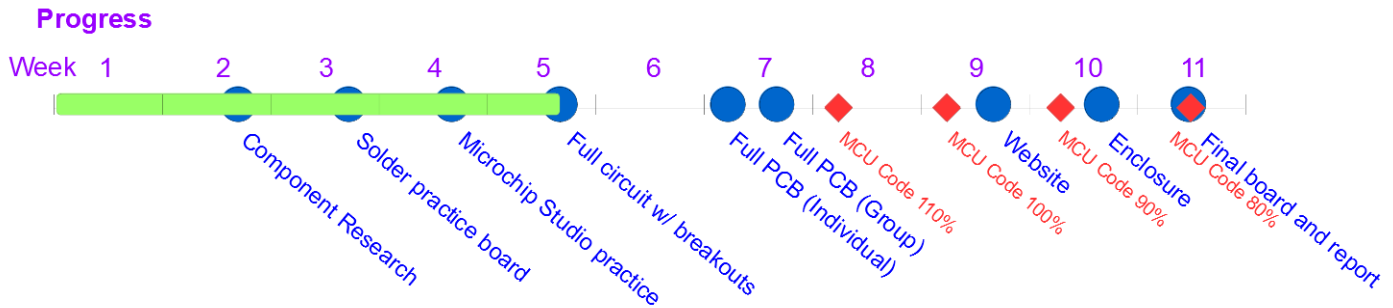


Task 5: PCB Design

Due: Feb 18, 2025 (Individual)
Feb 20, 2025 (Group)



This assignment will have both an individual and a group component. Please read all of the instructions carefully to avoid having to redo parts of your work.

For this assignment, you will practice designing in Eagle. First, you will make a custom component and a breakout board for it. Then, you will design your final webcam PCB in Eagle. Make sure your breakout board webcam is able to pass all of the tests in Task 4 first, so that you can be confident of the connections. You may use any resources, including my library and breakout schematics, available on Canvas, when designing your PCBs.

1. For this part, you will create a custom component in Eagle. Then, you will make a breakout board for it. This part must be completed individually, and each person must submit their own files. The component is the LTC2942-1, which is a chip that measures how much battery is left. Measuring voltage in a battery is a bad indicator of the remaining charge, as it drops very quickly towards the end of its life. However, this chip measures how much current is actually being used, so it can tell you how many mAh are left in your battery. The datasheet is available along with the assignment on Canvas. **Submission: each person must submit their .lbr, .brd, and .sch files to Canvas (under “Task 5, Part 1”). You do not have to submit the Gerber files for this part of the assignment.**
 - (a) Create a custom library component for the LTC2942-1, and put it in your own library. In your component, include at least the following:
 - Footprint
 - SMD pads
 - Silkscreen outline
 - Pin 1 indicator
 - Name and value placeholders
 - Symbol
 - Pin names
 - Name and value placeholders
 - Device
 - (b) Create a simple breakout board for the LTC2942-1. Include at least:
 - Headers for each of the pins (except for the Exposed Pad).
 - The bypass capacitor shown in the Typical Application circuit on page 1 of the datasheet.
 - Labels on the silkscreen for each of the headers.
 - A name for your board, as well as the date.

2. Here, you will design your final embedded webcam. First, everyone will design their own. Then, you and your partner will decide together on the best board to submit for manufacturing.
- (a) **INDIVIDUAL PART:** For this part, you will design your final webcam PCB individually. Please follow the specifications below. **Submission: each person must submit their .brd and .sch files, as well as a zipped folder containing their 10 Gerber files, to Canvas (under “Task 5, Part 2(a)”).**
- Your board must use two layers.
 - Your board must be no larger than 50×50 mm (i.e. it must fit within a 50×50 mm square). Your WiFi antenna or part of the barrel jack may be hanging off the board, which does not count in the measurement.
 - Your board must be a rectangle (or square).
 - Use the “Dimension” tool in the layout to find the dimensions of your board (in mm), and include those on the “tDocu” layer. Please include units.
 - Include at least two mounting holes (with 3mm diameter) in opposite corners. You can have more than two for stability. Make sure there are no components within 2.5mm of the center of the hole (to account for the screw head).
 - Include a 2×3 through-hole programming header (with 0.1 inch pitch) for the MCU. Look at the MCU breakout for reference. These will be the pins used for programming your MCU. (Note: If you use the same pin configuration as on my breakout, you will be able to use the same programming adapter that you made before. Otherwise, you will have to make a new one for your custom pin configuration.)
 - Include a 2×3 through-hole header (with 0.1 inch pitch) for communicating with the WiFi chip directly. This should include RX^{MCU} , TX^{MCU} , RX^{DEBUG} , TX^{DEBUG} , PWR, and GND.
 - Connect PWR to the input of your voltage regulator, so you can power your board using the 5V output of the USB-to-UART module.
 - The RX^{DEBUG} and TX^{DEBUG} pins will be used to program the ESP32 and to listen to debug information, while the RX^{MCU} and TX^{MCU} pins will be used to listen (and potentially talk to) the MCU directly.
 - Include the LEDs and buttons from the breakout webcam, as well as an LED indicating when power is connected. For the LEDs, use 0603 packages, just like in the ESP32 breakout. For the buttons, use the same through-hole buttons as in the breakout webcam.
 - There should be a total of 4 LEDs (3 status indicators and 1 power indicator).
 - There should be a total of 4 buttons (MCU reset, provisioning mode, WiFi reset, IO0).
 - Use an SMT barrel jack for a power supply. This will provide 5V. The datasheet of our barrel jack can be found on Canvas. Make sure to verify which side the cable plugs into!
 - Provide the following labels. These should be on the “tPlace” and “bPlace” layers, and they should be readable (e.g. not obstructed by a component or vias). The size should be at least 24 mil.
 - Labels for your buttons and LEDs.
 - Labels for each header pin.
 - A board title, including at least the name of the board, the version number of this camera (v1.0 for you), and the date.
- (b) **GROUP PART:** For this part, you will submit a single PCB for both you and your partner, which will get manufactured. You can choose the better of your two designs. This is a chance for you both to look over each other’s board, and to find possible mistakes and make corrections before the final order goes out. If you want to challenge yourself, try making your board as small as possible. For reference, in a previous year one group made a board that was 38×30 mm. **Submission: each team must submit their final .brd and .sch files, as well as a zipped folder containing their**

10 Gerber files, to Canvas (under “Task 5, Part 2(b)”). In the comments during submission, please include the dimensions of your board, as Length×Width (please don’t forget, as you will lose points if you do not include this).

FAQ

Q. What Gerber files should be included in the zip file?

A. The 10 files are: GBL, GBO, GBP, GBS, GML, GTL, GTO, GTP, GTS, TXT. There is also an extra auto-generated folder, which you can also keep, making a total of 11 items.

Q. How do I import a library into Eagle?

A. You have 2 options. You can put it in the same folder as every other Eagle library, which should be in the Eagle installation directory under “lbr”. Alternatively, you can put it somewhere else (for example with all your other course files), and add that directory to Eagle’s library path by going to the Control Panel (i.e. the main screen when you open Eagle), clicking on Options and selecting Directories. There, you should see the paths for various parts of Eagle, and you can add your folder to the Libraries path. Paths are separated by semicolons (or colons for Mac/Linux). For example, my Libraries entry reads “\$EAGLEDIR\lbr; C:\Users\Ilya\Google Drive\Teach\326\Eagle”.

Q. How should we connect the Groundbreak pin of the barrel jack?

A. Connect it to GND.

Q. Can the silkscreen layer hang off the board space?

A. Yes, however, it will not be printed on the PCB. Components cannot leave the board space.

Q. Although we set up the ground plane, there are still air wires telling us to connect to ground. Is there a setting that we should change?

A. For the GND connections, there is a good chance your GND plane got broken when you made other connections. You must either connect them manually or use vias (whose name you change to GND) to make the connection.