Technical Note

Hardware Tips for Point-to-Point System Design: Termination, Layout, and Routing

Introduction

Designers can benefit from a set of proven techniques for termination, layout, and routing for SDRAM and DDR devices used in point-to-point interfaces. Derived from electronic theory and Micron design experience, the guidelines in this technical note are meant to be used for signal integrity (SI) optimization in point-to-point systems.

This technical note is the practical companion to Micron technical note TN-46-06, "Termination for Point-to-Point Systems," which discusses transmission line theory and the effects of series resistance. Micron recommends that designers using SDRAM or DDR components in a point-to-point system consult TN-46-06 regarding theory and use this technical note as a primary memory-subsystem design recommendation for printed circuit boards (PCBs).

The guidelines and examples of Micron design requirements in this technical note should not be considered the only acceptable methods, nor are they applicable to all point-to-point designs. Micron recommends designers also analyze other factors that can affect SI, such as system data width, number of non-DRAM loads, controller I/O drive strength, and AC timing.

Definitions

In this technical note, V_{SS} refers to digital ground, $V_{SS}Q$ refers to DQ and signal ground; the two are equivalent unless otherwise noted. V_{DD} is digital power for the device core and $V_{DD}Q$ is power for the DQ and I/O signals; they also are equivalent unless otherwise noted.

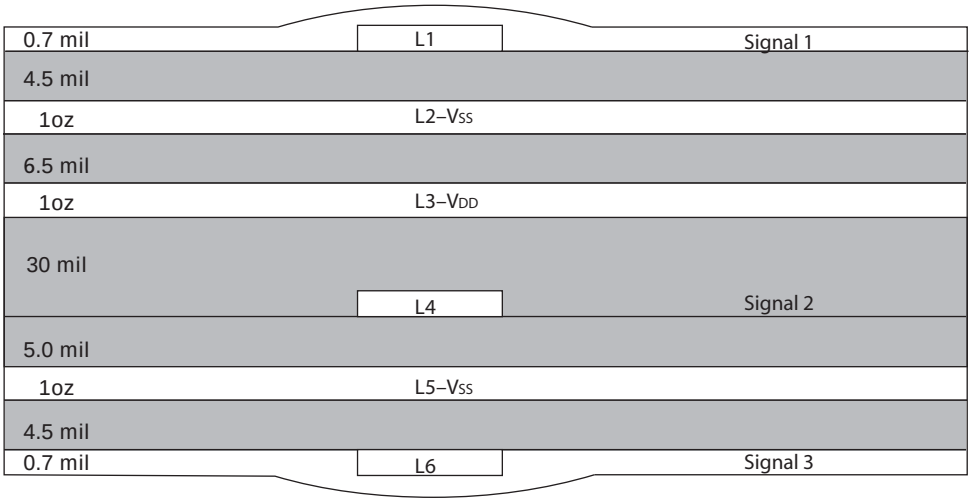
Recommended Six-Layer PCB Stackups

A well-designed PCB stackup is critical in eliminating digital switching noise. The ground plane must provide a low-impedance (Low-Z) return path for digital circuits.

Micron recommends a PCB design with a minimum of six layers: layers 1 (top) and 6 (bottom) for signals; layers 2, 3, and 5 for ground/power; and layer 4 for as ground/power or for signals, as shown in Figures 1 and 2 on page 2.

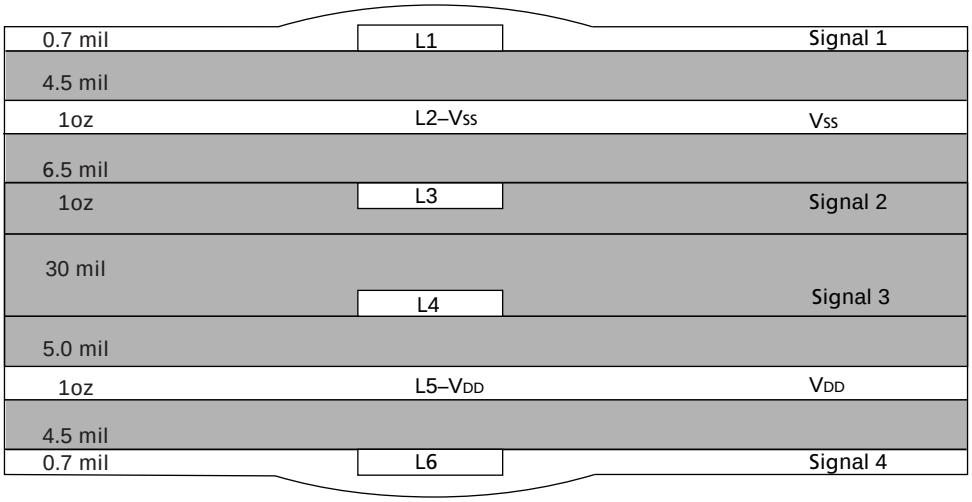
Figure 1 shows a three-signal-layer stackup, while Figure 2 shows a four-signal-layer stackup. Poor SI and other problems render three- or four-layer PCBs unusable except in very limited cases.

Figure 1: Micron-Recommended Three-Signal-Layer PCB Stackup



Notes:
1. Layer L1 has controlled impedance to L2 (V_{SS}).
2. Layer L4 has controlled impedance to L5 (V_{SS}).
3. Layer L6 has controlled impedance to L5 (V_{SS}).
4. Impedance on L1, L4, and L6 is 50–60Ω for a trace 4 mil wide.
5. To balance copper weight in signal areas and reduce PCB wrapping, copper-flood the L4 layer where there are no signals.
6. Micron recommends 10 mil drill sizes.
7. Micron recommends 0.35mm pads.
8. Solder mask thickness varies, but averages 7 mil.

Figure 2: Micron-Recommended Four-Signal-Layer PCB Stackup



Notes:
1. Recommended total PCB thickness = 1.57mm (62 mil).
2. Typical dielectric thicknesses; “prepreg” thickness can vary from 4–6 mil.

TSOP Packages

For TSOP-packaged SDRAM and DDR components, typical routing requires two internal signal layers, two surface signal layers, and two other layers (V_{DD} and V_{SS}) as solid reference planes. Memory devices have V_{DD} and $V_{DD}Q$ pins, which are both normally tied to the PCB V_{DD} plane. Likewise, component V_{SS} and $V_{SS}Q$ pins are tied to the PCB Vss plane. Each plane provides a low-inductance path to the memory devices to deliver $V_{SS}Q$. Dedicating a single plane for both power and ground does not enable strong signal referencing.

Most DDR designs reference DQ, strobe, and clock signals to V_{SS}, while address, command, and control signals are referenced to V_{DD}. DQ signals are generally routed on the outer layers of the PCB.

FBGA Packages

Systems using FBGA-packaged DDR components typically split PCB planes between V_{DD} and V_{SS} such that DQ, DQS, DM, and clock signals maintain a V_{SS} reference, while address, command, and control signals maintain a V_{DD} reference. These split planes require additional effort to ensure good power delivery to the memory device; the usual solution is to flood the outer PCB layers—where no signal is present—with V_{DD}.

Standard characteristic impedance (Z_O) of 50–60Ω is recommended for all traces. The 60Ω level also provides a good match to the sum of the output impedance of the controller/FPGA driver and any series resistors used; 60Ω ±6Ω (10 percent tolerance) is Micron's module and reference design target. Designers are advised to specify Z_O, enabling board manufacturers to adjust dielectric thickness and line width to achieve the specification.

Minimizing the length of the signal return path (loop area) reduces transient currents and electromagnetic interference (EMI). Micron recommends placing V_{SS} and V_{DD} layers adjacent to each other on PCB layers 2 and 3 of a six-layer stackup, and placing the return signal on the plane immediately under the trace (such as plane 2 and plane 5) for the smallest loop area.

PCB Dielectric Material

The dielectric constant of PCB materials for most memory applications is 3.6 to 4.5, varying slightly with frequency, temperature, material, and resin-to-glass ratio. FR-4, one commonly used dielectric material, averages 4.2 with signaling at 100 MHz. The FR-4 is copper-clad laminate that offers exceptional dimensional stability, dielectric thickness control, and high-quality manufacturing repeatability.

Micron recommends FR-4 for PCBs, because of its low cost, low moisture absorption, and low electrical conductivity in module fabrication.

DDR V_{REF}

DDR device receivers compare the difference between a steady reference voltage (V_{REF}) and the signal received. Because only the differential voltage (ΔV) is processed internally, it is important that the reference voltage matches data sheet limits and remains as close to constant as possible, ±1 percent of nominal ($V_{DD}Q/2$) after power up.

Nominal VREF is expected to track variations in the DC voltage; thus, with VDD specified as 2.5V nominal, VREF's range is 1.237–1.262V. However, VDD is specified as 2.3–2.7V for 2.5V nominal and 2.5–2.7V for 2.6V nominal (DDR400), as measured at the nearest VREF bypass capacitor. VREF also must adhere precisely to all other data sheet requirements for proper device operation.

Because VREF is an input to a differential-pair, common-source amplifier, there is no significant current draw. Any standard regulator integrated circuit (IC) with generation capability of ≥5mA is adequate. To minimize the noise effects on VREF, Micron recommends a decoupling capacitor at each DDR device input.

Accuracy Requirements

Meeting VREF's stringent accuracy requirements requires careful design consideration. In addition to tracking changes in VDDQ, VREF requires both minimal ripple and minimal noise. Peak-to-peak noise on VREF must not exceed ±2 percent of the DC value; ripple on VREF must maintain a maximum of ±25mV for DC error and a maximum of ±25mV for AC noise.

Micron recommends:

- Route VREF at least 2cm away from other signals to minimize coupling.
- Design traces as short and wide as possible between the voltage source and VREF.
- Place a single low-inductance (0.1µF) bypass capacitor as close to the device VREF pin as possible.

VREF Generation

VREF generation is a straightforward process. Micron recommends the following VREF generating solutions:

- For light loads (fewer than four DDR components), connect VDDQ to VSSQ through a simple resistor divider made up of two equivalent 1kΩ resistors, each with ±1 percent tolerance, as shown in Figure 3 on page 5. This enables VREF to track any changes in VDDQ. Resistor values should be calculated so that any error caused by component leakage currents (typically <3µA) is small relative to other error sources.
- For heavier loads (four or more components), design in a regulator IC configured to output 1.25V on VREF. Micron has successfully used various switching regulators with integrated MOSFETs in DDR reference designs.

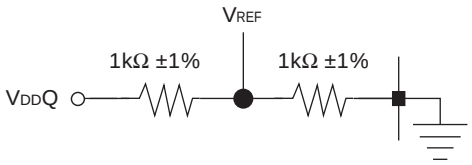
  The regulator can absorb or generate up to 3A for the VTT circuit and has a separate, linear, regulated output for VREF at 3mA. With an active driver for VREF, leakage currents can be ignored.

Appropriate DDR termination regulator vendors offering products with VTT, VREF, VDDQ, and VDD outputs include:

- Fairchild—ML6554 and FAN1655
- Linear Technology—LTC3413 and LTC3831
- National Semiconductor—LP2995 and LP2996
- Phillips—NE57810 and NE47814

Micron does not endorse one vendor over another and recommends engineers choosing one of these regulator ICs to verify performance through prototype testing.

Figure 3: VREF Generation with Resistor Divider



Miscellaneous VREF Design Guidelines

Micron recommends the following:

- Route VREF with a 20–25 mil minimum trace to reduce inductance
- Maintain at least a 15–25 mil clearance from VREF to other adjacent traces
- Place a 0. 1µF capacitor between VREF and VDDQ
- Place a 0.1µF capacitor between VREF and VSSQ
- Decouple at each device or connector to minimize noise at each component

DDR VTT

When designing with DDR memory, special power considerations must be addressed, because DDR requires three highly accurate voltages: VDDQ, VTT, and VREF. VTT is a memory bus termination voltage equaling VDDQ/2.

To place VTT in perspective, it is necessary to understand how system voltages are related. DDR devices generally have two PCB power supplies (rails): VDD, supplying both device VDD and VDDQ, and VSS, for device VSS and VSSQ. As stated in “Definitions” on page 1, VDD is the main high-current voltage used for both the DDR core and component I/O lines. VREF is a low-power reference voltage equal to VDD/2. DDR devices compare internal signals to VREF. Other voltage relationships are VDD = VDDQ, and VTT = VREF = VDD/2.

VTT transient current can be as high as ±3.5A during heavy activity on the DQ and address buses. This transient current averages 0A, but can be somewhat random in nature, depending upon address/data patterns.

Some system designs can operate without requiring VTT. The approximate system boundaries enabling VTT exclusion are:

- Two or fewer DDR components in the system
- Moderate current draw
- Trace length <2in (5cm)
- SI and drive strengths within data sheet limits (determined through simulation)

VTT is recommended for maximum voltage accuracy and current delivery and requires a regulator with ±3 percent tolerance; VTT is not applied directly to the device, but connected, as shown in Figure 4 on page 6.

VTT regulators are available as standalone devices and as devices with integrated VREF generation. Suitable VTT regulators (with or without VREF generation) are available from Intersil, Philips Semiconductors, California Micro Devices, Fairchild, Champion Microelectronics, National, and Texas Instruments.

Miscellaneous VTT Design Guidelines

Figure 4 shows recommended VTT layout for a system with multiple DDR components.

When a regulator has been selected, Micron recommends the following:

- VTT terminates command and address signals to VDD/2, using a parallel resistor (RT).
- VTT terminates DQ and DQS on PCB; DQS charge = VDDQ/2 when data transfers are not occurring,
- VTT does not terminate any DDR clock pairs. CK and CK# termination is a parallel 100–121Ω resistor between the two lines; Micron has found that only differential termination on CK and CK# produces optimal SI.
- VTT and VREF islands must be separated by a minimum of 150 mil if placed on the same PCB layer. Placing the islands on different layers is preferred.
- VTT islands must be placed at the end of the memory channel, as close as possible to the last device.
- VTT islands require at least two additional decoupling capacitors (4–7µF) and two bulk capacitors (100µF) at each end.
- VTT island surface trace = 150 mil MIN; 250 mil preferred.
- VTT power at initialization must be applied after VDDQ to avoid device latch-up; VTT power is nominally coincident with VREF power.
- Balanced-T routing (shown in Figure 5 on page 7) is recommended where a signal must go to more than one point; place the memory controller/ASIC and series resistor (RS) at the bottom of the T, the VTT terminator at the top of the T, and DDR devices at the left and right arms of the T.

Figure 4: VTT Island and Regulator PCB Layout for Multiple DDR Components

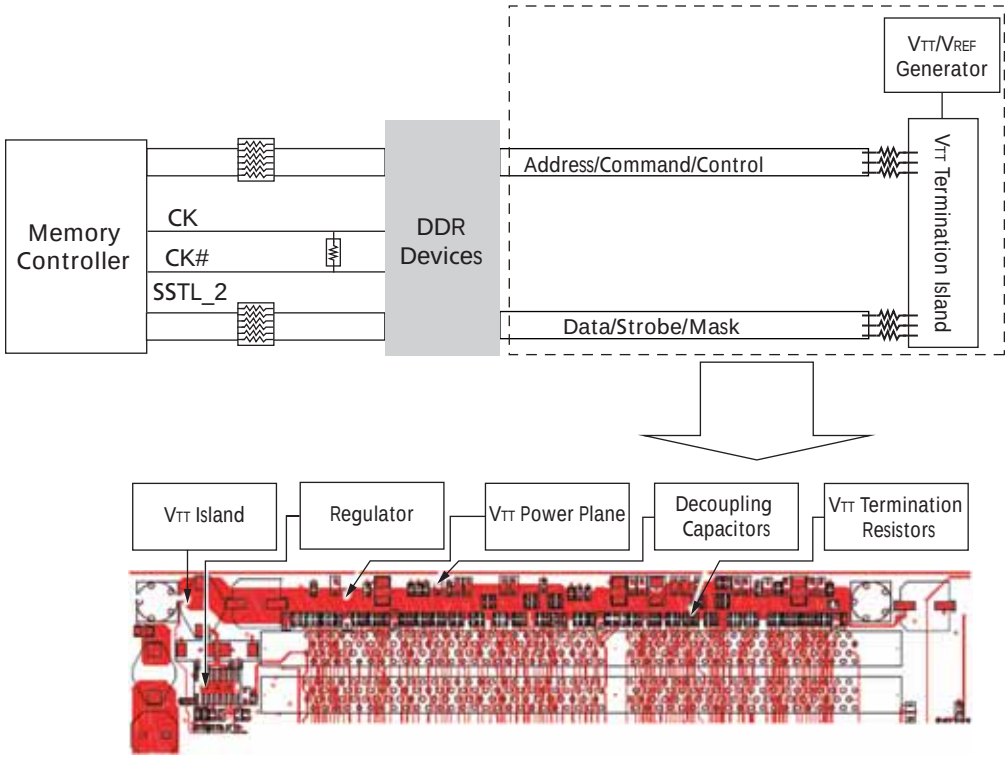
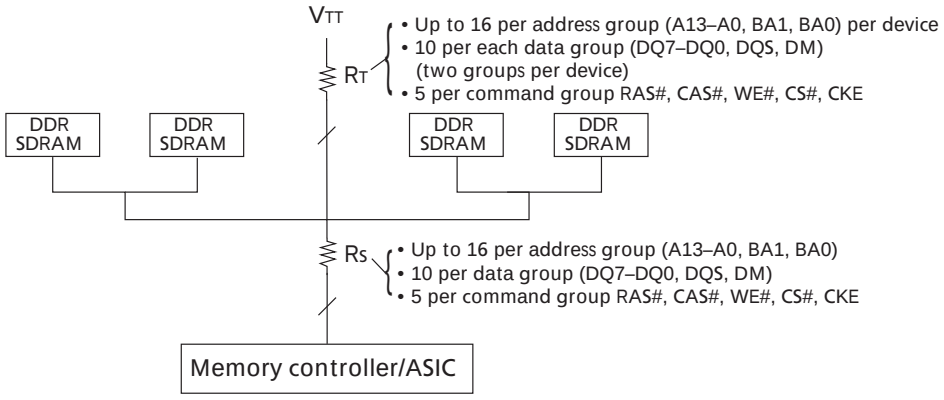


Figure 5: Balanced T Routing Using Four DDR Devices and SSTL-2 Type 1 Topology



Notes:
1. Position the V_{TT} source as close as possible to the R_T resistors.
2. All trace lengths are equal.
3. For unidirectional address, command, and control signals, optimal R_S placement will be determined through simulation. Generally, this is near the middle of the bus or close to the controller, but it is not required to be in exactly the same place for DQ and command/address groups if skew specifications are met.
4. For bidirectional data signals with parallel topology, place the R_S close to the source processor/FPGA.
5. Values for R_S and R_T are system-dependent; derive through simulation and prototype testing.
6. The R_T recommended range is 22–27Ω with 56Ω MAX. Use simulation results to optimize R_T and reiterate until simulations have V_{IH} and V_{IL} margins before testing prototypes.
7. For systems with four or more DDR devices with trace length >2in (5cm), parallel terminating resistors are strongly advised (also see Figure 8 on page 11).

Layout: Trace Widths, Interpair Spacing, and Intrapair Spacing

There are two types of trace spacing that play a role in system SI: interpair spacing and intrapair spacing.

Interpair spacing (S1) is the distance between two adjacent traces within a related set of signals with similar or equivalent functionality. The control signals group, clocks, address bus, data bus, and data/strobes are all signal sets. The data bus is sometimes broken down into data bytes, which are sets of eight signals, and the associated strobe and mask signal.

Intrapair spacing (S2) is the distance between the two outermost signals of different sets. For example, if the control signal set is routed together and adjacent to the address signal set, intrapair spacing is the distance between the two individual signals from control and address sets that are closest together.

Figure 6 illustrates the difference between S1, S2, and trace width (S3) using the control and address groups.

In general, Micron recommends an average S2 of 6 mil, with 4 mil minimum. Going lower than recommended minimum for S1 or S2 can increase cost (specific guidelines are shown in Table 1 on page 8). If all signals are routed at exactly 4 mil spacing for their full length, crosstalk is likely to disrupt SI, but if spacing limits are not met for short segments, SI is not likely to suffer.

Crosstalk is a function of trace spacing and slew rate; for systems with slew rates <1 V/ns, trace spacing can be closer. Slower systems generally have more timing budget, which allows more crosstalk without affecting SI.

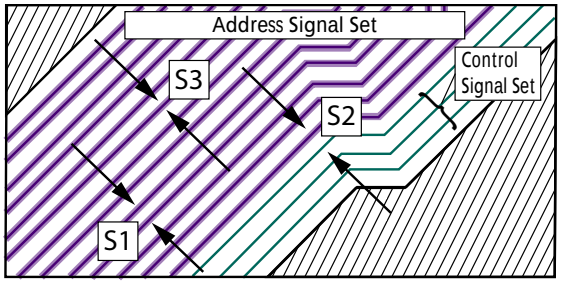
Trace Width (S3) Design Guidelines

Recommended S3 for functional signal sets:

- DQ lines = 4 mil minimum, 6 mil nominal
- DQS lines = 4 mil minimum, 6 mil nominal
- Address lines = 4 mil minimum, 6 mil nominal
- Command/control lines = 4 mil minimum, 6 mil nominal
- Clock lines = 4 mil minimum, 6–10 mil nominal

V_{DD}, $V_{DD}Q$, V_{SS}, and $V_{SS}Q$ must be composed of planes only, not traces. Short connections (~8 mil) are commonly used to attach vias to planes in Micron designs. Any connections required from V_{SS} to vias, for terminating resistors or decoupling capacitors, should be as short as possible to minimize trace inductance; Micron recommends an 8 mil trace width.

Figure 6: S1, S2, and S3 Spacing



Notes:
1. S3 is trace width.
2. Only RAS#, CAS#, and WE# control traces are shown. Other control signals are CS# and CKE.

Table 1: Interpair and Intrapair Spacing Design Guidelines

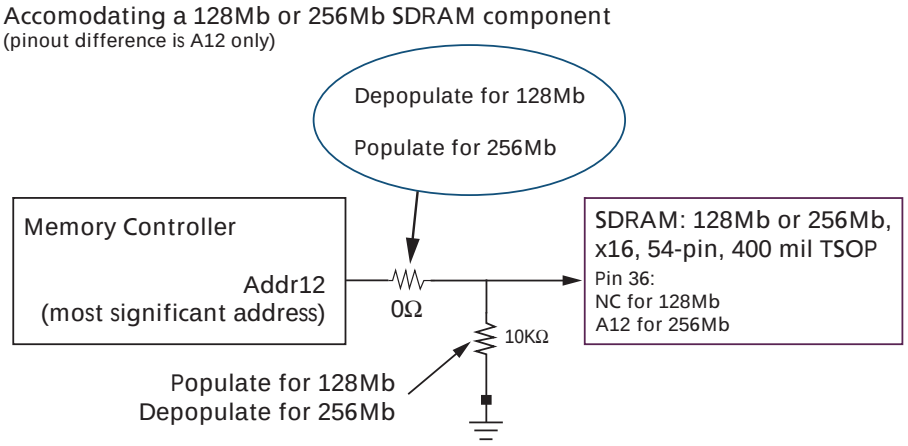
Signal Set	Signals	Spacing Type	Min	Nom	Max	Unit	Notes
Data/Data strobe	DQ to DQ	S1	8	12	–	mil	
	DQ to DQS	S2	8	12	–	mil	
	DQS in byte lane #1 to DQS in byte lane #2	S1	–	–	–	mil	1
	DQ and DM	S2	8	12	–	mil	
Address	Adjacent address lines	S1	6	12	–	mil	
	Address lines	S2	6	12	–	mil	
Command/Control	CAS#, RAS#, WE#, CS#, CKE	S1	6	15	–	mil	
Clock	CK#-to-CK	S1	4	–	6	mil	
	CK# (or CK in group of two) to DQS line	S2	–	–	–	mil	2
	Differential pair (CK, CK#) to any other signal	S2	8	12	–	mil	

Notes:
1. DQS signals are generally routed in the midst of related nibbles or bytes, so DQS-to-DQS spacing is not relevant.
2. Generally not an issue as they are not adjacent.

Expandability

Check component data sheets for "NC" pins in lower-density components that are used as upper-address pins in higher-density components. For example, pin 36 is "NC" in the 128Mb (8 Meg x 16) SDRAM component, but is A12 in the 256Mb (16 Meg x 16) SDRAM component. Increase design expandability by placing a 0Ω series resistor between the pin and the controller such that it can be depopulated when switching to the lower density component (shown in Figure 7), enabling the PCB to be reused without significant redesign.

Figure 7: Single PCB Layout to Accommodate Different Densities of Pin-Compatible SDRAM



Notes:
1. 128Mb SDRAM components require AUTO REFRESH cycles at a maximum interval of 15.625µs, while 256Mb SDRAM components require AUTO REFRESH cycles every 7.8125µs. The controller must adjust timing according to the refresh requirements for each device.
2. Some upper-column address lines change from "Don't Care" for lower-density components to the most-significant column address inputs for higher-density components. If the controller does not account for extra address pins, the system will experience addressing issues.

Unused DQ Pins

Occasionally, a x16 DDR device dedicated for error checking and correcting (ECC) will only use the lower 8 bits of data, and the upper DQ lines will be unused. In this case, if the x16 DDR component's DQ15:8 are not used, tie its corresponding UDM/DQMH mask pin HIGH to mask activity on this portion of the DQ bus. In addition, each pin in the DQ15:8 group should be tied LOW through a pull-down resistor (1–10 kΩ) to prevent noise from WRITE bursts. The same recommendation applies to the lower byte: If DQ7:0 are not used, tie the LDM/DQML mask pin HIGH and ground each unused DQ pin through a pull-down resistor.

For the x16 SDRAM device, unused DQ and DQM pins should be NC (no connect) or left floating. NC means the designated pins should not be connected to V_{SS}, V_{DD}, or any other pin. If only 8 bits of data are needed for the ECC on the SDRAM, design in a x8 component.

Termination: Topology, Value, Placement

Termination type (series or parallel) and values depend primarily upon device type (SDRAM or DDR), trace lengths, and number of components connected to the ASIC/FPGA/controller. Micron recommends series termination for SDRAM designs and DDR designs having fewer than four components, with trace lengths of ≤2in (5cm). Parallel termination is recommended for DDR designs with trace lengths >2in (5cm).

Series Termination

Series termination lowers power requirements (there is no DC current to be grounded), simplifies routing, and dampens overshoots occurring after signal transitions. However, determining resistor values can be difficult for series termination, and in some cases the optimum value depends on the output driver state (HIGH or LOW). Simulation will determine whether series termination is adequate.

Choose series resistor values such that the sum of the resistor impedance and controller output driver impedance equals Z_O.

If resistor packs are integrated as series impedance, use the same value of resistor for all signals in each group; it is also important to place the resistor pack physically in the same location (e.g., midpoint) for each data lane in the data bus.

Starting-point impedance values for the address, control, clock, and data signals are:

- SDRAM: For signals with lead-in termination <2in (5cm), place a 51–70Ω series resistor in the middle of the lead-in or close to the transmitting device. Z_O equals the value of the resistor.
- DDR: For signals with lead-in termination <2in (5cm) and full-strength drive enabled, place an ~50Ω series resistor in the middle of the lead-in or close to the transmitting device.
- DDR: For signals with lead-in termination <2in (5cm) and reduced-strength drive enabled, place an ~35Ω series resistor in the middle of the lead-in or close to the transmitting device, if required.

Verify that signaling meets data sheet requirements and those in "Simulation" on page 15. If the initial series termination values yield poor SI or data-eye aperture in simulation or prototypes, resistor values can be decreased to the ranges shown in Table 2.

Table 2: Minimal Series Termination Resistor Ranges

Place Rs terminators (if used) close to the memory device

Memory Type	Resistor	Location	Range	Units
SDRAM	CLK	midpoint	22–36	Ω
DDR	CK, CK#	midpoint[1]	–	Ω
DDR and SDRAM	DQ/DQS[2], address, command/control		10–33	Ω

Notes:
1. DDR does not use series termination on clock signals, only differential termination to minimize skew.
2. Do not share a DQS signal resistor pack with a non-data-group signal.

Simulations and prototype validation will confirm optimal values. Any additional timing margin that can be achieved through termination fine-tuning merits pursuit.

For bidirectional I/O signals, such as DQ, minimize ringing, overshoot, and undershoots by placing the RS halfway between source and sink devices.

For unidirectional I/O signals, such as address and control, optimal RS placement is close to the source processor/FPGA or at the midpoint of the signal line to minimize ringing and noise. Most of the energy is dissipated by reflection at the driver before it can propagate back down the trace a second time.

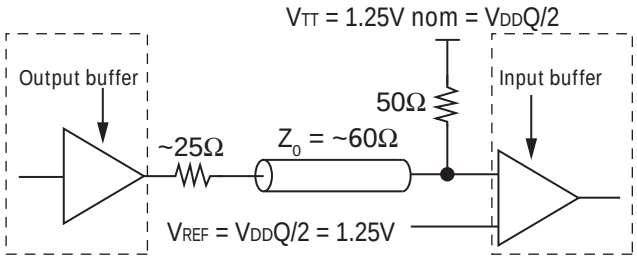
Parallel Termination

Parallel termination is only used with DDR and only required where at least one of the following is true:

- Five or more DDR devices
- >2in (5cm) trace lengths
- Poor simulation results
- Single- or multibit errors during prototyping (after simulation)

When parallel termination is required, Micron recommends the topology shown in Figure 8, which shows typical RT and RS values (SI determines optimal RS and RT values). Micron recommends values of $R_T \approx 2 \times R_S$, with RS typically 10–33Ω and RT typically 22–57Ω.

Figure 8: Single-Ended Parallel Termination for DDR Component(s)



Notes: 1. Dashed lines indicate inside of the controller or DRAM device.

All Micron DDR component address, data, command, and control signals are SSTL-2 compatible and should be terminated with single-ended parallel termination, when necessary. CKE is both SSTL-2 and LVCMOS compatible but can be terminated with single-ended parallel termination as well.

Point-to-point system topology progresses from no termination/unterminated, to series termination only, to single-ended parallel termination based on SI. As data rates increase, extra time spent optimizing termination will provide beneficial system results.

Routing

Though there are many signals on a DDR or SDRAM component, most of them have similar functionality and work together. Groups of I/O signals have one of four purposes: carry a binary address, transmit or receive data, relay a command to the device, or latch in address/data or a command.

The address group consists of row/column address and bank address pins. The command group includes the row address strobe (RAS#), the column address strobe (CAS#), and write enable (WE#). The control group includes chip select (CS#), and clock enable (CKE). Each data group/lane contains 10 signals: the eight DQ (DQ7–DQ0), the strobe (DQS), and the data mask (DM). Devices with x8 bus widths have only one data group, while x16 and x32 bus-width devices have two and four lanes, respectively.

To facilitate fanout of DDR data lanes, Micron recommends placing alternate adjacent 10-line data lanes on different critical PCB stackup layers. To decrease crosstalk, when data and address/control tracks coexist on the same layer they must be isolated from each other by at least 20 mil. Place data tracks on different layers from address and control lanes, if possible.

Related functionality makes minimizing skew critical, which requires signals of each group to be routed to similar electrical lengths. Routing address lines together on the same layer, and isolating data lanes from the address, command, and control groups will also help minimize skew.

Match trace lengths for the data group within ±50 mil of each other to diminish skew; serpentine traces (back and forth traces in an "S" pattern to increase trace length) can be used to match lengths. In addition, some controllers require byte lanes to have matched trace lengths. If the controller does not specify this, route byte lanes so that ±500 mil is the largest trace-length difference relative to the clock group trace length.

"S" pattern traces contribute the desired delay, but there is some coupling; simulations will validate timing.

Clock Signals

DDR Devices

All DDR differential clock pairs (CK and CK#) must be routed on the same layer. Placing the clock signals on an internal layer minimizes the noise (EMI).

For DDR systems, match CK trace length to CK# trace length ±20 mil, and CK/CK# trace lengths to DQS trace length ±500 mil. If multiple clock pairs are transmitted from the controller to components, all clock-pair traces should be equivalent within ±20 mil. Matching trace lengths to this level of accuracy helps meet the clock input midpoint voltage (V_{MP} [DC]) data sheet specification. As stated in "Miscellaneous V_{TT} Design Guidelines" on page 6, place a differential termination resistor (R_T) of 100–200Ω between CK# and CK near the DDR component input pins.

Figures 9 and 10 show recommended DDR routing topology and R_T placement for two clock pairs.

- If the trace lengths from split point to DDR components are less than ~1in (2.5cm), use a single 100–120Ω resistor (R_T) at the split point (Figure 9).
- If the lengths from the split point to the DRAM devices are greater than ~1in (2.5cm), use two resistors located near the respective DDR components (Figure 10). These resistors are in parallel, so each R_T should be 200–240Ω to keep the effective resistance at 100–120Ω.

Figure 9: Single CK–CK# Differential Resistor Placement at Split Point

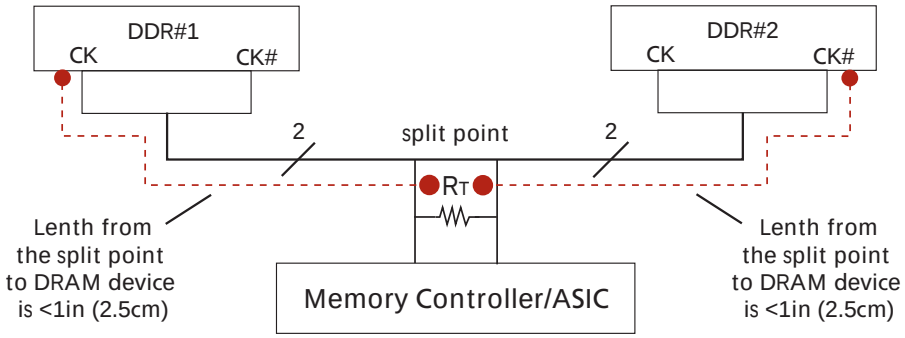
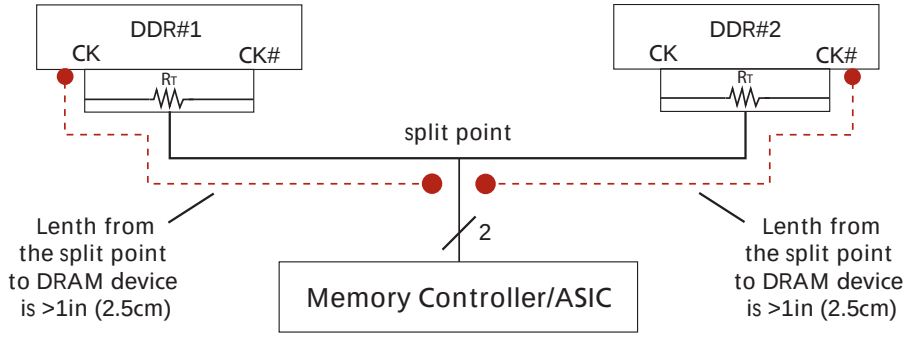


Figure 10: Two CK–CK# Differential Resistors Placement at Component



Notes: 1. All CK and CK# signal lines have characteristic impedance (Z_0) of 50–60Ω.

Match clock-pair traces to each signal trace in the address and command groups to within ±400 mil. If clocks cannot be matched to these groups within 400 mil, then all clock trace lengths must be increased as a group. The longest-to-shortest trace length difference must be ≤800 mil, so both longest and shortest traces determine how much length must be added to all clock lines.

SDRAM Devices

SDRAM systems have only a single-ended clock (CLK), so the important trace-matching relationship is not to a second differential clock trace but instead to the other groups. Match clock traces to data group traces within ±500 mil. If multiple clocks are transmitted from the controller to components, all clock-pair traces should be equivalent to within ±20 mil. Matching trace lengths to this level of accuracy helps minimize skew.

For both DDR and SDRAM, also match clock traces to each signal trace in the address and command groups to within ±400 mil. If clock traces cannot be matched to the trace lengths of these groups within 400 mil, then all clock trace lengths must be increased as a group. The longest-to-shortest trace-length difference must be ≤800 mil, so both longest and shortest traces determine how much length must be added to all clock lines.

Miscellaneous Routing Recommendations

A 400 mil difference in address-, command-, or signal-group trace lengths equates to 0.4in × (1,000ps of propagation delay per 6in of trace), or ~67ps of skew. If the timing budget can absorb this minor amount of lane-to-lane skew and other routing delays, the system will perform normally. Total routing-based delays must meet ^{t}DQSCK, controller DQS recovery limits, and other data sheet AC timing parameters.

Regardless of bus type, all DDR signal groups must be properly referenced to a solid VSS or VDD plane. For both READs and WRITEs the key relationship is between CK/CK#, DQ, DM, and DQS signals (the DDR data group), which operates at twice the speed of other signal groups, which makes SI more critical. DQ, DQS, and clock lines are best referenced to VSS to minimize noise. If a VSS layer is not easily accessible, address and command lines can reference a VDD layer, though it is generally more noisy.

Keep traces as short as possible. If trace length (from controller pad to DRAM pad) is <2in (5cm) for both DDR and SDRAM point-to-point applications, routing simplifies and signal quality usually increases in proportion. In most cases, trace lengths >2in (5cm) lead to more signal undershoot, overshoot, and ringing—all of which are detrimental to SI.

Additional Trace-Length Design Guidelines

- Match different DQ byte lanes to within 1in (2.5cm) of each other. A 1in trace-length difference equates to 167ps of propagation delay. Thus, the timing budget must be able to absorb 167ps for a 1in difference in byte-lane matching.
  - Within a byte lane, match all DQ and DQS traces to within ±50 mil.
  - Route data groups next to a VSS plane to minimize the return path/loop length.
- Maintain a solid ground reference (no splits, etc.) for each group to provide a Low-Z return path; high-speed signals must not cross a plane split.

Decoupling

Adequate power decoupling on the PCB is necessary to prevent excessive VDD noise and memory errors in a situation where power-supply draw can change by magnitudes in a single clock cycle.

Optimizing particular capacitor values and benefits can also be done through simulation or prototype. From this data, designers can determine the number and location of decoupling capacitors, which are more important factors than the exact value of each capacitor.

Micron recommends at least four low (2nH) effective series inductance (ESL) capacitors per DDR component to decouple VDD/VDDQ from VSS/VSSQ. Values from 0.01–0.22µF balance well in bypassing higher-frequency (0.01µF) and lower-frequency (0.22µF) noise. If a frequency spectrum characterizing power-supply noise can be generated, choosing capacitors to optimize decoupling at certain frequencies (where noise peaks) is preferred.

Micron recommends tantalum capacitors; while they have a higher purchase cost, they have a more stable service life than electrolytic capacitors. Electrolytic capacitors tend to steadily lose quality as they age.

Place ESL capacitors as close as possible to each corner of each DDR or SDRAM device; adjacent devices can share capacitors at the corners that fall between them, but back side components must have their own capacitors. This distributed decoupling minimizes capacitor ESL and localizes transient currents and returns.

See Micron technical notes TN-46-02, "Decoupling Capacitor Calculation for a DDR Memory Channel" and TN-00-06, "Bypass Capacitor Selection for High Speed Designs" for detailed decoupling discussions. Both are available on Micron's Web site: www.micron.com.

Simulation

During the layout phase of a new or revised design, Micron strongly recommends simulating I/O performance at regular intervals. Optimizing interface termination values through simulation can decrease noise and increase timing margins before building prototypes. Issues are often easier to solve when found in simulation, as opposed to those found later that require expensive and time-consuming board redesigns or factory recalls.

Micron has created many different types of simulation models to match different tools in use. Component simulation models presently posted on Micron's Web site include IBIS, Verilog, VHDL, Hspice, Denali, and Synopsys.

Verifying all simulated conditions is impractical, but there are a few key areas to focus on: DC levels, signal slew rates, undershoot, overshoot, ringing, and waveform shape. Also, verifying that the design has sufficient signal-eye openings to meet both timing and AC input voltage levels is extremely important.

Micron Technical Note TN-46-11, "DDR SDRAM Point-to-Point Simulation Process," provides a detailed discussion and is available at Micron's Web site. TN-46-11 provides more information on these key areas of DDR point-to-point simulation:

- Signal integrity
- Board skew and the contributing factors
- Return path discontinuities

Other Resources

Other available design resources at Micron's Web site:

- Technical presentations "Termination for Point-to-Point Systems" and "DDR Motherboard Design" give additional recommendations for SDRAM and DDR device layout, termination, and routing, including reference illustrations and photographs of different motherboards that highlight key areas.
- Gerber files for Micron modules are available at www.micron.com/products/modules/ddrsdram/designfiles.html. Designers using five or more components in a point-to-point environment might find these reference designs useful as examples of highly optimized signal routing.
- Several JEDEC reference designs are available on the site for module raw cards. JEDEC designs include the schematic, layout, layer stackup, components used, passive values, passive placement, and routing. Designers using five or more components in a point-to-point environment might find these reference designs useful; however, they are less specific than Micron module designs.

Conclusion

This document contains established system design recommendations to improve SI and reduce noise for DDR and SDRAM devices in point-to-point systems. If implemented, most systems will realize better memory functionality and stability, leading to enhanced production yields for their finished product.

8000 S. Federal Way, P.O. Box 6, Boise, ID 83707-0006, Tel: 208-368-3900
prodmktg@micron.com www.micron.com Customer Comment Line: 800-932-4992