Printed Circuit Board Design

ECE 362
https://engineering.purdue.edu/ee362/

© Rick
A Different View of the STM32F0

• You’re used to seeing logical block diagrams of the STM32, talking about its features, and programming it.

• Remember that it is a physical object.

• For a moment, we’re going to look at it from this perspective:
Physical Chip Package: LQFP64

- Details about the shape, leads, pins, heat profile, etc define the package.
- The package for our μC, the STM32F051R8T6, is LQFP64
  - Low-Profile Quad Flat Package
  - 10x10mm square
  - 64 pins. 0.5 mm lead pitch.
- Not all LQFP64 packages are the same.
- Where to find this information?...
1. Drawing is not to scale.

Table 67. LQFP64 package mechanical data

<table>
<thead>
<tr>
<th>Symbol</th>
<th>millimeters</th>
<th>inches(1)</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Min</td>
<td>Typ</td>
<td>Max</td>
<td>Min</td>
<td>Typ</td>
<td>Max</td>
</tr>
<tr>
<td>A</td>
<td>-</td>
<td>-</td>
<td>1.600</td>
<td>-</td>
<td>-</td>
<td>0.0630</td>
</tr>
<tr>
<td>A1</td>
<td>0.050</td>
<td>-</td>
<td>0.150</td>
<td>0.0020</td>
<td>-</td>
<td>0.0059</td>
</tr>
<tr>
<td>A2</td>
<td>1.350</td>
<td>1.400</td>
<td>1.450</td>
<td>0.0531</td>
<td>0.0551</td>
<td>0.0571</td>
</tr>
<tr>
<td>b</td>
<td>0.170</td>
<td>0.220</td>
<td>0.270</td>
<td>0.0067</td>
<td>0.0087</td>
<td>0.0106</td>
</tr>
<tr>
<td>c</td>
<td>0.090</td>
<td>-</td>
<td>0.200</td>
<td>0.0035</td>
<td>-</td>
<td>0.0079</td>
</tr>
<tr>
<td>D</td>
<td>-</td>
<td>12,000</td>
<td>-</td>
<td>-</td>
<td>0.4724</td>
<td>-</td>
</tr>
<tr>
<td>D1</td>
<td>-</td>
<td>10,000</td>
<td>-</td>
<td>-</td>
<td>0.3937</td>
<td>-</td>
</tr>
<tr>
<td>D3</td>
<td>-</td>
<td>7,500</td>
<td>-</td>
<td>-</td>
<td>0.2953</td>
<td>-</td>
</tr>
<tr>
<td>E</td>
<td>-</td>
<td>12,000</td>
<td>-</td>
<td>-</td>
<td>0.4724</td>
<td>-</td>
</tr>
<tr>
<td>E1</td>
<td>-</td>
<td>10,000</td>
<td>-</td>
<td>-</td>
<td>0.3937</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>millimeters</th>
<th>inches(1)</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Min</td>
<td>Typ</td>
<td>Max</td>
<td>Min</td>
<td>Typ</td>
<td>Max</td>
</tr>
<tr>
<td>E3</td>
<td>-</td>
<td>7,500</td>
<td>-</td>
<td>-</td>
<td>0.2953</td>
<td>-</td>
</tr>
<tr>
<td>e</td>
<td>-</td>
<td>0.500</td>
<td>-</td>
<td>-</td>
<td>0.0197</td>
<td>-</td>
</tr>
<tr>
<td>K</td>
<td>0°</td>
<td>3.5°</td>
<td>7°</td>
<td>0°</td>
<td>3.5°</td>
<td>7°</td>
</tr>
<tr>
<td>L</td>
<td>0.450</td>
<td>0.600</td>
<td>0.750</td>
<td>0.0177</td>
<td>0.0236</td>
<td>0.0295</td>
</tr>
<tr>
<td>L1</td>
<td>-</td>
<td>1,000</td>
<td>-</td>
<td>-</td>
<td>0.0394</td>
<td>-</td>
</tr>
<tr>
<td>ccc</td>
<td>-</td>
<td>-</td>
<td>0.080</td>
<td>-</td>
<td>-</td>
<td>0.0031</td>
</tr>
</tbody>
</table>

1. Values in inches are converted from mm and rounded to 4 decimal digits.
The footprint refers to the arrangement of pads on a printed circuit board to which the pins of a package are soldered.
PCB design, briefly

- PCB design, from a high-level perspective, is simply specifying wires to connect the right pads of footprints to each other.

- Let’s design a simple PCB with a single STM32 chip, capacitors, and some header pins.
Using KiCad

- These examples use KiCad 5.0.2.
  - Examples depend on version 5 or higher.
  - Industrial strength PCB design software.
  - https://kicad-pcb.org/
  - Free to use. Unencumbered results.
  - Runs on Linux.
    - And other operating systems.
  - Installed on EE 069 systems.
Main interface

- Choose **new project**, and invoke **schematic editor**
Start with a schematic

- First create a schematic to symbolically connect things together.
  - The symbols don’t have to look anything like the physical packages.
  - What are we going to connect?
    - Look at the STM32F0Discovery User Manual to see what’s necessary to build an STM32 system.
      - Figure 15 is the programmer.
      - Figure 16 is the STM32.
Necessary Connections

- \( V_{DD} \) (3V)
- \( V_{SS} \) (0V)
- NRST button, R/C circuit
- Pull BOOT0 low.
- Programming pins:
  - PB3 (55), PA13 (46), PA14 (49), NRST (7)
- Decoupling capacitors
Place a part

- In the schematic editor, place a symbol.
  - Click in the page.
  - Select the part.
Add other components

• In the search box of the symbol selector,
  - Type ‘R’ to add a resistor.
  - Type ‘C’ to add a capacitor.
  - Look for headers in the ‘conn’ library.

• Press the brown ground symbol button to add power ($V_{DD}$) or ground ($V_{SS}$) connections.

• Press the green wire button to draw wires between components.
Assign Values to Components

• Each component added starts off with:
  – An undefined reference designator
  – An undefined value

• For instance, a resistor would have a reference designator of "R?" and a value of "R".

• Change the value by right-clicking >> Properties >> Edit Value. (Or just hover over the part and press ‘v’.)
Annotate the Schematic

- Reference designators are names for parts on the PCB such as:
  - R1 and R2 for resistors.
  - C1 and C2 for capacitors.
  - U1 for a chip.

- To update all unset reference designators, select the "Annotate Schematic Symbols"
PCB layout videos

- Digikey is pretty interested in KiCad, so they sponsored the development of several youtube videos.
  - Probably better than what I can tell you.
  - Look for a guy wearing a bowtie.
Electrical Rule Checks

- ERCs find problems with the schematic:
  - Things that were left unconnected.
    - Maybe you misspelled the name of a wire in two places, so you have two unconnected wires.
  - Things that were not supposed to be connected together that were anyways.
    - Maybe a signal output and a power pin?
- Not always useful unless you are careful with your schematic.
Assign PCB Footprints

• When we selected a STM32F051R8Tx, the schematic editor knew that the only kind of package for the chip was an LQFP-64.

• Sometimes there are many different packages for a particular part.
  – For instance, a capacitor: surface mount, ceramic, electrolytic, axial leads, radial leads, etc.
  – PCB layout representation of the component including copper, silkscreen, documentation, etc. is called the Footprint.
  – Footprint assignment is where we pick one.

• Choose the icon to "Assign PCB Footprints to Schematic Symbols"
Generate a Netlist

- A netlist represents a set of footprints, pins (pads), and extra text to go along with them.
- It is list of physical connections between multiple pins rather than a high-level logical view of the schematic.
- Choose the "Generate netlist" icon.
PCB Layout

• Once the schematic is entered, footprints assigned, and the netlist is generated, THEN we can begin the PCB layout step.

• This is a long process and there is more to say than I can tell you this semester.
Step 1: Import a Netlist

- We start with a blank page.
  - Lots of things we could configure.
  - We’ll take the default options here. (2-layer board)
- Import the netlist and build a "rats nest".
- Spread out the components with the ‘M’ key to move and ‘R’ key to rotate them.
Step 2: Make a bounding box

- Sometimes, you want to aim for a particular board size. (Maybe you’ve found a PCB fab that has a size restriction.)
  - Select the "Add graphical lines" icon.
    - Draw a rectangle.
    - Use the ‘E’ key to edit the placement.
    - Align if needed.
  - If you use the Edge.Cuts "layer" for these graphical lines, it will define the edges on which the fab will cut your board.
- Complete physical placement.
Step 3: Create traces

- Most PCB editors have "autorouters."
  - They are all terrible. Do not use them.
- Select the "Route Tracks" icon.
  - Manually lay out each trace.
- Insert a "via" with the ‘V’ key.
- Use the Route => Interactive Router Settings to change how routing assistance works.
Step 4: Create Fill Zones

- Make all connections *except* ground.
- Replace the large areas not occupied by traces by copper fills. Two reasons for this:
  - It creates large ground planes to help reduce noise.
  - It saves the fab from having to etch off more metal than they need to.
- Select the add filled zones icon and create two fill zones that span the entire board.
  - Select F.Cu layer for front and B.Cu layer for the back.
  - Connect both to GND/VSS.
Step 5: Design Rule Checks

• Did we miss anything?

• Every PCB vendor has design rules that must be satisfied for a PCB to be producable. e.g.:
  - Minimum thickness of traces.
  - Minimum distance between traces.
  - Minimum turn chamfer on traces.
  - Minimum thickness of pads.
  - Minimum distance between pads.
  - Minimum/Maximum hole sizes.
Step 6: View, Check, Export

• Use the 3-D viewer to look everything over. Make adjustments.
• Export "Gerber" files.
• Send to fab.
  – What fab?
Doing things out of order here...

- That’s the brief view of how to create a PCB.
  - Let’s look at some of the details of what they consist of.
Chosing a PCB vendor

- Try a comparison site:  https://pcbshopper.com/
- Lab staff recommendations:
  - 4pcb.com: 2-layer, 60in², $33 each
  - 4pcb.com: 4-layer, 30in², $66 each
  - SeeedStudio.com: 2-layer, 100cm², 10 boards, $5.
Find PCB Vendor’s Design Rules

- Also called capabilities.
  - e.g. 4pcb.com capabilities → tolerances
Sending your design

- The copper trace information and drill files are commonly known as Gerber files.
  - Industry standard (RS-274-X)
  - View these files with a Gerber viewer
  - One file for each layer of the board + drills
Timeframe and Cost

- The Turn Time (turn) is determined by how much you’re willing to pay as well as the presence of special features.
  - 4pcb is domestic, but 1-2-day shipping from Asia costs hundreds of dollars.
  - Custom board shapes or cuts increase the tooling time and cost.
  - Number of boards determines tooling cost.
    - First one is expensive.
    - The next million are inexpensive.
PCB architecture

- Silkscreen: the printing
- Soldermask: repels solder
- Copper: the conductor
- Substrate: the board

"Layers stacked like lasagna."

Source:
https://learn.sparkfun.com/tutorials/pcb-basics/all
Substrate: FR4 (usually)

• Fiberglass reinforced multi-functional epoxy
  – Very typical substrate material
  – Typically 1.6mm thick (0.063")

• Other substrate materials are possible
  – Plastic: Makes the PCB flexible
  – Aluminum: Used for high thermal conductivity
Conductor: Copper

• A PCB starts off as a substrate with a solid sheet of copper glued to it.
  − Sometimes only on one side: single-layer
  − Sometimes on both sides: 2-layer board

• Copper thickness:
  − 1 ounce per square foot
  − High power circuits may need 2 or 3 oz/sqft
  − 1 oz/sq = 35μm (.0014") thickness
Etching

• Copper is removed from the board either chemically or mechanically.
  – It helps the process, and produces a better result if you leave as much metal behind as possible.
    • Should be connected to something rather than just floating.
  – You can do chemical etching yourself.
    • But you need seriously dangerous chemicals
  – You can do mechanical etching in BIDC.
Solder Mask

- Keeps solder from sticking to things it shouldn’t
- Protects copper from corrosion
- Available in different colors

- Notice zones of metal not etched off.
- Notice the places to solder to are tinned.
Silkscreen

- Usually white text on top of the soldermask
- Used to let you know what things are
  - Logos, notes, etc
- e.g. STM32 has pin names silkscreened on
Multilayer boards

- Consist of multiple 2-layer boards glued together by an additional sandwich of FR4.
  - No soldermask or silkscreen on inside layers.
  - Many layers are possible.
  - Often used to create power/ground planes.
    - Entire layer dedicated to one thing.
    - Improves electrical noise characteristics.
PCB terminology

• Pin: a plated-through hole used to connect the terminal of a part
  - Hole is drilled after etching, and an electrochemical process deposits copper on the edges of the hole.
• Pad: the conductive surface around the pin
  - either for a pin hole or surface mount connection
• Trace: a wire connection
• Via: a plated-through hole used for signal routing
  - Blind via: one outer layer connected to an inner layer
  - Buried via: two inner layers connected
Fastening components

- Two major types:
  - Through-hole: Devices have pins that are put through holes and soldered on.
    - Pin+Pad takes significant area = low pin density = large PCB
    - More solder = Higher capacitance = Lower frequencies.
  - Surface mount: Devices have pins that rest on pads and are soldered on.
    - Less solder = Lower capacitance = Higher frequencies.
    - Extremely compact = more connections in small space = smaller board
    - Results in less costly designs
Examples of components

- STM32 chip is surface mount
- Crystal oscillators are thru-hole
  - Anything you use in a breadboard is through-hole.
Mini-Project PCB Guidelines

- Must be more than just a breakout board.
  - Must have active components.
  - Should not be powered by STM32 dev board.
    - Power from external power supply.
  - Power/ground traces should be as large as possible.
  - Through-hole is fine. Larger surface mount OK.
  - Provide space and mechanical support for connectors, heat sinks, standoffs, etc.
Mounting STM32 on PCB

- Use two 33-pin single row 0.1" header sockets.
  - 33-pin sockets are not common.
  - Take 36-pin sockets and chop them down.
  - Room under the STM32 for parts.
- Buy a STM32F051R8T6 LQFP64 chip and solder it to a PCB designed for it.
  - Follow the design of the STM32F0DISCOVERY.
General Recommendations

- Minimum trace/space width: 10-12 mil
  - What’s a "mil"? It’s a milli-inch (0.001 inch)
- Power/ground should be sized for current.
- Decoupling capacitors: as close to each IC as possible.
- Space and mechanical support for connectors, heat sinks, standoffs.
- Use test pads, pin headers, vias, etc for signal monitoring and debugging support.
What happens when a trace is too small?

- Nothing good.
Layout Guidelines

- Ground layout is the most important PCB layout design consideration. Most Electromagnetic Interference (EMI) problems can be resolved using practical and efficient grounding methods.
  - Noise can be coupled into other circuits by mutual inductance. (Long traces next to each other.)
  - Ground return paths can be an inductive influence.
Shared grounds can be a problem

• Imagine a heavy digital load on the same ground as a sensitive analog circuit.
  – Intrinsic resistance of traces will have a voltage drop proportional to current.
  – Varying current will cause noise.

Sort of like the ADC noise problem on the STM32F0Discovery.
Ground layout tips

- Separate digital logic and analog circuit grounds as far as possible.
- Use multiple parallel ground pathways.
  - Taken to an extreme, this is a ground plane.
- Reduce trace inductance: make traces short and wide.
- Reduce trace impedance: make traces wide.
- Minimize signal reflection by making 135-degree turns instead of 90-degree turns in traces.
  - This is called "chamfering" or a "bevel".
Example of trace bevel

- Notice how there are no right angles in traces.
- Note the presence of test pads.
- Seems like a lot of wasted space?
  - Maybe.
- I wish someone made a board with all the Port A connectors in one place, narrower pin row spacing, pins on the outside, and silk screen on the inside.
Decoupling capacitor placement

- EMI can be caused by a number of factors
  - Switching action of push-pull logic circuits cause bursts of current source or sink.

Use 0.1\(\mu\)F ceramic capacitors.

Above 15 MHz use 0.01\(\mu\)F ceramic capacitors.

Use large (+10\(\mu\)F) electrolytic "bulk" capacitors on power supply that can recharge the decoupling caps.
Sometimes noise is suppressed

• When using solderless breadboards in development, remember that any two rows has 1 – 20pF of capacitance.

• Helps to use a ground plane below breadboards with high-frequency designs.

• Common senior design traps.
Signal layout

- Much "noise" is generated by the clock and other high frequency digital signals.
- Any trace running in parallel with the clock will have an induced square wave in it proportional to the current in the clock trace.
  - Depends on capacitance of clock net as well as inductance of clock inputs.
  - Try to make sensitive signals (analog?) signals cross the clock at right angles.
Component layout

• Put parts with many shared connections close together.
  - The time taken to initially place parts is just as important as the time taken to lay down traces.
Packages

- Various packages available.
  - Some you’ll be able to solder yourself.

Solder these by hand.

“Seek help” for these.
What to put on the silkscreen

- Your team name and ID
- Names of your team members
- Date modified, revision number
- Component IDs (e.g. R1, R2, C1, etc)