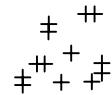


ECN#	ISS.	REV.	DESCRIPTION	DATE	INT.	PG#
.	1	A	INITIAL RELEASE	6-15-10	S	XXX
.	1	B	MOVED VIAS	10-3-12	S	.
.
.
.
.

NOTES

MATERIAL: FR4 1 OZ COPPER EACH LAYER.
 ALL HOLES PLATED THRU WITH 1 OZ COPPER MIN.
 FINAL COPPER ON EXTERNAL LAYERS = 2 OZ. MINIMUM.

For best thermal performance,
 Make GND copper as large as
 possible, except on layer 1
 directly under U1.



This layout shows the critical components
 and traces for minimum footprint with ENABLE
 tied to Vin. Alternate ENABLE configurations,
 and other small signal pins need to be connected
 and routed according to the customer application.

Gerber Format: RS274X, 3.5
 Copyright Enpirion 2012

All spacings have been set according to common
 PCB assembly practices. If necessary,
 these spacings can be modified according
 to specific manufacturing capabilities.

Solder mask and paste layer openings should be
 reviewed and optimized to meet customer
 PCB fabrication and assembly guidelines.

SIZE	QTY	SYM	PLATED	TOL
5	13	+	YES	+/-3.0

PLUGGING/FILLING OF 5mil HOLES IS ACCEPTABLE.

ENPIRION			
DRAWN BY	DATE	P/N 04588, EP5357XUI-CCC. BD	
S	6-15-10	FAB	
UNLESS OTHERWISE SPECIFIED TOLERANCES: .XX +/- .020 .XXX +/- .010 ANG. +/- 1 DEG.		SIZE A	ISSUE 1
		REV. B	DRAWING NO. .
		SCALE: 2	SHEET: 1 OF 1