

Routing Techniques

Tips for a proper, organized PCB

Good Routing Techniques

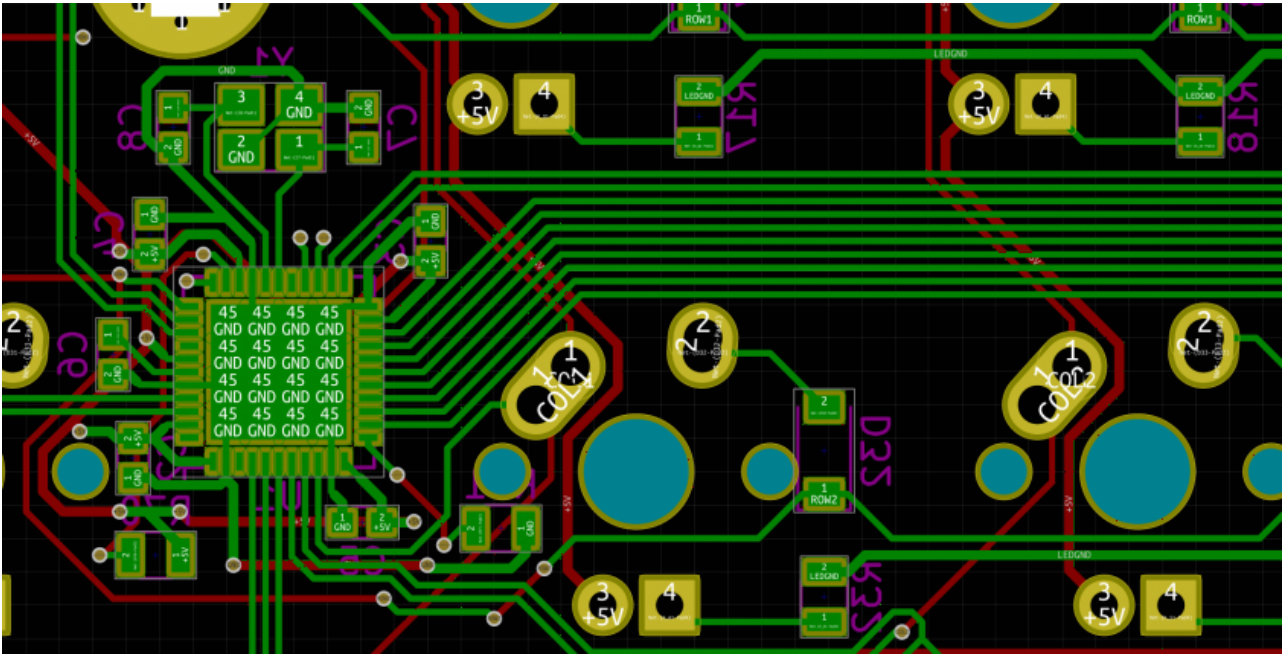
- Do not route traces close to the edge.
- Do not place EMI-emitting components, such as crystals, near a board edge.
- Surround pulsating traces with proper ground plane fill to reduce EMI.

Verticals and Horizontals

- Use front and back for horizontal and vertical traces respectively, or something similar to avoid clashing.
- Pick sides which do not force the abuse of vias for routing smd components.
- Let busses take precedence over singular traces (explained below).

Trace Bus

- Route many traces going in the same direction together for efficient use of space.
- This can be used to efficiently break out many traces from the controller.



Aesthetics

- Space things evenly.
- Bundle traces together if not susceptible to interference.

Revision #4

Created 25 January 2018 11:45:36 by ai03

Updated 29 June 2020 01:52:32 by ai03