

EECS 192: Mechatronics Design Lab

Discussion 5: PCB Peer Review

written by: Richard "Ducky" Lin 📧 Spring 2015

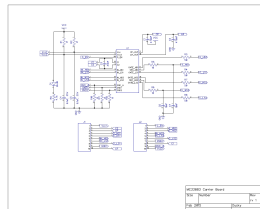
15 & 16 Feb 2017 (Week 5)

- PCB Peer Review
- Fabrication Data
- Summary

PCB Peer Review

PCB Peer Review

- ▶ Why peer review?
 - ▶ Get a fresh perspective on your board to catch bugs you've missed
 - ▶ Get a new opinion from someone with a different background
 - ▶ Facilitate transfer of knowledge
- ▶ Things to look for in your peer reviews:
 - ▶ Schematic style: messiness hides bugs!
 - ▶ Circuit safety and spec check
 - ▶ Layout sanity: DRC violations, don't design for minimums
 - ▶ Really, anything that looks off



Hopefully a fairly readable schematic

PCB Peer Review

Pair up with another team

(or another two teams, if you're in an odd group of three)

Bring up the PCB Peer Review Checklist

(www-inst.eecs.berkeley.edu/~ee192/sp15/docs/dis5-pcbchecklist.pdf)

but feel free to add additional criteria as you want

You'll have 30 minutes to review each other's boards

(so about 15 minutes per team in a group)

Note anything you really liked about the boards you reviewed

as well as pitfalls others should know and avoid

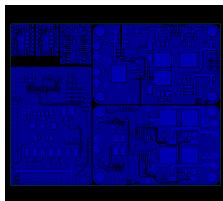
We'll discuss as a class after you're done in groups

PCB Fabrication Data

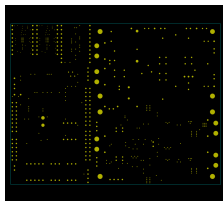
Gerbers

no, it's not baby food...

- ▶ The layers we're interested in are:
 - ▶ top / bottom copper
 - ▶ top / bottom silkscreen
 - ▶ top / bottom soldermask (negative image)
 - ▶ board outline
 - ▶ drill file
- ▶ The Gerber format (RS-274X) is a bi-level (2 "colors") vector image format
 - ▶ De-facto standard for PCB layer data
 - ▶ Contains detail "layer" information.
- ▶ The N/C drill file is officially called the Excellon format.
 - ▶ Contains drill hole information.
- ▶ You should export these from your design tool for submission to the board house



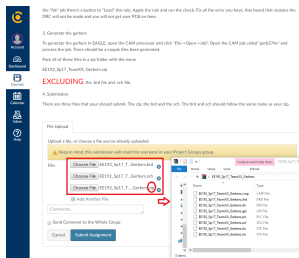
Top Copper Gerber



N/C Drill file

EAGLE CAM processor

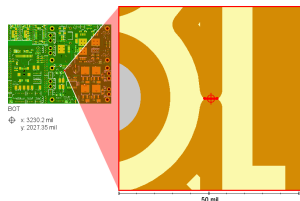
- ▶ The CAM processor generates the Gerbers and drill.
- ▶ There are several preset "job" we can use.
- ▶ On the menu, click "File", "Open", "Job"
- ▶ Gerbers
 - ▶ Pick "gerber274x"
 - ▶ Process Job
 - ▶ .cmp, .gpi, .plc, .sol, .stc, .sts
- ▶ drill
 - ▶ Pick "excellon"
 - ▶ Process Job
 - ▶ .drd, .dri
- ▶ You should export these from your design tool for submission to the board house



Complete PCB
submission example

InstantDFM

- ▶ DRC: Design Rules Check
DFM: Design for Manufacturability
 - ▶ or, can the board house make it and expect it to come out working
 - ▶ These typically check for minimum feature sizes (trace width / spacing, hole size)
 - ▶ If it fails, don't expect a functional board
- ▶ Bay Area Circuits has a online DFM tool: (instantdfm.bayareacircuits.com)
 - ▶ Run your Gerbers through it to ensure it's within limits for fabrication



InstantDFM showing minimum trace width

Deadlines and Submissions

- ▶ Make sure the size of the board fit your mechanical design.
 - ▶ 3D-space for heat dissipation
 - ▶ Each team should fit their entire design into a 4" by 12" area.
 - ▶ If you made multiple boards, remember to add an outline encapsulating all the design.
 - ▶ Mark the cutting trace with silkscreen, not copper.
- ▶ Submit gerbers and drills as a .zip on bCourses, together with the .sch file and .brd file.
- ▶ Tuesday (2/21), 11:59PM: Design files for review by course staff and peer review
 - ▶ We will check over your schematic and layout for obvious errors and return comments within 24 hours
- ▶ Thursday (2/23), 09:00PM: Final Gerbers due
 - ▶ This is what gets sent to the board house.
 - ▶ Watch your email carefully - we will do a quick spot check - be prepared to fix errors FAST.

Summary

Summary

- ▶ Do design reviews so others can catch bugs that you won't!
- ▶ Generate Gerber fabrication data for your boards for submission
- ▶ Verify your designs through InstantDFM

Parts Handout

- ▶ Get a BlueSMiRF (Bluetooth serial terminal)
 - ▶ this is how you `printf` on a moving platform

Checkoff Reminders

- ▶ Avoid alligator clip leads for your motor drivers. Your circuit should begin to resemble what would go on your car - make nice connectors with nice wiring which you can re-use when boards come in.