AMPHENOL TCS

TB-2090

GbX SIGNAL TRACE ROUTING GUIDELINES
FOR BACKPLANE AND DAUGHTERCARD CONNECTORS
HIGH SPEED DIFFERENTIAL AND LOW SPEED

Revision “A“

Specification Revision Status

Revision	SCR No.	Description	Initial	Date
“-“	44648	Initial Release	T.Do	7/14/04
“A“	S0043	Replaced template format	M.Lee	02/07/06

Amphenol TCS
A Division of Amphenol Corporation

Amphenol TCS
44 Simon Street
Nashua, NH 03060
603.879.3000

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 GbX connector series when used in differential signal applications and low speed applications. This document supercedes all other GbX 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 needs 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 TCS Applications Engineering.

2.2 Foils/Copper Weights - 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".

2.3 Pads/Lands/Annular Ring - A pad is the support around a hole. If you see a specification calling out an annular ring of 0.005", that will mean the amount of the pad left around the hole after processing.

2.4 Spacing - Spacing is the space 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 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 .008" width.

2.6 Backplane-When used within this document refers to the PCB associated with the male connector half of the connector system.

2.7 Daughtercard-When used within this document refers to the PCB associated with the female 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 Consider impedance (if applicable) when designing to insure line widths will meet requirements. Please contact TCS Application Engineering for impedance calculations.

3.1.3 Consider copper weights 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 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 Rule and Manufacturing/Produceability Guidelines

3.2.1 General Design Rules

3.2.1.1 Require a 0.0225" (0.57mm) drill, this is a #74 drill.

3.2.1.2 For specific connector footprint see customer use drawings.

3.2.1.3 For copper thickness requirements and finish hole size reference see table 1.

3.2.2 Daughtercard/Backplane 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 Recommended minimum panel thickness of 0.063" (1.60mm).

3.2.2.4 Non-functional pads on signal 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. Please contact TCS Application Engineering for applications with more than 2 ounce copper.

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 GbX Backplane-Daughtercard signal and shield PCB

Hole to Hole Grid	PCB Material Thickness	Copper Weight Ounces	Process	Min. Pad Size 0.000 A/R	Min. Pad Size 0.001 A/R	Min. Pad Size 0.002 A/R
High Speed Backplane 0.053" x 0.073" (1.35 x 1.85) **Low Speed Backplane 0.073" x 0.073" (1.85 x 1.85)**	0.062" to 0.350" (1.60 to 8.80)	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)
All Daughtercard 0.059" x 0.073" (1.50 x 1.85) **2 Pair Daughtercard 0.053" x 0.073" (1.35 x 1.85)**	0.062" to 0.260" (1.60 to 6.60)	1.0 (35μm)	Inner Layer	0.036" (0.91)	0.038" (0.96)	0.040" (1.02)
			Outer Layer	0.0375" (0.95)	0.0395" (1.00)	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 Backplane 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)

3.3.3 Daughtercard 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.

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

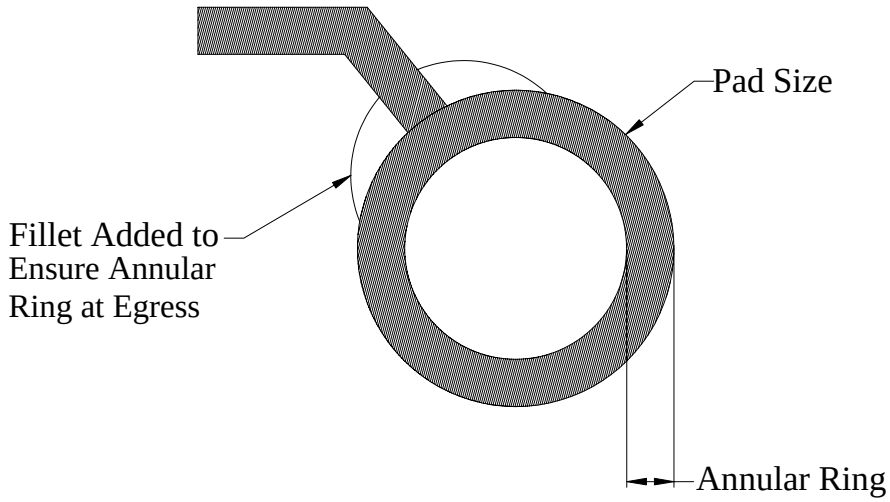

FIGURE 1, Preferred Fillet

"Pad" one half size of component pad located on a line central to trace so that fillet size equals minimum annular ring plus 0.005" (0.13)

NOTES:

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

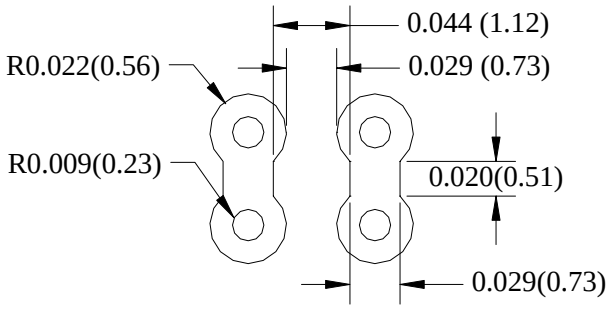

FIGURE 2, Typical High Speed Application Anti-pad Geometry

4.0 Anti-pad geometry

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

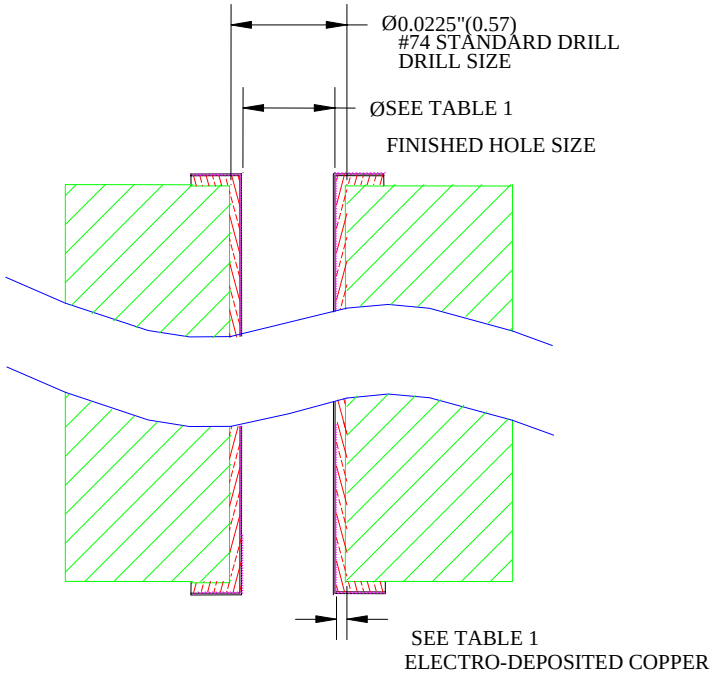

FIGURE 5, Signal and Shield Plated Through Hole

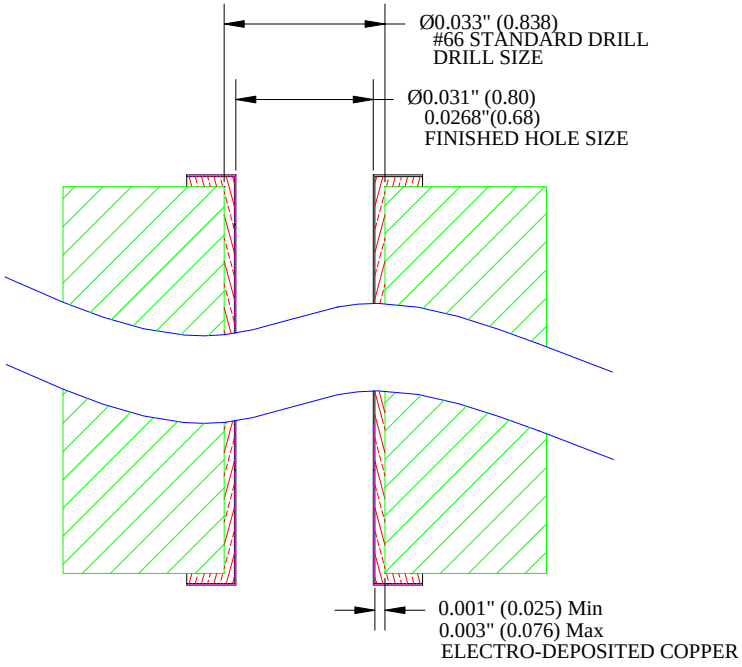

FIGURE 3, Power Plated Through Hole

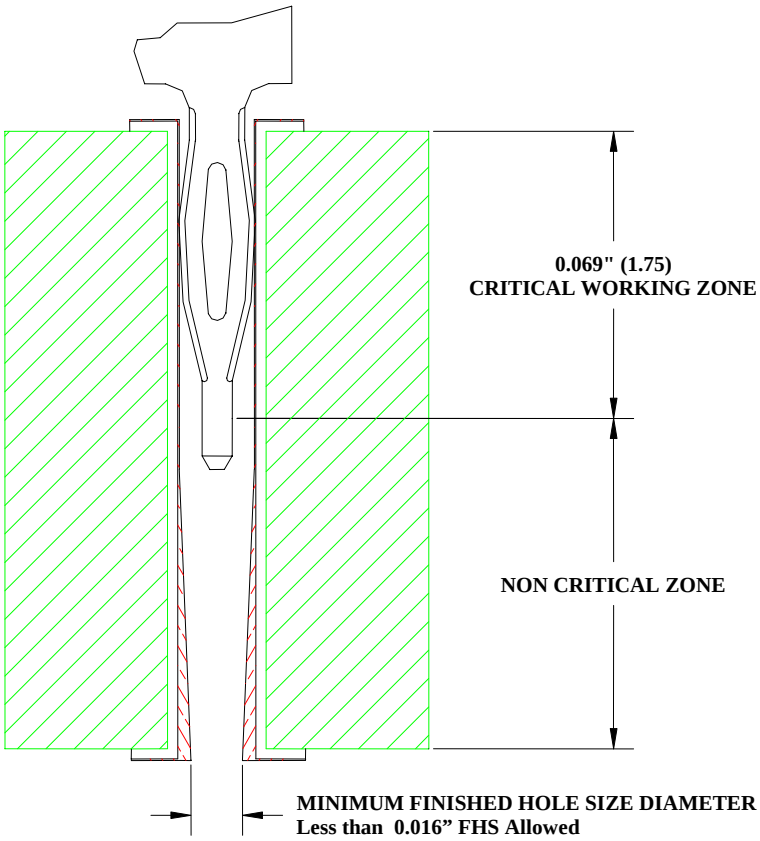

FIGURE 4, Compliant pin critical zone

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.

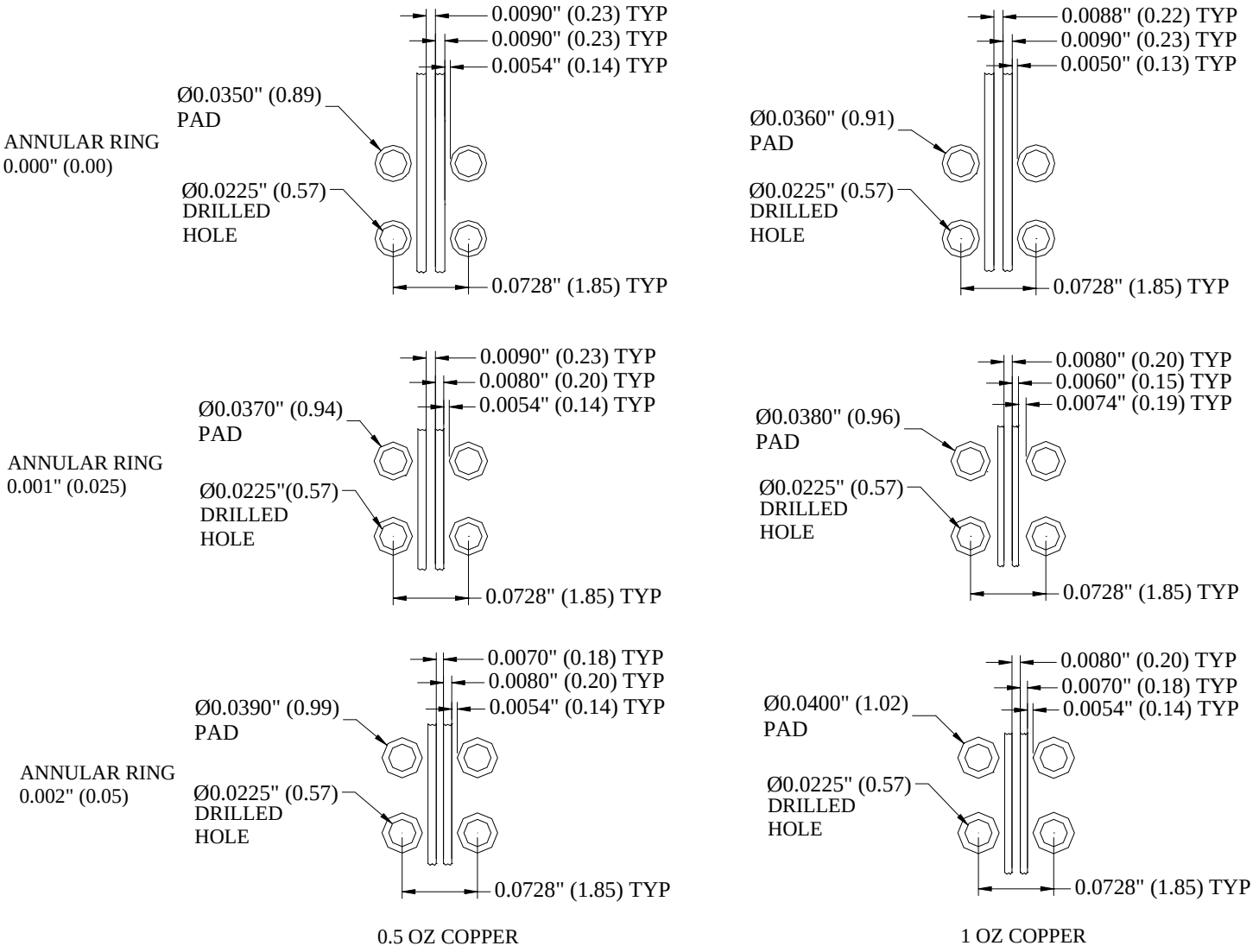

FIGURE 5, GbX TYPICAL HIGH SPEED DIFFERENTIAL ROUTING

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