

## PSpice Tutorial Part One: DC Voltage Divider

By Alex Barnett

IEEE @ The Illinois Institute of Technology Fall 2008

Our first circuit is an elementary voltage divider. However, I will present basic technique to be used throughout the tutorial. A basic understanding of Circuit Analysis will be assumed for this tutorial.

Start by opening Schematics. The first step for building a circuit is selecting components. Click on the toolbar button with the binoculars, near the top of the screen. The Part Browser should open in alphabetical order, allowing you to either type the first letter of the part you want, or you can search the part description. We'll start with a DC voltage source (VDC). Just click on "Place & Close" to exit the browser.

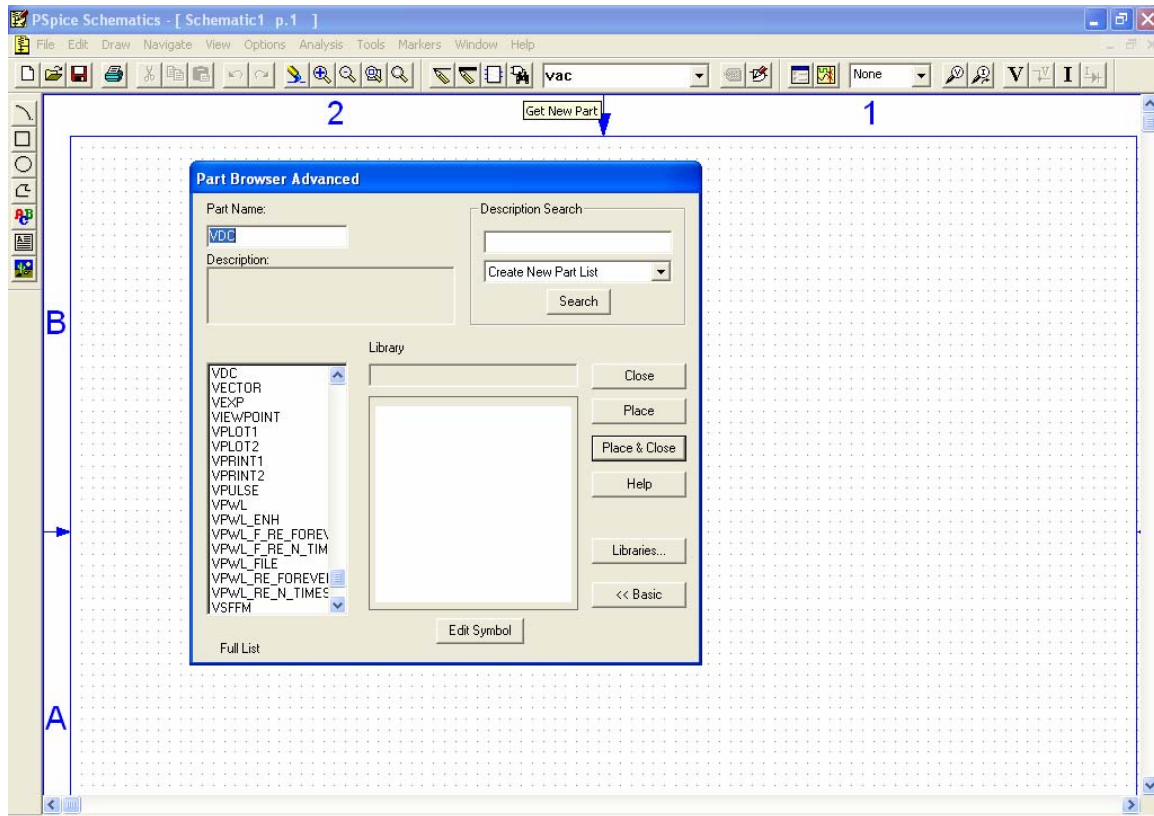
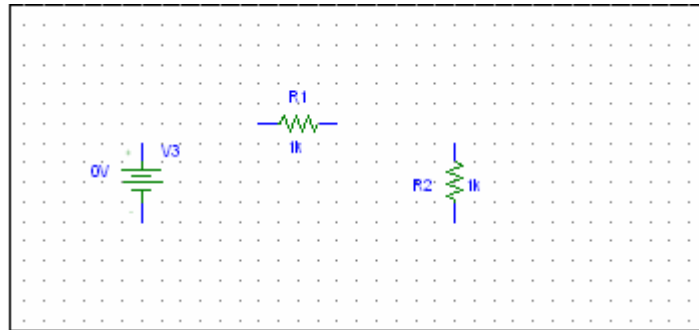


Figure 1: part browser

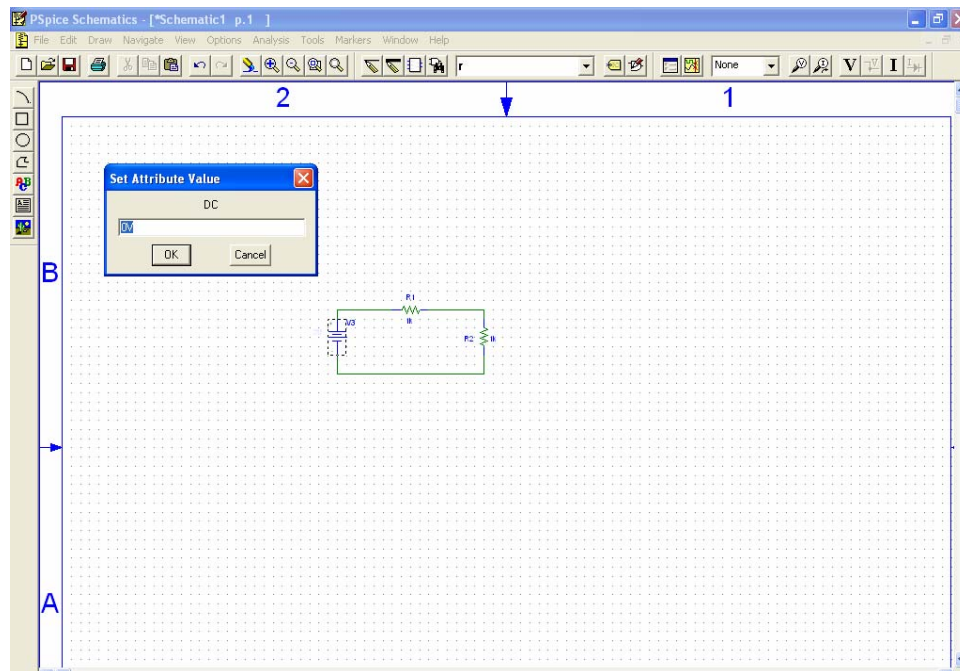
You are left with a black shadow of the part to release onto your schematic with the click of the mouse. You may arrange the components shown below. Any component can be rotated with  $\text{ctrl} + \text{r}$ . If the component has already been placed, simply click once on the component to activate it.



**Figure 2: Components for Circuit 1**

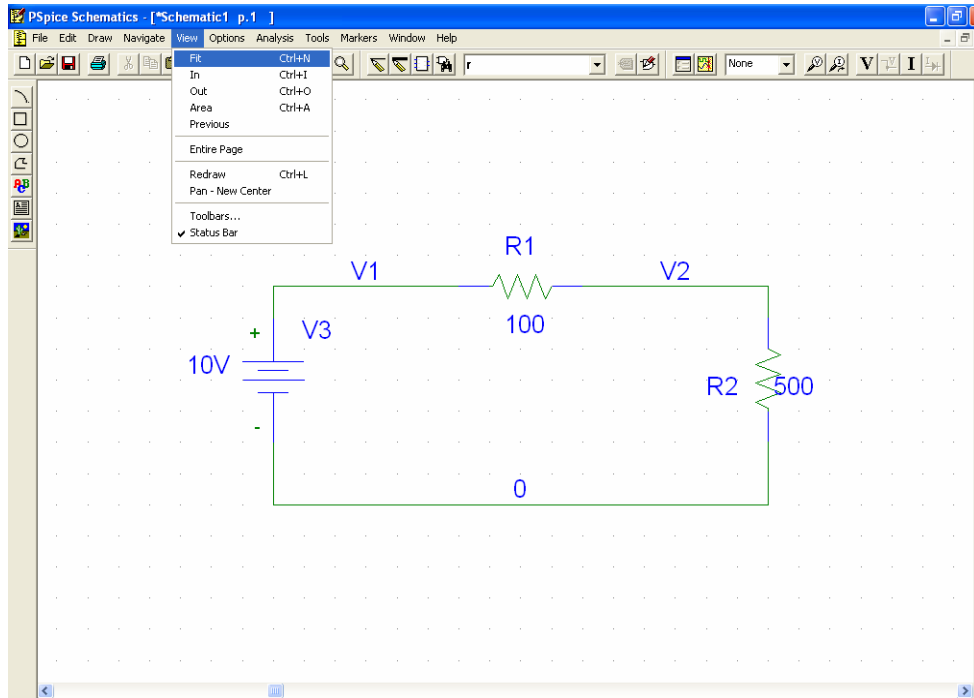
Next we will run wire for our first circuit. This is a good time to point out that the GUI of Pspice 9.1 Schematic allows you at least 2 ways of accessing each feature. Wire may be drawn by clicking `ctrl + "w"`, `Draw>wire`, or clicking on the small pencil icon on the toolbar. Keep this in mind as you explore the PSpice interface.

Once you have completed the circuit loop, click on the icon marked 0V near the voltage source. Usually, the most important attributes of a part are by default displayed on the schematic. Simply clicking on the attribute value allows you to change it. Use this technique to change our voltage source to 10V, R1 to 100 Ohms, and R2 to 500 Ohms.



**Figure 3: DC Voltage Attribute**

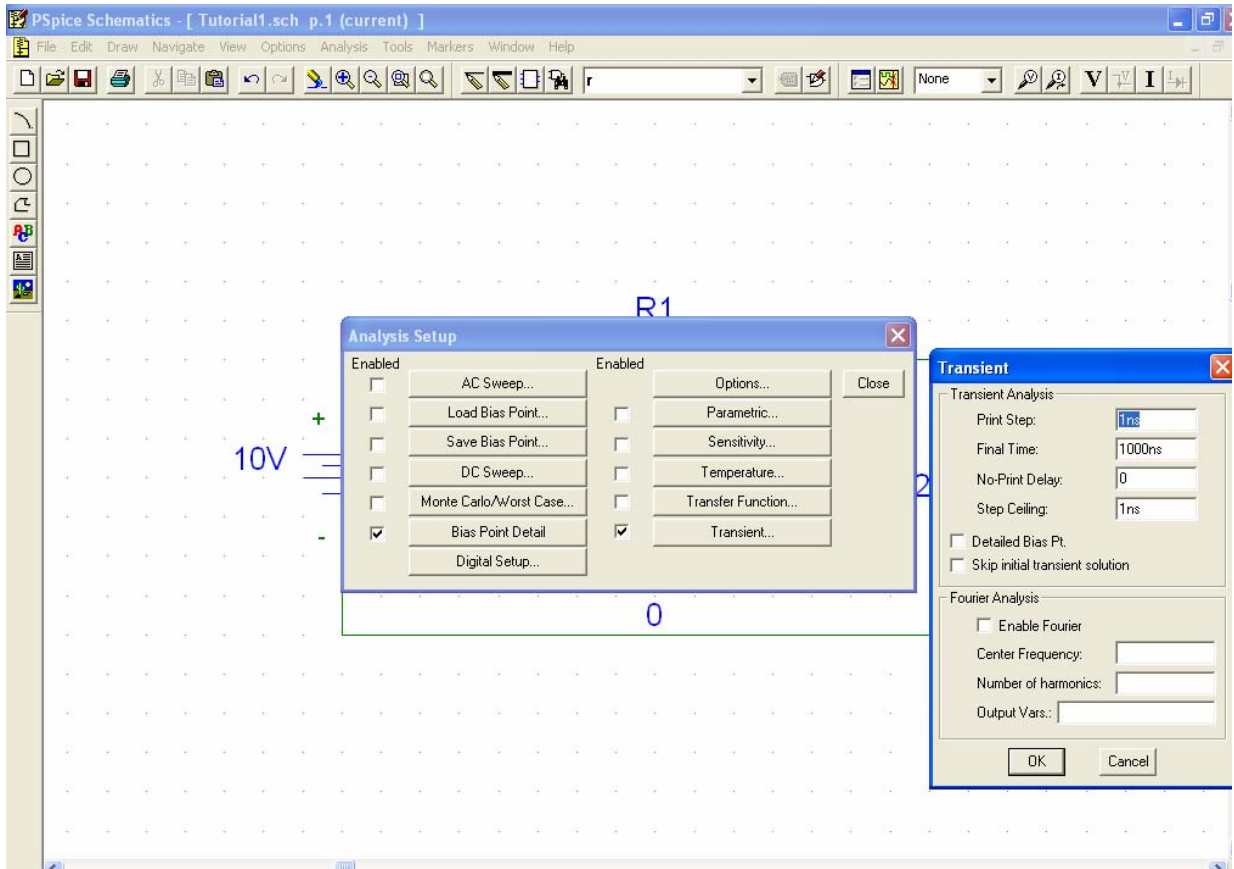
Any node on your circuit may also be given a label by clicking on it. Instead of using a ground from our parts catalog, I typically just label my ground node "0". PSpice understands this as the reference ground for the circuit. Label the nodes "V1", "V2", and "0." Then use `view>fit` to obtain an appropriate zoom. You should have the following view:



**Schematic 1**

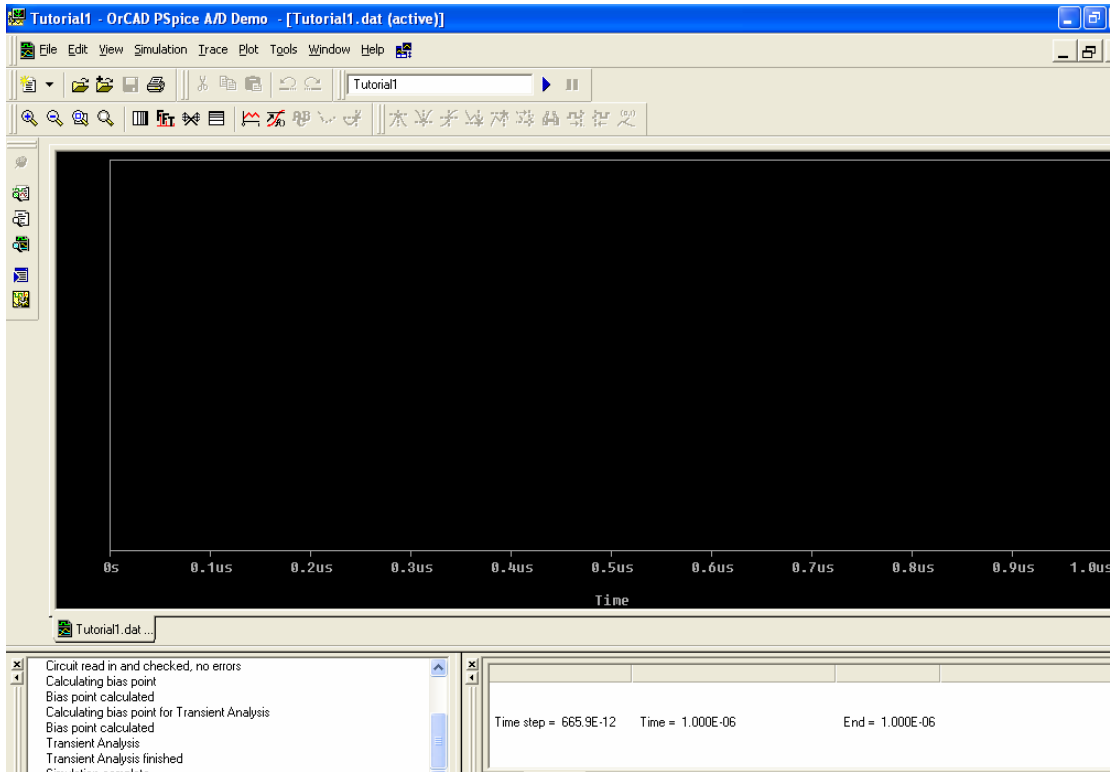
Next we will set up our simulation for this circuit. Although our circuit is in a steady state, we will perform a transient analysis simply because it produces the best looking graph.

Use *Analysis>Setup* to open the Analysis setup menu. Click on transient analysis to set up your menu. Setup the analysis with print step 1ns, final time 1000ns, no print delay 0, and step ceiling 1ns.



**Figure 4: Analysis Setup**

You must save your circuit before running any simulation. Save your circuit with a title and location of your choice. PSpice generates several files, so things will become messy quickly if you do not keep project folders. Next click *Analysis>simulate* to run your simulation. The “Probe” interface opens a new window. If all goes well, you will see the following display:



**Figure 5: Probe Window**

There is a toolbar button to add traces, which looks like a jagged curve on a graph. The “Add traces” menu opens, with all valid traces which PSpice has already calculated for you. To display our Voltage Divider we will need to display  $V(V1)$ , which means the voltage from node V1 to ground.  $V(V2)$  will be the voltage across R2, and  $V(V1)-V(V2)$  is the voltage across R1. You can also use  $V(R1:1)-V(R1:2)$  for the voltage across R1 dictated by R1’s pins. When finished, you should have something resembling Figure 6.



**Figure 6: Final Voltage Divider Plots**

We are now finished analyzing our Voltage Divider.