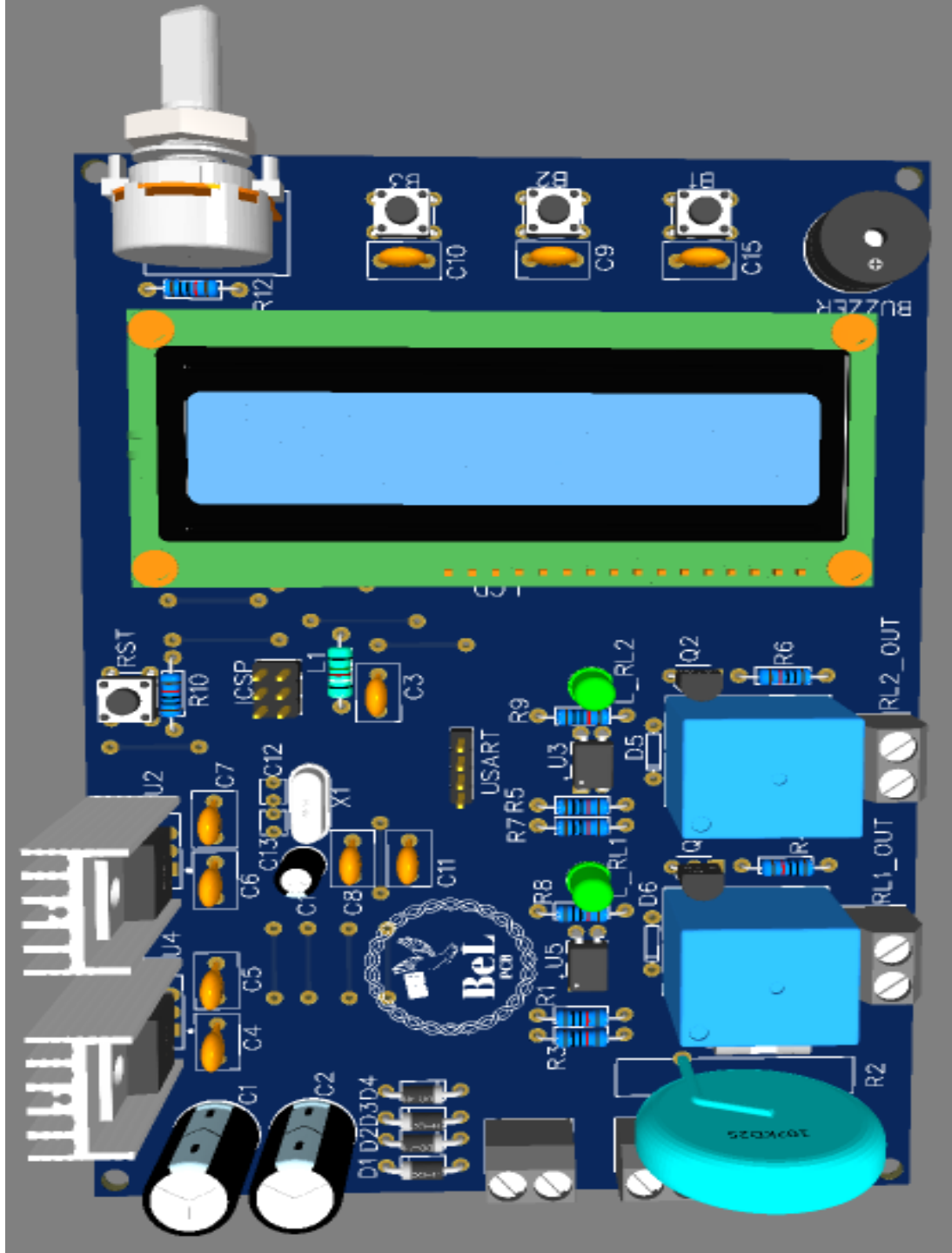
Bel PCB Workshop

6 DAYS LIVE BOOTCAMP



Purpose

- **Circuit Design**

we aim at giving you the possibility to DIY your Basic PCB Prototypes at home. Our design circuit will be that of an exposure unit for PCB fabrication with an SMD Atmega328P as controller.

Purpose

- **PCB Design**

During this phase, you will learn how to design PCB from your circuit. This part is the most crucial part because we have realized that most designs we receive don't take into account signal integrity, EMI, and trace width calculation using IPC standards.

Purpose

- **PCB Fabrication**

This is where you shall learn to DIY your PCB easily at home using chemicals and Presensitized PCB.

Outcome

- You will have a solid understanding of basics of Electronics design
- You will learn new PCB design tools
- You will be introduced to SMD component technologies
- You will be able to design your own PCB taking signal integrity into account
- You will have a deep understanding of PCB fabrication process
- You will have necessary skills for more complex hardware designs

LET'S GET STARTED

OUTLINE

- Circuit Design:
  - What's circuit design?
  - Circuit design steps
  - Design software
  - Case Study
- PCB Design:
  - Introduction into PCB Design
  - Good Practices
  - PCB Design Rules
  - Case Study

OUTLINE

- PCB Manufacture:
  - Manufacturing Process
  - Case Study

CIRCUIT DESIGN

To **Design** means to choose the suitable components that will implement the desired control while allowing the various voltage/current quantities to flow without being destroyed.

Circuit Design Steps

- Make a block Diagram of the solution
- Use reference Documents (datasheet)
- Analyze and design each block (design for constraints)
- Separate circuits into blocks/sections during schematic development

Circuit Design Steps

- Simulate and amend the circuit if necessary and chose right component.
- Test the circuit on breadboard or PCB prototype (for high power circuits).
- Bring necessary modifications from testing.
- Test the circuit again.
- Test it again to make sure everything is working well.

Good News:

You don't need to memorize all these but practice them instead!!!

Case Study: Exposure Unit Schematic

Proteus & EasyEDA design and simulation.

- **Design Calculations:**
  - ***Protection Circuit Design***

    1. *Varistor (VDR)*

- Taking a max input at the V_{in} = 12V voltage regulator at 20V
- Transformer rating 220V/12V
- V_{rms} = 20V/√2 = 14.14V
- Input max Voltage of Trans = 259V
- Voltage range 220 ≤ $V_{inTrans}$ ≤ 259
- We choose a VDR of 275V (Closest value to 259v)

Case Study: Exposure Unit Schematic

2. *Fuse*

Assuming max circuit current to be 1A

Determining $I_{InTrans}$ from transformer turn ratio

$$I_{InTrans} = \frac{1A \times 12V}{220V} = 54\ mA$$

Clamping voltage maximum current is 10A

Minimum available fuse rate 1.5A

Case Study: Exposure Unit Schematic

- ***Relay Circuit Design***

  1. *Relay*

- Start with load: 4 lamps, 18w each (48w, 220v)
- I_{rms} = 48/220 = 0.218A
- Relay contact current ≥ 0.218A (I_D = 30mA)
- Relay operating voltage 12V (higher coil resistance, lower current)

Case Study: Exposure Unit Schematic

2. Choix of Transistor

- JFET voltage control device (zero current at steady state), and small current required to charge gate capacitance during transient state.
- Choice of JFET is dependent on relay contact current (2N7000, V_{BR} = 60V, V_{GS} = 0.8 to 3V, I_D = 0.2A)

Case Study: Exposure Unit Schematic

3. Operating Point Setting

- Optocoupler act as a switch
- Choose minimum operating point of optocoupler (I_C = 1mA)
- Minimum saturation voltage, I_F = 5mA.
- Calculating foreword resistance (R_F)
- $R_F = \frac{V_G - V_F}{I_F} = \frac{5-1.4}{5\times10^{-3}} = 720\Omega$

PCB Design

❑**Introduction into PCB Design**

- It’s always a daunting process to start your PCB design layout.
- Many components are involved and its difficult to sort a starting point.
- The key resides into how well you structured your schematic.

PCB Design

- ***Terms***

PCB Design

❑Good Practices

- First and foremost thing, PCB planning!!!
- Plan PCB during schematic development.
- Avoid the use of auto placement and auto routing.
- The magic is in the datasheet.

PCB Design

- **Rules of Thumbs**
  - ***Phase 1: Component placement***
    - Ensure all the components are out of the board outline.
    - Place components as close as possible to the pins they should be connect to; That is decoupling capacitor & inductor, sensitive circuitry.
    - Solve conflicts to at least 80% to ease routine.

PCB Design

❑***Phase 2: Routine***

- Has most of the conflicts be resolve during components placements??

  Why? Return to Phase 1.
- Begin with priorities signals and direct connections signals
- Don't connect ground signals
- Avoid crosstalk (for switching signals)

NB: Design according to manufacturer specification

PCB Design

Voltage between conductors(DC or AC peaks)	Min spacing for Internal conductors	Min spacing for uncoated External conductors	Min spacing for coated External conductors
0-15	0.05mm	0.1mm	0.05mm
16-30	0.05mm	0.1mm	0.05mm
31-50	0.1mm	0.6mm	0.13mm
51-100	0.1mm	0.6mm	0.13mm
101-150	0.2mm	0.6mm	0.4mm
151-170	0.2mm	1.25mm	0.4mm
171-250	0.2mm	1.25mm	0.4mm
251-300	0.2mm	1.25mm	0.4mm
301-500	0.25mm	2.5mm	0.8mm

Table-electrical Conductor Spacing (source IPC-2221A)