Overview

Designing the QuickUSB circuit into a custom PCB is not as daunting of a task as you might believe. There are many reasons why you might choose to design QuickUSB into your own PCB, such as reducing pre-unit cost, reducing the physical size a design, switching to a different FX2LP package, etc. Whatever your reason for designing QuickUSB into your PCB, there is the increased risk that the USB circuit will not work because you may inadvertently introduce errors into the circuit. This application note describes the process of integrating QuickUSB into your design and provides a design checklist to help ensure that the QuickUSB circuit will perform as expected the first time.

The QuickUSB Circuit

The QuickUSB Schematic and QuickUSB Bill of Materials for the QuickUSB Module are both freely available on the QuickUSB website. The easiest way to integrate the QuickUSB circuit into your design is to copy it exactly, except for leaving off the Hirose FX8-80P-SV connector as it is no longer needed. Since you are designing the circuit into a custom PCB, you do have the freedom to alter parts of the design to better fit your application. The following sections will discuss the changes you can and cannot make to the QuickUSB schematic.

Safe Changes to the QuickUSB Circuit

Remove the Interface Connector

Since you are designing the QuickUSB circuit into a custom PCB, you no longer need the target interface connector J2. Instead, you make connections directly using traces on the PCB.

Use a Different FX2LP IC

The QuickUSB Module is designed with a Cypress CY7C68013A-128AXC IC, which is Cypress's High-Speed USB peripheral based on their FX2LP technology. They offer a number of variant ICs that implement the FX2LP core that are compatible with QuickUSB. The variants offered by Cypress allow you to select ICs in different packages (QFN, SSOP, TQFP, and VFBGA), different pin-outs (128-pin, 100-pin, and 56-pin), as well as different operating temperature ranges (commercial and industrial). For a complete listing of available FX2LP ICs, please visit the Cypress EZ-USB FX2LP website.

All of the Cypress EZ-USB FX2LP ICs should be compatible with QuickUSB, but the only one extensively tested by Bitwise Systems is the CY7C68013A-128AXC. If you choose to use a FX2 variant, keep in mind that you may lose functionality. For example, when using the 56-pin version of the FX2 you lose access to the GPIF address bus, Port E, RS-232, and the BKPT pin (used to indicate when it is safe for bus-powered designs to power downstream electronics). Figure 3-1 on Page 16 of the Cypress FX2 Datasheet shows precisely what pins are lost when moving between different packages. Make sure that your design will work with the selected package before choosing to use it.

Remove RS-232

If you do not use the RS-232 ports in your design then you are free to omit the RS-232 level converting electronics in the QuickUSB circuit. This means you may omit the following components from the QuickUSB Schematic: U2, C1, C2, C3, C6, and C7. You may leave the TxD0 and TxD1output pins open and should connect the RxD0 and RxD1 input pins to ground.

Remove the I2C Buffer

U3 in the QuickUSB Schematic is an I2C signal buffer. It prevents I2C devices connected to the QuickUSB I2C bus from seeing the FX2 communicate with the EEPROM (U4) when the FX2 powers on. You may omit this buffer if you are certain that communication with the QuickUSB EEPROM will not interfere with, or be interfered by, other electronics on the I2C bus. Additionally, no other devices on the I2C bus may use I2C address 0x51 as that is the address used by the EEPROM. If you choose to not use the I2C buffer in your design, you may omit the following components: U3, C10, and the two associated pull-up resistors (part of the R19 resistor network). Note that you must have exactly one 2.2 kOhm pull-up resistor on each of the SCL and SDA lines (the QuickUSB circuit has two sets of pull-up resistors—one on each side of the I2C buffer).

Remove the Series Termination Resistors

Most of the signals that route to the target interface connector have series terminating resistors on them. It is recommended to keep these resistors in your design. You may, however, remove these resistors as long as you fully understand the effects of removing them. If you choose to not include these resistors in your design, then you may omit the following resistor network components: R20, R21, R24, R25, R26, and R27.

Change the Power Supply

U5 on the QuickUSB Schematic is the power regulation chip used for the QuickUSB Module. It contains one linear dropout (LDO) regulator and one switching regulator. The LDO used to convert the unregulated 5V power from the USB connector to a regulated 3.3 V supply used to power the FX2, I2C buffer, EEPROM, and RS-232 level converter. The switching regulator is used to convert the unregulated 5V power from the USB connector to a regulated 5V supply that is routed to the target interface connector to provide power to bus-powered electronics.

You can alter the power supply of the QuickUSB circuit, but you must follow a few guidelines in order to ensure proper device operation:

- Power from the USB connector is an unregulated 5V ± 5% supply. Make sure that all regulators or devices connected directly to the VBUS pin of the USB connector can tolerate variations of voltage from 4.75V – 5.25V.
- USB can provide at most 500 mA of current. Allocate 100 mA for the QuickUSB circuit and allocate at most 400 mA for other devices powered from VBUS or derivative supplies.
- If you intend to draw more than 50 mA of current, use the BKPT pin output of the FX2 to determine when it is safe to draw large amounts of current from the USB port. The BKPT goes high after VBUS becomes present, stabilizes to >= 93%, and the FX has powered up and began to run the QuickUSB firmware.
- Be sure that you have reviewed and meet all of the requirements listed in the FX2 Reset and Power Considerations section of this document.

If you choose to implement your own power circuit, you may omit the following components from the QuickUSB circuit: U5, R16, R17, C20, C22, C23, C24, and C25.

Unsafe Changes to the QuickUSB Circuit

Change the EEPROM or I2C Pull-up Resistors

It is paramount to include 2.2 kOhm pull-up resistors on the I2C SCL and SDA signals. The pull-up resistor value must be 2.2 kOhm (not 2.0 kOhm or 2.5 kOhm, but 2.2 kOhm). Using any other valued resistor can cause the FX2 to fail to access the EEPROM causing the USB device to not enumerate.

Insufficiently Bypass FX2 Power Pins

C11, C12, C13, C14, C15, C16, C17, C18, C19, and C21 are all 0.1uF bypass capacitors for the FX2. It is important to include these in your design and place each one close to a power supply pin (VCC) on the FX2. Note that C21 is the bypass capacitor for the isolated analog 3.3V supply, which is the AVCC pin on the FX2. In addition, if you use a different FX2 package you will need to adjust the number of bypass capacitors in the design appropriately. In general, you should have one 0.1uF bypass capacitor associated with each VCC and AVCC pin on the FX2.

Tap into the USB D+/D- Signals

The D+ and D- pins on the USB connector must route directly to the FX2 without any taps into other electronics or test point taps. These two signals are sensitive high-speed data lines and should be treated as such.

Bus-Powered vs. Self-Powered Designs

Bus-powered designs are those that power all electronics using power provided from the USB port. USB is capable of supplying 500 mA of current at an unregulated 5V. You may power electronics on your custom PCB using the USB VBUS power supply, or supplies derived from VBUS, assuming your electronics will never draw more than 400 mA (100 mA is allocated for the QuickUSB circuit). You must make sure to not draw more than 50 mA of current before QuickUSB has a chance to power-up properly. The BKPT pin of the FX2 is used by QuickUSB to indicate when it has finished powering up and that it is safe for bus-powered electronics to begin drawing power from the USB port.

Self-powered (or mixed-power) designs are those that have external power supplies that may be present when the USB connection is not. Special care must be taken to ensure proper operation in this case as power could be provided by external sources, by the USB port, or both. To ensure proper operation in all cases, make sure to carefully review all the notes make in the FX2 Reset and Power Considerations section of this document. Designs that have external power supplies may still use power from the USB port. In such cases it is important to follow all of the guidelines described above for bus-powered designs in addition to the guidelines described here.

FX2 Reset and Power Considerations

The Cypress FX2 has special reset and power requirements to maintain compliant with the USB 2.0 specification. Cypress has published Application Note 063: EZ-USB FX2LP™/AT2LP™ Reset and Power Considerations describing how to meet these requirements. Please note that the QuickUSB firmware meets the "No D+ without VBUS" requirement by polling the WAKEUP pin and disabling the D+ pull-up when VBUS is removed. In order for your design to comply with this requirement you must pull the WAKEUP pin of the FX2 high to VBUS, or a voltage source derived from VBUS, using a 10

kOhm resistor on both bus-powered and self-powered designs. Do not pull the WAKEUP pin high to any other external power source.

Layout Considerations

- Use a dedicated ground layer and power layer.
- Keep the USB D+ and D- trace lengths as short as possible and matched in length.
- Place the FX2 bypass capacitors as close to their respective supply pin as possible.
- Keep the XIN and XOUT trace lengths as short as possible.
- Space all traces as least 3 times their trace width from each other to prevent crosstalk.
- Ensure adjacent signal layers in the PCB stack-up are separated by a dedicated ground or power plane to prevent crosstalk.
- Keep analog ground and digital ground separated in the design and connect them through a single ferrite bead inductor.

Design Checklist

- ☐ The design uses a Cypress FX2LP IC, such as the CY7C68013A-128AXC.
- ☐ The EA pin of the FX2 is directly connected to ground.
- ☐ The WAKEUP_B pin of the FX2 is pulled high to VBUS, or a voltage source derived from VBUS, through a 10 kOhm resistor.
- ☐ The nRESET pin of the FX2 is pulled high through a 10 kOhm resistor.
- ☐ All digital ground pins on the FX2 are connected to digital ground on the PCB.
- ☐ All analog ground pins on the FX2 are connected to analog ground on the PCB.
- ☐ The digital and analog ground planes are connected through a single ferrite-bead inductor.
- ☐ All VCC pins on the FX2 are connected to VCC on the PCB.
- ☐ All AVCC pins on the FX2 are connected to AVCC on the PCB.
- ☐ The SCL and SDA pins on the FX2 are pulled high through 2.2 kOhm resistors.
- ☐ A 24 MHz crystal is connected to the XIN and XOUT pins of the FX2, along with any stabilizing capacitors required by the crystal.
- ☐ Bus-powered designs do not draw power from VBUS, or power supplies derived from VBUS, until the BKPT pin of the FX2 becomes high unless the amount of current being drawn is 50 mA or less.
- ☐ If the IFCLK pin of the FX2 is being externally sourced (i.e. not internally generated by the FX2 and outputted but instead being inputted), then IFCLK must be: 1) free-running, 2) be between 5 and 48 MHz, and 3) present on the IFCLK pin before SETTING_FIFO_CONFIG[7] is set to zero to select external clock sourcing (which could be when the FX2 boots if it is set as the default power-on value).
- ☐ The write-protect (WP) pin of the EEPROM is connected to ground.
- ☐ The address lines of the EEPROM are as follows: A0=VCC, A1=GND, and A2=GND.

Bitwise Systems
6489 Calle Real, Suite E
Goleta, CA 93117
V (805) 683-6469
F (805) 683-4833
info@bitwisesys.com