# CIRCUIT DESIGNING

## CURRICULUM

### » INTRODUCTION TO CIRCUIT DESIGNING

- Need of Circuit Designing
- Introduction to Basic Components like Resistor, Capacitor, Inductor etc
- How to select components
- Basic Circuit Designing Process

### » INTRODUCTION TO PCB DESIGNNG

- Types of PCBs
- Machines used for designing
- The Designing Process

### » EXPRESS SCH

- Beginning a New Schematic
- Placing Items in the Schematic
- Placing Symbols and Ports
- Labeling components
- Editing the Schematic
- Creating New Components
- Working with Sheets and Ports
- Checking the Schematic for Errors

### » EXPRESS PCB

- Beginning a New Layout
- Placing Items in the Layout
- Editing the Layout
- Placing Power and Ground Planes
- Changing the Board's Perimeter
- Creating New Components
- Linking the Schematic and PCB
- Keyboard Shortcuts
- Troubleshooting errors
- Checking the Schematic for Errors

### » PROTEUS

- Introduction to Proteus
- Components Libraries
- Components Properties
- Interfacing for circuitry
- Burning IC in Proteus
- Circuit Simulation
- Connection Trouble Shooting

### » PCB PRINTING AND ETCHING

- Introduction to printing process
- Placing of circuit on copper clad
- Etching process for final PCB

### » DRILLING AND SOLDERING

- Drilling of designed PCB
- Soldering Process of Components
- Testing & Troubleshooting

### » OTHER CAD SOFTWARES FOR PCB

- PadPad
- Multisim
- Eagle
- OrCAD
- Altium

## » PCB Design with ALTIUM Schematic Entry, Libraries, and Designing Components Schematic Entry

- Creating a Project and Adding Files
- Adding Parts to your Schematic Sheet
- Component Design
- Schematic Component Design
- PCB Foot Print Design
- IPC Footprint Wizard
- PCB Component Wizard
- Setting a Component's Footprint
- Creating a Design
- Adding Parts
- Connecting Parts  Schematic Creating a PCB Project
- Adding a Schematic Sheet
- Project Window
- Datasheet Diagram of LP
- Adding Parts to your Schematic
- Adding a part to your schematic
- Library Menu
- Available Libraries Window
- Creating a Component
- Blank Component with Pins
- Adding a Rectangle body (left) and the Final Design (Right)
- Rectangle Properties
- Editing Component Properties
- Editing Component Pin Assignments
- Sheet Symbols
- Repeated Schematic Symbol Design
- Finished Schematic Part
- Package Dimensions
- Package Dimensions
- IPC Footprint Wizard
- | P a g e
- IPC Footprint Wizard Component Selection
- Footprint Dimensions
- More dimension Info

- Solder Fillet Density
- Footprint Name
- Footprint Save Location
- Final Footprint
- PCB Component Wizard
- Component Wizard Component Pattern
- Finding a Footprint
- Browsing for Footprints
- PCB Model Window
- Schematic of Power Module
- Changing Passive Component Footprint
- Adding the  Standard Footprint
- Wired and Net list connections
- Power, Ground, Port, and Off  Sheet Symbols
- Creating a Schematic Sheet Symbol
- Selecting a Schematic Sheet
- Top Level Schematic with Sheet Symbol
- Repeated Schematics
- Project Options Menu
- Un-routed Schematic Sheets
- Base Room for Copied Routing
- Copying Room Formats
- Copied Room Layout Format

---

**DUCAT** ®
development | consultancy | training

E-mail: info@ducatindia.com
Visit us: www.ducatindia.com
www.facebook.com/ducateducation