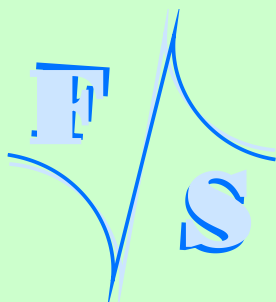
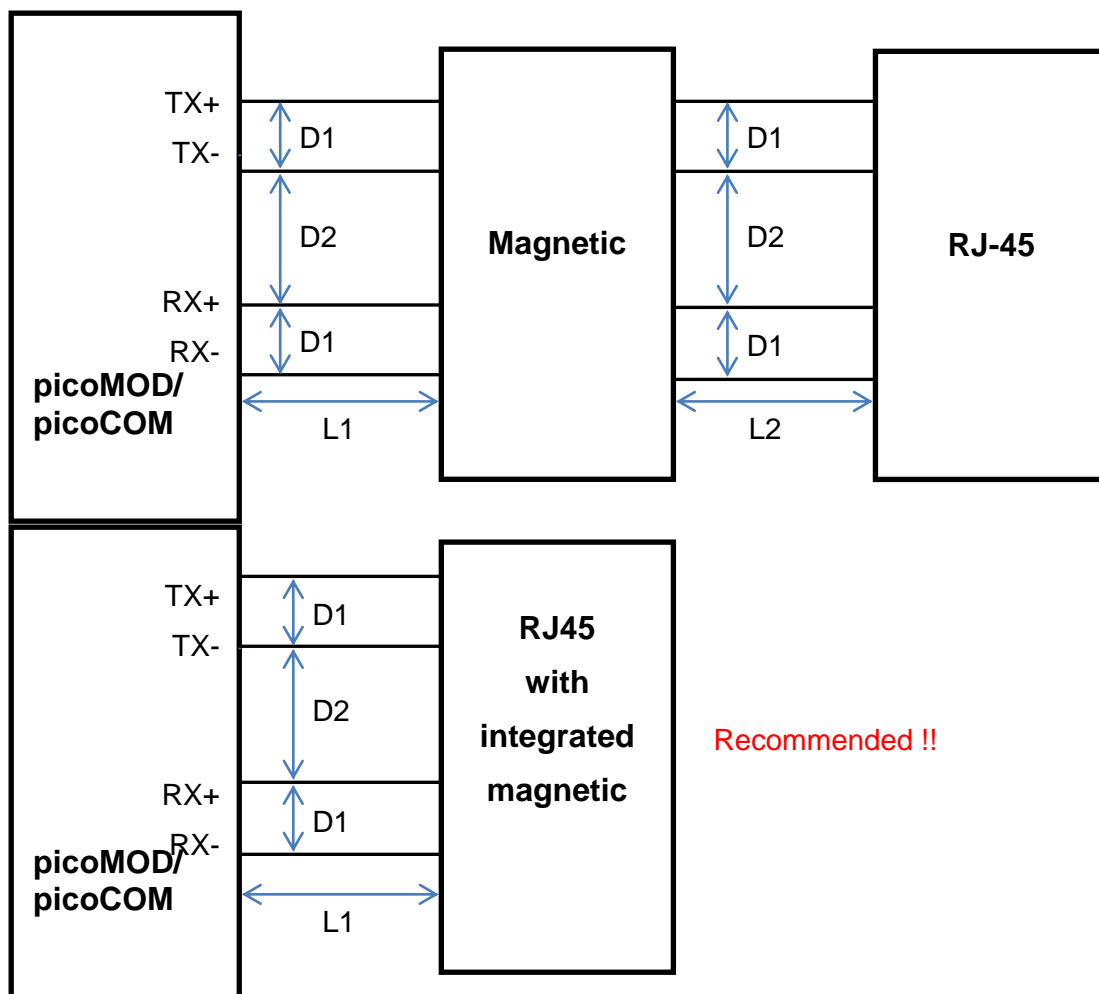


Ethernet routing

Guidelines V1.10

(2011-09-20)





Reference notes for routing Ethernet circuit on your base board

- ✓ Signals must be routed as differential pairs with 100 Ohm $\pm 10\%$ impedance. Differential impedance can be calculated by your PCB manufacturer. Distance D1 is 6.8mil (150..200 μ m). The trace width should be adjusted accordingly to yield the required trace impedance. Length mismatch in pair should be as small as possible and not larger as 50mil (1,25mm)
- ✓ Length difference between the pairs should be <700mil (18mm).
- ✓ Routing without vias is preferred. If the PCB layout really needs to use vias on the differential pairs, please match the vias to keep the differential pairs balanced.
- ✓ Distance D2 between pairs should >200mil (5mm) to avoid crosstalk.
- ✓ Distance to other signals and ground areas should >100mil (2,5mm).
- ✓ The power plane and digital ground plane should not be placed under the magnetic and RJ-45 connector.
- ✓ Length L1 between module and magnetic should be as short as possible and not larger then 4 inch (100mm).
- ✓ Length L2 between magnetic magnetic an RJ45 connector should be as short as possible and not larger then 1 inch (25 mm). We recommend to use a RJ45 with integrated transformer.
- ✓ More information about differential pairs can be found at http://www.usb.org/developers/docs/hs_usb_pdg_r1_0.pdf
Most guidelines are valid for Ethernet in principle.