STM32F103C8T6 Breakout board

Aim

The purpose of the breakout board is to implement and/or tailor the firmware written in the "Drivers-for-STM32-MCUs" repository (https://github.com/Suvashan1/Drivers-for-STM32-MCUs) on another microcontroller from ST Microelectronics and interface calibrated sensors such as the BMP180 pressure and temperature sensor, with communications interfaces. I have also aimed to present my skills in PCB (printed circuit board) design and assess my shortcomings as an engineer, to provide hardware design which is more efficient in future revisions. It is a PCB constructed entirely from scratch and serves to provide the same use as an Arduino nano or STM32 blue-pill, which are popular development boards used for prototyping with sensors.

Microcontroller used and core components

- **STM32F103C8T6 microcontroller from ST microelectronics** – This microcontroller had been used, as it provides an adequate amount of basic communication peripherals (2 I2C peripherals of which 1 was used, 2 USART peripherals of which both were used, and 2 SPI peripherals of which 1 was used). These communication peripherals will allow adequate testing of the firmware written. Other interfaces such as CAN and I2S are usable on the microcontroller, for later revisions.
  This series of microcontrollers are known as "mainstream performance line" by ST, as they use an ARM Cortex M3 processor, which is adequate for general usage, and cost-effective. Other microcontrollers were considered such as the STM32F407VGT6, which is native to the drivers already written, but are more expensive and requires more design considerations.
  A USB 2.0 FS interface is also available on this microcontroller, allowing for USB further, but non-reliant, firmware development in the future. Essentially, one can still power the board and program it, without the use of USB, since there is provision for BOOTx (x = 1,2) mode changes. The board can be programmed via SWD (Serial-wire debug), which is an interface that has been accounted for.

- **Power supply section and filtering** – A 3.3V regulator (AMS1117) was used, for voltage supply of the microcontroller, from an input supply pin (allowing an input voltage from the user up to 18V). A Schottky diode was added at the input line of the regulator, and prevents any reverse bias connection, from the input supply, and an additional diode was added from a 5V line of the USB port.
  According to the hardware design application note from ST (AN2586), **one (typically two, as per consideration of further noise suppression) 100nF capacitor must be added for each VDD pin and a 10uF capacitor.** Since we consider three VDD pins and a VBat (battery connection for the microcontroller, which should be connected to VDD if a battery is not intended), **nine capacitors have been added, as a decoupling network.** Decoupling and filtering for the VDDA(ADC voltage supply), had been considered, and referenced from AN2586. **A ferrite bead has been added (600Ω @ 100MHz), with decoupling 100nF and 1uF capacitors**, in connection to the VDD pin.

- **USB implementation and ESD protection –** Since USB functionality had been added, a USB-B micro had been chosen for aesthetic and size considerations. ESD protection has been added using a USBLC6-2SC6 IC. This prevents static discharge through the board.

- **HSE (High-Speed External) and LSE (Low-Speed External) crystal oscillators** – Both crystals (16MHz and 32.768kHz respectively) have been added, and their respective load capacitors added. Calculations of the capacitor values have been added on the schematic sheet, where the parasitic capacitance of the board had been chosen as 2pF. The LSE crystal allows the use of the RTC (Real-Time Clock) peripheral at another stage of development.

- **BOOTx modes and RESET button –** A 2-pole dip switch has been added to alter BOOT1 (PB2) and BOOT0 has been added. This allows the **use of DFU Mode (Direct-Firmware Upgrade)**, where the microcontroller may be programmed directly by USART. The most popular mode of programming an ST microcontroller, is by using SWD, which is the alternative to DFU mode. **A reset button has been added** to provide an option to perform a hard reset on the microcontroller.

Design considerations when drawing the schematic and component layout

- JLCPCB was the chosen fabrication service (PCBway was also considered but proved to be more expensive). A board of dimensions 50x50mm has a price of $5(R82) for a 4-layer PCB. Boards with larger dimensions fare worse in terms of cost, even if exceeded by 1mm. However, a larger board is never recommended, especially dealing with communications peripherals. A 50x31mm (L X B) board size has been used.
- **Crystal oscillators, decoupling capacitors and USB had been placed closer to the microcontroller**, than other peripherals. This prevents any harmonics from affecting those peripherals or the microcontroller. Schottky diodes of the power supply section specifically had also been placed further from the crystal oscillators, to prevent harmonics and electrical noise.
- **A "drive-resistor" is added to the output pin of the HSE crystal oscillator** on the microcontroller, to prevent the output pin from sinking excessive amounts of current and "over-driving" the crystal oscillator.
- Header pins of peripherals have been included, as close to the microcontroller as possible, and on the respective edges of their pins on the microcontroller.
- **Components which could be recalculated and replaced if the peripheral proves to be problematic have been assigned larger PCB footprints**, to provide easier reflow and hand-soldering. Examples of these can be seen in the pcbnew file, where 1206 pull-up resistors have been used for the I2C peripheral, as the speed of the peripheral is influenced by these resistors and may be replaced. Through experimentation, 2.2kΩ and 4.7kΩ are standard values. The drive-resistor of the HSE crystal oscillator may also be replaced if the crystal cannot sustain oscillations. This may be monitored in software.
- **An additional USART peripheral has been added to provide usage of the HC-05 bluetooth module** (https://www.robotics.org.za/HC05-PINS), and the I2C peripheral uses a 4-pin header for the use of a BMP180 (https://www.mantech.co.za/ProductInfo.aspx?Item=15M2316) pressure sensor. These peripherals have been accounted for, to provide simplistic yet thorough testing. **The HC-05 will be used to receive commands from a smartphone via Bluetooth and transmission of pressure readings from the BMP180 sensor may be processed, either transmission to the smartphone or using printf statement in the IDE.**
- The 10-pin header chosen for use of the SWD interface, is to be used with a STM32F407VG-DISC1 development board which I own. This development board will serve as debugging probe via ST-LINK and allow for programming and debugging of the microcontroller.