

PSPICE Tutorial

prepared by Nate Matter
Stanford University
Oct. 8, 2000

1 What is SPICE?

SPICE (Simulation Program for Integrated Circuits Emphasis) is an analog circuit simulator developed at Berkeley. Many different versions of SPICE are available from many different vendors. Common SPICEs include HSPICE, PSPICE, and B²SPICE. SPICE takes a circuit **netlist** and performs mathematical simulation of the circuit's behavior. A netlist describes the components in the circuit and how they are connected. SPICE can simulate DC operating point, AC response, transient response, and other useful simulations.

For EE122, you may use whichever version of SPICE you prefer, however the TA's will only offer help if you use MicroSim PSPICE. PSPICE uses a schematic capture program to generate the netlist, so you simply have to draw your circuit in order to simulate it. PSPICE is available on all the lab computers, and you can download a student version for windows from:

<http://www.orcad.com/Product/Analog/download.ASP>

2 PSPICE Tutorial

2.1 Inputting your Circuit

1. From the Start Menu, select Programs → PSPICE Student → Schematics. You are now running the schematic capture program, which you will use to enter the simple integrator shown in figure 1.
2. Add an ideal op-amp:
 - Open the Get New Part window by clicking on the binocular icon or using the Draw menu.
 - Search for available op-amp models by typing "opamp" in the description search box.
 - You should see a list of available op-amp models. Notice the 741 and 411 are both available in lab. It is possible to add more models if you ever need to do so.
 - Select OPAMP, the ideal op-amp model. Click on "Place and Close"
 - Place the op-amp on the sheet by left-clicking the mouse.
 - Double click on the op-amp. You should see a window listing the parameters for the part. Notice you can set the op-amp gain. You don't need to change any parameters for this part. If you need to know what the parameters mean for a part you are using, the help contents has a list of parameters and their meaning.
3. Add the resistors and capacitors:

- Use Get New Part to place the resistors and capacitors on the sheet. Use the R and C components (ideal resistor and ideal capacitor.)
 - Set the component values by double clicking on the value already displayed next to the part. You can also change the component value by double clicking on the component itself and changing the correct parameter.
4. Add an AC voltage source.
 - Use Get New Part to place an AC voltage source (VAC).
 - Set the AC magnitude (ACMAG) to 1V.
 5. Add a ground (GND_EARTH)
 6. Connect the components using the Draw Wire button, or use the Draw menu.
 7. Name the nodes to useful names
 - Double click on the wire between the output of the op-amp and the resistor. Name this node something like "Vout".
 - Rename the other nodes as well. The ground node is already labeled "0" and cannot be changed to anything else.

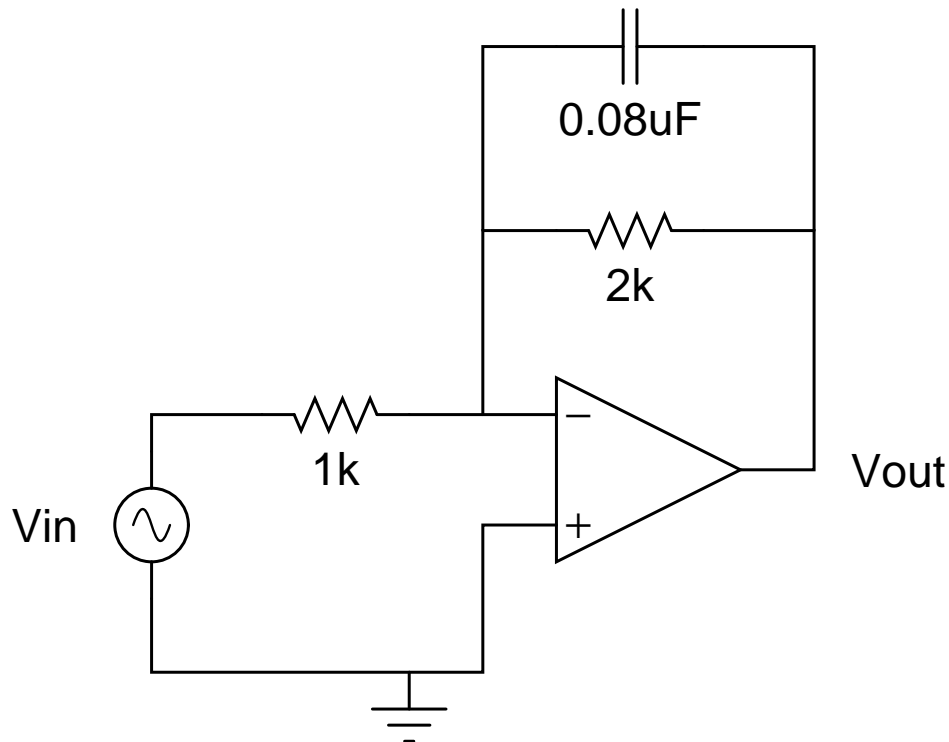


Figure 1: Integrator

2.2 Simulating Your Circuit

1. Set the simulation parameters.
 - Go to Analysis → Setup.
 - Enable AC Sweep
 - Click on the AC Sweep button.
 - Set the AC parameters to:
 - Sweep Type = Decade
 - Pts/Decade = 101
 - Start Freq = 10
 - End Freq = 1meg
2. Save your schematic. Save it in D:\SAVE STUFF HERE\your_folder_here
3. Simulate the circuit by clicking on the simulate button, or use the Analysis menu.

2.3 Plotting the Results

1. When the plotting window appears, plot the results:
 - Click on the Add Trace button or use the Trace menu.
 - Uncheck Alias Names
 - You can plot any of the parameters seen on the left. Select and plot V(Vout). Make sure the plot matches what you expect.
 - Set the y-axis to a log scale by double clicking on the axis.
 - Now, plot the same data using dB for the y-axis.
 - Trace → Delete All Traces
 - Set the y-axis back to the linear scale.
 - Add Trace
 - In the Trace Expression box, type: $20 * \log_{10}(V(Vout))$ Click OK.
 - Find the -3dB point on the plot using the markers:
 - Turn on the cursors using the Toggle Cursors button or the Trace → Cursor menu.
 - A small window has appeared showing the position of the cursors and their values. The left mouse button moves one cursor and the right mouse button moves the second.
 - Using the cursors, find the -3dB point by setting one cursor to 10Hz and move the second cursor until their difference is 3dB. Is this the frequency you expect?

3 A Few Abbreviations in SPICE

- f = fempto
- p = pico
- n = nano
- u = micro
- m = milli
- k = kilo
- meg = mega
- g = giga
- F = farad
- Hz = Hertz
- H = Henry
- ohm = ohm

Note: Labelling units (F, H, ohm, Hz) is optional in SPICE.

Now you know the basics of PSPICE, but the only way to learn it well is to spend time using it and exploring all the functionality. Don't be afraid to use the help contents!