

# ✓ Jitter Pre-order PCB Checklist

Use this Checklist to prevent costly iterations of your designs by finding mistakes before ordering

## Schematics Check

1.  Check all not-connected pins. Are you sure they shouldn't be connected?
2.  Verify microcontroller pins with datasheet (pinmuxing/pinconfig): are the selected pins suitable for the intended use case?  
Watch out for open-drain pins, current ratings, slew rate.
3.  For each connector/mounting hole: are all shield/GND pins connected to GND?

## Layout Check: Placement

4.  Review the 3D view: check for overlapping parts, or parts that are so close that it would be hard to place them.
5.  For each connector/mounting hole: is the mating connector/screw/cable going to fit? Sometimes the cable-side connector is wider than the PCB footprint!
6.  Is the board outline compatible with assembly in the final product / enclosure? The best way to test is to export the PCB to CAD software and try to fit it into the product.
7.  Is it possible to assemble the board into your product? Try to visualize in what order to assemble it: watch out for notches in the PCB outline and connectors extending past the board outline.
8.  Check for each sensor if the orientation with relation to the physical world is correct. E.g. the direction of an encoder strip.

## Layout Check: Routing

9.  Review each layer in high-contrast mode: can traces be made nicer / tidier?
10.  Highlight power nets: could they be routed to be shorter? Are traces wide enough for the rated current?

## Layout Check: Ground Plane

11.  Re-pour all planes in the design  
(KiCad: b, Altium: t,g,a)
12.  Verify that the ground plane covers the complete board
13.  Check if ground planes are connected to correct net
14.  Review ground plane(s): cutouts/holes/slots should be minimized. Verify there are no slots in the ground plane caused by groups of vias.

## Layout Check: Silkscreen

15.  Silkscreen text should be clearly readable, not overlapping.
16.  Verify presence of clear labels for connectors (add pinout info where possible).
17.  Verify presence of company/project Logo + correct PCB name/version

## Gerber Check

18.  Are all required layers exported as gerber file? Are there any non-required layers (remove them to avoid confusion)?
19.  Solder mask layers: are all pad openings as expected? You want solder mask between all pads if possible (kicad: setup - pads to mask clearance: tweak clearance + min width)
20.  Paste layers: Are all required stencil openings present. Reversely, are there any openings that shouldn't be there? Usually this is a matter of footprint settings.

If you have any questions, or need more assistance,  
send us an email at [info@jitter.company](mailto:info@jitter.company)

# Jitter

Checkout our other content at  
<https://jitter.company>