
Homework

This assignment is due by our second meeting (2/08/03).

1. Read chapters 13-14 of the EZ-USB FX technical reference manual from <http://www.cypress.com>.
2. Explore the new course web site at: <http://ece-www.colorado.edu/~mcclurel/usbdesign.html>
Please send me good URLs as you find them.
3. Learn about the PCB design process. Study the PCB manufacturing handout and understand the basic steps. Be able to define the following terms: PCB, stackup, core material, prepreg, dielectric, via, through-hole, SMT, pad, anti-pad, stripline, microstrip, plane, keep-out, microvia, blind via, buried via, solder mask, silkscreen, tooling, test points, 3070 test, bed-of-nails testing, flying probe testing, Gerber, photoplot, aperture, NC drill, plated through hole (PTH), non-plated through hole, photoresist, FR-4, registration, fiducial. Links to some glossaries are available on the course web site.
4. Examine the Cypress development boards and the reference design schematics from the Cypress web site. See if there are any reference schematics for the CY7C64613-52NC controller.
5. Develop schematics for a small PCB (~2"x3", determine PCB vendor size limits before making final decision). The schematics should meet the following requirements:
 - 5.1. Orcad Capture file format (you can leverage the schematics from Cypress)
 - 5.2. Letter size, 8.5"x11.0" (landscape)
 - 5.3. Organized as follows:
 - 5.3.1. Sheet 1: Left edge is index/table of contents; lower right is title block, including author names; center and top right is block diagram of the design.
 - 5.3.2. Sheet 2: Entire design, if design is small enough; otherwise, this sheet should contain the power system design and decoupling capacitors.
 - 5.3.3. Sheet 3+: Additional pages of design if it is more than can easily fit onto one sheet. Use off-sheet connectors/ports as necessary. Consider adding test points to unused pins.
 - 5.4. Contain neat comments regarding special functionality, load options, or layout requirements. Explain the intended usage of any switches or special options.
 - 5.5. Cypress CY7C64613-52NC controller, 3.3V regulator, jumper to allow selection of bus power (Vbus), power indicator LED, RS-232 connector, RS-232 charge pump line driver/receiver, I2C EEPROM (see chapter 4.11, 24LC128 or 24LC256 acceptable), and as much extra circuitry as is prudent in the PCB space and development time allowed. Surface mount or through hole circuitry is acceptable. The Cypress controller will be surface mount. The RS-232 and USB (type B) connectors will be through hole.
6. Determine the specific board size and layout requirements from the PCB vendor you plan to use. Make sure you follow all minimum hole size and trace width and spacing requirements.
7. Lay out the design as a two layer design. Keep the USB traces routed in parallel, equal length, and as short as possible. Note that the package pinout and size is shown in the CY7C64613 data sheet. Add silkscreen which includes reference designators for all components.
8. Generate all output files for PCB vendor, including gerber, drill file, etc. Print out artwork and verify placement of all elements and that silkscreen markings will be visible after components are loaded. Be ready to submit your design for manufacturing.