EE 590 Spring 2020
Directed Research
Project Report: RF Location-Broadcasting Bio-Chips System
Instructor: Manuel Monge

Yaxin Deng 4704064815
May 6, 2020

Instruction

This project is focusing on developing a system to localize the miniature wireless medical devices inside the patient's body. Instead of the traditional localization features including electromagnetic, acoustic, and imaging-based method, we introduced magnetic resonance to this new localization system to eliminate the limitations from physical properties of human tissues and organs, or performance of the imaging modality. This system allowed us to use an RF transmitters to output the frequency and compare it with magnetic field to identify the location inside the space. Then from the signal and frequency received, the individual equipment can be localized in a very accurate coordinate inside patients.

On the current research phase, we focus on the parallel process that design this system in silicon intergraded circuit, and verified the design by using the chips. This report is focus on introducing the PCB version of the design by using the prototype chips on the current market, and showing the detail for schematic and layout drawing by using Altium Design. The next step for the project will be to verify the function of the PCB and test that in the laboratory environment.

1. Schematic Design

For this design, we are designing a four layers board to achieve the equivalent function as our silicon integrated circuit design, which including microcontroller, antenna, transceiver, magnetic sensor on the board. Meanwhile, we consider using a coin battery as the permanent power supply for this PCB, and the size for the board should be minimized.

1.1 Main Chips and Functionalities

In this section, I listed the main chips which is using in the following table 1. In this design, we need to support the transceiver to work under 915MHz Industry Scientific and Medical frequency band. We choose ATtiny841 as our microcontroller to upload the firmware and communicate with magnetic sensor and transmitter.

Table 1: List of the Main Chips and Supporting Function

Manufacturing Number	Manufactory	Description	Function in this Project
ATTINY841-MU	Microchip Technology / Atmel	8-bit AVR Microcontroller with 8K Bytes In-System Programmable Flash	ATtiny841 is the microcontroller for this PCB. It allowed us to upload the firmware to support the system communications. This chip support three different functions, we use the SPI and I2C communication in this project. This Microcontroller has a built-in 8MHz Crystal, if consider a higher frequency clock, it can accept external crystal.
IIS2MDCTR	STMicroelectonic	High-accuracy, ultra-low-power, 3-axis digital output magnetometer	IIS2MDCTR is the magnetic sensor in this design. It communicates through I2C bus with the microcontroller. This part can detect the magnetic field inside the system, and generate the signal to microsensor for further operations.
SX1231HIMLTRT	Semtech	Low Power Integrated UHF Transceiver with On-Chip +20dBm PA	SX1231 is the transceiver to receive the signal from the antenna, and communicate with the antenna through SPI communication. This transceiver supports a 915MHz ISM frequency band. This chip required an external 32MHz crystal.
W3070	PulseLarsen Antennas	Ceramic Dual Band Monopole Antenna	W3070 is the ceramic antenna in this project. It can receive a frequency range between 880 – 960 MHz.

1.2 Power Supply Circuit

For the power supply circuit, we use a 3V coin battery as a permanent voltage supply. I list the detail in Table 2, this design combines the voltage boost converter and voltage regulator to stabilize the power provided from the battery, and maintain two voltage channels 3.3V and 2.5V. 3.3 V is the main power supply for chips, and the 2.5 is the power supply to support the isolated I/O pin and Vdd circuit on a magnetic sensor.

Table 2: Power Supply Circuit Parts

Manufacturing Number	Manufactory	Description	Function in this Project
TPS61201DRCR	Texas Instruments	TPS6120x Low Input Voltage Synchronous Boost Converter With 1.3-A Switches	This chip will boost the voltage supply from the voltage across the battery to a stabilized 3.3V. Depends on the voltage drop on the battery, it will provide an operation time range around 2 hours.
ADP222ACPZ-3325-R7	Analog Devices	Dual, 300 mA Output, Low Noise, High PSRR Voltage Regulators	This voltage regulator provides two stabilized channels as voltage supplies. This particular model gives us 3.3V and 2.5V power supply and up to 300mA.
3002	Keystone Electronics	Coin Cell Battery Holders SM COIN CELL BH 20mm	This support hooding 20mm dimensions battery. It allows to hold battery on the back of the board.
CR1220	Murata Electronics	4V 40mAh Coin Manganese Dioxide Lithium Batteries	This battery provides 3.0V 0.1mA as a power supply. The board size will be limited by the size of the battery.

1.3 Overall Design for Schematic

Based on the above chips, we designed the schematic for the board shown below in Figure 1. This design is based on properties shown in datasheets for the chips, and adding capacitors and resistors to finished the connection in between.

On this design, we will need to program this board through the ISP headers on the top layers. Consider the requirement of the Atmel chip, we need to finish the boot loading process before we can program the board through Arduino Software.

The voltage supply decoupling capacitors are modified depending on the voltage supply pin and size for the board. One part I still need to pay extra attention to test after the assembly is the capacitance C24 and C23, the original recommendation was 10uF titanium capacitor for eliminated the ripple for the voltage signal. However, a 15 or 18uF ceramic cap will provide the same feature.

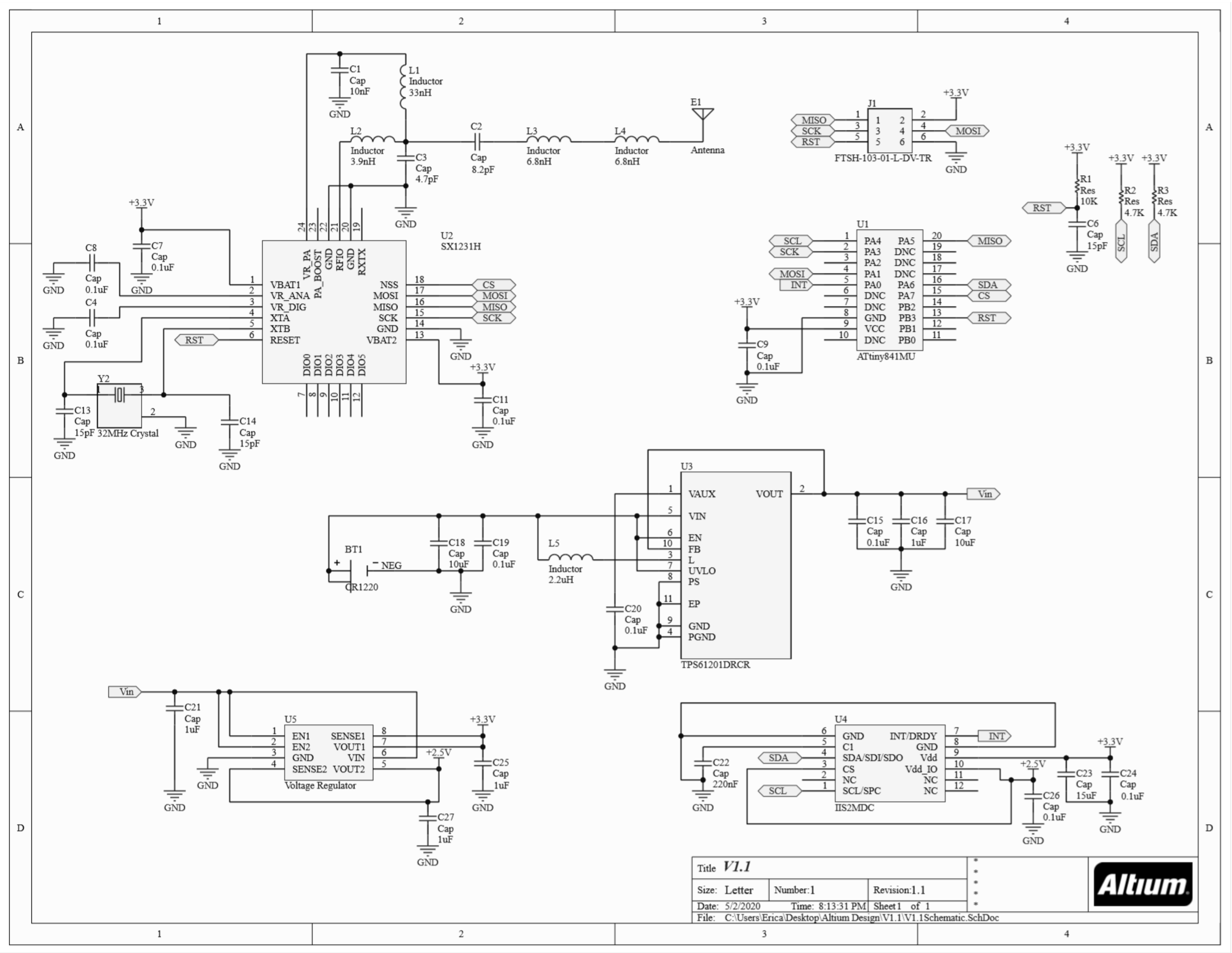


Figure 1: Overall Schematic Design

2. Layout Design

2.1 Overview Layout Design

Based on the schematic design above in Figure 1, I draw the layout shown below in Figures 2.1 and 2.2. This is a 4 layers design, the top layers are mainly the components, the back layer is used for the battery, and the middle two layers are GND and Vdd . The overall dimensions for this layout design are shown in Table 3 below.

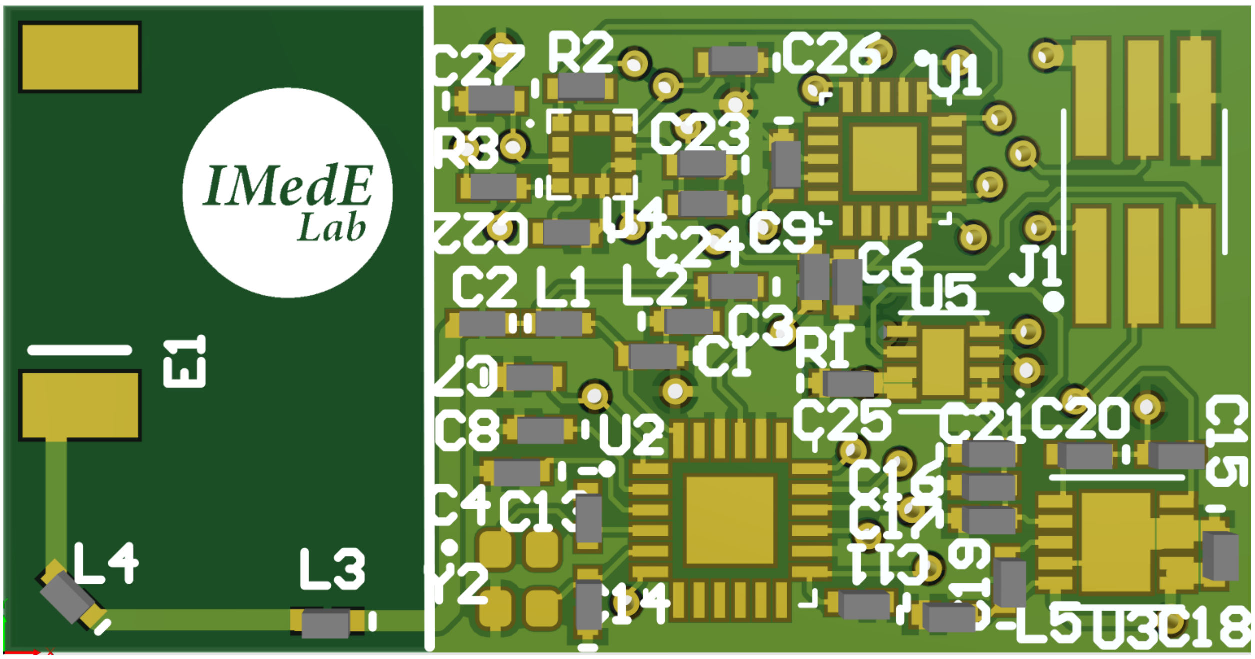


Figure 2.1: Front View for PCB

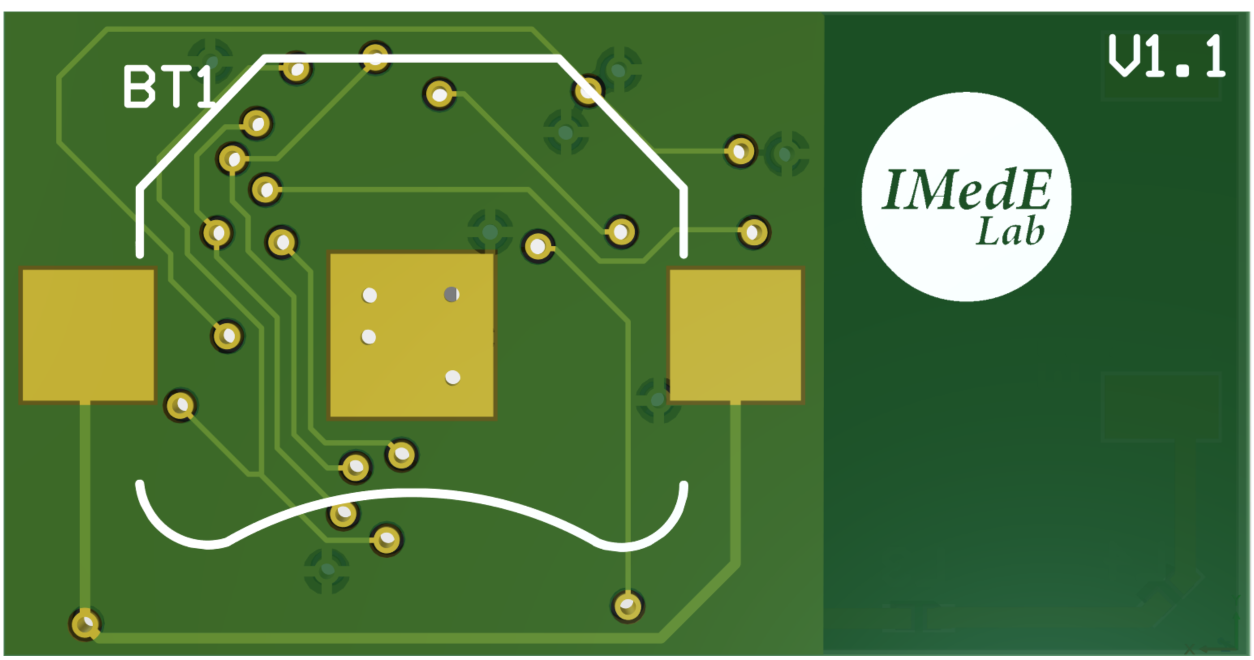


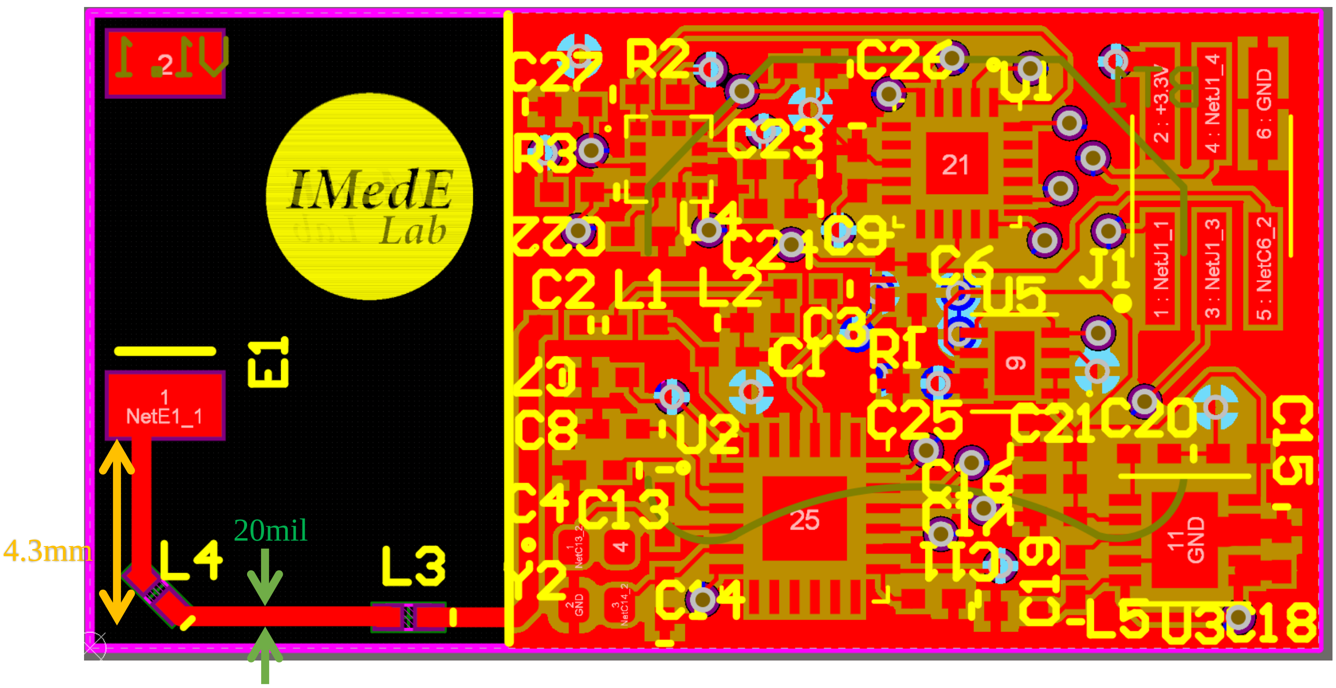
Figure 2.2: Back View for PCB

Table 3: Dimensions for this board design:

Layers	4	*Thickness*	1.6mm
Length	30.3mm	*Base Material*	FR-4 TG 130
Width	15.7mm	*Copper Weight*	1oz
Trace Width	5mil	*Clearance*	5mil
Via Diameter	25mil	*Edge Rail*	5mil
Via Hole Size	15mil		

2.2 Spatial requirement for layout design

At this design, the ceramic antenna requires a special layout for operating properly. Based on the recommendation layout in the datasheet, we scale the dimension to fit the antenna in our design better. So we choose the trace for 20mils due to the Inductor’s 0402 package. Meanwhile, the space between antenna and the trace is 4.3mm, and this distance is shorter than the recommendation data is because we do not have other components around the antenna. Ground plans should be clear under the antenna area.



3. Fabrication

For this particular board, we use a manufactory called Seeedstudio for PCB and assembly. The detailed manufacturing documents can be checked under the Gerber files and drill files on the output document folder.

For using the assembly function, we also generated the BOM, Pick and Place file, and assembly file through Altium and send them to the manufactory.

Conclusion

This PCB version design is been done on the design level, and currently is in the manufacturing process. In the current phase, all the selected components are following the requirement of the purpose of this project.

The next step for this project is to test the actual function operation for this PCB. To achieve that target will need to start with the boot loading process as well as the fuse bit uploading, then finally start the system testing in the lab.