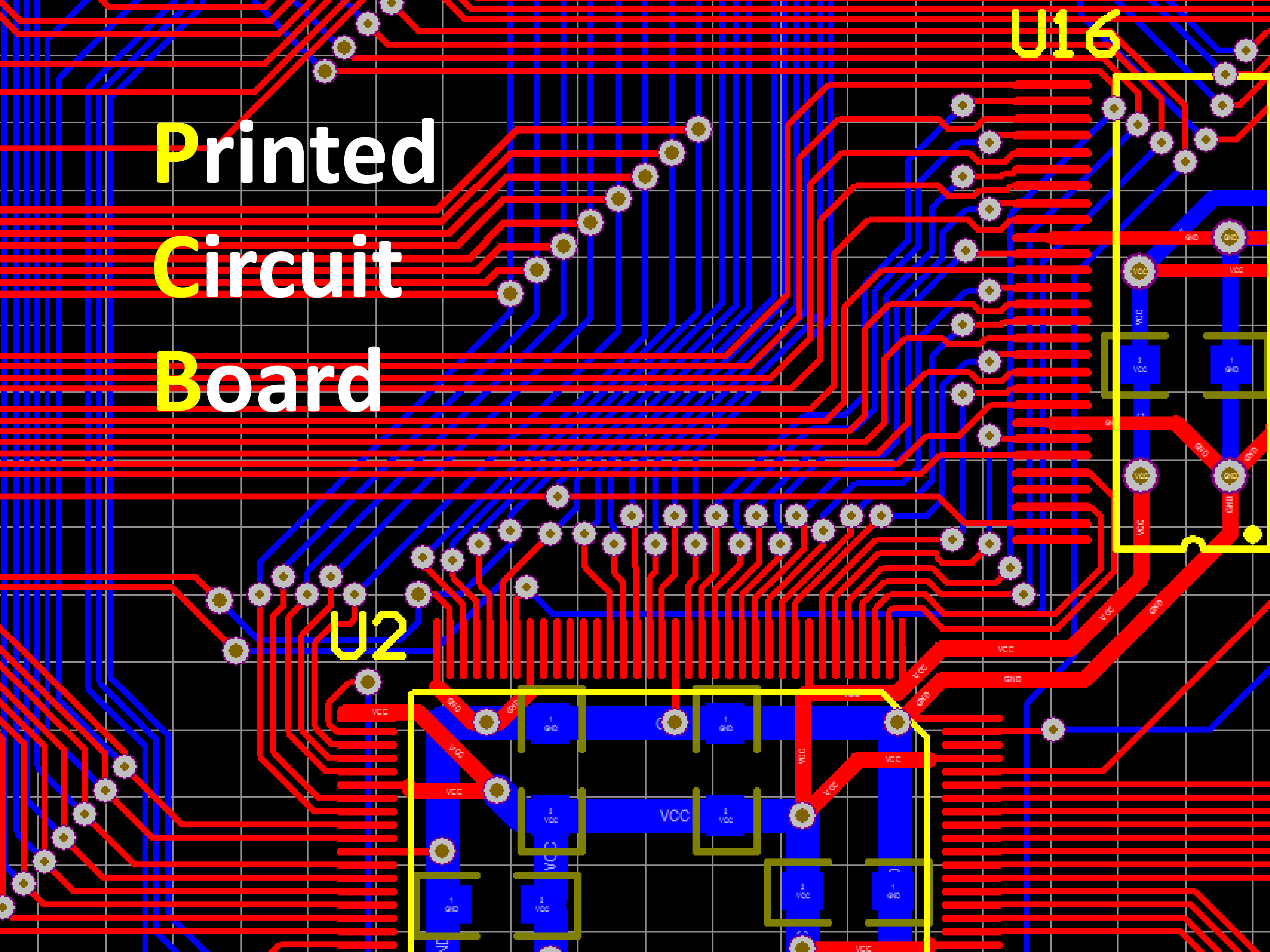
PCB Design

Using Altium Designer/DXP/Protel

Gabe A. Cohn

May 2010

Electrical Engineering
University of Washington


Printed
Circuit
Board
U16
U2
VCC
GND

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Terminology

- **Net** – representation of a signal (wire) and all of its connections
- **SMD (SMT)** – surface mount component
- **Trace (track)** – metal connections on PCB between components
- **Pad** – metal on PCB where component is soldered
- **Via** – plated through-hole in PCB to connect traces on 2 sides
- **Mil** – 1/1000 of an inch (don’t confuse with mm)
- **Pitch** – spacing between pins
- **DRC** – design rule check
- **Rat’s Nest** – point-to-point drawings of all un-routed connections

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Schematics

- Components
  - Find in Altium libraries
  - Draw your own
  - Double-check the pins
  - Connect “hidden” pins to the correct net
- Power
  - Use bypass caps for all ICs
  - Use large filter cap on power supply
- Testing
  - Test points on power lines
  - Test points on important signals
  - Use Power LED (for basic signs of life)

Schematic Hints

- Use multi-sheet schematics for large designs
- Wires must touch to make a connection
- Use NetLabels to label all wires
  - Can make your life easier in PCB layout
- For large complicated designs, connect by net rather than by connection
- Use buses
  - Must label all signals pulled from buses
- Ensure that VCC and GND are labeled correctly (everywhere)
- No shorts (like VCC and GND, for example)

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Footprints

- Choose all parts before finalizing schematic
- Choose (*hand-solderable*) packaging:
  - Through-hole: DIP, PLCC, etc.
  - SMT: SOIC, SOP, SOT, QFP, etc.
- Make sure pin numbers on footprint match component
- Carefully check all footprint from libraries
- All components must have a footprint
- Print out footprints (on paper) and test the fit

Drawing Custom Footprints

- Most library footprints are not for soldering *by hand* (so, edit them or draw your own)
- Refer to mechanical drawings in datasheets
- Select correct units (mil or mm)
- Make pads longer than pins (at least 1 mm)
- Make holes larger than pins
- Use silkscreen to show how to align part
- Check design rules:
  - Min spacing
  - Min. annular ring
  - Soldermask expansion

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Compile Design

- Annotate design by adding designators
  - Replace U? with U1, U2, etc.
  - By hand (if very few components) or automatically using Altium's tools
- Compile the design (will generate netlist)
- Read and **address** all errors and warnings
- Re-compile after making all changes

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Start PCB Layout

- Create a blank PCB (easiest to use the wizard)
- Choose a size
  - Compromise between routing ease and wasted space
- Move the new PCB document into the project and save it
- Do an initial “Import Changes from Schematic” *(described later)*

Design Rules

- Defined by PCB manufacturer
- Must choose a PCB fab. house now
- Get rules from PCB fab. house:
  - Trace width
  - Clearance
  - Via size
  - Annular ring
  - Hole size restrictions
  - Soldermask expansion (swell)
- Run DRC (design rule check) after each change
- DRC violations highlighted in green (in real-time)

Design Rules: Trace Width

- Must be greater than manufacturer’s min.
- Must be large enough to handle current
- Signal Nodes:
  - Preferred: 10 mil
  - Limit: 8 mil
- Power Nodes:
  - Preferred: 30 mil
  - Limit: 20 mil

Design Rules: Clearance and Soldermask Expansion

- Must be greater than manufacturer’s min.
- Preferred: 10 mil
- Limit: 8 mil

- Soldermask expansion is space around pads that are not covered by soldermask
- Don’t put SMT pads so close that there is no soldermask between them (very hard to solder)

Design Rules: Hole/Via Size and Annular Ring

- Ensure that pins/wires fit in holes
- Warning: the plating will make holes smaller
- Size and annular ring must be greater than manufacturer's min.
- Some manufacturers only allow certain sizes, and will round your sizes to the nearest drill (be careful)
- Preferred: 35 mil hole, 10 mil annulus
- Limit: 28 mil hole, 8 mil annulus

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Sync PCB with Schematic

- Import all components from schematics
- Keep schematics and PCB in sync throughout design
  - I always make changes to the schematics, and then sync the PCB layout (but Altium can go both ways)
  - Keep an eye on what is happening when syncing!
  - Save often!

Place Components

- Components start aligned on the right side of the screen (off of the PCB)
- Must manually place and orient each component
  - Keep connections short
  - Bypass caps must be next to power pins of each IC
  - Connectors generally on edge of board (facing the correct way)
- Minimize wire crossings in Rat's Nest
- Shortcuts (while dragging a component):
  - Space rotate component 90 degrees
  - L change which layer the component is on

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules – fix all violations
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Routing

- Lay down traces to connect all nodes
- Can be done manually or automatically
- Routing ease depends on component placement (routing is NP-Complete)
- Untangle the Rat's Nest as much as possible before routing:
  - Re-order signals (headers, buffers, IOs, prog. logic)
  - Alter the schematic (not necessarily the design)

Manual Routing

- Use interactive router
  - Won't let you violate design rules
- May need to adjust electrical grid
- Works well for buses and power lines
- Many digital designs must be manually routed
- It's fun! ☺
- Shortcuts:

  *　　　(on numpad) changes layers

Autorouting

- Ensure design rules are defined
- Define keepout regions (board edge, mounting holes)
- Try to autoroute first, then fix problems
  - Must carefully check the autorouter’s output
  - Fix all problems with the autorouter’s output
- Manually route some difficult/important nets, then run autorouter
  - Buses and power are often manually routed
- Sparse designs can typically be easily autorouted

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules – fix all violations
9. Generate manufacturing files

Steps

1. Draw schematics
2. Attach footprints for all components
3. Compile design
4. Assign design rules
5. Place components
6. Check design rules
7. Route PCB
8. Check design rules
9. Generate manufacturing files

Final Checklist

1. Schematics still in sync?
2. Check over connections (specially for ICs)
3. Re-check footprint dimensions
4. Include mounting holes
5. Add name, title, date, logos to silkscreen
6. Add board outline (mechanical layer)
7. Check all hole sizes (in drill report)
8. Run final DRC

Manufacturing Outputs

- Check with manufacturer for specifics
- Make using File->Fabrication Outputs
- Gerber files (one for each layer)
- NC Drill files
- Carefully check over the Gerber files in an Gerber viewer or CAM editor

Gerber Files

- Specify masks for each layer:
  - .GTL top layer (metal)
  - .GTS top soldermask
  - .GTO top overlay (silkscreen)
  - .GBL bottom layer (metal)
  - .GBS bottom soldermask
  - .GBO bottom overlay (silkscreen)
  - .GM1 mechanical layer 1
  - etc.

NC Drill Files

- Specify location and size of all holes
  - .DRL EIA (binary) drill file
  - .DRR drill report
  - .TXT text file describing holes

PCB Manufacturers

• Advanced Circuits	www.4pcb.com
• Sunstone Circuits	www.sunstone.com
• AP Circuits	www.apcircuits.com
• Sierra Circuits	www.protoexpress.com
• Prototron Circuits – In Redmond	www.prototron.com

Altium Resources

- All Documentation
  - http://wiki.altium.com/display/ADOH/Home
- Keyboard Shortcuts
  - http://wiki.altium.com/display/ADOH/Editor+Shortcuts
- Manual Routing
  - http://wiki.altium.com/display/ADOH/Interactively+Routing+a+Net

PCB Design Demos

www.gabeacohn.com/teaching/pcb

Gabe A. Cohn