

Hspice Quick Start

By:

**Michael Chan
Hamid Mahmoodi**

**Nano-Electronics & Computing Research Lab
School of Engineering
San Francisco State University
San Francisco, CA**

Spring 2011

HSPICE and WaveView Tutorial

Hspice is used for circuit simulation and WaveView is used to view output waveforms. To setup Hspice and WaveView type the following commands after login to your account:

```
ssh
source /packages/synopsys/setup/full_custom.csh
```

HSPICE simulation is run by typing `hspice input_file > output_file`, where *input_file* is the name of the SPICE netlist file and *output_file* is the name of the file the output report of hspice is saved in.

Example SPICE file:

A sample SPICE file containing the description of a CMOS inverter is given below. This file must be saved as a text file. Save this enlist in a text file named inv.sp and move it to your account on the server.

```
*Notice the first line is treated as a comment line
*netlist of CMOS inverter
.include PTM_HP_16nm.txt

*MOSFETs
m1 out in vdd vdd pmos l=16n w=100n
m2 out in 0 0 nmos l=16n w=50n

*Output capacitance
cload out 0 5f

*voltage sources
vdd vdd 0 1.0
vin in 0 0 pulse 0 1.0 0 50p 50p 2n 4n

.options list node post

*Transient Analysis
.tran 10p 10n

.end
```

The first line of the SPICE file is always a comment line. Therefore any statements on the first line will be ignored. Moreover any expression after * is treated as comment.

The .include line includes the model file PTM_HP_16nm.txt, which is assumed to be in the current running directory.

Where to get model files: Predictive model files for advanced CMOS technologies can be downloaded from the following website: <http://ptm.asu.edu/>
Click on the latest models and download 16nm PTM HP model to a text file and save it as PTM_HP_16nm.txt and move the file to your account on the server.

The next two lines in the netlist are a pmos and an nmos transistor, respectively. After the transistor name (which must begin with **m**), the source, gate, drain, and bulk nodes are given. The next entry is the model name (nmos and pmos). The length and width are specified. The process is a 16 nm process, so 16 nm is the minimum gate length. Source and drain perimeters and areas can also be specified here.

A capacitor of 5fF from node out to node 0 is given on the next line. Notice that node 0 is the ground line and any spice netlist must have a node called 0 node.

The two supply nets are defined next. The DC supply voltage is defined as Vdd and its value is set at 1.0 V.

An input pulse voltage source is defined from 0 to 1.0 V with 0 delay, 50ps rise time, 50ps fall time, 2n pulse width, and 4ns repetition period.

.options list node post instructs HSPICE to write an output file ending in .tr0 containing the simulation waveforms.

.tran indicates a transient analysis with a plot interval of 10ps and simulation duration of 10ns.

.end signifies the end of the SPICE netlist.

This command runs hspice on this file: **hspice inv.sp > inv.out** as shown below:

```
hafez:% hspice inv.sp > inv.out
>info:          ***** hspice job concluded
real 0.38
user 0.03
sys 0.01
```

If you see the message “hspice job concluded”, this is an indication that the simulation ran successfully without any errors in the netlist. If there are any mistakes in the netlist you would get the message “hspice job aborted”. In that case, please open the output file (inv.out) and search for error to see the cause of the error and fix the error (modify the netlist) and run hspice again.

After the simulation is finished, several files are created by HSPICE:

inv.ic: Text file containing the circuit initial conditions
inv.st0: Text file containing a summary of the simulation
inv.tr0: Binary file containing transient analysis waveforms

After simulation, the waveforms can be viewed using WaveView. Type **wv** to launch the WaveView program and plot the results. The following window will appear.

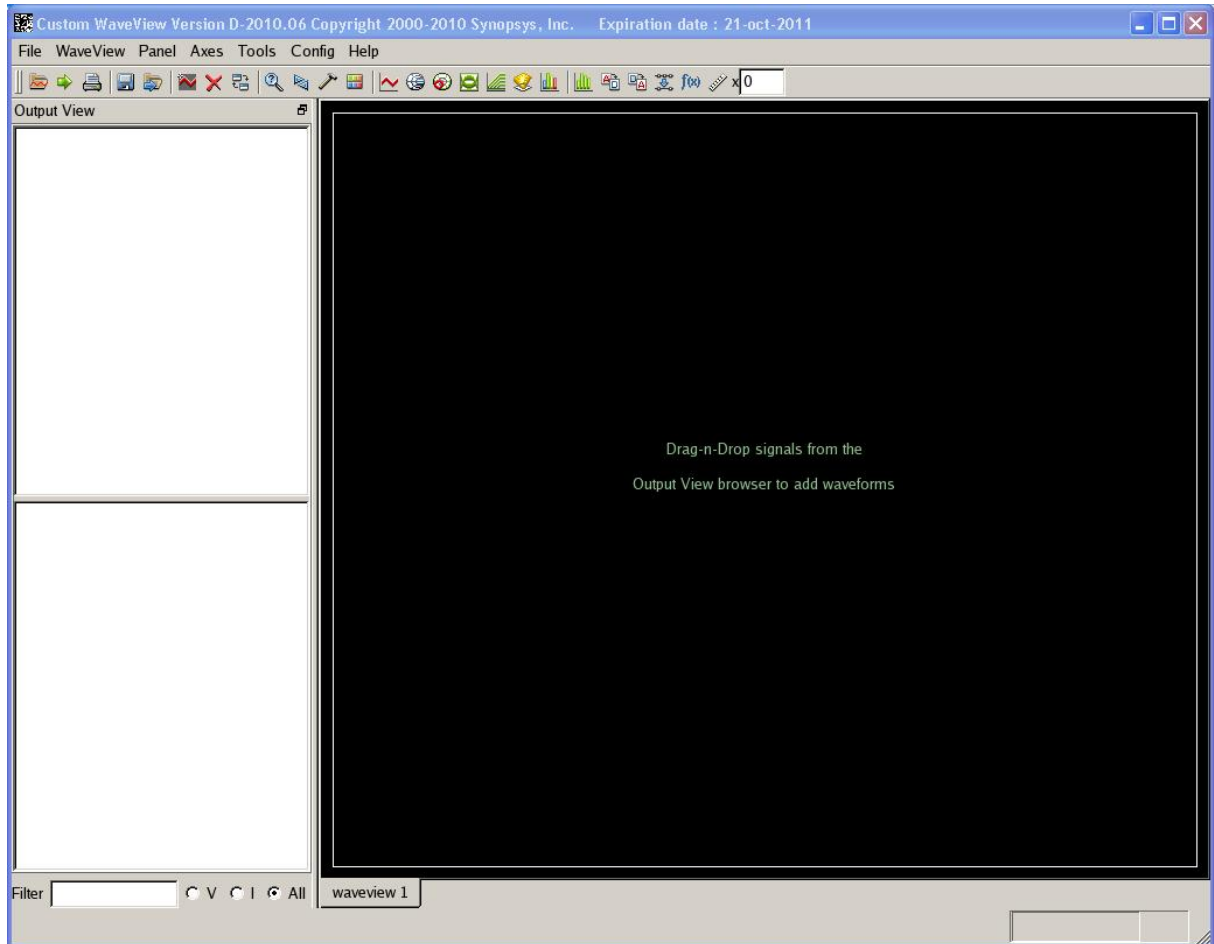


Figure 1: WaveView Window

In the WaveView window select **File > Import Waveform File**. Double click on inv.tr0, the file that contains the transient analysis.

In the WaveView window, double click **D0:inv.tr0** in the output view followed by clicking **toplevel** just below it. This should display your available signals to graph in the box below. See figure 2 below for reference.

Double click the signal names in the bottom left hand box to display their respective waveforms. In this case, double click v(in) and v(out) to display them. See figure 2 below for reference.

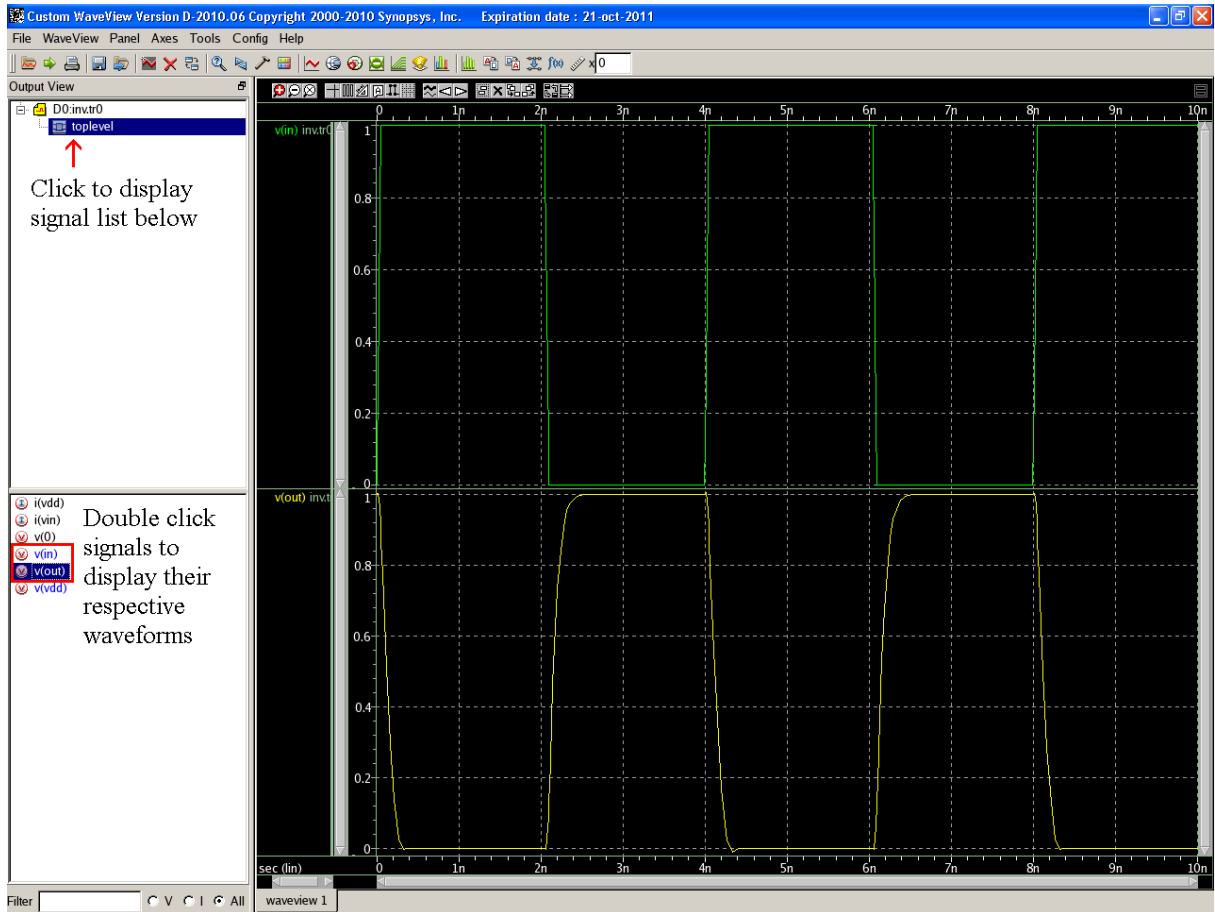




Figure 2: Vout and Vin Waveforms

WaveView Measurement Tools and Features:

The measurement tool in the WaveView window provides many methods for measuring the waveforms. Here is a list of several features and measurements you can do in WaveView:

Zooming In and Out:

To zoom in on your waveform, click  to do a vertical or horizontal zoom by dragging a box over the waveform with the mouse. Click  to unzoom. Also you can press **X** for a full unzoom of the waveform.

Grouping and Ungrouping Signals:

First off, it is handy to know how to group and ungroup waveforms in WaveView. To group waveforms together, select the names of the signals to be grouped in the waveform signal list and right click on the mouse to open a drop down menu and select **Group**. See figure 3 below for reference.

To ungroup waveforms, select the group of signals in the waveform signal list and right click on the mouse to open a drop down menu and select **Ungroup**. See figure 4 below for reference.

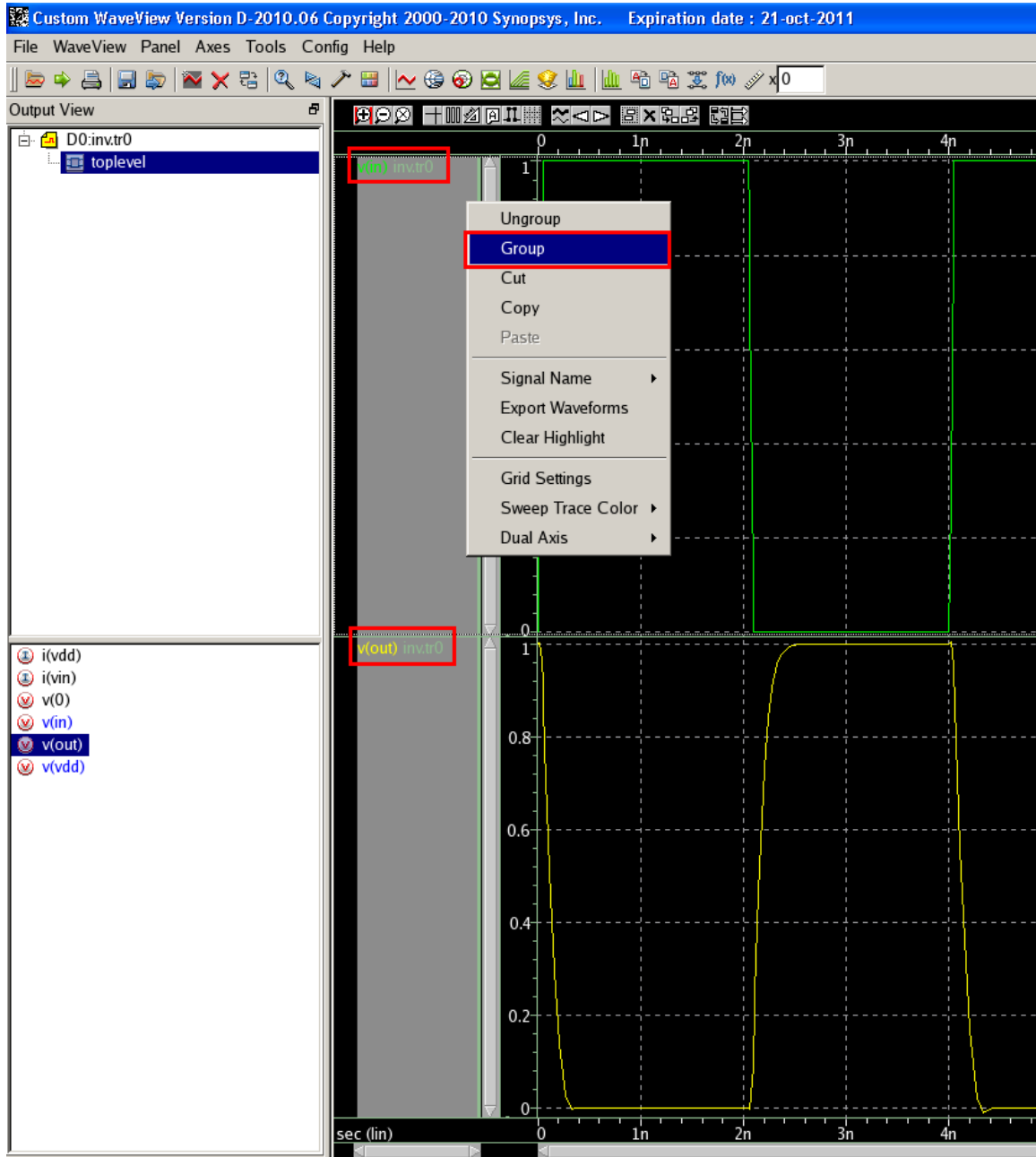


Figure 3: Grouping Waveforms

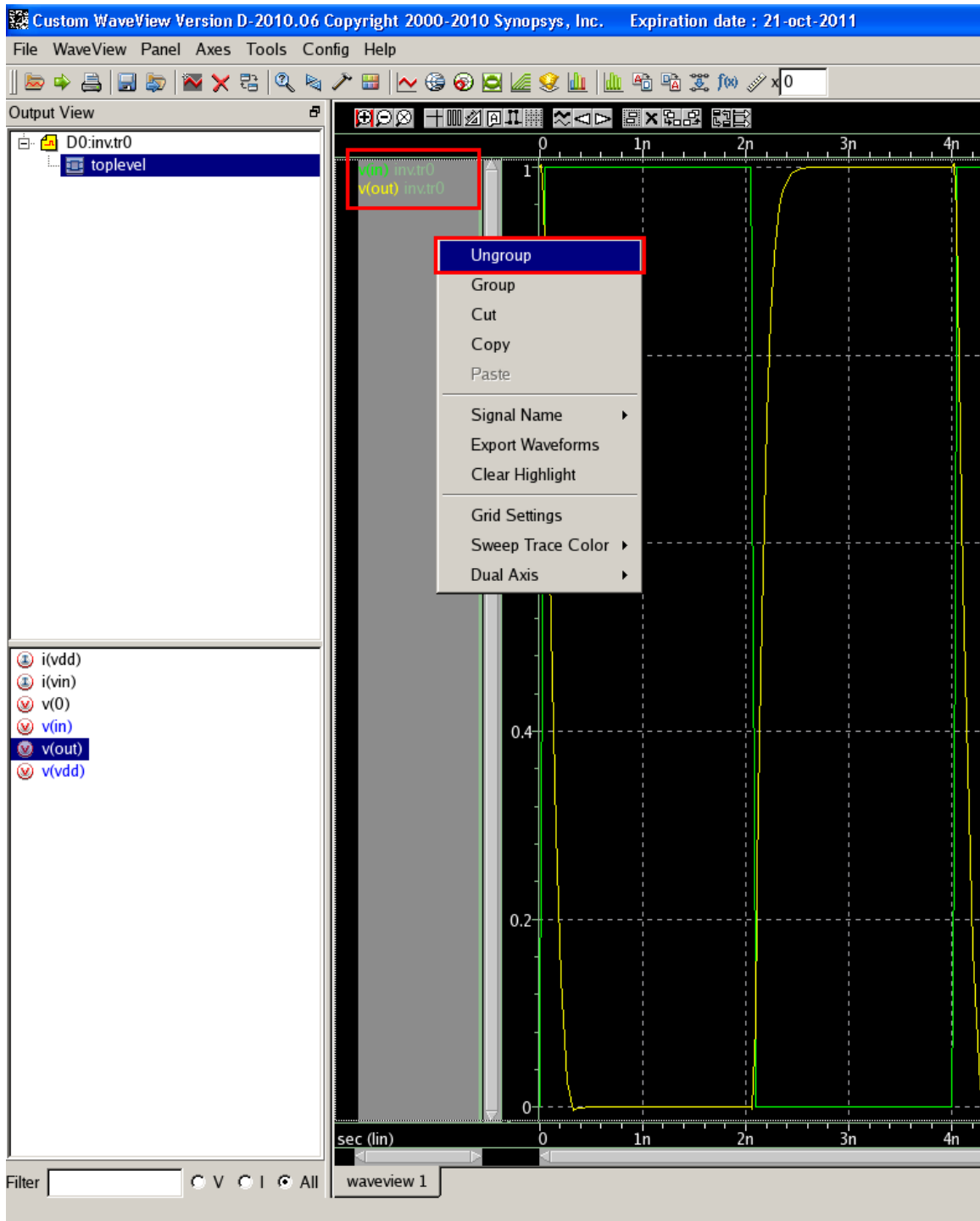



Figure 4: Ungrouping Waveforms

Delay Measurements of Vin and Vout at 50% to 50%:

To measure delay between the input and output signals of the inverter at 50% group the

v(out) and v(in) waveforms together so the two waves overlap each other. Open the measurement tool by going to **Tools → Measurement...** or by clicking  in the WaveView window. Click the **All** tab in the measurement tool window and fill out the options as shown below in figure 5. Click **Ok** when done.

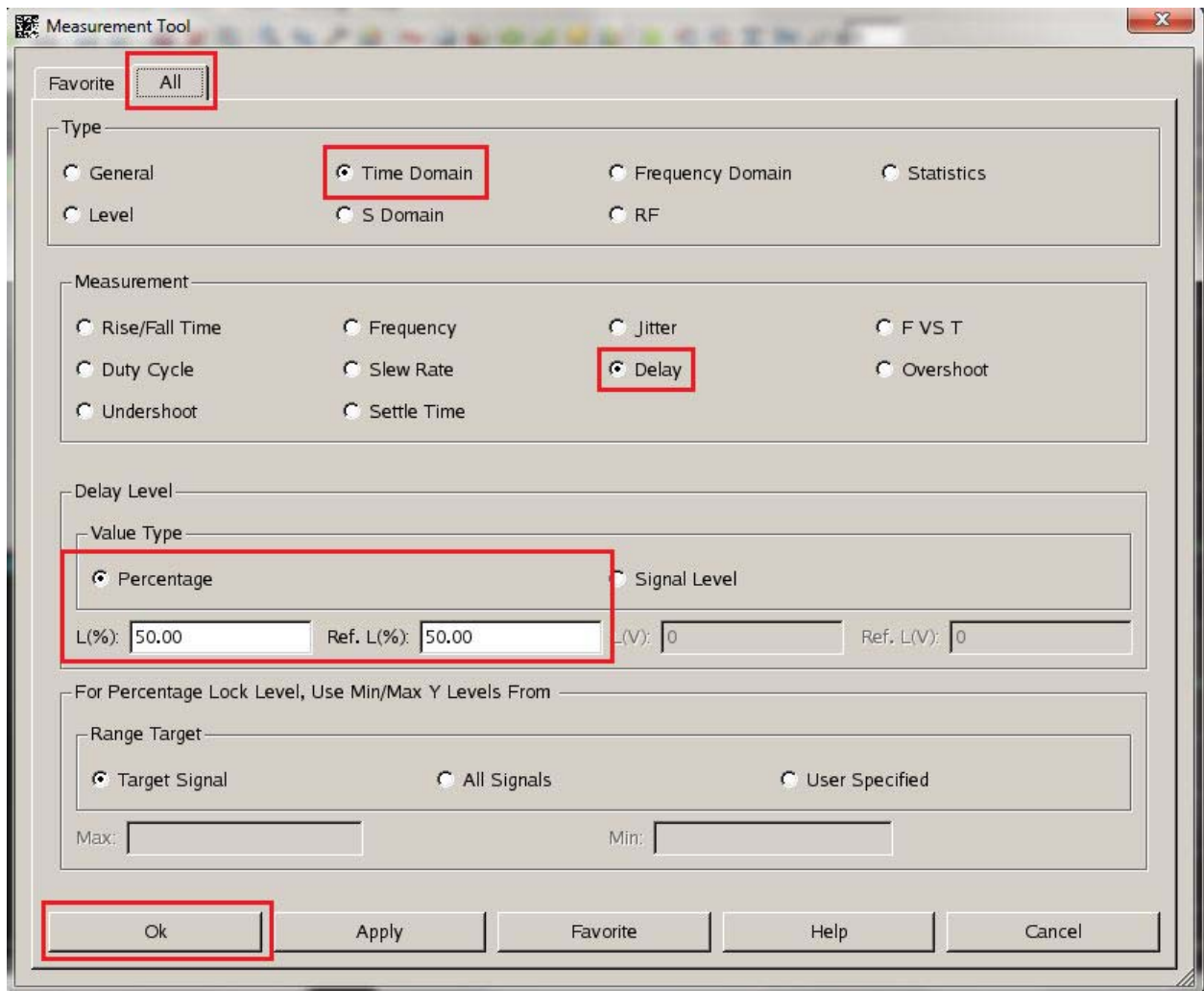


Figure 5: Delay Measurement Tool

After clicking **Ok**, a delay measurement box will appear on the waveform. Just drag the box to the waveform you want to measure (in this case the v(in) and v(out) overlapping waveforms) and the delay value will appear in the box. You can also drag the delay measurement box along different valid points of the waveform to get more delay values. See figure 6 below for the delay measurements box.

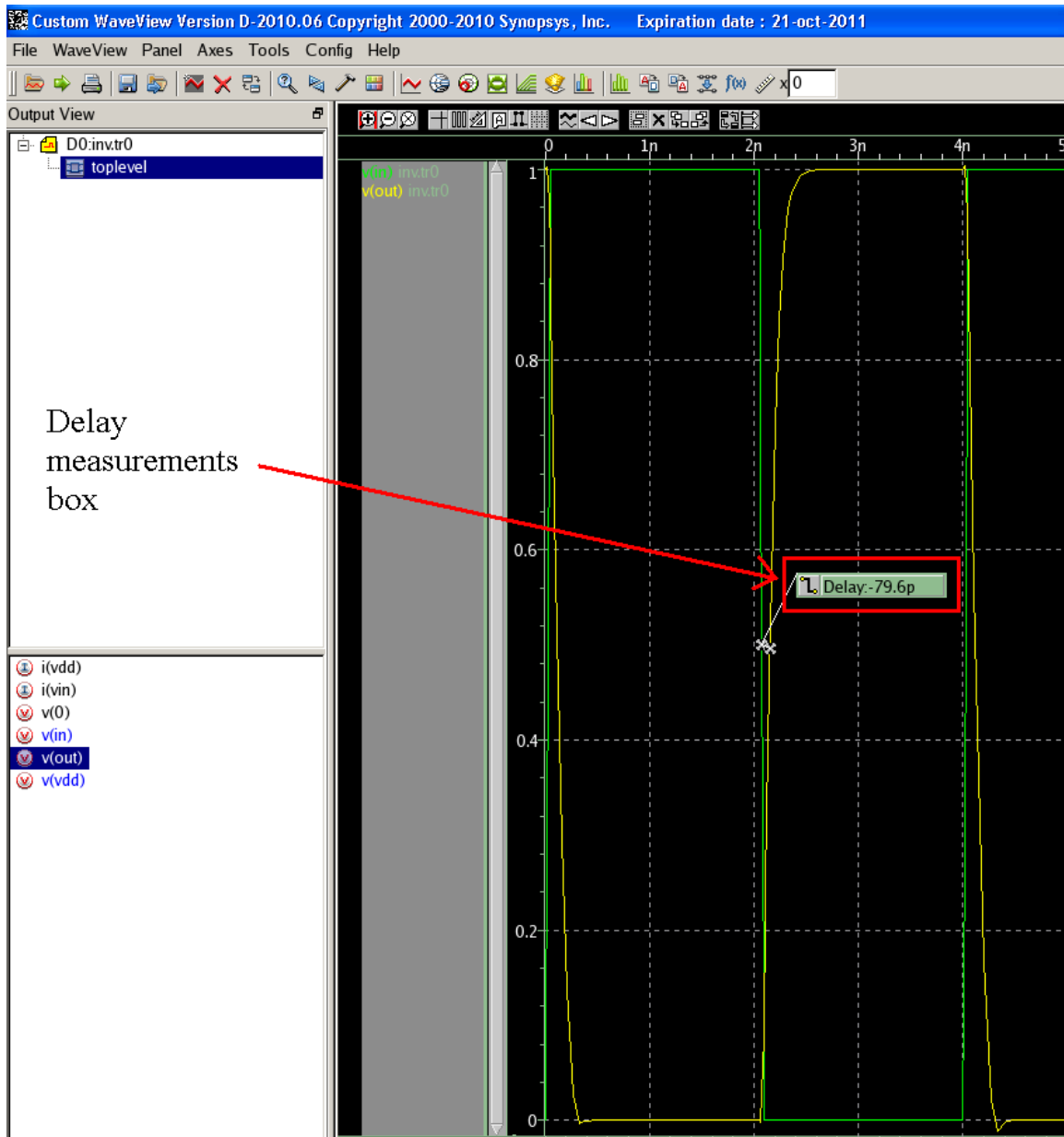



Figure 6: Delay Waveform Measurement

Rise/Fall Time Measurements at 90% and 10% for v(out):

To measure fall and rise time ungroup all the waveforms. Open the measurement tool by going to **Tools** → **Measurement...** or by clicking  in the WaveView window. Click the **All** tab in the measurement tool window and fill out the options as shown below in figure 7. Click **Ok** when done.

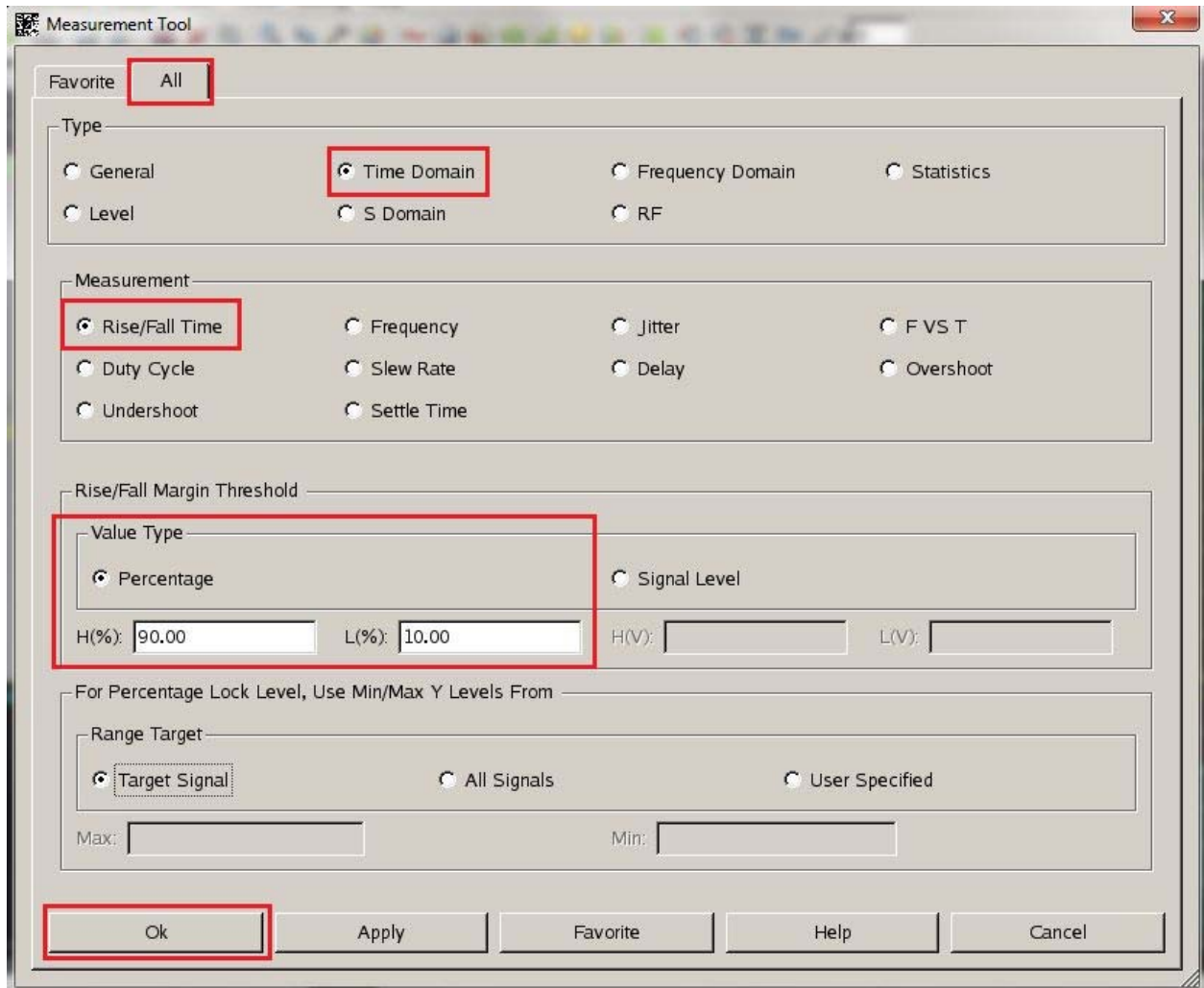


Figure 7: Rise/Fall Measurement Tool

After clicking **Ok**, a rise/fall measurement box will appear on the waveform. You can drag the rise/fall measurement box along the v(out) waveform to get the rising and falling delay times. Notice that when the rise/fall measurement box shows a rising red curve the tool is measuring rising delay time from 10% of the signal to 90% of the signal. Also when the rise/fall measurement box shows a falling green curve the tool is measuring the falling delay time from 90% of the signal to 10% of the signal. See figure 8 below for reference, it shows two measurement boxes.

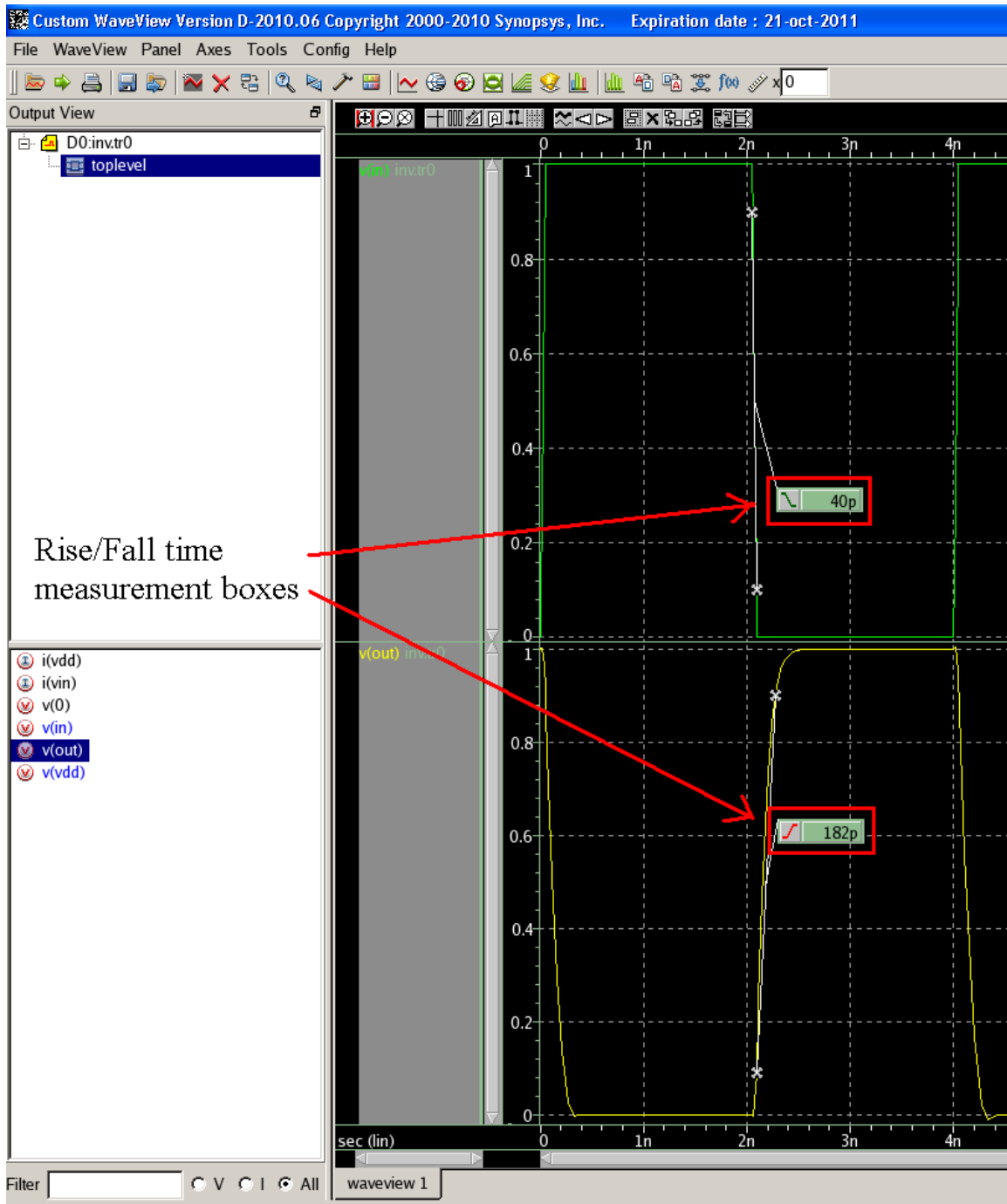



Figure 8: Rise/Fall Waveform Measurement

Frequency Measurement:

To measure signal frequency, ungroup all the waveforms. Open the measurement tool by going to **Tools → Measurement...** or by clicking  in the WaveView window. Click the **All** tab in the measurement tool window and fill out the options as shown below in figure 9. Click **Ok** when done.

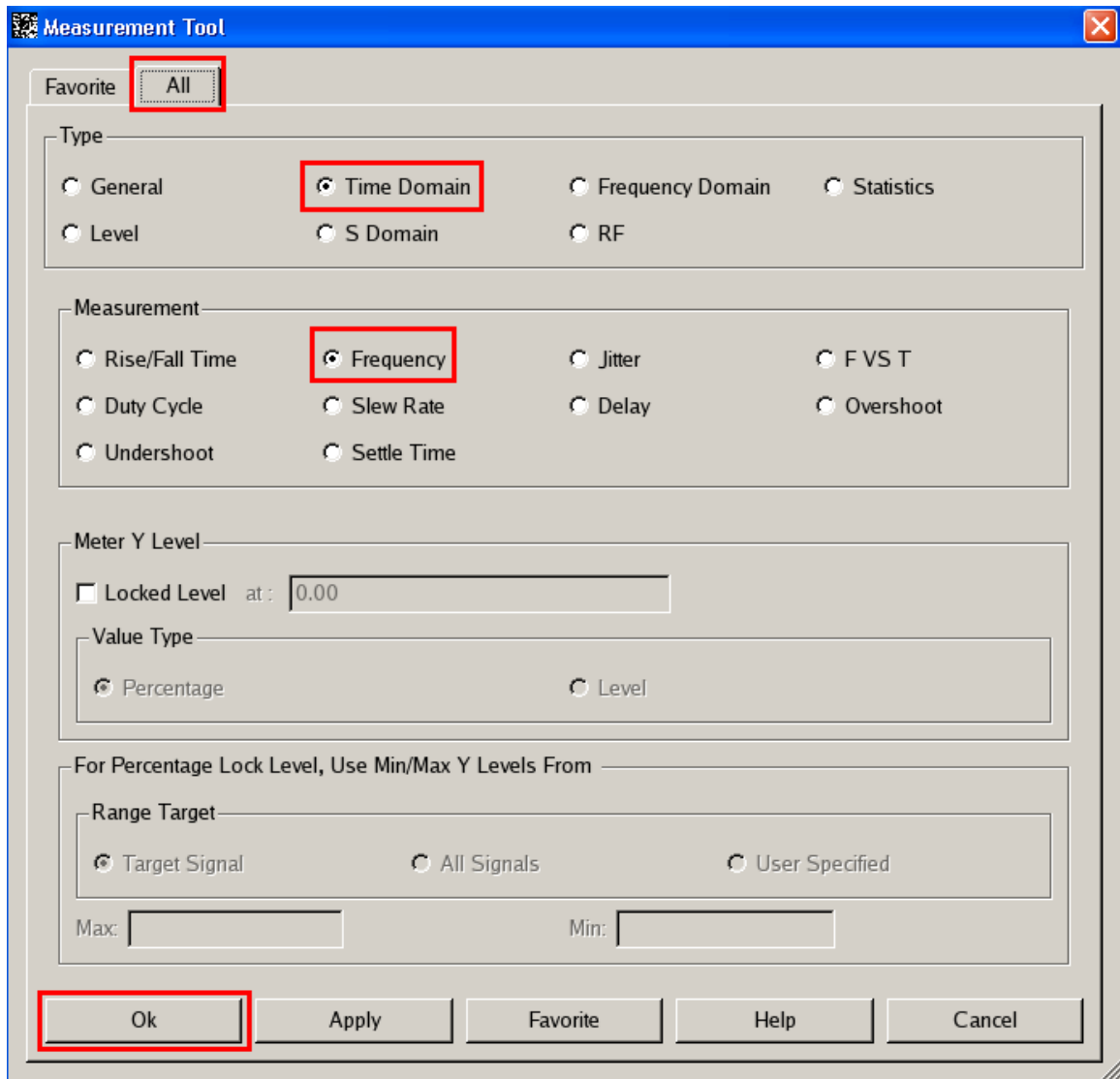


Figure 9: Frequency Measurement Tool

After clicking **Ok**, a frequency measurement box will appear on the waveform. Drag the box toward the waveform until it displays the frequency value. See figure 10 below for reference.

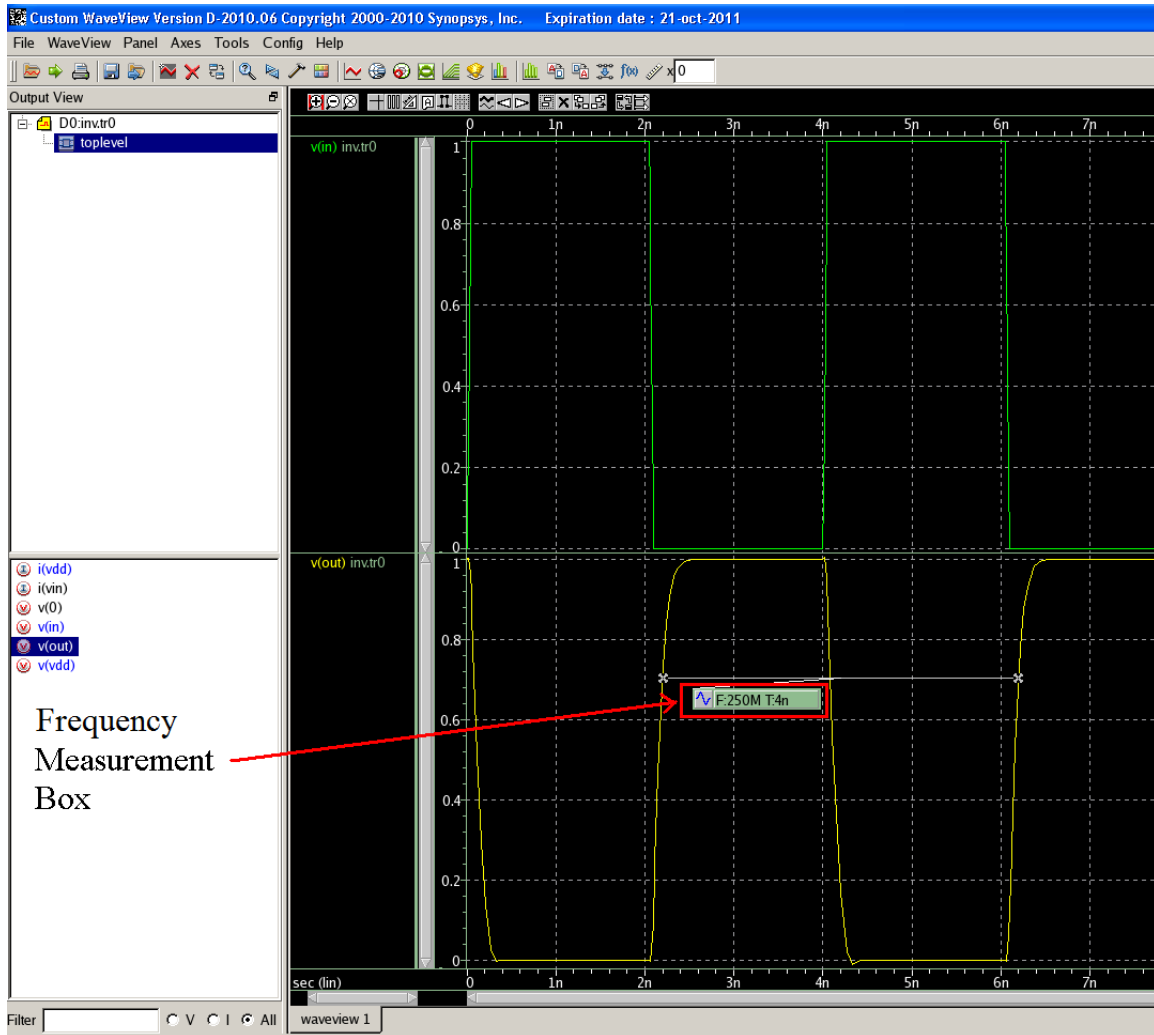


Figure 10: Frequency Waveform Measurement

Average Current Measurement:

To measure average current, ungroup all the waveforms. Delete the v(out) and v(in) waveforms and double click **i(vdd)** from the left side signal list so only that signal's waveform displays. You can delete a waveform by selecting its name in the right signal list and pressing **delete** on the keyboard and clicking ok. You can always recover these signals later by double clicking the signals in the left signal list. See figure 11 below for reference.

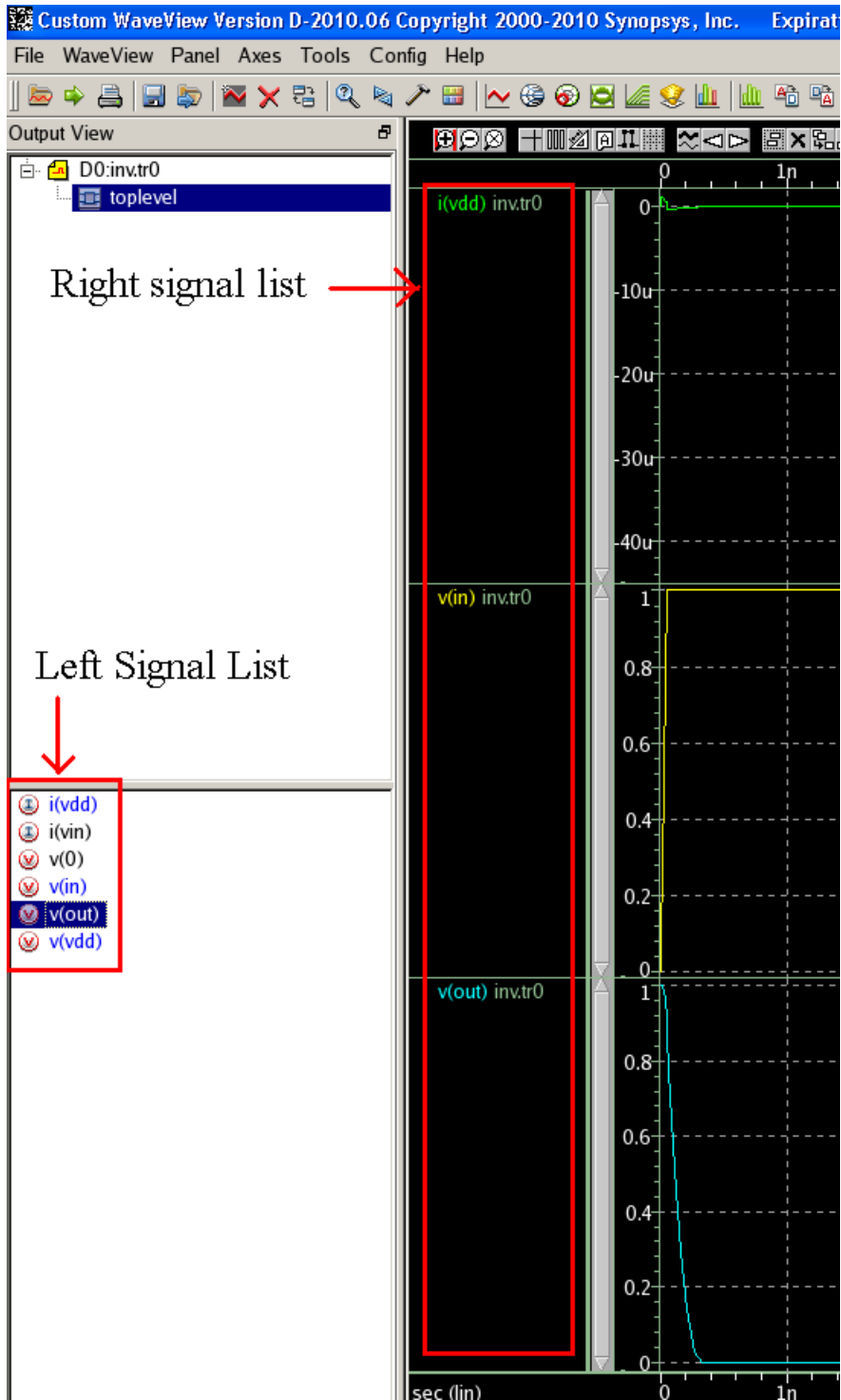



Figure 11: Left and Right Signal List

Open the measurement tool by going to **Tools** → **Measurement...** or by clicking  in the WaveView window. Click the **All** tab in the measurement tool window and fill out the options as shown below in figure 12. Click **Ok** when done.

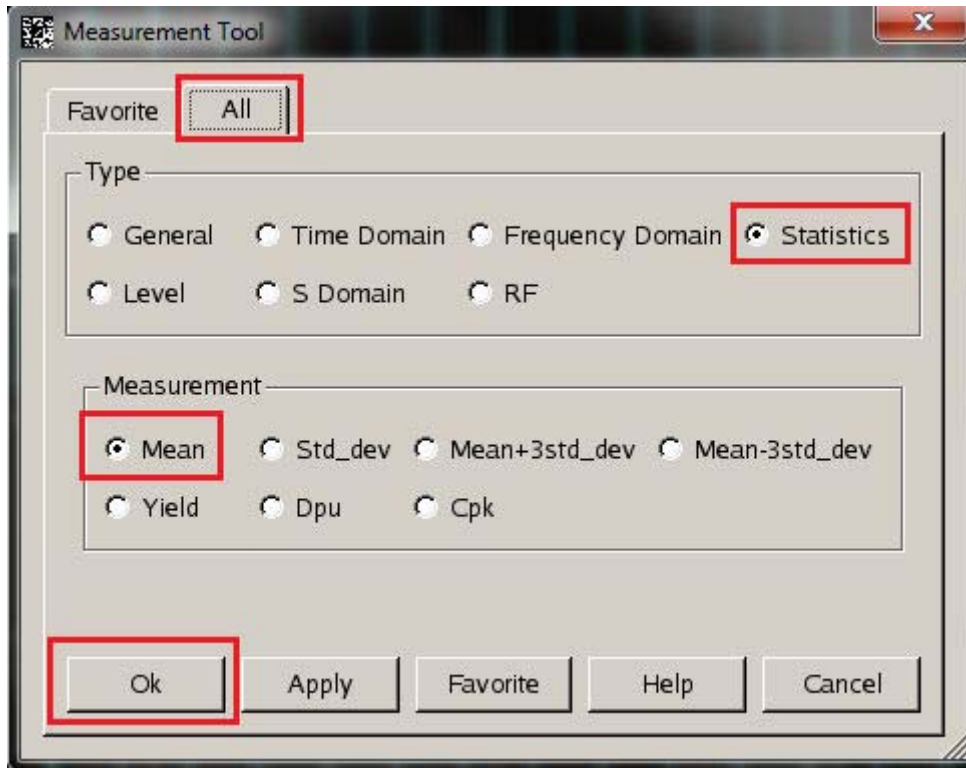


Figure 12: Average Measurement Tool

After clicking **Ok**, an average measurement box will appear on the waveform. Drag the box toward the waveform until it displays the average value. See figure 13 below for reference.

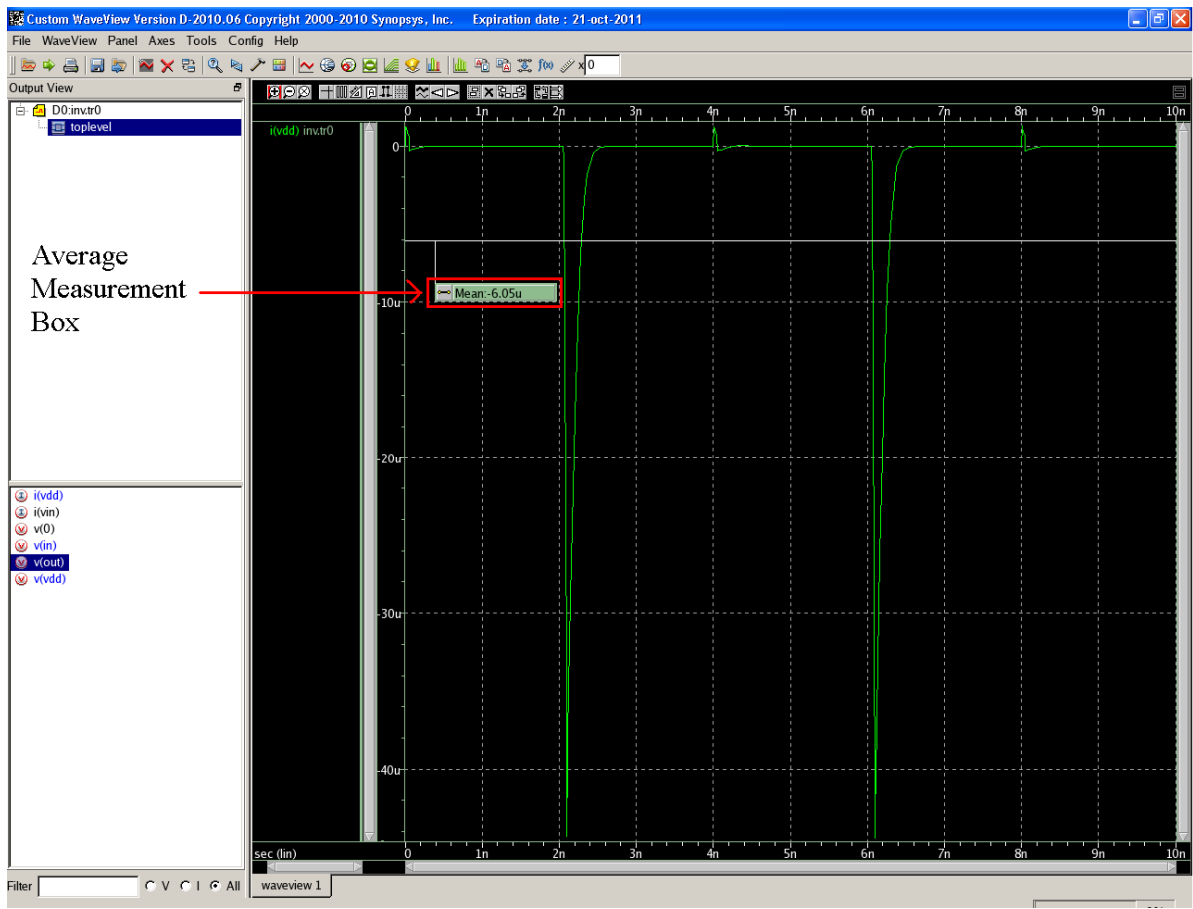


Figure 13: Average Waveform Measurement