

Analog Artist Tutorial

This tutorial is designed to introduce you to the tools we will use in ECE 746. It will introduce you to the **Cadence Environment**, specifically **Composer**, **Analog Artist** and the **Results Browser**.

The following typeface and color conventions will be used in this document:

- General Text: add cadence
- Commands / Menu Selections: **add cadence**
- Screen Output: *add cadence*

1. Starting the Cadence Environment:

% add cadence

Use the following commands to start Cadence applications:

openbook Cadence online documentation

icde & Basic digital and analog design entry
icds & Front end design
icms & Front end analog, mixed signal and microwave design
icca & Cell based chip assembly
layout & Basic layout with interactive DRC
layoutPlus & Basic layout plus automated design tools
msfb & Mixed-signal IC design
icfb & Front to back design

spectre <input file> & Spectre circuit simulator
verilog <input file> & Verilog HDL simulator
signalscan & Analog/digital waveform display

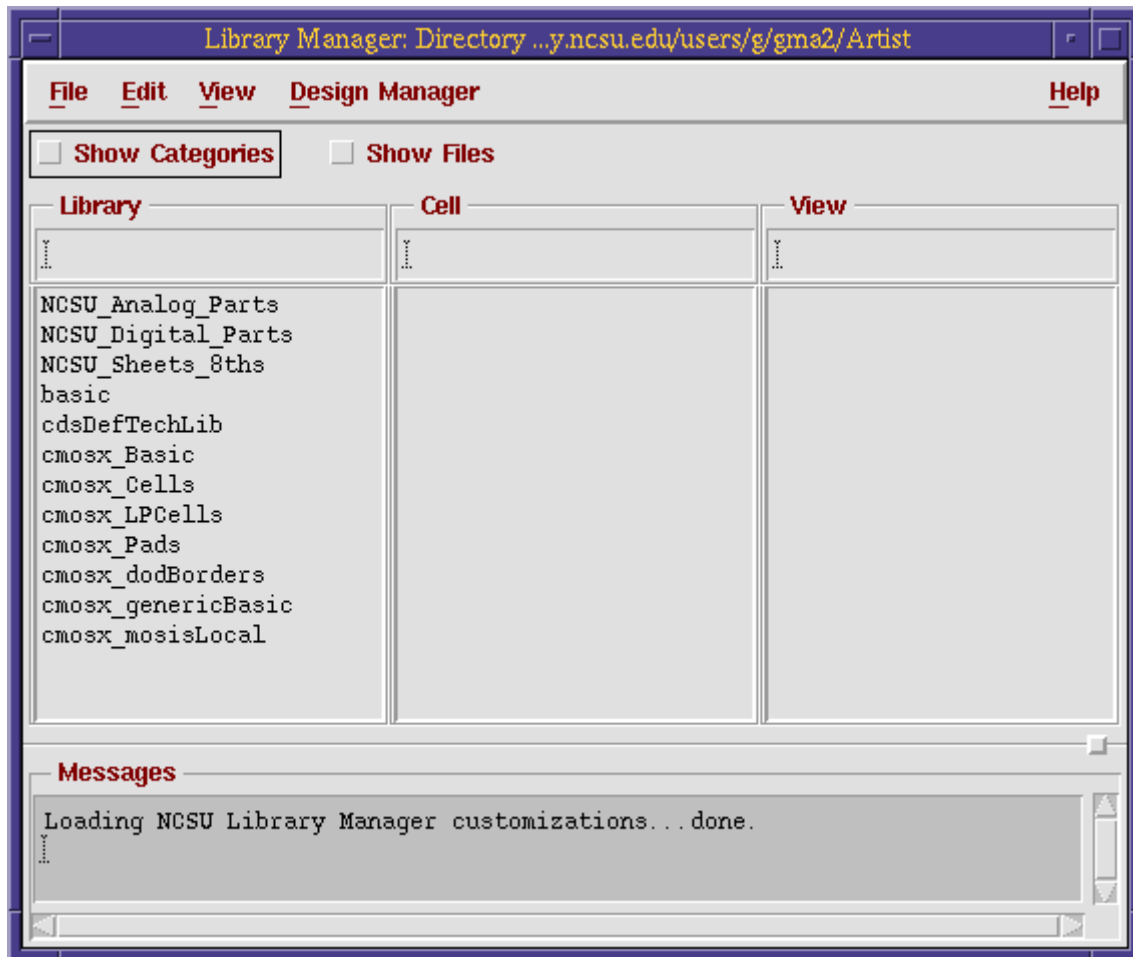
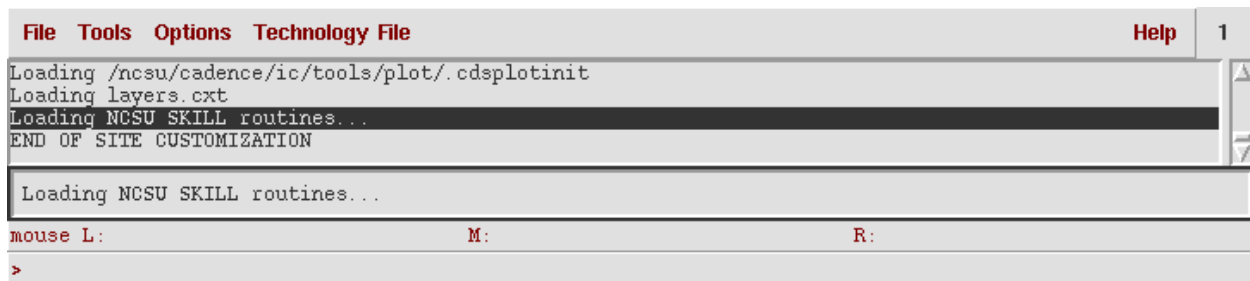
We are now using version 4.4.5 of the Cadence IC tools. If you require version 4.4.2, please type "add cad442" instead.

Problems with the tools? Wondering where SimWaves went? Check the FAQ at

<http://www.ece.ncsu.edu/cadence/doc/cdsuser/>

```
% mkdir Artist  
% cd Artist  
% icfb &
```

A few seconds later, you'll see **CIW** (Command Interpreter Window) and **Library Manager**



2. Create a Design Library:

In **Library Manager**, Click **File -> New -> Library**

In the **Name** field, enter “**ArtistTutorial**”

In the **Technology Library** box, select

Attach to existing tech library -> TSMC 0.30u

Press **OK**

Library Manager dialog box showing the configuration for creating a new design library.

Library

Name: ArtistTutorial

Path:

Technology Library

If this library will not contain physical design (i.e., layout) data you do not need a tech library. Otherwise, you must either attach to an existing tech library or compile one.

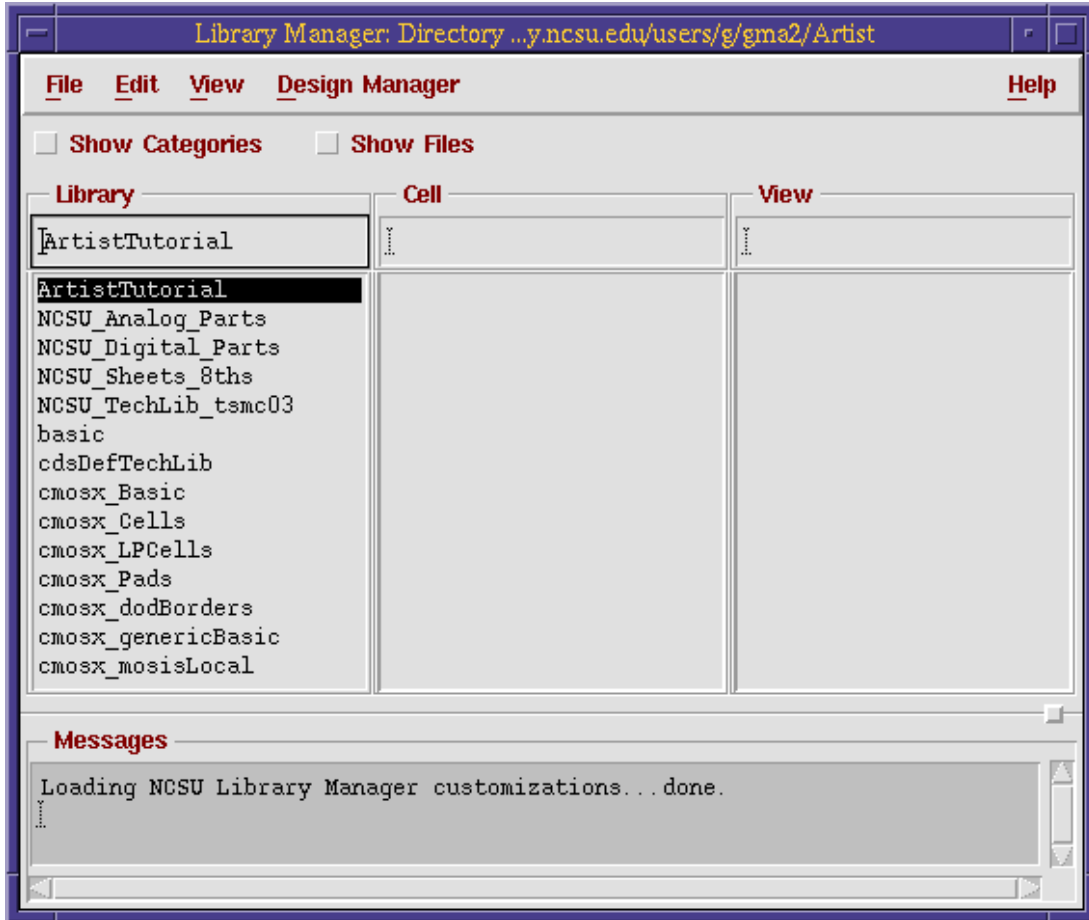
Choose option:

- No tech library needed
- Attach to existing tech library --> TSMC 0.30u CMOS025 (5M, HV FET)
- Compile tech library

Misc.

I/O Pad Type: Perimeter Area array

Then the Library Manager will refresh as follows:



3. Create a Schematic:

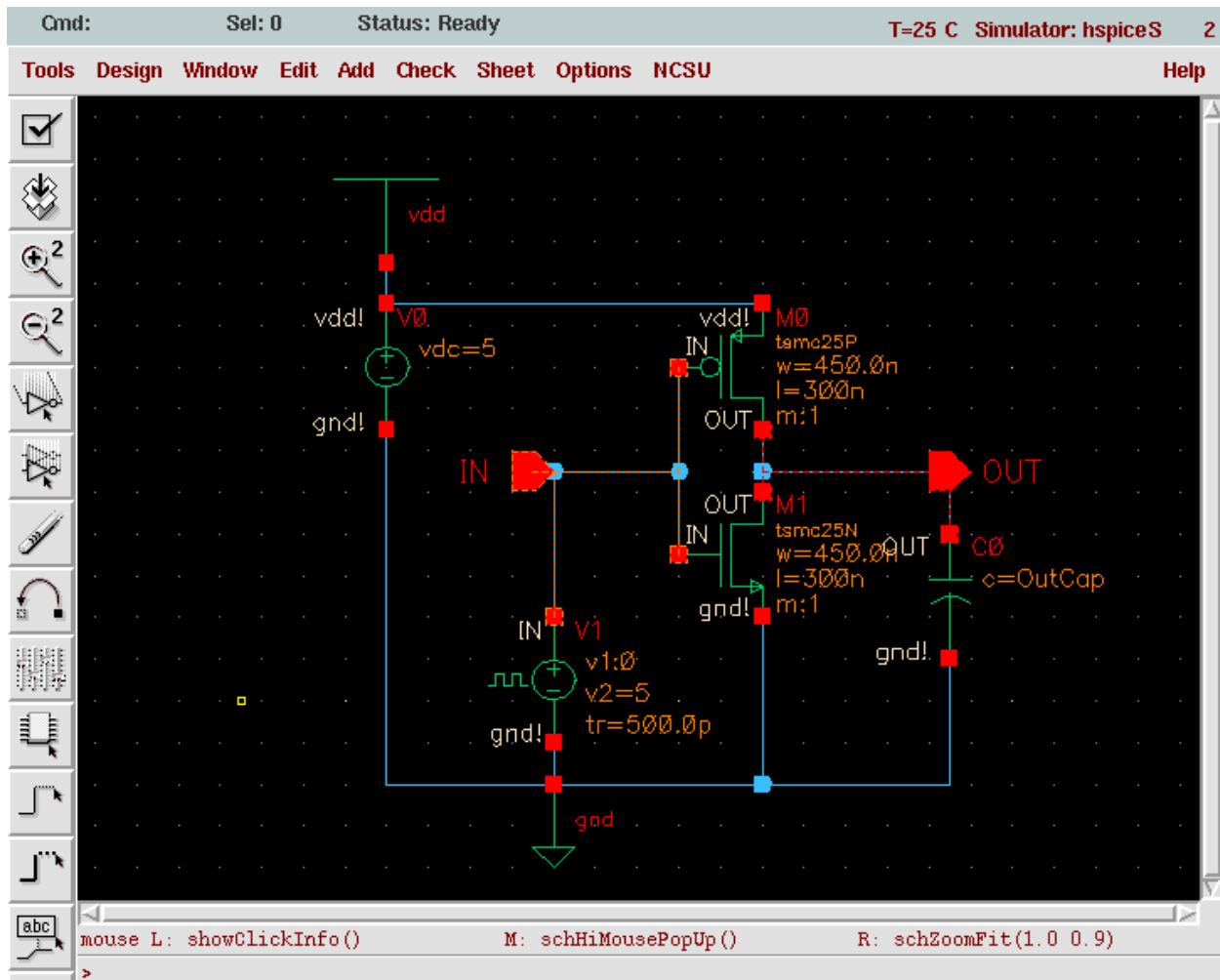
In **Library Manager**, select **ArtistTutorial**

From the Menu Bar, select **File -> New -> CellView**

Fill the Form as follows, then click **OK**

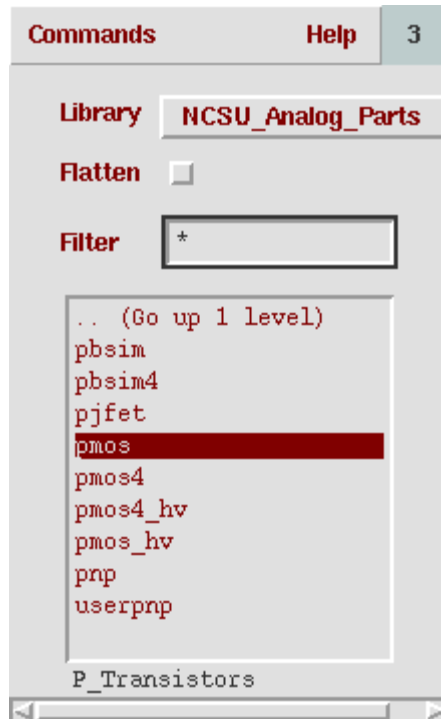
OK	Cancel	Defaults	Help
Library Name	ArtistTutorial		
Cell Name	myInverter		
View Name	schematic		
Tool	Composer - Schematic		
Library path file	csu.edu/users/g/gma2/Artist/cds.lib		

A blank Schematic window will then appear.
We need to generate a schematic as shown below:



To generate a schematic like this, you will need to go through the following steps:

From the **Schematic Window** menu, select **Add -> instance**
The **Component Browser**, will then pop up.



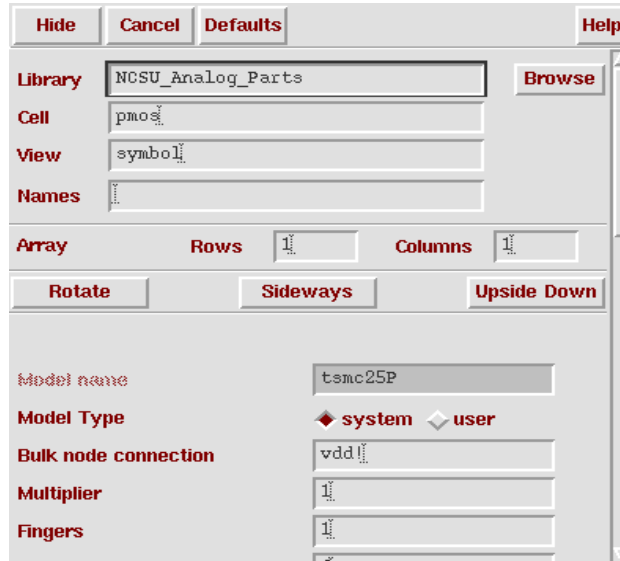
In the **Library** field, select **NCSU_Analog_Parts**
 We will place the following instances in the **Schematic Window** from the **NCSU_Analog_Parts** library as instructed below:

<i>N_Transistor</i>	:	<i>nmos</i>
<i>P_Transistor</i>	:	<i>pmos</i>
<i>Supply_Nets</i>	:	<i>vdd , gnd</i>
<i>Voltage_Sources</i>	:	<i>vdc, vpulse</i>
<i>R_L_C</i>	:	<i>cap</i>

Note: pay attention to the parameters specified in *vdc*, *vpulse*, and *cap*. These parameters are very important in simulation

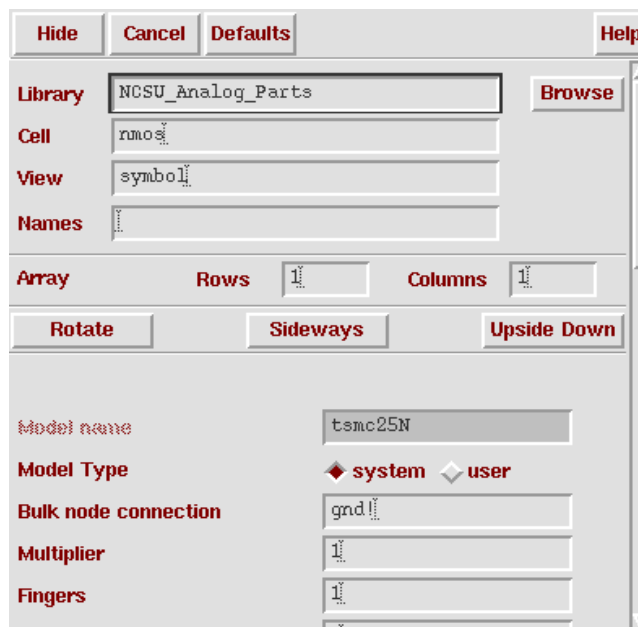
A. Place pmos instance:

In **Component Browser**, select **P_Transistor** and then **pmos**
Place it in the **Schematic Window**



B. Place nmos instance:

In **Component Browser**, select **N_Transistor** and then **nmos**
Place it in the **Schematic Window**



C. Place gnd instance:

In **Component Browser**, select **Supply_Nets** and then **gnd**
Place it in the **Schematic Window**

The screenshot shows the Component Browser dialog box. At the top, there are buttons for 'Hide', 'Cancel', 'Defaults', and 'Help'. Below these are four text input fields: 'Library' containing 'NCSU_Analog_Parts', 'Cell' containing 'gnd', 'View' containing 'symbol', and 'Names' which is empty. To the right of the 'Library' field is a 'Browse' button. Below the input fields are two spinners for 'Rows' and 'Columns', both set to '1'. At the bottom, there are three buttons: 'Rotate', 'Sideways', and 'Upside Down'.

D. Place vdd instance:

In **Component Browser**, select **Supply_Nets** and then **vdd**
Place it in the **Schematic Window**

The screenshot shows the Component Browser dialog box. At the top, there are buttons for 'Hide', 'Cancel', 'Defaults', and 'Help'. Below these are four text input fields: 'Library' containing 'NCSU_Analog_Parts', 'Cell' containing 'vdd', 'View' containing 'symbol', and 'Names' which is empty. To the right of the 'Library' field is a 'Browse' button. Below the input fields are two spinners for 'Rows' and 'Columns', both set to '1'. At the bottom, there are three buttons: 'Rotate', 'Sideways', and 'Upside Down'.

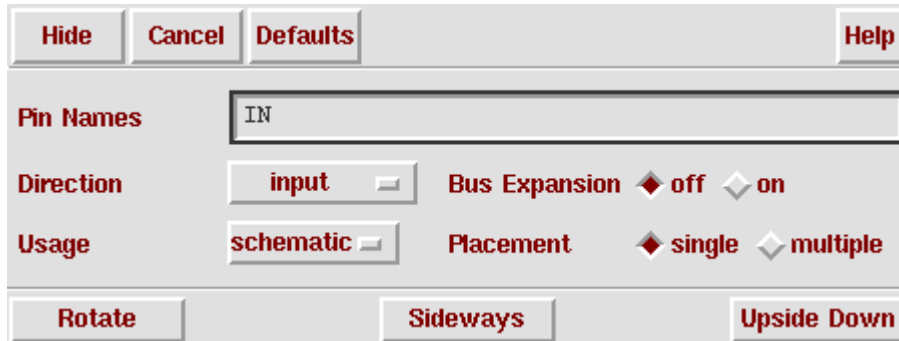
E. Place IN pin:

From the **Schematic Window** menu, select **Add -> Pin...**

In the Pin Name field , enter “**IN**”

In the Direction field, select **input**

Place it in the **Schematic Window**



The screenshot shows a dialog box for adding a pin. At the top are buttons for 'Hide', 'Cancel', 'Defaults', and 'Help'. The 'Pin Names' field contains 'IN'. The 'Direction' dropdown is set to 'input'. The 'Usage' dropdown is set to 'schematic'. The 'Bus Expansion' section has 'off' selected with a diamond indicator. The 'Placement' section has 'single' selected with a diamond indicator. At the bottom are buttons for 'Rotate', 'Sideways', and 'Upside Down'.

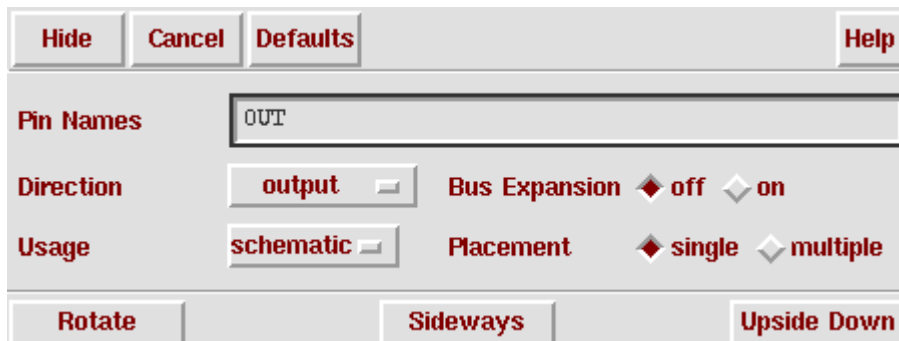
F. Place OUT pin:

From the **Schematic Window** menu, select **Add -> Pin...**

In the Pin Name field , enter “**OUT**”

In the Direction field, select **output**

Place it in the **Schematic Window**



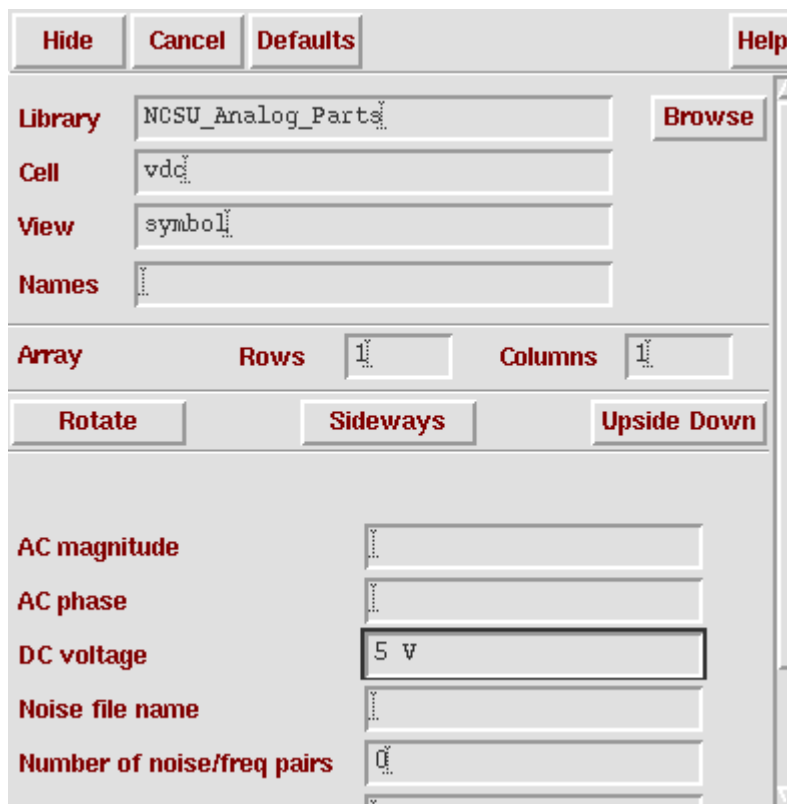
The screenshot shows a dialog box for adding a pin. At the top are buttons for 'Hide', 'Cancel', 'Defaults', and 'Help'. The 'Pin Names' field contains 'OUT'. The 'Direction' dropdown is set to 'output'. The 'Usage' dropdown is set to 'schematic'. The 'Bus Expansion' section has 'off' selected with a diamond indicator. The 'Placement' section has 'single' selected with a diamond indicator. At the bottom are buttons for 'Rotate', 'Sideways', and 'Upside Down'.

G. Place vdc instance

In **Component Browser**, select **Voltage Sources** and then **vdc**

In the DC voltage field, enter “**5 v**”

Place it in the **Schematic Window**



The screenshot shows the Component Browser dialog box for the vdc component. The dialog has a title bar with buttons for Hide, Cancel, Defaults, and Help. The main area contains several fields and buttons:

- Library:** NCSU_Analog_Parts (with a Browse button)
- Cell:** vdc
- View:** symbol
- Names:** (empty field)
- Array:** Rows: 1, Columns: 1
- Rotate:** (button)
- Sideways:** (button)
- Upside Down:** (button)
- AC magnitude:** (empty field)
- AC phase:** (empty field)
- DC voltage:** 5 v (highlighted)
- Noise file name:** (empty field)
- Number of noise/freq pairs:** 0

H. Place vpulse instance

In **Component Browser**, select **Voltage_Sources** and then **vpulse**

Enter the values as shown in the following form (next page)

Place it in the **Schematic Window**

Edit Object Properties [X]

OK Cancel Apply Defaults Previous Next Help

Apply To

Show system user CDF

Property	Value	Display
Library Name	NCSU_Analog_Part	off
Cell Name	vpulse	off
View Name	symbol	off
Instance Name	v1	off

User Property	Master Value	Local Value	Display
IvsIgnore	TRUE		off

CDF Parameter	Value	Display
AC magnitude		off
AC phase		off
Voltage 1	0 V	off
Voltage 2	5 V	off
Delay time	0 s	off
Rise time	500p	off
Fall time	500p	off
Pulse width	3n	off
Period	6ns	off
DC voltage		off
Noise file name		off
Number of noise/freq pairs	1	off
Temperature coefficient 1		off
Temperature coefficient 2		off
Nominal temperature		off
Frequency		off
Number of harmonics		off
Gibb's compensation	<input type="checkbox"/>	off
DC source		off

I. Place cap instance

In **Component Browser**, select **R_L_C** and then **cap**
In the Capacitance field, enter “**OutCap F**”
(This **Design Variable** will be used in **Artist**.)

Place it in the **Schematic Window**

The screenshot shows the Component Browser dialog box with the following configuration:

- Apply To:** only current, instance
- Show:** system, user, CDF
- Buttons:** Browse, Reset Instance Labels Display
- Instance Properties:**

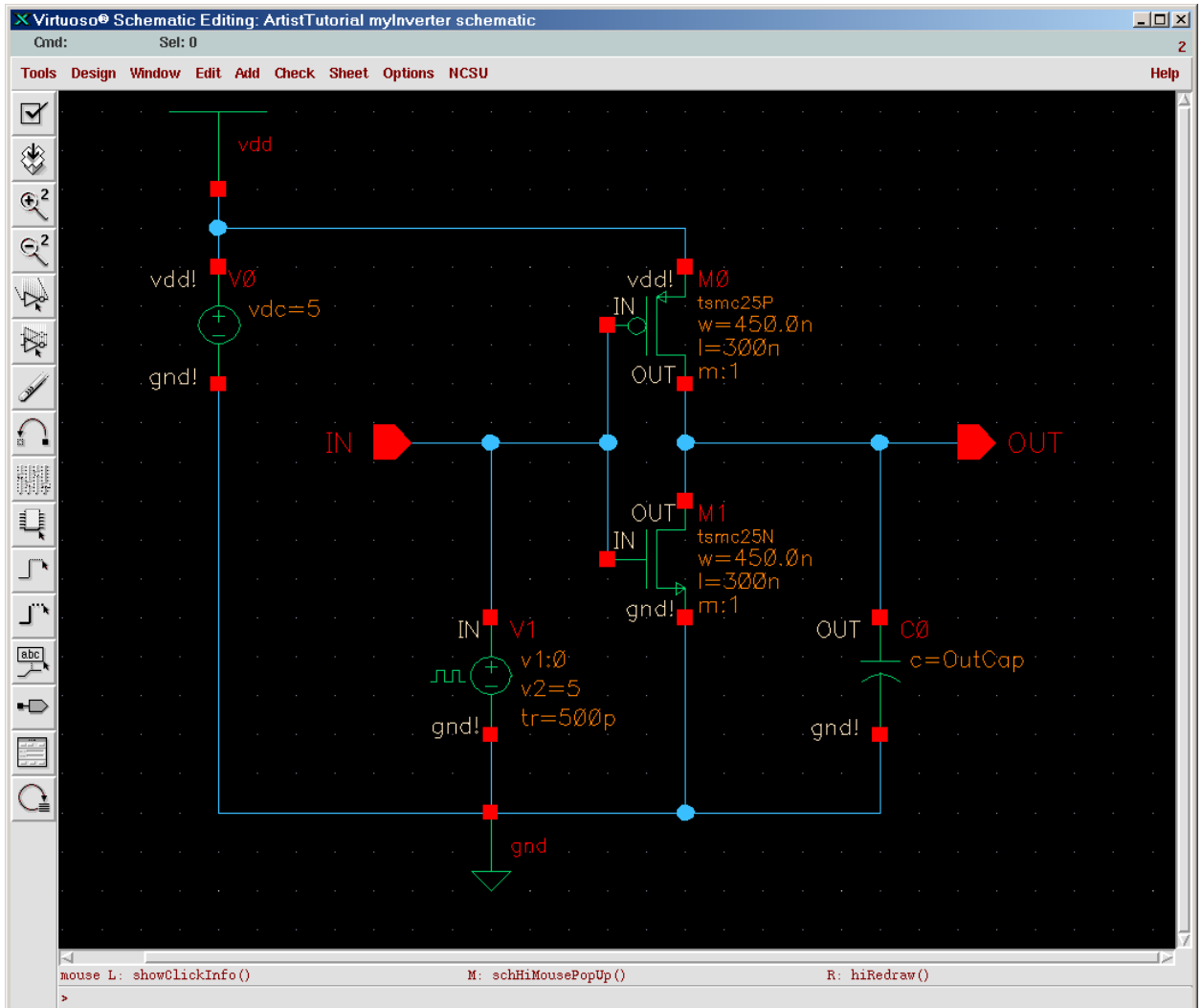
Property	Value	Display
Library Name	NCSU_Analog_Parts	off
Cell Name	cap	off
View Name	symbol	off
Instance Name	C0	off
- Buttons:** Add, Delete, Modify
- CDF Parameters:**

CDF Parameter	Value	Display
Capacitance	OutCap F	off
Initial condition		off
Model name		off
Width		off
Length		off
Multiplier		off

J. Place wires

In the **Schematic Window** menu, select **Add -> Wire (narrow)**
Place the wire to connect all the instances
Select **Design -> Check and Save**. CIW will report any errors.

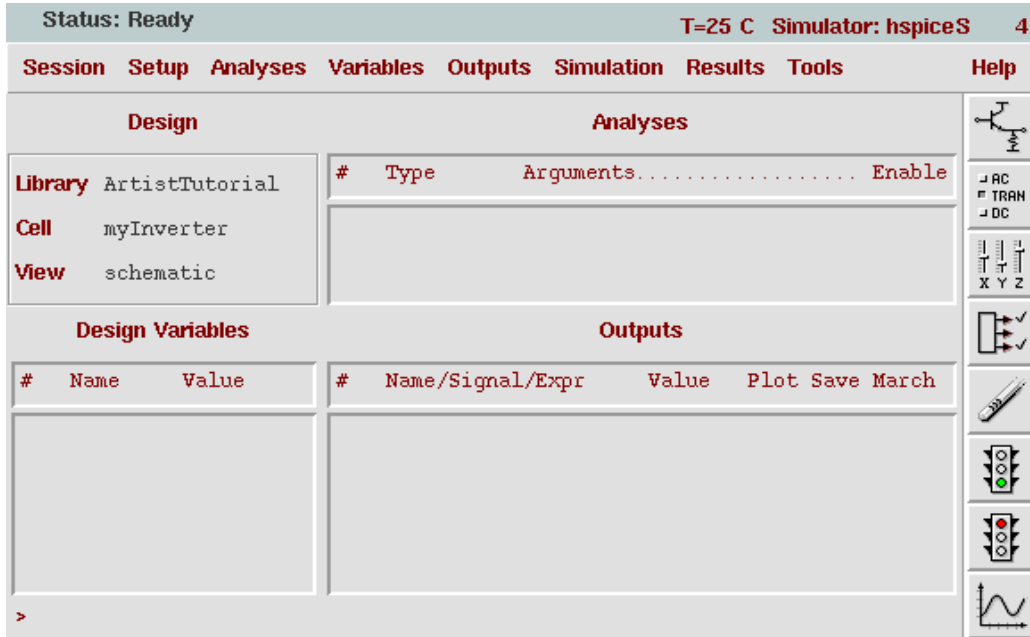
Your schematic should look like the one shown below.



4. Set up the Simulation Environment

You are now prepared to simulate your circuit.

From the **Schematic Window** menu, select **Tools -> Analog Environment**. A window will pop-up. This window is the **Analog Artist Simulation Window**.



A. Choose a Simulator

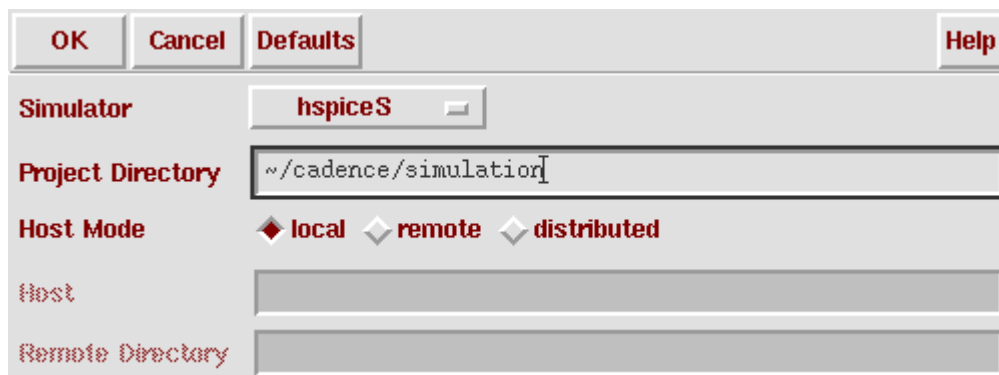
From the **Analog Artist** menu, select **Setup -> Simulator/Directory/Host**.

Enter the fields as shown below.

Choose **hspiceS** as your simulator.

Your simulation will run in the specified Project Directory.

You may choose any valid pathname and filename that you like.



B. Choose Analysis

We will do Transient Analysis on the circuit that we just produced.

From the **Analog Artist** menu, select **Analyses -> Choose...**
Fill out the form as follows:

Analysis dc noise ac tran

Transient Analysis

From To By

Max Step

Enabled

C. Add a Variable

From the **Analog Artist** menu, select **Variables -> Edit**

The **Editing Design Variables** form will appear.

Fill out the form as shown below, and then click **Add** to send this Variable to the Table of Design Variables.

(We entered the OutCap Design Variable in section 3.I.)

Selected Variable

Name

Value (Expr)

Table of Design Variables

#	Name	Value
---	------	-------

Add Delete Change Next Clear Find

Cellview Variables Copy From Copy To

D. Setup Output

When using Transient Analysis, the transient voltage will be saved automatically. We can save the current through capacitor C0 in the schematic by doing the following:

From the **Analog Artist** menu, select

Outputs -> To be Saved -> Select On Schematic

In the **Schematic Window**, click on the lower terminal (not the wire) of capacitor C0.

After you click on the terminal, the **Analog Artist Window** should look like this:

The screenshot shows the Analog Artist window with the following configuration:

- Status: Ready
- Temperature: T=25 C
- Simulator: hspiceS
- Session: 3
- Menu: Session Setup Analyses Variables Outputs Simulation Results Tools Help
- Design:
 - Library: ArtistTutorial
 - Cell: myInverter
 - View: schematic
- Analyses:

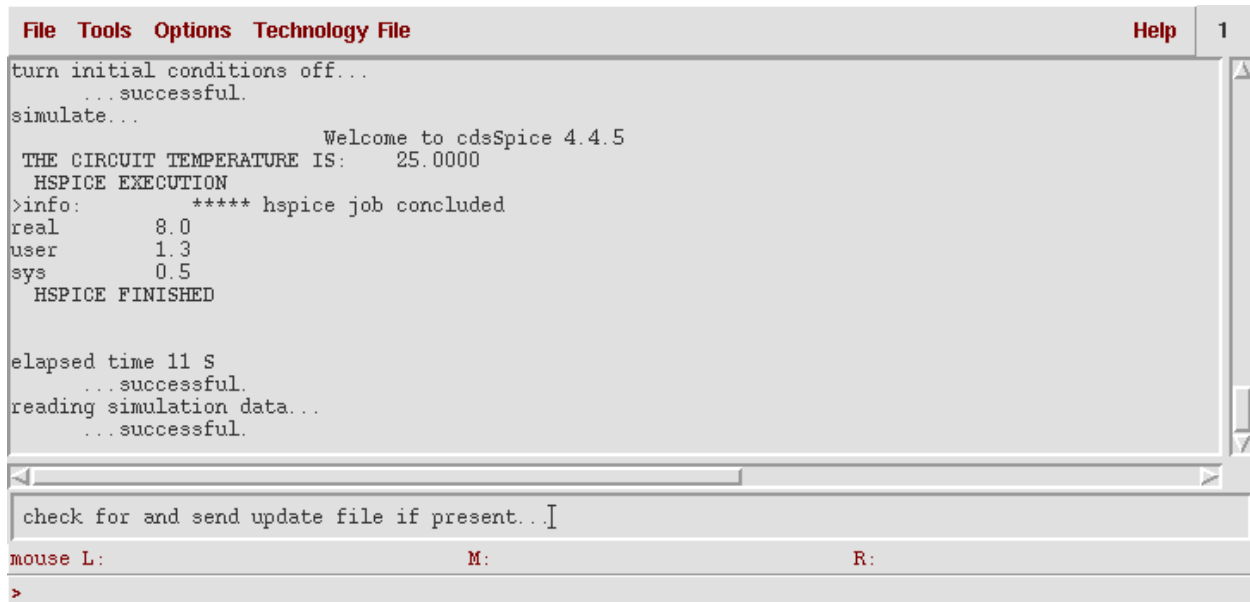
#	Type	Arguments.....	Enable
1	tran	0 15n 100p	yes
- Design Variables:

#	Name	Value
1	OutCap	10f
- Outputs:

#	Name/Signal/Expr	Value	Plot	Save	March
1	C0/MINUS		no	yes	no

5. Run Simulation

From the **Analog Artist** menu, select **Simulation -> Run**,
Look at the echoing information in the **CIW** window.
If the simulation succeeds, the window will display “...successful.”



The screenshot shows a terminal window with a menu bar at the top containing 'File', 'Tools', 'Options', 'Technology File', 'Help', and '1'. The main text area displays the following output:

```
turn initial conditions off...  
...successful.  
simulate...  
                                     Welcome to cdsSpice 4.4.5  
THE CIRCUIT TEMPERATURE IS: 25.0000  
HSPICE EXECUTION  
>info:      ***** hspice job concluded  
real       8.0  
user       1.3  
sys        0.5  
HSPICE FINISHED  
  
elapsed time 11 S  
...successful.  
reading simulation data...  
...successful.
```

Below the main text area is a scroll bar and a status bar containing the text 'check for and send update file if present...'. At the bottom of the window, there are labels for 'mouse L:', 'M:', and 'R:', followed by a right-pointing arrow '>'.

6. View Waveforms

From the **Analog Artist** menu, select

Results -> Direct Plot -> Transient Signal

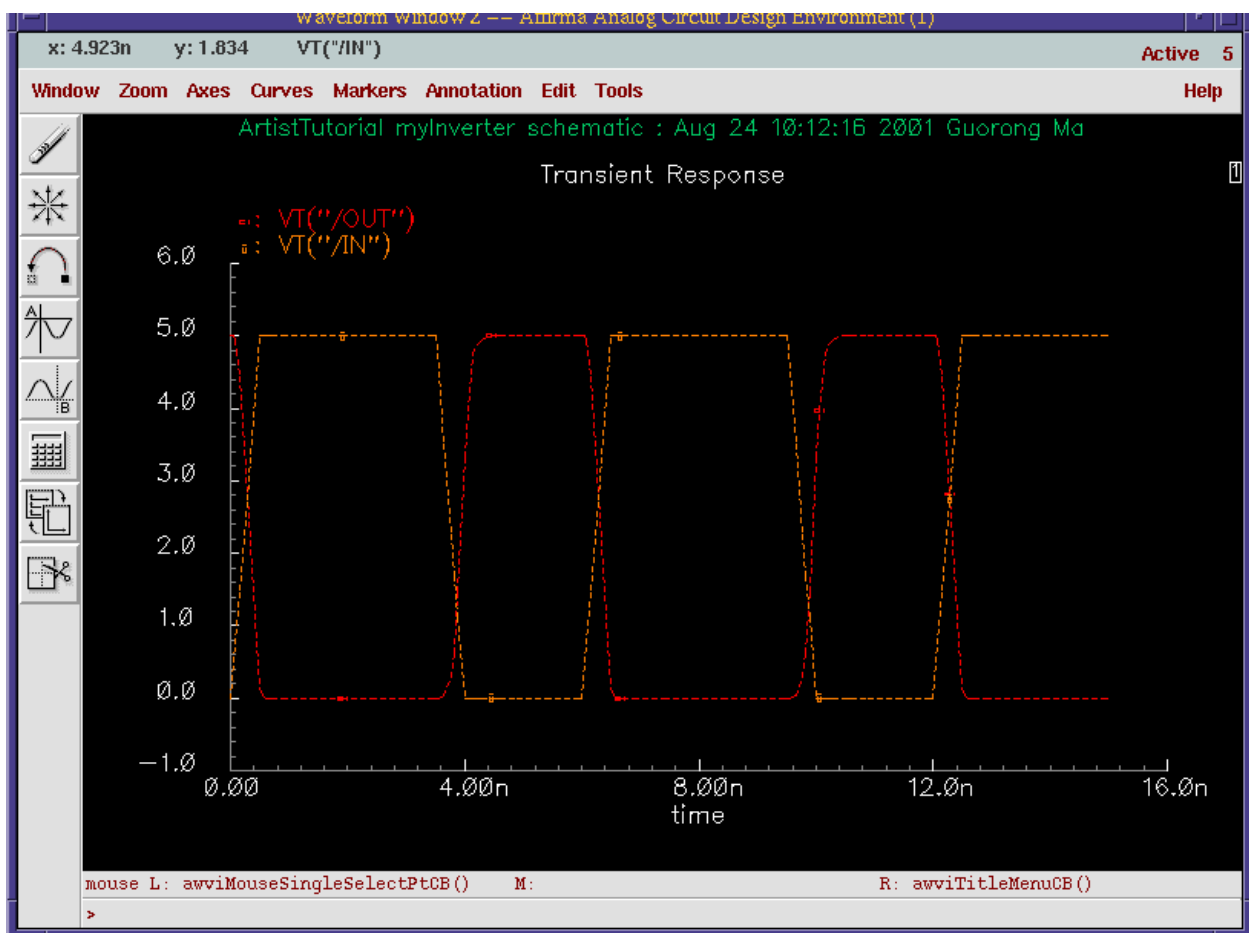
The **Waveform Window** will then pop up

In the **Schematic Window**,

Click on the IN wire and then **Click on the OUT wire**

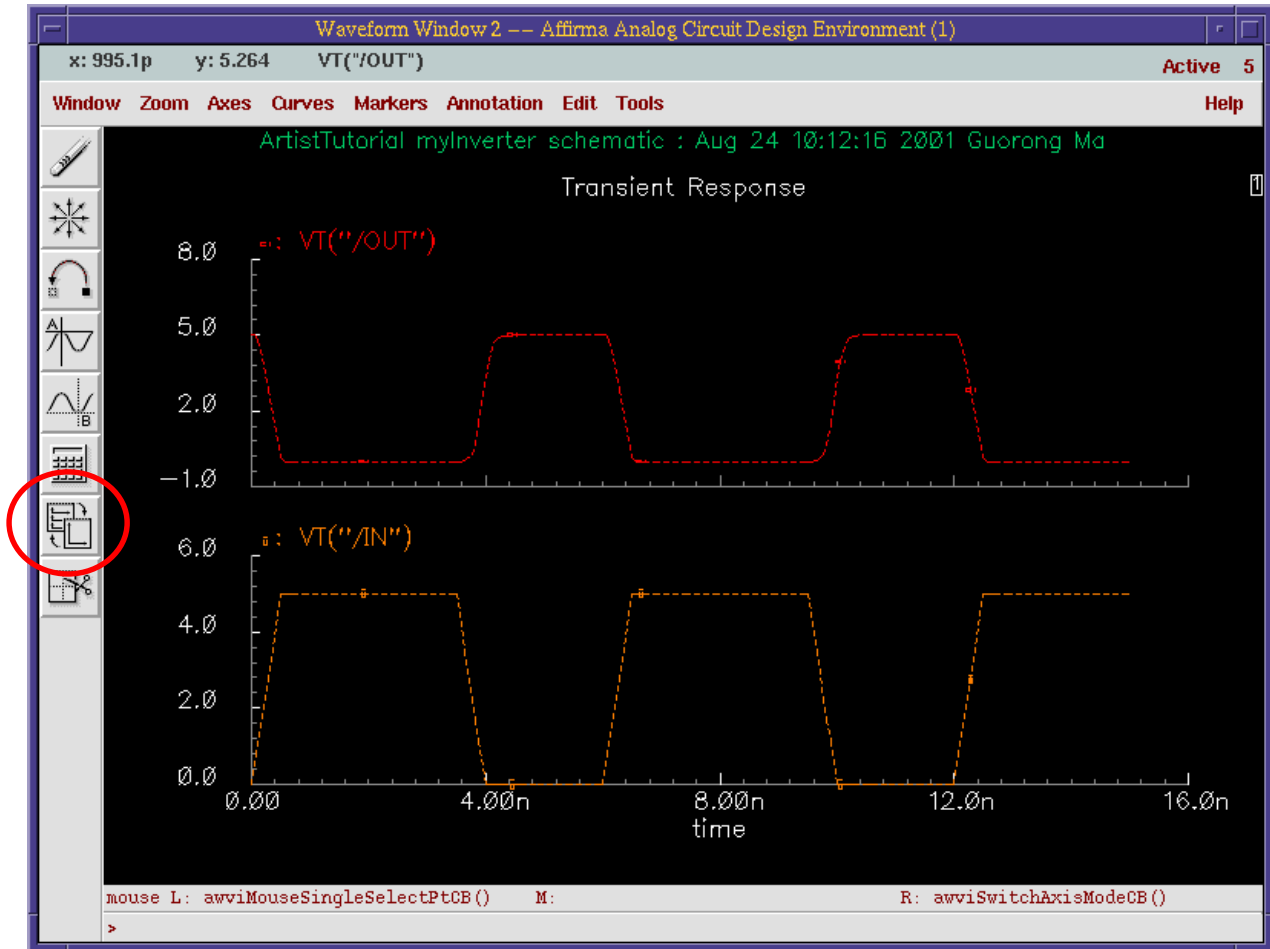
Press **ESC** on your keyboard

The two curves (IN and OUT) will then be displayed in this window:



Press the **Switch Axis Mode** icon (circled in Red) on the left side of the **Waveform Window**

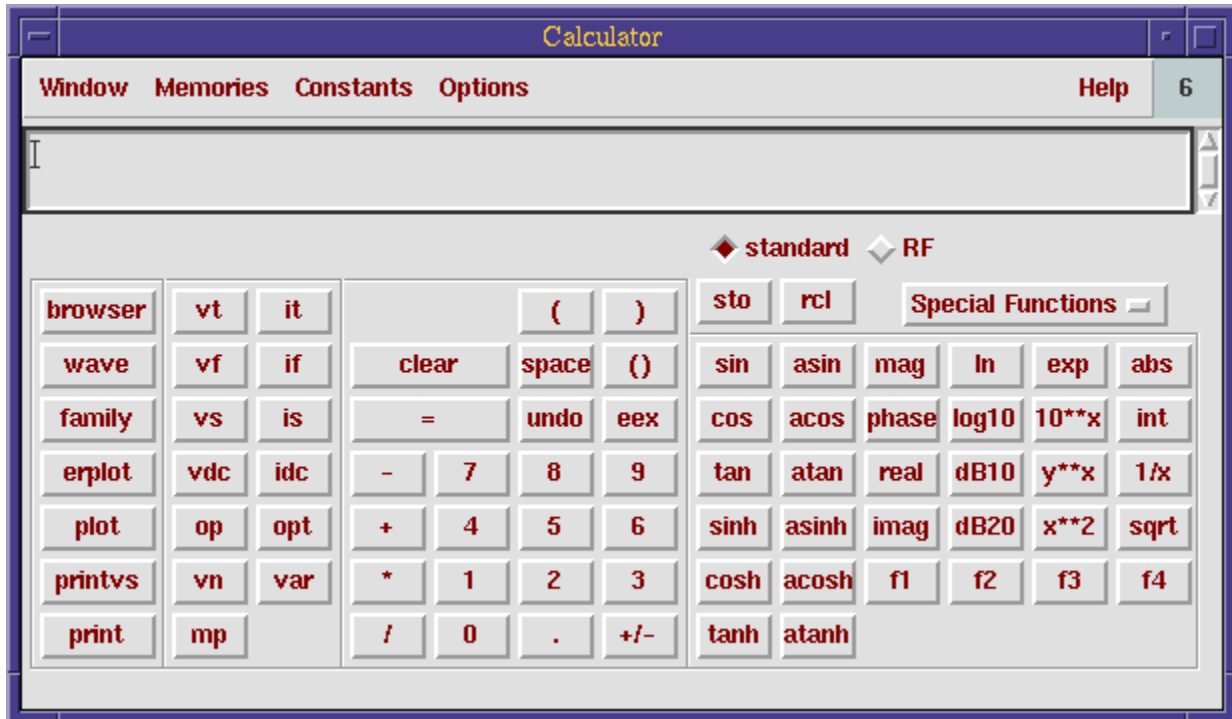
The waveforms will then be displayed separately as shown below:



7. Use Calculator

In **Artist Window**, go to **Tools -> Calculator**.

The **Calculator Window** will then pop up, as shown below:



In **Calculator Window**, go to **Options -> Set Algebraic**

We are going to use the calculator to plot both the current through the capacitor and the absolute value of the capacitor current.

In **Waveform Window**, select **Window -> Reset** to clear the input and output plots from the window.

In the **Calculator Window**, click on the **it** button (3rd column of buttons on the top).

In the **Schematic Window**, click on the lower terminal of the capacitor.

Returning to the **Calculator Window**, the text area at the top should like this:



In the **Calculator Window**, press the **plot** button to plot this waveform in the **Waveform Window**.

In the **Waveform Window**, press the **Add Subwindow** button (bottom button on left).

In the **Calculator Window**, press the **clear** button (4th Column, top) to erase the text area, press the **abs** button (last column, top), and press the **it** button.

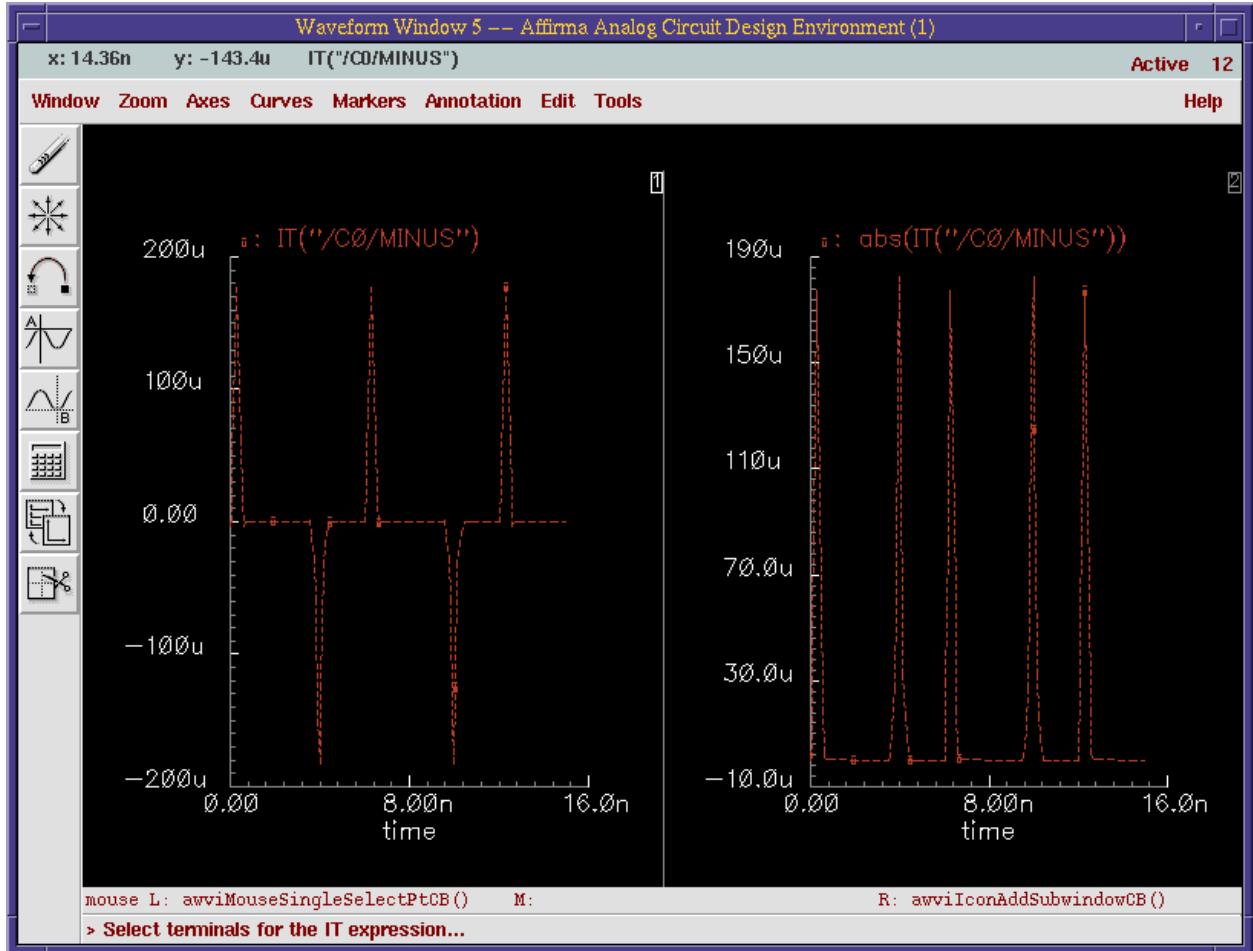
In the **Schematic Window**, click on the lower terminal of the capacitor.

Returning to the **Calculator Window**, the text area at the top should like this:



In the **Calculator Window**, press the **plot** button to plot this waveform in the **Waveform Window**.

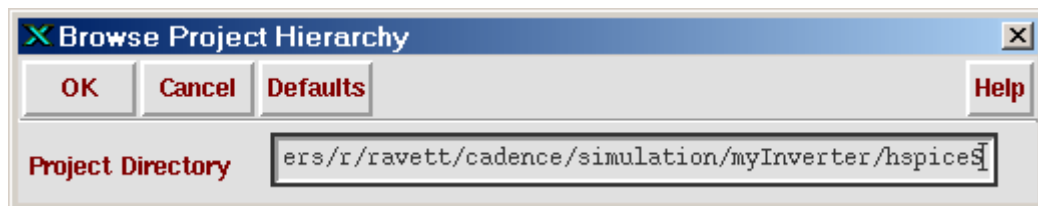
Your **Waveform Window** should now look like this:



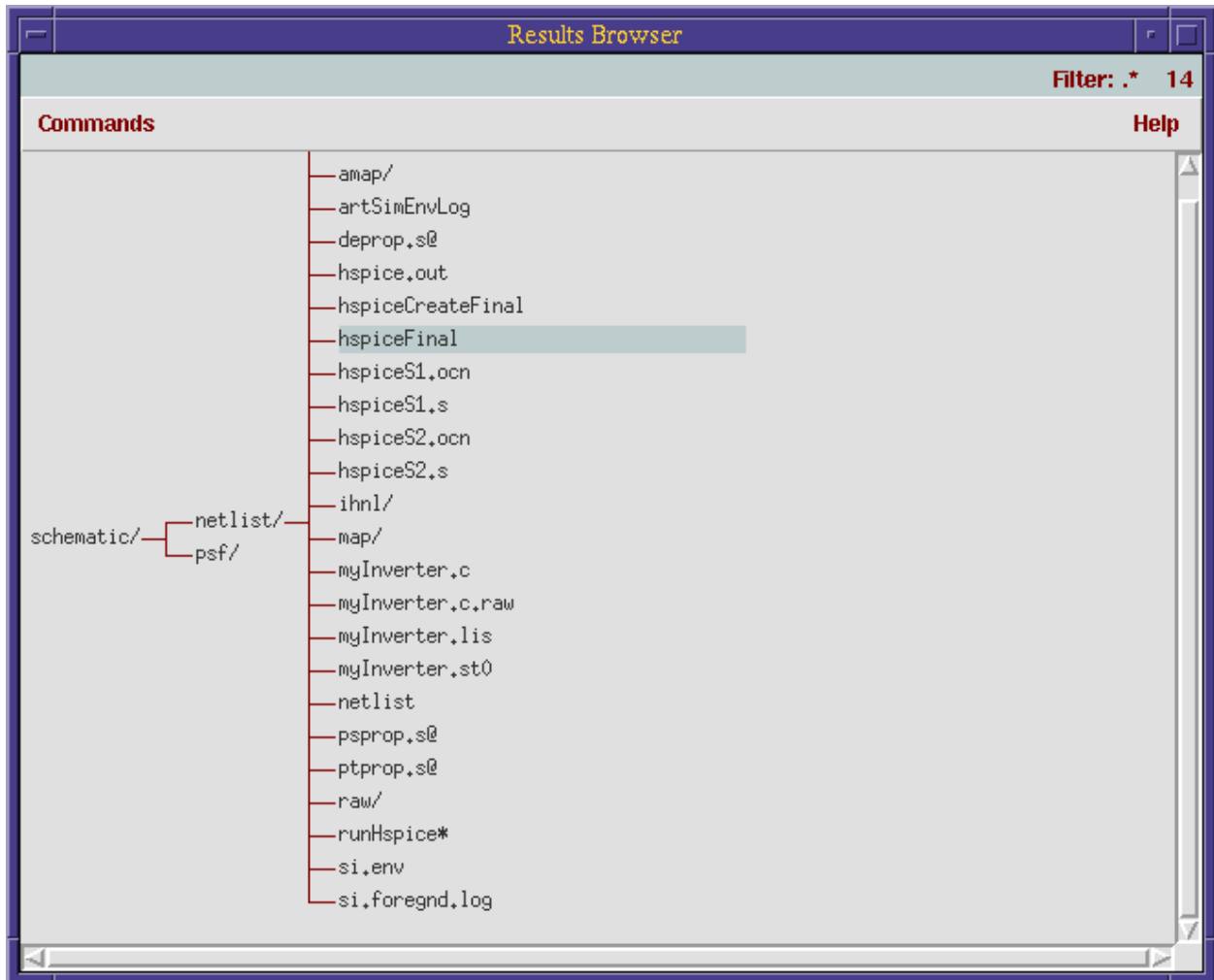
8. Use Results Browser

In **Artist Window** menu, select **Tools -> Result Browser**

In the pop-up window that appears, click **OK**

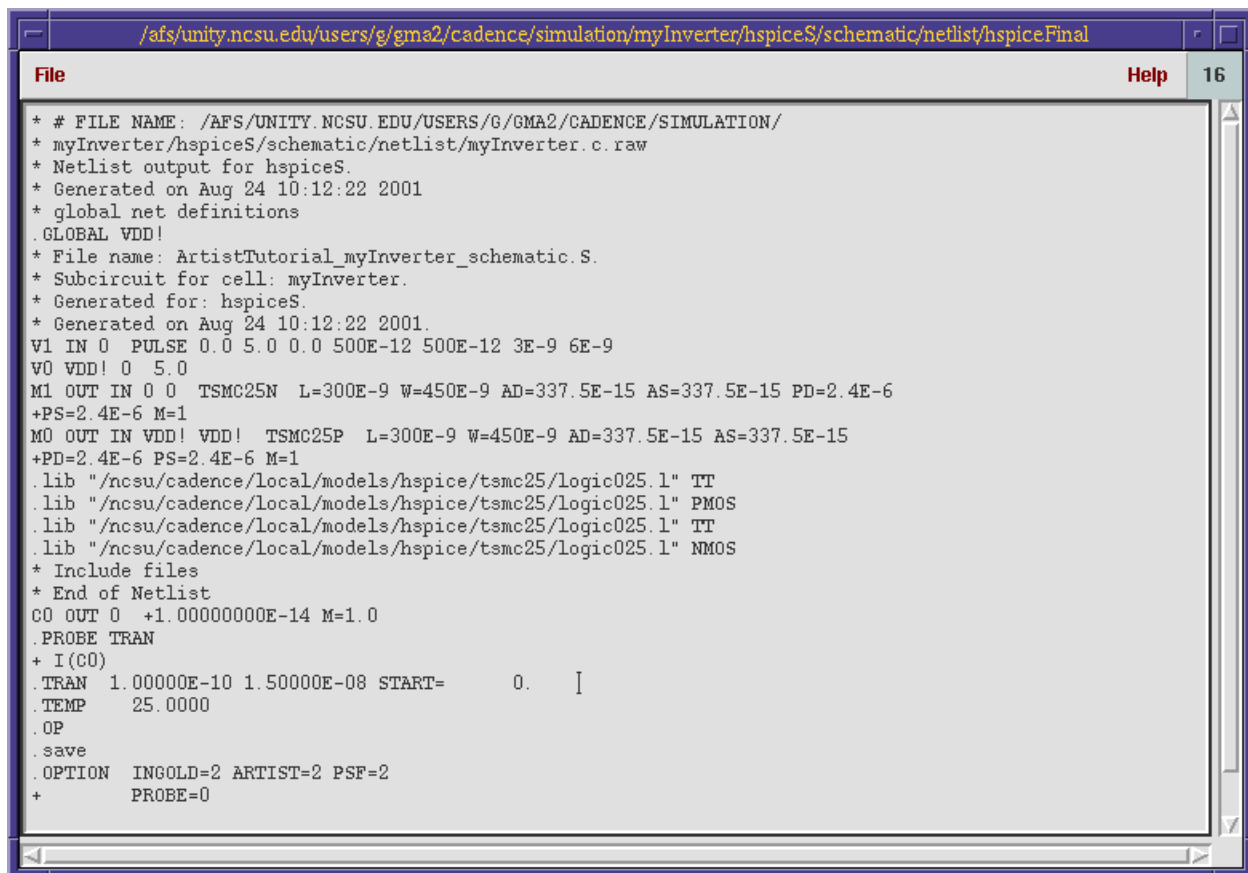


The **Results Browser Window** will then be displayed



In **Results Browser Window**, click **netlist/** and then click **hspiceFinal**

A text window will then show the hspice netlist file for your circuit.

A screenshot of a text editor window showing a netlist file. The window title is "/afs/unity.ncsu.edu/users/g/gma2/cadence/simulation/myInverter/hspiceS/schematic/netlist/hspiceFinal". The text content is a netlist for an inverter circuit simulation. It includes comments about the file name, generation date, and global definitions. The circuit components include a pulse source (V1), a PMOS transistor (M0), and an NMOS transistor (M1). The netlist also includes include files for logic models, include files, and simulation options like CO, TRAN, and OPTION.

```
* # FILE NAME: /AFS/UNITY.NCSU.EDU/USERS/G/GMA2/CADENCE/SIMULATION/
* myInverter/hspiceS/schematic/netlist/myInverter.c.raw
* Netlist output for hspiceS.
* Generated on Aug 24 10:12:22 2001
* global net definitions
.GLOBAL VDD!
* File name: ArtistTutorial_myInverter_schematic.S.
* Subcircuit for cell: myInverter.
* Generated for: hspiceS.
* Generated on Aug 24 10:12:22 2001.
V1 IN 0 PULSE 0.0 5.0 0.0 500E-12 500E-12 3E-9 6E-9
V0 VDD! 0 5.0
M1 OUT IN 0 0 TSMC25N L=300E-9 W=450E-9 AD=337.5E-15 AS=337.5E-15 PD=2.4E-6
+PS=2.4E-6 M=1
M0 OUT IN VDD! VDD! TSMC25P L=300E-9 W=450E-9 AD=337.5E-15 AS=337.5E-15
+PD=2.4E-6 PS=2.4E-6 M=1
.lib "/ncsu/cadence/local/models/hspice/tsmc25/logic025.1" TT
.lib "/ncsu/cadence/local/models/hspice/tsmc25/logic025.1" PMOS
.lib "/ncsu/cadence/local/models/hspice/tsmc25/logic025.1" TT
.lib "/ncsu/cadence/local/models/hspice/tsmc25/logic025.1" NMOS
* Include files
* End of Netlist
CO OUT 0 +1.00000000E-14 M=1.0
.PROBE TRAN
+ I(CO)
.TRAN 1.00000E-10 1.50000E-08 START= 0. ]
.TEMP 25.0000
.OP
.save
.OPTION INGOLD=2 ARTIST=2 PSF=2
+ PROBE=0
```

This is the end of the tutorial. On the last few pages we've added an extra section on using a PWL (piece wise linear) voltage source instead of a pulse source, for your edification. Our thanks go to Guorong Ma (gma2@unity.ncsu.edu) for writing the original version of this tutorial, which we have edited to bring to you. If there are any questions, please email either one of us: ravett@eos.ncsu.edu or mwbaker@eos.ncsu.edu

Thanks,

Ryan and Matt – Everyone's favorite ECE 746 TA's

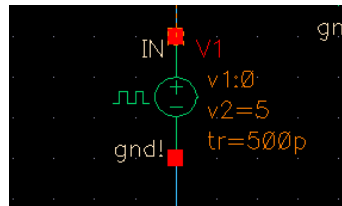
Appendix:

9. Using PWL voltage sources.

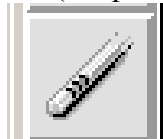
Assuming you have gone through the previous tutorial and have successfully built your inverter, we will now show you how to use a PWL voltage source instead of the pulse source that the tutorial source. A PWL voltage source is a source whose output voltage is dependant on a list of time and voltage pairs that is set by the designer. This list defines what the output voltage should be at the given times, with the voltage being linearly interpolated between these points. This allows a designer to simulate things like digital input streams.

First, we must remove the old pulse source:

In the **Schematic Window**, click on the pulse source object

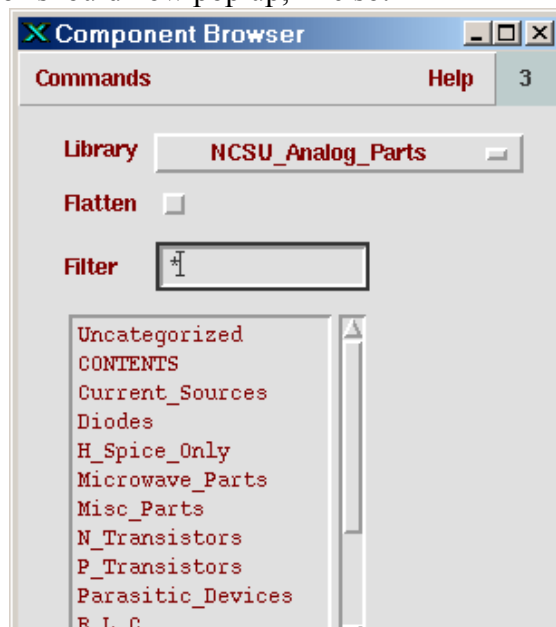


Then click on the delete button to remove it (the pencil eraser icon)



Now from the **Schematic Window** menu, select **Add -> Instance...**

The **Component Browser** should now pop up, like so:



In Component Browser, select **Voltage_Sources** and then **vpwl**. The Add Instance Window should now look something like this:

Add Instance [X]

Hide Cancel Defaults Help

Library: NCSU_Analog_Parts [Browse]

Cell: vpwl

View: symbol

Names: []

Array: Rows: 1 Columns: 1

Rotate Sideways Upside Down

Number of pairs of points: 0

AC magnitude: []

AC phase: []

Time 1: []

Voltage 1: 0 V

Time 2: []

Voltage 2: 0 V

DC voltage: []

Delay time: []

Noise file name: []

Number of noise/freq pairs: 0

DC current: []

Offset voltage: []

Scale factor: []

Time scale factor: []

Period of the PWL: []

Transition width: []

Temperature coefficient 1: []

Temperature coefficient 2: []

Nominal temperature: []

DC source: []

Repeated function: []

While this option list looks very intimidating, we are really only concerned with a few of these many options.

The most important parts of this list are the “**Number of pairs of points**”, which specifies how many (time, voltage) pairs you will be defining for this source, and the “**Time X**” and “**Voltage X**” entries where you actually define the point list.

For this tutorial we will do a simple 10-point list:

In the **Add Instance Window**, fill in the “**Number of pairs of points**” entry box with “**10**”.

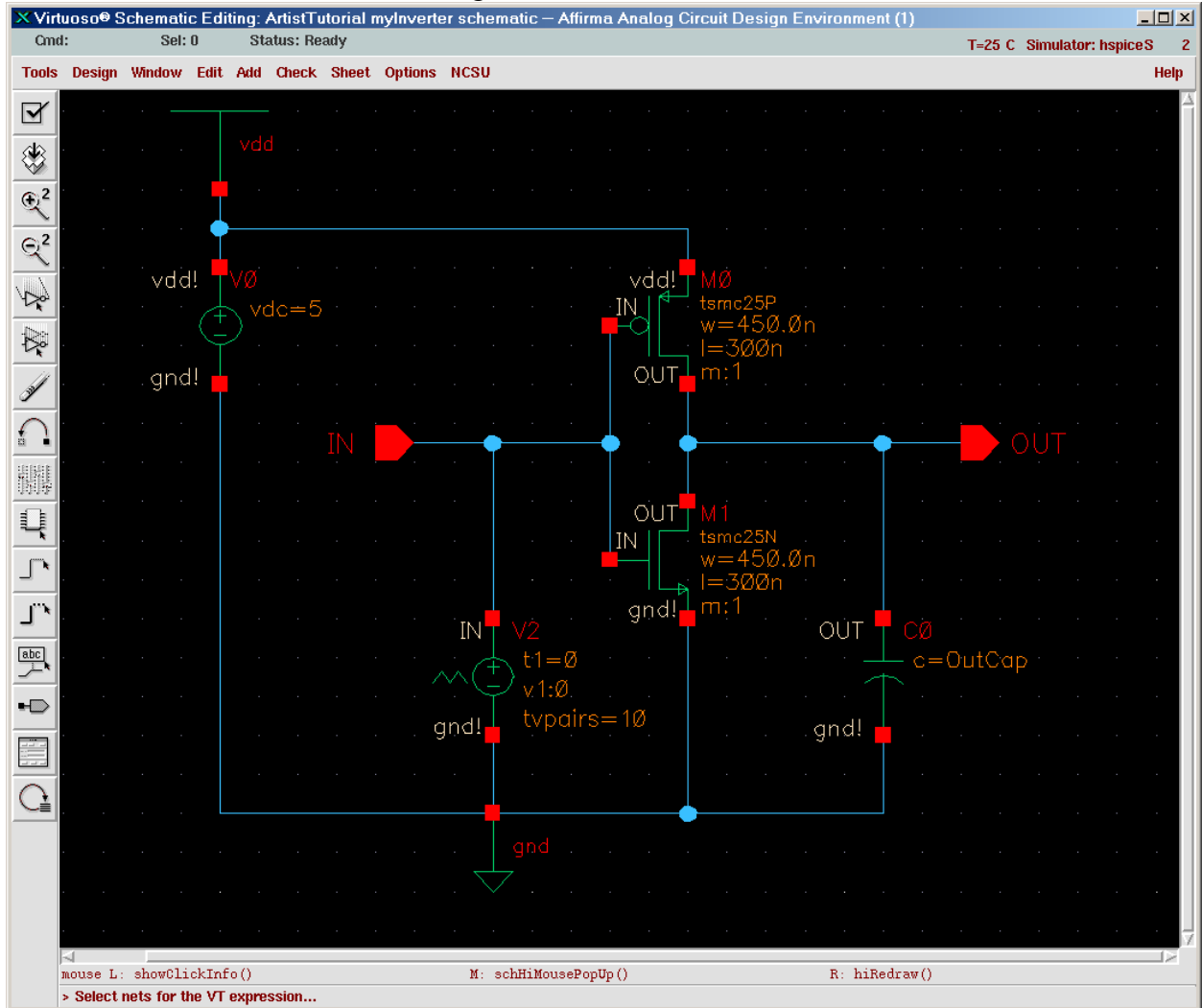
When the **Add Instance Window** finishes updating itself, fill in the time and voltage boxes with the following:

CDF Parameter	Value	Display
Number of pairs of points	10	off
AC magnitude		off
AC phase		off
Time 1	0	off
Voltage 1	0	off
Time 2	5n	off
Voltage 2	0	off
Time 3	5.5n	off
Voltage 3	5	off
Time 4	10n	off
Voltage 4	5	off
Time 5	10.5n	off
Voltage 5	0	off
Time 6	20n	off
Voltage 6	0	off
Time 7	20.5n	off
Voltage 7	5	off
Time 8	25n	off
Voltage 8	5	off
Time 9	25.5n	off
Voltage 9	0	off
Time 10	30n	off
Voltage 10	0	off

In the **Schematic Window**, add the source instance in the vacant spot left from where we removed the pulse source.

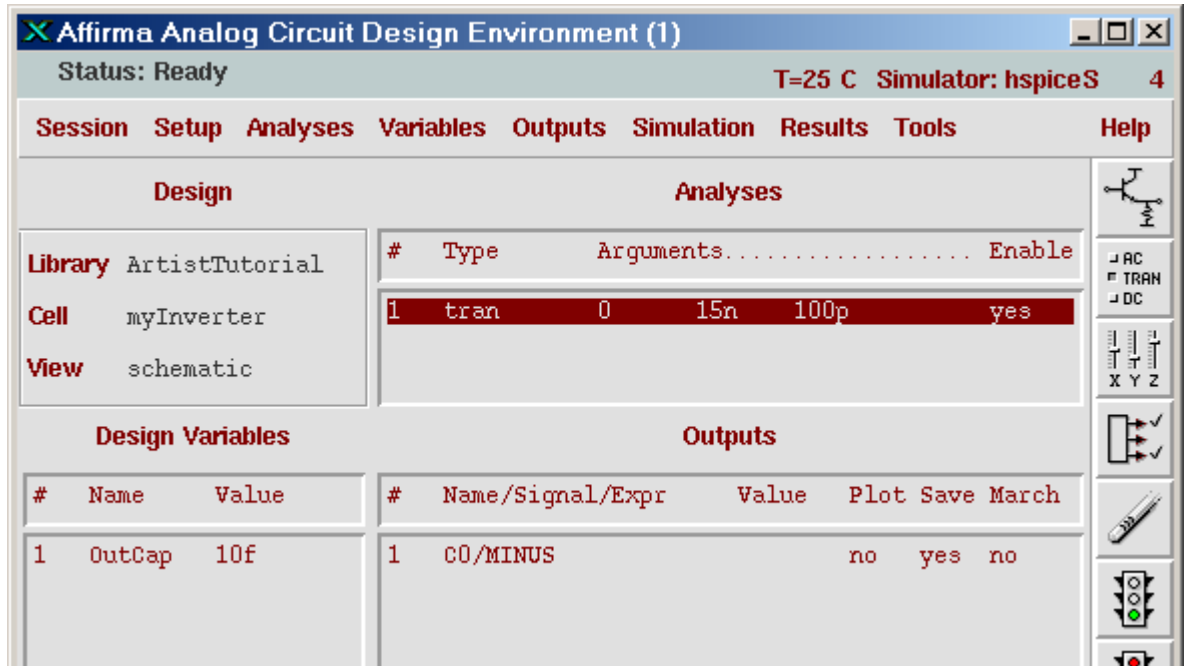
In the **Schematic Window**, click the **Check and Save** button, and fix any errors it highlights.

Your schematic should look something like this:

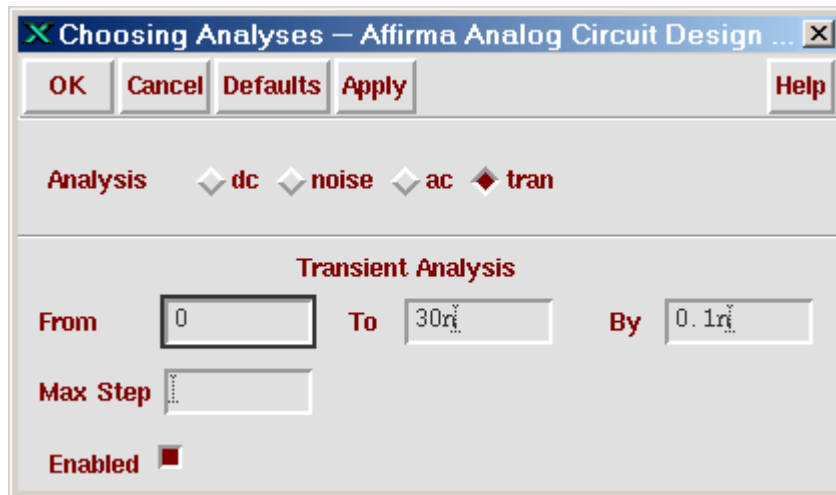


Now we need to test our new circuit to see the PWL source in action.

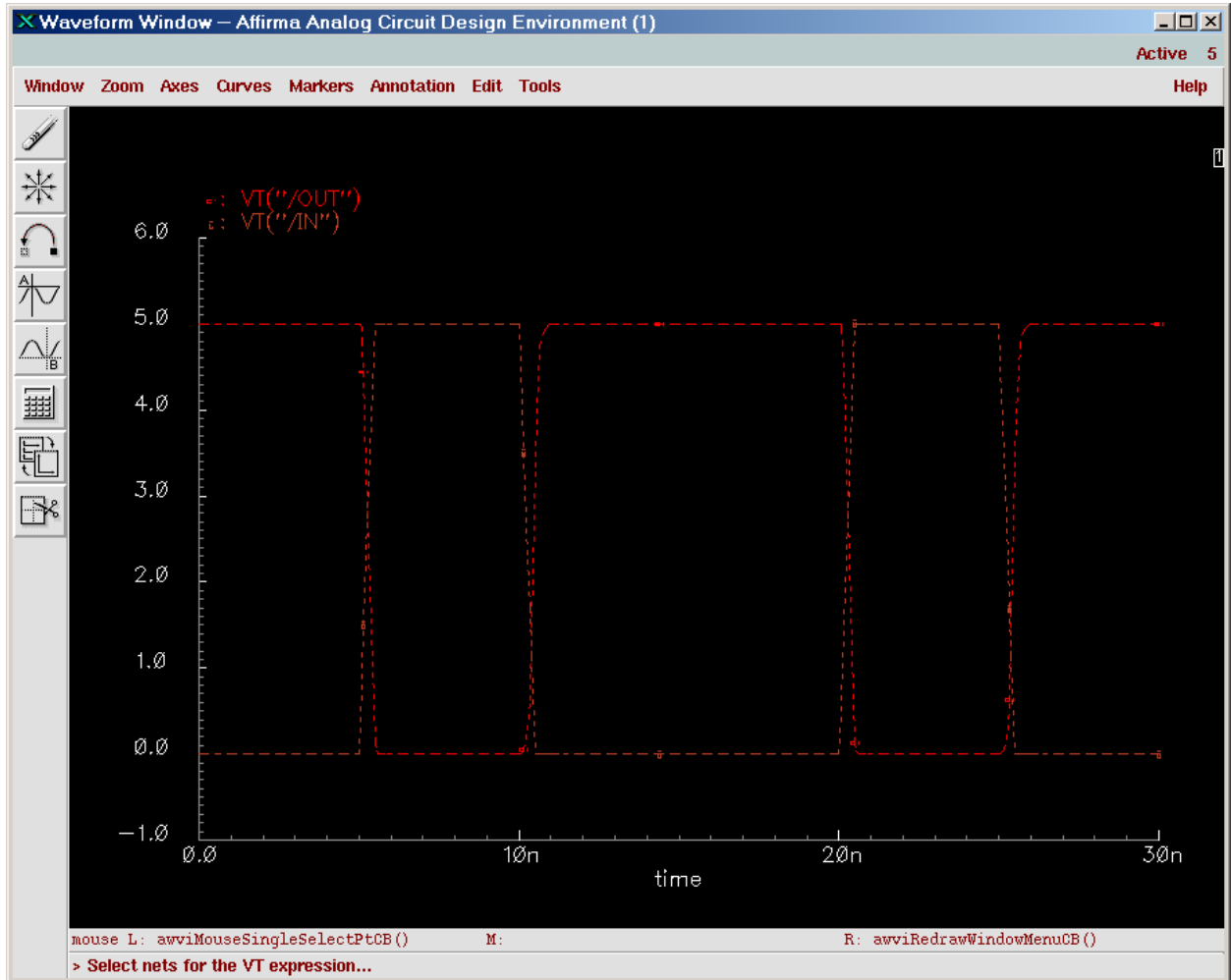
In **Analog Artist Window**, double-click on the entry highlighted below, which we set during the tutorial.



In the window that pops up, change the transient analysis so that it runs for 30 ns, like shown here:



Now, we can run the simulation just like we did in the tutorial. The plot in the **Waveform Window** should look like this:



Congratulations! You've just used a PWL voltage source.