Cadence Quick Start Guide

This guide assumes that you installed Cadence using the default settings. **MOUSE CLICKS** will be indicated with bold, small caps. *Entered text* with be indicated with italicized text.

1. Run Cadence

Environment	Action
Desktop	**START → ALL PROGRAMS → CADENCE PSD 15.0 → CAPTURE CIS**
Studio Suite Selection	Click the menu down arrow. Scroll down the list. **PCB DESIGN EXPERT WITH CAPTURE CIS** **OK**
Capture CIS	**FILE → NEW → PROJECT**
New Project	In the Location box, enter a directory path. Use the **BROWSE** button to select a location if necessary. Replace names with spaces with the *~1* notation. In the Name box, enter a project name without spaces. **ANALOG OR MIXED A/D** **OK**
Create PSpice Project	**CREATE A BLANK PROJECT** **OK**

2. Draw Schematic

Capture CIS	On the right side of the main program window is a schematic toolbar. On this toolbar, select **PLACE PART**.
Place Part	Select all of the Libraries in the bottom left list. **REMOVE LIBRARY**
CAP0062	(Ignore this warning) **OK**
Place Part	Add Library
Browse File	Make sure that you are in the *pspice* directory. Hold down CTRL and select the libraries: SOURCSTM.OLB

	DIG_PRIM.OLB OPEN
Place Part	Select **DIG_PRIM** and then select the desired part.
Capture CIS	On the SCHEMATIC1:PAGE1 window, place as many elements of the part as desired. Continue using PLACE PART until all of the circuit elements are laid out on the page. Once all of the parts are laid out, click PLACE PART and select DIGSTIM1 in the **SOURCSTM** library. Place one element of DigStim1 for each of the circuit inputs. If at any time while placing a part you want to stop placing, hit the *ESC* key. You can also select a part and hit *Delete* or *DEL* to remove it from the schematic.
	Once all elements are added, use the PLACE WIRE tool to make wire connections between all of the elements. Don’t forget to make wire connections for the outputs as well.

3. Define components

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, (e.g., *400n*) Enter an on-time (e.g., *200n*) 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.

4. Simulate Schematic

Capture CIS	In the top tool bar, select the New Simulation Profile tool.
New Simulation	Enter a name for the simulation. CREATE
Simulation Settings	PROBE WINDOW tab LAST PLOT OK
Capture CIS	Select the RUN PSPICE tool.
SCHEMATIC1	The simulation window will be initially blank. Select the ADD TRACE tool
Add Traces	Select the signals in the order you want them displayed. Hint: you will see the items added to the "Trace Expression" list at the bottom of the window. Hint: de-select ALIAS NAMES to remove the simulator names and only see your defined names. OK
SCHEMATIC1	View your simulation results.

This ends the formal quick start. If your simulation has any errors, you will need to return to the CAPTURE CIS schematic page to make corrections. Use your knowledge of how the circuit *should* behave to help you understand where it has gone wrong.

When the simulation is working correctly, print it out and annotate it.