

IEEE @ IIT Presents

Circuit Design Seminar

Part 2 of 3: PCB Routing

Introduction

- What is a PCB?
- Production Steps
- Sample Board
- PCB Design
- Know your Components
- Design Considerations
- Transmission Lines
- Layout Order (traces and components)
- Sample Seminar Project

What is a PCB?

- Printed Circuit Board
- Substrate for electronic components
- Single / multiple layers
 - Copper / fiberglass
- Traces (0.1" typical)
- Through-hole vs. SMD / SMT



PCB design is often overlooked: it's not just an art, it's a science.

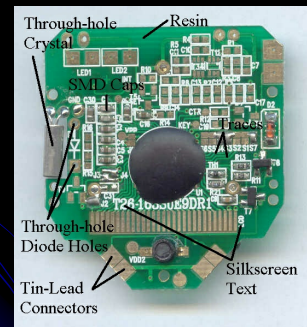
Production Steps

- Create a PCB design in a CAD environment
 - Not a straightforward process!
 - Depends on circuit type (analog, digital, mixed, etc.)
 - The focus of today
- Start with a fiberglass board
- Drill holes
 - Proper connections
- Plate fiberglass with copper
- Photolithography
 - CAD created design

More Production Steps

- Tin lead plate
- Etching (ferric chloride): removes unwanted copper
- Hot air level
- Solder mask application: prevents oxidation of copper, protects the traces
- Silkscreening, testing

Sample Board



PCB Design

- Analog, digital, mixed-signal, or other?
 - Digital circuits should have I/O traces routed first
- Components with higher pin count get higher priority
- How many layers?
 - Depends on circuit type and complexity
- Software can automatically place components
 - Not always optimal
 - Only as good as algorithms
- SMT dominates through-hole today

Know your Components

- SMD / SMT
 - Higher density, lower power
 - Typically smaller traces
- Through-hole
 - High power designs
 - Hobbyist projects

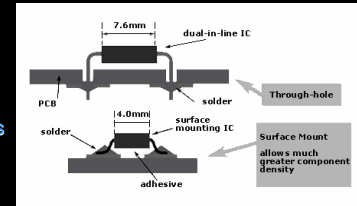


Image: http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/images/bod_ipcbt_imgq.gif

Design Considerations

- Propagation delay
- EMC (Electromagnetic Compliance)
- EM generated by high dv/dt nodes and di/dt loops
 - Shorten these paths, avoid loops
 - 45 degree corners, not 90
- Loops and sharp edges generate / receive EMI / EMC

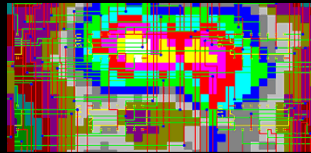
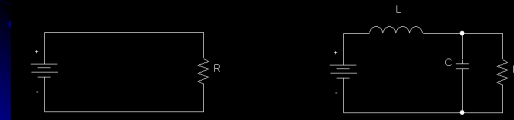


Image: http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/images/pod_ipcbt_imgac.gif

Transmission Lines

- Everything has a transient!
- No instantaneous changes in voltage / current
- Reflections



Transmission Lines

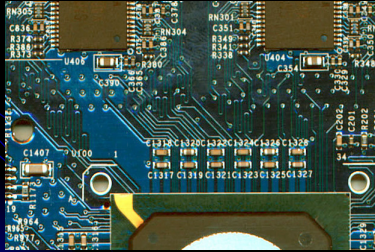
- Stray capacitance and inductance
- 2 layer vs. 4 layer boards
 - Power and ground planes can reduce EMI and improve noise rejection
- Every signal on the board has a return path
- Signal integrity, switching speed
 - 80-conductor IDE cables vs. 40-conductor
- Higher frequency devices are typically lower voltage: keep dv/dt near constant
 - 1.5V 4x AGP vs 0.8V 8x AGP

Layout Order

- Ultimately depends on your circuit
- Typical steps:
 - Analog
 - 2 layer boards common
 - Large components
 - Power and ground
 - Digital
 - 4 layer boards common
 - Large components
 - I/O buses
 - High speed signals (clk, etc.)

EMI / EMC

- Electromagnetic Interference / Compliance
- Loops in traces can act as antennas



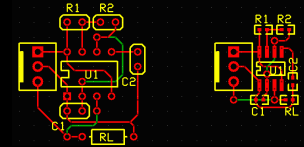
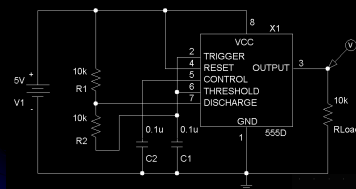
ExpressPCB

- Free software package
- Many on the market
- Commercial software packages do more
 - OrCAD PCB Designer

Advanced Topics

- Integrated antennas
- Thermal management
- Managing EMI / EMC in high speed designs

Sample Seminar Project



Questions?

What's next?

Coming up: Prototyping and circuit testing.

Rapid prototyping and circuit debug techniques will be discussed. Good soldering practice will be taught.

Thanks for coming!

References

IEEE Power Electronic Society Newsletter,
Q4 2006, p10-12.

<http://www.altera.com/literature/an/an224.pdf>

http://www.intetron.com/EMC_Tip11.htm

<http://www.xteradesign.com/xds/trace-routing.htm>