (continued)

Notes:

- Round symbol in the dependent source in the top schematic corresponds to the diamond shape in PSPICE
- Gain set to $1/20 = 0.05$
- Wires used to connect the control voltage, V_2, to the source

PSPICE results match the hand results!

PSPICE Demonstration

If time allows and computers are available to the class, the instructor will lead the class in creating and analyzing circuits using PSPICE.

Demonstration topics:

- Launching the software and creating a project
- Creating and analyzing schematics
- Labeling nodes
- Using dependent sources
- Changing and displaying properties. In general, display any property that you set or change.
- Adding text to a schematic
- Displaying results with a Bias Point analysis
- Calculating other voltage, current, and power values from the displayed bias point values
- Copying schematics to other documents