TB-2353

Lynx QD Routing Guidelines

Revision "A"

1. Specification Revision Status

Revision	SCR No.	Description	Initial	Date
A	S7939	Initial Release	MC	10/21/19

Amphenol TCS
A Division of Amphenol Corporation

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

www.amphenol-icc.com

TABLE OF CONTENTS

1. Specification Revision Status 1
2. Scope 4
2.1 Intent 4
2.2 Printed Circuit Board Design Rules Reflect Available Printed Circuit Board Fabrication Capabilities 4
2.3 Efficient routing 4
3. Important Considerations for Optimal Utilization of Lynx Connector 4
4. Definitions 5
4.1 Foils/Copper Weights 5
4.2 Pads/Lands/Annular Ring 5
4.3 Spacing 5
4.4 Trace/Circuit/Line Width/Lines/Conductor 5
5. Routing Guidelines 6
5.1 Minimum Spacing 6
5.2 Impedance 6
5.3 Copper Weights 6
5.4 Fillets 6
5.5 Trace Centering 6
5.6 Non-Functional Pads 6
6. Design Rules and Manufacturability Guidelines 6
6.1 General Design Rules 6
6.1.1 Drill 6
6.1.2 Footprint 6
6.1.3 Drilled Hole and Copper Thickness 6
6.2 Manufacturability Guidelines 6
6.2.1 Line Widths, Pad Sizes and Spacing 6
6.2.2 Filleting 6
6.2.3 Minimum PCB Thickness 6
6.2.4 Non-functional pads 6
6.2.5 Plane Clearances 7
6.2.6 Surface Traces 7
7. PCB Design Guidelines 7
7.1 Lynx QD Drill and Signal Pad Sizes 7
7.2 Routing escape 8
7.3 Typical Anti-pad Geometry 9
7.3.1 Additional Antipad Clearance – Lynx QD 9
7.4 Lynx QD Typical High-Speed Differential Routing 10

LIST OF FIGURES

Figure 1: Traditional landing pad geometry example 8
Figure 2: Via in pad (VIPPO) landing pad geometry example 8
Figure 3: Routing Escape Options 9
Figure 4: Typical Antipad Geometry 9
Figure 5: Antipad Clearance 10
Figure 6: Example Lynx QD Differential Routing 11
Figure 7: Example Lynx Differential Single side escape Routing 12

LIST OF TABLES

Table 1: Copper Thickness Requirement and Finished Hole Inner Diameter 7
Table 2: Example of Lynx QD Routing Channel 11
Table 3: Example of Lynx QD Routing Channel 11

2. Scope

2.1 Intent
The intent of this technical bulletin is to outline the standard signal trace widths, minimum spacing requirements, and finish hole size requirements for the Lynx QD connector series when used in differential signal applications and low speed applications. This document supersedes all other Lynx QD documents including customer use drawings when conflicts exist for the stated requirements outlined within this document.

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 Lynx QD connector. 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.

2.3 Efficient routing
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 may result in lower yields and higher PWB costs.

3. Important Considerations for Optimal Utilization of Lynx Connector

- 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 hole or plated shadow via in the Lynx footprint.

- Backdrilling is recommended on signal holes to reduce physical stub and extend signal transmission bandwidth. Typically, the backdrill diameter would be 10 – 11 mils larger than the drilled hole being backdrilled. For example, a. 0.65mm or 0.7mm drill would be used to backdrill a 0.4mm (15.7mil) drill hole. The resulting via stub should be as short as possible. Current PCB capabilities for backdrilling is 7mil +/-5mil (6mil +/-4mil for boards less than 90 mils thick). If backdrilling will pass through plane layers, then there should be a minimum of 6mils of clearance from edge of drill to edge of copper.

- The finished hole inner diameter of the signal and ground holes can be 7.9mil (0.2mm) or 9.8mil (0.25mm) depending on the pwb attributes and the pwb supplier's capability.

- The Lynx connector is a surface mount connector. The footprint can be designed for a traditional SMT geometry or a via in pad geometry (VIPPO). The choice will depend on the pwb attributes and the pwb supplier's capability. In general, the VIPPO footprint should offer slightly better signal integrity performance.

- All of the signal and ground vias may be allowed to plate shut.

4. Definitions

4.1 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”.

4.2 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.

4.3 Spacing

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

4.4 Trace/Circuit/Line Width/Lines/Conductor

These are different terms for a printed circuit board electrical connection. If you see the term 0.004” lines, it means the electrical connection from one point to another will measure 0.004” in width.

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.

5. Routing Guidelines

5.1 Minimum Spacing
The Lynx connector footprint has been designed to meet a minimum drill to metal (D2M) space of 8mils, and a minimum trace to trace space of 4mils

5.2 Impedance
Consider characteristic impedance (if applicable) when designing to ensure line widths will meet requirements. Please contact ATCS Application Engineering for impedance calculations.

5.3 Copper Weights
Consider copper weights when routing. Higher weights will impact minimum trace widths.

5.4 Fillets
Fillets are not recommended. Fillets at the interface (egress) of the trace to the pad are typically capacitive and reduce electrical performance.

5.5 Trace Centering
Center all traces between holes to optimize spacing.

5.6 Non-Functional Pads
For high speed applications, remove all non-functional pads.

6. Design Rules and Manufacturability Guidelines

6.1 General Design Rules

6.1.1 Drill

The Lynx footprint has been designed to support both 7.9 mil (0.20mm) & 9.8 mil (0.25mm) drilled holes.

6.1.2 Footprint
For specific connector footprint see customer use drawings.

6.1.3 Drilled Hole and Copper Thickness
For copper wall thickness requirements and finish hole size reference, see Table 3, Figure 2 and Figure 3.

6.2 Manufacturability Guidelines

6.2.1 Line Widths, Pad Sizes and Spacing
Line widths, pad sizes and spacing applicable for 1/2 ounce and 1-ounce copper weights.

6.2.2 Filleting
Filleting of pads is not recommended.

6.2.3 Minimum PCB Thickness
Recommended minimum PCB thickness of 0.035” (0.89mm).

6.2.4 Non-functional pads
For high speed applications, remove all non-functional pads.

6.2.5 Plane Clearances

Plane clearances are applicable for copper weights up to 2 ounces. Please contact ATCS Application Engineering for applications with more than 2-ounce copper.

6.2.6 Surface Traces

Surface traces are not recommended.

Finish Type	Copper thickness, in (mm) per side (1)	Drill size, in (mm)	Typical Finish Thickness (3)	Finished Hole Size, in (mm)
Lead Free HASL	0.0010 (0.0254) min 0.0025 (0.0635) max	0.0079” (0.20mm) 0.0098” (0.25mm)	500 micro inches maximum	0.006 +0.002/-0.006” (0.15 +0.05/-0.15) 0.008 +0.002/-0.008” (0.20 +0.05/-0.20)
Immersion Sn (Tin)	0.0008 (0.020) min 0.00275 (0.069) max		35 to 75 micro inches minimum	
Immersion Ag (Silver)	0.0008 (0.020) min 0.00275 (0.069) max		4 micro inches minimum	
Copper - OSP	0.0008 (0.020) min 0.00275 (0.069) max		N/A	
Ni Au (Nickel-Gold)	0.0008 (0.020) min 0.00275 (0.069) max		53 to 210 micro inches Ni-Au compositions combined	

Table 1: Copper Thickness Requirement and Finished Hole Inner Diameter

7. PCB Design Guidelines

7.1 Lynx QD Drill and Signal Pad Sizes

The Lynx QD footprint is designed to support 7.9mil and 9.8mil drill sizes. In general, the smaller the drill size the better the SI performance. Each design application needs to consider the trade-off for SI performance and manufacturability.

For the attachment surface pads, size and geometry is specified in TB-2345. For through vias (Outer layer/Inner layer) in general, pads should be as small as possible for best SI performance. Our assumption is that pads are typically 8 mils over drill size.

The Lynx connector system is versatile. You can use dog-bone style surface attach pads or via in pad (VIPPO). Refer to TB2345 for specific details on landing pad geometries

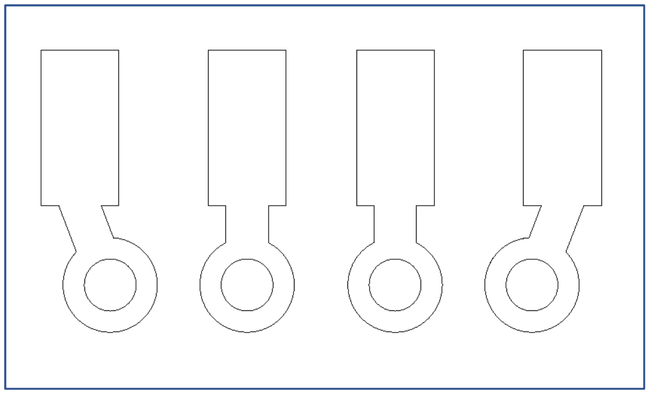


Figure 1: Traditional landing pad geometry example

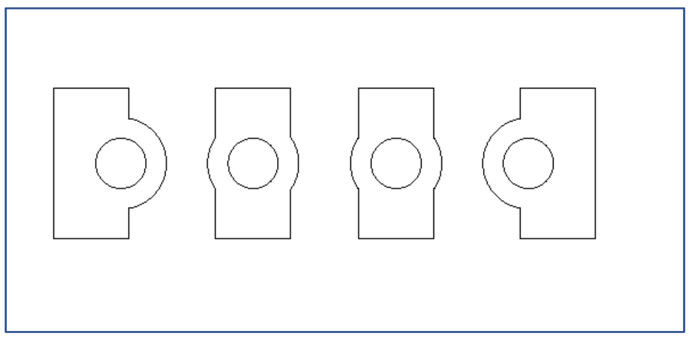
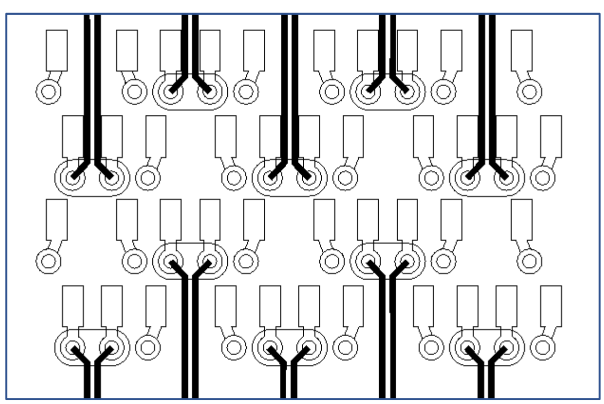


Figure 2: Via in pad (VIPPO) landing pad geometry example

7.2 Routing escape

Depending on the application, the Lynx footprint may be trace escaped from both sides of the footprint or just a single side. Refer to the below for example routing escapes



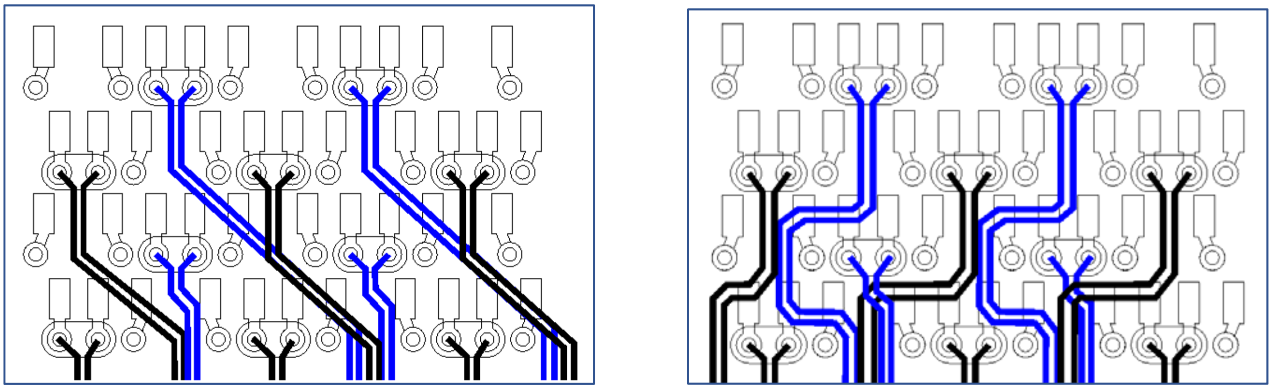


Figure 3: Routing Escape Options

7.3 Typical Anti-pad Geometry

The following shows the nominal antipad design for Lynx QD. The antipad can be either an oval shape or a rectangle. The choice will depend on the materials used (primarily the dielectric constant) and the target impedance. Typically, the rectangular antipad will provide a slightly higher via impedance. In most cases the width of the antipad should be maximized. The minimum size antipad will be based on the pwb supplies capabilities. Typically, the minimum would be 18mils over the via drill size. Variations to this design based on stackup design and performance requirements are allowed.

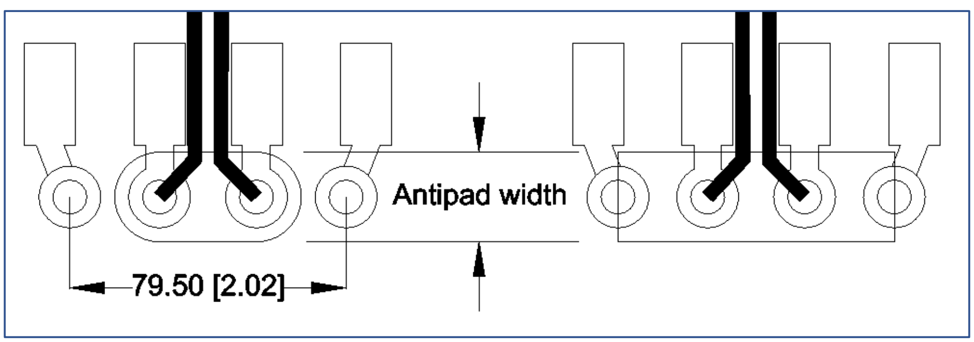


Figure 4: Typical Antipad Geometry

7.3.1 Additional Antipad Clearance – Lynx QD

For designs that use a traditional SMT pad geometry and incorporate a plane layer as the first layer below the SMT attach pads, it is recommended that an additional clearance be added to help raise the impedance of the SMT pads.

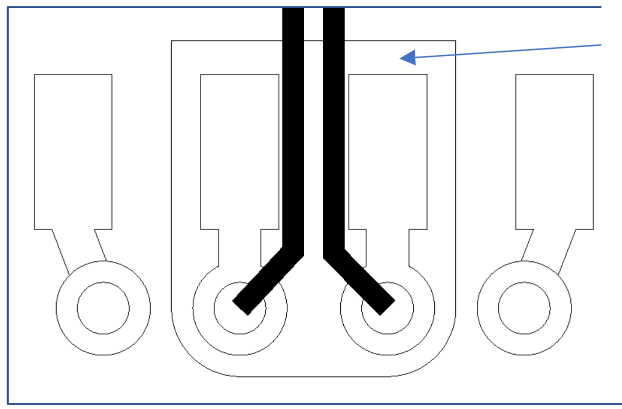


Figure 5: Antipad Clearance

7.4 Lynx QD Typical High-Speed Differential Routing

For complete hole pattern dimensions please refer to the customer use drawings. The available space for routing is determined by the distance between the ground vias. For the case of double-sided trace escapes, there are no practical routing constraints.

Note, The Lynx QD signal vias can be backdrilled for improved signal integrity. Contact your PCB fabrication vendor for recommendations on backdrill diameter.

Lynx QD Double sided escape

Center to Center (Via Space)	0.061 (1.55)
- Keep out around GND vias	0.026 (0.66)
= Resulting Space for Traces	**0.035 (0.89)**
Line Width	**0.006 (0.152)**
Space Between Lines	**0.006 (0.152)**
Resulting extra clearance on Each Side	**0.008 (0.216)**

Table 2: Example of Lynx QD Routing Channel

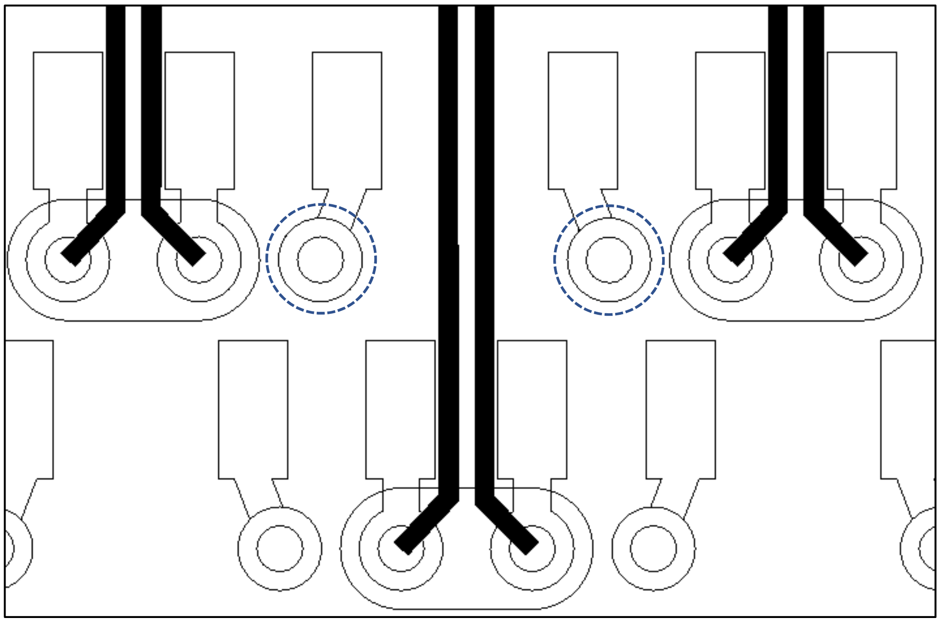


Figure 6: Example Lynx QD Differential Routing

Lynx QD Single side escape

Center to Center (Via Space)	0.061 (1.55)
- Keep out around ground vias	0.026 (0.66)
= Resulting Space for Traces	**0.035 (0.89)**
Line Width	**0.004 (0.101)**
Space Between Lines	**0.004 (0.101)**
Space Between Pairs	**0.011 (0.28)**

Table 3: Example of Lynx QD Routing Channel

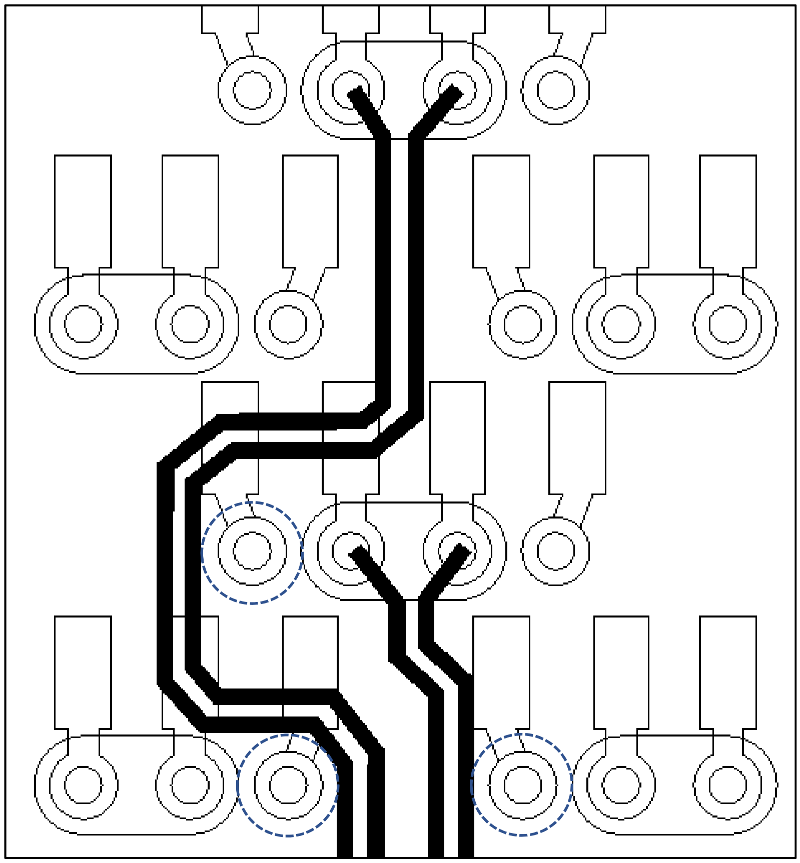


Figure 7: Example Lynx Differential Single side escape Routing