

GaN Systems Inc.
 1145 Innovation Dr. Suite 101
 Ottawa, ON
 Canada K2K 3G8
 Tel. +1-613-686-1996

| | |
|-----------------------------------|--|
| Project Description | GS66502B/04B HALF BRIDGE DAUGHTER BOARD |
| Board Title: | GS66502-04B HALF BRIDGE DAUGHTER BOARD |
| Part Number: | GS66502-04B-EVBDB |
| Revision: | A |
| Size (length x width): | See Design files |
| Shape: | Regular 85 x 35mm |
| Material: | 0.063" +/- [10%] FR4 High Tg 170 |
| Number of Layers: | 4 |
| Component Type | Mixed, Both sides |
| Hole diameter | Finished hole size |
| Gold Finger | No |
| Impedance | No |
| Plug Vias: | None |
| Minimum Trace Width/Gap | 0.254mm (10mi) |
| Minimum Hole Size | 0.3mm (12mi) |
| Copper Thickness | 2oz. Cu all layers |
| Board Surface Finish | ENIG |
| Soldermask: | SMOBC. Green on both sides |
| Silkscreen: | White, Top & Bot side |
| Paneling: | Not required. Allowable at discretion of board manufacturer if cost effective |
| Electrical Test: | YES |
| Special Info/Instructions: | <ol style="list-style-type: none"> 1. PCB Manufacturing must be compliant with IPC-6012 Class 2 2. Mechanical dimensions, drilling drawings and other manufacturing specifications provided in the *-FAB.PDF document. |

The following files have been bundled within the ZIP file for quotation and manufacture of the PCB.

| FILE NAME | DESCRIPTION | FILE FORMAT |
|---|-----------------------------|-------------|
| TOP.ART | TOP COPPER LAYER | GERBER |
| SILKSCREENTOP.ART | TOP SILKSCREEN | GERBER |
| SOLDERMASKTOP.ART | TOP SOLDERMASK | GERBER |
| PASTEMASKTOP.ART | TOP PASTEMASK | GERBER |
| M1.ART | INTERNAL LAYER 1 | GERBER |
| M2.ART | INTERNAL LAYER 2 | GERBER |
| BOTTOM.ART | BOTTOM COPPER LAYER | GERBER |
| SILKSCREENBOT.ART | BOTTOM SILKSCREEN | GERBER |
| SOLDERMASKBOTTOM.ART | BOTTOM SOLDERMASK | GERBER |
| PASTEMASKBOT.ART | TOP PASTEMASK | GERBER |
| FABRICATION.ART | FABRICATION/MECHNICAL LAYER | GERBER |
| NCDRILL-1-4.DRL | NC DRILL FILE | EXCELLON |
| NCDRILL-1-4-NP.DRL | NC DRILL FILE (NON-PLATED) | EXCELLON |
| IPC-D-356A.IPC | NETLIST FILE | IPC-D-356A |
| *-BRDSPEC.PDF | PCB SPEC FILE (THIS FILE) | PDF |
| *-FAB.PDF | FABRICATION DRAWING | PDF |
| FILES GENERATED BY: ALLEGRO 16.6 GERBER File Format: Unit (millimeters), RS274X, 3.3, TRAILING ZERO NDRILL FILE: Unit (metric), FORMART 3.3, TRAILING ZERO | | |