

ENGI 5131 --- Tutorial 2

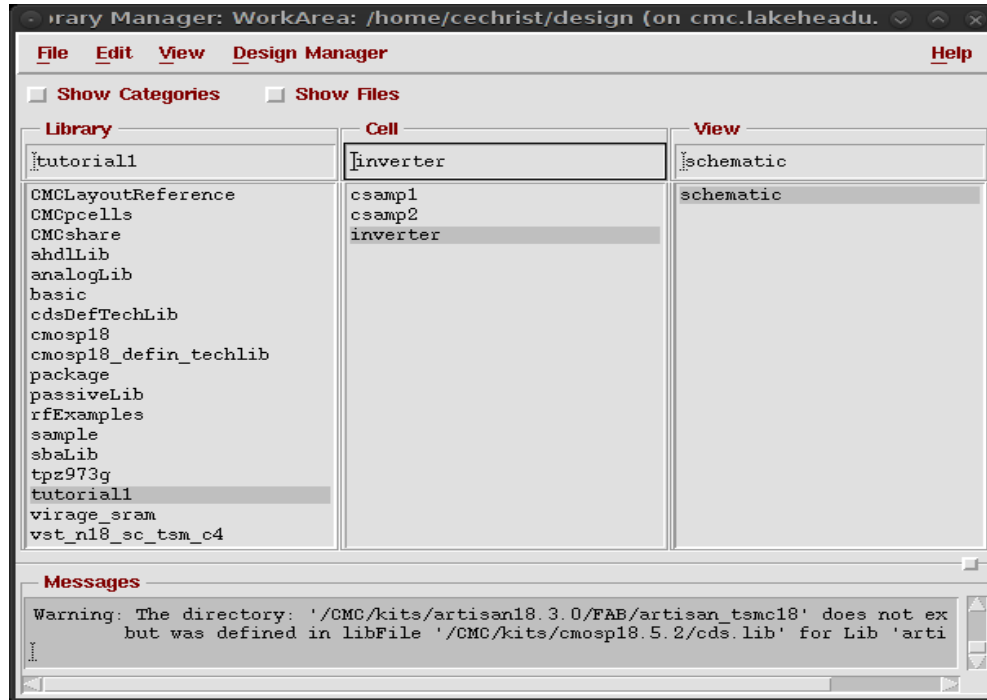
Transient Analysis
Lakehead University

Carlos Christoffersen, Greg Toombs
Winter 2013

cādence™

1. Constructing the Circuit

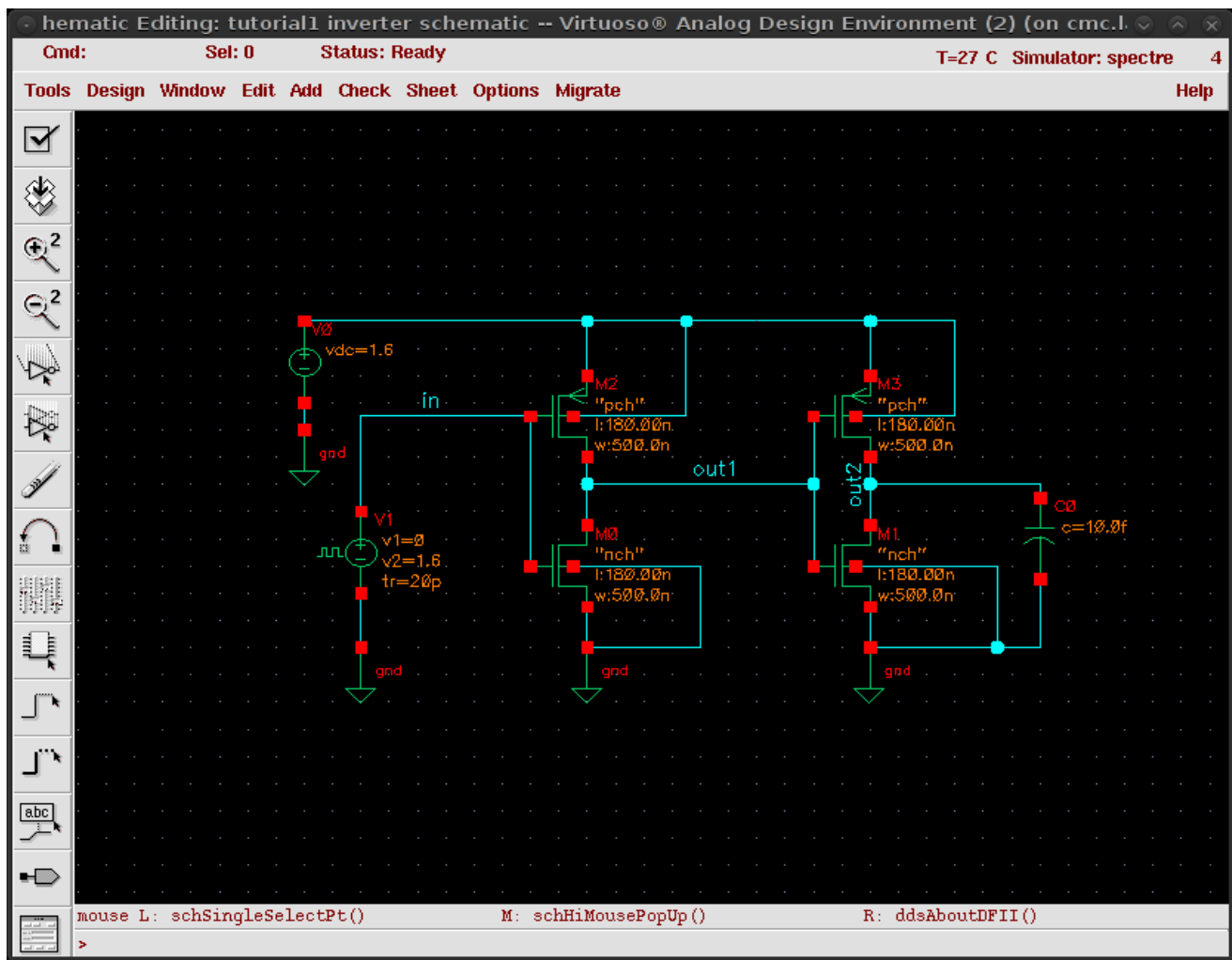
Start Cadence ICFB window as usual and open the library manager. You should be able to select the “tutorial1” library and create a new cell view there. In this tutorial the new schematic cell view is named “inverter”:



The Virtuoso Schematic Editor should open. Enter the circuit shown in the figure below. NMOS transistors are instances of the `cmosp18->nfet->symbols` cell and similarly PMOS transistors are instances of the `pfet` cell. These correspond to the thin-oxide transistors with a maximum supply voltage of 1.8 V. Use a “vdc” instance for the supply voltage source (set to 1.6 V) and “vpulse” instance for the input voltage source (both from the `analoglib` library). Set the following parameters for the input source:

- Voltage1 = 0
- Voltage 2 = 1.6
- Delay Time = 200p
- Rise Time = 20p
- Fall Time = 20p
- Pulse Width = 200p
- Period = 500p

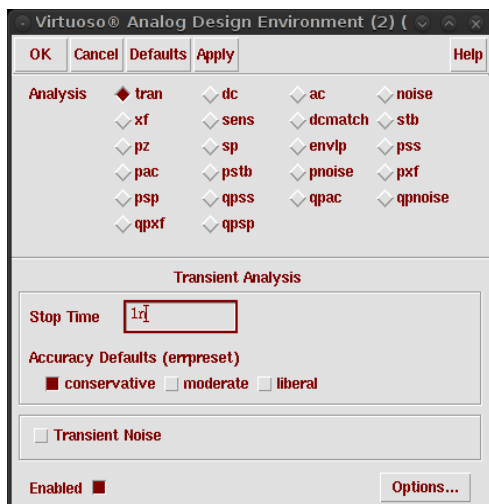
Note: p means pico = 10^{-12}



Note the input and output nodes have been named: “in”, “out1”, “out2”. Check and save the schematic once it is complete. Open the Virtuoso Analog Environment. Add the following library file:

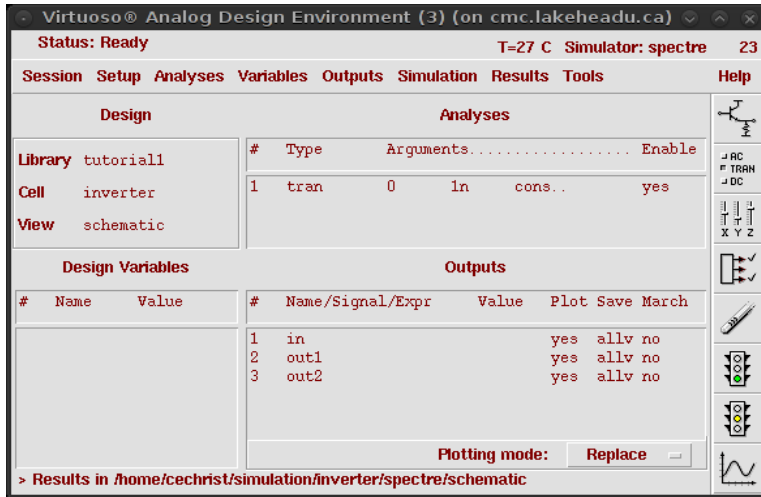
/CMC/kits/cmosp18/models/spectre/icfspectre.init

Now set the analysis to be performed to “Tran” with 1ns stop time and accuracy set to “conservative”

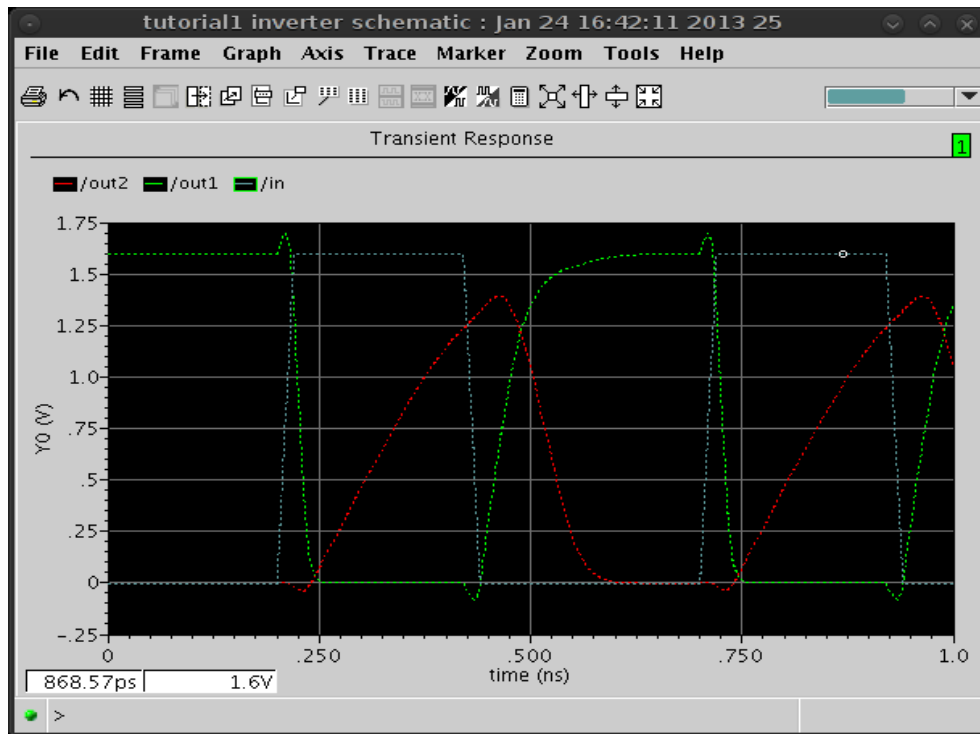


for accurate results. Transient analysis solves the complete nonlinear algebraic-differential equations of a circuit. Effects such as nonlinear distortion, intermodulation, saturation, clipping and oscillations (unstable behaviour) can be modeled with this analysis. Equations are numerically solved by default using the DC operating point as the initial condition. Start time is always 0 and the “stop time” parameter tells the simulator when to stop. The selection of time step is automatic by default, but it can be changed in the “Options” dialog box.

Select outputs to be plotted as shown below:



After running the transient analysis the following Wavescan plot should open:



During the laboratory session we'll use the trace and zoom features to determine $t_{+70\%}$ and $t_{-70\%}$ of the two inverters.