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:

csh
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.

```
***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 10f

***voltage sources
vdd vdd 0 0.7
vin in 0 0 pulse 0 0.7 0 10p 10p 1n 2n

.options list node post

***Transient Analysis
.tran 10p 4n

.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 10fF 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. He DC supply voltage is defined as Vdd and its value is set at nominal supply voltage for 16nm technology which is 0.7 V.

An input pulse voltage source is defined from 0 to 0.7 V with 0 delay, 10ps rise time, 10ps fall time, 1n pulse width, and 2ns 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 4ns.

.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 neltist) 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.
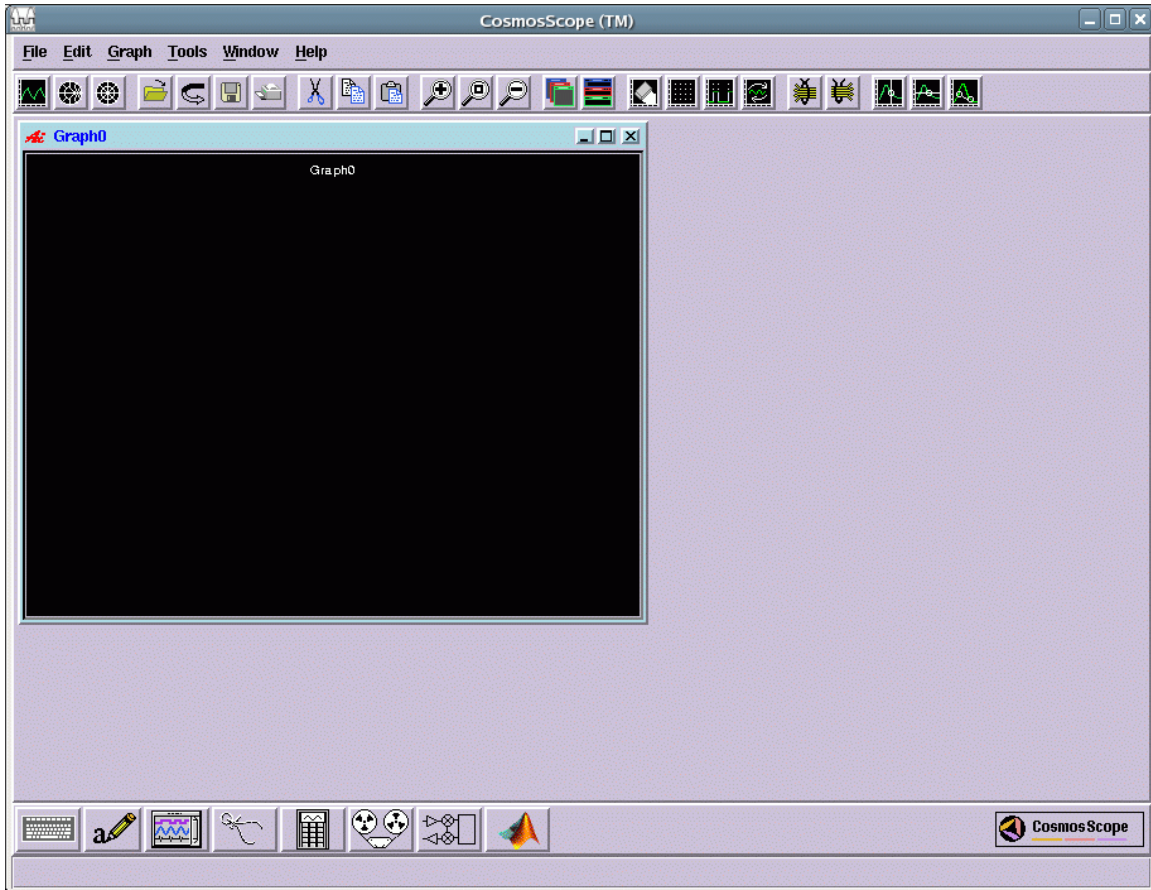

Click on **File > Open > Plotfiles**. Click on inv.tr0, the file that contains the transient analysis.

A signal manager window and the Plot File window will open up. In the Plot File window, plot v(in) by double clicking v(in), or by selecting v(in) and clicking **Plot**. The input waveform of the inverter will open up.
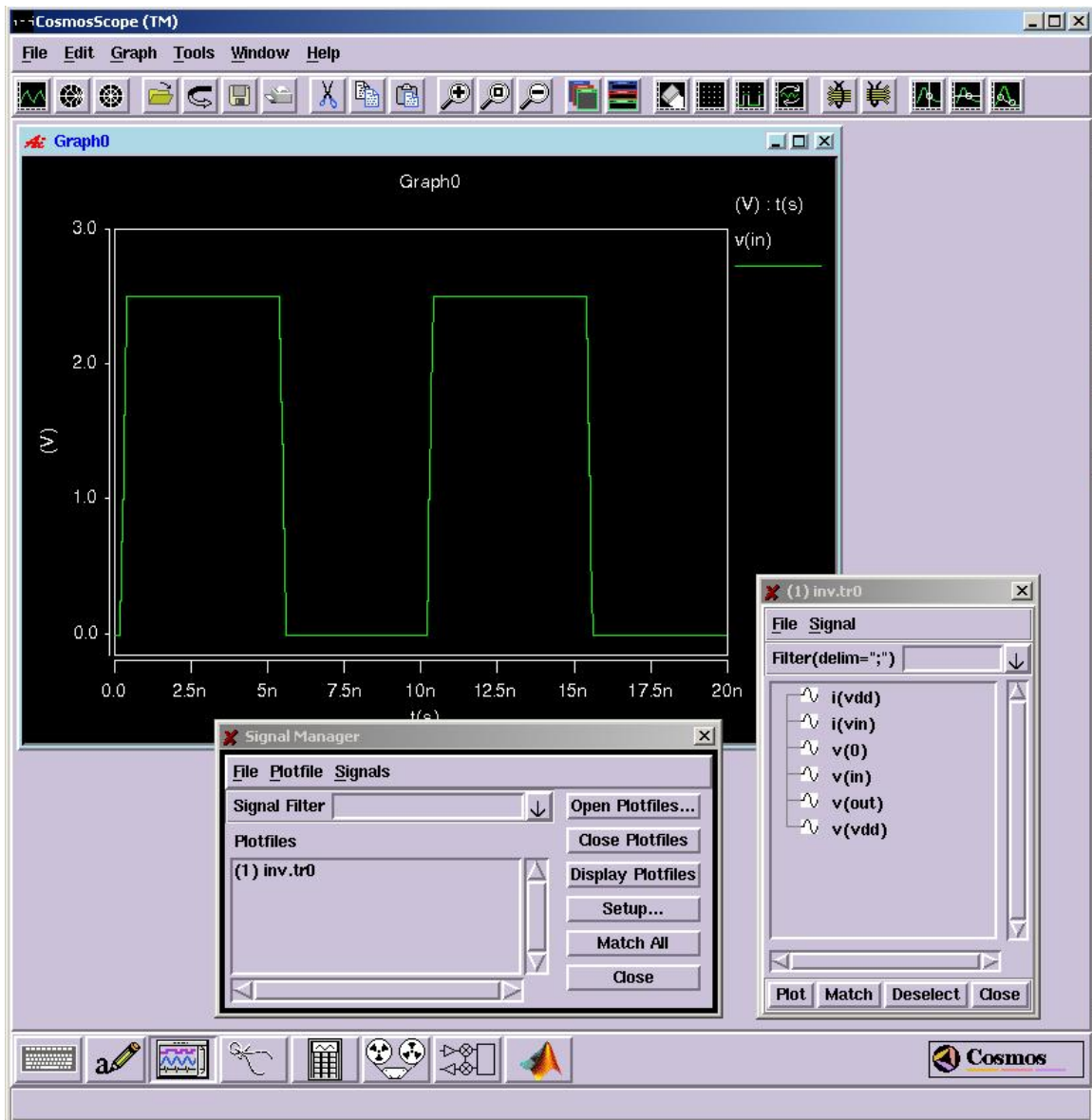
Now plot v(out), the same way you plotted v(in). This will open another graph with v(out).
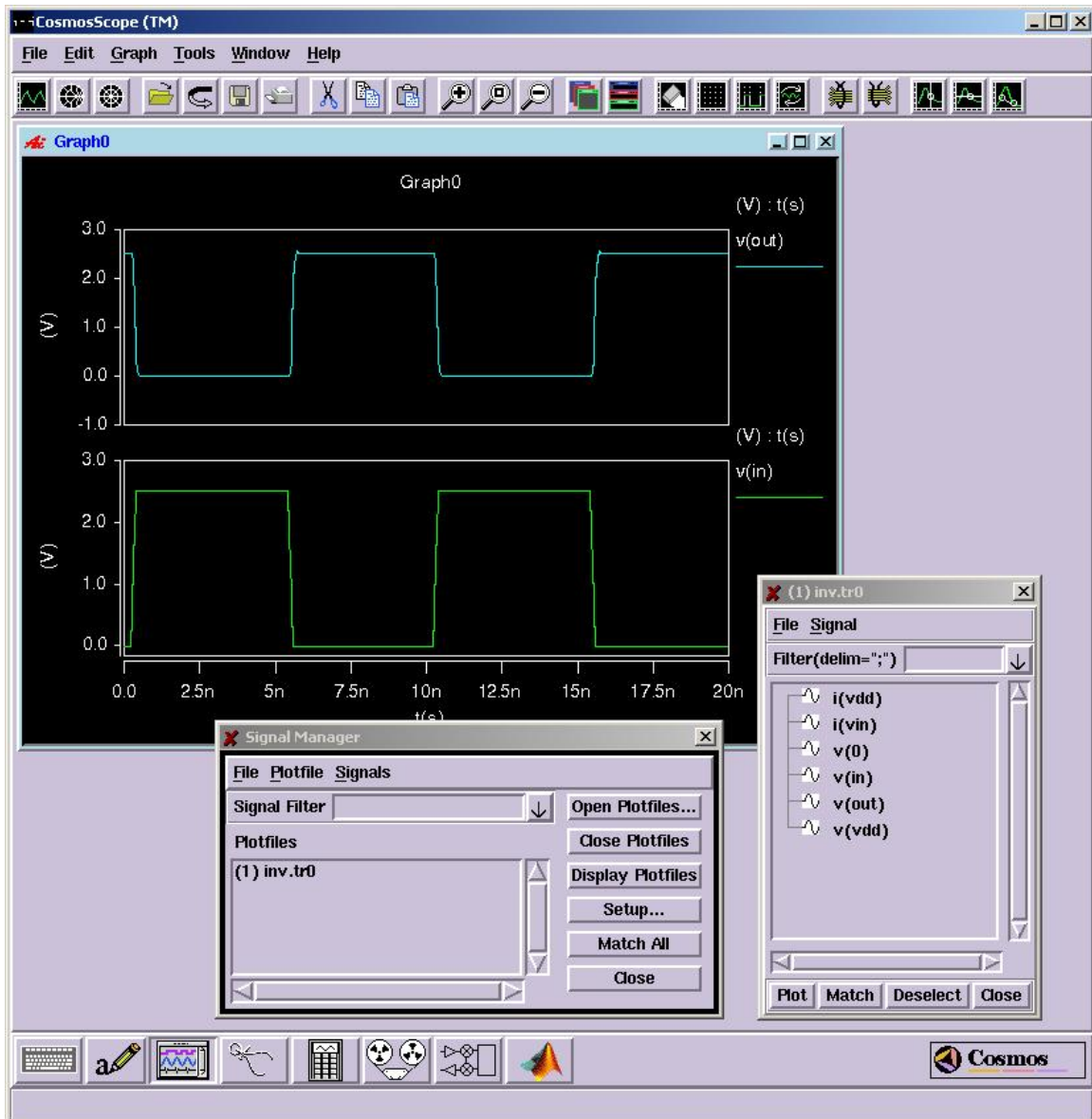
Now to compare them on the same graph, simply click and drag the title "v(out)" from the top graph to the bottom graph where v(in) is plotted.
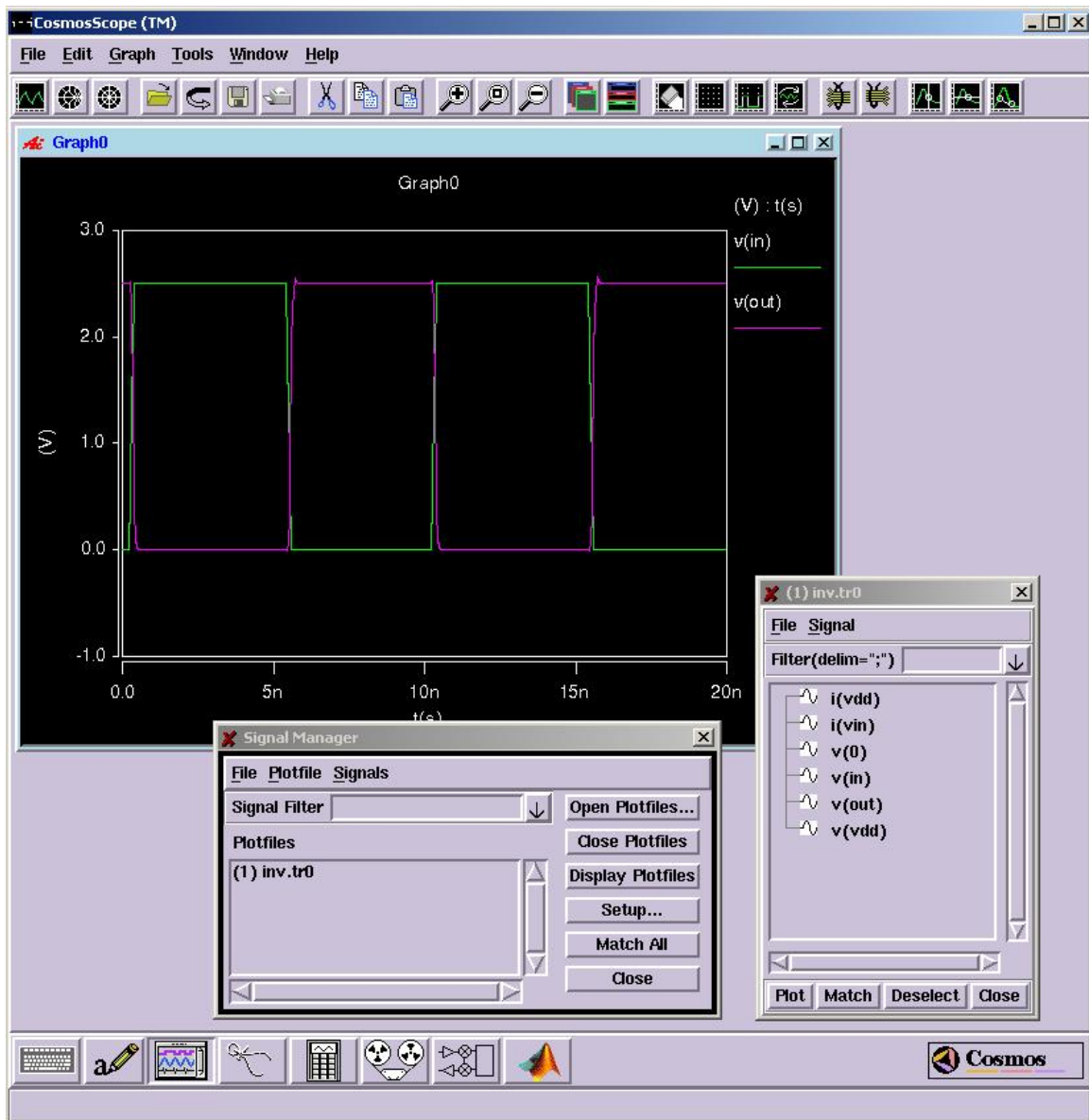
If you want to zoom in on a certain part of the plots, move your mouse cursor to the x-axis and click your starting point and drag it to your desired ending point.
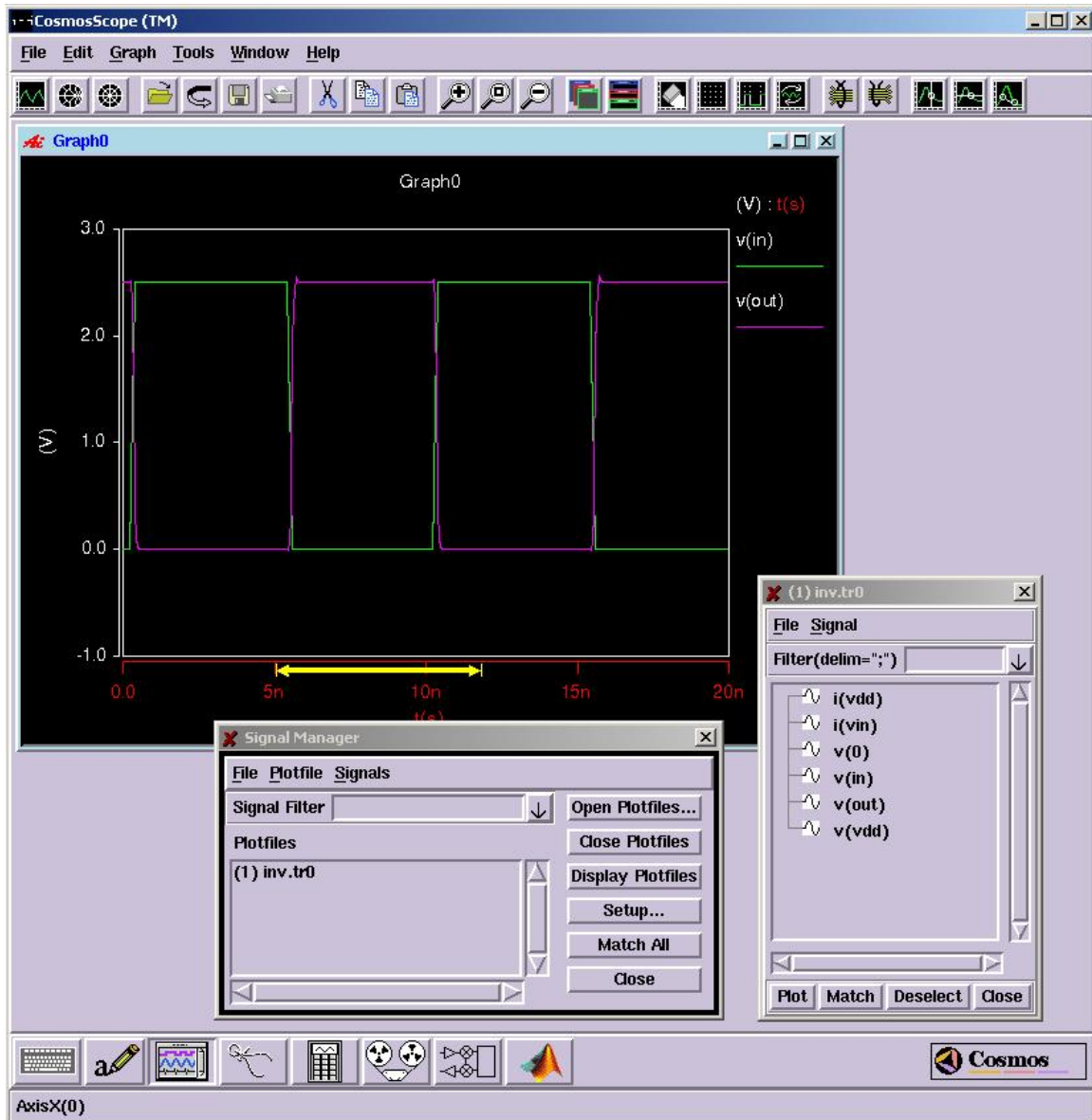
The graph will auto-adjust to your specified zoom length. To return to the default view, click on the magnifying glass with the square inside of it.