Battery Safety
Fabrication Data
Summary
Batteries
LiPo battery
- Fire hazard if mistreated
- Do not let the battery get punctured or shorted
- Never leave the battery plugged into anything unattended (including charger and battery checker)

Battery checker and buzzer
- Must always use whenever the battery is in use per NATCAR rules
- Note which pins correspond to which cells
Battery Charger

- Plug in both the battery main power connector and white balance connector
- Recommend charging at 2.2A (selected with dial)
- Unplug the battery once it is done charging (charger beeps and flashes)
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
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.
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 an online DFM tool: (instantdfm.bayareacircuits.com)
  - Run your Gerbers through it to ensure it’s within limits for fabrication
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 (2/26), 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

- **Friday (3/1), 01:59PM:** Final Gerbers due
  - You can send the PCBs in for printing independently as a team
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

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.