The complete schematic of EL7516 demo board is shown in Figure 1. With different components stuffed, this demo board can easily be configured into the applications shown in this technical brief. The standard demo board being sent out is the complete 5V to 12V converter shown in Table 1, which occupies less than 0.4 in^2 area with components on top side only. Layout diagrams are given at the end of this document.

1. V_{IN} = 2.5V-5.5V. Can be higher with higher voltage rated C_1
2. V_O = 12V. Can be set according to the following formula:

$$V_O = V_{FB} \times \left(1 + \frac{R_1}{R_2}\right)$$

where V_{FB} is 1.294V.

3. I_O - up to 750mA depending on input voltage, provided that diode and inductor can handle the corresponding currents (refer to data sheet for the chart)
4. Switching frequency - can be selected between 620KHz (FSEL <0.6V) or 12.5MHz (FSEL >2.7V)
5. R_3 and C_5 are the compensation network.

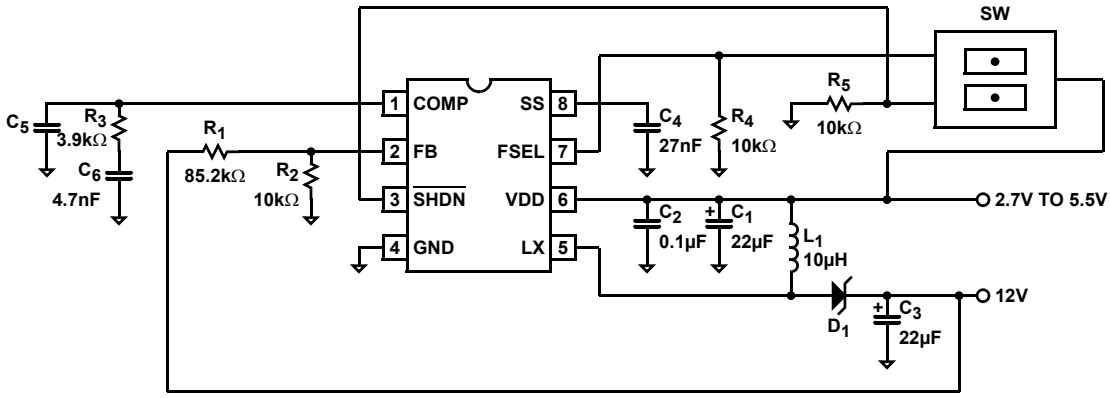

FIGURE 1. 3.3V TO 12V CONVERTER

TABLE 1. EL7516 BILL OF MATERIALS FOR V_O = 12V

REFERENCE DESIGNATOR	VALUE	MANUFACTURER	MANUFACTURER'S PHONE NUMBER	MANUFACTURER'S PART NUMBER
R_1	85.2kΩ/1%, 0603	Any		
R_2	10kΩ/1%, 0603	Any		
R_3	3.9kΩ, 0603	Any		
R_4, R_5	10kΩ/1%, 0603	Any		
C_1, C_3	22µF, 16V, 1812	TDK		C4532X5R1C226M
C_2	0.1µF, 16V, 0805	Any		
C_4	27nF, 0603	Any		
C_5	OPEN			
C_6	4.7nF, 0603	Any		
L_1	10µH	Coilcraft	847-639-6400	DO1608C-103
D_1	MBR0520	On Semiconductor	800-282-9855	MBR0520

Demo Board Layout

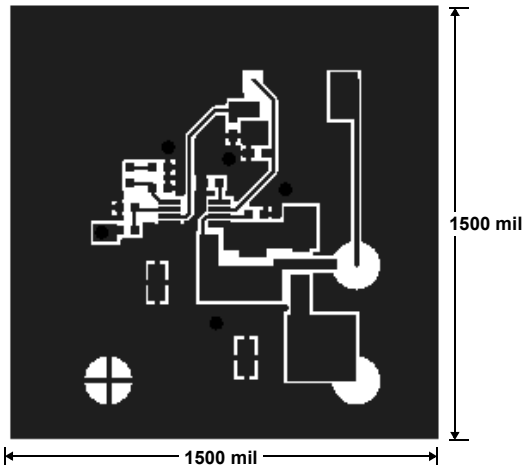

FIGURE 2. TOP LAYER

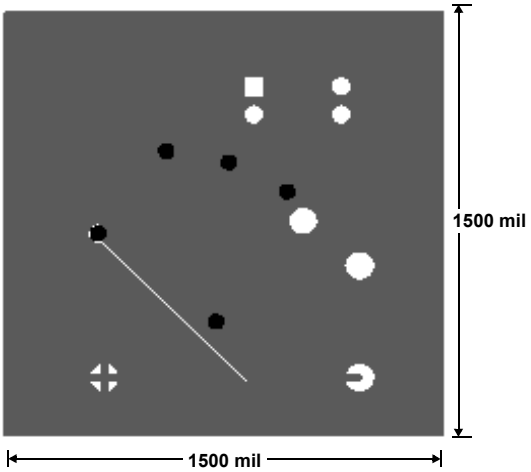

FIGURE 3. BOTTOM LAYER

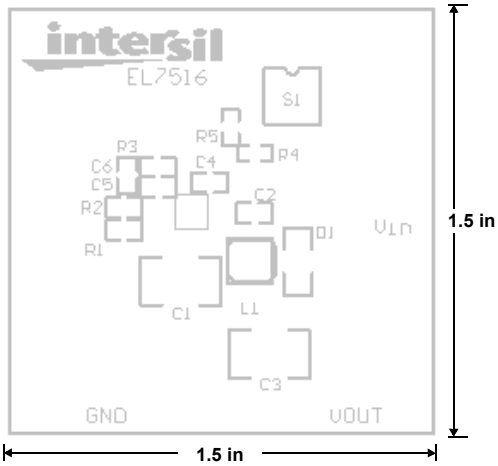

FIGURE 4. TOP SILKSCREEN

Layout Guidelines

To achieve highest efficiency, best regulation and most stable operation, a good printed circuit board layout is essential. It is strongly recommended that the demo board layout to be followed as closely as possible. Use the following general guidelines when laying out the print circuit board:

1. Place C_4 as close to the V_{DD} pin as possible. C_4 is the supply bypass capacitor of the device.
2. Keep the C_1 ground, GND pin and C_2 ground as close as possible.
3. Keep the two high current paths a) from C_1 through L_1, to the LX pin and GND and b) from C_1 through L_1, D_1, and C_2 as short as possible.
4. High current traces should be short and as wide as possible.
5. Place feedback resistor close to the FB pin to avoid noise pickup.
6. Place the compensation network close to the COMP pin.

The demo board is a good example of layout based on these principles; it is available upon request.