**ModelSim Help**

When creating a test script, start off with “vsim <design\_file>”, where <design\_file> is the name of your block diagram or vhdl file. For example, if your design file was lab7.bdf, the test script would start off with “vsim lab7”. Next have “view wave” because that is where the signal values are seen. Following that, add each input and output signal by having “add wave <signal>”.

If a signal is several bits, it can become tedious to look at it in binary form in the simulation output. We can tell ModelSim to show the signal in a different radix, shown with the following examples:

add wave –radix hex address  
 add wave –radix decimal signalA  
 add wave –radix unsigned signalB

After all the signals are added, the next step is to set values with the command force. The format is “force <signal> <value> <time>”, where <signal> is the signal getting the value, <value> is the value being assigned to the signal, and <time> is the time in the simulation at which the signal gets the value. A few examples are shown below.

force a 0 0 # sets signal a to be 0 at time 0  
 force b 1 20 # sets signal b to be 1 at time 20  
 force c 0 0, 1 30 # sets signal c to be 0 at time 0 and then 1 at time 30  
 force d 0 0, 1 10 –repeat 15 # sets signal d to be 0 at time 0 and then 1 at time 10  
 # this will repeat at time 15, 30, etc.

If the signal is multiple bits, that’s easy to handle:

force a 00011010 0 # sets signal a to be 00011010 at time 0

We can say the exact same thing, but specify the value in hexadecimal:

force a 16#1A 0 # sets signal a to be 00011010 at time 0

Once the values are set, all that is left to do is run, so have a “run <time>”, where <time> is the length that you want the simulation to run. A sample test script is shown below.

If you want to add comments in your .do file, use the # symbol.

moddsTest.do

vsim modds  
view wave  
add wave a  
add wave b  
add wave f  
  
force a 0 0, 1 30, 0 50, 1 60  
force b 0 0, 1 15 -repeat 30  
  
run 90