

TB-2134

APTERA™ BOARD PRODUCABILITY AND SIGNAL TRACE ROUTING
GUIDELINES FOR MIDPLANE, BACKPLANE AND DAUGHTERCARD
APPLICATIONS

Revision “-“

Specification Revision Status

Revision	SCR No.	Description	Initial	Date
“1“		Initial Release	Don Milbrand	7-15--05

Amphenol TCS

A Division of Amphenol Corporation

Amphenol TCS
44 Simon Street
Nashua, NH 03060
603.879.3000

Aptera, Crossbow, GbX, HD Plus, HDM Plus, HDM, HD-Optyx, NeXLev, Ventura, VHDM, VHDM-HSD, and DesignLink are trademarks or registered trademarks of Amphenol Corporation. All other products names are the marks of the respective owners.

1.0 SCOPE

- 1.1 The intent of this technical bulletin is to outline the standard signal trace widths, minimum spacing requirements, and finish hole size requirements for the 4mm and 2mm Plus connector series when used in differential signal applications. This document supercedes all other Amphenol documents including customer use drawings when conflicts exist for the stated requirements outlined within this document.
- 1.2 Efficient routing of signal traces between connector patterns improves yields and manufacturability. Spacing between trace/pad and trace/trace must to be considered to allow for proper feature modifications needed for the inner layer fabrication process. Failure to allow for this will result in lower yields and higher PWB costs.
- 1.3 All finish hole size requirements provided within this document are based on testing completed in FR-4 laminate.

2.0 DEFINITIONS

- 2.1 Fillets - An extension of the pad at the interface of the signal to the pad that will allow more pad area, in the event that the pad to hole relationship compromises the interconnect area. For further information regarding these routing guidelines, please contact ATCS Applications Engineering.
- 2.2 Foils/Copper Weights - Copper foil is measured in ounces per square foot. Common copper weights are 0.5 ounce, 1 ounce, 1.5 ounce and 2 ounce, approximate foil thickness is 1 ounce = 0.0014", 1.5 ounces = 0.0021", 2 ounces = 0.0028". Other copper weights available upon request.
- 2.3 Pads/Lands/Annular Ring - A pad is the support around a hole. A specification calling out an annular ring of 0.005" indicates the amount of the pad left around the hole after processing.
- 2.4 Spacing - Spacing is the distance between two electrical connections; it can be between two lines, two pads, a line and a pad etc.
- 2.5 Trace/Circuit/Line Width/Lines/Conductor - These are interchangeable terms for a connection. The term 0.008" lines, describes an electrical connection from one point to another that will measure .008" in width.
- 2.6 Midplane or Backplane-when used within this document refers to the PCB associated with the receptacle half of the connector system.
- 2.7 Daughtercard When used within this document refers to the PCB associated with terminal connector half of the connector system.

3.0 PROCEDURE

3.1 Routing Guidelines

- 3.1.1 Minimum spacing, specific pad/trace, and trace/trace between all features should be 0.005" (.127mm) to allow for manufacturing tolerances.
- 3.1.2 Impedance must be considered when designing line widths, and spaces to insure requirements will be met. Contact ATCS Application Engineering for assistance with impedance calculations.
- 3.1.3 Copper weights must be considered when routing. Higher weights will impact minimum trace widths.
- 3.1.4 Fillets at the interface (egress) of the trace to the pad are required to improve annular ring when the electrical design requires tight hole to pad configurations.
- 3.1.5 Center all traces between holes to optimize spacing.
- 3.1.6 For compliant pin applications the specified drilled hole size and copper thickness is mandatory, see table 1 and figure 3.
- 3.1.7 For high speed applications, remove all non-functional pads.

3.2 Design Rules and Manufacturing/Produceability Guidelines

3.2.1 General Design Rules

- 3.2.1.1 For specific connector footprint see customer use drawings.
- 3.2.1.2 For copper thickness requirements and finish hole size reference see table 1.
- 3.2.1.3 Require a 0.0225" (0.57mm) drill, this is a #74 drill for the .018" FHS.

3.2.2 PCB Produceability Guidelines.

- 3.2.2.1 Line widths, pad sizes and spacing applicable for 1/2 ounce and 1 ounce copper weights.
- 3.2.2.2 Filleting of pads recommended (to be added by fabricator) for 0.000" annular ring, see figure 1.
- 3.2.2.3 For applications with connectors pressed on one side only the recommended minimum board thickness is 0.063" (1.60mm). for press fit applications the minimum board thickness is .197" (5.0mm)
- 3.2.2.4 Non-functional pads on signal vias can be removed or added at designers option, see section 3.7.
- 3.2.2.5 Plane clearances are applicable for copper weights up to 2 ounces. Contact ATCS Application Engineering for applications using higher copper heights.
- 3.2.2.6 Surface traces are not recommended. If surface traces are used refer to the customer drawings for keep-out zones.

3.3 Routing Guidelines

3.3.1 Aptera pad requirements

Copper Weight Ounces	Process	Mid. Pad Size	Mid. Pad Size	Mid. Pad Size
		0.000 A/R	0.001 A/R	0.002 A/R
0.5 (17μm)	Inner Layer	0.035" (0.89)	0.037" (0.94)	0.039" (0.99)
	Outer Layer	0.0375" (0.95)	0.0395" (1.00)	0.0415" (1.05)
1.0 (35μm)	Inner Layer	0.026" (0.91)	0.038" (0.96)	0.040" (1.02)
	Outer Layer	0.0375" (0.95)	0.0375" (0.95)	0.0415" (1.05)

NOTES for section 3.3.1:

1. Outer layer pad sizes reflect panel plating process.
2. Use inner layer pad sizes for outers when pattern plating.
3. Values in () are metric equivalents. For printed circuit board layout use metric units.
4. For plane clearances see figure 2, for all other plane clearances contact TCS Application Engineering.

3.3.2 Trace Requirements

Copper Weight ounces	Max. Line 2 Track 0.000 A/R	Max. Line 2 Track 0.001 A/R	Max. Line 2 Track 0.002 A/R
0.5 (17μm)	0.009 (0.23)	0.008 (0.20)	0.008 (0.20)
1.0 (35μm)	0.009 (0.23)	0.008 (0.20)	0.007(0.18)

NOTES for sections 3.3.2 and 3.3.3:

1. Assume 0.005" minimum spacing.
2. Assume 100Ω characteristic impedance traces.
3. See figure 1 for annular ring requirements.
4. Values in () are metric equivalents. For printed circuit board layout use metric units.

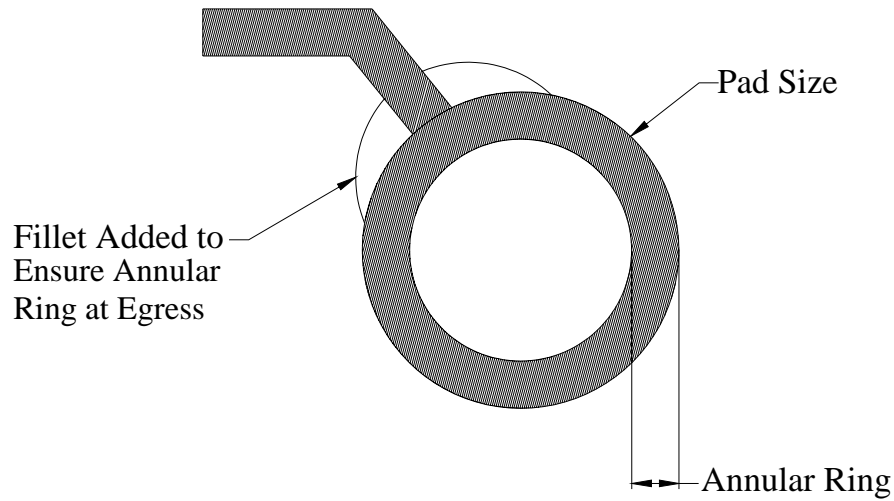


FIGURE 1, Preferred Fillet

Note: Fillet size equals minimum annular ring plus 0.005" (0.13)

Finish Type	Copper thick, inches (mm) per side	Drill size, in	Typical Finish Thickness (ref only)
Solder Finish ⁽¹⁾	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0225 (0.57)	300 to 500 micro inches
Immersion Sn (Tin)	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0225 (0.57)	35 to 75 micro inches minimum
Immersion Ag (Silver)	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0225 (0.57)	4 micro inches minimum
Copper - OSP	0.0010 (0.0254) min 0.0025 (0.0635) max (DC) 0.0030 (0.0762) max (BP)	0.0225 (0.57)	N/A
Ni Au (Nickel-Gold)	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0225 (0.57)	53 to 210 micro inches Ni-Au compositions combined

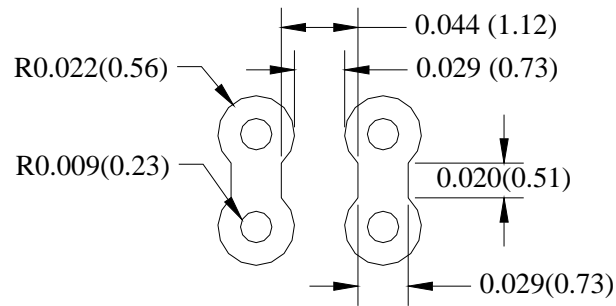
TABLE 1, Copper Thickness Requirement and Finished Thickness Reference

NOTES:

1. Solder finish includes: Tin/lead reflowed (plated and reflowed) and HASL.

3.3.3 Anti-pad geometry

- 3.3.3.1 For high speed applications refer to figure 2 for the anti-pad geometry.
3.3.3.2 For low speed applications use standard circular anti-pad geometry of pad size plus 0.012", see section 3.3.1

**FIGURE 2, Typical High Speed Application Anti-pad Geometry**

4.0 Compliant Pin Hole Guidelines
4.1 Cross section details

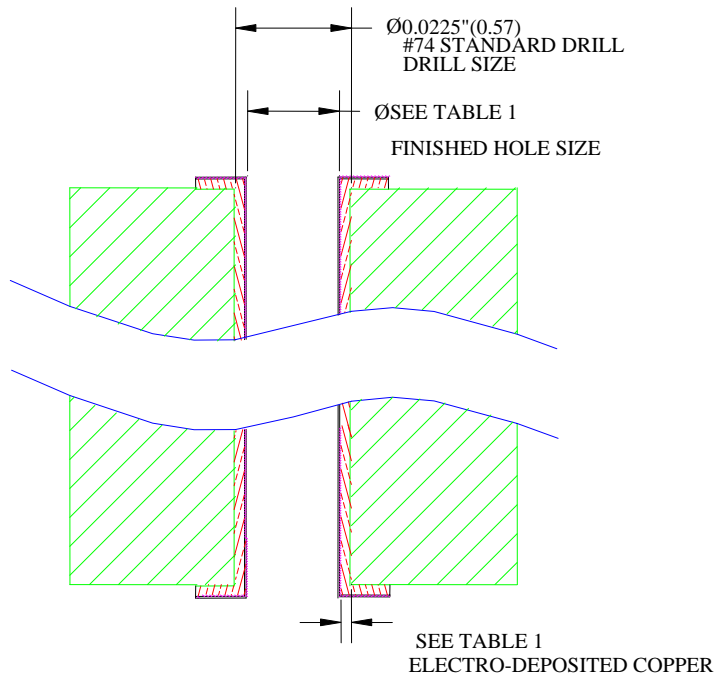


FIGURE 3, Compliant pin signal Plated Through Hole

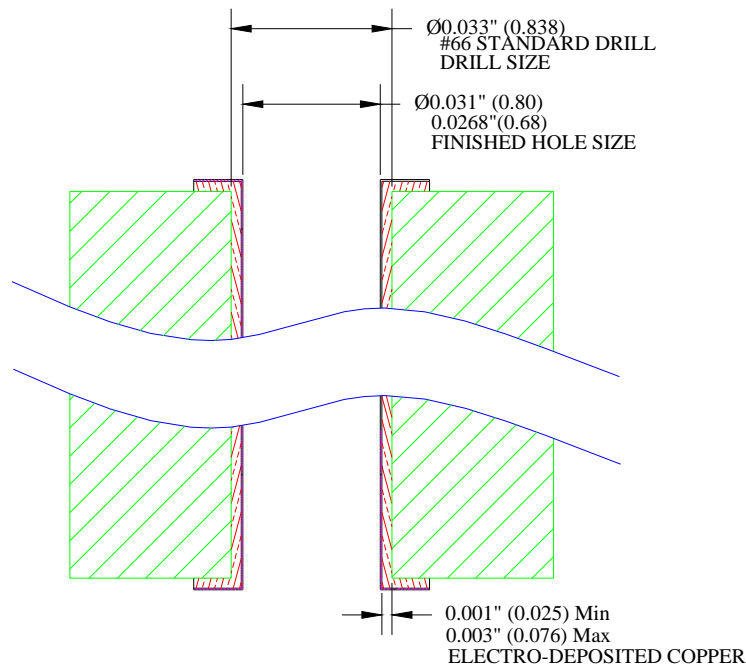
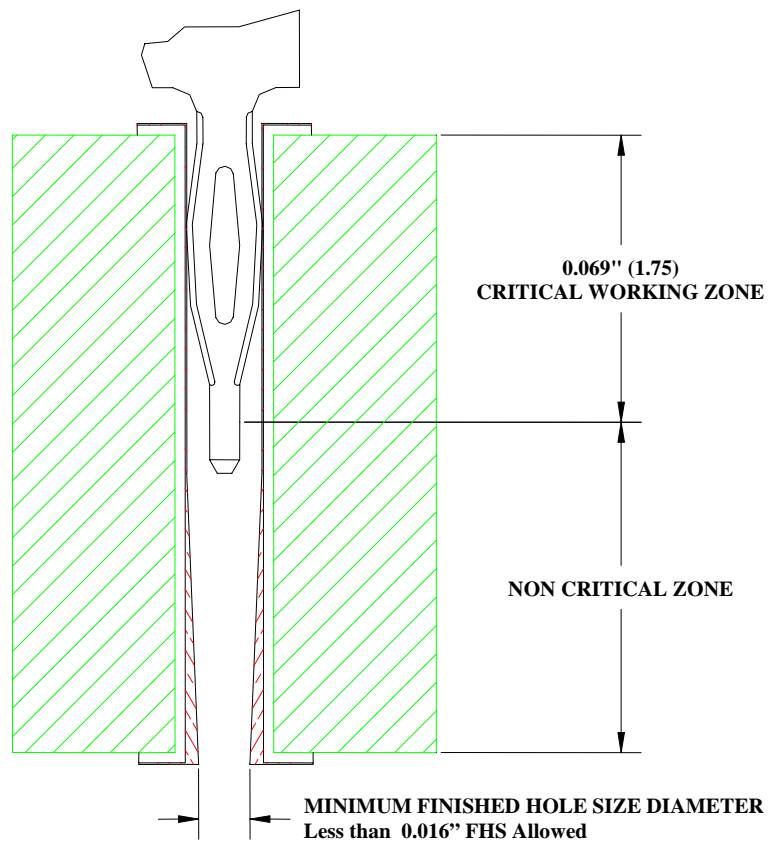


FIGURE 4, Compliant pin Power Plated Through Hole

4.2 Critical Areas

The "Critical Working Zone" shown in figure 4 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 plate through hole is allowed to go below the minimum required finish hole size of 0.016" for non midplane applications.

**FIGURE 5, Compliant pin critical zone**

5.0 Escape Routing Guidelines

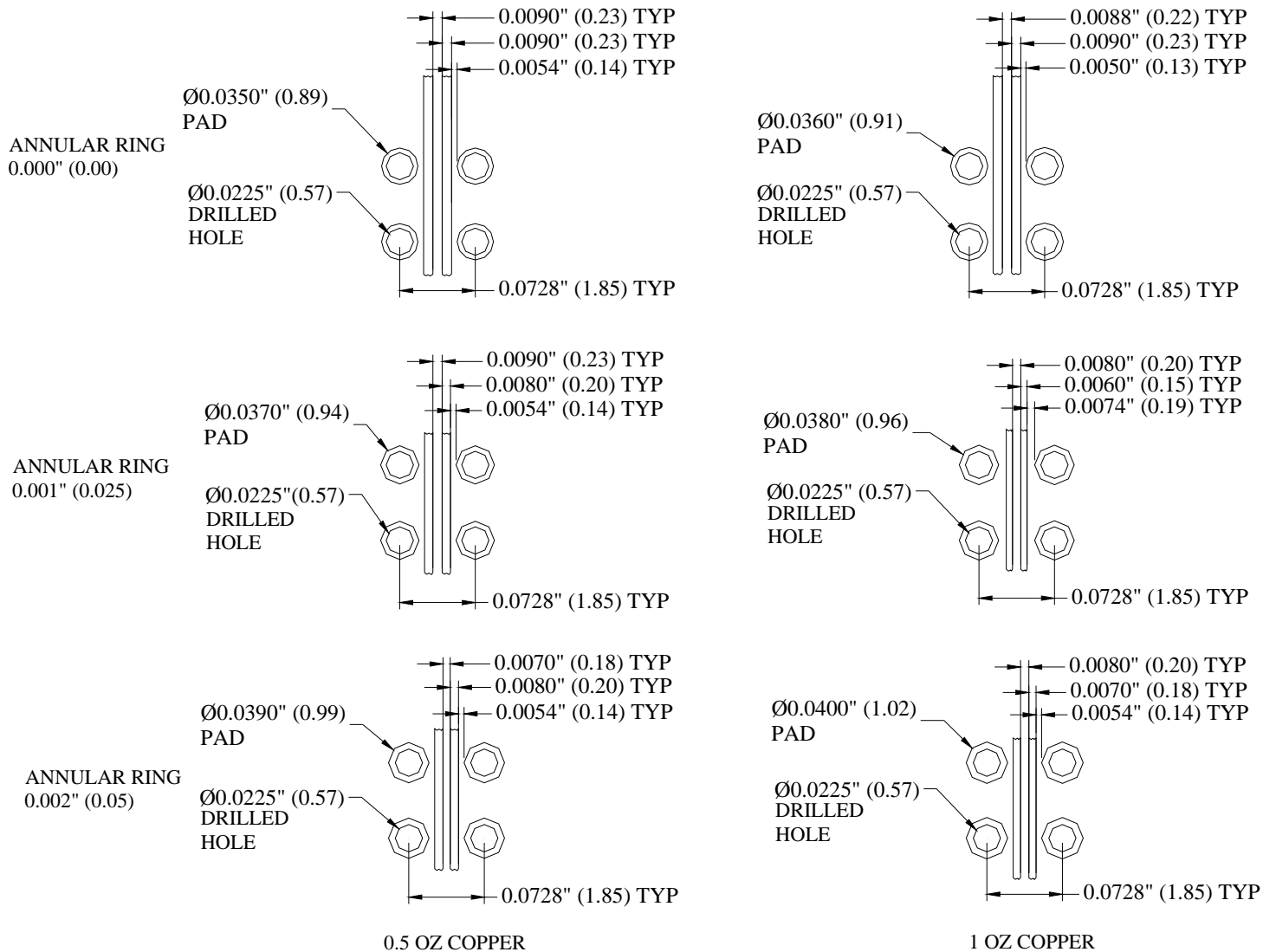


FIGURE 6, GbX TYPICAL HIGH SPEED DIFFERENTIAL ROUTING

NOTE:

1. For complete hole pattern dimensions please refer to the customer use drawings.