Before you send your gerbers to your TA:
- Your gerber files should pass PCBWay's audit
- Your gerber zip file should have your team number in the file name
- The cost of 10 boards should be $5 (as shown in the cart after audit)
  - Is the minimum hole 0.3mm? 0.2mm holes will cause problems
  - Is the board outline within 100mm by 100mm? Almost all boards made for this course can be made to fit in that square

This is a checklist covers many but not all of the common mistakes made by students. If you are short on time, the most important items are highlighted in blue.

Schematic
- How will you program the microcontroller? Does it have enough memory? Is it fast enough? Does the default bootloader let you program the chip?
- Are any components being overvolted? What is the maximum operating voltage of your microcontroller?
- What voltage are your logic signals? Are your logic signals push/pull or open collector/drain? Do you have pull up/down resistors where needed?
- Are decoupling/bypass capacitors placed properly? (video)
- Make your circuit easy to debug! Have you placed test points on key nodes? (Add a part > search "test point". Edit symbol properties and select a footprint. Search "test point" in the footprint editor)
- If you are apprehensive about a chip not working, you can break out the pins into a row of headers (Search "Conn" in "Choose Symbol" window)
- Do you have a low-pass filter in front of any ADCs you are using? Even if you aren't sure you need one, make one anyway. You can bridge over the resistor and remove the capacitor if you change your mind later.
- Have an external oscillator for the micro if you have timing requirements (either strict timing, or need timing over a long period of time). Also, know exactly what oscillator requirements your microcontroller needs
- Is there an easy way to power the circuit with a lab bench power supply? Having the ability to current limit your circuit while building and testing it is very useful. This can be as simple as having test points to solder leads to.

Components
- Are connectors used OK for rated current? Take a look at "screw terminals" for external connections.
- Are barrel connectors correctly sized? They aren't standardized well.
- Are components hard to solder? (QFN, BGA, etc. are really hard to solder because the pads are under the package)
- Are your SMD passive components too small? 0603 imperial (not metric!) are as small as you can feasibly hand-solder. 0805 is much easier.`!
- Does the footprint of what you bought/will buy match what you are using on your board? Ctrl-Shift-M to measure in the KiCAD PCB Editor
- Are your components in-stock?

Printed Circuit Board

- Decoupling capacitors are placed next to ICs
- Proper trace widths. Ask how much current will be going through. A good metric is at least 20 mils for power and at least 10 mils for signal. If they have some high current paths, they may need to be more, though. (KiCAD project window > Calculator > Trace Width Calculator)
- Traces should not make 90 degree bends! These produce a lot of noise.
- The maximum size your board can be is 100mm x 100mm
- No unrouted nets, no vias in pads
- Did you include mounting holes? How will the circuit mount to the rest of your project?
- Do you have a ground plane? Ground planes help reduce EMI
- Have you placed micro or mini USB connectors at the edge of your board? You won't be able to plug into the connector if it's in the middle of your board!
- Have you entered PCBway's capabilities in the design rule checker? (6 mil minimum trace width/trace spacing; 0.3mm minimum hole size) (In PCB Editor: File > Board Setup > Design Rules > Constraints

Copper

Minimum clearance:	0.15	mm
Minimum track width:	0.15	mm
Minimum annular width:	0.15	mm
Minimum via diameter:	0.3	mm
Copper to hole clearance:	0.3	mm
Copper to edge clearance:	0.3	mm

Holes

Minimum through hole:	0.3	mm
Hole to hole clearance:	0.3	mm

- Have you run a design rules check (DRC)?
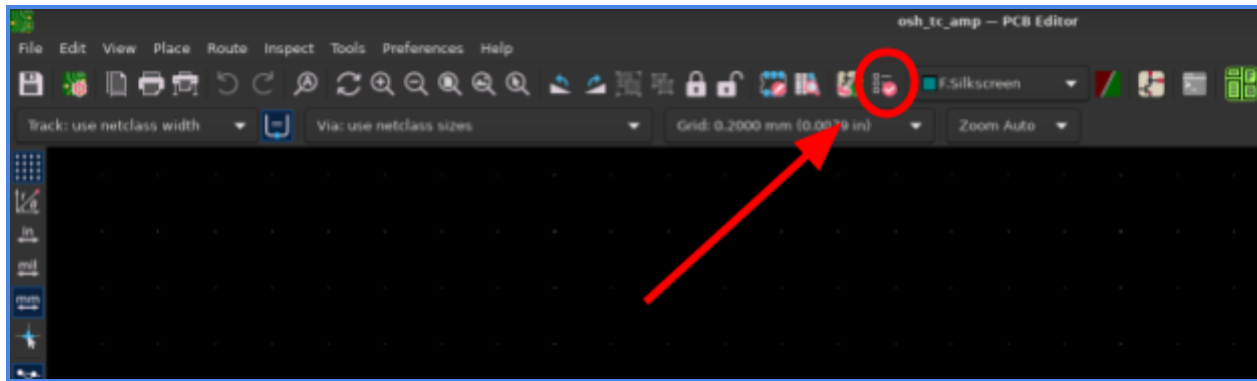
- Do you know how to generate gerber files?
- Do you know how to view gerber files? (KiCAD project window > Gerber Viewer or this online viewer)
- Do you know how to have PCBway check your gerber files?
- Things to include in your zip file
    - Front copper
    - Back copper
    - Front soldermask
    - Back soldermask
    - Top silkscreen
    - Bottom silkscreen
    - Drill file
    - Edge cuts
    - F. paste (if you want a stencil for the front)
    - B. paste (if you want a stencil for the back)