SUMMIT MICROELECTRONICS, Inc.

Application Note 61

Layout Guidelines for an SMB series Switching Regulator

Shadi Hawawini

Introduction

In a switching power supply layout, there are many considerations that must be accounted for, that a designer may not have to worry about for a linear or digital circuit. Such considerations are important because an improperly laid out PCB can contribute to, or cause instability in a system, and cause excessive electromagnetic interference (EMI).

These considerations, and suggestions for a sound PCB layout on a Summit Microelectronics SMB series programmable switching power manager, will be discussed in this app note.

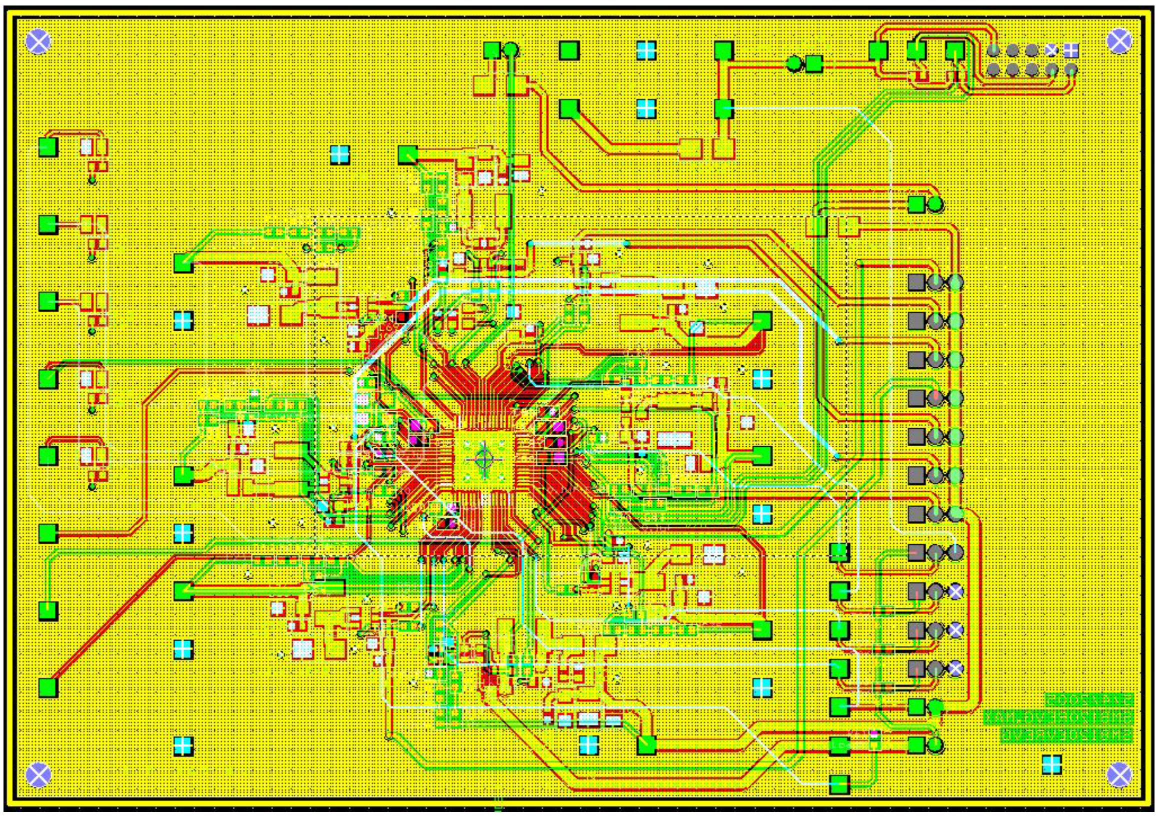

Figure 1: Typical IC Layout

Layout Considerations
While designing the layout for a switching power supply, the designer should be aware of the many precautions that need to be taken to ensure a properly designed PCB. These considerations can be generalized into two groups, which can include, but are not limited to, the amount of copper pour and trace requirements, as well as component placement.

Copper Pour and Trace requirements
In designing the PCB for a switch mode power supply it is very important to understand your power requirements and what traces will be carrying AC signals and what will be carrying DC signals. These criteria will affect the lengths and widths of traces as well as the amount of copper that must be poured.
As a traces width begins to increase its resistance and inductance begin to decrease. Conversely, as a traces length begins to increase, its resistance and inductance begin to increase as well. Therefore it is very important to ensure that traces carrying AC signals and supplying any power components must be made as short and wide as possible. Additionally, power traces should run adjacent to one another as to minimize the amount of noise coupling, because at high switching frequencies the PCB can act as an antenna, and even DC signals can begin picking up noise from neighboring traces. However, if traces must cross, they should be made to be exactly perpendicular to one another to reduce the magnetic flux components and magnetic field interaction.
Finally, understanding the power requirements and the amount of current needed in the system is important for determining the amount of copper to be poured. As an example, a boost channel at 24Vout from 5Vin supplying a 1A load will need roughly 7A through the transistors. If the transistor has an rds(on) of roughly 15mohms, then the transistor will need to dissipate 735mW of power, so the amount of copper to be poured will need to be able to dissipate the heat generated, as well as be thick enough (1-2oz) for the current to flow with very little resistance.

Component Placement
The second consideration for the layout of the switching power supply is placement of the components. In discussing the placement of components it is important to understand the major current loops. Of most concern to a layout is the Power switch loop, as well as the rectifier loop. These two loops are illustrated in Figure 2. The reason for these being the most important is due to the fact that they mainly contain AC signals and are therefore most likely to cause problems. It is important to try and isolate these loops to prevent coupling between traces.
In addition to isolating these loops, power components should be placed as close to one another as possible without causing additional heat generation, while the compensation components should be on the opposite side of the PCB separated by an internal grounded layer when possible.
Components should also be placed so that the current through them travels in as straight of a line as possible, in-line with the PCB trace. This will help reduce the amount of loops and reduce the amount of EMI from the board. For example using the circuit from Figure 2, the power components should be laid out as in Figure 3.

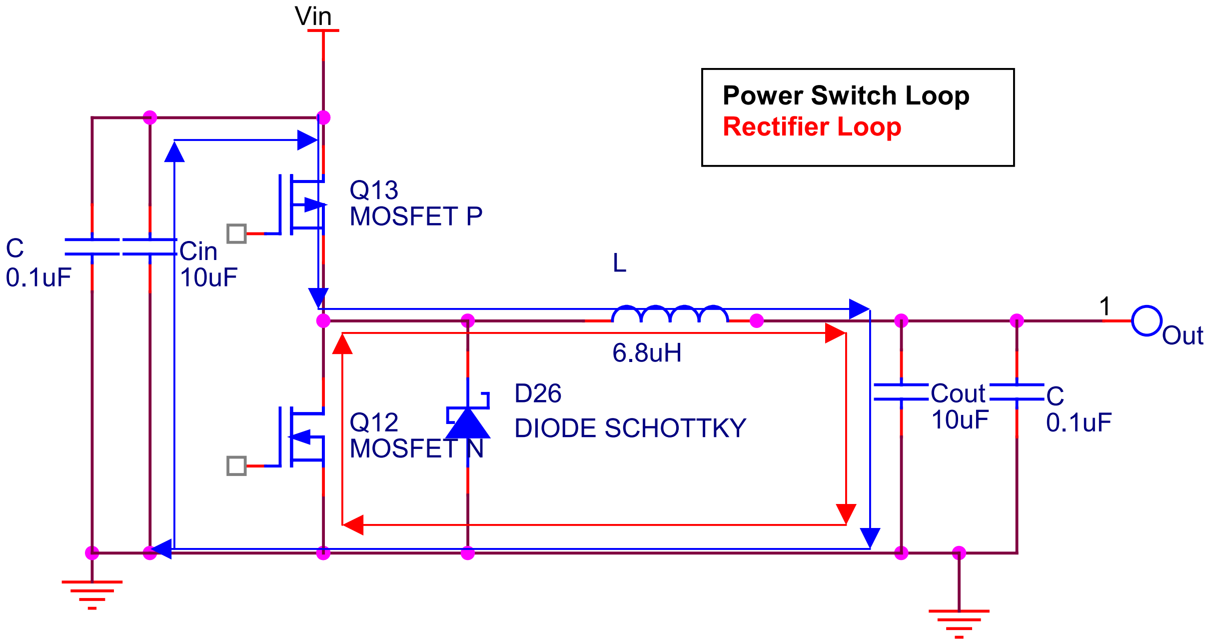

Figure 2: Synchronous Buck Current Loops

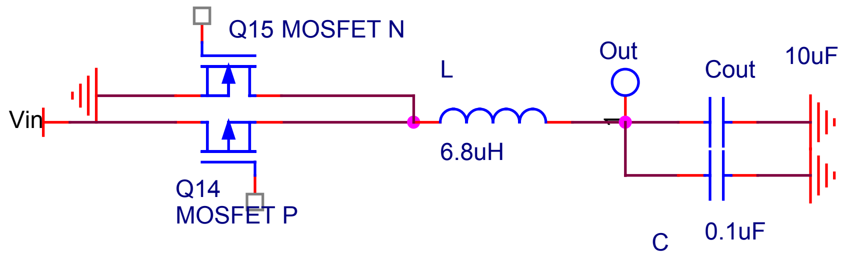

Figure 3: Synchronous Buck Layout Component Placement

The same care should be taken for the compensation components on the back of the PCB. The voltage mode SMB series part uses a type 3 compensation as in Figure 4, and is usually laid out as in Figure 5.

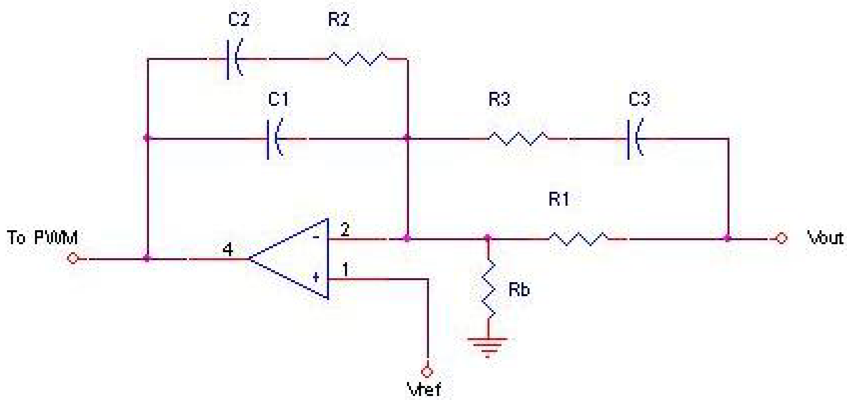

Figure 4: Type 3 Compensation Network

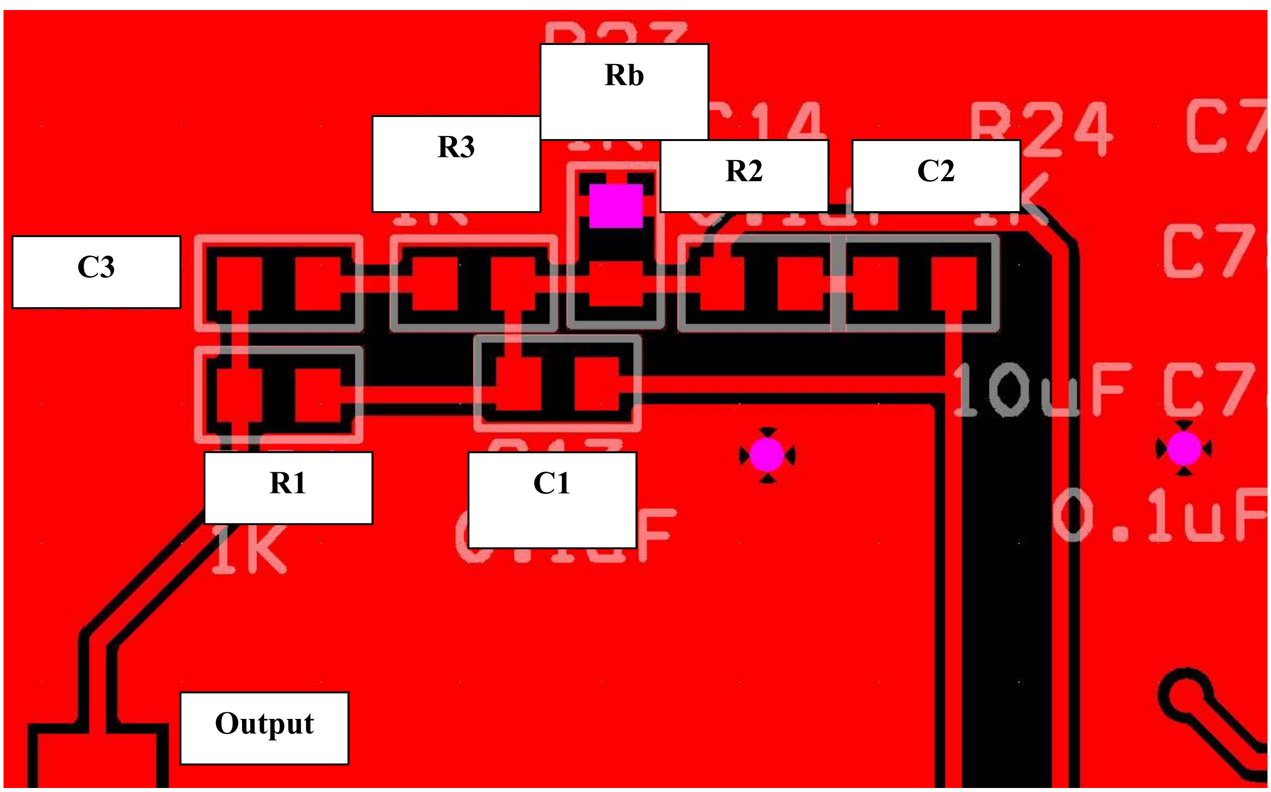

Figure 5: Typical Type 3 Compensation Layout

SMB Series Specific Layout Considerations and Conclusion

The following list is more specific for an SMB series programmable switching regulator, however many of the same theories apply. To summarize, the following should be considered:

1) Depending on the power needed for a particular application, increasing the copper pour for the FETs and inductor on that plane to help dissipate heat may be required.
2) The trace length from the HSDRV and LSDRV on the SMB series part to the gate of the transistor should be minimized as much as possible.
3) Compensation Components should be placed on the opposite side of the PCB from the Power Components and should have a grounded layer separating the two whenever possible. Additionally, the Compensation Components should be placed as far from the switch node as possible and as close to the high-z input to the Error Amplifier.
4) Ensure that there is plenty of grounding on the board to reduce EMI noise.
5) Place 0.1uF bypass Capacitors as close as possible between a component and ground. This is particularly important for the output, where the bypass capacitor should be placed directly in between where the output is being taken and ground.
6) Make sure that the power components are very close together with minimal trace lengths to reduce the overall length of the loop. The same care should be taken for the compensation components on the opposite side of the board.
7) VDD_CAP and HVSUP capacitors should be placed as close as possible to the SMB part.
8) Star grounding should be used whenever possible as opposed to daisy chaining grounding.
9) There should not be large areas that are bare; these areas should be filled with grounded copper.

With these design considerations taken into account a PCB board should exhibit far less EMI and not contribute to instability in the switching power supply design.

Revision 4.0 - This document supersedes all previous versions. Please check the Summit Microelectronics Inc. web site at http://www.summitmicro.com for data sheet updates.