PCB Design with KiCAD

Part 0: Introduction

Ben Hurwitz, Spring 2024

Hello, and welcome to The Hive's PCB Design with KiCAD video series. My name is Ben, and in this fifteen part walkthrough, I will guide you through the basic workflow of how to design a PCB, from conceptualization, through part selection, schematic capture, and layout, ending with a final board design that could be fabricated. The final four videos are not design-focused per-se, but discuss good library management practices and model creation.

In this video, we'll start with an overview of the series, offer some goals for what I hope you'll take away from this, and a short biographical sketch about me, your host.

Let's get started.

Series Overview

1. PCB basics and terminology
2. Electronics design software concepts
3. Circuit conceptualization
4. Schematic (5 parts)
   a) Adding parts
   b) Symbol creation
   c) Wiring
   d) Footprints
   e) ERC
5. Layout (3 parts)
   a) Setup
   b) Placement and routing
   c) Final DFM checks and DRC
6. Symbol libraries
7. Footprints (3 parts)
   a) Footprint libraries
   b) Custom footprints from scratch
   c) Custom footprints with the wizard

Goals for this tutorial

1. Introduce you to PCB design terminology
   - Understand the jargon, be able to have a conversation, and transfer knowledge to other PCB design software

2. Demonstrate the PCB design workflow
   - Learn how to actually design a circuit board

3. Make custom models, or locate pre-designed ones online.
   - Making your own is the very last resort.

4. Provide you with additional resources
   - Where to find answers to questions and additional tools/techniques

Who am I?

- As of this writing, I'm a 7th year PhD candidate in the Ocean Science and Engineering program trying desperately to graduate.

- My work was in developing a novel ocean sensor and the accompanying instrument, which turned into a lot of KiCAD development.

- I am not an expert. I just have experience with this software.

End of Part 0

And with that, we'll end the introduction to the series. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In the next video, I'll introduce PCBs and some of the jargon you'll encounter on your journey through PCB design and fabrication.