PCB Design Fundamentals

1. **Course number and name:** 020PCBES5 PCB Design Fundamentals

2. **Credits and contact hours:** 4 ECTS credits, 2x1:15 contact hours

3. **Instructor's or course coordinator's name:** Rayan MINA

4. **Instructional materials:** Printed Circuits Handbook 6th Ed., 2008, Clyde Coombs
   a. **Other materials:**
   Professor's material (PowerPoint slides)

5. **Specific course information**
   a. **Catalog description:**
   This course introduces the fundamentals of designing printed circuit boards (PCBs) using industrial EDA software tool. Students will learn the key concepts, tools, and techniques used in PCB design, including schematic capture, component placement, routing, design rules, and manufacturing considerations. The course will also cover topics such as signal integrity, parasitic, coupling, controlled impedance and power distribution. The course also includes a project realization of a complex circuit using Proteus software.

   b. **Prerequisites:** 020ELNES2 Digital Electronics

   c. **Selected Elective** for CCE and EE students

6. **Educational objective for the course**
   a. **Specific outcomes of instruction:**
   - Identify the PCB technology, design Process and materials used.
   - Master the use of Proteus EDA tool for PCB design.
   - Master the fundamental PCB Design topics: Schematic capture, Component placement, Routing, power distribution.
   - Identify and analyze the importance of signal integrity in PCB design.
   - Design and implement a complex circuit using PCB.

   b. **PI addressed by the course:**

PI	2.1	2.2	2.3	2.5	6.1	6.2	7.2
Covered	x	x	x	x	x	x	x
Assessed							

7. **Brief list of topics to be covered:**
   - Week 1: Introduction to PCB Design
     - Overview of PCB design process and materials
     - Introduction to EDA (Electronic Design Automation) tools

- PCB design standards and guidelines

– Week 2-3: Schematic Capture and Component Placement
  - Understanding schematic symbols and components
  - Creating and editing schematics
  - Symbol libraries and component management
  - Placement strategies and considerations
  - Managing footprints and component libraries
  - Placement optimization techniques
– Week 4: Routing Basics
  - Introduction to PCB routing techniques
  - Single-layer and double-layer routing
  - Net connectivity and constraints
  - Differential pairs and high-speed routing
– Week 5: Design Rule Checking (DRC)
  - Setting up design rules and constraints
  - Running DRC checks and resolving errors
– Week 6: Power Distribution
  - Power plane design and management
  - Decoupling capacitor placement and optimization
  - Power integrity analysis
– Week 7-8: Signal Integrity
  - High-frequency Signals and propagation delays
  - Transmission line theory
  - Impedance control and matching
  - Crosstalk analysis and mitigation
  - Layout design of mixed-signal boards
– Week 9-12: Project Realization