Driving Stimuli with Cadence

This guide assumes that you have already walked through the Quick-Start and are familiar with Cadence. **MOUSE CLICKS** will be indicated with bold, small caps. *Entered text* with be indicated with italicized text.

1. Exhaustive Testing

Sometimes you will want to test your circuit using an exhaustive search across every possible input. For example, you may have implemented a circuit for a given truth table, and you want to make sure that the circuit outputs correctly for every possible input. In this case, it is best to use the Cadence CLOCK stimulus as inputs to your circuit.

Once you have laid out your circuit, use the following procedure to set up an exhaustive test:

Capture CIS	Click **PLACE NET ALIAS** on the tool bar.
Place net alias	Enter a name for a wire and press *Enter*.
Capture CIS	Place the net name on the corresponding wire. Repeat until all of the wires have a name.
	Select an input stimulus. Right-click on the stimulus and select **EDIT PSPICE STIMULUS**
New Stimulus	Enter a Name for the stimulus **CLOCK** **OK**
Clock Attributes	**PERIOD AND ON TIME** Enter a period. The period should be a power of 2 from some base clock period. For example, use 100n as the base clock period for your least significant input bit. Then, the next most significant input bit should have a clock period of 200n, then 400n, and so on. If you have N bits, the most significant bit should have a period of $100*2^{(N-1)}$. Enter an on-time. This should be exactly half of the period. **APPLY** If the signal looks correct, then **OK**
Stimulus Editor	**FILE → SAVE** **YES** **FILE → EXIT**

Capture CIS	Continue adding PSpice stimuli until all of the input stimuli have been defined. For each stimulus, double click on the text name and change the Value of the name to match the implementation name.

Once you have generated your stimulus, run the simulation and create an output waveform. Make sure to arrange the signals so that the inputs are in order from most significant (top) to least significant (bottom), and successive outputs are plotted in the order of progression through the circuit.

Once you have an output waveform, you must print and annotate it. To annotate a waveform, draw vertical lines (use a ruler!) on the plot for a given test region. Then label the signals within the region to show what their values are. Finally, relate the values in each test region to the specified circuit behavior – how do you know that the output is correct?

2. Non-Exhaustive Testing

Sometimes you will want to test your circuit using a non-exhaustive search over only some of the possible inputs. For example, you may have implemented an 8-bit adder circuit, and you want to make sure that the circuit generally does the correct addition. It is not a good idea to test all of the possible input combinations because there are 2^{16} possibilities! In this case, it is best to use the Cadence SIGNAL stimulus as inputs to your circuit.

Once you have laid out your circuit, use the following procedure to set up a non-exhaustive test:

Capture CIS	Click **PLACE NET ALIAS** on the tool bar.
Place net alias	Enter a name for a wire and press *Enter*.
Capture CIS	Place the net name on the corresponding wire. Repeat until all of the wires have a name.
	Select an input stimulus. Right-click on the stimulus and select **EDIT PSPICE STIMULUS**
New Stimulus	Enter a Name for the stimulus **SIGNAL** **OK**

Stimulus Editor	You can use the Pencil Tool to "Add a new point or transition to a stimulus." Select the tool and then click on the stimulus and various places. You will see that it switches from 0 to 1 or 1 to 0 at the point where you click. The editor will also show you all of the other signals that you have generated, and you can modify them as well. You can also click and drag to the right to make an area where the signal changes. Use this method to enter in specific input combinations for simulation.
	FILE → SAVE **YES** **FILE → EXIT**
Capture CIS	Continue adding PSpice stimuli until all of the input stimuli have been defined. For each stimulus, double click on the text name and change the Value of the name to match the implementation name.

Once you have generated your stimulus, run the simulation and create an output waveform. Make sure to arrange the signals so that the inputs are in order from most significant (top) to least significant (bottom), and successive outputs are plotted in the order of progression through the circuit.

Once you have an output waveform, you must print and annotate it. To annotate a waveform, draw vertical lines (use a ruler!) on the plot for a given test region. Then label the signals within the region to show what their values are. Finally, relate the values in each test region to the specified circuit behavior – how do you know that the output is correct?