

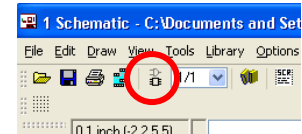
Task 8

Task Purpose: Design a Printed Circuit Board – Part 2

Due Date: December 6

In this second half, you are to lay out a board, route the traces, and submit the Gerber files for free DFM review.

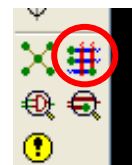
To create a board from your schematic, you need only click on the board icon at the top of the schematic. This will create a board with the same name as the schematic but a different suffix. On the board display, you will see a frame around the board area, and all of your parts, connected by what are called air wires, off to the side. To layout and route the board:



1. Size the board frame so it is no larger than 4 inches wide by 3.2 inches tall. This is the maximum size that the free version can route. I find it convenient to keep the + that indicates (0,0) at the bottom left corner of the frame.
2. Move the parts into the frame that represents the board area. In placing parts, you should attempt to minimize the length of the air wires, and the amount of crossing they do. Note:
 - You should place components that are connected together near each other. For example, the resonator for the FTDI chip should be located near the pins it connects to.
 - Make frequent use of the rats nest command. This causes the air wires to reroute, minimizing their length.
 - Decoupling capacitors should be placed as near to the device as possible.
 - You should rotate parts to minimize the crossing of air wires.



3. Once all of the parts have been placed, the next step is to route the connections. The autorouter will route the connections, changing air wires into traces on the board. Routing is usually an iterative process. When you click on the auto route icon, you will be presented with the Autorouter Setup screen.

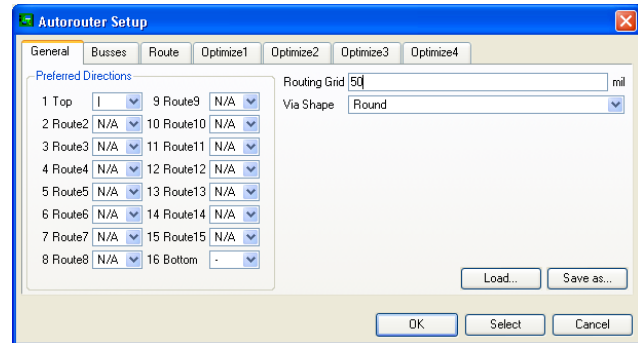


The left side of this gives the preferred direction for signals. The screen shows you the default direction for signals in each layer (you will only have the top and bottom layers in your design.)

On the right side, you will see that the default routing grid is set to 50 mils. The routing grid determines how finely the autorouter will attempt to place traces.

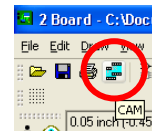
Routing time increases as the routing grid gets finer. My tendency is to use as coarse a grid as possible that achieves 100% routing.

When I ran the auto router with 50 mil grid on my layout, I was only able to successfully route 60% of the traces. When this happens, I remove the traces (use the group and rip-up tools), reduce the routing grid, and try again.



Many things can keep the autorouter from being successful: bad placement of parts, trace widths that are too wide (particularly with fine pitched parts), etc. If you are unable to successfully route the board with a routing grid of 10 mils, you should improve your parts placement on the board.

4. Once I have successfully routed the board, and am happy with the parts placement and the routes, the next step I take is to make the board more attractive and readable. The smash command lets you move labels on parts. You should put useful text labeling on the board, and I usually include a revision number. Indicating pin 1 of connectors and devices is useful also. You might also think about mounting holes and things like that.
5. The final step is to create what are called Gerber files that can be sent to a board house. These are files in a standard format from which the board house can make your board. You need to run two different CAM jobs.



The first CAM job is to run *excellon.cam*. This produces files that indicate where holes are drilled in your board. (You can see that the holes and drills layers are the only ones included in this job.) This CAM job will produce two files: *name.dri* and *name.drd*.

The second CAM job to run is called *gerb247x.cam*. Boards are described as having a component side (what you would think of as the top) and a solder side (what you would think of as the bottom.) This job produces files that describe the traces on the component and solder sides, the solder mask that is on the component and solder side, and the silk screen (the printing) that is on the component side. The solder mask, or solder stop, is a coating that keeps the solder from spreading across the board. Each of these sections has a different tab, which sets the layers and options.

Before running this job, you will need to make two changes in the setup. On the solder side, and on the solder stop mask SOL, you want to un-check the mirror box.

Once you have done this, you can process the CAM job, and it will produce the following set of files: *name.cmp*, *name.sol*, *name.sts*, *name.stc*, *name.plc*, and *name.gpi*.

All of the files produced by these two CAM jobs (a total of 8 files) should be put into a zip folder, and then submitted for DFM (Design for Manufacturability). (Note that the .dri file and the .gpi file are not absolutely necessary, but I never remember which two I don't need, so I send them all.)

Go to the site <http://www.4pcb.com/> , and click on the Free DFM link. It will ask for an e-mail address and then have you upload a folder with the zipped versions of the files produced by the CAM jobs. It will then give the screen below. Arrows indicate where I am not using the default:

Design File Information
Please take a few moments to help us identify your files and give us some general information.

Filename:	File Content:	
task7.cmp	Top Copper	View File
task7.drd	NC Drill	View File
task7.dri	Drawing/Other	View File
task7.gpi	Drawing/Other	View File
task7.plc	Top Silkscreen	View File
task7.sol	Bottom Copper	View File
task7.stc	Top Soldermask	View File
task7.sts	Bottom Soldermask	View File

General Information

* Required Field

Material Type: FR4 Prototype available only with FR4

Part Number: class

Revision: 1

Layers: 2

Finish Plating: Leaded Solder All prototypes upgraded using Lead Free HAL finish plating.

Copper Weight (outer, finished): 1 oz Prototype available only with 1 or 2 ounce

Gold Fingers: None

Solder Mask Sides: both sides

Solder Mask Color: Green Prototype available only with Green

Silkscreen Sides: top side

Silkscreen Color: White Prototype available only with White

CNC Route Points: 4

Material Thickness: 0.062" Prototype available only as .031", .062", .093" and .125"

Plated Slots: Not available on prototypes

Counter Sinks: Not available on prototypes

Counter Bores: Not available on prototypes

Plated Edges: Not available on prototypes

Array ?

X dimension: 4 in

Y dimension: 3.2 in

Tab-Rout:

Scoring:

Company Name: * Notre Dame

First Name: * Mike

Last Name: * Schafer

Phone: * 574-631-4851

Additional Email 1:

Additional Email 2:

Additional Email 3:

Submit

You should receive an e-mail response that indicates if your board is OK. It will tell you of any violations (I always get silkscreen width violations) and also give you links to plots of the various layers.

If your report comes back with no “showstoppers”, the board is makeable. (This does not imply that the board is correct. Any mistakes on the schematic will be faithfully represented on the board.

Task Report:

Your task report should include:

1. Eagle schematic file which includes the team name in the file name.
2. Eagle board file.
3. A copy of your library if you created any parts.
4. Results of the DFM.