Hardware Design for Embedded Systems

Embedded Systems Engineering WS10

Armin Wasicek
Overview

**Printed Circuit Boards (PCBs)**
Workflow for designing and manufacturing PCBs
„Ideal passive components“ and simplified ECD for a circuit path
Noise, shielding, pitfalls, etc.
Further references
Printed Circuit Boards (PCBs)

**Substrate** (e.g., epoxy and cotton paper, epoxy and woven glass) plated with **conducting layers** (e.g., copper)

- different substrate types exhibit different characteristics w.r.t. humidity absorption, thermal fluctuation, leakage current, HF properties, etc.
Breadboards and PCBs

A *breadboard* (protoboard) is a construction base for a *one-of-a-kind electronic circuit*

Initial costs (e.g. design cost) of PCBs are typically higher than the cost of breadboard constructions

- PCBs enable faster fabrication and assembly, better characteristics w.r.t. EMC, etc.
- careful design, in particular for EMC and HF properties, saves the costs of subsequent improvements
Printed Circuit Boards (PCBs) (2)

Multiple layers (e.g. thickness of 35µm, 70µm, …)

- **Single-layer** (used in cheap consumer electronic devices) – easy and cheap fabrication, cheap materials, bad EMC characteristics
- **Dual-layer** – design and fabrication with reasonable effort, more freedom for routing
- **Multi-layer** boards (up to 12 layers in mobile phones) – usually dedicated layers for power supply and ground, feasible for highly integrated boards, good EMC characteristics

Solder mask (negative mask)

Position prints
Overview

Printed Circuit Boards (PCBs)

**Workflow for design and manufacturing**

„Ideal passive components“ and simplified ECD for a circuit path

Noise, shielding, pitfalls, etc.

Further references
Workflow for Designing a PCB

Schematic

Simulation
Workflow for Designing a PCB

Schematic

Simulation

Board Simulation

Hardware Design for Embedded Systems
Workflow for Designing a PCB

1. Manual placement of devices (e.g. quartz next to µC)
2. Manual routing of critical paths (e.g. power supply, clock)
3. Determination of mounting holes
4. Execution of auto-router
5. Subsequent improvements (e.g. ground planes)
6. Labeling (e.g. version number)
7. Design rule check (e.g. track width)
Workflow for Designing a PCB

1. Schematic
2. Simulation
3. Board
4. Simulation
5. Production
6. Testing
Workflow for Designing a PCB

Schematic -> Simulation

Board -> Simulation

Production -> Testing
Line Testing / Short Circuit Testing

**Line Testing**
Test connections, e.g., resistance
- Measure presence of current
  - Measurement < 10 Ω → Good connection
  - Measurement > 10 Ω → High-resistive connection
  - Measurement > 2 MΩ → Circuit break

**Short circuit testing**
Test different nets against each other
- Measure absence of current
  - Measurement > 2 MΩ → No short circuit
  - Measurement < 2 MΩ → High-resistive short circuit
  - Measurement < 100 Ω → Short circuit
The dark areas of the mask remain on the carrier.
The dark areas of the mask remain on the carrier.
The dark areas of the mask remain on the carrier.
Photo – Positive – Process

The dark areas of the mask remain on the carrier.
The areas under the mask are removed by cauterization.
The areas under the mask are removed by cauterization.
The areas under the mask are removed by cauterization.
The areas under the mask are removed by cauterization.
The areas under the mask are removed by cauterization.
Overview

Printed Circuit Boards (PCBs)
Workflow for designing and manufacturing PCBs
„Ideal passive components“ and simplified ECD for a circuit path
Noise, shielding, pitfalls, etc.
Further references
Crash-Course „ideal passive Components“

There are no ideal components in reality, designers have to deal with (parasitic) effects.

Real components (and also a piece of conductor path) can be modeled as a circuit consisting of ideal components.

- Ohmic Resistance „R“
- Capacitor „C“
- Coil „L“

Close to the technically achievable limits, parasitic effects of „ideal passive components“ gain in importance.
Ideal Ohmic Resistance „$R$“

resistance independent from frequency

current results in voltage drop \( U = I \cdot R \)

current results in heat dissipation \( P = U \cdot I \)
Ideal Capacitor „C“

reactance depends on frequency:

\[ X_c = \frac{-1}{2 \cdot \pi \cdot f \cdot C} \]

acts as open-circuit for DC voltage or AC voltage with low frequency

acts as short-circuit for AC voltage with high frequency

energy can be stored and restored in the electric field

no heat dissipation
Ideal Coil „L“

Reactance depends on frequency:

\[ X_L = -2 \cdot \pi \cdot f \cdot L \]

Acts as short-circuit for DC voltage or AC voltage with frequency → very low

Acts as open-circuit for AC voltage with frequency → very high

Energy can be stored and restored in the electromagnetic field

No heat dissipation
Circuit Paths on a PCB

... are no short-circuits, but are ...

- Resistors - dependent from length and cross sectional area
- Coils - dependent from length and geometry
- Capacitors - dependent from length and distance to other conductors
Simplified Equivalent Circuit Diagram for a Small Piece of a Conductor Path
Overview

Printed Circuit Boards (PCBs)
Workflow for designing and manufacturing PCBs
„Ideal passive components“ and simplified ECD for a circuit path
*Noise, shielding, pitfalls, etc.*
Further references
Noise Sources (1/2)

**Internal noise** is caused mostly by signals with high frequency (e.g. Oscillator of Microcontroller) or high currents (e.g. power supply)

Minimize length of circuit paths that could act as antennas for noise signals (e.g., ground-planes)

Maximize distance to sensitive signals (e.g. measurement signals), separation analog and digital circuits

Subdivide the system in (nested) „System Zones“ and use filters for blocking noise at the boundaries (e.g. supply)

Eliminate noise sources (e.g., no floating input pins)
Noise Sources (2/2)

**External noise** is received mostly through I/O-connectors or electromagnetic waves

Shield I/O connectors (or even the cables) and/or use filters for blocking noise

Use a star-topology for ground, i.e. connect all ground lines in a common point
Shielding

For higher frequencies a thinner shield is sufficient (e.g., conductive foil, tape, or paint)

For higher frequencies (shorter wavelengths) the tolerable gap dimension in the shield decreases

Ca. 1/10 of wavelength

Create compartments separated with vertical metal strips on the PCB
"Decoupling is stopping a portion of a circuit from being affected by switching that happens in another portion."
Example of a shielded circuit

Metal case protects against undesired electro-magnetic waves

Feed-through capacitor for decoupling the power supply
Subdivision of a system

Further shielding could be achieved with compartments of conductive material soldered on the PCB.
Path to Ground

Separate high noise/current lines and analog/sensitive signals
Things to consider for High-Current Tracks

High currents heat up tracks on the PCB
Warmer copper has a higher resistance
The increased resistance causes more heat dissipation

For high frequencies the current accumulates in the outer layers of the conductor („Skin-Effect“)
  • alternating magnetic field (due to AC) within a conductor causes eddy current
  • impedance increases
Overview

Printed Circuit Boards (PCBs)
Workflow for designing and manufacturing PCBs
„Ideal passive components“ and simplified ECD for a circuit path
Noise, shielding, pitfalls, etc.

Further references
Further References

Eagle (http://www.cadsoft.de/)

p-cad (http://www.pcad.com/)

OrCAD (http://www.orcad.com/)

PSpice (http://www.pspice.com/)

Vendor-specific application notes

...
THE END

Thanks for your attention!