

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 in² 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.

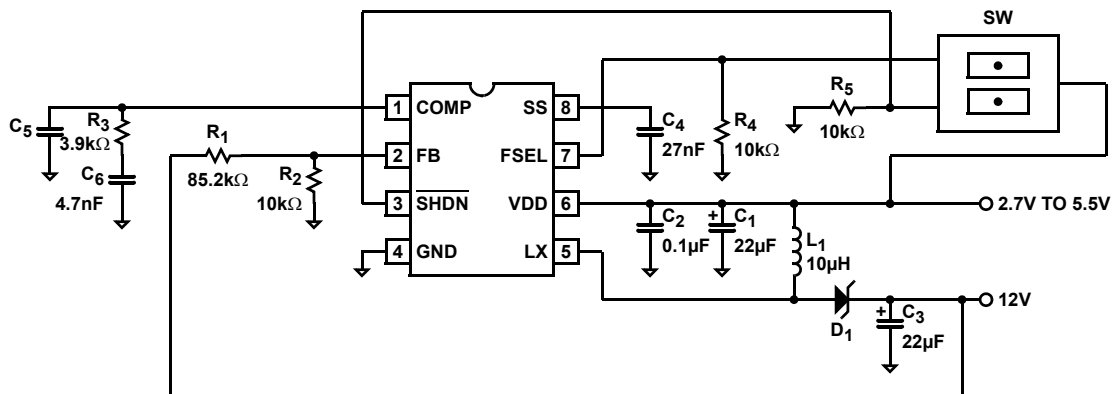


FIGURE 1. 3.3V TO 12V CONVERTER

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

REFERENCE DESIGNATOR	VALUE	MANUFACTURER	MANUFACTURER'S PHONE NUMBER	MANUFACTURER'S PART NUMBER
R 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

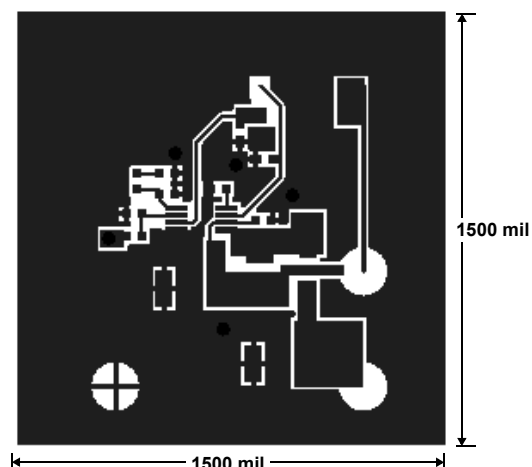


FIGURE 2. TOP LAYER

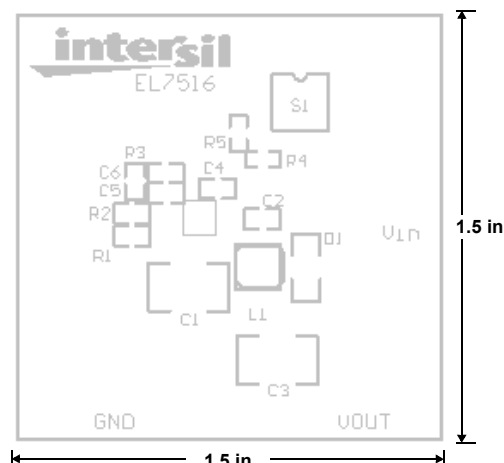


FIGURE 4. TOP SILKSCREEN

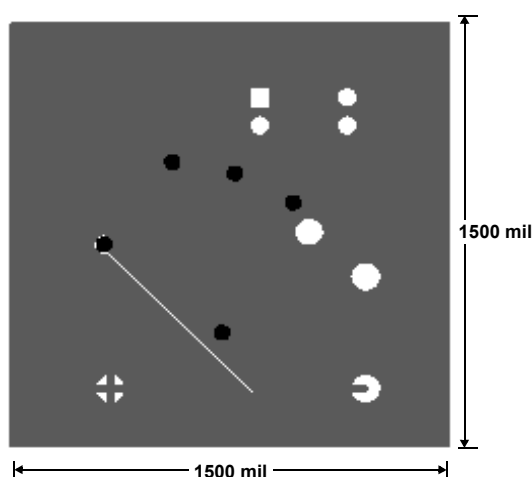


FIGURE 3. BOTTOM LAYER

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.

For information regarding Intersil Corporation and its products, see www.intersil.com