Signal Integrity and PCB layout considerations for DDR2-800 Mb/s and DDR3 Memories

Fidus Systems Inc.

900, Morrison Drive, Ottawa, Ontario, K2H 8K7, Canada

Chris Brennan, Cristian Tudor, Eric Schroeter, Heike Wunschmann, and Syed Bokhari

Session # 8.13

Presented at

cadence designer network

Silicon Valley 2007

Abstract

The paper addresses the challenge of meeting Signal Integrity (SI) and Power Integrity (PI) requirements of Printed Circuit Boards (PCBs) containing Double Data Rate 2 (DDR2) memories. The emphasis is on low layer count PCBs, typically 4-6 layers using conventional technology. Some design guidelines have been provided.

1. Introduction

DDR2 usage is common today with a push towards higher speeds such as 800 Mbps [1] and more recently, 1066 Mbps. DDR3 [2] targets a data rate of 1600 Mbps. From a PCB implementation standpoint, a primary requirement is delay matching which is dictated by the timing requirement. This brings into it a number of related factors that affect waveform integrity and delay. These factors are interdependent, but where a distinction can be made, they can be termed PCB layer stackup and impedance, interconnect topologies, delay matching, cross talk, PI and timing. Cadence ALLEGRO™SI-230 and Ansoft's HFSS™ are used in all computations.

VDD / Vref
Clock
Address
Command/ Control
DataStrobe (differential)
DataMask
Data
Controller
CKP,CKN
ADDR<15,0>
CKE, CS, ODT, RAS,CAS,WE,BA0-2
DQS0,DQS1,DQS2,DQS3
DM0,DM1,DM2,DM3
DQ <7,0>, DQ<15,8>,DQ<23,16>, DQ<31,24>
VDD / Vtt / Vref
Memory

Technology	**DDR2**	**DDR3**
Max Clock Freq. (MHz)/Data rate(Mbps)	533/1066	800/1600
Power Requirement		
VDD (Volts)	1.8 +/- 0.1	1.5 +/- 0.075
Vtt (Volts)	0.9 +/- 0.04	0.75 +/- TBD
Vref (Volts)	0.9 +/- 0.018	0.75 +/- 0.015
Input Thresholds		
Vih/Vil (Volts)	0.9 +/- 0.2	0.75 +/- 0.175
Delay Matching Requirement		
Match ADDR/CMD/CNTRL to Clock tightly	Yes	Yes
Match DQ<7,0>, DM0 to DQS0 tightly	Yes	Yes
Match DQ<15,8>, DM1 to DQS1 tightly	Yes	Yes
Match DQ<22,16>, DM2 to DQS2 tightly	Yes	Yes
Match DQ<31,23>, DM3 to DQS3 tightly	Yes	Yes
Match DQS0-3 to Clock loosely	Yes	Not required

Table 1: Comparison of DDR2 and DDR3 requirements

Signals common to both technologies and a general comparison of DDR2 and DDR3 is shown in Table 1. It must be noted that "matching" includes cases where the clock net may be made longer (termed DELTA in ALLEGRO SigXP). We have assumed a configuration comprising a Controller and two SDRAMs in most illustrations that follow.

2. PCB Layer stackup and impedance

In a layer constrained implementation, a 4 layer PCB (Figure 1) is a minimum with all routing on TOP and BOTTOM layers. One of the internal layers will be a solid ground plane (GND). The other internal plane layer is dedicated to VDD. Vtt and Vref can be derived from VDD. Use of a 6-layer PCB makes the implementation of certain topologies easier. PI is also enhanced due to the reduced spacing between power and GND planes.

The interconnect characteristic impedance for DDR2 implementation can be a constant. A single-ended trace characteristic impedance of 50 Ohms can be used for all single-ended signals. A differential impedance of 100 Ohms can be used for all differential signals, namely CLOCK and DQS. Further, the termination resistor pulled up to VTT can be kept at 50 Ohms and ODT settings can be kept at 50 Ohms.

In the case of DDR3 however, single ended trace impedances of 40 and 60 Ohms used selectively on loaded sections of ADDR/CMD/CNTRL nets have been found to be advantageous. Further, the value of the termination resistor pulled up to Vtt needs to be optimized in combination with the trace impedance through SI simulations. Typically, it is in the range 30 – 70 Ohms. The differential trace impedance can remain at 100 Ohms.

Layers	Thks.(mils)	Cross Section Diagram	Layer Type	Layer Definition
			mask	
	1.4		plating	
L01	0.600		.5 oz foil	SIG
	3.0		Prepreg	
L02	0.6			GND
	50.8		0.5/0.5 Core	
L03	0.6			PWR
	3.0		Prepreg	
L04	0.600		.5 oz foil	SIG
	1.4		plating	
			mask	
Total:	*58*	***Finish thickness over laminate +-10%***		
	62	***over plate***		

Layers	Thks.(mils)	Cross Section Diagram	Layer Type	Layer Definition
			mask	
	1.4		plating	
L01	0.600		.5 oz foil	SIG
	4.0		Prepreg	
L02	0.6			GND1
	4.0		0.5/0.5 Core	
L03	0.6			PWR1/SIG
	40.0		Prepreg	
L04	0.6			PWR2/SIG
	4.0		0.5/0.5 Core	
L05	0.6			GND2
	4.0		Prepreg	
L06	0.600		.5 oz foil	SIG
	1.4		plating	
			mask	
Total:	*58*	***Finish thickness over laminate +-10%***		
	62	***over plate***		

Figure 1 : Four and Six layer PCB stackup

3. Interconnect Topologies

In both cases of DDR2 and DDR3, DQ, DM and DQS signals are point-to-point and do not need any topological consideration. An exception is in the case of multi-rank Dual In Line Memory Modules (DIMMs). Waveform integrity is also easily addressed by a proper choice of drive strengths and On Die Termination (ODT). The ADDR/CMD/CNTRL signals, and sometimes the clock signal will involve a multipoint connection where a suitable topology is needed. Possible choices are indicated in Figure 2 for cases involving two SDRAMs. The Fly-By Topology is a special case of a daisy chain with a very short or no stub.

For DDR3, any of these topologies will work, provided that the trace lengths are minimized. The Fly-by topology shows the best waveform integrity in terms of an increased noise margin. This can be difficult to implement on a

4-layer PCB and the need for a 6-layer PCB arises. The daisy chain topology is easier to implement on a 4 layer PCB. The tree topology on the other hand requires the length of the branch AB to be very close to that of AC (Figure 2). Enforcing this requirement results in the need to increase the length of the branches which affects waveform integrity. Therefore, for DDR3 implementation, the daisy chain topology with minimized stubs proves to be best suited for 4-layer PCBs.

For DDR2-800 Mbps any of these topologies are applicable with the distinction between each other being less dramatic. Again, the daisy chain proves to be superior in terms of both implementation as well as SI.

Where more than two SDRAMs are present, often, the topology can be dictated by constraints on device placement. Figure 3 shows some examples where a topology could be chosen to suit a particular component placement. Of these, only A and D are best suited for 4-layer PCB implementation. Again, for DDR2-800 Mbps operations all topologies yield adequate waveform integrity. For a DDR3 implementation, in particular at 1600 Mbps, only D appears to be feasible.

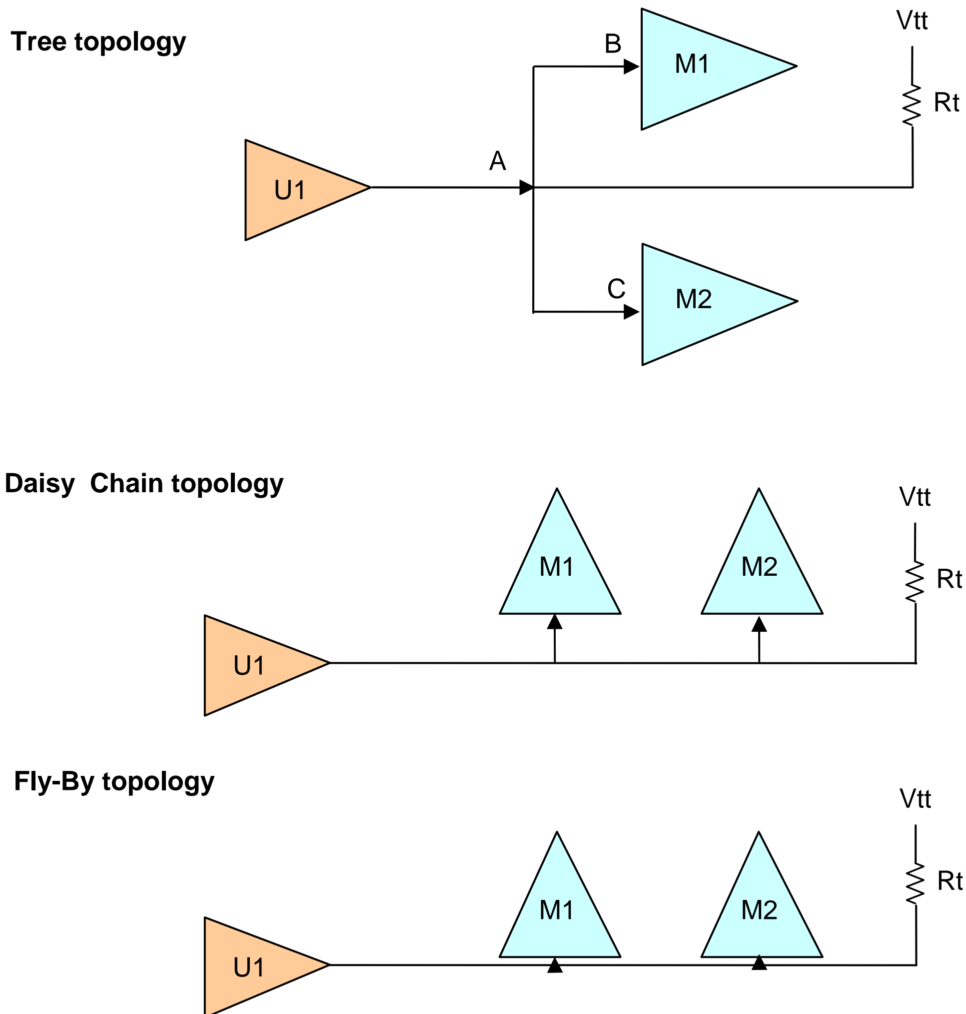


Figure 2: ADDR/CMD/CNTRL topologies with 2 SDRAMS

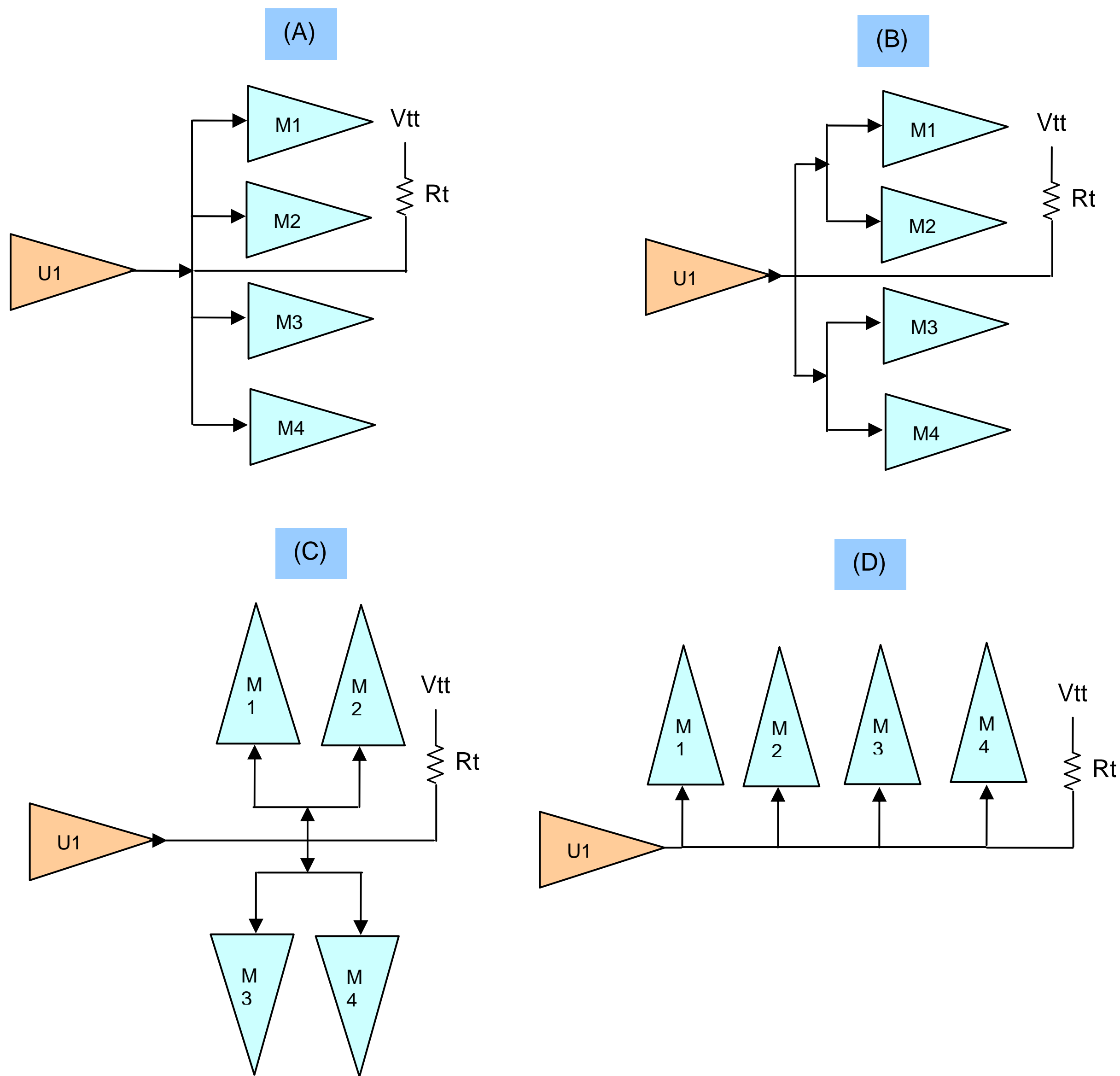


Figure 3: ADDR/CMD/CNTRL topologies with four SDRAMS

4. Delay matching

Implementing matched delay is usually carried out by bending a trace in a trombone shape. Routing blockage may require layer jumping. Unfortunately, while physical interconnect lengths can be made identical in layout, electrically, the two configurations shown in Figure 4 will not be the same.

The case of trombone delay has been well understood, and the case of a via is obvious. The delay of a trombone trace is smaller than the delay of a straight trace of the same center-line length. In the case of a via, the delay is more than that of a straight microstrip trace of length equal to the via length. The problem can be resolved in two different ways. In the first approach, these values can be pre-computed precisely and taken into account while delay matching. This would become a tedious exercise which could perhaps be eased with user

defined constraints in ALLEGRO 16.0. In the second approach, one would use means to reduce the disparity to an acceptable level.

Figure 4: Illustration of Trombone traces and Vias

Figure 5: Circuit for estimation of trombone effect and resulting waveforms.

Consider the case of a trombone trace. It is known that the disparity can be reduced by increasing the length of L3 (Figure 4). Details can be found in reference [3]. A simulation topology can be set up in SigXP to represent parallel arms of a trombone trace as coupled lines. A sweep simulation is carried out with L3 (S in Figure 5) as a variable and the largest reasonable value that reduces the delay difference with respect to a reference trace is selected. For microstrip traces, L3 > 7 times the distance of the trace to ground is needed.

Delay values are affected in a trombone trace due to coupling between parallel trace segments. Another way to reduce coupling without increasing the spacing is to use a saw tooth profile. The saw tooth profile shows better performance as compared to a trombone although it eventually ends up requiring more space. In either case, it is possible to estimate the effect on delay precisely by using a modified equation for the computation of the effective trace length [3]. This would need to be implemented as a user defined constraint in ALLEGRO.

Consider the case of a through hole via on the 6 layer stackup of Figure 2. Ground vias placed close to the signal vias play an important role in the delay. For the illustration, the microstrip traces on TOP and BOTTOM layers are 150 mils long, and 4 mils wide. The via barrel diameter = 8 mils, pad diameter is 18 mils and the anti-pad diameter is 26 mils.

Three different cases are considered. In the first case, the interconnect with via does not have any ground vias in its immediate neighborhood. Return paths are provided at the edges of the PCB 250 mils away from the signal via. In the second case, a reference straight microstrip trace of length = 362 mils is considered. The third case is the same as case 1 with four ground vias in the neighborhood of the signal via. Computed s-parameters with 60 Ohm normalization are shown in Figure 6. It can be seen that the use of 4 ground vias surrounding the signal via makes its behavior more like a uniform impedance transmission line and improves the s21 characteristic. In the absence of a return path in the immediate neighborhood, the via impedance increases. For the present purpose, it is important to know the resulting impact on the delay.

A test circuit is set up similar to Figure 5. The driver is a linear source of 60 Ohms output impedance and outputs a trapezoidal signal of rise time = fall time = 100 ps and amplitude = 1V. It is connected to each of the 3 interconnects shown in Figure 6 and the far end is terminated in a 60 Ohm load. The excitation is a periodic signal with a frequency of 800 MHz. The time difference between the driver waveform at V = 0.5 V and the waveform at the receiver gives the switched delay.

Results are illustrated in Figure 7 where only the rising edge is shown. It can be seen that the delay with four neighboring ground vias differs from that of the straight trace by 3 ps. On the other hand, the difference is 8 ps for the interconnect with no ground vias in the immediate neighborhood.

It is therefore clear that increasing the ground via density near signal vias will help. However, in the case of 4 layer PCBs, this will not be possible as the signal traces adjacent to the Power plane will be referenced to a Power plane. Consequently, the signal return path would depend on decoupling. Therefore, it is very important that the decoupling requirement on 4 layer PCBs addresses return paths in addition to meeting power integrity requirements.

The clock net is differential in both DDR2 and DDR3. In DDR2, DQS can be either single ended or differential although it is usually implemented as differential at higher data rates. The switched delay of a differential trace is less than that of a single ended trace of identical length. Where timing computations indicate the need, the clock and DQS traces may need to be made longer than the corresponding ADDR/CMD/CNTRL nets and DATA nets. This would ensure that the clock and DQS transitions are centered on the associated ADDR/CMD/CNTRL nets and DQ nets.

Since DQ and DM nets run at the maximum speed, it is desirable that all of these nets in any byte lane be routed identically, preferably without vias. Differential nets are less sensitive to discontinuities and where layer jumping is needed, the DQS and CLOCK nets should be considered first.

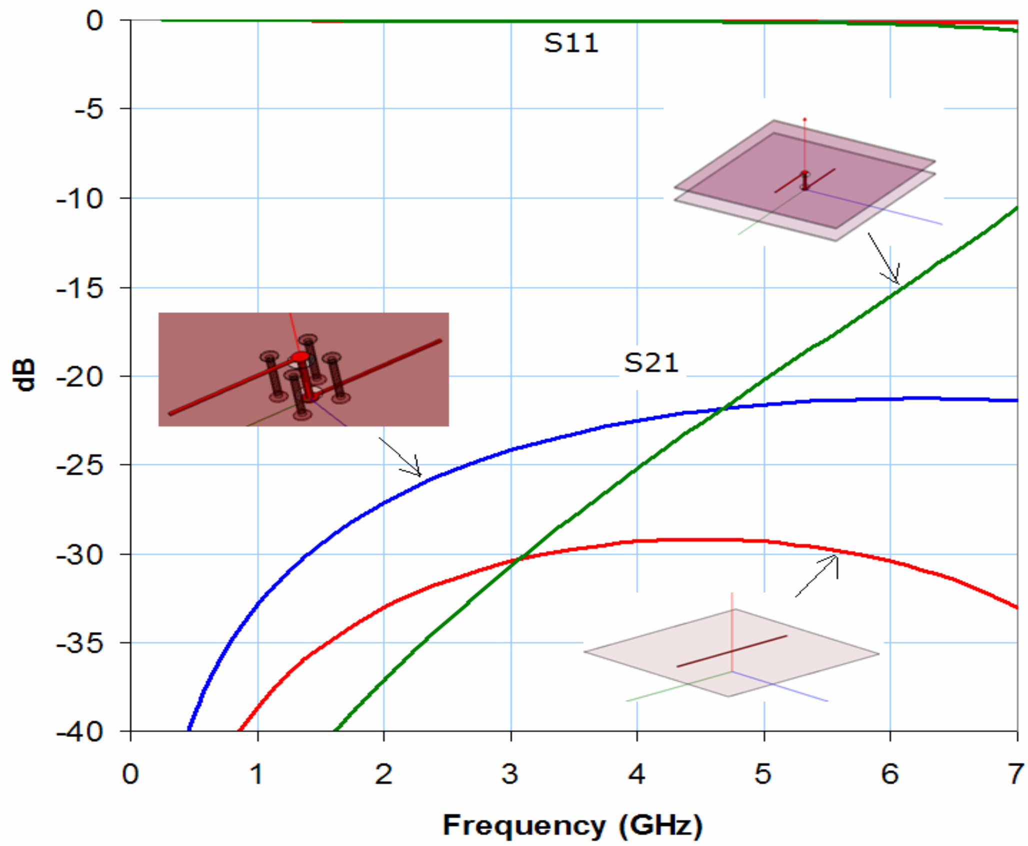


Figure 6: s-parameters of interconnects with vias (60 Ohm normalization)

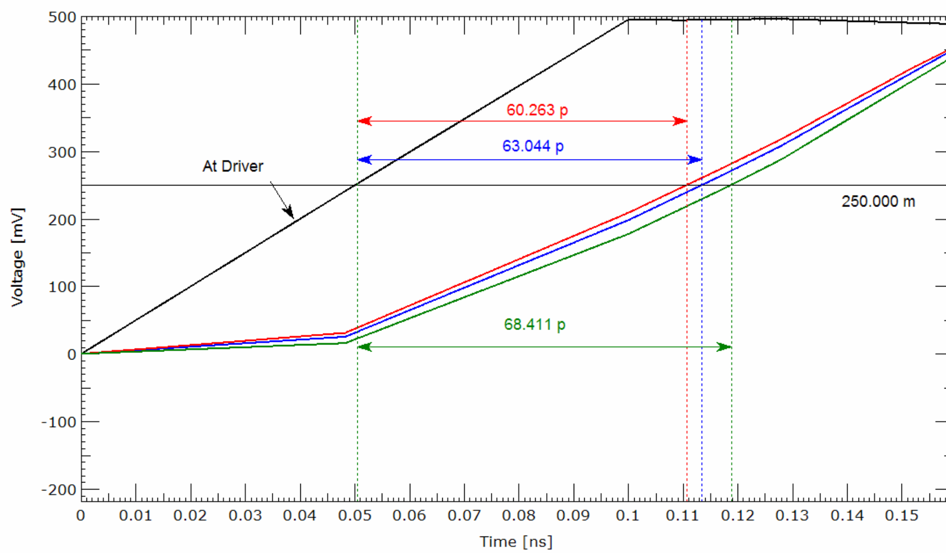


Figure 7: Driver and Receiver waveforms for the 3 cases of Figure 6. (Plot colors correspond)

5. Crosstalk

Cross talk contributes to delay uncertainty being significant for microstrip traces. This is generally reduced by increasing the spacing between adjacent traces for long parallel runs. This has the drawback of increasing the total trace length and therefore a reasonable value must be chosen. Typically the spacing should be greater than twice the trace distance to ground. Again, ground vias play an important role. Near and far end coupling levels are illustrated in Figure 8. Use of multiple ground vias reduces coupling levels by 7 dB. To derive the interconnect budget, a simulation of a victim trace with two aggressors on both sides is adequate. Using a periodic excitation on all nets will yield the cross talk induced jitter. Using a pseudo random excitation on all nets will show the effect of both cross talk as well as data dependencies. Time domain results are not shown here, but it is easily done by setting up a 5 coupled line circuit in SigXP with the spacing between traces set up for sweeping. Reasonable spacing values that keep the jitter in the waveform due to both cross talk as well as pattern dependence at an acceptable level are chosen.

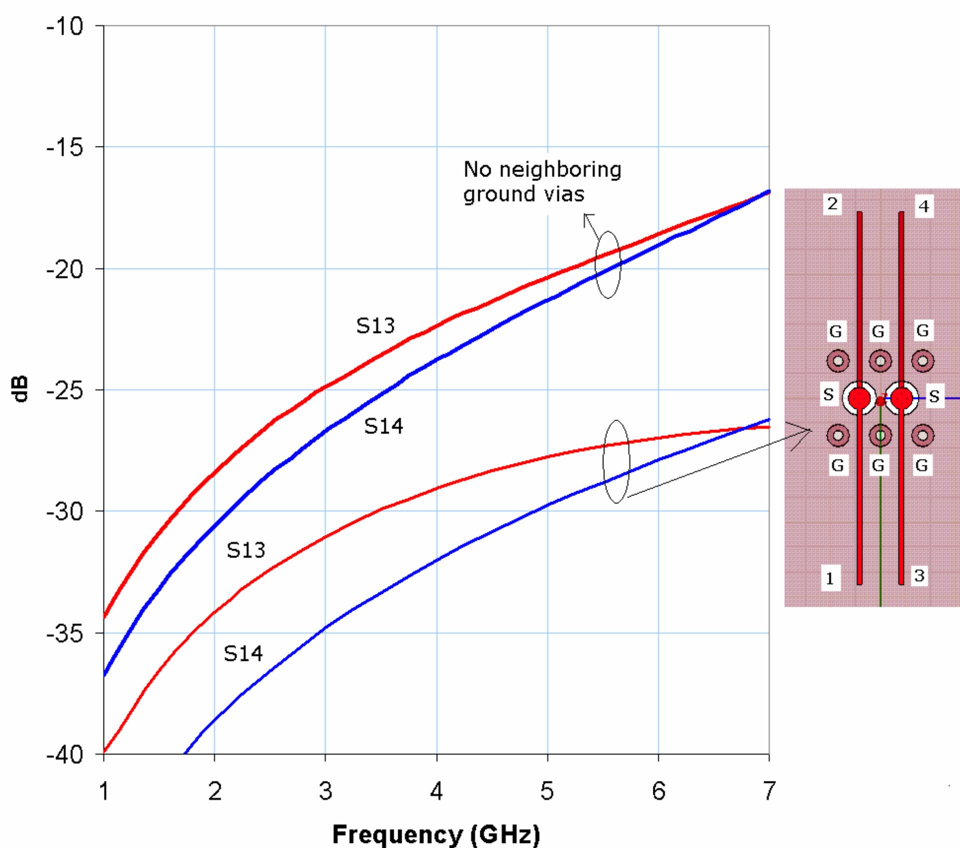


Figure 8: s-parameters of coupled traces (60 Ohm normalization)

6. Power Integrity

Power Integrity here refers to meeting the Power supply tolerance requirement under a maximum switching condition. Failure to address this requirement properly leads to a number of problems, such as increased clock jitter, increased data dependent jitter, and increased cross talk all of which eventually reduce timing margins.

The theory for decoupling has been very well understood and usually starts with the definition of a "target impedance" as [4]

$$Z_{target} = \frac{Voltage\ tolerance}{Transient\ Current} \tag{1}$$

An important requirement here is knowledge of the transient current under worst case switching condition. A second important requirement is the frequency range. This is the range of frequencies over which the decoupling network must ensure that its impedance value is equal to or below the required target impedance. On a printed circuit board, capacitance created by the Power-Ground sandwich and the decoupling capacitors needs to handle a minimum frequency of ~100 kHz up to a maximum frequency of ~100-200 MHz. Frequencies below 100 kHz are easily addressed by the bulk capacitance of the voltage regulator module. Frequencies above 200 MHz should be addressed by the on-die and in some cases on-package decoupling capacitance. Due to the finite inductance of the package, there is no need to provide decoupling on the PCB to handle frequencies greater than 200 MHz. The actual computation of power integrity can be very complex involving IC package details, simultaneously switched signals and the PCB power distribution network. For PCB design, the use of the target impedance approach to decoupling design is simpler and provides a practical solution with very little computational effort.

The three power rails of concern are the VDD, VTT and Vref. The tolerance requirements on the VDD rail is ~ 5% and the transient current is determined as the difference between Idd7 and Idd2 as specified by JEDEC [1,4]. This is accomplished by using plane layers for power distribution and a modest number of decoupling capacitors. It is preferable to use decoupling capacitors of 10 different values distributed in the range of 10 nF to 10 uF. Further, the capacitor pad mounting structure should be designed for reduced mounted inductance.

The Vref rail has a tighter tolerance, but it draws very little current. Its target impedance is easily met using narrow traces and one or two decoupling capacitors. It is important however that the capacitors be located very close to the device pins.

The VTT rail proves to be challenging because it not only has a tighter tolerance, but it also draws a transient current close to that of the VDD rail. The transient current is easily calculated as described in reference [5]. Again, the target impedance requirement can be met using an increased number of decoupling capacitors.

On a 4 layer PCB, the planes are too far apart and consequently the advantage of inter-plane capacitance is lost. The number of decoupling capacitors needs to be increased and higher frequency capacitors with values less than 10 nF may be needed. These computations are easily done using ALLEGRO SI Power Integrity option.

7. Timing

Timing computation is carried out as described in reference [6]. A table needs to be setup for the following eight cases:

1. Write Setup analysis DQ vs. DQS
2. Write Hold analysis DQ vs. DQS
3. Read Setup analysis DQ vs. DQS
4. Read Hold analysis DQ vs. DQS
5. Write Setup analysis DQS vs. CLK
6. Write Hold analysis DQS vs. CLK
7. Write Setup analysis ADDR/CMD/CNTRL vs. CLK
8. Write Hold analysis ADDR/CMD/CNTRL vs. CLK

An example is shown for the case of Write setup analysis in Table 2. Actual numbers have been omitted as they are not precisely known yet for DDR3. These numbers are obtained from data sheets of Controller and memory manufacturers. The numbers in the interconnect section are determined by SI simulations. All the eight cases need to be analyzed for DDR2. For DDR3, 5 and 6 are not needed due to its write leveling feature. In the PCB implementation, length match tolerances must ensure that the total margin is positive.

Element	Skew Component	Setup	Units	Comments
Controller	a.)DQ vs. DQS skew at transmitter output		ps	From controller design data
	b.) Data / Strobe PLL jitter		ps	Used if not included in transmitter skew
Total Controller	a + b		ps	
SDRAM (or DIMM)	Setup requirement (tDSb @ Vih/Vil level)		ps	From SDRAM datasheet; this number is to be adjusted based on DQ and DQS slew rates
	DQ slew rate		V/ns	Measured as per JEDEC specification from SI simulation results
	DQS slew rate		V/ns	Measured as per JEDEC specification from SI simulation results
Total SDRAM setup requirement	tDSb + slew rate adjustment		ps	Includes slew rate adjustment
Interconnect	a.) Data Xtalk		ps	2 aggressors (one each side of the victim); victim – repetitive; aggressor- PRBS
	b.) DQS Xtalk		ps	2 aggressors (one each side of the victim); victim – repetitive; aggressor- PRBS
	c.) Length matching tolerance		ps	Extracted from SI simulation results longest data net, worst case PVT corner
	d.) Characteristic impedance mismatch		ps	can be omitted if routing of DQ and corresponding DQS signals are done on same layer
Total Interconnect	Interconnect skew (a + b + c + d)		ps	
Min. Total Setup Budget	0.24*tck		ps	From SDRAM datasheet (includes clock duty cycle variation)
Setup margin	Min. Total Setup Budget – (Total Controller + Total SDRAM + Total Interconnect)		ps	Must be positive

Table 2: Illustration of DDR3 Write Setup timing analysis summary for DQ vs. DQS

8. PCB Layout

Implementation on a PCB involves a number of tradeoffs to meet SI requirements. Often, the question is how far does one need to go? PCB layout tasks are facilitated using the following approach:

1. Set up topology and constraints in ALLEGRO Constraint Manager.
2. Design Controller BGA breakout. A controller pin arrangement with ADDR/CMD/CNTRL pins in the middle and DQ/DQS/DM byte lanes on either side is best suited. Within these groups, individual pins may need to be swapped to ensure routing with minimum cross-over.
3. Attempt routing with reduced stub length and a minimum trace spacing as obtained from cross talk simulation. Often, most stubs can be eliminated but it will not be possible for all the pins. One may try two traces between BGA pads of the memory devices. This would require narrow PCB traces which can increase manufacturing cost. Yet, it will not be possible for all signals unless micro via and via-in-pad technology is used. Complete routing with coarse length matching tolerances.
4. Place Vref decoupling capacitors close to the Vref pins. Vtt decoupling can be placed at the far end of the last SDRAM and will not come in the way of routing. VDD decoupling can be placed close to devices where possible without blocking routing channels. The smaller valued capacitors should be placed closer to the devices. With a proper decoupling design, it will not be necessary to cram all capacitors close to the devices. All decoupling capacitors should use a fan out for the footprint designed for reduced inductance. This is typically two short wide traces perpendicular to the capacitor length. This can be automated by using a user defined capacitor footprint that can be attached to all the decoupling capacitors in the schematic.
5. Implement fine length matching and insert multiple ground vias where signal traces jump layers. It is better to use the delay matching option in ALLEGRO and one must include z-axis delay. Typically, P and N nets of differential pairs should be matched with a tolerance of +/- 2ps and the tolerance for all other matched nets can be +/- 10 ps or more based on the timing margin computation.

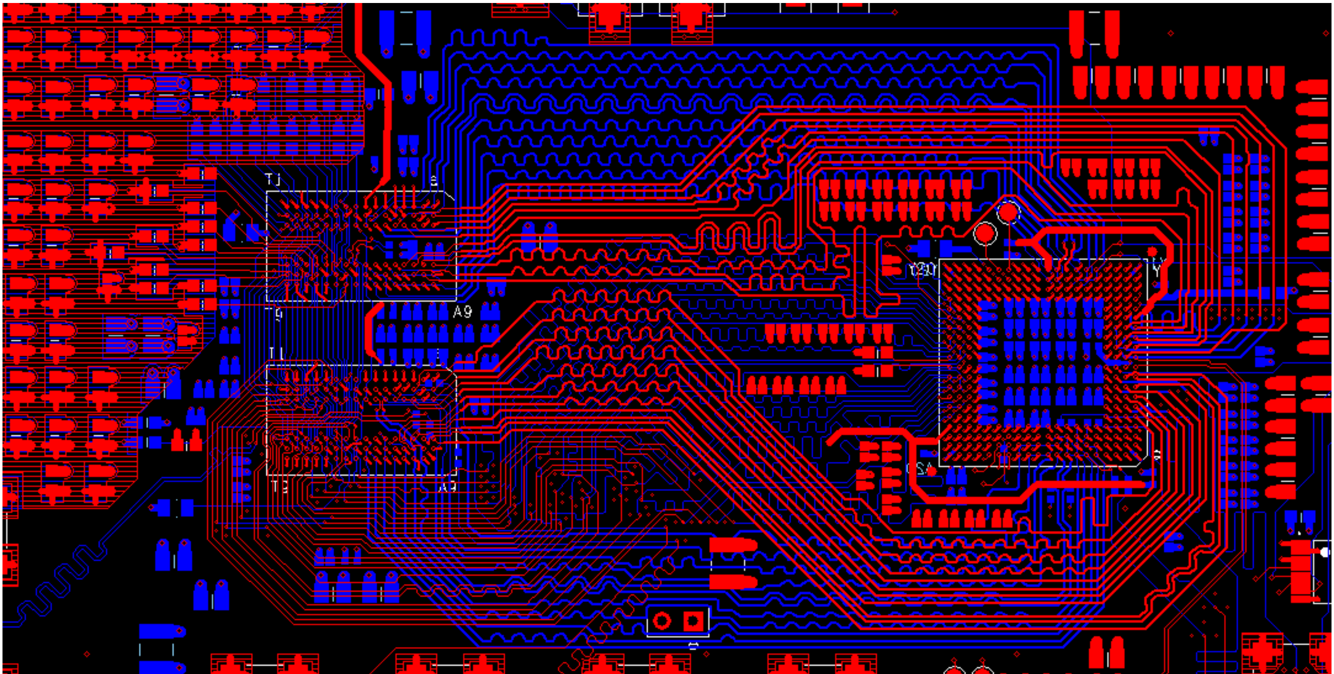
9. DIMM

Considerations described above apply to the case of PCBs containing one or more DIMMs. The only exception is that the decoupling requirement for the memories can be relaxed as it is already accounted for on the DIMM PCB. SI analysis of registered DIMMs is also much simpler where the DIMM is treated as a single load. While the routing topology for ADDR/CMD/CNTRL nets is usually a daisy chain with reduced stubs, tree topologies can also be used for registered DIMMs. Analysis of un-buffered DIMMs can become tedious as the timing requirement at all the SDRAMs must be analyzed. DIMM routing on 4-layer PCBs is relatively simpler compared to the case of SDRAMs.

10. Examples

The detail described above has been used in the implementation of a DDR2 PCB, a DDR3 PCB and a DDR3 – DIMM PCB. The controller is from MOSAID [7] which is designed to provide both DDR2 as well as DDR3 functionality. For the SI simulations, IBIS models have been used. Models for the memories are from MICRON Technology, Inc [8]. The IBIS models for the DDR3 SDRAMs were available at 1333 Mbps speed. These were used at 1600 Mbps. For the unbuffered DDR3 DIMM (MT_DDR3_0542cc) EBD models from Micron Technology were used. All waveforms are for the typical case and are computed at the SDRAM die. The 6 layer PCB stackup of Figure 2 is used with routing on TOP and BOTTOM layers only. The memory consists of 2 SDRAMs

routed as a daisy chain. In the case of the DIMM, a single unbufferred DIMM is used. Snapshots of TOP/BOTTOM layer routing and Signal Integrity waveforms are shown in Figures. 9-11.



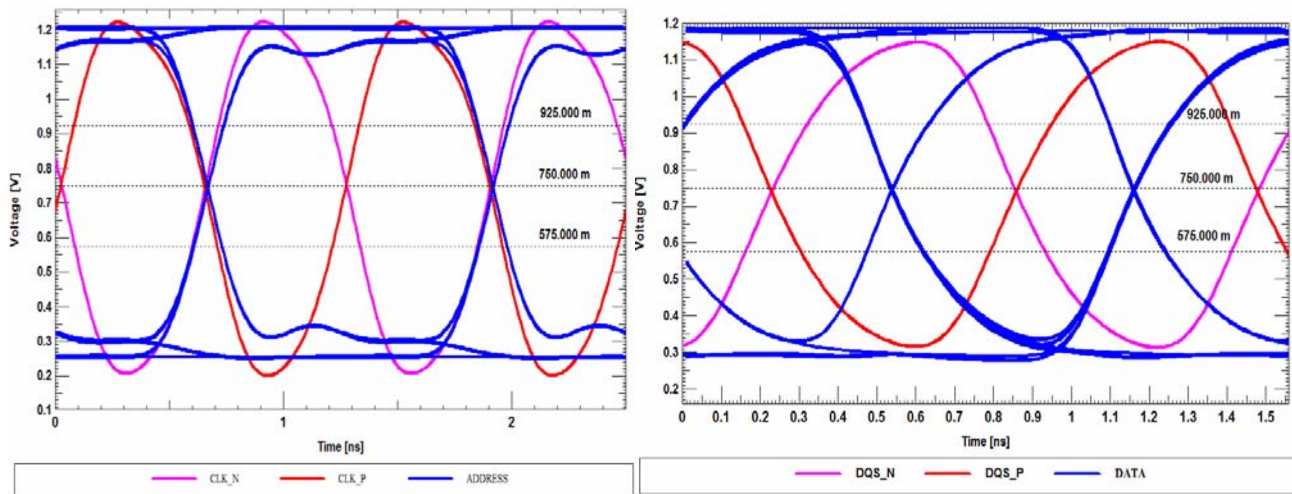
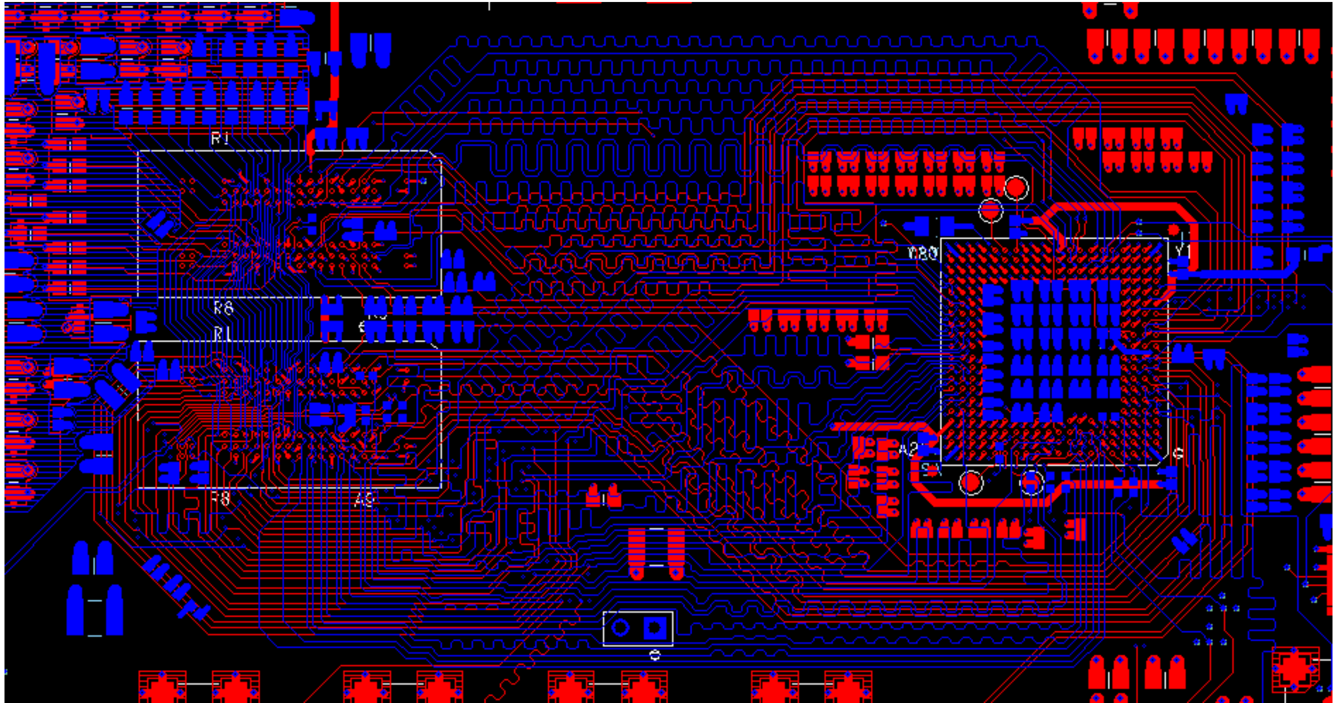


Figure 9: Illustration of TOP and BOTTOM layers of a DDR3 PCB with computed waveforms at the farthest SDRAM. Waveform on left is an ADDRESS net compared to that of the CLOCK net. Waveform on the right is a DATA net compared to that of a DQS net. Clock frequency = 800 MHz and data rate is 1600 Mbps.



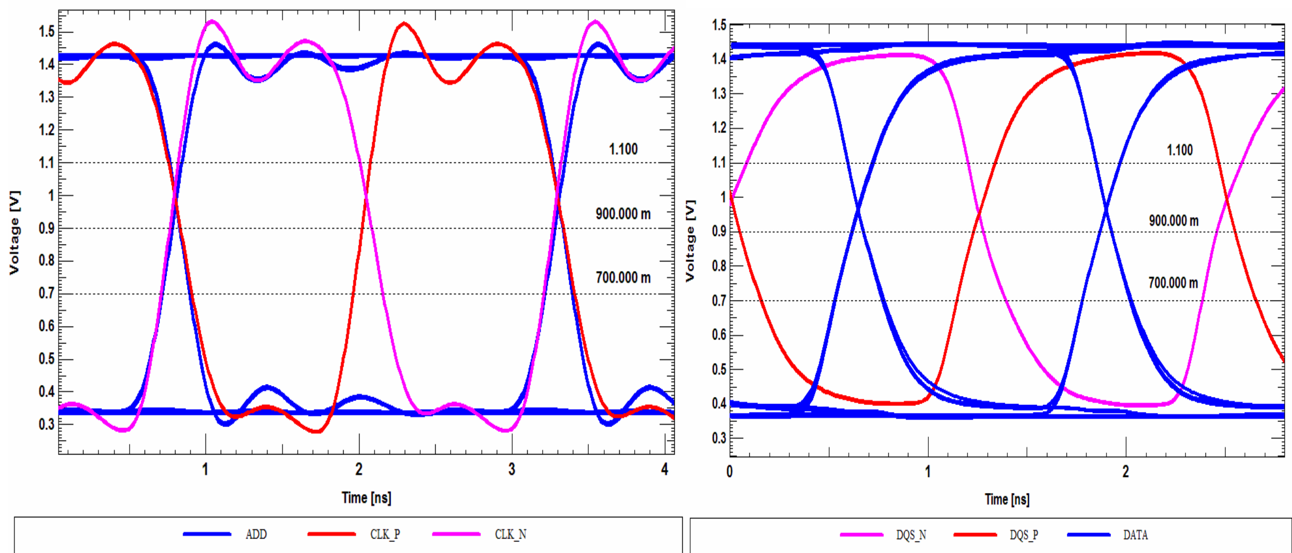
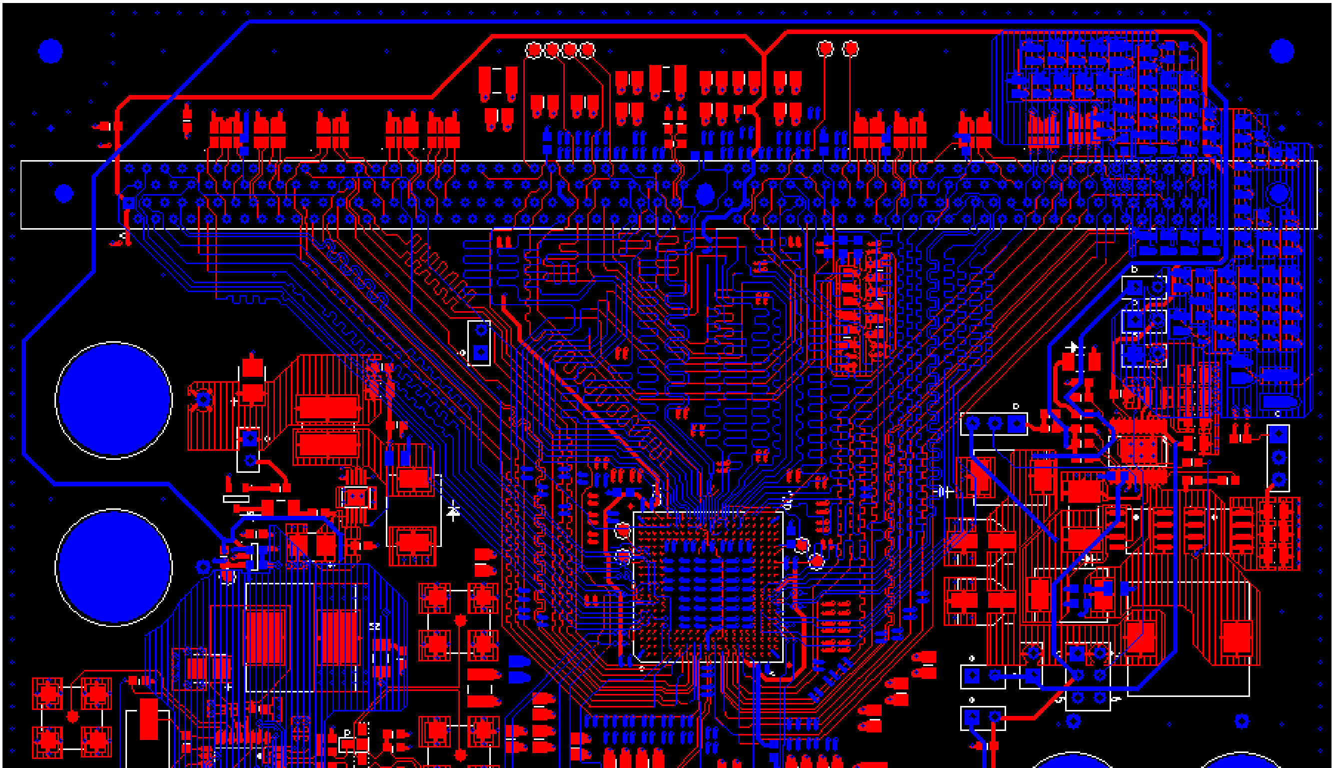


Figure 10: Illustration of TOP and BOTTOM layers of a DDR2 PCB with computed waveforms at the farthest SDRAM. Waveform on left is an ADDRESS net compared to that of the CLOCK net. Waveform on the right is a DATA net compared to that of a DQS net. Clock frequency = 400 MHz and data rate is 800 Mbps.



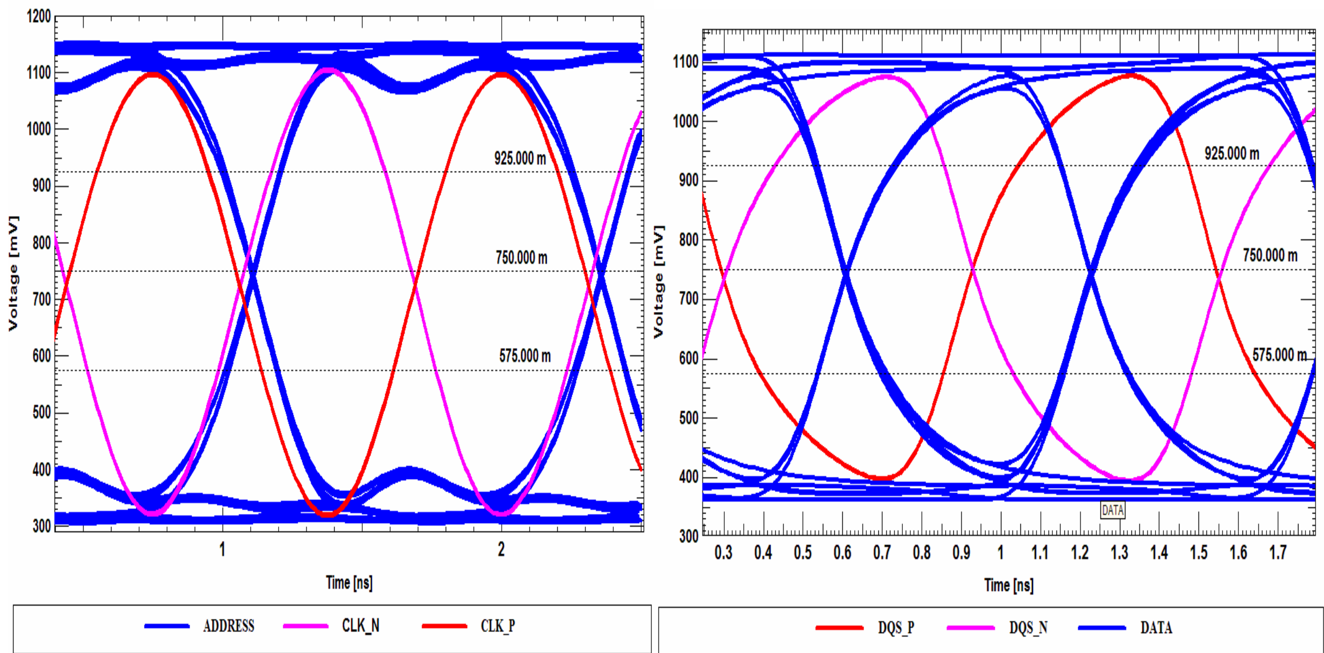


Figure 11: Illustration of TOP and BOTTOM layers of a DDR3 – DIMM PCB with computed waveforms at the 8th (last) SDRAM on DIMM. Waveform on left is an ADDRESS net compared to that of the CLOCK net. Waveform on the right is a DATA net compared to that of a DQS net.

Lastly, Figure 12 shows a comparison of computed and measured DATA eye patterns of an 800 Mbps DDR2. In all cases waveform integrity can be seen to be excellent.

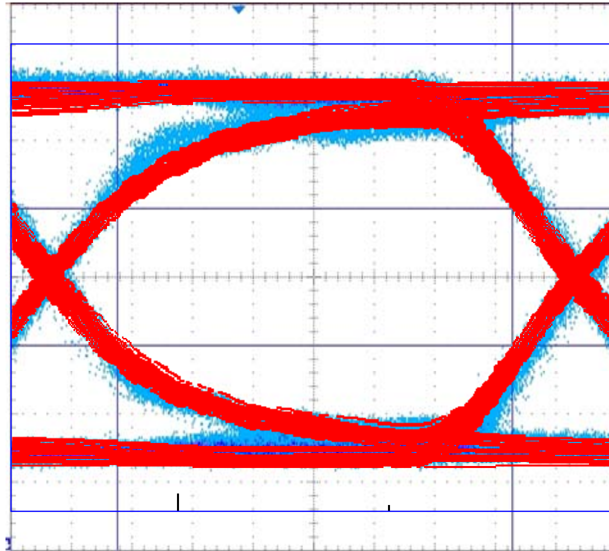


Figure 12: Computed (Red) and Measured (blue) waveforms of a data net of an 800 Mbps DDR2 PCB.

11. Conclusion

In this paper, all aspects related to SI, and PI of DDR2 and DDR3 implementation have been described. Use of Constraint Manager in ALLEGRO™ makes implementation easy. While a four layer PCB implementation of 800 Mbps DDR2 and DDR3 appears to be feasible, DDR3-1600 Mbps will prove to be challenging. It will become clearer as the memory devices become available and one has a good handle on timing numbers.

References

[1] DDR2 SDRAM Specification, JEDEC JESD79-2B, January 2005.
[2] DDR3 SDRAM Standard, JEDEC JESD79-3, June 2007.
[3] Syed Bokhari, “Delay matching on Printed Circuit Boards”, Proceedings of the CDNLIVE 2006, San Jose.
[4] Larry D Smith, and Jeffrey Lee, “Power Distribution System for JEDEC DDR2 memory DIMM, Proc. IEEE EPEP conference, Princeton, N.J., pp. 121-124, October 2003.
[5] Hardware and layout design considerations for DDR2 SDRAM Memory Interfaces, Freescale semiconductor Application Note, Doc. No. AN2910, Rev. 2, 03/2007.
[6] DDR2 design guide for 2 DIMM systems, Technical Note, Micron Technology Inc. TN-47-01, 2003.
[7] http://www.mosaid.com/corporate/products-services/ip/SDRAM_Controller_whitepaper_Oct_2006.pdf
[8] http://www.micron.com/products/dram/ddr2/partlist.aspx?speed=DDR2-800
[9] http://www.micron.com/products/dram/ddr3/partlist.aspx?speed=DDR3-1066

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

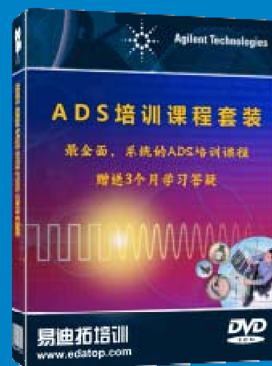
易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装



该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html

HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com