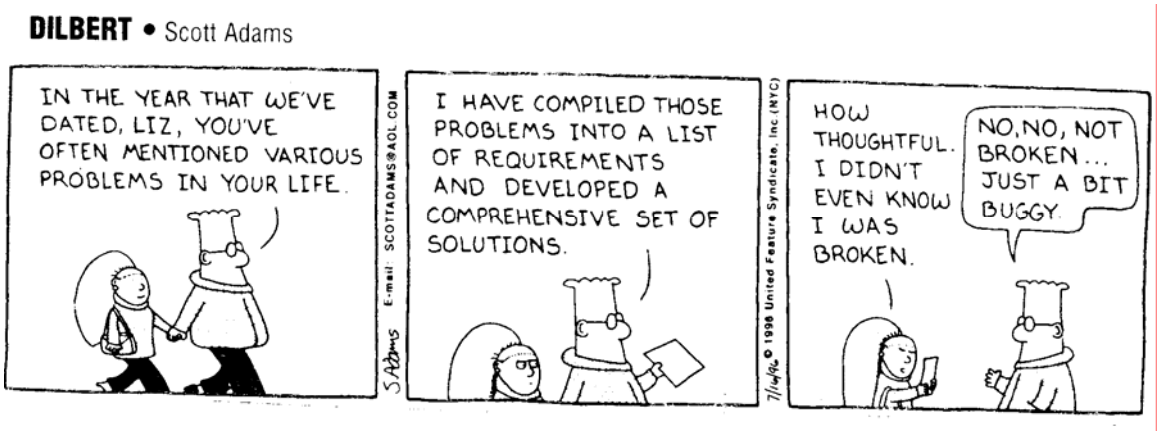
Harris

Introduction to CMOS VLSI Design (E158)

Problem Set 4



1. Do problems 5.5 and 5.10 from *CMOS VLSI Design*.

For this class, please use the AMI 0.6 μm transistor models for all simulations because that is the process in which you are doing your designs. An example of the FO4 inverter simulation referencing this library is in
\\Charlie\Courses\Engineering\E158\ProblemSets\fo4

To invoke HSPICE on the ECF or MicroP's computers, select Start * Programs * Engineering * HPSICE Y-2006.03 * HSPICE Y-2006.03. In the HSPICE window, choose File * Simulate. Select your spice file (e.g. fo4.sp). Then click Save again to save the results listing in a file with the same base name (e.g. fo4.lis). Open the .lis file in a text editor to view the results. Occasionally HSPICE has trouble checking out a license. If this happens, just rerun the simulation and it will usually find the license. If you find yourself running HSPICE many times, it may be more convenient to run hspui or invoke HSPICE from the DOS cmd prompt.

The graphical waveform viewer is invoked as Start * Programs * Engineering * Synopsys * CosmosScope™ X-2006.03 * Cosmos-Scope. Use File * Open * Plot Files… to open the .tr0 transient results. Click on signals to graph the waveforms. Be sure there are no spaces in the directory or filenames; Cosmos-Scope will have trouble reading files with spaces.

2. Please indicate how many hours you spent on this problem set. This will not affect your grade, but will be helpful for calibrating the workload for the future.