

GaN Systems Inc.  
1145 Innovation Dr. Suite 101  
Ottawa, ON  
Canada K2K 3G8  
Tel. +1-613-686-199

<b>Project Description</b>	<b>GS66508/16T EVALUATION DAUGHTER BOARD</b>
<b>Board Title:</b>	GS66508/16T HALF BRIDGE DAUGHTER BOARD
<b>Part Number:</b>	GS66508-16T-EVBDB
<b>Revision:</b>	A
<b>Size (length x width):</b>	85mm x 35mm
<b>Shape:</b>	Rectangular
<b>Material:</b>	1.6mm +/- [10%] FR4 High Tg
<b>Number of Layers:</b>	4
<b>Component Type</b>	Mixed, Both sides
<b>Hole diameter</b>	Finished hole size
<b>Gold Finger</b>	No
<b>Impedance</b>	No
<b>Plug Vias:</b>	None
<b>Minimum Trace Width/Gap</b>	0.254mm (10mi)
<b>Minimum Hole Size</b>	0.3mm (12mi)
<b>Copper Thickness</b>	2oz. Cu all layers
<b>Board Surface Finish</b>	ENIG
<b>Soldermask:</b>	SMOBC. Green on both sides
<b>Silkscreen:</b>	White, Top and Bot Sides
<b>Paneling:</b>	Not required. Allowable at discretion of board manufacturer if cost effective
<b>Electrical Test:</b>	NOT REQUIRED
<b>Special Info/Instructions:</b>	<ol style="list-style-type: none"><li>1. PCB Manufacturing must be compliant with IPC-6012 Class 2</li><li>2. Mechanical dimensions, drilling drawings and other manufacturing specifications provided in the *-FAB.PDF document.</li></ol>

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 PASTE STENCIL MASK	GERBER
M1.ART	INTERNAL LAYER 1	GERBER
M2.ART	INTERNAL LAYER 2	GERBER
BOTTOM.ART	BOTTOM COPPER LAYER	GERBER
SOLDERMASKBOTTOM.ART	BOTTOM SOLDERMASK	GERBER
SILKSCREENBOT.ART	BOTTOM SILKSCREEN	GERBER
PASTEMASKBOTTOM.ART	BOTTOM PASTE STENCIL MASK	GERBER
FABRICATION.ART	FABRICATION/MECHNICAL LAYER	GERBER
NCDRILL.DRL	NC DRILL FILE	EXCELLON
NCDRILL-NP.DRL	NC DRILL FILE (NON-PLATED)	EXCELLON
IPC-D-356A.IPC	IPC D 356A NETLIST	IPC-D-356A
*-BRDSPEC.PDF	PCB SPEC FILE (THIS FILE)	PDF
*-FAB.PDF	FABRICATION DRAWING	PDF
<b>FILES GENERATED BY: ALLEGRO 16.6</b> GERBER File Format: Unit (millimeters), RS274X, 3.5, LEADING ZERO NDRILL FILE: Unit (metric), FORMART 3.5, LEADING ZERO		