

**Amphenol** SOCAPEX

AMPHENOL R-VPX EVOLUTION  
AND EVOLUTION 2  
ROUTING GUIDELINES

**TABLE OF CONTENTS**

1. Scope .....	4
1.1 Intent .....	4
1.2 Efficient routing .....	4
1.3 Finished Hole Sizes .....	4
2. Definitions .....	4
2.1 Fillets.....	4
2.2 Foils/Copper Weights.....	4
2.3 Pads/Lands/Annular Ring .....	4
2.4 Spacing.....	4
2.5 Trace/Circuit/Line Width/Lines/Conductor .....	4
2.6 Backplane.....	4
2.7 Daughtercard.....	4
3. Routing Guidelines .....	5
3.1 Minimum Spacing .....	5
3.2 Impedance .....	5
3.3 Copper Weights.....	5
3.4 Fillets.....	5
3.5 Trace Centering.....	5
3.6 Non Functional Pads .....	5
4. Design Rules and Manufacturability Guidelines .....	5
4.1 General Design Rules.....	5
4.1.1 R-VPX Evo and Evo 2 Drill .....	5
4.1.2 Footprint.....	5
4.1.3 Drilled Hole and Copper Thickness .....	5
4.2 Daughtercard/Backplane Manufacturability Guidelines .....	6
4.2.1 Line Widths, Pad Sizes and Spacing.....	6
4.2.2 Filleting .....	6
4.2.3 Minimum PCB Thickness .....	6
4.2.4 Non-functional pads.....	6
4.2.5 Plane Clearances .....	6
4.2.6 Surface Traces.....	6
5. Routing Guidelines .....	6
5.1 Copper Thickness Requirement and Finished Thickness Reference.....	6
5.2 Drill and Finished Hole Size .....	10
5.3 Compliant Pin Critical Zone.....	12
5.4 Typical Antipad Geometry.....	15
5.5 R-VPX Evo and Evo 2 Typical High Speed Differential Routing .....	16
5.5.1 Example R-VPX Evo and Evo 2 Daughtercard Routing .....	16
5.5.2 R-VPX Evo and Evo 2 Daughtercard routing – Advanced features for improved signal integrity performance.....	17
5.5.3 Example R-VPX Evo and Evo 2 Backplane Routing .....	19
5.6.1 Elongated Antipad for Improved Return Loss .....	20
5.6.2 Example R-VPX Evo and Evo 2 Backplane Routing.....	22
5.6.3 Printed Circuit Board Skew Compensation .....	23

5.6.4	Example R-VPX Evolution and Evolution 2 Daughtercard PCB Ground Via Scheme .....	24
5.6.5	Example R-VPX Evolution 2 Backplane PCB Ground Via Scheme .....	25

**LIST OF FIGURES**

Figure 1: Preferred Fillet.....	9
Figure 2: BP Tail (0.0256”) Drill and Finished Hole Size.....	10
Figure 3: Nano Tail Drill (0.0177”) and Finished Hole Size .....	11
Figure 4: Femto Tail Drill (0.0157”) and Finished Hole Size.....	11
Figure 5: BP Tail (0.0256”) Compliant Pin Critical Zone .....	12
Figure 6: Nano Tail (0.0177”) Compliant Pin Critical Zone .....	13
Figure 7: Femto Tail (0.0157”) Compliant Pin Critical Zone.....	14
Figure 8: Antipad Clearance – Backplane Connector.....	15
Figure 9: Antipad Clearance – Daughtercard Connector.....	15
Figure 10: Example Daughtercard High Speed Differential Routing.....	16
Figure 11: Example Daughtercard Typical High Speed Differential Antipad Design.....	17
Figure 12: Example Daughtercard Typical High Speed Differential Antipad Design.....	18
Figure 13: Example Backplane High Speed Differential Routing .....	19
Figure 14: Option 1 Elongated Rectangular Backplane Antipad Clearance.....	20
Figure 15: Option 2 Elongated Rectangular Backplane Antipad Clearance .....	20
Figure 16: Performance Improvement of Elongated Antipad.....	21
Figure 17: Example Backplane Typical High Speed Differential Antipad Design.....	22
Figure 18: Example Backplane Typical High Speed Differential Antipad Design.....	22
Figure 19: Skew compensation recommended approach.....	23
Figure 20: Daughtercard PCB ground via dimensions .....	24
Figure 21: Daughtercard PCB near-end crosstalk .....	24
Figure 22: Backplane PCB ground via dimensions .....	25
Figure 23: Backplane PCB near-end crosstalk .....	25

**LIST OF TABLES**

Table 1: Copper Thickness Requirement and Finished Thickness Reference .....	6
Table 2: Example R-VPX Evolution Daughtercard Routing Channel.....	16
Table 3: Example R-VPX Evolution Backplane Routing Channel.....	19

## 1. Scope

### 1.1 Intent

The intent of this document is to outline the standard signal trace widths, minimum spacing requirements, and finish hole size requirements for R-VPX Evolution and Evolution 2 product family of interconnect systems when used in differential signal applications and low speed applications.

### 1.2 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.

### 1.3 Finished Hole Sizes

All finished hole size requirements provided within this document are based on testing completed in FR-4 laminate.

## 2. Definitions

### 2.1 Fillets

An extension of the pad at the interface of the trace to the pad that will allow more pad area, in the event that the pad to hole registration compromises the interconnect area. See Figure 1 for details. For further information regarding these routing guidelines, please contact AAO 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 0.008" width.

### 2.6 Backplane

When used within this document refers to the PCB associated with the male connector half of the connector system mounted to a fixed PCB in a chassis.

### 2.7 Daughtercard

When used within this document refers to the PCB associated with the female connector half of the connector system mounted to a plug-in card in a chassis.

### 3. Routing Guidelines

#### 3.1 Minimum Spacing

Minimum spacing, specific pad/trace, and trace/trace between all features should be 0.005" (0.127 mm) to allow for manufacturing tolerances.

#### 3.2 Impedance

Consider characteristic impedance (if applicable) when designing to ensure line widths will meet requirements. Please contact AAO Applications Engineering for impedance calculations.

#### 3.3 Copper Weights

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

#### 3.4 Fillets

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.

#### 3.5 Trace Centering

Center all traces between holes to optimize spacing.

#### 3.6 Non Functional Pads

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

### 4. Design Rules and Manufacturability Guidelines

#### 4.1 General Design Rules

##### 4.1.1 R-VPX Evolution and Evolution 2 Drill

BP Tail Hole: ISO 0.65 mm (0.0256") [Evolution backplane]

Nano Tail Hole: ISO 0.45 mm (0.0177") [Evolution 2 backplane]

Femto Tail Hole: ISO 0.40 mm (0.0157") [Evolution and Evolution 2 daughtercard]

##### 4.1.2 Footprint

For specific connector footprint see customer use drawings.

##### 4.1.3 Drilled Hole and Copper Thickness

For copper wall thickness requirements and finish hole size reference, see Table 1.

## 4.2 Daughtercard/Backplane Manufacturability Guidelines

### 4.2.1 Line Widths, Pad Sizes and Spacing

Line widths, pad sizes and spacing applicable for 1/2 ounce and 1 ounce copper weights.

### 4.2.2 Filletting

Filletting of pads recommended (to be added by fabricator) for 0.000” annular ring (tangency), see Figure 1.

### 4.2.3 Minimum PCB Thickness

BP Tail: Recommended minimum pcb thickness of 0.063” (1.6 mm).

Nano and Femto Tail: Recommended minimum pcb thickness of 0.043” (1.1 mm).

### 4.2.4 Non-functional pads

Non-functional pads on signal can be removed at designer’s option.

### 4.2.5 Plane Clearances

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

### 4.2.6 Surface Traces

Surface traces are not recommended. If surface traces are used refer to the customer drawings for keep-out zones.

## 5. Routing Guidelines

### 5.1 Copper Thickness Requirement and Finished Thickness Reference

Finish Type	Copper thickness, in (mm) per side	Drill size, inches (mm)	Typical Finish Thickness	Finished Hole Size, inches (mm)
<b>Solder Finish</b> <sup>(1)</sup>	0.0010 (0.0254) min BP tail 0.0025 (0.0635) max BP tail 0.0008 (0.020) min Nano tail 0.00275 (0.069) max Nano tail 0.0008 (0.020) min Femto tail 0.00275 (0.069) max Femto tail	BP Tail: 0.0256 (0.65) Nano Tail: 0.0177 (0.45) Femto Tail: 0.0157 (0.40)	300 to 500 micro inches	BP Tail: 0.0220 +/- 0.002 (0.56 +/- 0.05) Nano Tail: 0.0142 +/- 0.002 (0.36+/-0.05) Femto Tail: 0.0122 +/- 0.002 (0.31+/-0.05)
<b>Immersion Sn (Tin)</b>	0.0010 (0.0254) min BP tail 0.0025 (0.0635) max BP tail 0.0008 (0.020) min Nano tail 0.00275 (0.069) max Nano tail 0.0008 (0.020) min Femto tail 0.00275 (0.069) max Femto tail	BP Tail: 0.0256 (0.65) Nano Tail: 0.0177 (0.45) Femto Tail: 0.0157 (0.40)	35 to 75 micro inches minimum	BP Tail: 0.0220 +/- 0.002 (0.56 +/- 0.05) Nano Tail: 0.0142 +/- 0.002 (0.36+/-0.05) Femto Tail: 0.0122 +/- 0.002 (0.31 +/- 0.05)
<b>Immersion Ag (Silver)</b>	0.0010 (0.0254) min BP tail 0.0025 (0.0635) max BP tail 0.0008 (0.020) min Nano tail 0.00275 (0.069) max Nano tail 0.0008 (0.020) min Femto tail 0.00275 (0.069) max Femto tail	BP Tail: 0.0256 (0.65) Nano Tail: 0.0177 (0.45) Femto Tail: 0.0157 (0.40)	4 micro inches minimum	BP Tail: 0.0220 +/- 0.002 (0.56 +/- 0.05) Nano Tail: 0.0142 +/- 0.002 (0.36+/-0.05) Femto Tail: 0.0122 + 0.002/- 0.001 (0.31 + 0.05/- 0.025)

<b>Copper - OSP</b>	0.0010 (0.0254) min BP tail 0.0025 (0.0635) max BP tail 0.0008 (0.020) min Nano tail 0.00275 (0.069) max Nano tail 0.0008 (0.020) min Femto tail 0.00275 (0.069) max Femto tail	BP Tail: 0.0256 (0.65) Nano Tail: 0.0177 (0.45) Femto Tail: 0.0157 (0.40)	N/A	BP Tail: 0.0220 +/- 0.002 (0.56 +/- 0.05) Nano Tail: 0.0142 +/- 0.002 (0.36 +/- 0.05) Femto Tail: 0.0122 +/- 0.002 (0.31 +/- 0.05)
---------------------	--	---	-----	--

<b>Ni Au (Nickel-Gold)</b>	0.0010 (0.0254) min BP tail 0.0025 (0.0635) max BP tail 0.0008 (0.020) min Nano tail 0.00275 (0.069) max Nano tail 0.0008 (0.020) min Femto tail 0.00275 (0.069) max Femto tail	BP Tail: 0.0256 (0.65) Nano Tail: 0.0177 (0.45) Femto Tail: 0.0157 (0.40)	53 to 210 micro inches Ni- Au composition s combined	BP Tail: 0.0220 +/- 0.002 (0.56 +/- 0.05) Nano Tail: 0.0142 +/- 0.002 (0.36 +/- 0.05) Femto Tail: 0.0122 +/- 0.002 (0.31 +/- 0.05)
----------------------------	--	---	---	--

**Table 1: Copper Thickness Requirement and Finished Thickness Reference**

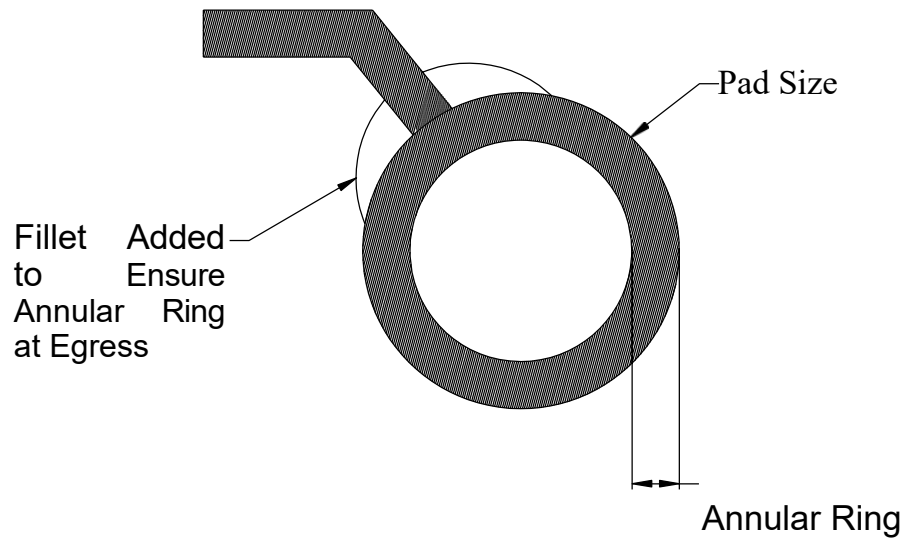
**NOTES (Table 1):**

1. Solder finish includes: Tin/lead reflowed (plated and reflowed) and HASL. Cu and HASL finished must combine to achieve the finished hole size specified.

**Copper thickness per side is an Amphenol requirement. The copper plating must be within specified range in Table 1: Copper Thickness Requirement and Finished Thickness Reference 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 1 below, of the plated thru hole, and cannot exceed the MAX copper plating thickness specified in Table 1: Copper Thickness Requirement and Finished Thickness Reference**

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 referencerange.
3. Typical Finish Thickness is an Amphenol reference value. 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. However, for ENIG, the nickel plating thickness is not a reference but is a requirement.
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.

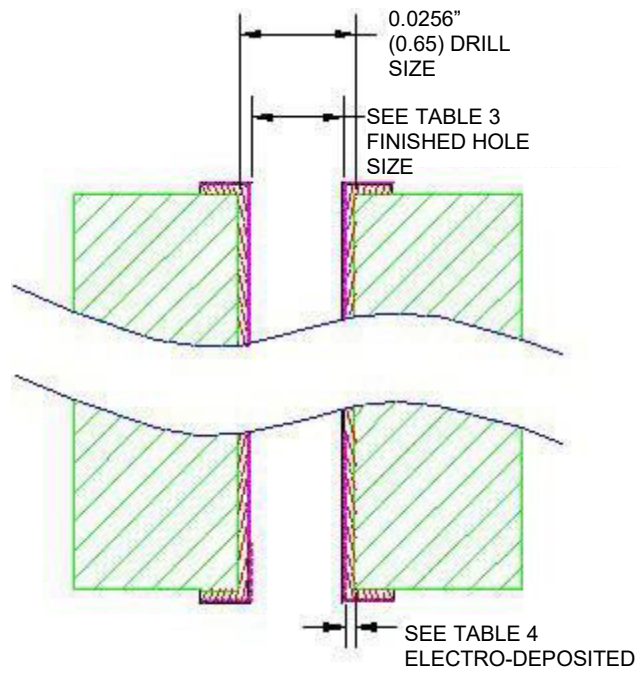




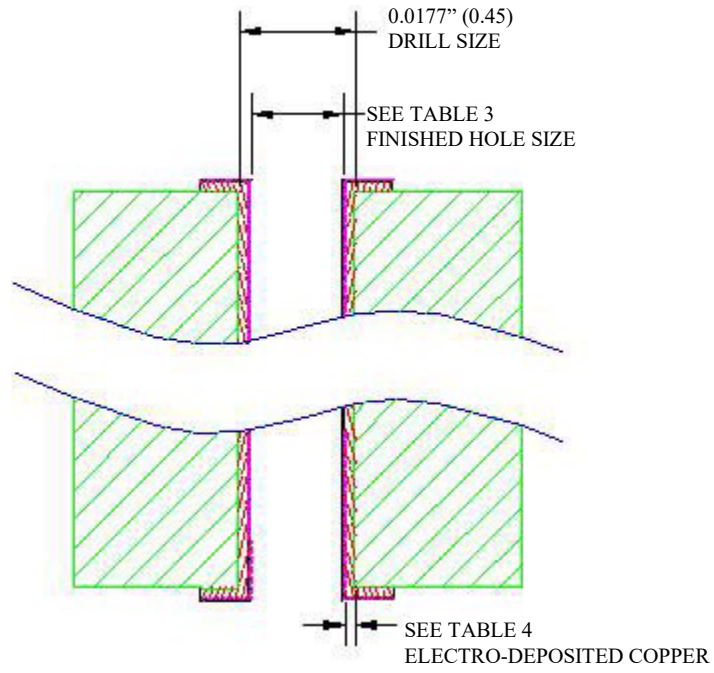
**Figure 1: Preferred Fillet**

Fillet Diameter equals one half size of plated through hole pad diameter located on a line central to trace so that fillet size equals minimum annular ring plus 0.005" (0.13 mm).

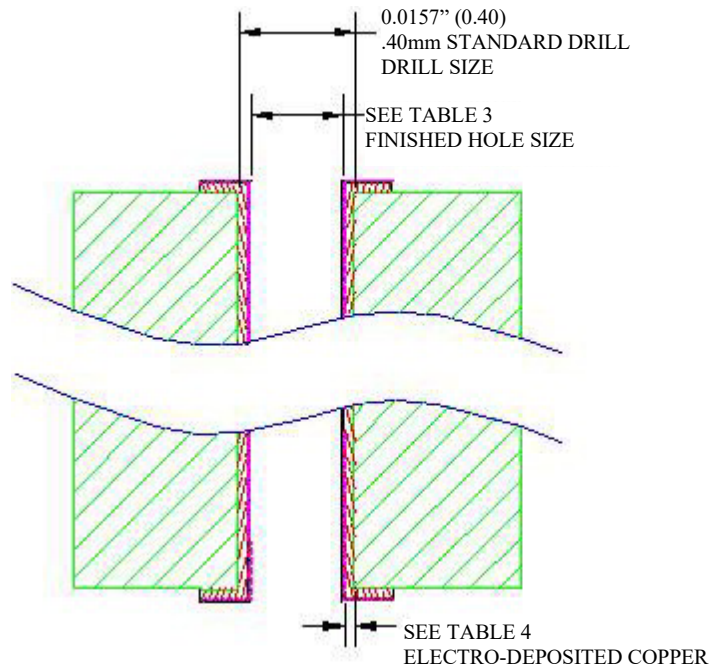
## 5.2 Drill and Finished Hole Size



**Figure 2: BP Tail (0.0256") Drill and Finished Hole Size**



**Figure 3: Nano Tail (0.0177") Drill and Finished Hole Size**



**Figure 4: Femto Tail Drill (0.0157") and Finished Hole Size**