sparkfun ELECTRONICS

WEBSITE: sparkfun.com
6175 LONGBOW DRIVE, SUITE 200
BOULDER. COLORADO USA
ZIP CODE: 80301
[303] 284.0979 [GENERAL] P
443.0048 F

Creating a Schematic

Step-by-Step Guide to creating a USB to Serial Interface using the FT232

1. With Eagle open, go to File->New and select Project. Give the project a name, but no version number: USB Converter
2. Right click on the USB Converter project (now located in the projects folder) and select New->Schematic.
3. Click the 'Add' button in the toolbar and then find the part named “FRAME-LETTER” in the list of parts. (Tip: The 'search' box in the bottom of the parts window doesn't work very well. It's better to manually find the part.)
4. Double click the part to select it. This will take you back to the schematic screen. Place the screen at the origin. There's a dotted cross-hair that identifies the origin.
5. Save the schematic. Be smart about saving schematics; use the project name followed by a version number. For this schematic use the name 'USB Converter-v10.' That stands for major version 1, minor version 0.
6. Now add all of the parts that are in the reference design. Normally you have to search through the parts to find what you need, and after a while you become accustomed to the available parts. Here's what you'll need for this project though:

Part Name	Quantity
FT232RLSSOP	1
INDUCTOR (0603)	1
CAP (0603)	4
USB (USBPTH)	1

*Tip: You can press the Escape key after adding your part to the schematic to go back to the Parts List

7. Use the 'Move' command (Shortcut Key 'F7') to arrange the components on the schematic so that it looks similar to the sample circuit in the datasheet.

Creating an Eagle Schematic

SparkFun Electronics Summer Semester

8. Assign values to the components that require them (caps and ferrite bead). Use the 'Value' command to do this (Shortcut Key 'F5'). Use the values from the sample circuit in the datasheet.
9. Check out the name of the inductor: U$1. This is just ugly! The proper prefix for an inductor is the letter 'L.' Use the 'Name' command (Shortcut Key 'Alt+F5') to rename the inductor L1.
10. Some of the labels for the part names and values may not be optimally placed; perhaps they overlap each other or you just don't like their locations. We can move the labels indepently of the part by first 'Smashing' the labels from the part, then 'Moving' the labels. Use the 'Smash' command (Shortcut Key 'F6') to smash the labels of all the parts; after smashing a part you'll notice that a crosshair appears on each label. Use the 'Move' command and move the labels by clicking and dragging their corresponding crosshair.
11. Add Power and Ground to the schematic by finding and adding the part named 'VCC' and the part named 'GND' to the schematic.
12. In a schematic we connect parts together using 'Nets' (think of them as wires). Use the 'Net' command (Shortcute Key 'F9') to connect the VBUS pin of the USB connector to one end of the Ferrite Bead.
    1. Start a net at the endpoint of a pin by clicking the pin. Pins are red!
    2. While running a Net from one pin to another you will have to add in some angles to the Net to reach from place to place. Only use Right Angles. You can change the direction of the angle by holding 'Ctrl' and clicking the right mouse button while running the trace.
    3. End a Net by clicking on either another net or another pin.
13. Before moving on to the next net, check to make sure that the net has been connected properly by using the 'Show' command (Shortcut Key 'F1'). Select the 'Show' command and then click the Net; the entire signal will be highlighted. Make sure the green wire and the red pins of the signal are highlighted. If the pins aren't highlighted, the net is not connected to them.

14. Now connect the opposite end of the Ferrite Bead to the VCC pin of the FT232 part using a net. Also connect the VCC part that was added to the schematic to this net.
    1. You will create two nets that will merge into one signal. When the Net is added from the bead to the FT232 module it will be just like the first net you added.
    2. Next run a net from VCC to the net you just created. A couple things might happen, first you will be given a prompt that reads “Merge segment net N$X into supply net VCC?” This is Eagles way of verifying that you want to connect two signals. Click 'Yes' to accept.
    3. Now the Net should end; however when you move your mouse the net may still be going. Press Escape to end the net.
        1. Go to Options in the menu bar, and open the Set... window. Go to the 'Misc' tab and enable the option 'Auto end net and bus.' This will stop a net after you've connected it to a new signal.
    4. Now there should be a 'T' shaped signal that connects the VCC to the FT232 module, the Ferrite Bead, and the VCC part. We want a better indicator that the signals are connected at the intersection of the 'T,' so we'll add a Junction. Use the Junction command (Find the Dot in the Toolbar) and add a junction at the intersection of the T.
        1. Go to Options in the menu bar and open the Set... window. Go to the 'Misc' tab and enable the option 'Auto set junction.' This will automatically create junctions when signals are merged.
    5. Verify the new VCC net by using the 'Show' command and make sure the proper signals are all highlighted.
15. Run a net from the VCCIO pin to a pin on one of the 100 nF capacitors (if the capacitor isn't nearby, move one so that it is).
    1. Oops! We connected the wrong pin. Check the sample circuit again, it is actually the 3V3OUT pin that needs to be connected to the capacitor. Use the 'Delete' command (Shortcut Key 'F3') to delete the net; each segment of the net must be deleted individually.
    2. Now add the correct net from the 3V3OUT pin to the capacitor and double check it with the 'Show' command.
16. Move the GND part under the capacitor that was just connected. Connect the opposite end of this capacitor to the GND part.
17. Run a net from the VCCIO pin to the VCC net created earlier and double check the connection.
18. Move the 10 nF capacitor so that it is located beneath and to the right of the USB part. Connect one end of the Capacitor to the VBUS net created earlier.

19. The other end of the capacitor needs to be connected to ground, but there's a better way to do this than to drag a net all the way over to our existing ground. A Net doesn't actually have to be *physically* connected in the schematic to connect two parts. If nets that aren't connected have the same *name* then they will be connected in the design. So we just need another net with the same name.
    1. We can do this three different ways: add another GND part from the parts list, copy the existing GND part and place the new one where we want it, or just add a 'stub' net to the end of the capacitor and name it GND.
    2. Start by adding a net to the end of the capacitor; drag it down from the end a bit and then click add the stub. Press Escape to end the net without connecting it to anything.
    3. Use the 'Name' command (Shortcut Key 'Alt+F5') and click the stub to change the name. Assign the name GND and click OK. You'll be prompted to check if you really want to connect the new net to GND, confirm this.
    4. Now use the 'Show' command and select your GND stub; notice that the other GND net that was created earlier is also highlighted. This verifies that the two signals are connected in the design even though we don't see a physical connection in the schematic.
    5. It's bad design creation to leave a GND net without a Ground symbol, though, so we'll also copy the GND part. Use the 'Copy' command (Shortcut Key 'F8') and click the GND part by the FT232. Now just place the GND part on the end of the GND stub just created. Make sure the pin doesn't overlap the net, the pin and the net should only touch at their endpoints.
20. Connect the GND pin of the USB connector to the GND net just created.
21. Copy the GND part once more and place it beneath the two remaining unconnected capacitors. Run a net from one end of each of these capacitors to the new GND part. This time there won't be a prompt to connect the signals, so verify the connections with the 'Show' command. Make sure the new nets are connected to all the other GND nets in the schematic.
22. Repeat step 20 with the VCC part and the opposite ends of the two capacitors.
23. Run a net from D- on the USB connector to the USBDM pin on the FT232. You'll have to cross a net that's already been created. The nets won't merge unless you click on a previous net. Crossing nets is fine, but avoid placing nets on top of each other.
24. Run a net from D+ on the USB connector to the USBDP pin on the FT232.
25. Connect the 4 ground pins (AGND, GND7, GND18, GND21) of the FT232 to a GND net.
26. Connect the TEST pin of the FT232 to a GND net.

27. The design is about finished, but it seems there's something missing. The purpose of the design is to create a module that will convert a USB signal to a Serial signal. We have a place where the USB signal comes (the USB connector), but we have nowhere for the Serial signal to go.
    1. Add a serial header to the design. A serial header has 4 pins (VCC, GND, Tx, Rx).
    2. Use the 'Add' command in the Toolbar and find the part M04. Select the M04PTH footprint.
28. Run a net from the TXD pin on the FT232 to pin 3 of the header.
29. Run a net from the RXD pin on the FT232 to pin 4 of the header.
30. Connect pin 1 of the header to VCC.
31. Connect pin 2 of the header to GND.
32. There's no name for the 4 pin header. Add a note to the schematic so we know what it is.
    1. Use the 'Text' command in the Toolbar to add a note. Assign the text Serial Header.
    2. We can place the text in different layers. By default the text is probably green and assigned to the Nets layer. Click the middle mouse button and choose the Info layer. Place the text near the 4 pin header.
33. For vanities sake the schematic should be centered on the frame. Use the 'Select' command (Shortcut Key 'Alt+F7') and select the entire design. Moving a group of parts is a bit different than moving individual parts; hold the 'Ctrl' key and right-click near the center of the group. While holding the 'Ctrl' key move the group to the desired location. Click to finish the move.
34. The schematic is finished, but we need to check the design before moving on. Run the ERC (Electrical Rule Check) to check for errors on the board. This won't check the validity of the design (you have to do that!), it's more like a spell-check for a schematic. The ERC will check for unconnected nets, warn if pins may look connected but aren't, check if a pin isn't the right type, etc...
    1. There will *likely* be errors reported in the DRC. This doen't mean you can't move on; just read all the errors and make sure they are intended mistakes.
35. The design is finished! Save the schematic and move on to the PCB Layout.