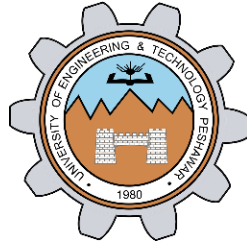


LAB 01: INTRODUCTION TO PROTEUS



Spring 2024

Submitted by: **Hassan Zaib**

Registration: **22pwsce2144**

Class Section: **A**

"On my honor, as student of University of Engineering and Technology, I have neither given nor received unauthorized assistance on this academic work."

Submitted to:

Engr. Usman Malik

Month Day, Year (08 March 2024)

Department of Computer Systems Engineering

University of Engineering and

Technology, Peshawar

Introduction to Proteus

Objectives:

The primary objective of this lab is to introduce students to the Proteus Design Suite, a comprehensive software tool used for electronic design automation (EDA). By the end of this lab, students will be able to create and simulate basic circuits using Proteus.

Overview of Proteus:

Proteus Design Suite is a sophisticated software suite that provides tools for the design and simulation of electronic circuits. It is widely used by electronic design engineers and technicians for creating schematics and layouts for printed circuit boards (PCBs). The suite includes features for both schematic capture and PCB design, making it an all-in-one solution for electronic design automation.

A Guide to Using Proteus

Starting Proteus:

1. Launching the Software:

- Open Proteus from your desktop or start menu.
- You will be greeted with the main window of Proteus.

Figure 1: Main window of Proteus

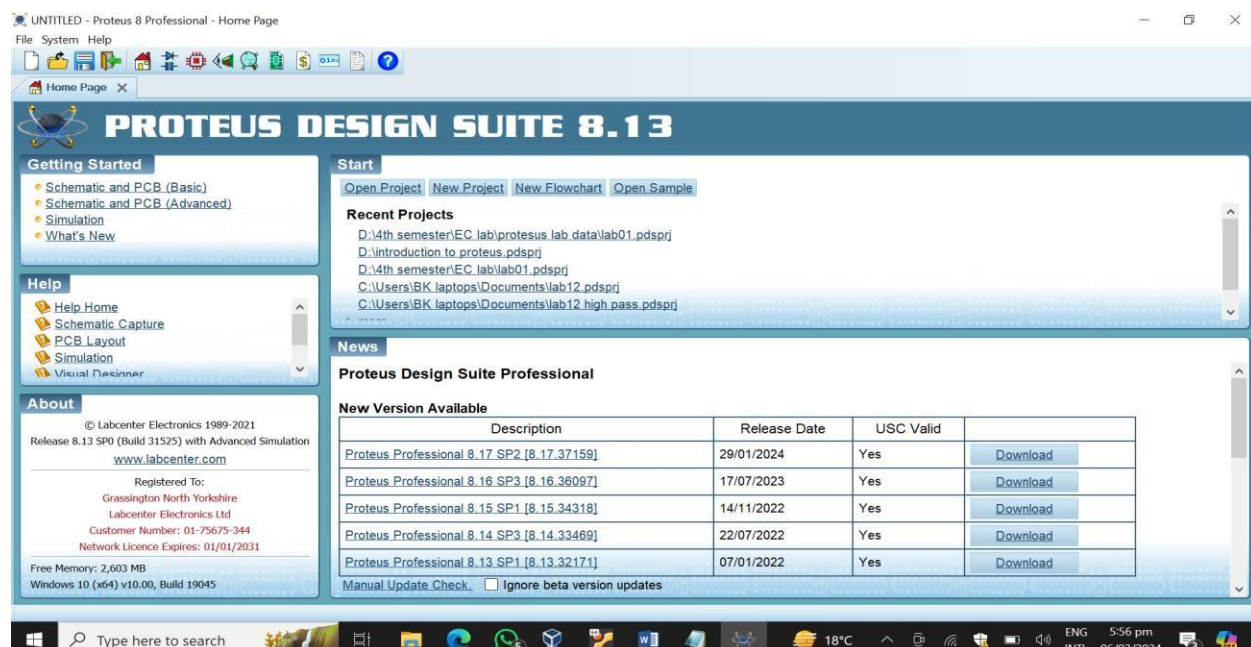


Figure 1: Main Window of Proteus

2. Creating a New Project:

- Click on "New Project" to begin creating a new project.

Figure 2: Starting a new project in Proteus



Figure 2: Creating New Project

3. Naming the Project:

- Enter a name for your project in the provided field.

Figure 3: Naming the project

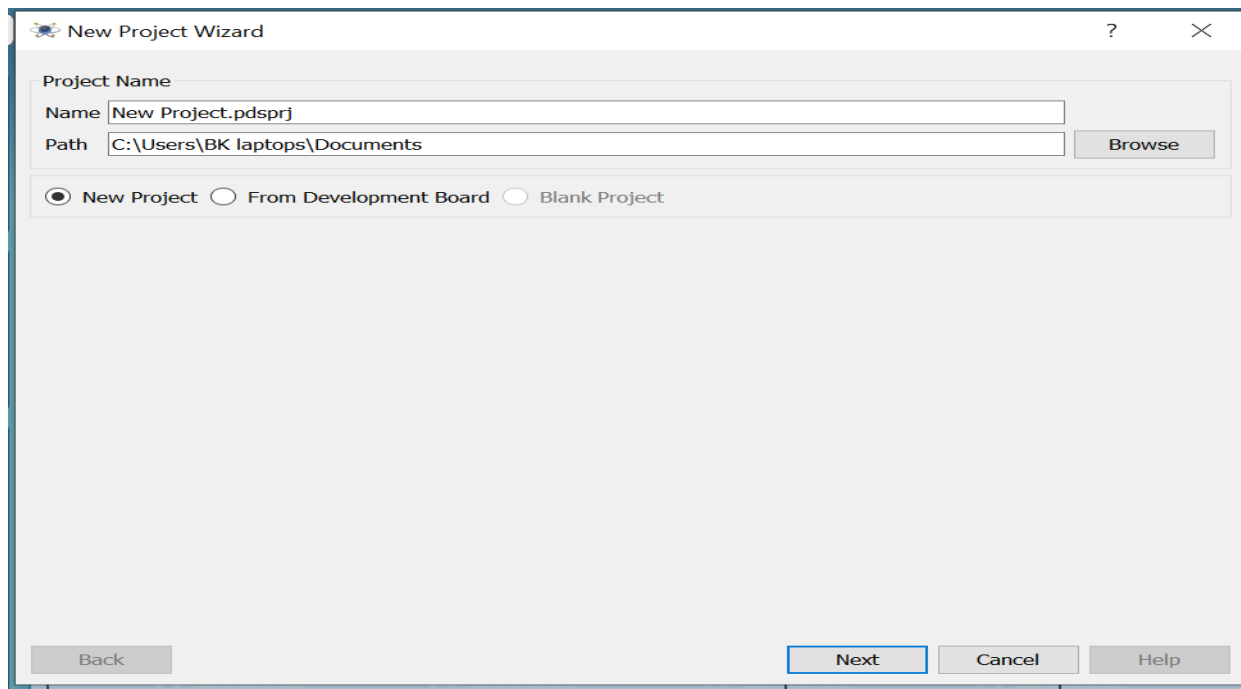


Figure 3: Naming the project

4. Selecting Page Size:

- Choose A4 as the page size for your project layout.

Figure 4: Selecting Portrait A4 page

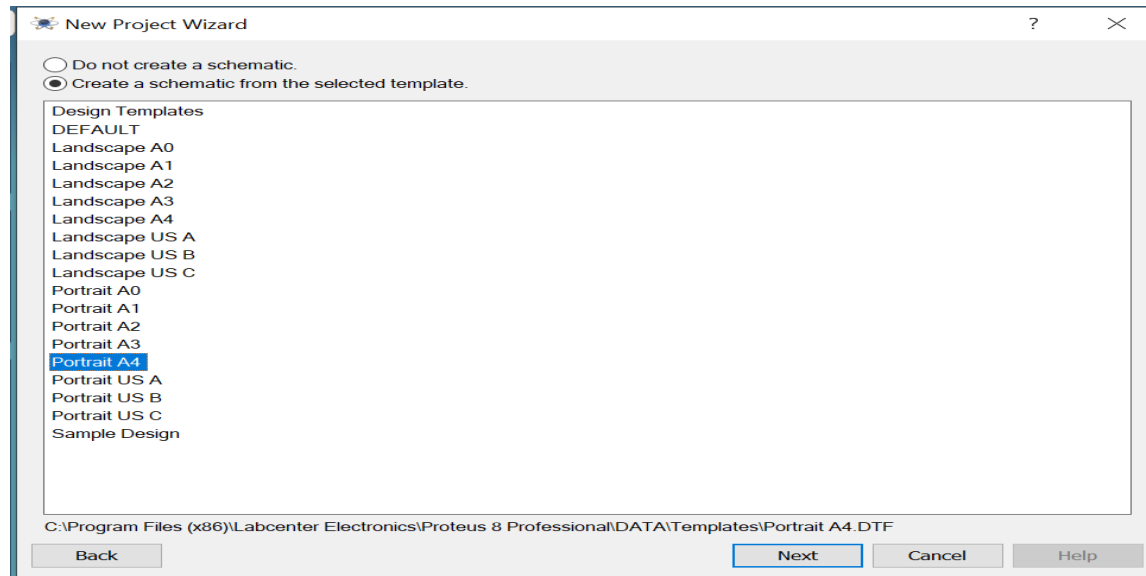


Figure 4: Selecting Page Size

5. Choosing PCB Layout:

- Select "Single Eurocard (2 layer)" for the PCB layout. This layout is suitable for many standard PCB designs.

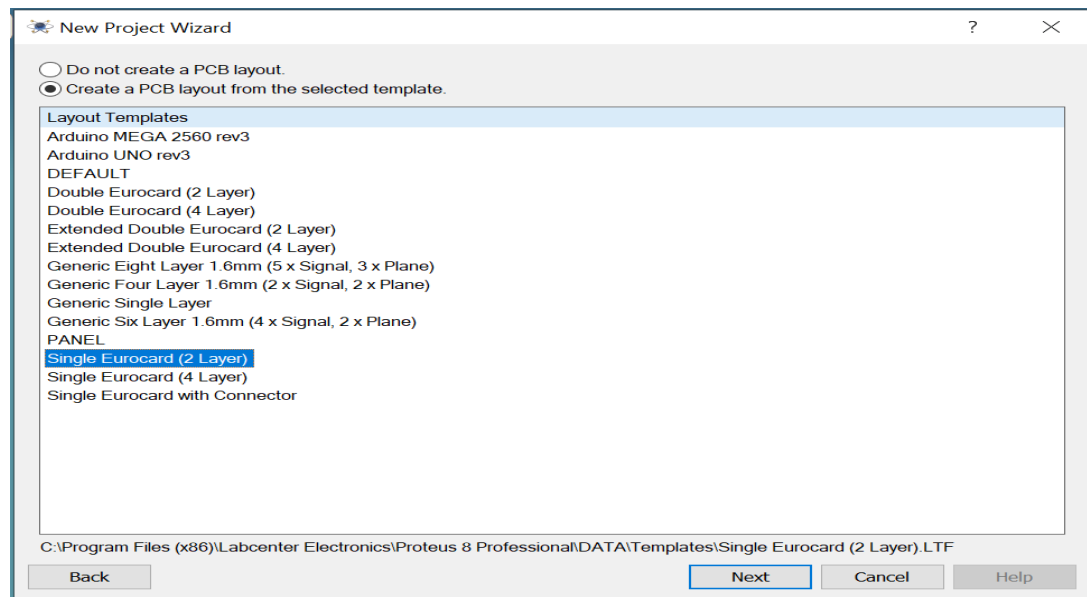


Figure 5: Single Eurocard (2 layer) selection

6. PCB Layout Screen:

- After selecting the PCB layout, the main PCB design screen will be displayed.

Figure 6: PCB Layout screen

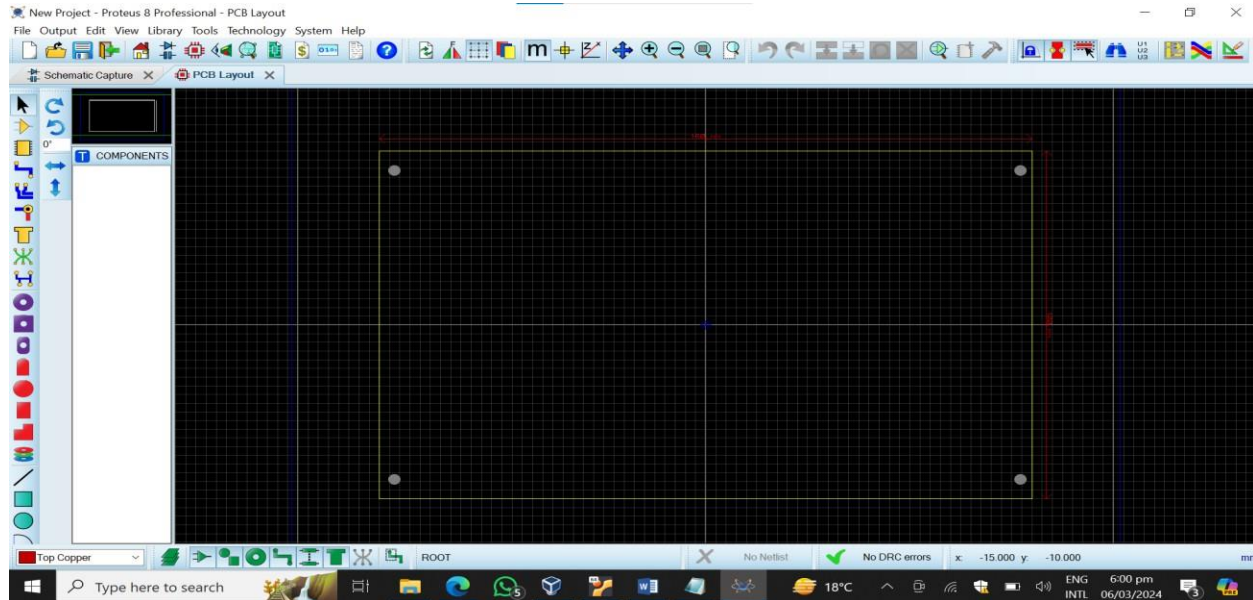


Figure 6: PCB Layout

7. Schematic Capture:

- Switch to the schematic capture interface to start creating your circuit.

Figure 7: Schematic Interface

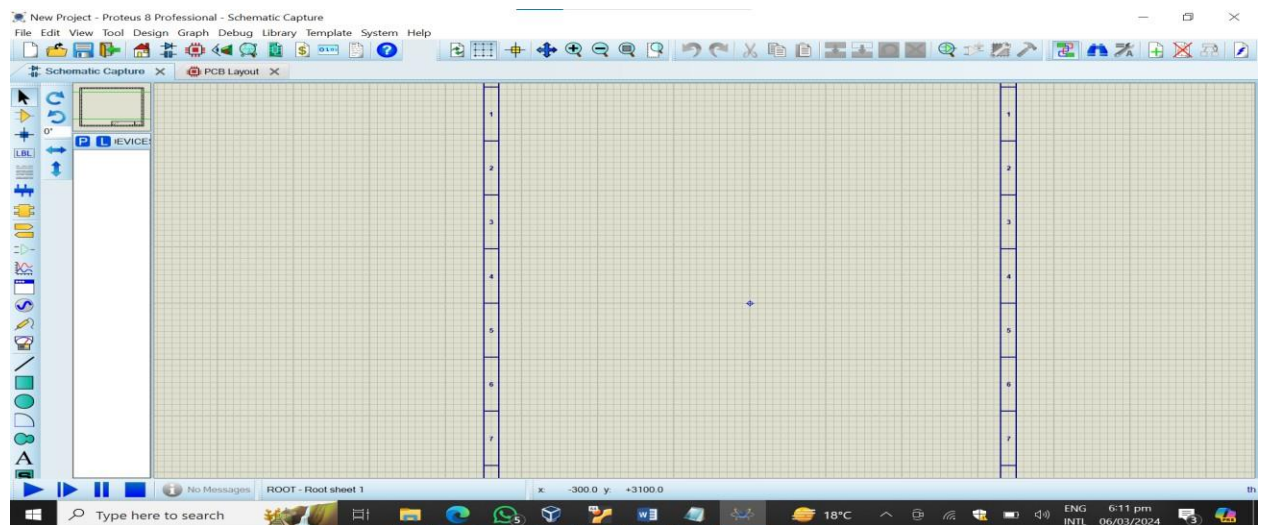
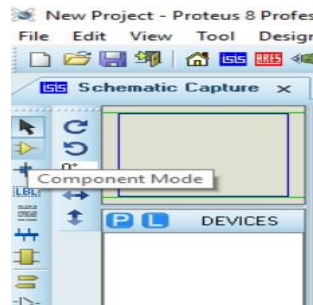


Figure 7: Schematic Interface

8. Adding Components:

- Click on the component mode to add electronic components to your circuit. You can search for components using the search bar in the library.

Figure 8: Component Mode



Creating a Simple Circuit:

1. Adding a Resistor:

- In the component mode, search for "Resistor" and add it to your schematic.

Figure 9: Adding a resistor to the schematic

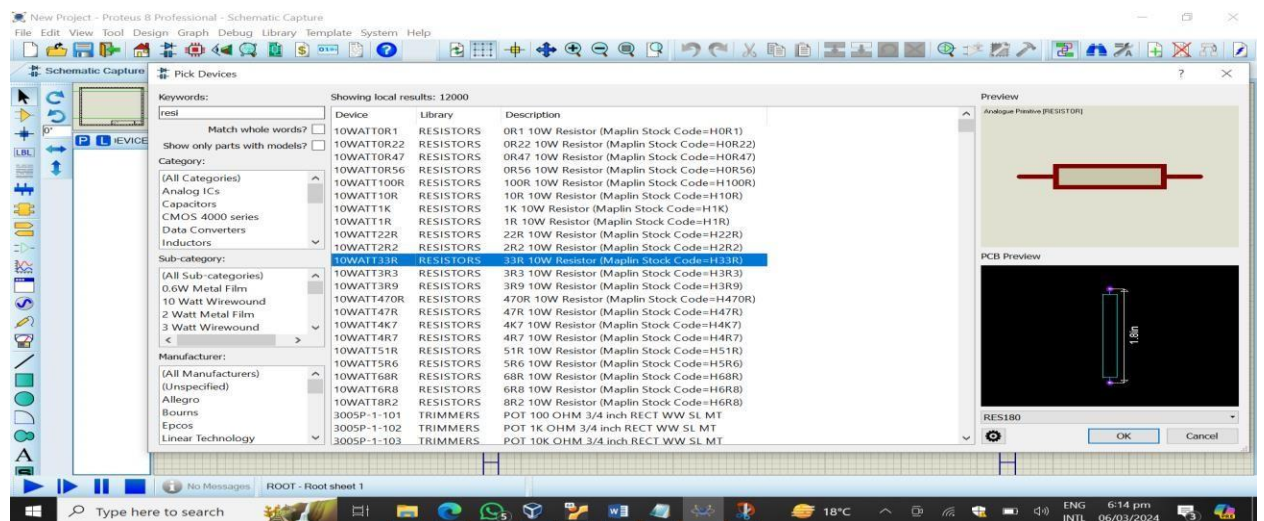
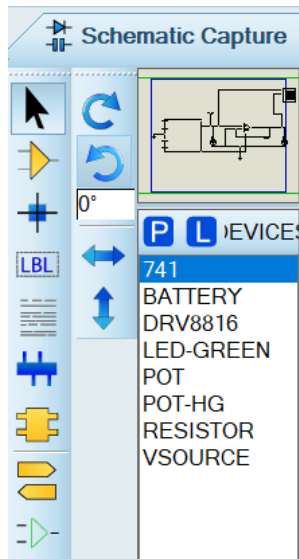


Figure 9: Pick Devices

2. Adding a Power Source:

- Similarly, search for a power source (e.g., DC Voltage Source) and add it to the schematic.

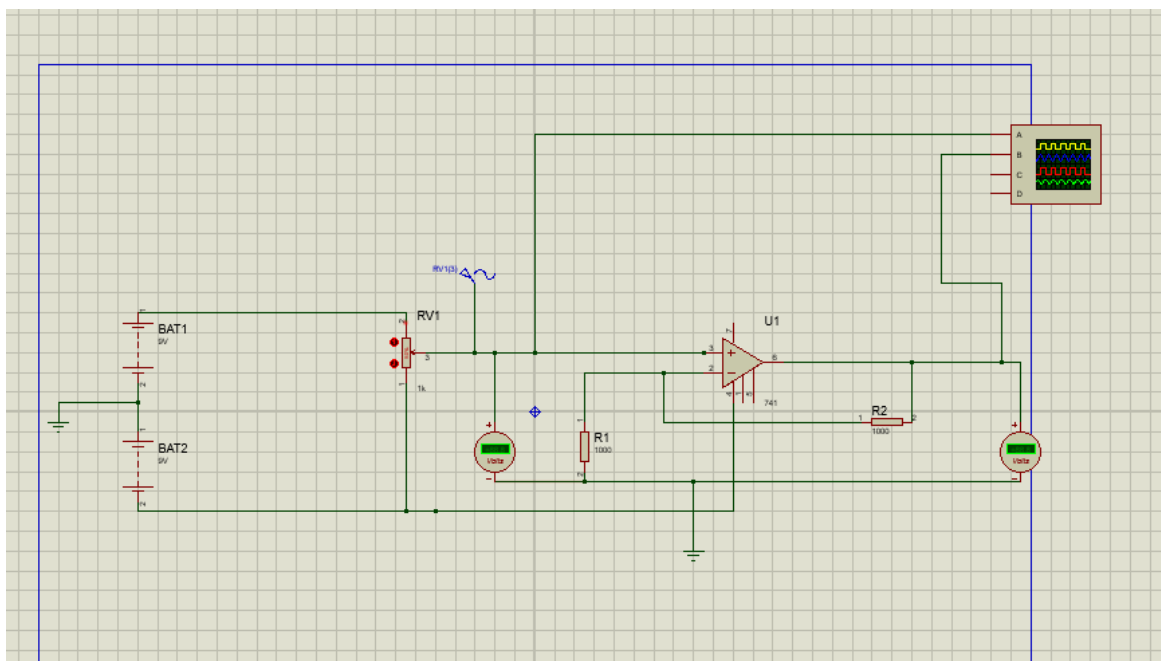
Figure 10: Adding a power source to the schematic



3. Connecting Components:

- Use the wiring tool to connect the components. Click on the terminal of one component and drag the wire to the terminal of the next component.

Figure 11: Connecting components with wires



4. Simulating the Circuit:

- Once your circuit is complete, you can simulate it by clicking on the "Play" button. This will run the simulation and display the results.

Figure 12: Running the simulation

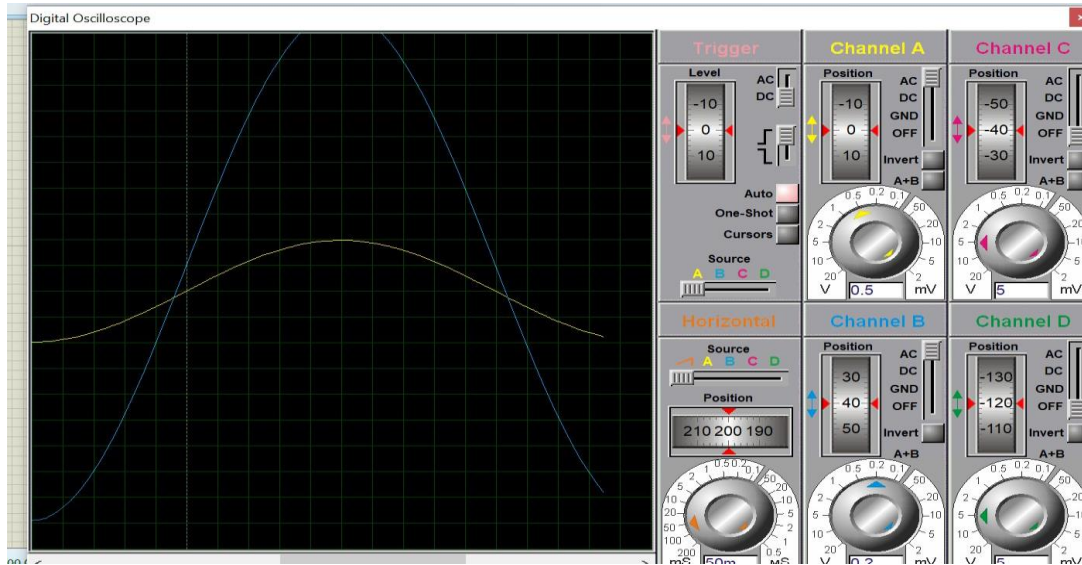


Figure 13: Oscilloscope (graphical representation of output signal)

PCB Layout:

Now for pcb layout first add all the component in packaging tool.

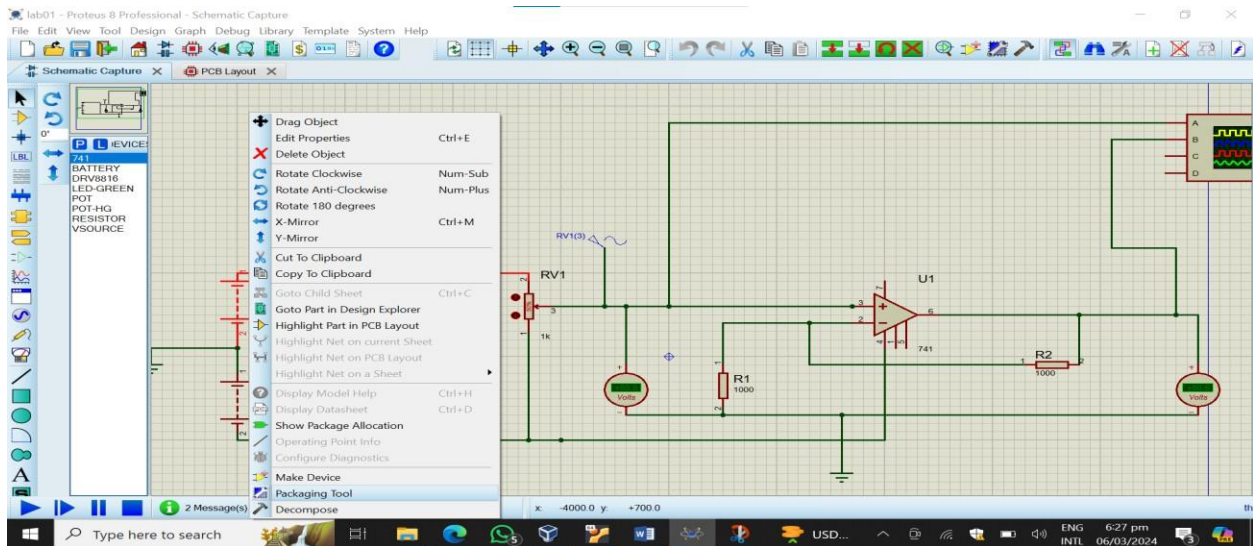
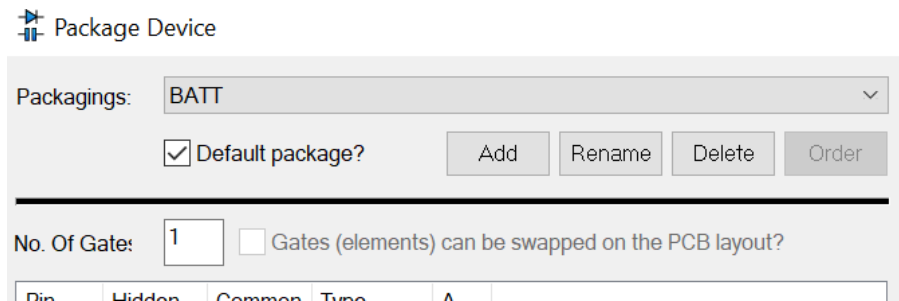


Figure 14: Packaging tool

Right click on the component and select packaging tool.



Package Device

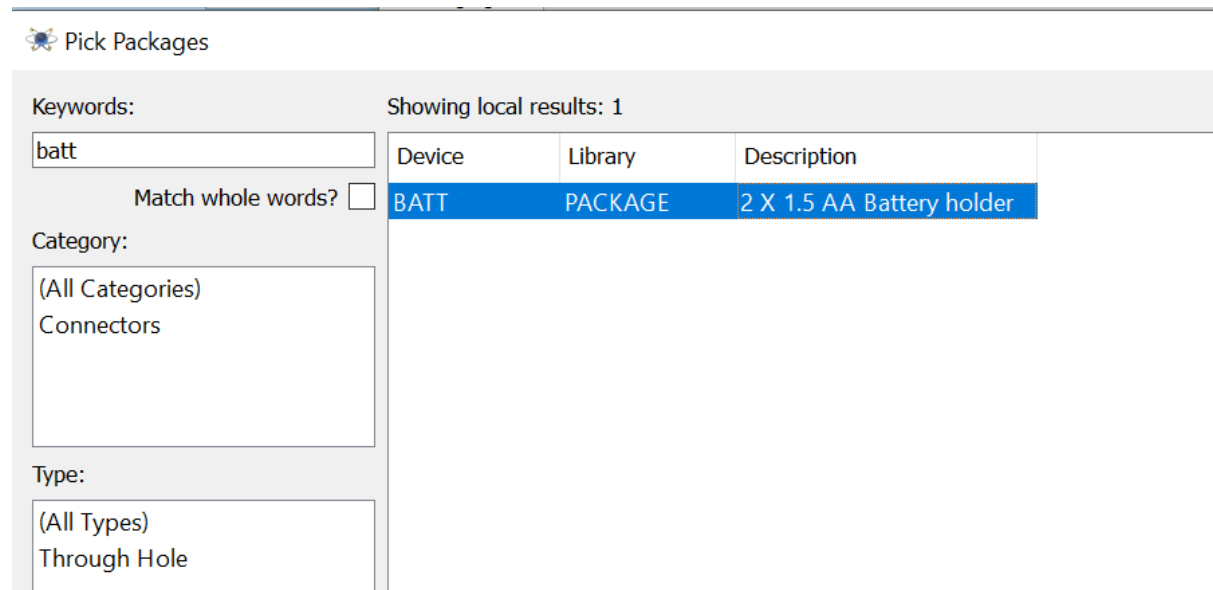
Packagings: BATT

☒ Default package? Add Rename Delete Order

No. Of Gate: 1 ☐ Gates (elements) can be swapped on the PCB layout?

Pin Hidden Common Type A

Figure 15: Add the package



Pick Packages

Keywords: batt

Match whole words? ☐

Category: (All Categories) Connectors

Type: (All Types) Through Hole

Showing local results: 1

Device	Library	Description
BATT	PACKAGE	2 X 1.5 AA Battery holder

Figure 16: Search the name of package

After select the component press enter.

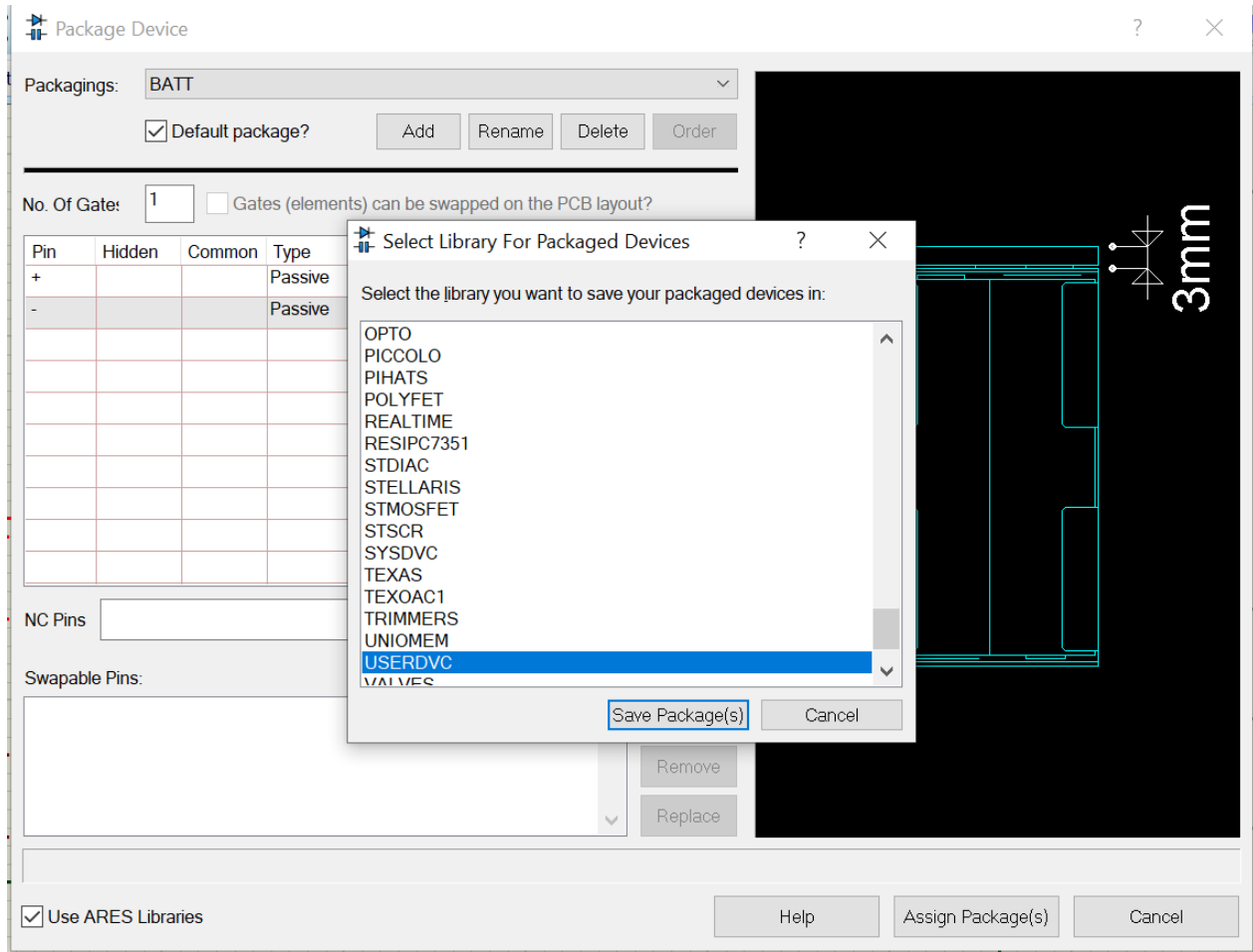


Figure 17: Assign Package

After assigning package click on userdvc and press enter.

Now similarly add all packages of component.

The click on PCB layout.



Figure 18:Pcb layout

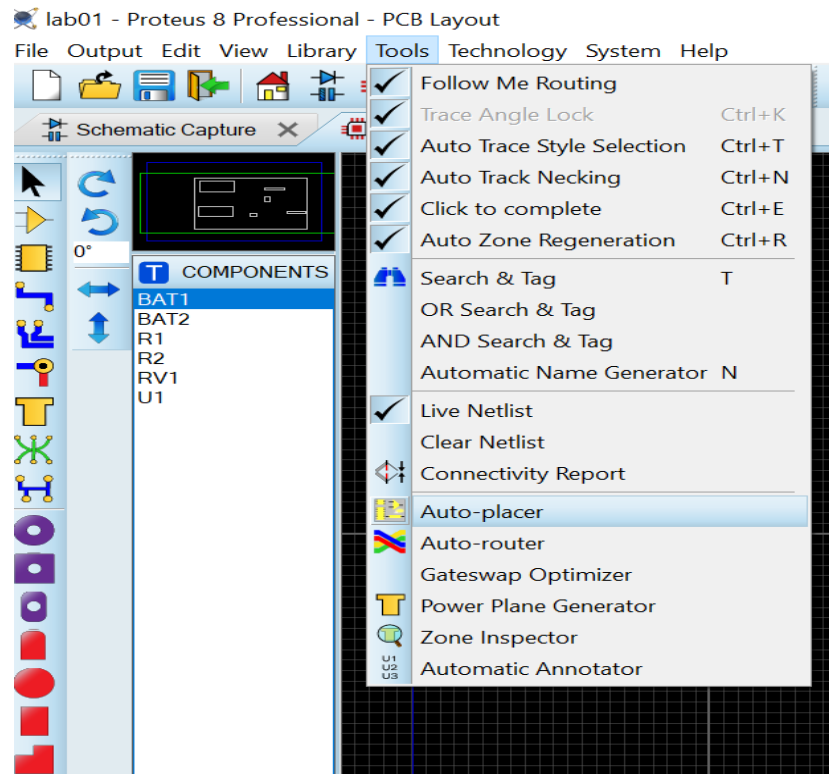


Figure 19: Click on auto placer

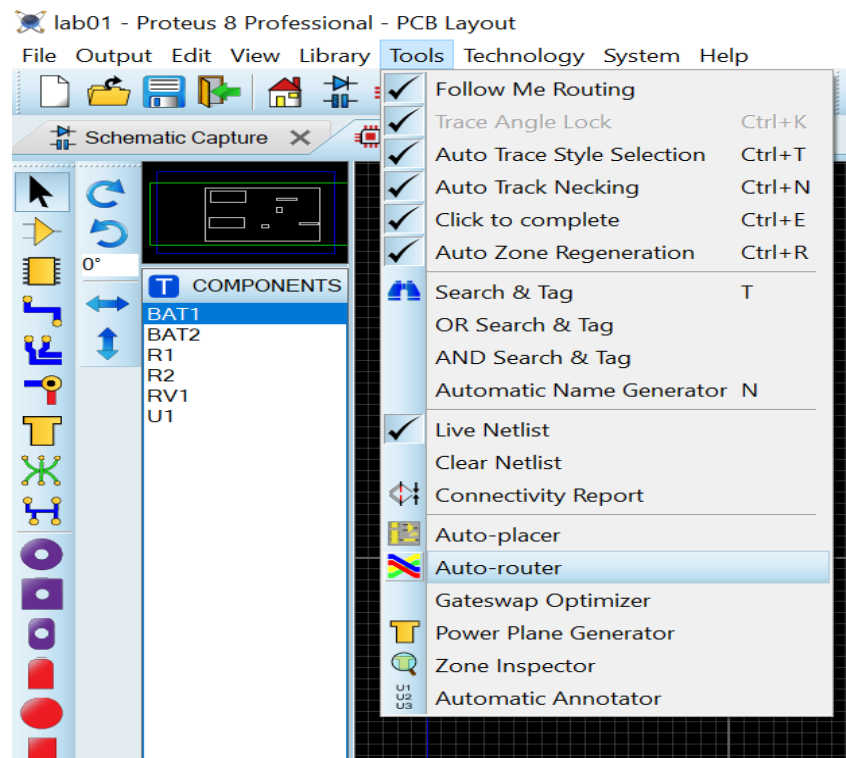


Figure 20: Click on auto router

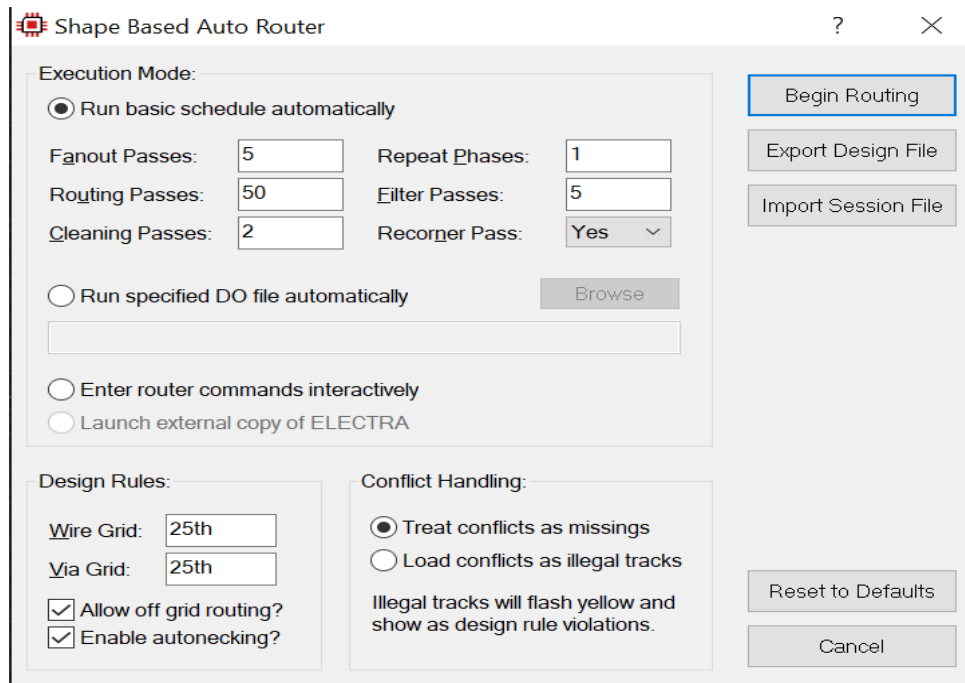


Figure 21: Click on begin routing

Now your circuit is ready. It will auto place and auto connect all the components

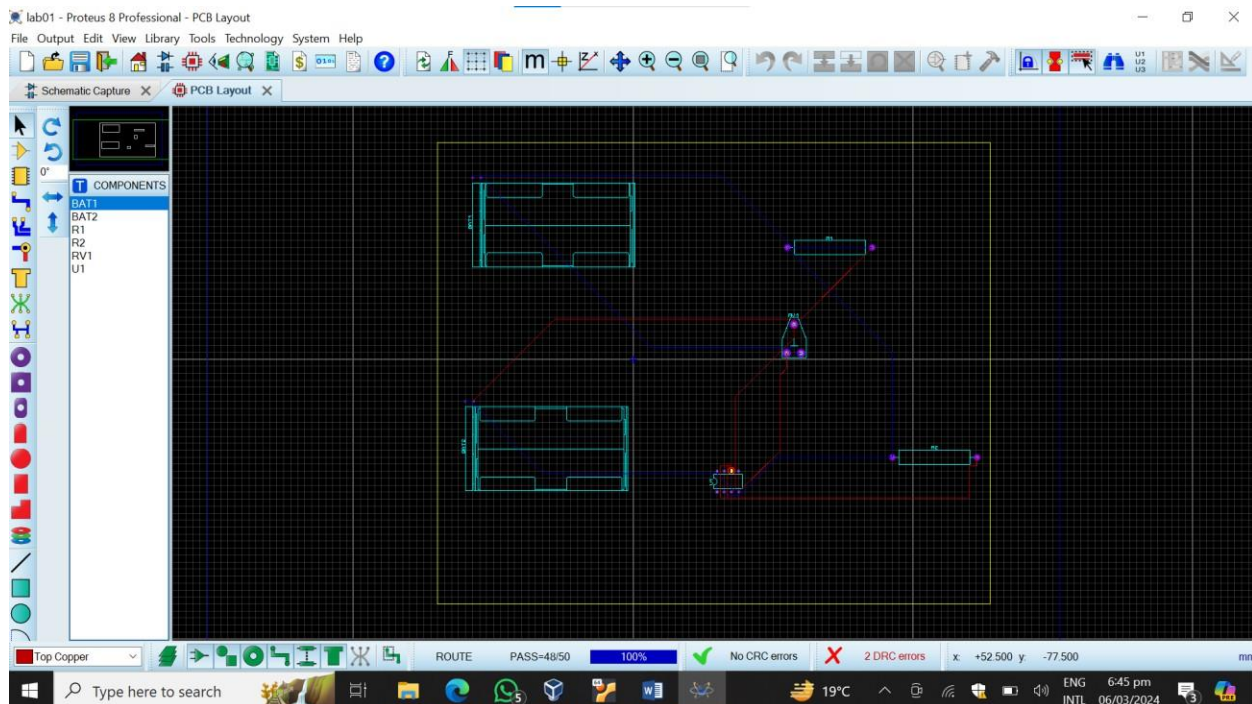


Figure 22: PCB Layout

You can check it by click on 3d Visualizer.

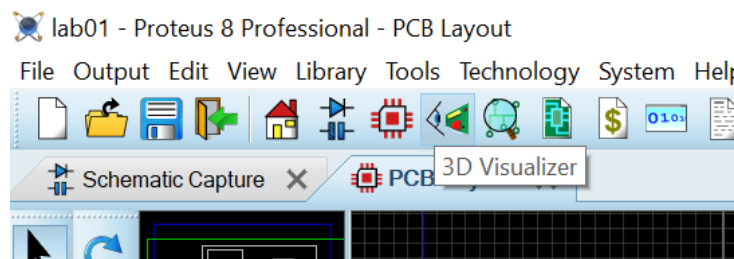


Figure 23: 3D Visualizer

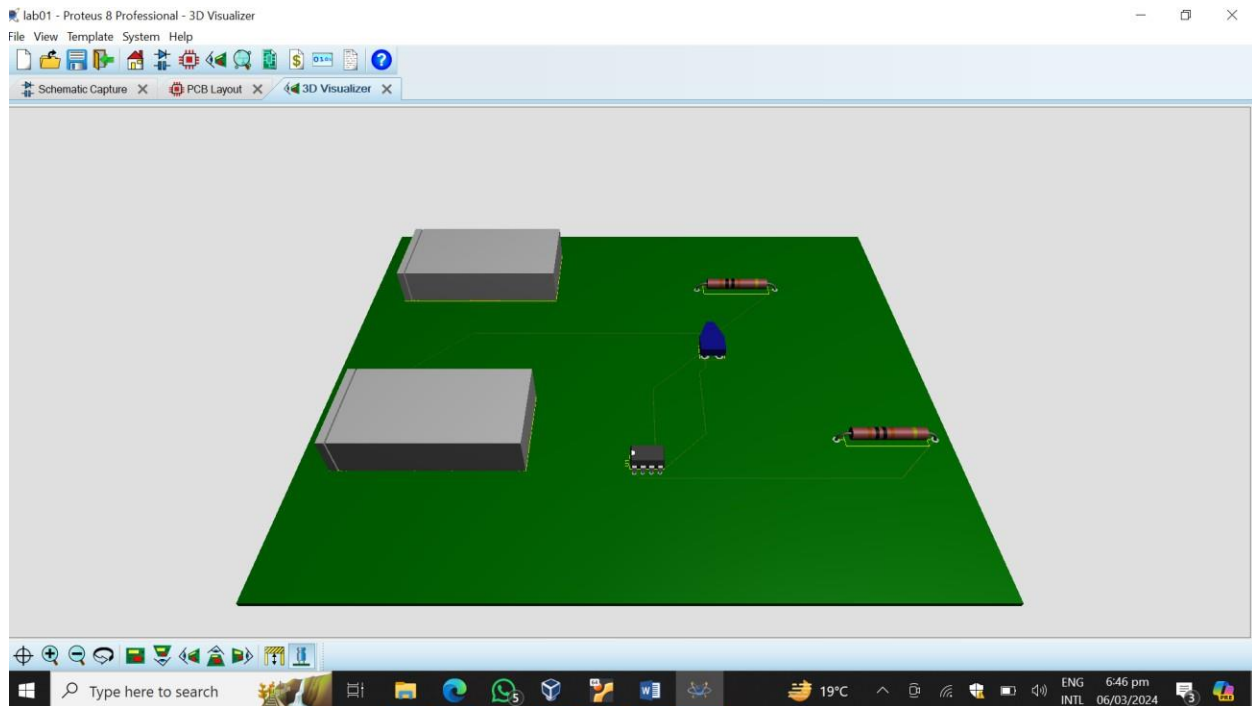


Figure 24: 3D view

Conclusion:

This lab provided an introductory overview of Proteus Design Suite, including how to create and simulate basic circuits. By following the step-by-step guide, students should now be familiar with the interface and basic functionalities of Proteus, enabling them to design and simulate their own electronic circuits.