Introduction to PSPICE

To install OrCAD PSpice Release 16.5 Lite Edition

1. Go to http://www.cadence.com/products/orcad/pages/downloads.aspx#pspice and click on the "Download FREE – OrCAD 16.5 Demo software" under the "Capture and PSpice only" section. You will have to fill out a short form and then will receive an email with download instructions.
2. Save the .exe file somewhere convenient and double click the executable and follow the steps in the installation wizard.

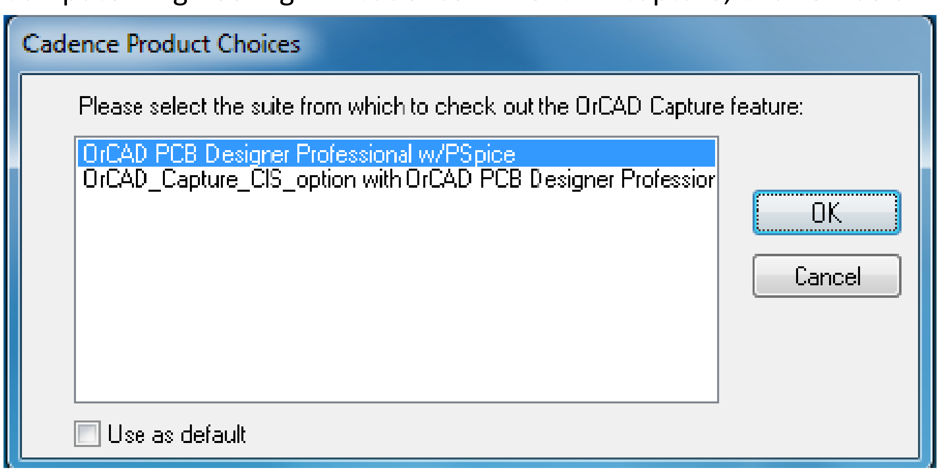
The steps to simulation

In any basic circuit simulation, there are six steps to simulating a circuit:

1. Create a simulation project
2. Draw the schematic to simulate
3. Establish a simulation profile
4. Set up a simulation type
5. Simulate the circuit
6. Analyze the results in Probe

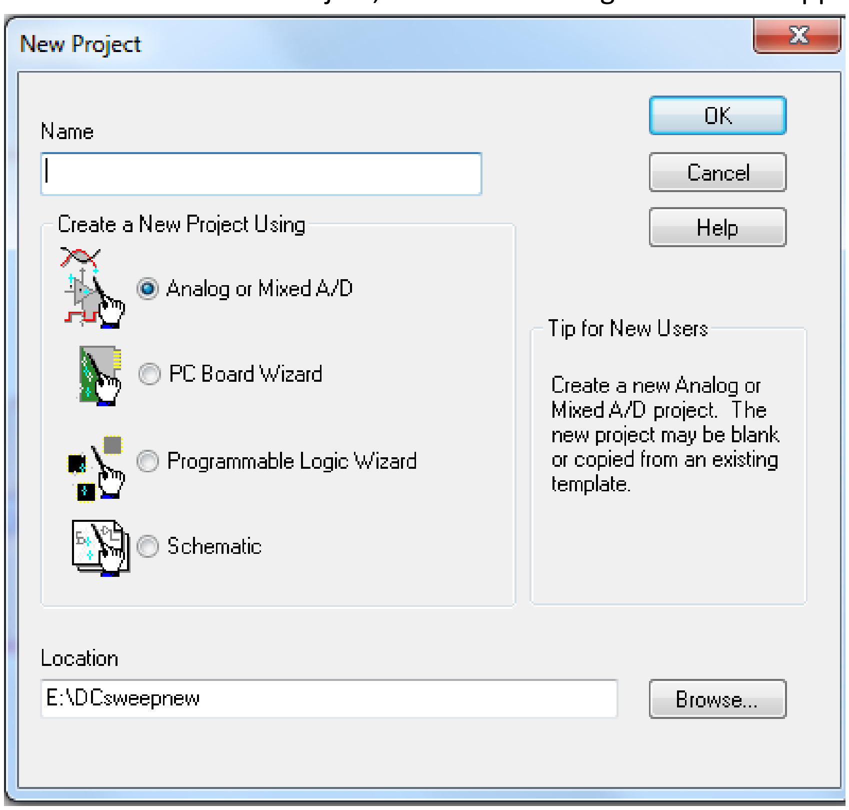
To start a simulation session

1. On the start menu select Engineering Software ---> Electrical Electronic and Computer Engineering ---> Cadence ---> OrCAD Capture, Click OK below.

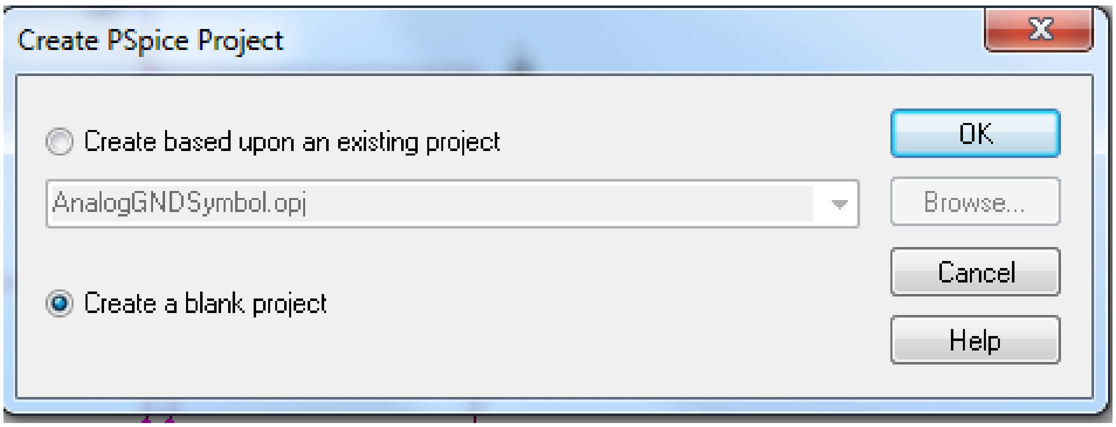


2. The window is shown below:

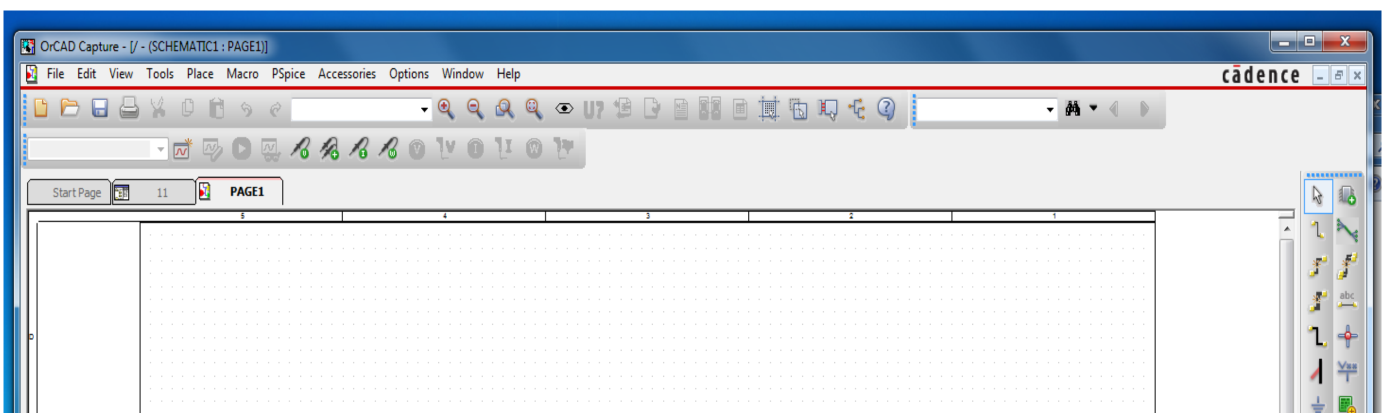
3. Click File---> New---> Project, and the following window will appear:



4. Give the project a descriptive name (e.g. CMOS_inverter). Select Analog or Mixed A/D. You will not be able to simulate your project if the schematic button is selected.
5. Specify a location where the project is to be stored. Click Browse and specify the location where the files are to be stored, e. g., your own drive or usb but not C: drive.
6. Click OK. The window below appears. Select Create a blank project and click OK.



7. The following window appears:

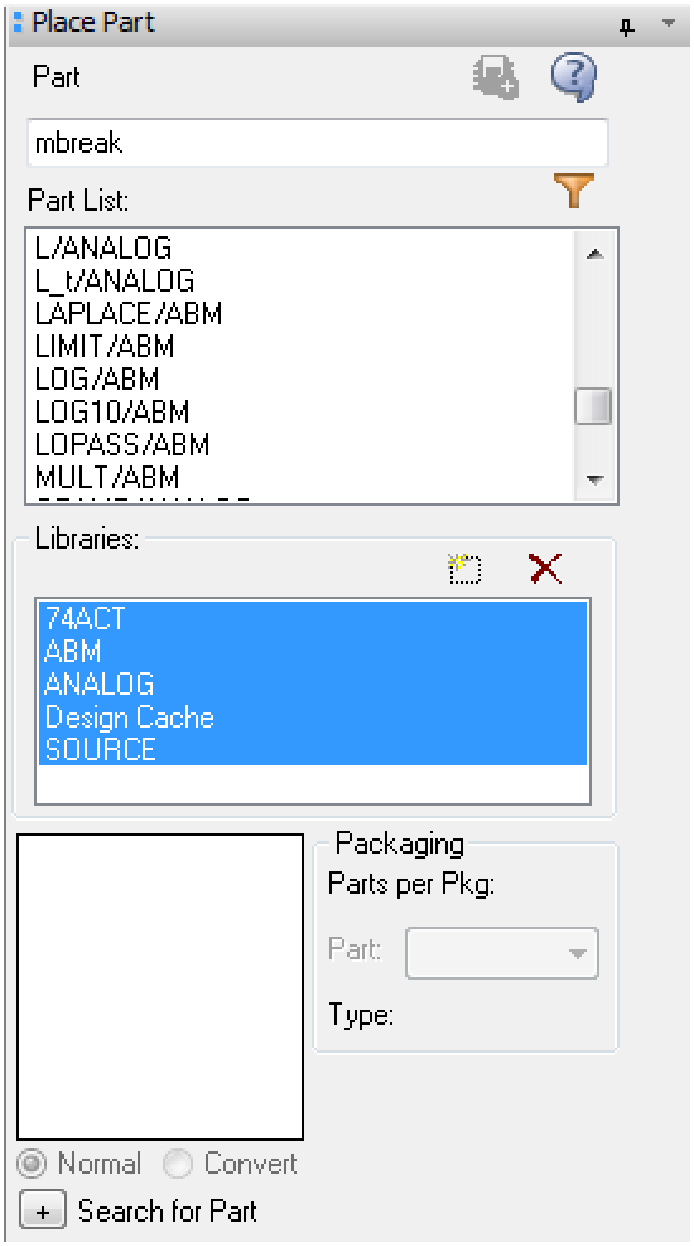


Across the top are tabs to select the Start Page, the Project Page and the Schematic Page. Toolbars are across the top and down the right side. These toolbars and menus will change depending on which window is selected. Be sure the Schematic Page Tab (likely named Page1) is selected.

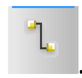
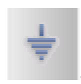
You are now ready to begin creating a circuit for simulation.

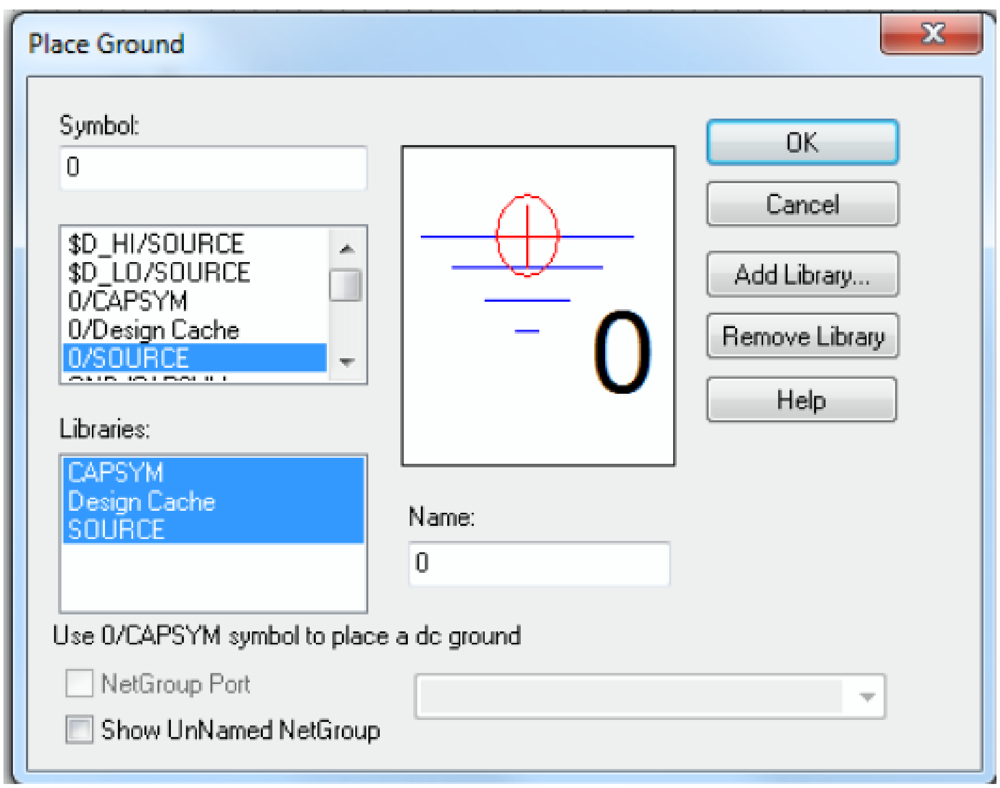
To create a schematic for a CMOS inverter simulation

1. Click on Schematic Page Tab. If the Schematic Tab is not visible, go to DESIGN RESOURCES---> .\filename.DSN ---> SCHEMATIC ---> PAGE 1 in the Project Tab and double click.
2. To place parts, click the Place Part Icon  in the upper right tool palette.
3. The Place Part side panel shows that:



If you do not see all of the libraries listed, click the Add Library icon  and select all of the libraries in the PSpice folder. Parts from other locations will not simulate properly. In this lab, you need libraries of BREAKOUT and SOURCE.

4. Select the part you wish to place in the schematic and press Enter. You can quickly find the part by starting to type the name in the part window. In this lab, you need MbreakN, MbreakP, VDC and VPULSE.
5. Select the part and use Right Click to rotate and mirror.
6. Insert as many as needed. You can also select a group of parts and CTRL-drag to copy those parts.
7. Hit ESC or Right Click and select END MODE to stop inserting parts.
8. To wire parts together, click the Place Wire icon .
   Place your cursor over the boxes at the ends of the parts and draw wires connecting parts. When done, hit ESC or right click and select END WIRE.
9. To insert a ground node, click the Ground Icon .



10. To change component values that are displayed
    Double click the displayed value.
    Change the desired value in the dialog box that appears.

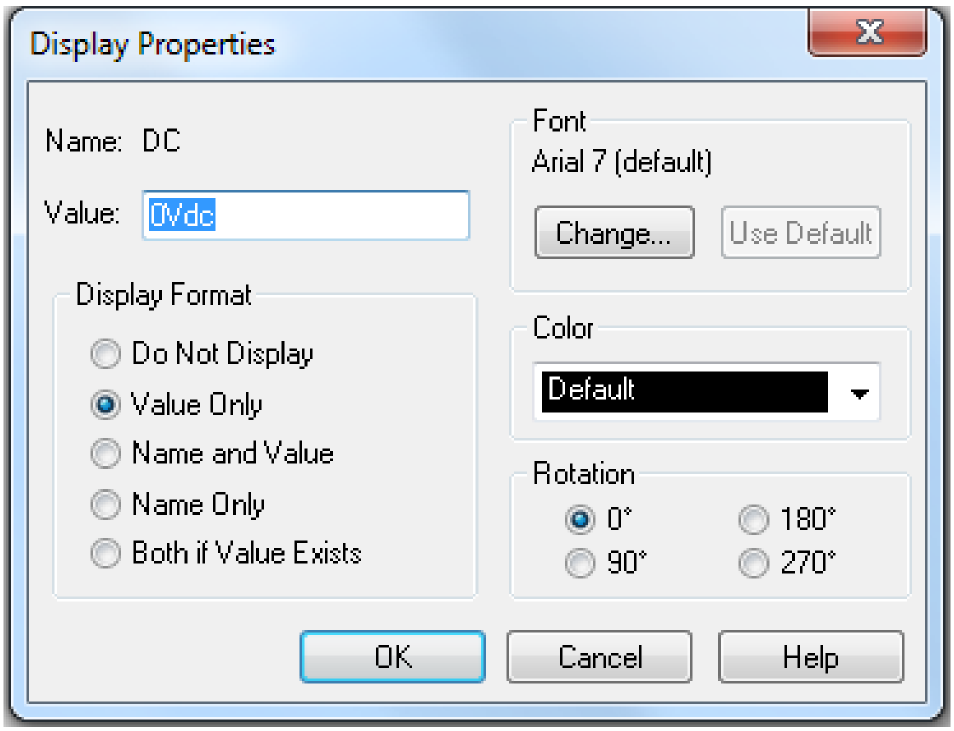
11. To change component values that are not displayed

    Select the part (a box is drawn around the entire part when it is selected). Double click the part or Right click and select EDIT PROPERTIE.

    A Property Editor database screen appears in a new tab. Scroll right until the desired value is seen on the top line. Click the property (second line) you wish to change.

    Type in the new value.

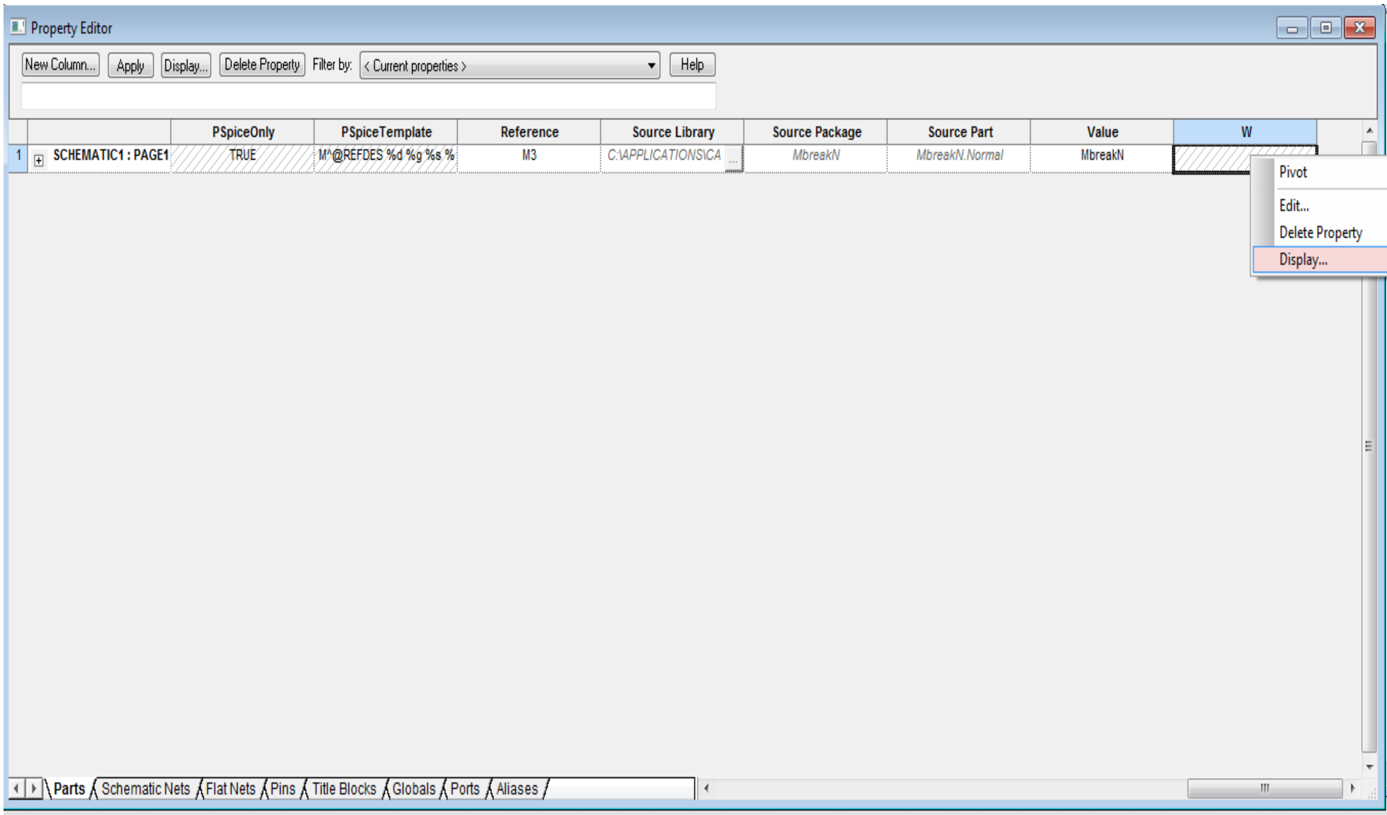
    Right click on the tab and select CLOSE to close the tab.

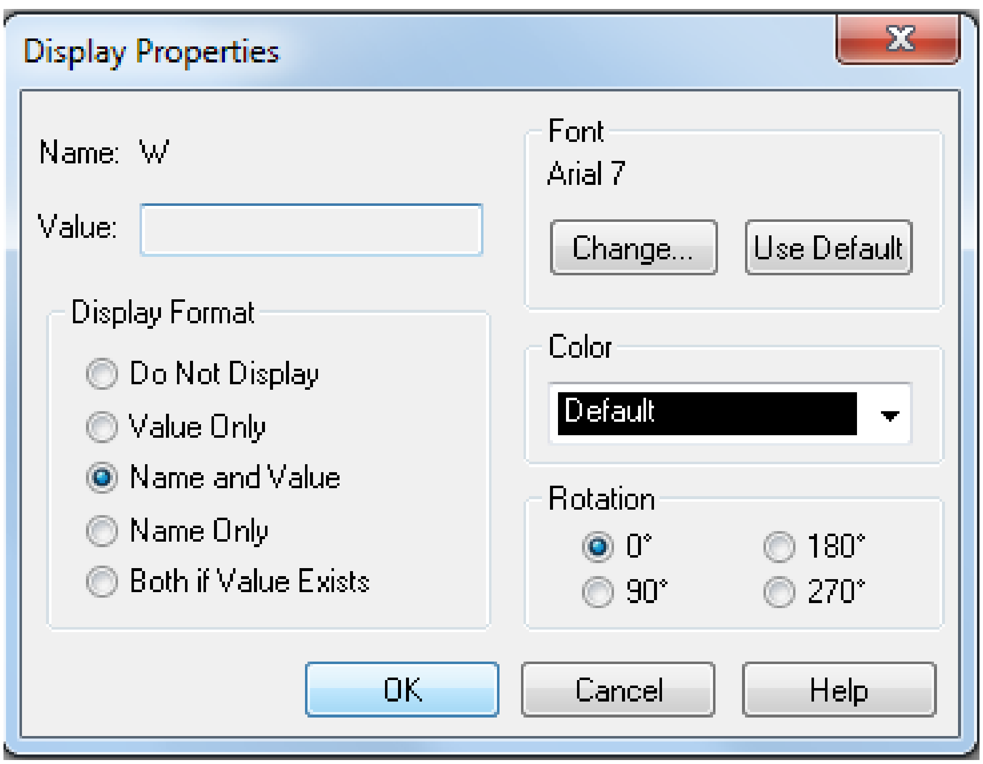


12. To change model values

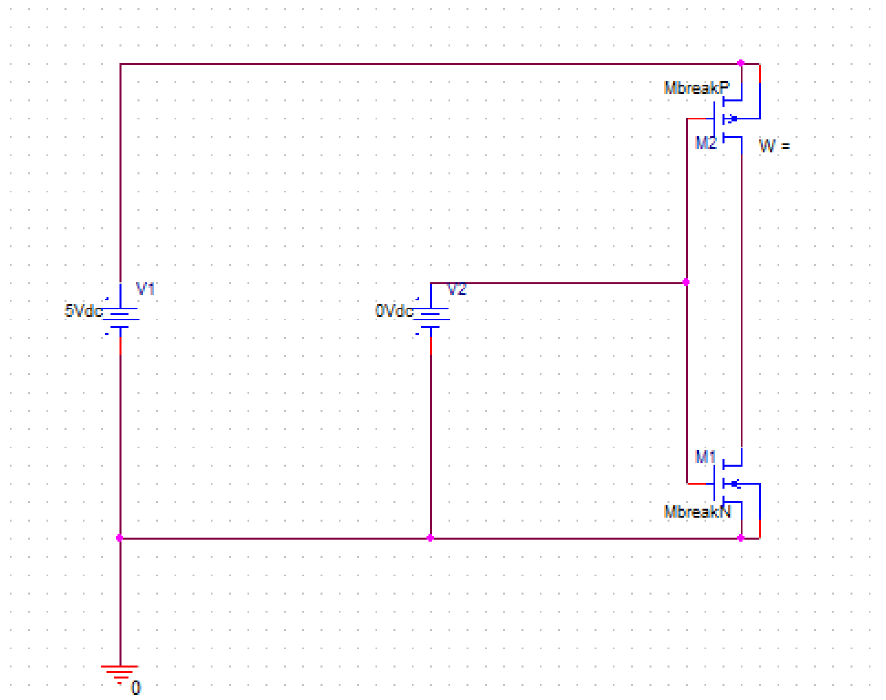
    1) Select model and double click, you will see the following,

    Right click the following area, and click Display in the dialog box that appears. The parameter W can be modified (Note that you can only edit the parameters that can be seen from property editor).


Property Editor
New Column...
Apply
Display...
Delete Property
Filter by:
< Current properties >
Help
PSpiceOnly
PSpiceTemplate
Reference
Source Library
Source Package
Source Part
Value
W
SCHEMATIC1 : PAGE1
TRUE
M^@REFDES %d %g %s %
M3
C:\APPLICATIONS\CA
MbreakN
MbreakN.Normal
MbreakN
Pivot
Edit...
Delete Property
Display...
Parts
Schematic Nets
Flat Nets
Pins
Title Blocks
Globals
Ports
Aliases


Display Properties
Name: W
Value:
Font
Arial 7
Change...
Use Default
Display Format
Do Not Display
Value Only
Name and Value
Name Only
Both if Value Exists
Color
Default
Rotation
0°
180°
90°
270°
OK
Cancel
Help

Select Name and Value, click OK. You will see



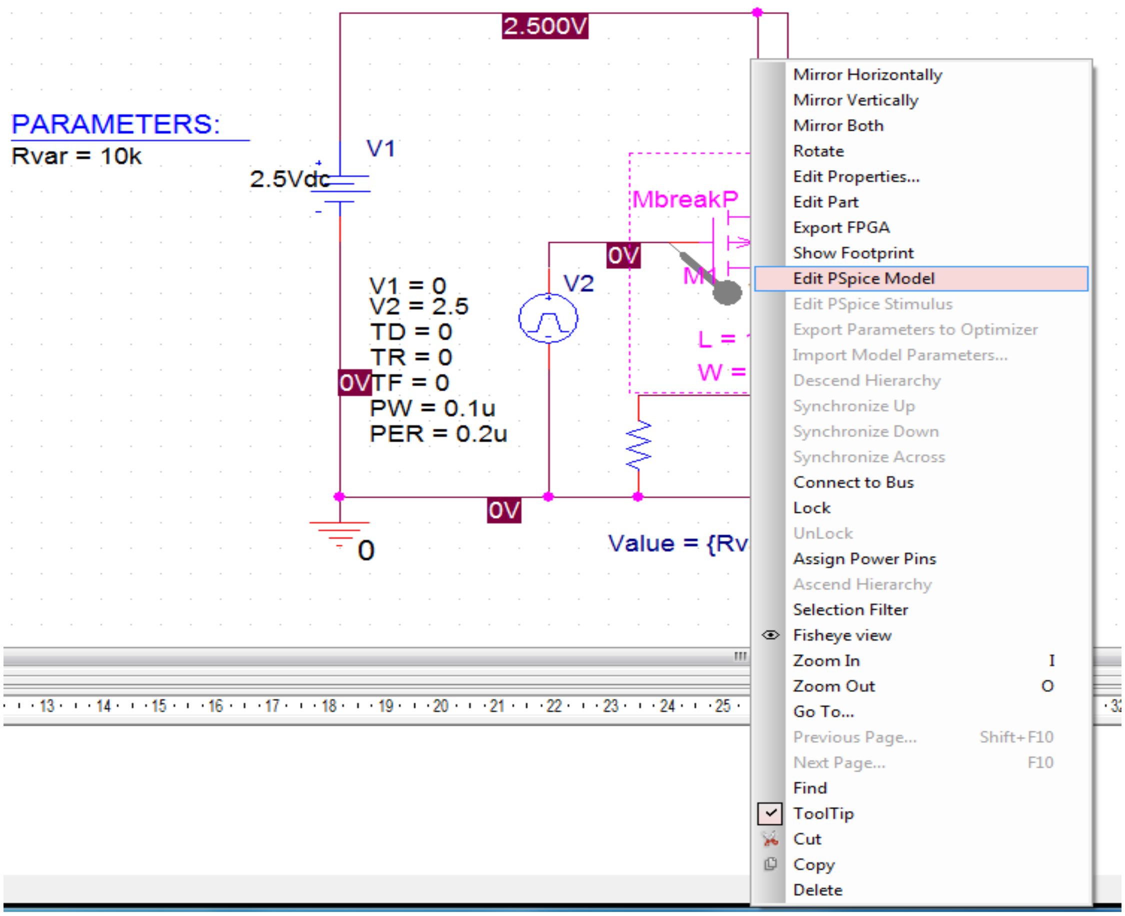
You can now double click "W=" in the schematic, and edit the value you want.

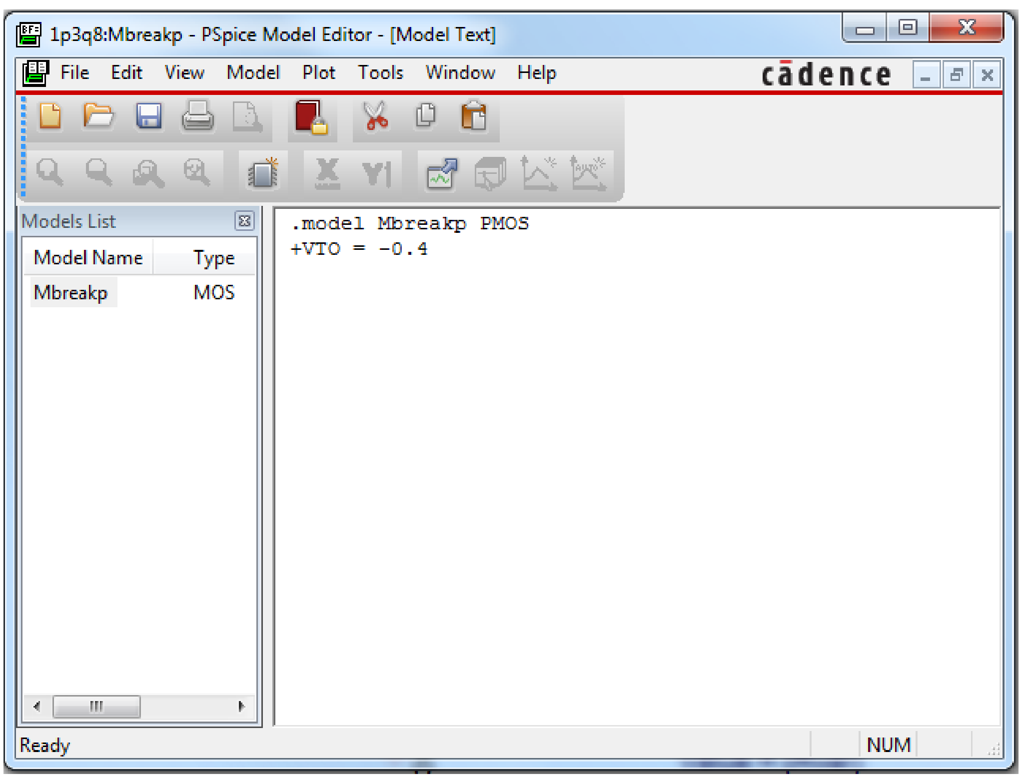
2) When you want to edit parameters that cannot be seen from property editor (e.g., V_{TO}), you can do it as follows:

   First, Select Mbreakp and Right click it, select "Edit PSpice Model" (see the following figure).

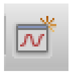
   Then, Input "+VTO = -0.4" (for Mbreakp) in PSpice Model Editor (see the following figure).

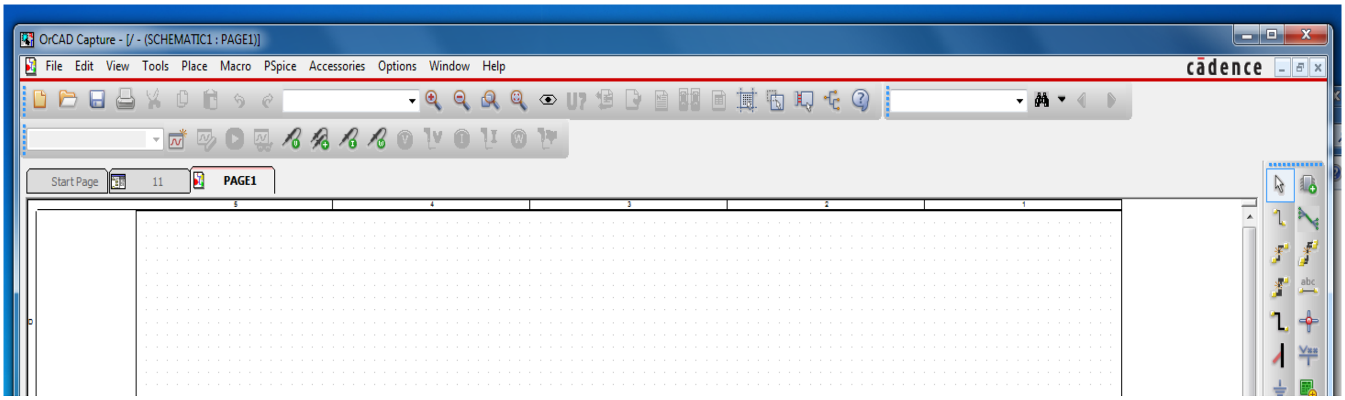
   At last, save it and Close.


2.500V
PARAMETERS:
Rvar = 10k
V1
2.5Vdc
MbreakP
0V
V1 = 0
V2 = 2.5
TD = 0
TR = 0
TF = 0
PW = 0.1u
PER = 0.2u
V2
0V
0
Value = {Rv
Mirror Horizontally
Mirror Vertically
Mirror Both
Rotate
Edit Properties...
Edit Part
Export FPGA
Show Footprint
Edit PSpice Model
Edit PSpice Stimulus
Export Parameters to Optimizer
Import Model Parameters...
Descend Hierarchy
Synchronize Up
Synchronize Down
Synchronize Across
Connect to Bus
Lock
UnLock
Assign Power Pins
Ascend Hierarchy
Selection Filter
Fisheye view
Zoom In I
Zoom Out O
Go To...
Previous Page... Shift+F10
Next Page... F10
Find
ToolTip
Cut
Copy
Delete


1p3q8:Mbreakp - PSpice Model Editor - [Model Text]
File Edit View Model Plot Tools Window Help
cādence
Models List
Model Name Type
Mbreakp MOS
.model Mbreakp PMOS
+VTO = -0.4
Ready
NUM

To set up a simulation profile

1. Select PSPICE → NEW SIMULATION PROFILE or click the new simulation profile icon  on the toolbar.

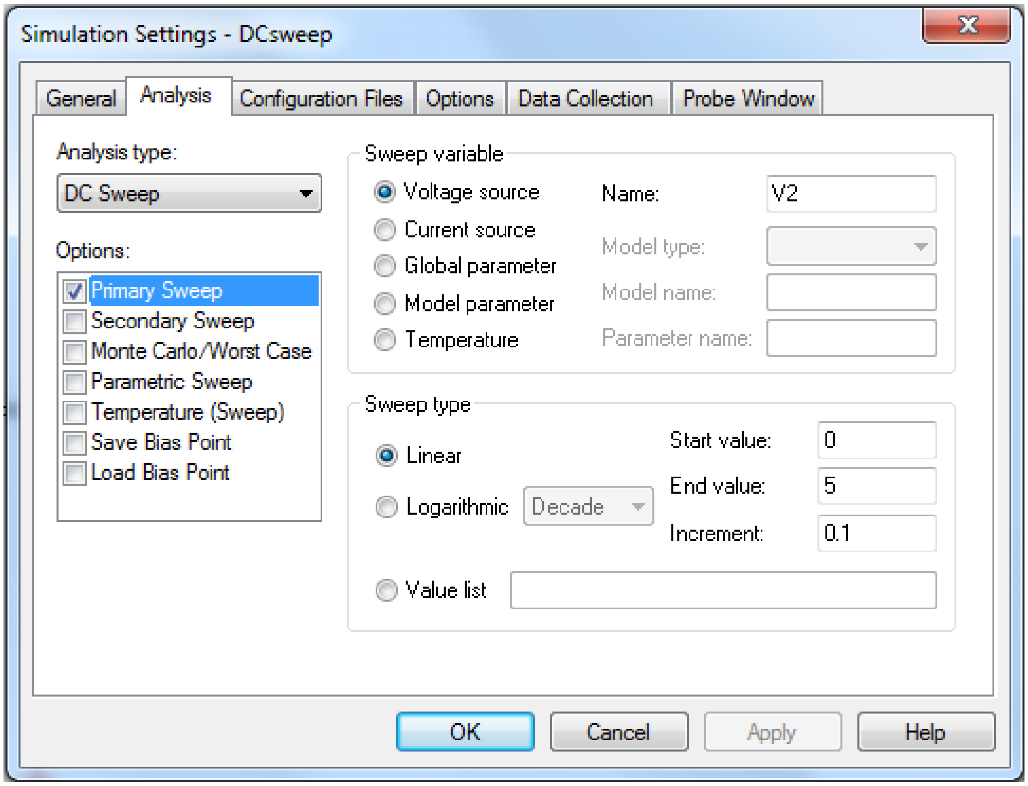


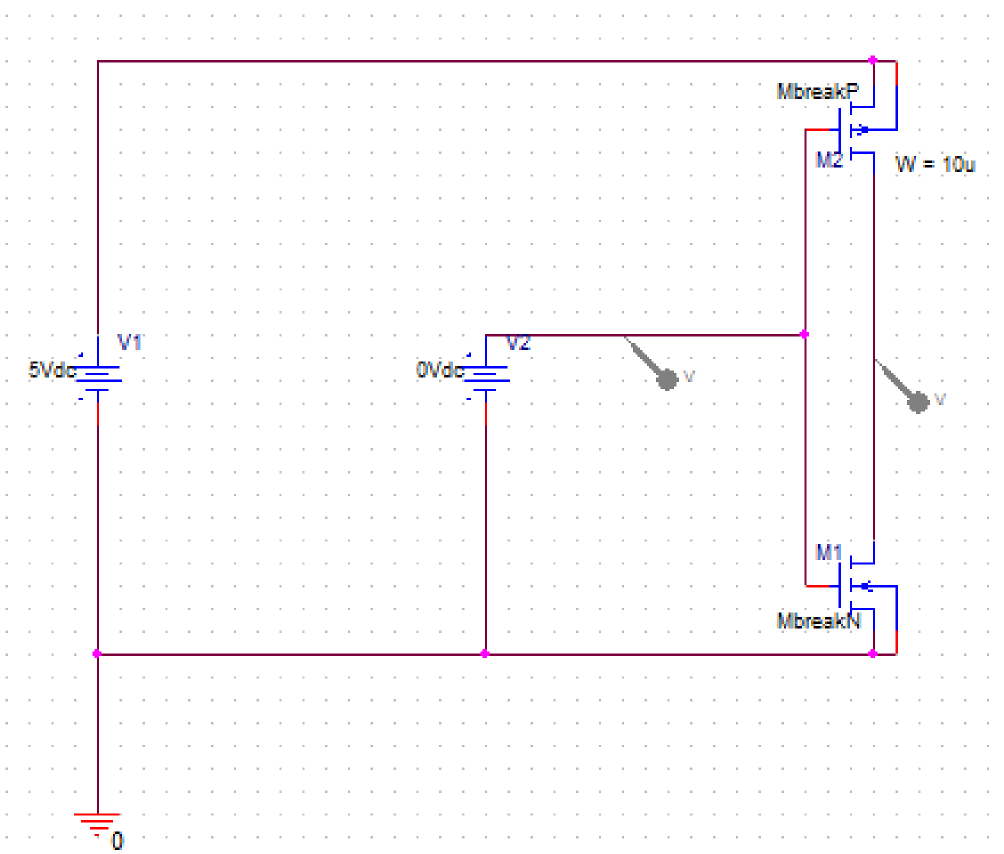
2. Give a descriptive name to the type of simulation (e.g. DC sweep, transient, etc…).
3. Click CREATE and the simulation settings menu will appear:
4. Select the ANALYSIS TYPE from the pull down menu.
5. Set the parameters.
6. Click OK.

Simple DC Sweep (Sweeping sources only)

Use a simple DC sweep when you want to plot a voltage, current, or power as a function of another DC voltage or current. The sources used in this type of simulation will typically be VDC and IDC sources.

1. Create a new simulation profile called “dc sweep” (or whatever you want).
2. Under ANALYSIS Tab, select an ANALYSIS TYPE of DC SWEEP.
3. Be sure PRIMARY SWEEP is checked.
4. In the SWEEP VARIABLE section, select the radio button for voltage source or current source and enter the source name (such as V1, I3, etc…) to be varied.
5. Enter the appropriate variables in the SWEEP TYPE Section.
6. Click OK.
7. Determine the variables of interest to be plotted in Probe.
8. For example, you can try to simulate the output voltage of the CMOS inverter shown below. Do not forget to put voltage Probe.

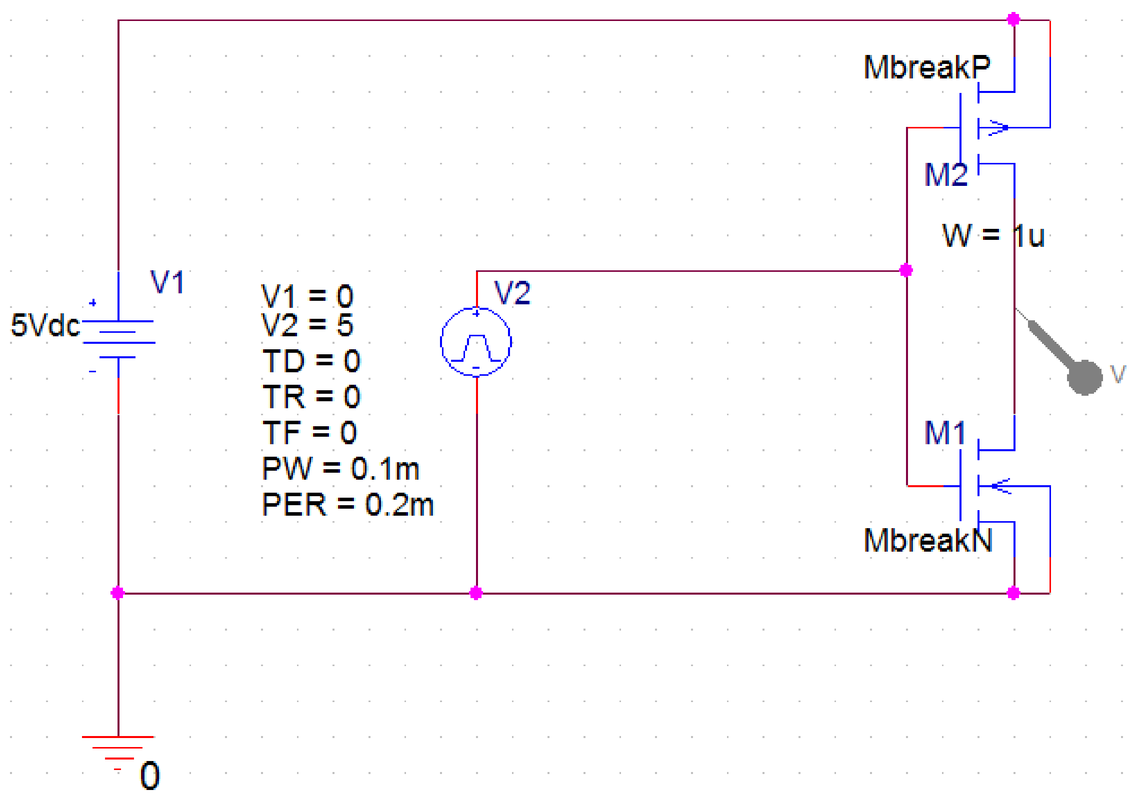

Simulation Settings - DCsweep
General
Analysis
Configuration Files
Options
Data Collection
Probe Window
Analysis type:
DC Sweep
Options:
Primary Sweep
Secondary Sweep
Monte Carlo/Worst Case
Parametric Sweep
Temperature (Sweep)
Save Bias Point
Load Bias Point
Sweep variable
Voltage source
Current source
Global parameter
Model parameter
Temperature
Name:
V2
Model type:
Model name:
Parameter name:
Sweep type
Linear
Logarithmic
Decade
Start value:
0
End value:
5
Increment:
0.1
Value list
OK
Cancel
Apply
Help

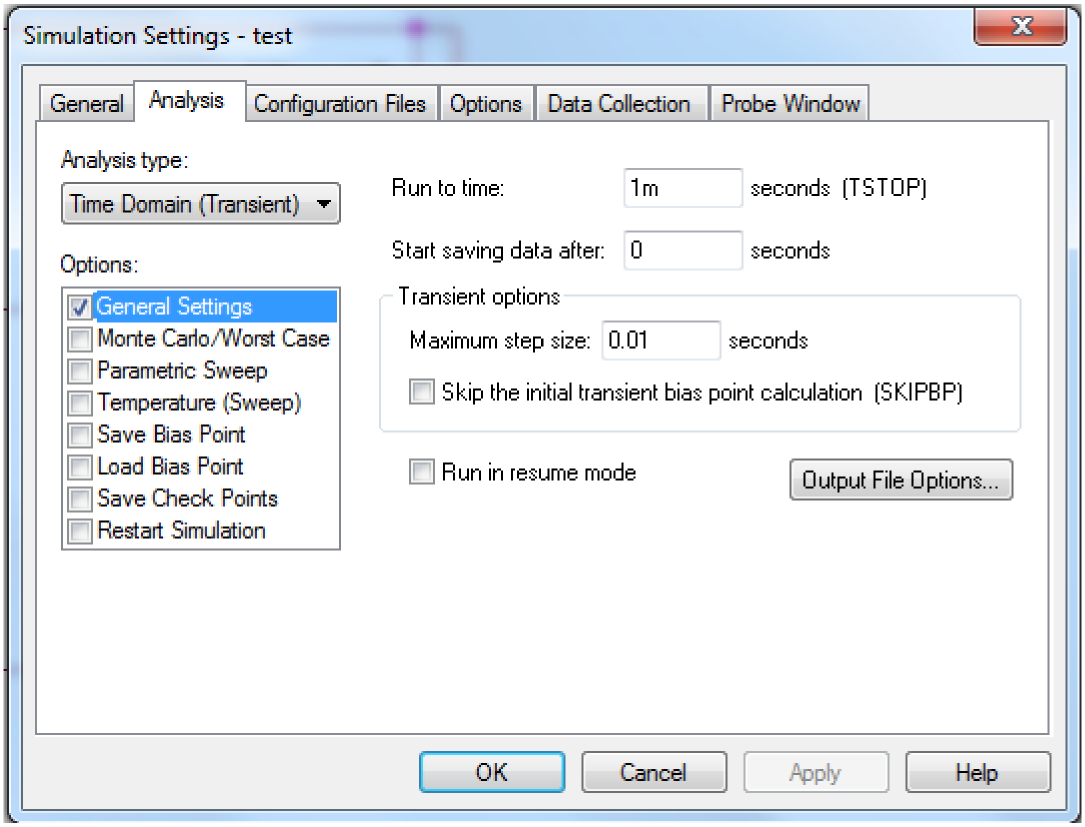

MbreakP
M2
W = 10u
V1
5Vdc
V2
0Vdc
V
V
M1
MbreakN
0

Transient Analysis

Use a transient analysis when you want to plot a voltage, current, or power as a function of time. The sources used in this type of analysis will be any voltage or current source that varies with time; examples include VSIN, VPULSE, ISIN, and VRAMP.

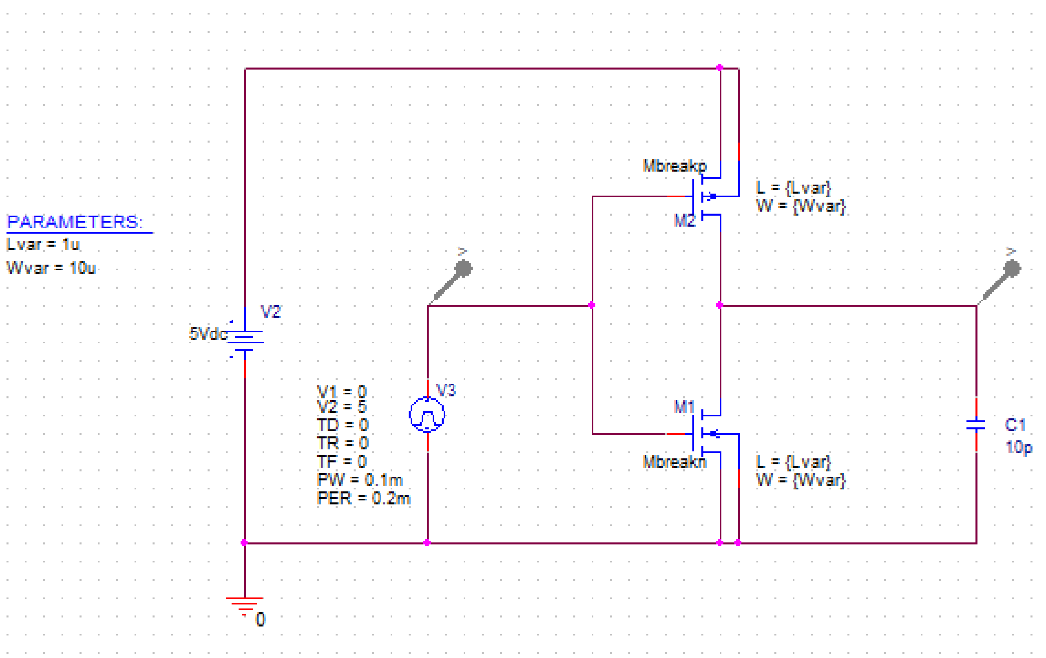
1. Create a new simulation profile called "transient" (or whatever you want).
2. Under ANALYSIS Tab, select an ANALYSIS TYPE of TIME DOMAIN (TRANSIENT).
3. Enter the stop time (TSTOP) for the simulation in the RUN TO TIME box.
4. Enter the maximum step size in the TRANSIENT OPTIONS section – making this variable ~TSTOP/1000, will help ensure that the traces in Probe are relatively smooth.
5. Click OK.
6. Determine the variables of interest to be plotted in Probe.
7. Simulate the circuit below.



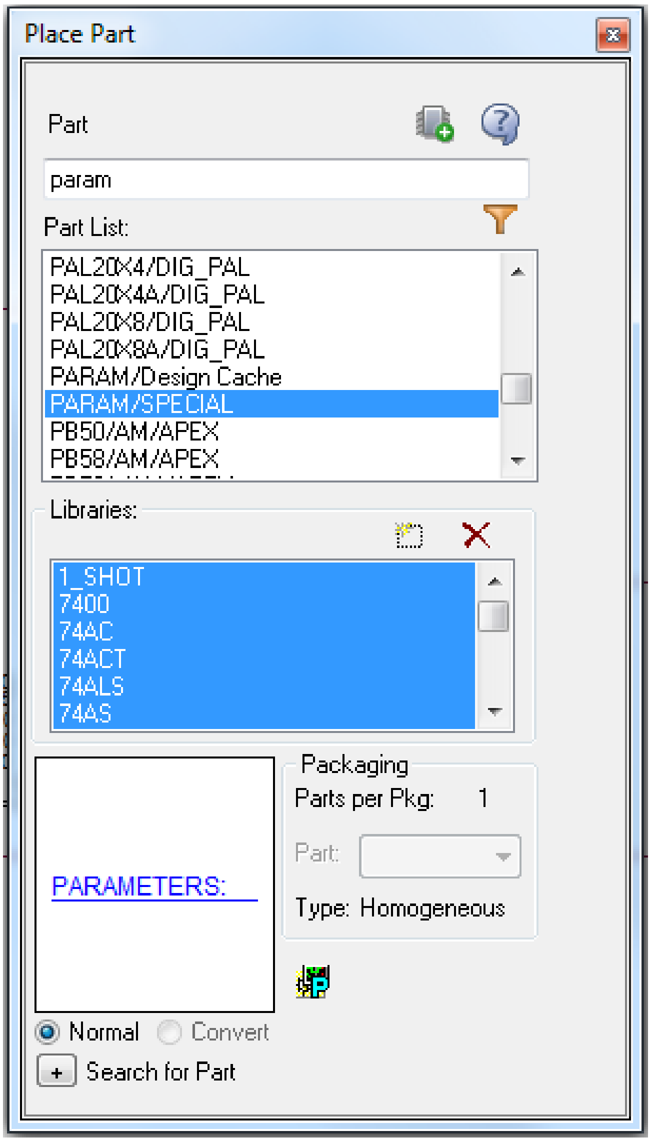


Parametric Simulation

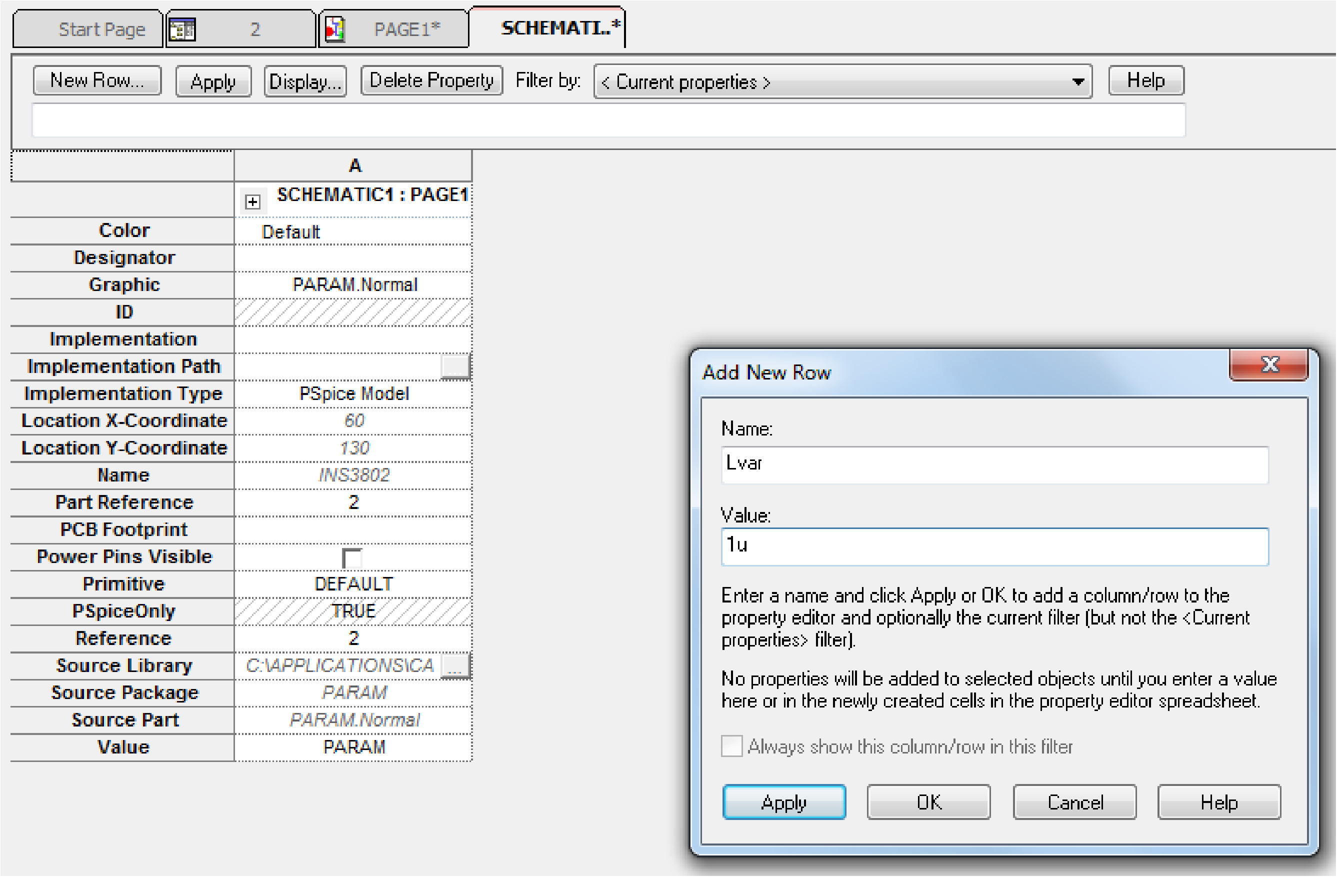
1. Use this type of analysis when you want to plot a voltage, current, or power as some component or model parameter varies, such as L or W of Mbreakp.



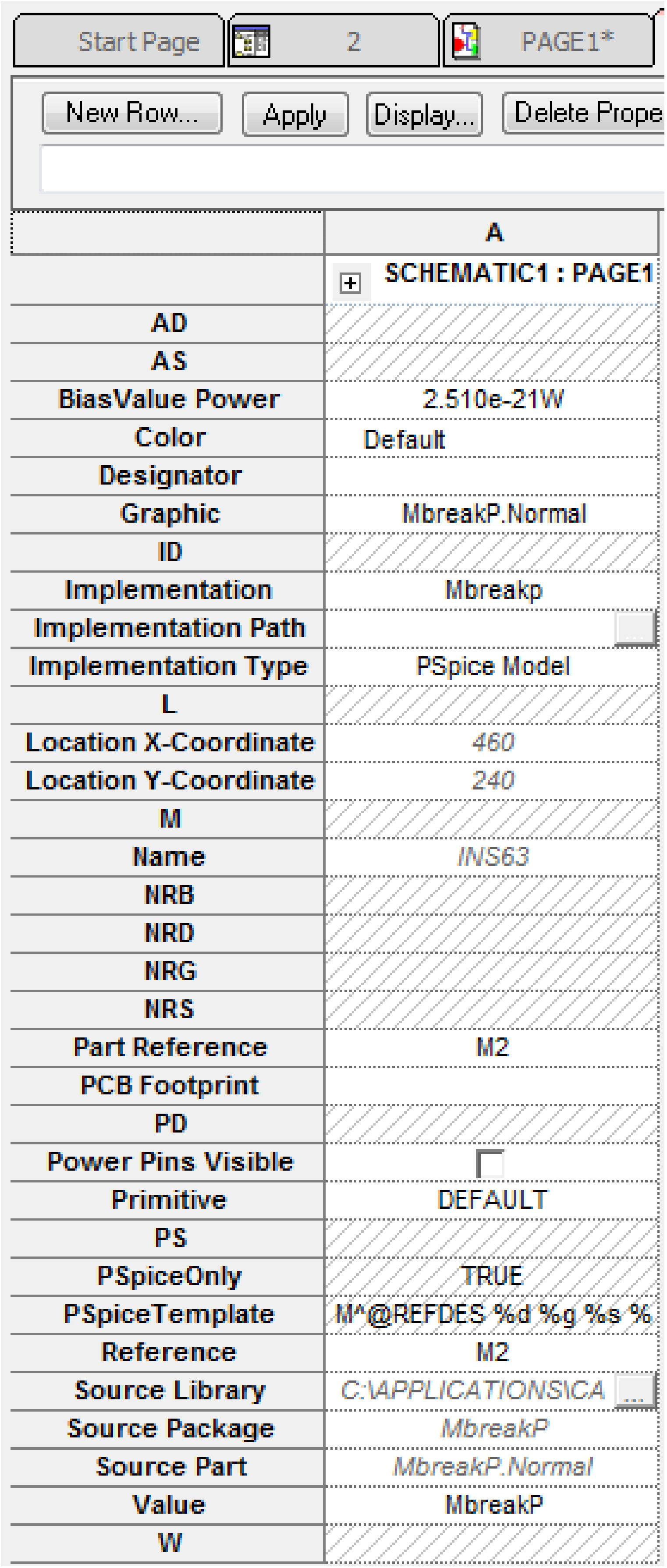
2. Place the PARAM part (located in the SPECIAL library) somewhere on the schematic



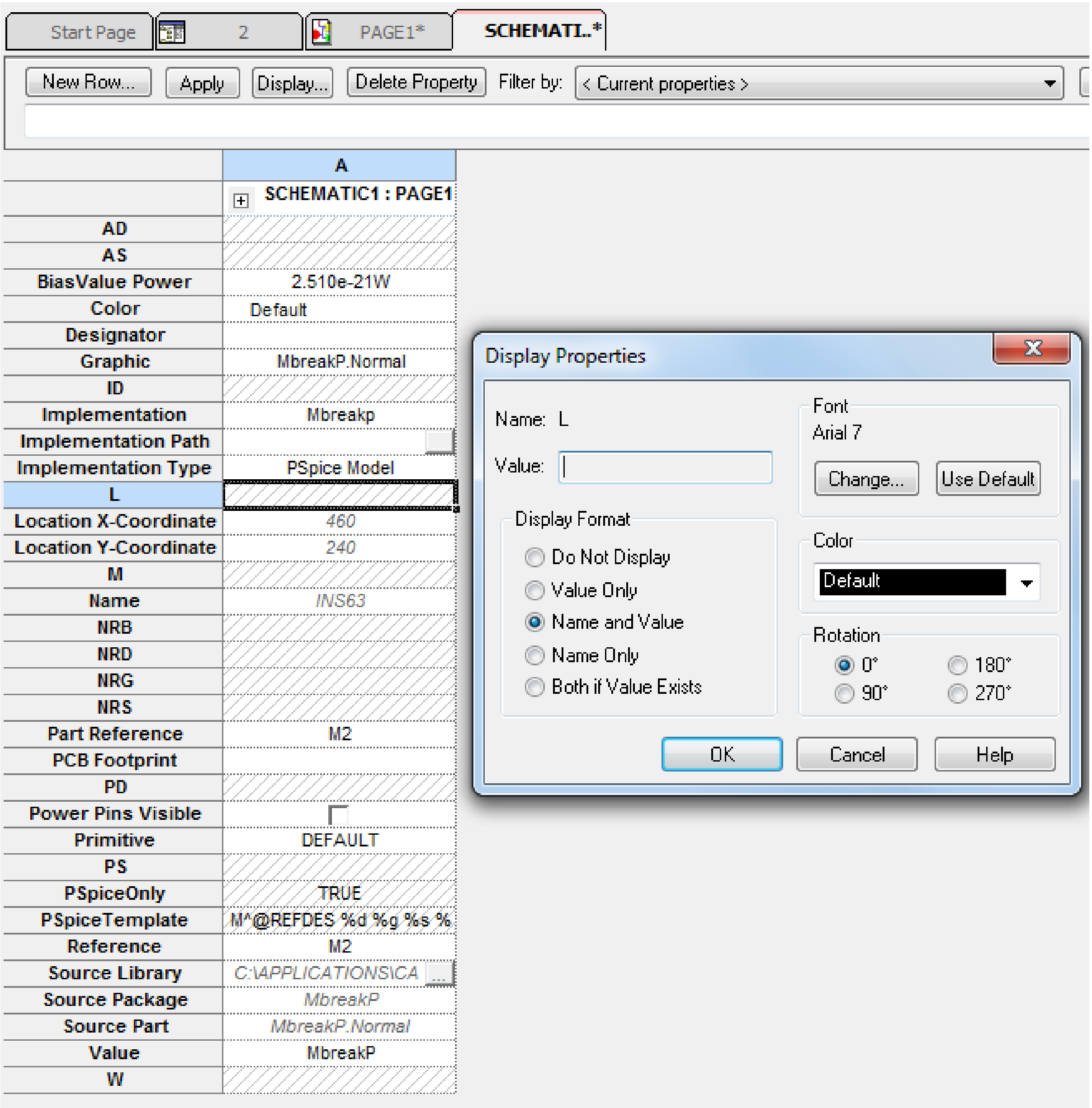
3. Double click PARAMETERS:, you will see the following, and click New Row, Edit Name: and Value as follows (e.g., Wvar and Lvar)



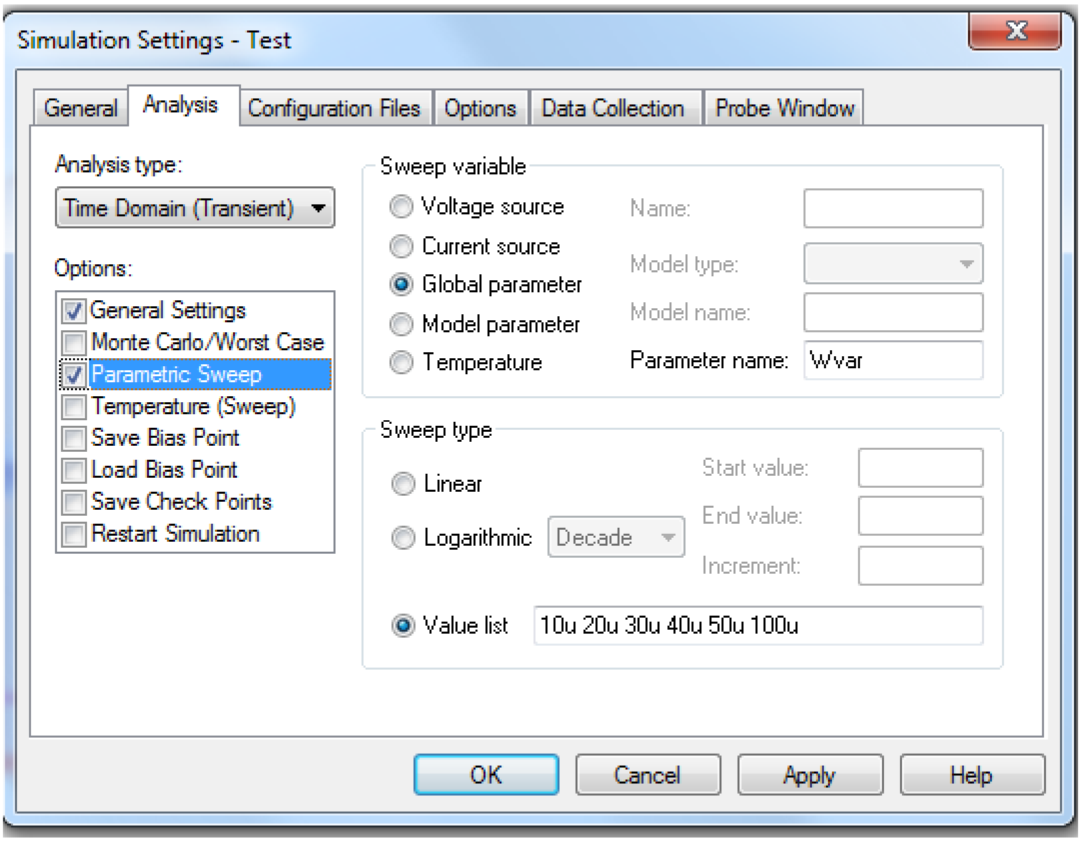
4. Double click MbreakP and MbreakN (see below), right click the area (right of L or W), and click Display in the dialog box that appears. The parameter W and L can be modified. Select Name and Value, click OK. You can double click W and L, and edit the value as Wvar and Lvar.



	A
	SCHEMATIC1 : PAGE1
AD	
AS	
BiasValue Power	2.510e-21W
Color	Default
Designator	
Graphic	MbreakP.Normal
ID	
Implementation	Mbreakp
Implementation Path	
Implementation Type	PSpice Model
L	
Location X-Coordinate	460
Location Y-Coordinate	240
M	
Name	INS63
NRB	
NRD	
NRG	
NRS	
Part Reference	M2
PCB Footprint	
PD	
Power Pins Visible	
Primitive	DEFAULT
PS	
PSpiceOnly	TRUE
PSpiceTemplate	M^@REFDES %d %g %s %
Reference	M2
Source Library	C:\APPLICATIONS\CA
Source Package	MbreakP
Source Part	MbreakP.Normal
Value	MbreakP
W	

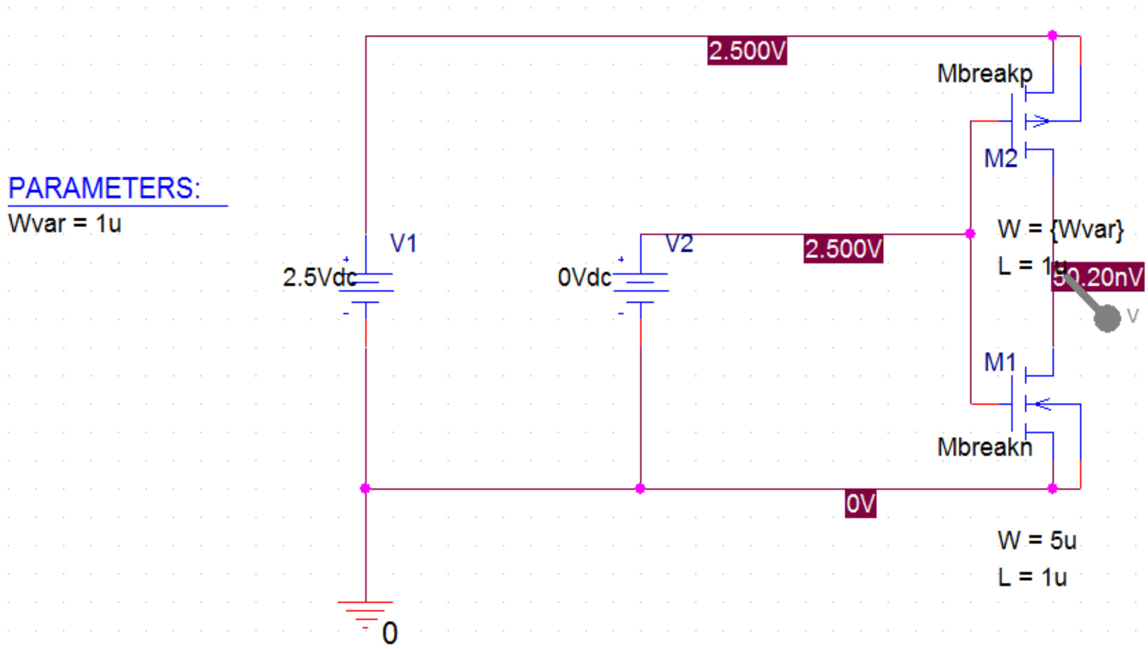


5. Create a new simulation profile called “transient” (or whatever you want).
6. Under ANALYSIS Tab, select an ANALYSIS TYPE of TIME DOMAIN (TRANSIENT).
7. Enter the stop time (TSTOP) for the simulation in the RUN TO TIME box.
8. Enter the maximum step size in the TRANSIENT OPTIONS section – making this variable ~TSTOP/1000, will help ensure that the traces in Probe are relatively smooth.
9. Click OK.
10. In Parametric Sweep, select Global parameter, input the name of parameter (e.g., Wvar) in Parameter name.
11. Set sweep type using Linear or value list (see follows).
12. Determine the variables of interest to be plotted in Probe.
13. Simulate the circuit.

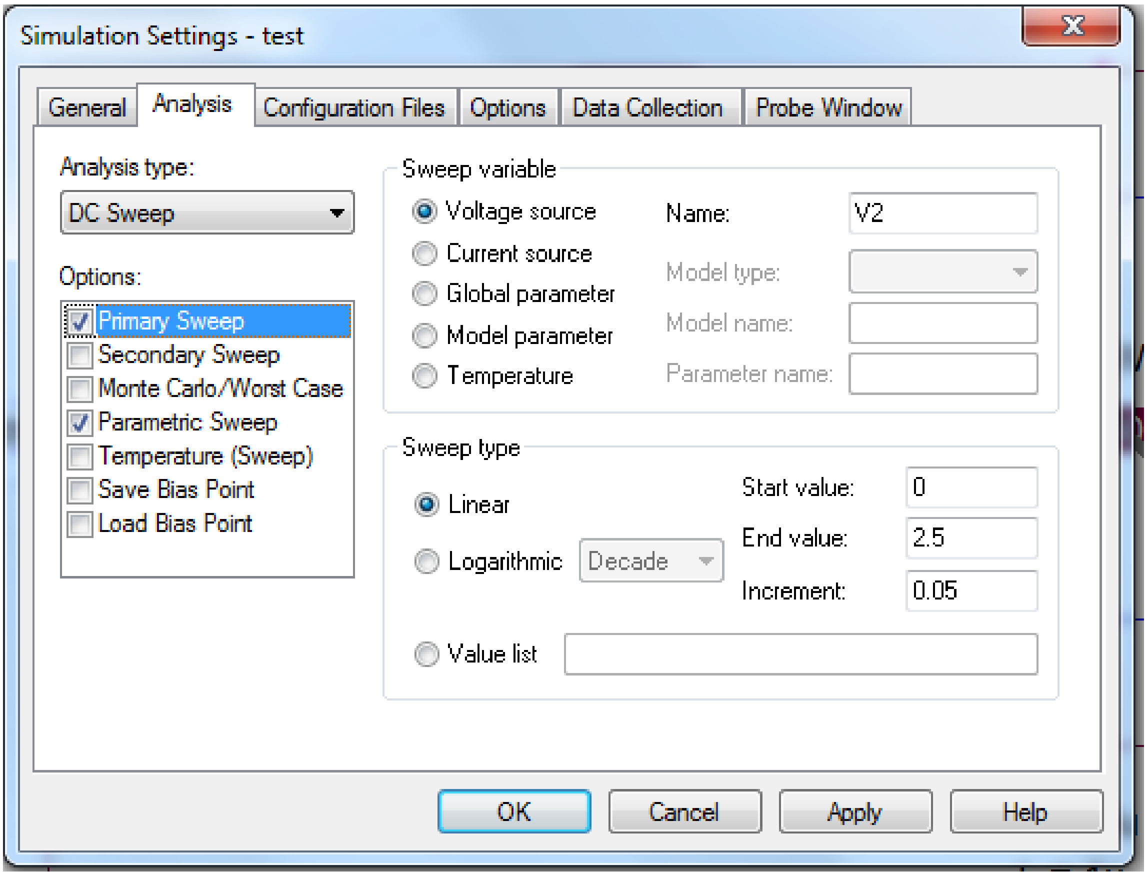


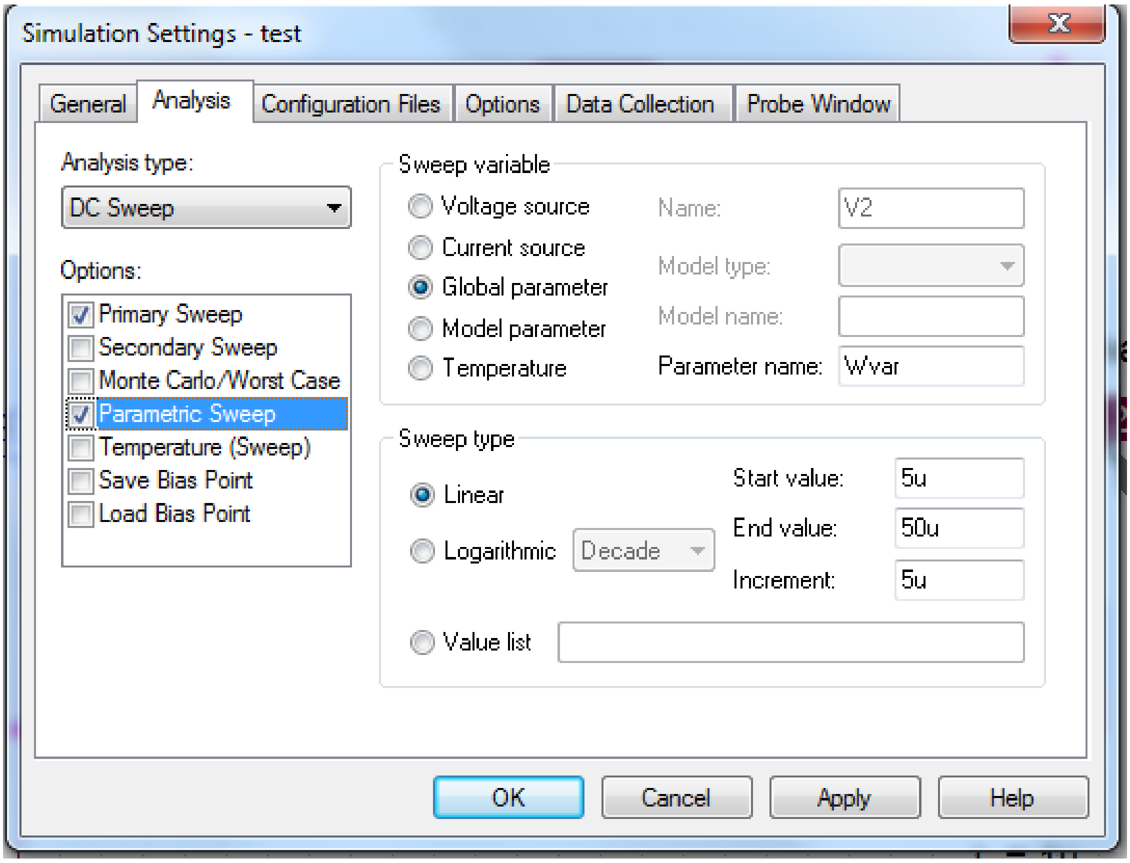
Parametric simulation of VTC of the inverter

1. If you want to do a VTC of the inverter parametric simulation, e.g., for the following schematic, we can edit them in "edit simulation profile" as follows:

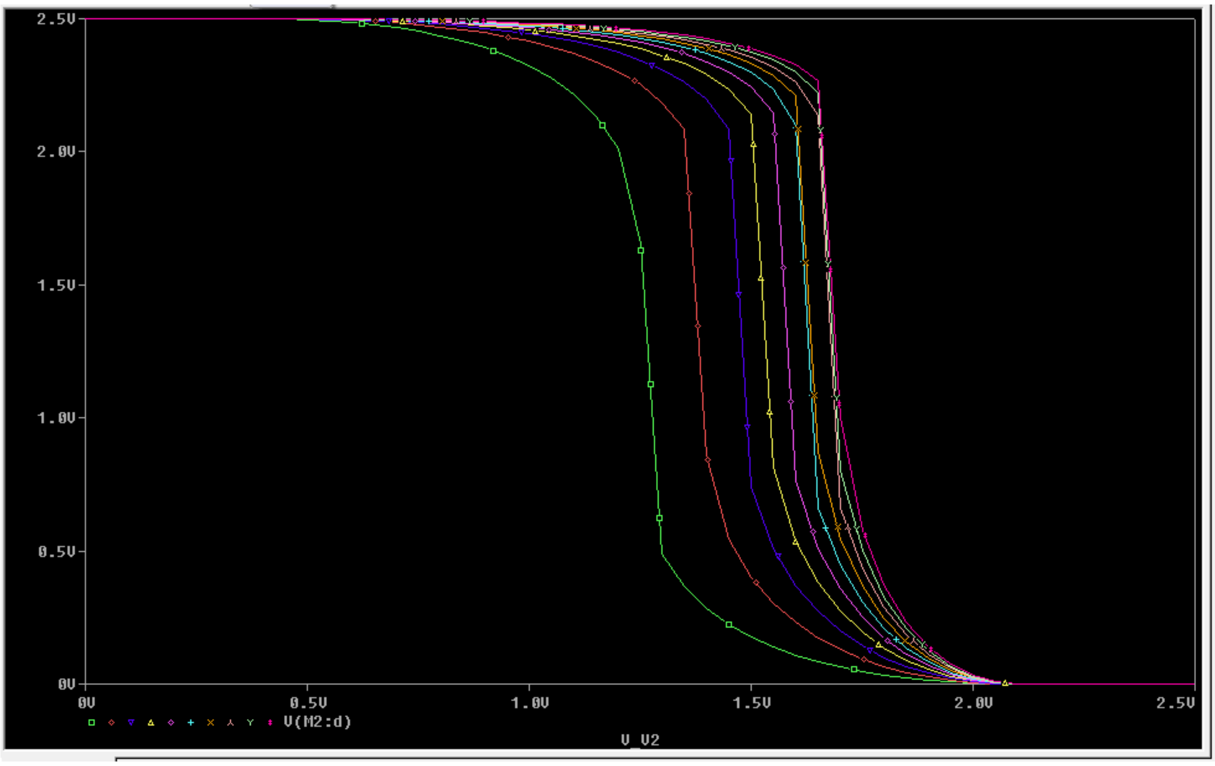


Where we change +VTO=0.4V for MbreakN, and +VTO=-0.4V for MbreakP.





The simulation results are as follows:



To Edit Waveform

1. Place voltage, voltage differential, current, and power markers from the toolbar where needed. 

   Voltage Markers can be placed on any wire and they will give the voltage at that node with relation to ground.

   Current Markers must be placed at one of the terminals of a device and they will give the current entering that pin.

   Power markers must be placed directly on the part and they will give the power dissipated by that device (thus a negative value means the part is supplying power).

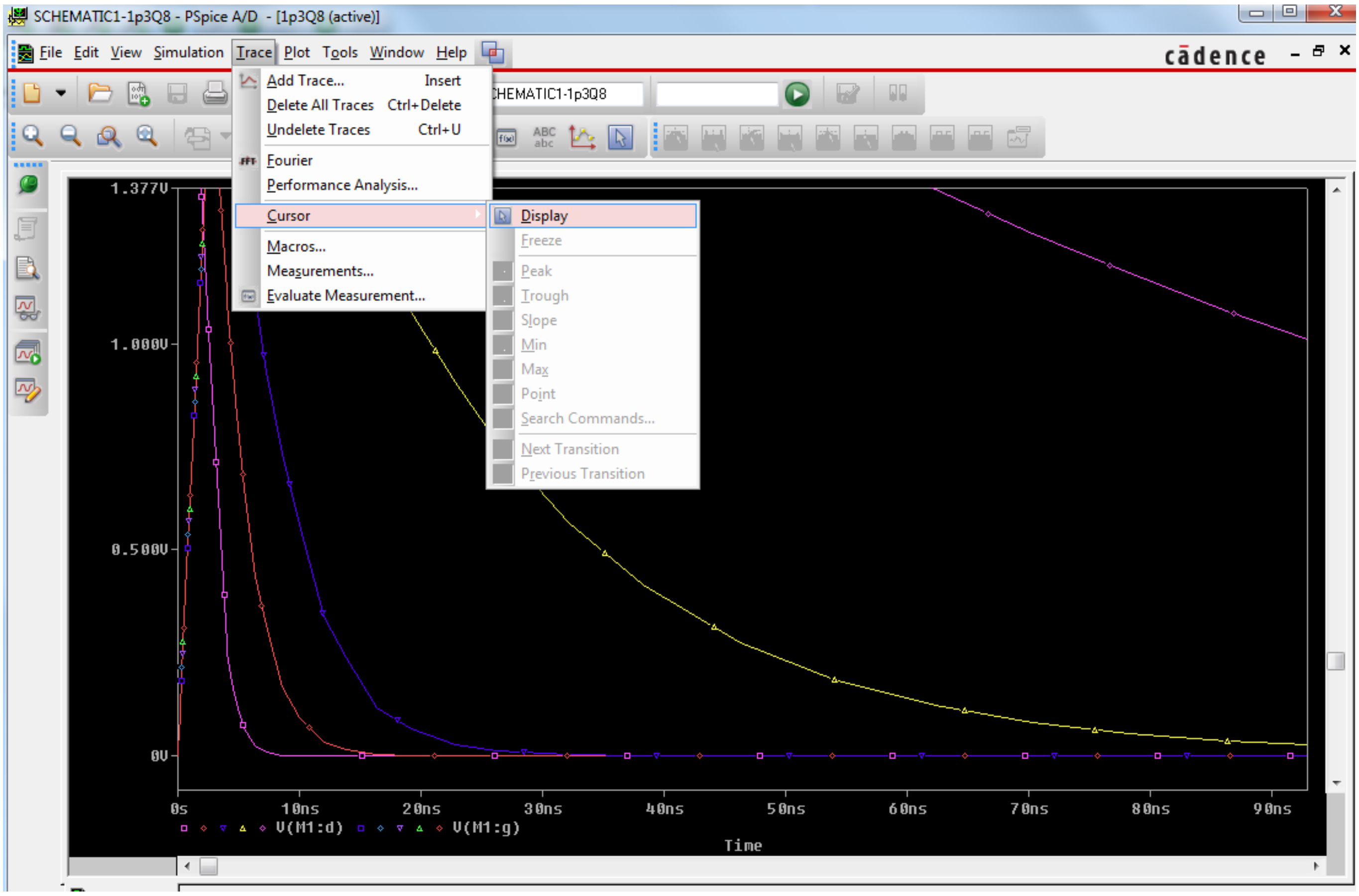
2. Click the run icon on the toolbar. 
3. Probe will be open with the marked values displayed on one graph
4. To add a new y-axes in Probe select PLOT→ ADD Y AXIS
5. To add a new plot window in Probe select PLOT →ADD PLOT TO WINDOW
6. To move a trace to the new axis/window:

   Click on the trace name to be moved (it should turn red when selected) and select EDIT → CUT

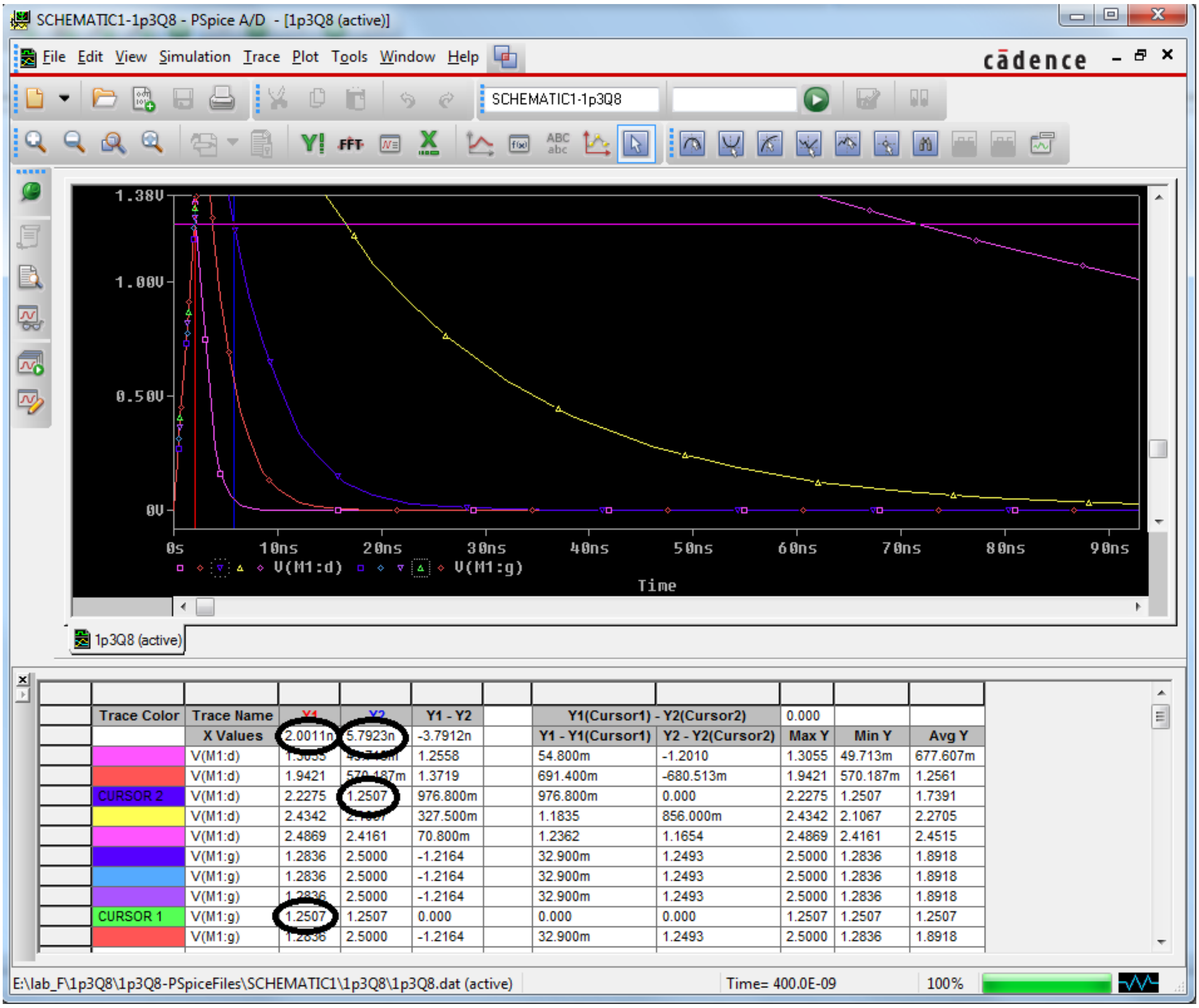
   Click on the axis/window to move the trace to (it should have a >> at the bottom when selected) and select EDIT → PASTE

7. To add a new trace select TRACE → ADD NEW TRACE and select the variable to plot. Algebraic expressions can also be entered here using the operators in the right hand side of the add trace window.
8. To measure the delay as follows:

Click Trace→Cursor→Display, we have



Left click, then Right click in the plot, you will see two measure cursors (red and blue ones). You can choose the marker below the X-axis (e.g., the markers in front of V(M1:d) of the following figure) to measure corresponding results. Drag the cursors, and let the Y value be 1.25 (see the following figure), we can obtain two corresponding X values. Therefore, the delay can be calculated. In this way, all delays can be obtained.



Print Simulation Results

The following method helps you to print simulation results in an excel file as follows:

1. In plot, click Edit→Select All→Copy
2. Open a new excel file, Paste, you will see many columns data. Select them.
3. Click Insert→Scatter. The figure in excel can be shown.

Common Mistakes

These are the most common problems I see students make when using PSpice.

1. **Not having a 0 Ground**. If you are getting errors that say you have a bunch of floating nodes, check to be sure that the ground you have is indeed named "0"!
2. **Not selecting Analog A/D when setting up the project**. If you can draw the schematic but then do not have an option to set up a simulation profile, you

selected Schematic instead of Analog A/D. To fix this, select the entire schematic, copy it, restart PSpice, create a new project with a new name (selecting Analog A/D, of course), and paste the schematic into this project.

3. **Not having the correct libraries included**. Only the components in the PSpice library will simulate correctly. You must have these libraries and no others included.
4. **Using the wrong sources**. DC sources (VDC, IDC) for bias and DC analysis, AC sources (VAC, IAC) for AC analysis, and time varying sources (VSIN, VPULSE) for transient analysis. Do not use VSIN for a frequency response or VAC for a transient analysis!
5. **Having a zero-impedance path from a voltage source to ground**. Before PSpice does any kind of analysis, it always does a DC bias point analysis first. If it finds a voltage directly across an inductor, it cannot compute the bias point. This can be solved by placing a very small resistance in series with the inductor. If you are getting an error that tells you that you have a voltage-inductor loop, you have this problem. This can also result from a direct short across a voltage source.
6. **SI prefix confusion**. PSpice is not case sensitive and you can use SI prefixes.