



TGA2760-SM

PCB Instruction



General

- Material = 8mil RO4003 boards with 0.5oz copper
- Solder mask = blue
- The zip folders contain gerber files and drill files representing four different board types built up using the following:
 - `_top_metal.gerber`
 - `_top_solder_mask.gerber`
 - `_top_ink.gerber`
 - `_bottom_metal.gerber`
 - `_drill.gerber`
 - `_board_outline.gerber`
 - `_drill.drl`
- The layout was done using a *mil* scale.
- *No* solder mask on the backside of the boards.



General

- Drill holes:
 - Please remove solder mask from all drill holes by modifying the `_top_solder_mask.ger` file
 - There are two types of drill holes for these boards: plated-thru-holes and copper-filled. See following slides for details.



TGA2760-SM - QTY: 50

- Holes Inside Green Box:

- Copper filled vias
- QTY = 391
- Drill at 6 mil (0.1524mm)

- Holes Outside Green Box:

- Plated Thru Holes
- Finished sizes =
 - 20 mil (0.508mm)
 - 35 mil (0.889mm)
 - 82 mil (2.083mm)
- TGA2781_SM_apps_board_DRILL_.drl

- Gold Finish = ENIG

