Potential Areas for Improvement

If you want to be an over-achiever, the copper area and trace length should be minimized. Probably not worthwhile...

I don't like the routing here... Is there a better way?

Problem in PCB, NEEDS to be fixed

Increase spacing between traces Here

Add ~5V Zener Diode & Ceramic Cap

Footprint isn't quite the right size, it will need to be made smaller to be soldered onto the board well

Footprint too big, find another or customize and make smaller Should be powered from 5V, not 3.3V

Increasing trace spacing here wouldn't be a bad idea

May or may not be backwards... Investigate...
Trace spacing must be increased

Bigger through holes, or one bit through hole might make this iron solderable... currently hot air is necessary

PFET Footprint has wrong pin-number order
May or may not be backwards... Investigate...
You're welcome Mechanical Engineers