TB-2322

Paladin® Routing Guidelines

Revision "C"

Specification Revision Status

Revision	SCR No.	Description	Initial	Date
"A"	S4824	Initial release	M.Osbourne	08/02/2016
"B"	S7439	Update of Ni Au FHS tolerance from +/- 0.002 to +0.002/-0.001, Added Cu thickness max for BP	S.Yoeuth	01/17/2019
"C"	S8262	Updated section 6.5.1 to say wider in the lower layers	S.Yoeuth	04/23/2020

Amphenol TCS
A Division of Amphenol Corporation

Amphenol TCS
200 Innovative Way, Suite 201
Nashua, NH 03062
603.879.3000

www.amphenol-icc.com

1.1 TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

2. Scope

2.1 Purpose

The purpose of this technical bulletin is to describe the various required and recommended features of the printed circuit board footprint design and trace escape routing for optimal use of the Paladin connector. This document is intended to be used in conjunction with specified customer use drawings as called out below.

2.2 Printed Circuit Board Design Rules Reflect Available Printed Circuit Board Fabrication Capabilities

The technical capabilities of each individual printed circuit board manufacturer will determine the printed circuit design rules that are appropriate for acceptable yield manufacturing of a given size and complexity of printed circuit board utilizing a given insulating laminate material system. This document firstly outlines Amphenol rules that apply to all board designs using the Paladin interconnect. Secondly Amphenol board recommendations are intended to make designs more efficient for a given set of rules. Finally, detailed examples are shown, each of which proceeds from a particular set of technical PCB fabrication capabilities and electronic system performance requirements.

3. Important Considerations for Optimal Utilization of Paladin Connector

- Note: Corresponding pin-pair designations run B-A, D-C, F-E, H-G, etc. on the RAF connector footprint, and A-B, C-D, E-F, G-H, etc. on the BMA connector footprint (where applicable) of the mated connector.

- Pin-pair designations for direct ortho configurations are detailed in TB-2320.

- Provide for an exposed conductive surface ground layer on layer 1, free from solder mask or other insulating coating, as detailed in section 4.4. This will improve crosstalk isolation between signal pairs.

- Note: The BMA footprint differs from RAF footprint. See customer use drawings for footprint dimension details.

- The RAF footprint provides differential pair routing channels in both X and Y directions, i.e. it provides secondary differential pair routing channels.

- Note: Since the electrical function, routing, and drill-to-inner-layer registration impact of all signal, ground, and ground shadow vias depends upon their outer conductive diameter, it is highly recommended that a specific metric or number drill be called out in the PCB fabrication documentation for the drilling of each plated press-fit hole or plated shadow via in the Paladin footprint.

- Backdrilling is recommended on signal holes to reduce physical stub and extend signal transmission bandwidth.

- The finished hole inner diameter of the press-fit signal and ground holes need be controlled only for the top 0.70mm (0.028”) from the layer 1 side (Paladin Connector mounting side) of the PCB. See Table 1 and Figure 2.

- The indicated ground shadow vias are important for the electrical performance of the Paladin Connector System. Unlike the press-fit holes, the specified positions and drilled diameters of these shadow vias may be adjusted to some degree to accommodate the particular fabrication and electrical requirements of a given application scenario. See section 5.3

- Ground shadow vias may be allowed to plate shut. The location and drill diameter determine connector performance.

- Use of dual-diameter signal holes can provide improved trace routing and electrical performance (see section 5.2)

4. Design Rules and Manufacturability Guidelines for Press-Fit Holes

4.1 Press-Fit Hole Locations

Reference the customer use drawings.

4.2 Press-fit Finished Hole Diameter Requirements

4.2.1 Description of femto press-fit pin (15.7 mil drill)

The "Femto" compliant pins is designed for a 0.40mm (0.0157") drill hole size plated through hole (PTH). For plating considerations consult PCB fabricator or ATCS Application Engineering. This applies to signal and ground PTHs.

4.2.2 Dimensional Requirements for Press-Fit Holes in Compliant Pin Critical Working Zone

Finish Type	Copper thickness, in (mm) per side (1)	Drill size, in (mm) (2)	Typical Finish Thickness (3)	Finished Hole Size, in (mm) (4)
Lead Free HASL	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0157 (0.40)	500 micro inches maximum	BP: 0.0122+/- 0.002 (0.31+/-0.05) DC: 0.0122+/- 0.002 (0.31+/-0.05)
Immersion Sn (Tin)	0.0008 (0.020) min 0.00275 (0.069) max	0.0157 (0.40)	35 to 75 micro inches minimum	BP: 0.0122+/- 0.002 (0.31+/-0.05) DC: 0.0122+/- 0.002 (0.31+/-0.05)
Immersion Ag (Silver)	0.0008 (0.020) min 0.00275 (0.069) max	0.0157 (0.40)	4 micro inches minimum	BP: 0.0122+/- 0.002 (0.31+/-0.05) DC: 0.0122+/- 0.002 (0.31+/-0.05)
Copper - OSP	0.0008 (0.020) min 0.00275 (0.069) max	0.0157 (0.40)	N/A	BP: 0.0122+/- 0.002 (0.31+/-0.05) DC: 0.0122+/- 0.002 (0.31+/-0.05)
Ni Au (Nickel-Gold)	0.0008 (0.020) min 0.00225 (0.0572) max, BP 0.00275 (0.069) max, DC	0.0157 (0.40)	53 to 210 micro inches Ni-Au compositions combined	BP: 0.0122 +0.002/- 0.001 (0.31 + 0.05/-0.025) DC: 0.0122+/- 0.002 (0.31+/-0.05)

Table 1: Copper Thickness Requirement and Finished Hole Inner Diameter

NOTES (Table 1):

1. Copper thickness per side is an Amphenol requirement. The copper plating must be within specified range in Table 1 according to the specific Finish Type selected by the end user. Copper plating at the knee of the hole is limited to a maximum of 10% above the average copper wall thickness measured in the working zone, see Figure 2 below, of the plated thru hole, and cannot exceed the MAX copper plating thickness specified in Table 1.
2. Drill size is an Amphenol requirement. Amphenol requires this drill to be used as outlined in Table 1. The drill size specified does not include a tolerance nor does it include a drilled hole tolerance. These tolerance allowances are included in the overall tolerance outlined in the Finish Hole Size tolerance reference range.
3. Typical Finish Thickness is an Amphenol reference value of all finishes except ENIG (Nickel plating thickness for ENIG finish is a requirement). Amphenol highlights the reference values on these finishes as a guideline for processing and inspection of PCB holes. Actual finish thicknesses will vary depending on the finish type selected by the end used design guidelines.
4. Finish Hole Size is an Amphenol nominal value. The Finish Hole Size tolerance for each finish type accounts for the accumulation of tolerances in the actual Drill Hole Diameter as a result of using the specified drill noted in the table above and Finish Thickness over the range of holes in a connector-hole pattern on a PCB. Cu thickness, finish plating thickness, and Drill Size must be adhered to per Amphenol's requirements in this document, but not cumulatively exceeded where these tolerance buildups exceed the finish hole requirements called out.

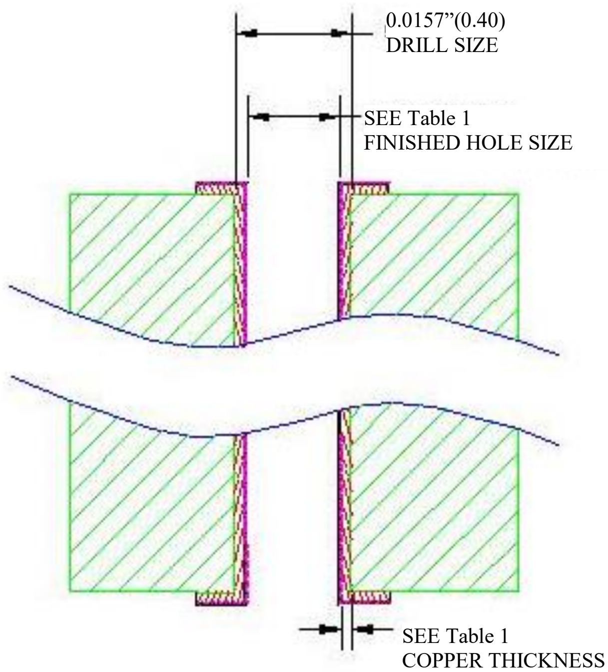

Figure 1: Paladin Drill and Finished Hole Size

4.2.2.1 Compliant Pin Critical Working Zone

The "Critical Working Zone" shown in Figure 2 is defined as the compliant working zone where the plated through hole requirements must meet the specifications defined within this document. In the "Non Critical Zone", the plated through hole is allowed to go below the minimum required finish hole size for non-midplane applications. Back drilling is allowed in the "non-critical zone" only.

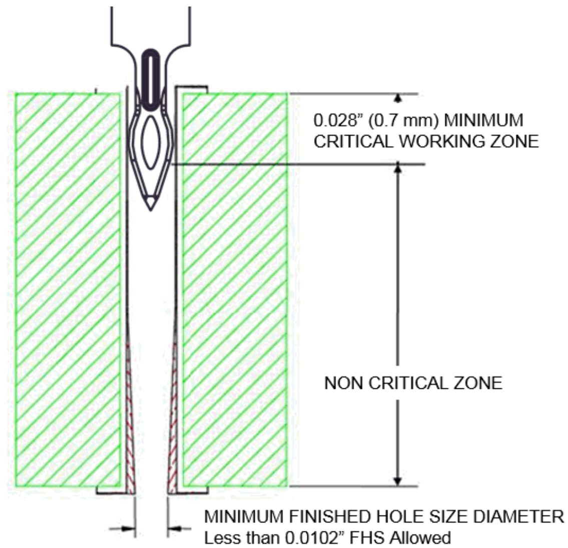

Figure 2: 0.0157" Compliant Pin Critical Working Zone

4.3 Minimum PCB Thickness

Minimum PCB thickness is 1.0mm (0.039 inches). Note that for boards less than 1.25mm (0.049 inches) thick the press-fit pins may protrude from the rear-side of the board.

4.4 Conductive Surface Layer

It is required that the top surface layer which the Paladin connector is pressed into has a conductive ground layer free from insulating coating in the connector footprint region. See Figure 3.

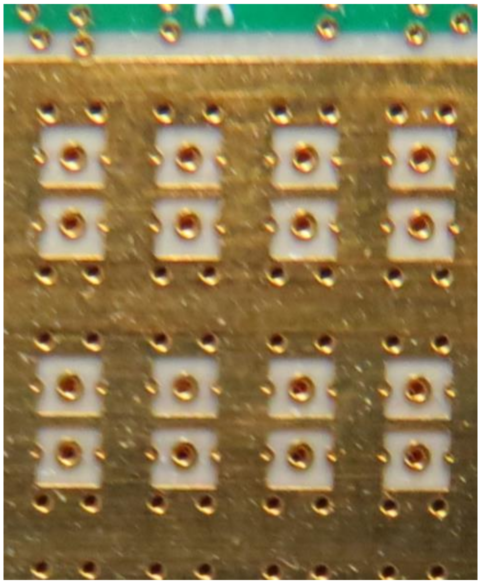

Figure 3: Conductive Ground Layer on Connector Footprint

5. General Recommendations

5.1 Surface Traces

Surface traces are not recommended in the region of the connector footprint on the primary side where the connector is attached.

5.2 Dual Diameter Holes

The compliant pin is particularly suited for use with dual-diameter holes, which should be considered for high speed signals (see Figure 4). The larger diameter is drilled first to a controlled depth followed by the second smaller drill. The larger diameter portion of the through hole should follow the requirements outlined in Table 1. The requirements for the smaller diameter portion of the through hole will vary depending on the specific application. Contact your local field applications engineer for support on optimizing dual diameter through holes.

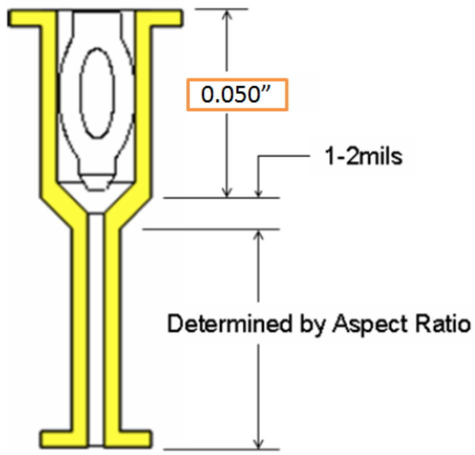

Figure 4: Diagram of a dual diameter hole with the Paladin medium tail compliant pin.

5.3 Shadow Vias

Ground shadow vias are important for the electrical performance of the Paladin Connector System. Unlike the press-fit vias, the precise position and drill diameter of these shadow vias may be adjusted to some degree to accommodate the particular fabrication and electrical requirements of a given application. There is a minimum copper wall thickness requirement of 0.025mm (0.001"), however there is no maximum wall thickness requirement (may be plated shut). The ground via diameter requirement varies depending on the specific application. Contact your local field applications engineer for support on optimizing the shadow via geometry.

6. Signal Integrity Aspects of Footprint Design

6.1 Surface Conductive Layer

It is required that the top surface layer which the Paladin connector is pressed into has a conductive ground layer free from insulating coating in the connector footprint region. This is important for optimizing signal transmission and reducing reflections and crosstalk. See Figure 3.

6.2 Relative Dielectric Constant of Laminate Material

Note that different laminate insulating materials have differing dielectric constant values. The dielectric constant value will affect the characteristic impedance of the differential plated holes, trace escape, and routed traces. For this reason, it is usual to adjust such parameters as antipad size, location and drill diameter of ground shadow vias, trace escape configuration, trace width, and dielectric thicknesses to optimize the electrical signal integrity performance.

6.3 Location and Finished Plated Hole Sizes for Signal Press-Fit Holes

The signal press-fit contacts on the RAF and the BMA are specified and must match the pattern of press-fit contacts of the corresponding connector. The internal diameter of these finished plated holes is important for the proper functioning of the press-fit connector attachment and should be controlled as per requirement for the specified working range depth from the connector side of the board. The electrical characteristics of these plated holes, on the other hand, is determined by the drill diameter, which should therefore be specified (either metric or numbered drill) for drilling these holes.

6.3.1 Backdrilling

Signal plated through holes should be backdrilled to reduce the length of the electrical stub formed in the region between the signal escape layer and the bottom PCB layer. Stub resonances are undesirable because they increase return loss and reduce signal transmission bandwidth.

6.3.2 Always Remove Non-Functional Pads on Signal Plated Through Holes

Non-functional pads on signal plated through holes are undesirable because they typically lower the characteristic impedance below desired values. In typical applications, the only pads attached to a signal plated through hole are those on the primary connector attach surface, and the internal escape layer where there are pads to attach the signal plated through hole to signal traces.

6.3.3 Dual-Diameter Signal Plated Through Holes

Use of dual-diameter signal plated through holes can provide improved electrical signal integrity performance and easier trace routing. The upper, wider portion of the dual-diameter hole is larger and receives the press-fit signal pin, while the lower, narrower portion connects down to the signal trace escape layer.

6.4 Location and Finished Plated Hole Sizes for Ground Press-Fit Holes

The ground press-fit contacts on the RAF and the BMA are specified and must match the pattern of press-fit contacts of the corresponding connector. The internal diameter of these finished plated holes is important for the proper functioning of the press-fit connector attachment and should be controlled as per requirement for the specified working range depth from the connector side of the board. The electrical characteristics of these plated holes, on the other hand, is determined by the drill diameter, which should therefore be specified (either metric or numbered drill) for drilling these holes.

6.5 Diameter and Location of Ground Shadow Vias

Note that there is no finished hole size requirement for these vias and they can be plated shut. There is some degree of flexibility in the location and drill diameter used for these vias. They perform important electrical functions in reducing footprint crosstalk and controlling characteristic impedance. Note that their electrical characteristics depend on the diameter of the drill used, and therefore a specific drill size (either metric or numbered drill) should be specified for drilling these vias.

6.5.1 Reverse Dual-Diameter Ground Shadow Vias

Use of reverse dual-diameter ground shadow vias can provide improved electrical signal integrity performance and easier trace routing. A typical application will combine dual-diameter signal plated through holes (Refer to Section 6.3.3) and reverse dual-diameter ground shadow vias, which are narrower in the upper layers of the board, and wider in the lower layers.

Appendix A: Definitions and Considerations

Foils Thickness or Copper Weight

Consider copper weight when routing. Higher weights will impact minimum trace widths. Copper foil is measured in ounces (or weight). Common copper weights are 0.5 ounces, 1 ounce, 1.5 ounces and 2 ounces (3 ounces up to 10 ounces are available for special order). 1 ounce = 0.0014”, 1.5 ounces = 0.0021”, 2 ounces = 0.0028”.

Pads/Lands/Annular Ring

A pad is the support around a hole. A specification describing an annular ring of 0.005” means the amount of the pad left around the hole after processing.

Spacing

Spacing is the space between two electrical connections; it can be between two lines, two pads, a line and a pad etc.

Trace/Circuit/Line Width/Lines/Conductor

These are different terms for a connection. If you see the term 0.008” lines, it means the electrical connection from one point to another will measure 0.008” width.

BMA

Backplane Module Assembly Connector.

RAF

Right Angle Female Connector.

DOM

Direct orthogonal male connector.

Trace Centering

Center all traces between holes to optimize spacing.

Non Functional Pads

A non-functional pad is a signal pad around a plated drilled hole that does not connect to a functioning signal trace.

Characteristic Impedance

Characteristic Impedance is a high frequency electrical property of a signal transmission path modeled as a transmission line. A differential pair signal path has both a differential and common-mode characteristic impedance. In general it is advantageous for efficient signal transmission to appropriately match the characteristic impedances of various successive portions of a signal transmission path. Consider characteristic impedance when designing to ensure line widths will meet requirements. Please contact ATCS Application Engineering for impedance calculations.