NAAN MUDHALVAN COURSE FOR

ELECTRONICS AND COMMUNICATION ENGINEERING

COURSE NAME	PRINTED CIRCUIT BOARD DESIGN
TOTAL DURATION	60 HOURS
MODE OF DELIVERY	PHYSICAL CLASSROOM TRAINING AT RESPECTIVE COLLEGES
TRAINER TO STUDENT RATIO	1:60
TOTAL MARKS	70 (External) + 30 (Internal)

OVERALL COURSE OBJECTIVE:

This course aims to equip students with the fundamental knowledge and practical skills necessary to design and layout single and double-layer printed circuit boards (PCBs) using Orcad/Cadence EDA software.

LEARNING OUTCOMES:

Upon successful completion of this course, students will be able to:

- List the types and applications of PCBs in various industries.
- Identify common electronic components and their packaging types.
- Apply the PCB design workflow from schematic capture to fabrication and assembly.
- Apply design rules and standards for single and double-layer PCBs.
- Use Orcad tools effectively for schematic capture, PCB layout design, and footprint creation.
- Design and implement any microcontroller-based circuit using Orcad.
- Employ optimal component placement strategies considering heat dissipation, signal integrity, and manufacturability.
- Implement proper routing techniques for single and double-layer PCBs, adhering to design rules and signal integrity considerations.
- Identify the problems while designing the PCB and troubleshoot them.
- Generate Gerber files for PCB manufacturing.
- Apply best practices in PCB design, including design for manufacturability (DFM), thermal management, and documentation practices.
- Design and layout a single or double-layer PCB for a specific electronic project using the acquired knowledge and Orcad tools.
- Create comprehensive PCB design documentation, including schematics, layout files, and bill of materials (BOM).

MODULE-WISE CONTENT:

Module 1: **PCB Design Fundamentals** [5 HOURS]

Introduction to PCB Design: Basics of electronic components (resistors, capacitors, inductors, ICs) and circuits-Introduction to PCB design - Evolution of PCBs -Types -Single layer - Double layer –Multi layer – Board materials - Applications across various industries - Different packaging types - Through-hole mounting type package (DIP, ZIP, SIP, PGA, etc.), Surface mounting type package (SOP, QFP, SOJ, QFJ, BGA)-Symbol & Code - PCB Design Workflow:

Schematic capture to fabrication and assembly - Electronic Components and Footprints- Understanding component data sheets- Introduction to PCB design software (e.g., Eagle, Orcad, KiCAD, Mentor Graphics, Easy EDA, Circuit Maker, Protel etc.)

Module 2: **PCB Design Software (Orcad/Cadence) Fundamentals** [10 HOURS]

Introduction to Orcad: Overview of Orcad tools (Capture CIS, PCB Editor) - User interface, navigation, and environment setup - PCB Terminologies: Key terms like layers, pads, jumper, traces, vias, etc. Units used in PCB design - Design rule basics (track width, spacing, clearances) - Layer stackup concepts - Orcad Capture CIS: Schematic capture process - Component placement, symbol creation, and netlist generation - Design rule checks (ERC) and error correction - PAD Stack Editor: Defining padstacks for different component types (THT, SMD) - Customizing pad shapes, sizes, and layers - Managing drilling and plating requirements - Creating PCB Footprints: Designing footprints using the Package Symbol Editor - Linking footprints to schematic symbols for accurate placement

Module 3: **Single Layer PCB Layout and Routing with Orcad** [10 HOURS]

Creating an 8051-based circuit schematic - Utilizing Orcad libraries and custom component creation - Implementing proper signal connections and power distribution - 8051 Library Creation: Building a custom library for the 8051 microcontroller - Defining symbol and footprint for accurate representation - Footprint Linking: Associating schematic symbols with their PCB footprints - Solder mask – Silk screen - Troubleshooting and debugging common issues - Importance of RoHS (Restriction of use of Hazardous Substances)- Waste management of hazardous materials in PCB- Environment Management Standards(EMS)

Module 4: **Double Layer PCB Layout and Routing with Orcad** [10 HOURS]

Component Placement: Strategies for optimal placement on the PCB - Considering heat dissipation, signal integrity, and manufacturability - PCB Routing: Manual and automatic routing techniques - Design rule compliance and signal integrity considerations - Implementing proper routing practices for single and double-layer PCBs - Gerber Creation: Generating Gerber files for PCB manufacturing - Understanding Gerber file formats and specifications - Exporting design data for fabrication - Best Practices: Design for manufacturability (DFM) principles - Signal integrity considerations (trace length matching, impedance control) - Thermal management techniques - Documentation and version control – Bill of Materials

Module 5: **Manual PCB Fabrication** [5 HOURS]

Manual PCB Fabrication: Schematic - PCB Layout - Transfer to copper clad board - Etching - Drilling- component placement – Testing – Finishing - Design, fabricate and test a power supply circuit using copper clad sheet

Module 6: Experiments to be completed during the course: [10 HOURS]
(minimum 5 experiments)

1. Create a schematic, generate net list and simulate an RC coupled amplifier. Place the components of RC coupled amplifier and route the connections between the components without auto-routing option and verify using design rule check.
2. Create a schematic, generate net list and simulate a High Pass filter. Place the components of High Pass filter and route the connections between the components without auto-routing option and verify using design rule check.

3. Design a PCB layout for Astable multivibrator circuit and verify using design rule check. Generate gerber file, BOM.
4. Design a PCB layout for regulated power supply, verify using design rule check and generate gerber file, BOM.
5. Create symbols and foot print for 1N4007 diode.
6. Create symbols and foot print for IC741.
7. Create symbols and foot print for BC107 transistor.
8. Create symbols and foot print for a connector.
9. Design a PCB circuit which uses a via.
10. Design a PCB circuit which uses a jumper.

Module 7 : **MINI-PROJECT (any one from the list of Mini-Projects)** [10 HOURS]

Design Project: Design and layout a single or double-layer PCB using Orcad tools. Implement the learned concepts and tools for a specific electronic project - Documentation and Presentation: Create detailed PCB design documentation

List of Mini-Projects:

1. Design a single layer PCB for a circuit that blinks an LED at a controlled rate using a timer IC.
2. Design a PCB for a light dependent resistor (LDR) based automatic light switch.
3. Design a PCB for a temperature sensor circuit that displays readings on an LCD screen.
4. Design a single layer PCB for a small audio amplifier circuit using discrete components.
5. Design a PCB for a simple buzzer alarm circuit triggered by a push button.
6. Design a single layer PCB for a basic infrared remote control transmitter circuit.
7. Design a PCB for a circuit implementing basic logic gates (AND, OR, NOT) using discrete components.
8. Design a single-layer PCB for a simple metronome circuit with adjustable tempo control.
9. Design a PCB for a circuit that measures and displays battery voltage level.
10. Design a PCB for a circuit that encodes text messages into Morse code using LEDs or a buzzer.
11. Design a double-layer PCB for a circuit that sequences multiple LEDs in a chasing pattern.
12. Design a PCB for a basic FM radio receiver circuit with an antenna and audio output.
13. Design a double-layer PCB for a simple DC motor driver circuit with variable speed control.
14. Design a PCB for a basic audio mixer circuit with multiple input channels and volume control.
15. Design a double-layer PCB for a regulated power supply circuit with multiple voltage outputs.
16. Design a PCB for a data logger circuit that records sensor data onto an SD card.
17. Design a double-layer PCB for a simple wireless communication module using Bluetooth or Wi-Fi.
18. Design a PCB for a weather station sensor that measures temperature, humidity, and pressure.

19. Design a double-layer PCB for a basic line follower robot using infrared sensors and motor control.
20. Design a PCB for a simple smart home device controller with functionalities like
21. Design a PCB for street light control system.
22. Design a PCB for Digital Logic Circuit Trainer Kit.
23. Design a PCB for RP 2040 microcontroller with LCD interfacing in double layer.
24. Design a PCB for 8051 Microcontroller trainer kit.
25. Design a PCB for PIC Microcontroller trainer kit.

PCB Design

Overall Course Outcomes:

- List the types and applications of PCBs in various industries.
- Identify common electronic components and their packaging types.
- Apply the PCB design workflow from schematic capture to fabrication and assembly.
- Apply design rules and standards for single and double-layer PCBs.
- Use Orcad tools effectively for schematic capture, PCB layout design, and footprint creation.
- Design and implement any microcontroller-based circuit using Orcad.
- Employ optimal component placement strategies considering heat dissipation, signal integrity, and manufacturability.
- Implement proper routing techniques for single and double-layer PCBs, adhering to design rules and signal integrity considerations.
- Identify the problems while designing the PCB and troubleshoot them.
- Generate Gerber files for PCB manufacturing.
- Apply best practices in PCB design, including design for manufacturability (DFM), thermal management, and documentation practices.
- Design and layout a single or double-layer PCB for a specific electronic project using the acquired knowledge and Orcad tools.
- Create comprehensive PCB design documentation, including schematics, layout files, and bill of materials (BOM).
- Design a multipage circuit schematic using active and passive components
- Place and Interconnect the Through-Hole (TH) components and Surface Mount Devices (SMD) in the layout by following the PCB standards
- Design a single and double-layer PCB layout & generate the Gerber File for PCB fabrication by calculating the trace width of the interconnect in the layout
- Create the footprint for the Through-Hole (TH) components and Surface Mount Devices (SMD)

LEARNING OUTCOME	ASSESSMENT CRITERIA	USECASES
• Hands-on experience with the 555 timer IC and its application as a stable multivibrator (oscillator). • Evaluate how resistors and capacitors influence timing circuits. • Analyse how component values	• Identify and gather all necessary components for the LED blink timer circuit. • Create a schematic diagram of the circuit using PCB design software. • Design the PCB layout, ensuring proper placement	**Use case1:** Design a single layer PCB for a circuit that blinks an LED at a controlled rate using a timer IC. **Scenario**: To design a PCB for an LED blink timer circuit to be used in a microwave oven.

affect the LED's blinking rate. • Design a basic PCB design, including component placement and trace routing. • Practice soldering techniques to securely attach components to the PCB.	and routing of components. • Verify the design by running design rule checks and simulations. • Generate and review the Gerber files for the PCB manufacturing process. • Assemble the PCB by soldering the components onto the board. • Test the completed PCB to ensure the LED blink timer circuit functions correctly.	**Task:** The student must design a PCB for an LED blink timer circuit for a microwave oven by gathering components, creating a schematic, laying out the PCB, verifying the design, generating Gerber files, assembling the circuit, testing its functionality.
• Evaluate how LDRs work and how their resistance changes with light. • Analyse how transistors work as switches, with the base current controlling the collector current. • Examine relays and how they switch circuits. • Practice basic PCB design, focusing on placing components and routing traces on a single layer. • Design a circuit, assemble it on the PCB, and troubleshoot any issues that prevent the light from switching correctly.	• Select appropriate components for an artificial lighting system for greenhouse. • Create a schematic diagram of the lighting system, including power supply. • Design the PCB layout, ensuring proper placement and routing for optimal performance. • Perform design simulations and tests to verify functionality and efficiency. • Generate Gerber files for PCB fabrication and assembly. • Present the final PCB design and system for greenhouse applications.	**Use case2:** Design a PCB for a light dependent resistor (LDR) based automatic light switch. **Scenario**: To design a PCB for an artificial lighting system used in green house. **Task:** Students will research and select components for an efficient artificial lighting system for greenhouse use, designing a PCB layout. They will design through simulations, generate Gerber files and assembling the circuit, testing its functionality.
• Connect temperature sensors to the ATmega328P and process their	• Select appropriate components, such as temperature sensors, microcontrollers,	**Use case3:** Design a PCB for a temperature sensor circuit that

signals. • Control LCD displays using the HD44780 controller. • Practice advanced PCB design, including multi-layer routing and using ground planes. • Assemble complex circuits and troubleshoot to ensure temperature readings show correctly on the LCD.	and display units. • Create a schematic diagram of the temperature monitoring circuit. • Design the PCB layout, ensuring proper placement and routing of components. • Verify the design through simulations and tests. • Generate Gerber files for PCB fabrication. • Assemble the PCB by soldering the components. • Test the assembled PCB to ensure accurate temperature monitoring.	displays readings on an LCD screen. **Scenario**: To design a PCB for a temperature monitor for an egg incubator. **Task:** Design a PCB for a temperature monitor for an egg incubator, including researching the necessary components, creating a schematic, designing the PCB layout, performing design verification and testing, generating Gerber files, assembling the PCB, and documenting and presenting the final project.
• Evaluate how text-to-speech functionality works. • Apply basic principles of audio amplifier circuits using discrete components. • Practice the embedded systems design principles, where a microcontroller interacts with separate circuits. • Develop advanced PCB design skills, considering critical factors like grounding, trace width and length, and potential use of jumper wires for single-layer limitations.	• Select appropriate discrete components such as transistors, resistors, capacitors, and an audio IC for a small audio amplifier circuit. • Design the schematic diagram of the audio amplifier circuit. • Plan and layout the PCB design, optimizing for a single layer. • Place components on the PCB layout, ensuring proper spacing and connections. • Route traces carefully to minimize noise and interference. • Verify the design by simulating the	**Use case4:** Design a single layer PCB for a small audio amplifier circuit using discrete components. **Scenario**: To design a PCB for a simple amplifier circuit for text to speech conversion. **Task:** Students will select components for a small audio amplifier circuit, designing a single-layer PCB layout with careful component placement and trace routing. They will then fabricate, assemble, and test the PCB.

	circuit performance. • Generate Gerber files for manufacturing the single-layer PCB. • Fabricate and assemble the PCB, soldering components onto the board.	
• Examine how microcontrollers can read digital inputs (push button) and control digital outputs (buzzer). • Apply the basic circuit components (resistor, button, buzzer) and their connection principles for a simple alarm system.	• select components for the buzzer alarm circuit, including a buzzer, push button, resistor, and power supply. • Create a schematic diagram of the alarm circuit, indicating connections between components. • Design the PCB layout, considering compactness and ease of assembly. • Verify the design through simulations or breadboard testing. • Generate Gerber files for manufacturing the PCB. • Fabricate the PCB and assemble the components. • Test the buzzer alarm circuit's functionality by simulating button presses and monitoring the buzzer response.	**Use case5:** Design a PCB for a simple buzzer alarm circuit triggered by a push button. **Scenario**: To design a PCB for an emergency alarm system aimed at enhancing women's safety in public transportation. **Task:** The student should Design a PCB for a simple buzzer alarm circuit triggered by a push button. This task includes selecting suitable components, creating a schematic, designing the PCB layout, verifying the design, generating Gerber files, assembling the PCB, testing the alarm circuit's functionality.
• Apply the principles of microcontrollers and their role in generating control signals. • Analyse the	• Selecting the components required for an infrared remote control transmitter circuit, such as an	**Use case6:** Design a single layer PCB for a basic infrared remote control transmitter circuit.

working of infrared communication and how it's used in remote controls.	infrared LED, resistor, capacitor, and a microcontroller or IC. • Create a schematic diagram of the transmitter circuit, indicating the connections between components. • Design a single-layer PCB layout, optimizing component placement and trace routing for efficient performance. • Verify the design through simulations or breadboard testing to ensure functionality. • Generate Gerber files for manufacturing the single-layer PCB. • Fabricate the PCB and assemble the components onto the board. • Test the infrared remote control transmitter circuit by sending signals to a compatible receiver device.	**Scenario**: To design a PCB for an infrared (IR) based remote control system for controlling home appliances. **Task:** Students will design a single-layer PCB for a basic infrared remote control transmitter circuit by selecting components, creating a schematic, and optimizing the layout for efficient performance. They will then fabricate and test the PCB.
• Evaluate the working of basic logic gates (AND, OR, NOT) function through hands-on experimentation. • Analyse the role of transistors and resistors in building logic circuits. • Build electronic circuits using discrete components based on schematics.	• Select discrete components such as transistors, resistors, and diodes for the basic logic gates (AND, OR, NOT). • Create a schematic diagram of the logic gates circuit, illustrating the connections and functionality. • Design the PCB layout, ensuring	**Use case 7:** Design a PCB for a circuit implementing basic logic gates (AND, OR, NOT) using discrete components. **Scenario:** Designing a PCB for a basic gates digital IC lab kit. **Task:** Students will design a PCB for a circuit

	efficient component placement and trace routing for each logic gate. • Verify the design through simulations or breadboard testing to confirm the logic gates' functionality. • Generate Gerber files for PCB fabrication, ensuring compatibility with manufacturing processes. • Fabricate the PCB and assemble the discrete components according to the layout. • Test each lASogic gate's functionality individually and as part of the circuit.	implementing basic logic gates (AND, OR, NOT) using discrete components by selecting components, creating a schematic, and optimizing the layout for efficient performance. They will then fabricate and test the PCB.
• Design complex embedded systems with multiple microcontrollers and functionalities. • Code microcontrollers to process sensor data, trigger alarms, and control audio/visual alerts. • Integrate separate PCBs with communication modules.	• Select components such as oscillators, timers, and potentiometers for the metronome circuit. • Create a schematic diagram of the metronome circuit, including adjustable tempo control. • Design the single-layer PCB layout, optimizing for efficient assembly and performance. • Verify the design through simulations or breadboard testing to ensure	**Use case 8:** Design a single-layer PCB for a simple metronome circuit with adjustable tempo control. **Scenario**: To design a PCB for a metronome circuit with adjustable tempo control for Intelligent Ambulance Alert System. **Task:** Students will design a single-layer PCB for a simple metronome circuit with adjustable tempo control by researching components, creating a schematic, and optimizing the layout for assembly. They will then

	adjustable tempo functionality. • Generate Gerber files for PCB fabrication, adhering to manufacturing requirements. • Fabricate the PCB and assemble the components according to the layout. • Test the metronome circuit's functionality, adjusting the tempo control to validate variable tempo settings. • Present the final working metronome circuit, explaining its operation and demonstrating adjustable tempo control.	fabricate, test the PCB's functionality of the functioning metronome circuit.
• Apply the principles and safety aspects of designing circuits for high-voltage applications like EV battery packs. • Connect various sensors (voltage, current) to microcontrollers to gather data in an EV system. • Apply techniques for sensor data acquisition, microcontroller processing, and displaying results on an LCD.	• Select components such as voltage sensors, microcontrollers, and display units for the voltage measurement circuit. • Create a schematic diagram of the circuit, including connections for measuring and displaying battery voltage level. • Design the PCB layout, ensuring efficient placement of components and clear trace routing for accurate voltage	**Use case 9:** Design a PCB for a circuit that measures and displays battery voltage level. **Scenario**: Designing a PCB for a circuit that measures and displays battery voltage levels is crucial for electric vehicles. **Task:** Students will design a PCB for a circuit that measures and displays battery voltage level by selecting components, creating a schematic, and optimizing the layout for efficient assembly. They

	measurement. • Verify the design through simulations or testing to confirm the circuit's functionality in measuring and displaying battery voltage. • Generate Gerber files for PCB fabrication, adhering to manufacturing standards and requirements. • Fabricate the PCB and assemble the components according to the layout. • Test the circuit's functionality, ensuring accurate measurement and display of battery voltage level.	will then fabricate, test the PCB's functionality.
• Program the ATmega328P microcontroller using platforms like the Arduino IDE, gaining hands-on experience. • Evaluate the working of electronic components like resistors, LEDs, and possibly a piezo buzzer, understanding how to connect them to the microcontroller. • Apply the working of Morse code, and how it communicates text using dots and dashes.	• Select components such as microcontrollers, LEDs or buzzers and resistors for the Morse code encoding circuit. • Create a schematic diagram of the circuit, incorporating control logic for accurate text-to-Morse encoding and decoding. • Design the PCB layout, optimizing for efficient assembly and clear trace routing to ensure reliable functionality. • Verify the design through simulations or testing to confirm accurate text	**Use case 10:** Design a PCB for a circuit that encodes text messages into Morse code using LEDs or a buzzer **Scenario**: Designing a PCB for a Electronic Morse code translator for blind people. **Task:** Students will design a PCB for a circuit that converts text messages to Morse code using LEDs or a buzzer, integrating microcontrollers and control logic for accurate encoding and decoding.

	message encoding into Morse code using LEDs or a buzzer. • Generate Gerber files for PCB fabrication, ensuring compatibility with manufacturing processes and standards. • Fabricate the PCB and assemble the components according to the layout, including LEDs or a buzzer for Morse code output. • Test the circuit's functionality, verifying accurate text-to-Morse encoding and decoding with visual or auditory feedback.	
• Program microcontrollers like the ATMEGA328 using tools like the Arduino IDE. • Setup and connect LEDs with current-limiting resistors to microcontrollers. • Apply the principles of basic electronic components such as resistors, capacitors, crystals, and LEDs and choose appropriate values for different circuits.	• Create a schematic diagram of the LED chaser circuit. • Select appropriate components such as LEDs, resistors, transistors, and a microcontroller or timer IC. • Plan component placement on both layers of the PCB. • Route traces for power, ground, and signal paths following design rules. • Design a solid ground plane and distribute power traces evenly. • Minimize signal interference and crosstalk between traces. • Add silkscreen markings for component	**Use case 11:** Design a double-layer PCB for a circuit that sequences multiple LEDs in a chasing pattern. **Scenario:** Designing a PCB for a directional LED chaser circuit means creating a layout for controlling LEDs to light up sequentially in a chosen direction in lift. **Task:** To design a double-layer PCB for a circuit that sequences LEDs in a chasing pattern, start with a detailed schematic layout, ensuring proper component selection and logical placement. Then, meticulously route traces for power, ground, and

	designators and labels. • Perform a design rule check (DRC) and verify schematic-to-layout correspondence. • Generate Gerber files for PCB fabrication. • Fabricate a prototype PCB and test the LED chaser circuit. • Iterate the design process based on test results for optimization.	signals, adhering to design rules to maintain signal integrity. Finally, validate the design through prototype testing, refining the PCB layout as necessary for optimal performance and manufacturability.
• Apply the basic principles of FM radio technology. • Implement dedicated ICs like the TDA7088 for FM radio reception.	• Create a detailed schematic diagram of the FM radio receiver circuit. • Select appropriate components for tuning, demodulation, audio amplification, antenna, and audio output. • Design the PCB layout, placing components to minimize noise and optimize antenna placement. • Route traces for RF signals, audio signals, power, and ground, ensuring proper trace width and spacing. • Perform design rule checks, generate Gerber files, fabricate the prototype PCB, assemble components, and test the circuit.	**Use case 12:** Design a PCB for a basic FM radio receiver circuit with an antenna and audio output. **Scenario:** Designing a PCB for basic FM radio receiver circuit for advertisement. **Task:** Design a PCB for a basic FM radio receiver by creating a detailed schematic and selecting components for tuning, demodulation, and amplification. Place and route components to minimize noise and optimize signal integrity, ensuring proper trace spacing. Perform design rule checks, generate Gerber files, fabricate the prototype, assemble, and test the circuit.
• Program microcontrollers	• Create a schematic	**Use case 13:** Design a double-layer PCB for a

like the ATmega328P. • Design a circuit using various components like microcontrollers, motor driver ICs, resistors, capacitors, and user input elements.	diagram with all required components. • Select appropriate motor driver IC, resistors, capacitors, and connectors. • Design the PCB layout, optimizing space and placing the speed control potentiometer. • Route traces for power, ground, control signals, and motor outputs. • Perform design rule checks, generate Gerber files, fabricate the PCB, assemble components, and test the circuit.	simple DC motor driver circuit with variable speed control. **Scenario**: Designing a PCB for simple DC motor driver circuit with variable speed control for E-vehicle. **Task:** Design a double-layer PCB for a DC motor driver circuit by creating a schematic, selecting components, and optimizing the layout for space and thermal management. Route power, ground, and control signal traces, then perform design checks. Generate Gerber files, fabricate the PCB, assemble it, and test for functionality and speed control.
• Mix audio signals from multiple sources and adjust their levels using volume controls in audio mixing. • Implement the passive components like potentiometers for volume control, capacitors for blocking DC voltage and passing AC audio signals, and resistors for setting gain and biasing the op-amp. • Utilise an op-amp (e.g., LM358) to combine audio signals from different channels,	• Create a schematic for the audio mixer with input channels and volume control. • Select components like op-amps and potentiometer for volume adjustment. • Design the PCB layout, arranging components for efficient routing. • Route traces for audio signals, ensuring proper isolation and minimal noise. • Perform design rule checks and electrical rule checks for	**Use case 14:** Design a PCB for a basic audio mixer circuit with multiple input channels and volume control. **Scenario:** Designing a PCB for a basic audio mixer circuit. **Task:** Design a PCB for a basic audio mixer by creating a schematic with multiple input channels and volume control, selecting appropriate components. Optimize the PCB layout, routing traces for audio signals and power. Perform design checks,

amplify the mixed audio signal, and drive an output device.	validation. • Generate Gerber files, fabricate the PCB, assemble, and test the mixer for functionality.	generate Gerber files, fabricate the PCB, assemble it, and test for proper audio mixing and volume control.
• Implement voltage regulators like the LM317 work to provide stable output voltage and how to adjust it using resistors and potentiometers. • Apply the basic power supply design, including AC/DC conversion with an external transformer, filtering and smoothing voltage with capacitors, and protecting circuits with diodes for reverse polarity.	• Design a schematic for the power supply with regulators for multiple voltage outputs. • Select components like regulators, capacitors, resistors, and connectors for each output. • Design the PCB layout, considering component placement and heat dissipation. • Route traces for power lines, ground connections, and individual voltage outputs. • Perform design rule checks and electrical rule checks for validation. • Generate Gerber files, fabricate the PCB, assemble components, and test the power supply for stability and accuracy of multiple voltage outputs.	**Use case 15:** Design a double-layer PCB for a regulated power supply circuit with multiple voltage outputs. **Scenario:** Designing a PCB for a regulated power supply circuit for electronics appliances. **Task:** Design a double-layer PCB for a regulated power supply with multiple voltage outputs by creating a detailed schematic and selecting components like regulators and connectors. Optimize the layout for efficient routing of power lines and individual outputs, then validate the design through checks, fabrication, assembly, and testing for stable and accurate voltage regulation.
• Program the ATmega328P to work with a moisture sensor using analog-to-digital conversion (ADC). • Connect a moisture sensor to a microcontroller with a voltage divider circuit and	• Create a schematic for the data logger circuit with a microcontroller, sensors, and SD card module. • Select components like a microcontroller with SD card support, sensors,	**Use case 16:** Design a PCB for a data logger circuit that records sensor data onto an SD card. **Scenario**: Designing a PCB for a data logger circuit that records

read analog data, converting it to digital values for processing.	and connectors. • Design the PCB layout, placing components logically for sensor connections and SD card slot. • Route traces for sensor signals, power lines, data lines to the SD card, and communication lines. • Perform design rule checks and electrical rule checks for validation. • Generate Gerber files, fabricate the PCB, assemble components, and test the data logger circuit for accurate sensor data recording onto the SD card	sensor data onto an SD card for irrigation fields. **Task:** Design a PCB for a data logger circuit by schematically integrating a microcontroller, sensors, and an SD card module. Optimize component placement for efficient routing of sensor signals and SD card data lines. Fabricate and test the PCB to validate accurate recording of sensor data onto the SD card.
• Evaluate the capabilities of a powerful microcontroller like the RP2040 with dual-core processing. • Integrate a GPS module for location tracking. • Apply the Bluetooth or Wi-Fi communication protocols for data transmission.	• Design schematic for the wireless module with components like a microcontroller, Bluetooth/Wi-Fi module, and antenna. • Select components suitable for wireless communication, including the microcontroller, Bluetooth or Wi-Fi module, antenna, and connectors. • Design the PCB layout, optimizing for signal integrity and antenna performance. • Route traces for power, ground, communication lines between components, and antenna	**Use case 17:** Design a double-layer PCB for a simple wireless communication module using Bluetooth or Wi-Fi. **Scenario**: Designing a PCB for a simple wireless communication module for livestock monitoring using Bluetooth or Wi-Fi **Task:** Design a double-layer PCB for wireless communication with a microcontroller, Bluetooth/Wi-Fi module, and antenna. Optimize layout for efficient routing, ensure compatibility, and validate stable connectivity through fabrication and testing.

	connections. • Perform design rule checks and electrical rule checks for validation. • Generate Gerber files, fabricate the PCB, assemble components, and test for reliable wireless connectivity using Bluetooth or Wi-Fi.	
• Communicate with the BME280 sensor using the I2C communication protocol on the RP2040. • Read and process sensor data such as temperature, humidity, and pressure from the BME280 sensor. • Control an LCD display to present the processed data on the RP2040.	• Design schematic for the weather station sensor with temperature, humidity, and pressure sensors, and a microcontroller. • Select appropriate sensors, a microcontroller, and passive components for the PCB. • Design the PCB layout to optimize space and minimize interference between sensor circuits. • Route traces for sensor connections, power lines, and communication lines to the microcontroller. • Perform design rule checks (DRC) and electrical rule checks (ERC) for accuracy and compatibility. • Generate Gerber files, fabricate the PCB, assemble components, and test the sensor for precise weather data measurement	**Use case 18:** Design a PCB for a weather station sensor that measures temperature, humidity, and pressure. **Scenario:** Designing a PCB for a weather station sensor that measures temperature, humidity, and pressure in fire crackers industries. **Task:** Design a PCB for a weather station sensor by integrating components for temperature, humidity, and pressure measurement. Ensure proper sensor selection, layout optimization for accurate data collection, and validate functionality through testing. Fabricate the PCB, assemble components, and test for reliable weather data acquisition.

• Analyse the digital sensor data from IR sensors for line detection using the RP2040. • Process the sensor data to make decisions about robot movement based on the IR sensor inputs. • Control the L298N motor driver with the RP2040 to manage the direction and speed of DC motors in response to the sensor data.	• Create a schematic for the line follower robot with infrared sensors, motor drivers, and a microcontroller. • Select components like infrared sensors, motor drivers, a microcontroller, motors, wheels, and passive components. • Design the PCB layout for efficient routing of sensor connections and motor control signals. • Route traces for sensor connections, motor control signals, power lines, and microcontroller communication. • Perform design rule checks (DRC) and electrical rule checks (ERC) for functionality and compatibility. • Generate Gerber files, fabricate the PCB, assemble components, and test the robot's line following and motor control. • Write and upload control software to the microcontroller for line following behaviour and motor control based on sensor inputs.	**Use case 19:** Design a double-layer PCB for a basic line follower robot using infrared sensors and motor control. **Scenario**: Designing a PCB for a basic line follower robot for nuclear power plant using infrared sensors and motor control **Task:** Design a PCB for a line follower robot with infrared sensors and motor control by creating a schematic layout integrating these components. Select and place components strategically for efficient routing of sensor connections and motor control signals. Validate the design through checks, fabrication, assembly, and testing for accurate line tracking and motor control behaviour.
• Read user input from buttons using the RP2040 to control connected devices. • Control a relay	• Define functionalities for device control, sensor integration, and communication	**Use case 20:** Design a PCB for a simple smart home device controller with functionalities.

with the RP2040 for on/off switching of a smart bulb. • Control an RGB LED using the RP2040 for visual status indication.	protocols. • Create schematic with microcontroller, sensors, communication modules, and relays. • Select components like microcontrollers (e.g., Arduino), sensors (e.g., temperature, motion), communication modules (e.g., Wi-Fi), and relays. • Design PCB layout for efficient space usage and signal routing. • Route traces for power lines, ground, sensors, communication, and relay outputs. • Prototype and test the controller with Gerber files, PCB fabrication, component assembly, and functional testing. • Develop control software for device management, sensor readings, and communication protocols	**Scenario:** Designing a PCB for a simple smart home device controller for smart bulb. **Task:** Design a PCB for a line follower robot with infrared sensors and motor control by creating a schematic layout integrating these components. Select and place components strategically for efficient routing of sensor connections and motor control signals. Validate the design through checks, fabrication, assembly, and testing for accurate line tracking and motor control behaviour.
• Evaluate the resistance changes of an LDR with varying ambient light levels. • Relay module switches power based on the LDR circuit's control signal. • Use a comparator for comparing LDR voltage with a reference voltage	• Plan automatic street light control based on LDR and relay switching. • Design a schematic with LDR, relay, microcontroller (if used), and power supply. • Choose components like LDR, relay, microcontroller (if used), power	**Use case 21:** Design a PCB for street light control system. **Scenario**: Designing a PCB for an automatic street light control system using LDR and relay. **Task:** Design a PCB for automatic street light

to detect specific light level thresholds for relay activation.	supply components, and connectors. • Arrange components for efficient routing of LDR, relay, power lines, and ground on the PCB layout. • Connect LDR, relay control signals, power lines, and ground traces, ensuring signal integrity. • Perform design rule checks (DRC) and electrical rule checks (ERC) to validate compatibility and correctness.	control with LDR and relay by planning, schematics, and component selection. Arrange components efficiently, validate the design, and ensure compatibility before fabrication.
• Apply adders (U1) to perform binary addition, observing sum, carry out, and potential overflow. • Evaluate subtraction functionalities, understanding borrow out and potential underflow. • Implement sequential circuits like flip-flops (U3-U5), learning about D, JK, and T flip-flops and control using switches for clock, reset, or data inputs.	• Create schematic with adder, sub-tractor, and basic flip-flops. • Select appropriate ICs and passive components. • Design PCB layout for clear signal routing. • Route traces for signals, power, and ground. • Validate design with checks for accuracy and compatibility. • Fabricate PCB, assemble components, and test circuit functionality.	**Use case 22:** Design a PCB for Digital Logic Circuit Trainer Kit. **Scenario**: Design a PCB for a Digital Logic Circuit Trainer Kit featuring adder, sub-tractor circuits, and basic flip-flops. **Task:** Design a PCB for a Digital Logic Circuit Trainer Kit featuring adder, sub-tractor circuits, and basic flip-flops. Incorporate these components into the schematic and arrange them logically on the PCB layout for efficient routing. Validate the design through checks, fabrication, assembly, and testing to ensure functionality in arithmetic operations and sequential logic.
• Write code for the RP2040 (U1) to	• Create schematic for RP2040	**Use case 23:** Design a PCB for RP 2040

communicate with the RTC module (RTC1) using I2C or SPI protocol to retrieve time data. • Process the retrieved time data and formatting it for display on the LCD, including hours, minutes, seconds, and potentially date format. • Control the LCD using its GPIO pins to send data and control signals for displaying the formatted time.	microcontroller and LCD interfacing. • Choose components for microcontroller, LCD, and necessary passive elements. • Design double-layer PCB layout for efficient placement of components. • Route traces for power, ground, data lines, and control signals. • Validate design with checks for signal integrity and electrical compatibility. • Fabricate PCB, assemble components, and test LCD interfacing with RP2040 microcontroller	microcontroller with LCD interfacing in double layer. **Scenario**: Designing a PCB for a RP 2040 microcontroller with LCD interfacing in double layer for displaying Real-time-clock. **Task:** Design a double-layer PCB for RP2040 microcontroller with LCD interfacing by creating a schematic, selecting components, and optimizing layout for efficient routing. Validate design with checks for signal integrity, then fabricate, assemble, and test the circuit for functionality.
• Configure input/output (I/O) pins on the 8051 microcontroller (U1) to read switch states (SW1-SW8). • Control LEDs (LED1-LED8) based on switch states using the 8051 microcontroller in the educational project. • Analyse the interaction of microcontrollers with external components like switches and LEDs through I/O pins.	• Develop a schematic for the 8051 Microcontroller trainer kit with provisions for 8-bit LED display and switch inputs. • Select appropriate 8051 microcontroller, LEDs, switches, and passive components for the PCB. • Design the PCB layout considering the arrangement of LEDs, switches, and necessary connections. • Route traces for LED connections, switch inputs, power lines, and ground connections on	**Use case 24:** Design a PCB for 8051 Microcontroller trainer kit. **Scenario**: Designing a PCB for 8051 Microcontroller trainer kit using 8bit LED and Switch. **Task:** Design a PCB for a 8051 Microcontroller trainer kit with 8-bit LED display and switch inputs by creating a schematic, selecting components, and optimizing layout. Route traces for LEDs, switches, power, and ground, then validate and test the functionality for a complete trainer kit experience.

	the PCB. • Validate the design with checks for signal integrity, compatibility, and proper functionality. • Fabricate the PCB, assemble components accurately, and test the LED and switch functionalities on the trainer kit	
• Scan and decode user input from a matrix keypad (K1) using the PIC microcontroller for access code entry. • Implement code validation logic with conditional statements and loops to check for the correct access code. • Control a relay module (RL1) based on the entered code to activate/deactivate an external device, such as a siren, for alarm functionalities. • Drive an LCD display (LCD1) to show system status messages like armed, disarmed, or alarm triggered.	• Develop a schematic for the trainer kit with provisions for the keypad, LCD display, and switch inputs. • Select appropriate components such as the PIC microcontroller, LCD, relay, switches, and passive components for the PCB. • Design the PCB layout, considering the arrangement of the relay, switches, and necessary connections. • Route traces on the PCB for the matrix keypad connections, switch inputs, power lines, and ground connections. • Validate the design by checking signal integrity, compatibility, and functionality. • Fabricate the PCB, accurately assemble	**Use case 25:** Design a PCB for PIC Microcontroller trainer kit. **Scenario**: Double-Layer PCB Design for PIC Microcontroller Security System Trainer Kit. **Task:** Design a PCB for a PIC Microcontroller trainer kit for security system by creating a schematic, selecting components, and optimizing layout. Route traces for switches, Relays, power, and ground, then validate and test the functionality for a complete trainer kit experience.

	components, and test the relay and matrix switch functionalities on the trainer kit.	

Course Duration: 60 Hours

Test Projects:

1. Design a single-layer PCB for a circuit that blinks an LED at a controlled rate using a timer IC.
2. Design a PCB for a light-dependent resistor (LDR) based automatic light switch.
3. Design a PCB for a temperature sensor circuit that displays readings on an LCD screen.
4. Design a single-layer PCB for a small audio amplifier circuit using discrete components.
5. Design a PCB for a simple buzzer alarm circuit triggered by a pushbutton.
6. Design a single-layer PCB for a basic infrared remote control transmitter circuit.
7. Design a PCB for a circuit implementing basic logic gates (AND, OR, NOT) using discrete components.
8. Design a single-layer PCB for a simple metronome circuit with adjustable tempo control.
9. Design a PCB for a circuit that measures and displays battery voltage level.
10. Design a PCB for a circuit that encodes text messages into Morse code using LEDs or a buzzer.
11. Design a double-layer PCB for a circuit that sequences multiple LEDs in a chasing pattern.
12. Design a PCB for a basic FM radio receiver circuit with an antenna and audio output.
13. Design a double-layer PCB for a simple DC motor driver circuit with variable speed control.
14. Design a PCB for a basic audio mixer circuit with multiple input channels and volume control.
15. Design a double-layer PCB for a regulated power supply circuit with multiple voltage outputs.
16. Design a PCB for a data logger circuit that records sensor data onto an SD card.

17. Design a double-layer PCB for a simple wireless communication module using Bluetooth or Wi-Fi.
18. Design a PCB for a weather station sensor that measures temperature, humidity, and pressure.
19. Design a double-layer PCB for a basic line follower robot using infrared sensors and motor control.
20. Design a PCB for a simple smart home device controller with functionalities like light control or temperature regulation.

Student Assessment Plan:

Each of the above-mentioned test projects will be divided into at least 5 tasks by the training partner for each specific institution. Such tasks will be jointly evaluated by the faculty and the training partner and the following weightage is to be followed.

- 70% weightage to the external practical assessment.
- 30% weightage to the internal assessment.

Final Test Project/External Assessment Plan:

The Final Test Project will be chosen from the list given above, jointly by the college faculty and the Training Partner. The Final Test Project will be assessed on the following tasks, for 70 marks:

Task	Description	Marks
1	Workplace Safety & Setup	10
2	Component Selection	5
3	Multipage Schematic Design	10
4	Placement & Routing	20
5	DRC Checks	5
6	Microcontroller Programming	10
7	Testing and Verification	10