Integrated Circuit Design

Homework #1

Due: 2020/04/29 13:00 (No late homework is allowed.)

1. Setup HSPICE following the steps listed in *20200408_T1_Linux_and_Hspice.pdf*.
2. Use *mosistsmc180.sp* as the transistor model.
3. Write a SPICE deck for NAND2 using *.subckt* . Save it in Plain Text format using the name *nand2.sp* .
4. Write a SPICE deck for a multiplexer ($D = D_0\bar{S} + D_1S$) using three NAND2 gates.
5. Run HSPICE using the deck prepared in Step 4. Test all possible output cases:
    (D_0, D_1, S) = (0, 0, 0), (0, 0, 1), (0, 1, 0), (0, 1, 1), (1, 0, 0), (1, 0, 1), (1, 1, 0), (1, 1, 1).
6. Use COSMOS scope to show the results in Step 5.
7. Take a snapshot of the screen showing the waveforms in Step 5.
8. Paste the snapshot taken above in a document. Save this document in PDF format using the name, *multiplexer_wfs.pdf*.
9. Save the SPICE deck in Plain Text format using the name, *multiplexer.sp* .
10. Create a directory named *~/ICD2020/HW1* and put the above three files under the directory.
11. Compress the directory use the following command:
    tar cvzf *YourID#_HW1.tar.gz* HW1
12. Submit your *YourID#_HW1.tar.gz* file to ceiba.