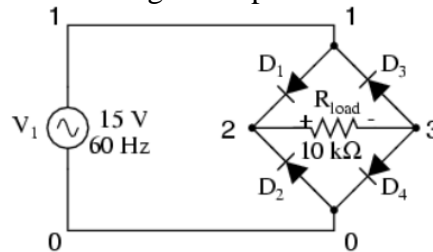


EE311 Laboratory Assignment Problems

Design layout of the following problems using MAGIC which is solved using SPICE and verify their layouts by performing post layout simulations. It is anticipated that all the problems have been solved by you by this time using SPICE and the circuit information is readily available with you. Therefore, making layouts using MAGIC should not be an issue. Thus, please follow submission instructions at the end of the assignment.

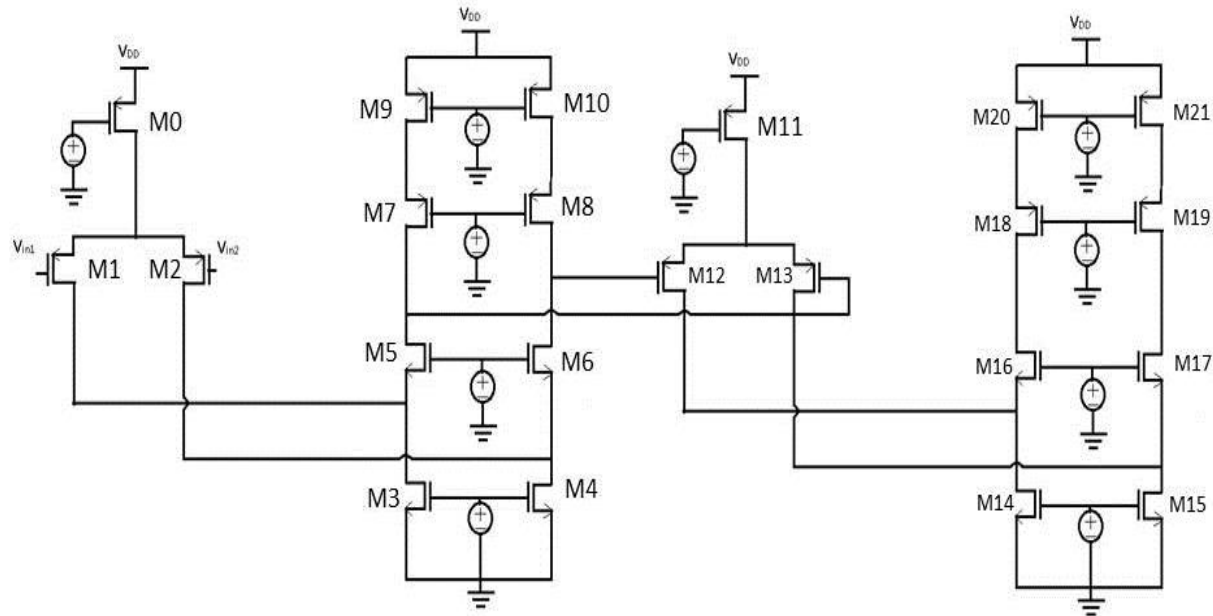
1. A bridge rectifier circuit is given below.
 - (a) Find out voltage at each node.
 - (b) Plot output waveform for the given input.



2. Design a Band Pass, Low Pass and High Pass circuit. Design parameters for these circuits maybe chosen on your own.
3. Plot nMOS and pMOS transfer characteristics for different V_{ds} and V_{gs} .
4. Design of CMOS inverter using SPICE.
 - (a) Calculate Rise Time and Fall Time using transient analysis, propagation delay with and without load capacitor, C_L .
 - (b) Plot VTC of CMOS Inverter for DC input and calculate noise margin (NM).
5. Design of 3 input CMOS NAND, 3 input CMOS NOR and CMOS EX-OR gates using SPICE. Study the effect of change of MOSFET sizes on the gate delays and appropriate comments on it.
6. Design 8 bit adder, 8 bit register, 16 bit counter (of your choice) using SPICE.
7. Design 16 bit fixed point multiplier using SPICE. (**Refer CMOS VLSI Design: A Circuits and Systems Perspective by Neil Weste and David Harris**)
8. Design nMOS amplifier using voltage divider biasing.
 - (a) Apply a 50mV peak-to-peak 4kHz sinusoidal signal at the input. Measure the output signal at the drain of the MOSFET
 - (b) Using the same circuit, connect a load of 1Meg Ohm at V_{out} . Measure the output signal at the drain of the MOSFET, and calculate the gain. Repeat this procedure for load values of 50k, 5k, 1k, 500, and 50 ohms. Comment on these variation of characteristics of circuit for each load

The diagram shows a CMOS differential amplifier. The PMOS network consists of two parallel branches. The left branch contains PMOS transistors M6 and M4 in series. The right branch contains PMOS transistors M5 and M9 in series. The NMOS network also has two parallel branches. The left branch contains NMOS transistors M2 and M1 in series. The right branch contains NMOS transistors M3 and M8 in series. A central NMOS transistor M1 is connected between the gates of M2 and M3 and the common source node. The gates of M2 and M3 are driven by differential-mode input signals V_{in1} and V_{in2} . The gates of M4 and M5 are connected to a PMOS current mirror formed by M6 and M9, which is biased by a tail current source M0. The gates of M6 and M9 are connected to the output nodes. The gates of M7 and M8 are connected to the common source node. The output nodes are connected to the gates of M10 and M11, which are also connected to the common source node. The common source node is connected to ground through a tail current source M1.

10. Design a two stage amplifier given below using SPICE.



11. Draw layout of an INDUCTOR in MAGIC and simulate its characteristics using SPICE. (Do it only after completing previous 10 questions otherwise do not attempt it. No marks will be given if you attempt it first and do not complete first 10 problems.)

NOTE:

1. The problems should be completed by **25 September 2019, 23:59Hrs** and submitted to gtiitgcourses@gmail.com. No submission will be entertained by us after the above mentioned date.
2. The email must have subject line as per the format given, **"NAME_ROLLNO_MAGIC_ASSIGNMENT SUBMISSION"**
3. It is strongly advised not to copy assignments other than at the end of the semester during evaluation, otherwise severe penalty will be imposed.
4. The viva based on the assignments will be conducted by EE311 Laboratory TAs. The final decision will be of TAs only while assignment evaluation and only in exceptional cases, course instructors will intervene.
5. For any query, you can approach TAs after taking an appointment with them.
6. For any dispute or issue to be resolved, you can always approach course instructors, Prof. Harshal B. Nemade and Dr. Gaurav Trivedi.

All the best!