sparkfun ELECTRONICS

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

Laying out a PCB

Step-by-Step Guide to laying out the PCB for the FT232 USB to Serial Converter

1. Before starting the layout we need to create the board file. Open the schematic and press the 'Generate Board from Schematic' button located along the top of the schematic window.
2. A prompt will appear with the notification that a board doesn't exist and asking if it should be created. Click 'Yes.' In the new window you'll notice a thin white rectangle and 8 smaller parts. The white rectangle is the physical outline of the board, and the 8 parts correspond to the parts added in the schematic. All of the yellowish colored lines are called Airwires; these correspond to unrouted traces and will disappear as we route the board.
3. Start by redrawing the board shape. Use the 'Delete' tool (Shortcut Key F3) to delete the white rectangle. Each segment will have to be deleted individually.
4. Now redraw a more appropriately sized board. Select the 'Line' tool in the toolbar on the left. Starting at the origin of the board (the dotted crosshair), draw a rectangle that will be big enough to fit all 8 components of the board.
    1. The rectangle needs to be placed on the 'Dimension' layer. To do this, click the layer drop-down box at the top of the window and select 'Dimension.'

*Hint: You can also access the layers by clicking the middle mouse button

    1. While drawing the rectangle, the angle might not be set to 90 degrees; to change this just right-click while drawing the wire until the appropriate angle is shown.
    2. Press the escape key to stop drawing.
2. Click the layer drop-down box again. Notice at the top of the list there are a bunch of layers named 'Route X.' These are rarely used and end up causing a lot of annoyance. To get rid of them:
    1. Open the DRC
    2. Erase the parenthesis around 1*16 in the setup box of the Layers tab.
    3. Press 'Check.'
    4. Open the DRC again.
    5. Put the parenthesis back around 1*16 in the setup box of the Layers tab.
    6. Press 'Check' again.

    Now the erroneous layers are gone!

3. Before we start moving components around, it would be nice to see which component is which (though it might be obvious). Click the 'Layer' tool in the toolbar (this is different than the drop-down box), and scroll down to find the layer name tNames. Click the number 25 next to the name so that the number is highlighted in blue, this turns the names layer on. Press OK to confirm.
4. Ah, now things are more clear! The names on the parts in the PCB correspond to the names on the parts in the schematic. We can start moving things now. Start with the FT232 module. If we check the schematic we can see that the module is named 'IC1.' Use the 'Move' tool (Shortcut Key F7) to move this part to the center of the board.
5. Next move the USB connector onto the board, the USB connector is named X1. Notice the yellow outline around one end of the connector. This indicates the physical dimension of the part; the yellow part of the connector needs to stick off the end of the board so that a USB cable can be plugged in without running into the board. Move the USB connector onto the board but make sure that the yellow outline is hanging outside the boards edge.
6. As a side note, notice the bright white shapes around the parts. These are silk-screen markings that will be printed on the PCB. They are used to help identify how a part should be placed on a board when the board is populated.
7. Now move the 4 pin header onto the board. It should go on the opposite side of the board as the USB connector so that we can connect to the header while a USB cable is plugged into the board.
8. Move the rest of the components onto the board one at a time. Make sure none of the 'pads' are touching one another. Try placing the components so that the airwires are as short as possible. A part can be rotated while being moved by right-clicking. Try positioning the parts to keep the airwires from getting tangled up; keep the airwires as short as possible.
9. Often times when parts are moved the airwires need to be recalculated. To do this the Ratsnest command must be used. Press F8, or click the Ratsnest icon on the toolbar.
10. Before routing the board, add a ground plane. To do this, select the 'Polygon' tool from the toolbar (make sure not to use the rectangle tool). Next, select the 'Top' layer from the layer drop-down menu (or middle click and select the layer). Draw a rectangle around the board.
11. Repeat step 13, but place the rectangle on the bottom layer.
12. Now rename the two rectangles so that they are connected to ground. Use the 'Name' command (Shortcut Key 'Alt + F5') to rename the red and blue rectangles as GND.
13. Run the Ratsnest command again. Whoah! The whole board changed color. This is because the rectangles named ground fill in and connect to any pin in the design with the same name.

14. Now we're ready to start routing the board. In the schematic we created 'Nets' to connect pins; in the board there is an airwire everywhere where a corresponding net is in the schematic. To connect the pins in the board we must create a *trace* that connects the pins. Connecting pins along an airwire will cause the airwire to disappear; however the airwire can be used as a guide while routing the trace to find the destination. Here are some general guidelines for routing:
    1. Start with the shortest airwires and progress to the longest ones.
    2. Before starting a trace, use the 'Show' tool and click on the airwire you're attempting to trace. This will highlight the airwire and the pins that need to be connected. This will help you plan a path for the route.
    3. Never have two routes from different signals intersect.
15. Select the 'Route' tool (Shortcut Key F9) and click on a pin to route. Drag the trace to the another highlighted pin. If the trace is not bending at 45 degree angles, right-click until it is. Finish the trace by clicking again. The trace will automatically end when connected properly. Here are some notes to help while routing:
    1. A trace can be placed on either the Top or the Bottom layer, but since all of the components are on the top a trace must go from the top to the bottom, then back to the top if this is to happen. To go from the Top to the Bottom with a trace, press the middle button while routing the trace to create a *via.* A via is like a tunnel for the trace.
    2. Traces on the top are red, traces on the bottom are blue.
    3. If airwires aren't disappearing when you think they should, try running the Ratsnest again.
    4. Notice that right after the Ratsnest is run, the number of remaining airwires is shown in the bottom left corner of the screen.
    5. The USB and 4 pin header have holes instead of pins. A hole can connect to a trace from the top or the bottom; a pin, like on the capacitors and FT232, can only connect on the top.
    6. Don't overlap any pins with a trace(except the ones being routed to).
    7. Hold the 'Alt' key while running a trace to route on a finer grid.
    8. The 'Delete' command won't get rid of a trace. Instead, if you need to re-route a trace you must first 'Ripup' the original trace. Use the shortcut 'Alt+F9' to select the 'Ripup' tool.
    9. Use the scroll wheel to zoom in and out while routing.
    10. Press and hold the middle mouse button while routing to pan around the board without modifying the trace.
    11. When you think you've routed all of the traces, run the ratsnest. Make sure the status bar at the bottom left of the window reads: “Ratsnest: Nothing to do!”

16. When the board is done being routed, there's still some work to do. First off, what the heck is this thing? Once you start getting multiple boards out at the same time, it's easy to forget which board is which when they start arriving. We need to add labels. It's also very smart to label any pins or switches on the board to make it easier to connect to. If you fail to label the pins, you'll find you have to refer back to your schematic much more often.
    1. Add a title to the board. Use the Text tool from the toolbar and label the board (USB to Serial Converter). Place the text on the tPlace layer.
    2. Is the text too big? Go to the 'Change' tool in the toolbar, then go to 'size' and select a smaller size. Try not to go smaller than .04 with the text.
    3. Label each pin of the 4 pin header. The labels should be VCC, GND, Tx and Rx. Make sure to label them correctly.
17. Add a version number somewhere as well (v10). This helps identify the board in case multiple version are made with very minor changes.
18. Before we call the design complete we must run the DRC, or Design Rule Check. Click the DRC tool.
    1. Click 'load' and then select the SparkFun DRC. This will ensure that the board can be built by SparkFun PCB manufacturer. Different PCB manufacturerers will have different DRC rules that must be followed.
    2. Click 'Check.'
    3. If there are any errors with the board layout, they will be listed. Some of them may be able to be ignored, this must be determined by the designer; but it's always good to be very aware of the PCB fabrication house rules.
19. The design is now finished! Next the gerber files need to be created. The gerbers are a collection of files the fabrication house uses to create the PCB. Start by clicking the 'Gerber' tool near the top of the window.
20. In the new window, go to File->Open->Job... and select sfe-gerb274x.cam.
21. Click “Process Job.” The gerbers are created and saved in the same directory as the project.
22. Before we're ready to send the gerbers off to the fabrication house we need to double check that the gerbers were created properly. We use a program called a 'Gerber Viewer' to do this. Open the Pentalogix ViewMate software.
    1. Go to file->import->gerber...
    2. Navigate to the directory where your project was stored and select all of the files with the extension .GXX. There are 7 files.
    3. Inspect the gerbers. You can turn different layers on and off to ensure everything is correct.
23. Once the gerbers have been verified the design is ready to be sent off. Usually this requires just zipping the the gerber and drill files into a folder and submitting them online. Typically just include all of the files except the schematic and board file.