Physics 3150, Laboratory 1 Voltage Dividers, Circuit Simulation, and the Oscilloscope

by Ed Eyler Last revised Jan. 19, 2016

Purpose:

- 1. To gain a basic familiarity with the basic equipment used for Physics 3150 labs breadboards, signal generators, and oscilloscopes.
- 2. Use the web-based PartSim program, a graphical user interface to NGSPICE, to conduct numerical simulations of the circuit for both dc and ac excitation.

References:

As soon as you have an opportunity, read the description of the Tektronix TDS210 oscilloscope provided on the course web page, which is copied from *Hands-On Electronics* by Kaplan and White. It is equally applicable to the TBS1022 and TBS1052 scopes, which are more-capable successors to the TDS210 with added features. An especially important addition is that a front-panel USB connector is now provided, allowing oscilloscope data to be easily stored on flash drives in a comma-separated-variables (.csv) format that can easily be converted to a spreadsheet file. For a quick introduction to the basics of electronics, Chapter 1 of either the Eggleston or Meyer textbooks are very helpful. Finally, to learn more about NGSPICE computer circuit simulations and the PartSim user interface, refer to the PartSim web page and, optionally, to the NGSPICE tutorials at http://ngspice.sourceforge.net/tutorials.html.

Equipment:

- 1. Digitally synthesized signal generator, (Instek AFG2105 or AFG2112). We also have numerous analog function generators, including a basic version built into the breadboard, but the digital instruments are far more accurate and stable.
- 2. Digital oscilloscope, Tektronix TBS1022 or TBS1052.
- 3. Digital multimeter ("DMM") used as voltage meter and current meter.
- 4. Global Specialties PB-503 "breadboard" with jumper wire kit.
- 5. Computer with internet access, to run the online SPICE simulation program available at www.partsim.com/.
- 6. A selection of 1/4W or 1/2W resistors.
- 7. Coaxial cables.

Theory:

There's not a lot to say about the dc voltage divider shown in Fig. 1, which is perhaps the simplest of all circuit configurations. If no loaded is connected, the output voltage is a fixed fraction of the input voltage:

$$V_{\rm out} = \frac{R_2}{R_1 + R_2} V_{\rm in}.$$

If a finite load impedance is present, it appears in parallel with R_2 , changing its effective value. If R_2 is small enough that this effect is insignificant, the voltage divider is said to be *stiff*, meaning its behavior is independent of the load conditions within the desired tolerance.

In this lab we will also perform basic circuit simulations with SPICE, using a nice web-based graphical user interface provided by the PartSim program. The SPICE simulations are essentially just time-stepped numerical solutions of the circuit equations, and are reasonably straightforward when no nonlinear devices such as diodes or transistors are present. Detailed descriptions are provided in the tutorials and documentation listed under "References" above.

Procedure:

A. Breadboard-based circuit measurements

Before beginning, make sure you understand how the breadboard terminals are connected. A very brief introduction is provided as an Appendix to this lab writeup, and your instructor will provide more details. As you construct your circuits, try to develop the habit of keeping the physical layout of the components and wires simple and easy to follow, which will become vital to your sanity later in the semester.

1. Construct a voltage divider using two 1 k Ω resistors. If none are available, 1.1 k will work just as well. If you're not used to the color coding for resistor values, check with your T.A. or see pp. 5–6 of the Eggleston text. Note that the input and output voltages are referred to ground, indicated by the symbol with three horizontal bars. This is very common in circuit designs, and is used to designate a common reference point that is physically connected to earth ground through the power supply cord.



Figure 1. Screen view from PartSim, showing a voltage divider with the output from a dc simulation.

Connect a 5 V dc supply to the circuit, such as the one provided on the breadboard. Measure

both V_{out} and Vi_n with a DMM. Confirm that the voltages are in the expected 1:2 ratio.

2. Rebuild the circuit using a pair of 510 k Ω resistors, and remeasure the voltages. You will find that the output voltage isn't quite right, due to the finite input resistance of the DMM input terminals. This resistance is often referred to as the *input impedance* of the instrument, although more properly the impedance is a complex-valued, frequency-dependent value that depends on both the resistance and the capacitance of the input.

3. Now return to the 1 k Ω resistors. To familiarize yourself with the oscilloscopes, repeat the measurements of part (1) using a digital oscilloscope and a 10x probe, with V_{in} provided by a signal generator that is operated first at 100 Hz, then at 100 kHz. First learn to trigger the oscilloscope at a fixed signal voltage level, so that a stable signal appears on the screen. Now learn to use the built-in "Measurement" menu system to display the rms or "root mean-squared" voltages at V_{in} and V_{out} . Note the difference between peak-to-peak and rms voltage measurements.

4. Finally, place a 0.1 uF capacitor in parallel with R2, as shown in Fig. 2 below. What effect does the capacitor have at 100 Hz, if any? What does it do to the 100 kHz signal? Finally, adjust the signal generator to find the frequency at which the circuit acts as a 1:4 divider rather than as a 1:2 divider. You will compare this result with numerical simulations in the next section. Try saving a few of the 'scope displays on a USB drive, both as pictures and as .csv data, to learn how the process works.

B. SPICE simulation

While there is no substitute for actually building a complicated circuit to see if it works well, computer simulation has become a powerful tool for determining whether it is likely to work, and to eliminate designs that cannot possibly work. On occasion, we will use the PartSim program to aid in designing and analyzing our circuits. Here's a step-by-step procedure for looking at the voltage divider, including the addition of a capacitor to turn it into a low-pass filter.

1. Using either a lab computers or your own personal computer or tablet, find the partsim.com website with a web browser. Click on the HELP menu item, and take a few minutes to watch the "Getting Started" tutorial that shows the basics of defining a schematic diagram with PartSim, then using its automated interface to generate a text-based "netlist" for an NGSPICE simulation and to display its results of this simulation.



Figure 2. A complete web browser screen showing PartSim, with a low-pass filter schematic ready for simulation.

- 2. Set up an online account so that you can save your work on the cloud, making it accessible from anywhere.
- 3. Set up a new project, and save it using the name "Voltage divider." Be sure to save your work every few minutes, so that if you make a major mistake, you can recover your most recent usable version and resume from there.
- 4. Place the components one by one, using 'R' to rotate, and right-clicking on the components to set their values. When doing so, enter the values without spaces, using the closest Roman equivalent for Greek prefixes. For example, enter "0.1uF," not "0.1 μ F". You can find the ground symbol under "Ports". For the voltage source, select an AC source, which has the flexibility to produce both ac and dc test signals. By right-clicking on it, turn off the ac voltage, and set it to 5V dc.
- 5. To connect the components, click on one end of each. Do not just move them up against one another this will *not* be interpreted as a connection.
- 6. Each set of connected terminals is called a "net". To give the input and output nets convenient names, right-click on the wire connecting R1 and R2, and set the Net Name to Vout. Likewise, define the wire from the ac source to R1 as Vin.
- 7. Now use the "Spice Run" button to automatically generate code for an NGSPICE simulation, requesting a "dc bias" simulation. You should see a display similar to the one in Fig. 1 above, showing an output of 2.5 V.
- 8. Finally, explore the effects of capacitor C1 by re-entering the source properties menu to turn off the dc voltage and define an ac voltage of 5V. Re-run the Spice calculation to provide an "AC Analysis" as shown in Fig. 3, with a range of ac frequencies from 100 Hz to 100 kHz. You should now be presented with a plot showing both the magnitude and phase of Vout, as in Fig. 4. Use the cursor to determine the frequency at which Vout is at ½ of its low-frequency amplitude. Does this match well with what you measured?

| R | tun Simulation | 3 |
|-----------------------------|--------------------------------------|-----|
| | | |
| DC Bias DC Sweep | 🗷 AC Analysis 🛛 🗆 Transient Response | |
| Configuration | | |
| DC Sweep AC Response Transi | ent Response | |
| Sweep Type: DEC | • | |
| Start Frequency: | Stop Frequency: | |
| 100Hz | 100Khz | |
| Total Points Per Decade: | | |
| 10 | | |
| Magnituda Phase Plat | | |
| Magnitude Unit: REAL | Phase Unit: RAD | |
| | | |
| Magnitude Unit: REAL | Phase Unit: RAD | |
| | | |
| | | |
| | Cancel | Run |

Figure 3. Data entry menu for ac frequency sweeping.



Figure 4. Output of an ac SPICE simulation, showing the magnitude and phase of Vout as a function of frequency. This configuration is called a low-pass filter, for obvious reasons.

Questions:

- 1. Comment on the accuracy of your results from part A.1. Are they within the tolerance of the resistors you used? You can determine the tolerance using the last band of the color code; for example, a gold band denotes a 5% tolerance, although usually the error is much less than this. Are any other uncertainties expected to contribute appreciably here?
- 2. Use your results from the divider with two 510 k Ω resistors, together with the formula for parallel resistor combinations, to deduce the input resistance of the DMM. Typically the nominal resistance or "impedance" of a DMM is either 1 M Ω or 10 M Ω . Does your result seem consistent with this?
- 3. In part A.3, try to think of two or more reasons explaining why the rms voltage is generally the preferred way to measure and quote your results when measuring the amplitude of an ac signal, if this measurement is available, rather than the peak-to-peak voltage.

Appendix: Breadboard layouts

Most breadboard panels for electronic prototyping share a common design. The photos below show the front and back of one of the breadboard panels used in the Physics 3150 lab, with a single integrated circuit inserted in the intended position over one of the slots in the panel. The dark lines on the back of the board are electrical connections. As you can see, each pin of the IC is connected to a row of four holes on the breadboard. A few of these connected rows are indicated by vertical red lines on the photo. Wires or small-diameter component leads can be inserted into any of these holes to make a connection.

In addition, there are several long lines of connected holes intended as "buses" for power or signal distribution. The connections do not span the wide gaps, so each line across the breadboard is actually a pair of two separate buses, as indicated by the horizontal blue lines on the photo. These must be connected with a jumper wire if you want the entire row to share a common signal. For the similar-looking power buses on the workbench that hosts the breadboard, you may find that such a jumper has already been provided internally. If in doubt, check!

For any but the simplest circuits, it's important to keep your design clean and easily interpreted. It's also important to make sure that your components cannot accidentally short-circuit to one another, causing erratic operation or perhaps destruction of your circuit. As a general rule, keep your layouts two-dimensional, not three dimensional, adding insulation as needed where wires cross one another. Towards this end, it's worth noting that resistors are very cheap, so don't worry about trimming the leads to obtain a better layout. On the other hand, capacitors are considerably more expensive and the leads should be left at full length when possible.



