Notes and Tips for PCB Design

By Dimitris Alexopoulos

Part 1: Choosing The Right Components

Component requirements:

- **LED**: low current (e.g. 5mA); suitable forward voltage; common anode is useful
- **Accelerometer**: small size (e.g. 2x2mm); low current consumption (e.g. 0.9µA in standby, 155µA in high power); wide supply voltage range; I2C digital communication, physical interrupt pin
- **Microcontroller**: small size (e.g. 5x7mm); internal oscillator; suitable voltage operation; low current consumption (e.g. 20nA in sleep mode); number of I/O pins; programmable weal pull-up resistors; programmable interrupt-on-change pins; PWM outputs (for regulating intensity); I2C digital communication
- **Capacitor**: suitable material (e.g. made of tantalum); suitable voltage
- **Diode**: high maximum forward current; low forward voltage (the voltage you will lose across the diode)
- **Transistor**: suitable maximum current, DS Voltage, GS Voltage; suitable Gate-Threshold Voltage, Drain-Source On-Resistance
- **Power Supply**: small size; suitable maximum input voltage maximum output current; small dropout voltage (e.g. 200mV)
- **Regulator**: may need a capacitor in input and output (check the typical application circuit in datasheet)

Disclaimer: each project has different requirements; this is some guidance

Note:

- For passive components like capacitors or resistors you can use basic parts (cheaper).
- For connections (like battery) we need through-hole headers with a number of pins (e.g. HDR-TH_2P).
- USB charging resistors differ for each manufacturer.
- We use 0 Ohm resistor to replace in future use.
- Use two diodes to connect to different power sources to the circuit
- Control LED with transistors (protects the circuit).

Definitions:

- **NF**: not to be fitted
- **ESR**: Equivalent Series Resistance; view capacitor's datasheet for its value

- **Dropout Voltage**: minimum voltage across the regulator for proper function (input voltage = dropout voltage + output voltage)

How to calculate LED resistors:

R = Voltage across the resistor / Current, where Voltage across the resistor = Vin - Vd

How to calculate the maximum output current needed for my PCB:

1. Check current consumption of components (e.g. for accelerometer at High Resolution Mode 155µA, 4.4mA for microcontroller).
2. It can be useful to place resistors in series between the microcontroller and the transistors to limit the maximum current (gate resistors).
3. It can also be useful to place pull-down resistors in series between the microcontroller and the transistors to prevent initial current from turning the power ON from switching the initial state of the transistors.
4. Calculate maximum current through the pull-up resistors using Ohm's Law (e.g. (3.3V)/(10kΩ) = 330µA)
5. Check the maximum current of the Debug Connector
6. Check the current the regulator itself draws (e.g. Quiescent GND current 17µA)

Part 2: Drawing Schematic

While drawing your schematic you may consider the following useful tips:

- If you want to later separate a part of your circuit you can use connectors to that part.
- Check which Microcontroller pins support Pulse-Width-Modulation (PWM).
- Check which Microcontroller pins support Interrupt feature.
- For connecting parts (like the accelerometer to the microchip) check datasheets and look for specification and pin allocation tables.
- We do not care about the polarity of ceramic capacitors, but we do of tantalum capacitors (plus sign to high voltage).
- When using I2C interface you need to connect external pull-up resistors (smaller resistance for faster communication, bigger resistance for less power consumption); check recommended values.
- For Debugging and Firmware Flashing check Debugger Datasheet and its pin allocation table; Headers and pull-up resistors are needed.
- Interrupt Support makes coding easier and more efficient, as you do not have to continuously check what is the level of the connected signal. You can set an event which will only happen when someone presses the button.

- You can do the debouncing of a button (filter the anomalies) through hardware and software
- Button may need an external pull-up resistor. If you use the internal one, make sure to enable pull-up interrupt in your firmware.
- We use headers for debugging the board.

Note: when possible use components that already exist on your board (it's cheaper)

Part 3: Placing Components On PCB

The steps you should follow are the below:

1. Fix Canvas Properties (unit mm)
2. Set Board Outline
3. Place Parts

While placing parts you may consider the following useful tips:

- Make convenient placement of components. For example, GND of one component should be close to that of another.
- To place a pin in a more convenient place you may flip the component and update the PCB.
- It is useful to hide the component designators and place all of them based on the schematic.
- Check 3D view to see whether the footprints are of realistic scale (footprints may be wider than the actual component).
- Check the blue connection lines for more convenient placement.
- Yellow lines on footprints are white lines on PCB (outline of component).
- Rotating components can be also helpful to the connections (e.g. -rarely- 45degrees).

Attention! Do not place parts on bottom layer of metal components as they might short-circuit.

Consider Heat dissipation! P = V*I (e.g. 3.7V*100mA=0.37W); above 1W it is usually a problem.

Part 4: PCB Layout

Quicker, simpler and easier Layout on 4-layer PCB, other than 2-layer PCB. However, it is a bit more expensive.

While dealing with your PCB Layout you should consider the following steps and tips:

1. *Setting up the rules*:
    - check the capabilities of the manufacturer
    - via drill diameter must be bigger than manufacturer's minimum drill hole size
    - same for via diameter, track width (1oz copper thickness) and clearance
2. *Connect everything with tracks and check if they all fit nicely:*
    - routing width set to 0.3mm
    - do not place the VIAs too close to the tracks
    - via diameter must be bigger/equal to the parameters from design rules
    - do not place vias too close to each other even if they are of the same net (check manufacturer's rules)
    - connect the ground vias and ground parts to the PCB's solid ground plane
    - draw copper area at the same layer as the ground parts
    - you may need to place multiple (ground) vias for one component especially when it is for important power delivery (e.g. for big capacitors)
    - keep vias in the same raw if possible to make future tracks easier
    - it is better to connect power and ground separately with vias for each component instead of linking them all together
    - decide which connections will be on which layer (e.g. power on separate layer, ground on separate layer, route signals on separate layer)
    - copper area should not cover a part of the board that may be cut off later
    - highlight the net to make it easier for you to find where you should connect a component
    - it helps disabling all other layers to have a better look of the usable space (except multi-layer of course)
    - in order not to make connecting tracks impossible for a part of the PCB it is useful to draw vertical tracks and horizontal tracks on separate layers and connect them through vias
3. *Run DRC (Design Rule Check) to fix any errors:*
    - make sure that all nets and components are connected

Note:

- **VIA:** a VIA hole in a PCB consists of two pads in corresponding positions on different layers of the board, that are electrically connected by a hole through the board. The hole is made conductive by electroplating.
- **Minimum clearance:** minimum distance between objects (e.g. between two tracks)
- Smaller sizes (e.g. via diameter) makes the PCB more expensive.

Part 5: Improving PCB Layout

Some useful tips on how to improve your PCB layout or, better, how to improve clearance of your PCB layout:

- Do not place I2C and PVM tracks too close to each other as the PVM signal can be noisy.
- Do not place tracks too close to the board outline.
- Interrupt Signals should be as isolated as possible since there cannot be any noise around them or else they may falsely trigger.
- May be useful to use wider tracks in some connections (e.g. power, ground).

**notes will be updated