Integrated Circuit Design

Homework #2

Due: 2019/03/27 13:00

1. Setup HSPICE following the steps listed in *20190313_T2_Hspice.pdf*.
2. Use *mosistsmc180.sp* as the transistor model.
3. Write a SPICE deck for NAND3.
4. Run HSPICE using the deck prepared in Step 3. Test all 8 input combinations.
5. Use COSMOS scope to show all input/output combinations.
6. Take a snapshot of the screen showing the waveforms in Step 5.
7. Paste the snapshot taken above in a document. Save this document in PDF format using the name*, nand3wfs.pdf*.
8. Save the SPICE deck in Plain Text format using the name, *nand3.sp*.
9. Create a directory named *~/ICD2019/HW2* and put the above two files under the directory.
10. Compress the directory use the following command:
    tar cvzf *YourID#_HW2.tar.gz* HW2
11. Submit your *YourID#_HW2.tar.gz* file to ceiba.