DDR4 DIMM THROUGH-HOLE CONNECTOR
BOARD ROUTING RECOMMENDATIONS

TABLE OF CONTENTS

1.0 SCOPE

This specification covers the high-speed PCB routing recommendations of DQ and DQS signals for 151105 miniDIMM connector. The connector is surface mount type. The pins of the connector are soldered for mechanical retention to the PC board.

DISCLAIMER: Molex does not guarantee the performance of the final product to match the information provided in this document. All information in this report is considered proprietary, confidential and the property of Molex. This guide is not intended as a substitute for engineering analysis.

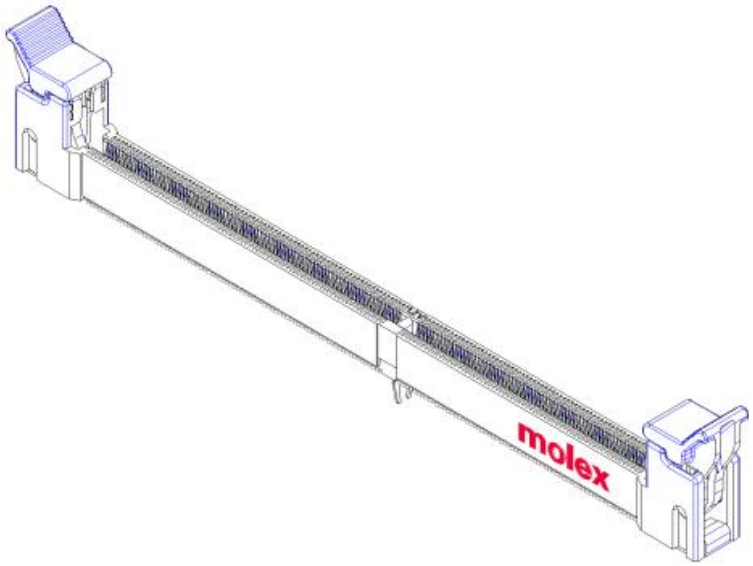


FIGURE 1

2.0 PC BOARD REQUIREMENTS

2.1 MATERIAL THICKNESS

The recommended PC board thickness shall be 1.57mm to 3.18mm. Suitable PC board material shall be glass epoxy (FR-4).

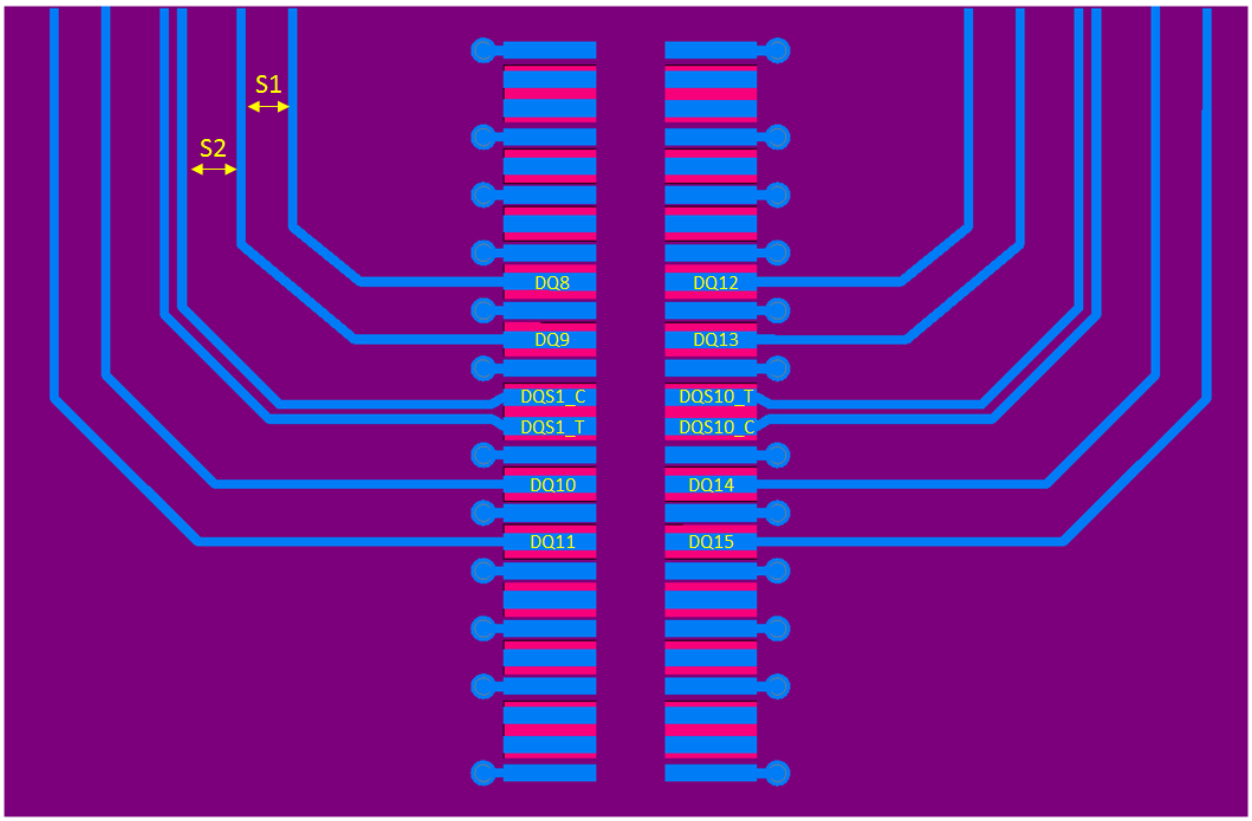
2.2 LAYOUT

The holes for the connector assembly must be precisely located to ensure proper placement and optimum performance of the connector assembly. Refer to the applicable Sales Drawing for the recommended hole pattern, dimensions and tolerances.

3.0 HIGH-SPEED ROUTING

3.1 GENERAL ROUTING EXAMPLE (other configurations are possible)

TOP LAYER MICROSTRIP ROUTING



Parameter	MM (INCH)
Single-ended trace width	0.1651 (0.0065)
Differential trace width / spacing	0.1016 (0.0040) / 0.1016 (0.0040)
DQ to DQ spacing (S1)	0.6604 (0.0260)
DQ to DQS spacing (S2)	0.6604 (0.0260)

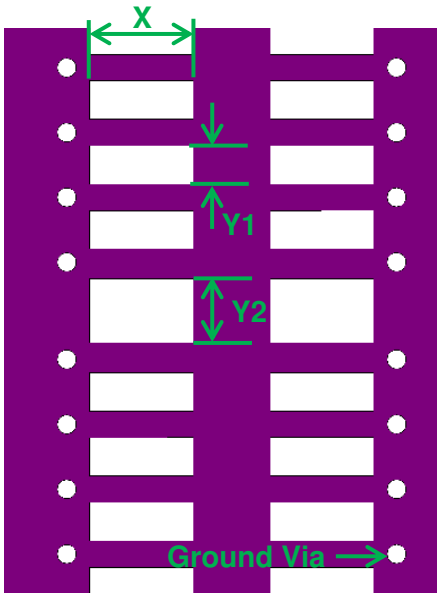
The routing examples shown are for reference only. They were extracted to show a portion that contains the routing of DQ and DQS signals. 2 layers were overlaid (1 signal and 1 reference ground layer).

Note: Dimensions can vary from recommendation to meet board thickness, routing and electrical requirements.

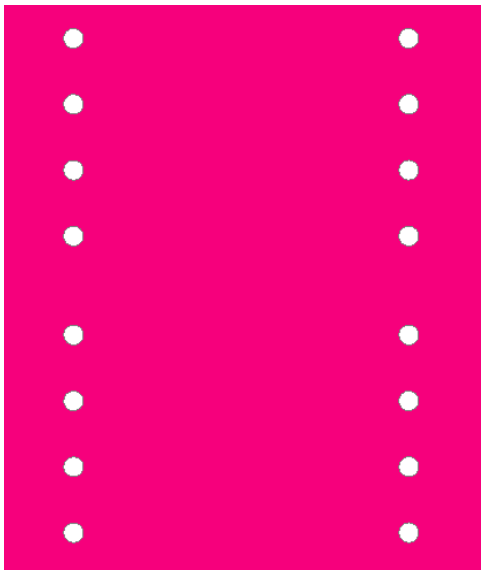
3.2 HIGH-SPEED REFERENCE PLANE ANTI-PAD

TOP LAYER MICROSTRIP ROUTING

Layer 2 Ground Plane



All Other Ground Planes



ANTI-PAD DIMENSION	MM (INCH)
X	1.6 (0.06299)
Y1	0.6 (0.02362)
Y2	1.0 (0.254)
GROUND VIA DIMENSION	MM (INCH)
DRILL DIAMETER	0.3 (0.01181)
FINISH DIAMETER	0.25 (0.00984)
PAD DIAMETER	0.45 (0.01771)

Note: Anti-pad was implemented for impedance matching. Dimensions can vary from recommendation to meet electrical requirements.

3.3 TRACE TO PAD ATTACHMENT AND GROUND VIA PLACEMENT

There are several ways to connect the traces to their corresponding signal pads. Two possible methods are illustrated in Figures 3 and 4.

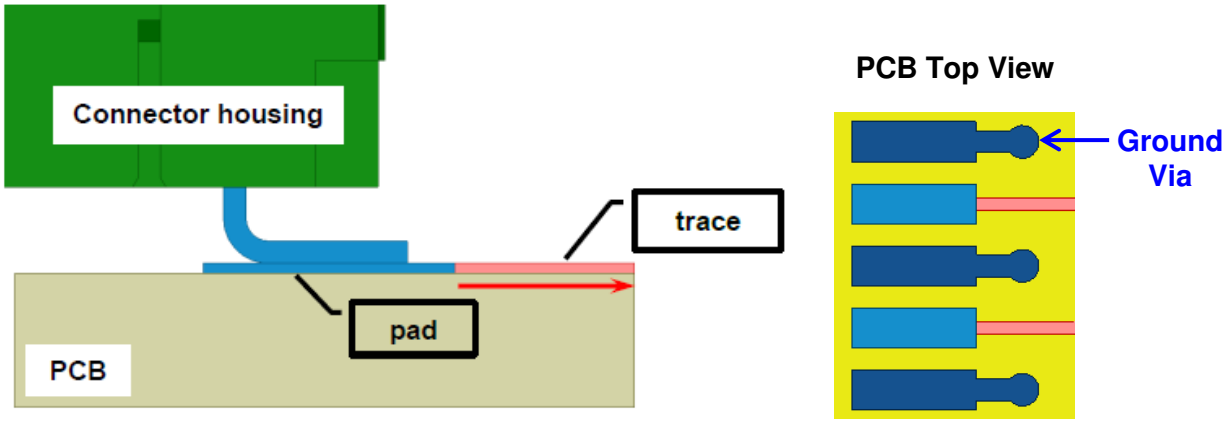


Figure 3

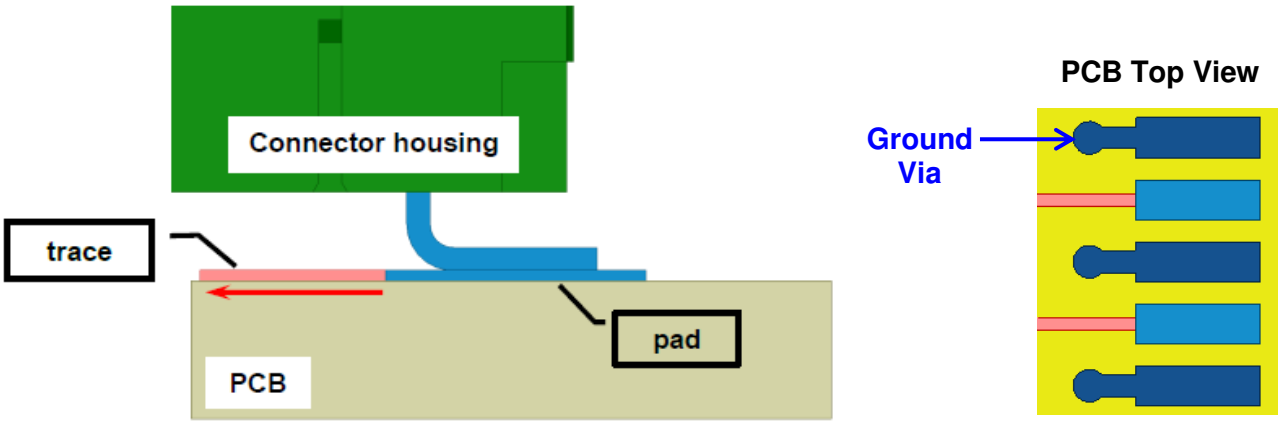
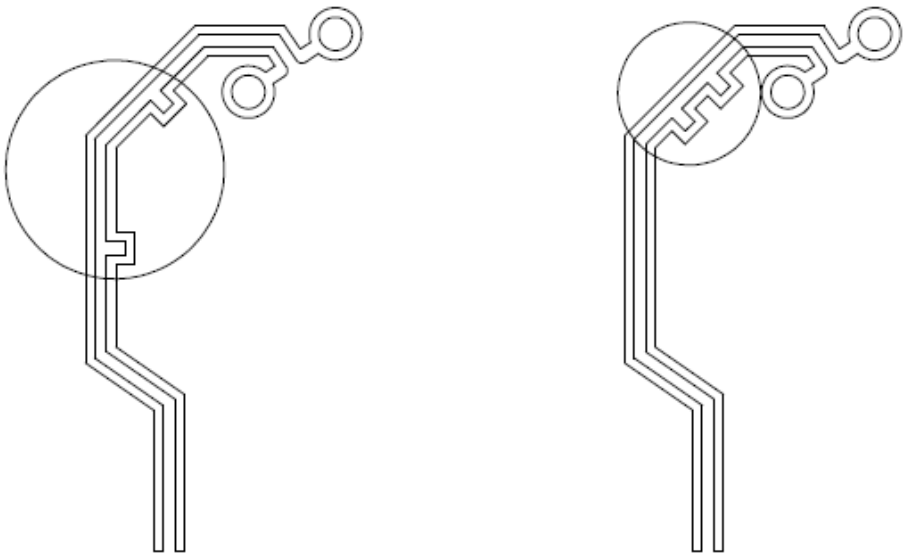


Figure 4

As seen in Figure 3, trace routed outwards from pad of connector will result in minimum pad stub while worst case pad stub occurs when trace is routed inwards as shown in Figure 4.

In addition, “run-away” ground vias from ground pads should follow the direction where the signal traces were attached to their corresponding signal pads.

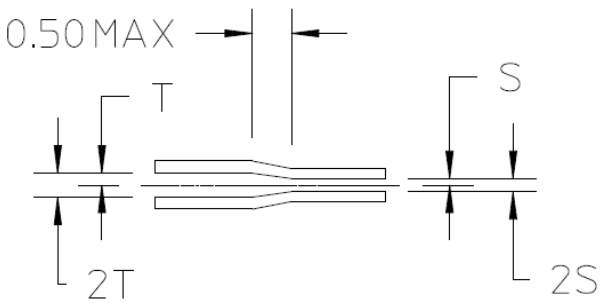
3.4 SKEW COMPENSATION



PREFERRED NOT RECOMMENDED

It is recommended that skew compensation be distributed verses grouped in one or more locations. This applies for both intra skew compensation of each DQS pair and inter skew compensation between all DQ and DQS within the same data lane group.

3.5 TRACE COMPARISON FOR DIFFERENTIAL SIGNALING



TRANSITION SHOULD BE SYMMETRIC