1.0 OVERVIEW OF THE CONNECTOR

SpeedEdge is a line of Board to Board Connectors that connect one printed circuit board to another printed circuit board to which it's parallel; this type of connector is also known as a "mezzanine" connector. SpeedEdge is primarily for high speed Differential signals, but is also appropriate for Single-Ended signals, Low Speed signals and Power connections.

SpeedEdge can be made in varying heights to accommodate varying separation distances between two parallel boards in a variety of circuit sizes: 22, 60 & 82. Check with Molex for availability.

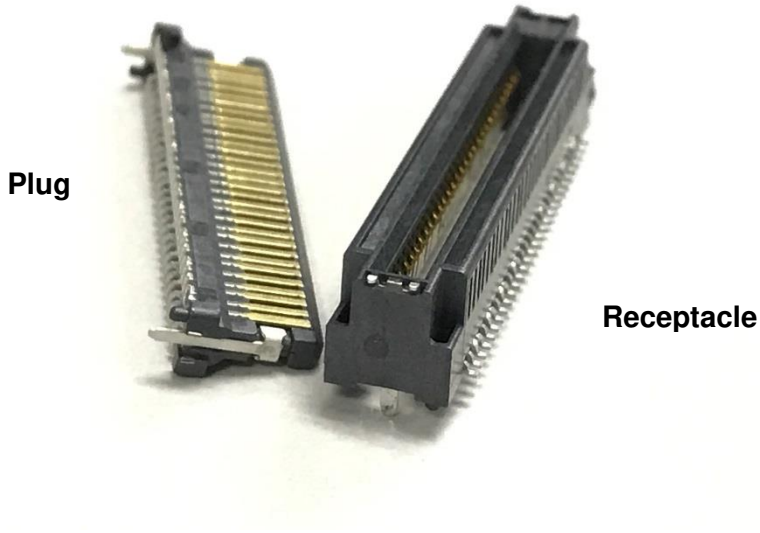


Figure 1: SpeedEdge connectors

2.0 PCB DESIGN REQUIREMENTS AND ROUTING STRATEGIES

Board Design Considerations for SpeedEdge

2.1. Recommended Footprint Design Considerations

1. The SpeedEdge terminals are arranged in dual lines. Refer to applicable sales drawings for more information.
2. Footprint optimization helps to improve electrical performance. Blind vias for signal routing are recommended to achieve best performance. Board stack-up illustrated are referenced to 8 layers board stack-up. Host board and plug board signals are routed with striplines on L2. Edge card signals are routed with striplines on L2 and L7.
3. On host boards and plug boards, ground voids are on **L3 ground plane** for better impedance control. **Ground strip** is used in L3 for good ground return.
4. Plug board includes linear ground via array for improved isolation between front and back rows.

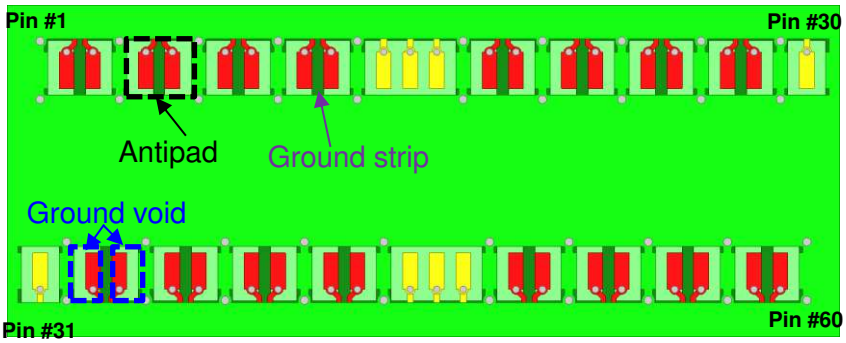


Figure 2: 60 Circuits receptacle footprint pattern

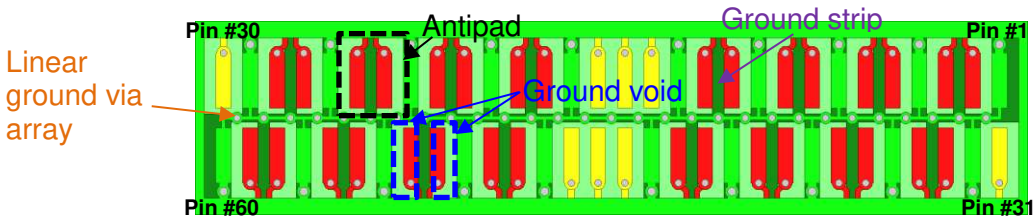


Figure 3: 60 Circuits plug footprint pattern

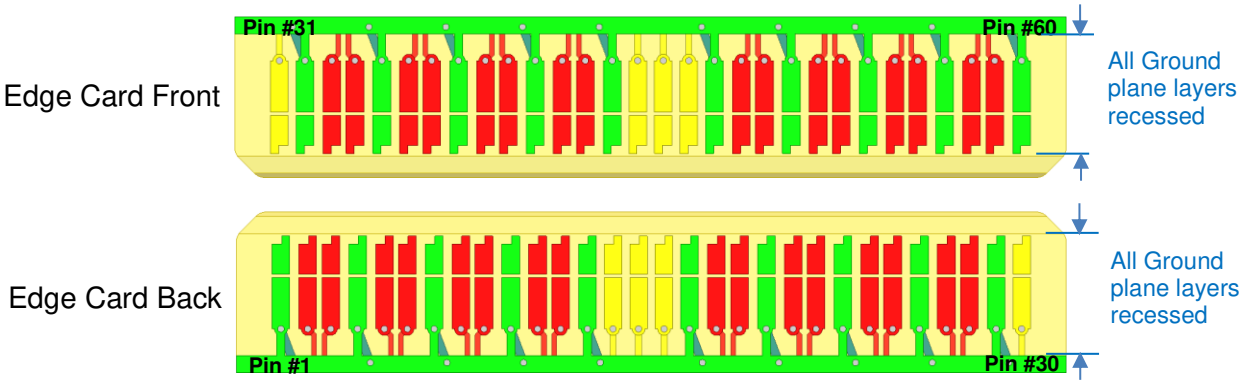
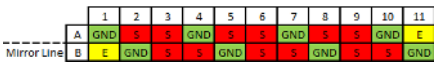
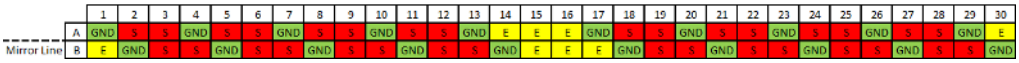


Figure 4: 60 Circuits edge card footprint pattern

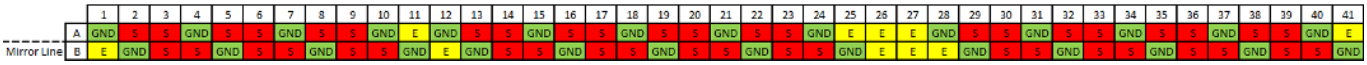
2.2. Example of Connector Pin configuration pattern



22 Circuits



60 Circuits



82 Circuits

YELLOW = Extra Pins

GREEN =GND (Ground)

RED = DIFF Gbps

Figure 5: Example of SpeedEdge pinout pattern

2.3. Recommended Anti-pad, Ground Strip and Ground Void Dimensions for SpeedEdge Footprint:

1. The anti-pads are rectangular in shape. For Top layer, anti-pad is a single rectangle. For L3 ground void, two rectangular voids are located beneath the SMT pads.
2. The size of the rectangle and the distance of the additional ground vias are tuned to get 100 ohm impedance.
3. Recommended anti-pad dimensions top layer are shown in Table 1.
4. Recommended L2 differential trace and ground strip dimensions are shown in Table 2.
5. Recommended ground void dimensions for L3 layer are shown in Table 3.
6. Recommended ground recess dimensions for Edge Card are shown in Table 4.

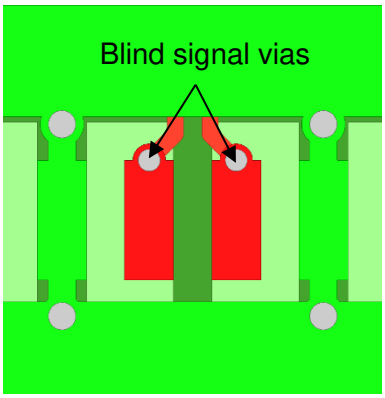


Figure 6: Anti-pad with trace and ground strip

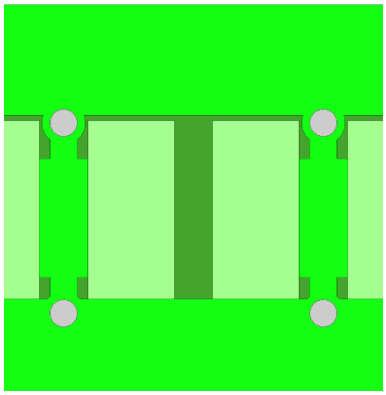


Figure 7: Anti-pad with only ground plane

Top Layer Anti-pad Design

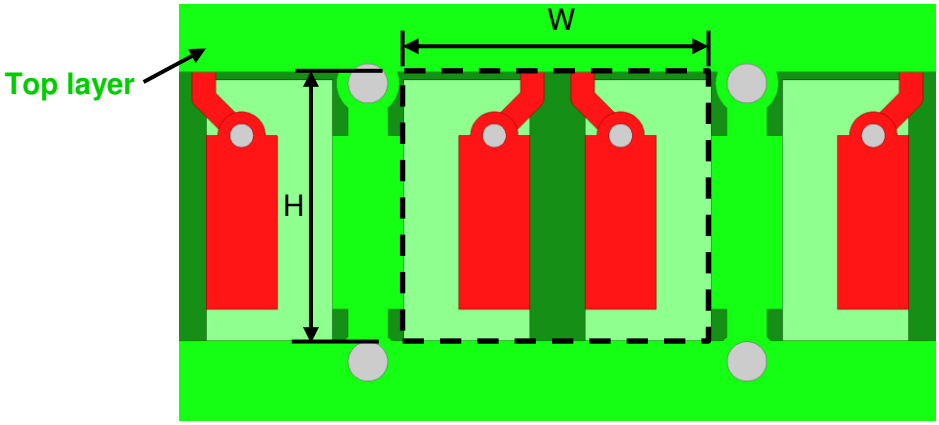


Figure 8: Receptacle anti-pad dimensions

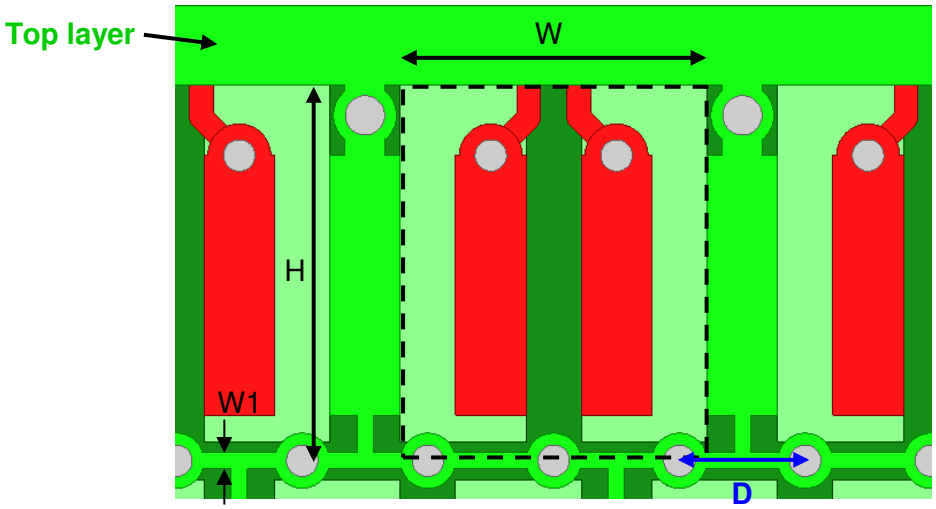


Figure 9: Plug anti-pad dimensions

Summary for recommended dimensions

Feature	Receptacle	Plug
Ground via drill size diameter	0.25mm	0.25mm
Anti-pad height, H	1.7mm	2.34mm
Anti-pad width, W	1.95mm	1.95mm
Via array distance, D	N/A	0.80mm
Via array drill size diameter	0.20mm	0.20mm
Via array trace width, W1	N/A	0.1016mm

Table 1

L2 Layer Differential Trace & Ground Strip Design

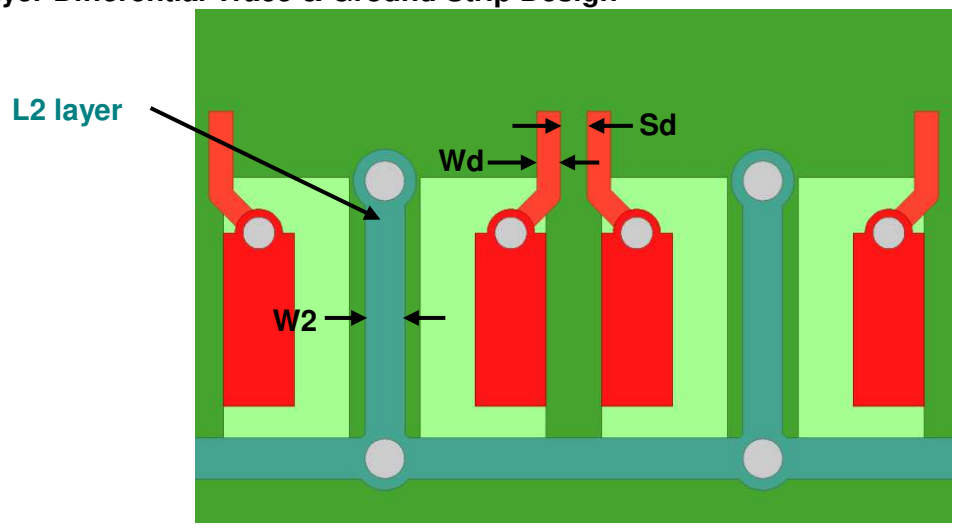


Figure 10: Receptacle board differential trace and ground strip dimensions

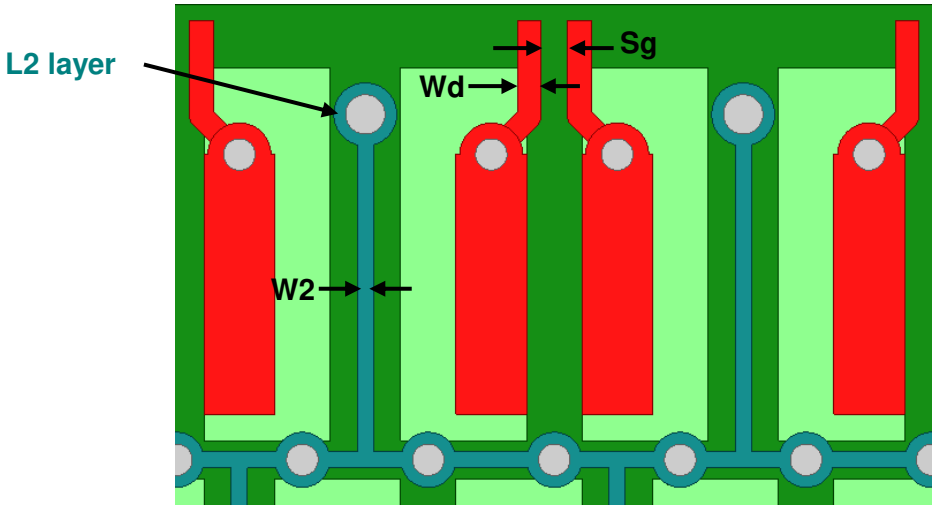


Figure 11: Plug board differential trace and ground strip dimensions

Summary for recommended dimensions

Feature	Receptacle	Plug
Ground strip width, W2	0.13mm	0.1016mm
Differential trace width, Wd	0.15mm	0.15mm
Differential trace gap, Sg	0.17mm	0.17mm

Table 2

L3 Layer Differential Trace & Ground Strip Design

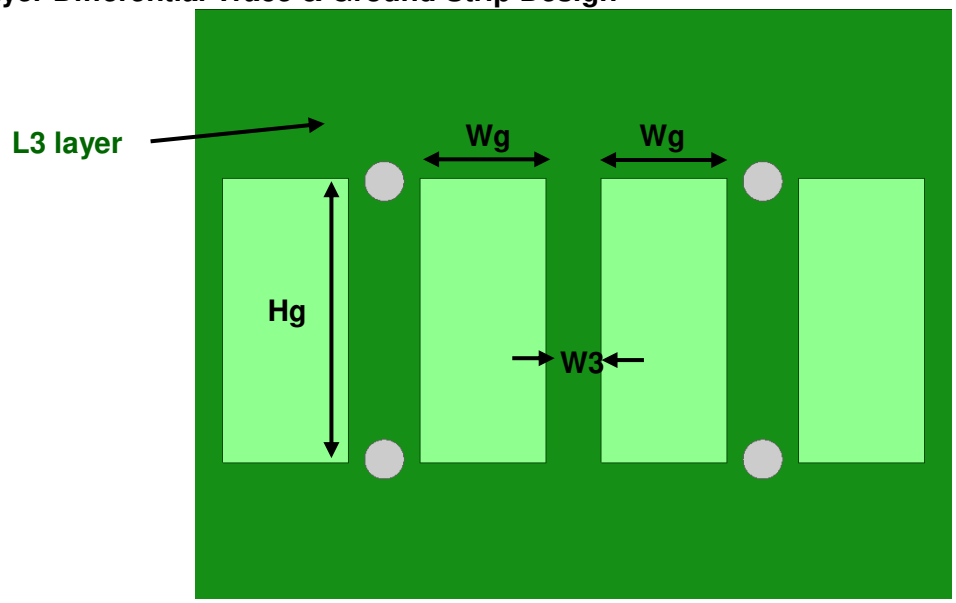


Figure 12: Receptacle board differential trace and ground strip dimensions

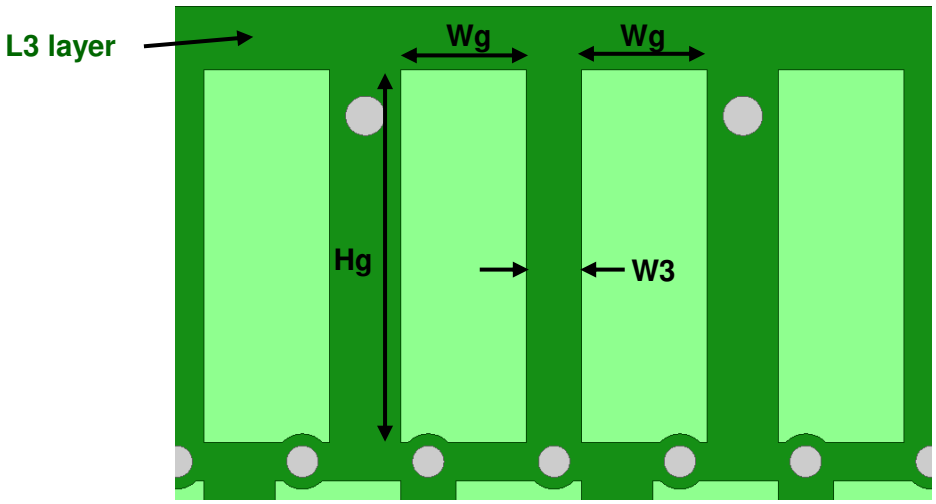


Figure 13: Plug board differential trace and ground strip dimensions

Summary for recommended dimensions

Feature	Receptacle	Plug
Ground void height, Hg	1.70mm	2.37mm
Ground void width, Wg	0.80mm	0.80mm
Ground strip width, W3	0.35mm	0.35mm

Table 3

Edge Card Ground Recess

Edge card example shown in this routing guide has an 8-layer board stack-up. Gold fingers are located on top and bottom layers. Trace routing are located on L2 and L7 for front and back of the edge card respectively. All ground plane layers are recessed with by Wr measured from the edge of the tie-bar.

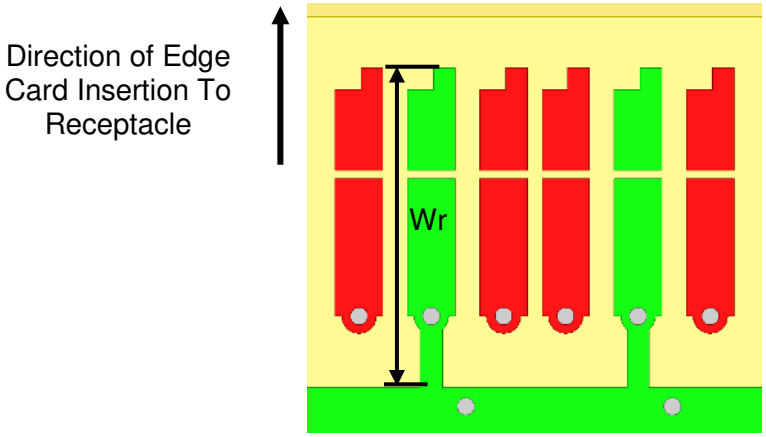


Figure 14: Edge card ground recess from edge of tie-bar, Wr, for all ground plane layers

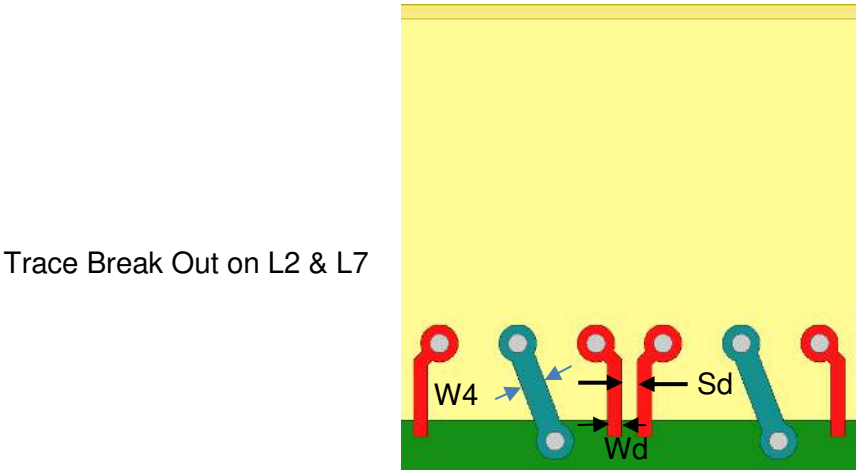


Figure 15: Edge Card differential trace and ground strip dimensions for L2 and L7 layers

Summary for recommended dimensions

Feature	Edge Card
Ground recess distance, Wr	3.725mm
Ground trace width, W4	0.254mm
Differential trace width, Wd	0.15mm
Differential trace gap, Sg	0.17mm

Table 4

2.4. Recommended routing for high speed differential signal trace.

1. Use symmetric signal traces.
2. Use zero skew traces.
3. On signal reference layers, a ground strip is used for impedance control and good ground return.
4. Routing can be done with blind vias (preferred), back drilled vias or through vias.
5. Short section of single-ended trace from via break-out may need to be tuned for impedance control.
6. Trace bending angle, α ≥ 45degrees. Refer to Figure 8.
   i. Spacing between the same pair, A ≥ 5 x of distance to reference plane.
   ii. Length segment B, C ≥ 5 x Trace Width.

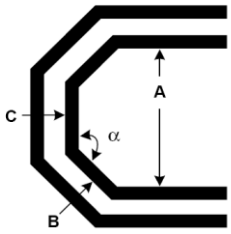


Figure 16: Trace bending angle

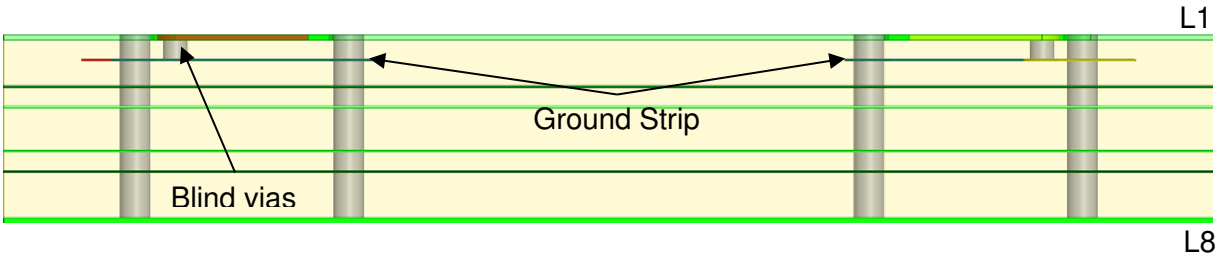


Figure 17: Signal layers routing with blind vias

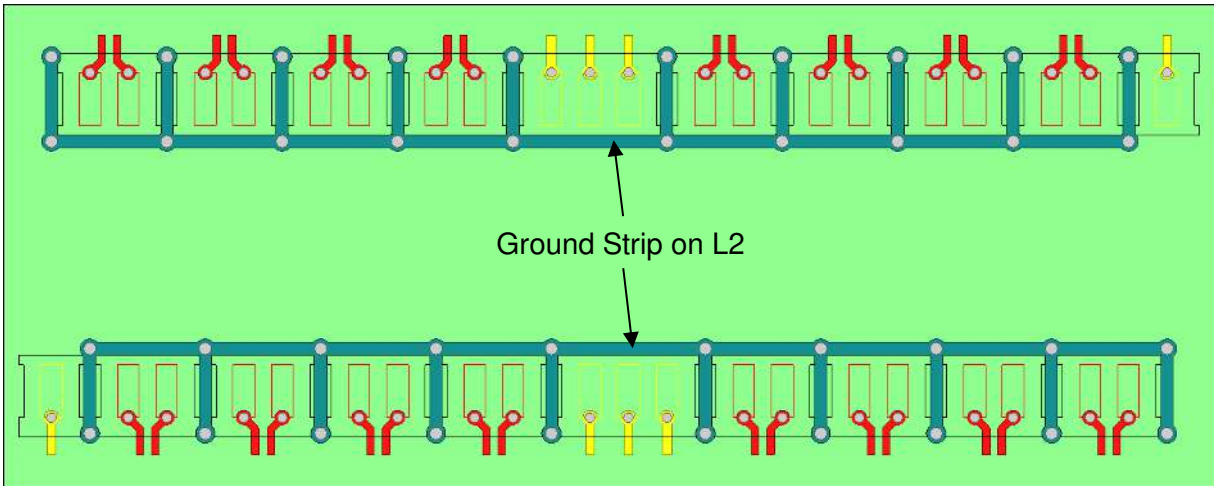


Figure 18: Typical trace escape route pattern on host board (Receptacle) L2 layer

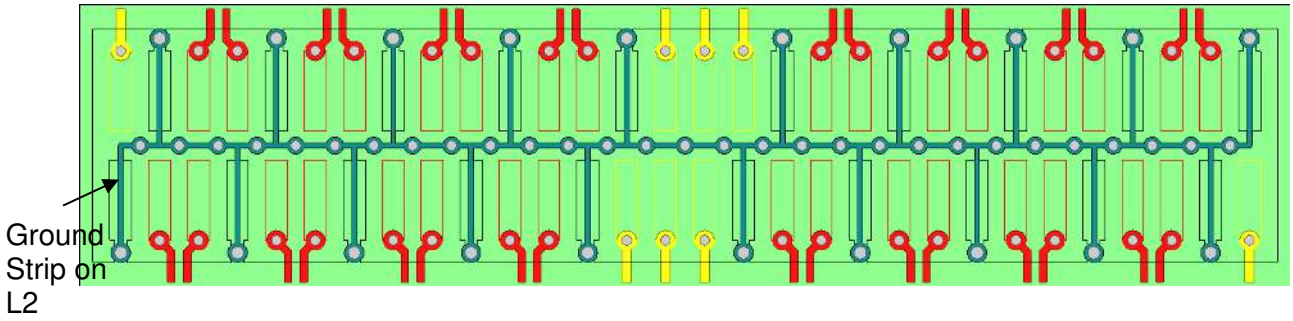


Figure 19: Typical trace escape route pattern on plug board (Plug) L2 layer

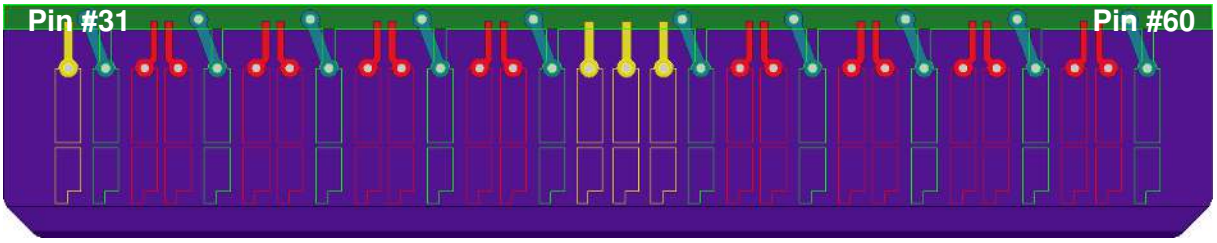


Figure 20: Typical trace escape route pattern on edge card L2 layer (Front)

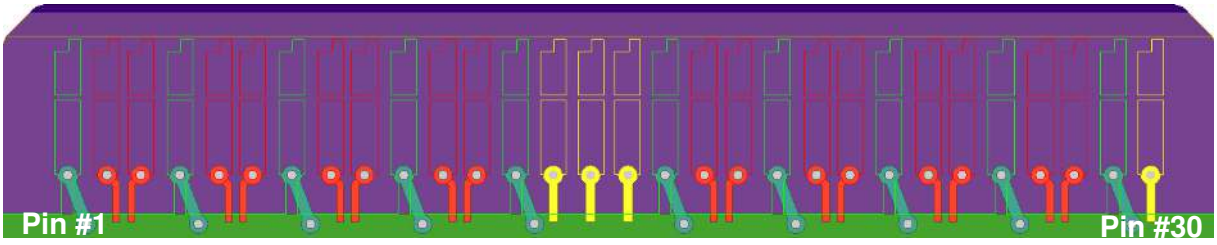


Figure 21: Typical trace escape route pattern on edge card L7 layer (Back)