

Transient Simulation of a CMOS NAND Gate using PSPICE

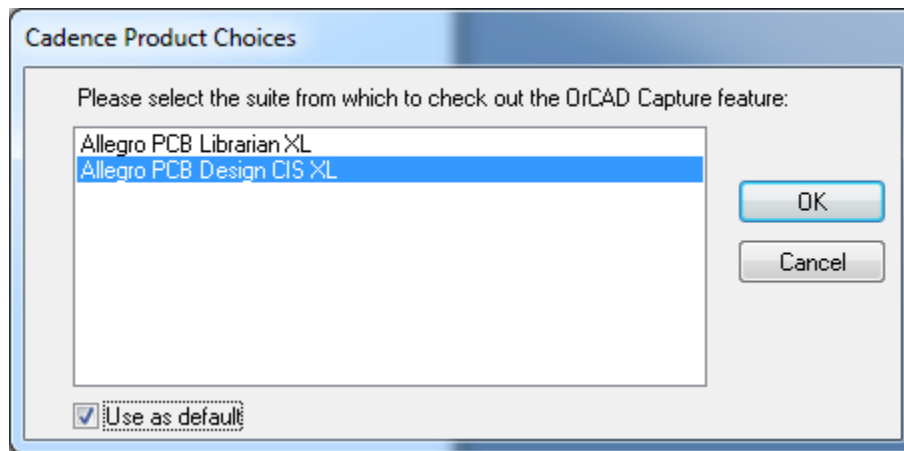
Dr. Elias Kougianos

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"

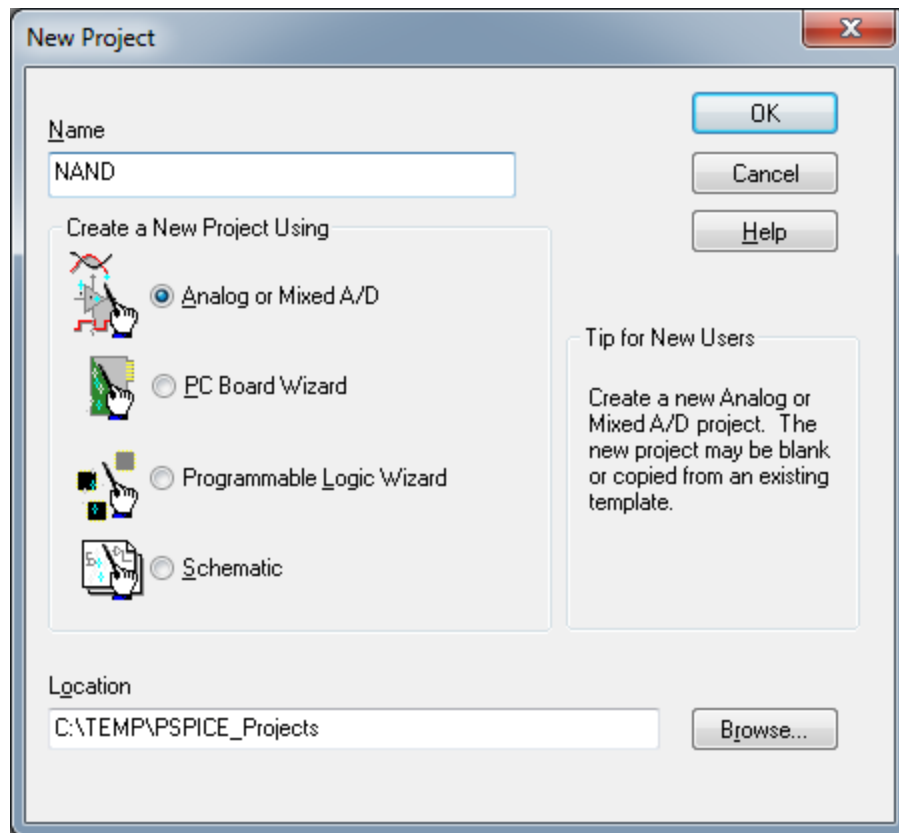
If the following screen comes up, make the selections as shown and check the "Use as default" button.



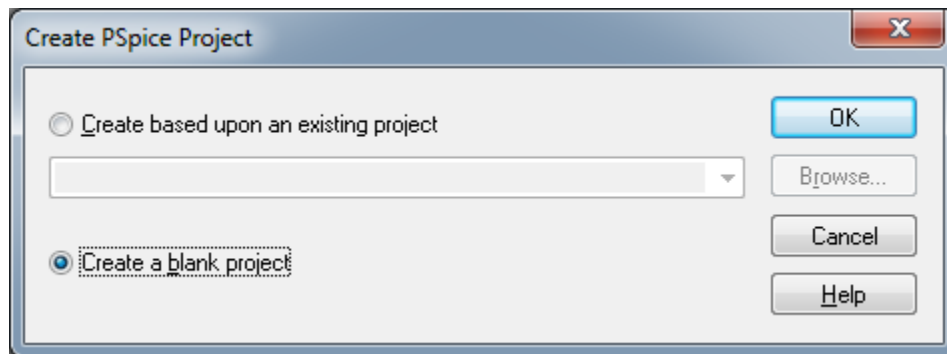
To create a new project go to:

File->New->Project

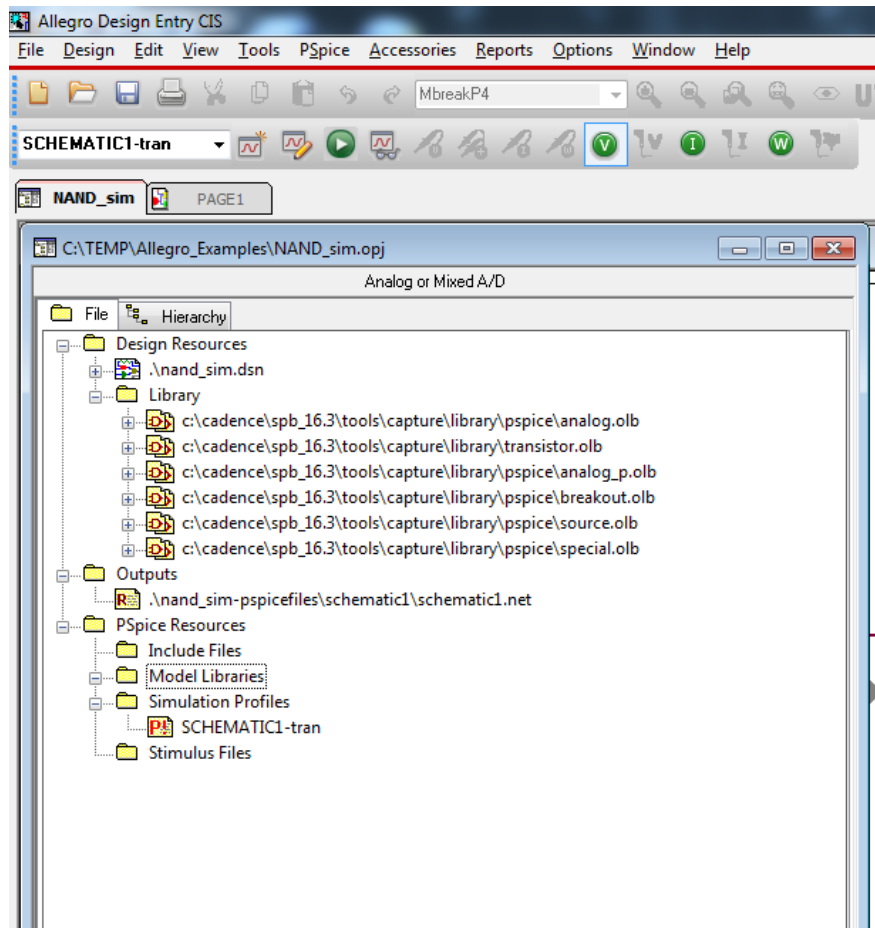
You will need to give a name to the project (in this case "NAND") and a location (folder on the hard disk). The completed form should look similar to:



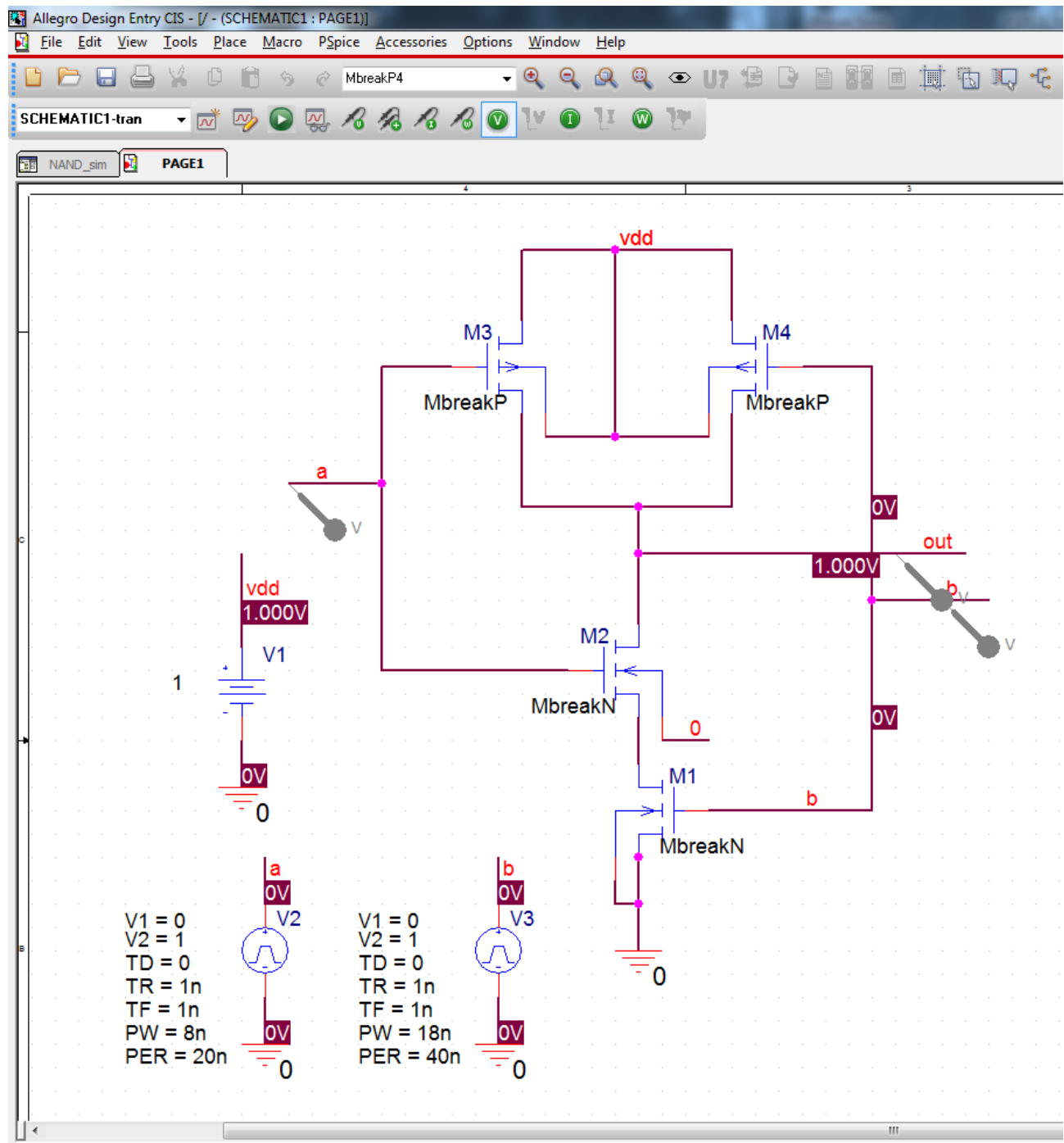
When the “Create PSpice Project” dialog comes up, select “Create a blank project”:



Add the required libraries to the project:



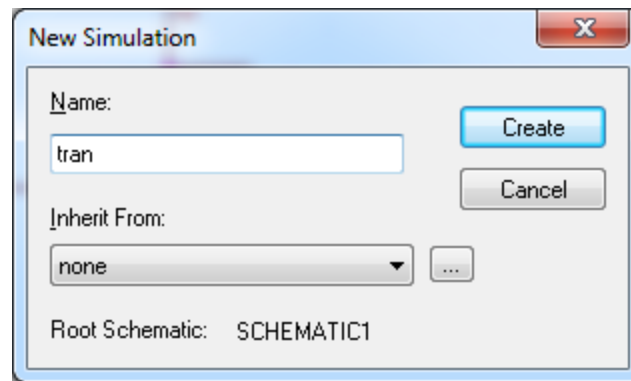
Create the schematic:



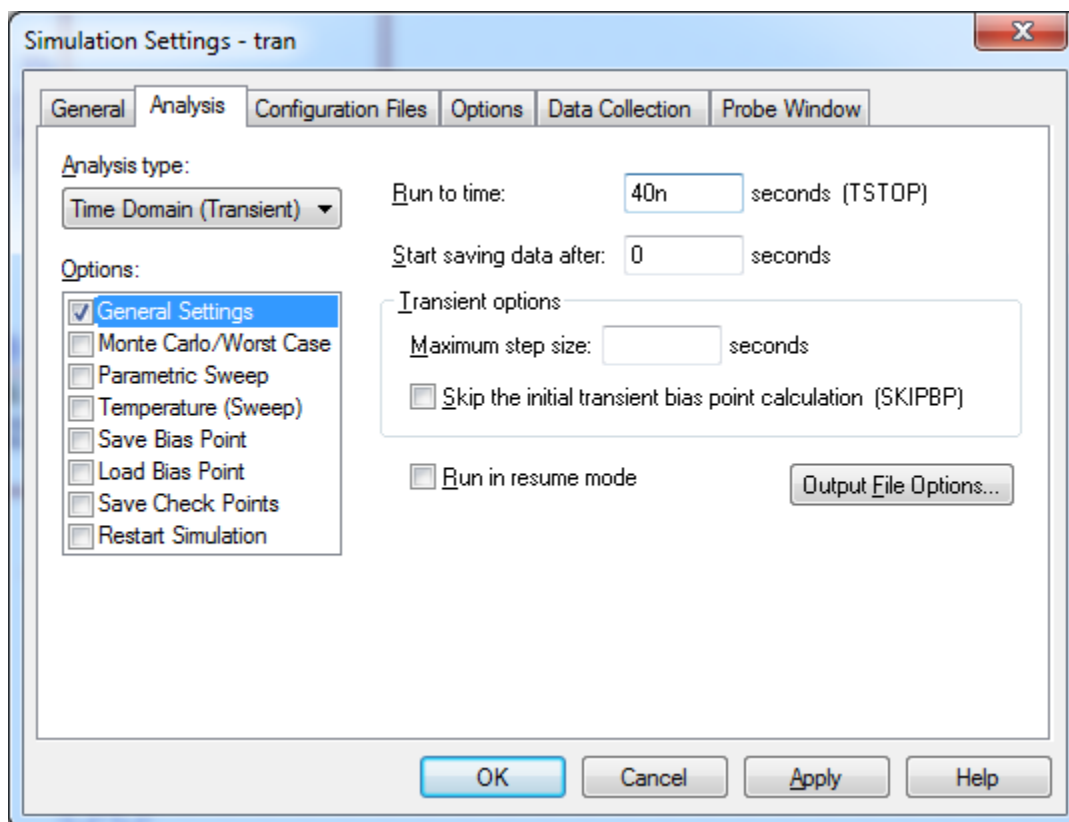
Prepare the simulation, Go to:

PSPICE->New Simulation Profile

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



Fill out the settings form as follows:



Finally, run the simulation:

PSPICE->Run (or F11)

Simulation results:

