**Introduction:**

LTspice allows you to add the transistor capacitances values into the device model. This can be done using the” .model” directive, i.e. “.model nmos NMOS(kp=100u, VTO=0.5, l=10u, w=36000u, Phi=0, lambda = 0.0, Cgso=1e-9,Cgdo=1e-9)”.

In this lab you will be tasked to perform frequency analysis on amplifier circuits. The frequency analysis can be done as follows:

1. We will use the amplifiers designed in Lab 3 and 5.
2. Right clicking on the input voltage source🡪choose advanced settings🡪lookup the AC small signal analysis and enter a value of 1 inside the AC level box.
3. Choose the Simulate tab and select edit simulation Cmd 🡪 Choose the AC analysis 🡪select a Decade as the sweep type 🡪 Choose the start-stop frequencies based on your theoretical analysis.
4. Run the spice simulation, you should observe that a screen will show up with the swept frequencies displayed on the x-axis.
5. On the LTspice schematic screen probe the transistor output. You should notice a Bode plot displayed on the blank screen.
6. The screen will show you both the Magnitude and Phase responses. To have the magnitude response only, right click on the phase angle axis and choose “don’t show phase angle”

**Task#1:**

1. Assuming the following transistor capacitance values for Cgso = 1e-9, Cgdo = 1e-9, and a 10µF for coupling capacitors, perform a theoretical low and a high frequency analysis on the 3 amplifier stages you have designed in labs 3 (only the CS stage biased using a voltage divider) and 5. (This should be done using node-pole association method, Miller’s Theory and the Direct analysis.)
2. Validate your theoretical calculations using LTspice.
3. Decrease the value of the transistor capacitances by 2 orders of magnitude and observe the effect on the bandwidth using LTspice. Discuss this theoretically.
4. Increase the value of the transistor capacitances by 2 orders of magnitude and observe the effect on the bandwidth using LTspice. Discuss this theoretically
5. Discuss these results thoroughly in your report, where the theoretical calculations using the 3 different approaches are to be compared with the simulation results.