UM3501

EV Board (Evaluation Kit)

User's Guide

V1.0

Version	Date	Prvoider	Approve	Note
1.0	2012-05-11			Initial version.

Table of Contents

1 . Board Information

This UM3501EVB uses a UM3501 adjustable output buck converter to step down 2.5-V or higher input voltages. The EVB operates over an input voltage range of 2.5V to 5.5V. The goal of the EVB is to demonstrate the small size of the UM3501 power supply solution and provide flexibility in interchanging the supporting passive components. This EVB include an enable jumper that allows the user to disable the device.

1.1 Schematic

Fig 1.1 is UM3501 EVB design schematic.

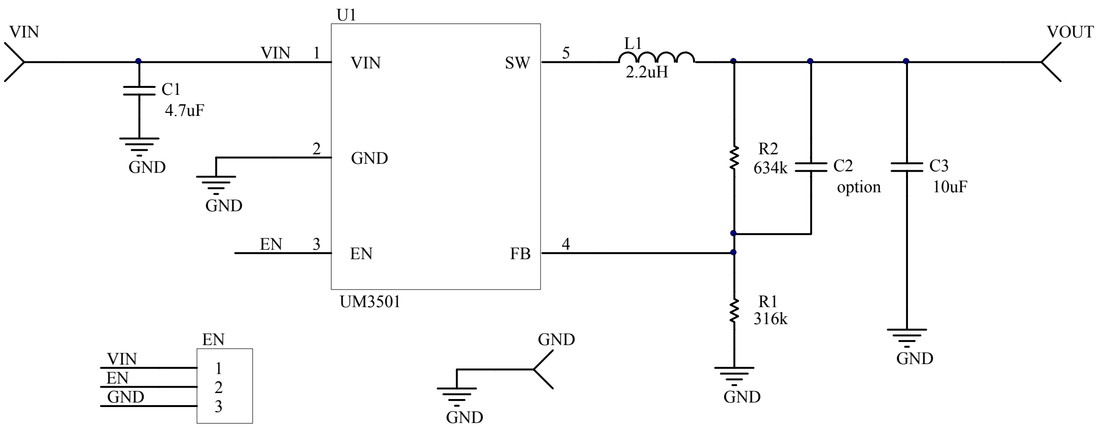

Fig 1.1 UM3501 EVB schematic

（Recommended Value of R1 and R2 for 1.8V output）

1.2 PCB Layout

Fig 1.2, Fig 1.3 is PCB layout and components location diagram of UM3501 EVB.

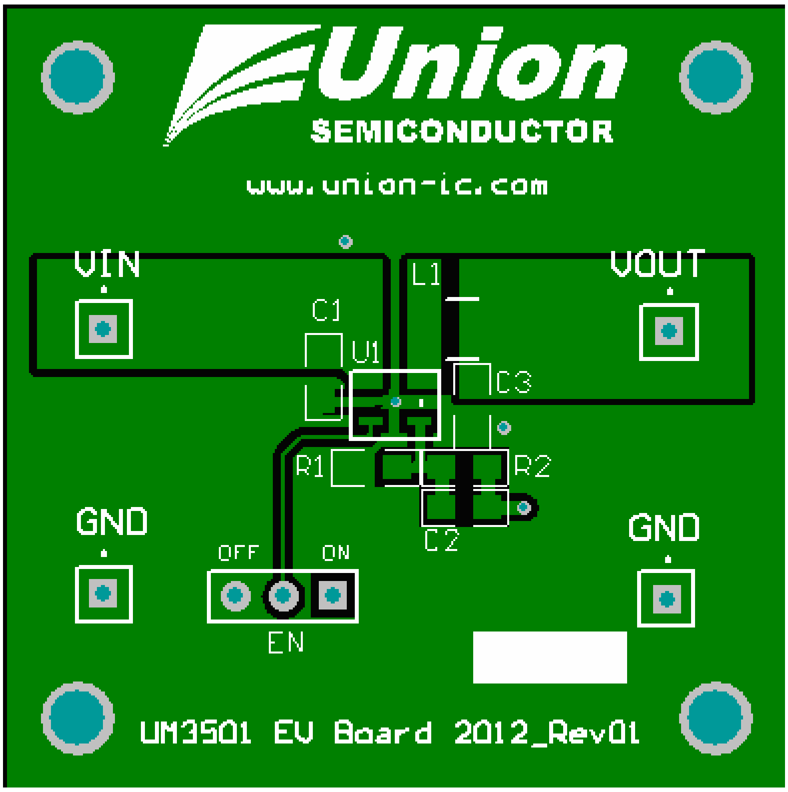

Fig 1.2 UM3501 EV board PCB top layer

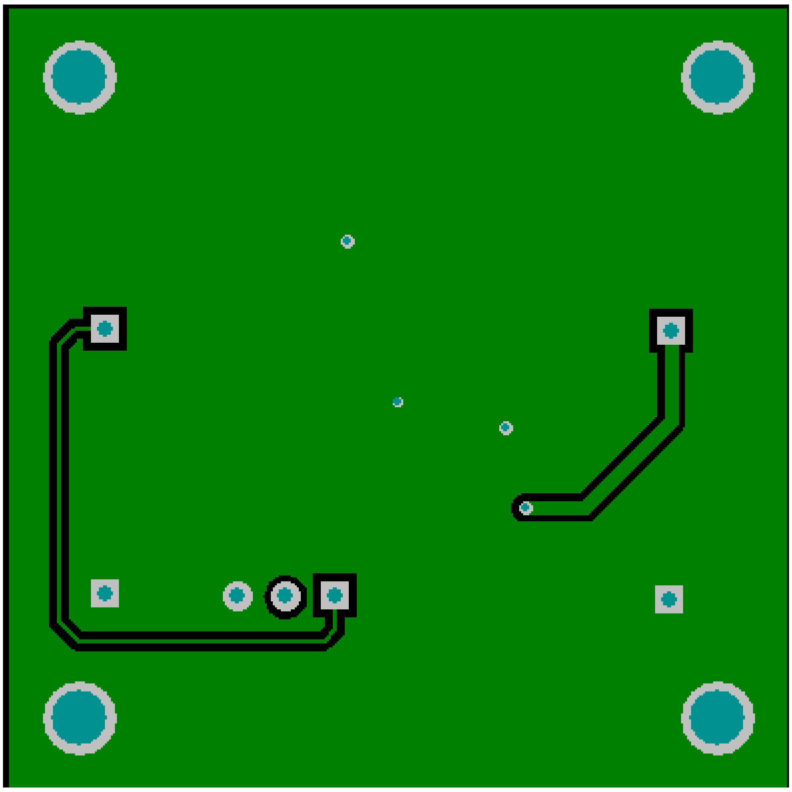

Fig 1.3 UM3501 EV board PCB bottom layer

1.3 Interface Define

Tab 1.1 is the directions of UM3501 EV board interface signals

Tab 1.1 UM1661 EVB board interface

Interface	Function	Note
VIN	Power supply input.	2.5V – 5.5V power supply.
GND	Ground.	
EN	Chip enable/disable option.	EN=ON (1): enable. EN=OFF (0): disable.
VOUT	The board output.	UM3501 output.

2 . Board Operation

An input power supply and a load must be connected to the appropriate EVB connectors in order for the EVB to operate. The absolute maximum input voltage is 6V. The UM3501 is designed to operate with a maximum input voltage of 5.5 V. Short pins 2−3 on jumper EN (labeled ON) to enable the device. Connect a load not to exceed 600 mA to the output of the EVB. Fig 2.1 shows the output ripple and switching waveform for UM3501's operation. (V_{IN}=3.6V, V_{OUT}=1.8V, 300mA load current).

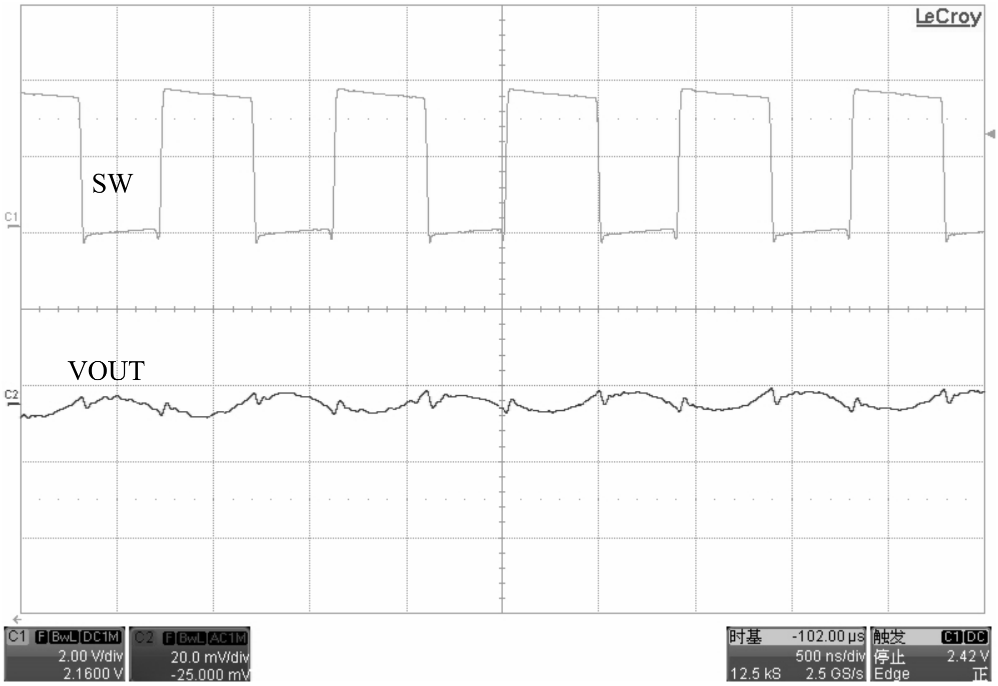

Fig 2.1 UM3501 waveform diagram (V_{IN}=3.6V, V_{OUT}=1.8V, I_{LOAD}=300mA)

2.1 Power Supply

UM3501's input supply voltage range from 2.5V – 5.5V, you need to confirm there is sufficient margin with the current limit when use a DC source to supply the device. Input power cable should be thicker to reduce the loss of input voltage when the load current is large.

2.2 Enable

EN jumper should be set to ON to enable the UM3501.EN jumper set to OFF to shut down the device.

2.3 Output Setting

The output voltage of UM3501 is set by the external resistor divider. See Figure 1.1.The output voltage is calculated as V_{OUT}=0.6V×(1+R2/R1) with R1+R2≤1MΩ. For stability, R1+R2 should not be greater than 1 MΩ. To keep the operating quiescent current to a minimum, the feedback resistor divider should have high impedance.

For example: Use 634 k ohms R1 and 316 k ohms R2 to get a 1.8V output.

V_{OUT}=0.6V×(1+R2/R1)=0.6V×(1+634 /316)=1.80V

3. Board Component

Tab 3.1 is the recommended BOM list of UM3501 EV board.

Tab 3.1 UM3501 EV board recommended BOM list

Reference	Description	Part No.	Manufacturer.
U1	Union Buck Converter.	UM3501	Union
C1	Capacitor,4.7uF,6.3V,Ceramic,X5R,0805.	JMK212BJ475MG	Taiyo
C2	Default for UM3501 Evaluation	-	-
C3	Capacitor,10uF,6.3V,Ceramic,X5R,0805.	C2012X5R0J106M	TDK
L1	Inductor,2.2uH,850mA,33mΩ,SMT.	CDRH2D18/LD-2R2	Sumida
R1,R2	Resistor, 1%,0603 or 0805	Std	Std

3.1 Input Capacitor

The input capacitors reduce the current peaks drawn from the battery or input power source and reduce switching noise in the IC. Ceramic capacitors with X5R or X7R temperature characteristics are highly recommended due to their small size, low ESR, and small temperature coefficients. A 4.7µF X5R or X7R capacitor (C1) from VIN to

GND is recommended for most application. For optimum noise immunity and low input ripple, the input capacitor value can be increased. Note that some ceramic dielectrics exhibit large capacitance and ESR variation with temperature and DC bias. Ceramic capacitors with Z5U or Y5V temperature characteristics should be avoided. See Table 3.1 for suggested capacitors and manufacturers.

3.2 Output Capacitor

The output capacitor limits the output ripple and maintains the output voltage during large load transitions. A 10µF X5R or X7R ceramic capacitor (C3) typically provides sufficient bulk capacitance to stabilize the output during large load transitions and has the ESR and ESL characteristics necessary for low output ripple. For optimum load-transient performance and very low output ripple, the output capacitor value can be increased; however, care should be taken with regards to output voltage slew rate requirements. Note that some ceramic dielectrics exhibit large capacitance and ESR variation with temperature and DC bias. Ceramic capacitors with Z5U or Y5V temperature characteristics should be avoided. Tantalum capacitors are not recommended. See Table 3.1 for suggested capacitors and manufacturers.

3.3 Inductor

The step-down converter operates with a typical switching frequency of 1MHz. This operating frequency allows the use of physically small inductors while maintaining high efficiency. A 2.2µH to 4.7µH inductor is recommended for most applications. Inductor saturation current is another important parameter during inductor selection.

Below formula shows how to calculate the actual peak inductor current under certain application. The saturation inductor current should be higher than the calculated value under the worse case.

$\Delta I_L = V_{OUT}*(V_{IN}-V_{OUT})/(V_{IN}* L *fosc)$

$I_{PEAK} = I_{LOAD} + \Delta I_L/2$

Where ΔI_L is Peak-Peak inductor ripple current,

I_{PEAK} is peak inductor current,

I_{LOAD} is maximum load current in the application.

For example: 1.8V output at 3.6V input voltage, when choose a 2.2uH inductor, the value of the inductor Peak current at 600mA load current can be calculated as blow:

$\Delta I_L = V_{OUT}*(V_{IN}-V_{OUT})/(V_{IN}* L *fosc)=1.8*(3.6-1.8)/(3.6*2.2*1)=0.409(A)$

$I_{PEAK} = I_{DC} + \Delta I_L/2=0.6+0.409/2=0.805\ (A)$

For optimum load transient and efficiency, low DCR inductors should be selected. See Table 3.1 for suggested inductors and manufacturers.

4. Layout Considerations

For all switching power supplies, the layout is an important step in the design, especially at high-peak currents and switching frequencies. If the layout is not carefully done, the regulator shows stability problems as well as EMI problems. Good layout for the UM3501 can be implemented by following a few simple design rules.

1) Use wide and short traces for the high current paths.

2) The input capacitor, as well as the inductor and output capacitor, should be placed as close as possible to the IC pins. In particular, the input capacitor needs to be placed as close as possible to the IC pins, directly across the Vin and GND pin.

3) The feedback resistor network must be routed away from the inductor and switch node to minimize noise and magnetic interference. To further minimize noise from coupling into the feedback network and feedback pin, the ground plane or ground traces must be used for shielding.