Schematic Capture / PCB Design

Overview

In this activity, you will use Electronic Design Automation (EDA) software to enter a schematic design and generate a printed circuit board (PCB) definition. This will give students experience in designing PCBs and an understanding of how hardware designs are created via schematics and how components are integrated within them. While modern PCB design can be extremely complex, the basic process is simple, and even with a small working knowledge you can design boards for a variety of applications. The primary learning objectives of this lab are to understand the process of schematic capture, PCB layout, and fabrication, which will serve as a foundation for communication with other engineers working on cyber-physical systems and as a starting point for developing your own electronic assemblies.

A selection of schematics from typical computer engineering applications are available at the end of this document. Your goal is to recreate the schematic and design a simple PCB using one of the following EDA software:

- **Altium Designer** – An advanced industry standard software. Free to use for educational use through the University. A good choice for those who intend on doing firmware/hardware design in a high-tech environment. Works only on Windows systems.
- **KiCad** – A free and open-source alternative with a strong community engagement and support. A good choice for those looking to do hobby or small-scale hardware design, without needing to pay for licensing. Cross-platform.

Instructions

This activity is broken into four steps: installing the software, creating the schematics and design files, generating a report of the schematic development, and completing a demonstration. The requirements for your implementation include:

- Accurate recreation of one of the provided schematics. Some guidelines to follow include:
  - Be consistent with your use of through-hole or surface-mount components.
  - Utilize net labels to make your design readable.
  - Add notes to clarify any important information.
  - Include your name, date, and other relevant information.
- Feasible implementation of a PCB design, with the following additional requirements:
  - Use a layer stack up of two or four copper layers.
  - Include at least one copper pour in your design.
  - Include a board outline.
  - Include appropriate ports (connectors or pin headers) for power, digital, and/or analog IO.
  - Generate suitable fabrication outputs as if you were to place an order for your board.

Submission

Your submission will be composed of the following elements:

- **Demonstration** of PCB Design (in-person or recorded, see below),
- **Project files** within a ZIP archive on Canvas,
- **Report** in PDF format on Canvas.

Demonstration

Your demonstration will be completed either in person during office hours or as a link to an unlisted video resource (i.e. YouTube video). It is your choice how you convey this demo to us. Regardless of how you demo, it must include the following elements:

- Narrated overview of circuit schematic and PCB layout,
- Clear showcase of the board outline and copper pour(s),
- Clear showcase of how fabrication files are produced for the design.

Report

In addition to a short introduction and conclusion, your report should include a page for each of the following:

- **Original image of your chosen schematic** – describe the primary purpose of the circuit (you should read a bit about the circuits from the links provided), and a high-level description of how this purpose is accomplished.
- **Screenshot of your schematic capture** – write about the process you went through to recreate the circuit in your chosen EDA software. Highlight any challenges you faced and any notable discoveries you made.
- **Screenshot of your PCB layout** – write about the process of laying out your PCB. What did you learn and what challenges did you face? What are some strengths of your design and where could it be improved?
- **Screenshot of an example Gerber file** – write about the process of producing output files to send to a fabrication facility.

Altium Setup

Here is a quickstart video thanks to your TA Raven! Here are many tutorials from Mike Stapleton (UF ECE senior design instructor).

First, you will need to install the supporting software, including Altium Designer and libraries. Note that this will require approximately **5 gigabytes** of storage just to download the files, before installation.

Download

To set up Altium Designer, you will need to download the following files:

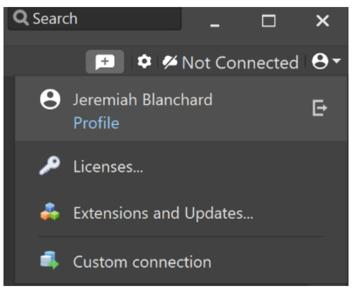
Altium Designer Installer 21.6.4 or newer
Altium Component Library (2.1GB)
ECE Component Library (167K)

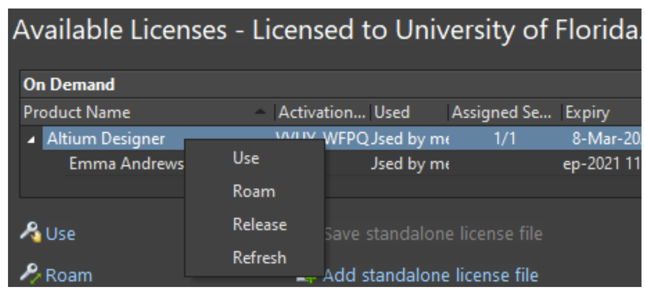
For libraries, note that the URL is http, **not https**. UF-hosted files require access from the VPN.

Installation

Fill out the form on the [student licenses website](), making sure to use your UFL email address. Once the form is submitted, you will receive an email (may take several minutes) asking you to verify your email address. Click on the provided link to verify your email address. You will then receive another email (may take several minutes) containing your license information. Follow the activation in the email to create your Altium account. After your account is created you will be provided with a link to download the installer. Download the installer and then run it. During the installation process, you will be prompted to sign into your Altium account. Enter your UFL email address as the username and the password you just created. Select the PCB Design, Extensions, and Importers/Exporters features in the installer.

Once installed, run Altium Designer. Open the License Manager by clicking on the profile icon in the top right, then selecting “Licenses”. If you signed into your account during installation, your license will be auto-populated. Otherwise, sign into your Altium account, and your license will appear. Right click and select "Use" or click on “Use” under the Available Licenses pane to activate the license. If not signed in, sign into your Altium account from this page.





KiCad Setup

Here is a [quickstart video]() thanks to your TA Lucas! There are many useful online tutorials for KiCad (such as [this one]()).

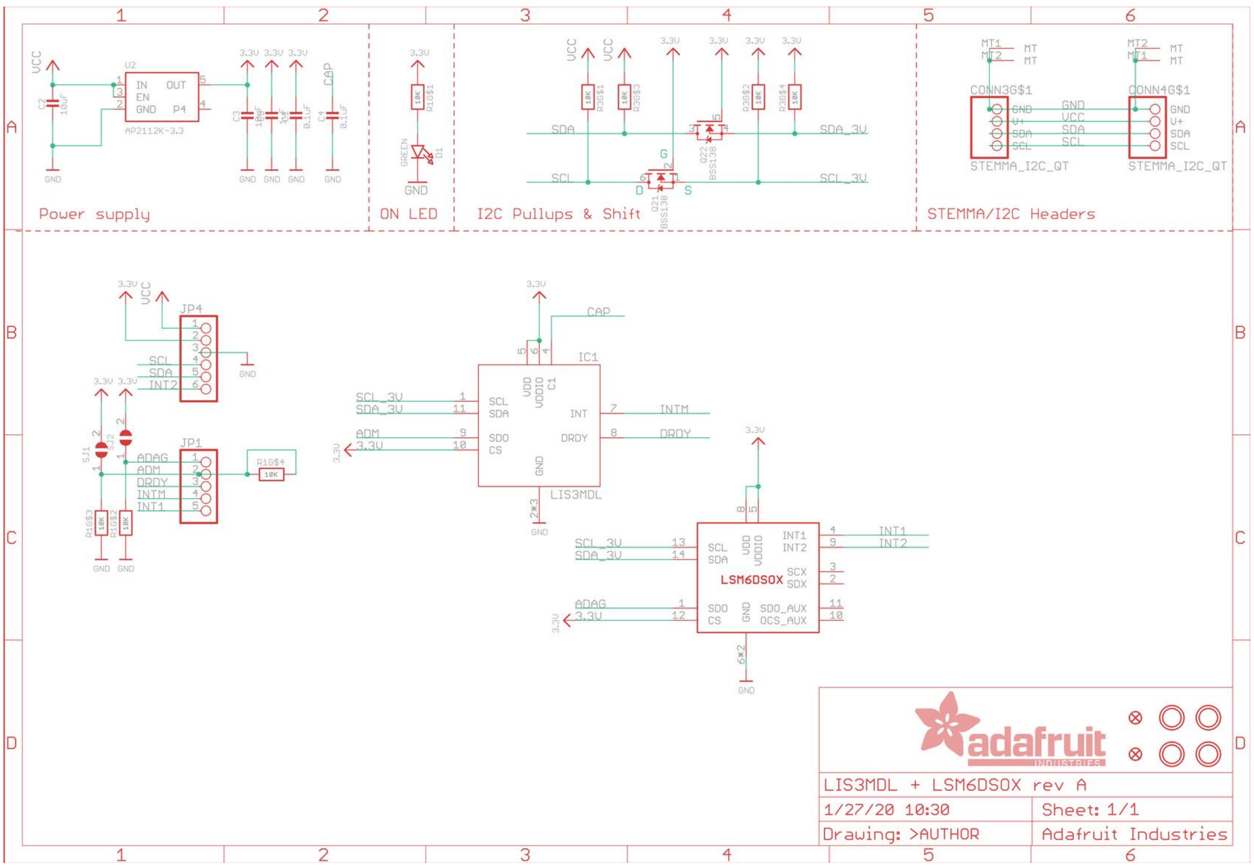
Download the [KiCad installer]() for your platform and proceed as instructed to install. The default options should be suitable for most use-cases, which includes the built-in component libraries that you will need to find and use parts in your designs.

Schematic Options

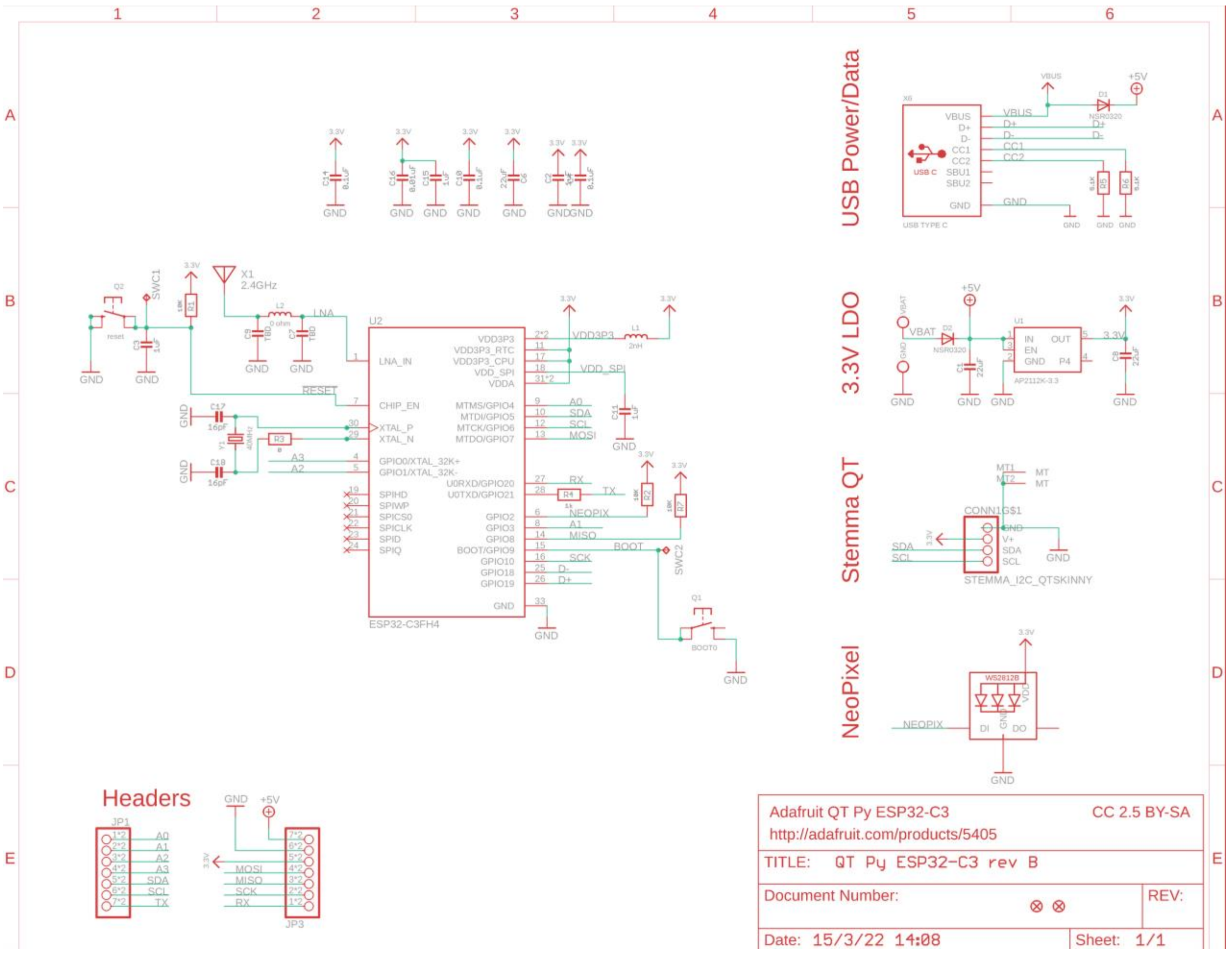
Browse the below selection of sample schematics and choose one for your implementation based on your interests and goals. If there is a schematic you'd like to recreate that is not on this list, please contact the instructor for approval.

Microcontroller & digital IC application circuits

These schematics outline the very basic requirements to get a microcontroller or digital integrated circuit up and running, including power supply filtering, crystal oscillators, flash memory, etc. A good choice for those interested in embedded systems, IoT, mixed-signal applications, etc.



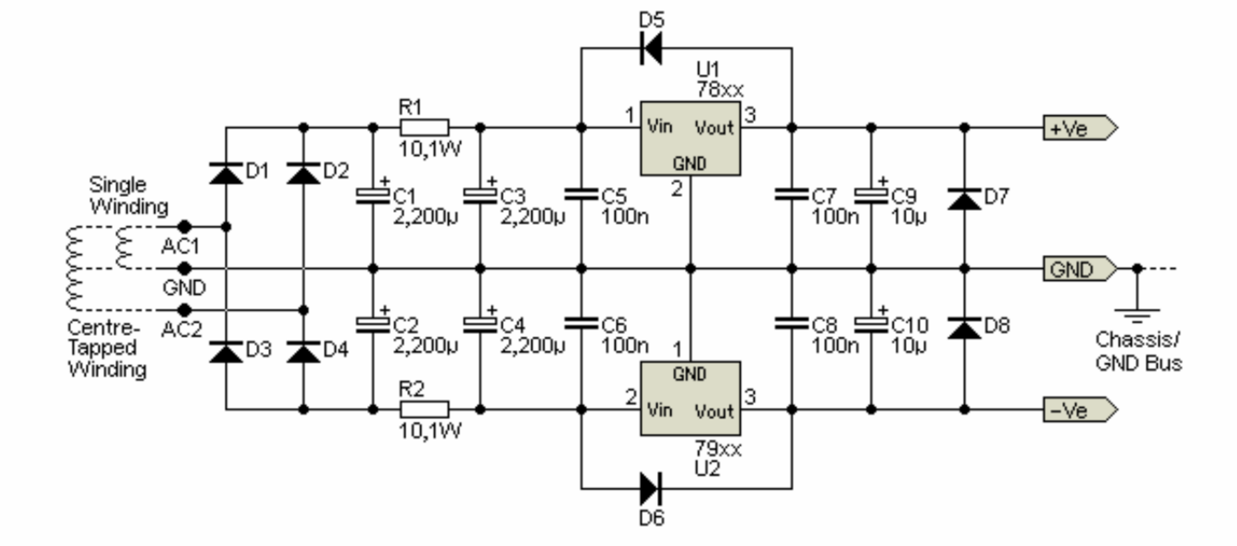
Schematic 1 - From Adafruit's [product number 4517](). A 9DoF IMU breakout board.



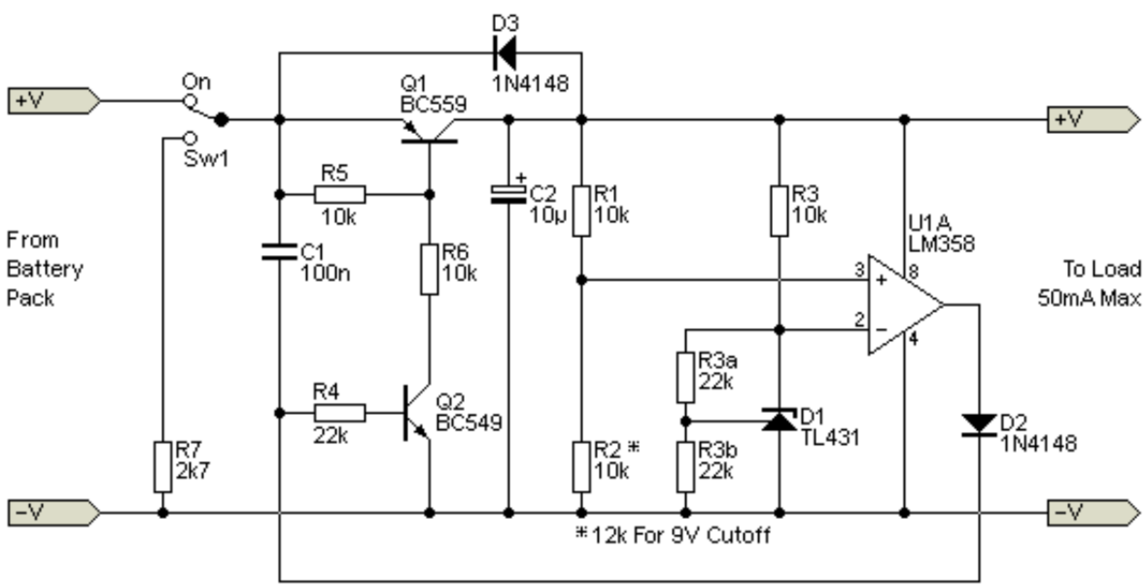
Schematic 2 - From Adafruit's [product number 5405](http://adafruit.com/products/5405). An ESP32-C3 development board.

Audio-electronics application circuits

These schematics highlight some analog electronic designs for audio applications. A good choice for those interested in analog circuit design, analog-to-digital conversion, or similar sensor applications.



Schematic 3 - A simple linear power supply for audio applications. Borrowed from [Elliott Sounds Products project 05-mini](#).



Schematic 4 - Undervoltage protection for LiPo battery. Borrowed from Elliott Sound Products [project 184](#).