PCB design in brief

Lars Svensson (larssv@chalmers.se)

What & why?

- PCB: “Printed Circuit Board”
- Electrical and mechanical substrate for off-the-shelf and purpose-designed parts
    - Packages affect PCB choices
- PCBs affect most important system parameters / costs !

MKM105

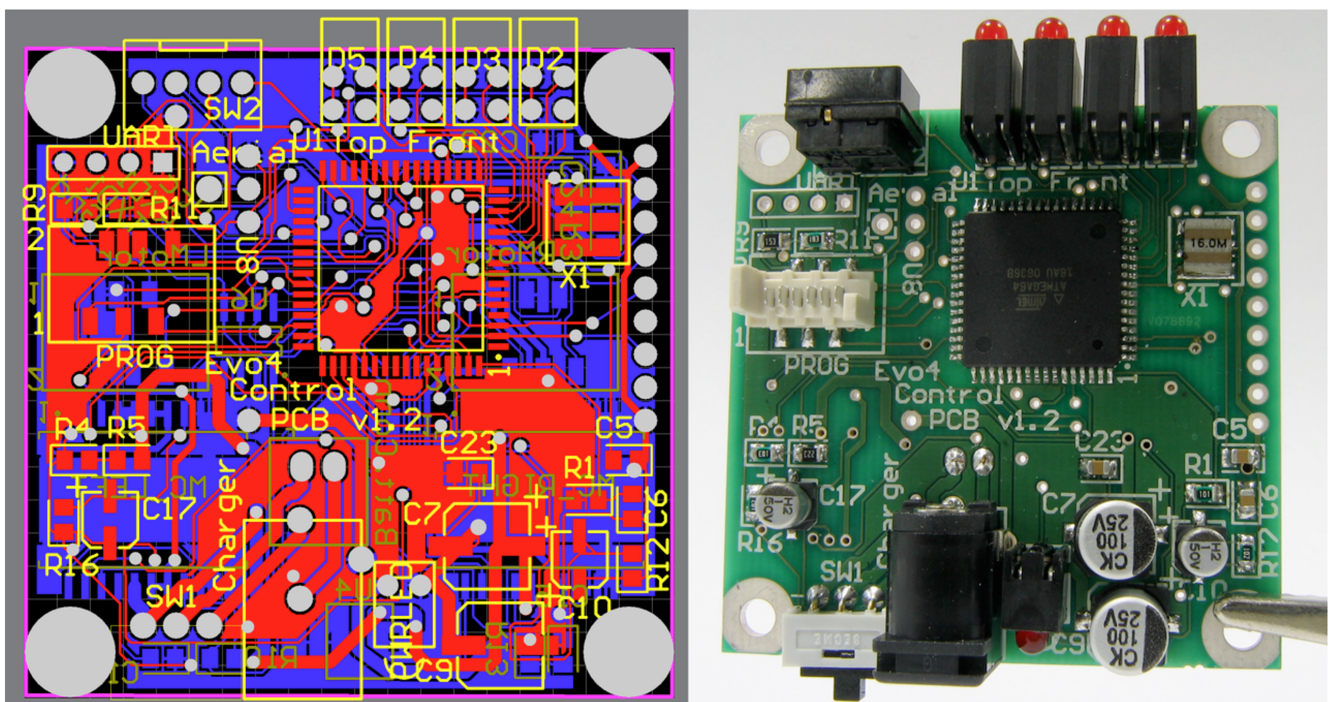
Overview

- General PCB discussion
- Signal propagation on PCBs
- Practical PCB design

- Refer to Mitzner: “Complete PCB design using OrCAD Capture and PCB editor” (available as eBook in library)
    - Reading directions in PingPong

Physical construction

- Sandwich of alternate layers of conductors and isolators
  - Conductor layers patterned to create wires
  - Vertical connections between conductors
- Components mounted on one or both sides



[Wikipedia]

Requirements

- System level requirements:
  - electrical
  - mechanical
  - thermal
  - cost
  - …
- PCBs influence most of these!

Electrical requirements

- System level: processing speed
  - PCB level: signal delay
  - PCB level: signal bandwidth
- System level: power requirement and delivery
  - PCB level: Supply current capacity

Mechanical requirements

- System level: support weight of components/parts
  - PCB level: board may have to be stiff and unyielding
- System level: survive certain abuse
  - PCB level: board and connections must not be too brittle

Thermal requirements

- System level: handle dissipated power
  - PCB level: help evacuate heat from components
  - PCB level: survive thermal expansion & contraction
  - PCB level: avoid metal fatigue in solder joints
  - …

Cost requirements

- System level: Cheap, cheap, cheap!
  - PCB level: cost of PCB itself (more layers means more expensive)
  - PCB level: cost of populating PCB
  - PCB level: component package compatibility
  - PCB level: cost of environmental compliance

Common materials

- Conductors: almost always copper
  - Thicker is electrically better, at a price
- Carriers: several alternatives; examples:
  - FR4, glass-reinforced epoxy (default choice)
  - Teflon (high frequencies)
  - Thin polymers (flexible/bendable)
  - Sometimes aluminium (durability, heat evacuation)

Fabrication

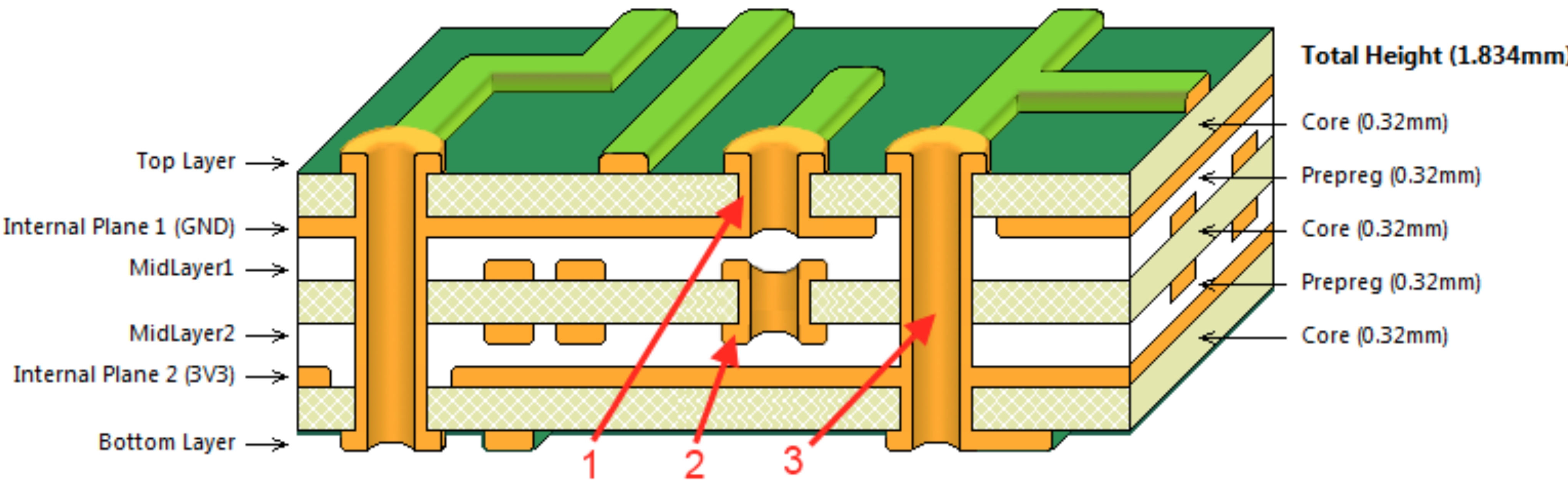
- Typical steps:
    - Construct / make bare PCB
    - Populate PCB with components
    - Solder components to PCB

- Variations for all of these!
- Here: most common cases only

1. Construction

- Start from copper-clad FR4 boards ("cores")
- Define conductor patterns ("traces") by photolithography
- Etch away rest of copper
- Optionally sandwich several boards w/ epoxy separators ("prepreg"); bake to set epoxy
- Drill holes through board, copperplate inside of holes to connect layers

Example end result



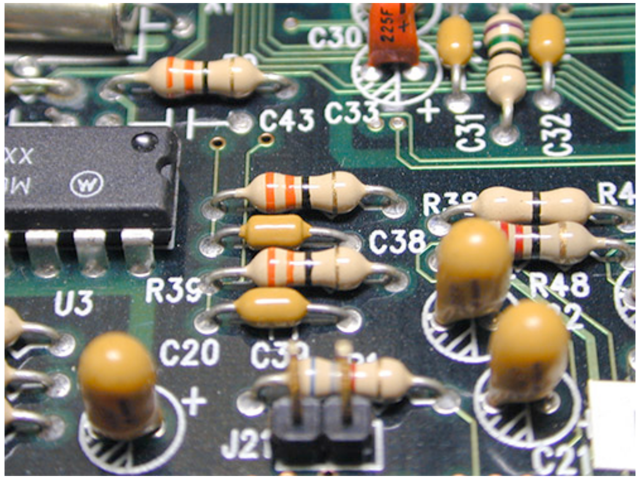
1. Blind "via" (vertical connection)
2. Buried via
3. Through-hole via

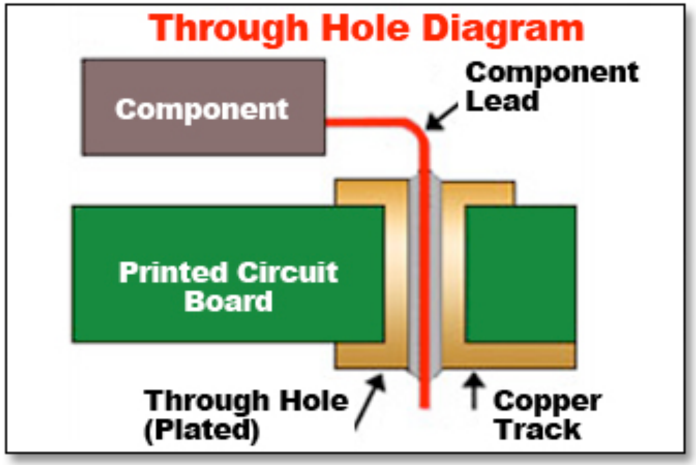
[techdocs.altium.com]

2. Populate

- Place / mount components on board for subsequent electrical connection to traces
- Two main mounting styles:
  - Through-hole
  - Surface mount

Through-hole mounting





- Component pins fitted through holes in PCB
  - Pin + hole provides alignment
- Fixed and connected with solder, traditionally applied from PCB back side
  - Mechanical + electrical + thermal connection

[www.motionxcorp.com]

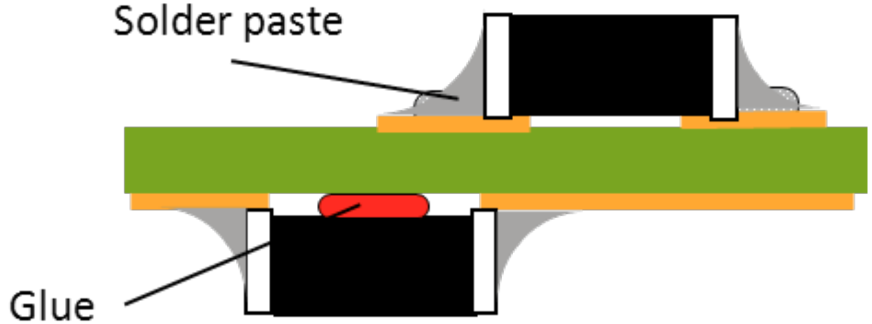
[www.amselectronics.com]

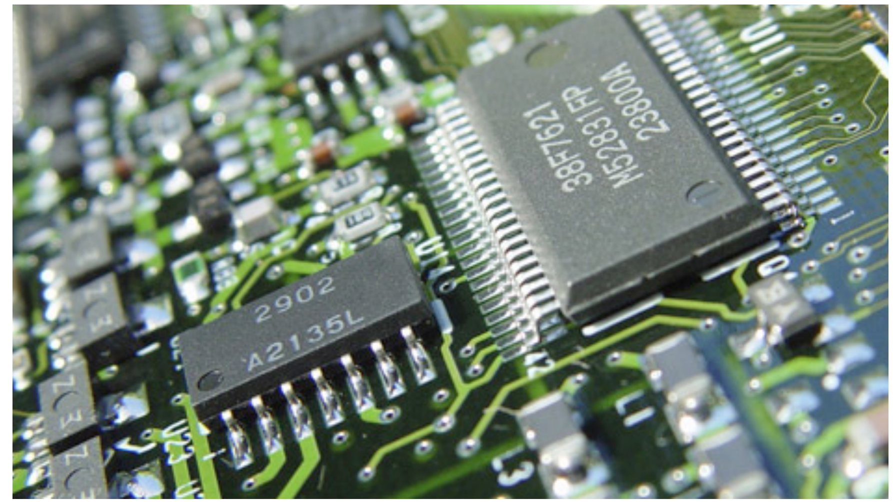
Through-hole + and –

+ Hand-tool (dis)assembly workable
+ Component leads offer some flexibility
– Through-holes occupy space on <u>all</u> PCB layers
    – Other wires must be routed around holes
– Drilling is expensive (serial process)

• Mainly used for prototypes or for bulky/heavy components

Surface mounting (SMT)





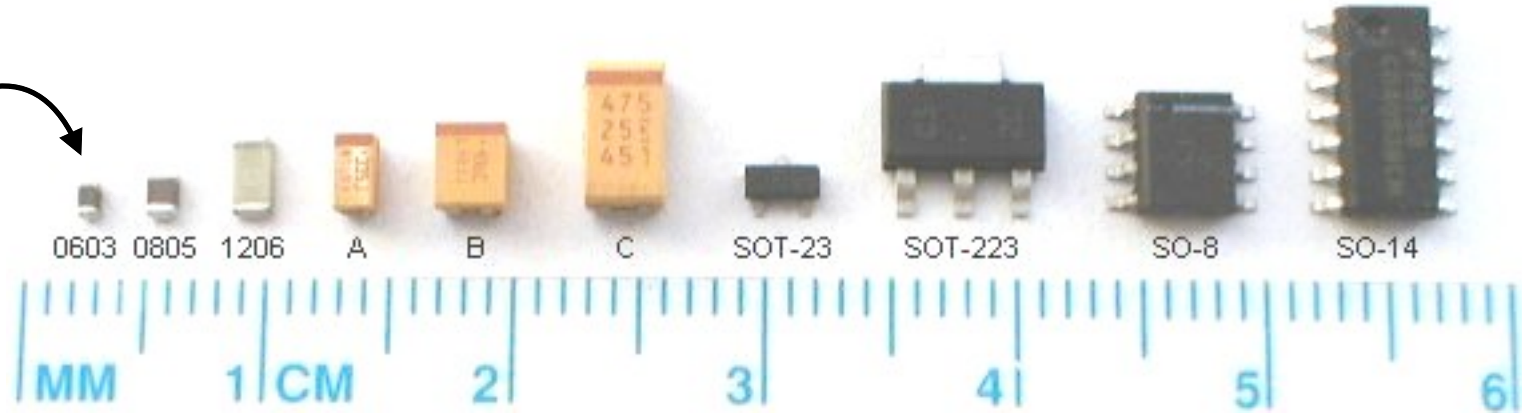
- Fix components to PCB with glue or solder paste
- Align component pins with PCB traces
  - Careful alignment w/o pins to help!

[www.powerguru.org]

[www.clarydon.com]

Surface mount + and –

60 x 30 mils



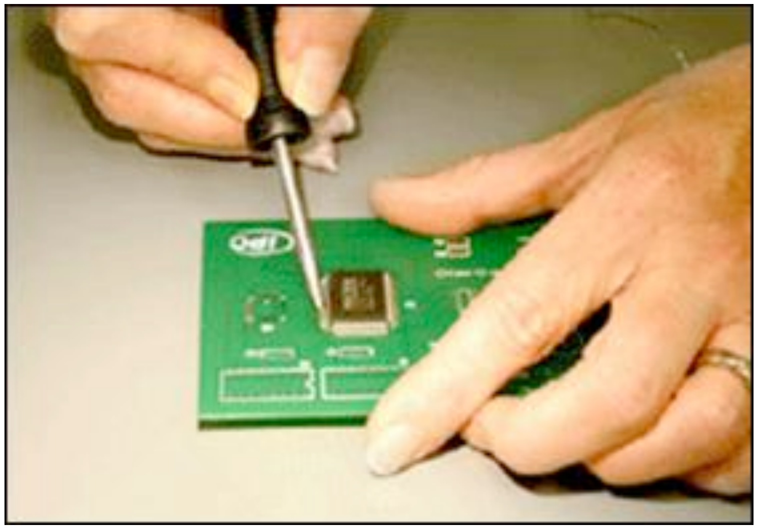
- \+ Smaller components, finer pitches possible
- \+ Both sides of PCB populatable w/o interference
- \+ Internal PCB layers undisturbed by components
- – May need more careful thermal design
- • The default choice today!

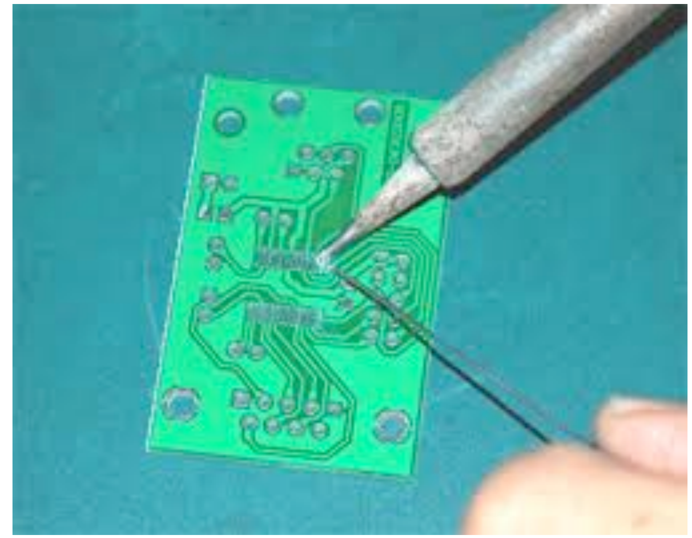
[fpga4fun.com]

3. Solder

- Electrically connect components to PCB traces by wetting with molten metal which is then allowed to cool and solidify
- Methods
  - Hand soldering
  - Wave soldering
  - Reflow soldering
- Solder material properties?

Hand soldering





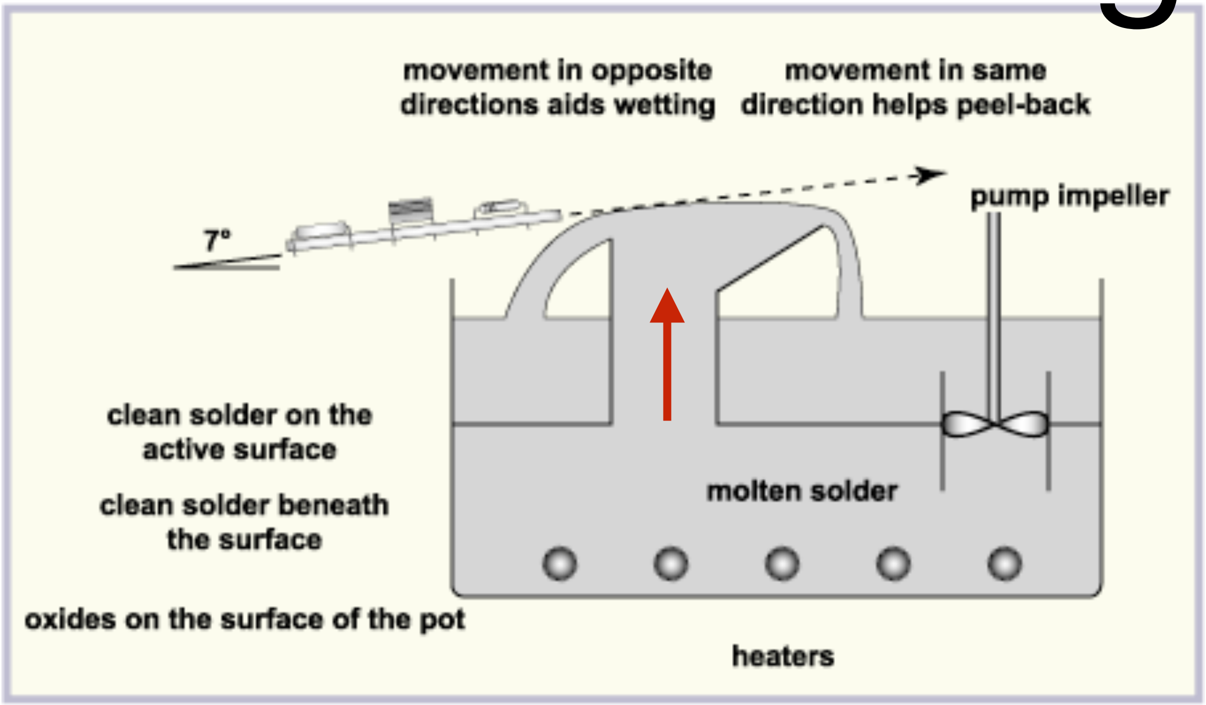
- Add solder material, add heat …

\+ Zero setup time, cheap equipment

– Time-consuming, error-prone, difficult at small geometries, impossible for some packages

- Useful for prototyping, rework

Wave soldering



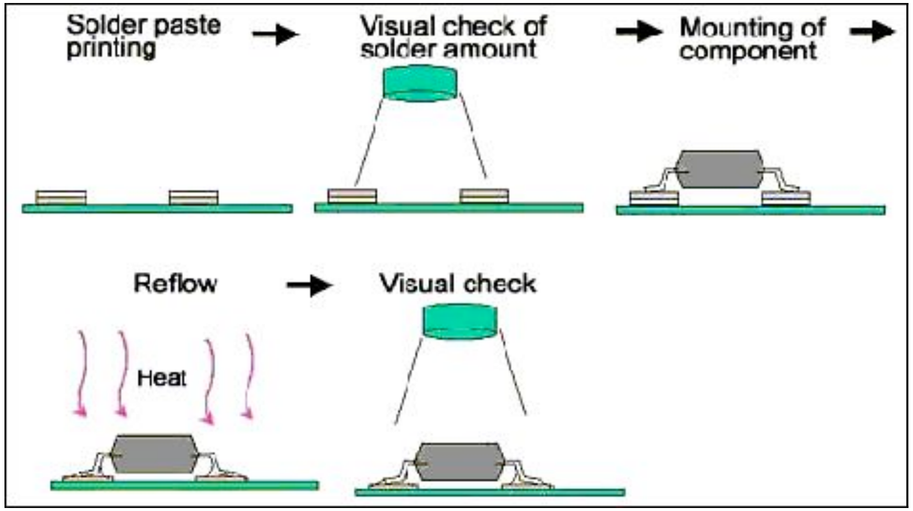
- Wave of molten solder metal “licks” one side of PCB

+ Very fast, so good for mass production

– Mainly single-side, through-hole components

[fixtrade.nl]

Reflow soldering



- Screen-print solder paste, place (and optionally glue) components on PCB
- Heat by IR lamps or by hot air

\+ Compatible with dense surface-mount PCBs

– Complex setup, expensive equipment

[www.eichenzell.com]

Solder materials

- Metal alloys, mostly based on tin (Sn)
  - Low melting point (<≈ 200°C)
  - Good “wetting” of copper conductors
  - Not brittle when solid
  - Stable over time
- Previously large percentage (e.g. 37%) of lead (Pb)
  - Outlawed for environmental reasons (“ROHS Directive”; see Wikipedia!)
  - Now other additives (Cu, Ag, Bi, Zn, …)

Signal propagation on PCBs

Signal propagation

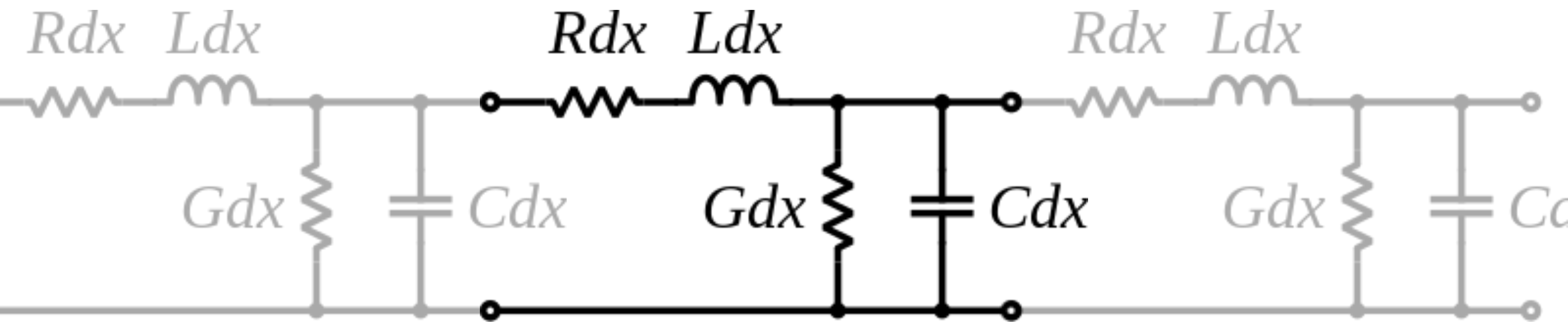
- Simple case: voltage change travels from near to far end of a wire
- Distance involved, so not instantaneous (speed-of-light!)
- From dimensions and materials, signal propagation in general may be calculated with 3D field solvers
  - Very extensive calculations :-(
  - To determine speed, simplified models often satisfactory

Delays significant?

- Compare distance with “fastest signal feature”
  - If distance smaller, no worries
- Ex: 1-GHz clock signal
  - Period: 1 ns
  - Rise/fall times ~ 100 ps
  - Corresponds to ~ 3 cm, in vacuum
  - … but speed-of-light in FR4 (etc) lower, so ~ 1? cm

What is the wire speed?

Uniform-wire speed



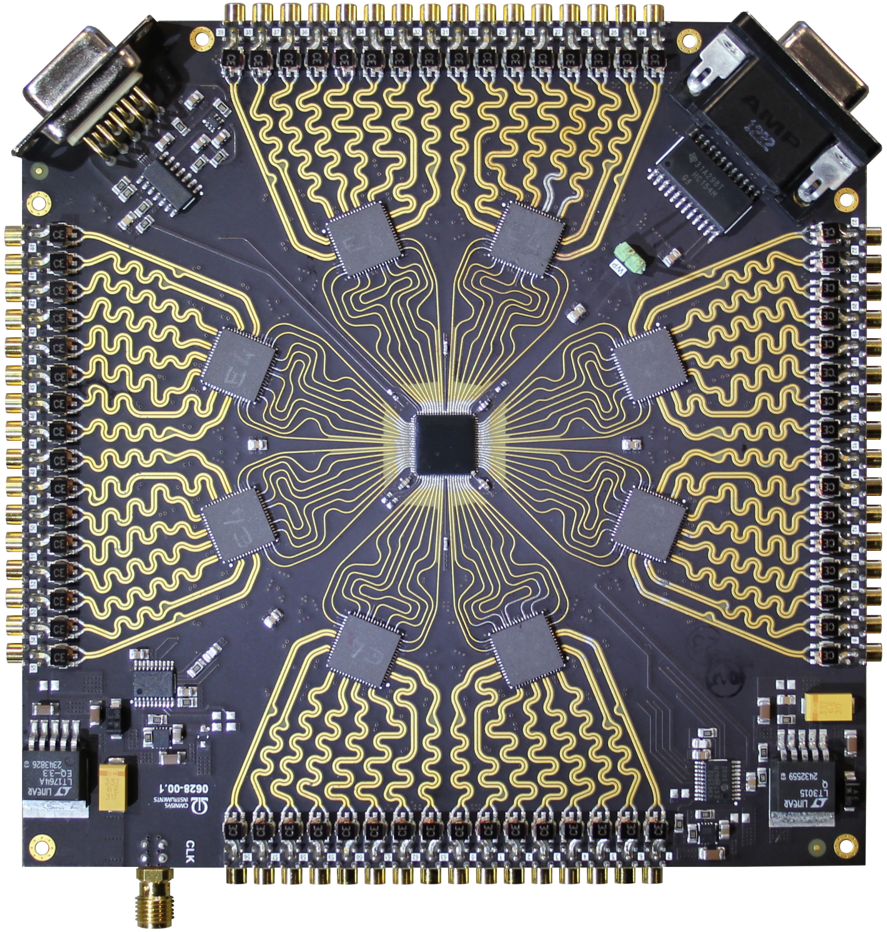
- Model wire/ground with L, C, R, G per unit length
    - Telegrapher's equation determines voltage waveforms at all points, given boundary conditions
- In PCB traces, G may be ignored, and sometimes R (at least to first order)
- Then, propagation speed is v = 1 / sqrt(LC)

≈ speed of light in surrounding medium

Why care about a few ps?

- For a critical digital signal, they may determine if your system works
    - Need to avoid setup + hold violations at far end
- Analog high-frequency signals may carry important information in signal phase

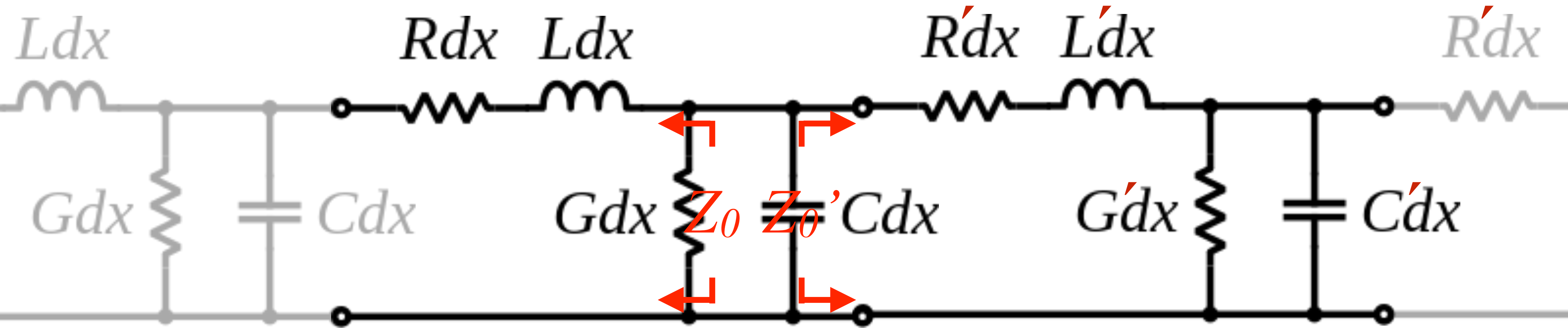
Length matching



- Extreme example for multi-GHz analog signals
- Ensure equal delay for all signals

 [Erik Ryman]

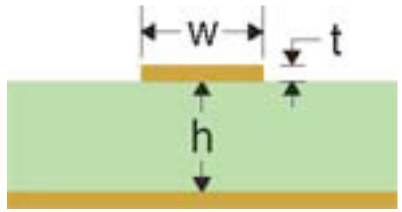
Reflections



- Uniform L, C cause constant wire speed v
- Discontinuity causes <u>reflections</u> unless
  L / C = L' / C'
- <u>Characteristic impedance</u>: Z_0 = sqrt(L/C)
    - Avoid reflections at end of line by matching Z_0

PCB trace impedance

Cross cut

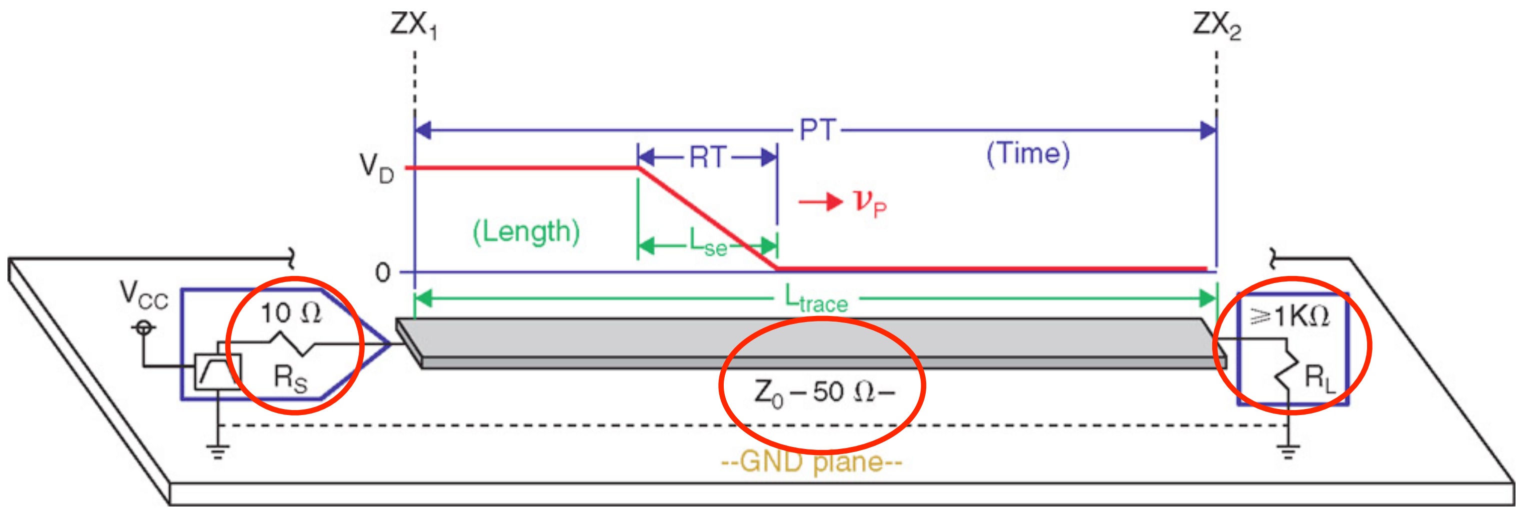


$$Z_0 = \frac{k}{\sqrt{\varepsilon_r + 1.41}} \ln\left(\frac{5.98h}{0.8w + t}\right)$$

- Z_0 depends (weakly!) on dimensions
    - "Microstrip" example (there are other styles, see Mitzner Table 6.6); note "ground plane"
- PCB designer can select *w* for each line, but *t* for entire board only
    - *h* may be selected in multi-layer boards…

[Mitzner]

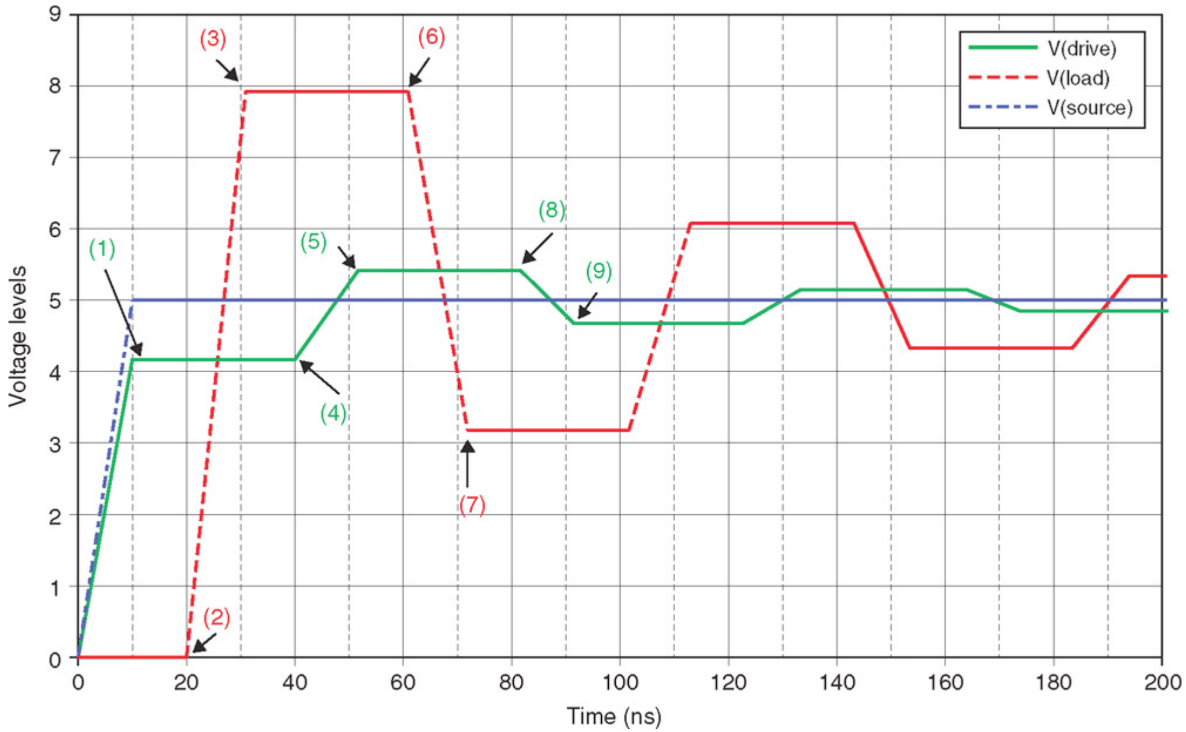
Reflections on microstrip



- Typical situation: $R_S < Z_0 << R_L$
- Reflection at far end causes wave travelling back towards driver
- Reflection at driver sends signal forwards…

[Mitzner]

Time-of-flight ringing



- Exponentially-damped ringing
- Period depends on distance
- Amplitude, decay rate depends on impedance mismatches

[Mitzner]

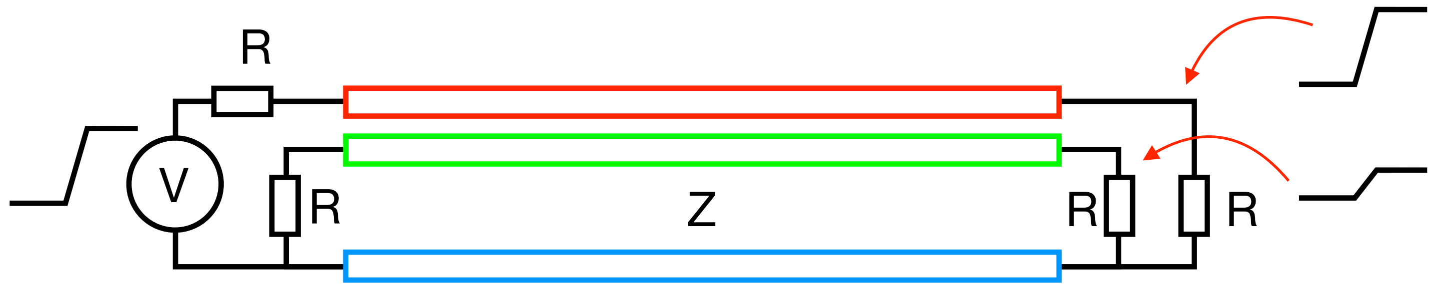
How avoid ToF ringing?

- Select $R_S \approx Z_0$ and/or $R_L \approx Z_0$
    - High-speed signal interface circuits often have <u>tunable</u> R_S and R_L
- Beware that received amplitude is reduced
    - Logic threshold levels may have to be adjusted accordingly

Final note on reflections

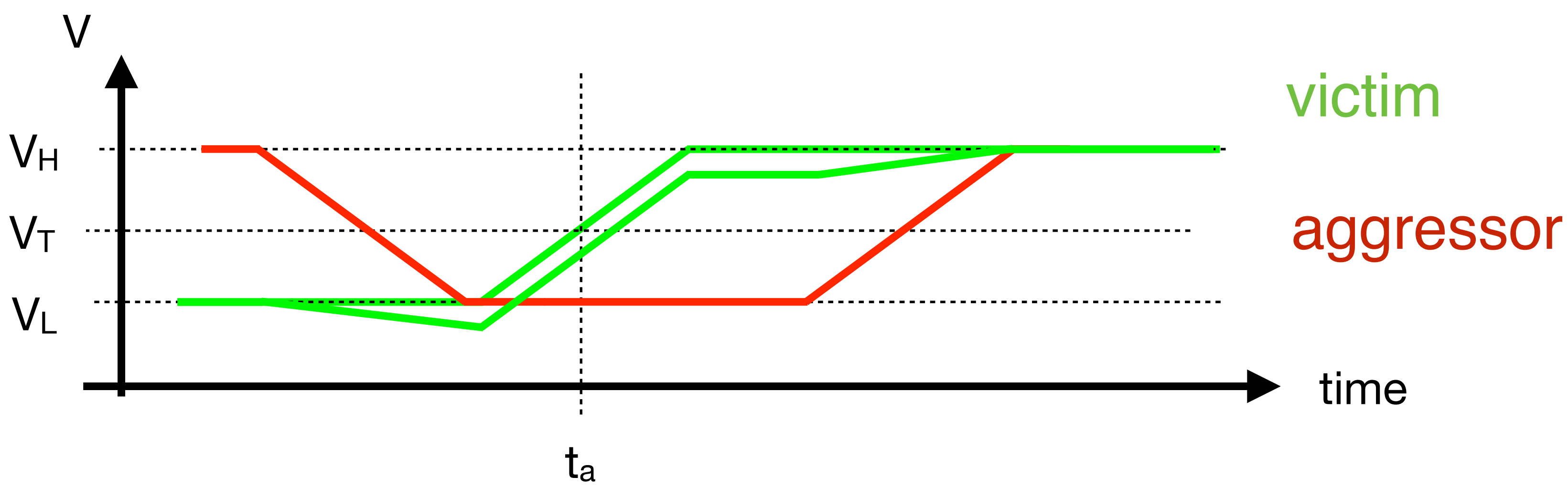
- Reflections not only at end of line!
- Any Z_0 variation causes reflection:
  - Conductor width change
  - Conductor direction change (corner)
  - Conductor fork
  - Ground plane irregularity…
- With care, may be kept small

Signal interactions



- Neighbouring signal traces may suffer from crosstalk
  - Inductive/capacitive coupling from aggressor to victim
- May cause timing and / or logic errors

Timing errors?



- Negative-going crosstalk delays positive-going edge!
  - Neg-neg etc. left as exercises

How reduce crosstalk?

- Reduce coupling from aggressor to victim
  - Space traces farther apart
  - Introduce grounded shield trace between aggressor and victim
- Dual-rail signalling
- All have an area cost!

Power supply connections

- “Dual” of signal propagation problem
  - Voltage to be kept stable even if boundary conditions change (e.g., current surge)
- Same field equations apply (Maxwell rules!)
- Typically, use full copper planes to distribute power (inner PCB layers?)
  - Telegrapher’s equation (one-dimensional) not directly applicable
  - “Full” plane: minus signal vias, other traces…

Several cards?

- When needed, segregate parts onto several cards
- What parts to aggregate on same card?
  - Many interconnections
  - Delay-critical interconnections
- Number of inter-card signals T often assumed to follow “Rent’s Rule”
  - $T \sim g^p$, $0.5 < p < 0.8$, g is # of gates

PCB connectors

- Enormous variety available
  - Component distributor Farnell lists 64980 board-to-board connectors!
- Differ in dimensions, electrical/ mechanical properties, plug/unplug resilience, temperature range, etc.
- Electrical drivers selected accordingly
  - Handle impedances, plugging, etc

Practical PCB design

Design

- CAD tools similar to “physical” parts of ASIC design
    - Allow manual placement of parts, connectors, etc
    - Automatic signal routing (check impedances, delays!)
    - Libraries of parts w/ pinouts

Fabrication

- Simple 1- and 2-layer boards: quick and cheap (may be mfg at home)
- More layers, smaller parts increase demands on alignment, machinery
- Example: http://pcbshopper.com/
- Populating and testing adds to cost!

Summary

- PCBs: maybe not glamorous, but essential for electronic systems
- Affect “all” system parameters, costs
- Viewed as somewhat of a black art? (See Johnson + Graham book titles)
- Ruled by Ohm, Kirchhoff, and Maxwell, just as other electronic designs
    - A bit more Maxwell due to sizes and frequencies

Summary, cont.

- Up to ~10 layers is common (at a cost)
- Trace width narrow enough to support components with ~0.3 mm lead pitch
- CAD support essential for non-trivial designs