LTspice Simulations

EE3002 Analog Circuits

- Download the LTspice Simulation exercises from https://alfy7.github.io/LTspice-Basics/
- These exercises were designed to explain most of the features of LTspice by showing how they are used when simulating different circuits. The exercises are meant to be done in order for proper continuity. Most of the circuits would have been covered in previous/ongoing courses.
- Each exercise has a PDF of the steps to be followed, as well as the LTspice simulation file. It is recommended to try and work out all the exercises.
- For this course, use the TSMC 0.18μm MOSFET model file from http://www.ee.iitm.ac.in/~nagendra/cadinfo.html. Note that V_{DD}=1.8V, W_{MIN}=0.27μm, L_{MIN}=0.18μm. Download the model file for *Eldo and others*.
- Copy the model file to the same directory as the LTspice simulation file and add the spice directive *.include tsmc018.lib* to include the file in the simulation. [Verify file extension]
- When adding NMOS and PMOS to the schematic, specify the model name as *cmosn* and *cmosp* respectively. Make sure to specify the drain/source area and perimeter as instructed in the problem statement.
- Try out the following simulations with this model file before starting the project.
  - Run a DC operating point simulation with a mosfet biased in saturation, and verify the small signal parameters obtained from simulation and hand calculation
  - Run a DC sweep simulation to get the $I_D - V_{DS}$ curves
  - Run a DC parametric sweep simulation to get the $I_D - V_{DS}$ curves for different W, keeping W/L fixed.
  - Design a common source amplifier and run an AC simulation to measure its gain and bandwidth. Use a current mirror to generate the bias voltage. [Make sure that the mosfets are in saturation]
  - For the same amplifier, run a transient simulation with sinusoidal inputs of different amplitudes to see the swing limits
- For the Project
  - First estimate the required DC gain and pole/zero frequencies using the given specifications.
  - You may modify the simulation of Exercise 6 and design the complete opamp to verify if the circuit will work as expected [simulation of small signal circuit using ideal models].
  - Now try to estimate the mosfet sizes using the calculated small signal parameters. Use reasonable assumptions to get an initial starting point. [You may create new symbols with parameters W and L that can automatically calculate drain/source area & perimeter]
  - Run a DC operating point simulation to get the small signal parameters from simulation. Now, adjust the bias currents/mosfet sizes to get closer to the required parameters.
  - You can also use the small signal parameters obtained from simulation to fine tune the hand calculated values, and get better estimates of other component values.
  - Run AC analysis and use the obtained graphs to estimate pole/zero locations from simulation. Compare this with the hand calculated values and then adjust bias currents/component values as needed.
  - Use parametric sweeps to fine tune component parameters.