

Cornflower: Bike Safety

# Printed Circuit Board (PCB) Design

By Huyen Pham, ECE '22

## Introduction

A printed circuit board (PCB) is a physical, solid platform that consists of electrical components connected by embedded metal wires called traces and large metal areas called planes. The board can have a single layer (including a top and a bottom layer) or multiple layers stacked together [4]. A PCB is used to construct electrical circuitry for various commercial applications and follows a standard PCB Design process.

The Hardware Design process can be split into 5 main steps, as shown in Figure 1 [4]. After designers have designed a circuit schematic for their application, the design needs to be validated against customer requirements and system constraints. After the design is validated to meet specifications, it can be passed onto the next step of the design process so that the final product can be manufactured [1]. Before reaching the final layout, designers need to develop PCB prototypes to iteratively test their design. This tech note will introduce the steps of a standard PCB Design process that outputs such PCB prototypes.

components that meet specifications. Based on the desired functionality and calculations, components can be chosen from various semiconductor manufacturers such as Texas Instruments, Analog Devices, NXP, and On Semi. Designers should also consider the temperature range in which the PCB operates, component tolerances, packaging size, etc. To ensure the components are a good fit, designers should check the datasheets, test on the manufacturer's evaluation boards, and run the design in simulation tools [1].

While selecting components, a parts library is developed. The library contains all component symbols, simulation models, PCB footprints, and three-dimensional (3D) models. The PCB footprint of a component is its physical outline when attached onto the PCB. The simulation model of a component includes its PCB footprint, dimensions, and pin/lead arrangements; while the 3D model shows its physical shape.

## Step 2: Schematic Capture

Schematic capture involves designing a circuit by wiring the selected components together in a schematic/diagram to analyze the design visually. Signals should flow from the upper left corner to the lower right one of the schematic. The schematic needs to include the names, values, ratings, and manufacturer part numbers of all selected components [2]. In parallel with designing the circuit, a bill of materials (BOM) is created. The BOM includes all the component details from the diagram so that the circuit is easily converted into a PCB layout in Step 3.

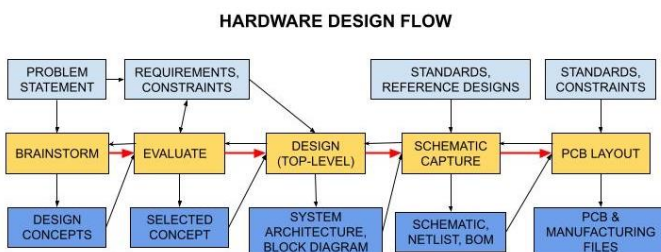


Figure 1. Hardware Design Flow [4]

## PCB Design Flow

### Step 1: Part Selection

Before designing, designers need to select

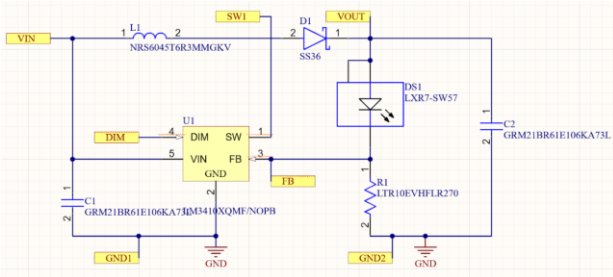


Figure 2. Sample schematic

This step also requires simulation of the schematic to ensure that the circuit functions properly. The simulation can be done using SPICE, which is the industry standard for simulation software.

### Step 3: Board Layout

After the schematic is simulated, it can be transformed into a board layout. The board layout visually shows the physical dimensions and positions of components in a manufactured prototype board. This step includes defining the board outline, placing components on the board, and creating copper paths (i.e. traces) between components. The process of drawing traces to connect components is called routing. In Figure 3, the yellow lines show the PCB footprints, and the red lines show the traces.

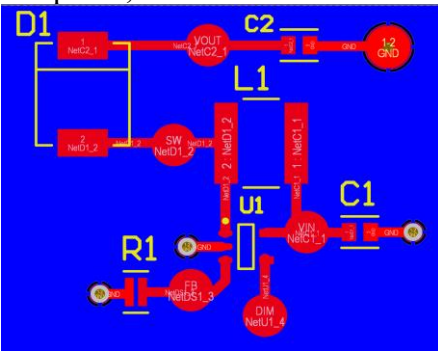


Figure 3. Sample board layout

The main steps in board layout are listed below [2]:

- Set up the snap grid, visible grid, and default trace/pad/via sizes. The snap grid is where the PCB is laid out, and components and traces can snap into fixed grid positions. The visible grid is an optional on-screen grid of solid or dashed lines or dots, which helps the designer align components and tracks.
- Import the schematic with all components onto the PCB.
- Place components in functional “building blocks”. Group electrically sensitive blocks

into bigger groups. For example, digital and analog blocks, or high and low frequency blocks, should be separated.

- Route each “building block” separately. Then move the blocks into position on the main board. Route the remaining signal connections between blocks.
- Review the layout, first with a Design Rule Check (DRC) in the software, then with other designers. Compare each electrical net connection on the board against the schematic.

A netlist is a list of connections and components extracted from the schematic. When importing the netlist to the PCB layout, a net name is assigned to each component pin, trace, and pad/via. The assigned net names allow for auto routing, DRC, and connectivity display. The PCB design software also displays a rats nest, which draws straight lines between connected components, at the start of the board layout. The rats nest will disappear once all component connections are made with traces in the software [2]. In Figure 3, the yellow texts show the component designators, while the white texts on the red traces show the net names.

When routing, designers should generally use wide and short traces. Traces should curve by 45 degrees and “snake” around the outer edge of the board instead of going point-to-point. Designers should also follow the clearance limit, which is the minimum distance between a trace and a pad or another trace on the PCB [2].

### Step 4: Board Verification

After designing the PCB layout in software, the layout is exported to the PCB assembly vendor that fabricates the physical board (i.e. the PCB prototype). The PCB prototype needs to be verified in the lab by taking measurements from the board to ensure that its real performance meets the specifications. Once the prototype is verified, it can be passed onto the manufacturing phase, when the final boards are assembled. At the time of manufacture, the finished PCB should also be checked again using an automated test machine.

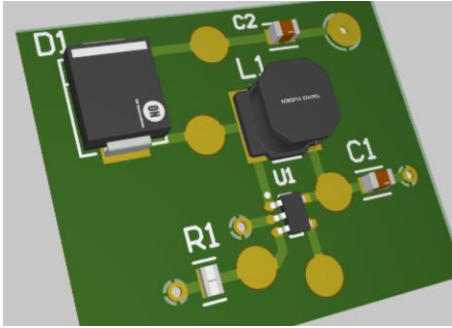


Figure 4. Sample PCB design (3D) of the layout in Figure 3

In Figure 4, the large, filled circles are pads. Pads are large welding holes that pass through all layers of the PCB to accommodate a wire or component pin. The small, hollow circles are vias. Vias are small plated through-holes (PTH) with copper land to electrically connect traces and copper between layers. The green plane is the solder mask directly on top of the bare copper traces. A solder mask is a thin polymer coating which surrounds pads and vias to prevent solder from bridging between pins. The silkscreen layer on top of the PCB contains all component outlines and designators (i.e. white texts in Figure 4). The PCB design is extracted from the software as Gerber files for the manufacturer. In addition, the designer should give more details including: reference code and revision number; desired turn-around time; board quantity; board thickness; type of PCB; surface finish material; copper weight; track/space clearance; etc [2].

## Conclusion

Hardware designers follow a standard PCB Design Process to transform their design from concepts to physical boards. The process ensures that the final board will not only meet specific product requirements but also experience minimal design errors. This tech note explains the steps of the PCB design process and includes some design standards. More comprehensive design standards can be found in external industry papers.

## References

1. PCB Design Fundamentals: Prototyping and the PCB Design Flow. (2021, May 12). [White Paper]. NI. <https://www.ni.com/en-us/innovations/white-papers/10/pcb-design-fundamentals--prototyping-and-the-pcb-design-flow.html>

2. Jones, D. L. (2004, June 29). PCB Design Tutorial. <http://alternatezone.com/electronics/files/PCBDesignTutorialRevA.pdf>

3. Printed Circuit Board: An Introduction and the Basics of Printed Circuit Boards. (2019). Cadence PCB Solutions. <https://resources.pcb.cadence.com/blog/2019-printed-circuit-board-an-introduction-and-the-basics-of-printed-circuit-boards>

4. Gupta, R. (2017, August 25). Hardware Design Flow— Learn This before Getting into PCB DESIGN! <https://www.youtube.com/watch?v=7y2JDnO8e28>

---

---