

HSPICE and CosmosScope Tutorial

Hspice is used for circuit simulation and CosmosScope is used to view output waveforms. To setup Hspice and CosmosScope type the following commands:

```
source /packages/synopsys/setup/hsp.csh
source /packages/synopsys/setup/cosmoscope.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 of hspice is saved in.

Example SPICE file:

A sample SPICE file containing the description of a CMOS inverter is given:

CMOS Inverter (inv.cir)

```
.lib "tsmc_018um_model.txt" cmos_models

m1 out in vdd vdd cmosp l=0.18u w=1.8u
m2 out in 0 0 cmosn l=0.18u w=0.9u
vdd vdd 0 2.5
vin in 0 0 pulse 0 2.5 200p 200p 200p 5n 10n
cload out 0 20f

.options post
.tran 200p 20n
.print tran v(in) v(out)
.end
```

The first line of the SPICE file is always a comment line. Therefore any statements on this line will be ignored.

The .lib line includes the model file tsmc_018um_model.txt, which is assumed to be in the current running directory. (**Where to get model files:**

some model files can be downloaded from this website:

<http://www.ee.vt.edu/~ha/cadtools/hspice/hspice.html>

Predictive model files for more advanced technologies can be downloaded from the following website: <http://www.eas.asu.edu/~ptm/>)

The next two lines 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. Next is the model. The length and width are specified. The process is a 0.18um process, so 0.18um is the minimum gate length. Source and drain perimeters and areas can also be specified here.

The two supply nets are defined next.

A pulse voltage source is defined from 0 to 2.5 V with 100ps delay, 100ps rise time, 100ps fall time, 2n pulse width, and 4ns repetition period.

A capacitor of 20fF from node out to node 0 is given on the next line.

.options 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 200ps and simulation duration of 20ns.

.print specifies the nodes to be printed to the .out file.

.end signifies the end of the SPICE stack.

This command runs hspice on this file: **hspice inv.cir > inv.out**

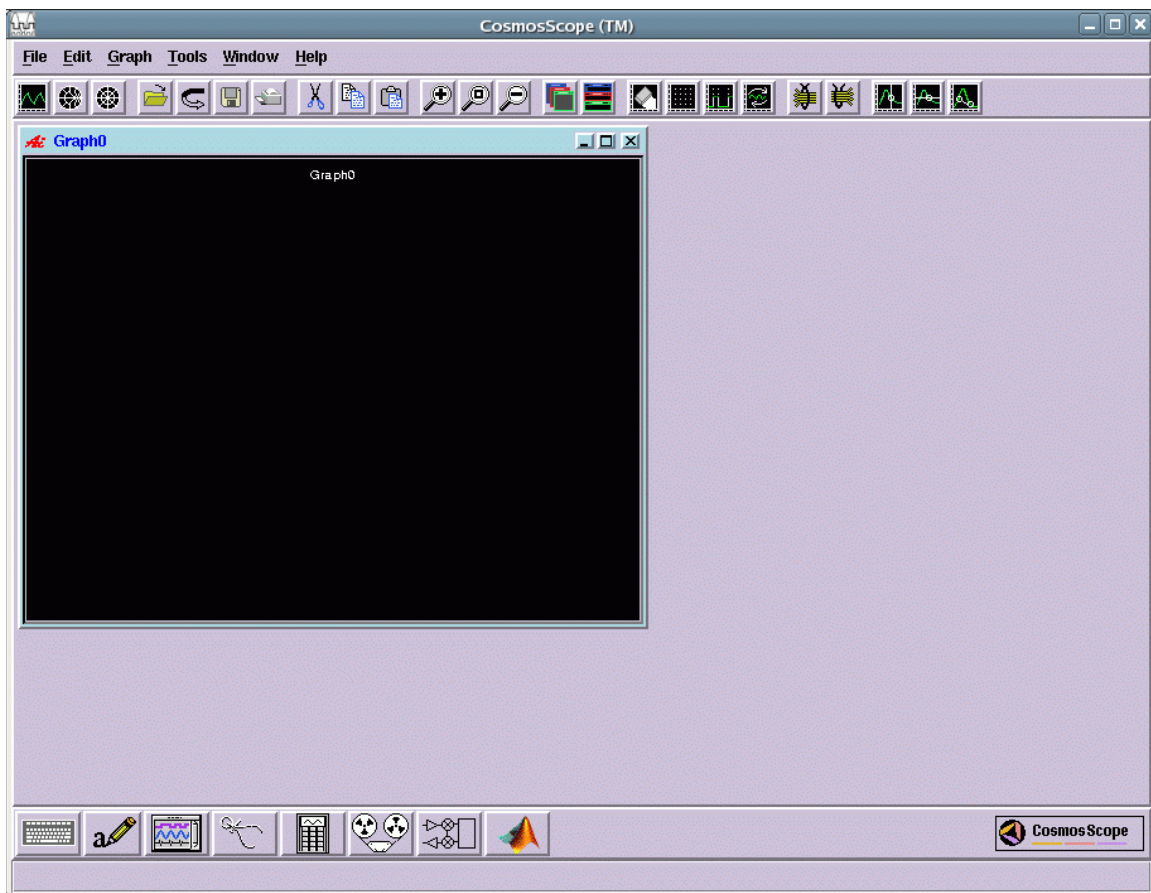
Several files are created by HSPICE:

inv.ic: Text file containing the circuit initial conditions

inv.st0: Text file containing a summary of the simulation

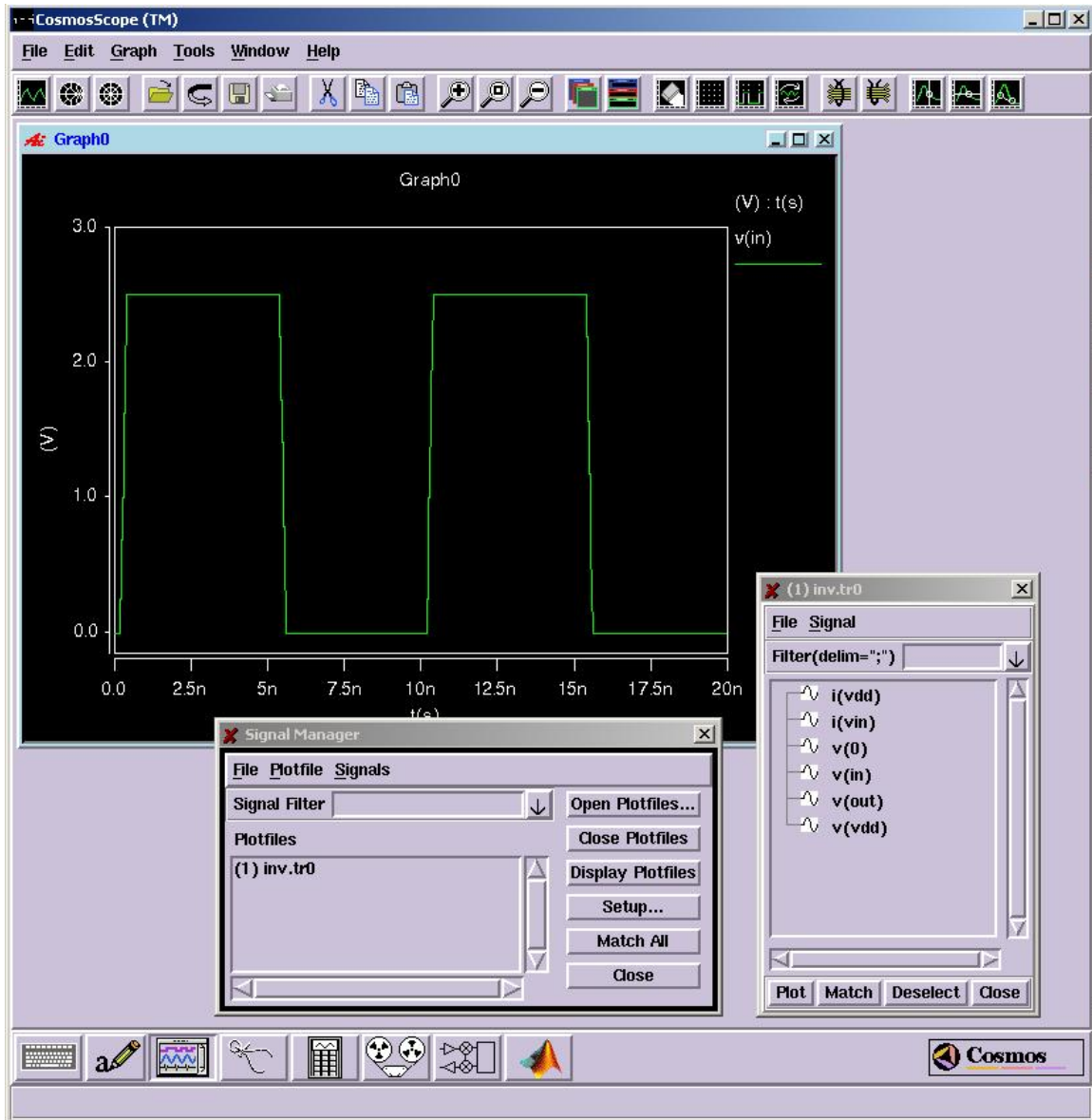
inv.tr0: Binary file containing transient analysis waveforms

After simulation, the waveforms can be viewed using CosmosScope. Type **scope** or **cscope** to load the results. The following window will appear.

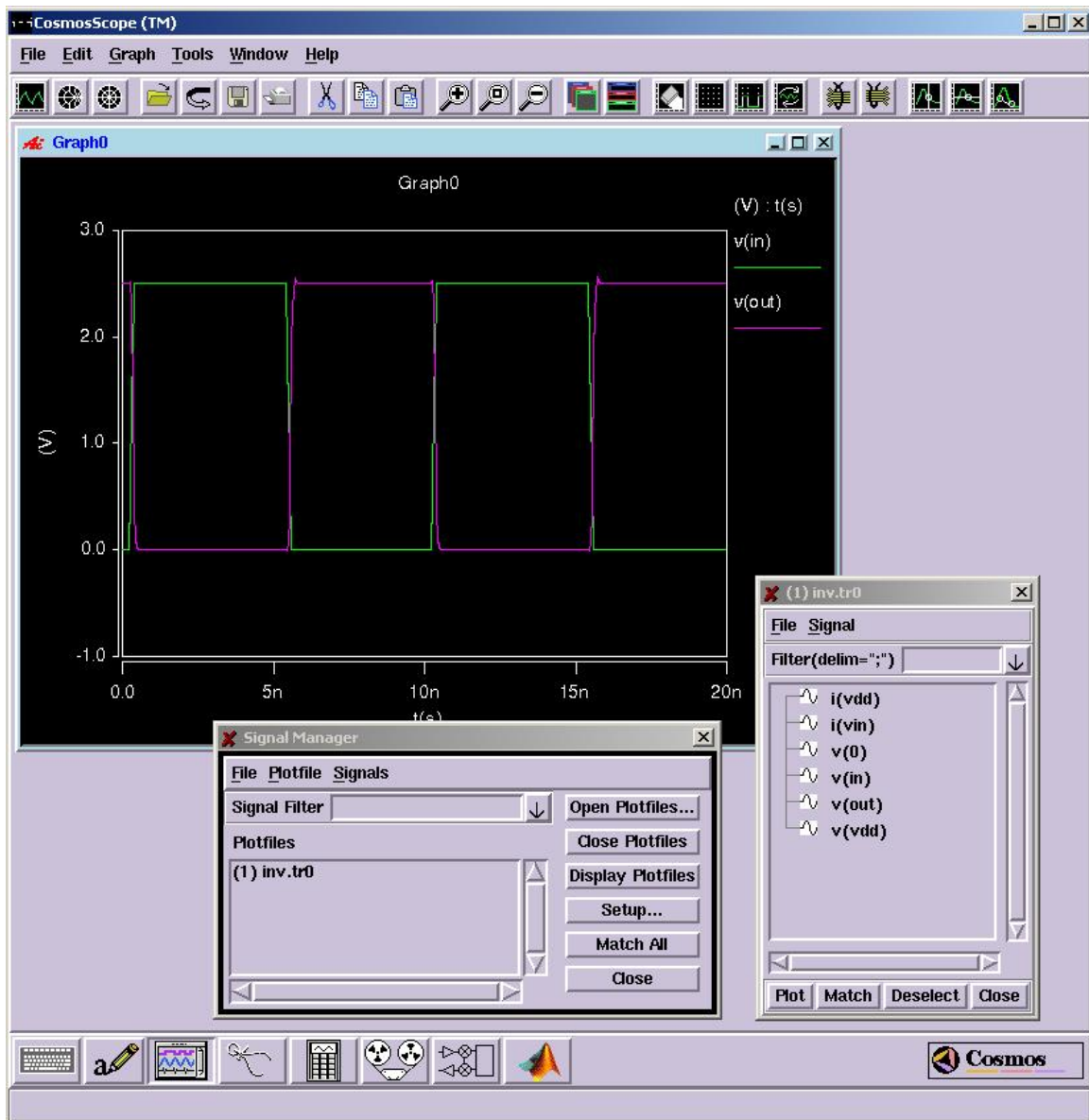


Click on **File > Open > Plotfiles**. Click on inv.tr0, the file that contains the transient analysis.

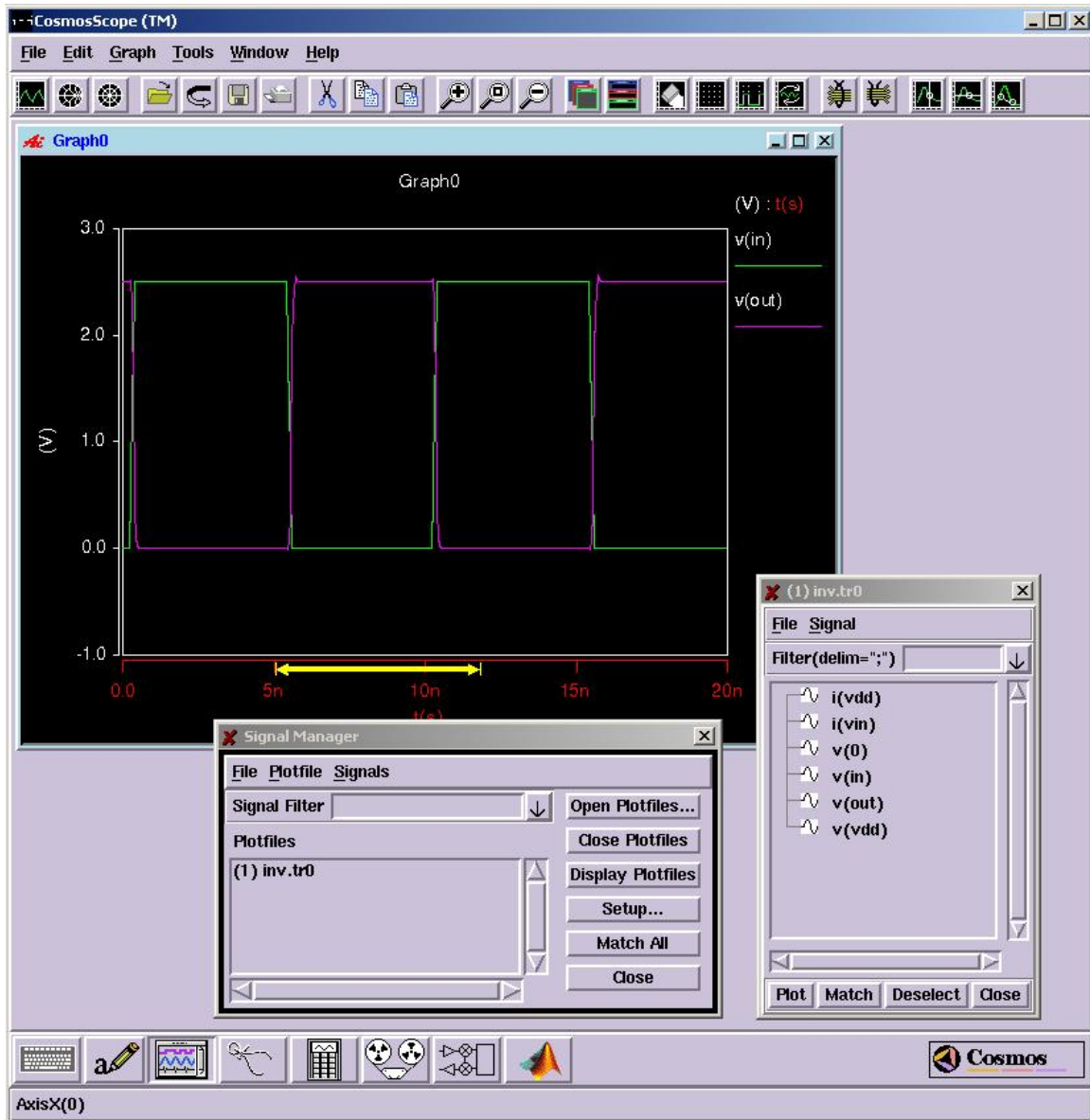
A signal manager window and the Plot File window will open up. In the Plot File window, plot v(in) by double clicking v(in), or by selecting v(in) and clicking **Plot**. The input waveform of the inverter will open up.



Now plot v(out), the same way you plotted v(in). This will open another graph with v(out).



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.