AN 16.8

Power and Layout Considerations for EMC2106

1 Overview

The EMC2106 device contains a fan driver which will cause internal power dissipation and heat generation. In addition, the high noise environment in which the EMC2106 is designed to operate may contribute unnecessary error to the temperature measurements**.** This application note describes design and layout techniques that can be used to increase the performance and dissipate the power generated by the EMC2106.

2 Audience

This application note assumes that the reader is familiar with hardware design and the functionality of the EMC2106. The goal of the application note is to provide information on thermal management and noise immunity when using the EMC2106 device.

3 References

The following documents should be referenced when using this application note:

- SMSC EMC2106 Datasheet
- SMSC Application Note 16.4, “Using Anti-Parallel Diodes (APD) with EMC Devices”
- AMKOR Application Note: “Application Notes for Surface Mount Assembly of Amkor’s *Micro*LeadFrame (MLF) Packages” (www.amkor.com)

4 Maintaining Accuracy of External Diode Connections

This section provides information on maintaining accuracy when using diodes as remote sensors with EMC2106 device.

SMSC supplies a family of Environmental Monitoring and Control (EMC) devices including EMC2106 that are capable of accurately measuring the temperature of CPUs (including CPU diodes requiring the transistor or BJT model), GPUs and discrete diodes (2N3904, 2N3906). Most devices include an internal temperature sensor along with the ability to measure one or more external sensors. Recommendations for the printed circuit board layout are provided to help reduce error caused by electrical noise.

Temperature measurement is performed by measuring the change in forward bias voltage of a diode when different currents are forced through the junction. The circuit board itself can impact the ability to accurately measure these small changes in voltage.

4.1 Layout Considerations

Apply the following guidelines when designing the printed circuit board:

1. Route the remote diode traces on an outside layer of the PCB.
2. Place a ground plane on the layer immediately below the diode traces.
3. Keep the diode traces short. It is possible with careful layout to route the diode traces 8 - 12 inches, however, longer traces pickup more noise.
4. Keep the diode traces parallel, and the length of the two traces identical within 0.3 inches.
5. Use a diode trace width of 0.01 inches with a 0.01 inch spacing between traces.
6. If possible, place a 0.01 inch wide ground guard trace on both side of the differential pair of diode traces at 0.01 inch spacing. The guard traces should be connected to the ground plane at least every 0.25 inch. Note: do not connect the guard traces to the CPU ground pins. These pins may inject noise due to locally high currents.
7. Separate the diode traces from any other signal traces by at least 0.020 inches, if the guard traces are not applicable for the PCB.
8. Keep the diode traces away from sources of high frequency noise such as power supply filtering or high speed digital signals.
9. When the diode traces must cross high speed digital signals, make them cross at a 90 degree angle.
10. Avoid joints of copper to solder that can introduce thermocouple effects.
11. Minimize use of via and layer change.
12. For single diode (non-APD) traces, the DP line is more sensitive to noise than the DN line. For long runs, routing the DP line on the outside of the PCB is advised.

These recommendations are illustrated in Figure 4.1.

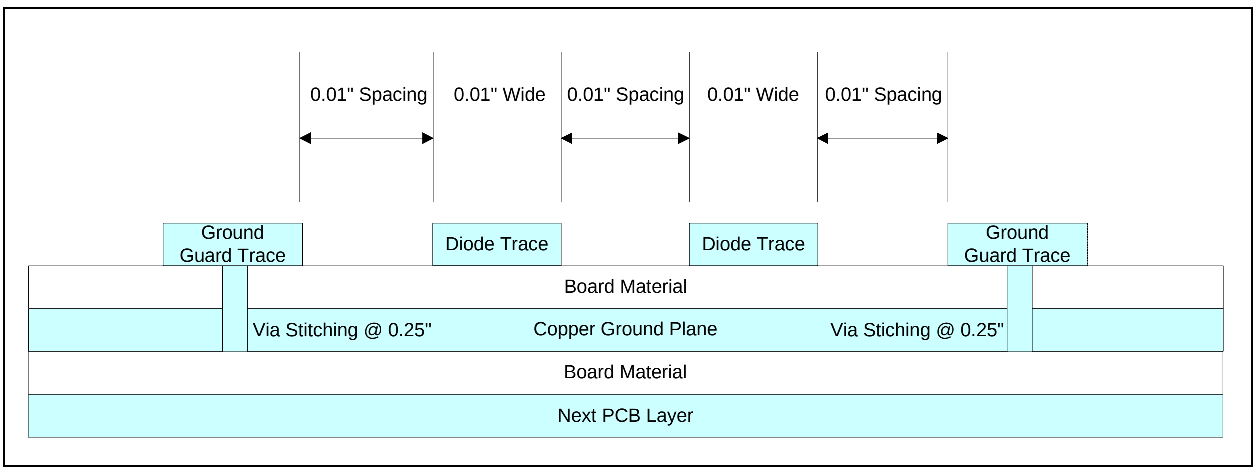


Figure 4.1 Routing the Diode Traces

4.2 Remote Sensors Connected by Cables

When connecting remote diodes with a cable (instead of traces on the PCB) use shielded twisted pair cable. The shield should be attached to ground near the EMC2106, and should be left unconnected at the sensor end. *Belden 8451* or *88641* cables are good choices for this application. If shielded cable cannot be used, route the cables away from noise sources.

4.3 Capacitors on Diode Traces

In the board layout, provide pads to install a capacitor across the DP and DN traces as close to the EMC2106 package pins as possible. The value of the capacitor should not exceed 2200pF.

It is recommended that pads for a capacitor also be placed at the diode, and that this capacitor is generally not populated. In certain noisy environments it may be necessary to install a small capacitor (18 - 100pF) at the diode.

4.4 Anti-Parallel Diode Support

The EMC2106 has Anti-Parallel Diode (APD) capability on one remote diode channel. This feature, illustrated in Figure 4.2 below, allows two 2N3904 type diodes to be measured on one pair of DP/DN pins. Routing and capacitor guidelines for APD applications are available in SMSC application note 16.4. “Using Anti-Parallel Diodes (APD) with EMC Devices”.

.

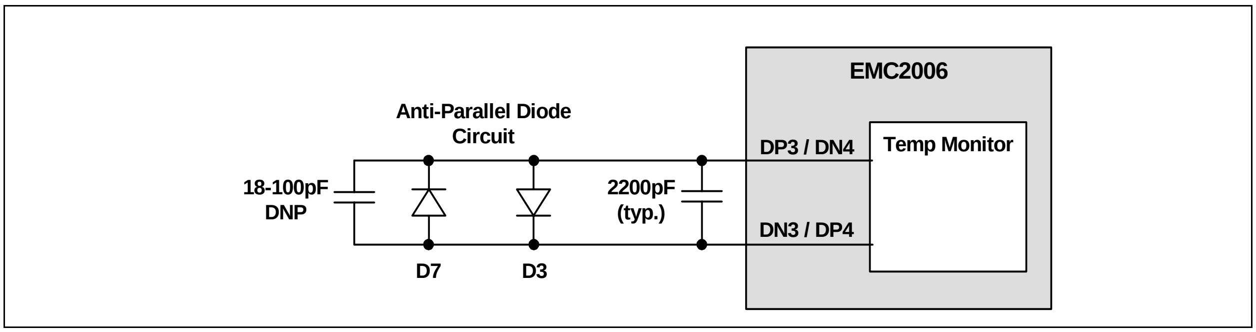


Figure 4.2 Anti-Parallel Diode Configuration

4.5 Power Supply Decoupling

Accurate temperature measurements require a clean, stable power supply. Locate a 0.1µF capacitor as close as possible to the VDD power pin with a good ground. To create a low pass filter, a 22 ohm series resistor should be added per Figure 4.3, "Power Supply Decoupling".

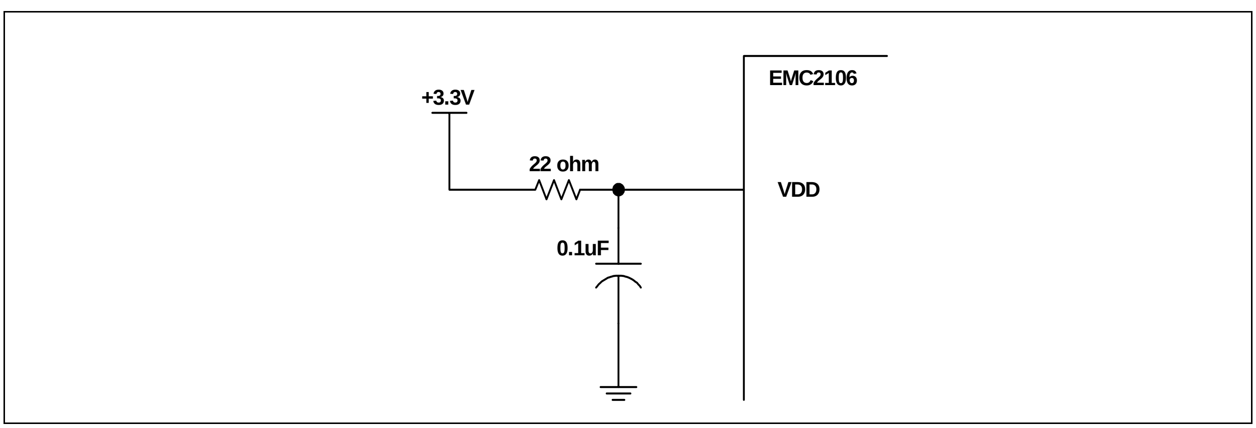


Figure 4.3 Power Supply Decoupling

4.6 Manufacturing

Circuit board assembly processes may leave a residue on the board, which can result in unexpected leakage currents that may introduce temperature measurement errors. For example, processes that use water-soluble soldering fluxes have been known to cause problems if the board is not cleaned thoroughly.

5 Power Considerations

The EMC2106 devices contain a high side fan driver rated at 600mA maximum drive current. If the Fan Driver is fully loaded and driving at the maximum power dissipation, the EMC2106 can dissipate as much as 0.75W of on-chip power from the 5V supply.

The value of $Theta_{JA}$ for this package is approximately 60^{o}C/W when soldered to a PCB without utilizing vias to conduct heat away from the die. The thermal resistance can be reduced to a much lower level with a sufficient quantity and diameter of vias connected to the ground plane.

When the guidelines in this section are followed, the EMC2106 is rated to operate up to an ambient temperature of 85°C and drive a fan rated up to 5V@600mA in all voltage/current settings.

5.1 Maximum Die Temperature

The maximum rated die temperature for the EMC2106's fabrication process is 125^{o}C, however, SMSC recommends designing the thermal solution to maintain a die temperature below 110^{o}C. This will provide thermal margin for different PCB configurations, via solder-fill variations and solder voids between the exposed pad and the thermal landing.

Note: The internal temperature sensor will report the average die temperature. This sensor will not report an accurate ambient temperature when the fan driver is dissipating power. The internal sensor may be measured to validate the thermal design. See Section 5.3, "Validating the Thermal Design".

5.2 Thermal Pad Design

The EMC2106's QFN package has an "exposed die paddle" which must be soldered to a thermal landing on the PCB. The thermal landing must be connected to the PCB ground plane with four 20mil vias ($Theta_{JA}$ = 34^{o}C/W) or nine 12mil vias ($Theta_{JA}$ = 37^{o}C/W). The AMKOR application note provides details on manufacturability issues such as achieving solder-fill in the vias and minimizing voids due to gassing.

Note: SMSC strongly recommends using 20mil vias to minimize die temperature. The larger vias are especially effective for applications with high power dissipation.

To help conserve routing resources, blind vias may be used to connect the QFN exposed die paddle to the ground plane. This makes it possible to route other signals under the QFN package on the bottom layers of the PCB. If the vias are drilled completely through the PCB, the bottom side pad diameter may be the minimum allowed by the PCB manufacturing process.

5.3 Validating the Thermal Design

A simple test can be performed to validate the thermal solution. With no other active heat sources on the PCB, apply a load to the fan driver that result in 1W of power dissipation in the EMC2106. Apply power to this configuration for 1 hour to allow the temperatures to stabilize and measure the ambient air temperature near the EMC2106 and the internal temperature sensor to determine the value of $Theta_{JA}$. The difference between the two temperatures is the thermal solution's $Theta_{JA}$ value.

Power dissipation of 1W can be achieved by applying the following conditions to the Fan Driver. Set the fan driver voltage to 3.0V (write value 0x99 to register 0x40) and connect a 6 ohm power resistor from the fan driver to ground.

Note: A power dissipation level of 1W is in excess of the normal operating conditions for this IC, however, it will not cause permanent damage to the EMC2106 as long the die temperature does not exceed the maximum rated die temperature of 125^{o}C.

6 5V Linear Fan Driver

The EMC2106's high side fan driver circuit can provide up to 600mA of current from a 5V supply. The power dissipation of this circuit is dependent upon the current draw from the load as well as the output voltage where it is programmed. For linear drive systems, the on-chip power dissipation takes on a parabolic curve as shown in Figure 6.1, "EMC2106 Fan Driver Power Example with Linear Load".

This illustration assumes a constant load impedance. This is a simple mathematical example only and does not represent measured performance, nor does it include fan driver R_{ON}. Most linear fan drive systems have a non-linear load impedance that is proportional to the fan speed. In addition, there is a point in the curve where there is not enough energy to maintain the fan magnetic field or electronics and the fan will stop spinning, reducing the current load to zero. Consult the fan manufacturer for the voltage/current for the specific fan to be driven.

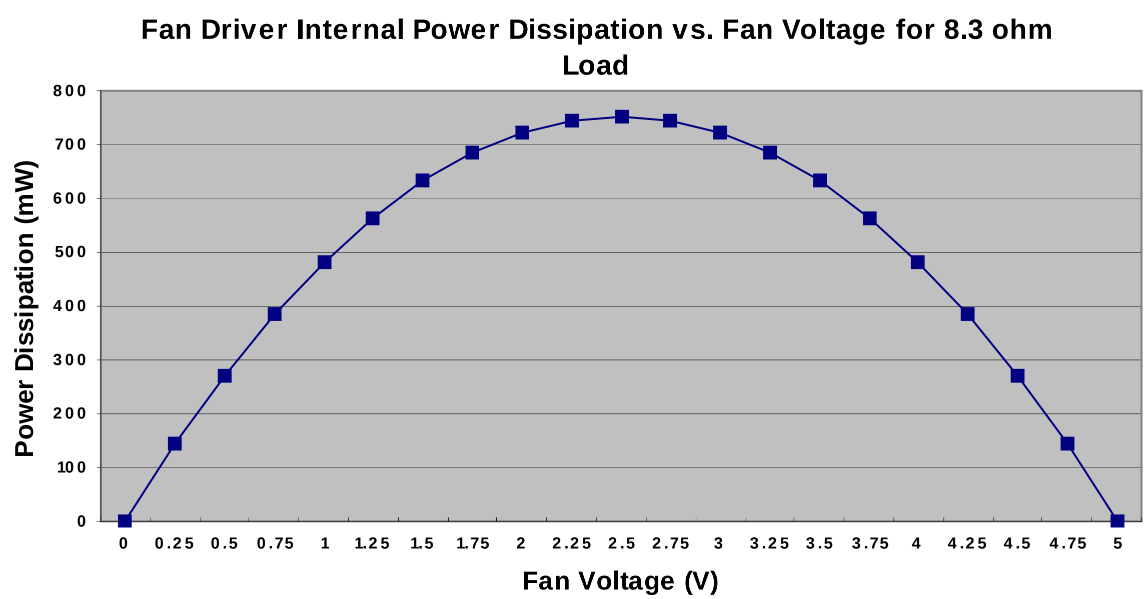


Figure 6.1 EMC2106 Fan Driver Power Example with Linear Load

6.1 Layout Considerations

The VDD_5V and FAN pins may carry currents up to 600mA so the cross-sectional area of the associated traces must be large enough to avoid voltage drop or excessive trace heating.

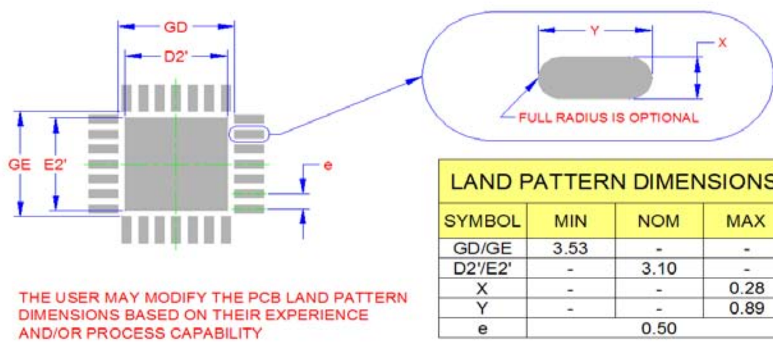
The high side fan driver should obey the following layout guidelines:

1. The FAN signals should be loaded with a capacitor to ground. The value of this capacitor can vary from 10uF to 100uF. The ESR should be less than 1 ohm. This capacitor should be placed as close to the load as possible.

2. The VDD_5V supply should be bypassed with a 10uF capacitor to ground and a 0.1uF capacitor to ground. These capacitors should be placed as close to the device as possible.
3. The FAN pin cannot sink more than 100uA of current, therefore it should not be loaded with capacitors, resistors or inductors connected to VDD_5V or any supply higher than the programmed output voltage.

7 Other Layout Considerations

7.1 EMC2106 Recommended PCB Footprint



LAND PATTERN DIMENSIONS			
SYMBOL	MIN	NOM	MAX
GD/GE	3.53	-	-
D2'/E2'	-	3.10	-
X	-	-	0.28
Y	-	-	0.89
e		0.50	

Figure 7.1 EMC2106 Recommended PCB Footprint Design for 28SQFN-5x5 Body

7.2 Solder Paste Stencil

A better solder joint can be achieved by applying solder paste with a stencil approximately 75% the size of the solder flag. Various shapes and sizes are used for the opening in the stencil, and an array of smaller openings is often cut into the solder paste template for QFN packages, as shown in Figure 7.2. For more details, refer to AMKOR - *Application Notes for Surface Mount Assembly of Amkor's MicroLeadFrame (MLF) Packages*.

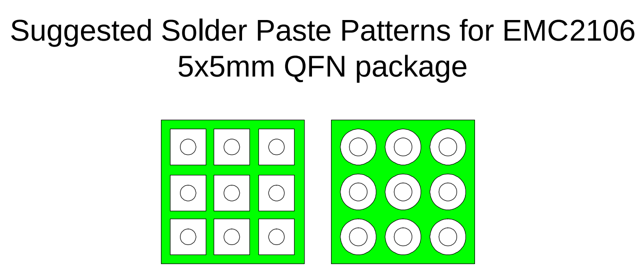


Figure 7.2 Suggested Solder Paste Patterns for EMC2106 5x5mm QFN package

7.3 General Purpose I/O Pins

The GPIO pins should obey the following layout guidelines:

1. The General Purpose I/O pins default as Inputs and should be terminated to some known state if they are not used. If this is not practical, then the BIOS should configure the not-connected pins as push-pull outputs.

7.4 Other Pins

The remaining pins should obey the following layout guidelines:

1. The following pins are open drain and require a pull-up resistor:

- ALERT# (Pin 6)
- SYS_SHDN# (Pin 9)
- SMDATA (Pin 10)
- SMCLK (Pin 11)

2. The following pins are programmable I/O pins and will require pull-ups when used as open drain outputs:

- OVERT3#/GPIO5/PWM4 (Pin 5)
- CLK_IN/GPIO1 (Pin 7)
- OVERT2#/GPIO4/PWM3 (Pin 8)
- GPIO6 (Pin 12)
- PWM2/GPIO3 (Pin 13)
- TACH2/GPIO2 (Pin 14)
- OVERT1#/PWM1 (Pin 16)

80 ARKAY DRIVE, HAUPPAUGE, NY 11788 (631) 435-6000, FAX (631) 273-3123