



Application Note

SL1620 General PCB Design and DDR4 & DDR3 Interface Layout Guidelines

Abstract: This document provides PCB design and layout guidelines for integrating the Synaptics SL1620 with DDR4 and DDR3 memory, detailing routing rules, power supply considerations, impedance requirements, and best practices to ensure optimal signal integrity and performance.

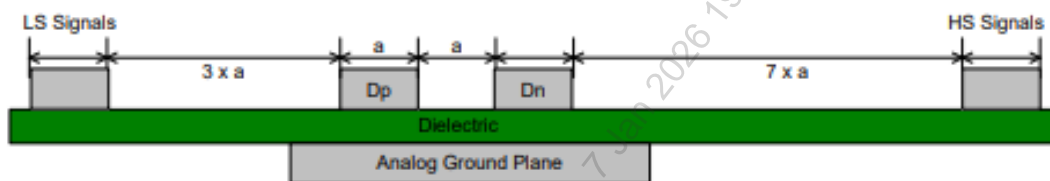
Contents

1.	PCB Layout Guidelines.....	3
2.	Power Supply Guidelines.....	4
3.	4L non-HDI PCB Design Rules.....	5
	3.1. NSMD Pad.....	5
	3.2. SMD Pad.....	5
	3.3. Via.....	5
	3.4. Spacing.....	5
4.	PCB Routing Rules for SL1620 DDR4/DDR3 Interface.....	6
	4.1. General Rules.....	6
	4.2. Placement.....	6
	4.3. Routing.....	7
	4.4. Package Trace Length Compensation.....	7
	4.5. Layer PCB Routing Example: Recommended Design Rules.....	11
	4.5.1. Placement and Routing Recommendations.....	12
5.	Revision History.....	13

Downloaded by Anonymous () on 7 Jan 2026 19:24:28 UTC

1. PCB Layout Guidelines

- Trace impedance of 100ohm differential (+/-10%) is required. For USB2.0, the trace impedance of 90ohm differential (+/-10%) is required.
- Matching to < 0.5mm (about 0.02 inch) between two different signals. Trace lengths should match by 0.25mm (about 0.01 inch) or less for differential pairs (same pair) of high-speed signals.
- The skew between any data lane and clock lane should be matched within +/-10ps on both package and PCB.
- Do not route trace over plane void or anti-pads. Return path should be VSS and continuous.
- Ensure ground return vias adjacent to the differential pair core vias to minimize crosstalk between lanes.
- Void the planes above the BGA pads to minimize the capacitive discontinuity.
- To minimize crosstalk, take care of signal traces which are routed close to the data differential pairs. The minimum recommended spacing is $3x a$ for low-speed non-periodic signals and $7x a$ for high-speed periodic signals. A continuous ground plane below the differential lines is required.



- TX and RX pairs should not be routed side-by-side in the same signal layer.
- Crosstalk should be accumulated total with all aggressors and meet the
- -30dB requirement until Nyquist frequency.
- The insertion loss of the trace is < 3dB (at Nyquist). For PCIE3.0, USB3.0 and Ethernet, the insertion loss of trace can be < 6dB. It should be monotonic, with no large insertion loss variations (+/-2dB) in the Nyquist frequency range. Trace should be shortest and low-loss as possible.

2. Power Supply Guidelines

- Supply bypass capacitors are recommended to minimize power supply noise. Noise analysis of the power delivery network is required to determine the actual values. Depending on their size, each capacitor will have a different equivalent series resistance (ESR) and equivalent series inductance (ESL) that will determine the given capacitor's effectiveness over a frequency range. In general, several low-value capacitors (ceramic-type capacitors) should be placed as close as possible to the package pins. Larger-value capacitors (tantalum/electrolytic-type capacitors) can be placed farther away.
- The supply bypass capacitors should be connected as close as possible to the package pins to ensure a tight return path and maximize their effectiveness. When connecting from the package pin to the bypass capacitors, use as wide a plane and trace as possible to reduce inductive and resistive losses. Typical capacitor placement can be under the package (other side of the board) or on the same side but close. An example of bypass capacitors is shown below.

Component	Value ¹
Power Supply Bypass Capacitors	0.01 μ F, 0.1 μ F, 4.7 μ F, 10.0 μ F

1. Smaller-value capacitors must be placed between the ferrite bead and the package.

- Both power plane and ground plane should be maintained continuously and have solid return path (not in bits and pieces). Pay attention to the void areas caused by vias. If the planes are cut down by vias, we need to compensate for the loss of the plane shape to make sure the effective width of the plane.

3. 4L non-HDI PCB Design Rules

3.1. NSMD Pad

- Minimum pin pitch = 0.4 mm (15.75 mil)
- Footprint pad / Paste mask of pad = 10mil (bga10)
- Solder mask of pad = 10 mil

3.2. SMD Pad

- Minimum pin pitch = 0.381 mm (15 mil)
- Footprint pad / Paste mask of pad = 12 mil (bga12)
- Solder mask of pad = 8 mil

3.3. Via

- Minimum through hole pad / Drill size = 14 / 8 mil

3.4. Spacing

- Minimum trace-to-trace = 3 mil
- Minimum trace-to-via = 3 mil
- Minimum trace-to-pad = 3 mil

The example stack-up/trace width and spacing of 4L non-HDI PCB is shown below.

Stack-up			Thickness(mil)		DK	Df	Material
	SM_TOP (0.5_PT_10Z)	0.5			3.4		
L1	PP (1080/RC=69%) / 0.076mm	2.99	1.654	0.333oz +Plating	4.14	0.019	\$1000-2M
L2	Core+PP (1.3mm)	51.18	1.26	1oz	4.6	0.018	\$1000-2M
L3	PP (1080/RC=69%) / 0.076mm	2.99	1.26	1oz	4.14	0.019	\$1000-2M
L4	SM_BOT (0.5_PT_10Z)	0.5	1.654	0.333oz +Plating	3.4		
Thickness(mil):		63.988	63.988				
FinishPCBThickness(mm):		1.6 (+/-0.16) mm					
PressPCBThickness(mm):		1.48 (+/-0.08) mm					
Number	Type	Control Layer	Reference Layer	Adjust Line Width (mil)	Adjust Spacing (mil)	Adjust Line To Copper (mil)	Design Impedance (ohm)
1	Single-End	L1/L4	L2/L3	3.6			55.01
2	Single-End	L1/L4	L2/L3	4			52.93
3	Single-End	L1/L4	L2/L3	4.5			50.15
4	Differential	L1/L4	L2/L3	3.5	3		84.62
5	Differential	L1/L4	L2/L3	3.6	4		90.11
6	Differential	L1/L4	L2/L3	3.4	5.8		99.3
7	Differential	L1/L4	L2/L3	4	4		88.41

4. PCB Routing Rules for SL1620 DDR4/DDR3 Interface

Notes:

- This guideline recommends optimal layout practices for the DRAM section and is not a set of strict limitations.
- Synaptics reference layouts follow the guidelines, ensuring good operating timing margins.
- It is strongly recommended to copy DDR routing from the reference design.

4.1. General Rules

The general rules of usage are:

- All signals must be routed with a solid reference layer.
- Ground reference routing for all signals is preferred.
- No signal can be routed through discontinued reference.
- The impedance of single-ended signals should be targeted within a range of 50 ohms to 55 ohms, with a tolerance of +/- 10%.
- All differential signals should have a differential impedance of 85 ohms with a tolerance of +/-10%.
- Include via delay in length matching calculations.
- No more than two vias are recommended on any signal between SoC and DRAMs, excluding low speed DDR RSTn.

4.2. Placement

- The DDR4 signal pin assignments of the SL1620 are designed to allow the DDR4 chip to be placed next to SoC on top layer.
- Underneath the SoC, the decoupling capacitors for VDDM should include a combination of various capacitors that provide lower impedance from DC to 300MHz. It is highly recommended to follow the placement and values specified in the Synaptics Reference Design Kits.
- The RC components connecting to the SL1620 MO_VREF should be placed as close to the pin as possible.
- A 1% 240 ohm resistor (pulldown to GND) connecting to MO_CAL should be placed as close to the pin as possible.
- The RC components connecting to SDRAM_CK0+/- should be placed as close to the SDRAM pins as possible.
- A small capacitor (2.2pF) on SDRAM_CK0+/- needs to be placed near the first DRAM (the one closest to the SoC).
- A couple of 1nF and 10nF capacitors should be placed as close as possible to each individual power pin of the SoC and all DRAMs for decoupling purpose.
- A couple of 1nF and 10nF capacitors should be placed at the edge of the power plane to reduce resonance.
- Place 1uF/10uF capacitors at the power rail input for both SoC and DRAM, to avoid power trace bottlenecking.

4.3. Routing

- The trace length difference of the DDR4/DDR3 signals (within each own group) should be maintained as follows:
 - **DQ-to-DQ (including DQS) for each Byte group:** Keep the shortest possible for each trace; trace length matching is not necessary.
 - **Address / Command (ADCM) -to-ADCM (including CLK):** Keep the shortest possible for each trace; trace length matching is not necessary.
 - **Intra-pair of Differential signals (DQS+/-, CLK+/-):** Maintain a trace length difference within 15 mils.
- The stub length from the via of each CKO+/- trace to the resistor pad should be minimized as much as the board topology, stack-up and placement constraints allow. While the general guideline is to not exceed 300 mils, in many cases the trace can be as short as 50 mils.

4.4. Package Trace Length Compensation

The following are considerations for compensation:

- Trace lengths between chip die and package pins were not matched well due to package geometry, resulting in different propagation delays for each signal on the package substrate. These differences will impact timing margin.
- When matching the trace length for each Byte group, pin delay should be accounted for.

Byte Lane 0 Group				
Pin	Signal Net	DDR4 Signal	DDR3 Signal	Equivalent PCB Trace Length Based on Propagation delay Entered (mil)
P2	MO_DQSP[0]	MO_DDR4_DQS0p	MO_DDR3_DQS0p	190.33
N2	MO_DQSn[0]	MO_DDR4_DQS0n	MO_DDR3_DQS0n	195.54
V2	MO_DM[0]	MO_DDR4_DMO	MO_DDR3_DMO	197.23
T6	MO_DQ[0]	MO_DDR4_DQ0	MO_DDR3_DQ2	125.18
T1	MO_DQ[1]	MO_DDR4_DQ1	MO_DDR3_DQ3	194.06
U3	MO_DQ[2]	MO_DDR4_DQ2	MO_DDR3_DQ0	162.14
T2	MO_DQ[3]	MO_DDR4_DQ3	MO_DDR3_DQ5	172.39
P4	MO_DQ[4]	MO_DDR4_DQ4	MO_DDR3_DQ6	151.78
U2	MO_DQ[5]	MO_DDR4_DQ5	MO_DDR3_DQ1	168.93
P6	MO_DQ[6]	MO_DDR4_DQ6	MO_DDR3_DQ4	118.64
R2	MO_DQ[7]	MO_DDR4_DQ7	MO_DDR3_DQ7	175.45

Byte Lane 1 Group				
Pin	Signal Net	DDR4 Signal	DDR3 Signal	Equivalent PCB Trace Length Based on Propagation delay Entered (mil)
W1	MO_DQSp[1]	MO_DDR4_DQS1p	MO_DDR3_DQS1p	206.70
W2	MO_DQSn[1]	MO_DDR4_DQS1n	MO_DDR3_DQS1n	201.21
W6	MO_DM[1]	MO_DDR4_DM1	MO_DDR3_DM1	123.28
U6	MO_DQ[8]	MO_DDR4_DQ8	MO_DDR3_DQ13	126.55
AA1	MO_DQ[9]	MO_DDR4_DQ9	MO_DDR3_DQ8	209.73
W7	MO_DQ[10]	MO_DDR4_DQ10	MO_DDR3_DQ11	105.79
Y2	MO_DQ[11]	MO_DDR4_DQ11	MO_DDR3_DQ12	189.13
W3	MO_DQ[12]	MO_DDR4_DQ12	MO_DDR3_DQ15	173.81
AA2	MO_DQ[13]	MO_DDR4_DQ13	MO_DDR3_DQ14	195.80
W4	MO_DQ[14]	MO_DDR4_DQ14	MO_DDR3_DQ9	141.00
AB2	MO_DQ[15]	MO_DDR4_DQ15	MO_DDR3_DQ10	183.74
Byte Lane 2 Group				
Pin	Signal Net	DDR4 Signal	DDR3 Signal	Equivalent PCB Trace Length Based on Propagation delay Entered (mil)
AC2	MO_DQSp[2]	MO_DDR4_DQS2p	MO_DDR3_DQS2p	214.38
AC1	MO_DQSn[2]	MO_DDR4_DQS2n	MO_DDR3_DQS2n	217.80
AF2	MO_DM[2]	MO_DDR4_DM2	MO_DDR3_DM2	218.32
AE4	MO_DQ[16]	MO_DDR4_DQ16	MO_DDR3_DQ18	171.01
AF1	MO_DQ[17]	MO_DDR4_DQ17	MO_DDR3_DQ19	226.97
AC6	MO_DQ[18]	MO_DDR4_DQ18	MO_DDR3_DQ16	158.43
AD3	MO_DQ[19]	MO_DDR4_DQ19	MO_DDR3_DQ21	188.75
AC4	MO_DQ[20]	MO_DDR4_DQ20	MO_DDR3_DQ22	182.60
AE3	MO_DQ[21]	MO_DDR4_DQ21	MO_DDR3_DQ17	202.93
AC7	MO_DQ[22]	MO_DDR4_DQ22	MO_DDR3_DQ20	156.91
AD2	MO_DQ[23]	MO_DDR4_DQ23	MO_DDR3_DQ23	211.67

Byte Lane 3 Group				
Pin	Signal Net	DDR4 Signal	DDR3 Signal	Equivalent PCB Trace Length Based on Propagation delay Entered (mil)
AH1	MO_DQSp[3]	MO_DDR4_DQS3p	MO_DDR3_DQS3p	245.19
AJ1	MO_DQSn[3]	MO_DDR4_DQS3n	MO_DDR3_DQS3n	242.23
AK2	MO_DM[3]	MO_DDR4_DM3	MO_DDR3_DM3	238.19
AH2	MO_DQ[24]	MO_DDR4_DQ24	MO_DDR3_DQ27	216.37
AL3	MO_DQ[25]	MO_DDR4_DQ25	MO_DDR3_DQ24	237.31
AE6	MO_DQ[26]	MO_DDR4_DQ26	MO_DDR3_DQ31	165.81
AJ2	MO_DQ[27]	MO_DDR4_DQ27	MO_DDR3_DQ28	225.48
AG4	MO_DQ[28]	MO_DDR4_DQ28	MO_DDR3_DQ29	196.17
AL4	MO_DQ[29]	MO_DDR4_DQ29	MO_DDR3_DQ30	216.70
AJ3	MO_DQ[30]	MO_DDR4_DQ30	MO_DDR3_DQ25	203.98
AM4	MO_DQ[31]	MO_DDR4_DQ31	MO_DDR3_DQ26	227.84

Downloaded by Anonymous () on 7 Jan 2026 19:24:28 UTC

ADDRESS / COMMAND / CONTROL Group				
Pin	Signal Net	DDR4 Signal	DDR3 Signal	Equivalent PCB Trace Length Based on Propagation delay Entered (mil)
M3	MO_CKp	MO_DDR4_CKp	MO_DDR3_CKp	179.68
M4	MO_CKn	MO_DDR4_CKn	MO_DDR3_CKn	180.65
C4	MO_A[0]	MO_DDR4_A0	MO_DDR3_A11	198.08
H6	MO_A[1]	MO_DDR4_A1	MO_DDR3_A2	165.55
K6	MO_A[2]	MO_DDR4_A2	MO_DDR3_A6	149.73
J3	MO_A[3]	MO_DDR4_A3	MO_DDR3_A15	181.94
H2	MO_A[4]	MO_DDR4_A4	MO_DDR3_A0	228.18
G2	MO_A[5]	MO_DDR4_A5	MO_DDR3_A12	197.44
F3	MO_A[6]	MO_DDR4_A6	MO_DDR3_A5	209.42
F2	MO_A[7]	MO_DDR4_A7	MO_DDR3_A4	225.36
F4	MO_A[8]	MO_DDR4_A8	MO_DDR3_A9	204.51
C1	MO_A[9]	MO_DDR4_A9	MO_DDR3_A7	248.83
K7	MO_A[10]	MO_DDR4_A10	MO_DDR3_A3	131.16
B4	MO_A[11]	MO_DDR4_A11	MO_DDR3_A8	225.23
K3	MO_A[12]	MO_DDR4_A12	MO_DDR3_A10	190.45
H3	MO_A[13]	MO_DDR4_A13	MO_DDR3_A1	194.53
D3	MO_ACTn	MO_DDR4_ACTn	MO_DDR3_A13	244.12
C3	MO_BA[0]	MO_DDR4_BA0	MO_DDR3_BA0	230.86
D2	MO_BA[1]	MO_DDR4_BA1	MO_DDR3_BA1	227.76
C2	MO_BG	MO_DDR4_BG0	MO_DDR3_BA2	253.59
H1	MO_CASn	MO_DDR4_CASn	MO_DDR3_CASn	226.71
J2	MO_CKE	MO_DDR4_CKE	MO_DDR3_CKE	212.16
M2	MO_CSn	MO_DDR4_CSn	MO_DDR3_CSn	212.96
L2	MO_ODT	MO_DDR4_ODT	MO_DDR3_ODT	215.68
L1	MO_RASn	MO_DDR4_RASn	MO_DDR3_RASn	207.24
A5	MO_RSTn	MO_DDR4_RSTn	MO_DDR3_RSTn	227.17
F1	MO_WEn	MO_DDR4_Wen	MO_DDR3_WEn	238.27
B5	MO_NC	MO_DDR4_NC	MO_DDR3_A14	221.50
M7	MO_CAL	MO_DDR4_CAL	MO_DDR3_CAL	114.90

4.5. Layer PCB Routing Example: Recommended Design Rules

1. Footprint

Please follow the dimension drawing in the Synaptics Datasheet. Pay attention to the size of the pad and solder mask opening.

- **Pad size:** 10 mil
- **Solder mask opening size:** 10 mil

2. Layout Design Rule

- **PCB board thickness:** 1.6mm
- **4-layer PCB**
- **Single-ended trace width:** 4 mil
- **Single-ended trace impedance:** 55+/-10% ohms
- **Differential-pair trace width/spacing:** 4.5/4 mil
- **Differential-pair trace impedance:** 85+/-10% ohms
- **Minimum via (mechanical drilling) size:** 8 mil
- **Maximal annulus of via:** 16 mil
- **Minimum spacing (excluding BGA area):**
 - **Pad to pad:** 4 mil
 - **Pad to trace:** 4 mil
 - **Pad to via:** 4 mil
 - **Trace to trace:** 4 mil
 - **Trace to via:** 4 mil
 - **Via to via:** 4 mil

3. PCB stack-up

Layer Name	Thickness	Dk @ 1GHz
Top SM	1.18 mil	3.8
Cu1	0.333oz + Planting	
Dielectric	3.2 mil	4.5
Cu2	1 oz	
Core	48.38	4.5
Cu3	1 oz	
Dielectric	3.2 mil	4.5
Cu4	0.333oz + Planting	
Bottom SM	1.18 mil	3.8

4.5.1. Placement and Routing Recommendations

This placement and routing of Synaptics reference design were based on intensive signal integrity (SI) simulations, including model extraction and transient analyst, to ensure timing closure. This is particularly sensitive for 4-layer PCBs due to the non-homogeneous nature of the micro-strip line.

Single set of fixed routing rules: It is not possible to provide one single set of fixed detailed routing rules for all designs.

Recommendation: Follow the PCB layout of Synaptics reference design.

Custom PCB layout: If a custom PCB layout is used, it is crucial to close timing with proper SI simulation to ensure proper functionality.

Downloaded by Anonymous () on 7 Jan 2026 19:24:28 UTC

5. Revision History

Revision	Description
A	Initial release.
B	Release as Public document.
C	Updated Routing section.
D	Added the following sections: <ul style="list-style-type: none"> • 1 PCB Layout Guidelines • 2 Power Supply Guidelines • 3 4L non-HDI PCB Design Rules
E	Minor update to document title.
F	Minor update to latest template.

Downloaded by Anonymous () on 7 Jan 2026 19:24:28 UTC



Copyright

Copyright © 2023–2025 Synaptics Incorporated. All Rights Reserved.

Trademarks

Synaptics and the Synaptics logo are trademarks or registered trademarks of Synaptics Incorporated in the United States and/or other countries.

All other trademarks are the properties of their respective owners.

Contact Us

Visit our website at www.synaptics.com to locate the Synaptics office nearest you.

PN: 506-001393-01 Rev F

Notice

Use of the materials may require a license of intellectual property from a third party or from Synaptics. This document conveys no express or implied licenses to any intellectual property rights belonging to Synaptics or any other party. Synaptics may, from time to time and at its sole option, update the information contained in this document without notice.

INFORMATION CONTAINED IN THIS DOCUMENT IS PROVIDED "AS-IS," AND SYNAPTICS HEREBY DISCLAIMS ALL EXPRESS OR IMPLIED WARRANTIES, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, AND ANY WARRANTIES OF NON-INFRINGEMENT OF ANY INTELLECTUAL PROPERTY RIGHTS. IN NO EVENT SHALL SYNAPTICS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, PUNITIVE, OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR IN CONNECTION WITH THE USE OF THE INFORMATION CONTAINED IN THIS DOCUMENT, HOWEVER CAUSED AND BASED ON ANY THEORY OF LIABILITY, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, AND EVEN IF SYNAPTICS WAS ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. IF A TRIBUNAL OF COMPETENT JURISDICTION DOES NOT PERMIT THE DISCLAIMER OF DIRECT DAMAGES OR ANY OTHER DAMAGES, SYNAPTICS' TOTAL CUMULATIVE LIABILITY TO ANY PARTY SHALL NOT EXCEED ONE HUNDRED U.S. DOLLARS.