Dear Students,

- We have 5 experiments in Circuit Theory Laboratory and you must attend at least 3 experiments to have a lab score. Otherwise you will not get a credit for the lab part of the course.
- You can have only 1 make-up experiment if you cannot attend the experiment due to medical reasons. You have to send your report showing your health status.
- Experimental sheets and material list will be shared with you. You must write a preliminary work report and submit it before each lab. You must bring your lab equipment and materials to experiments. Otherwise you are not allowed in the laboratory and you will get no credit for that experiment.
- You need OrCAD PSpice for simulations. The links are given below to download this simulator.

  Link:
  Alternative link:

- After you install PSpice, watch the given video tutorials and prepare your homework until deadline.

  1st video tutorial
  2nd video tutorial
  3rd video tutorial

HOMEWORK <mark>(Due date: Saturday, October 23th, at 10:00 a.m.)</mark>

Prepare your own PSpice tutorial (a pdf file) using only simple circuit elements (Resistors, capacitors, inductors, current and voltage sources). Your tutorial must cover followings:

1. How to open a new project?
2. How to add necessary libraries?
3. How to draw a circuit on a schematic?
4. How to do a DC simulation (bias point)? How to use voltage/current/power probes and displays?
5. How to do a DC sweep simulation?
6. How to do a Transient simulation (Time domain)?
7. How to do an AC simulation (AC/frequency sweep)?
8. How to change trace properties (thickness, color) on a simulation graph?

Your tutorial should not be very long (Max 2/3 pages for each item). Give only brief informations bullet by bullet and show each simulation type with one simple example. Insert screenshots for the software interface, schematics and simulation results.

I give you this homework just to practice what you watched in video tutorials. Do not get lost in details.