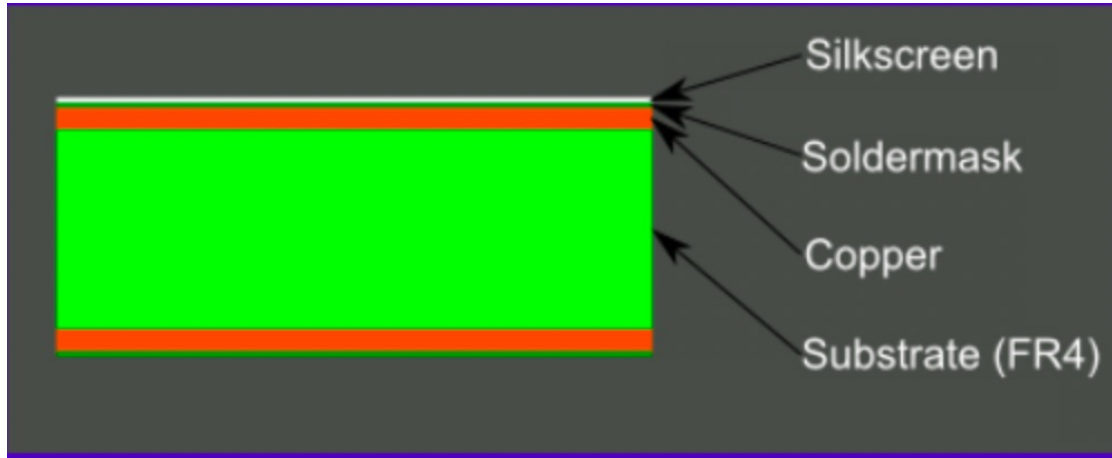


Module 4

Printed Circuit Boards

PCB architecture



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

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

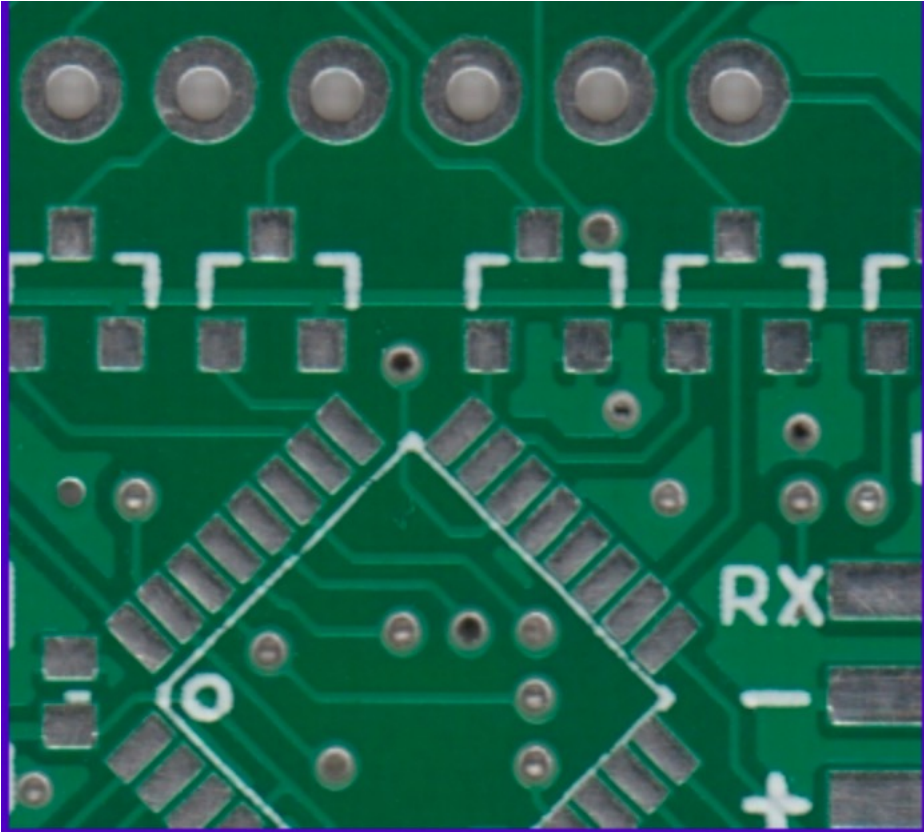
Conductor: Copper

- 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 = $35\mu\text{m}$ (.0014") thickness

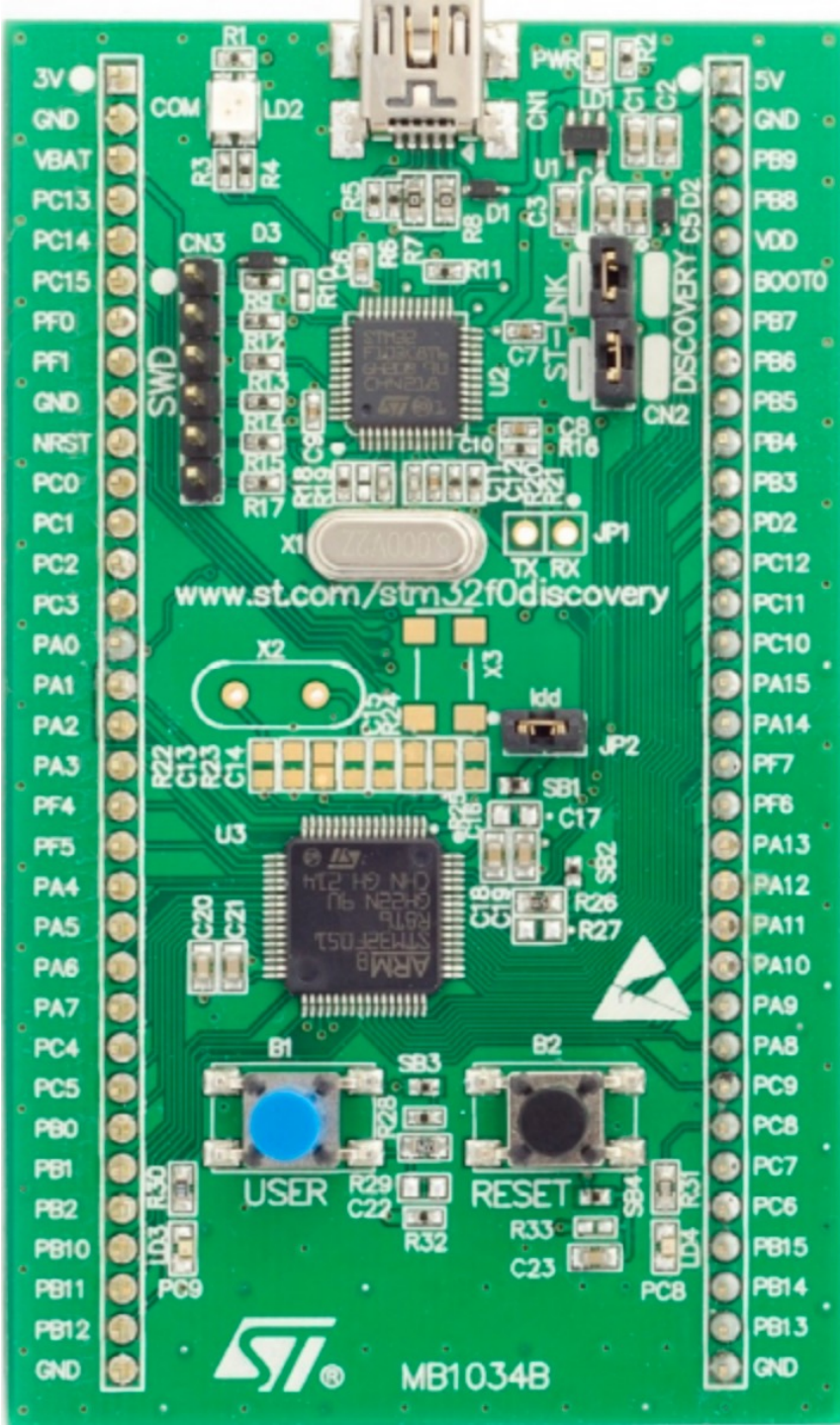
Etching

- 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.
 - Especially useful for ground and power planes.
 - 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

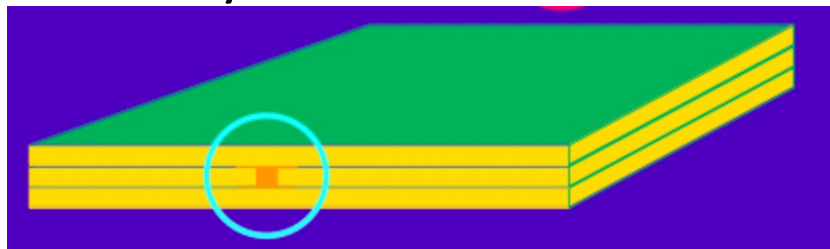
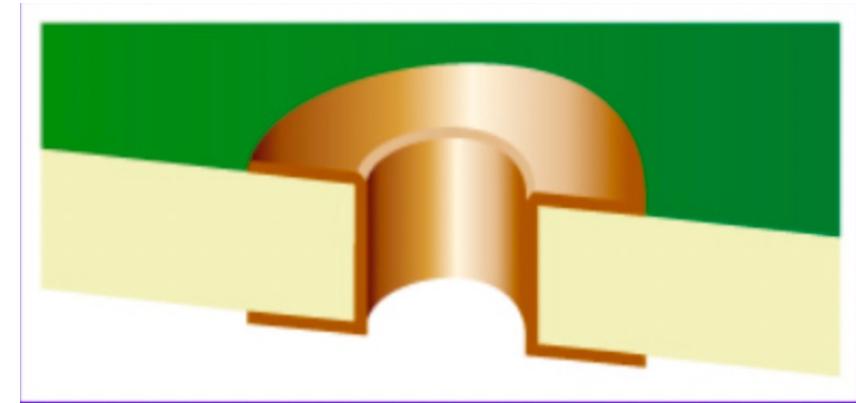
- Usually white text on top of the soldermask
- Used to let you know what things are
 - Logos, notes, etc
- e.g. STM32 has pin names silkscreened on

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
- When designing a PCB: Measure and double check component sizes

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 an 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.

Mount STM32 dev board on PCB

- Maybe you just want to put the dev board on another PCB.
 - You don't get as much bonus credit for this.
 - Use two 30-pin single row 0.1" header sockets.
 - 30-pin sockets are not common.
 - Take 36-pin sockets and chop them down.
 - There may be room under the dev board for parts.
- Maybe you want to design your own PCB on which you solder an STM32F091RCT6 LQFP64 chip.
 - You get the maximum PCB bonus credit for this.
 - Follow the design of the STM32 dev board.
 - Many things to check for this...

Designing a PCB for STM32 chip

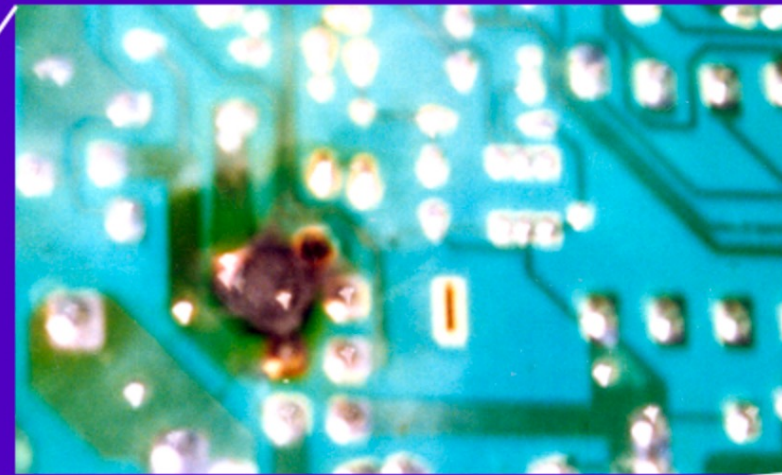
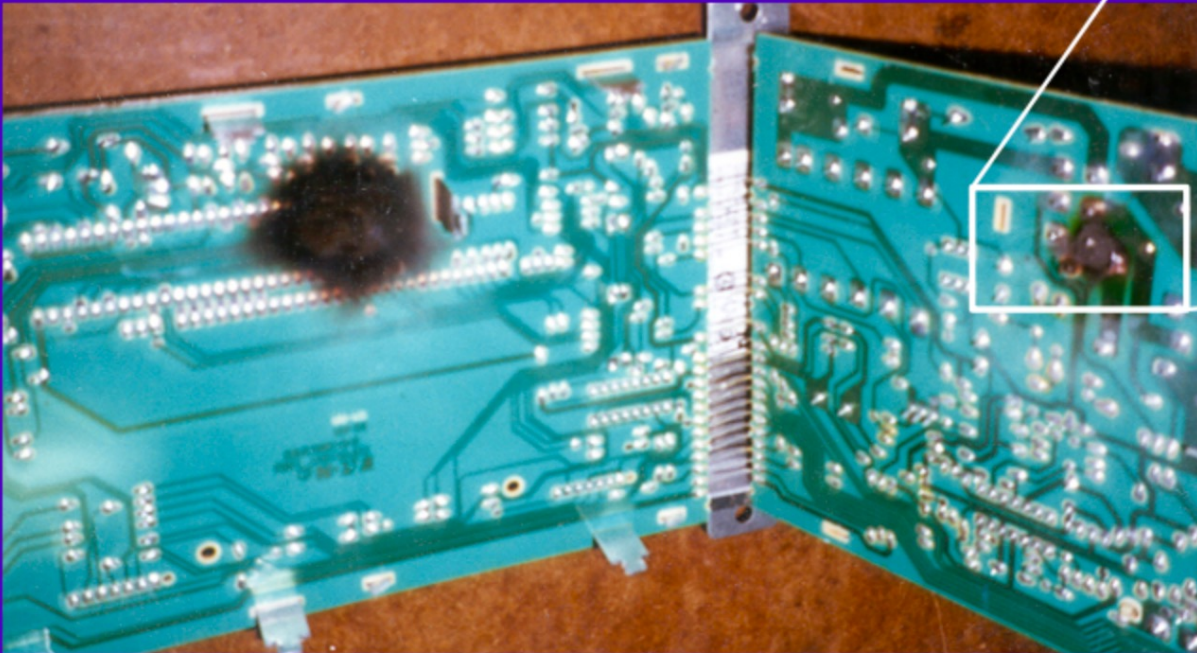
- Most important:
 - Connect all VDD as well as VDDA to 3V.
 - Connect all VSS as well as VSSA to 0V.
- Put a programming header on your board

General Recommendations

- Minimum trace/space width for power: 10-12 mil
 - What's a "mil"? It's a milli-inch (0.001 inch)
- Power/ground should be sized for current.
 - <https://www.4pcb.com/trace-width-calculator.html>
- 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.

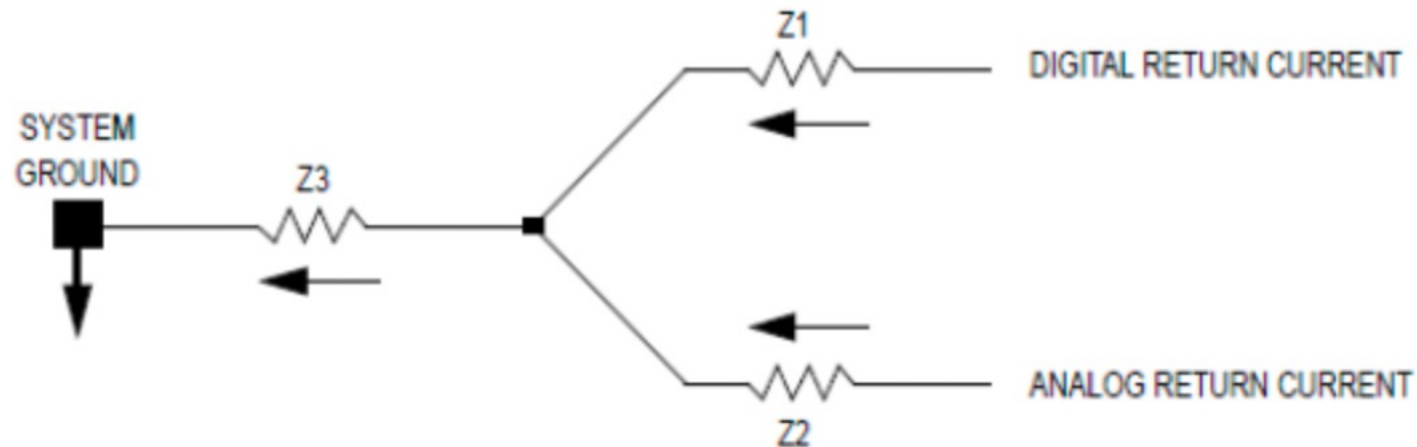


Layout Guidelines

- 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.

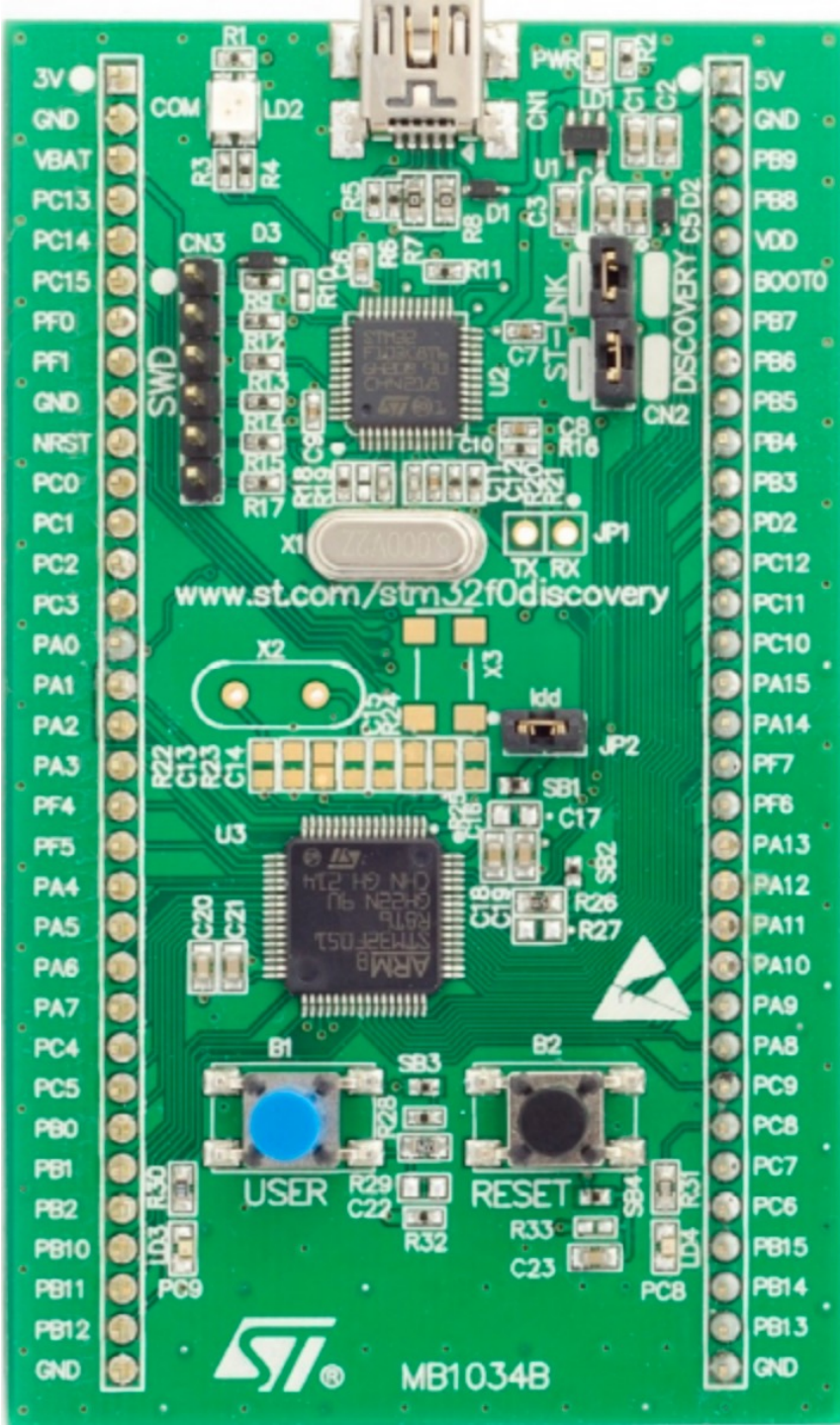


Ground layout tips

- 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".

Ground/Power planes

- Any decent PCB layout tool can create what are known as "copper fills" or "copper pours"
 - These take up all copper zones unused for traces
- Can connect each one to ground or one to ground and the other to power.
- Connect any "islands" together with traces.

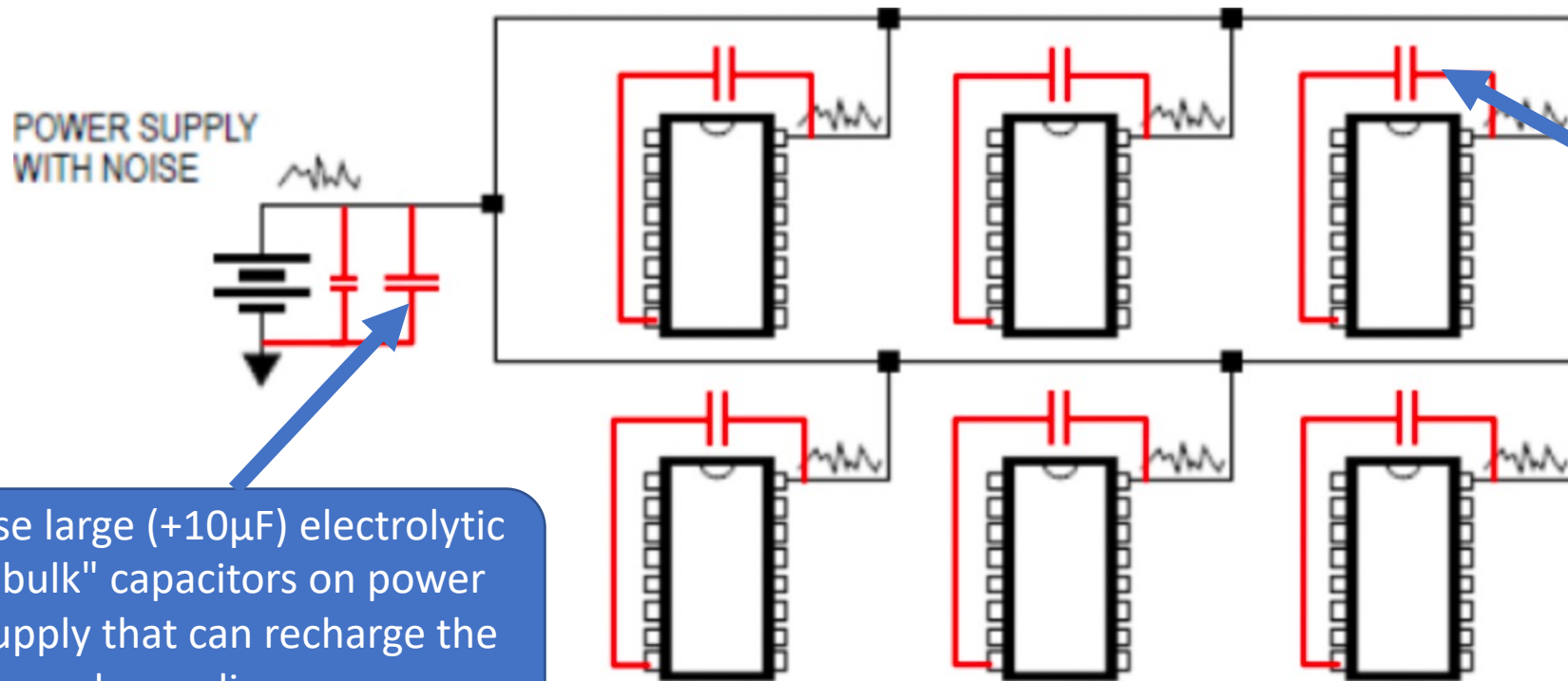


Example of trace bevel

- 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 large (+10 μ F) electrolytic "bulk" capacitors on power supply that can recharge the decoupling caps.

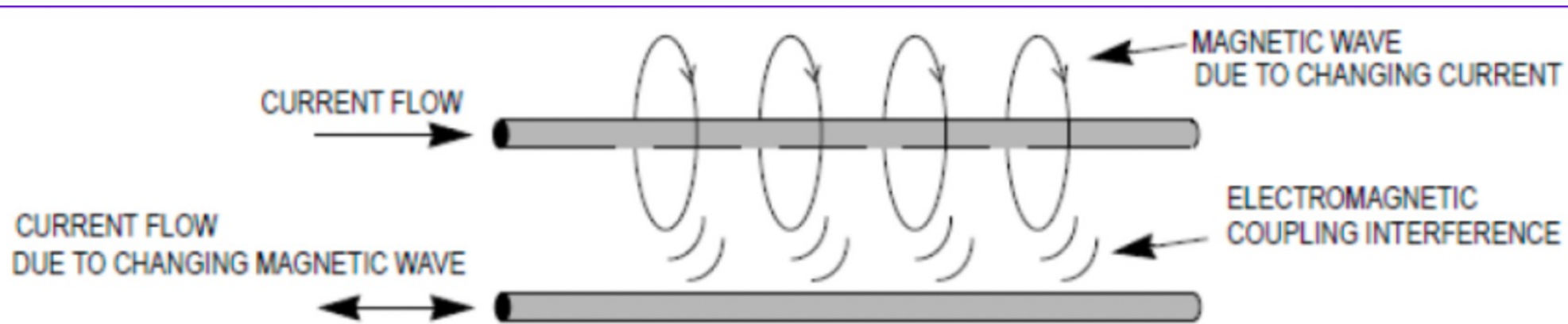
Use 0.1 μ F ceramic capacitors.
Above 15 MHz use 0.01 μ F ceramic

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.



Dual Inline Package
(DIP)



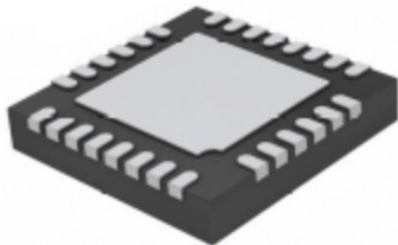
Small Outline Integrated
Circuit (SOIC)



Shrink Small-Outline
Package (SSOP)



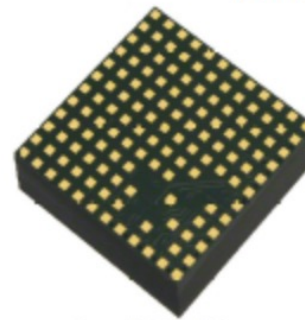
Quad Flat Pack
(QFP)



Quad Flat, No-Leads
(QFN)



Ball Grid Array
(BGA)



Land Grid Array
(LGA)

Solder these by
hand

"Seek help" for these.
(More likely, you
should simply avoid.)

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)

Soldering Parts

- An LQFP64 can be soldered by hand.
 - Use lots of flux and "drag solder" with a large iron.
 - Lead solder works better for this than lead-free.

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 type for our μ C, the STM32F091RCT6, 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?...

