

ECE 220 PSpice Homework

Voltage Divider and Current Divider

Michael W. Marcellin

In this tutorial assignment, you will learn to perform a simple circuit simulation in PSpice. You will learn more advanced techniques in your laboratory assignments.

PSpice is a circuit simulator. In the old days, you had to enter your circuit diagram information by hand as a netlist. Nowadays, we have a graphical interface to simplify the entering of a circuit diagram. This graphical interface is called Capture. We really interact mostly with Capture, but the whole package is often still called PSpice for historical reasons.

You can use the computers in room ECE 232 to do all of your PSpice work. These computers are supported by the IT staff. They ensure that all software works properly. If you so choose, you may download PSpice and install it on your own computer. You can get instructions to install PSpice on your computer from the ECE 220 website by following the link for Lab Resources.

Voltage Divider

The first circuit we will simulate using PSpice is a simple voltage divider. This circuit is shown below in Figure 1.

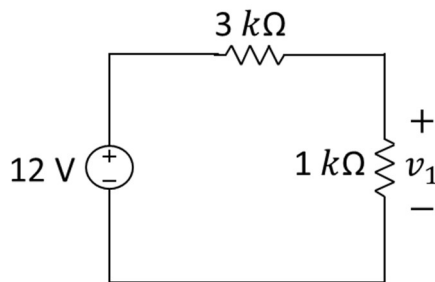


Figure 1. Voltage divider.

From class, we know (or will learn) that $v_1 = \frac{1k}{1k+3k} 12 = 3 \text{ V}$. There is really no reason to simulate this simple circuit other than to practice PSpice where we already know the answer, and can easily check our work.

To begin, click

Start -> Cadence Release 17.2-2016 -> Capture CIS (Lite)

In some versions of PSpice, a window may open with two choices. If this happens, choose Allegro PCB Design CIS L

Note: You may see a different version number than "17.2."

To open a new project, click

File -> New -> Project...

A new window will open. Type a name for your project. Click the button for "Analog or Mixed A/D" Choose a location to save your work and click "OK"

Another new window will open, click the button for "Create a blank project" click "OK"

A white screen with a grid of black dots will appear. You draw your circuit on this screen.

To place components on the screen, click

Place -> Part...

A new panel will open on the right portion of the screen.

Begin to type a part name (see useful part names below). The list of parts will automatically scroll. **** When you see the name of the part you want, double click it. Your mouse cursor will change to reflect the circuit symbol you have chosen. You can rotate the symbol by pressing the 'r' key on the keyboard. After you get the desired rotation, you place the rotated part on the screen by clicking at your chosen location. When you are finished, press 'esc' on the keyboard. You may then select a different part to place.

Useful parts for this exercise include:

VDC (voltage source) – ignore /Design Cache, /ANALOG, /BREAKOUT, etc. These are library names.

R (resistor)

Let's begin by placing a voltage source on your schematic. The circuit symbol that PSpice uses for a voltage source is shown below in Figure 2. Note that this circuit symbol is different than the one we use in class, but is commonly used in other textbooks.

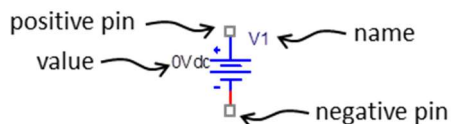


Figure 2. PSpice circuit symbol for voltage source.

Once you place a voltage source on your schematic, be sure to press esc on your keyboard. You can then adjust the parameters of the voltage source. There are 5 attributes that you can select on the voltage source: the name, the voltage value, the positive voltage pin, the negative voltage pin, and the voltage source symbol itself. For this circuit element, you will usually only change the voltage value (required), and the name (optional). Double click carefully on the voltage value. A new window will open where you can change the voltage value from 0Vdc to 12Vdc as needed for the circuit of Figure 1.

**** If there are no parts, go to the Lab Resources page of the class website and click the link "What to do if there are no parts in PSpice." Follow the instructions you find there.

A word about units. Pay close attention to this. There are a couple of opportunities to make mistakes. PSpice uses the following abbreviations for unit prefixes:

Prefix	Abbreviation
femto	f
pico	p
nano	n
micro	u
milli	m
kilo	k
mega	meg
giga	g
tera	t

The ones that will give you trouble are micro, milli, and mega. PSpice is not case sensitive. So, “m” is not micro – it is milli
“M” is not mega – it is milli

PSpice ignores the units themselves (or anything else that may come after the prefix). For example, for a current source, 12mA dc is the same as 12m. One case where this can get you into trouble is with a capacitor. Think about 1F... How will PSpice interpret this? Is it 1 Farad? Nope... it is the same as 1fF which is 1 femto-Farad.

Now place the resistors on your schematic and change their values to those shown in Figure 1. Note: You can place both resistors without pressing esc in between.

To wire your parts together, click

Place -> Wire

Click on a device pin to begin wiring. Move the mouse to an intermediate location and click to put a bend or a corner in a wire. Click on another device pin or wire to complete the connection. Be very careful that your wires actually make a connection. Hint: watch for a red dot.

Important Note: In PSpice, every circuit must have a ground. The simulation will not function without a ground. To include a ground, click

Place -> Ground

A new window will open. Choose one of the grounds named 0 (e.g., 0/SOURCE, 0/CAPSYM, 0/Design Cache) and click “OK”

None of the other choices for ground will work.

If this ground is missing, see the instructions for installation of PSpice libraries mentioned earlier in this document.

Place a ground on your schematic and connect it as shown below in Figure 3. This “ground” should be thought of as a point in the circuit that we are defining to be 0 V. We can put this

point anywhere in the circuit that we choose. For now, place it as shown in the figure.

A good analogy for the PSpice 0 V ground can be explained in terms of the (arbitrary) reference point used to define altitudes. Altitudes do not make sense without a reference. For example, if I want to talk about the altitude of the ceiling in my room, I could say that the ceiling is 10 feet above the floor. I could also say that it is 2,410 feet above sea level. In the first case, the reference (0 feet) is the floor. In the second case, the reference (0 feet) is sea level. Voltages in PSpice are much the same. They are specified with respect to the reference (0 V) defined by the ground.

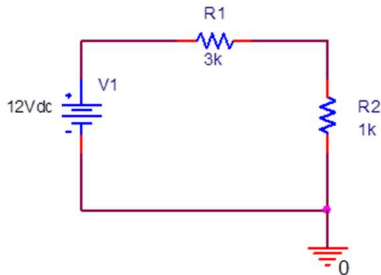


Figure 3. PSpice schematic for the voltage divider of Figure 1.

More notes: There are many other ways to accomplish the tasks described above. For example, see the icons on the right portion of the screen. There are also many hot-keys. We mentioned 'r' above which rotates parts. Additionally, 'v' will flip a part vertically, while 'h' will flip a part horizontally. Pressing 'g' will open the window for placing grounds. To zoom in, press 'i'. To zoom out, press 'o'. You can drag parts on the schematic to relocate them. You can also drag labels, values, etc. Any item can be removed from the drawing by clicking it and pressing delete on the keyboard. You can redraw the screen to clean up any extraneous marks by pressing 'F5'.

Let's now use PSpice to find the voltages and currents in our circuit. Click

PSpice -> New Simulation Profile

Enter a name for this profile (e.g., VDiv)

Click Create

A new window will open. You might have to look at the icons at the bottom of the screen to find it.

In the drop down menu entitled Analysis Type:, choose Bias Point

This tells PSpice that we are going to compute the static DC values of voltages and currents in the circuit.

Check the box below the drop down menu that says "Save Bias Point."

Click "OK"

Here we go.

Click

PSpice -> Run

A new window will open. You may need to look at the icons on the bottom of the screen to find it. For this simple simulation, the only thing of interest is the text in the bottom center which may give you some hints if you have errors in your circuit drawing. If the simulation fails to run,

check your work. Common errors include: no ground, wrong ground (see discussion on grounds above), wires/pins not connected properly (upon failure, PSpice usually will draw some circles on your schematic to highlight this latter problem). You may close this window and return to your schematic.

Near the top of the screen there are 2 green buttons, labeled **V** and **I**, respectively. Click the button labeled **I** a few times. It turns on and off labels near one pin of each circuit element. This label tells you the current flowing into that pin of the device. Note carefully the position of these labels and make sure you understand the sign convention. When you are finished, leave the labels turned off.

Repeat with the button labeled **V**. This turns on and off labels that tell you the voltage value at each node in the circuit. These voltages are given with respect to ground, which is defined to be 0 V. If you want to know the voltage across a circuit element, you can take the difference between 2 node voltages. For example, the voltage the across the 1 kΩ resistor is $3 - 0 = 3$ V, with positive polarity on the top, as expected. Similarly, the voltage across the 3 kΩ resistor is $12 - 3 = 9$ V, with positive polarity on the left. Make sure you understand this. Think about KVL.

a) Print a copy of your schematic with the **voltage labels** turned on (File -> Print).

Experiment with moving the ground to a different node. Recall from the discussion above that this defines a new reference for 0 V. Run the simulation again (PSpice -> Run). Make sure you understand how the voltage labels change. Note that the voltages across the individual circuit elements do NOT change.

Current Divider

Repeat the steps above for the current divider circuit shown below in Figure 4. The part name for the current source is IDC.

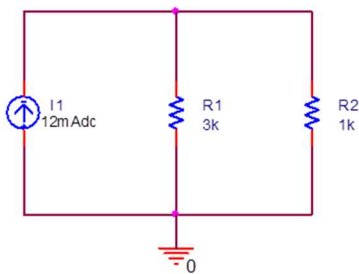


Figure 4. Current divider circuit.

b) Print a copy of your schematic with the **current labels** turned on.

Turn in the printouts from a) and b) above in class.

One last note: You can easily copy/paste drawings from PSpice into MS Word or Powerpoint. To copy a drawing of interest, stretch a rectangle around it and then press ctrl-c.