



**PAM2862 EV Board User Guide**

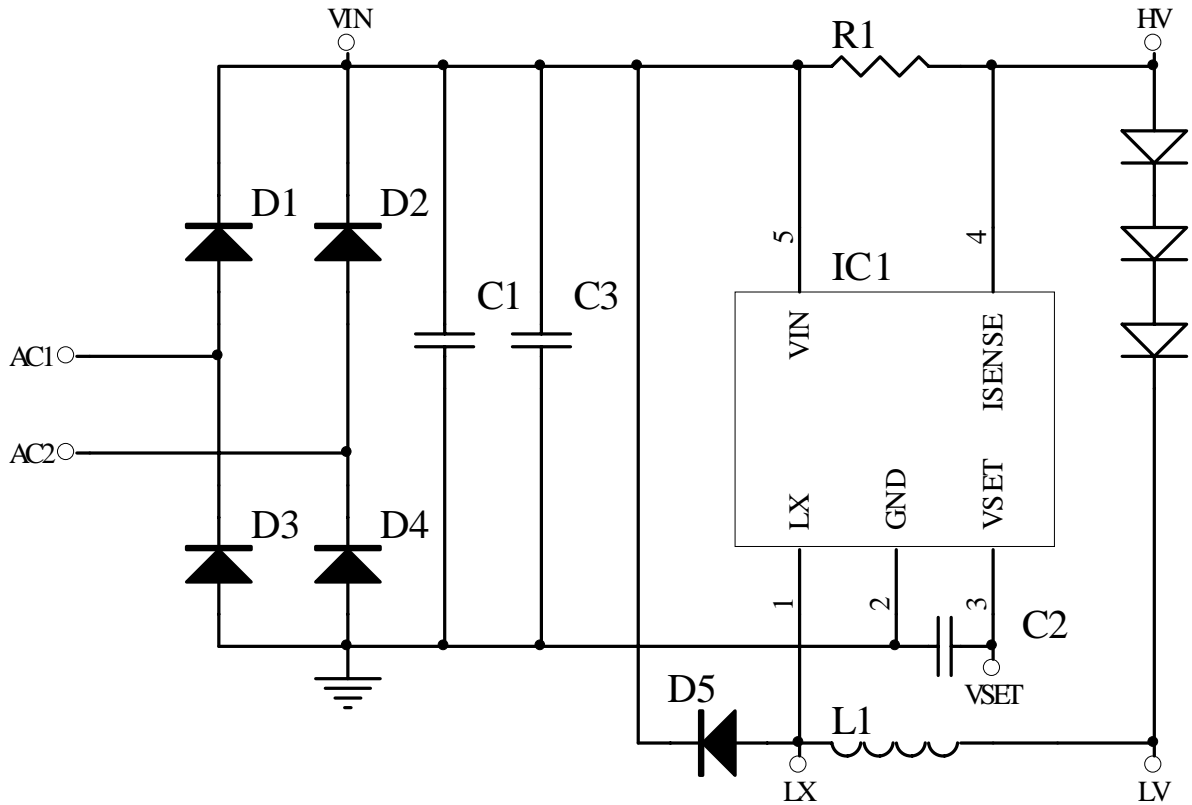
AE Department

**1. Revision Information**

Date	Revision	Description	Comment
2008/8/18	V1.0	Initial release	

DO NOT DISTRIBUTE OR REPRODUCE WITHOUT PERMISSION FROM P.A.M

## 2. EV Board Schematic



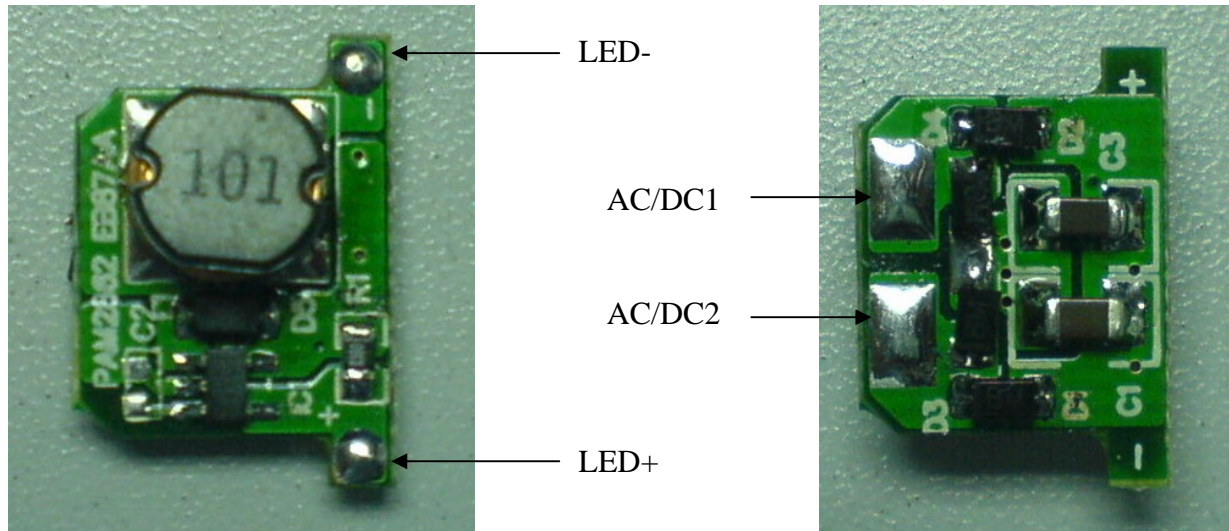
## 3. EVB PAM2862 EB87AA Description

PAM2862 EB87AA is an evaluation board for the PAM2862 (SOT-23-5 package) MR16 application, a high power LED buck driver. The board is targeted to be used in providing a simple and convenient evaluation environment for the PAM2862. Efficiency on the board make it easy to be evaluated.

Use single power supply (12V DC or AC) to drive 1-3 serials 1W power LED.

DO NOT DISTRIBUTE OR REPRODUCE WITHOUT PERMISSION FROM P.A.M

## 4. EV Board View



EV board operational sequence:

- a. Connect AC/DC1 and AC/DC2 to power supply (DC12V or AC12V).
- b. Connect LED- to power LED cathode and LED+ to power LED anode.
- c. This demo has a diode bridge, so not worry about input power connect in reverse.

## 5. EV Board BOM List

Item	Value	Type	Rating	Description	Vender and Part No.
C1, C3	10uF	X5R/X7R, Ceramic/1206	25V	Input coupling CAP	Torch TMK316BJ106KL-T
C2	NC				
L1	100uH	SDR75	0.7A	Inductor	Wurth WE-PD2 type L 74477520 100uH
D1,D2,D3, D4,D5	STPS140Z	SOD-123	1A/40V	Schottky Diode	ST STP140Z
R1	0.3Ω	0805	1%	Iset Resistor	
IC1	PAM2862	SOT-23-5		Power management IC	PAM2862
PCB		PAM2862 EB87AA			

DO NOT DISTRIBUTE OR REPRODUCE WITHOUT PERMISSION FROM P.A.M

## 6. External Components Selection

### Input Capacitors (C1,C3)

- (1) Low ESR needed, 10uF, X5R/X7R ceramic recommend.
- (2) For DC power supply 10uF is enough. For AC power supply the capacitance is more large more better, or output average current will decrease.

### Soft start Capacitors (C2)

- (1) This capacitor is to increase soft start, 10nF for 1.5mS.
- (2) The capacitor can not use in normal application.

### Iset Resistors (R1)

- (1) R1 set the power LED current ,  $I_{LED}=0.1V/R1$ .
- (2) R1,  $0.3\Omega \pm 1\%$  for 1W LED (about 333mA).

### Inductor (L1)

- (1) Low DCR needed, 100uH (rating 0.7A) recommend

### Schottky Diode (D1,D2,D3,D4,D5)

- (1) STP140Z (1A, 40V, SDO-123) recommend

## 7. PCB Layout Guidelines

### Grounding:

- (1) The filter capacitors C1, C3, bridge diode D3,D4, power IC should make a star connect for GND.

### Others:

- (1) Connect L1, LX, D5 with short and wide connections. Minimize the switching circuit area to avoid unexpected radiation.
- (2) Place the Iset resistor R1 as close to the sense pin as possible.
- (3) Make sure the current flow path has a wide trace. When current path need vias, use the multiple vias to decrease impedance.

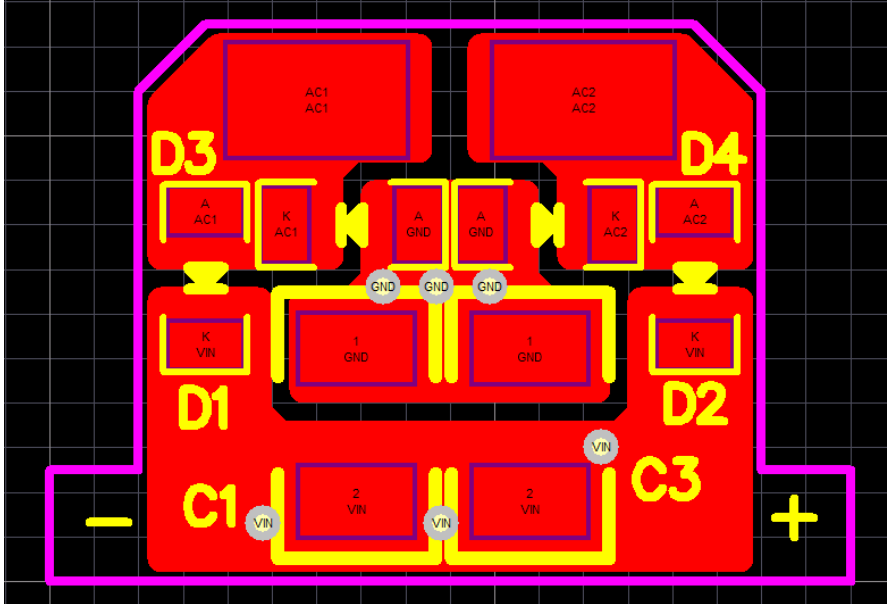
### Thermal Dissipation:

- (1) PAM2862 integrate the power MOSFET. Make sure the heat dissipation area is large enough and have a low thermal resistance to atmosphere.
- (2) The inductor L1 and diode D1,D2,D3,D4,D5 also generate quantity of heat, pay attention to their thermal dissipation.

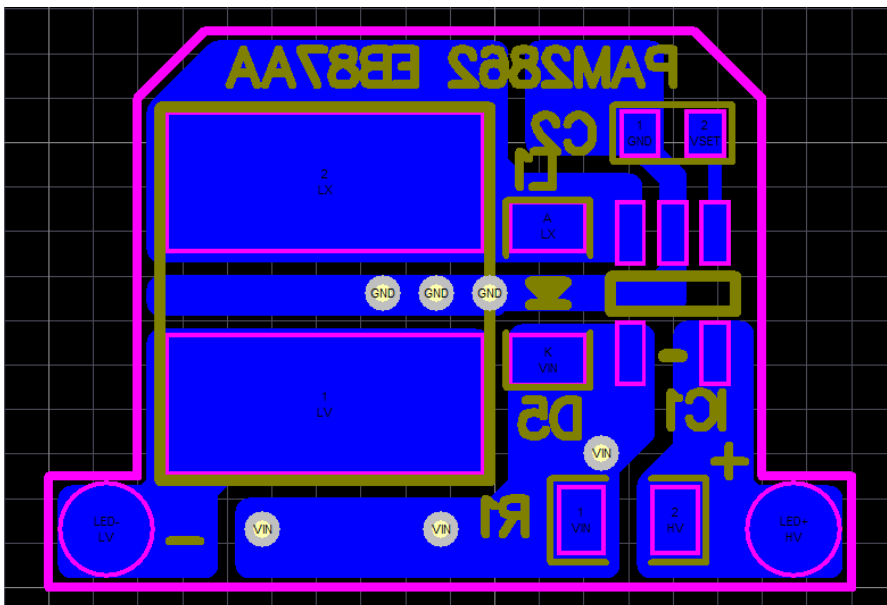
DO NOT DISTRIBUTE OR REPRODUCE WITHOUT PERMISSION FROM P.A.M

### 8. PCB layout Example

#### Top layer



#### Bottom layer



DO NOT DISTRIBUTE OR REPRODUCE WITHOUT PERMISSION FROM P.A.M