Lab #1: Introduction to Electronic Design Automation (EDA) II

Name_____

All Schematics and Screenshots must be uploaded in your CMPE 310 BOX I created for you.
AND PRESENTED to TA before leaving lab.

Pay careful attention to Allegro PCB Design Tutorial. The objective of this project is to walk you through the process of creating PCB output layout files from layout circuit netlist created using Capture CIS that are suitable for PCB fabrication using Allegro PCB Design.

You should complete this lab individually. When you have completed the assignment, raise your hand to get TA's attention, so that you can present your PCB layout to the TA and get your lab signed off before leaving class.

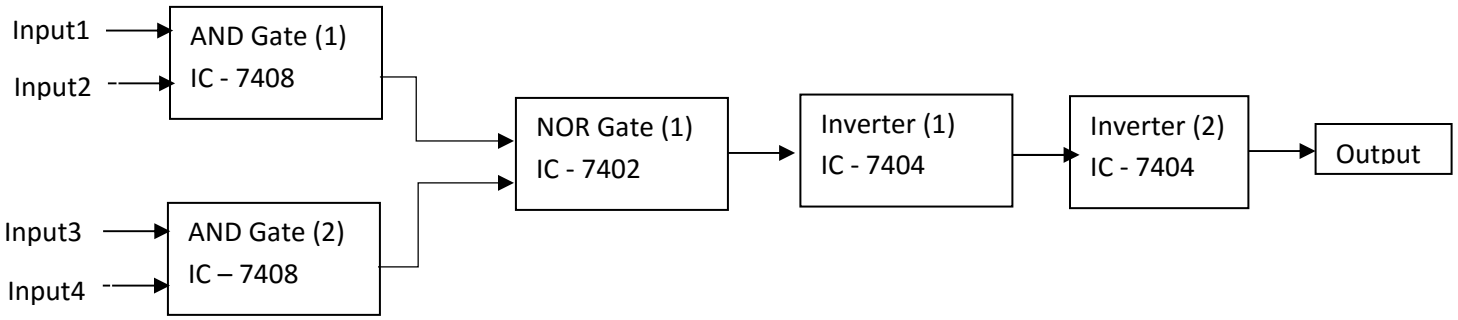
Reading/reference material

- Allegro PCB Design tutorial

Concepts

Illustrate the importance developing skills in using an EDA tool to create PCBs.

Problem: Using the AOI (AND-OR-Inverter) schematic design from Lab#0, transfer the AOI schematic design to output layout files that are suitable for PCB fabrication using Allegro PCB Design tool.



*****All Schematic design and Screenshot must be uploaded in your CMPE 310 BOX I created for you**.***