

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<sup>2</sup> 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 ( $F_{SEL} < 0.6V$ ) or 12.5MHz ( $F_{SEL} > 2.7V$ )
5.  $R_3$  and  $C_5$  are the compensation network.

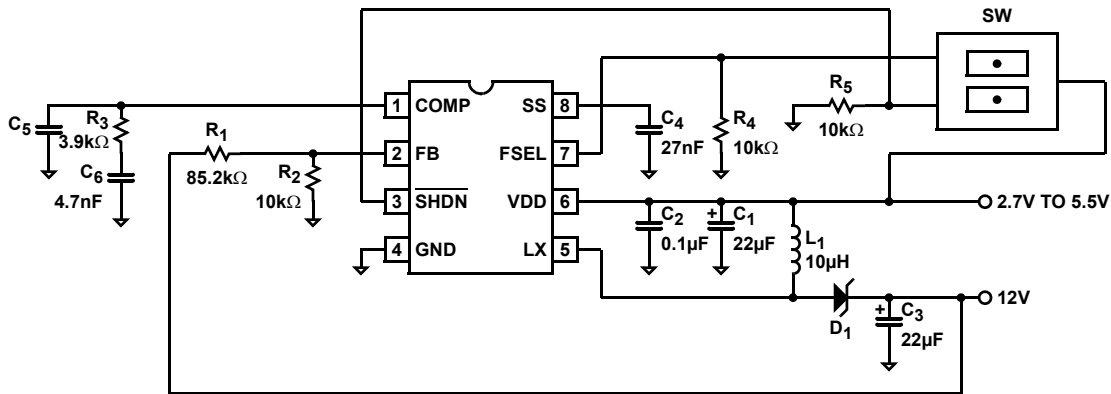


FIGURE 1. 3.3V TO 12V CONVERTER

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

REFERENCE DESIGNATOR	VALUE	MANUFACTURER	MANUFACTURER'S PHONE NUMBER	MANUFACTURER'S PART NUMBER
R <sub>1</sub>	85.2kΩ/1%, 0603	Any		
R <sub>2</sub>	10kΩ/1%, 0603	Any		
R <sub>3</sub>	3.9kΩ, 0603	Any		
R <sub>4</sub> , R <sub>5</sub>	10kΩ/1%, 0603	Any		
C <sub>1</sub> , C <sub>3</sub>	22μF, 16V, 1812	TDK		C4532X5R1C226M
C <sub>2</sub>	0.1μF, 16V, 0805	Any		
C <sub>4</sub>	27nF, 0603	Any		
C <sub>5</sub>	OPEN			
C <sub>6</sub>	4.7nF, 0603	Any		
L <sub>1</sub>	10μH	Coilcraft	847-639-6400	DO1608C-103
D <sub>1</sub>	MBR0520	On Semiconductor	800-282-9855	MBR0520

**Demo Board Layout**

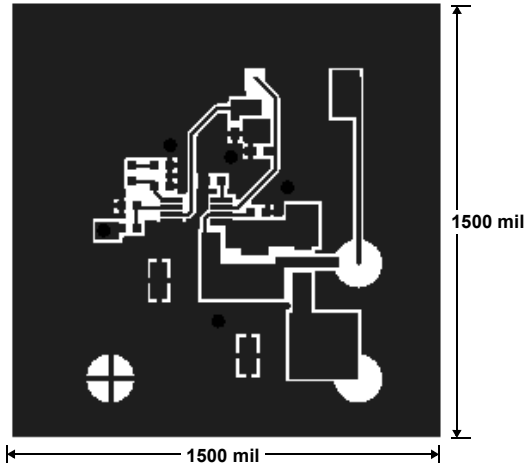


FIGURE 2. TOP LAYER

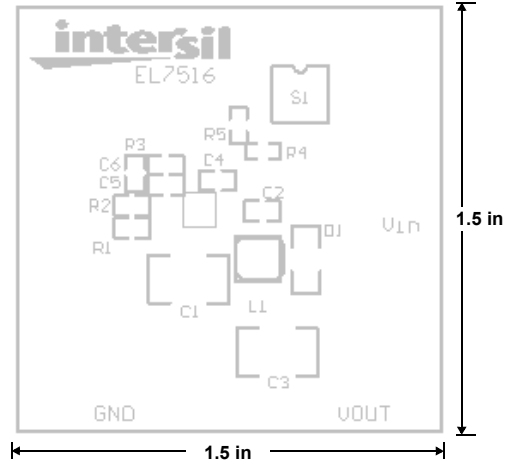


FIGURE 4. TOP SILKSCREEN

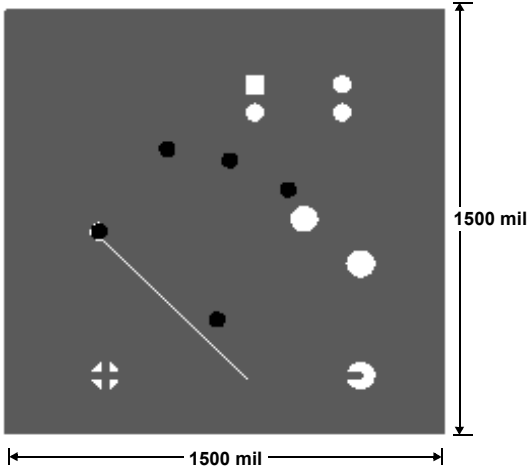


FIGURE 3. BOTTOM LAYER

**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.

*Intersil Corporation reserves the right to make changes in circuit design, software and/or specifications at any time without notice. Accordingly, the reader is cautioned to verify that the Application Note or Technical Brief is current before proceeding.*