


PLOTTING WAVEFORMS AND USING CALCULATOR

Once the simulation is done, the window having the desired input and output waveforms will pop up and you can place at the desired place by clicking wherever you want. If your simulation is going to take a long time (say 10 hours) and if you are interested in looking at the outputs in the middle of the simulation and see if the outputs are as desired, you can do it at any point without stopping the simulation by clicking on the 'Plot output' (last tab in the right side of the Analog environment).


In Waveform window

- Click Graph... Split All Strips or just simply Click 'split all strips' or  to separate multiple waveforms plotted one over the other initially.
- To re-arrange the order of the waveforms click and drag the waveforms that need to be swapped.

To evaluate expressions using the Calculator


- In Analog environment click on 'Tools' and select 'Calculator'.

To evaluate the crossing point of a digital signal:

- In Calculator window, select the desired function say, (vt) and
- Click on the wire of the parameter you want to choose in schematic window.
- Then in the calculator, click on special functions and select 'cross'.
- In 'Threshold crossing' window, enter the 'Threshold Value', 'Edge Number' and also select 'Edge Type' (rising/falling/either) and click ok. Now you will see an expression to evaluate the crossing point in the calculator window.
- Now go back to Analog environment and click on 'setup outputs' or  and in the new window that appears, enter a name for the expression and click on 'Get Expression', finally click ok. You will now see the expression to be evaluated along with the other outputs to be plotted in the Analog environment.
- Next click on the 'Results' tab in Analog environment and choose 'Printing/Plotting options' and select 'Mark design variables' and 'scalar outputs' options to display the calculated value in the waveform window. Click ok and run the simulation.

Example Calculation of Inverter Output Rising Delay: from input a to output z, assuming $V_{DD} = 1.2V$

- Go through steps...
- In Calculator window, select the desired function say, (vt) and
- Click on the output 'z' in schematic window.
- Then in the calculator, click on special functions and select 'cross'.
- In 'Threshold crossing' window, enter 'threshold value' = 0.6, 'Edge Number' = 1 and also select 'Edge Type' –falling and click ok. Now you

- will see an expression to evaluate the crossing point in the calculator window.
- Copy this expression.
 - Now click on vt and select input 'a' in schematic window.
 - In 'Threshold crossing' window, enter ' threshold valve' = 0.6, 'Edge Number' = 1 and also select 'Edge Type' –falling and click ok. Now you will see an expression to evaluate the crossing point in the calculator window.
 - Now subtract the copied expression from this expression. (hit '-' and paste the copied expression).
 - Final expression = (cross(VT("/a") 0.6 1 "falling" nil nil) - cross(VT("/z") 0.6 1 "rising" nil nil))
 - Now go back to Analog environment and click on 'setup outputs' or  and in the new window that appears, enter a name for the expression and click on 'Get Expression', finally click ok. You will now see the expression to be evaluated along with the other outputs to be plotted in the Analog environment.
 - Next click on the 'Results' tab in Analog environment and choose 'Printing/Plotting options' and select 'Mark design variables' and 'scalar outputs' options to display the calculated value in the waveform window. Click ok and run the simulation.

To measure values directly from the waveforms using cursors:

1. Select Markers and choose Crosshair marker A or just press 'a' on keyboard, place the marker wherever you want to measure.
2. Select Markers and choose Crosshair marker B or just press 'b' on keyboard, place the marker wherever you want to measure.
3. You can see the values at the bottom of the window.