<table>
<thead>
<tr>
<th>Type</th>
<th>Copper</th>
<th>Single Ended</th>
</tr>
</thead>
<tbody>
<tr>
<td>THICKNESS</td>
<td>Width</td>
<td></td>
</tr>
<tr>
<td>Differential</td>
<td>Impedance</td>
<td>Width &quot;Space&quot; Impedance</td>
</tr>
</tbody>
</table>

1. Copper stackup (copper thickness specified is after plating).

2. Impedance requirements shown in the stackup below.

3. Finish: solder with electroless nickel immersion gold (ENIG).

4. Conductors: minimum finished conductor width: 0.005" spacing: 0.005.

5. PCB thickness: 0.079 +/- 10%.

6. Etch or white paint on bottom side only.

7. No solder mask is permissible on pads unless provided this way.

8. Silk screen: silk screen shall be white non-conductive epoxy ink. Silk screen allowed.

9. Vendor marking: vendor name, approved initials, or logo, followed by the date.

10. No under cuts allowed on exposed copper.

11. Manufacturer and flame retardant class (4-F or 0) to be marked in

12. PCB: UL94-V0 minimum.

13. Board to be built to standard IPC-6012 Class II (latest release revision).

Secondary Side

Primary Side

Notes (unless otherwise specified)