MULTISIM DEMO 5.1: INTRODUCTION TO TRANSIENT CIRCUITS

In the previous portions of the tutorial, we began to use time-varying simulations in Multisim with simple steady-state circuits. Since Chapter 5 of the text begins to deal with transient responses of circuits, we will now look at how to use the oscilloscope in analyzing these types of circuits.

In Multisim, let's build the circuit shown below. The primary component of interest to us is the switch. We can use a SPDT time-delay switch which can be found under:

Group: Basic Family: SWITCH Component: TD_SW1



Place the component in the schematic and double click on it. This will bring up a window with two fields of values to enter. The Time-delay switch works as such: It starts out in the "On" position. At time TOFF, it turns "Off" which means it switches to the other pole. Then at TON, the switch turns back on. For our circuit, let's set TON to 8 sec, and TOFF to 5 sec, as shown in Fig. 5.1.2. Build the rest of the circuit around the switch, making sure that you connect to the same terminals as shown in Fig. 5.1.3. When the switch arm is flat (connecting to the bottom pole), it is "On" and when the switch arm is angled up (connecting to the top pole), it is "Off".

	TD_SW1		×
	Label Display Value Fault Pi	ns User Fields	
	Time On (TON):	8 sec	3
	Time Off (TOFF):	5 sec	÷.
	Replace OK	Cancel Info	Help
Figu	re 5.1.2 Time dela	v switch contro	ol wind



Now we're ready to run the Interactive Simulation. However if you go and press F5, you will most likely notice that the progression of time in the simulation is extremely slow. It may take 10 real-life seconds for the Interactive Simulation to reach 1 second of Interactive Simulation time. This is annoying especially since the switch doesn't do anything until the five second marker. This issue can be amended by going up to Simulate>Interactive Simulation Settings... Doing this will bring up the window shown in Fig. 5.1.4 below. Just like you would do to adjust the time step for a Transient Analysis, manually set the Maximum time step (TMAX) to 0.001 seconds or something even larger depending on the speed and memory of your computer.

1	Interactive Simulation Settings	
	Interactive Simulation Settings Defaults for Transient Analysis Instruments Analysis Options Initial conditions Reset to default Automatically determine initial conditions Image: Condition Settings Instrument Analysis Start time (TSTART) Start time (TSTOP) 1e+030 Sec Set maximum timestep (TMAX) Image: Condition Setting Sec Image: Condition Sec Image: Condition Sec Condition Sec Image: Condition Sec Sec Image: Condition Setting Sec Sec	
-	More Options Set initial time step Initial time step (TSTEP) 1e-005 Sec Estimate maximum time step based on net list (TMAX) OK Cancel Help	
Figure 5.	1.4 Readjusting TMAX in the Interactive Simulation Settings window.	

Now that the size of TMAX is adjusted, open up the oscilloscope screen, adjust the Timebase value to 2 s/Div and simulate. After 14 seconds, your oscilloscope screen should look like that in Fig. 5.1.5 on the next page.



Now stop the simulation and change the time values of the switch. Change the return time (TON) to 6 seconds. Re-simulate and note the difference in the plot. It should look like Fig. 5.1.6 shown below. The signal never gets a chance to reach its maximum value of about 5 V as it does in the previous simulation because the switch closes too early.

