### BOARD's DRILL SCHEDULE

<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>.00795</td>
<td>1252</td>
<td>YES</td>
<td>---</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>.05</td>
<td>50</td>
<td>YES</td>
<td>---</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>.06496063</td>
<td>2</td>
<td>NO</td>
<td>---</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>.12</td>
<td>2</td>
<td>YES</td>
<td>---</td>
<td>---</td>
<td></td>
</tr>
</tbody>
</table>

### Board Characteristics

1. All dimensions in mm, unless specified
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 Gerber.
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

5-Layer Flex PCB

- **Film 1 - Component Side**
- **Film 2 - Signal 2**
- **Film 3 - Power**
- **Film 4 - Signal 3**
- **Film 5 - Bottom**

---

**Note:**
- Board Characteristics
- Layer Order
- Drilling Schedule
- 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 Gerber.
- Soft Au finish (ENIG) for Al wire bonding and hand soldering on all pads.
- Total flex circuit thickness cannot exceed 0.012", not including stiffener.