

Printed Circuit Board Design with Open-Source Software

Hannah Nelson, Adviser: Dan White Ph.D.

Abstract

Printed Circuit Board (PCB) design allows for the creation of the modern-day electronic systems. Open-source PCB design software allows people to easily access, modify, share, and collaborate on electronic projects. This project created several designs demonstrating various PCB design techniques for a keyboard interface, a transmission line test card, and a 2.4 GHz antenna.

KiCad EDA is a cross-platform and open-source electronics design automation suite and was used for creating the boards in this project. Open-source software has few restrictions and no licensing costs, other commercial PCB design packages include Altium Designer.

Printed Circuit Board Design Process [1]

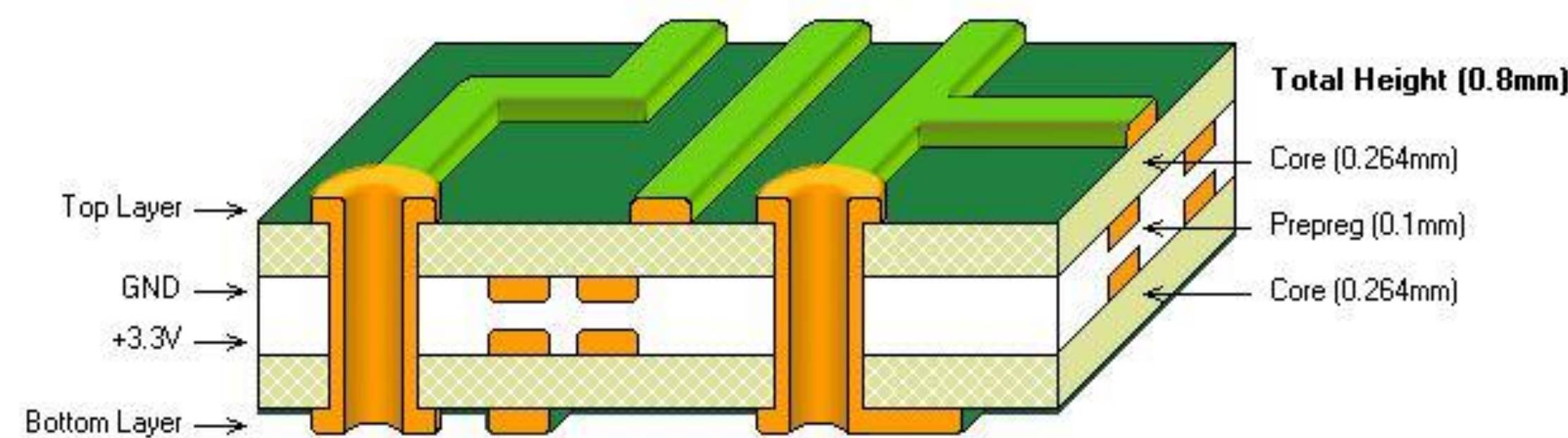
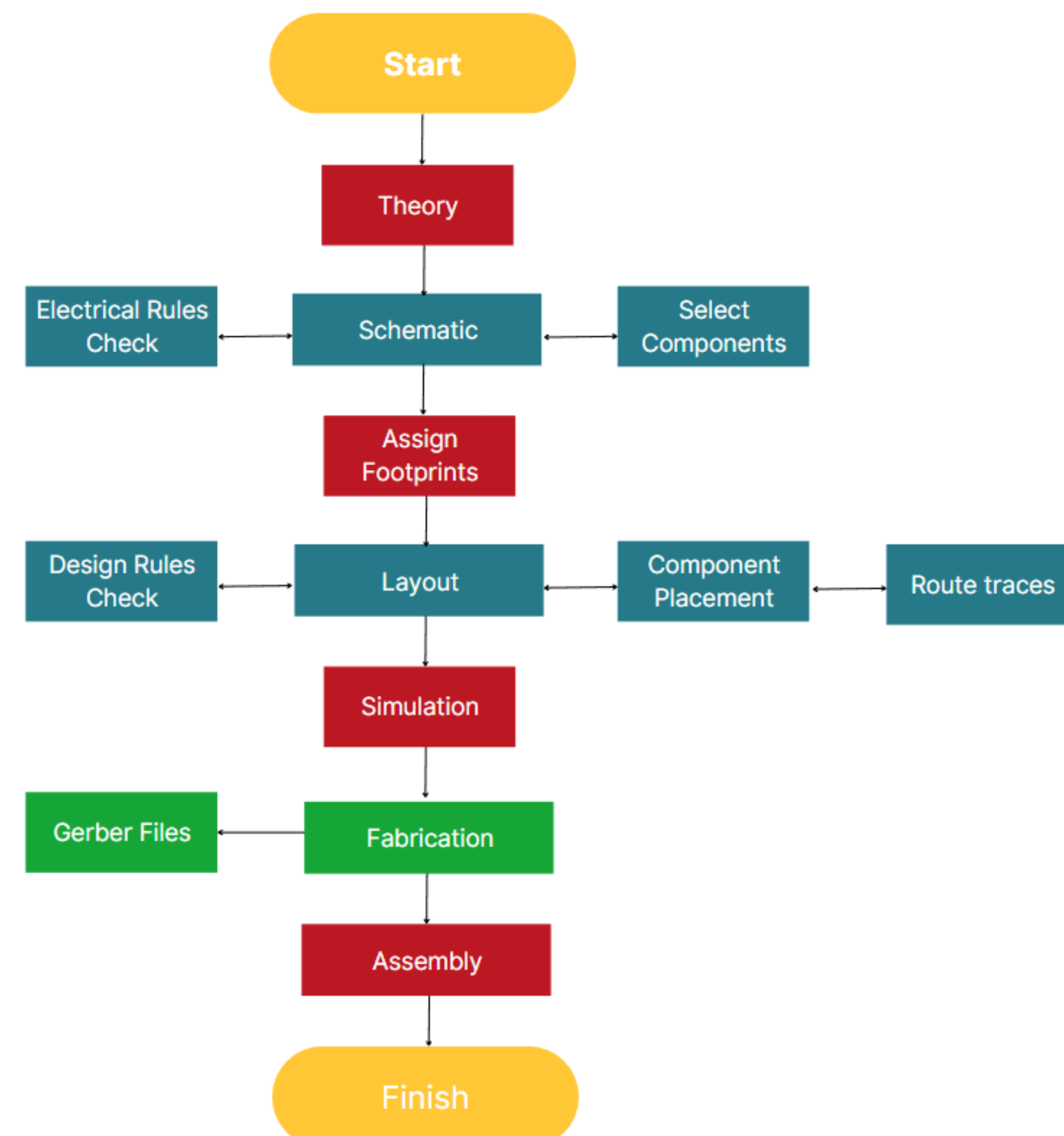
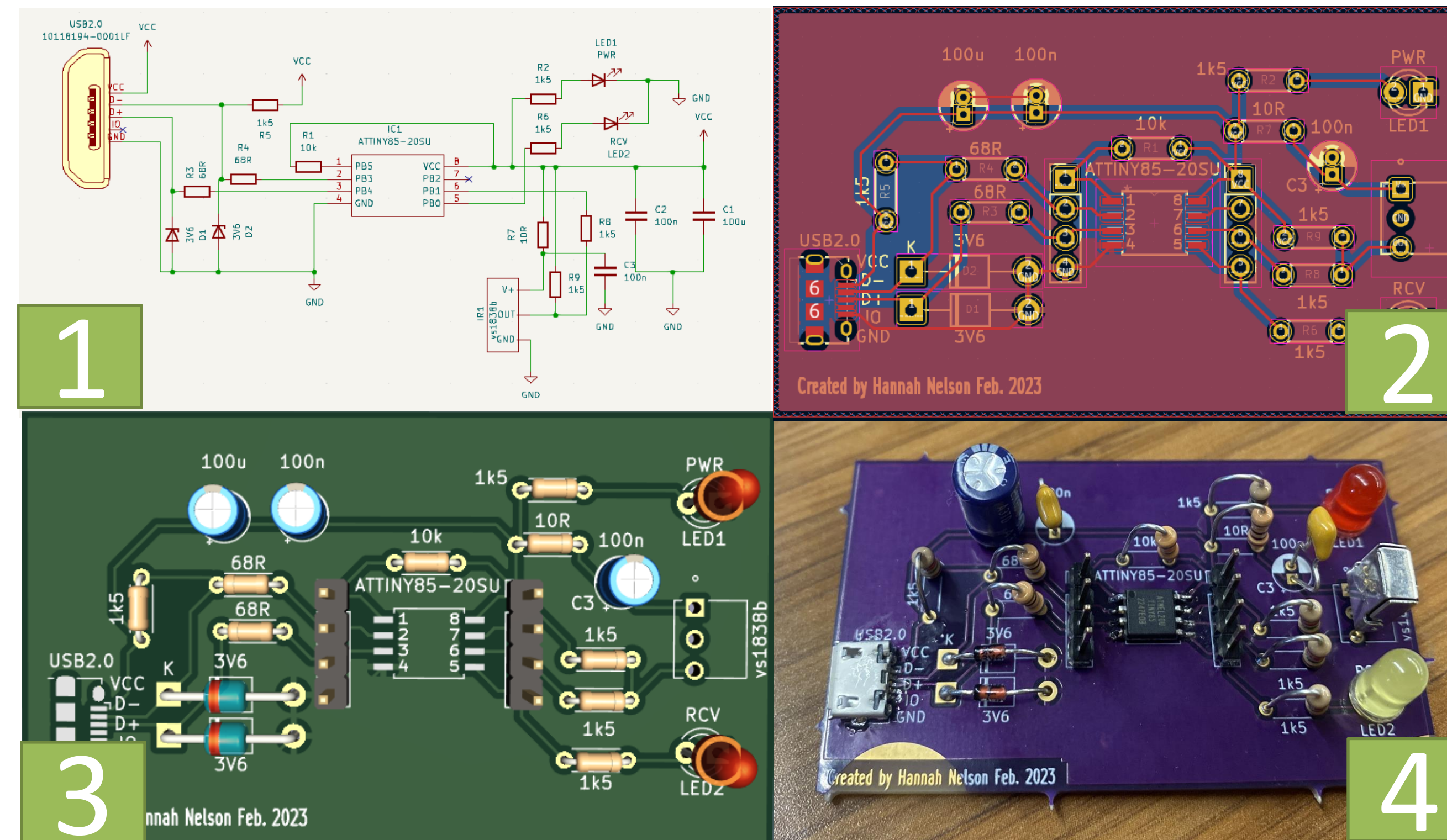


Figure 1: Four Layer PCB Stack-up

References

- [1] "How to Design a Circuit Board in 10 Easy Steps." *Altium*, 18 Mar. 2023, <https://resources.altium.com/p/10-easy-steps-comprehensively-designing-circuit-board-altium-designer>.
 [2] Ghosh, Poulomi. "IPC-2221 Standards in PCB Design." *Sierra Circuits*, 12 Apr. 2023, <https://www.protoexpress.com/blog/pcb-2221-circuit-board-design/>.
 [3] Bevelacqua, Pete. "Microstrip Patch Antennas." *Microstrip Antennas: The Patch Antenna*, <https://www.antenna-theory.com/antennas/patches/antenna.php#fringing>.



Infrared Keyboard to USB Interface Board

- 1) **Schematic View** showing electrical connections between components
- 2) **Board Layout View** showing trace routing between electrical footprints
- 3) **3D Rendering** showing a preview of finished board layout
- 4) **Manufactured PCB** showing the finished board with soldered components

Terminology for Printed Circuit Board Design [2]

- **Stack-up** refers to the number of conductive and insulating layers a PCB can have like what is shown in Figure 1. A two-layer board was chosen for all given projects.
- **Trace Width** affects both the current carrying capacity and characteristic impedance.
- **Trace Spacing** reduces coupling between signals and prevents manufacturing error that may cause shorts within the circuit.
- **Silkscreen** ink (white) is useful for component labeling, polarity indicators, and other ways to identify components and connections between components.
- **Board Material** provides mechanical strength, affects electrical properties and maximum operating temperatures. A common board material is FR-4 (Flame Retardant) made from fiberglass cloth and epoxy resin.

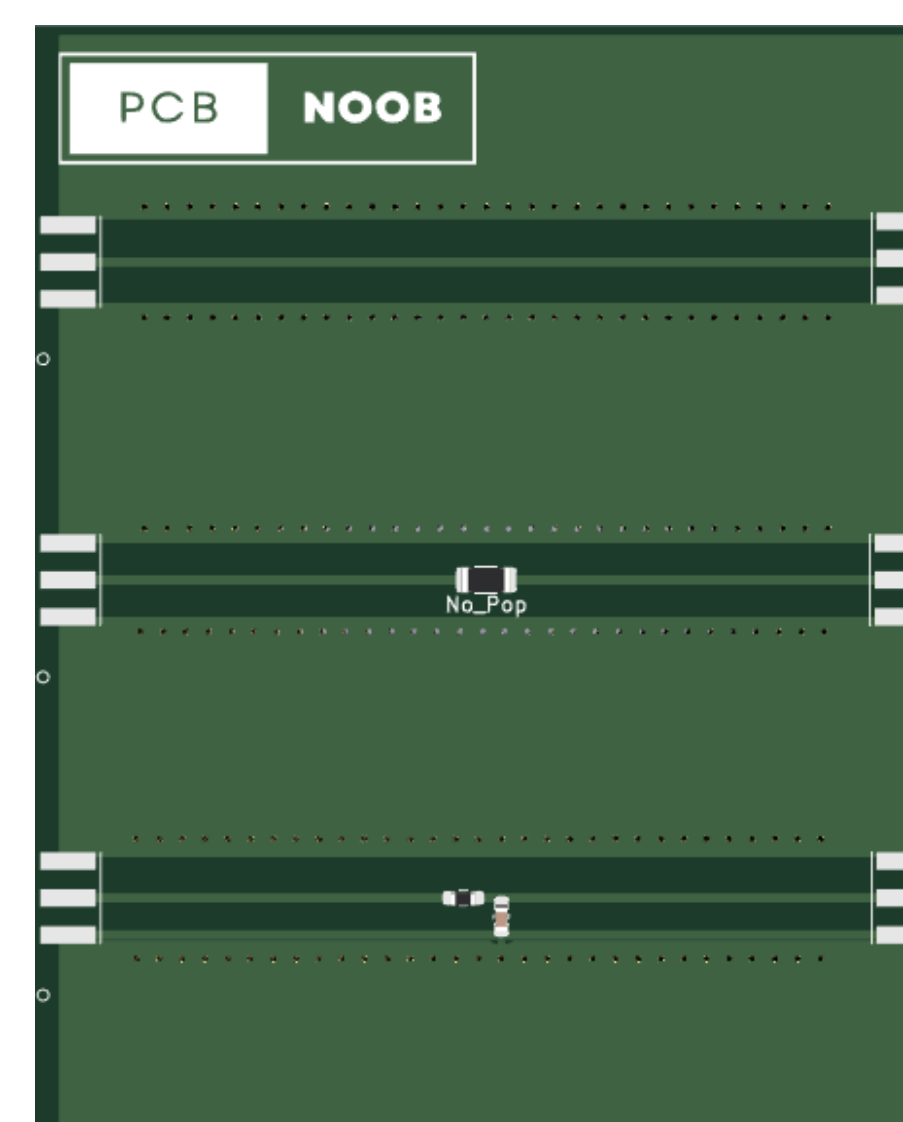


Figure 2: Vector Network Analyzer Calibration Board

Type	Expected	Measured
Short	0 Ω	1.9 Ω
Open	∞ Ω	1010 Ω
Load	55 Ω	57.2 Ω

Figure 3: VNA Calibration Circuit Expected vs. Measured Impedance

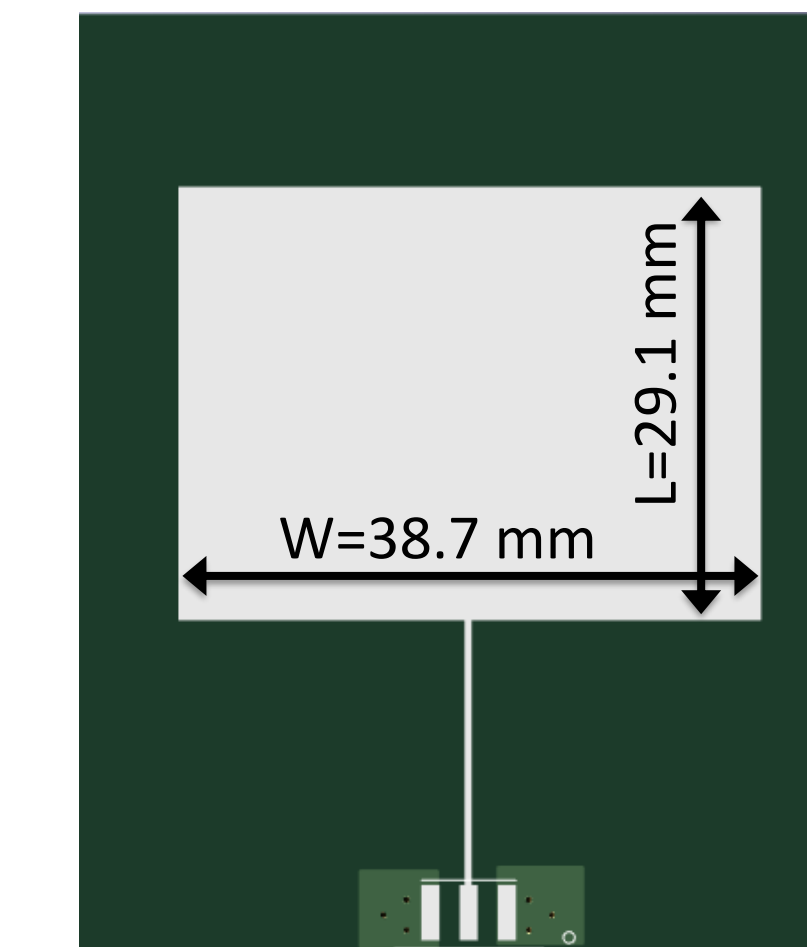


Figure 4: Half-Wave Patch Antenna PCB

$$f_c \approx \frac{c}{2L\sqrt{\epsilon_r}} = \frac{1}{2L\sqrt{\epsilon_0\epsilon_r\mu_0}}$$

$$L \approx \frac{1}{2f_c\sqrt{\epsilon_0\epsilon_r\mu_0}}$$

$$W \approx \frac{c}{2f_0\sqrt{\frac{\epsilon_r+1}{2}}}$$

Equation 1: Determining Dimensions Through Cutoff Frequency [3]

Patch Antenna Performance Analysis

Measurements were performed using a Vector Network Analyzer (VNA) to determine the antenna's input impedance characteristic, S11, over the design frequency range. Figure 5 shows that the antenna's resonance frequency (minimum S11) was approximately 2.328 GHz, which is 3% below the design frequency of 2.4 GHz.

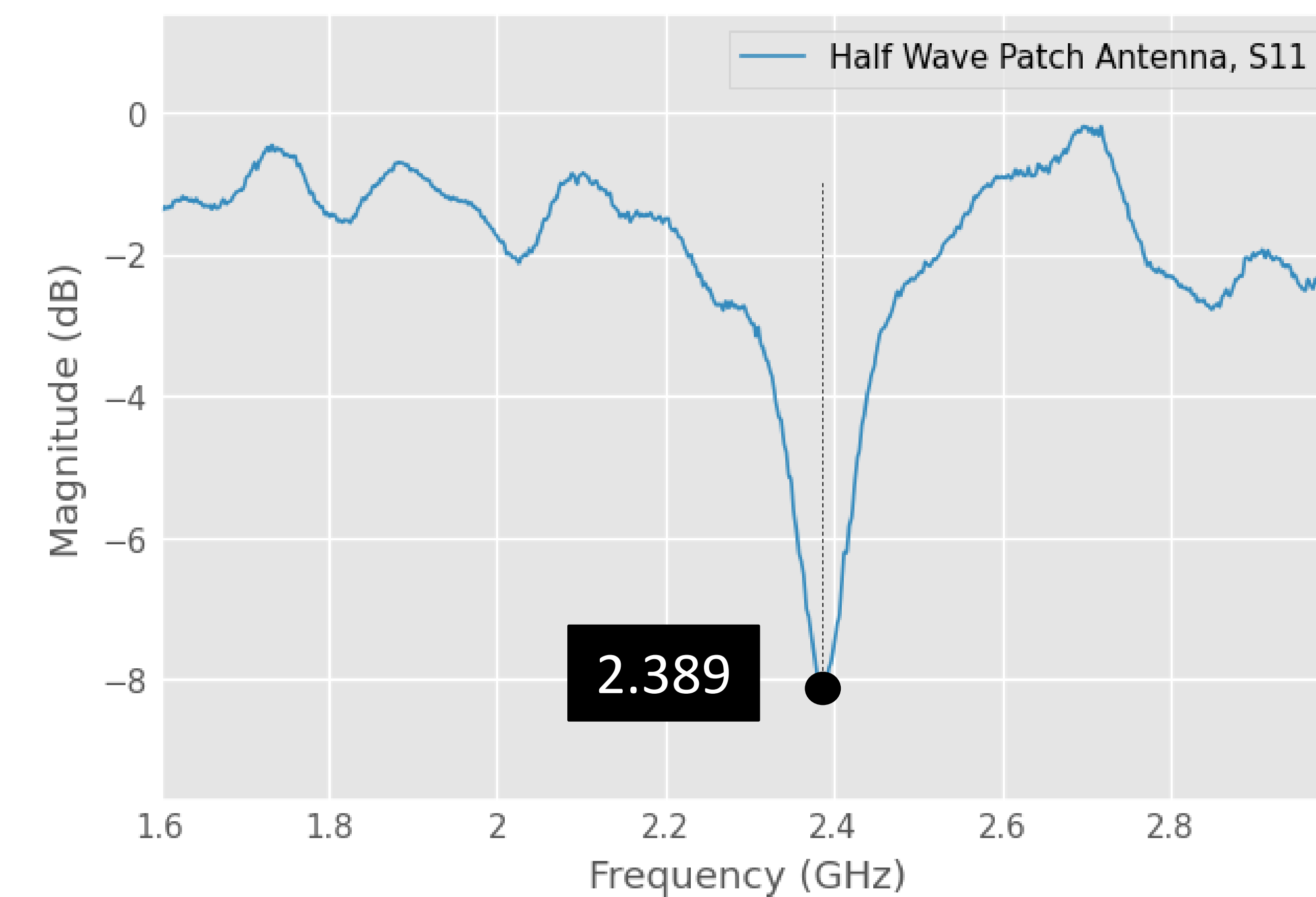


Figure 5: Magnitude of S11 over frequency for half-wave patch antenna

