

Technical Brief: LTspice: A Voltage-Controlled Resistor

Béla Géczy

LTspice is one of the world's leading circuit simulation tools. In 1998, Linear Technology hired Mike Engelhardt to re-write Spice to allow simulation of linear circuits with primary emphasis on switching power supplies. Mike Engelhardt has been writing physical simulators for 39 years.

There are many functions included within the free download of the software and available on the internet. One such function that has been difficult to find is a voltage-controlled resistor (VCR). This Technical Brief offers one simple way of making a VCR using LTspice.

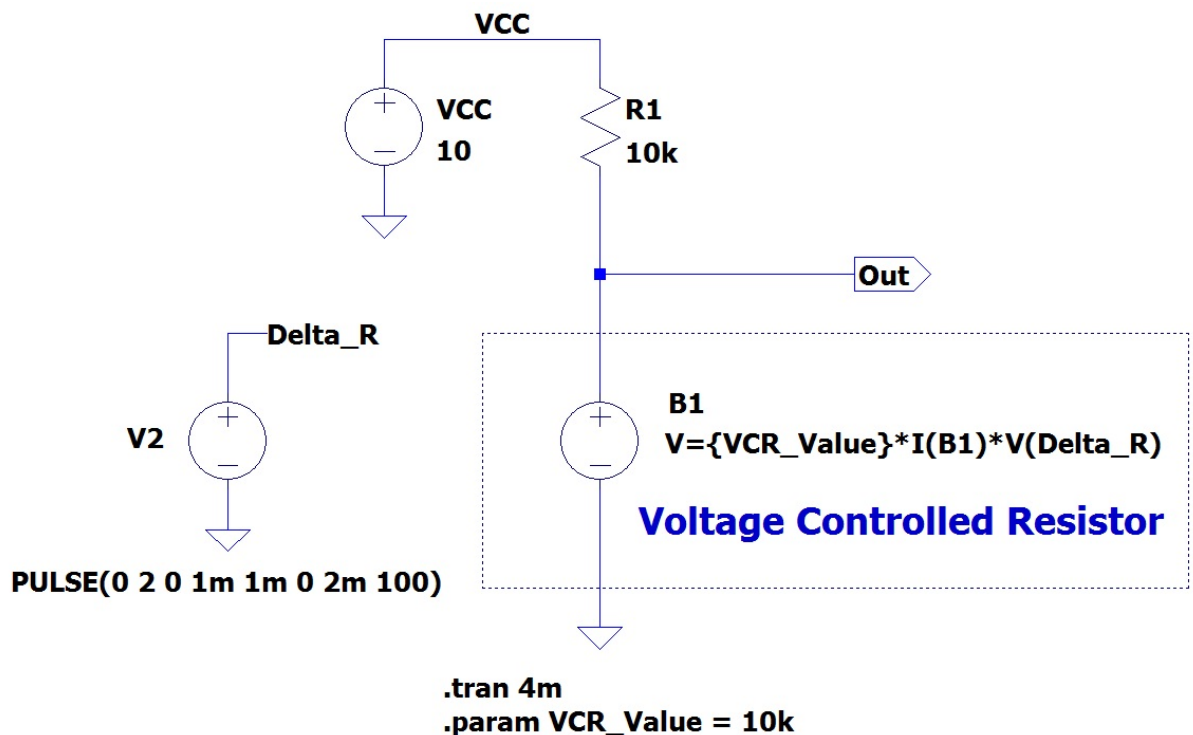


Figure 1

Figure 1 demonstrates the method suggested here. The VCR is drawn within the dotted outline. The VCR is comprised of a "B" Source which is an Arbitrary Behavioral Voltage Source. The "B" source contains the function you wish to use for the VCR. B1 is also used to determine the current through our VCR and is multiplied by the resistor value in order to make a VCR. In this example, the function of the VCR is $VCR_Value \times \Delta_R$.

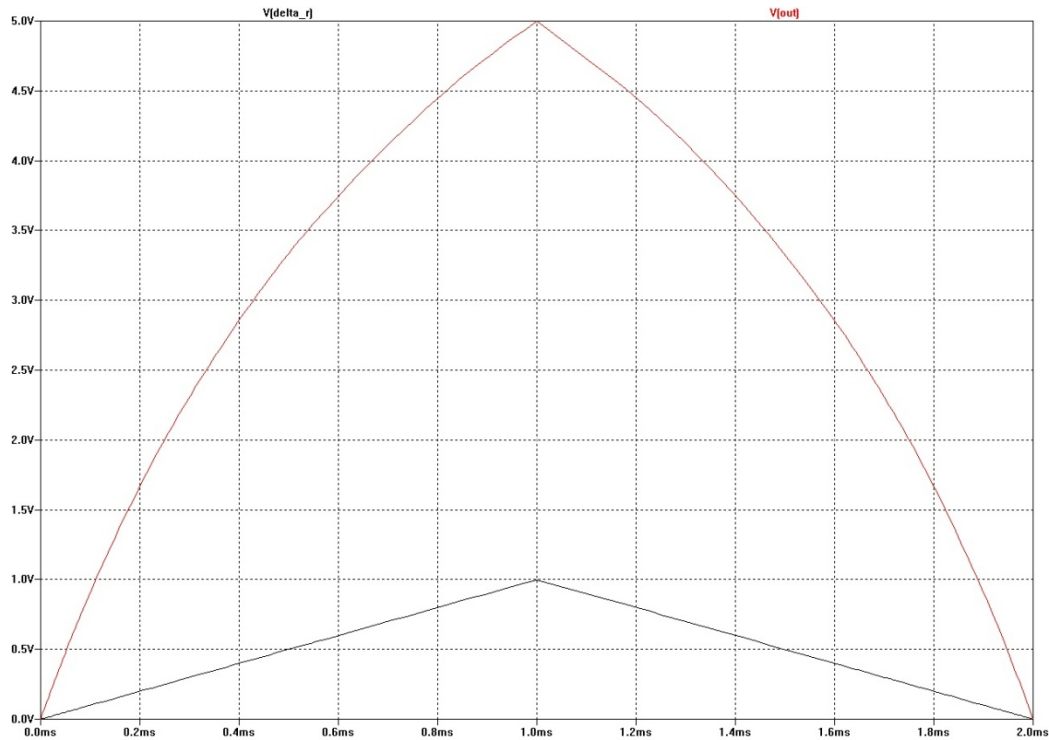


Figure 2

The schematic is a simple voltage divider comprised of a 10k Ohm resistor (R1) and our 10k VCR along with a 10V power source. The .param function defines the value of the VCR and the voltage from the Voltage source V2 (or any node in your circuit) is the multiplier to obtain the variable resistor value. In this case, the voltage Delta_R is a triangle wave with a value that starts at zero volts and has a maximum of one volt.

Figure 2 shows a plot of the two waveforms, V(Delta_R) and V(Out). As can be seen, V(out) starts at zero when the value of V(Delta_R) = 0 and VCR = 0, then climbs to 5V when the value of V(Delta_R) = 1V and VCR = 10k.

The VCR can follow any function available in LTSpice which can be found in the help file (you can view the list under LTSpice/Circuit Elements/B).