Final Report on Engineering System Design

Chenqi Guo Group 10

Introduction

For this class, we need to build a project about Web-Cam, both the hardware system and the software.

Our Web-Cam can take pictures from time to time and transmit the images to the web through WiFi chip.

For the hardware, we use ATSAM4S8B as MCU, AMW136 as WiFi chip and OV2640 as camera. First, we select the corresponding components, like crystal and voltage regulator for our project. Second, we use breakout board to achieve the required function, coding for the MCU and setting the WiFi. Third, we make a whole PCB board to combine those modules together. Finally, we use 3D printer to make an enclosure for our Web-Cam.

For the software, we use Eagle to draw our PCB design, use Atmel Studio to code for MCU, use Tera term to communicate with WiFi chip, use Brackets to make our web pages, use Onshape to design our 3D enclosure.

Design Process

1. How we approach our project

1.1 PCB Design

As discussed in class, to start with a hardware engineering system, we should first make the

breakout board for each module, and then test on each module to make them work together as a whole system. In this project, we have three modules: the wifi module, the MCU module and the camera module. After getting our breakout board work, we started to construct our final PCB design. We use software called EAGLE to draw the schematic graph of our project circuit, and then convert this schematic graph into PCB board.

For the final PCB design, instead of simply combining each module used before together, we should make some necessary modification:

1) We should add at least two mounting holes in opposite corners on the board for assembling later.

2) We should add a barrel jack for the power supply.

3) We should get rid of the unnecessary headers and connect the corresponding pins together directly.

4) We should get rid of the unnecessary crystal.

5) We should add two buttons for reset MCU and set WiFi into setup mode on the PCB board.

6) We should connect the RTS, CTS, RX, TX pins on WiFi chip with CTS, RTS, TX, RX pins on MCU respectively.

Besides, we also need to make the wires in the PCB board neat and without acute angles. Our group design 2 PCB boards, and selected the better one as our final PCB.

The final PCB board can be seen in figure 1.1.1.

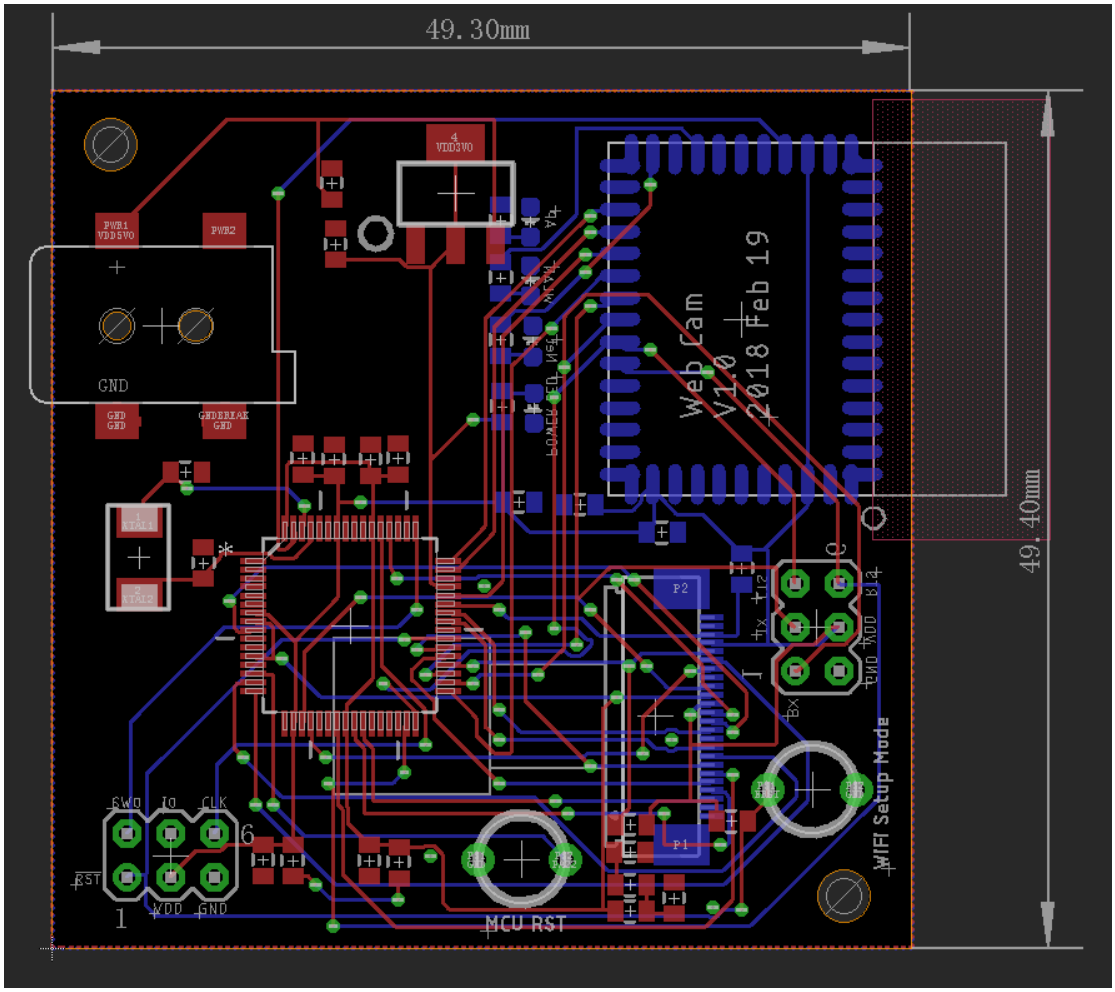

Figure 1.1.1 Final PCB board

1.2 C Code

For building Webcam software, Atmel SAM4S8B microcontroller is based on C programming language. Atmel Studio 7.0 is used for development system and it provides ASF(Atmel Software Frameworks) that we can apply this to our project.

1) Reading our development board on Atmel Studio 7.0: check the board connection and target voltage.
2) We set ASF wizard and open IO port example; modifying example code to our board operation test.
3) We should construct code structure for an WiFi(AMW136) chip and Camera(OV2640)

chip with ATSAM4S8B; get reference USART serial communication and OV7740 CMOS sensor example project.

4) For effective management software, making each part of .c and .h file.

5) Coding for wifi communication on wifi.c, and pin definitions, parameters and variable declarations within wifi.h.

6) We also wrote code for camera on camera.c and camera.h

7) Constructing conf_board.h, conf_clock.h and main.c for initialize on general board.

8) Checking and setting the WiFi communication with tera-term terminal tool and camera operation with debugging.

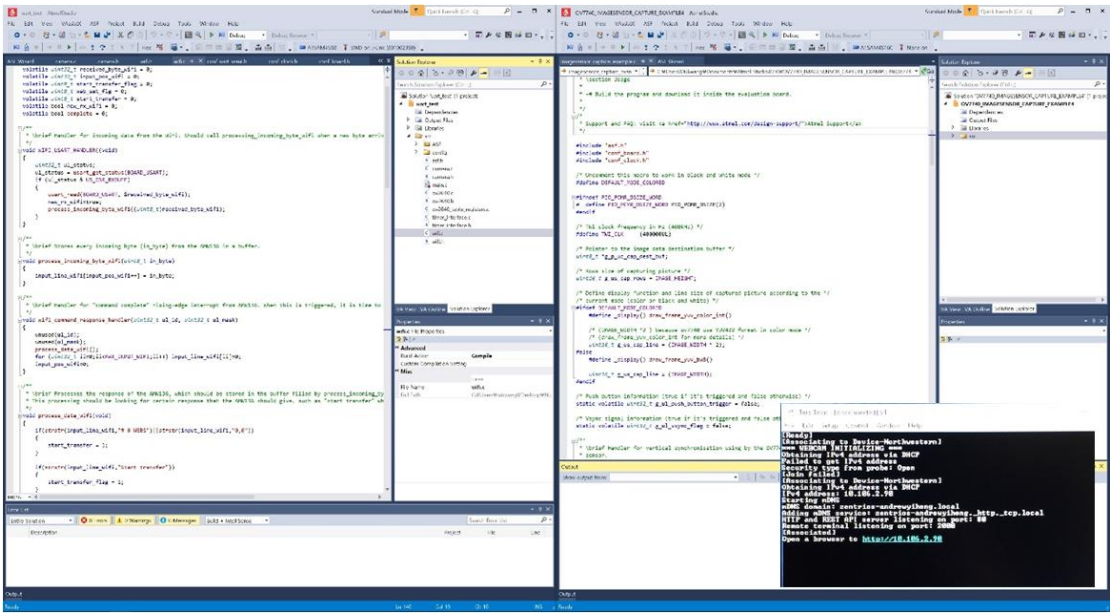

Figure 1.2.1 Software development environment

1.3 Website

For successful webcam service, we should design three web pages: a homepage which introduces system based on our board system; a webcam page for camera operation and timestamp for real time operation with start/stop button; and an info page which introduces our group member information with pictures. We used Brackets which is HTML, CSS, and JS editor.

1) We drew a wireframe for each page we plan to make.

2) index.html: This is our homepage that it includes title, navigation button, current time, system information and our development board picture. We made navigation tab button style and clock box on homepage_style.css and function of clock on time.js.

3) info.html: This is member introduction page with our pictures with info_style.css

4) webcam.html: This is for our webcam operation. We made this page based on professor's webcam page. We modified about start/stop button working with adding timestamp function in webcam_functions.js which contains the necessary functions for Websockets operation.

Additionally, we made common background color and menu buttons animation such as hover and visited on css file. The Menu has also can connection to our AMW136 server for management. After design website, we checked mobile web service based on Nexus 6P within Chrome Devtools.

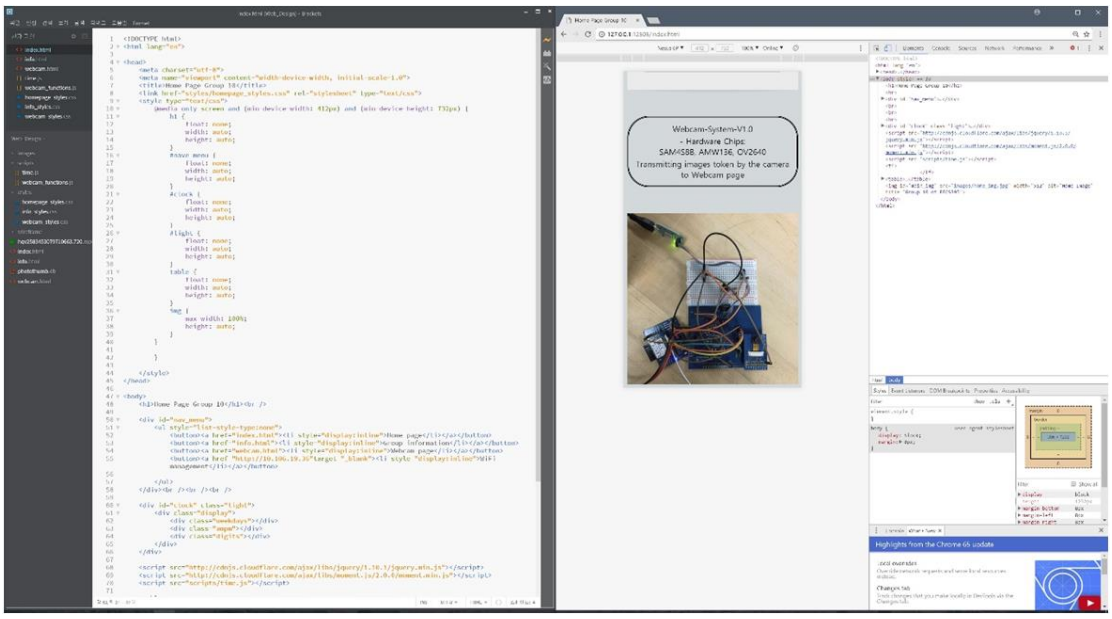

Figure 1.3.1 Web design environment (Brackets and Chrome Devtools)

1.4 3D design

We also need an enclosure for our Web-Cam. In this part, we use On-shape and 3D printer.

To achieve the basic function for the Web-Cam and to make it look better, there are few requirements that we should meet:

1) To make the enclosure printed faster but also strong enough, we set all of the walls to be 1mm thick.

2) There should be cutout s for the power supply, camera and WiFi antenna.

3) There should be holes for two buttons and LEDs.

4) There should be wall mount holes which can orient camera correctly.

5) All structures on the PCB should be supported.

Since our final PCB board turned out to be not usable, at last we use professor's PCB board.

The 3D design of this enclosure by Onshape is shown in figure 1.4.1.

In fact, this is the 3rd Version. In the 1st Version, I set the wall to be 2mm thick, but it turned out that there is no enough room inside the box to put our PCB board. In the 2nd Version, the position of the camera cutout did not align well with the camera, and the enclosure was too high to be assembled with the screws we are provided in the lab. The position of button holes are not aligned well either. Finally in the 3rd Version, we made a pretty well enclosure for our Web-Cam.

Figure 1.4.2 shows the enclosure printed by 3D printer.

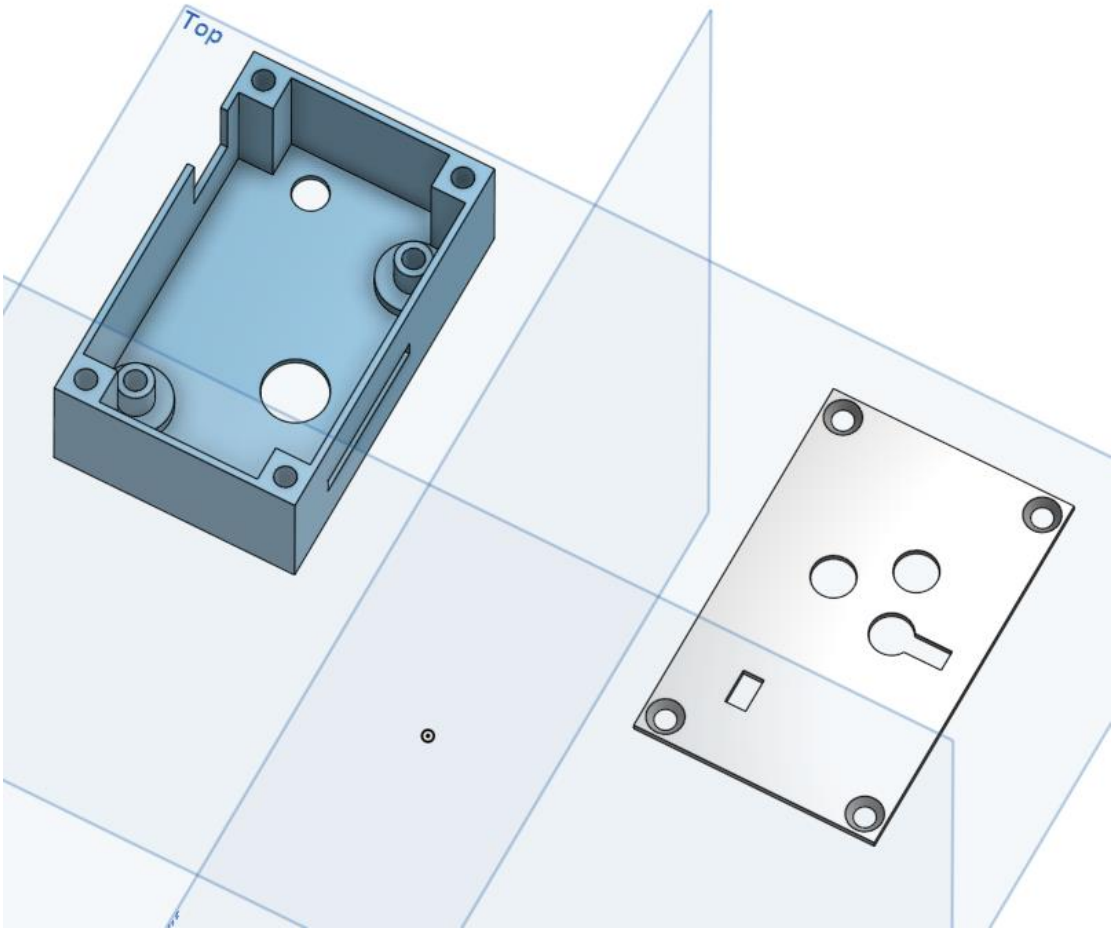

Figure 1.4.1 Onshape: 3D design of enclosure

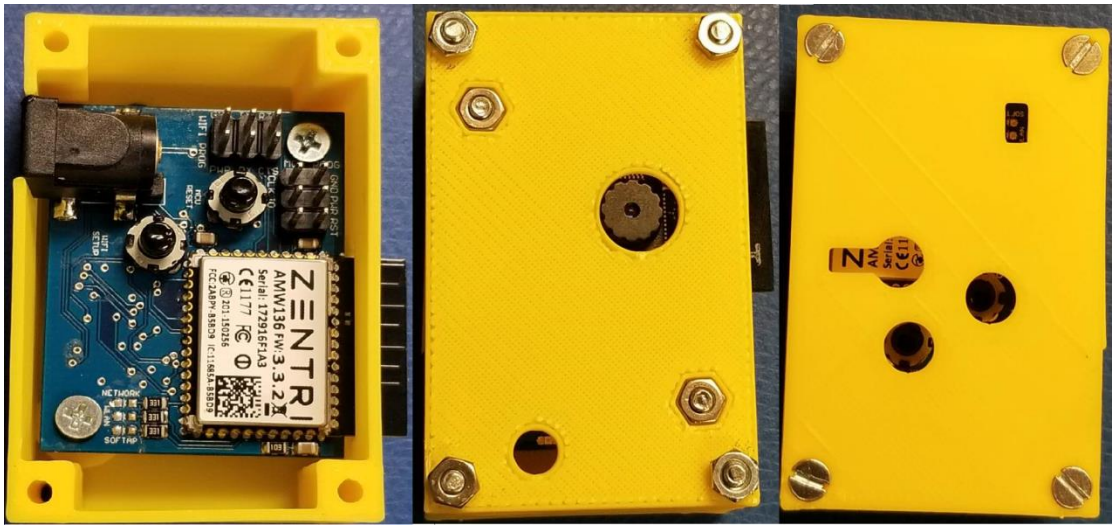

Figure 1.4.2 Physical enclosure: inside, up and down

2. What I learned through the design process

2.1 Challenges I met

Through the whole project, we met challenges almost in every task. Here are the challenges that we struggled with:

1) In task 1, we need to select the components that we want to use in our hardware system, and compute the power cost. We met with difficulty in selecting the voltage regulator, because we need to consider about the current output.

2) In task 2, we need to practice soldering. It was hard to prevent the bridge between pins at the beginning.

3) In task 3, we need to solder the breakout board and test on the MCU and WiFi chip. At the beginning we just cannot type in Tera term.

4) In task 4, we need to make the camera module. At first the reset button for the MCU just cannot work.

5) In task 5, we should design the final PCB board. As discussed in part 1.1, we need to make some modifications for the breakout board and consider about many details. The biggest challenge is how to add some via holes properly so the grounded pins can all be connected to the ground plane properly. Also, the wires should be neat.

6) In task 6, we should make our web page. The biggest challenge lies in the java script file. It was difficult for us to add the time stamp and set a button that can control the webcam stream.

7) In task 7, we should design the enclosure for the final PCB. The challenge is how to design the dimension and positions of holes to fit with our PCB.

8) In firmware design, the coding is hard. We struggled with write_image_to_file function,

start_capture function and find_image_len function.

2.2 How I overcome these challenges

We solved the challenges by:

1) In task 1, to select the voltage regulator, we decided to add the awake current of the three main components in our project, and used this current to determine the voltage regulator.

2) In task 2, to prevent the bridge between pins, we found that if we use less solder, we can reduce the probability of making bridge.

3) In task 3, to type in Tera term, we updated the driver on the computer, and re-soldered the WiFi chip (we used too little solder before so there was a poor contact on the WiFi chip).

4) In task 4, the reason that the reset button cannot work is because the pin for resetting MCU is NRST, so we should let the button connected to NRST and ground.

5) In task 5, we just checked our PCB every time we made some modification and until there was no error. This is very time consuming.

6) In task 6, we just use the Date variable and WebSocket message to add the time stamp, and use variable buttonClicked as a flag to control the webcam stream.

7) In task 7, we just measured our PCB and components carefully to make the dimension and positions of holes to fit with our PCB.

8) In firmware design, to code the write_image_to_file function, start_capture function and find_image_len function, we searched and learned on the Internet for some syntax, and finally made it.

3. Changes I want to make

3.1 Approach to the problem

This project was for webcam software and hardware design. It is perfectly working as webcam service. However, it might be insufficient to use real time webcam because frame-drop and low sensitivity of CMOS sensor. Furthermore, about the board design, our technique for aligning the components and circuit on 50X50mm board size was not enough. Thus, our board might be unstable to solder well and operation. We expect that if we start over this project, we can modify our system for better performance.

3.2 Our actual design

For enhanced system performance and environment, we can consider some design.

1) OV2640 CMOS Sensor: our system camera frame rate is not enough to use effective webcam. OV2640 provides several modes such as SVGA 30fps and 15fps preview for 60Hz light environment, 25fps and 14.3fps preview for 50Hz light environment, and Night mode environment. Thus, we can choose SVGA 30fps JPEG mode and modify with OV2640 table registers.c. About the problem of frame recognition difficulty, CMOS sensor has low sensitivity. We can choose Night mode for enhanced sensitivity and it might be allowed to operation in the dark environment. This mode is only work with one type is set to fixed low frame rate, for example 3.75fps or the other type is set to auto frame rate, for example from 30fps to 3.75fps. However, it is still insufficient to use by good performance. In this case, we can also choose other CMOS sensor which has better performance.
2) For stable PCB board design, we can cluster component with each module. We use two layers PCB design and also construct 3 parts such as microcontroller(ATSAM4S8B), WiFi

part(AMW136) and camera part(OV2640). Thus, if we align onboard components related with these chips, it will be effective to solder and debugging.

4. Why our final PCB did not work

Our final PCB is not perfect and it did not work. Part of the reasons lie in the design of PCB, part of the reasons lie in the soldering.

For the design of our PCB, I set many via holes close to the pins of chips. Therefore when we were soldering our final PCB, we could hardly avoid short circuit at those points. In fact, when we checked our PCB board to see where the problems were, we found that between pins of MCU there were short circuits. Therefore when we want to test our MCU using Atmel Studio, we just cannot read our MCU. Besides, when we test our WiFi chip, it was weird that sometimes it worked well, sometimes it did not. Thus I think there may also be some poor contact or short circuit for our WiFi chip.

Finally, we decided to use professor's PCB instead of ours.

Team Work

I think we contribute fairly to our team effort. I contribute to soldering, PCB design, C code and 3D design; my partner contributes to soldering, PCB design, Website design and C code. We worked together and finally get our Webcam work.

Learning

I take this course because I want to know how to design a whole engineering system, from

hardware to hardware. Also, I want to learn how to solder chips and components on a board, how to design PCB, how to do the embedded programming, how to code a Website. All of these are required knowledge and techniques as an engineer.

In this class, I think I learned as much as I expected. And I do not think there are any topics that I want to learn but did not.

One suggestion for this class is that I think it will be better if we have more office hour, since some of the tasks are difficult and we may need some extra help.

I think the workload is appropriate. We can finish all of the tasks if we just spend our whole weekends in the lab, and sometimes in the evening after class during week days. But we need help in office hour to finish some of the tasks.

Conclusion

This is a hard and time-consuming class. We spend almost all of our weekends for the project, but I think it worth. We learned really a lot and have a very good experience on engineering system design. Professor Ilya is also very nice. He answered our questions in detail and helped us solved some problems we met patiently.

For the structure of the class, we spent about one week on every topics, from soldering to testing the chips and breakout boards, from PCB design to Website design, from C code (which I think is the most difficult part in this class) to 3D print. It really helps us a lot that we can review the class on youtube.

My teammate is also very nice. We worked together to solve all of the challenges. I also learned how to collaborate with others, how to finish a project as a group.