# Problem Statement-->

# Marking Scheme:

Every step performed will earn you a single point [1 Point], Every skipped step will earn you zero point [0 point].

The steps performed mentioned in the special consideration will earn you 5 Points!

Hardware Designers who are not proficient in PCB Design can skip the PCB design stage and work with other mentioned steps but, they are directed to perform calculations and document them for all the major component network such as power, feedback, signal etc.

The components list is provided for ease and Manufacturer Part Number is provided for your reference. You are free to use online tools but on a condition that you must mention the usage and the extent of usage as well.

The steps and their clarity may earn you extra points, make sure to document them carefully.

The designer is free to use any cad tool they desire, Although KiCad is suggested due to its open-source nature.

## **# Assignment Description:**

This Assignment is intended on evaluating the steps taken to approach the problem written above first and then steps taken to make the solution. The product is of less consideration than the method incorporated to make the solution.

The Engineer must make a device around the Quectel M66 SoM, depicting the features of an Arduino board with it. Header pins must be given as interface for different components and sensors. The intention is to make the Quectel M66 SoM work as an Arduino UNO (Not literally).

Power supply to the board can be supplied by a DC adaptor rated at 5 – 12 V @ 2.5 Amp.

The board must have a USB TypeC port which shall be connected to the SoM’s UART as a 3-Wire setup [*Tx, Rx, GND*], accommodating **Tx** as **D+** and **Rx** as **D-** on the USB Receptacle port. The port must be protected to *ESD* and *Hot Plugging*.

The design must have the following:

* Quectel M66 SoM
* MUP-M618/MUP-M619
* SPI Headers – Native 3.3V
* I2C Headers – Native 3.3V
* 5V Compatible I2C and SPI Headers
* On-Board Power Regulation for the components. - May use an LDO or a Switching Converters. Be considerate while choosing any of these two.
* Protection Circuitry – Basic protection which shall be there to withstand prototyping. This step will showcase your skills of foreseeing the renowned issues.
* LEDs to showcase available power on the power bus, Signals and others. - Be creative on this point.

Layouting of the components on the PCB has special observance.

## Q1: Using the following components:

* Quectel M66 SoM [MPN: M66-FB-04-STD]
* Micro (U)SIM Card + Micro SD Card Holder [MPN: MUP-M618 or MUP-M619]
* Ceramic Chip Antenna [MPN: Pulse Larsen W3009] / Ceramic Patch Antenna [MPN: Molex 208890-0001]
* USB Type-C Interface Port [MPN: Palconn-UTC16-G]
* SPI Header Pins – Exposed SPI Header Pins to interface SPI Peripherals.
* I2C Header Pins – Exposed I2C Header Pins to interface I2C Peripherals.
* UART Header Pins - Exposed UART Header Pins to interface UART Peripherals. Note that Full UART interface is required [Hardware Flow Control].

## Design the following sections in the given order;

1. ***Block Diagram*** – Showing all the components and their data/power flow using lines and arrows.
2. ***Gathering of Datasheets and design notes*** - for the given components, also making a separate personal note which will depict all the calculations and considerations taken while designing the system.
3. ***Schematic*** – A Multi Sheet Schematic, dedicated for each major components/section, you are free to use labels and nets as per your convenience, make sure it is readable.
4. ***PCB Designing*** – *4 Layer Board*
   1. *Board Setup* – Set the board and constraints parameter based on JLC PCB’s capabilities. Constraints being the following; Trace Width for Power and Signal, Clearance between trace and via, Creepage distance etc.
   2. *Stackup* – Set the board stackup using JLC PCB’s [JLC04161H-7628 Stackup.](https://jlcpcb.com/impedance)
   3. Define Board edge and make sure the dimension does not exceed 40mm x 40mm, ∓10mm is okay.
5. ***DRC Check*** *–* Do a DRC check and export it as a report in a readable text format. Must take 2 reports, one right after the Layouting + Routing is complete and one after doing the rectifications based on DRC report.
6. ***Fabrication Files*** – Following fabrication files must be ensured:
   1. **Gerber** – Export the layers into Gerber archive to ensure successful PCB Fabrication, Special checks shall be done on the export of layers.
   2. **3D Cad File** – Export 3D Cad model of the board in STEP or VRML format. Special check shall be performed to ensure all the models are present in the final file.

## Special Considerations:

Stars will be given to those who incorporate the following in the design:

1. On board GPS Antenna using a ceramic patch/chip antenna. Pay special considerations while positioning the antenna and routing the feed. *Consider 50Ω as a trace impedance.*
2. On board GSM Antenna using a Ceramic Chip Antenna. *Consider 50Ω as a trace impedance.* [MPN: W3070]
3. SPI, I2C, UART Logic Level conversion at the header pins to 5V compatible controller such as Arduino and others.

If point 1 & 2 are skipped, Place a SMDUFL connector for both the GSM and GPS feed lines.