APPLICATION NOTE
Design & Installation of VersaLink PCB Connectors

WRITTEN BY: ___________________________ DATE: 05/06/2021
Bryan Samowitz

APPROVED BY: ___________________________ DATE: 05/06/2021
Guido Hunziker
# REVISION HISTORY

<table>
<thead>
<tr>
<th>REVISION</th>
<th>DATE</th>
<th>REVISED PAGES</th>
<th>REVISIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>05/06/21</td>
<td></td>
<td>Released</td>
</tr>
</tbody>
</table>
Table of Contents
1.0 Purpose ........................................................................................................................................... 4
2.0 Referenced Documents .................................................................................................................... 4
3.0 Responsibility .................................................................................................................................... 4
4.0 Methods of Performance Optimization ............................................................................................ 5
  4.1 PCB Layout & Design ..................................................................................................................... 5
  4.2 Installation onto Board .................................................................................................................. 6
  4.3 Trace Routing ............................................................................................................................... 6
  4.4 Board Material ............................................................................................................................ 6
  4.5 Other Considerations .................................................................................................................. 6

Table of Figures
Figure 1 – Optimized PCB Layout ........................................................................................................ 5
Figure 2 – Optimized PCB Cross Section ............................................................................................... 5
1.0 Purpose

This document describes the proper way to design and install VersaLink PCB connectors & contacts onto a PCB to achieve optimal signal integrity.

2.0 Referenced Documents

<table>
<thead>
<tr>
<th>Document Number/Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GVLM-BR/BS &amp; Similar</td>
<td>VersaLink MicroD PCB Connectors</td>
</tr>
<tr>
<td>853-052/054, 853-065/067, 853-079/080 &amp; Similar</td>
<td>VersaLink Bridge PCB Connectors</td>
</tr>
</tbody>
</table>

Table 1. Reference Documents

3.0 Responsibility

This document is the responsibility of the Engineering team.
4.0 Methods of Performance Optimization

4.1 PCB Layout & Design

To achieve optimal impedance matching at the PC tail/PCB transition, we recommend the following layout:

Figure 1 – Optimized PCB Layout

Figure 2 – Optimized PCB Cross Section
### 4.2 Installation onto Board

Trim the PC tails so a stub length of .020 inches or less is obtained per Figure 2. This minimizes the length of capacitive stubs, which lower impedance and cause the signal to resonate.

### 4.3 Trace Routing

Route the traces for the high-speed lines on the opposite side of the PCB from the connector to further reduces the length of capacitive stubs.

The signal trace width needed for the desired impedance is dependent on the PCB Stack-up, PCB material, and whether the traces are routed as differential pairs or single-ended lines.

It is recommended that the trace be tear drop shaped as it comes off the solder pad.

### 4.4 Board Material

The dimensions provided are based on using a board material with a Dk value of 3.00.

### 4.5 Signal Plated Through Holes

For the Ø.0200 PTH – The important part of this is for the hole to be drilled with a #74 drill, resulting in a .0225 starting hole diameter, which controls the gap between the PTHs. The final hole can be as small as Ø.017 inches.

### 4.6 Other Considerations

When being used in a connector that does not have PCB stand-offs, design for the PCB to be .020 inches from the base of the VersaLink PCB per Figure 2.

Figure 1 – The two ground holes on the .175 spacing only apply to the 853 VersaLink Bridge connectors. They should not be included for any other VersaLink Connector.

The board design described by this document does not meet the requirements of IPC-A-610 Class 3 due to the size of the signal solder pad.