APPLICATION NOTE

APPLICATION NOTE
Design & Installation of High Speed Micro-D PCB Connectors

WRITTEN BY: _____ DATE: _____

     Keef Howerton

APPROVED BY: _____ DATE: _____

     Guido Hunziker

1211 AIRWAY, GLENDALE, CALIFORNIA 91201

APPLICATION NOTE

Design & Installation of High Speed Micro-D P Connectors

CODE NUMBER	SIZE	AN0003	REV.
06324	A		A
SCALE N/A		SHEET 1 OF 7	

REVISION HISTORY

REVISION	DATE	REVISED PAGES	REVISIONS
A	4/10/20		Initial Release

| Cage Code:

06324 | Document Description
APPLICATION NOTE
Design & Installation of High Speed Micro-D PCB Connectors | Document #: AN0003
Revision: A
Page 3 of 7 |
| --- | --- | --- |

Table of Contents

Table of Figures

1.0   Purpose

This document describes the proper way to design PCBs for the installation of PC tail and surface mount leads found on High Speed Micro-D connectors.

2.0   Referenced Documents

Document Number/Name	Description
GHSM-BSS	High Speed Micro-D Straight Surface Mount PCB Connectors
GHSM-HBR	High Speed Micro-D Hybrid Right Angle PCB Connectors
GHSM-1003	High Speed Micro-D Straight Surface Mount PCB Layouts
GHSM-1004	High Speed Micro-D Hybrid Right Angle PCB Layouts

Table 1. Reference Documents

3.0   Responsibility

This document is the responsibility of the Engineering team.

4.0   Methods of Performance Optimization

4.1   Straight PC Tails (Thru Hole)

4.1.1 PCB Design

To achieve optimal impedance matching at the PC tail/PCB transition, utilize layouts shown in GHSM-1004 with the additional manufacturing information shown below:
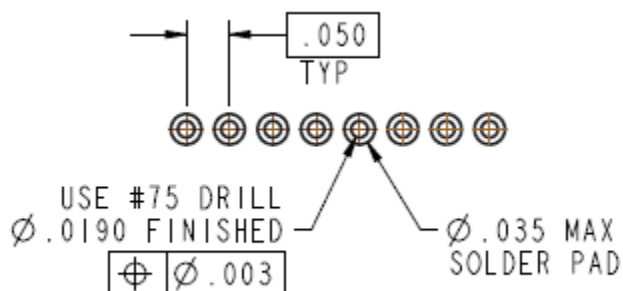

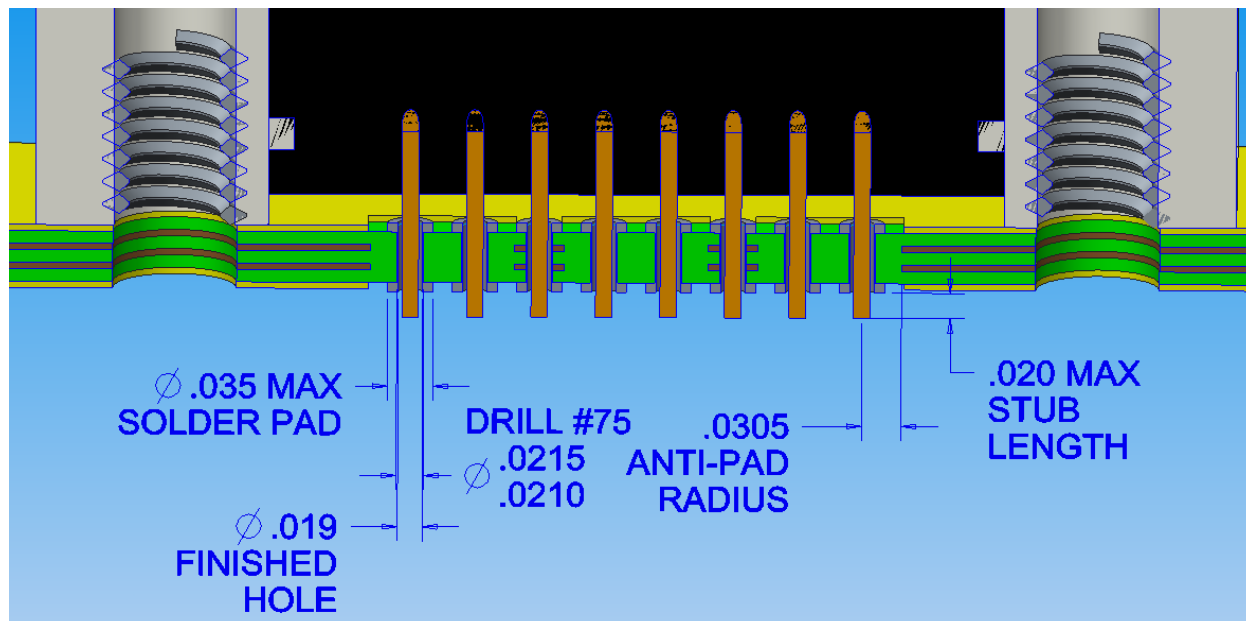**Figure 1 – Optimized PCB Layout for Thru Hole**

Figure 2 – Optimized PCB Cross Section for Thru Hole

4.1.2 Installation onto Board

To minimize the length of capacitive stubs, which lower impedance and cause the signal to resonate, we recommend trimming the PC tails to a length of .020 inches max per Figure 2.

4.1.3 Trace Routing

To reduce the effective length of capacitive stubs further, route the traces for the high-speed lines using PC tail terminations on the opposite side of the board from the connector.

Avoid right angles at trace to pad and other similar transitions throughout the signal path layout.

Signal trace widths and spacing are dependent on the desired impedance in addition to the PCB Stack-up, PCB materials, and routing traces as coupled or de-coupled. Reference Figure 3 for an example layout.
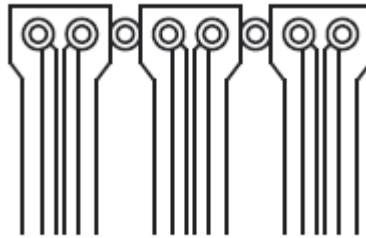

Figure 3 – Example PCB Thru Hole Layout, Termination Side

4.2 Surface Mount Leads

4.2.1 PCB Design

To achieve optimal impedance matching at the SMT lead/PCB transition, utilize layouts shown in GHSM-1003 and GHSM-1004.

4.2.2 Trace Routing

Like Thru Hole, SMT best practices include avoiding right angles at trace to pad and other similar transitions throughout the signal path layout.

SMT anti-pad geometry, in addition to signal trace widths and spacing, are dependent on the desired impedance in addition to the PCB Stack-up and PCB materials. Some applications will not require anti-pads. Reference Figure 4 for an example layout.
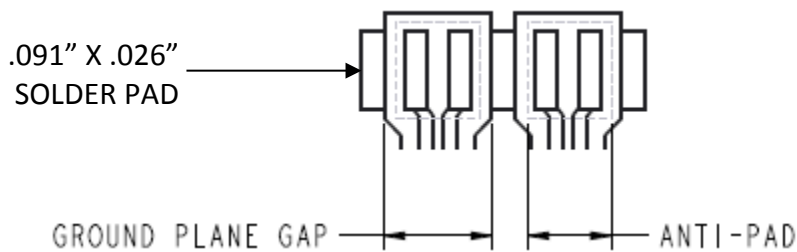

.091" X .026" SOLDER PAD

GROUND PLANE GAP

ANTI-PAD

Figure 4 – Example PCB SMT Layout, Component Side