

W5100 Layout Guide

1. Goal

- Make a low noise and stable environment for W5100 working.
- Reduce the possibility of EMI, EMC.
- Make a better circuit for W5100 by simplifying the signal trace.

2. Placement

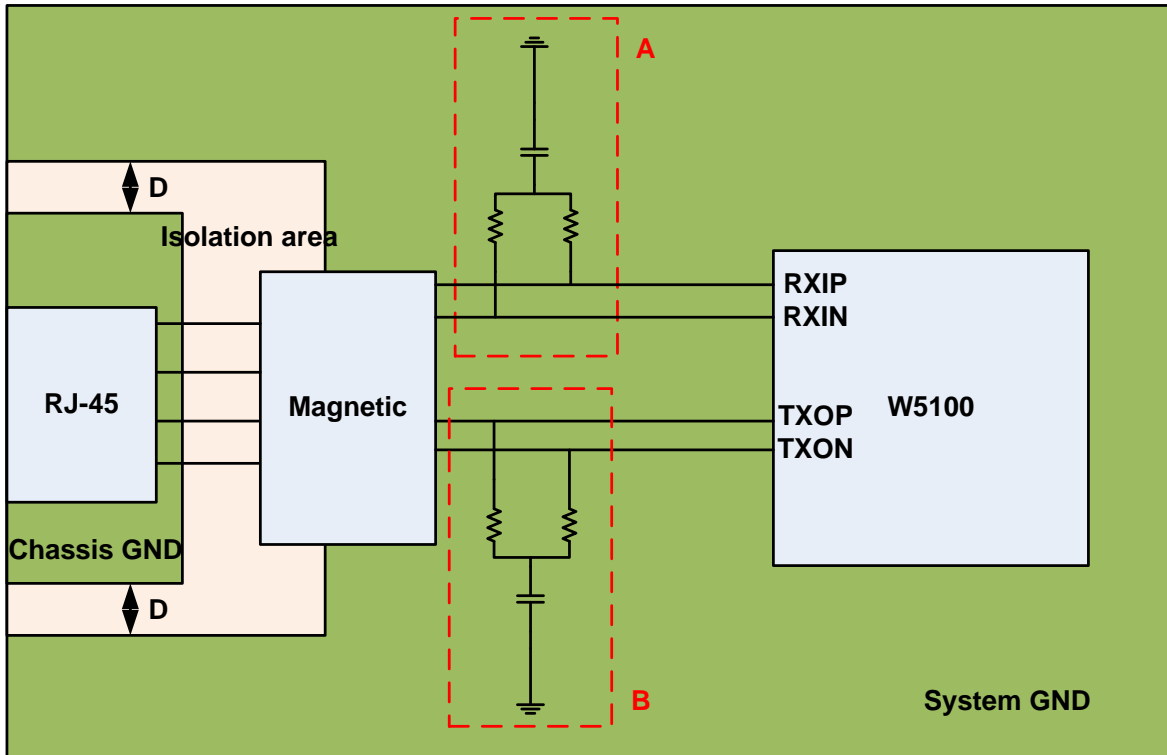


Figure 1

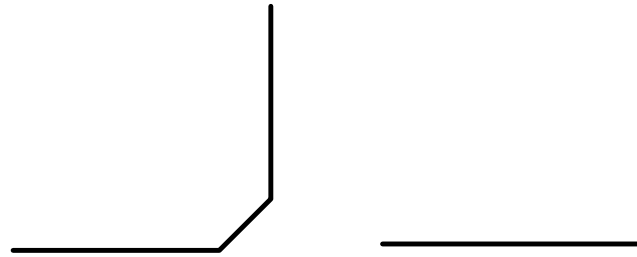
- Block A and B may be better placed as close to magnetic as possible. Let the trace between W5100 and magnetic as short as possible, and keep the Tx+/- (So as Rx+/-) signal traces to be symmetry. The traces should not be too long and 12cm will be the maximum of path's length. Besides, the distance between RJ-45 and magnetic should be as short as possible.
- Crystal shouldn't be placed close to Input /Output ports, edge of PCB board and magnetic devices. The most important thing is that crystal should not be placed close to high-frequency devices or traces, such as MII interface signals, Tx+/-, Rx+/- and Power signals.

3. Power and Ground Plane

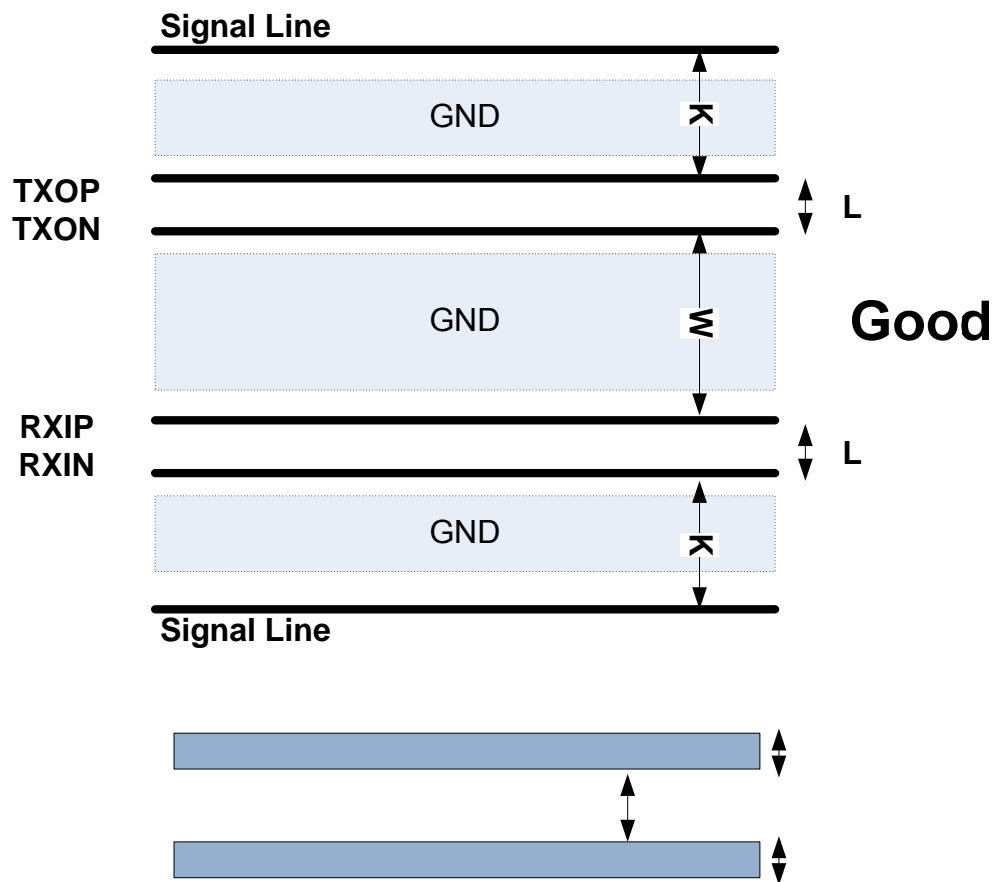
- It is better that do not try to partition GND at all.
- No power and GND planes can be underneath the isolated area for the RJ-45 connector and magnetic. Also RJ-45 connector has its isolated GND (Chassis GND) to connect to RJ-45's case.
- Try to keep the GND plane as large as possible.

4. Trace Routing

- Avoid right angle signal trace:



- For Tx+/-, Rx+/- traces:
- Avoid signal noise or loss on these traces.
- Tx+ & Tx- should be equal length to each other.
- Rx+ & Rx- should be equal length to each other.
- The line width and distance between Tx+/- and Rx+/- :



D: Line width is as wide as possible in the range of (6mil ~ 12 mil), ex: 8mil.

L: Width between differential pair should be small, ex: 4mil.

W: Isolation width between TX+/- and RX+/- is as wide as possible, ex: 30mil.

GND used as isolation is recommended.

K: Isolation width between TX/RX and noisy signal/power is as wide as possible, ex: 30mil. GND used as isolation is recommended.

For W & K need better isolation, ex: shielding with GND.

Try to avoid via for TX+/-, RX+/- traces. Via will degrade signal quality.

- Try to avoid digital signals (like Clocks or MII signal traces) interfere with analog signals (like Tx+/-, Rx+/- traces) and power lines. Never running noisy digital signals in parallel with TX+/- and RX+/-.

- The traces of power, ground, and those need de-couple cap should be shorter and wider.
- For some critical signals, clock and the other high speed signal traces should be as short and wide as possible. (Surely that is compared with normal signal traces.) And it's better having the GND plane under them, and it is even better with the GND plane around it.
- De-couple cap should be placed as close to IC as possible, and the traces should be short.
- Try to keep the distance between Tx+/- & Rx+/- differential pairs for good isolation. When these two pair of traces run together in parallel, don't place them too close for unwanted interference. Shielding by GND planes can get a better isolation to these two differential pairs.
- The signal trace length difference between Tx+ and Tx- (Same as Rx+ and Rx-) should be kept as small as possible, better within 1 inch.

5. For better analog performance

- Both Analog GND pins and Digital GND pins must maintain a good GND return path. One GND plane is recommended.
- When using 25MHz crystal as clock source, the spec of crystal is better under 50ppm

6. ESD Protecting

For ESD protection, we suggest to keep a distance(D) at least 80 mil for good isolation, which avoid ESD energy jumping by traces nearby IC. (See Figure 1)