



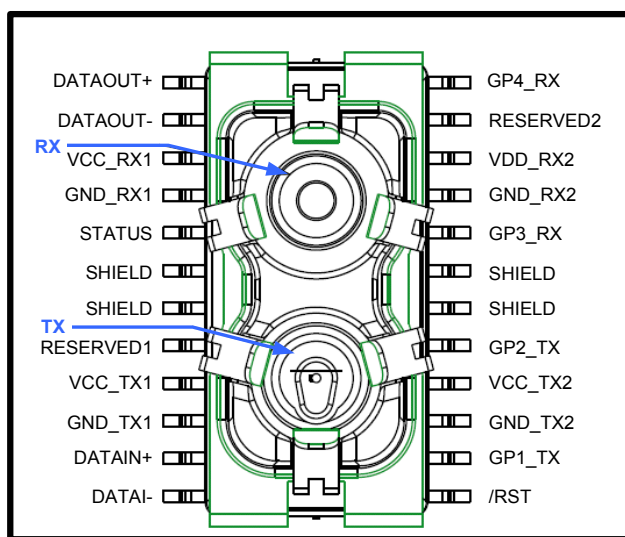
Application Note MLX75605CA

Revision	Date	Author	Change
1.0	15/10/2008	ADE	Creation
1.1	26/10/2008	CVA	Addition of footprint dimensions

Introduction

This document describes the connections between MLX75605 MOST150 Fiber Optics Transceivers and MOST control part. It explains how to connect the different pins, their role, necessary external components, and some layout recommendations.

Pin List and Descriptions





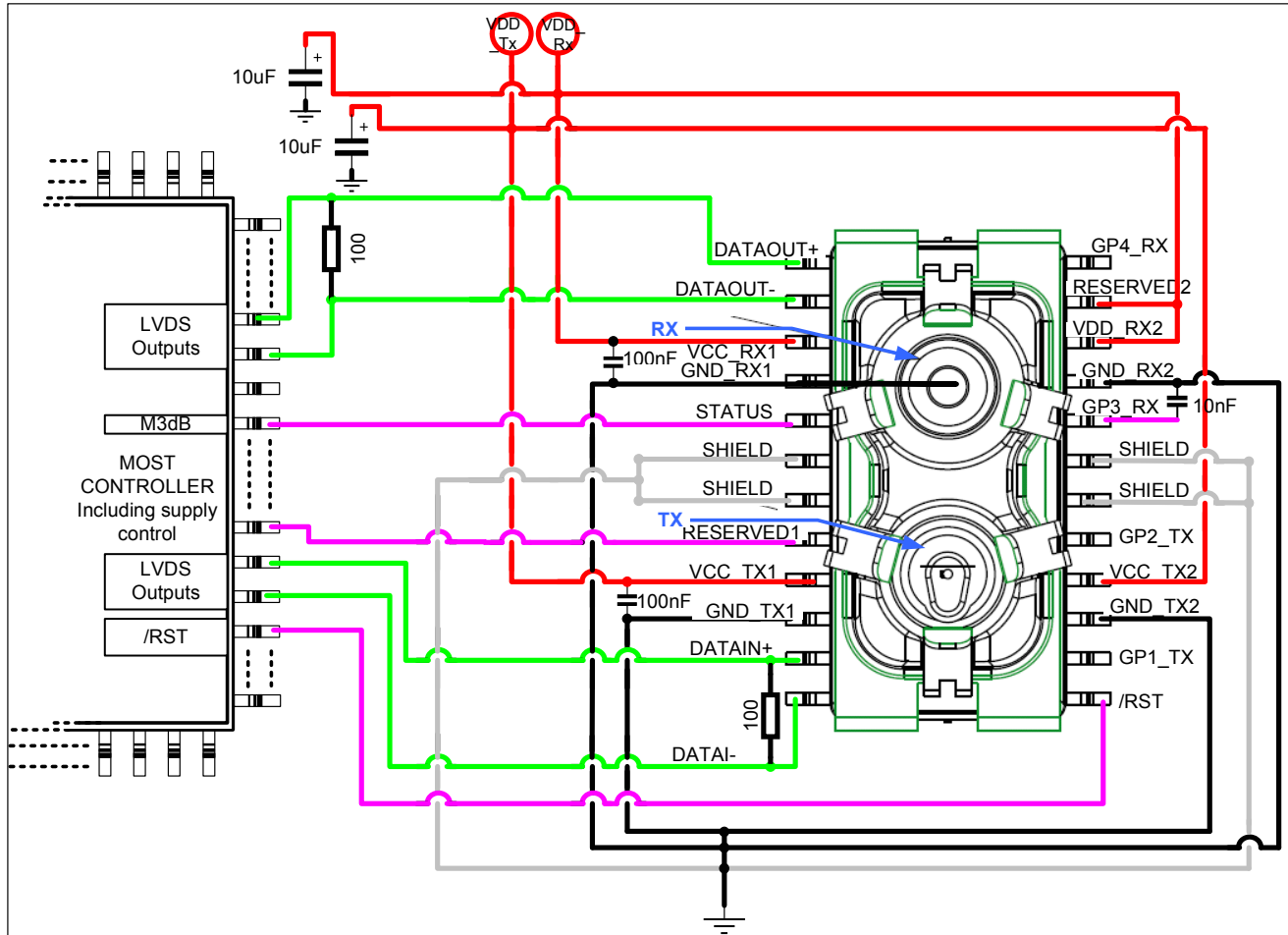
Pin Number	Pin name	Description	Connection
1	DataOUT Plus	LVDS Positive Output. Routing should be mirrored together with minus input.	To LVDS Plus input of MOST controller
2	DataOUT Minus	LVDS Negative Output Routing should be mirrored together with plus input.	To LVDS minus input of MOST controller
3	VCC_Rx1	Supply voltage for Receiver	Connected to Rx VDD
4	GND_Rx1	Ground for Receiver	Connected to Rx GND
5	STATUS	Indicates if modulated light is received within operation frequency range. If the received light power level and frequency is greater than threshold, then voltage on the status pin is «LOW». With the status pin the power supply of the whole MOST devices can be switched ON.	Connected to Status input of controller
6	Shield	Shield pin connected to die pad of package	Connected to Sh_GND
7	Shield	Shield pin connected to die pad of package	Connected to Sh_GND
8	Reserved1	Connect to GND through 0-ohm resistor. Used for factory test purpose. M3dB function	Connected to possible M3dB function of controller (Not available on MLX75605CA revision)
9	VCC_Tx1	Supply voltage for Transmitter	Connected to Tx VDD
10	GND_Tx1	Ground for Transmitter	Connected to Tx GND
11	DataIN Plus	LVDS Negative Input	From LVDS minus output of MOST controller
12	DataIN Minus	LVDS Positive Input	From LVDS plus output of MOST controller
13	/RST	Optical Output power disabling. Active-low logic signal disables the optical output	Connected to /RST of MOST controller
14	Tx_GP1	Currently used for Test output pin. Should be left unconnected	Should be floating
15	GND_Tx2	Ground for Transmitter	Connected to Tx GND
16	VCC_TX2	Supply voltage for Transmitter	Connected to Tx VDD



17	Tx_GP2	Factory Test input. Should left unconnected.	Should be floating
18	Shield	Shield pin connected to die pad of package	Connected to Sh_GND
19	Shield	Shield pin connected to die pad of package	Connected to Sh_GND
20	GP3_RX	Factory test output. Automatic Gain Control analog output Should be left unconnected	Should be floating. (Connected to 10nF for MLX75603CA only)
21	GND_Rx2	Ground for Receiver	Connected to Rx GND
22	VCC_Rx2	Supply voltage for Receiver	Connected to Rx VDD
23	Reserved2	Connect to VCC_RX2 through 0-ohm resistor. Used for factory test purpose.	Connected to Rx VDD.
24	GP4_RX	Should left unconnected.	Should be floating



Schematics





External Components

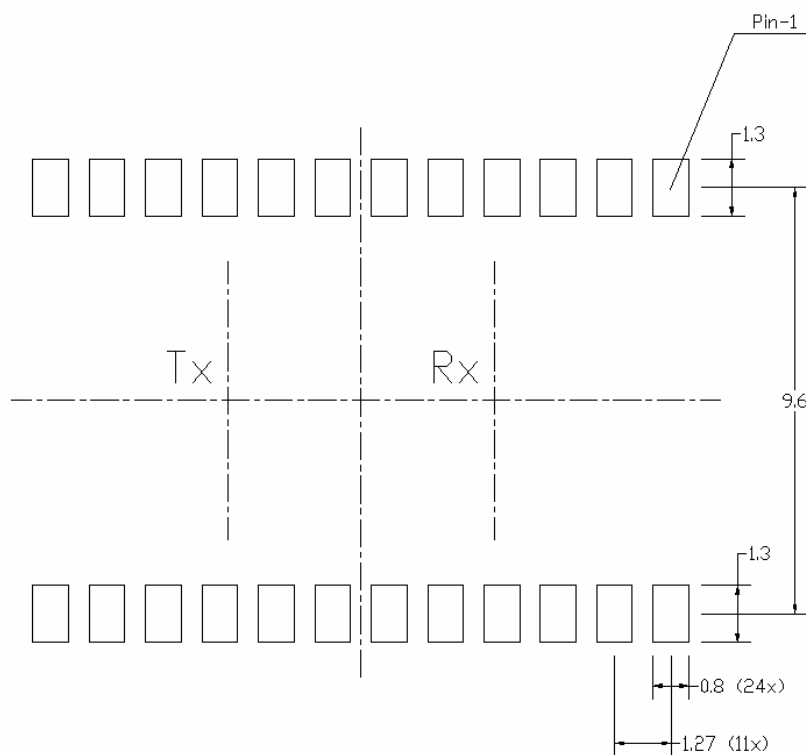
- To guarantee correct impedance matching between controller and transceiver, it is necessary to add 100ohms between both LVDS I/Os. To maximize their efficiency, the resistors have to be placed close to the LVDS receiver buffer. This means that, for data coming from transceiver, resistor has to be placed close to the MOST controller and for data arriving at transceiver, resistor should be placed close to this one.
- As Transmitter can be disabled by supply shut down, Receiver and transmitter supplies have to be fully decoupled.
- 10uF has can be used for global decoupling.
- Then, 100nF bypass capacitors can be connected for each device: one for Receiver chipset and one for transmitter. It is recommended that, these capacitors are with low equivalent serial resistance, in order to increase decoupling efficiency with high current. These capacitors have to be placed close to the device to minimize routing impact.
- For MLX75605CA only, a 10nF decoupling capacitor should be added on pin #20. This capacitor should be considered as a containment action for a small design weakness in Automatic Gain control on the receiver chip.

Layout Recommendations

- It is important to decouple supply lines to the maximum. This means that, supply lines have to be routed in a star configuration.
- For VDD side, it is recommended to short both VCC from each device, respectively transmitter and receiver, together routing under the device. Pinout of supply and ground lines have been made such a way that, they face each other. This allows direct connection with the shortest route.
- For GND side, it is advised to fully decouple ground from transmitter and ground from receiver to guarantee no cross talk.
- A ground plate has been placed all around FOT and especially around LVDS High speed tracks to fully isolate them.
- Moreover, device allows shield to be grounded to Ground plate. This allows optimizing EMc performance.



Footprint – recommendation



All dimensions given in mm.

11x means that this dimension is repeated 11 times in this footprint drawing (only the bottom half of the drawing)

Due to the symmetry axis this is also valid for the upper half of the drawing

24x means that this dimension is repeated 24 times in this footprint drawing (for the bottom half as well as for the upper half of the drawing)