Senior Design Workshop

PCB Design with EAGLE

Dr. Feng Li
Dept. of Electrical & Computer Engineering

University of Idaho
Outline

• PCB introduction
• CadSoft EAGLE introduction
• Schematic design
• Board layout and routing
• CAM processor export
• DFM
• Fabrication
• Assembly
What is PCB?

- **Printed Circuit Board**
  - Mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate.

- **Two purposes:**
  - A place to mount the component
  - Provides the means of electrical connection between the components

- **PCBs can be**
  - Single sided (one copper layer)
  - Double sided (two copper layers)
  - Multi-layer
PCB Terminology – Component Packages

- Through-hole
  - Soldered to the opposite side of the board
- SMD/SMT (surface mount device/technology)
  - Soldered in the same side of the board
  - Can be mounted on both sides of the PCB
  - Smaller than the thru-hole type-smaller & denser PCB
- Some common form factors
  - 0805 (means 0.08”x0.05”)
  - 1206
  - 1210

PCB Terminology

• Pad: a small surface of copper where component will be soldered to the board
• Via: plated hole that allows the current to pass thru the board
• Track (trace): conductive path connecting 2 points (pads, vias)
• Soldermask: a layer of insulating lacquer covering both surfaces of the board to prevent the solder to short-circuit two tracks from different nets.
• Silkscreen (overlay): letters printed on the final board, e.g., solder mask
PCB Terminology

- SMD pads
- Thru-hole pads
- Silkscreen
- Copper tracks
- Soldermask (stop mask)
- Via
- Thru-hole (full stack) via
- 1-2 blind via
- 2-3 buried via

http://en.wikipedia.org/wiki/Via_(electronics)
PCB Process Flow

- Function Design
- Form factor
- Design rules
- Schematic
- Layout
- Routing
- CAM Gerber output
- Electronic rules
- Component library
- DFM
- Production
- Assembly
- DIY and/or workshop

EAGLE

FAB house
What is EAGLE?

- **Easily Applicable Graphical Layout Editor**
  - Easy to use CAD tool for designing PCBs
  - CadSoft

- Consist of three main modules:
  - Schematic editor
  - Layout editor
  - Autorouter

- It runs under Windows, Linux, and MacOS

- It comes with an extensive library of components
Three Versions

• EAGLE light
  • Limitations:
    • 1 schematic sheets;
    • 2 signal layers (top & bottom)
    • 100x80mm routing area
  • It can be used for free

• EAGLE standard
  • 99 schematic sheets
  • 4 signal layers
  • 160x100mm routing area

• EAGLE Professional: full featured version
  • Up to 1600x1600mm
  • Up to 16 routing layer
  • Up to 999 sheets per schematic
Download and Installation

• Current edition: 7.1.0
• Professional version (6.5.0) available in the Senior Design Lab
  • Contact John Jacksha (J.J.)

• 7.1 and 6.5 may have compatibility issues
  • Begin with 6.5 light version and use Pro version in lab
  • Begin with 7.1 light version and order your standard or professional version
Installing Optional Libraries

• SparkFun library
  • contains footprints for many common devices
  • helps narrow the choices
  • https://github.com/sparkfun/SparkFun-Eagle-Libraries
  • C:\Program Files (x86)\EAGLE-6.5.0\lbr

• Adafruit
  • http://github.com/adafruit/Adafruit-Eagle-Library
  • C:\Program Files (x86)\EAGLE-6.5.0\lbr

• Component order
  • http://www.digikey.com
EAGLE User Interface – Control Panel

Menu bar

File manager

Status line

Information window: display what is selected in file manager
Example Project: *LED Flasher Using 555 Timer*
Creating and Naming a New Project

- File->New->Project (you can name the folder as flasher)
Creating a New Schematic

• Right click folder “flasher” -> New -> Schematic
Save and Rename of New Schematic

• [Schematic]File->Save as... (flasher.sch)

• Note: Do not create a board file yet.
Schematic Editor Window

- Title Bar
- Menu Bar
- Toolbar
- Parameter toolbar
- Command Toolbar
- Command Bar
- Working Area
- Status Bar
Schematic Editor Window Command Tools

- **Info** – Info on a part
- **Layer settings** – Change Layers
- **Move**
  - **Group** – Define a bunch of parts to a group to move.
  - **Delete** – Delete a part
  - **Name** – Name a part
- **Show** – Highlight a part, wire, or trace. Very Useful
- **Rotate**
- **Add** – Add a part from the library
- **Value** – Define a value (20 Ohms, etc)
- **Net** – Connect component
- **Label** – Label items
- **ERC** – Electrical Rule Check
Finding and Add a Component

- [Schematic] Add
Search for a Component

• Enter “*555*” in the Search field
• Find “NE555”
• Click “OK”
Drop a Component

- Move the cursor toward the middle of the schematic window
- Click to drop the component
- Press “ESP” key to escape from adding component;
- Click “Cancel” to escape from the adding Window.
- You can delete a Component by clicking “Delete”
Add Components – Resistors and Capacitor

• SparkFun-Resistors: RESISTORAXIAL-0.3 (x4)
• SparkFun-Capacitors: CAP_POLPTH2
Add Component – LED and a Screw Terminal for Power Supply

• Adafruit: LED5MM (search “LED5mm”)
• Adafruit: 1X2-3.5MM (search “terminal”)

![Diagram of adding component and terminal](image-url)
Rotating and Moving Components

- Rotate
- Move
- Zoom...
Connecting the components

• Net
Setting Component Values

- Value
Electrical Rule Check

• ERC
• Use “Show” to show net name
• Search for “capacitor” and find CAP-PTH-SMALL
• Add the capacitor and connect it to Pin5 and the ground
Net classes

- [Schematic]Edit->Net classes
- Can also be done in Layout Editor later

1mil = 0.001inch
Net Classes

• Info
Laying Out the Board

- [Schematic]File->Switch to board
- “This is no board, so would like to create one from the schematic?” – “Yes”
- Yellow lines: airwires-connections that will have to be converted into tracks
- Rectangle: borders

Note: 1. Do NOT close either schematic or board window. They must both remain open while working. Change in on editor window will lead to change in the other window.

2. Change background color: [Board]Options->User interface...->Background->White
Board Tools

Layer Settings – Change layers
Move – Select to Move components

Group – Group multiple components to move at once.

Route – Create a trace (this is the actual wire that carries the electrical signal)
Rectangle – Draw and define a rectangle
Via – Brings a trace from the top layer to the bottom layer

Ratsnest - Calculates the shortest possible airwires and polygons
Show – Highlight a trace or part. Useful to see what connects where.

Ripup – Remove Traces
Text – Add text
Polygon – Draw and define a polygon.
Auto Router – Somewhat unreliable
Design Rule Check – Very Important!
More Later
Move

- Move
- Group
- Group move
• Move
• Rotate
• Move
• (resizing the board)
• Zoom to fit
Ratsnest – calculate the shortest airwires (and polygons)
Add Mounting Holes
Add Mounting Holes
Autorouter

• [Board] Autorouter
Board Layout
Rip Up the Tracks (Optional)

• “Rip up”
• “Group”
• “Ripup: Group”
Tweak the Result – Add a Ground Plane

• Polygon – draw a square around the outline of the board

• (Right click the border of the polygon) Properties – Layer: 16 Bottom, Width 0.01
Add a Ground Plane

- “Ratsnest” - the polygon fill with blue
- “Name” - GND
Add a Ground Plane

- Autorouter
- Ratsnest: Nothing to do!
Design Rule Checker

- [Board]DRC
CAD Layers

• 1 Top (top tracks)
• 16 Bottom (bottom tracks)
• 17 Pads
• 18 Vias
• 19 Unrouted
• 20 Dimension
• 21 tPlace (top silkscreen)
• 22 bPlace (bottom silkscreen)
• 23 tOrigins
• 24 bOrigins
• 25 tNames
• 26 bNames
• 27 tValues
• 28 bValues
• 44 Drills (for pads (of through hole components) and vias)
• 45 Holes (for mounting holes)

Note: CAD layer is different from (conductive) layer.
Add Text on the Top (Silkscreen)

- [Board]Layers – Pads, Dimension, tPlace, tOrigins, tNames, tValues
Smash – Separate the Text From Devices

• [Board]Smash
• Group
• Smash: Group
Change Size

- Change -> Size -> 0.05
Change Font

- Change \rightarrow Font \rightarrow Vector
- Group - Change: Group
Change Ratio

- Change->Ratio->12
- Group – Change:Group
Move Labels

• Move (labels)

• Layers – Pads, Dimension, tPlace
Add Text to the Top (silkscreen)

- Change-Layer-tPlace
- Change-Size-0.05
- Change-Font-Vector
- Change-Ratio-12
Add Text to the Top

- Text->GND
Add Text to the Bottom

- Layer->Pads, Dimension, bPlace

- Change->layer->bPlace
Add Text to the Bottom

• Text – www.uidaho.edu
Final Board Layout
Gerber Files

• A 2D vector image format
• Universal format and industry standard for PCB fabrication
• TXT files with coordinates that tell the PCB machines to go to location X, Y and do something (drill, expose, etch, print, etc).
• Formats:
  • RS-274X (Extended Gerber or X-Gerber)
    • Human readable ASCII format consisting of a sequence of commands and coordinates
  • RS-274D (obsolete)
• Different text files for different layers.
• Generated by the CAM (computer aided manufacturing) processor
## Typical Gerber File Types

<table>
<thead>
<tr>
<th>Description</th>
<th>EAGLE (default)</th>
<th>EAGLE (Sparkfun)</th>
<th>Orcad</th>
<th>Protel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top copper layer (component side)</td>
<td>cmp</td>
<td>gtl</td>
<td>top</td>
<td>gtl</td>
</tr>
<tr>
<td>Bottom copper layer (solder side)</td>
<td>sol</td>
<td>gbl</td>
<td>bot</td>
<td>gbl</td>
</tr>
<tr>
<td>Top solder mask</td>
<td>stc</td>
<td>gts</td>
<td>smt</td>
<td>gts</td>
</tr>
<tr>
<td>Bottom solder mask</td>
<td>sts</td>
<td>gbs</td>
<td>smb</td>
<td>gbs</td>
</tr>
<tr>
<td>Top overlay (silkscreen)</td>
<td>plc</td>
<td>gto</td>
<td>sst</td>
<td>gto</td>
</tr>
<tr>
<td>Bottom overlay (silkscreen)</td>
<td>pls</td>
<td>gbo</td>
<td>ssb</td>
<td>gb0</td>
</tr>
<tr>
<td>NC Drill file (coordinate info)</td>
<td>drd</td>
<td>txt</td>
<td>thruhole.tap</td>
<td>drl</td>
</tr>
</tbody>
</table>

Note: Extensions are different for different PCB design tools, but information is the same.

EAGLE default CAM job needs to create drill file and gerber files separately. SparkFun CAM job needs only one step. [https://github.com/sparkfun/SparkFun_Eagle_Settings/tree/master/cam](https://github.com/sparkfun/SparkFun_Eagle_Settings/tree/master/cam) Download the file sfe-gerb274x.cam, and save to some folder
Gerber File Generation – CAM Processor

- [Board] File -> CAM Processor
- [CAM Processor] File -> Open -> Job -> sfe-gerb274x.cam
Gerber File Generation – CAM Processor

• [CAM Processor] Process Job
Gerber Review, Submit and Order

• Review: verify the CAM output in a Gerber viewer to make sure everything was positioned correctly
  • Viewplot: http://www.viewplot.com (can be downloaded and installed)
  • Online viewer: www.gerber-viewer.com

• Design for manufacturability (DFM)(Files upload)
  • FreeDFM: http://www.freedfm.com
  https://www.my4pcb.com/net35/FreeDFMNet/FreeDFMHome.asp

• Some PCB fab houses
  • Advanced Circuits http://www.4PCB.com
    • quick, reliable, and relatively cheap
    • $33/board with a week turn around
  • Dirty Circuits http://www.dirtycircuits.com
Assembly (Soldering)

• Through-hole components
  • DIY at ECE Senior Design Lab (BEL 216)

• Surface-mount devices
  • GJL 001 ECE lab (free)
    • < 30 components
    • Case size > 0402
    • Label pin numbers

• Mr. Greg Klemesrud: gregk@uidaho.edu
Reference


Special Thanks to

• Dr. Michael Santora
• Dr. Steven Beyelein
• Mr. John Jacksha (J. J.)
• Mr. Greg Klemesrud