Board Characteristics

1. Material - RF775
2. Minimum trace width: 4 mils
3. 0.5 Oz Copper on all layers
5. Polyimide Coverlay (Kapton Soldermask) on Top and Bottom, as per Gerbers.
6. Soft Au finish (ENIG) for Al wire bonding and hand soldering on all pads.
7. Place Stiffener on Top (Component) side of Flex PCB.
8. Total flex circuit thickness cannot exceed 0.012", not including stiffener.

Layer Order:
- Film 1 - Component Side
- Film 2 - Signal 2
- Film 3 - Power
- Film 4 - Signal 3
- Film 5 - Bottom

Stiffener 1mm Thick

<table>
<thead>
<tr>
<th>DRILL SYMBOL</th>
<th>DRILL SIZE</th>
<th>COUNT</th>
<th>PLATED</th>
<th>TOLERANCE</th>
<th>COMMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.00795</td>
<td>51</td>
<td>Yes</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.05</td>
<td>50</td>
<td>Yes</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.094488189</td>
<td>1</td>
<td>Yes</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.12</td>
<td>2</td>
<td>Yes</td>
<td>---</td>
<td></td>
</tr>
</tbody>
</table>

5-Layer Flex PCB

Specifications:

- Material: RF775
- Minimum Trace Width: 4 mils
- 0.5 Oz Copper on all layers
- Silkscreen on Component Side
- Polyimide Coverlay (Kapton Soldermask) on Top and Bottom, as per Gerbers
- Soft Au finish (ENIG) for Al wire bonding and hand soldering on all pads
- Place Stiffener on Top (Component) side of Flex PCB
- Total flex circuit thickness cannot exceed 0.012", not including stiffener