STM32 PCB DESIGN Wıth ALTIUM

# Introduction

In this report, STM32 microprocessor will be designed schematically in Altium. All components schematic & PCB libraries are created in Altium by me, but 3D designs will be added from third part sites. I will use STM32 LQFP48 package with 48-pin.

# Components

## LQFP48 STM32 48-Pin

### PCB Library (.PcbLib)

I have created STM32 with LQFP package by the help of Altium, which supplies IPC Compliant Footprint Wizard. The important parameter is the size of the microcontroller that I can find on the net easily.

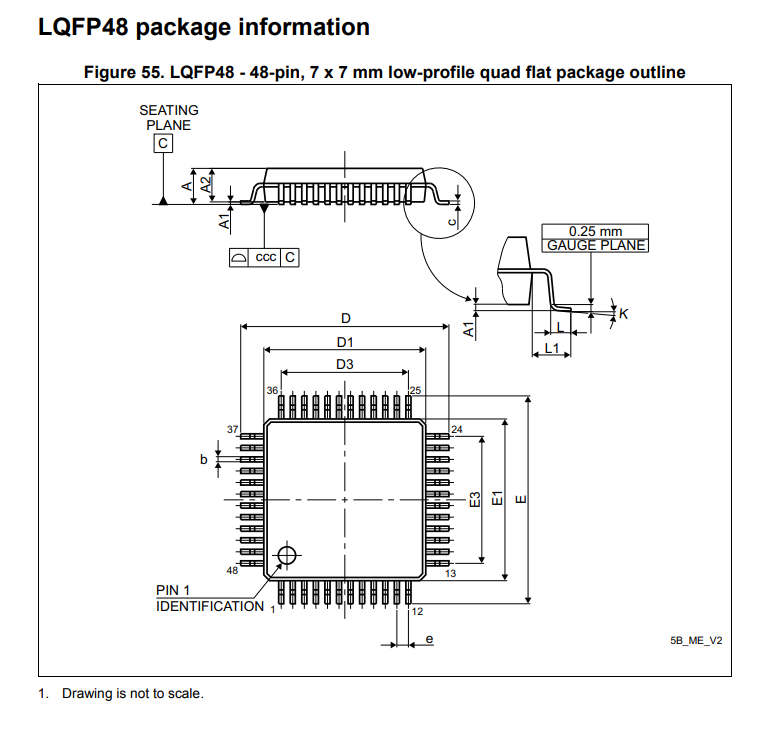


Figure 1. STM32 LQFP48 package information

We need sizes of the package which is also provided by the manufacturer:

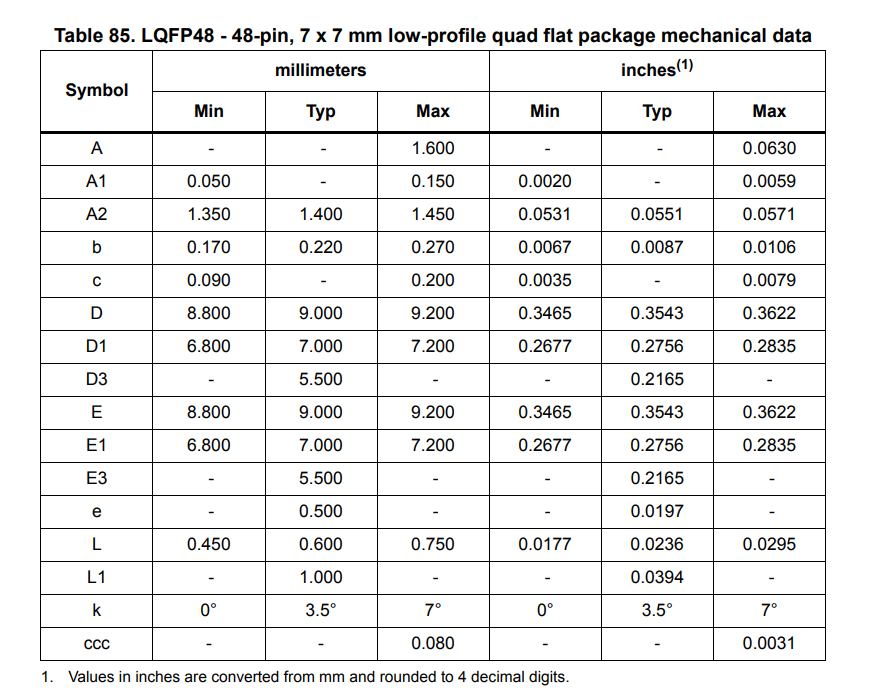


Figure 2. Sizes of LQFP48 package

Now, we can design STM32 in Altium without drawing one by one and we do not need to 3D body with the help of IPC Compliant Footprint Wizard (follow the path in PCB library documents → Tools → IPC Compliant Footprint Wizard)

After entering the given values to the program generates 2D and 3D body:

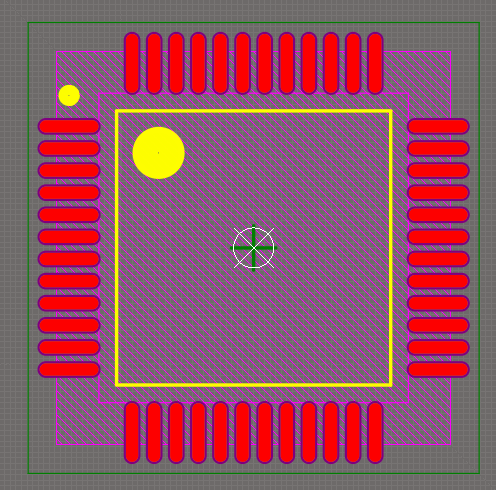


Figure 3. LQFP48 2d footprint

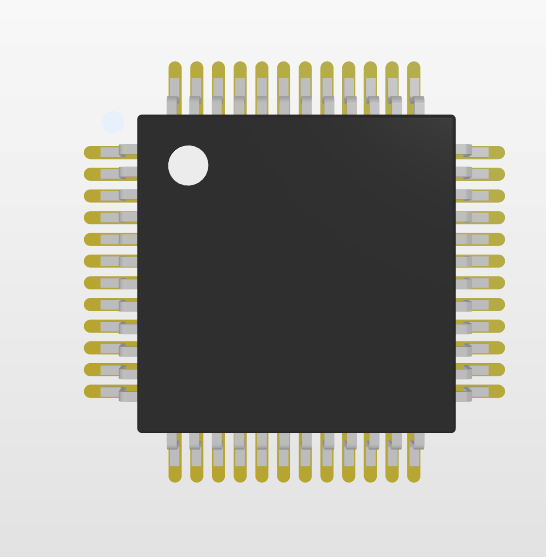
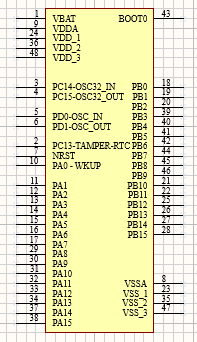
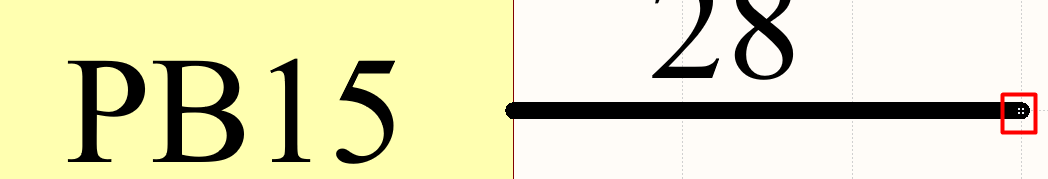


Figure 4. 3D body of LQFP48 package

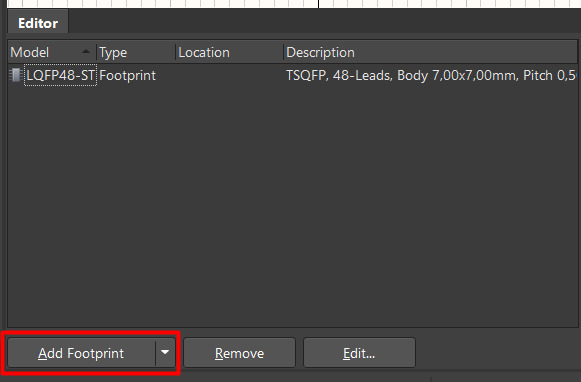
### Schematic Library (.SchLib)

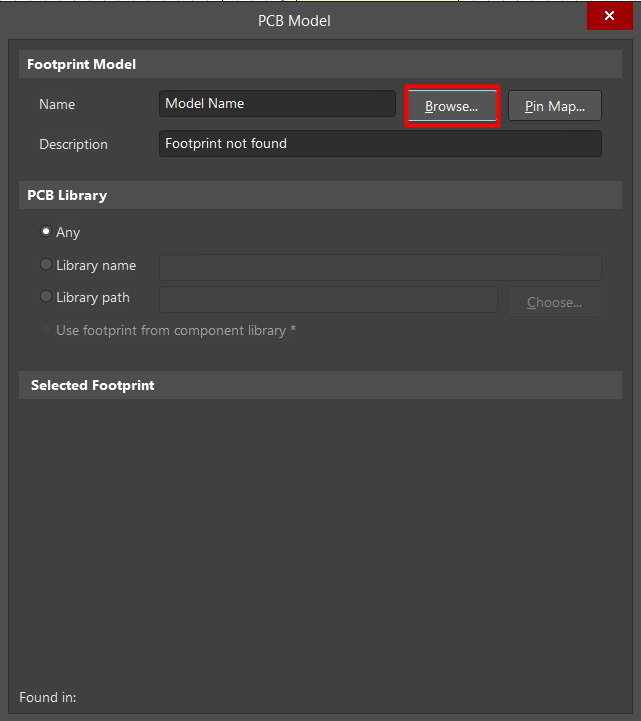
To create schematic: Go to SCH Library → AddI draw a rectangular and added 48 pins to around of the rectangular. It should be mentioned that I did not design schematic in sequence. In other words, it is not going like 1-2-3-4-…-48. Instead, I have divided pins into the groups like power pins, serial communication pins, ground pins etc. Be careful when adding pins that white rectangular of pin should be outside as below.



The last step is adding PCB footprint to the schematic:

From Editor tab → Add Foodprint: → Browse → Choose 3D model print and Click OK → OK





It is now added. Don’t forget writing **Designator** value of the component which is U? for the integrated components.

U? → integrated components

R? → Resistors

C? → Capacitors

J? → Jumpers

B? → Buzzers, buttons

Y? → Crystal Oscillators

L?→ Inductors

## 5V Regulator D7805 DPAK package

I will use 5V regulator but DPAK package

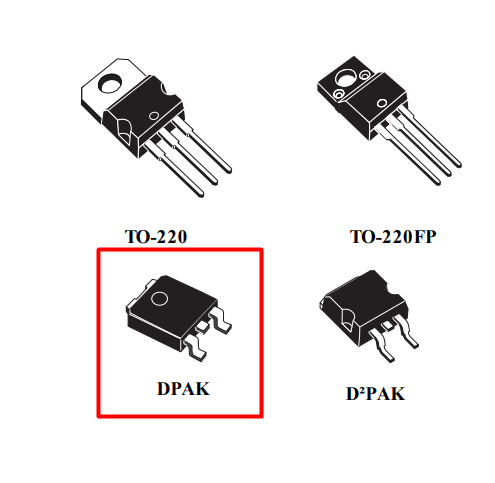


Figure 5. Package types of regulator

### PCB Library (.PcbLib)

Again, Altium provides us to use IPC Compliant Footprint Wizard to generate the regulator in DPAK package

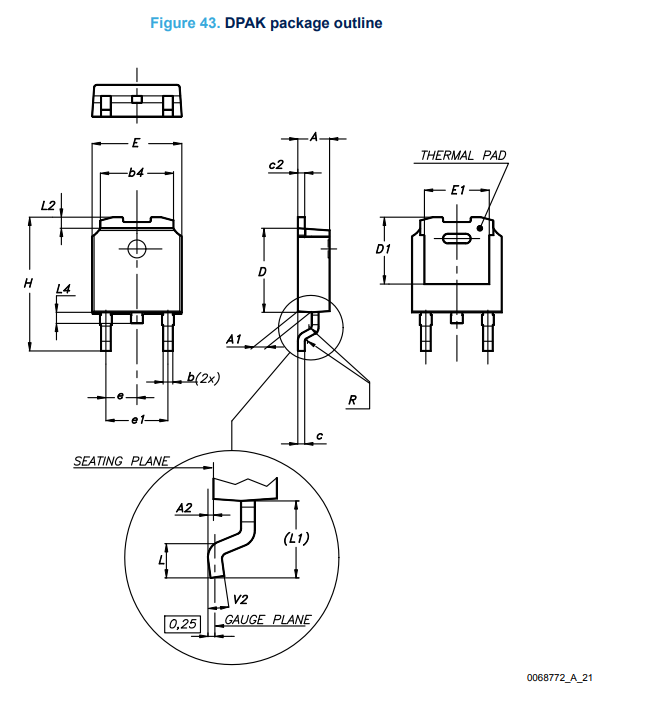
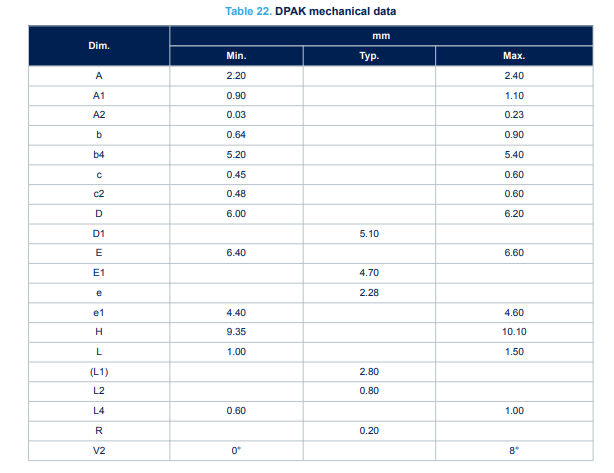


Figure 6. DPAK package information



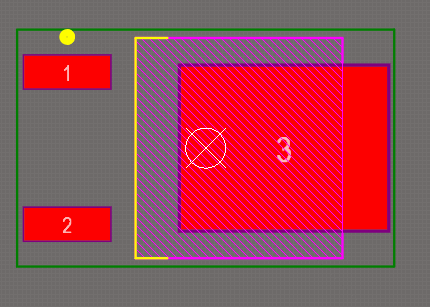


Figure 7. PCB schematic of DPAK regulator

### Schematic Library (.SchLib)

Now, we need to pin configuration schematic to create schematic of the regulator

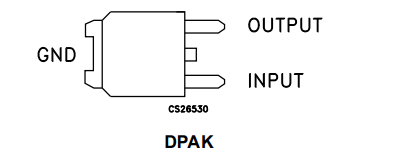


Figure 8. Pin configuration of the regulator

To create schematic go to Schematics Library Documents → SCH Library → Add → Draw the schematic

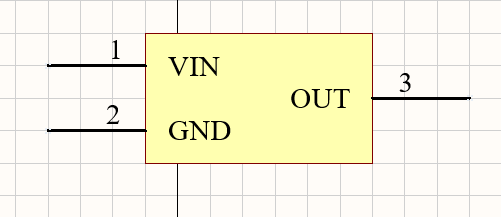


Figure 9. Schematic of the regulator

Add schematic PCB footprint as shown in part **2.1.2**

## MPU-6050 QFN Package

### PCB Library (.PcbLib)

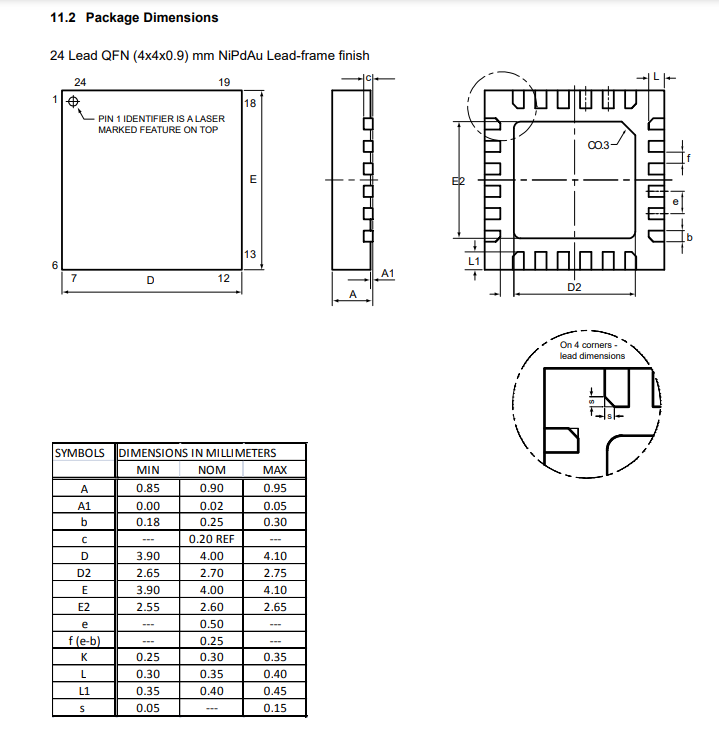


Figure 100. MPU-6050 package dimensions

Again, use Footprint wizard, QFN package is used to create PCB model of the component:

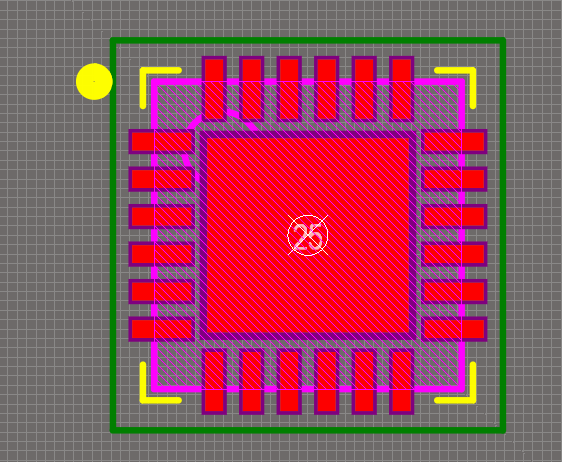


Figure 11. PCB schematic of MPU-6050

### Schematic Library (.SchLib)

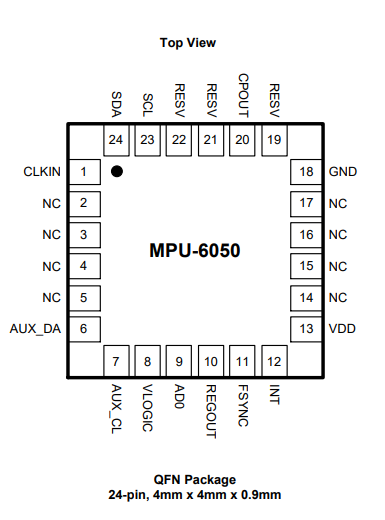


Figure 12. MPU-6050 PINs

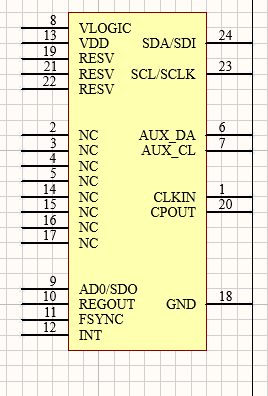


Figure 13. Schematic of MPU-6050

Add PCB footprint and MPU-6050 is completed.

## BMP280 Digital Pressure Sensor

For this BMP280 sensor, Altium IPC Footprint Wizard does not have package model of the sensor. Therefore, it should be drawed by third person (me 😊 )

### PCB Library (.PcbLib)

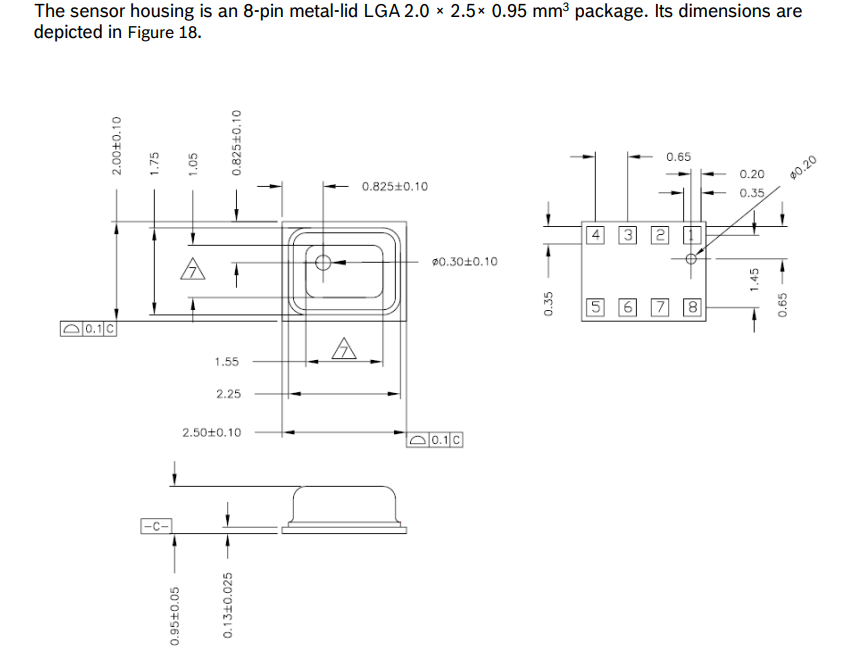


Figure 14. Dimensions of the sensor

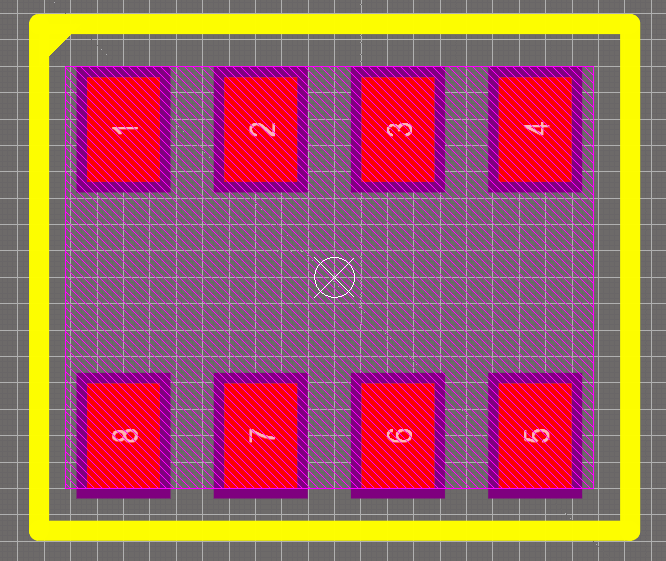


Figure 15. PCB schematic of BMP280

Now we need to insert 3D body to the footprint, I am using [Digi-Key](https://www.digikey.com/) or [ComponentSearchEngine](https://componentsearchengine.com/). I prefer to use Digikey library

We need to go to the Documents & Media → [EDA Models](https://www.digikey.com/en/models/6136307) → Select Download Format → Choose .step format

After downloading .step file, click **P → 3D Body → Select .step file you downloaded**

Put 3D body according to the pins and check if it is proper or not.

### Schematic Library (.SchLib)

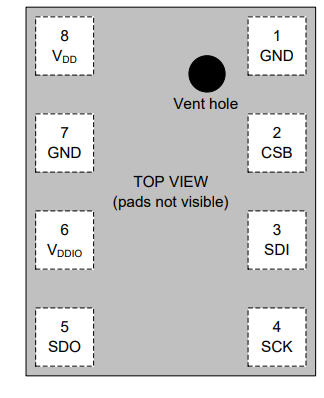


Figure 16. PIN description of BMP280

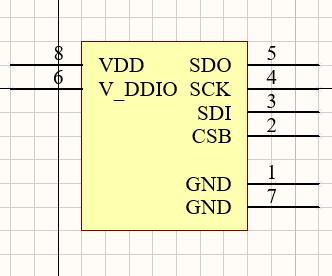


Figure 17. Schematic design of BMP280

Add footprint: Editor → Add Foodpring → Browse → Select PCB footprint → OK → Ok