PRINTED CIRCUIT BOARD FABRICATION NOTES

Efficient Power Conversion
909 N. Pacific Coast Hwy. ste230
El Segundo, CA 90245
U.S.A.
Tel: +1 (310) 615 0279
Fax: +1 (310) 615 0284

Printed Circuit board fabrication instructions, unless otherwise specified:

1. Any changes to the design must be approved by EPC in writing.
2. Fabricate using Gerber artwork. Do not add any text, logos, etc. to the top side of the board.
3. Acceptability of PCB Fabrication, except as specified in this drawing, must comply to IPC-6012B, class 2.
   Compelete array flatness must be within 7.5 mil per inch
4. Material, Copper Clad Laminate sheet in accordance with IPC-4101.
   Material: Use FR370-HR Substrate or similar. Must have the following Properties:
   a. Tg(min) = 180 degrees C
   b. Td(min) = 340 degrees C
   c. Flame rating 94V0
   d. Dielectric = 4.0/5.0
5. All holes to be located by pad centers & be within .003 inches dia. of true position.
   Layer to layer registration to be within 0.003 inches
6. All vias to be tented on both sides of the board
   All vias (0.0177 inches hole and smaller) are to be filled (non conductive) and plated over.
7. All holes to be Copper (Cu) Plated thru per IPC-2221, 0.000787 inches min. thickness
8. Solder mask created 1:1, do not enlarge or modify.
   Do not clip solder mask.
   Solder Mask over bare Copper, both sides, using Taiyu 4000HFX L.P.I or PSR-2000/ LF02/CA-25 or equiv.,
   Color green per IPC-SM-840, class T, RoHS Compliant
   LDI mask must be used to register tolerance.
   Solder Mask to copper layer registration to be within 0.003 inches
9. Finish, all exposed Copper, both sides, after solder mask, to be Electroless Nickle Immersion Gold (Ni thickness typical 150 micro inches and Au 3-5 micro inches) in accordance with IPC-4552
10. Silkscreen Both sides using white permanent ink.
    Silkscreen to copper layer registration to be within 0.003 inches
11. Cut out according to mechanical layer 3 (cutout), milling tool for J60 slot = 1.27 mm diameter.