A Practical Approach to Education of Electromagnetic Compatibility at the Undergraduate Level

Yang Zhao [1 **] Kye Yak See [2]
[1]School of Electrical Engineering, Nanjing Normal University, Nanjing 210042, China
[2]School of Electrical Engineering, Nanyang Technology University, Republic of Singapore

I. INTRODUCTION

Because of worldwide trend of imposing electromagnetic compatibility (EMC) regulations on electronic products, EMC design has been gaining more attention than before in the electronic industries. Unfortunately, in most universities, very little or none has been taught at the undergraduate level in the area of EMC compliance and design. Due to the lack of EMC knowledge, these future electronic engineers, when they graduate and enter the industry, will have to learn the EMC design technique through the painful trial-and-error process. Not only that, the companies that hire these engineers also suffered from higher design cost and unnecessary production delay, which are crucial competing factors in the electronic markets. A basic knowledge of the electromagnetic interference (EMI) generation mechanism and the mitigation techniques will help to avoid EMI problems as far as possible at the product design stage. This is a rational and economical approach when compared to leaving the EMI problems to be tackled and mitigated after the product is developed and assembled.

In view of this short-coming in EMC training at the undergraduate level, two years ago, a special training program is initiated in Nanyang Technological University of Singapore for the second-year students during their in-house practical training (IHPT) module. The IHPT module is a compulsory subject for all second-year engineering students. The objective of this module is to prepare the students to be more practice-oriented when they graduate from university. Under the IHPT module, students have a wide variety of projects to choose. The authors proposed a project entitled "printed circuit board (PCB) layout design for EMC compliance". The authors know that it is impossible to cover all aspects of EMC in an eight-week duration. It is better for student to focus on certain EMC design aspect that is most beneficial to them. Since almost all electronic circuits and systems are packaged at the PCB level, the authors feel that PCB layout design is a very good topic to start with. By going through this eight-week program, the students are expected to (1) realize the need of meeting the EMC standards (2) understand to study the mechanism of EMI radiation from the PCB (3) to operate basic equipment for EMI measurement (4) to learn the proper PCB layout techniques for EMI control.

II. SUMMARY OF THE EDUCATION PROGRAM

In the eight-week program, a standard digital circuit, as shown in Fig.1, is first given to all the students who have chosen the project. A 9V battery powers the circuit through a MC7805T voltage regulator that supplies 5V dc power to all gates. A 10MHz crystal is employed to provide

* Corresponding author: Dr.Y. Zhao, Tel: +86-25-6307596 e-mail: zhaoy2995@sina.com

0-7803-7846-6/03/$17.00 ©2003 IEEE

the clock signal. The 74ALS04 inverter gate is to serve as a buffer and to provide the sufficient drive currents for the subsequence gates. Based on the given circuit, the students are asked to route the circuit on a single-layer double-layer PCB, and each of them will be given two chances to fabricate the PCB. For the first PCB fabrication, no information is given to the students and they have the freedom to do their own PCB layout. Once the PCB is fabricated with all the components in place, they need to perform a functional test to ensure that all the digital signal are within specifications. Next, the radiated EMI levels generated by PCB, in the frequency range of 30MHz ~ 1GHz, are measured using a GTEM based EMI measurement system, and its radiated EMI requirement to be complied with is EN 55022 Class B limit. All the students, without exception, failed to get the first PCB layout passed the limit. In the following week, the students attend a one-week EMC design short course with emphasis on PCB layout technique. After that, they proceed with the second PCB layout for the same circuit. Without surprise, more than 90% of the students managed to get the second PCB passed EN55022 class B limit with good margin

III. PCB LAYOUT WITH AND WITHOUT EMC CONSIDERATION

The first layout is done without any EMC consideration. Basically once the component footprints are finalized, the auto-routing algorithm of PCB layout software carries out routing of the interconnecting tracks automatically. Usually auto-routing functions take care of the physical aspects such as looking for the most direct and convenient paths among the components, but failed to consider the EMC design aspects. Because of that, the layout tools tend to overlook the return current paths for signal and power track. Since currents flow in closed loops, without proper consideration, many unexpected large loop areas are created and become efficient radiating loop antennas at certain frequencies.

The second layout is done after the students attended the EMC design course. In the course the authors stressed the importance of knowing where are the return paths for signals and dc power distributions. Once the return path are identified, they should be routed as close as possible the corresponding signal and +5V tracks. One example is to use ground plane as shown in Fig. 2.

IV. RADIATED EMI RESULTS.

The measured radiated emissions for layout with and without ground plane are shown in Fig.3 and Fig.4 respectively. To illustrate how to control common-mode (CM) radiation through cable, radiated emission from PCB alone is first measured, then a 30cm wire is soldered to the ground of PCB and radiated emission is measured again. Fig.3a shows that without EMC consideration,, radiated emissions have already exceed EN 55022 class B limit. Due to bad layout practice, significant ground bounce is developed in the circuit ground. This is clearly reflected in Fig.3b, where noticeable increase of 5 to 20 dB in radiated emission is observed when wire is soldered to the ground. With as much conducting area as possible for circuit ground, the radiated emissions have been suppressed greatly as seen in Fig.4a. Even with wire soldered to the ground, the overall radiated emissions are still well below the limit as seen in Fig.4b.

V. CONCLUSION

In this paper , a practical approach is adopted to conduct EMC design training in Nanyang Technological University of Singapore for the second-year students. The program has received very good feedback from the students and most of them expressed that they have benefited from the program. The authors believe that appropriate EMC training in university education at undergraduate level is of much benefit to the future electronic engineers, not only on technology, but also on economics.

REFERENCE

[1]ECC title47 part 15, 18, and 68, US code federal regulation, Federal communication Committee, Washing DC

[2]V.P.Kodali, Engineering electromagnetic compatibility: principle,measurement and technique, IEEE Press, 1996

[3]J.S.Scott, and C.V.Zyl, Introduction to EMC, Butterworth_Heinemann Linance House, oxford, 1997

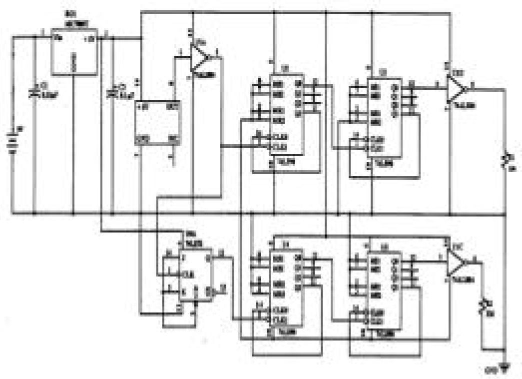


Fig.1 Schematic of the standard digital circuit

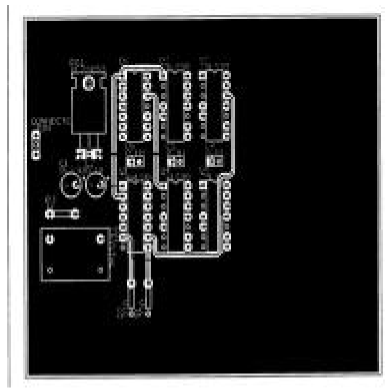
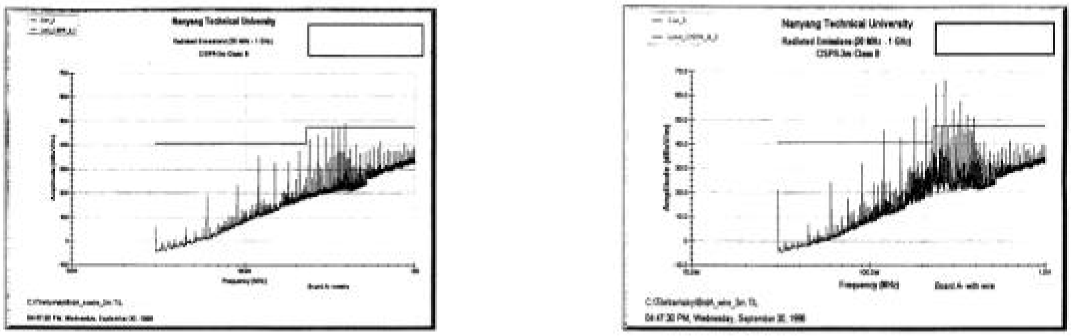
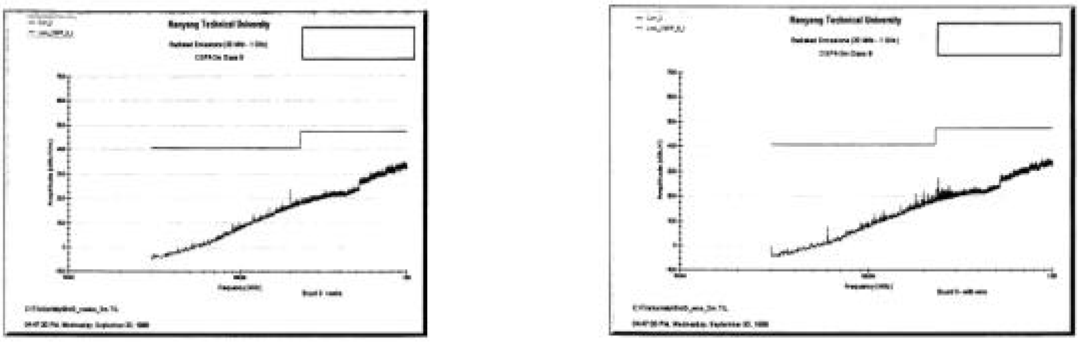


Fig.2 PCB layout with large conducting area as ground plane



(a) PCB alone **(b) PCB with attached wire**

Fig.3 Radiated EMI levels without EMC consideration



(a) PCB alone **(b) PCB with attached wire**

Fig.4 Radiated EMI levels with EMC consideration