**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>80</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>.094488189</td>
<td>2</td>
<td>YES</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/gap: 5 mils.
3. 0.5 Oz Copper on both layers. Dielectric thickness: max 2.5 mils.
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. Materials can be changed as required by application.

**Layer Order**

2-Layer Flex PCB

- Film 1 - Component Side
- Film 2 - Bottom

**Diagram**

- Stiffener 1mm Thick

---

**UNIVERSITY OF CHICAGO**
**ELECTRONICS DEVELOPMENT GROUP**

**4CCD - PA Flex Specification Drawing**

**CONTRACT NO.**

**3007-B**

**DRAWN**

M. Bogdan

**CHECKED**

M. Bogdan

**APPROVED**

M. Bogdan

**ISSUED**

M. Bogdan

**SCALE**

1/1