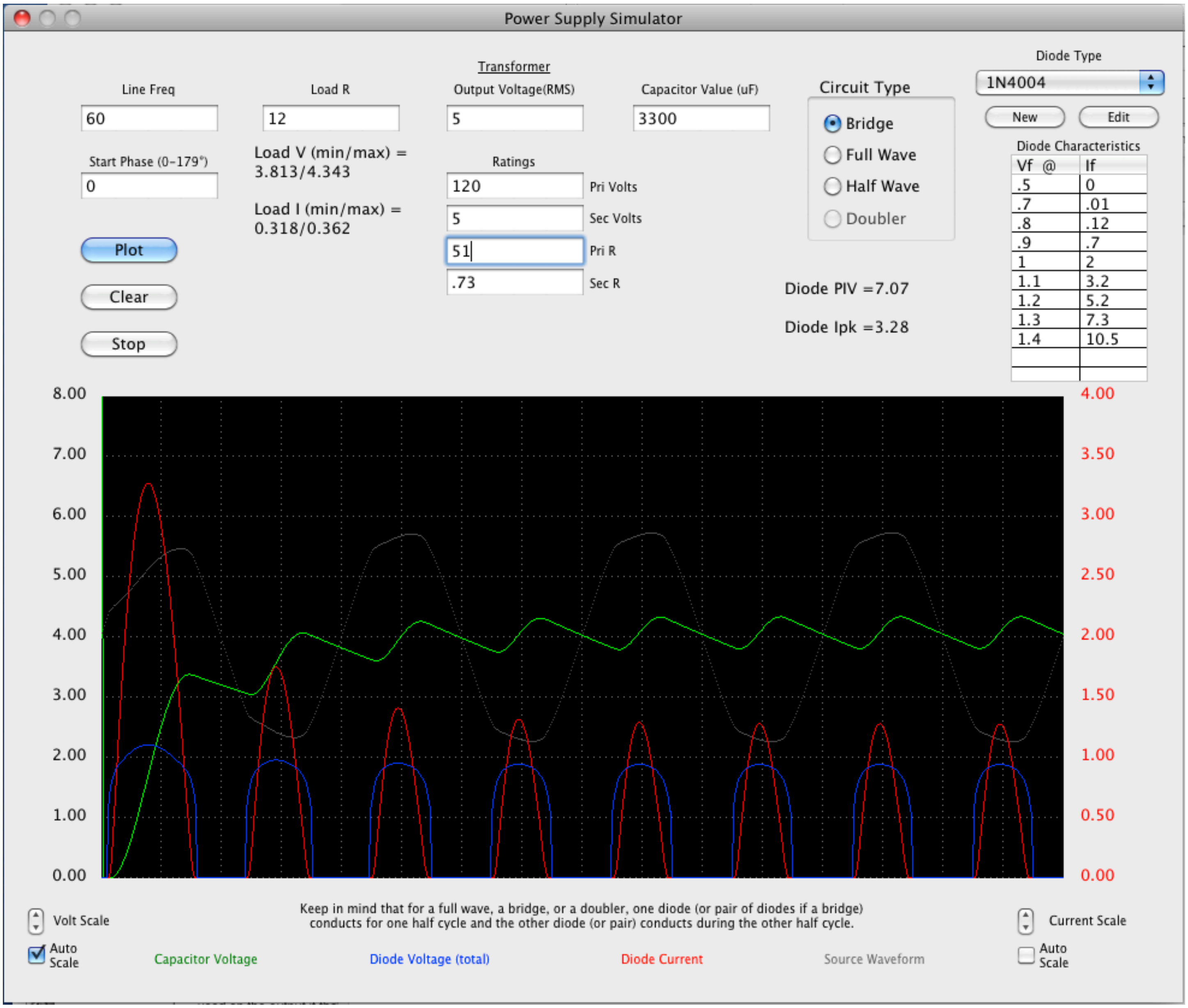
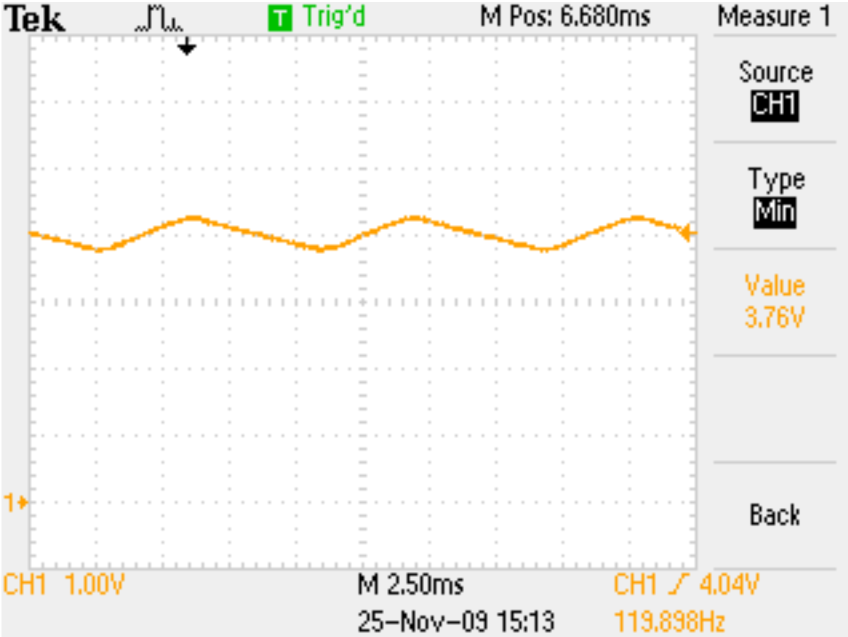
Power Supply Simulator

Designing a power supply can sometimes be a bitch. For me anyway. No matter how carefully you calculate the needed component specs, once you put it all together it's either overkill or underkill. There seems to always be something that could use a little tweaking.

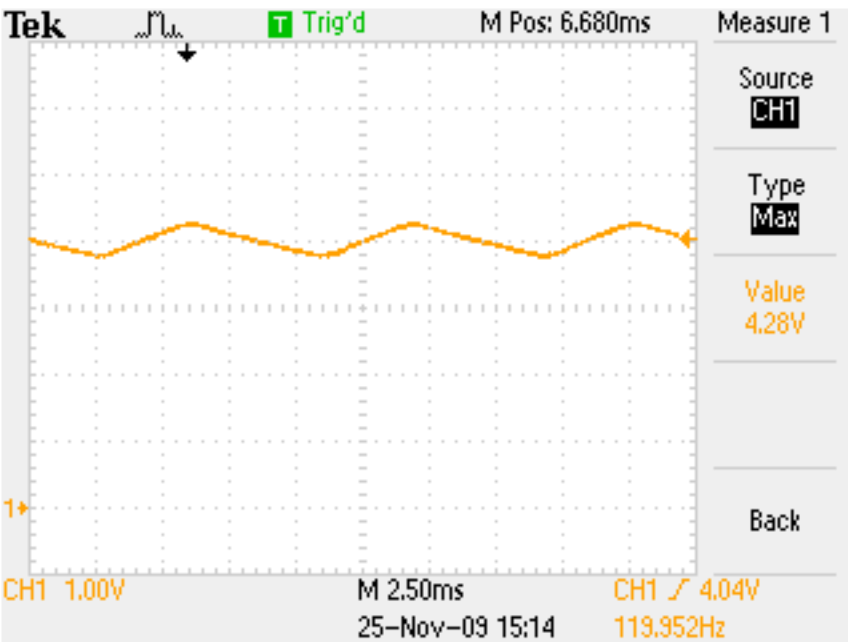
This software simulates a power supply so you can do your tweaking on your screen rather than on your breadboard. And it's accurate. If you enter the component values that you really will be using, the waveforms it generates will be the same as what you'll see on a scope. I've compared it against several power supply designs and it's always right on the money. I'll show you...



The above is a screen shot of a power supply simulation. The scope images of the real power supply follows.

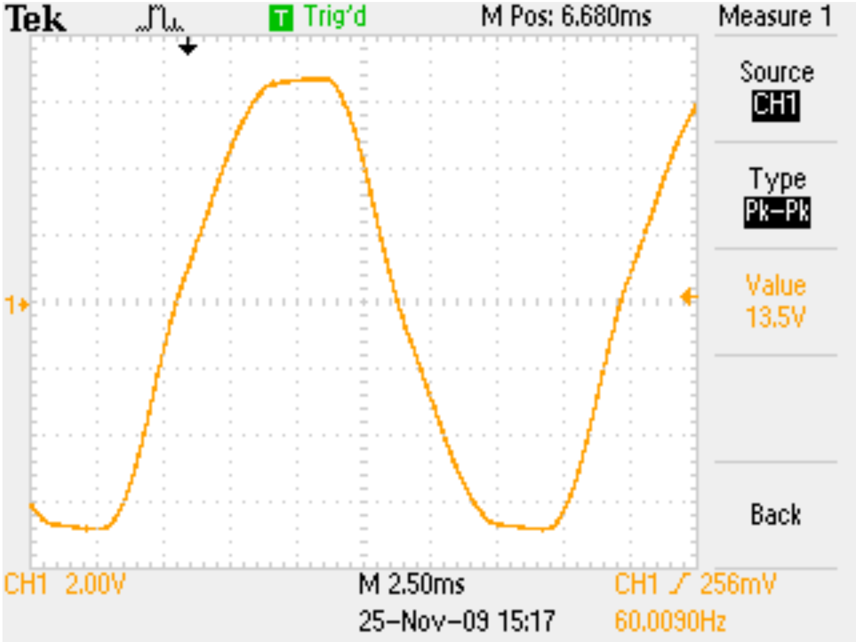


Notice the minimum voltage of 3.76, which compares favorably with the simulation result of Vmin of 3.81v (less than 1% difference).

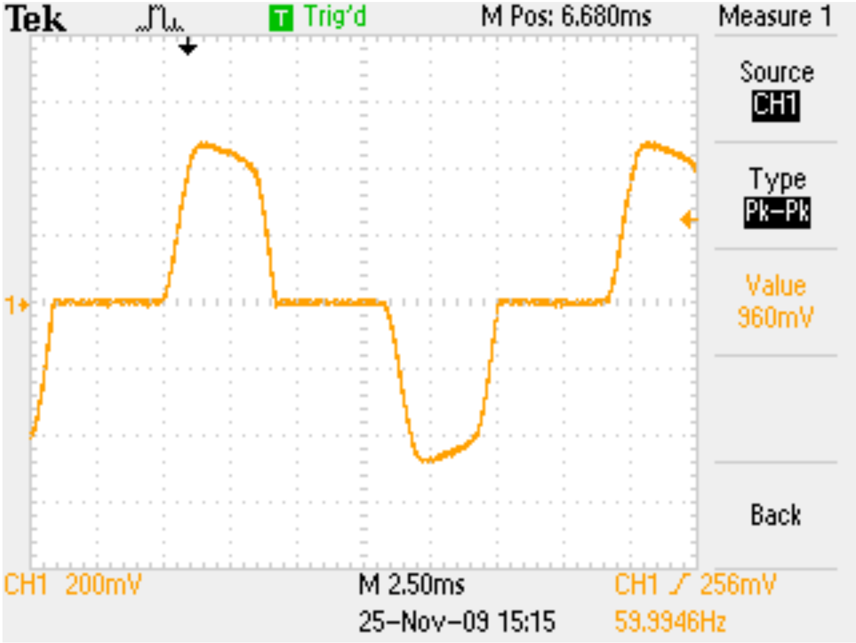


Notice the maximum voltage of 4.28, which compares favorably with the simulation result of Vmax of 4.34v (again less than 1% difference).

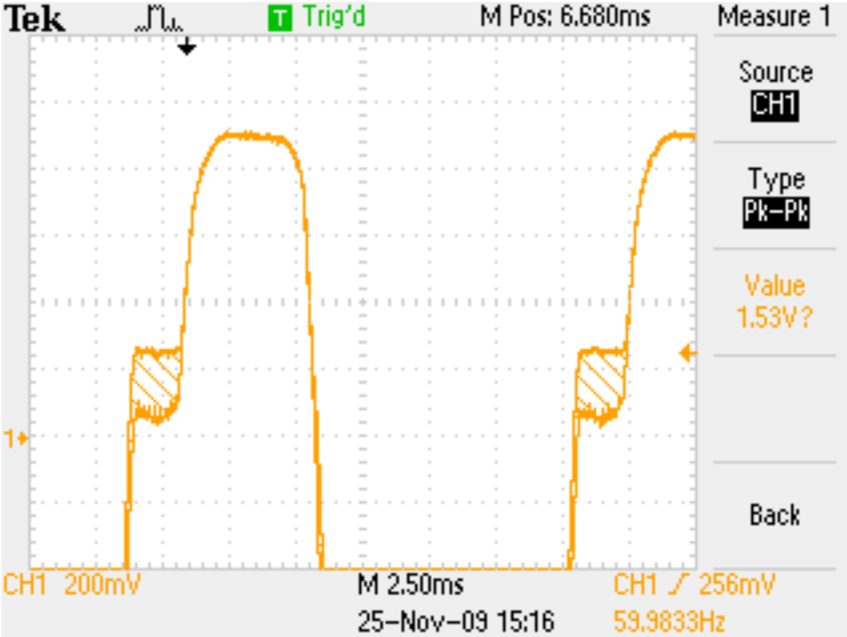
The diode currents in the real power supply and the simulation are close, but the current waveforms look a bit different. I can account for that because, for some reason, my power company is not providing me with a true sinusoidal power line voltage. It's a bit clipped on the tops, like this:



This is straight off the secondary, UNloaded. If it had real peaks then I believe the diode current waveforms in the real power supply would also have peaks. As it is, the diode current wave in the real power supply looks like: (This is the transformer secondary current which is the same as diode current, which explains the positive & negative excursions...also, the scope trace is the voltage across a 0.51 ohm resistor in series with the secondary.)



Current peaks of (0.48v/0.51 ohms) 0.94 amps. The simulation shows peaks of 1.3A (red trace), but they are of slightly shorter duration. The simulation uses a real sine wave with good peaks, not my rounded peaks from Duke Power Company.



This shows the voltage across one diode. The positive excursion of 0.9v is during diode conduction. For the 2 diodes in the bridge that are conducting simultaneously, this makes 1.8v lost due to the diode voltage drops. Matches nicely with the 1.9v shown in the simulation (blue trace).

How To Operate

It is really quite straightforward. Just enter the parameters for the components and hit “Plot”. A few items will need clarification...

The “Start Phase (0-179°)” entry field allows you to simulate the power supply being switched on at some phase angle other than at the zero-crossing point (which is 0°). 90° is worst case, as it is at that point that the supply voltage is at it’s peak. Switching on the supply at that point puts maximum stress on the circuit, especially the diodes.

The “Load R” field. Be careful with this one. If your power supply will be feeding a voltage regulator, the load seen by the power supply will be a constant current, not simply a resistance. You need to calculate an equivalent R the power supply will be driving, using the voltage from your regulator and load it will be driving. The graph results will be close to the real thing if you’re accurate. (In a future version I intend to provide an option for a constant current load.)

The 4 fields under the transformer ratings are the specs for the transformer. The software needs the specified primary volts and secondary volts in order to calculate the turns ratio, which is needed to calculate the reflected primary resistance into the secondary. The transformer “Output Voltage” field can be manipulated to see the results of low or high line voltage. The ratings fields should not be used for this. Once the ratings are entered for a particular transformer they should remain the same for that transformer, regardless.

Under the “Diode Type” section you can choose from 1 of 3 diode types that are created for you the first time the program is run. To utilize a different diode, click on “New” and enter the diode type number and the voltage-current characteristics for your new diode. Get this info off the diode data sheet graphs. Be accurate. One thing, you should always include a diode voltage for zero current at the start. No graph will have this, so make a good guess. A good figure is a tenth or two below the minimum diode currents shown in most graphs. Once you have entered a new diode and clicked the “Save” button (in the diode edit window...not shown here), that diode will be added to the list of diodes should you ever want to use it again.

“Volt Scale” and “Current Scale”: These up/down arrows allow you to choose your own scale for the graph. The voltage scale is on the left and the current scale is on the right. When you click on an Up/Down arrow the “Auto Scale” checkbox is UNchecked automatically. Otherwise (when the box IS checked), the software decides the best scale every time you click “Plot”. Be aware that if you change parameters you can cause a different scale to be used automatically and you may not notice it. If it doubt, UNcheck the Auto Scale boxes and choose your own scale.

The Source Waveform shown in the graph does not represent the actual voltage from the secondary, only it’s shape. It is always a nominal 4 divisions peak-peak, but you may find it useful to see how badly the peaks of your secondary voltage are clipped as your power supply becomes more loaded.

If you check the box “Help Tags”, you will get info for each parameter field by hovering your mouse over a field. In the Windows version this checkbox is not available (since I have no clue how to turn OFF the help system in Windows).

The Mac version can be downloaded from www.macupdate.com (search for PowerSupplySimulator).

In case you're interested in how this thing works.... I break up each sine wave into 200 time slices and work on each slice in a linear manner. No wierdo diode formulas were used (since you cannot find those in a diode's datasheet) and no time constants were used. It's all linear - constant current for capacitor charge/discharge and instantaneous voltage changes from the transformer. What was difficult was the iteration method for determining the voltage drops across the diodes as the capacitor is charged. Alternatively, the simultaneous equations needed for that would be quite hairy, so I didn't even try that approach. In any case, the iterative/time slice method works quite well, as can be seen by comparing it to a real power supply.

There are some limits to the component values you may use. If you start entering zero values for some items, you'll eventually break the software. Be reasonable and it will treat you fairly. Also, if the time constant of your filter capacitor and the load resistance is much less than 8.3ms the simulation will begin to fall apart (with a time constant that low your ripple voltage will be getting close to 50% of your output voltage). If your time constant is as low as around 100uS weird things will start to happen. The graph will go bananas. Again, be reasonable.

And I realize that this software won't win any user-interface awards.