Freescale Semiconductor
Application Note

Document Number: AN4476
Rev. 0, 03/2012

System Design Guideline for 5V 8-bit families in Home Appliance Applications

by: **T.C. Lun , Dennis Lui**
System and Application, Microcontroller Solutions Group

Contents

1 Introduction

In recent times, there has been a tremendous increase in the use of electronic and programmable electronic devices. So the Electromagnetic Compatibility of a full system has become one of the major technical issues developers have nowadays. If it is ignored early in the design cycle, and problems are encountered during testing or product in the field, fixes become very expensive. EMC problem on a printed circuit board (PCB) can be solved at the layout stage. This can be done at a relatively lower cost if we consider EMC practice at the beginning of the design cycle. Figure 1 shows that the cost for EMC fix after the production phase is very high and take a lot of time on engineering fixes.

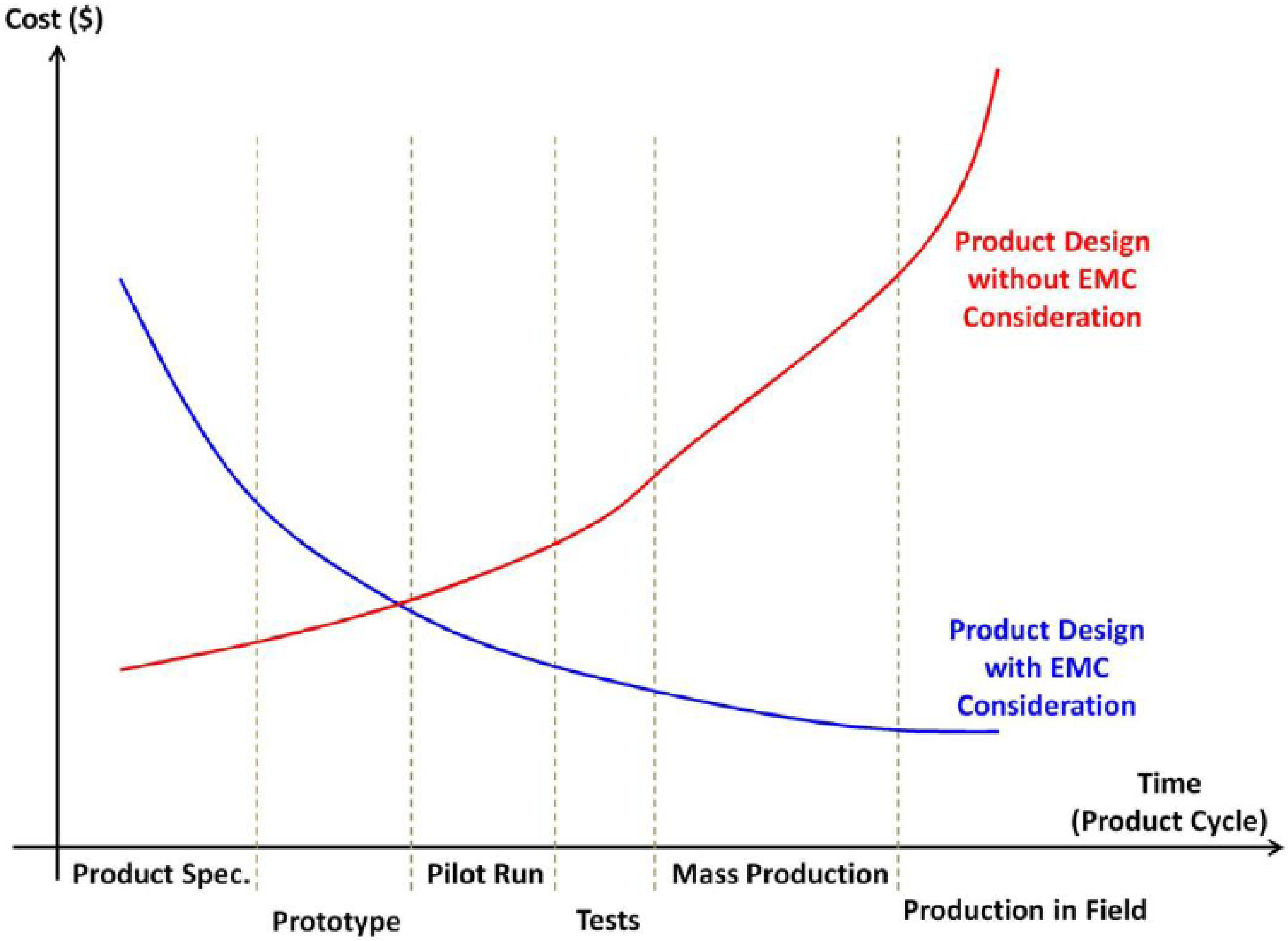

Figure 1. Cost of EMC fixing

Good practice of EMC consideration such as component selection, circuit design, PCB layout and system/product design can provide inherent EMC performance. This application note introduces some EMC basic concepts in Hardware Techniques and then we use a real case PCB layout example to explain how to achieve good EMC performance during the design phase.

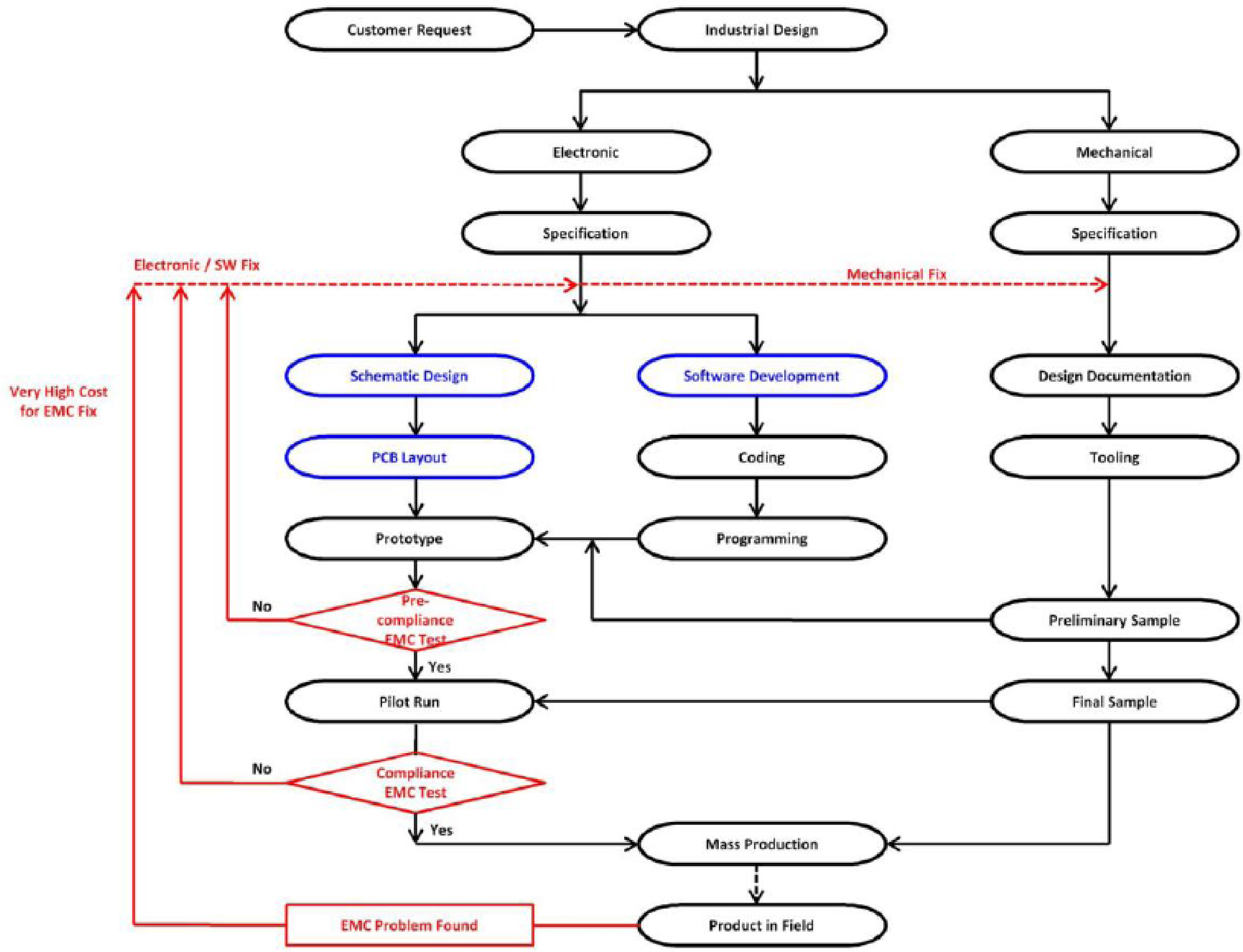

Figure 2. Product design flow

Figure 2 shows the typical product design flow, where the cost of the EMC is very high if we do not pay particular attention at the beginning of the schematic design, software development and PCB layout. In some worst case, it needs to change the mechanical casing design to let the product to pass the EMC compliance test particular for air discharge ESD test.

2 Hardware Techniques

In this application note, we introduced the need of PCB layout and software consideration at the beginning of the product design phase and some fundamental concepts related to the PCB layout design for EMC / EMI compliance. To explain better, the layout and software concept for EMC / EMI compliance, we use the actual microwave oven control board as the use case to show how to implement the board design with EMC / EMI good practice. We selected three 5V 8-bit (PT60, FL16 and AC16/60/128) MCUs to show how to implement the good EMC/EMI board PCB layout.

2.1 Single Layer, Double Side Loading PCB

For cost consideration, most of the home appliance (HA) manufacturers use the single-layer, double-side loading control board for their products.

Although single-layer, double-side loading PCB can reduce the cost of the control board but it is certainly more challenging than a double-layer and multi-layer PCB because of the more design flexibility.

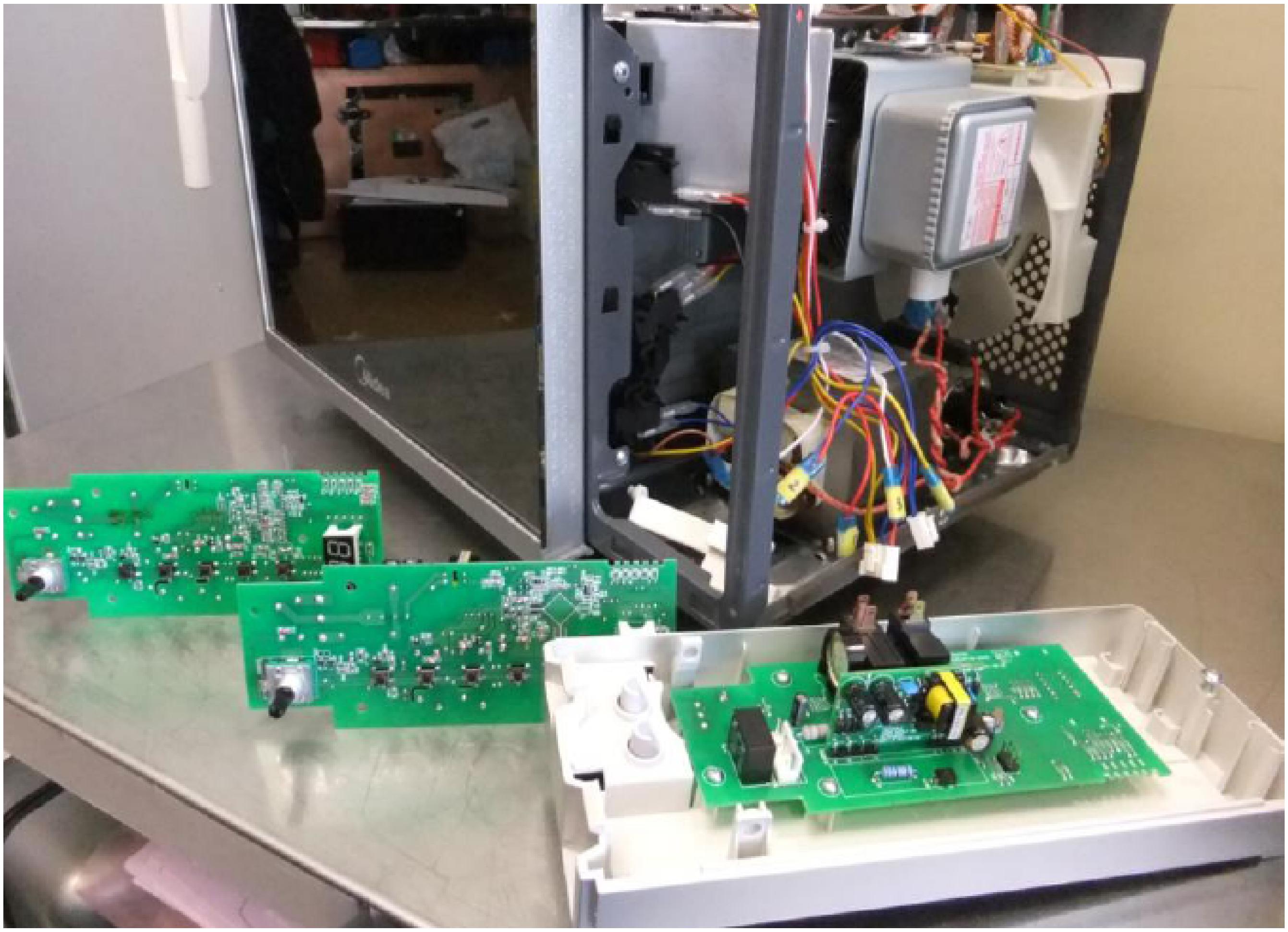

Figure 3. Control boards for microwave system

In figure 3, it shows the MC9S08PT60 (PT60), MC9S08FL16 (FL16) and MC9S08AC16/60/128 (AC60) control boards and the microwave oven. During the mechanical product design, the board size is fixed. And the control board is implemented by single-layer, double-side loading with the separated switch mode power supply daughter board mounted vertically. There are many constraints in the board design such as the size limitation, low cost concern, fixed position of push buttons location, LED 7-segment display and rotary 4 encoder. Following sections will show you how to implement the PCB layout for good EMC practice under such bounded conditions.

2.2 Placement Methods

Figure 4, 5 and 6 shows the placement of the PT60, FL16 and AC60 control boards. Since it is a single-layer, double-sided loading board, and need to fulfill the product design requirements. Here is the list of those mechanical constraints.

- Screw hole position is fixed
- Vertical mounted separated SMPS board
- Separated high-power AC main circuitry and low power DC circuitry.
- Placement of the push buttons, LED 7-segment display and rotary encoder are fixed.
- Except the SMD, push buttons, LED 7-segment display and rotary encoder, all others components must be placed in the TOP side.

By fulfilling all of above constraints, the components placement on the board is shown in figure 4, 5 and 6 which require a lot of effort to finalize the acceptable version.

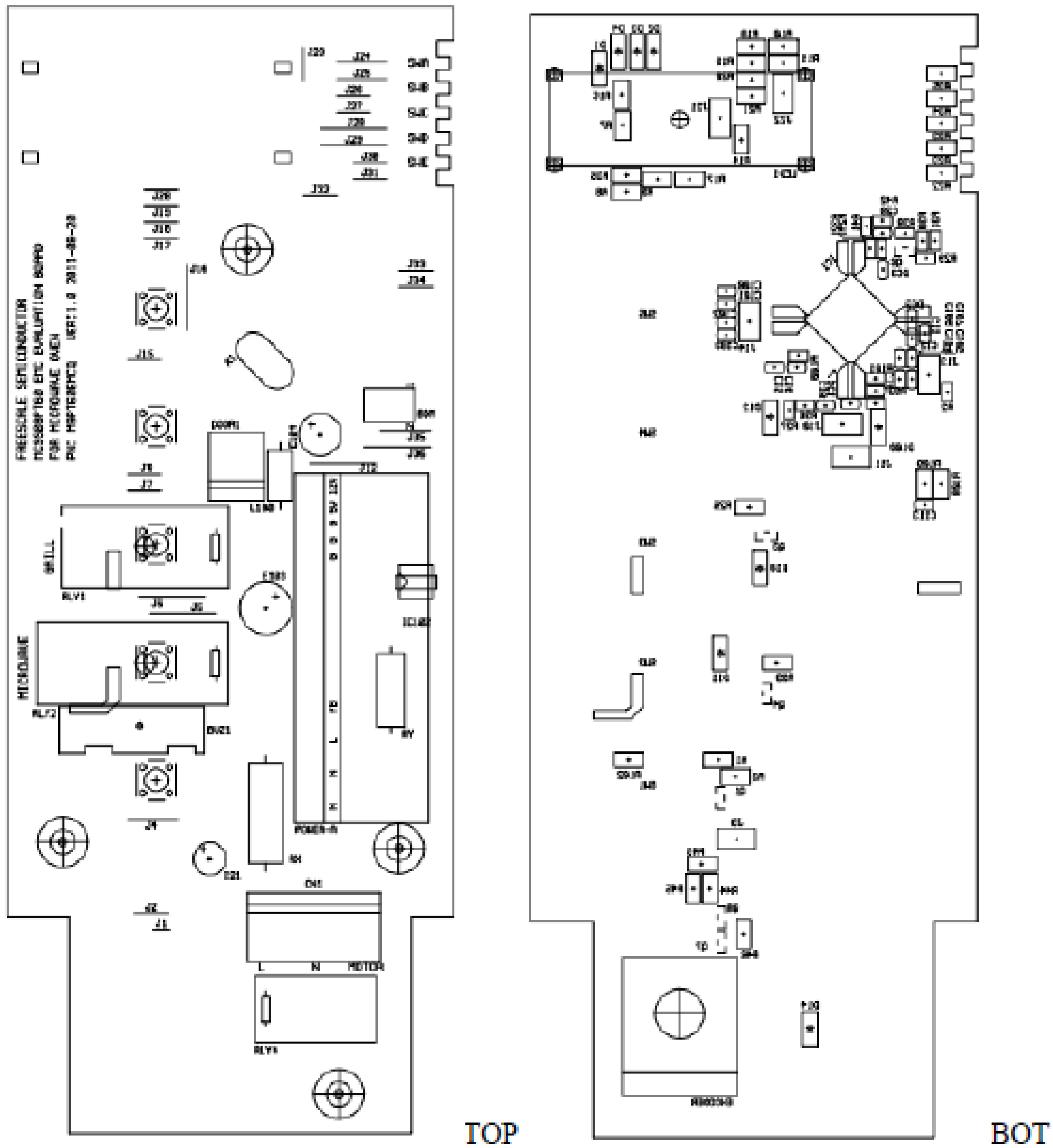

Figure 4. Component placement for PT60

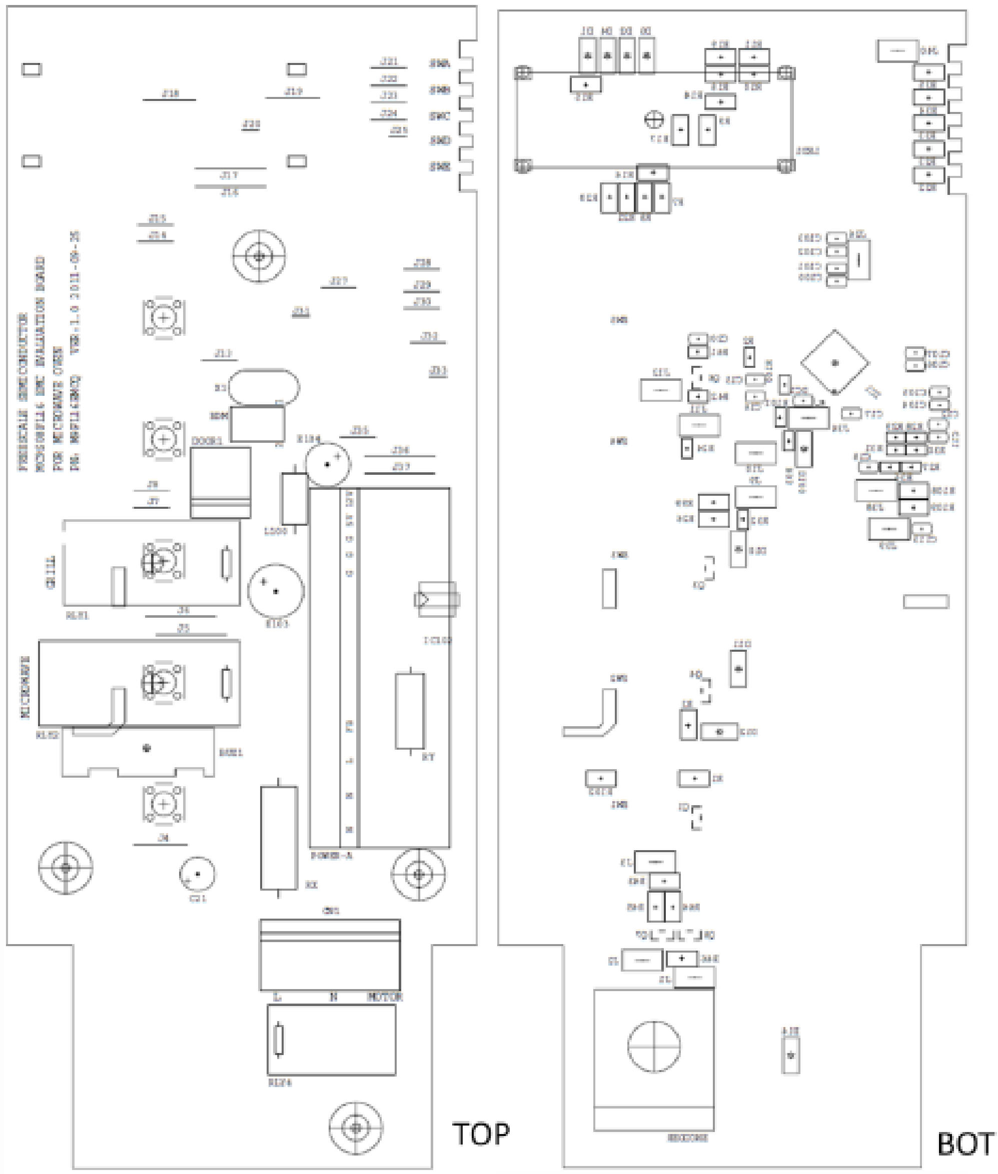

Figure 5. Component placement for FL16

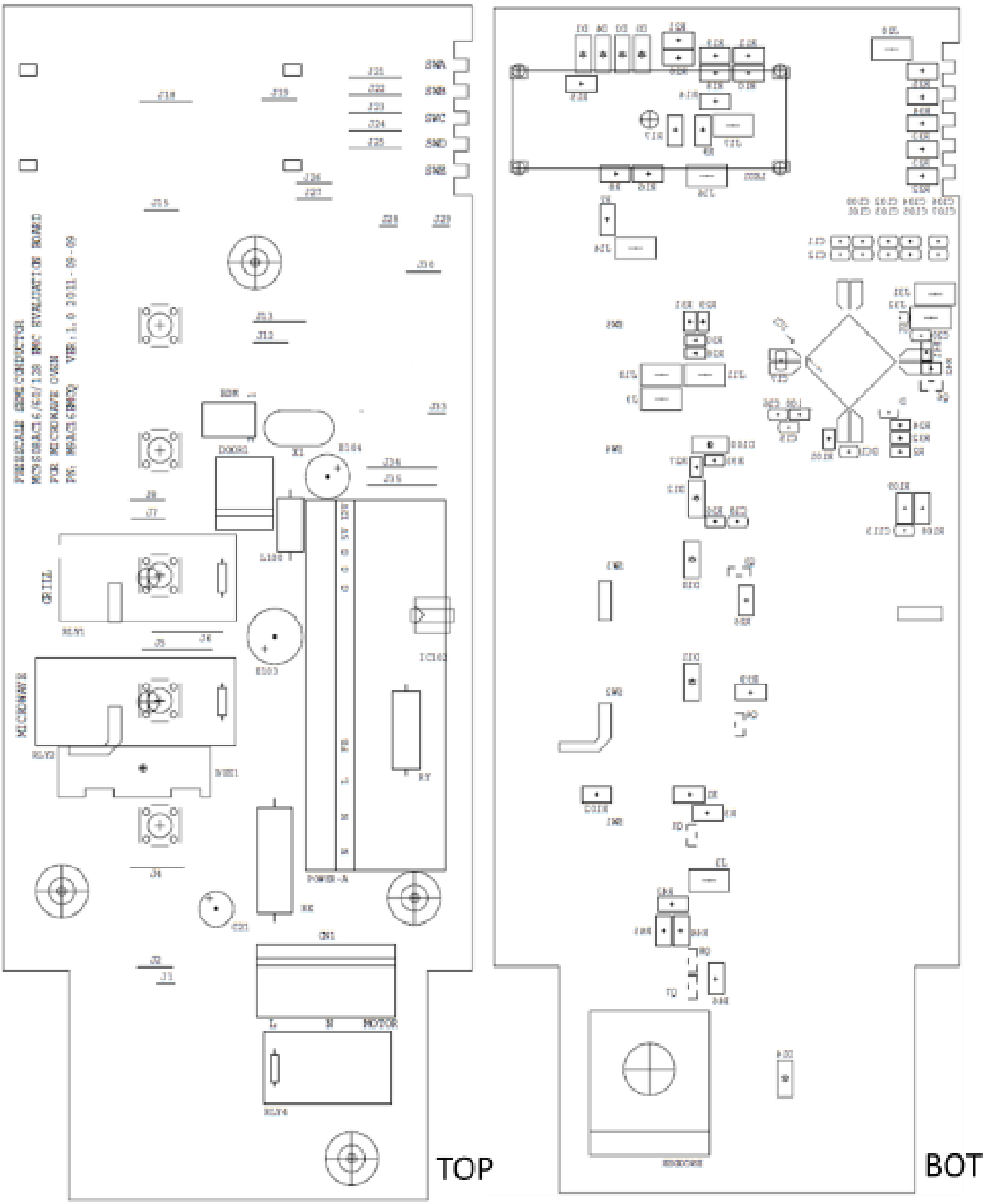

Figure 6. Component placement for AC16/60/128

The first EMC technique is how to select the critical components which needs to placed near the MCU. To consider the ground system, common impedance coupling and input port of the MCU is more sensitive with external noise, we need to minimize the ground and power loops area as well as the impedance of the power and ground. Therefore, the first step is to identify all components that connect to the MCU input ports, which are also connecting to power or ground. Those critical components are shown in figure 7, 8 and 9.

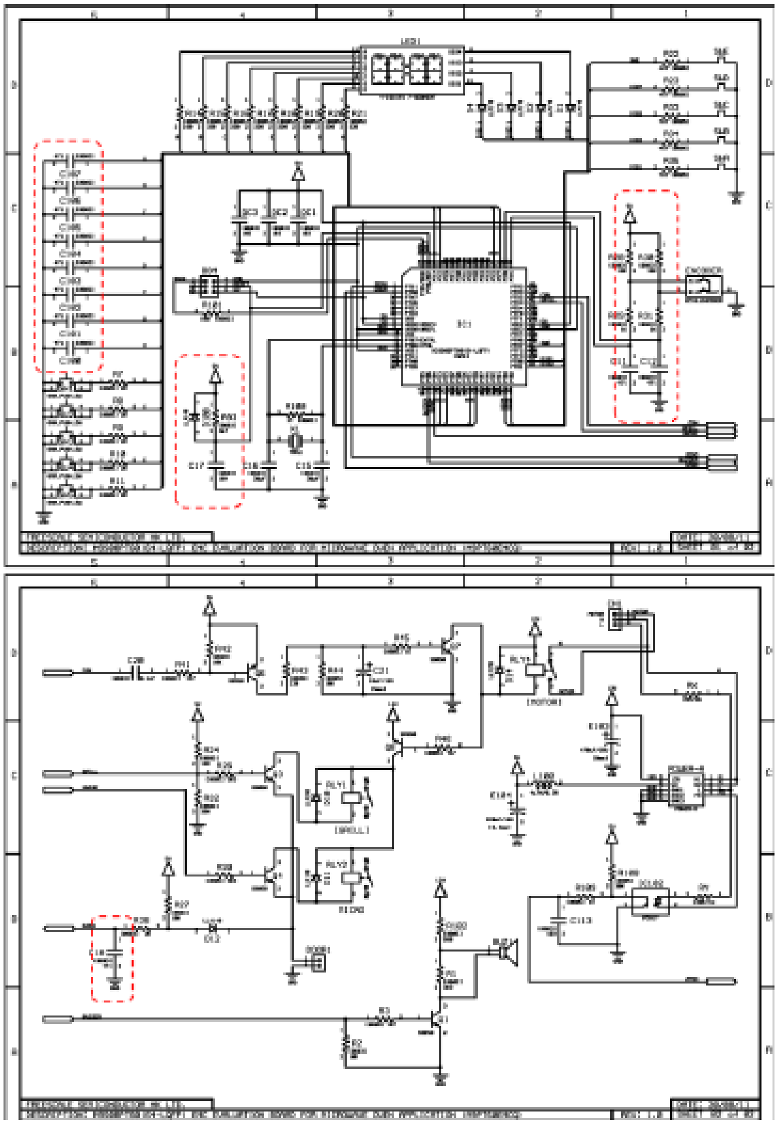

Figure 7. Critical component selections for placement in PT60 board

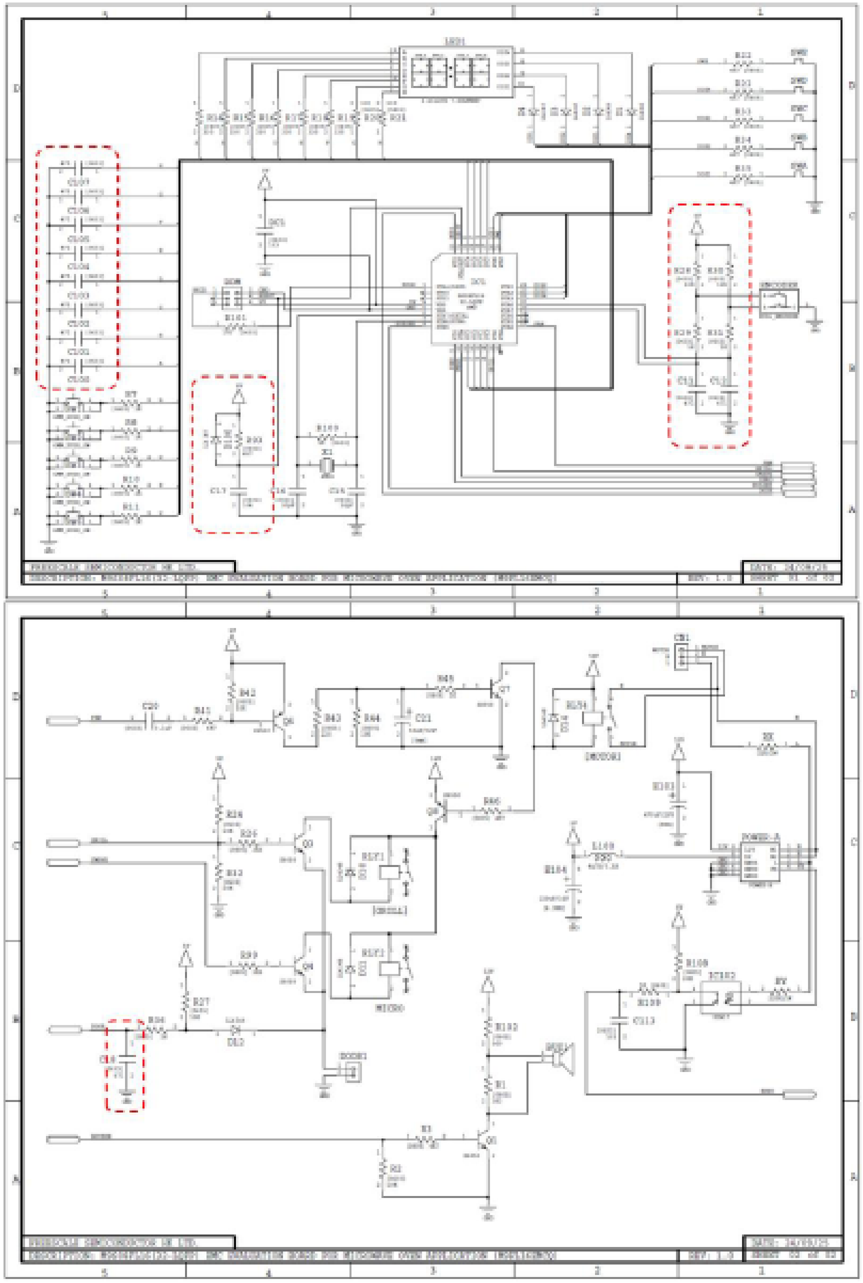

Figure 8. Critical component selections for placement in FL16 board

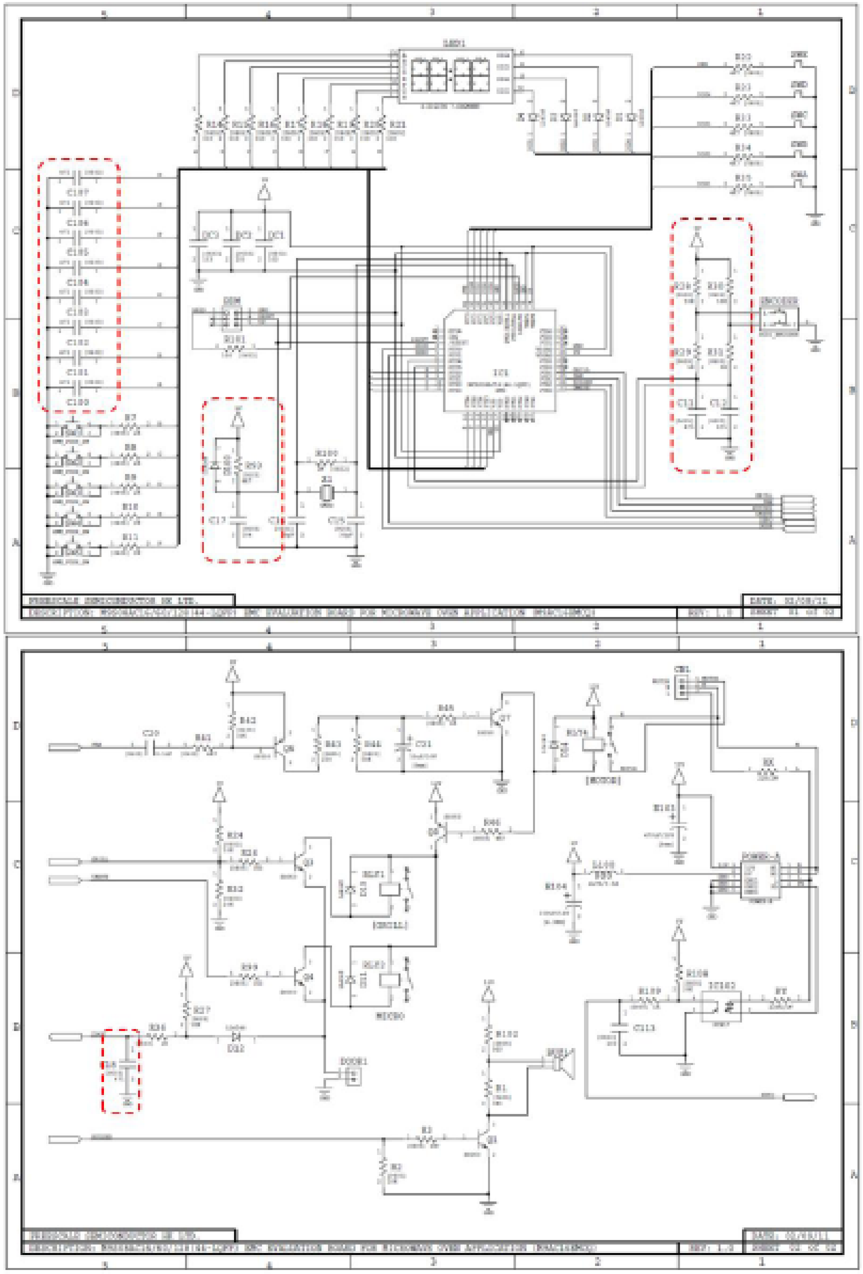

Figure 9. Critical component selections for placement in AC16/60/128 board

As a result, we place those critical components near the MCU as in figure 10, 11 and 12. The advantage of such placement can minimize both power and ground loop areas and reduce common impedances coupling (i.e. power and ground). It will increase the EMC performance of the board and the system.

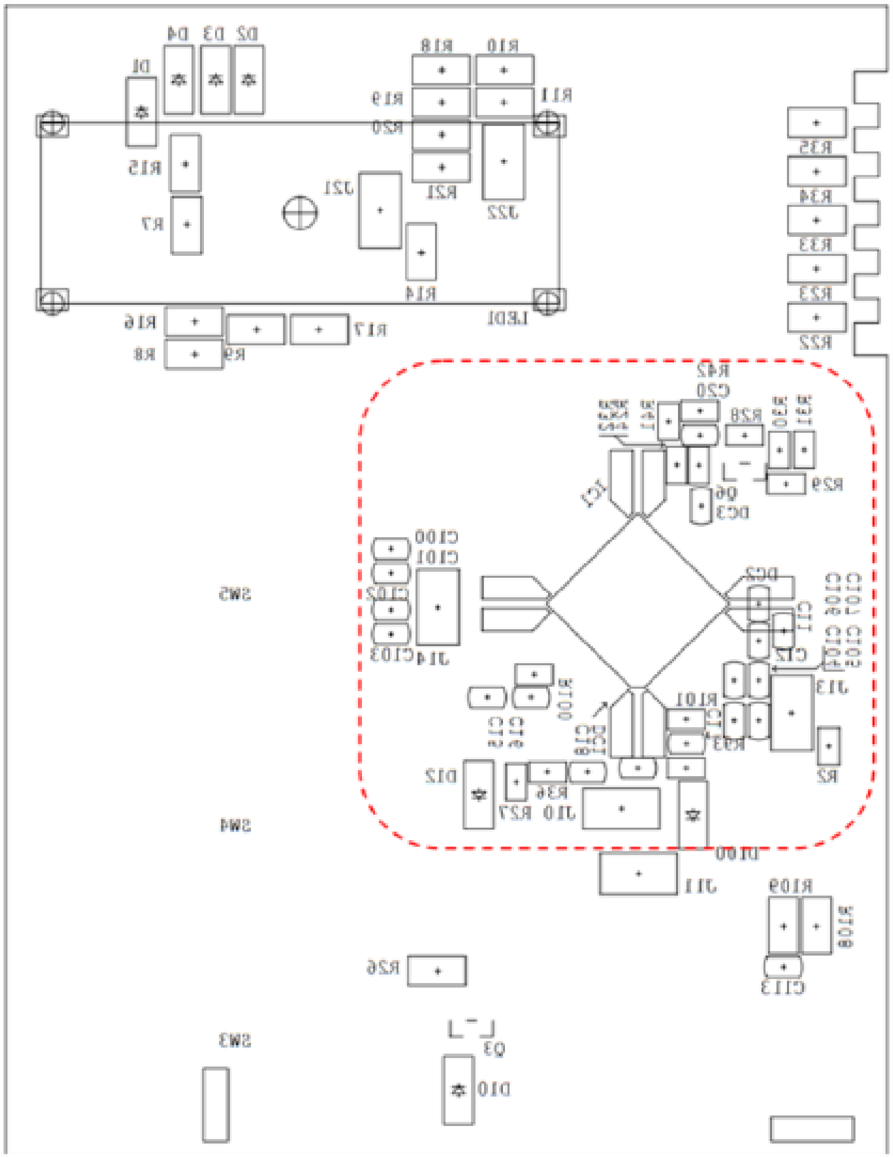

Figure 10. Selected critical component and place them near the MCU in PT60 board

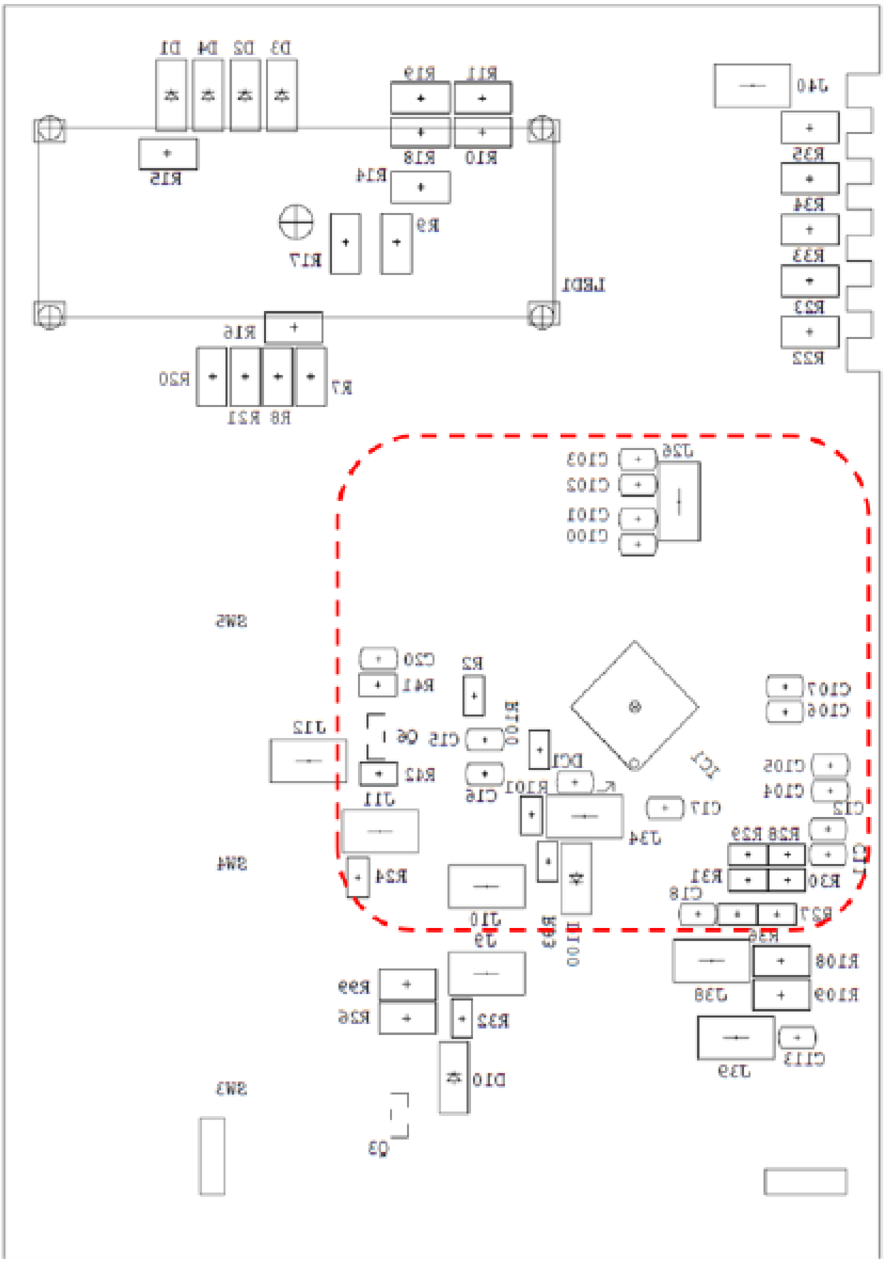

Figure 11. Selected critical component and place them near the MCU in FL16 board

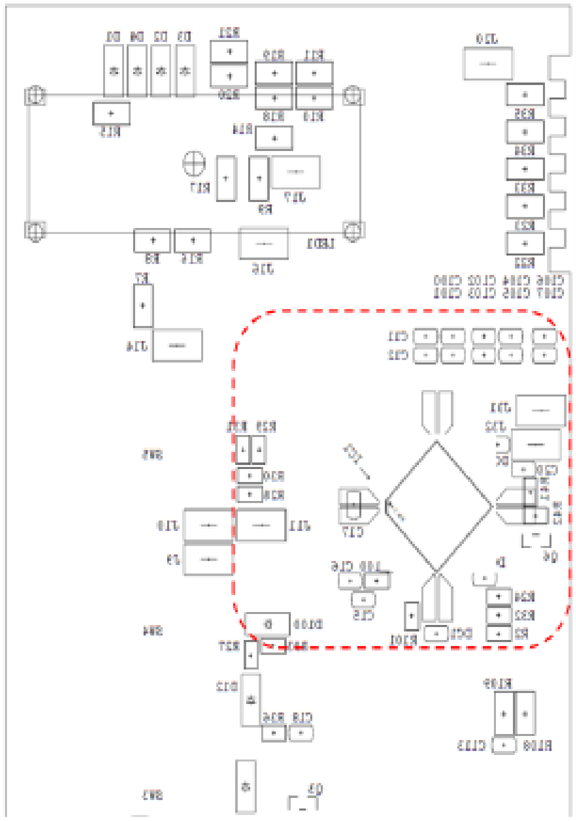

Figure 12. Selected critical component and place them near the MCU in AC16/60/128

2.3 Power Supply Boards

Better EMC performance needs to reduce the loop area of the return path. In the power supply board, the current is large compared with the other circuit, therefore more attention in the PCB layout is needed to minimize the noise generated from the board. Figure 13 shows the loops consideration and Figure 14 shows the PCB layout implementation. This is not only to minimize the loops but also to consider the isolation between AC and DC portion of the switch mode power supply.

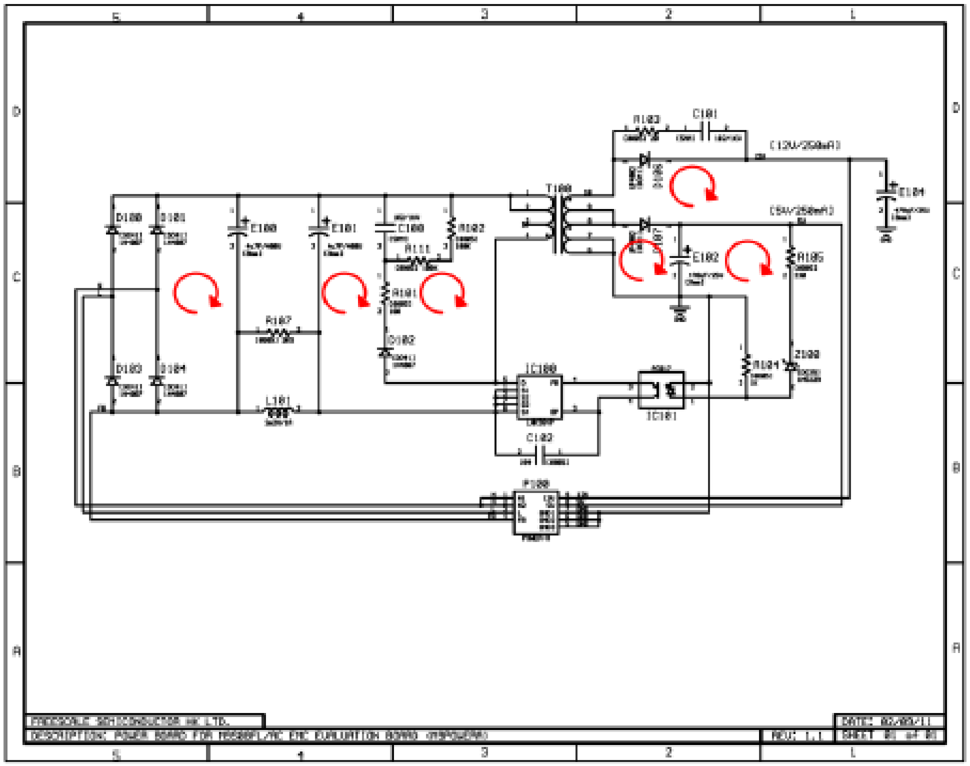

Figure 13. Loop consideration in SMPS circuit

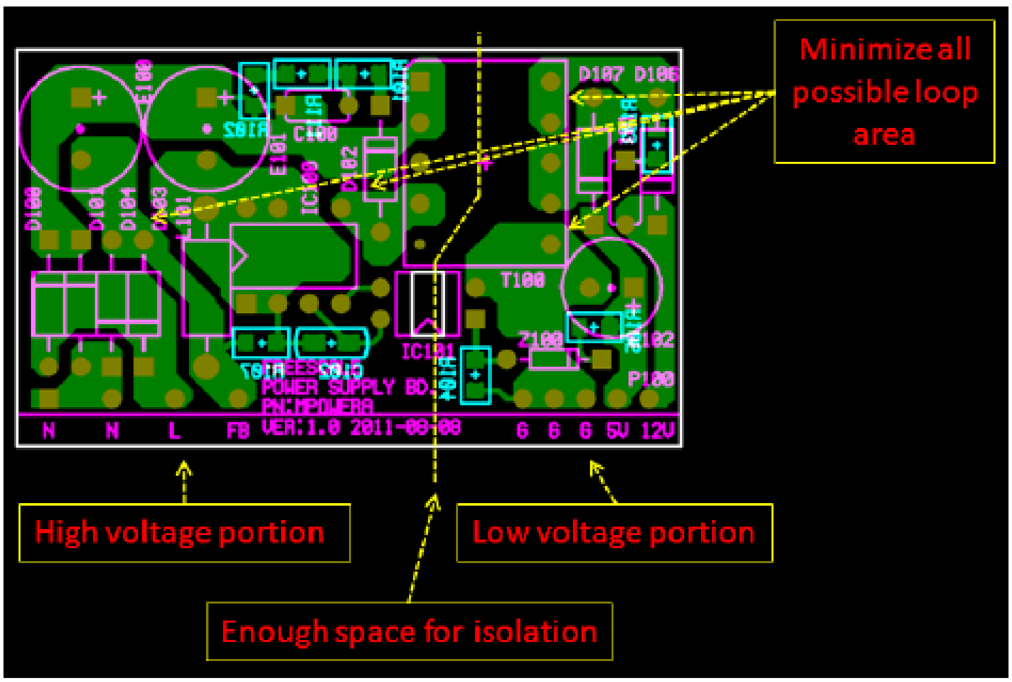

Figure 14. PCB layout implementation to minimize the return path loops

2.4 Main Control Board

Due to the space limitation, the AC to DC supply board is separated with the main control board which is mounted to main board vertically. Following are some important points that we need to take care:

- Good enough space to isolate the AC high-voltage portion and DC low voltage portion circuitry.
- In some case, a slot in the PCB needs to be added, to provide higher isolation between them.

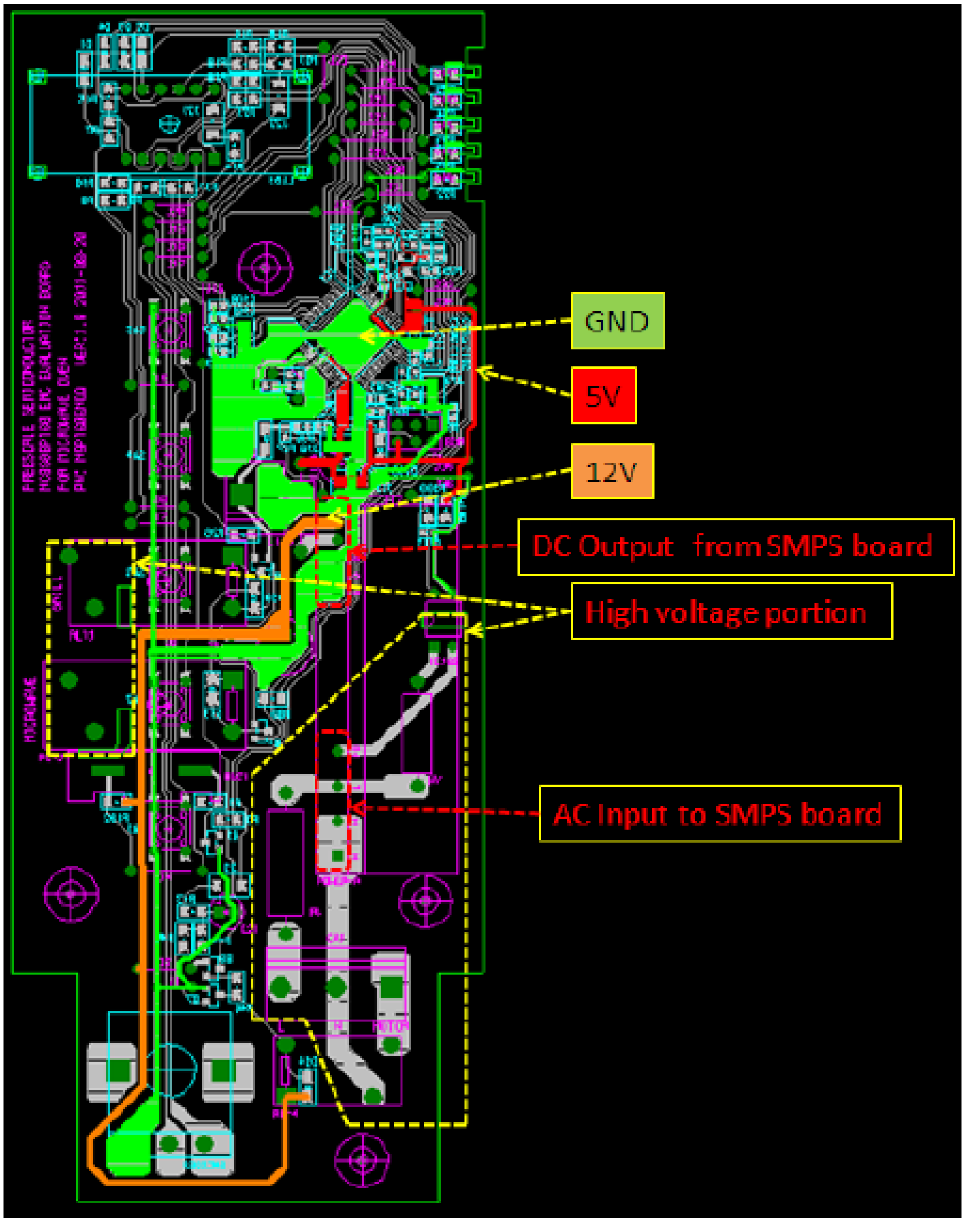

Figure 15. Overall placement and layout in PT60 board

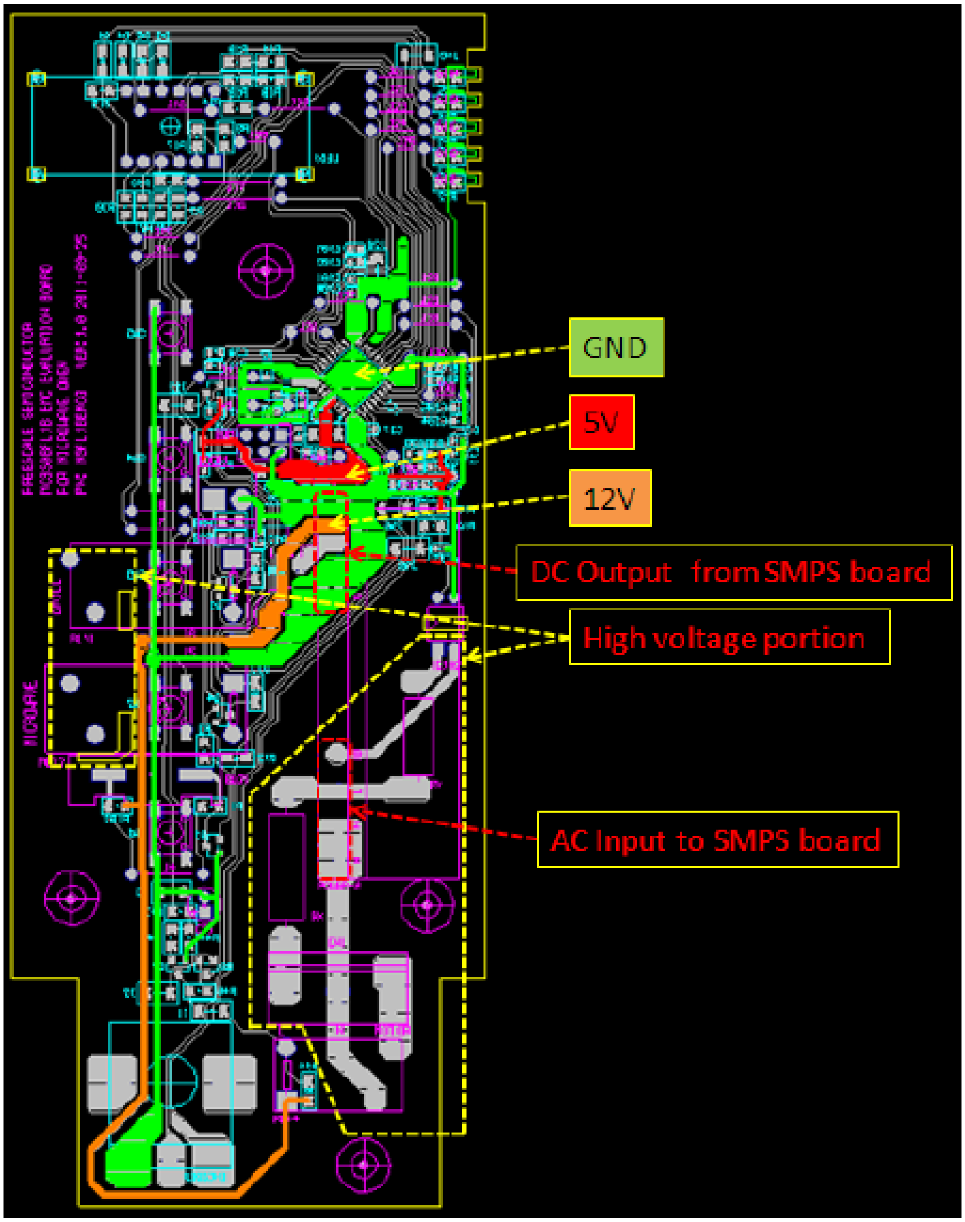

Figure 16. Overall placement and layout in FL16 board

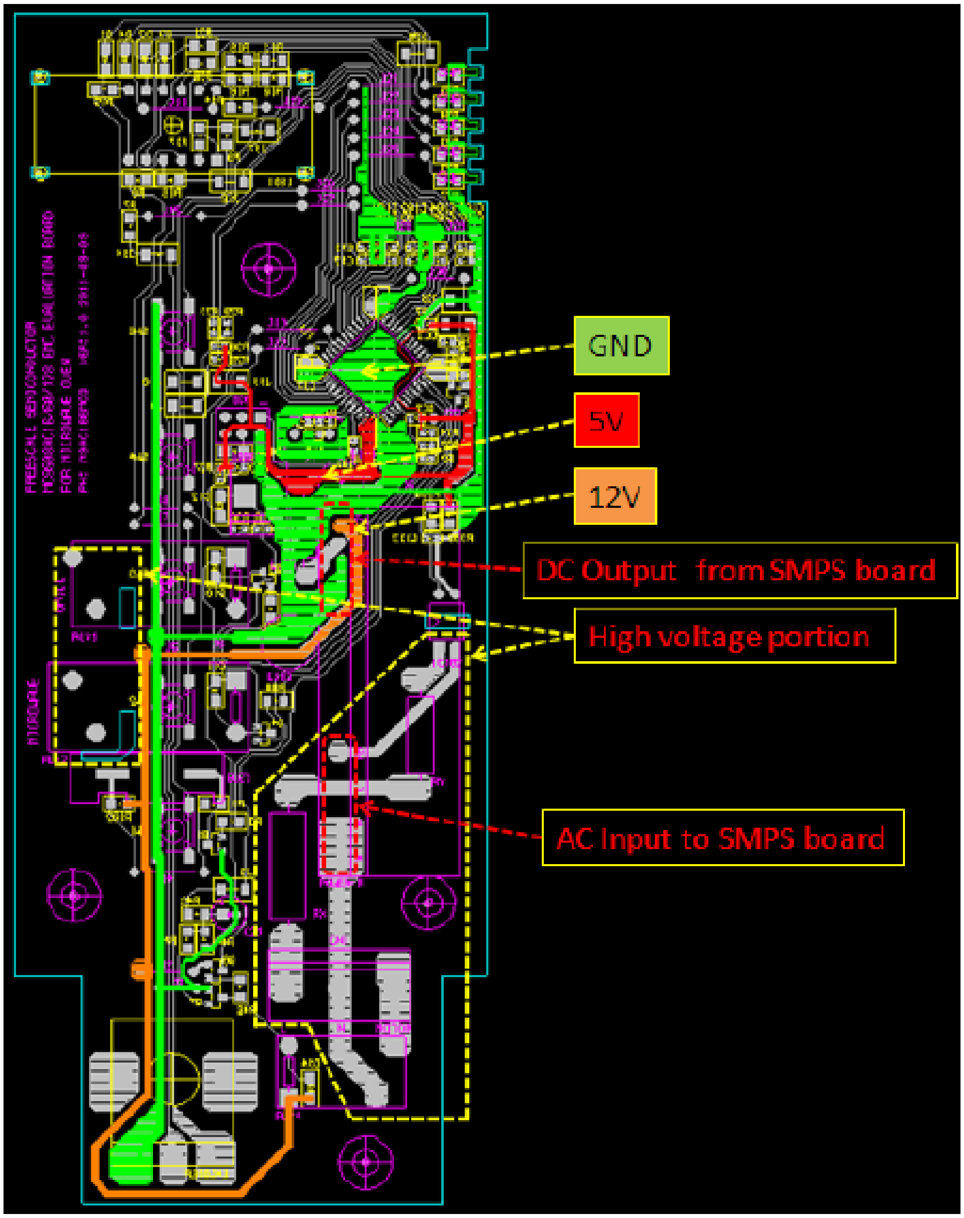

Figure 17. Overall placement and layout in AC16/60/128 board

2.5 Power Supply Routing and Grounding

Figure 15, 16 and 17, shows that the DC supply is coming from the vertical mounted SMPS board. In the output header of the SMPS board, we use the PCB layout technique to separate the ground into two portions. In figure 18, 19 and 20, it shows the ground from SMPS is divided into two sides by PCB layout technique. One side acts as the 12V ground return path except the switches and encoder. Another side acts as the 5V ground return path for MCU and those critical components. With that, the noise from the 12V ground will not be coupled to the 5V ground through the ground trace(s).

The reason to connect the 12V ground to the push buttons SW1-5 is to prevent the ESD discharge energy couples to the 5V GND directly. In some improper PCB layout with good enough ESD energy passing through the MCU may cause the MCU reset, or even hang up.

In the microwave oven, there is high noise energy in the ground of DOOR1 header during the door open and close operation. It is recommended to connect it near to the DC supply from SMPS.

In the MCU, to build the ground plane in the bottom and connect all Vss pins together is one of the good practices for EMC consideration. In this way, all Vss pins of the MCU are in the same potential level and provides the good high frequencies current return path (i.e. low inductance) back to the bypass capacitor. On the other hand, we use three corners to connect the ground from the ground plane to external circuitry that can reduce the ground loop for those critical components near the MCU.

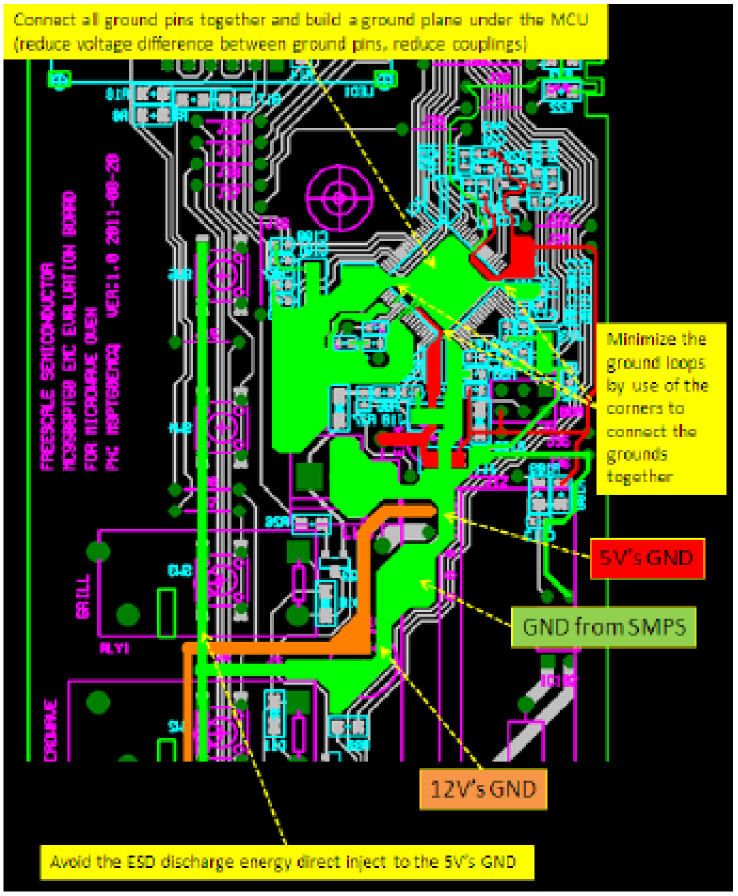

Figure 18. Power and ground routing in PT60 board

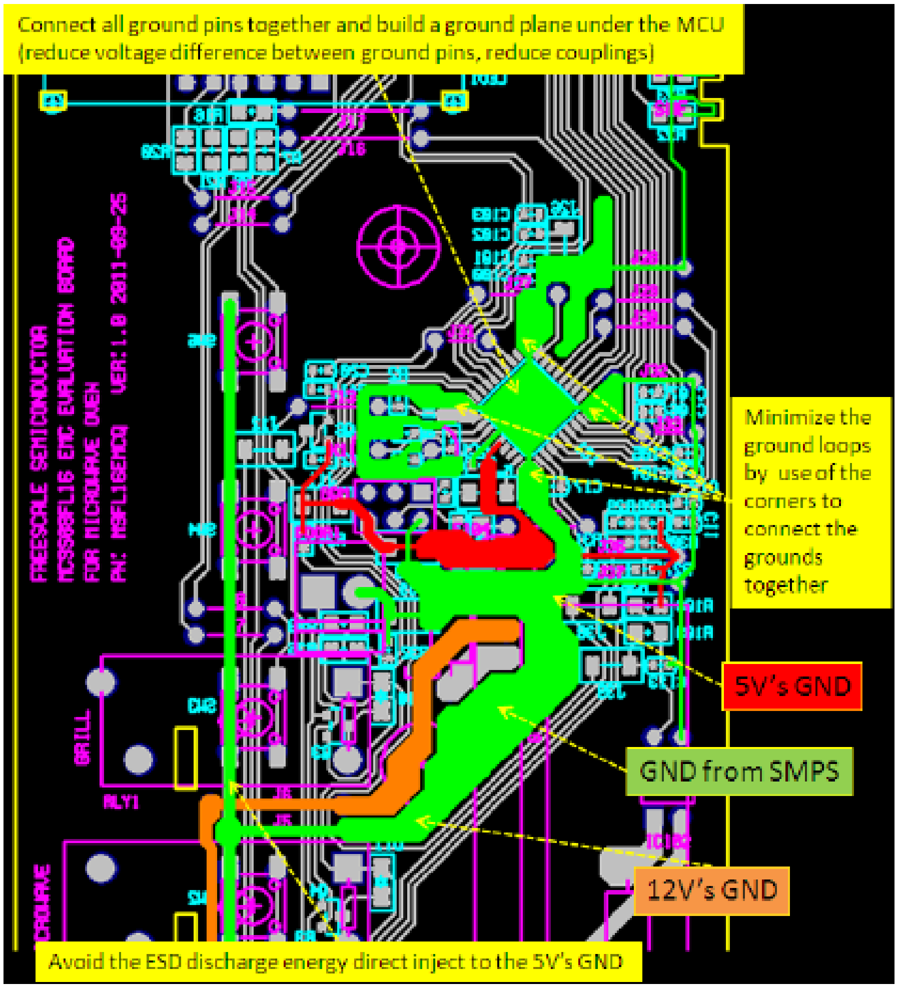

Figure 19. Power and ground routing in FL16 board

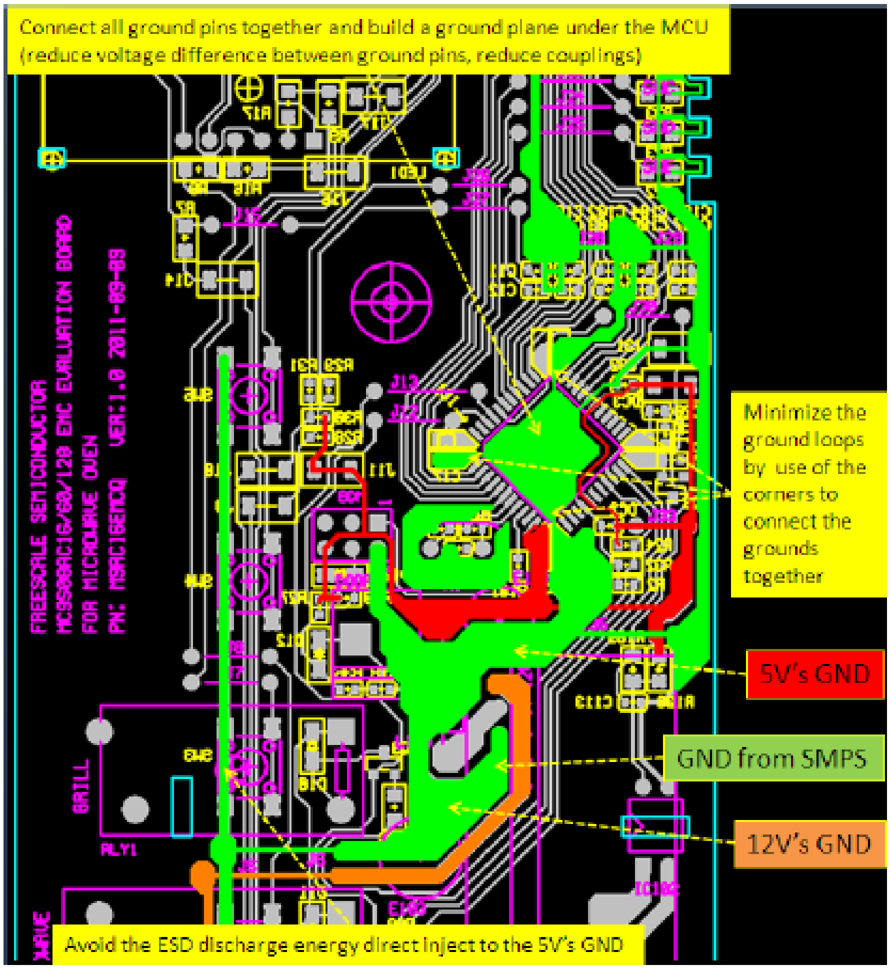

Figure 20. Power and ground routing in AC16/60/128 board

2.6 Bypass and Decoupling

We use two electrolytic capacitors E103 and E104 to provide the decoupling of the 12V and 5V supplies. This can help to minimize the common impedance coupling between 12V and 5V circuitries.

In 5V supply, we added the inductor L100 and associated with decoupling capacitor E104, which can eliminate the noise from other circuitry.

In PT60 MCU, we added three bypass capacitors near its Vdd and Vss pairs. PCB layout designer needs to be aware about the guideline given below:

- Connect Vdd and Vss traces from power source to the decoupling capacitor E104 first and then connect them to the bypass capacitor before going to MCU.
- Place Vdd and Vss in parallel to minimize loop area.
- Place the bypass capacitor to the Vdd and Vss as close as possible.

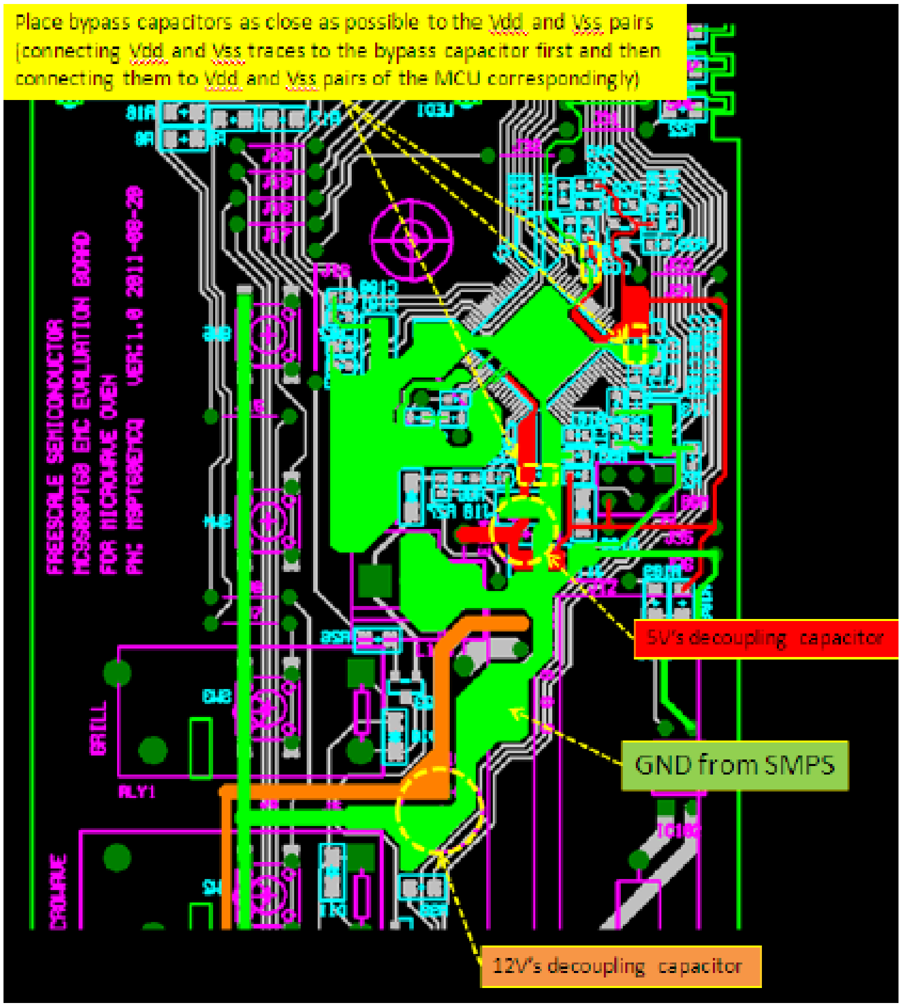

Figure 21. Bypass and decoupling in PT60 board

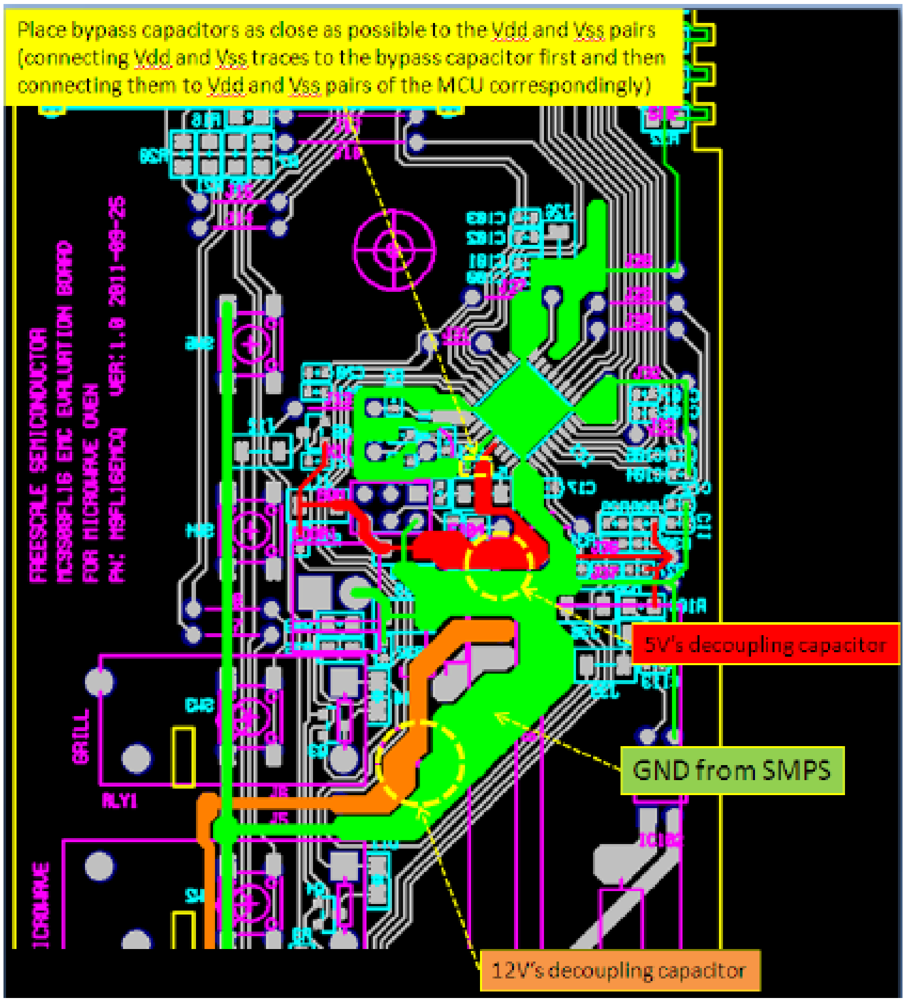

Figure 22. Bypass and decoupling in FL16 board

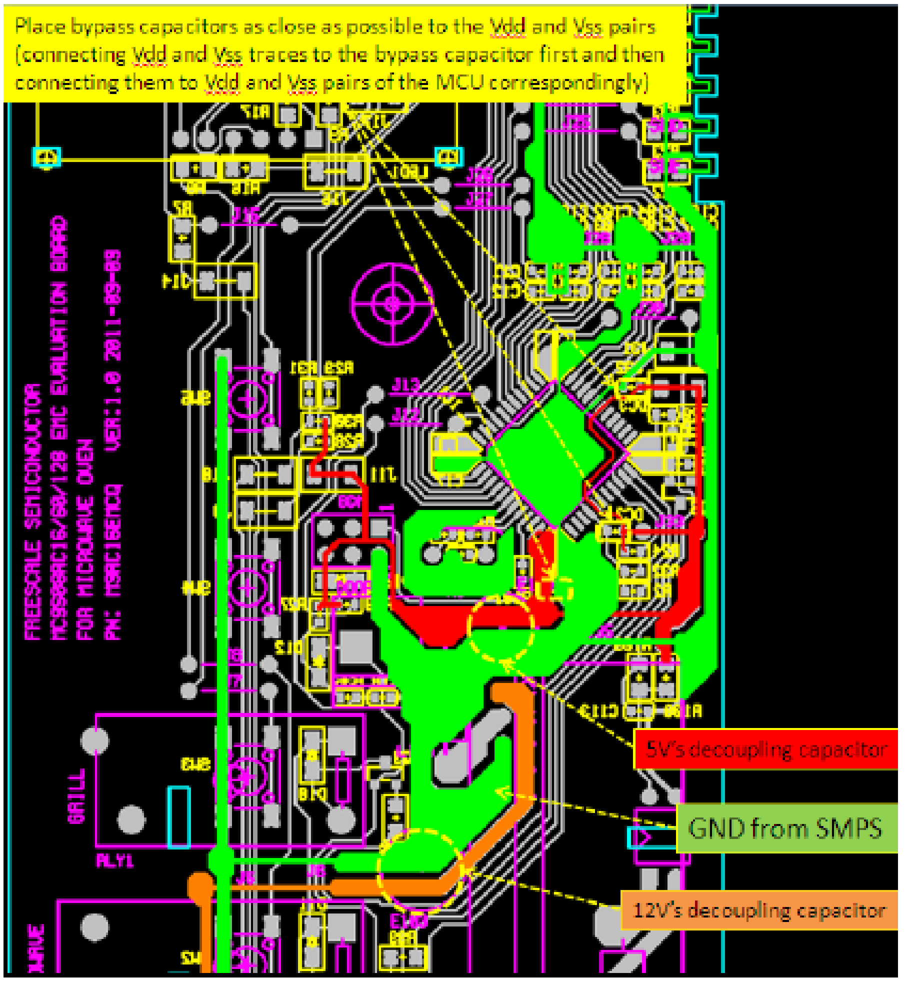

Figure 23. Bypass and decoupling in AC16/60/128 board

2.7 Crystal Oscillator Circuit

Crystal is easily affected by external noise, so we have to provide special attention in the layout. Additionally, the EXTAL and XTAL pins of the MCU are also very sensitive to external noise as well, placing the ground and guard ring (no current ground) along with the trace connecting to EXTAL and XTAL pin can minimize the noise coupling to the crystal circuit. The guidelines are listed as below:

- Do not place any trace except ground near the crystal circuit including the bottom of the crystal circuit.
- Place oscillator circuit (in this case is the crystal, feedback resistor and loading capacitors) to the EXTAL and XTAL pin as close as possible.
- Better EMC performance if we can use the internal oscillator instead.
- In double-layer or multi-layer PCB, the ground of the loading capacitor should be connected to the ground plane.
- Use the minimum bus frequency to fulfill the system requirement.
- Apply the minimum trace length for oscillator circuit.
- Use suitable value of the feedback resistor and loading capacitors.

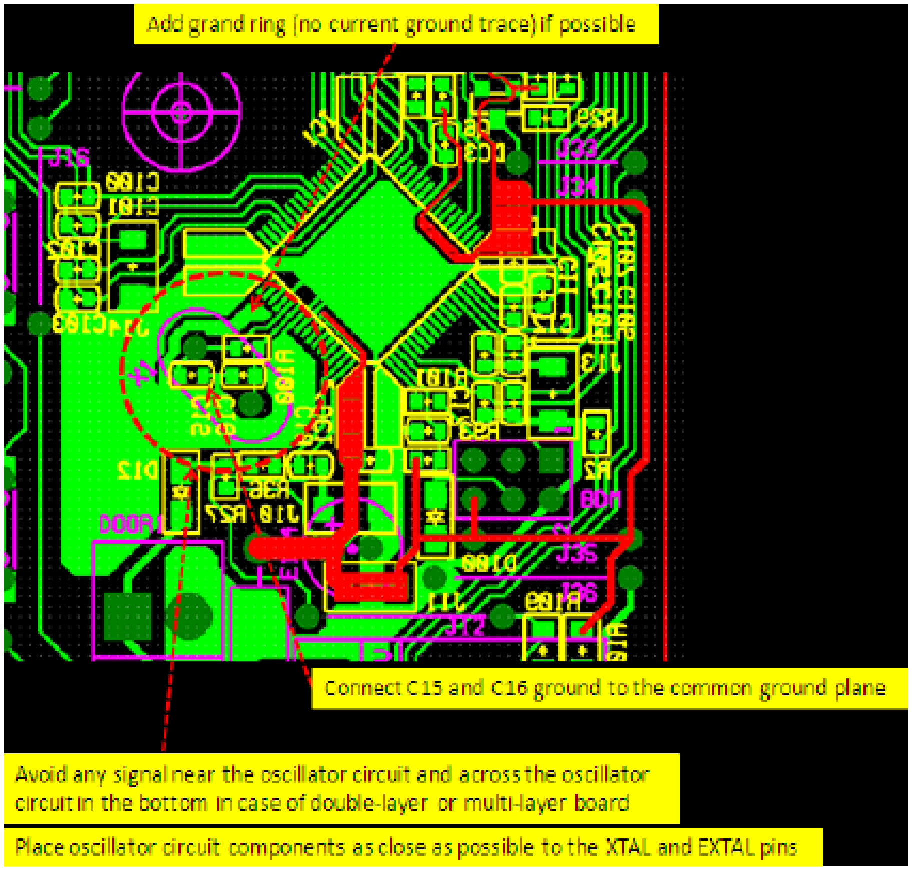

Figure 24. Oscillator circuit and layout in PT60 board

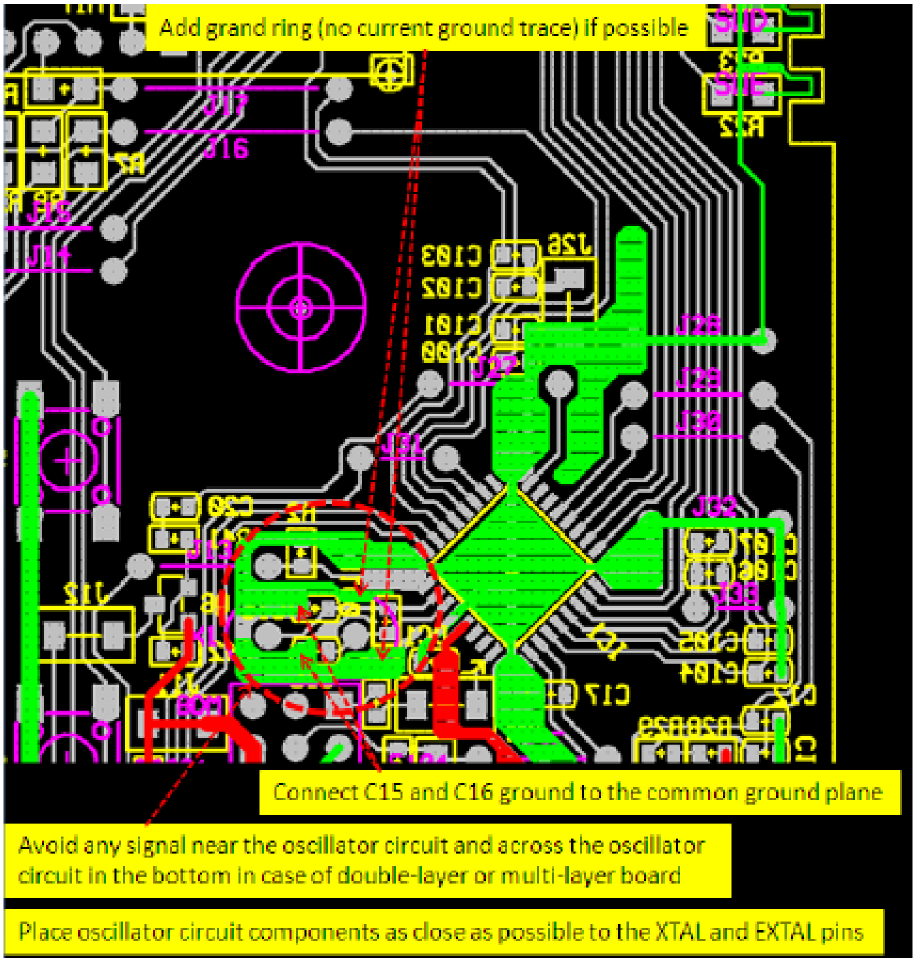

Figure 25. Oscillator circuit and layout in FL16 board

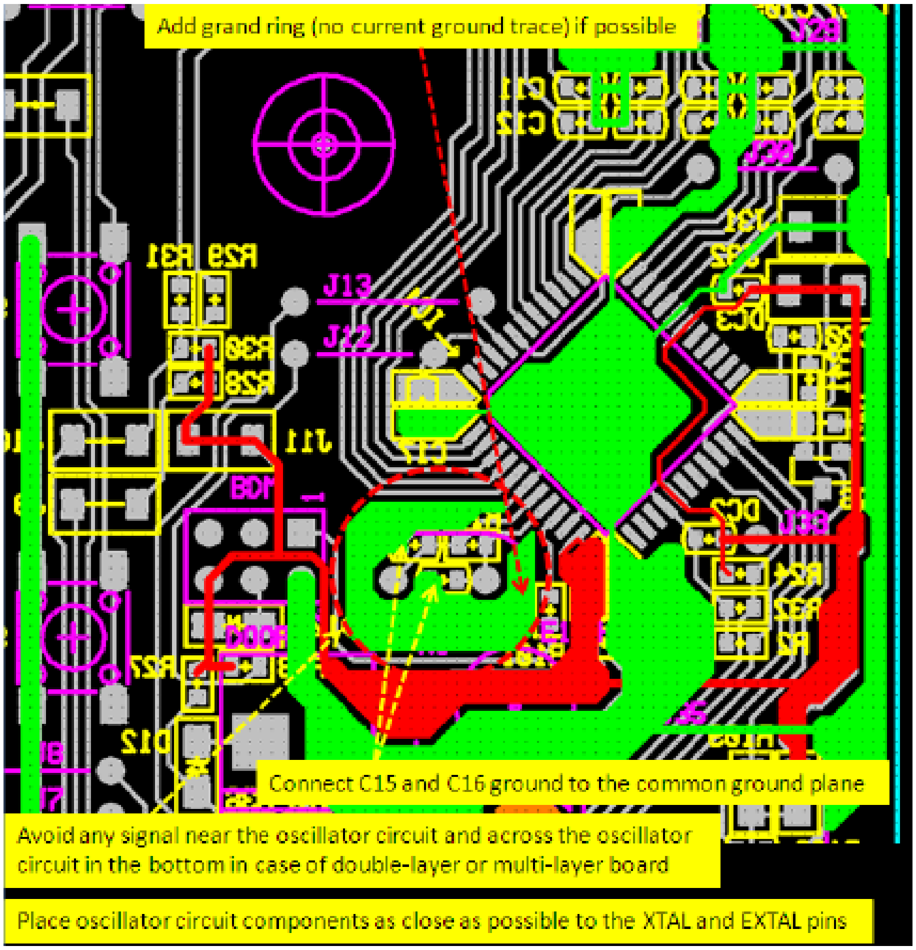

Figure 26. Oscillator circuit and layout in AC16/60/128 board

2.8 Spacing and Isolation

Figure 15, 16 and 17, shows there are the AC high-power region and DC low-power region. We need to provide enough isolation space between them. In some case, we may need to add a slot for even better isolation. Similarly, we need to apply enough isolation space between trace and screw holes or trace and board edge in particular for ESD consideration.

2.9 I/O

In MCU, the input port is also sensitive to noise. In most of the case, we add RC filter for input port. The value of the RC filter depends on the input signal and its characteristic (digital or analogue, rate of change). Typical value of the series resistor is in the range of 100 ohm to 1KΩ while the value of filtering capacitor is in the range of 1000pF to 0.1μF. As we mentioned in the critical component placement section, the RC filter should be placed near the input port as close as possible.

RESET, IRQ and BKGD/MS are special pins in PT60/FL16 and AC16/60/128 MCU. The typical circuits of RESET, BKGD and BDM header are shown in figure 7, 8 and 9. The corresponding layout is shown in figure 27, 28 and 29. The key concept is placed them near the MCU and to minimize the loops area of traces (Vdd , Vss) to reduce the noise coupling to those pins.

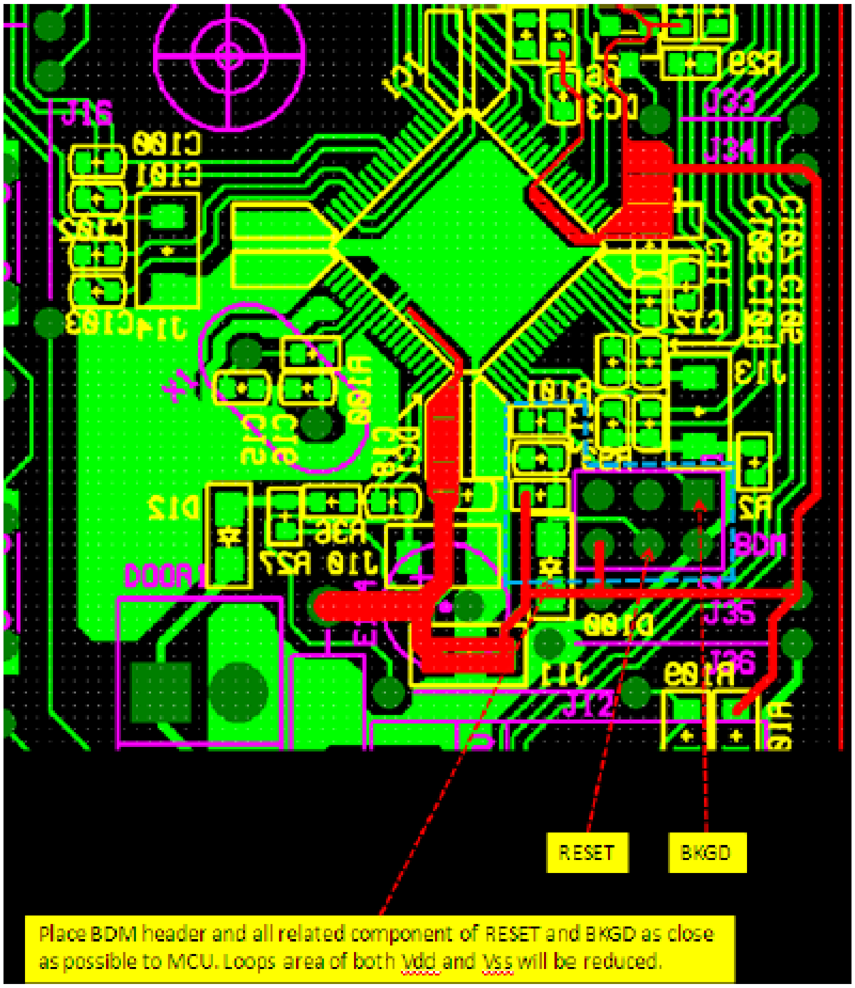

Figure 27. RESET, BKGD and BDM header and the layout in PT60 board

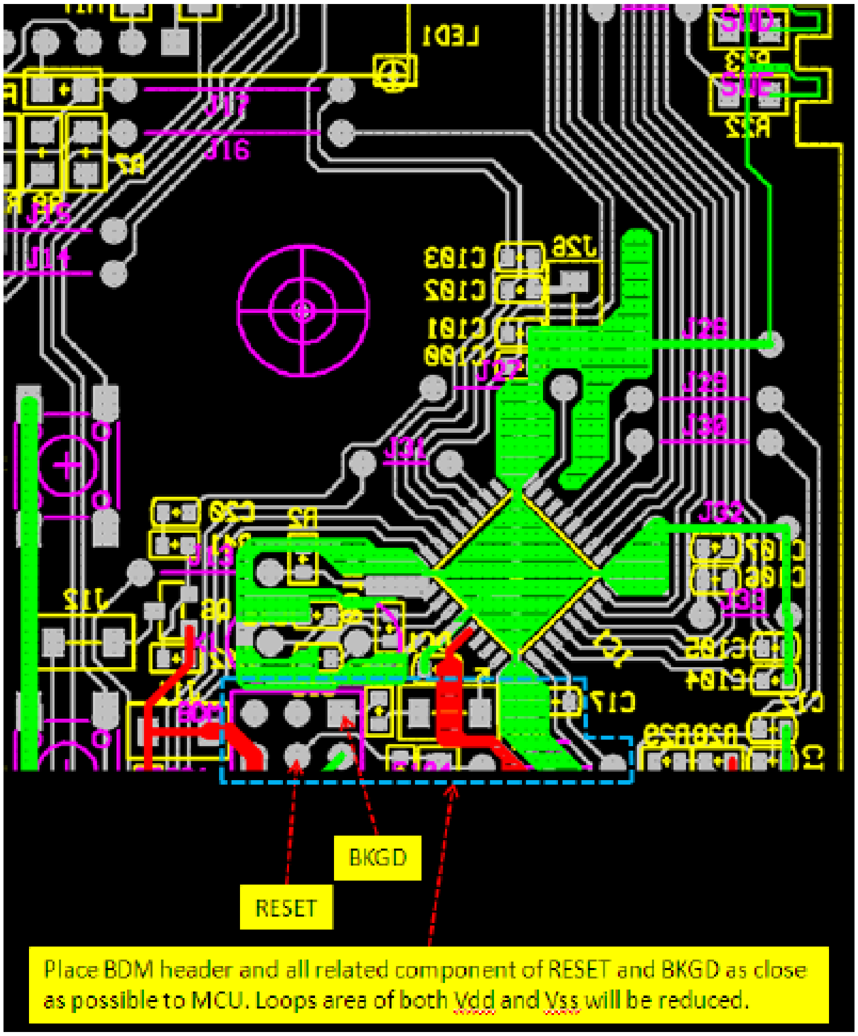

Figure 28. RESET, BKGD and BDM header and the layout in FL16 board

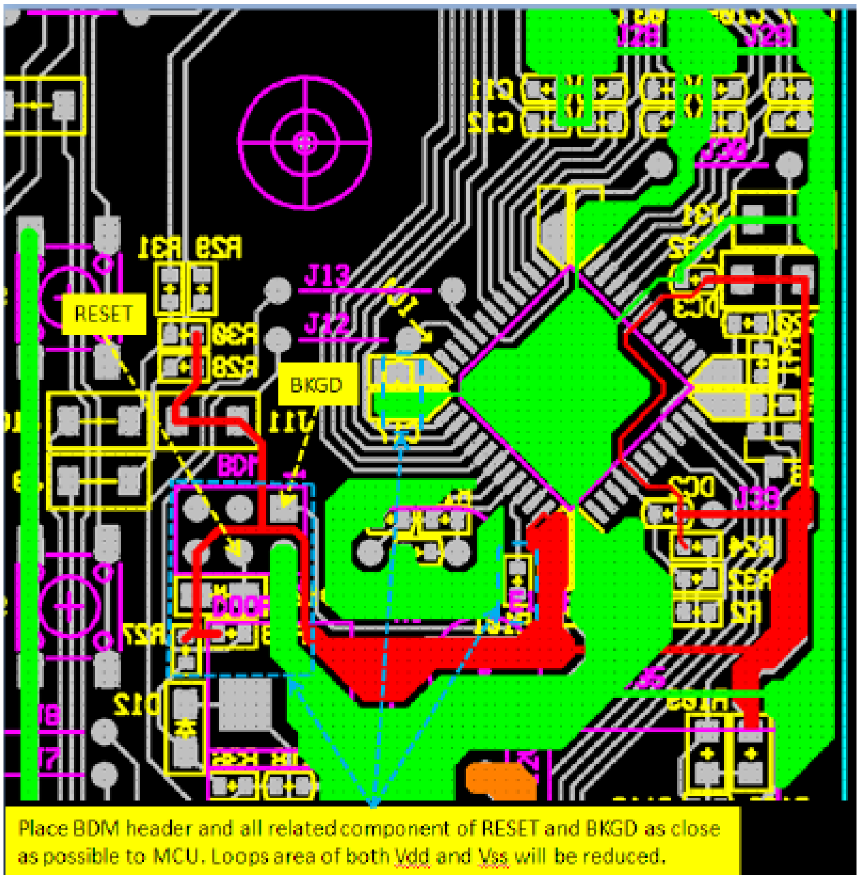

Figure 29. RESET, BKGD and BDM header and the layout in AC16/60/128 board

Unused I/O does not need to connect anything (i.e. float) and set it as output logic high or low, and the Direction Data Register (DDR) is needed to refresh periodically to avoid being changed with noise. The floating port pins are not allowed in some applications, therefore 10KΩ pull-down resistor should be connected to the unused pin. Do not try to connect any unused I/O to power or ground directly.

3 Software Techniques

In many cases, when the noise is coupled into the MCU, the system response depends on the software implementation. Good defensive software design can protect the system during the noise injection. Below are some important techniques.

- Refresh data direction register periodically
- Enable Watchdog to avoid code runaway
- Use of the slew rate to increase the rise edge of the digital signal
- Fill unused memory to avoid code runaway
- Define all interrupt vectors even those are not used
- Enable input glitch filter (PT60 build-in feature)

There is a new feature in PT60 MCU, the input glitch filter provides a simple low-pass filter for the I/O pins. The glitch width threshold can be programmable from 1 to 128ms.

4 Conclusion

There is always a challenge to build a low-cost PCB solution like a single-layer, double-sided loading with limited size and need to pass EMC compliance tests. Good planning to minimize the EMC risk at the beginning of the project is necessary to avoid the cost of EMC fix.

Although most of the MCU manufacturers spend a lot of time in improving their chip EMC performance, good PCB layout, hardware design, software design as well as good product designs are needed to minimize coupling paths from the noise source to the receptor.

5 References

- Ott, H., “Noise Reduction Techniques in Electronic Systems. New York: Wiley, 1976.
- Ott, H., “Digital Circuit Grounding and Interconnection,” Proceedings of the IEEE Symposium on Electromagnetic Compatibility, pp. 292-297, Aug. 1981.
- C.R. Paul., “Introduction to Electromagnetic Compatibility,” John Wiley Interscience, 1992.
- Martin O,Hara., “EMC at Component and PCB Level,” Newnes, 1998. -
- Keith Armstrong., “PCB Design Techniques for Lowest-Cost EMC Compliance, Part1,” Electronics and Communication Engineering Journal, August 1999. -
- Mark I. Montrose., “Printed Circuit Board Techniques for EMC Compliance,” IEEE press series 2000.
- T,C, Lun., “Designing for Board Level Electromagnetic Compatibility,” Freescale Application Note, AN2321. 2002.
- Ross Carlton, Greg Racino, John Suchyta., “Improving the Transient Immunity Performance of Microcontroller-Based Applications,” Freescale Application Note, AN2764. 2005.

How to Reach Us:

Home Page:
www.freescale.com

Web Support:
http://www.freescale.com/support

USA/Europe or Locations Not Listed:
Freescale Semiconductor
Technical Information Center, EL516
2100 East Elliot Road
Tempe, Arizona 85284
+1-800-521-6274 or +1-480-768-2130
www.freescale.com/support

Europe, Middle East, and Africa:
Freescale Halbleiter Deutschland GmbH
Technical Information Center
Schatzbogen 7
81829 Muenchen, Germany
+44 1296 380 456 (English)
+46 8 52200080 (English)
+49 89 92103 559 (German)
+33 1 69 35 48 48 (French)
www.freescale.com/support

Japan:
Freescale Semiconductor Japan Ltd.
Headquarters
ARCO Tower 15F
1-8-1, Shimo-Meguro, Meguro-ku,
Tokyo 153-0064
Japan
0120 191014 or +81 3 5437 9125
support.japan@freescale.com

Asia/Pacific:
Freescale Semiconductor China Ltd.
Exchange Building 23F
No. 118 Jianguo Road
Chaoyang District
Beijing 100022
China
+86 10 5879 8000
support.asia@freescale.com

Document Number: AN4476
Rev. 0, 03/2012