**PCB specifications:**

Type of material: Rogers RO3010

Thickness: 1.5 mm

Board size: 38.1 x 35.6 mm (1.5 x 1.4 inch)

Quantity: 10 pcs

Conductive Layers: 2

Min track/spacing: 6 mil

Min hole size: >0.3 mm

Solder mask: none

Silkscreen: none

Surface Finish: Immersion gold

Copper layer: 1 oz

**File list:**

|  |  |
| --- | --- |
| Filename | Description |
| UTCPW\_MountingBoard-B\_Cu.gbr | Backside copper layer |
| UTCPW\_MountingBoard-Edge\_Cuts.gbr | Board edge cut layer |
| UTCPW\_MountingBoard-F\_Cu.gbr | Frontside copper layer |
| UTCPW\_MountingBoard-F\_Paste.gbr | Frontside solder paste layer |
| UTCPW\_MountingBoard-User\_Eco1.gbr | Top side z-axis milling region |
| UTCPW\_MountingBoard-job.gbrjob | Gerber job file |
|  |  |
| UTCPW\_MountingBoard-NPTH.drl | Non-plated through holes drilling file |
| UTCPW\_MountingBoard-PTH.drl | Plated through holes drilling file |
|  |  |

Notes:

1. No solder paste on the backside, so no \*-B\_paste.gbr gerber file is uploaded.
2. No solder mask and silk screen, so no \*\_mask.gbr or \*\_Silkscreen.gbr gerber files are uploaded.
3. -User\_Eco1.gbr file is used to define the top side milling region. The intended milling depth is 0.5 mm.