

AX58101(2) EtherCAT Slave Controller Application Design Note

Revision 1.00
Sep 12th, 2025



Revision History

Revision	Date	Description
1.00	2025/09/12	Initial release

Table of Contents

1. Introduction.....	5
2. EEPROM Selection & Bootstrap Pins Considerations.....	5
3. Oscillator Selection Considerations	7
4. Ethernet PHY Selection Considerations	8
5. Ethernet Magnetic Selection Considerations.....	9
6. 4-Layer PCB Design	10
7. Power and Ground Planes Considerations	10
8. Ethernet Magnetic Layout Considerations	13
9. Thermal Considerations.....	15
9-1. Improve the Cooling Plane.....	15
9-2. Improve the Air Convection.....	15
10. EMI Considerations.....	16
11. ESD, EFT and Surge Considerations	18

List of Figures

Figure 1.	I ² C EEPROM Reference Circuit.....	5
Figure 2.	AX58101(2) Bootstrap Pins Reference Circuit	6
Figure 3.	25MHz Crystal Reference Circuit	7
Figure 4.	The principle connection between AX58101(2) ESC optional MII interface and Ethernet PHY .	8
Figure 5.	RJ-45 + Transformer Reference Circuit	9
Figure 6.	An Example of 4-Layer PCB Design	10
Figure 7.	Typical Chassis/Digital Ground Planes	10
Figure 8.	Typical Digital/Analog Power Planes for Single RJ-45 with Integrated Magnetic	11
Figure 9.	An Example of Power Pins and Decoupling Capacitors Circuits	12
Figure 10.	Ethernet TX± and RX± Differential Pairs Layout.....	13
Figure 11.	EMI Consideration Reference Circuit	17
Figure 12.	An Example of Separated Magnetic and RJ-45 Connector Magnetic Circuit for ESD, EFT Considerations	18
Figure 13.	A Sample ESD, EFT Protection Circuit for Separate RJ-45 + Magnetic	19
Figure 14.	A Sample of isolated DC-DC converter for ESD, EFT and Surge Protection.....	19

1. Introduction

The AX58101(2) is a 2/3/4-port EtherCAT Slave Controller (ESC), licensed from Beckhoff Automation, with two integrated Fast Ethernet PHYs which support 100Mbps full-duplex operation and HP Auto-MDIX. The AX58101(2) provides a cost-effective solution for industrial automation, motion/motor/digital I/O control, Digital to Analog (DAC)/Analog to Digital (ADC) converters control, sensors data acquisition, robotics, etc. industrial fieldbus applications.

This document indicates important information about external components selection, schematic and PCB layout design notes. ASIX Electronics highly recommends designers to read through this design note firstly before starting the hardware designs.

2. EEPROM Selection & Bootstrap Pins Considerations

The AX58101(2) supports I²C EEPROM with EEPROM size from 1 Kbit (128 bytes) to 4 Mbit (500Kbytes). The AX58101(2) HWCFGEE contents from offset 0x00 to 0x7F are mandatory, as well as the general category (at least minimum I²C EEPROM size is 2Kbit, and for the complex devices with many categories should be equipped with 32 Kbit EEPROMs or larger). Please refer to Section 3 of AX58101(2) datasheet to select a proper I²C EEPROM part based on AX58101(2) LED_RUN/EEP_SIZE pin bootstrap configuration.

Pins		Signal Name	Description
LQFP80	LQFP128		
79	126	EEP_SIZE	I ² C EEPROM Size 0: 1 Kbit to 16Kbit 1: 32Kbit to 4Mbit

The following is the I²C EEPROM reference circuit of AX58101(2) applications.

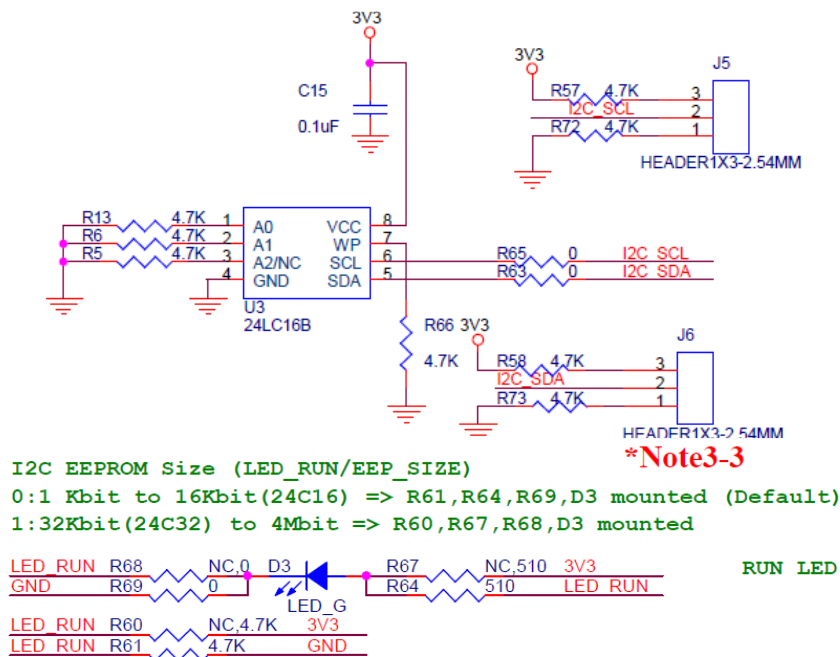


Figure 1. I²C EEPROM Reference Circuit

3. Oscillator Selection Considerations

The following are the specification of NSK NXX25.000AE12F-KAB6-12 SMD 25MHz crystal with CL 12pF, ESR 40 Ohm and Drive Level 100uW.

Parameter	Symbol	Typical Value
Nominal Frequency	Fo	25.000000MHz
Oscillation Mode		Fundamental
Frequency Tolerance (@25°C)		±20ppm
Equivalent Series Resistance	ESR	40 Ohm max.
Load Capacitance	CL	12pF
Driver Level		100uW
Operation Temperature Range		-40°C ~ +105°C
Aging		±3ppm

The following is an example of the 25MHz crystal clock circuit. The 1M feedback resistor is optional for 25MHz crystal circuit. Please reserve the R75, R80 resistors circuit for flexible designs.

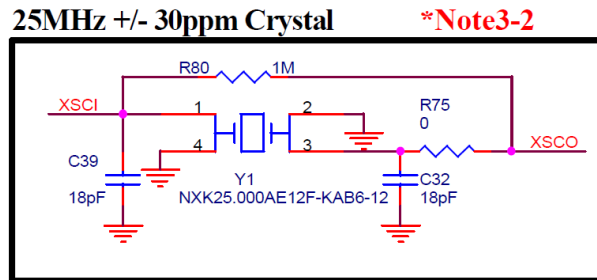


Figure 3. 25MHz Crystal Reference Circuit

Note: Please make sure the XSCO 25MHz clock output signals are within 25MHz +/- 20ppm. If the XSCO 25Mhz clock output signals are out of the specification, please fine-tune the load capacitors (C32, C39) to meet the specification.

4. Ethernet PHY Selection Considerations

The AX58101(2) ESC, licensed from Beckhoff Automation, supports two embedded PHYs and one optional MII interface (while the AX58102 supports two MII interfaces) to provide flexible network topologies. The optional MII interface of the AX58101(2) ESC omits a transmit FIFO in order to reduce processing and forwarding delays. Due to this design, the ESC has additional requirements for the Ethernet PHY; however, these requirements can be easily fulfilled by several PHY vendors. Please refer to Beckhoff's PHY Selection Guide to select a suitable Ethernet PHY.

The principle connection between the AX58101(2) ESC optional MII interface and the Ethernet PHY is as follows: the Ethernet PHY and the ESC must share the same quartz clock source. Since the ESC does not incorporate a TX FIFO, the TX_CLK is not connected. The TX signals can be delayed inside the ESC by configuring the AX58101 bootstrap pins for TX_CLK phase shift compensation. The LINK signal is connected to the PHY LED output to indicate a 100 Mbps (Full Duplex) link status.

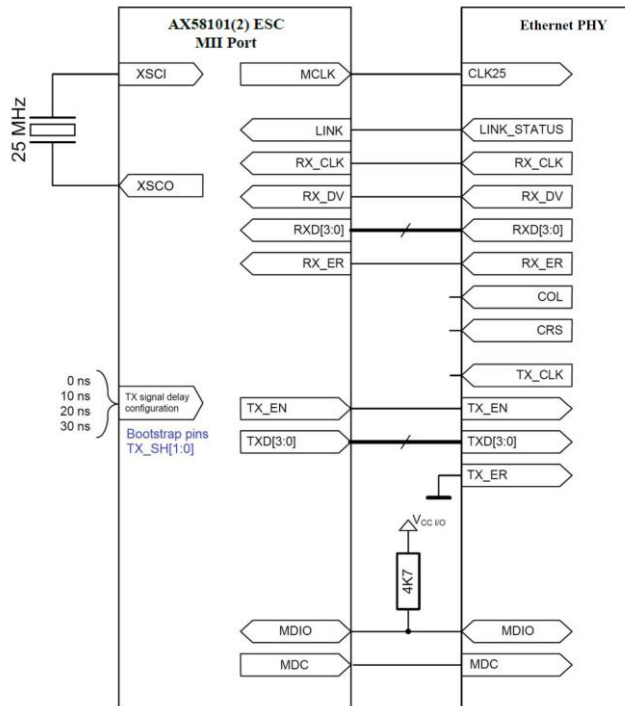


Figure 4. The principle connection between AX58101(2) ESC optional MII interface and Ethernet PHY

For external PHY selection, please select a PHY device that supports TX_CLK fixed phase and PHY addr. at least 0~4.

Vendor / Device	ET1200 suitable	ET1100 suitable	IP Core suitable	# Ports	Basic HW test ¹	TX_CLK fixed phase ²	PHY addr. ³	PHY addr. offset ⁴	Link loss reaction time	Enhanced Link Detection	Auto-TX-Shift (IP Core only)	Comments
-----------------	-----------------	-----------------	------------------	---------	----------------------------	---------------------------------	------------------------	-------------------------------	-------------------------	-------------------------	------------------------------	----------

6. 4-Layer PCB Design

We strongly suggest customers to design AX58101(2) on the printed circuit board (PCB) with at least 4 layers. The 4-layer PCB design can help reduce some potential EMI, thermal and signal integrity issues, etc.

The following is an example of 4-Layer PCB design.

Layer 1	Component (Top)	Magnetic and major signals
Layer 2	Ground	Digital/analog ground planes
Layer 3	Power	Digital/analog power planes
Layer 4	Component (Bottom)	Magnetic and other signals

Figure 6. An Example of 4-Layer PCB Design

7. Power and Ground Planes Considerations

1. Digital GND and Analog GND can be placed together in one-piece so as to enlarge the GND, and thus helping dissipation of noises.
2. The RJ-45 chassis ground and the digital ground should be isolated through a 1206 1M ohm resistor and a 1206 1000pF/2KV decoupling capacitor, and the gap between the 1206 package should be larger than 60 mils.

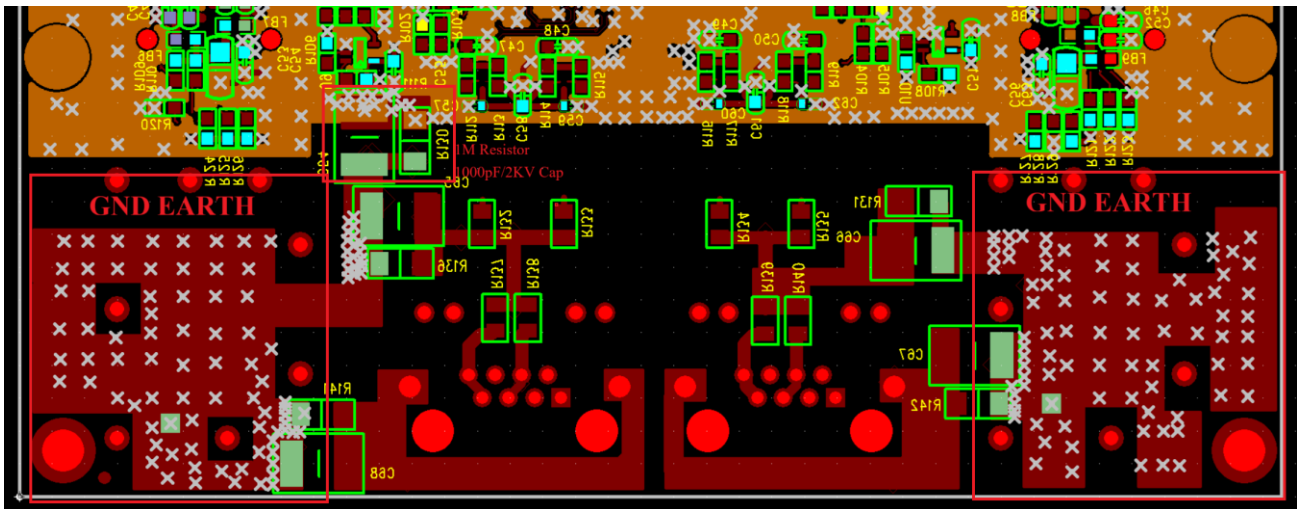


Figure 7. Typical Chassis/Digital Ground Planes

3. All the digital and analog power planes for different voltage supplies should be handled separately with different blocks to supply power to IC chip and peripheral components.

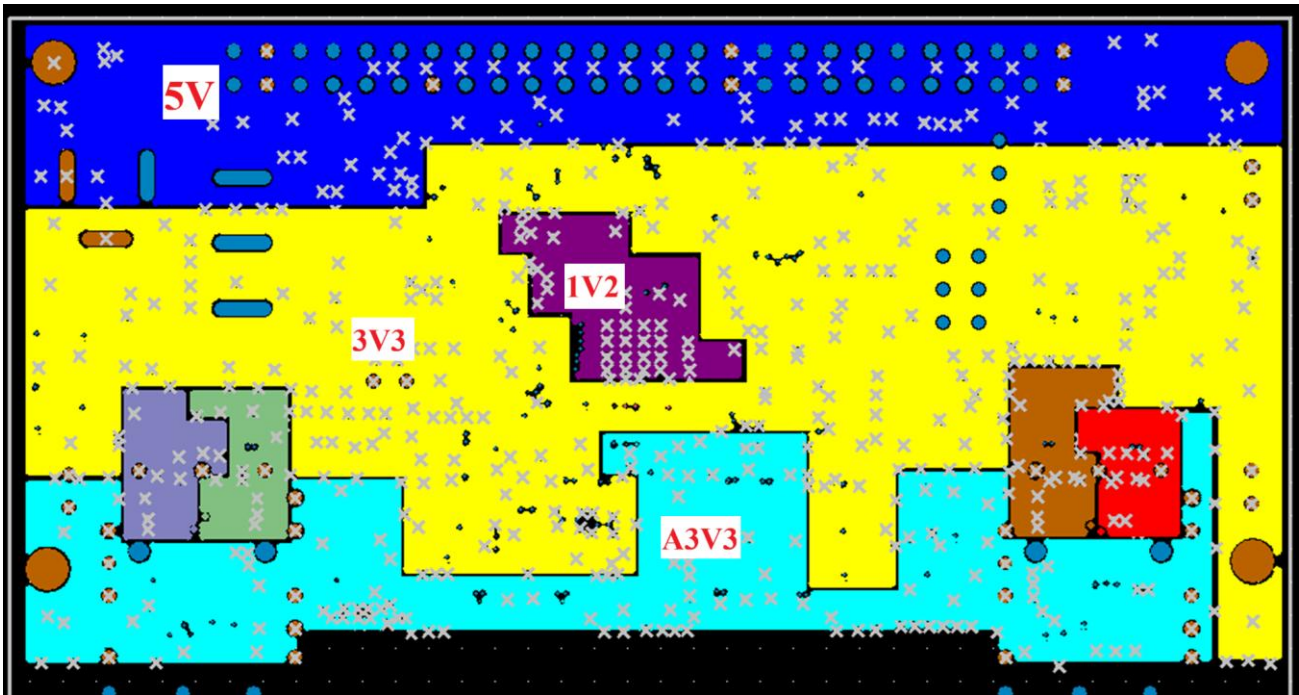
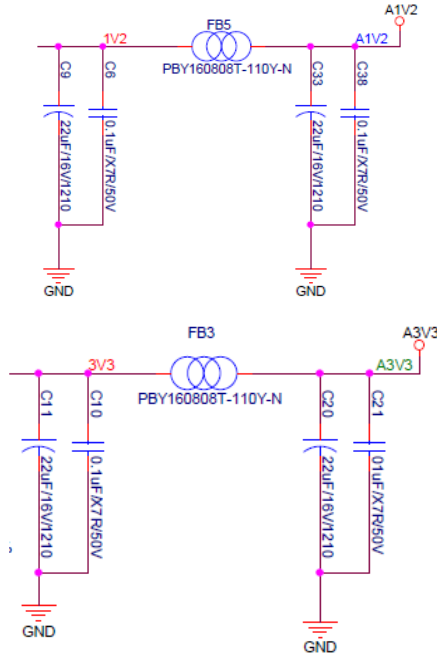


Figure 8. Typical Digital/Analog Power Planes for Single RJ-45 with Integrated Magnetic

Note: The above figures are the Digital and Analog Power Planes diagram of an illustrative PCB board design. For exact layout pattern, ASIX Electronics provides the PCB layout and Gerber files of AX58101(2) EVB for customer reference.

4. The digital and analog power planes should be isolated with a Ferrite Bead to isolate the noise source, and all power planes should be implemented with a large compensating capacitor to provide a stable power source.



5. All power pins should be implemented with a decoupling capacitor, and the decoupling capacitor should be as close to the respective power pin of AX58101(2) as possible.

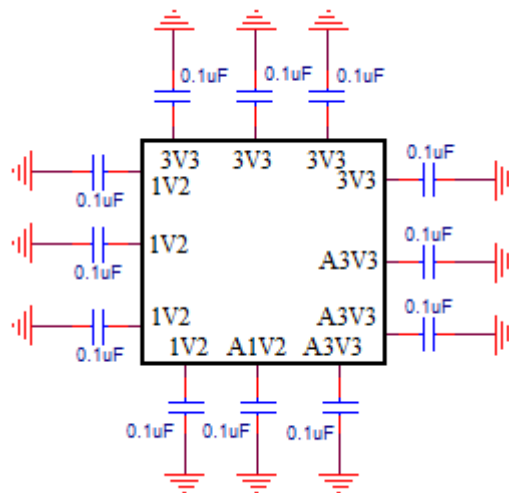


Figure 9. An Example of Power Pins and Decoupling Capacitors Circuits

8. Ethernet Magnetic Layout Considerations

This section describes some general Ethernet layout considerations for the differential signals of the Ethernet controller, the Ethernet magnetic and the RJ-45 connector.

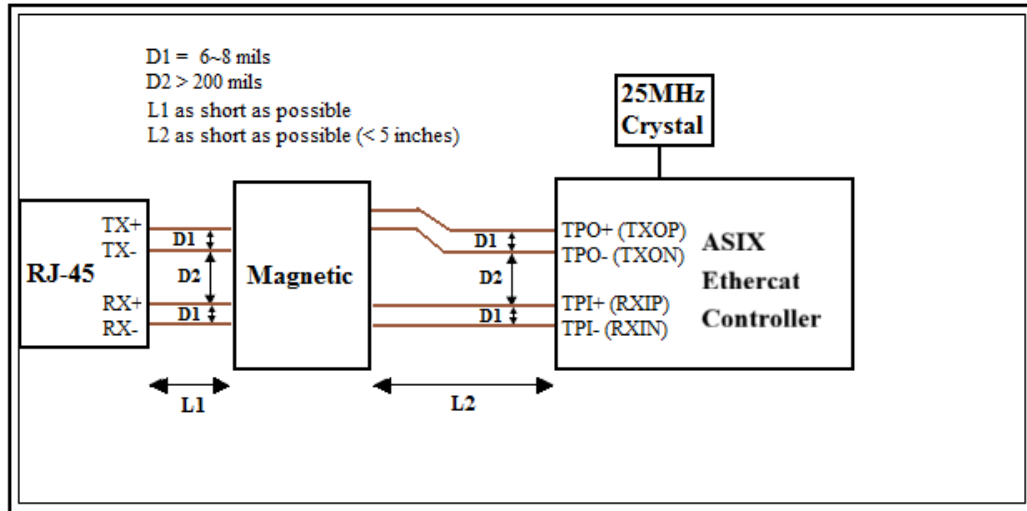


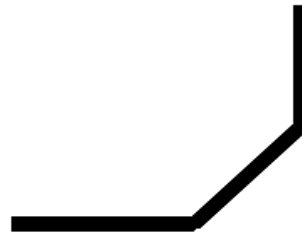
Figure 10. Ethernet TX± and RX± Differential Pairs Layout

1. The ideal Ethernet differential impedance between TX + and TX- traces (between RX+ and RX-) should be 100 Ohm. The Ethernet differential impedance between the TX+/TX-/RX+/RX- trace and GND should be 50 Ohm respectively. The designer can use the trace width of TX+/TX-/RX+/RX- traces to fine tune the Ethernet differential impedance value (a wider trace has a smaller impedance value). If the PCB manufacturer provides TDR measurement, it helps control the PCB Impedance.
2. The crystal clock source and the switching noises from digital signals should be kept away from the TX± and RX± differential pairs.
3. The Ethernet magnetic should be placed as close to the RJ-45 connector as possible.
4. The Ethernet controller should be placed as close as possible to the magnetic. If there are some limitations on the PCB layout, the trace length from the Ethernet controller to the magnetic should not be longer than 5 inches.
5. The TX± and RX± differential pairs should be routed as close as possible. The trace spacing D1 between TX+ and TX- (or between RX+ and RX-) pair should be in 6 ~ 8 mils. The trace width should be adjusted accordingly to yield the required trace impedance.
6. The spacing D2 between the TX± and RX± differential pairs should be larger than 200 mils. If the PCB layout is really difficult to meet this requirement, the D2 spacing should be as larger as possible.
7. Route the TX± and RX± differential pairs as straight as possible and keep them in parallel for differential pairs.
8. Keep the trace length difference between the TX+ and TX- (or RX+ and RX-) pair within 700 mils.
9. Route the TX± and RX± differential pairs running symmetric, equal length and close whenever possible.

10. Avoid using vias on the traces of the TX± and RX± differential pairs. If the PCB layout really needs to use vias on the differential pairs, please match the vias to keep the differential pairs balanced.
11. The power plane and digital ground plane should not be placed under the magnetic and RJ-45 connector.
12. Avoid routing the signal trace with right angle, instead, the signal trace should be routed with multiple 135° angles.



Worst



Better

9. Thermal Considerations

This section describes some information about how to reduce the operating temperature on the AX58101(2) applications.

9-1. Improve the Cooling Plane

There are two major heat sources on the AX58101(2) applications. One is the AX58101(2) controller and the other is the external voltage regulator.

You can connect the VCC/GND pins of the AX58101(2) controller with wide traces to the respective power/ground planes to increase the cooling effect and reduce the operating temperature of the AX58101(2) controller.

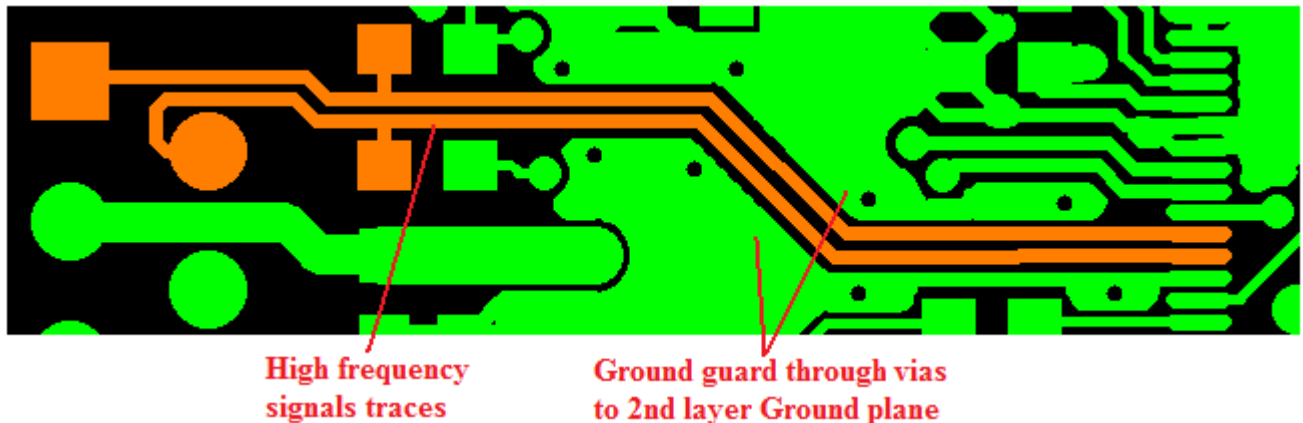
You can also add a cooling plane for the external voltage regulator to reduce the operating temperature of the external voltage regulator.

9-2. Improve the Air Convection

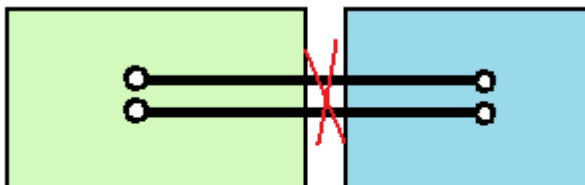
You can place the AX58101(2) controller at the location with good air convection and stay away from the high-operating-temperature IC such as voltage regulator or external MCU/DSP to reduce the operating temperature of the AX58101(2) controller.

10. EMI Considerations

1. The high frequency signals traces such as Ethernet differential signals, clock signals, etc. shall be surrounded by GND to prevent interference to other signal wires. Better EMI effect can be attained by a one-piece GND underneath the IC furnished with PTH holes to enlarge the GND area.



2. The chosen connector must be shielded so that EMI integrity of the design is not compromised. The shield must be electrically connected to chassis ground to extend the chassis barrier for high frequency emissions. If an unshielded connector were used, the EMI would pass through the nylon material of the connector. The shield will also prevent less external EMI from entering the chassis.
3. To reduce electromagnetic emissions and susceptibility, it is imperative that traces from the transceiver to the magnetic sand from the magnetic to the RJ-45 be routed as differential pairs. The objective is to close the loop area formed by the two conductors. The radiated field from the loop or the voltage picked up by the loop by external fields is governed by the field strength and the area formed by the two conductors. Reasonable board design uses 5~10 mils trace widths separated by 10 mils. Transmit differential pairs should be routed adjacent to a VDDO power plane.
4. To avoid crossing any signal traces over any reference plane cuts; otherwise, it might cause some unpredictable EMI problems.



Don't cross any signal traces over any reference plane cuts!!

- It is essential to maintain the continuous stable ground plane when board design to protect EMI emissions. The PCB designer should look upon the path currents, which take on the ground reference plane, and reduce the loop area. The continuous ground plane helps provide for a low inductance signal return path, and helps maintain a stable power supply. Simply, the most effective method of reducing EMI causes is to decrease the loop area. The main target of ground plane is designed to carry DC current only. Do not use this area as a return path for high speed signals.

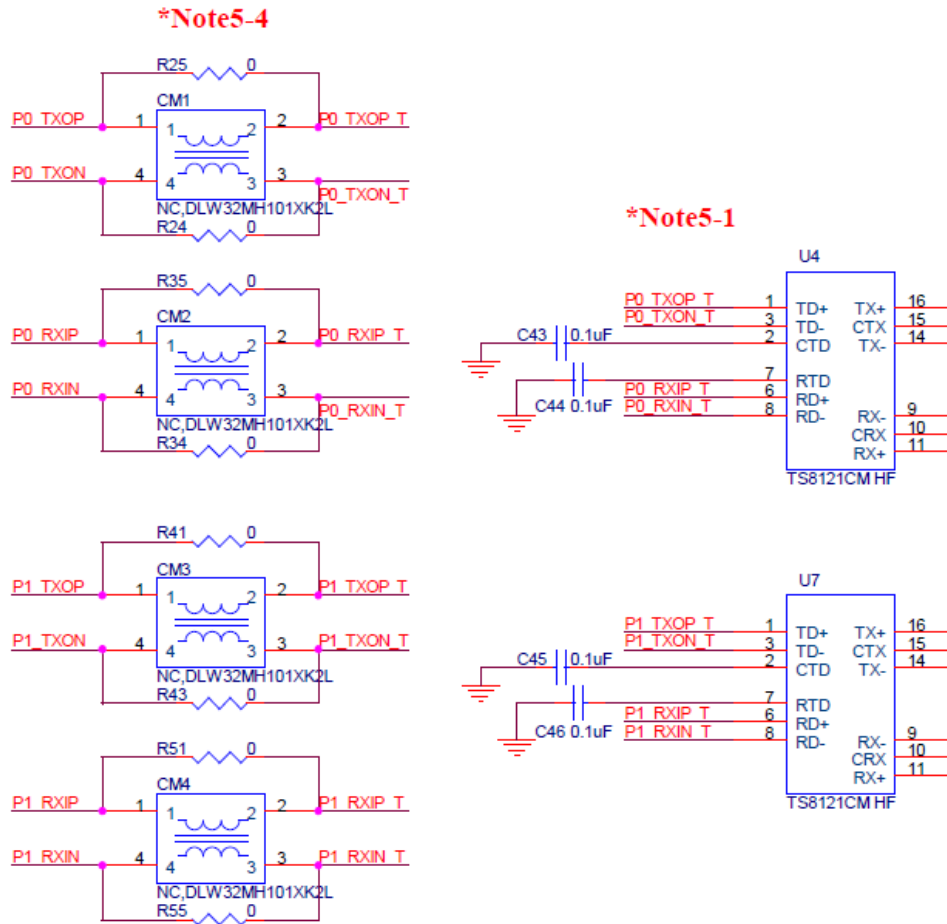


Figure 11. EMI Consideration Reference Circuit

11. ESD, EFT and Surge Considerations

This section describes some information about the ESD, EFT and Surge design guideline. Users can refer to the following circuit to avoid the ESD, EFT and Surge issue.

The high-energy pulses can cause abnormal system behavior and/or damage the silicon chips. The high energy can enter the system through the RJ-45 cable. There are eight wires in the RJ-45 cable and are grouped into 4 pairs as 2 active pairs (pin 1,2 & 3,6) and 2 unused pairs (pin 4,5 & 7,8).

If the high energy enters the system through the active pairs (pin 1,2 & 3,6) of RJ-45 cable, the Ethernet transformer will isolate the high energy. The unwanted high energy will be redirected to Chassis ground through a high voltage capacitor (2KV). If the high energy enters the system through the unused pairs (pin 4,5 & 7,8) of RJ-45 cable, these pins are connected to Chassis ground through a high voltage capacitor (2KV) so the unwanted high energy will be redirected to Chassis ground too.

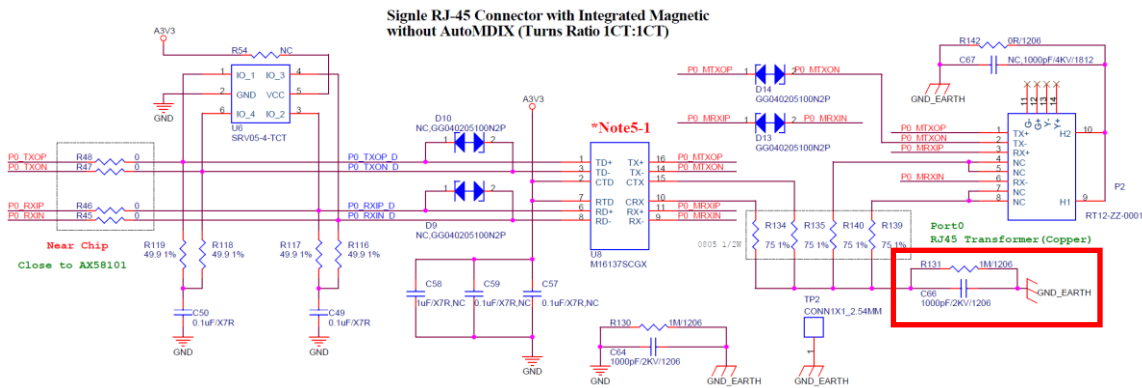


Figure 12. An Example of Separated Magnetic and RJ-45 Connector Magnetic Circuit for ESD, EFT Considerations

The fatal discharge may generate from a charged Ethernet cable, or out from outside devices such as human body, etc. Ethernet ICs become vulnerable to damage from Electrostatic Discharge within those unpredictable factors. Protecting Ethernet interfaces from cable discharges can create a challenge for board designers. Be sure a good protective circuit must effectively fasten a transient to a safe voltage, and must present an acceptable capacitive load on high-speed differential transmission lines as well.

Below are some sample circuits for ESD, EFT protection. Please refer to the ESD, EFT Protection TVS Diode datasheet for detailed ESD, EFT protection circuit.

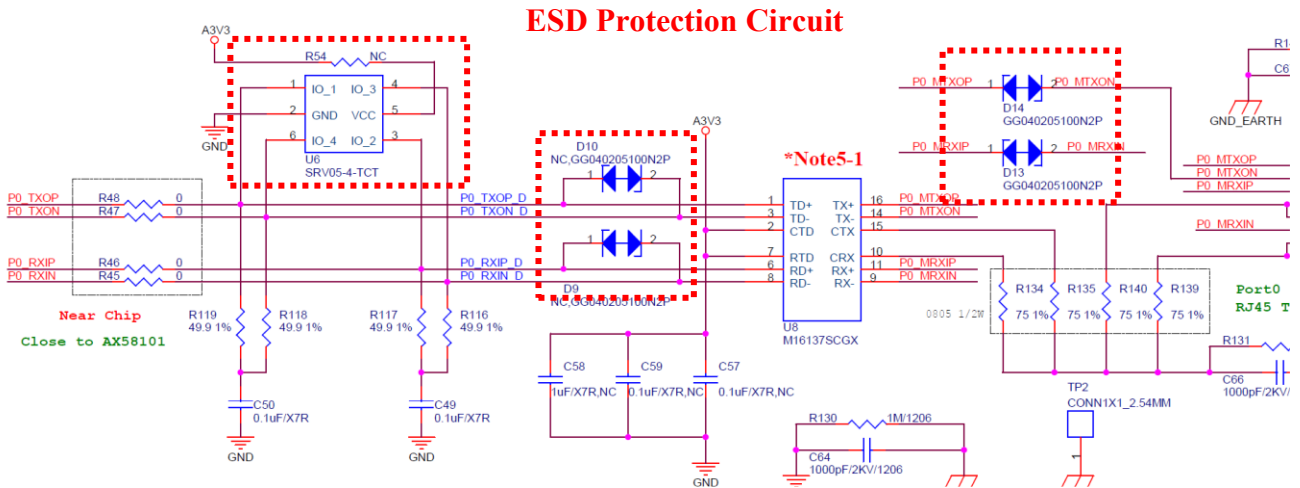


Figure 13. A Sample ESD, EFT Protection Circuit for Separate RJ-45 + Magnetic

The EtherCAT system is usually connected to an external DC power supply to provide power to the EtherCAT modules. The input protection circuits are required to protect the EtherCAT system from various faults that may occur either on the field or the EtherCAT system side.

The isolated DC-DC converter ESD, EFT and Surge protection solution is shown in Figure 13. It requires only a single isolated DC-DC converter to protect the EtherCAT system from ESD, EFT and Surge.

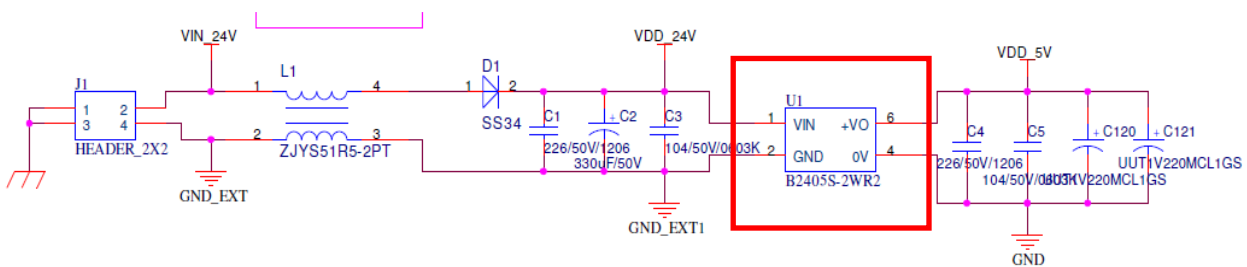


Figure 14. A Sample of isolated DC-DC converter for ESD, EFT and Surge Protection



AX58101(2) EtherCAT Slave Controller Application Design Note

Copyright © 2025 ASIX Electronics Corporation. All rights reserved.

DISCLAIMER

No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose, without the express written permission of ASIX. ASIX may make changes to the product specifications and descriptions in this document at any time, without notice.

ASIX provides this document “as is” without warranty of any kind, either expressed or implied, including without limitation warranties of merchantability, fitness for a particular purpose, and non-infringement.

Designers must not rely on the absence or characteristics of any features or registers marked “reserved”, “undefined” or “NC”. ASIX reserves these for future definition and shall have no responsibility whatsoever for conflicts or incompatibilities arising from future changes to them. Always contact ASIX to get the latest document before starting a design of ASIX products.

TRADEMARKS

ASIX, the ASIX logo are registered trademarks of ASIX Electronics Corporation. All other trademarks are the property of their respective owners.

EtherCAT® is a registered trademark and patented technology, licensed by Beckhoff Automation GmbH, Germany.



**4F, No.8, Hsin Ann RD., Hsinchu Science Park,
Hsinchu, Taiwan, R.O.C.**

TEL: +886-3-5799500

FAX: +886-3-5799558

Email: support@asix.com.tw

Web: <https://www.asix.com.tw>