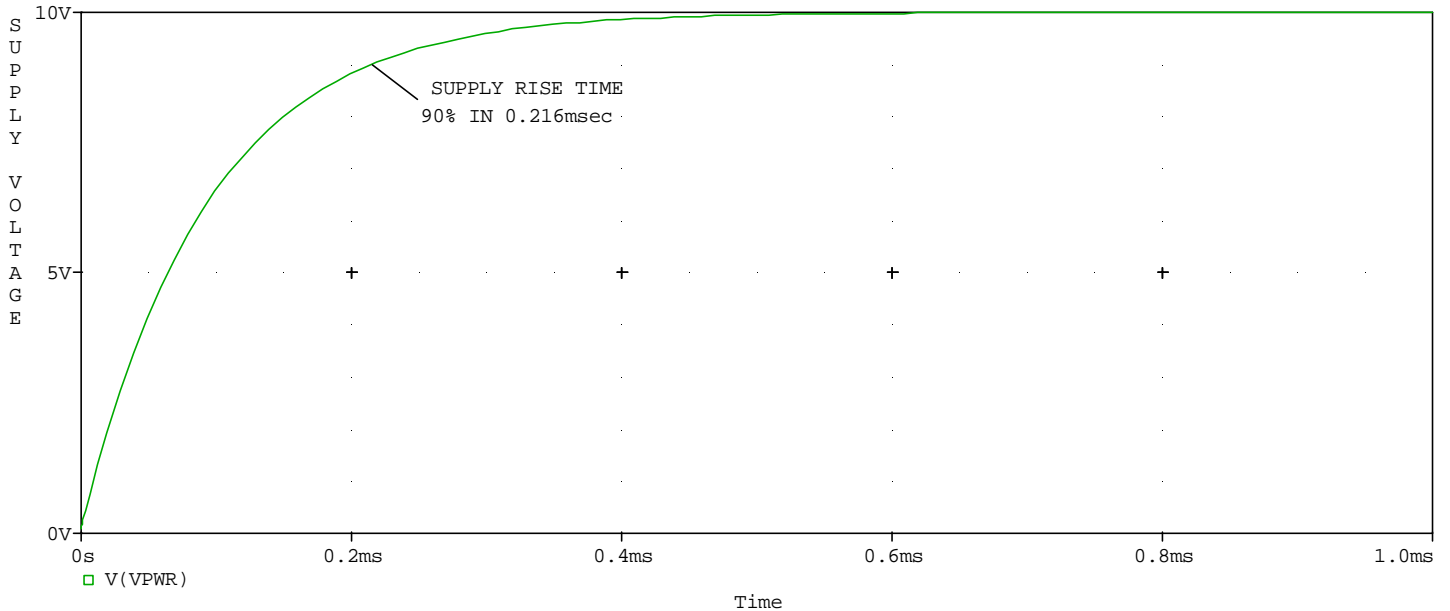
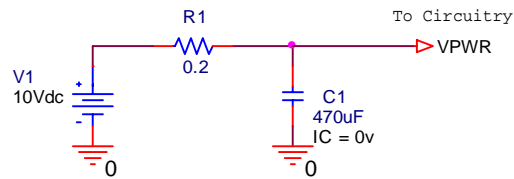


## Pspice Simulation Tip #1



## Managing Power Supply Voltages

One problem when using Pspice for simulation of Transient (Time Domain) Analysis is that at time = 0 the Power Supplies are stable. In the real world Power Supplies must ramp up to a stable voltage. During the time that the supplies ramp up some circuits may latch up or have unpredictable results. A common problem might be glitches on the output pins.

A solution to the problem of an idea power supply that is stable and at full voltage at the start of simulation is to use the simple circuit shown above. This circuit will actually benefit you in a couple of ways. One it will limit the power that the supply provides (remember power sources are idea) and second, it will have to charge the capacitor at time = 0 as you expect in normal supply operation. This means the supply will ramp up to full power at a controlled rate. You can adjust the RC time constant to control how fast the power rises. Note, that you must set the initial condition on the capacitor (IC = 0) due to it being charged during the bias calculation.

Also, remember with this circuit you will be looking into the impedance of the circuitry you are powering.

A variation to this circuit as shown below will allow you to run additional tests. By adding a Time to Open Switch in place of the resistor. You can provide the switch an ON and OFF resistance and at some point in time open the switch and let the capacitor be discharged by your circuitry. You will see the behavior of your circuit during power loss or power supply shut down.

When working with multiple supplies you can easily expand this basic idea and control the power sequence to the supplies and also the failure of a specific supply.

