

# **ECE480**

**FS14**

**Design Team 8**

Application Note:

Design of

Printed Circuit Board (PCB)

Yue Guo

A44263141

# Introduction & History

PCB (printed circuit board) connects electronic components by using conductive tracks and other features etched from metal sheets. It may contain single, double or multiple layers. Conducting components on different layers are connected by holes called vias. A well-designed PCB board can increase the qualities and decrease the costs of devices, reduce the physical sizes of them, and is highly reliable.

PCB was created by Austrian engineer, Paul Eisler, and had firstly been used on a radio. In 1943, PCBs were used on radios in military radios, and popularized in civil devices from 50s, 20<sup>th</sup> century.<sup>1</sup>



An example of a PCB-based Boost Converter<sup>2</sup>

# Different Types of PCBs

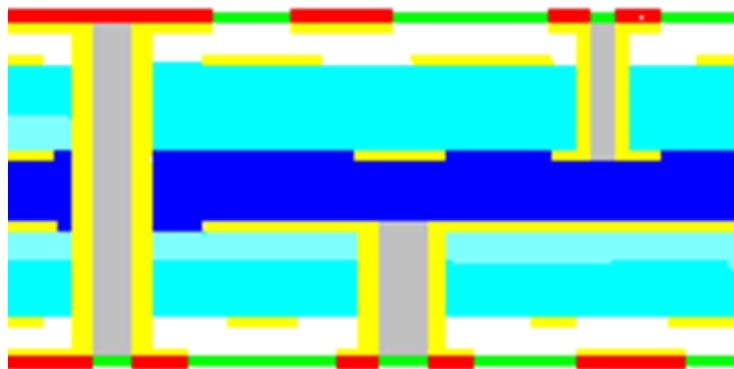
Different kinds of PCBs have different features. Base on the mechanical properties of the panel of PBC, there are Rigid PCB, Flex PCB, and Flex-Rigid PCB. We can see the comparison between Rigid and Flex PCB below.

	<b>Rigid PCB</b>	<b>Flex PCB</b>
<b>Cost</b>	Lower	Higher
<b>Reliability</b>	Higher	Lower
<b>Flexibility</b>	Lower	Higher
<b>Thickness</b>	Higher	Lower

Comparison of Rigid and Flex PCB

In our project, since we do not need the PCB board to be very thin and flexible, I choose Rigid PCB as the final layout.

Also, PCBs have different numbers of layers. In the early designs, all the components were on one side of a PCB and the copper wires are on the other side. This is called Single-Sided Board. This kind of PCBs was strictly limited by many conditions, for example, there could be no cross-overs between different wires. Double-Sided Board can solve this problem very well since both sides of the board can be patterned with copper wires and when they need to be connected, vias will connect them through the board. Nowadays many devices use Multi-Layer Boards. They increase the area on PCBs by using multiple interfaces and layers in the board. In the specific project of ECE480, I am planning to make a 2 to 4 layers PCB.

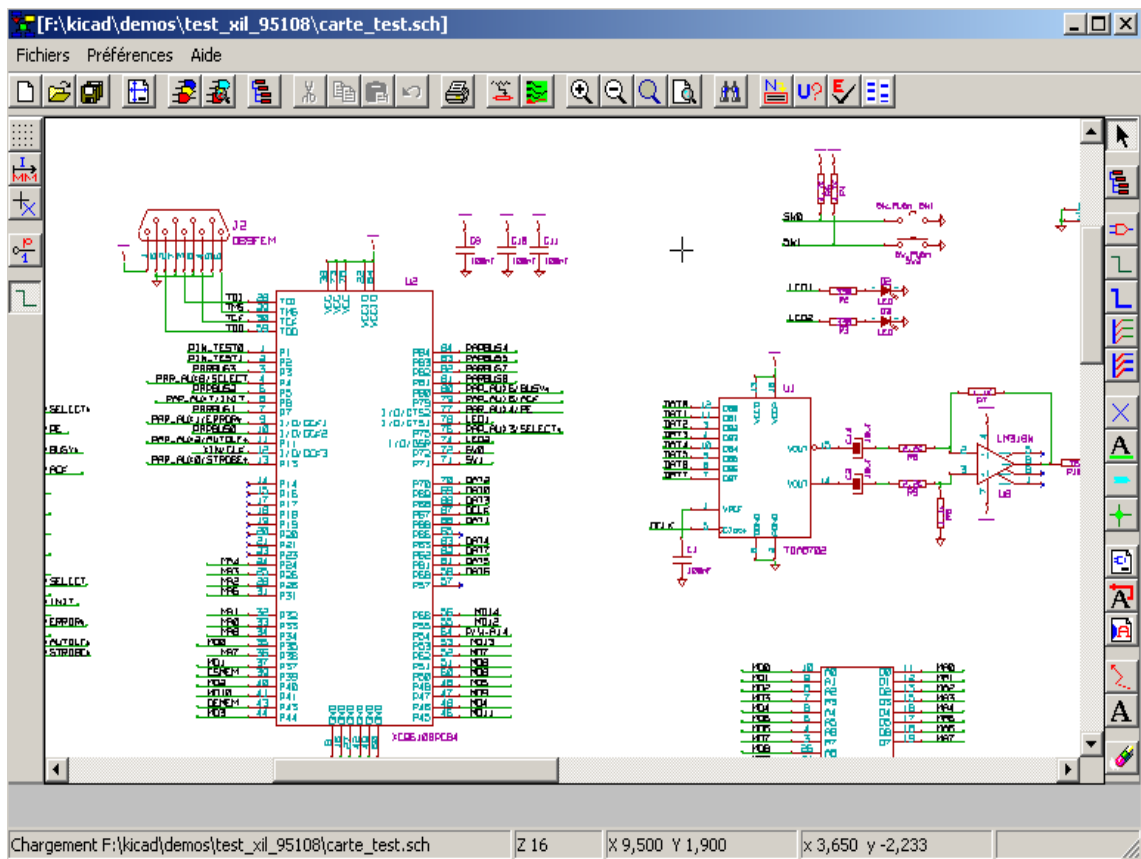


The structure inside a multi-layer board<sup>3</sup>

# EDC Software: KiCAD

In the ECE480 designing project, I choose KiCAD as the EDC software to design the PCB board because it is free and easy to learn, also recommended by our sponsor, the MSU solar car team.

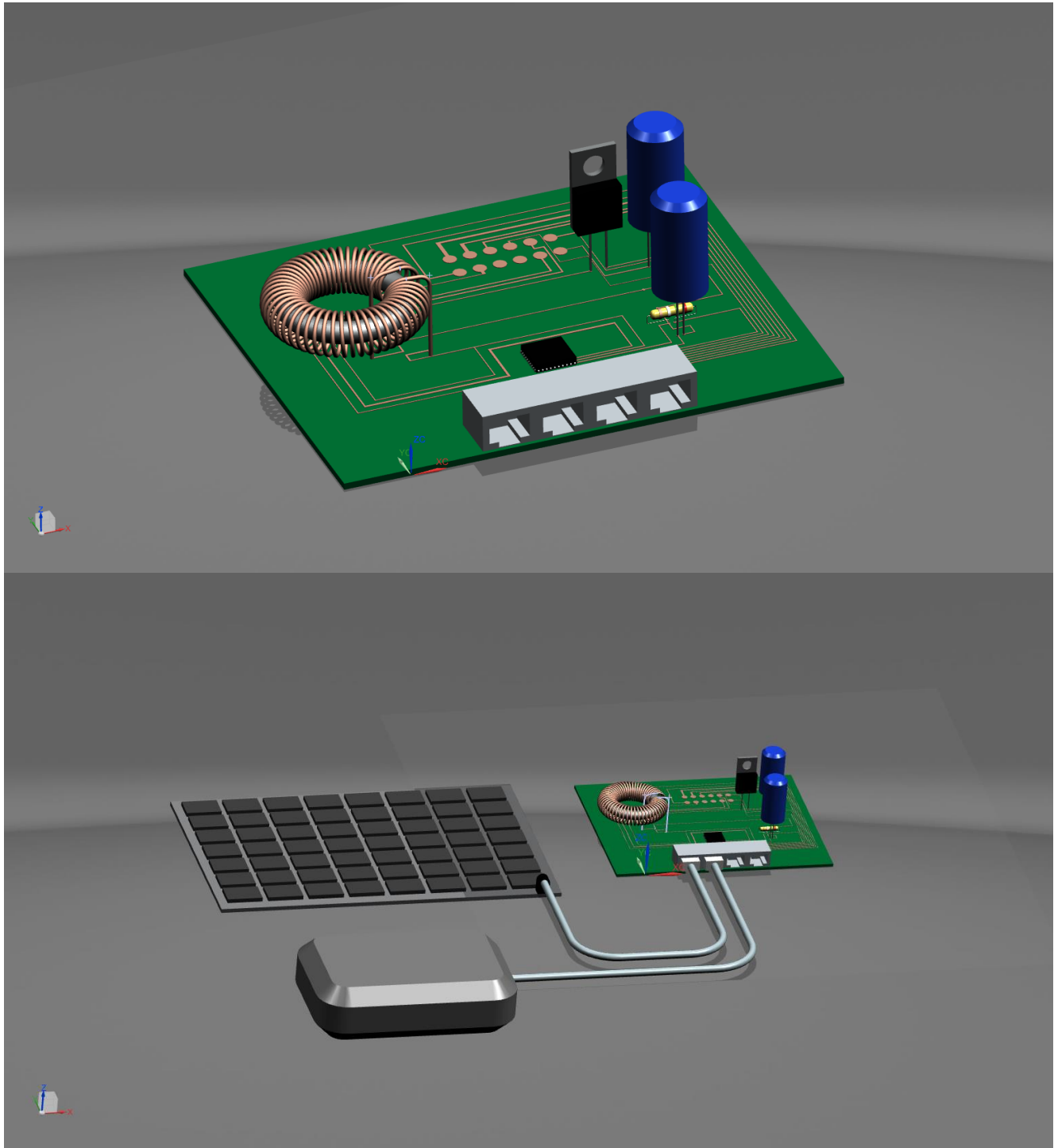
KiCAD is relatively user-friendly software. Also, it is capable of being used on Windows, Mac and Linux systems. The only disadvantage of it is that it does not have the simulation module, but in our project, most of the simulations are processed by using LABVIEW and MATLAB.



KiCad user interface<sup>4</sup>

# Conceptual design

The diagrams show the final PCB design with some of the components attached. The size will be roughly 120\*100mm, and the width of copper wires will be about 750um. With the varying of parts we decide to use, these numbers may also vary.



## Reference

1. *Wikipedia, Printed Circuit Board*, [http://en.wikipedia.org/wiki/Printed\\_circuit\\_board](http://en.wikipedia.org/wiki/Printed_circuit_board)
2. MSU Solar Car Team, <http://www.egr.msu.edu/~solar/blog/>
3. *Baidu*, <http://image.baidu.com/i?tn=baiduimage&ct=201326592&lm=-1&cl=2&nc=1&ie=utf-8&word=PCB>
4. *KiCad Official Site*, <http://www.kicad-pcb.org/display/KICAD/KiCad+EDA+Software+Suite>