1 PCB Design Using Altium Designer/DXP/Protel

Gabe A. Cohn

May 5, 2010

CSE 477 – Hardware Capstone

University of Washington

2 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

10.

•

•

3 Terminology

- Net representation of a signal (wire) and all of its connections
- 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

4 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

10.

- •
- •

5 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)

6 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)

7 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

10.

•

•

•

8 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

9 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)

5/5/2010

- Make holes larger than pins
- Use soldermask to show how to align part
- Check design rules:
 - Min spacing
 - Min. annular ring
 - Soldermask expansion

10 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

10.

- •
- •

•

11 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

12 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

10.

- •
- •

_

13 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

14 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

15 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)

16 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

17 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

10.

•

•

18 Start PCB Layout

- Create a blank PCB (easiest to use the wizard)
- Choose a size
 - Compromise between routing ease and wasted space

- 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!

19 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

20 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

10.

- 21 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

10.

22 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)

23 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

24 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

-

25 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

10.

.

•

26 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

10.

•

27 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
- 9.

28 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
- •

29 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.

30 **NC Drill Files**

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

31 PCB Manufacturers

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

32 PCB Design Demos

www.gabeacohn.com/teaching/pcb

5/5/2010

Gabe A. Cohn