Laboratory Goals

- Introduce windows-based PSPICE as a valuable design tool
- Create simple discrete circuits using PSPICE
- Simulate output response for the designed circuits

Pre-lab reading

- Read the pre-lab introduction below
- Visit the Cadence website (maker of PSPICE)

Equipment needed

- Lab notebook, pen
- Workstation PC, with PSPICE application

Parts needed

- No electronic parts are needed for this lab

Lab safety concerns

- There are no specific safety concerns for this lab

1. Pre-Lab Introduction

PSPICE is useful way of verifying your lab test results, and experimenting with changes to your own circuit designs. It is also widely used in industry for simulating designs prior to production. Here are a few words about PSPICE:

Part libraries:
The complete version of PSPICE has thousands of parts, but finding those parts can be difficult. You need to either know what the part is called in PSPICE, or be able to provide a good description of the part. The search engine is picky about your descriptions.

For parts such as resistors, capacitors, and inductors, PSPICE uses the names commonly used in industry. For example resistors are **R**; capacitors are **C**; and inductors are **L**. You can find these parts in the parts library by typing **R**, **C**, or **L** in the **Part Name** box of the **Get New Part** menu.

Parts like transistors or op-amps must be specified exactly as the actual parts are named. For example, you need to type **2N2222** in the **Part Name** box for the transistor of that name, not **2222**. Similarly for the '324 op-amp, you need to specify **LM324**. If you type **transistor** in the **Description Search** box of the **Get New Part** menu, you may get hundreds of results.

The best way to learn about the part libraries, and locating parts in them is just to explore this part of PSPICE on your own!

Simulations:
Before performing any simulations *that require graphical output*, you must first tell PSPICE how you want to display the results. The two most common types of AC simulations are the **Transient** and **AC Sweep**. (These are found by selecting **Setup** from the **Analysis** menu.) The **Transient** simulation displays voltage or current vs. time; the **AC Sweep** simulation displays voltage or current vs. a range of frequencies. *Note that you do not have to use these options for DC analysis.*

2. PSPICE Design and Simulation for Circuit 1

(Refer to Figure 1 below)

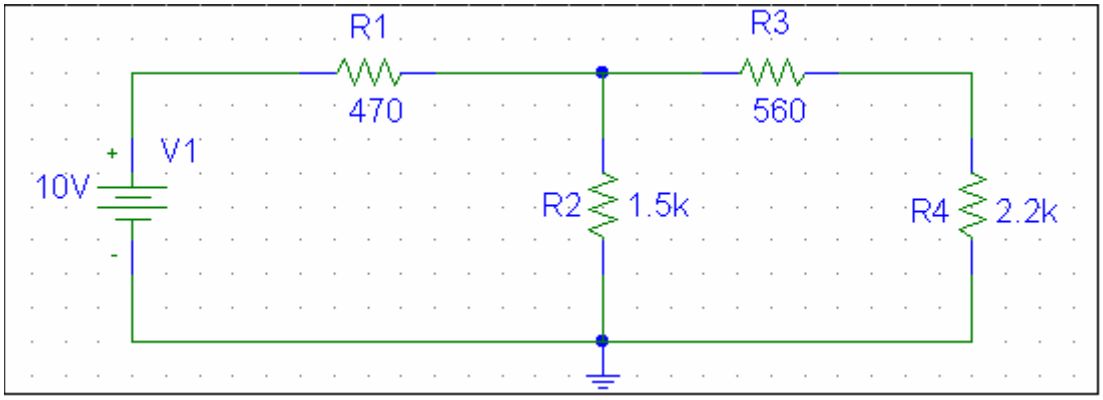

Figure 1 – PSPICE Schematic of a Resistive Network

You will be following along with your teaching assistant's presentation during this lab

- Create a folder in **My Documents** for your PSPICE designs to be stored. Use the name of your lab section in the folder name (e.g., "206_001")

- Open the PSPICE application called **Schematics**
- Click the **Get New Part** button (or use **CTRL+G**)
- If the pop-up window is titled **Part Browser Basic**, click the **Advanced** button
- In the **Part Name** box, type the letter **R**, to locate the generic resistor
- Click the **Place & Close** button
- Move the resistor (**R1**) to where you want it placed, then click the mouse button
- Press the **Ctrl+R** buttons to vertically rotate the next resistor to be placed
- Place resistor **R2** on the schematic with a mouse click
- Rotate **R3** horizontally, and place it on the sheet
- Rotate **R4** vertically, and place it on the sheet
- Press the **ESC** button to stop working with the part (or right-click the mouse)

- Click the **Get New Part** button
- In the **Part Name** box, type the letters **VDC**, to locate the DC voltage source
- Click the **Place & Close** button, then place the part on the sheet
- Press the **ESC** button to stop working with the part

- Get the **EGND** (earth ground) part from the library (note all schematics must contain a ground, or they won't simulate!!)
- Attach the ground to the bottom of **R2**
- Press the **ESC** button to stop working with the part

- Click the **Draw Wire** button
- Place the mouse pointer on the right end of **R1** and click the mouse button. Drag the wire over to the left end of **R3**, then click again to connect the two parts
- Finish wiring the components as shown in Figure 1 (don't worry if your schematic doesn't look exactly like the figure-we will fix it soon)
- Press the **ESC** button to stop working with the wiring tool

- Try deleting a wire by clicking on it, then clicking the **Cut** button
- Undo the cut by clicking the **Undo** button

- Highlight any wire or component by clicking on it once
- Click and hold the mouse button on the selected part, then drag the part to where you want it. Release the mouse button
- Move any of the remaining parts, or wires until you are happy with the look of your schematic
- Click the **Zoom to fit page** button to increase the size of your drawing

- Double click the **0V** label on the **VDC** part. Change the **0V** in the **Value** box to 10V and click **OK** (note that the voltage could also be set to a negative voltage)
- Double click each resistor value and change them to the values shown in Figure 1 (e.g., for **R1**: enter **470** for 470 Ohms; **R2**: enter **2.2k** for 2.2k Ohms)

- Save the circuit schematic in your folder (give it a name you'll remember!)

- Click the **Simulate** button (or press **F11)** to perform a DC analysis
- Minimize the new **PSpice A/D** window
- Click the **V** or **I** buttons to display node voltages and branch currents of the circuit (if they are not already displayed)

- To print your schematic:

    - Move the mouse pointer to the upper left corner of the schematic
    - Press and hold the mouse button, then drag a box just larger than all of your schematic, then release the button
    - Select the **Print** option from the **File** menu
    - Click **OK** to print from the print menu

*You can also import your schematic to a Word document by dragging a box around it and then selecting the **Copy to Clipboard** option from the **Edit** menu*

3. PSPICE Design and Simulation for Circuit 2

- Create the voltage-controlled voltage source shown in Figure 2 below. (Note: E1 is called **E** in the parts list; and the analog ground is called **AGND**)
- Double click on the **V1** label of the **VDC** icon. Label it "VBATT"
- Set **V1** to 10V
- Save the circuit schematic in your folder
- Press **F11** to simulate DC analysis

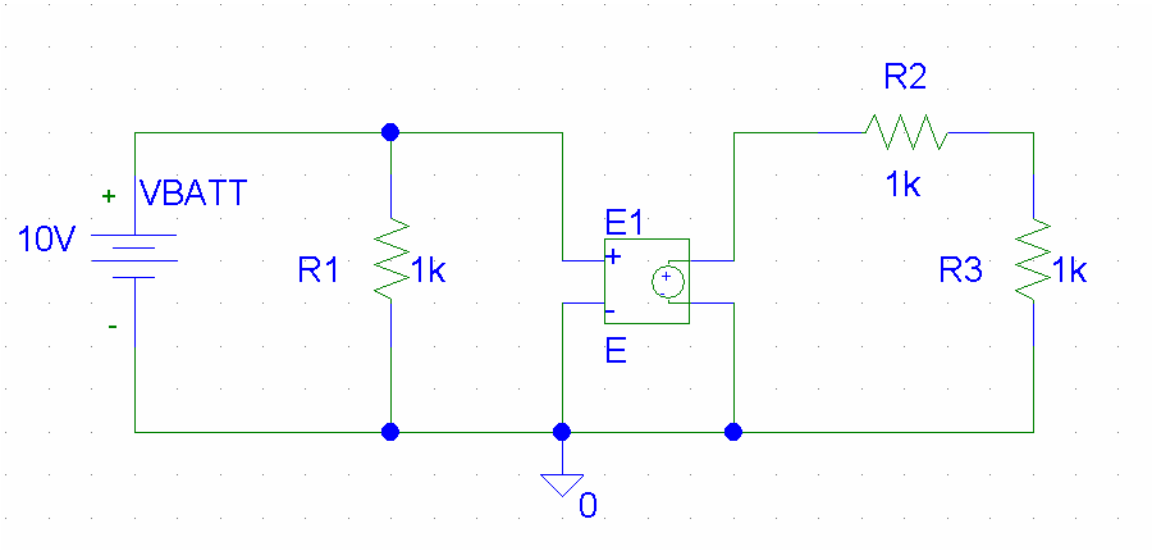

Figure 2 – PSPICE Schematic of a Voltage-Controlled Voltage Source

4. PSPICE Design and Simulation for Circuit 3

- Next, you will use what you have learned to create the inverting op-amp shown in Figure 2 below.

- Construction hints:

  - Click on the **New Schematic** button to start a new sheet
  - The most recently used parts are also available in the window to the right of the **Get New Part** button, saving you a trip to the part libraries
  - You will need to rotate (**Ctrl+R**), then flip (**Ctrl+F**) the op-amp part in order for it to look like the op-amp below
  - To add the text shown, click the **Draw Text** button, type the text, then move the text to where you want it and click the mouse

- The parts you need from the library are:
  - **LM324** (op amp)
  - **VSRC** (This part can be set for either DC or AC. We will set it for + or – 15VDC)
  - **VSIN** (This is the input signal. We will set it to: DC=0, AC=1V, VOFF=0, VAMPL=1V, and FREQ=1k)
  - **R** (Make **R1** and **R3** = 1k, and **R2** = the value you used in lab 3)
  - **AGND** (Analog ground. This part is similar to the **EGND** used before)

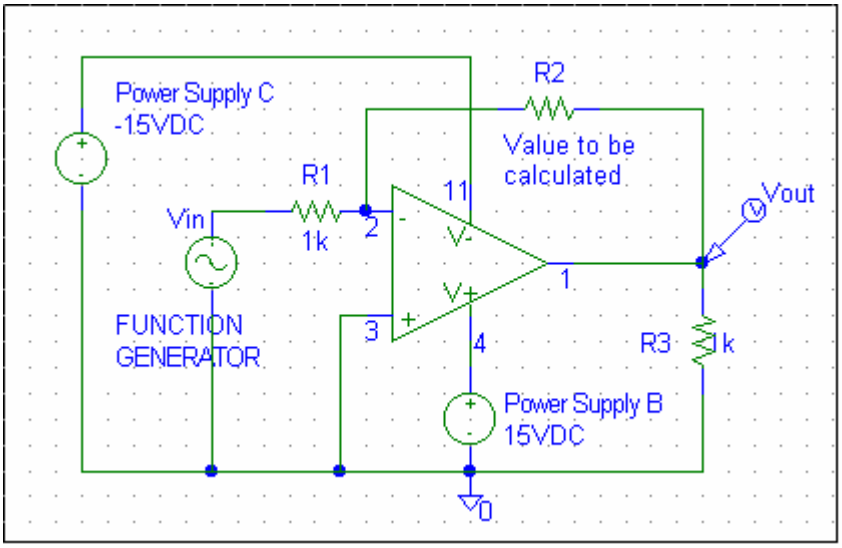

Figure 2 – PSPICE Schematic of an Inverting Op Amp

- Click the **Voltage/Level Marker**, then place it as shown in Figure 2

- To configure the output graph:

    - Click the **Setup** option from the **Analysis** menu
    - Click on the box next to the **AC Sweep** option (a check mark appears)
    - Click the **AC Sweep** option
    - Click the **Decade** option to create a logarithmic x-axis scale
    - Enter 2000k (or 2MEG) in the **End Freq** box (i.e., set the x scale from 10Hz to 2MHz), then click **OK**
    - Close the **Analysis Setup** window

- Save the circuit schematic in your folder
- Click the **Simulate** button (or press **F11)** to perform the AC analysis (a graph of the marker voltage vs. frequency appears)

- Minimize the **PSpice A/D** window

- Delete the voltage marker from the schematic
- Replace the marker with the **VDB** voltage marker, found in the **Markers**, **Mark Advanced** menus (dB is a common gain measurement used instead of voltage)
- Click the **Simulate** button (or press **F11)** to perform a new AC analysis

- To add a second plot to the page (for the phase):

    - Select the **Add Plot to Window** option of the **Plot** menu
    - Select the **Add Trace** option from the **Trace** menu
    - Uncheck all boxes except **Analog** and **Voltages**
    - Select the node voltage called **V(R3:2)**
    - Under **Trace Expression**, insert the letter **p** so the node now reads **Vp(R3:2)**
    - Click OK to add the trace

- Select the **Print** option from the **PSpice A/D File** menu
- Print the op-amp schematic

5. Analysis

Write a brief summary report for this lab. Be sure to also include the following topics:

Compare the results from Lab 2 to the PSPICE model you created in step 1 above. Record your findings in your lab notebook

Compare the results from Lab 4 to the PSPICE model you created in step 3 above. Record your findings in your lab notebook

Explain any difficulties you had with this lab. (Please include suggestions to improve the lab, if you have them).