Full Custom Design Flow Tutorial Using Synopsys Cosmos Tool

By

Eli Lyons
Vish Ganti
Hamid Mahmoodi

San Francisco State University
School of Engineering

Nano-Electronics & Computing Research Center

Fall 2008

Table of Contents

Introduction..4
Part 1. Setting up your workspace...4
Figure 1. shell commands...5
Figure 2. CosmosSE window...6
Figure 3. Add Reference Library...6
Figure 4. Reference Library Manager..7
Part 2. Creating a schematic..9
Figure 5. Inverter schematic for simulation...9
Figure 6. Properties of Voltage pulse...10
Part 3. Schematic Simulation..11
Figure 7. Cosmos command window..11
Figure 8. DC analysis in Cosmos Guide...12
Figure 9. Adding a Library in Cosmos Guide...13
Figure 10.a Cosmos Scope window...14
Figure 10.b Graphs of Vin and Vout..15
Figure 10.c Vin and Vout graphed together..16
Figure 11. Transient Analysis in Cosmos Guide...17
Figure 12. Simulation of both DC and Transient analysis...18
Figure 12. Inverter schematic for use in higher level design.......................................19
Part 4. Creating the Layout...20
Part 5. Running DRC..21
Part 6. Running LVS..22
Figure .Execute Hercules window...23
Figure . Hercules Output...24
Figure . file browser window for LVS..25
Figure . Contents of 90nminv.LVS_ERRORS...26
Part 7. Extracting parasitics..28
Figure . Enterprise command window...28
Figure . Star-RCXT window..29
Figure . Database tab of Star SingleShot Tech Form..30
Figure . Extraction tab of Star SingleShot Tech Form..31
Figure . Processing tab of Star SingleShot Tech Form...32
Figure . Star-RCXT processing output window...33
Figure . Parasitic Back-Annotation window...34
Figure . Schematic with parasitics added...35
Part 8. Gate level design and simulation..36
Figure 13.a Generated symbol for inverter...36
Figure 13.b Modified inverter symbol..37
Figure 14. Library manager window..38
Figure 15. Ring oscillator schematic..39
Figure 16. Transient analysis for ring oscillator in Cosmos Guide...............................40
Figure 17.a Cosmos scope of ring oscillator...41

Figure 17.b Graph of oscillation......42
Figure 17.c Edit axis attributes window......43
Figure 17.d Adjusted graph of Oscillation......44
Part 9. Trouble Shooting......45
CosmosLE will not open in the CAD lab: 45
Reference library, tech file, or runset will not load:......45
Schematic or layout has glitches:......45
Window does not close when close window icon is clicked:......45

Introduction

In the first three parts of this manual we will design and simulate a CMOS inverter using CosmosSE in conjunction with Hspice and CosmosScope to visually assemble the circuit schematic, simulate it, and view the output waveforms. For further help you are encouraged to go to “Help” in the menu bar of CosmosSE.

Starting in Part 4 we will use CosmosLE to create a layout, and use Hercules to run a design rule check (DRC) on the layout based on the technology process. We will also use Hercules to make sure our inverter layout matches our schematic by running a Layout Versus Schematic (LVS) check. Finally, we will use the inverter we create in a gate level design of a ring oscillator.

Part 1. Setting up your workspace

The first step is to login. Please refer to the login tutorial if you are having trouble logging in or running the following commands. If you are using a linux machine off campus to login try using the following command to ensure you can use x-server:

```
ssh -l username -X hafez.sfsu.edu
```

To setup all the software we will use the following commands in the shell window. These commands must be run every time you use the Synopsys software :

```
csh
source /packages/synopsys/setup/hsp.csh
source /packages/synopsys/setup/cosmos.csh
source /packages/synopsys/setup/hercules.csh
source /packages/synopsys/setup/star_rcxt.csh
```

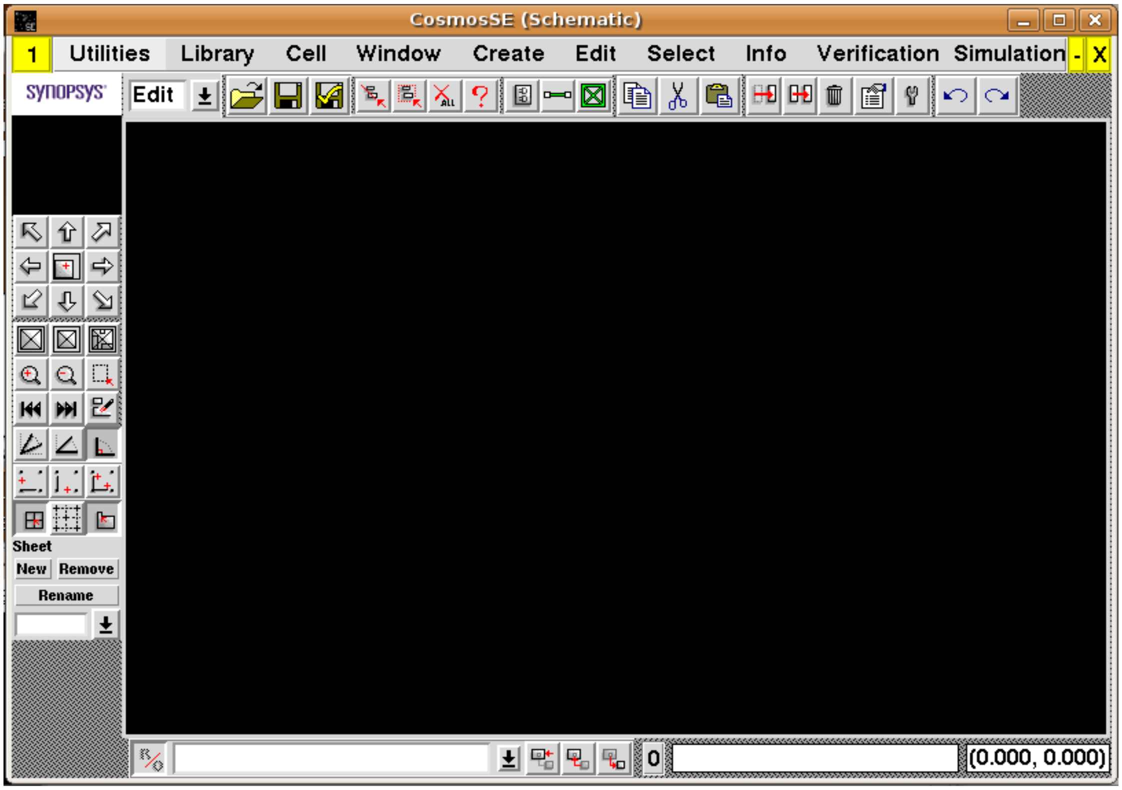
To run an instance of CosmosSE simply type “CosmosSE”. Your command window should look like the one shown in figure 1. Most likely you will also have the warnings shown, don't worry about them as long as CosmosSE opens.

Figure 1. shell commands

```
File  Edit  View  Terminal  Tabs  Help
eli@yadadai:~$ ssh -l engr848-01 -X hafez.sfsu.edu
engr848-01@hafez.sfsu.edu's password:
Last login: Tue Nov 11 10:49:46 2008 from adsl-75-35-113-90.dsl.pltn13.sbcglobal
.net
-bash-3.1$ csh
[engr848-01@hafez ~]$ source /packages/synopsys/setup/hsp.csh
Setting up environment variables for HSPICE version 2004.03
[engr848-01@hafez ~]$ source /packages/synopsys/setup/cosmos.csh
Setting up environment variables for Cosmos LE version W-2004.09
Setting up environment variables for CosmosScope version B-2008.09
[engr848-01@hafez ~]$ source /packages/synopsys/setup/hercules.csh
Setting up environment variables for Hercules version B-2008.09
[engr848-01@hafez ~]$ source /packages/synopsys/setup/star_rcxt.csh
Setting up environment variables for Star RCXT version B-2008.06-SP2-1
[engr848-01@hafez ~]$ CosmosLE
* Warning: You do not seem to be running a RedHat based linux distribution
     NOTE: Synopsys recommends running on a RedHat 7.2 or greater
           based distribution.
* Warning: Please feel free to use your current installation of linux
           for Cosmos tools.  However if you encounter OS specific
           issues, Synopsys may have difficulty resolving them.
```

CosmosSE should open in a new window shown in figure 2.

Figure 2. CosmosSE window



In the menu bar go to Libary>Create. In the "Library Name" field enter "mylibrary" and in the "Tech File Name" field copy and paste this file directory in the field:

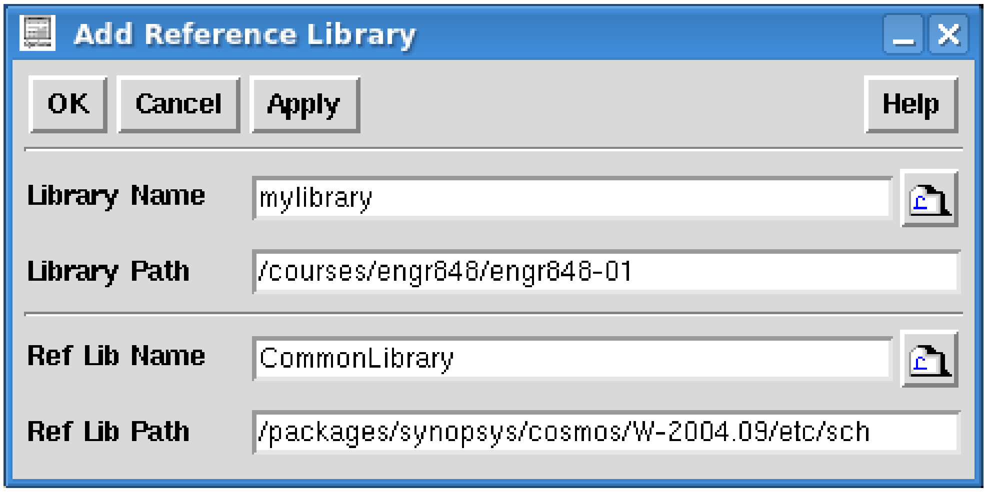
/
packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit/tech
file/saed90nm_1p9m.tf

Note that you could choose the technology file by clicking the open folder icon and browsing for it as well.

Click OK.

In the menu bar go to Library>Add

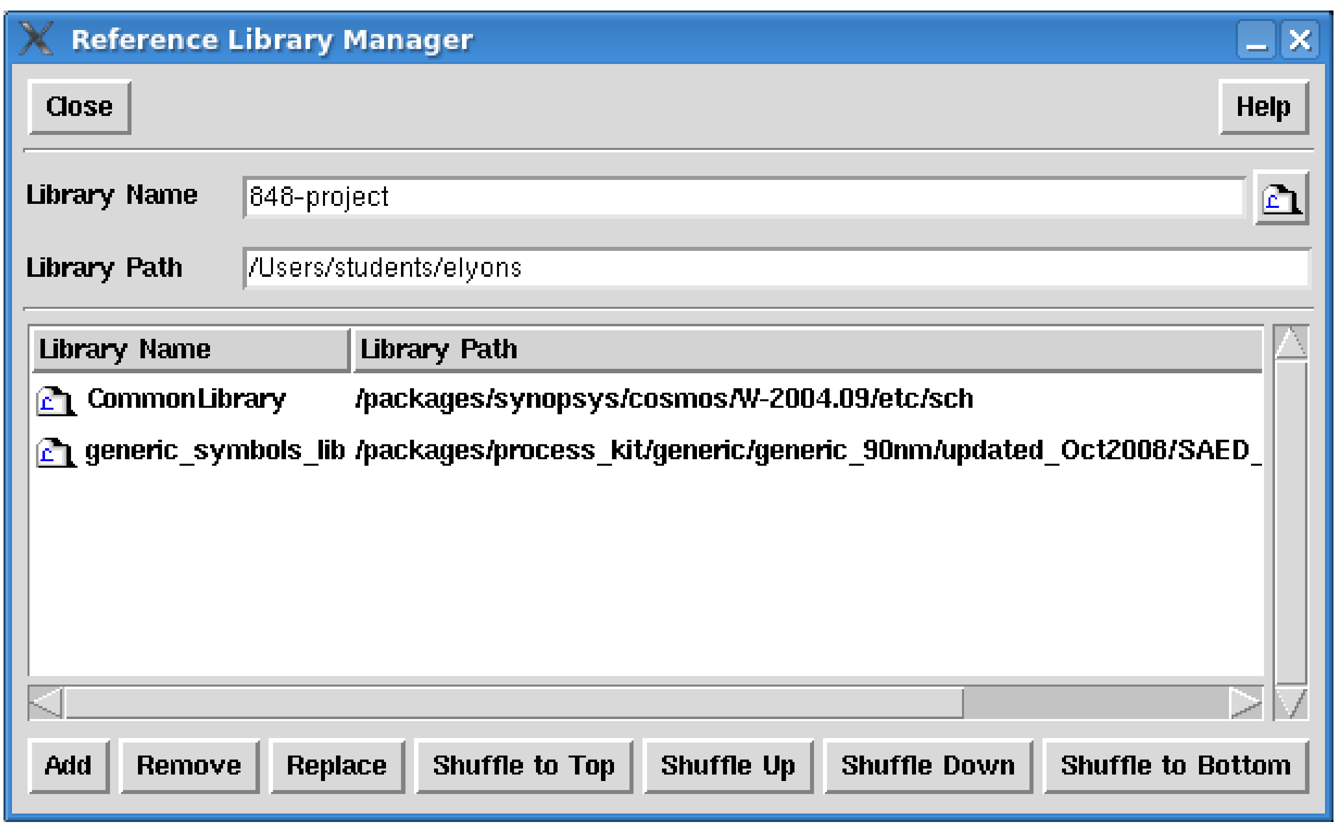
Figure 3. Add Reference Library



Again, you may click the folder icon on the right of the “Ref Lib Name” and select CommonLibrary in the directory /packages/synopsys/cosmos/W-2004.09/etc/sch or you can fill in the fields manually. When you are done click OK.

In the menu bar go to Library>Manager. Click “Add” and select generic_symbols_lib in the directory /packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit

Figure 4. Reference Library Manager



In the menu bar go to Cell>Create. Enter a name in the "Cell Name" field and click OK.

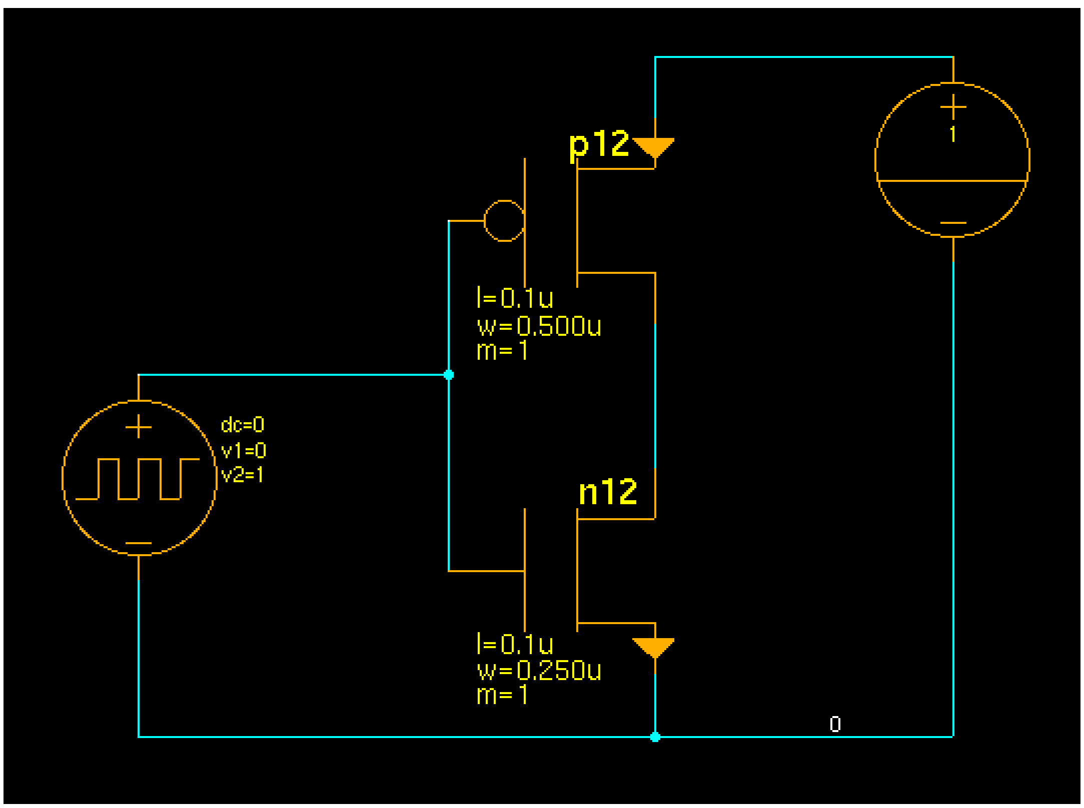
Part 2. Creating a schematic

There are three ways to place components. Press <shift-i>, click the Create Instance icon in the edit toolbar, or in select Create>Instance in the menu bar. Select the cell instance part you want to place by clicking it once and then clicking inside the schematic.

Use g_nmos3t, g_pmos3t, c_vpulse and c-vgeneric for the inverter parts. To create wire move the mouse over a part terminal, hold down the right click button and select “Create Wire”. When you are finished your inverter should look like the schematic below. To edit the properties of your parts move the mouse over the part, right click and select “properties”. Change the properties to the values in the schematic below. In our case the process is a 0.90um process, so 0.1um is the minimum gate length. You can change the names of the nets by right clicking on the nets and selecting “attributes”. Change the names of the nets to VDD, VIN,VOUT and '0' (zero).

IMPORTANT: YOU MUST specify a ground in your schematic, which is done by changing the attribute of your ground net to “0”. This can be seen in the model below.

Figure 5. Inverter schematic for simulation



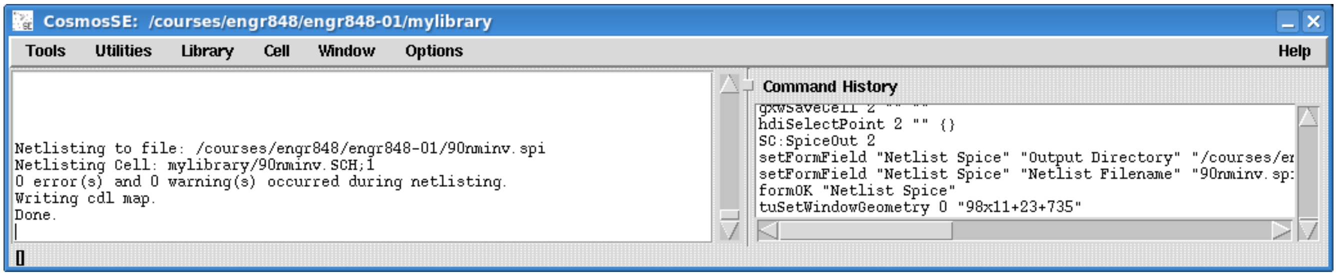
Change the properties of the voltage pulse input to the values shown in figure6

Figure 6. Properties of Voltage pulse

Part 3. Schematic Simulation

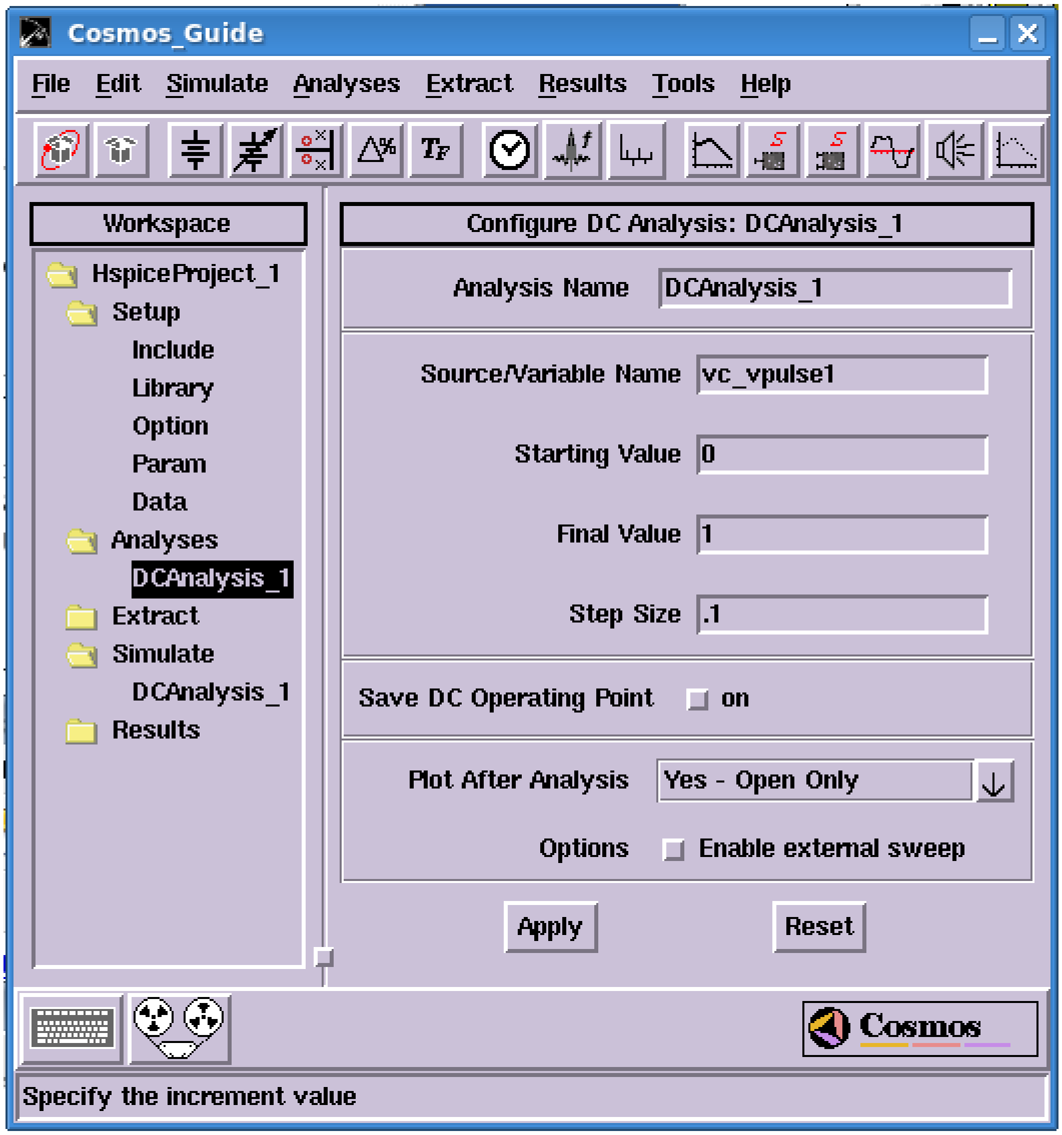
Before simulation you need to build your netlist by going to Simulation>Spice in the menu bar. The necessary fields should already be filled out. Click OK. You can make sure there were no errors by looking in the Cosmos command window.

Figure 7. Cosmos command window



Next go to Simulation>DC Sweep Analysis. Cosmos Guide should open, if it does not run it is probably due to a poor internet connection. Enter the fields with the values shown below, notice that the source we are sweeping is vc_vpulse1. If you look at the attributes of input pulse in the schematic, the name is c_vpulse1, however in the netlist this is a voltage and we must make that clear by adding the "v" in front of the name in Cosmos Guide. Also take note that we have chosen to "Plot After Analysis". Once all the fields have been filled out click "Apply".

Figure 8. DC analysis in Cosmos Guide

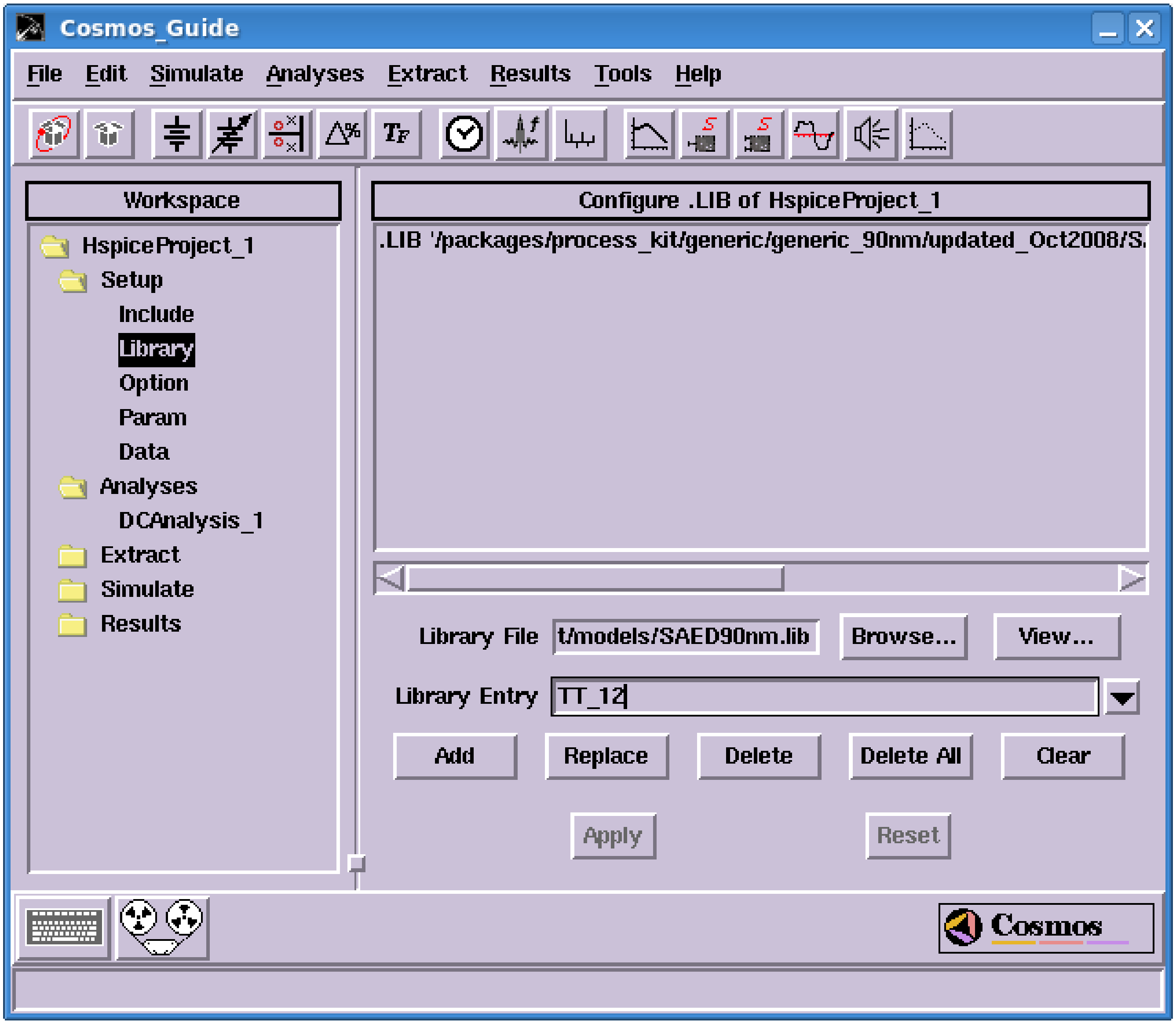


Open "Setup" in "Workspace" on the left of Cosmos guide. Click "Browse" for a library file and select the "SAED90nm.lib" in the following directory: /packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit/models/

In the Cosmos Guide you should see a drop down arrow next to "Library Entry" that allows you to select which model you want to use for the simulation. Choose "TT_12", which simulates the model of a typical NMOS and typical PMOS. Click "Apply". If the drop down arrow does not render properly in the GUI click inside the text field and use the keyboard arrows to select “TT_12”.

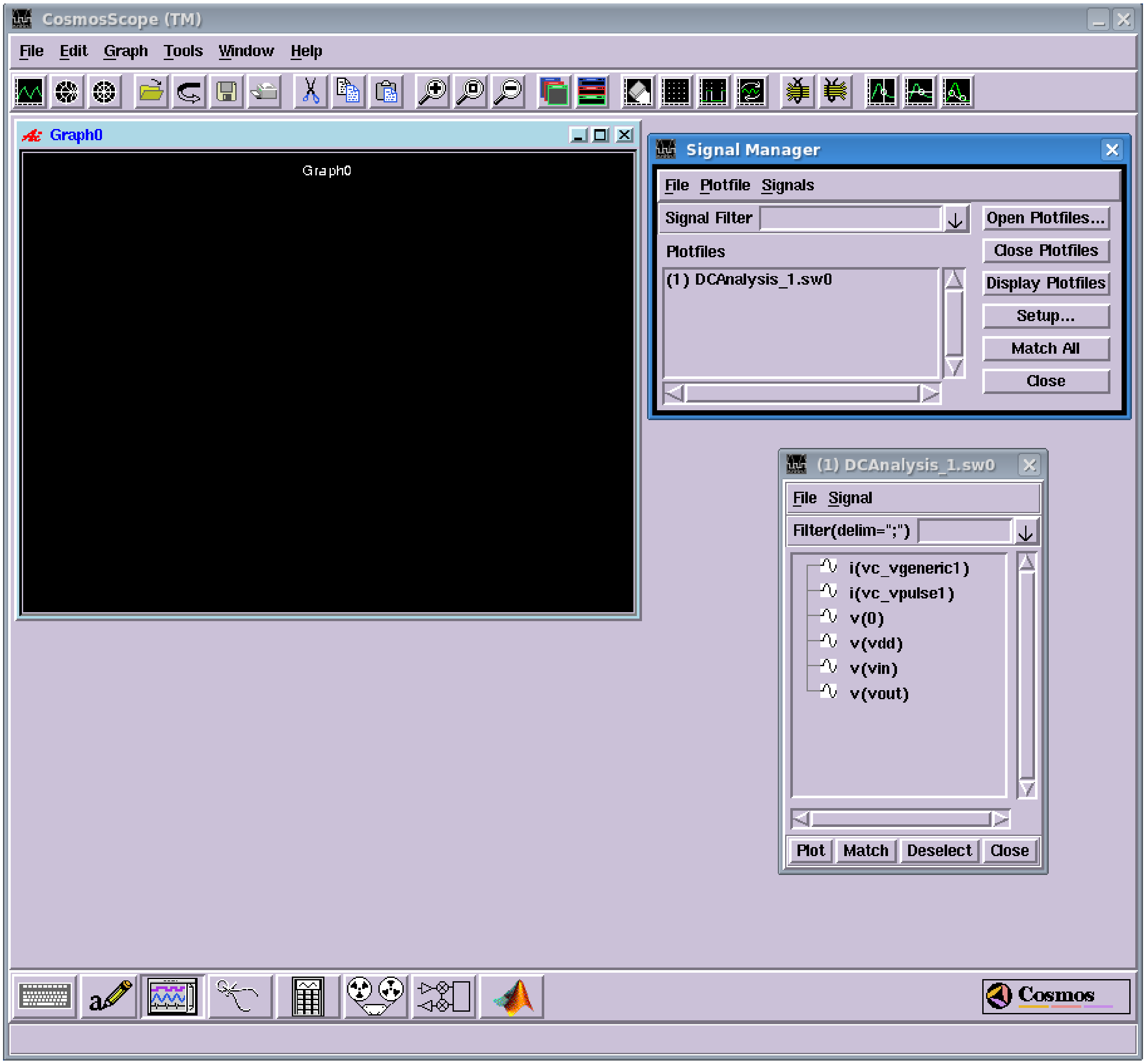
You probably noticed there are also entries for FF, SF...etc. These are models of fast NMOS and PMOS or one slow and one fast..etc. This library provides you with entries to simulate all combinations of the corners of the CMOS technology.

Figure 9. Adding a Library in Cosmos Guide



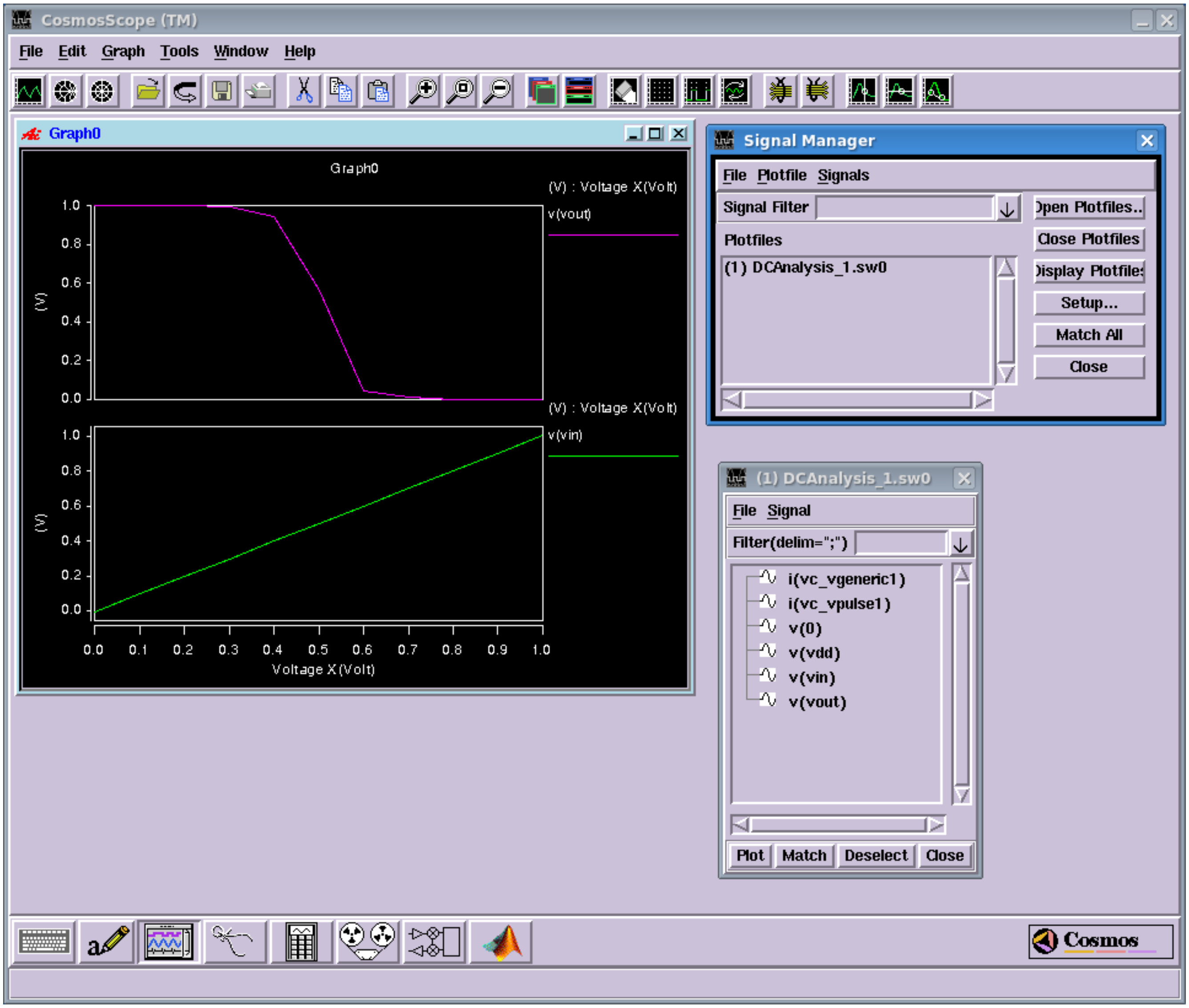
Select Simulate>Run All Projects>Serial
The simulation should run and CosmosScope should open automatically.

Figure 10.a Cosmos Scope window



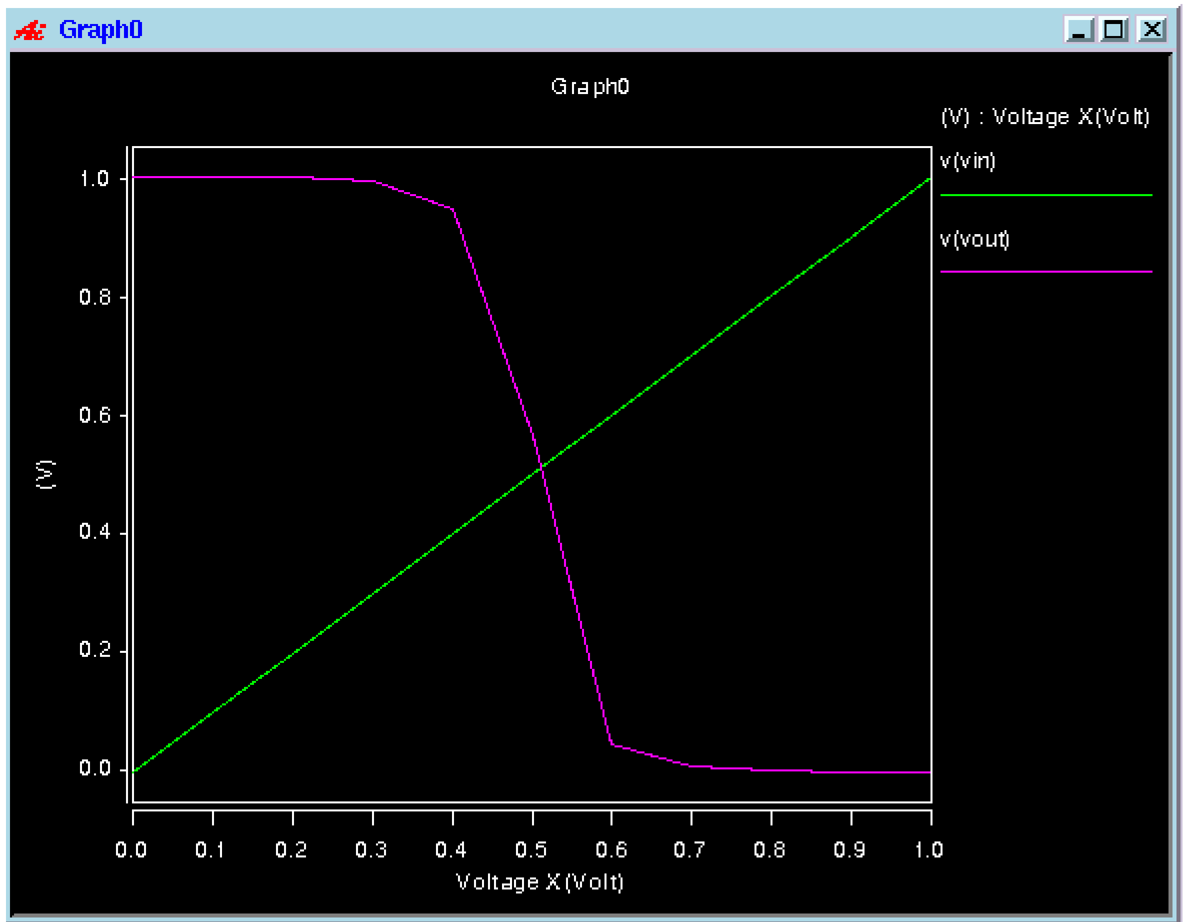
A signal manager window and the Plot File window will open up. In the Plot File window, plot v(in) by double clicking v(in), or by selecting v(in) and clicking Plot. The input waveform of the inverter will open up. Now plot v(out), the same way you plotted v(in). This will open another graph with v(out).

Figure 10.b Graphs of Vin and Vout



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.

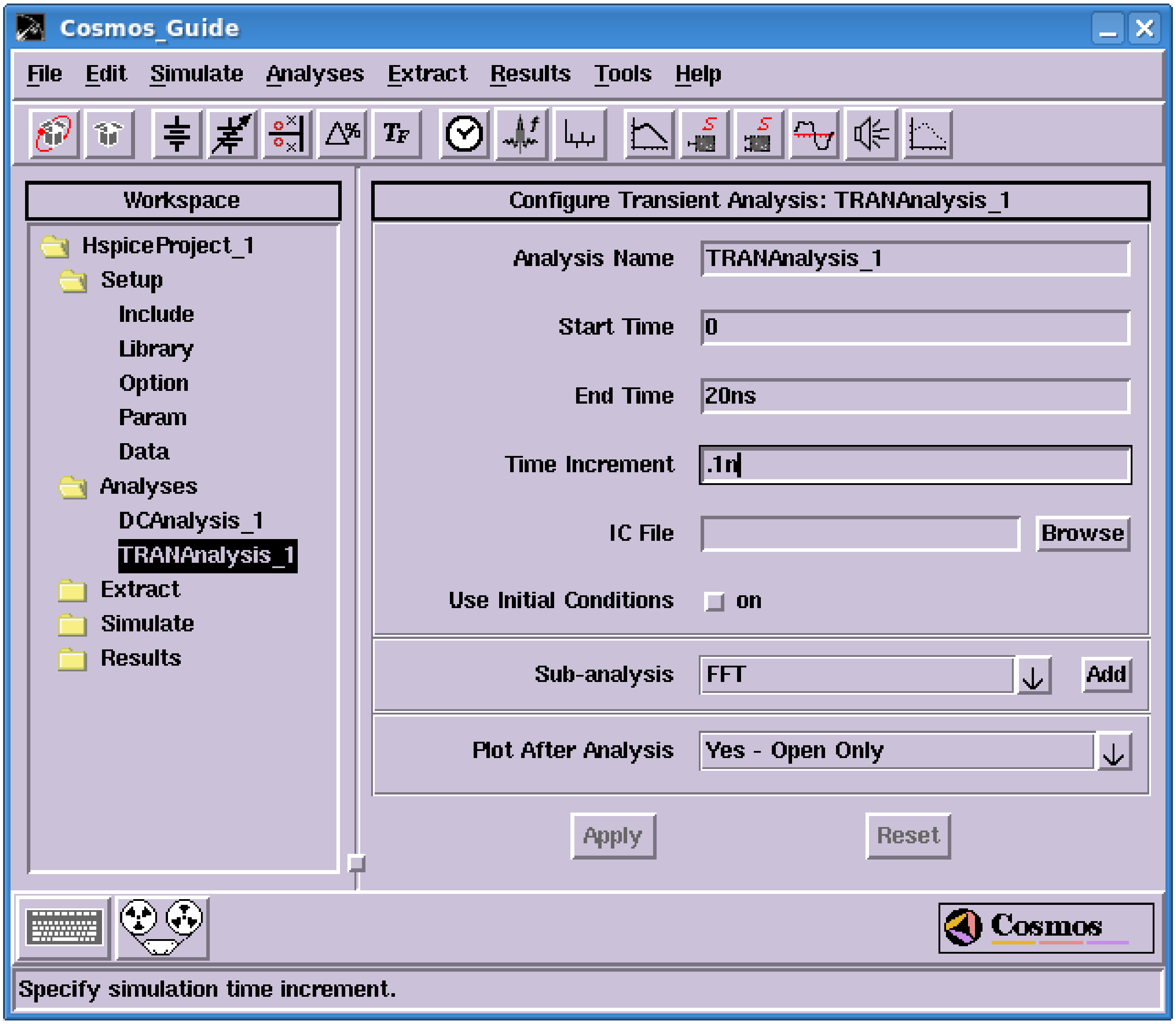
Figure 10.c Vin and Vout graphed together



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.

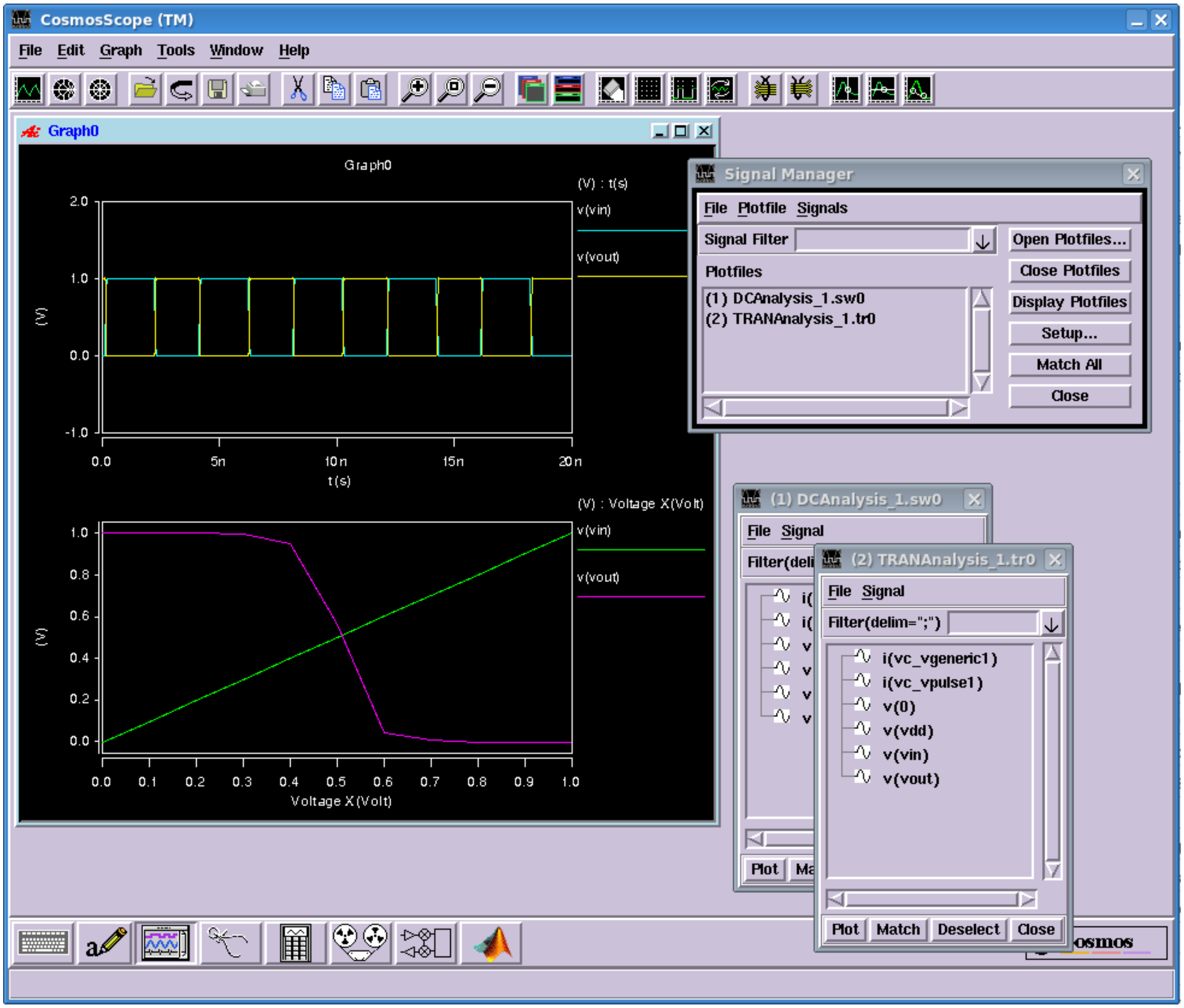
To simulate a transient response go to the Cosmos Guide window and select Analyses>Transient Analysis>Transient Analysis or click the transient analysis icon which looks like a clock.

Figure 11. Transient Analysis in Cosmos Guide



Select Analyses>Simulate>Run All Projects>Serial
Following the same process for the voltage sweep graph, now we can see both the Transient plot and Voltage sweep plot.

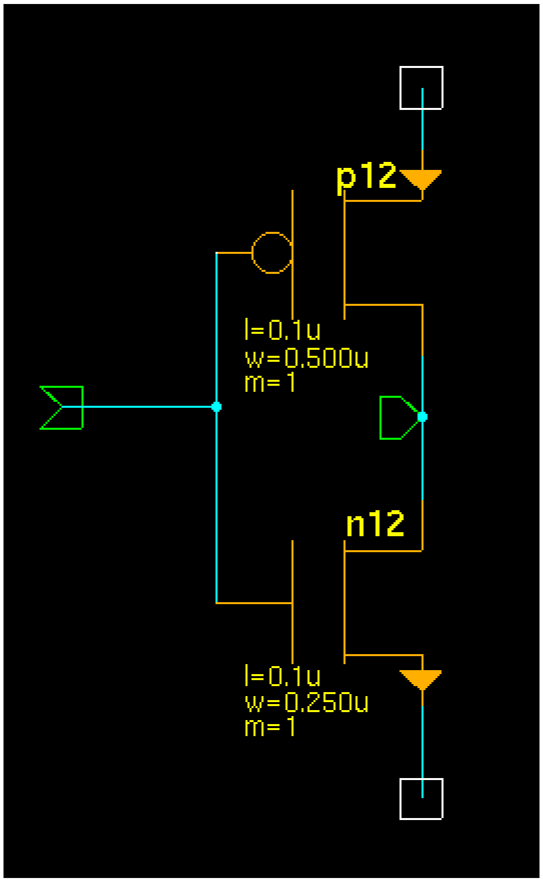
Figure 12. Simulation of both DC and Transient analysis



Now that we have simulated the inverter, we can can see that it is working properly. Now close Cosmose Guide and go back to the CosmosSE schematic window. Now we are going to make an actual physical layout for this inverter but before we do that we must modify the schematic. First delete the power sources since these will not appear on the layout we are going to make. Change the net named '0' to VSS. Now create a port by going to Create>Connector or click on the icon of a green box with a square in it. Place a port on the VDD net and and another on the VSS net. Now change the “Port Type” to Input and place it on the VIN net. Likewise choose “Port Type” to Output and place it on the VOUT net. Now your schematic should look similar to the one below.

Update the netlist by going to Simulation>Spice. Take note of the directory the spice netlist is being saved as it will be used later.

Figure 12. Inverter schematic for use in higher level design



Part 4. Creating the Layout

Part 5. Running DRC

Part 6. Running LVS

Before you do the LVS verification it is recommended you create a folder inside your student folder to hold the resulting files that Hercules creates when it runs LVS. In this tutorial the folder created was 90nminvLVS.

In either the schematic or layout window go to Verification>User LVS Runset.

Select the file “rules.lvs.9m_saed90.ev” as the Runset File in the following directory:

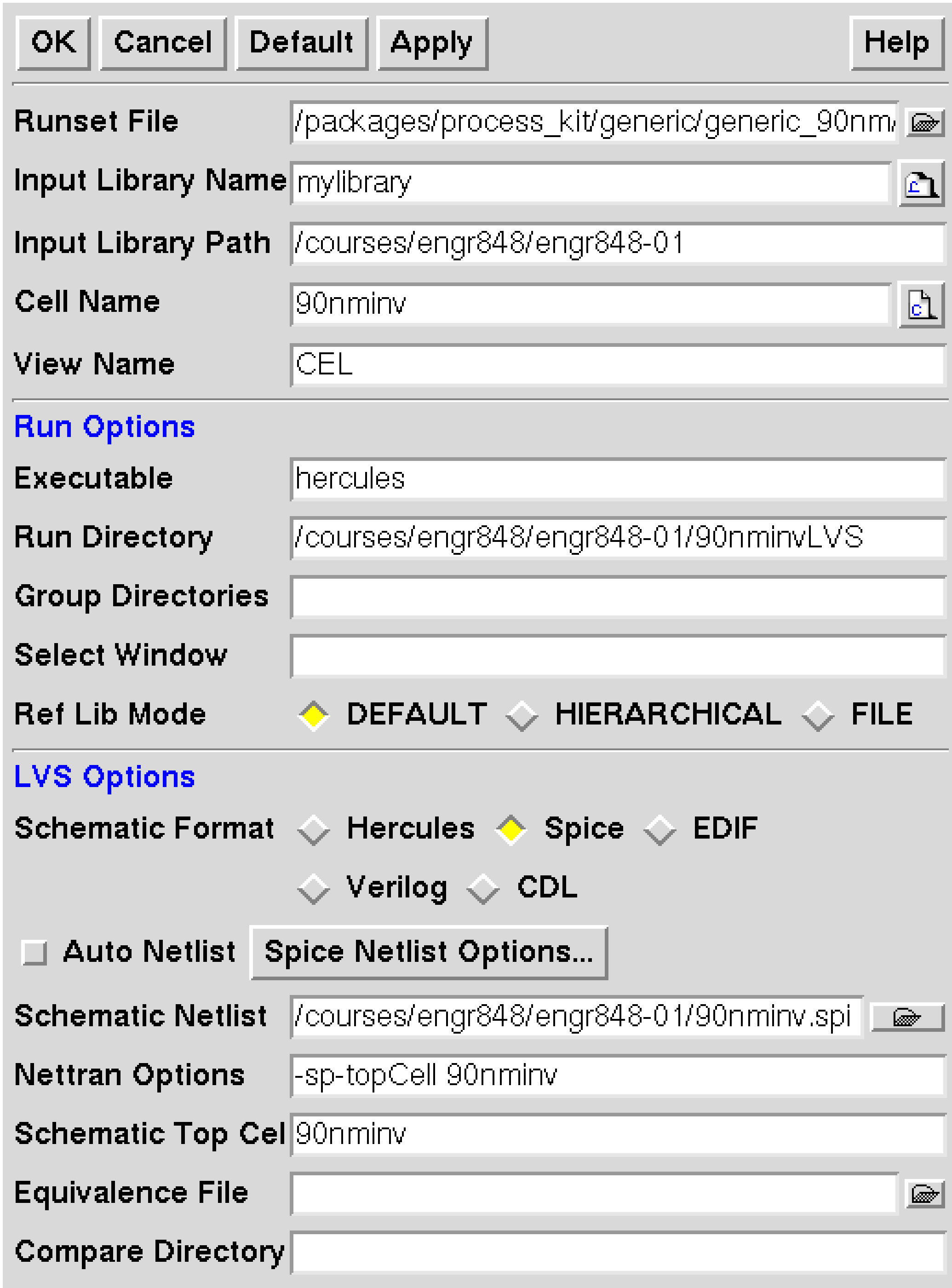
/packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit/rules/hercules/lvs/

Under Run Opttions enter the directory you want Hercules to output to. In our case it is /courses/engr848/engr848-01/90nminvLVS

Under LVS Options select Spice and select the spice netlist we created from the schematic earlier. The default directory is your student folder.

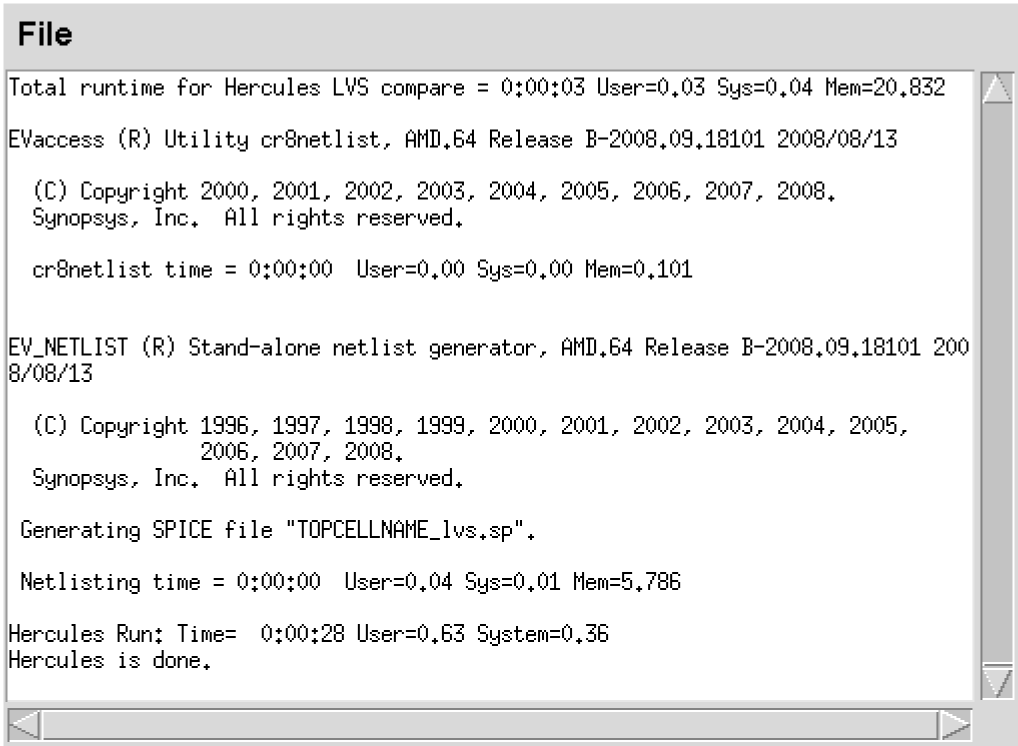
Make sure the fields in your Execute Hercules window mirror the ones in figure and click OK.

Figure .Execute Hercules window



A new window called Hercules Output should open and show what Hercules is doing. It should scroll through many processes and will say "Hercules is done." when it completes.

Figure . Hercules Output



Now open the directory that you specified as the Hercules Run directory, in our case /courses/engr848/engr848-01/90nminvLVS. There should be many new files created by Hercules there now. Open 90nminv.LVS_ERRORS

Figure . file browser window for LVS

If you have done everything correctly You should see a "PASS" in the 90nminv.LVS_ERRORS file. If it says "FAIL" read the errors it reports and try to fix them on the schematic or layout. If the error is in the schematic make sure to rebuild the spice netlist. Run Hercules again and see if 90nminv.LVS_ERRORS now says "PASS".

Figure . Contents of 90nminv.LVS_ERRORS

```
+---------------------------------------------------------------------+
|                    Hercules LVS Comparison Report                   |
+---------------------------------------------------------------------+

COMPARE (R) Hierarchical Layout Vs. Schematic
          AMD.64 Release B-2008.09.18104 2008/08/21
Copyright (C) Synopsys, Inc. All rights reserved.


-----------------------------------------------------------------------
LVS error file     = 90nminv.LVS_ERRORS
Layout error file  = 90nminv.LAYOUT_ERRORS
Schematic netlist  = 90nminv.sch_out
Layout netlist     = 90nminv.net
Equivalence file   = [automatic]
Runset file        = rules.lvs.9m_saed90.Ent.ev
Working directory  = /courses/engr848/engr848-01
Compare directory  = ./TOPCELLNAME.run_details/compare
Compare start time = 2008-11-11 15:50:56



-----------------------------------------------------------------------
Top block compare result: PASS

                    #####     ##      #####   #####
                    #    #   #  #     #       #
                    #####  ######   #####   #####
                    #      #      #       #       #
                    #      #      #  #####   #####

                [90NMINV == 90NMINV]

-----------------------------------------------------------------------
Comparison summary

   1 successful equivalencies
   0 failed equivalencies

Schematic and layout agree at all equivalent points.
```

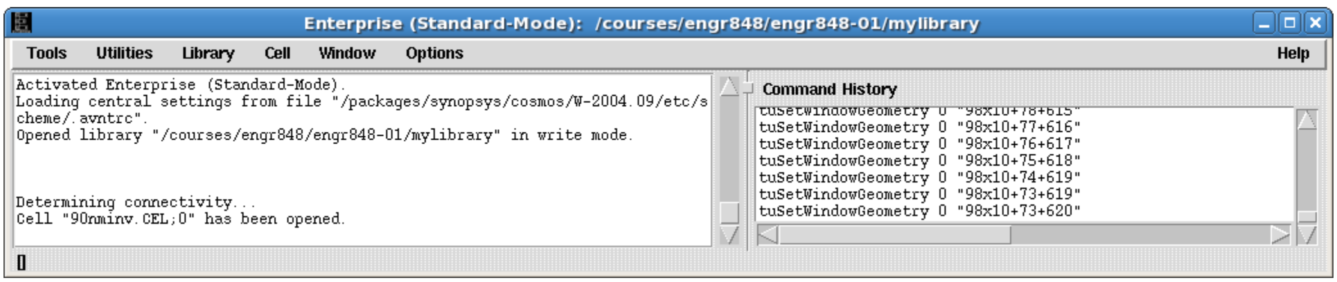
End of LVS comparison report

Part 7. Extracting parasitics

Create a new folder in your account folder called “90nminvSTAR”. This is where we will save the output files created by Star-RCXT.

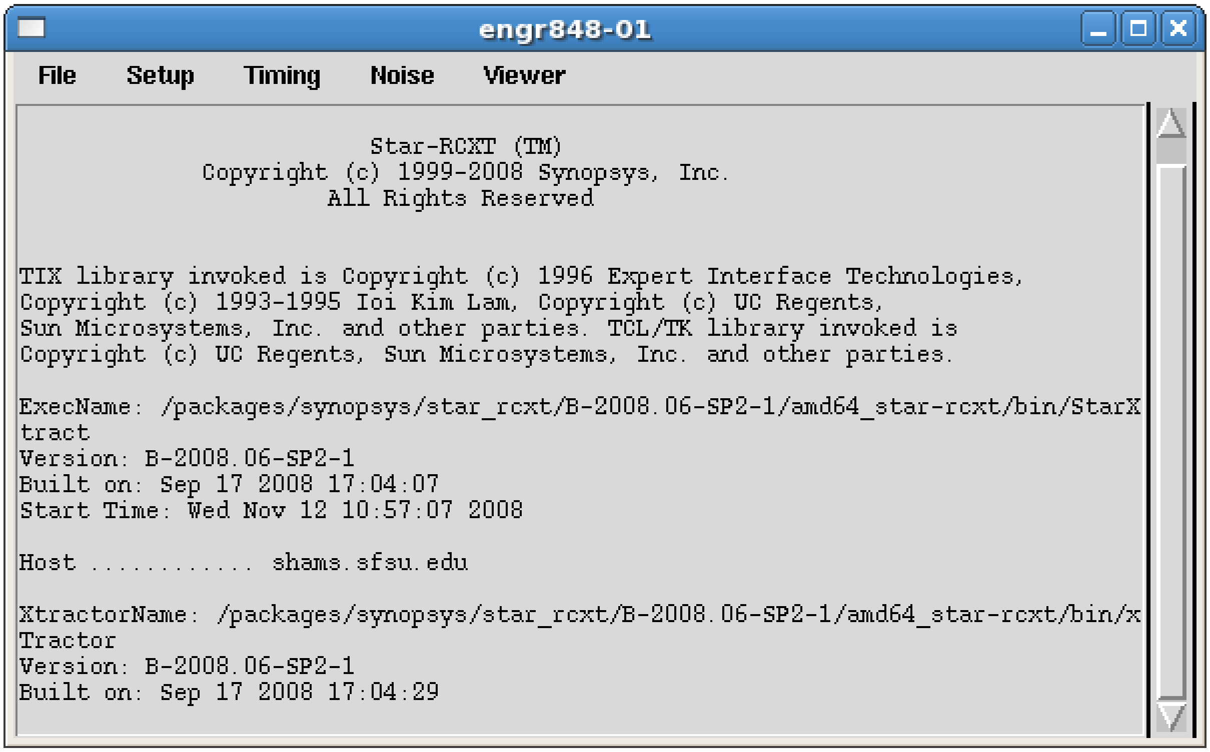
Locate the Cosmose Enterprise Command window which should be located at the bottom of the screen.

Figure . Enterprise command window



Now go to Utilities>StarRC Xtract and figure should pop up.

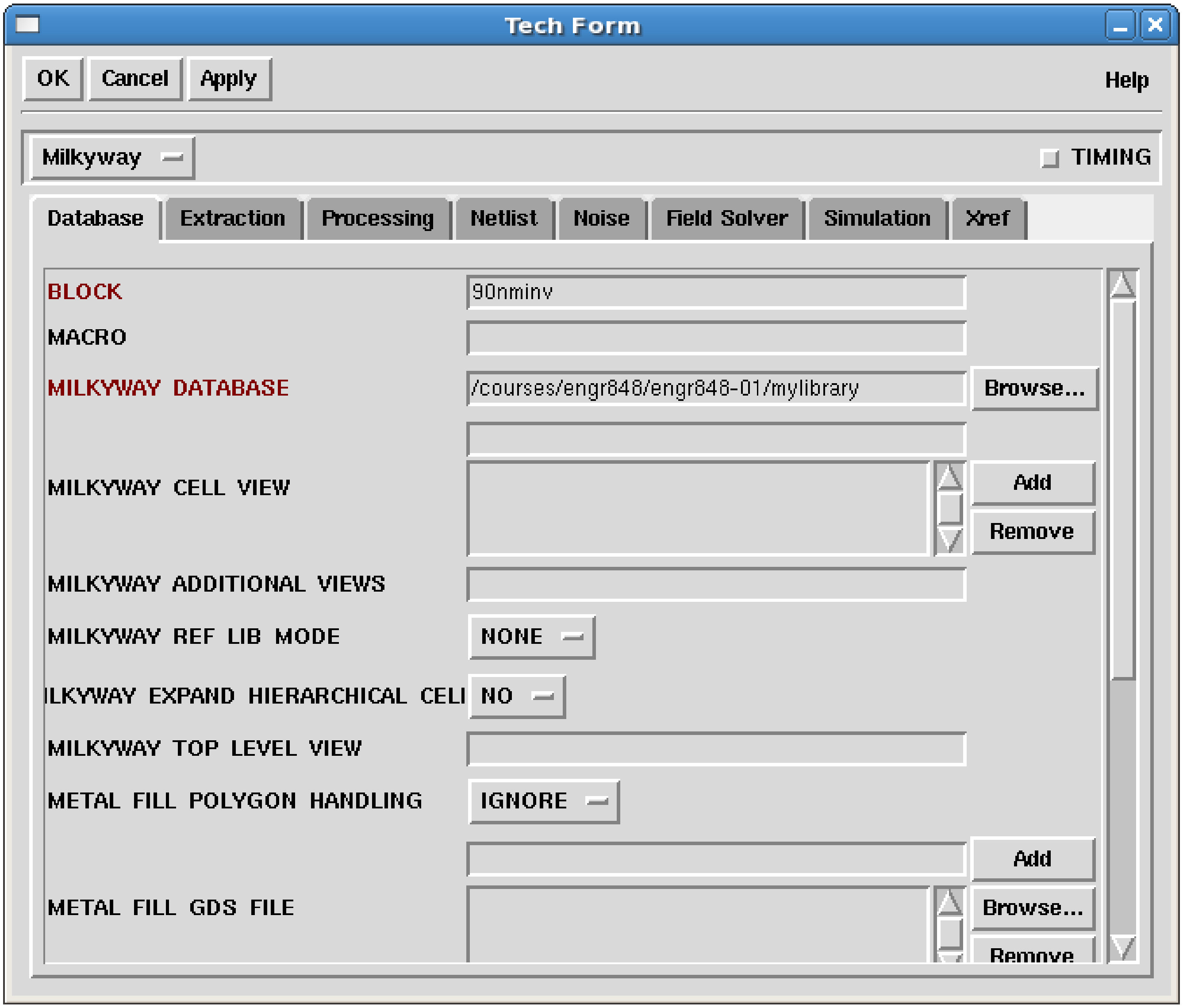
Figure . Star-RCXT window



Go to Setup>SingleShot

Fill out the BLOCK field with the name of the layout cell, in our case “90nminv”. In the field for MILKYWAY DATABASE, browse for and select the library our cell is in “mylibrary”.

Figure . Database tab of Star SingleShot Tech Form



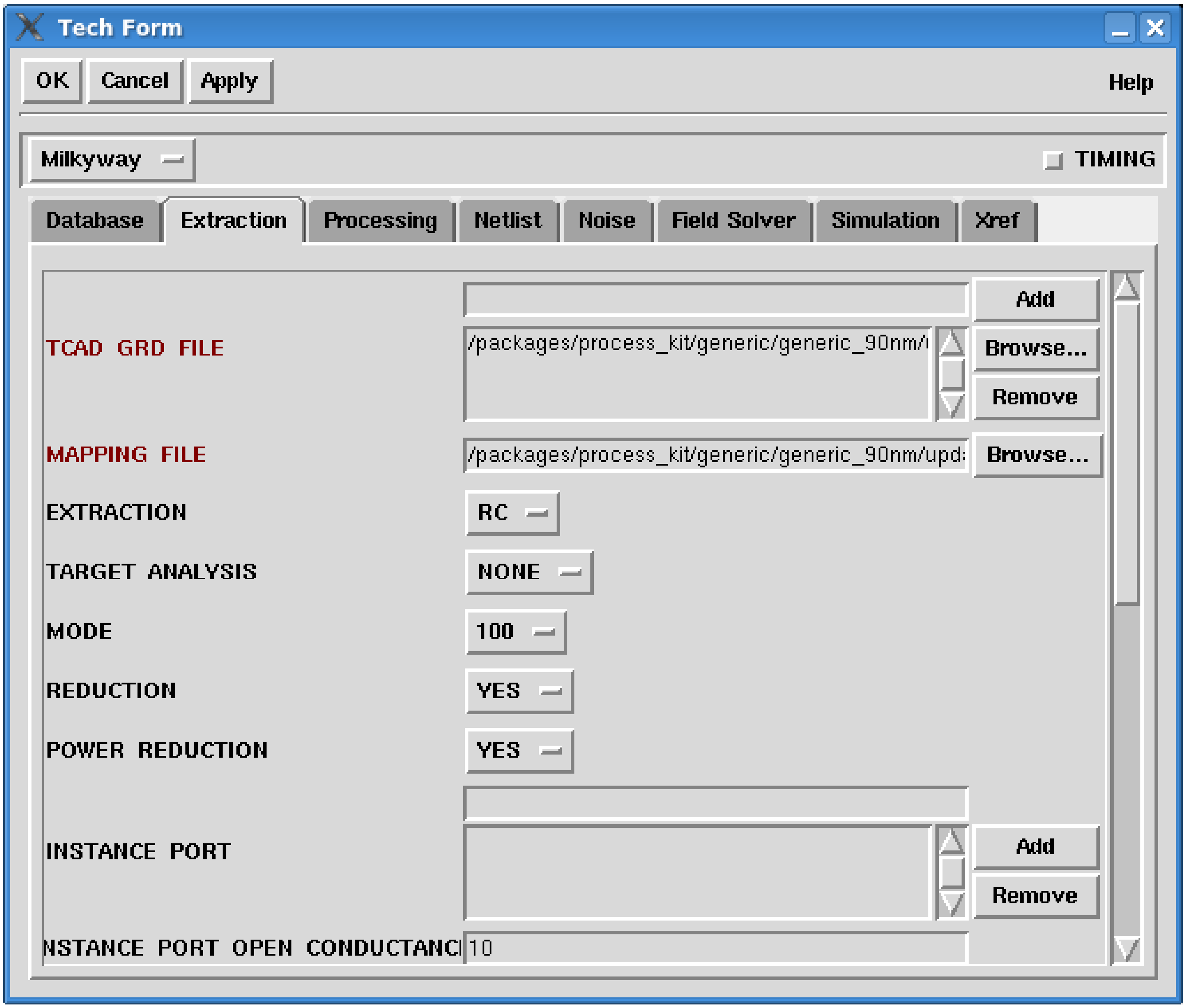
When you are done filling out the Database tab, click on the Extraction tab. For the TCAD GRD FILE, select "saed90nm_9lm.nxtgrd" in the directory:

/
packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit/rule
s/starrcxt

For the MAPPING FILE, select "saed90nm.map" in the directory:
/packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_

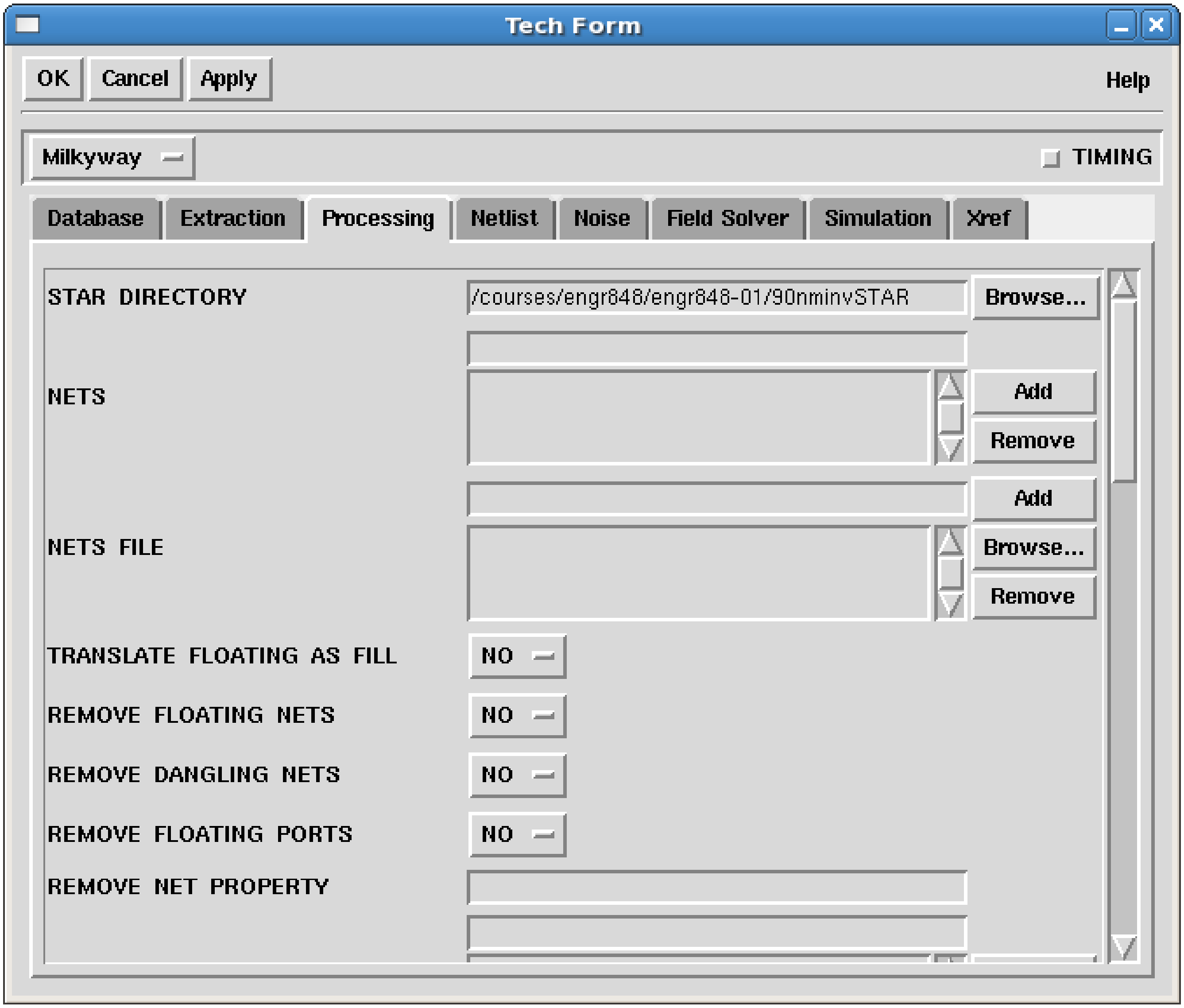
Kit/rules/starrcxt

Figure . Extraction tab of Star SingleShot Tech Form



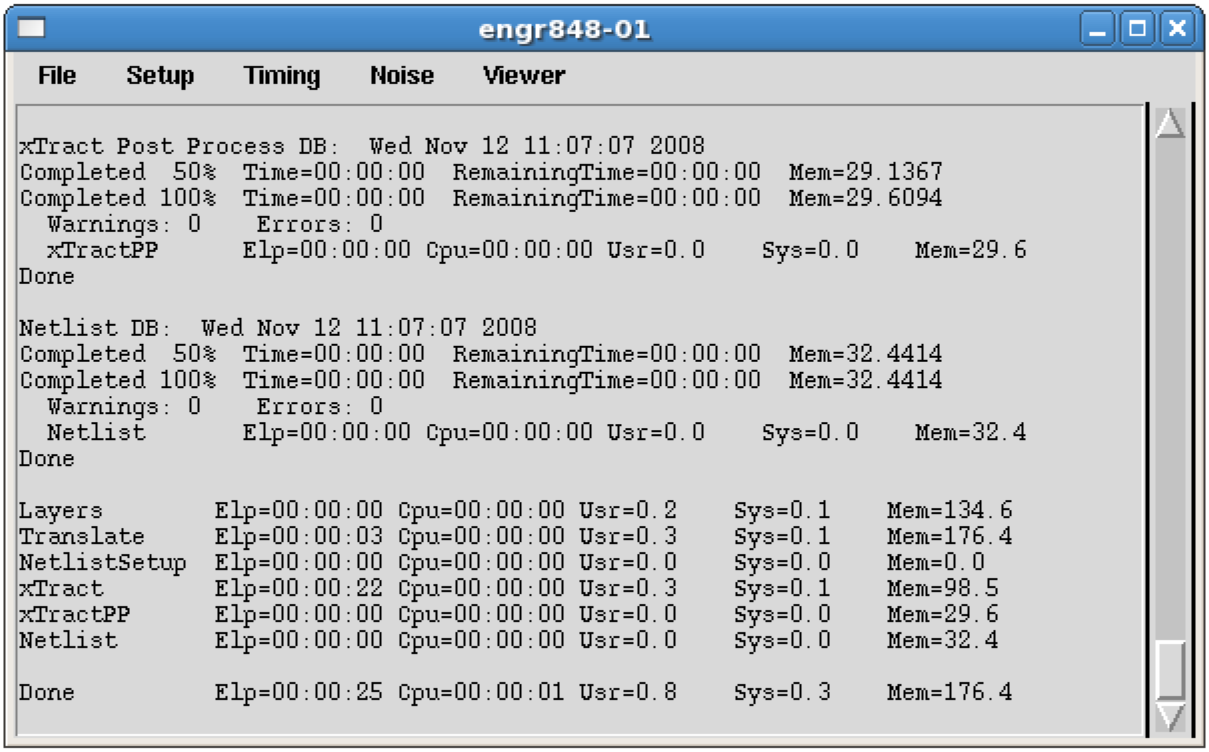
Open the Processing tab and select the folder “90nminvSTAR” we created earlier for our STAR DIRECTORY.

Figure . Processing tab of Star SingleShot Tech Form



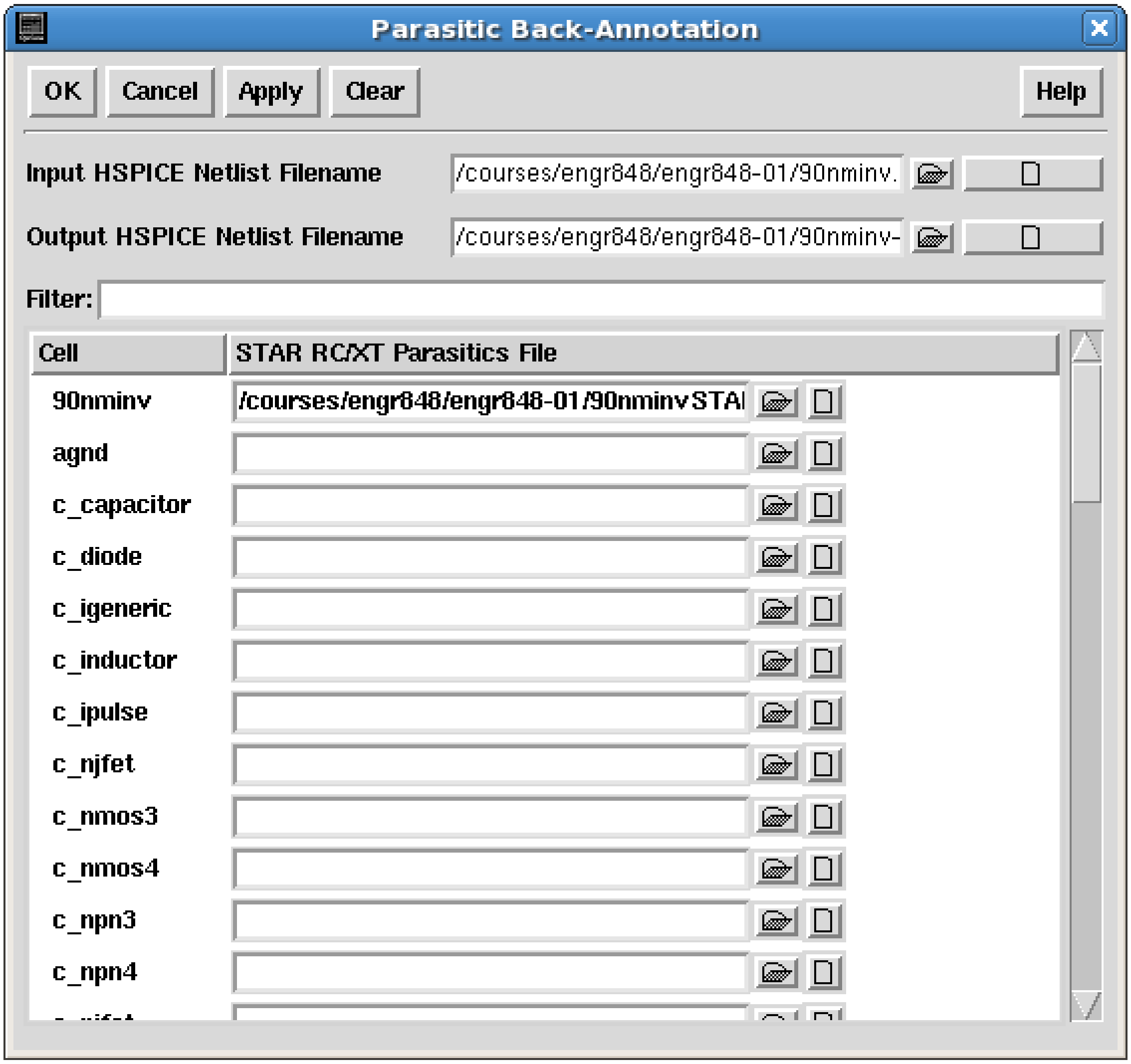
Once you have clicked OK on the SingleShot Tech Form, the Star-RCXT window should start showing the processes for the extraction. Star may pause between processes, so wait until it is completely done like shown in figure.

Figure . Star-RCXT processing output window



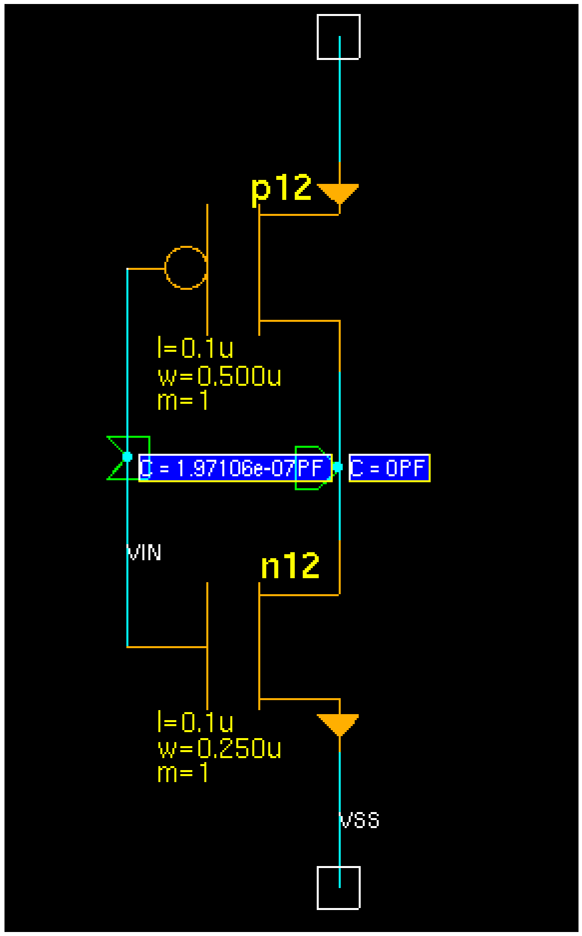
Go to Simulation>Parasitics and in the 90nminv field select the file “90nminv.spf“ in the 90nminvSTAR folder you created in your account folder.

Figure . Parasitic Back-Annotation window



Now you should see that a capacitance has been added to your inverter schematic.

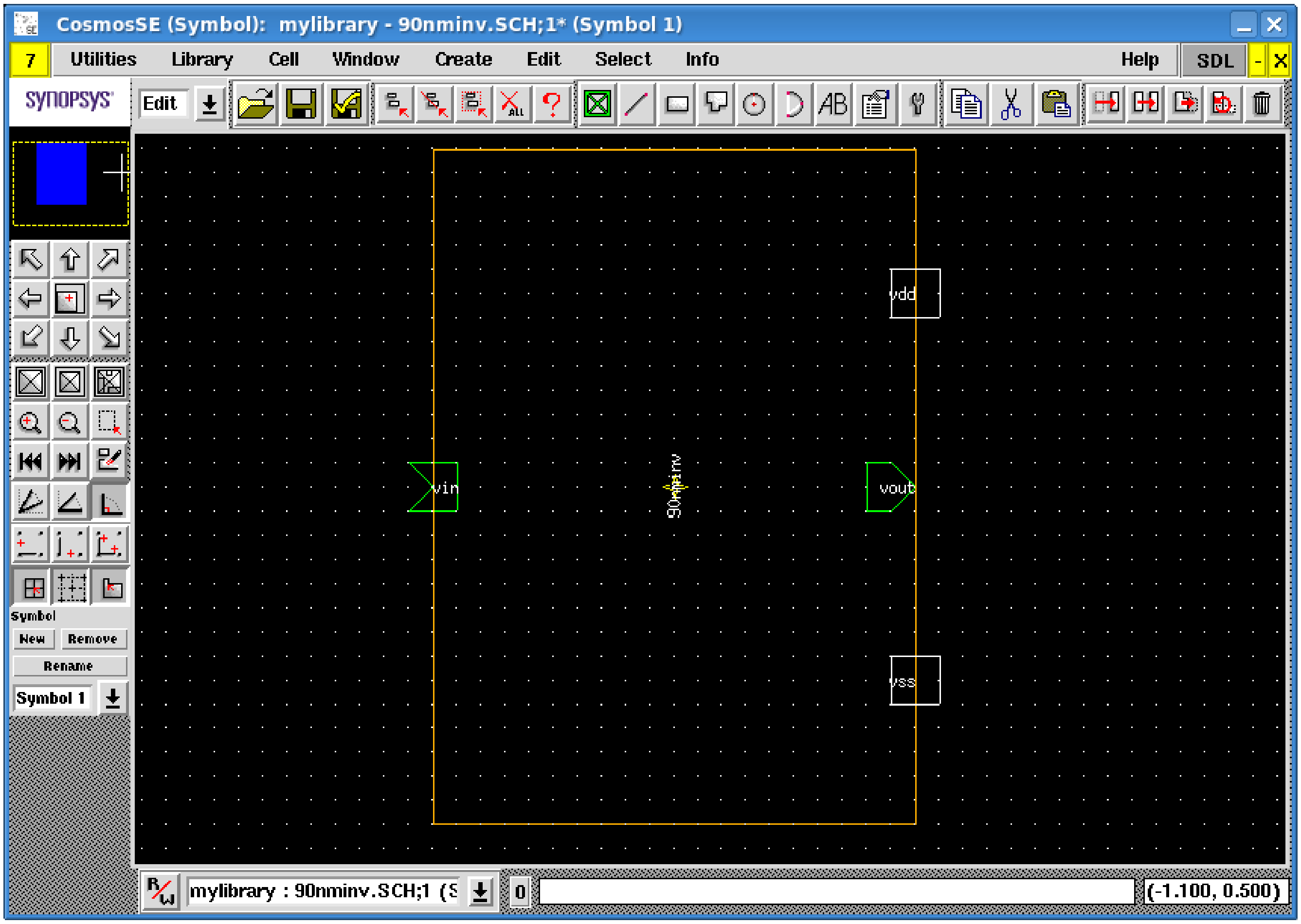
Figure . Schematic with parasitics added



Part 8. Gate level design and simulation

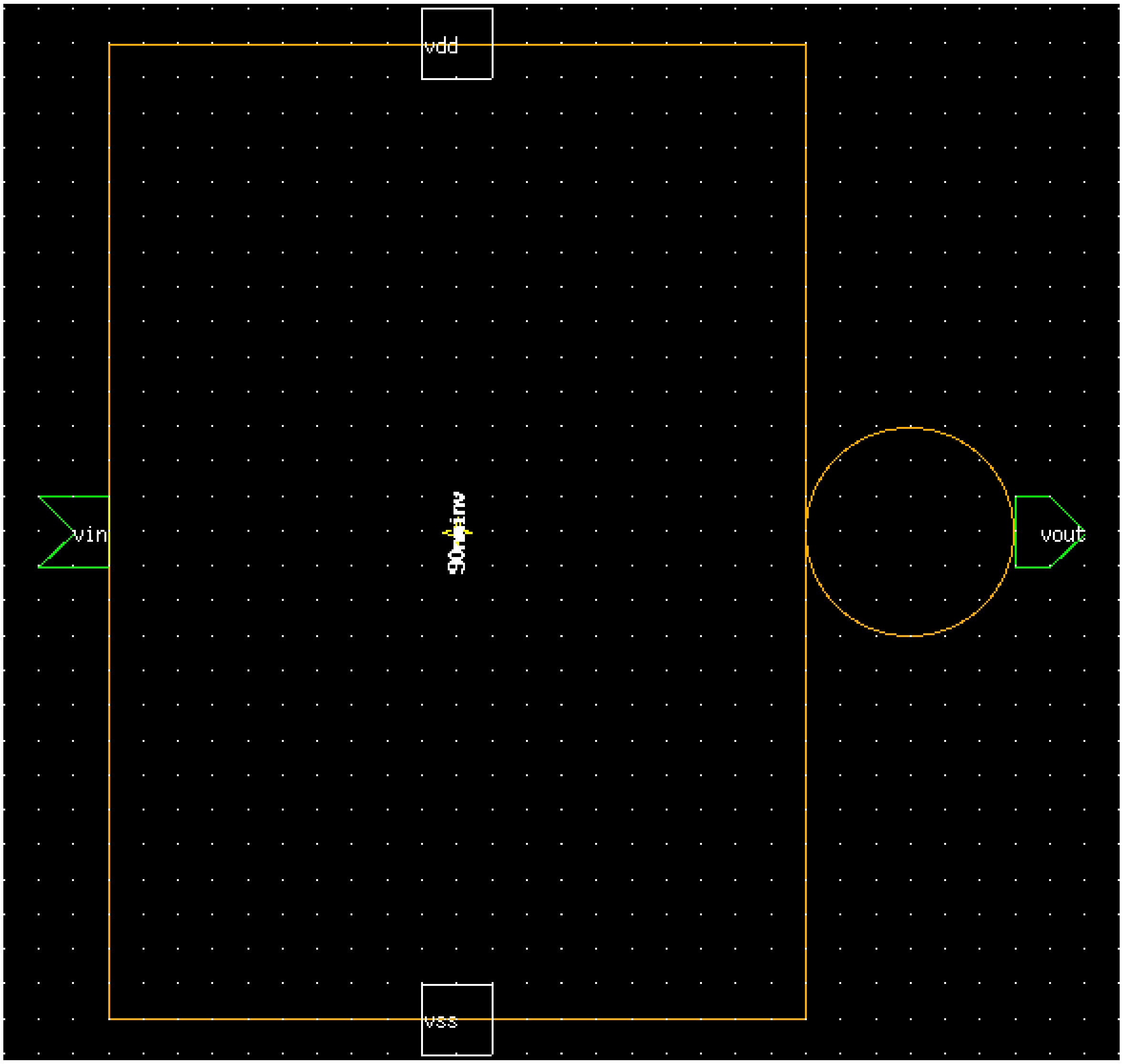
Now that we have a transistor level model of an inverter we can continue on and use that cell in a gate level design of a ring oscillator. First we need to make a symbol for our inverter. In the menu bar go to Cell>Generate Symbol. Cosmos will now generate a symbol of the inverter that can be used higher layers of circuit design.

Figure 13.a Generated symbol for inverter



You can edit the symbol to look more like the standard inverter symbol we are used to seeing by moving the ports and drawing a circle with the new tools you see in the tool bar.

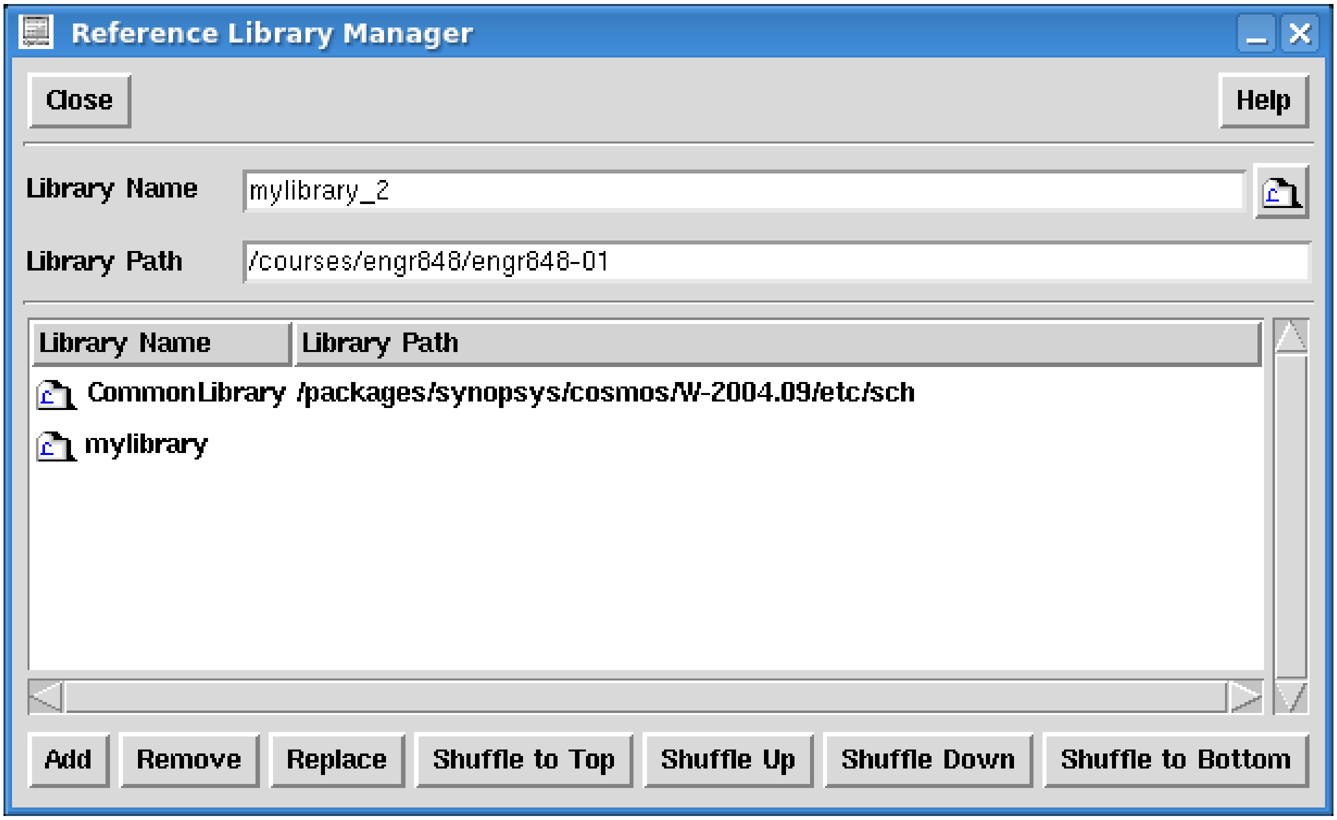
Figure 13.b Modified inverter symbol



In the menu bar go to Libary>Create. In the "Library Name" field enter "mylibrary_2" and in the "Tech File Name" field copy and paste this file directory in the field: "/packages/process_kit/generic/generic_90nm/updated_Oct2008/SAED_EDK90nm/Technology_Kit/techfile/saed90nm_1p9m.tf"

In the menu bar go to Library>Manager. Click "Add" and select the Common library, then add the first library we made called "mylibrary". Click "Close".

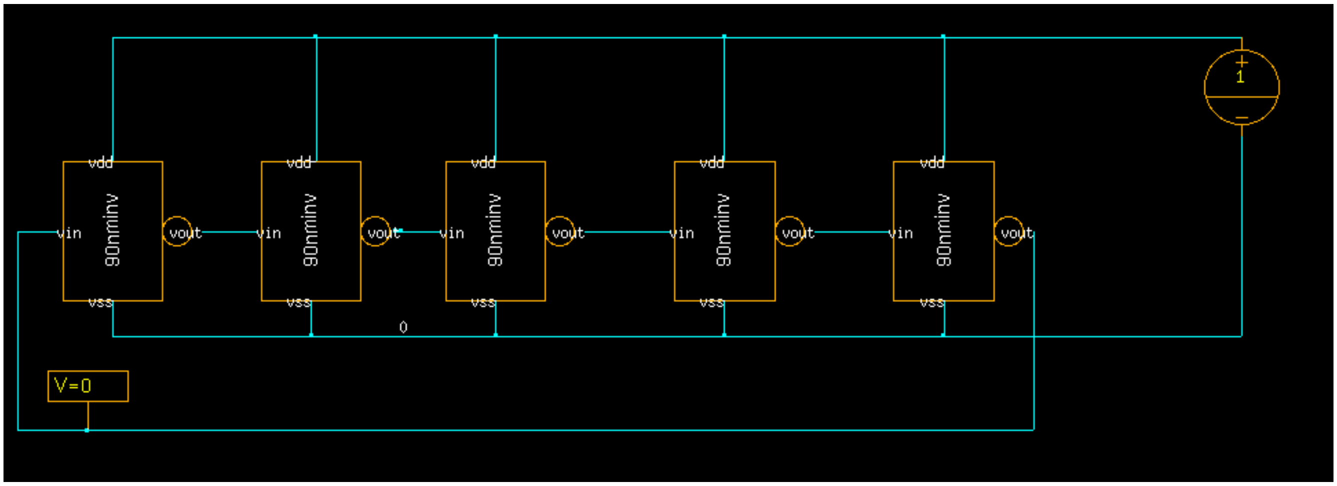
Figure 14. Library manager window



Create a new cell called “ring_oscillator”. Now go to Create>Instance to place parts like you did in the first schematic. Now you should see your invertor in the list of cells. Place five of them in series with the output of the last inverter connected to the input of the first like in the schematic below. Connect them to a 1V DC power supply and again make sure the ground net is '0'. Again go to Create>Instance and select the cell named “nodeset”. Place it on the net near the first invertor like in the schematic. Nodeset allows us to specify initial conditions for that node, which we need to do to make the oscillator oscillate.

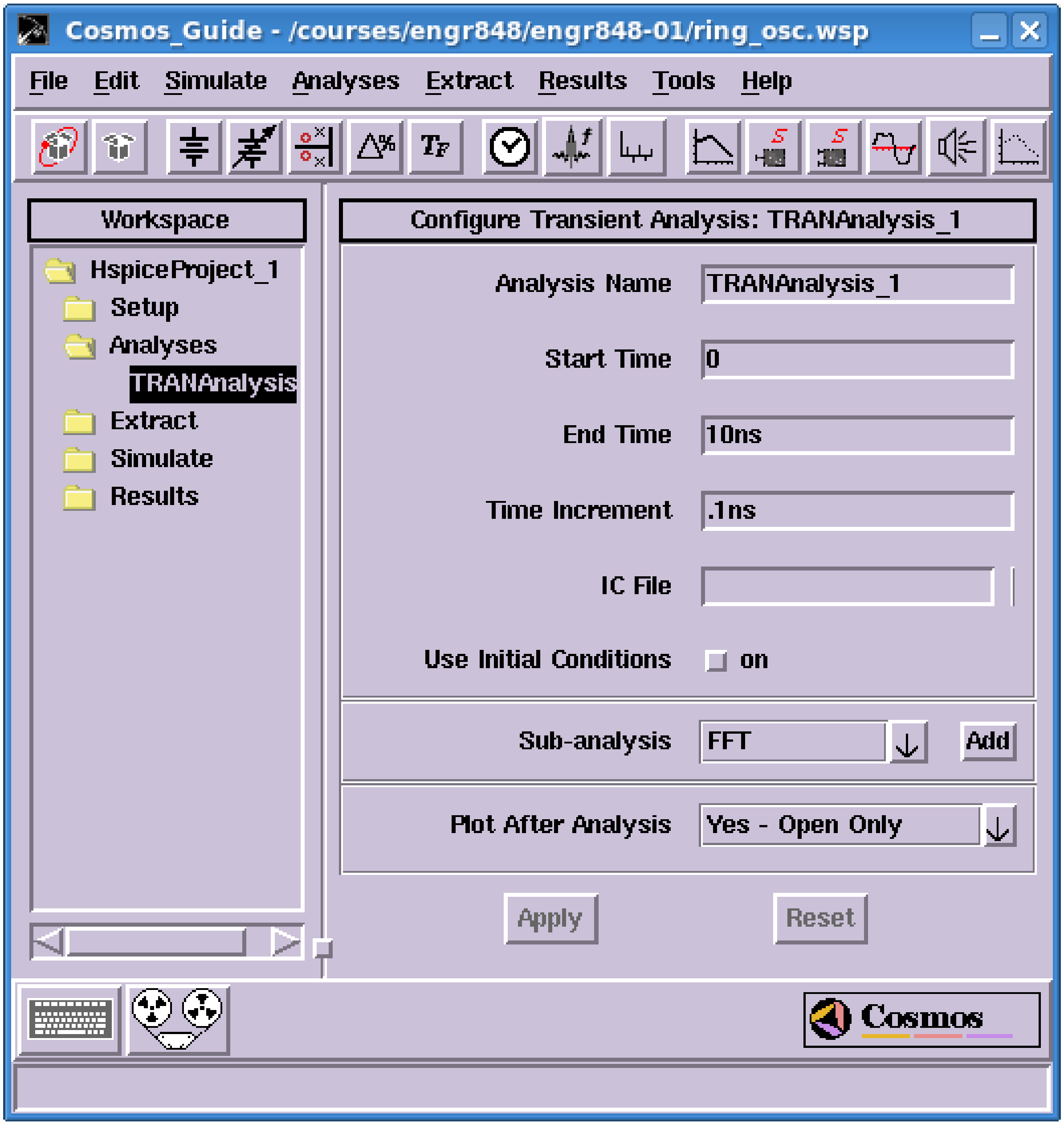
If we didn't specify the initial conditions, the oscillator would stay around 0.5V and not oscillate at all. Change the properties of the nodeset so the initial voltage is zero. You may also want to change the name of the nets. Make sure everything is connected correctly and you have a ground before you proceed to the next step.

Figure 15. Ring oscillator schematic



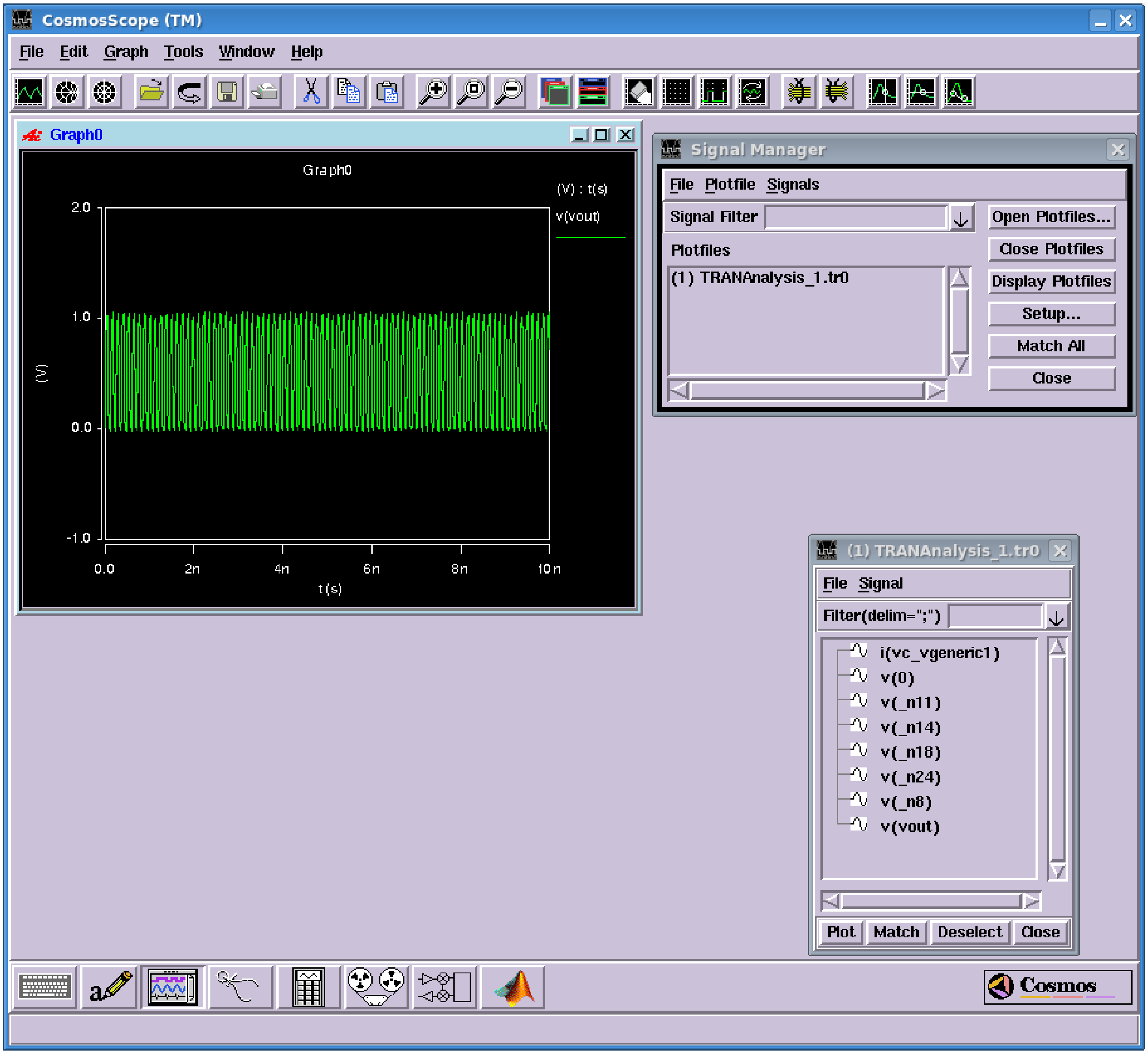
When you are done with the schematic create the netlist by selecting Simulation>Spice in the menu bar. Next select Simulation>Transient Analysis. Fill out the fields like in the figure 16 and click “Apply”.

Figure 16. Transient analysis for ring oscillator in Cosmos Guide



Now go to setup and add the model library like we did in part one on page 13, again selecting TT_12 as our model. When you are done, run the simulation. If you have no errors Cosmos Scope should open and the wave for voltage out of the oscillator should look the same as the one in figure 17.a

Figure 17.a Cosmos scope of ring oscillator



This waveform has a very high frequency, so we'd like to adjust the time axis of the graph so we can see the waveform better. If you move the mouse over the time axis it will turn red like in figure 17.b. Right click and select “Attributes”. Change the 10n in the range field to 2n and click Apply. Your graph should now look like the one in figure 17.d.

Figure 17.b Graph of oscillation

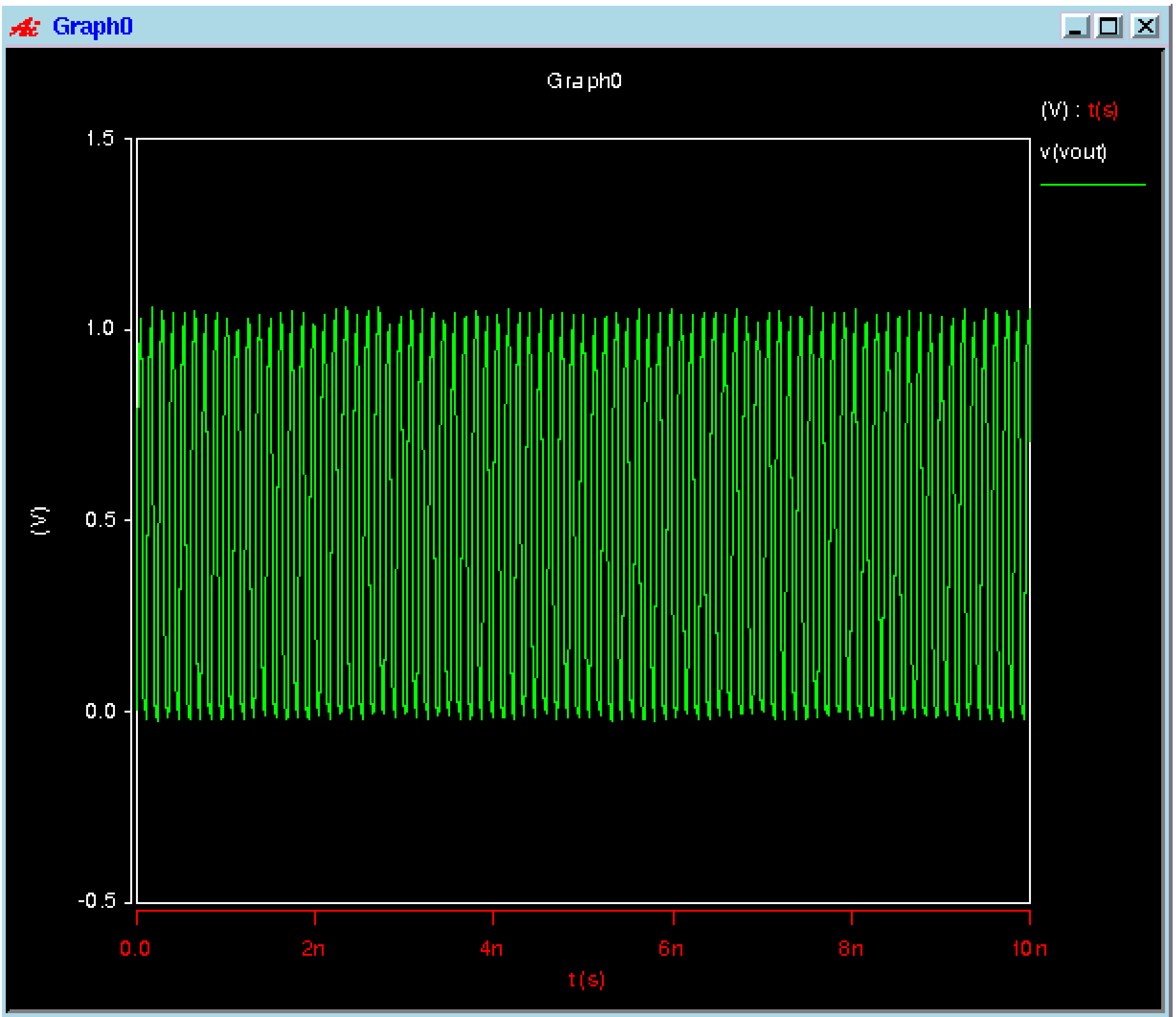


Figure 17.c Edit axis attributes window

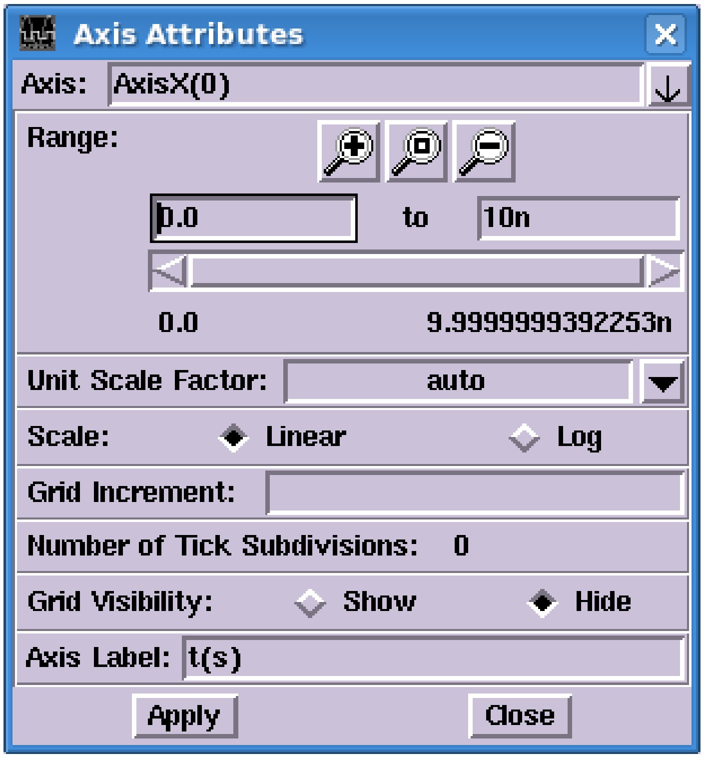
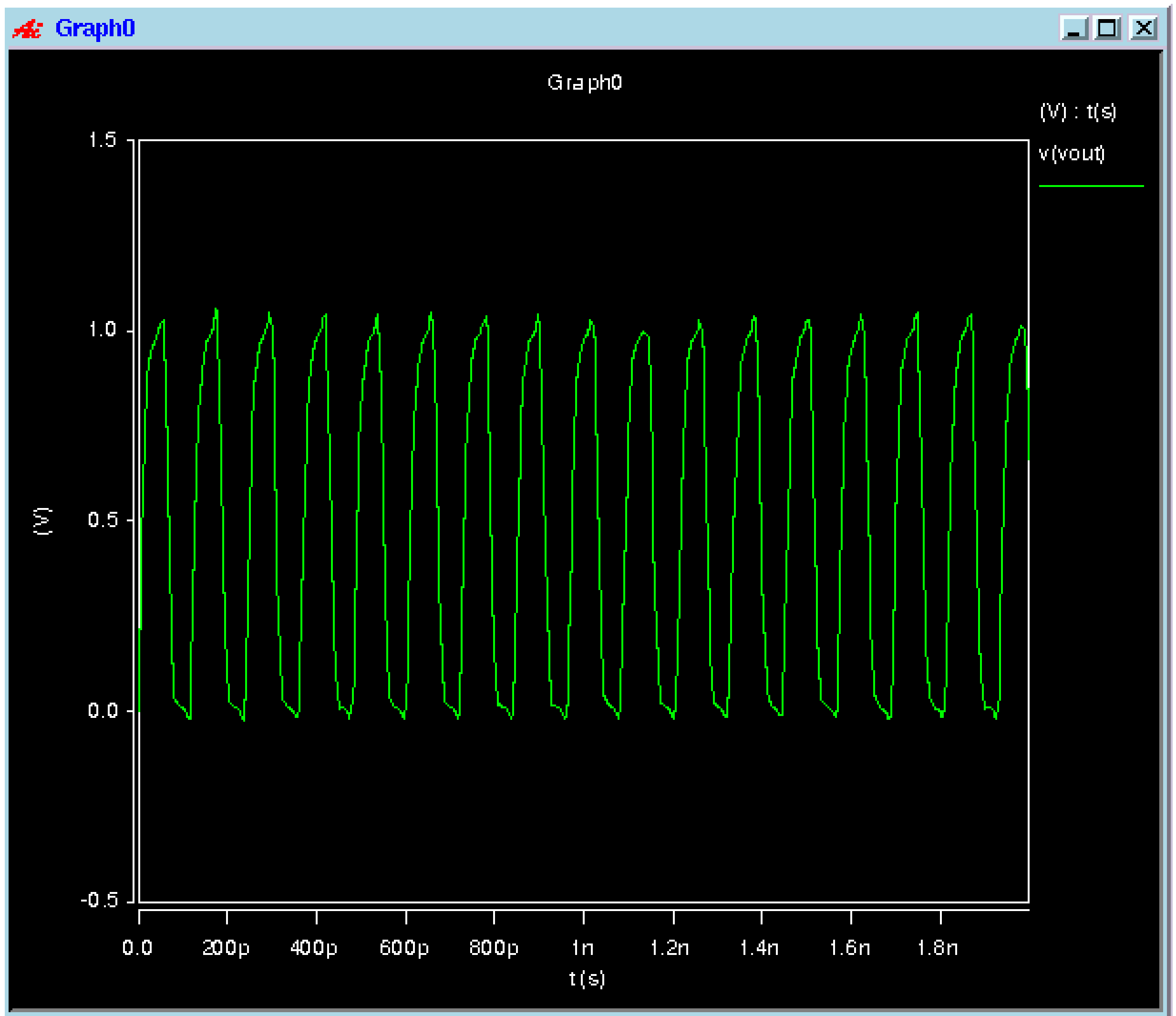


Figure 17.d Adjusted graph of Oscillation



Now that we have simulated the ring oscillator we can find the frequency of oscillation. How could we decrease the frequency? How about increasing it? Try altering the schematic or part properties and simulate the new design. What differences do you see?

Part 9. Trouble Shooting

CosmosLE will not open in the CAD lab:

Try restarting both your SSH session and X-win. Make sure the setup commands have been run successfully.

Reference library, tech file, or runset will not load:

Check to see the the library path is correct after you select a library in the file browser. Sometimes there is a glitch in the GUI. If there is a glitch try typing in the file path manually.

Schematic or layout has glitches:

Inside the schematic or layout window scroll away from the object and then return to the object. It should be refreshed.

Window does not close when close window icon is clicked:

This is an issue with X-win. Inside the window you want go to close go to File>Quit.