

## Printed Circuit Boards (PCB) Design and manufacture

### INSTRUCTOR:

Dr. Mustafa M. Shiple

### PREREQUISITE:

- Windows operating system environment.
- Basic electronic circuits.

### COURSE OBJECTIVES:

- Course goals:
  - To understand the principles and techniques of PCB designs (through OrCad tools).
  - To design and manufacture PCBs.
  - To learn to use computers in PCB fabrication labs (CNC machines and photo-plotters).
  - To develop teamwork skills

### LABORATORY: will be held in electronics dep. at PCB Fab.

- Students have to take their notes and write discussions and clarifications. No material available for lab.
- Lab etiquette:
  - *Cleanliness*: Keep food and drinks away from the machines. Put trash in wastebaskets.
  - *Safety*: there will be acids and photo-sensitive materials. Keep any light sources or camera flashes out of lab.
  - *Manners*: Use the available machines and computers in delicate manner.

### Time schedule:

| # | Title                                     |
|---|---|
| 1 | Introduction to schematic and part editor |
| 2 | Flat, Hierarchical design                 |
| 3 | Preparing for OrCAD Layout                |
| 4 | Introduction to layout and smart rout     |
| 5 | Fabrication process steps                 |

### TEXTS AND SUPPLIES:

- **On-line Materials**
  - OrCAD Flow Tutorial : <http://drshiple-courses.weebly.com/pcb-design-and-manufacture.html>

## **Course in details:**

### **Day 1<sup>st</sup>: Introduction to schematic and part editor**

- Creating a new project.
- Defining the OrCad interface windows and file extensions.
- Assigning global signals and defining the circuit construction.
- Assigning reference designators.
- Building parts and symbols.

### **Day 2<sup>nd</sup>: Flat design Hierarchical design**

- Building a hierarchical design (top to down and down to top methodologies)
- Creating multi-sheet flat designs.
- Copying work between projects.

### **Day 3<sup>rd</sup>: Preparing for OrCAD Layout**

- Post-process schematic designs to prepare them for layout. ( netlist creation , design checking, documentations, bill of material and reports editing)

### **Day 4<sup>th</sup>: Introduction to layout and smart rout**

- Using spreadsheets to manage design data and rules.
- Develop skills in part placement as well as interactive and automatic routing.
- Post-process board designs to prepare them for manufacturing.

### **Day 5<sup>th</sup>: Fabrication process steps**

- Introduction to PCB technology.
- Anatomy of laminates, and phototool generation including screen preparation.
- PCB fabrication techniques-single, double sided and multilayers.
- Drilling operation by CNC etching: chemical principles and mechanisms.
- Plating operations. Post operations; stripping solder masking.
- PCB component assembly processes (DIP and SMT technologies).