Politecnico di Torino
Department of Electronics and Telecommunications

Electronic Systems Engineering

Master's degree in Electronics Engineering

Safe Light

Project Report

Professor:

Pasero Eros

Designers:

Student 1: Pouyan Asgari s289607@studenti.polito.it
Student 2: Reza Khoshzaban s288966@studenti.polito.it

Release 1.0

April 29, 2023

Contents

1 Project Specifications

1.1 Goal of the Project

The goal of this project is to design a low-power, low-cost, portable, and rechargeable light-intensity monitoring device with a long-lasting battery specified for various uses such as monitoring purposes or security measurement. For monitoring purposes, it can be used to measure the light intensity in agricultural cases, smart gardens and greenhouses (ex. Growing mushrooms), or any other place that controls the amount of light matters. And regarding security purposes, it can be used to detect intrusion into containers or rooms. For example, when shipping an expensive cargo, a light sensor is useful to know if the shipping container has been opened. Another example is keeping safe the artworks that can be damaged due to exposure to light, like paintings, photographs and etc. The device is equipped with a Bluetooth interface to communicate quickly with a specific mobile phone application to observe the light intensity, control the intensity sensitivity, and alert the user in case of an emergency. For the latter use case, there is also a Buzzer on the device as an environmental alert.

1.2 Features and Ratings

The main features of the design are:

- Low-cost device
- Portable device
- Low-power consumption device
- Long-lasting and rechargeable battery
- Resistance-based light sensor
- Buzzer as an environmental alarm
- Bluetooth Interface controllable
- 3 controllable functions:
  - Power ON/OFF
  - Light intensity level settings
  - Deactivate the alarm in case it is activated 4

2 Design Flow

In this section, the steps performed throughout the development of this project and possible future perspectives are indicated

2.1 Completed Steps

The following are the completed tasks within the frame of the project:

- Selection of a suitable name for the project and definition of the intended scope for it.
- Definition of the specifications considered to be implemented in the project
- Research and selection of the needed components to be used for implementation of the considered specifications
- Design of the schematics of different blocks of the project circuitry divided into 3 pages in **Microcontroller**, **Power**, and **Peripherals** using OrCAD CAPTURE
- Creation of the padstacks and footprints of the used components according to their datasheets using OrCAD PCB Designer and OrCAD PADSTACK Editor
- Extraction of the PCB netlist from the schematics
- Design of the PCB using OrCAD PCB Editor
- Debug of the board and extraction of GERBER files

2.2 Next Steps

The following are the steps that could be performed for manufacturing and using the device:

- Manufacturing including the component soldering on the PCB
- Software development to be implemented on the microcontroller
- 3D design of the device package
- Testing and verification of the device performance

3 Block Diagram

3.1 Name

The name of the project **Safe Light** is driven by the fact that the initial perspective of the project was for it to be a security measurement as indicated in the project introduction. Since the project is basically a smart light sensor, the name is a mash-up of the device and the intended use of it in this project.

3.2 Block Diagram

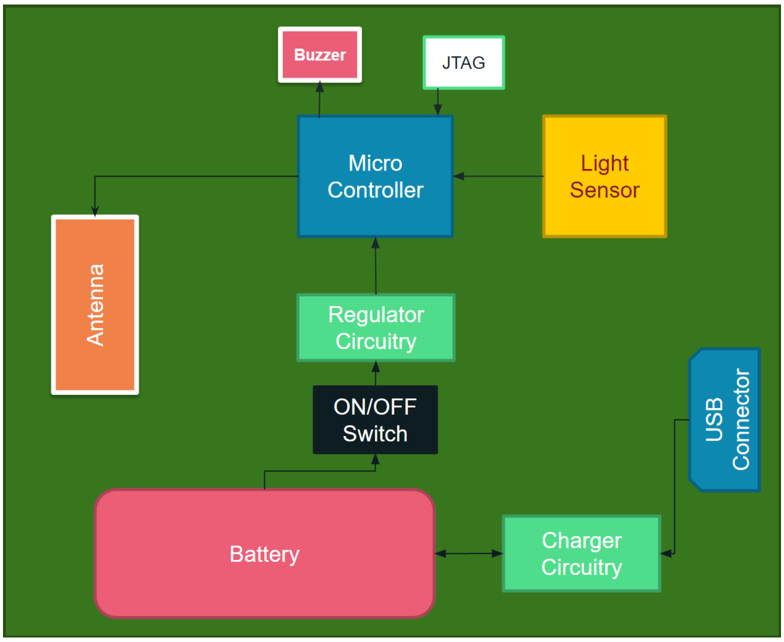


Figure 1: Safe Light Block Diagram

The block diagram in Figure 1 contains various blocks which are described below:

- Microcontroller: This is the heart of the device which contains the microcontroller component itself, the crystals needed for the clock generation, and the decoupling circuitry containing capacitors and inductors
- Antenna: This block contains the patch antenna for Bluetooth functionality and its RF front-end circuitry containing capacitors and inductors.
- Light Sensor: Contains the photodiode sensor with its respected circuitry containing a resistor
- JTAG: Contains a JTAG connector for external software access to the microcontroller alongside its circuitry containing a capacitor and a resistor
- Buzzer: A buzzer implemented for alerting purpose for security measurement intended to be activated upon light intensity fluctuation controlled by the microcontroller. This block

also contains the circuitry needed for the component including a BJT transistor and a couple of resistors

- Regulator Circuitry: Contains an external voltage controller for the microcontroller and its circuitry containing several capacitors which act as steps between the power source and the microcontroller
- ON/OFF Switch: contains a slide switch to power ON and OFF the board.
- Charger Circuitry: Includes a charging controller and its circuitry containing capacitors and a LED to indicate the charging state. Also in this block exists a Fuel Gauge which measures the fuel remaining in the battery and reports it to the microcontroller to be shown in the software application.
- USB Connector: a USB mini connector used to charge the Li-Po battery.
- Battery: a rechargeable Li-Po battery connected to the board using a battery connector.

4 Component Selection

This section provides a detailed description of all the components used in the project, including their specifications and the reasons for their selection.

With the exception of the light sensor and a few other components that will be discussed later in this chapter, SMD components have been favored over through-hole components for most of the components due to the following reason:

- SMD components are generally smaller and more compact, which can help reduce the size and weight of the PCB.
- SMD components can be mounted directly onto the surface of the PCB, which can allow for higher component density and more efficient use of space.
- SMD components can be automatically placed and soldered using pick-and-place machines, which can speed up the manufacturing process and reduce the cost of production.
- SMD components generally have better high-frequency performance and can operate at higher speeds than through-hole components, making them better suited for certain applications.
- SMD components can be more robust and resistant to mechanical stress and vibrations, which can be particularly beneficial for portable or ruggedized electronic devices.
- SMD components are widely available and compatible with a range of PCB materials.

4.1 Light Sensor

The selected sensor for sensing the light intensity is **PDV-P8001** manufactured by *Advanced Photonix* and the main features are listed below:

- Dark Resistance After 10 sec at 10 Lux is minimum 200 KΩ
- Illuminated Resistance is minimum 3 KΩ and maximum 11 KΩ
- Rise time @ 10 lux is 55ms and Fall time after 10 lux is 20 ms
- Operating Temperature: -30°C - 75°C
- Light wavelength sensitivity: 400 nm - 700 nm
- Max voltage: 150 v

The reasons for choosing this component are the following:

- Low Cost
- Good Rise/Fall time
- Convenient operating temperature and resistance sensitivity
- Illuminated Resistance in a suitable range
- The component has through-hole pins, so the sensor can be placed close to the case of the surface of the device so it can sense the light more easier

Figure 2: Light Sensor: PDV-P8001

4.2 Microcontroller

The used microcontroller for this project is **CC2640F128RSMT** manufactured by *Texas Instruments* and the main features are listed below:

- Powerful ARM Cortex M3
- Up to 48-MHz Clock speed
- 2-pin cJTAG and JTAG Debugging
- 16-bit Architecture
- Dimensions: 4-mm x 4-mm RSM VQFN32 (10 GPIOs)
- I2C protocol available
- Low Power device
- Voltage supply: 1.8 V to 3.8 V
- 4 general purpose timer modules
- 12 bit ADC
- 2.4 GHz RF transceiver compatible with Bluetooth low energy

The CC2640R2F device is a wireless microcontroller (MCU) targeting Bluetooth 4.2 and Bluetooth 5 low-energy applications. The device is a member of the SimpleLink ultra-low power CC26xx family of cost-effective, 2.4GHz RF devices. Very low active RF and MCU current and low-power mode current consumption provide excellent battery lifetime and allow for operation on small coin cell batteries and in energy-harvesting applications. The SimpleLink Bluetooth low energy CC2640R2F device contains a 32-bit ARM Cortex - M3 core that runs at 48 MHz as the main processor and a rich peripheral feature set that includes a unique ultra-low power sensor controller. This sensor controller is ideal for interfacing external sensors and for collecting analog and digital data autonomously while the rest of the system is in sleep mode. Thus, the

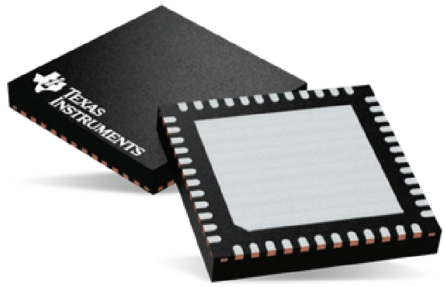


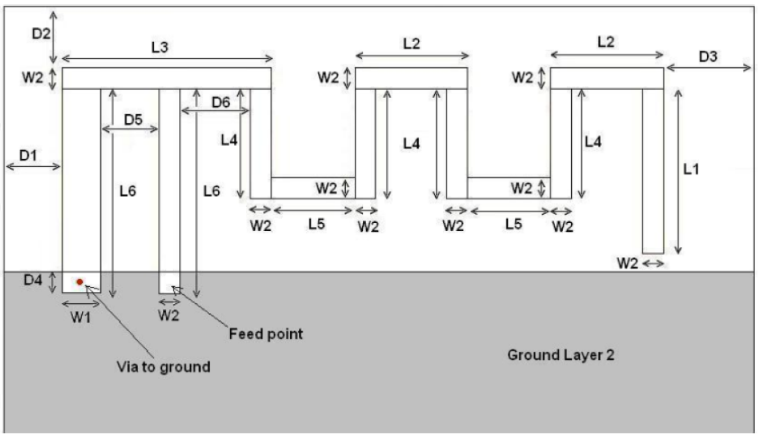
Figure 3: Microcontroller: CC2640F128RSMT

CC2640R2F device is great for a wide range of applications where a long battery lifetime, small form factor, and ease of use are important.
The reasons for choosing this component are the following:

- Various features of the microcontroller
- Bluetooth integration
- Low power consumption
- The reason for the choice of the RMS model of the microcontroller was its minimal dimension while having enough DIOs for the intended device specifications

4.3 Patch Antenna

The PCB antenna AN043 is designed to match an impedance of 50 at 2.45 GHz. Thus no additional matching components are necessary and additional costs for component purchase are avoided.



L1	3.94 mm
L2	2.70 mm
L3	5.00 mm
L4	2.64 mm
L5	2.00 mm
L6	4.90 mm
W1	0.90 mm
W2	0.50 mm
D1	0.50 mm
D2	0.30 mm
D3	0.30 mm
D4	0.50 mm
D5	1.40mm
D6	1.70 mm

Figure 4: PCB Antenna Dimentions

The AN043 is one of the recommended PCB antennas by Texas Instruments. Filling a large part of the PCB, this antenna is the best choice among the possibilities, since it is the smallest one. Moreover, being a PCB antenna, it avoids additional costs for component purchase.

4.4 Voltage Regulator

The selected external voltage regulator is **MAX1759** manufactured by *MAXIM* and the main features are listed below:

- Regulated Output Voltage (Fixed 3.3V or Adjustable 2.5V to 5.5V)
- 100mA Guaranteed Output Current
- +1.6V to +5.5V Input Voltage Range
- Low 50 uA Quiescent Supply Current
- 1uA Shutdown Mode
- Load Disconnected from Input in Shutdown
- High 1.5MHz Operating Frequency
- Uses Small Ceramic Capacitors
- Short-Circuit Protection and Thermal Shutdown
- Small 10-Pin uMAX Package

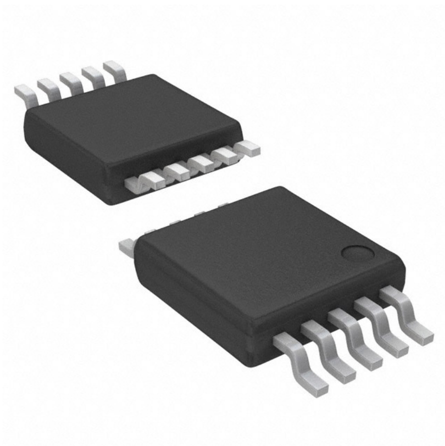


Figure 5: Voltage regulator: MAX1759

The reasons for choosing this component are the following:

- Device operation range (+1.6V to 5.5V) which suits the selected battery for the device
- Generates a fixed 3.3V voltage in its output which is in the voltage supply range of the used microcontroller
- Low-cost component among other possible selections

4.5 Battery Charger

The latest selected external voltage regulator is **MAX1555** manufactured by *MAXIM* and the main features are listed below:

- Charge from USB or AC Adapter
- Automatic Switch over when AC Adapter is plugged IN
- On-Chip Thermal Limiting Simplifies Board Design
- Charge Status Indicator
- 5-Pin Thin SOT23 Package
- Electrical characteristics (TA = 0C to +85C)
  - DC Voltage Range: 3.7V to 7.0V
  - USB Voltage Range: 3.7V to 6.0V
  - BAT Regulation Voltage: 4.158V to 4.242V
  - DC to BAT Voltage Range: 0.1V to 6.0V
  - CHG, POK Logic-Low Output: 300mV

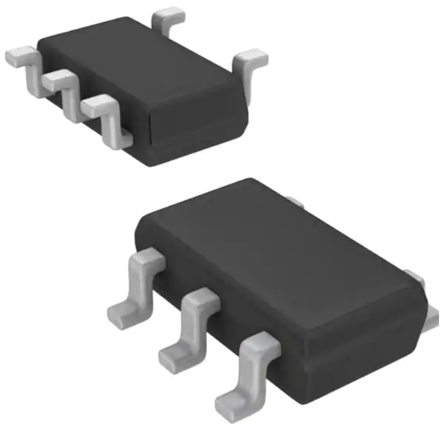


Figure 6: Charger controller: MAX1555

Pin specifications of the charger controller:

- USB: USB Port Charger Supply Input. USB draws up to 100mA to charge the battery. Decouple USB with a 1uF ceramic capacitor to GND
- GND: Ground
- POK: Power-OK Active-Low Open-Drain Charger Status Indicator
- CHG: Active-Low Open-Drain Charge Status Indicator. CHG pulls low when the battery is charging. CHG goes to a high-impedance state, indicating the battery is fully charged when the charger is in voltage mode and the charge current falls below 50mA

- DC: DC Charger Supply Input for an AC Adapter. DC draws 280mA to charge the battery. Decouple DC with a 1uF ceramic capacitor to GND
- BAT: Battery Connection. Decouple BAT with a 1uF ceramic capacitor to GND.

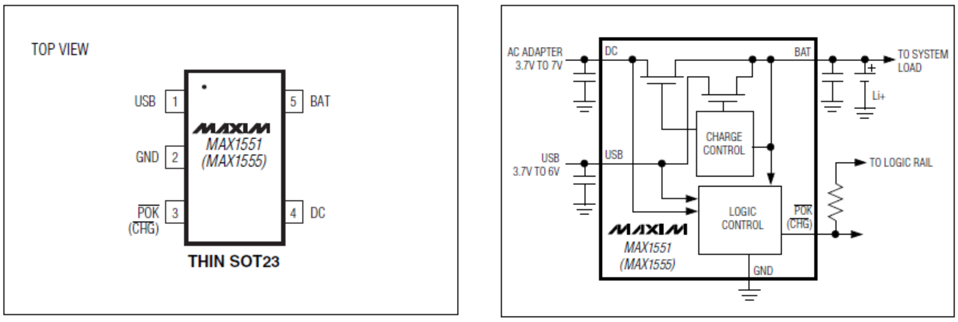


Figure 7: MAX1555 pins layout

The reasons for choosing this component are the following:

- The reason for exchange of **MCP73832T-2ACI/OT** by *Microchip* which was initially selected in the project's preliminary presentation with this component is that after further analysis, it was understood that since both components are compatible with the implemented device and they both cost the same price, it would be better if all the power components were manufactured by the same manufacturer for compatibility reasons and avoiding possible conflicts in the design.
- High Efficiency: MAX1555 has a high charging efficiency which reduces power loss during the charging process, resulting in longer battery life.
- Small Form Factor: The component comes in a small package, making it ideal for space-constrained designs like this project.
- Safety Features: This device includes several safety features, such as thermal shutdown and overvoltage protection, to help prevent damage to the battery or the device.

4.6 Fuel Gauge

The latest selected external voltage regulator is **MAX17048** manufactured by *MAXIM* and the main features are listed below:

- 1 Cell
- Precision ±7.5mV/Cell Voltage Measurement
- ModelGauge Algorithm
  - Provides Accurate State-of-Charge
  - Compensates for Temperature/Load Variation

  - Does Not Accumulate Errors, Unlike Coulomb Counters
  - Eliminates Learning
  - Eliminates Current-Sense Resistor

- Ultra-Low Quiescent Current
  - 3uA Hibernate, 23uA Active
  - Fuel Gauges in Hibernate Mode
  - Automatically Enters and Exits Hibernate Mode

- Reports Charge and Discharge Rate

- Battery-Insertion Debounce
  - Best of 16 Samples to Estimate Initial SOC

- Programmable Reset for Battery Swap
  - 2.28V to 3.48V Range

- Configurable Alert Indicator
  - Low SOC
  - 1 percent Change in SOC
  - Battery Undervoltage/Overvoltage
  - VRESET Alert

- I2C Interface

- 8-Bit OTP ID Register (Contact Factory)

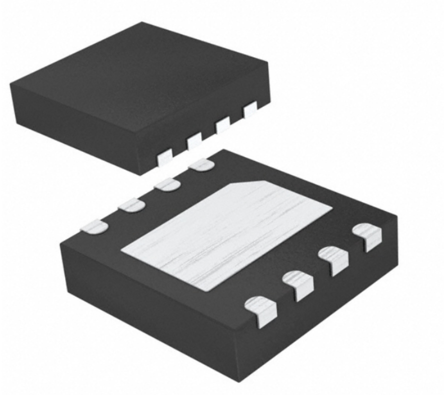


Figure 8: Fuel Gauge : MAX17048

The reasons for choosing this component are the following:

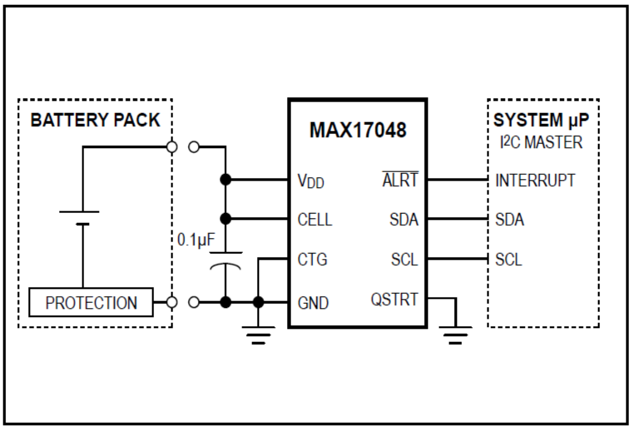


Figure 9: MAX17048 layout

- The **BQ27441DRZR-G1A** by *Texas Instruments* component initially selected for the project was replaced with this component. This decision was made after a thorough analysis which revealed that both components were compatible with the device being used and had a negligible cost difference. It was decided that using power components from the same manufacturer would enhance compatibility and minimize potential design conflicts.
- Accurate state-of-charge measurement: The MAX17048 uses the ModelGauge algorithm to provide accurate state-of-charge measurement for your battery, compensating for temperature and load variation.
- Configurable alert indicator: The MAX17048 includes a configurable alert indicator that can be set to notify the user of the low state of charge, changes in the state of charge, battery undervoltage/overvoltage, and more.

4.7 Other Components

4.7.1 Buzzer

The selected Buzzer for alarming is **AT-1127-ST-2-R** manufactured by *PUI Audio, Inc.* and the main features are listed below:

- Current – Supply: 70mA
- Voltage Range: 2 - 4 V (Rated voltage : 3 V)
- Size: 9.00 mm
- Height (Max): 4.5 mm
- Operating temperature: -20°C - 70°C
- Minimum Sound Pressure Level (SPL): 90 dB

The reasons for choosing this component are the following:

Figure 10: Buzzer: AT-1127-ST-2-R

- Low Cost
- Convenient operating temperature and Sound Pressure Level
- Operating voltage and current are in a suitable range and the powering system can handle it (thanks to the transistor)
- The component has through-hole pins, so the sensor can be placed close to the case of the surface of the device so the produced sound can be exposed conveniently to the environment without any fluctuation

4.7.2 Transistor

A transistor is used to control the buzzer in the circuit. The microcontroller can only supply a limited amount of current, which is not sufficient to drive the buzzer directly. Therefore, a transistor is used as a switch to amplify the current supplied by the microcontroller and drive the buzzer.

The selected Transistor is **MMBT3904** manufactured by *onsemi* and the main features are listed below:

- It is a common NPN transistor and it comes in a surface-mount package, which offers several benefits over through-hole mounting
- It has a low voltage drop and can handle a current of up to 200 mA (This means it can handle the current required by the buzzer and is compatible with the voltage levels used in the circuit.)
- It is a widely available and inexpensive transistor

The reasons for choosing this component are the following:

- Low Cost

The figure 12 circuit shows how the Transistor and Buzzer are connected.

The transistor is used as a switch to turn the buzzer on and off. The base of the transistor is connected to the microcontroller through the resistor R2, which limits the current flowing

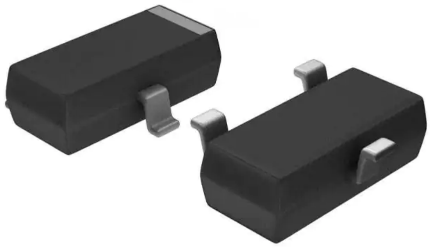


Figure 11: Transistor: MMBT3904

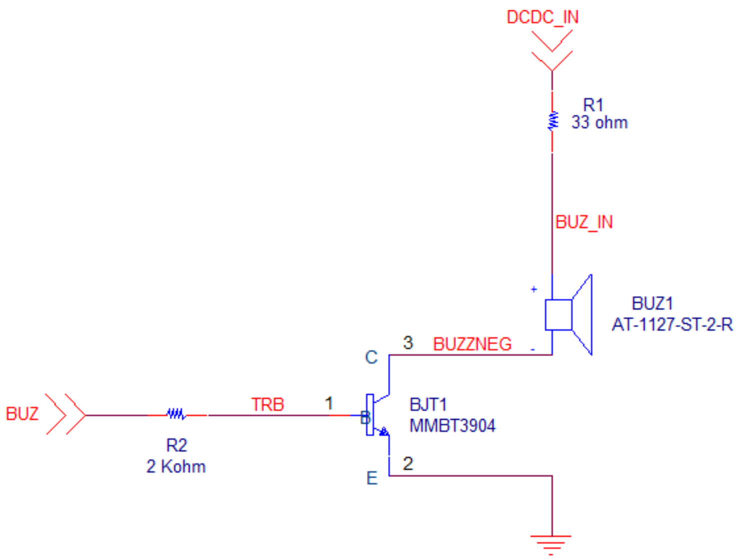


Figure 12: Buzzer connection circuit

into the base and protects the microcontroller from potential damage. When the microcontroller output pin connected to the base of the transistor goes high, current flows into the base and turns the transistor on, allowing current to flow from the collector to the emitter.

The emitter of the transistor is connected directly to the ground, which provides a path for current to flow out of the transistor when it is turned on. The collector of the transistor is connected to the negative pin of the buzzer, which is the pin that is connected to the internal electromechanical transducer that produces the sound. The positive pin of the buzzer is connected to DCDC_IN (connected to the BAT+ when the Switch is on) through the resistor R1, which limits the current flowing into the buzzer and protects it from potential damage.

When the transistor is turned on, current flows from VCC through the buzzer and the collector-emitter path of the transistor to the ground, producing a sound. When the transistor is turned off, no current flows through the buzzer, and it remains silent. Therefore, by controlling the output pin connected to the base of the transistor, the microcontroller can turn the buzzer on and off as needed, producing an alarm sound when the light intensity is too low or too high.

In the following, the approach for calculating the resistor's values is described. To begin with, resistor R2 is selected with the objective of limiting the base current to a safe and low value of 0.7 mA for the microcontroller. The transistor utilized in this project is characterized by a Current Gain of 100. Consequently, the current flowing through the collector to the emitter, which powers the buzzer, will be approximately 70 mA, equivalent to the rated current for the buzzer. Moreover, it is assumed that the MCU pin provides 2 V to activate the buzzer. Finally, based on Ohm's law, the optimal value of R2 can be calculated as follows:

$$V_{MCU} - (I_b * R2) - V_{be} = 0$$
$$2 - 0.7 = 0.7mA * R2$$
$$R2 = 1857\Omega$$
$$R2 \approx 2k\Omega$$

The value of resistor R1 is selected to restrict the current flowing through the buzzer and protect it from potential damage. It is important to limit the current that passes through the buzzer (from the collector to the emitter of the transistor) to 70 mA when it is in the "on" state, to prevent any harm to the buzzer and manage the power consumption. By considering the V_CE of the transistor in the saturation mode to be 0.3 V and the coil resistance of the buzzer to be 15 ± 3 Ω (based on their datasheets), the value for R1 can be calculated using the following equation:

$$V_{DCDC\_IN} - V_{CEsat} = 70mA * (15 + R1)$$
$$3.7 - 0.3 = 70mA * (15 + R1)$$
$$R2 = (3.4/0.07) - 15 = 33\Omega$$

4.7.3 LED

The previous LED that was used in the presentation step was found to be significantly larger in size compared to the other electronic components on the board. So it has been replaced by another LED which is more convenient. The **APTD1608LCGCK** manufactured by *Kingbright* has been selected to indicate the charging status, based on its main features and benefits listed below:

- Low power consumption: It has a low forward voltage of 2.2V and a low forward current of 2 mA, meaning it consumes less power than other LEDs.
- Good brightness: This LED has a luminous intensity of 6 to 8 mcd, which provides adequate visibility while also ensuring that it does not interfere with the light measurement objective of the project.
- Small size: It has a compact LED with dimensions of 1.6mm x 0.8mm x 0.95mm. Its small size makes it easy to integrate into compact circuit designs.
- Easy to mount: It is a standard, surface mount
- device LED that is commonly used. Cost-effective: The APTD1608LCGCK is a cost-effective LED that provides a good balance of performance and price

Figure 13: Led: APTD1608LCGCK

4.7.4 Passive Components

- Capacitors : The following capacitors manufactured by *Murata Electronics* are selected with values and package according to the need of the device circuitry
  - GCM188R71C334KA37D for 330nF
  - GRM1555C1H1R2CA01D for 1.2pF
  - GCM1555C1H120GA16D for 12pF
  - GCM21BR71C105KA58K for 1uF
  - GCM188R91E104KA37J for 100nF

  The capacitors included above are used for different purposes in the circuit which are as follows:
  - Bypass capacitors : typically small capacitors (in the range of pico- to microfarads) that are placed close to the power supply pins of an IC. They are designed to filter out high-frequency noise and voltage fluctuations that can affect the performance of the IC.
  - Bulk capacitors: larger capacitors (in the range of micro- to millifarads) that are used to smooth out the DC voltage on the power supply rail. They are placed further away from the IC and can be used to store charge to ensure a stable and consistent voltage supply.

  Different package sizes are selected for optimization reason since as an example, a 12 pF capactor which is a bypass capacitor, does not need to be a 0603 package, so 0402 package is used for this case. On the other hand a bulk capacitor like the 100nF are considered in 0603 package to accomodate the needed capacitance securely.

- Resistors : The following resistors manufactured by *YAGEO* are selected as **0603** package with values according to the need of the device cicuitry
  - RC0603FR-0733RL for 33Ohm
  - RC0603FR-13200R for 200Ohm
  - RC0603FR-13220RL for 220Ohm
  - RC0603FR-132KL for 2KOhm

    - RC0603JR-133K3L for 3.3KOhm
    - RC0603JR-0733KL for 33KOhm
    - RC0603FR-07100KL for 100KOhm

- Inductors : The following inductors manufactured by *Murata Electronics* are selected as **0603** package with values according to the need of the device circuitry
    - LQM18DN100M70L for 10uH
    - LQP03TN15NJ02D for 15nH
    - LQP03TN2N0B02D for 2nH
    - BLM15AG121SN1D for Ferrite beed

- Crystals :
    - TSX-3225 by *EPSON* for 20MHz
    - SC-20S by *Seiko Instruments* for 32.768KHz

4.7.5 Switch

The selected switch for turning the device On/Off is **EG1218** manufactured by *E-Switch* and the main features and reasons for selecting this component are listed below:

- Compact design: The EG1218 switch is compact in size, which is advantageous in a project that is designed to be portable and space-efficient.
- Ease of use: The switch is designed for ease of use, with a straightforward ON/OFF functionality that makes it simple for users to control the device. Additionally, the switch features through-hole pins, which allows for easy placement on the surface of the package, further enhancing the user's ability to operate the device with ease.
- Durability: The EG1218 switch is designed to withstand repeated usage, which is important in a project that may require frequent turning on and off of the circuit.
- Compatibility: the voltage rating (30 V) and current rating (200 mA) of this switch is suitable for the project, ensuring that it operates safely and effectively within the parameters of the project's requirements.

4.7.6 JTAG

The **FTSH-105-01-L-D-K** JTAG connector manufactured by *Samtec Inc.* was selected for this project because of its compact and low-profile design, while its high-quality connection ensures reliable programming and debugging of the microcontroller. Additionally, its durable construction provides long-lasting performance, and its versatility offers compatibility with a wide range of programming and debugging tools.

4.7.7 Battery Connector

The battery connector used for the device is **M50-3530242** by *HARWIN* which is a through hole component used in order to connect the external battery to the board. The component is cost-effective and is compatible with the intended battery. M50 connectors are spaced 1.27mm between rows and based on 0.40mm square/round pins.

Figure 14: Switch: EG1218

Figure 15: JTAG: FTSH-105-01-L-D-K

Figure 16: Battery Connector: M50-3530242

4.7.8 USB Charger Connector

he USB charger connector used for the device is **1051640001** by *MOLEX* which is a USB Type A receptacle connector that is designed for use with USB 2.0 applications. It has a 4-pin contact configuration, with a rated current of 1.5A and a rated voltage of 30V. The connector is RoHS compliant and has a flammability rating of UL 94V-0. It is designed to be mounted onto a printed circuit board (PCB) using through-hole soldering. The connector has a durable construction, with a gold-plated brass contact material and a black high-temperature thermoplastic housing material.

The reason for the selection of the component is that its specification includes a high durability and contact integrity, which ensures a long life cycle of the connector and also it is a low-cost component comparing to its competitors.

Figure 17: USB Connector: MOLEX 1051640001

4.7.9 Battery

While the device offers the flexibility to connect with any external battery satisfying the voltage and discharge current requirements, a specific battery model is recommended for optimal performance, which is the following. The **365** rechargeable Lithium-Ion battery manufactured by *Adafruit Industries* is selected as the suggested battery. The main features and reasons for this choice are listed below:

- It's rechargeable and compatible with the charging circuitry of the device.
- The discharge rate is 2.2 A, which means it can easily supply a high current to power the buzzer when the alarm is activated.
- It has a standard charge current of 1A, which means it can be recharged relatively quickly and efficiently.
- This battery has a large capacity of 4.4Ah, which means it can power the device for an extended period without needing to be recharged (calculations are reported in the power consumption section).
- It is relatively low cost compared to other batteries, considering its high capacity.

Figure 18: Battery: 354 by Adafruit Industries

5 Complete Schematics and bill of materials

This section includes the complete final schematics of the project in full detail and the bill of materials (BOM) for the design and a cost estimation taking into account all the component costs and other possible costs.

5.1 Complete schematics

The schematics created and extracted by OrCAD CAPTURE are divided into three pages as follows:

- Microcontroller:
  - TI CC2640R2F RSM: includes the schematic of the microcontroller
  - Crystals: includes the schematic of the two crystals used for the microcontroller
  - Decoupling Circuitry: includes the circuitry for decoupling
  - Bluetooth Antenna: includes the circuitry for the Bluetooth antenna alongside the antenna itself
- Peripherals and Sensor
  - Photodiode: includes the light sensor and its circuitry
  - Buzzer: includes the buzzer and its circuitry
  - Jtag Debugger: includes the JTAG component and needed circuitry for it
- Power Management
  - Battery connector and switch: includes the switch component and the battery connector connections
  - Voltage regulator: includes the voltage regulator and its needed circuitry
  - Battery charger: includes the charger controller, the USB port, and the LED and their connections and circuitry

– Fuel gauge: includes the fuel gauge for the battery and its circuitry

Among the different pages and the blocks inside each page, OFF-PAGE connectors are seen which are used to connect components and circuits from one page to the other or in the case of blocks on the same page for better readability

There is also a comments section on each page which includes useful comments for a better understanding of the schematics and tips for the placement of the components and some useful specifications.

5.1.1 Microcontroller

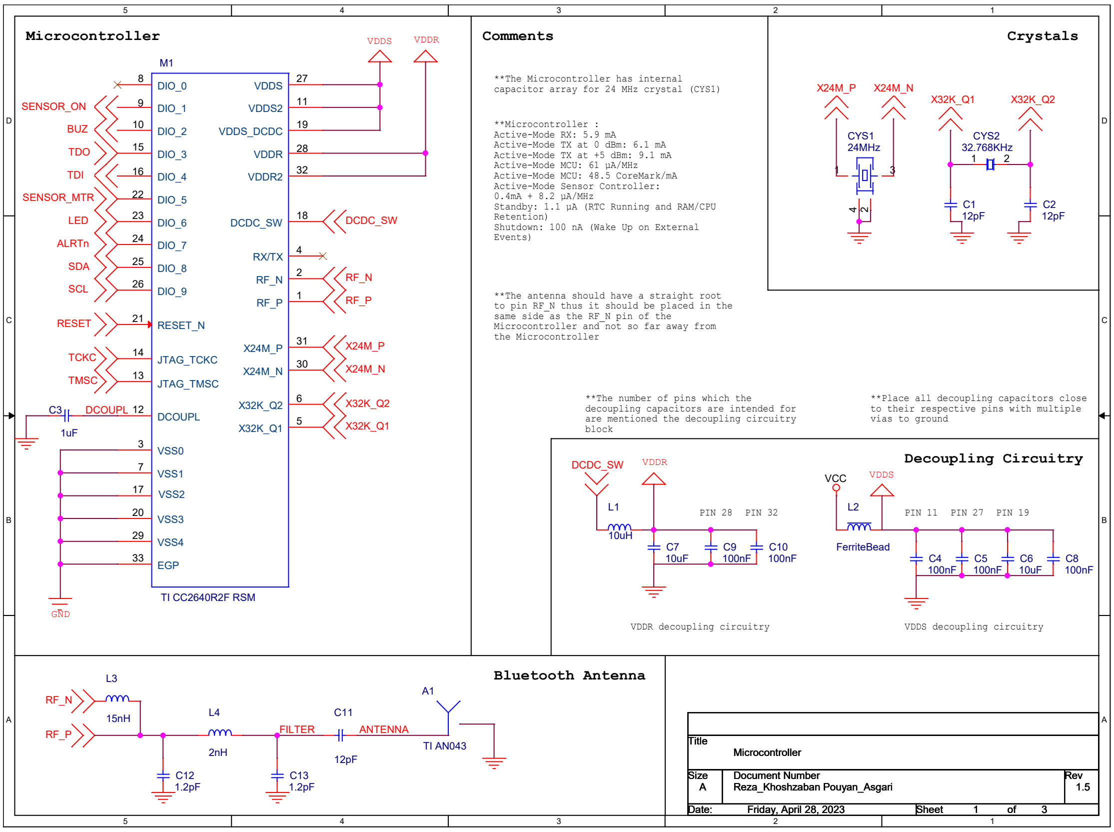


Figure 19: Microcontroller Schematics page

To transmit data from the Bluetooth module to the antenna, the RF N and RF P pins are connected to the RF front-end circuit. The light sensor is connected to the microcontroller on 2 DIO pins: DIO_5 and DIO_1, where the latter is used for powering the sensor through the "SENSOR_ON" connection and that is why it is connected to this pin since it has high drive capability pin according to the microcontroller's datasheet. DIO_5 has analog capabilities and that is the reason the "SENSOR_MTR" connection is connected to it since it is the value received from the light sensor to calculate the light intensity. The Buzzer is connected to DIO_2 because that pin has high drive capabilities and the JTAG connector connects the microcontroller to the

TCK, TDO, TMS, and TDI pins. Capacitors are used to decouple the microcontroller's pins and various capacitors are used for this purpose. The 24MHz crystal is used as the frequency reference for the radio and does not require decoupling capacitors since the microcontroller has an internal capacitor array for 24 MHz crystal. The decoupling capacitors ought to be placed close to the pins indicated in the schematics since they are specified for those pins.

5.1.2 Peripherals and Sensor

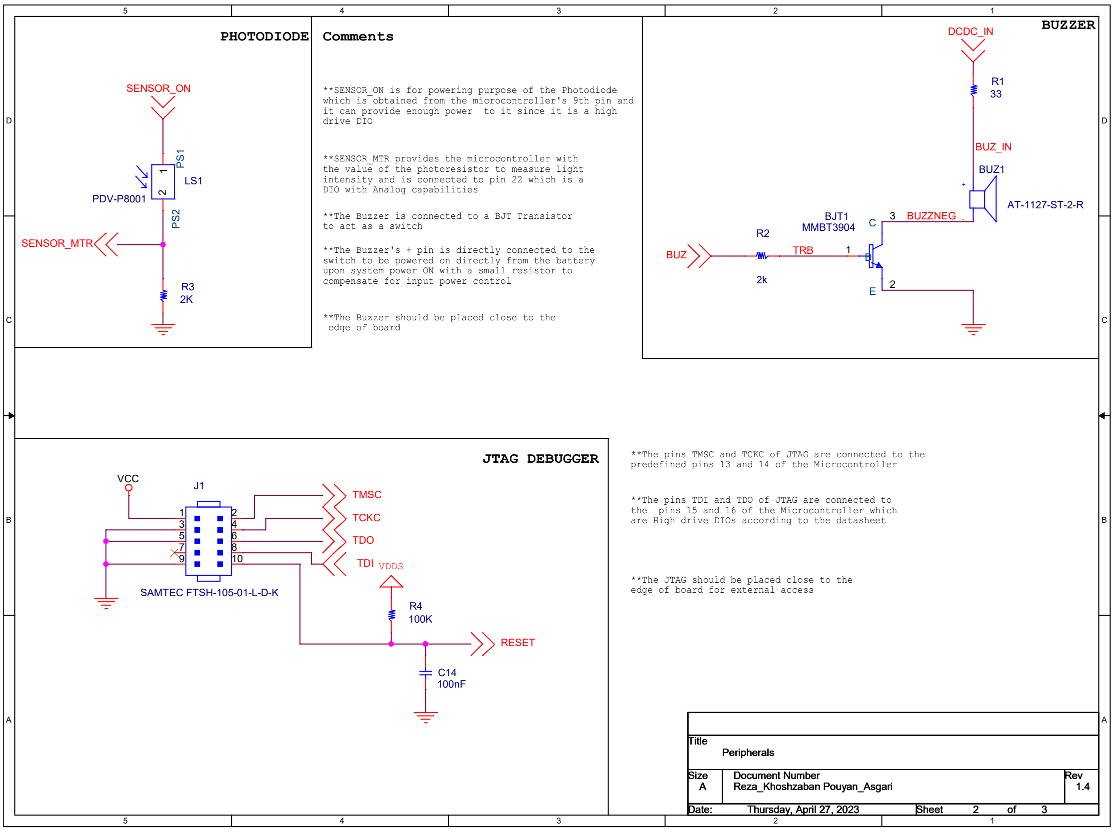


Figure 20: Peripherals and Sensor Schematics page

In this schematics, the light sensor and the two peripherals including the JTAG debugger and the Buzzer are exhibited:

- PHOTODIODE :
  - The microcontroller's 9th pin, which is a high drive DIO, is used to power the Photodiode using SENSOR_ON. This pin is capable of providing enough power to the Photodiode.
  - Pin 22 of the microcontroller is utilized for connecting the photoresistor to measure the intensity of light. This pin, which is a DIO with analog capabilities, receives information from the SENSOR_MTR and provides it to the microcontroller for further processing.

- BUZZER :
  - The buzzer is wired to a BJT transistor in order to function as a switch.
  - The positive pin of the buzzer is linked straight to the switch for direct power supply from the battery when the system is turned on, and a slight resistor is added to manage the input power.
  - The buzzer ought to be placed close to the edge of the board because of its big size compared to the board size
- JTAG :
  - The TMSC and TCKC are connected to the pre-defined pins 13 and 14 of the microcontroller
  - TDO is the output signal from the JTAG slave, which contains the data that the JTAG master reads during the communication. TDI, on the other hand, is the input signal to the JTAG slave, which carries the data that the JTAG master writes during the communication.
  - This reset circuit and the used capacitor and resistor are used to ensure that the microcontroller is properly initialized and starts executing code from a known state, which is essential for the reliable operation of the system.

5.1.3 Power Management

- This section contains various components:
  - MAX1555 battery charger, which serves as an interface with the USB connector
  - MAX1759 voltage regulator, which provides stable voltage to the circuit
  - MAX17048 battery level meter connected to the DIO pins of the microcontroller to report the amount of fuel left in the battery to the microcontroller to be monitored in the software application.
  - Charging indicator LED.
- To indicate to the user that the battery is charging, the LED is connected between the external voltage from the USB and the CHG pin of MAX1555.
- The pull-up resistor values were determined using the worst-case condition based on the relation: R = (Vmax-Vled)/Iled
- A switch has been added to disconnect the system from the battery to preserve the circuit.

5.2 Bill Of Materials and Cost Estimation

5.2.1 Bill Of Material (BOM)

The bill of materials (BOM) below was initially generated by OrCAD Capture and was modified for extra clarity. It is a comprehensive list of all the components needed to assemble a Printed Circuit Board (PCB). The BOM provided here lists all the necessary parts required for the assembly of the PCB. The BOM includes the Quantity required, the Reference Designator which is used to identify and place each component on the PCB, and the specific Part name and number for each component.

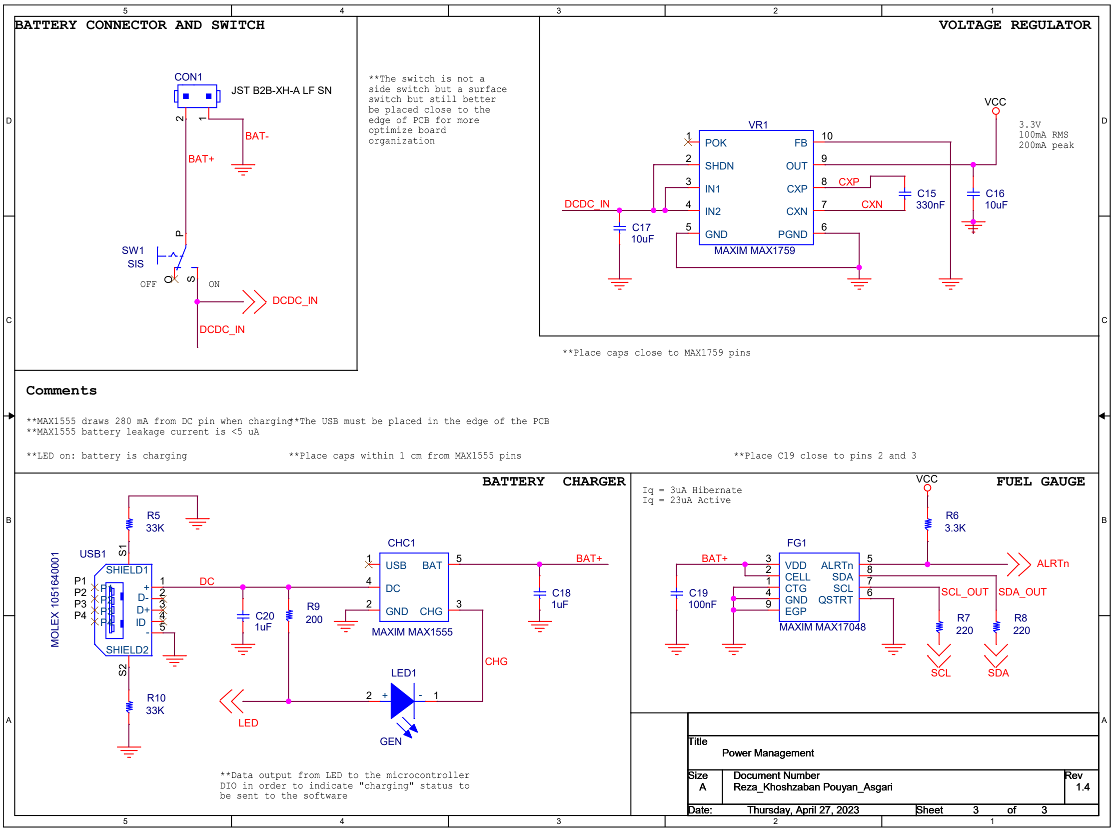


Figure 21: Power management Schematics page

The Bill of Materials (BOM) is a comprehensive list of all the components needed to assemble a Printed Circuit Board (PCB). The BOM provided here lists all the necessary parts required for the assembly of the PCB. The BOM includes a unique Item number for each component, the Quantity required, the Reference Designator which is used to identify and place each component on the PCB, and the specific Part name and number for each component and their package. In case the name of the package is the same as its part, /*/ is used. Due to limitations of representation of full BOM in this report, The full BOM is presented as a separate document as BOM.pdf

Item	Quantity	Reference	Part	Package
1	1	A1	TI AN043	*
2	1	BJT1	MMBT3904	*
3	1	BUZ1	AT-1127-ST-2-R	*
4	1	CHC1	MAXIM MAX1555	*
5	1	CON1	JST B2B-XH-A LF SN	*
6	1	CYS1	24MHz	TSX-3225_xtal
7	1	CYS2	32.768KHz	SC-20S_xtal
8	3	C1,C2,C11	12pF	0402_C
9	3	C3,C18,C20	1uF	0805_C
10	7	C4,C5,C8,C9, C10,C14,C19	100nF	0603_C
11	4	C6,C7,C16,C17	10uF	0805_C
12	2	C12,C13	1.2pF	0402_C
13	1	C15	330nF	0603_C
14	1	FG1	MAXIM MAX17048	*
15	1	J1	SAMTEC FTSH-105-01-L-D-K	*
16	1	LED1	APTD1608LCGCK	*
17	1	LS1	PDV-P8001	*
18	1	L1	10uH	0603_L
19	1	L2	FerriteBead	0402_L
20	1	L3	15nH	0603_L
21	1	L4	2nH	0603_L
22	1	M1	TI CC2640R2F RSM	*
23	1	R1	33	0603_R
24	2	R2,R3	2K	0603_R
25	1	R4	100K	0603_R
26	2	R5,R10	33K	0603_R
27	1	R6	3.3K	0603_R
28	2	R7,R8	220	0603_R
29	1	R9	200	0603_R
30	1	SW1	SIS EG1218	*
31	1	USB1	MOLEX 1051640001	*
32	1	VR1	MAXIM MAX1759	*

5.2.2 Cost Estimation

The cost estimation takes into account various aspects of the development of this device including the price of each component used, the PCB manufacturing price, and a wild guess at the cost for board assembly and soldering as well as delivery. All the prices are in EURO currency and the prices are according to the DIGIKEY official website.

In the following table, the price of each component used is documented :

Item	Quantity	Price	Total price
MMBT3904	1	0.09 €	0.09 €
AT-1127-ST-2-R	1	0.79 €	0.79 €
MAXIM MAX1555	1	2.45 €	2.45 €
JST B2B-XH-A LF SN	1	0.14 €	0.14 €
TSX-3225	1	0.39 €	0.39 €
SC-20S	1	0.63 €	0.63 €
GCM1555C1H120GA16D	3	0.09 €	0.27 €
GCM21BR71C105KA58K	3	0.16 €	0.48 €
GCM188R91E104KA37J	7	0.16 €	1.12 €
GRM188R61E106KA73J	4	0.23 €	0.92 €
GRM1555C1H1R2CA01D	2	0.09 €	0.18 €
GCM188R71C334KA37D	1	0.16 €	0.16 €
MAXIM MAX17048	1	3 €	3 €
SAMTEC FTSH-105-01-L-D-K	1	3.85 €	3.85 €
APTD1608LCGCK	1	0.43 €	0.43 €
PDV-P8001	1	2.06 €	2.06 €
LQM18DN100M70L	1	0.16 €	0.16 €
BLM15AG121SN1D	1	0.09 €	0.09 €
LQP03TN15NJ02D	1	0.09 €	0.09 €
LQP03TN2N0B02D	1	0.09 €	0.09 €
TI CC2640R2F RSM	1	5.25 €	5.25 €
RC0603FR-0733RL	1	0.09 €	0.09 €
RC0603FR-132KL	2	0.09 €	0.18 €
RC0603FR-07100KL	1	0.09 €	0.09 €
RC0603JR-0733KL	2	0.09 €	0.18 €
RC0603JR-133K3L	1	0.09 €	0.09 €
RC0603FR-13220RL	2	0.09 €	0.18 €
RC0603FR-13200RL	1	0.09 €	0.09 €
SIS EG1218	1	0.97 €	0.97 €
MOLEX 1051640001	1	0.95 €	0.95 €
MAXIM MAX1759	1	6.68 €	6.68 €

- The total cost of the components is **32.14 €**
- As reported by Eurocircuits, the cost of the manufacturing of the PCB is estimated to be **23.68 €**
- Approximate price for package manufacturing is around **5 €**
- The approximate total cost of manufacturing this project is around **61 €**

6 Power Consumption

To calculate the power consumption of this device, four different modes are defined as follows:

- **Sleep Mode**: In this mode, the microcontroller turns off most of its internal components and peripherals, including the processor. This results in the lowest power consumption of the three modes. The light sensor and buzzer are also powered off in this mode.
- **IDLE Mode**: This particular mode allows the microcontroller to enter IDLE mode while the light sensor and buzzer remain inactive. Its purpose is to enable users to conserve battery life by putting the device to IDLE when not in use, without having to power it off.
- **Sensing Mode**: This is the device's most frequently used mode, in which the microcontroller and light sensor remain active, but the buzzer is not activated.
- **Alarm Mode**: This mode is the most power-intensive setting for the device, as all components, including the buzzer, remain active. This mode is only intended for alarming purposes and will not be continuously activated. However, estimating the device's long-term battery life requires calculating power consumption in this mode.

The power consumption calculation for each mode is outlined in the following subsections, and finally, an average battery life estimate for the device is derived by combining the results of the three modes.

6.1 Sleep Mode

In this mode, the microcontroller is in the sleep mode and both the buzzer and light sensor are turned off. Therefore, the current consumption will only be the sleep mode current of the microcontroller, which is 1 μA. The other components (voltage regulator, fuel gauge, ...) use approximately 100 μA. Using the battery's capacity of 4.4 Ah, we can estimate the battery life as follows:

Battery life = Capacity / Current consumption
Battery life = 4.4 Ah / (1 + 100) μA = 43564 hours ≈ 1815 days = 60 months = **5 years**

6.2 IDLE Mode

In this mode, the microcontroller is in the IDLE mode and both the buzzer and light sensor are turned off as in the previous mode. So the current consumption will only be the IDLE mode current of the microcontroller, which is 650 μA. The other components (voltage regulator, fuel gauge, ...) use approximately 100 μA. Using the battery's capacity of 4.4 Ah, we can estimate the battery life as follows:

Battery life = Capacity / Current consumption
Battery life = 4.4 Ah / (650 + 100) μA = 5866.66 hours ≈ 244 days = **8 months**

6.3 Sensing Mode

During this mode, the microcontroller and light sensor operate while the buzzer remains inactive, resulting in a power consumption equal to the active mode current of the microcontroller and the supply current of the light sensor. The microcontroller necessitates a current of 1.45 mA + 31 $\mu A/MHz$ when in active mode, according to the datasheet. Therefore, with a 24 MHz crystal, the total consumed current in this mode will be approximately 2.2 mA. The light sensor is attached to a high drive GPIO pin of the microcontroller, which can supply up to 2.8 V of voltage. Taking into account the resistance change of the light sensor (ranging from 3 $k\Omega$ to 11 $k\Omega$) and the 2 $k\Omega$ resistor connected to it, the total current consumed by these components ranges from 0.215 mA to 0.560 mA (on average 0.388 mA). By utilizing the battery capacity of 4.4Ah, the battery life can be estimated as follows:

$$\text{Battery life} = \text{Capacity} \ / \ \text{Current consumption}$$
$$\text{Battery life} = 4.4 \text{ Ah} \ / \ (2.2 + 0.388 + 0.1) \text{ mA} = 1636.9 \text{ hours} \approx 68 \text{ days} = \textbf{2 months}$$

6.4 Alarm Mode

In this mode, both the buzzer and light sensor are active, in addition to the microcontroller. Therefore, the current consumption will be the active mode current of the microcontroller (2.2 mA as calculated before), the supply current of the light sensor (0.388 mA on average), and the operating current of the buzzer, which is around 70 mA. Using the battery's capacity of 4.4Ah, we can estimate the battery life as follows:

$$\text{Battery life} = \text{Capacity} \ / \ \text{Current consumption}$$
$$\text{Battery life} = 4.4 \text{ Ah} \ / \ (2.2 + 0.388 + 0.1 + 70) \text{ mA} = 60.5 \text{ hours} \approx \textbf{2.5 days}$$

6.5 Average Usage

For instance, to demonstrate typical device usage, the following scenario was taken into account, and the average duration of the battery under these circumstances was computed.

- 10 seconds per day in Alarming mode
- 18 hours per day in the Sensing mode
- 6 hours per day in the IDLE mode

To calculate the estimated battery life, the following formulas are used:

$$\text{Battery usage per day} = (\text{Current consumption} * \text{Time}) \text{ for each mode}$$
$$\text{Battery life} = \text{Battery capacity} \ / \ \text{Battery usage per day}$$

So :

$$\text{Batter usage per day} = (10 \text{ s} * 72.688 \text{ mA}) + (18 \text{ h} * 2.688 \text{ mA}) + (6 \text{ h} * 0.75 \text{ mA})$$
$$= 0.202 \text{ mAh} + 48.384 \text{ mAh} + 4.5 \text{ mAh} = 53.086 \text{ mAh}$$
$$\text{Battery life} = 4.4 \text{ A} \ / \ 53.086 \text{ mA} = 83 \text{ days} \approx \textbf{3 months}$$

7 PCB Layout

This section contains the description of the design choices of the PCB which are explained in detail in the following:

7.1 Component Padstack and Footprint Design

The initial step to be taken when it comes to PCB design after the component selection step is to design the Footprint of each component. This step was performed in two steps for each component which started with creating the Padstacks of the component using **CADENCE PADSTACK EDITOR** and according to the information provided in the datasheet of each component. A Padstack is a collection of physical and electrical attributes that define the connection between a component and a printed circuit board (PCB). It includes information about the size, shape, and placement of the pads on the PCB, as well as details about the conductive traces that connect the pads to the rest of the circuit. Padstack is an important part of the design process for PCBs, as it ensures that components are correctly mounted and connected to the correct signals on the board.

Next, the footprint of each component was created using **Allegro PCB EDITOR** according to the dimensions provided in the datasheet of each component and the previously designed Padstacks were imported and placed as pins for the footprint. The footprint included various layers including Etch, silkscreen, assembly, solder mask, and paste mask view.

7.2 Component Placement

After the design of the footprints and their assignment to the components in the schematics, the netlist of the device was exported from **OrCAD CAPTURE** to **Allegro PCB EDITOR** and after user preferences and design parameter settings, the components were placed according to previously mentioned criteria such as the straight and free path between the RF Antenna and the microcontroller and components like the Buzzer, JTAG, Switch and the LED is close to the edge of the board. The 24MHz external clock generator crystal was placed close to the microcontroller to reduce noise in the path and have a short trace to the microcontroller and the coupling capacitors for the microcontroller were also placed close to their respective pins which are specified in the schematics for similar reasons. The USB is placed in such a way that according to its datasheet, the board layout would not cover beneath the connection section of the component so upon connection to the external power source, the connection would be solid and fixed.

7.3 Board Layout and Ground Plane

After the placement of each component was considered, the board layout was created using a triangle shape in **Board Geometry** layer and the ground plane was created as a triangle shape in the **Etch bottom layer** and was net assigned to **GND**. The ground plane does not cover the patch antenna since the presence of a ground plane beneath the patch antenna can alter the effective permittivity of the substrate material, which in turn affects the resonant frequency of the patch. Additionally, the ground plane can create parasitic capacitance and inductance that can further de-tune the antenna.

7.4 Routing

The routing is the next step where routing the traces amongst the components is performed which takes into account several regulations while doing. First of all, the crucial point always to be considered is that the traces should not make sharp turns while trying to change the direction of the trace. a **45°** porition for turning is prefered and a **90°** turning must be avoided. As seen in figure: 23 all the traces have been routed in a way to avoid any sharp edges in the design. Another point while routing is upon layer change to the bottom layer, it is considered to have traces along the patch of the current flow to have the maximum performance out of the design, but there were a couple of changes in which a vertical trace was unavoidable where they were placed to be as short as possible. The change from the top etch layer to the bottom etch layer is performed using **VIAs (vertical interconnect access)**. A custom VIA was created for this design since the default VIA for Allegro is not well optimized and due to the use of a large solder mask, traces located in close proximity are left unprotected, which can result in these exposed traces breaking or causing short circuits with neighboring traces over time. The traces associated to power as well as the antenna trace are designed to have a wider width to accomodate higher current.

7.5 Silkscreen Adjustment

After the routing stage, a silkscreen adjustment was performed on the board in order to relocate some of the silkscreens of the components to a more suitable location in which they would still be readable and not overlap even after the components are mounted.

7.6 DRC

In the next step, a **DRC: Design Rule Check** was performed in order to identify if there exists any isolated shape or trace to trace or SMD violations according to the predefined design and parameter constraints, and any error was dealt with in this stage. As an example, the antenna was considered an isolated shape which was resolved after slightly extending the Antenna trace in the copper layer of the antenna from its respective pin.

7.7 Gerber Files Extraction

The final step of the PCB design was to extract the Gerber files using **Allegro PCB EDITOR** alongside drill report extraction to be sent to *EUROCIRCUITS* for PCB evaluation and board price determination. In order to obtain the needed Gerber files, first the layers from which the Gerber files were supposed to be extracted were defined which included all the views of the board from TOP to BOTTOM including all the layers such as soldermask and silkscreen. The obtained views by Eurocircuit are reported in the figures 27 and 28.

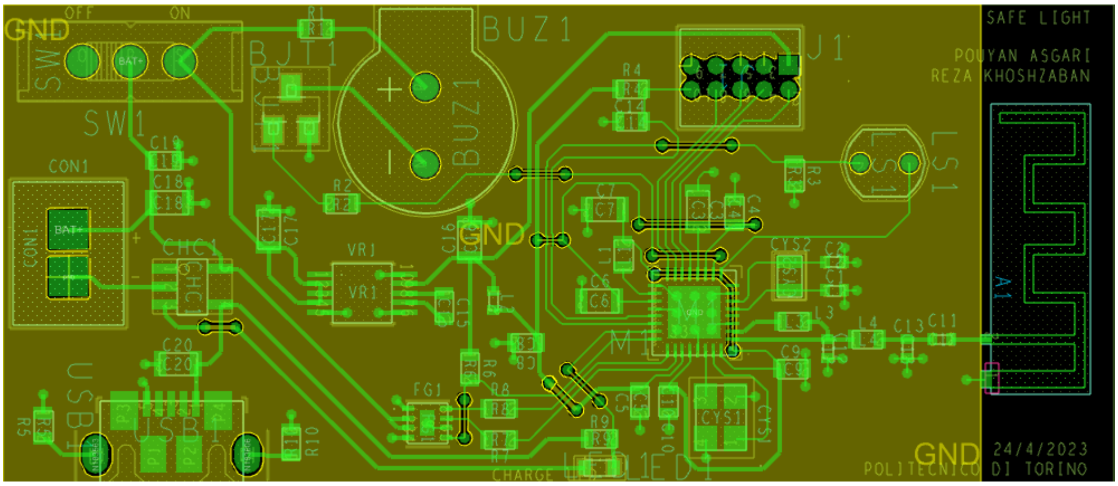


Figure 22: Safe Light PCB Full overview

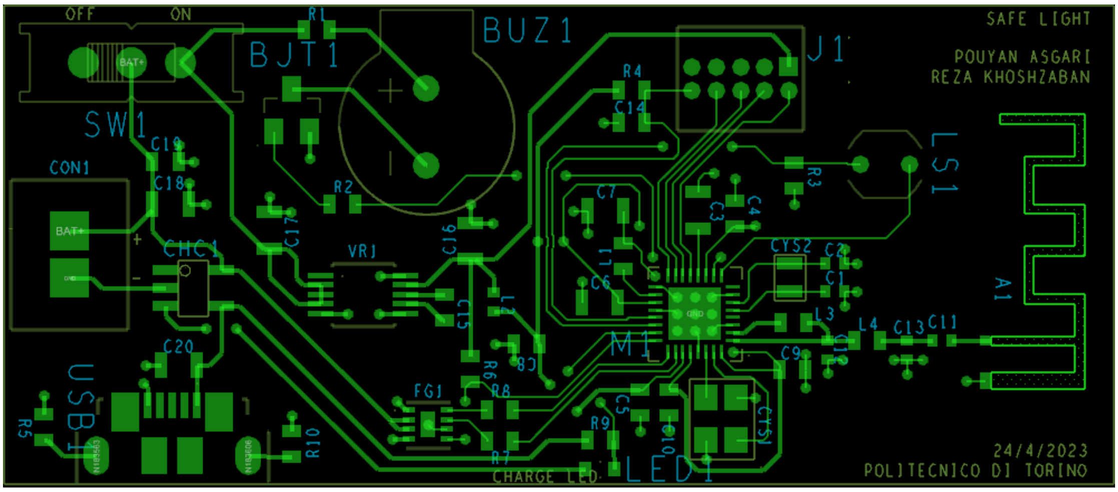


Figure 23: Safe Light PCB Silk Etch TOP view

.

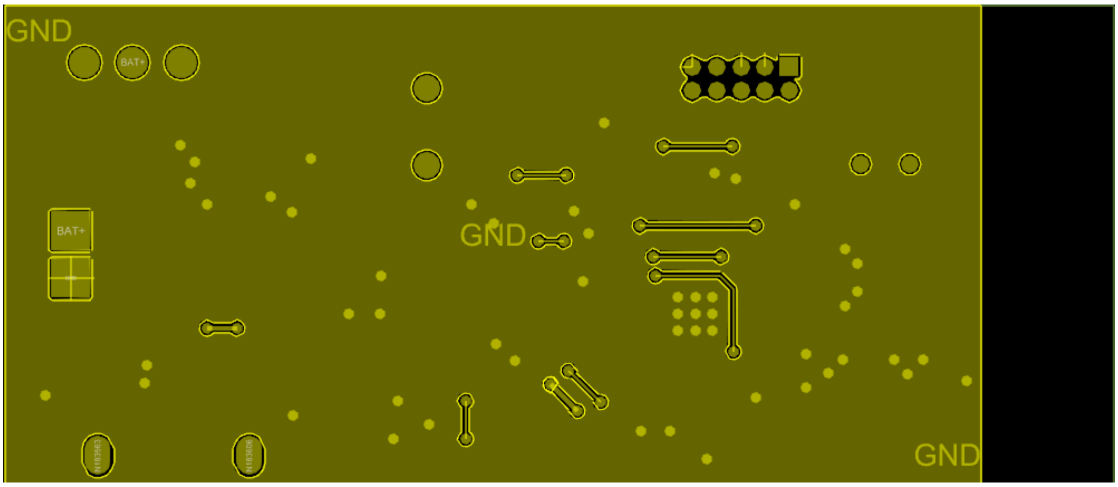


Figure 24: Safe Light PCB Silk Etch BOTTOM view

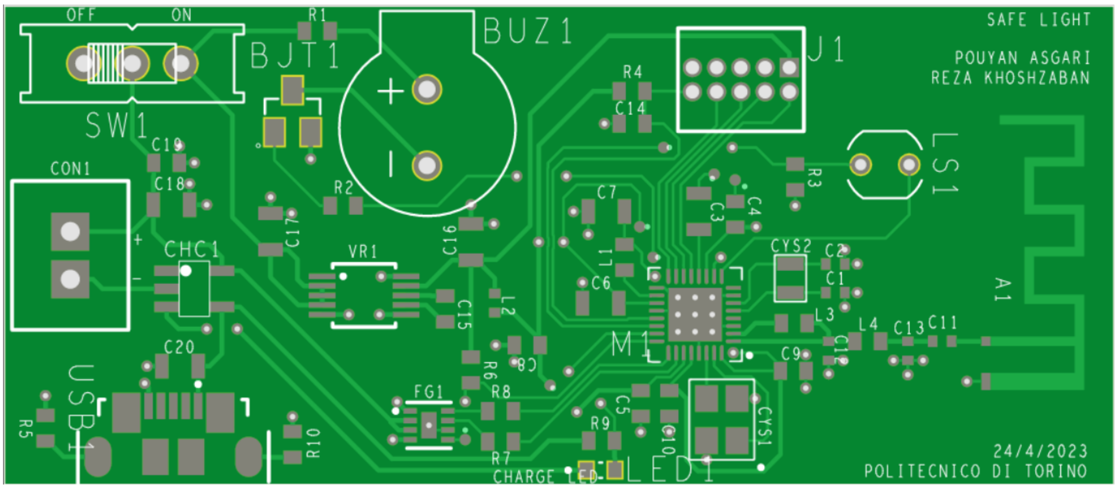


Figure 25: Safe Light PCB Eurocircuit Top view

.

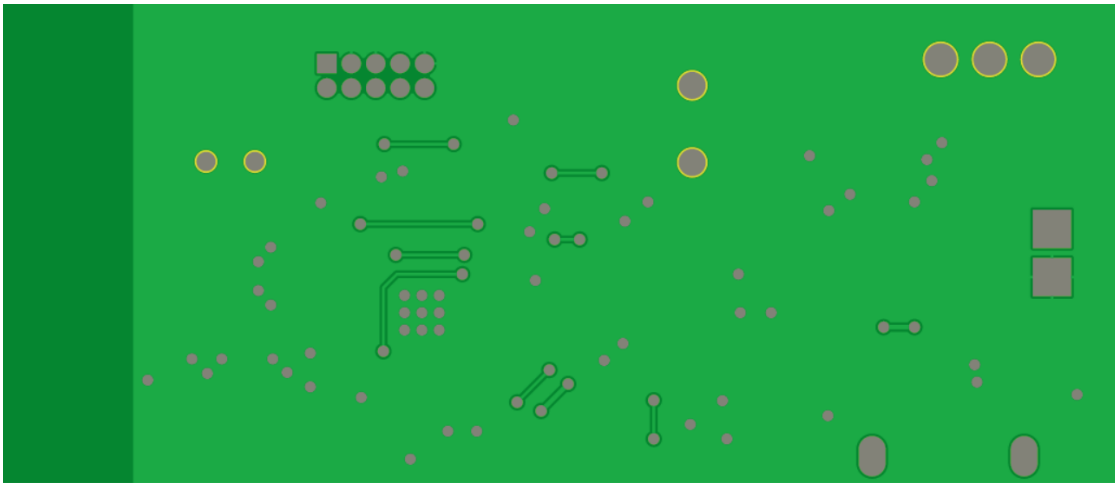


Figure 26: Safe Light PCB Eurocircuit Bottom view

7.8 Eurocircuit Report

The report given by *Eurocircuits* is reported below which contains information about the dimensions of the PCB and its manufacturing cost which is dependent on the Pattern class of the board. The pattern class of the board is 8 which is a pretty high class curtasy of the Outer layer isolation distance and the outer layer annular ring which despite several attempts to improve the class, the understanding of the cause of this issue was not understood and was considered to be in need of a more professional solution, but it was understood that the reason for this is the small size of the device PCB since the device is a small one which is suitable to fit in hard to reach environments.

Technology & options			
Board definition			
Number of layers	2	Delivery format	No
PCB width (X)	58.15 mm	PCB height (Y)	25.05 mm
eC-registration compatible	No		
Board definition			
Top soldermask	Green	Bottom soldermask	Green
Top legend	White	Bottom legend	White
Surface finish	Any lead free finish		
Bare board testing	Yes		
Board technology			
Pattern class	8	Drill class	Drill C
Outer layer trackwidth (OL-TW)	0.150 mm	Hole density	<1000/dm2
Outer layer isolation distance (OL-TT-TP-PP)	0.125 mm	Holes <= may be reduced	0.45 mm
Outer layer annular ring (OAR)	0.100 mm		

Figure 27: Eurocircuit report

Material definition

Board thickness	1.55 mm	Board buildup	Standard buildup
Base material	FR4IMP	Material Tg	145-150°C
Outer layer copper foil	18µm(End-Cu +/-35µm)	Inner layer copper foil	0
Extra PTH runs	0	Extra press cycles	0
Reversed buildup	No	Inner layer core thickness	Standard

Advanced options

Milling	No	Peelable mask	No
Edge connector gold surface mm²	0	Edge connector bevelling	No
Copper up to board edge	No	Plated holes on the board edge	No
Carbon contacts	No	Viafill	No
Top heatsink paste	No	Bottom heatsink paste	No
Specific tolerances	No	Specific marking	No
Press-fit holes	No	Depth routing	No
Round-edge plating	No	Chamfered mechanical holes	No

Pricing

Printed circuits

Basket nr.	Delivery term	Quantity	Unit price	Transport price	Transport mode	Total price	VAT	Gross
B3918177	7 Working days	1	20.00 EUR	3.68 EUR	Express	23.68 EUR	22.00 %	28.89 EUR

Figure 28: Eurocircuit report

8 Conclusion

In this section some generic design choices will be addressed alongside a conclustion of the steps and a few future prespectives for the project.

8.1 Generic design choices

- Device dimensions : The device was planned to be a small device max 10 cm * 10 cm in order to be compliant with the fact that the device muight be used in hard-to-reach areas as a light-meter for laboratory farms or as a security measurement for specific areas.
- The design includes one and only one LED which only lights up upon battery recharge. This decision was made due to the fact that the device is a light sensor and any amount of light was considered to be avoided when the device was ON specially when used as a security measurement to avoid false alarms and effect the light sensor.
- Due to the fore-mentioned fact about the avoidance of using LEDS for ON-state indication of the device, it was considered that the device's POWER-ON state would be indicated by a buzzing when powered on and the bluetooth connectivity.

8.2 Overall Conclusion

This project was initialized and developed courtesy of the Electronic Systems Engineering course at Politecnico Di Torino and throughout this course and developement of this project, the prcedure of a PCB design process was experienced such as the cruciality of component selection and many aspects to be considered during schematics implementation and PCB design and the aspects that can effect the quality and functionality of the project. The developed project is considered to be a prototype and further improvements and design choice modifications can and will be considered upon further analysis before physical manufactoration of the board.

8.3 Future prespectives

For now, the prototype design is usable for applications such as smart farm where light intensity is important for farming specific plants and aviculture where keeping the day cycle realistic is crucial and also as a security measurement in containers.

A more precise and advanced light sensor and a micro-form of the design can be used in future to extend to use of the design for academic purposes for molecular laboratories in various experiments and medical fields to keep the patients with specific diseases in a specific range of light intensity.

8.4 A word form the designers

Every design choice and decision we made throghout this project from component selection all the way to the gerber files, we tried to be as compliant as possible with the points and tips given throughout the course and the labs and for us, this being our first ever experience of a PCB design project, we beileve we tried as hard as we could to take into account everything we had the knowledge about and learned as much as we could thanks to this project.