

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 DESIGNS WITH LESS THAN 4 LAYERS, CONTACT ENPIRION APPLICATIONS SUPPORT.

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



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

EP5348UI Critical Component Circuit

This layout shows the critical components
 and traces for minimum footprint with ENABLE
 tied to Vin. Alternate ENABLE configurations, & other small
 signals should be routed according to specific application.

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	STD
8	4	X	YES	STD

NOTE: USE 13 MIL DRILL FOR 8 MIL PLTD HOLES
 USE 10 MIL DRILL FOR 5 MIL PLTD HOLES
 PLUGGING/FILLING OF THESE HOLES IS ACCEPTABLE.

Gerber Format: RS274X, 3.5
 Copyright Enpirion 2010