

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

Project Description	650V GAN UNIVERSAL MOTHERBOARD
Board Title:	GS665EVBMB
Part Number:	GS665EVBMB
Revision:	B
Size (length x width):	160X100mm
Shape:	Rectangular
Material:	0.063" +/- [10%] FR4 high Tg170
Number of Layers:	2
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	0.43mm (17mi)
Copper Thickness	2oz. Cu all layers
Board Surface Finish	ENIG Immersion Gold
Soldermask:	SMOBC. Green on both sides
Silkscreen:	White, Top and BOT sides
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
SILKSCREENBOTTOM. ART	BOTTOM SILKSCREEN	GERBER
SOLDERMASKTOP. ART	TOP SOLDERMASK	GERBER
PASTEMASKTOP. ART	TOP SOLDERPASTE MASK	GERBER
BOTTOM. ART	BOTTOM COPPER LAYER	GERBER
SOLDERMASKBOTTOM. ART	BOTTOM SOLDERMASK	GERBER
FABRICATION. ART	FABRICATION/MECHANICAL DRAWING	GERBER
NCDRILL. DRL	NC DRILL FILE	EXCELLON
NCDRILL-NP. DRL	NC DRILL FILE (NON-PLATED)	EXCELLON
IPC-D-356A. IPC	NETLIST	IPC-D-356A
*-SPEC. PDF	PCB SPEC FILE (THIS FILE)	PDF
*-FAB. PDF	FABRICATION DRAWING	PDF
FILES GENERATED BY: ALLEGRO 16.6 GERBER File Format: Unit (millimeters), RS274X, FORMAT 3.3, TRAILING ZERO NDRILL FILE: Unit (metric), FORMART 3.3, TRAILIG ZERO, Enhanced Excellon		