



SAB35403

PCB DESIGN

LAB 1

Introduction to the PCB Design Flow by Example

LECTURER: DR REZAL BIN MOHAMED

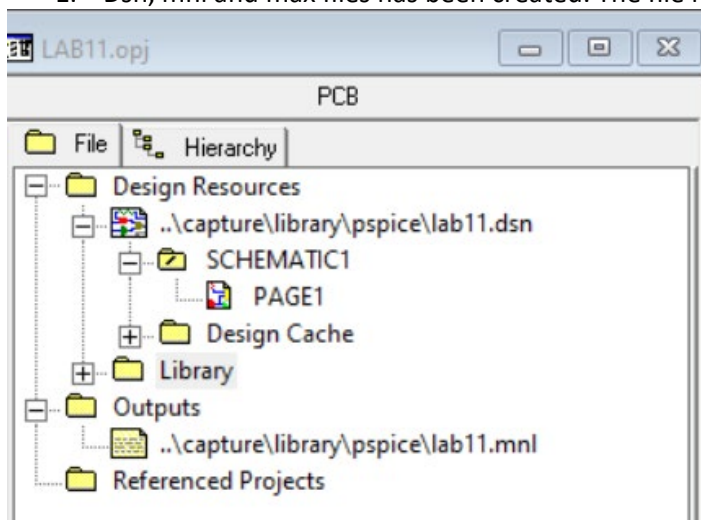
NO	STUDENT NAME	STUDENT ID
1	KHAIRUL ANWAR BIN KHAIRUL SALLEH	54215121186

Objectives:

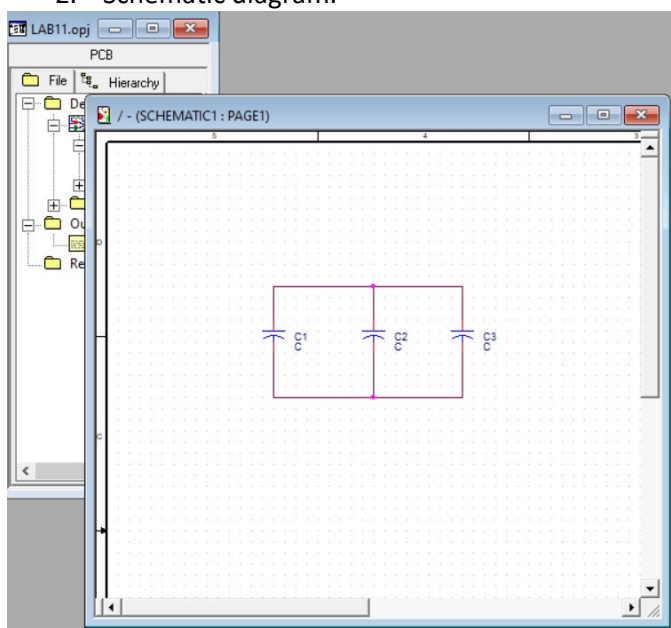
1. Start Capture and set up a PCB project using the wizard.
2. Design the circuit using OrCAD Capture.
3. Generate a Layout netlist using Capture and save it as a .MNL file.
4. Start Layout and select a PCB technology (TCH file) template
5. Save the Layout project as a .MAX file.
6. Import the .MNL netlist from Capture.
7. Make a board outline.
8. Position the parts.
9. Autoroute the board.
10. Postprocess the board to make the Gerber files used to manufacture the PCB

Result:

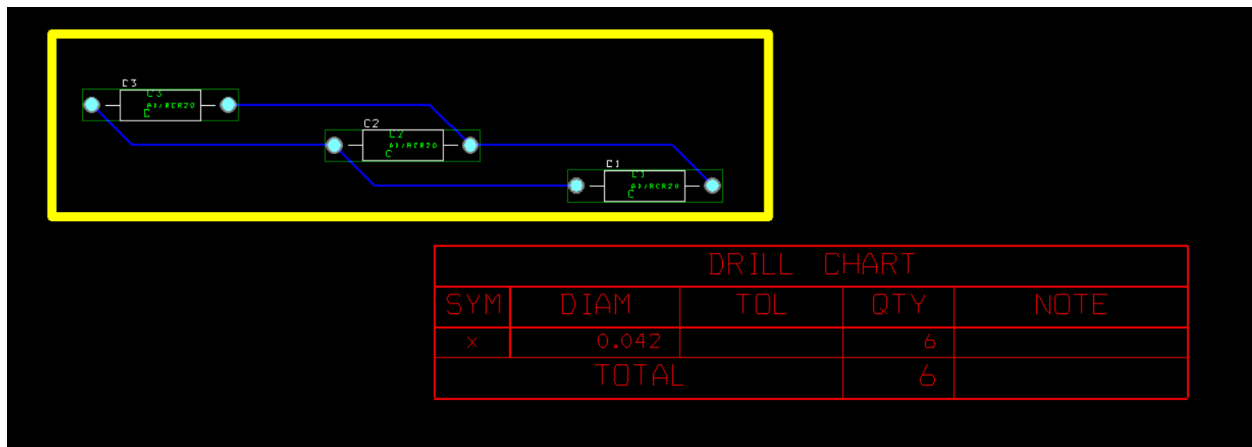
1. Dsn, mnl and max files has been created. The file name is lab 11.



2. Schematic diagram.



3. Max file.



Analysis:

Objective Clarification

The lab report sets the stage by clearly outlining objectives, providing a roadmap for the tasks to be undertaken. These objectives emphasize gaining a practical understanding of the PCB design flow.

Structured Design Flow

A structured approach to PCB design is evident in the report. It systematically introduces fundamental steps in the design process, with a particular emphasis on the transition from schematic design to layout.

Practical Application

The inclusion of a practical example adds significant value to the report. It signifies that students are not merely learning theoretical concepts but are actively applying their knowledge in a hands-on manner, fostering crucial skill development.

Overview of Design Flow

The report effectively communicates an overview of the design flow. It meticulously details steps such as initiating Capture, creating a circuit schematic, generating a Layout netlist, and subsequent actions in the Layout environment.

Technology Integration

The report demonstrates a seamless integration of specific tools like OrCAD Capture and Layout. This integration ensures that students not only understand theoretical concepts but also gain proficiency in using industry-standard tools, aligning their skills with real-world applications.

Conclusion:

- The conclusion effectively summarizes the lab's objectives, emphasizing the imparted understanding of the PCB design flow.
- Application of theoretical knowledge in executing the PCB design flow on a practical example is affirmed through the creation of DSN, MNL, and MAX files.
- The successful import of the .MNL netlist, creation of a board outline, and strategic component positioning align with the goal of initiating the board design process.

- Proficiency in basic autorouting techniques is claimed, although more detailed insights into this step would enhance the report's completeness.