HANDS-ON PCB ENGINEERING (HOPE) FINAL PROJECT

TABLE OF CONTENTS

Project Description

Over the past few weeks, we have introduced you to the basics of PCB design and manufacturing. In the coming weeks, we will introduce you to more advanced topics and techniques. Now, it is your turn to create your own PCB design. Over the rest of this course, you will work with a team of students to ideate a project PCB, create a schematic and layout, assemble it, and program it for a final demonstration expo during RRR week. During this final demonstration, you will have the opportunity to present to current industry engineers and UC Berkeley EECS Department professors. **While we allocate some time during class for you to work on this project, expect at least 2-3 hours of work outside of class each week to stay on top of project deadlines.**

Unlike projects in other university courses, we do not expect you to develop a perfect product by the conclusion of this semester. The beauty of PCBs lies in the fact that they can be low-cost solutions for certain smaller challenges, so we recommend defining those challenges and designing a board around solving those challenges.

With all parts of this project, **if you believe your design does not comply with our requirements, but has the merit to be a HOPE Final Project, please contact staff ASAP**. It is easy to grant exceptions to requirements early; 2 hours before the final project deadline is not so easy.

Group Overview

Project Groups due February 23, 2024

For this project, you will work in groups of 3 students. In addition, given that we give time in labs to work on project designs and hold design reviews, we require that every group member is part of the same lab section. However, we will allow cross-sectional groups on a case-by-case basis.

We can grant exceptions for teams as large as 4 students in extenuating circumstances (i.e. this is a project for a research group in which all 4 students are members, or the project is a multi-board project).

On the other hand, **we strongly discourage working solo on a HOPE final project**. The timeline, complexity, and requirements of the project are meant to be a challenge for 3 students and are even more difficult for a single student. In addition, in both your future classes and in industry, you will work with teams of engineers on a single board design, so this is great practice to build those skills. However, if you want to complete your HOPE Final Project solo please reach out to the staff and declare your intent on the Group Formation Form. Be sure to include why you want to do your project

solo and as well your openness to having another student added to your project if we have ungrouped students. Staff will review the request and reserve the right to not approve the project as a solo project. However, a well-thought-out proposal and project design and efforts to complete other assignments of the class on time go a long way to ensuring your solo project is approved.

If you need help finding a group for a project, please fill out the Project Group Search Form.

Deliverables

- Project Group Form - One per group
- Project Group Creation Checkoff Bcourses Quiz - Complete individually

Project Concept+Proposal

Project Proposal Due: March 1, 2024

Now it is time to get creative! As mentioned earlier, find a challenge/s that you want to design a board to overcome. Aim for a project concept and proposal that you and your team are passionate/excited about because you will spend a lot of time designing it and will have to defend its merit and the design choices you make to achieve the design goals. Also, have fun brainstorming projects; everything requires circuit boards so your options are nearly limitless. This project is open-ended to ensure that you can design a board related to any topic or interest you and your group may have. Past projects include:

- ESC motor control
- Microfluidics control
- Guitar-playing robot mainboard
- Ocean Buoy Radio
- IOT Sensor Boards
- Digital Clock
- LiPo Battery Management Board
- LoRa Transceiver
- Plant Growth Monitoring
- LCD Display Controller
- Closed loop fan controller
- Handheld Sonar

Please see the Design Requirements Section for details on what we require for this project. The goal of the Project Proposal is to explain to the staff, in detail, the challenge/s or functionalities your project will achieve. It is also a time to describe how you meet all the Design Requirements for the final project. **With that said, detail is imperative, vague descriptions and general parts lists without links/part numbers will not receive full credit** and will also cause teams to struggle later in the project. Also, while your written descriptions will provide us with the details of what parts you are considering and what your design will do, **we expect a drawing/block diagram of how the major circuit components are connected** (i.e. what buses you will use, what voltage levels will you have).

Within 1 week after you submit your Project Proposal, HOPE staff will return a written review of your proposal. This review will highlight any issues we may foresee with your current project, give suggestions for alternate components, ask for clarification on a technical design decision, or reinforce any overlooked requirements. If your group wishes to discuss the comments of the project proposal review further, please come to staff office hours or reach out to staff to schedule a meeting.

Deliverables

- Project Proposal Form - One per group
- Block Diagram submitted to Project Proposal bCourses assignment- One per group
- Project Proposal Checkoff Bcourses Quiz - Complete individually

Design Requirements

Below are the 5 categories on which we grade each project. **2 of them are required by default and then you must choose at least one of the remaining 3 for the project.** If you wish to substitute one or both of the required categories with another category from the list below, you must talk with course staff before submitting the Project Proposal and we will review on a case-by-case basis during Proposal Reviews. Given that the goal is to learn PCB design, **the use of pre-built PCBs, known as module or breakout boards, in project designs is generally discouraged and cannot fill the requirements**. However, they are permitted on a case-by-case basis.

We understand that the constraints of placing all components on the PCB may not fit with the goals of your project and you may want sensors or actuators that are off-board. **In cases where your project must keep sensors or actuators off board, to receive credit for the respective categories, you must design a proper connector and**

interface on the board to connect to the external sensor (i.e. Soldering wires into through-hole pads will not receive credit).

General Requirements

- Board size may not exceed 203mm x 203mm (8 in x 8 in)
- Projects need to be within the standard manufacturing capabilities of our manufacturing sponsors (JLCPCB and Bay Area Circuits)
- Total budget for non-PCB parts is $100. This budget must be enough to bring up TWO copies of your project design, so $50 per board of non-PCB parts
- All parts are in Stock and not from the Marketplace from these suppliers: Digikey and Mouser
- For specialty parts, no resistors or transistors from these suppliers, please obtain permission from HOPE Staff for parts from these providers: Amazon Prime and Adafruit.
    - Permission must be obtained by the Monday of the week when Final Design Files are due. Any request for specialty parts after that date will not be considered.
- All boards will be 2-layer with 1 oz copper thickness and FR4 dielectric material by default.
    - On a case-by-case basis, alternate dielectric materials, layer numbers, and copper thicknesses will be considered
- Every distinct voltage on the board and GND must have at least one associated test point/pad/pin
- Every board must have at least one power or GND plane/pour.
- Has at least one trace switch layer through a via.
- One via size used throughout the board. Areas with high current just use multiple vias
- **Layout passes HOPE DRC requirements detailed here**

Compute - Required

Your project is required to have some form of compute element, either a microcontroller or FPGA. **Microcontroller module boards and single-board computers such as Raspberry Pi Pico are not allowed**. The microcontroller must have one BOOT switch/button and one RESET switch/button (follow the HOPE Reference schematics). The default microcontrollers we suggest are ESP32-S2-SOLO-N4 or the Raspberry Pi RP2040. See Reference Schematic Section. With other ESP32 models such as the WROOM, there

are additional circuit requirements including a UART bridge. Staff will try to provide support for these other microcontroller selections but cannot guarantee it.

Additionally, **every board will require a USB-C interface to their computer**. An example schematic will be provided for how to wire this interface to the SOLO-N4 and one for the RP2040 is available online. This is to standardize debugging and interfacing for staff. Other interface types such as Micro-USB or non-USB interfaces will be reviewed on a case-by-case basis. A board with no interface to the onboard computer will not receive the points associated with the Compute topic.

Finally, **every project must use one data bus** for communication to an IC, sensor, actuator, other microcontroller, etc. A GPIO pin connection to an analog signal or a digital, unbiased signal won't receive credit. Example data buses are included below:

- I2C
- SPI (If using RP2040, the QSPI bus to FLASH memory does not count)
- Discrete UART (USB interface does not count)
- CAN
- PCIE
- Ethernet

Power - Required

Your project is required to have at least one power system to supply power to the board without the use of a benchtop power supply. That supply can be USB, a battery, a wall outlet, solar panels, etc. In addition, at least one voltage conversion or switching current division scheme must happen on board, such as 12V to 5V step down or 1.8V to 3.3V step up or relay/MOSFET switches to divide current between different actuators. **Do not use resistive dividers to step voltage down or divide current for power applications, they won't get you credit for this topic.**

Finally, all designs must have some form of power circuit protection and/or transient/ESD protection as discussed in lecture. If you are unfamiliar with this topic, please come visit staff in office hours.

Sensing

This category includes the use of any sensors to record, display, and read data related to the board state or the external environment. Examples include voltage sensors, current sensors, temperature, humidity, pressure, microphones, GPS, Lidar, etc. Again, no breakout boards can be used to receive credit for this topic.

Also, the emphasis of this topic is not solely on reading data from a sensor. **The second half of this requirement is some action from the board with respect to what is recorded on the sensors**. The goal is to have a design that completes the following statement: "After collecting/reading the _____ data from this _____ sensor, our board moved/changed/sent/switched _____ ."

Actuation

This category includes any type of actuator that leads to an electrical, mechanical, or optical response from your board that affects the external environment or board itself. Examples include speaker systems, unique LED display shows, motor actuation, gear actuation, radio transmissions, laser firing, etc.

Similar to the Sensing category, the sole requirement is not to spontaneously power a motor on or turn on an LED. For any actuation system, there must be a deterministic controller on the actuator that responds to some feedback from the board (ex a sensor, an internal timer, an interrupt, a button press/switch). Open loop or non-deterministic controllers will be considered on a case-by-case basis.

Other

(Other can only count as a topic for a max of one functionality, ie can't use Other for 2 of the 3 topics)

This topic covers unique aspects of projects that add a level of complexity and merit equivalent to one of the first 4 topics. We will evaluate this topic on a case-by-case basis and it will need to be approved at the time of the Project Proposal. **If your team has not confirmed with HOPE staff that they have an element that can be classified as Other by the Project Proposal deadline, the HOPE staff reserves the right to not grant credit for an Other topic element that does not meet the complexity we expect from Other-level project elements.**

Past examples of Other topic functionalities include:

- A intricate enclosure made with additive manufacturing techniques that not only encloses the project but properly mounts and secures it to aid in the project goals
- Multiple boards that interact with each other and are co-dependent
- Onboard analog processing/custom filter designs
- Folding/Origami PCB designs

Project Schematic

Due: March 15, 2024

This part of the project is to show us a completed schematic of all the components wired together and connected. For this submission, all components of the project should be preliminarily selected, connected on the schematic, and assigned proper values, **THIS INCLUDES PASSIVE COMPONENTS and power sources/power flags**. Also, please label wires and buses with descriptive names to help staff during the review. While this is not the final schematic and BOM we will submit to the fab house, the more complete and fleshed out this submission is, the more feedback you will get and the better grade you will receive. See the rubric in the Grading section for more details.

Deliverables

- **Submit One Set of Deliverables per group to BCourses Assignment and complete the Checkoff quiz individually**
- Project BOM - template exported as a PDF
- Project Schematic - exported as PDF
    - Instructions
    - In the schematic editor, go to File > Plot... Select an output directory and make sure the Output Format is set to PDF. Click Plot All Pages to Plot.
- Project Schematic Checkoff Bcourses Quiz - Complete individually

Project Layout

Due: April 1, 2024

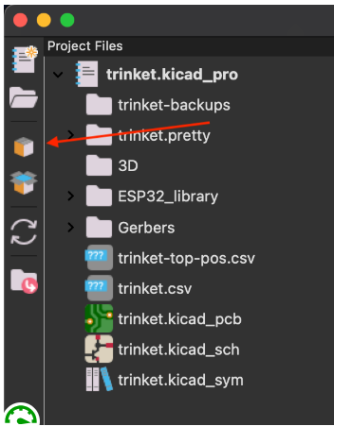
Layout must pass HOPE DRC Requirements detailed here

This part of the project is to show us a completed layout of all the components both placed on the board and routed together. **Power planes and GND pours are also a must**. For this submission, all components of the project should be selected, connected on the schematic, assigned proper values, placed on the PCB layout, and routed together. **NO RATS NEST.** Also, please label wires and buses with descriptive names to help staff during the review. The only major item the staff will not take into consideration during grading is the state of a project's silkscreen.

This submission should see improvements over the Project Schematic submission, implementing advice from HOPE staff, and other general improvements. **This should not just be a completed layout of the schematic you submitted in the previous**

submission. While this is not the final layout, schematic, and BOM we will submit to the fab house, the more complete and fleshed out this submission is, the more feedback you will get and the better grade you will receive. See the rubric in the Grading section for more details.

Deliverables

- **Submit One Set of Deliverables per group to BCourses Assignment and complete the Checkoff quiz individually**
- Project Layout Checkoff Bcourses Quiz - Complete individually
- Project BOM - template exported as a PDF
- Project Schematic - exported as PDF
    - Instructions
    - In the schematic editor, go to File > Plot.... Select an output directory and make sure the Output Format is set to PDF. Click Plot All Pages to Plot.
- Project Layout - submitted as two jpg/png screenshots
    - One screenshot with F.Cu and F.Silkscreen showing, and the other with B.Cu and B.Silkscreen. *Turn off all other layers, including Fab, Eco, User, etc.* Make sure your Edge. Cuts are also visible for both pictures.
- Project Zip File
    - 2. **Submit your project as a zip by using the "archive all" tool** in the KiCad main menu:
    - 

Project Design Review

Slides Due: April 2, 2024

In Class Project Design Reviews: April 3 and 4, 2024

This task of the project will be an in-class peer design review. You will be asked to discuss your component choices, your design decisions in your schematic, your layout design, etc. Each project group will receive 5 minutes to present and 2 minutes for staff and fellow students to ask questions and provide feedback. You must have a complete schematic and layout for this task. **This task is not a presentation on the idea of your project, the challenges you aim to overcome, or whose research lab the project is associated with. This presentation should be technical, not qualitative.**

Deliverables

- **Submit One Set of Deliverables per group to the BCourses Assignment and complete the Checkoff quiz individually**
- Project Design Review Checkoff Bcourses Quiz - Complete individually
- Please submit a set of presentation slides to the Bcourses assignment for the PDR
    - Follow this template

Project Final Design

Due: April 11, 2024

Your layout must pass DRC with the standard HOPE design rules, see HOPE Project Layout Design Rules.

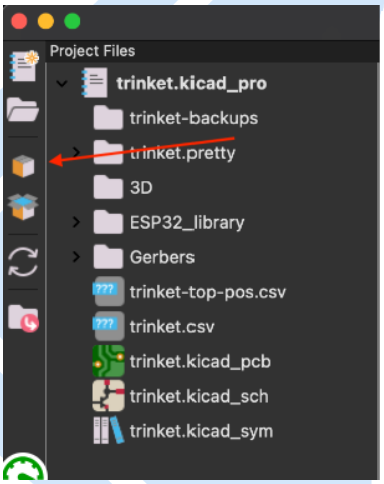
This submission should be the finalized gerber and drill files for the board, as well as the finalized BOM and project zip. The files submitted for this part of the project will be sent to the manufacturer to begin production of the boards, **DO NOT BE LATE.** We will buy all components specified on the submitted BOM up to the budget limit. Your budget must account for enough components for two copies of your project board. If a group does not account for a BOM equivalent to two copies of the project, HOPE staff will buy enough of each component to make two copies up to the budget limit, so a team may not have all their required components purchased during assembly if two copies of the project exceed the $100 budget.

A note on multi-board projects: By default the $100 budget applies to two copies of the project, not two copies of each PCB for the project. What this means is if you have two boards in your project each with $30 of components, the total cost would be $120 of components and your project will be over budget so not all components may be purchased.

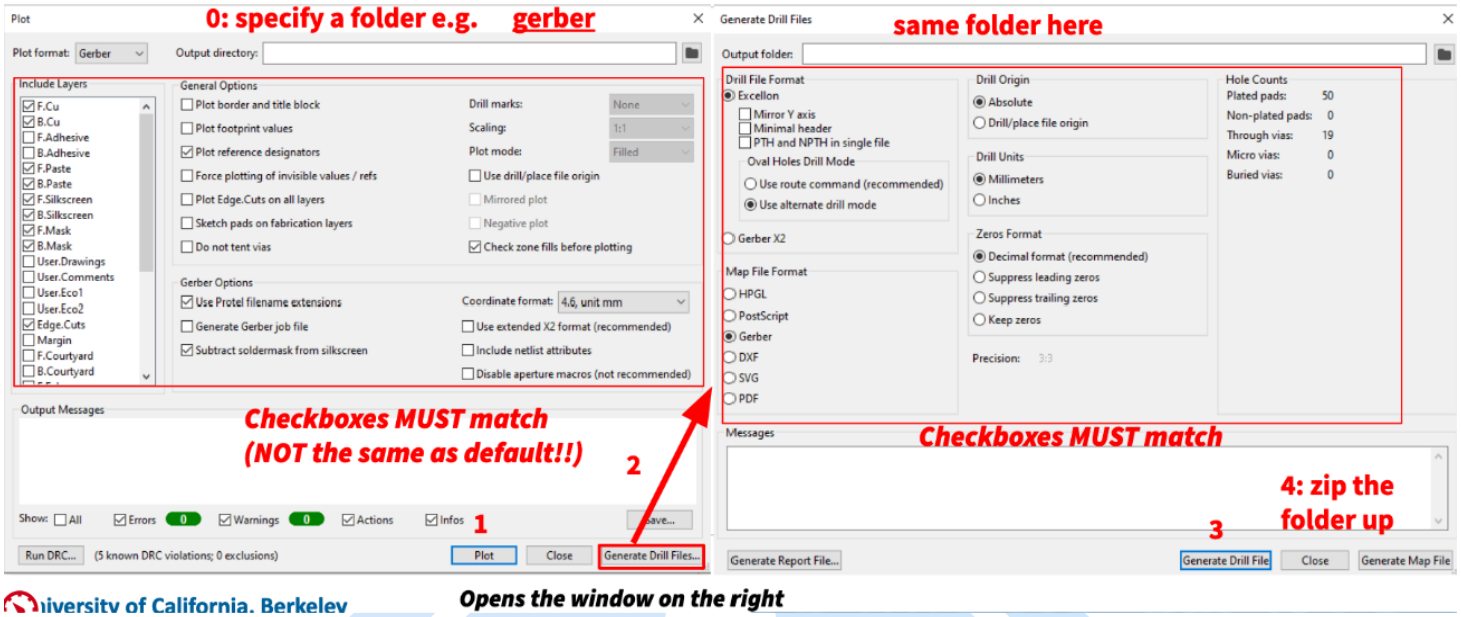
HOPE staff will review requests for a larger budget on a project on a case-by-case basis but extra budget allocations are extremely rare. Requests must be

submitted by email to HOPE staff at least 1 week before the Project Final Design Deadline.

Deliverables

- **Submit One Set of Deliverables per group to BCourses Assignment and complete the Checkoff quiz individually**
- Final Project Submission Checkoff Bcourses Quiz - Complete individually
- Your project submission should include **at least 3 files (project zip, manufacturing zip, BOM csv/xlsx), in addition to a comment with a link to a passing instantdfm run**
- Entire KiCad Project folder zipped (ie. project.zip)
    - Use the "archive all" tool:
    - 

    - Click the closed box icon. "archive all", in the left-hand toolbar of the KiCad main project menu
    - Include any non-KiCad-standard libraries, graphics, etc if you used them.
- Export manufacturing files (gerbers and drills), and zip both the gerbers and drills in one file (ie. manufacturing.zip)



- Project BOM matching the format of the Template BOM (separately uploaded) (ie. bom.csv or bom.xlsx)
    - Make a copy of the template
    - **REQUIRED: Upload your BOM on Digikey to see if there are any glaring issues (ie. out of stock parts, Marketplace Products, etc.). (You can do this too for Mouser to check).**
- Want to see: Part prices, Total price, quantities, part designators, descriptors, part links + datasheet where applicable
- **If you are ordering a stencil and have been approved for it by HOPE staff beforehand, please add a comment in your submission**
- Feel free to submit anything else to help us understand your project/progress
    - Include screenshots of your layout front/back and PDF/image of your schematic to make it easier to review
    - Enclosure designs, drawings, etc.

Project Assembly

This is a two-week period after boards are received from the manufacturer until the Final Project Expo. During this period, teams will be given their components and boards. It will be each team's responsibility to assemble and program their projects outside of class and during HOPE staff office hours to prepare for the Final Project Expo.

Project Final Presentation

Project Final Presentation Checkoff Quiz: May 2, 2024

Project Final Presentations: May 3, 2024

Be present for the HOPE Showcase from 2-4 PM in Cory Courtyard on Friday, May 3, 2024, to give a 5-10 minute presentation with Q/A to HOPE staff, industry engineers, and professors about your final project. **Part of your grade for this assignment is the quality of the presentation, so practice, practice, practice and make a strong set of slides.** While other groups are presenting, you will be tabling with your project demonstrating it to fellow students, guest attendees, and engineers who are not viewing presentations. **You must be at the showcase for the full 2 hours, but if there is a major conflict (ex. class showcase) please reach out to staff so we can work out a solution.**

Deliverables

- **Submit One Set of Deliverables per group to BCourses Assignment and complete the Checkoff quiz individually**
- Project Final Presentation Checkoff Bcourses Quiz - Complete individually
- Added project to the final project website
    - Link: TBA (ex Last semester's showcase site)

Deadlines

Deliverables are due at 23:59 PM (PST) on bCourses (or Google Forms) at the following dates:

2/23 - Project Group Form and Project Group Checkoff Quiz

3/1 - Project Proposal Form and Project Proposal Checkoff Quiz

3/15 - Project Schematic, Draft BOM and Project Schematic Checkoff Quiz

4/1 - Project Layout and Project Layout Checkoff Quiz

4/2 - Project Design Review Slides and Project Design Review Checkoff Quiz

4/3-4/4 - Project Design Reviews (In Class)

4/11 - Project Final Design and Project Final Design Checkoff Quiz

4/24-4/25 - In-class Final Project Parts/Board Pickup

5/2 - Project Final Presentation Checkoff Quiz

5/3 - HOPE Project Expo and Final Project Presentations

Grading Policies

Plagiarism Policy

Our staff have a zero-tolerance policy for plagiarism of past HOPE project schematics/layouts or other online schematics/layouts. Egregious copying of a schematic and/or layout is not acceptable and will result in an immediate 0 on that part of the project. We understand that more advanced parts of designs may require online inspiration and using the same ICs that other parties suggest as well is okay. Reference designs are part of our field; if you use them as a subpart of your project, please disclose them. As long as the majority of the project is unique to your design, staff will not have a problem with it.

One example of a violation of this policy would be duplicating an open-source KiCad (or other PCB ECAD or Module board manufacturer) project available online and moving/relabeling components.

If you use an online project as inspiration, please include a reference to it.

When in doubt, ask the instructors!

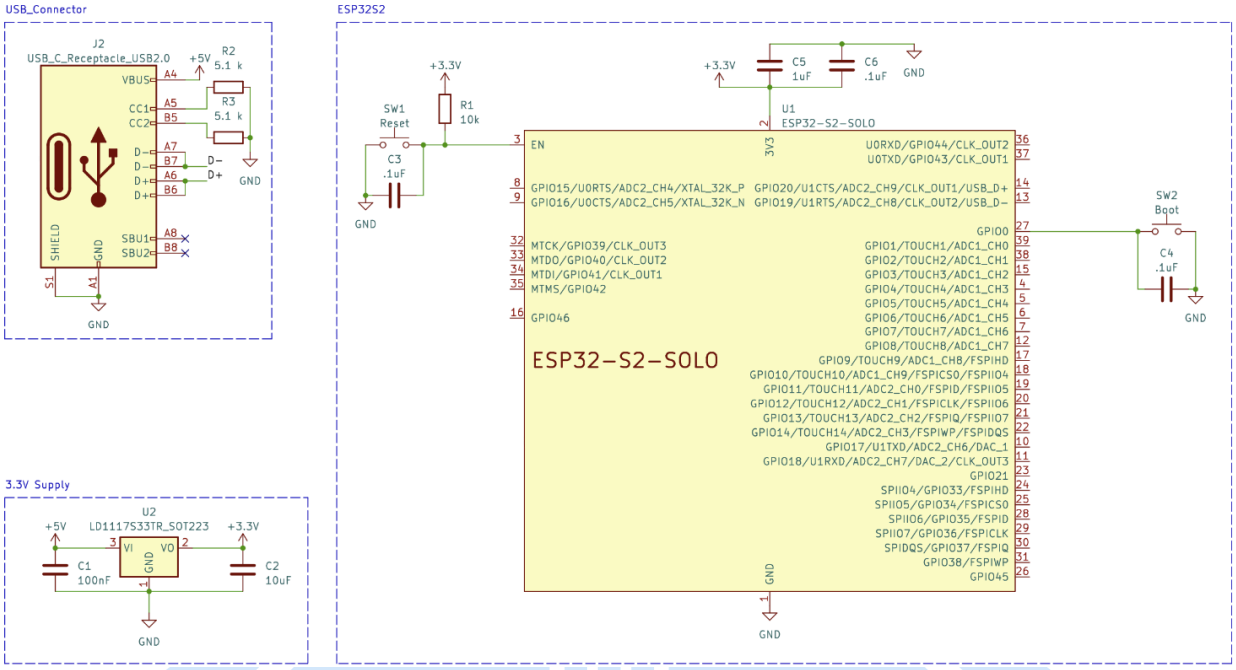
Large Language Model Policy

ChatGPT and other large language models are a part of our world now and can be very useful for answering questions quickly and finding resources. Recognize that despite their power, LLMs are not very good engineers, so do not use them to write your project proposal or any technical descriptions for this final project. First, they may write about a design that you are not familiar with and won't be able to explain to the staff at the design review. Second, they will more often than not write a very vague description of a PCB project, ignoring detailed circuit descriptions, and therefore won't land you a better grade on the proposal.

ESP32-S2-SOLO-N4 and RP2040 Reference Schematics

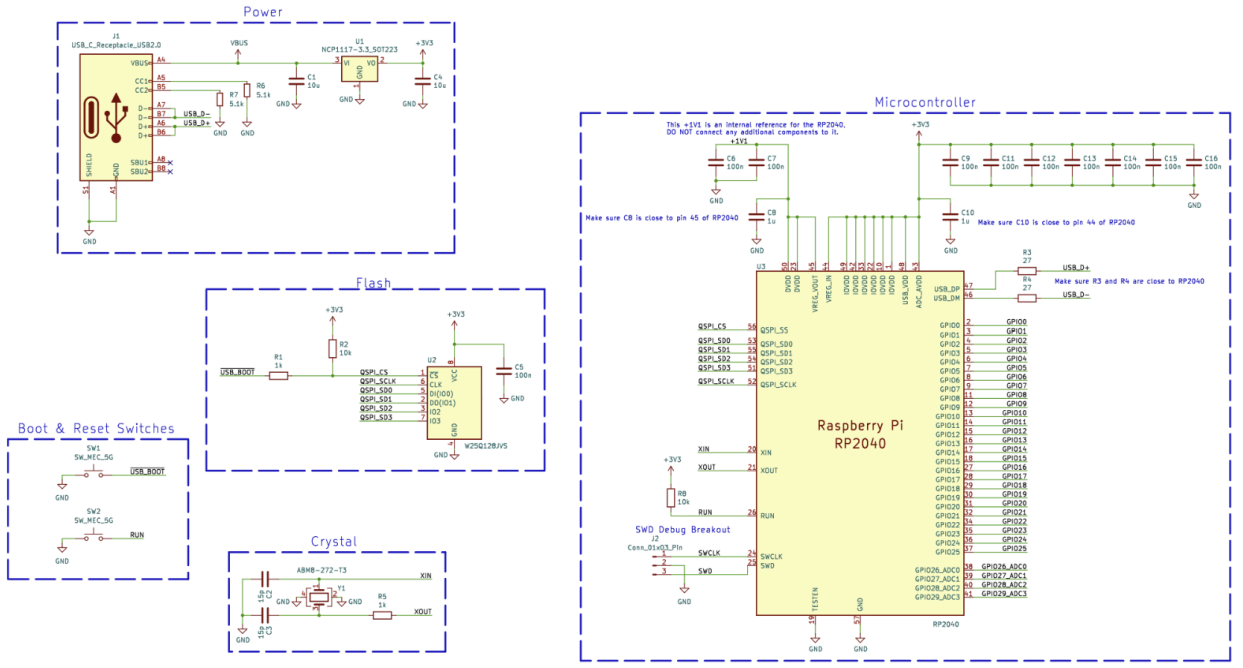
ESP32-S2-SOLO-N4

[Reference Design Files + Schematic PDF]



RP2040

[Reference Design Files + Schematic PDF]



FAQs

1. Am I allowed to use parts I may already have?
    - Yes. We do recommend putting these parts in your Bill of Materials and indicating they have already been purchased (set cost and quantity to zero)
2. Am I allowed to purchase parts that may not be approved or may lead to my project exceeding the budget?
    - Yes,
3. Am I allowed to use stencils?
    - This is up to HOPE staff discretion but the purchase of stencils and solder paste is the student' pr. (Supernode and Jacobs may have paste to borrow and Supernode has an oven)
4. Am I allowed to use non-KiCad ECAD software to design my PCB?
    - Generally not recommended as staff may not provide support.
5. If I happen to be in a group across sections do I need to do the design review twice?
    - No, come to the most convenient one for group members. If not possible, relay information to each other. The design review is a group-based grade.
6. Will we be required to write code for our project?
    - Yes, part of your final grade for this project will be its final functionality and completeness. This includes the firmware/code running on the PCB.
7. My club/research project requires a certain breakout board or single-board computer for this project. What should I do?
    - We understand that projects for clubs and/or research may have constraints that you cannot change. If this is the case, you can include these components as long as the basic requirements are still met if they were not included in the project. If this is not possible, we generally recommend you choose a different final project but please feel free to reach out to HOPE staff to discuss alternatives.
8. How many hours will this project take?
    - This always depends on the complexity of the design; however, generally:
        - Iterating to a finalized schematic should take about 8 hours to complete. In one sitting, a good schematic takes about 6 hours
        - Iterating the layout from the first draft to completion takes about 15 hours to complete. In one sitting a good layout takes about 8 hours
        - Parts selection should take about 3-5 hours
        - Assembly+Coding should be 5-10 hours