Transient Simulation of a CMOS NAND Gate using PSPICE

Dr. Elias Kougianos

Ver. 1.0

The PSPICE simulation environment is available on the General Access Labs (GAL) in Discovery Park. To start the PSPICE simulation environment go to:

START->All Programs->Cadence->Release 16.3->"Design Entry CIS"

If the following screen comes up, make the selections as shown and check the "Use as default" button.

Cadence Product Choices		
Please select the suite from which to check out the OrCAD Capture feature:		
Allegro PCB Librarian XL		
Allegro PCB Design CIS XL	Οκ	
	Cancel	
Use as default		

To create a new project go to:

File->New->Project

You will need to give a name to the project (in this case "NAND") and a location (folder on the hard disk). The completed form should look similar to:



When the "Create PSpice Project" dialog comes up, select "Create a blank project":

Create PSpice Project	×
Create based upon an existing project	ОК
	Browse
Create a blank project	Cancel
	Help

Add the required libraries to the project:



Create the schematic:



Prepare the simulation, Go to:

PSPICE->New Simulation Profile

Give the profile a meaningful name. We used "tran" since this is a transient analysis"

New Simulation		X
<u>N</u> ame:		Create
tran		
Inherit From:		Cancel
none	•	• •••
Root Schematic:	SCHEMATIC1	

Fill out the settings form as follows:

Simulation Settings - tran	×
General Analysis Configurati Analysis type: Time Domain (Transient) • Options:	on Files Options Data Collection Probe Window <u>R</u> un to time: 40n seconds (TSTOP) Start saving data after: 0 seconds
General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point	<u>I</u> ransient options <u>M</u> aximum step size: seconds <u>Skip the initial transient bias point calculation</u> (SKIPBP)
Load Bias Point Save Check Points Restart Simulation	Bun in resume mode Output File Options
	OK Cancel Apply Help

Finally, run the simulation:

PSPICE->Run (or F11)

Simulation results:

