



Application Note

SL1680 General PCB Design and LPDDR4 & LPDDR4(x) Interface Layout Guidelines

Abstract: This document provides PCB design and layout guidelines for integrating the Synaptics SL1680 with LPDDR4 and LPDDR4x memory, detailing routing rules, power supply considerations, impedance requirements, and best practices to ensure optimal signal integrity and performance.

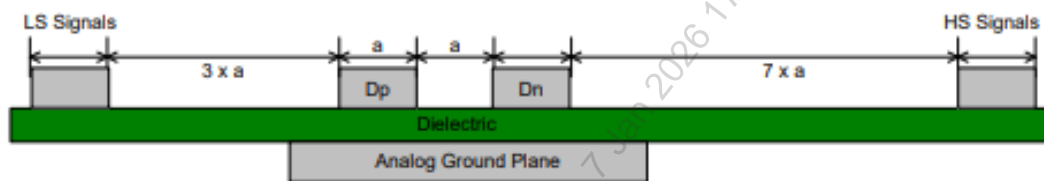
Contents

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 SL1680 LPDDR4/LPDDR4x Interface.....	6
4.1.	General Rules.....	6
4.2.	Package Trace Length Compensation.....	7
4.3.	Data (DQ/DQM/DQS) Bus.....	9
4.4.	CA Bus.....	10
4.5.	Board 6-Layers Stack up.....	11
5.	Revision History.....	12

Downloaded by Anonymous () on 7 Jan 2026 17:43:32 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 / 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




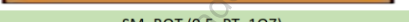
3.3. Via

- 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		1.654	0.333oz +Plating		4.14	0.019	\$1000-2M
L2	PP (1080/RC=69%) / 0.076mm	2.99					
L2		1.26	1oz		4.6	0.018	\$1000-2M
L3	Core+PP (1.3mm)	51.18					
L3		1.26	1oz		4.14	0.019	\$1000-2M
L4	PP (1080/RC=69%) / 0.076mm	2.99					
L4		1.654	0.333oz +Plating		3.4		
	SM_BOT (0.5_PT_10Z)	0.5					
Thickness(mil):		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 SL1680 LPDDR4/LPDDR4x 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.

4.1. General Rules

The general rules of usage are:

- All signals must be routed with a solid reference layer.
- Route single-end DQ[x] on the same layer within each BYTE group. If different layers are used, account for varying propagation delays during electrical length matching.
- Bit order can be scrambled within the same byte group for easier routing.
- The impedance of single-ended signals should be 50 ohms +/- 10%. (It may be 55 ohms +/- 10% in the BGA break-out area).
- The impedance of differential pairs should be 85 ohms +/- 10%.
- Include via delay in length matching calculations.
- Place 1nF/10nF capacitors near each power pin of the SoC and all DRAMs for effective decoupling.
- Place 1nF/10nF capacitors 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.
- Place ground vias as close as possible to the cluster of signals vias.
- No more than two vias are recommended on any signal between the SoC and DRAMs, excluding low-speed DDR RSTn.

4.2. 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. To compensate for this mismatch, proper PCB routing is essential. Refer to the table below for the electrical length of individual traces.
- Note that the compensation should be made in addition to any special trace matching or tuning requirements mentioned in the sections that follow.
- The detailed constraint information can be found in the Synaptics reference layout files.

Downloaded by Anonymous () on 7 Jan 2026 17:43:32 UTC

North Side

ADDRESS / COMMAND / CONTROL Group			Byte Lane 0 Group			Byte Lane 1 Group		
Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)	
J31 M0_CKP	25.50		B42 M0_DM[0]	44.52		C32 M0_DM[1]	31.99	
L31 M0_CKN	22.91		E39 M0_DQ[0]	36.61		C34 M0_DQ[8]	35.85	
G31 M0_A[0]	29.15		G39 M0_DQ[1]	31.54		G35 M0_DQ[9]	32.42	
N31 M0_A[1]	19.25		C40 M0_DQ[2]	36.41		B30 M0_DQ[10]	35.97	
C27 M0_A[2]	30.88		C38 M0_DQ[3]	33.75		C30 M0_DQ[11]	31.59	
C26 M0_A[3]	28.97		N39 M0_DQ[4]	26.02		B35 M0_DQ[12]	41.39	
G27 M0_A[4]	24.24		A42 M0_DQ[5]	49.31		C36 M0_DQ[13]	36.53	
E27 M0_A[5]	25.15		C43 M0_DQ[6]	42.97		A35 M0_DQ[14]	40.36	
A28 M0_CKE	39.25		B43 M0_DQ[7]	43.77		B36 M0_DQ[15]	36.14	
B28 M0_CSN	38.56		J39 M0_DQSP[0]	30.82		J35 M0_DQSP[1]	27.64	
			L39 M0_DQSN[0]	29.63		L35 M0_DQSN[1]	26.43	

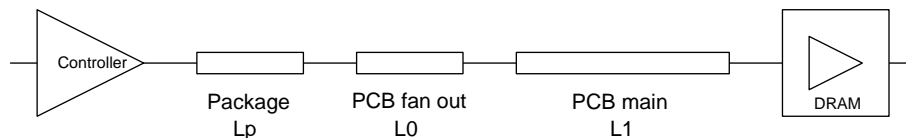
ADDRESS / COMMAND / CONTROL Group			Byte Lane 2 Group			Byte Lane 3 Group		
Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)	
L23 M1_CKP	22.06		E15 M1_DM[0]	27.23		B4 M1_DM[1]	37.57	
J23 M1_CKN	20.80		C11 M1_DQ[0]	35.66		A3 M1_DQ[8]	44.76	
B22 M1_A[0]	33.51		C14 M1_DQ[1]	33.54		C6 M1_DQ[9]	35.94	
G23 M1_A[1]	22.25		C12 M1_DQ[2]	31.63		B3 M1_DQ[10]	39.53	
C19 M1_A[2]	31.89		B11 M1_DQ[3]	33.03		C4 M1_DQ[11]	34.35	
G19 M1_A[3]	23.93		B16 M1_DQ[4]	37.67		G11 M1_DQ[12]	29.36	
A22 M1_A[4]	38.10		A16 M1_DQ[5]	38.72		C8 M1_DQ[13]	35.35	
C20 M1_A[5]	30.65		C18 M1_DQ[6]	30.30		A10 M1_DQ[14]	38.38	
C24 M1_CKE	34.36		B18 M1_DQ[7]	30.92		B10 M1_DQ[15]	33.68	
B24 M1_CSN	31.28		L19 M1_DQSP[0]	21.51		J15 M1_DQSP[1]	22.01	
			J19 M1_DQSN[0]	20.55		L15 M1_DQSN[1]	20.61	

West Side

ADDRESS / COMMAND / CONTROL Group			Byte Lane 0 Group			Byte Lane 1 Group		
Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)	
AC11 M2_CKP	20.35		D2 M2_DM[0]	44.38		P3 M2_DM[1]	32.69	
AC9 M2_CKN	21.27		C2 M2_DQ[0]	47.67		L7 M2_DQ[8]	38.12	
AC13 M2_A[0]	13.71		D3 M2_DQ[1]	38.47		R5 M2_DQ[9]	29.49	
AB2 M2_A[1]	35.51		F3 M2_DQ[2]	38.23		L3 M2_DQ[10]	34.61	
AC7 M2_A[2]	22.73		C1 M2_DQ[3]	48.88		R7 M2_DQ[11]	25.36	
W3 M2_A[3]	28.58		H3 M2_DQ[4]	41.44		W7 M2_DQ[12]	25.10	
AB1 M2_A[4]	34.18		K2 M2_DQ[5]	40.11		T1 M2_DQ[13]	34.48	
Y3 M2_A[5]	29.18		K1 M2_DQ[6]	38.29		M3 M2_DQ[14]	36.78	
V3 M2_CKE	31.17		L2 M2_DQ[7]	33.26		T2 M2_DQ[15]	34.91	
V2 M2_CSN	38.68		R11 M2_DQSP[0]	27.30		W11 M2_DQSP[1]	21.83	
			R9 M2_DQSN[0]	22.54		W9 M2_DQSN[1]	20.86	

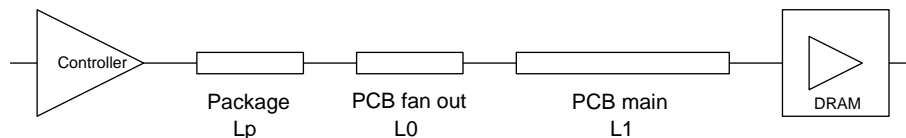
ADDRESS / COMMAND / CONTROL Group			Byte Lane 2 Group			Byte Lane 3 Group		
Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)		Pin Signal	Package Propagation Delay (ps)	
AL11 M3_CKP	14.52		AP3 M3_DM[0]	26.63		BB1 M3_DM[1]	33.03	
AL9 M3_CKN	17.28		AK3 M3_DQ[0]	28.32		AY3 M3_DQ[8]	31.47	
AF3 M3_A[0]	25.53		AK2 M3_DQ[1]	28.30		AW5 M3_DQ[9]	28.48	
AG7 M3_A[1]	20.63		AM3 M3_DQ[2]	28.60		AV3 M3_DQ[10]	25.96	
AG5 M3_A[2]	26.51		AH2 M3_DQ[3]	28.37		AT3 M3_DQ[11]	24.18	
AG3 M3_A[3]	30.07		AR7 M3_DQ[4]	22.19		AW7 M3_DQ[12]	26.45	
AL7 M3_A[4]	21.37		AR2 M3_DQ[5]	26.11		BC3 M3_DQ[13]	31.77	
AH1 M3_A[5]	32.60		AT2 M3_DQ[6]	27.49		BB2 M3_DQ[14]	29.91	
AD2 M3_CKE	35.88		AR1 M3_DQ[7]	32.14		BC2 M3_DQ[15]	31.21	
AD3 M3_CSN	28.44		AR11 M3_DQSP[0]	18.70		AW9 M3_DQSP[1]	20.78	
			AR9 M3_DQSN[0]	18.01		AW11 M3_DQSN[1]	18.86	

4.3. Data (DQ/DQM/DQS) Bus



- All signals use point-to-point routing topology.
- DQS must be routed in pairs on the same layer to match the propagation delay and should use VSS instead of PWR as the reference plane for optimal current return.
- The total trace length ($L_0 + L_1$) is recommended to be no longer than 1200 mils to minimize attenuation.
- For each byte lane, the length matching for DQ(DM) to DQ(DM) ($L_p + L_0 + L_1$) should be within ± 5 ps.
- For each byte lane, the length matching for DQS to DQ(DM) ($L_p + L_0 + L_1$) should be within ± 10 ps.
- For each byte lane, the length matching for DQSp to DQSn ($L_p + L_0 + L_1$) should be within ± 0.5 ps.
- The spacing (air gap between traces' edge) between single-ended signals should be:
 - more than 1x the trace width in the PCB fan out section (L_0).
 - more than 2x the trace width in the PCB main section (L_1).
- The spacing (air gap between traces' edge) between differential (DQSp / DQSn) to other signals should be:
 - more than 1x the maximum intra-pair air gap or trace width in the PCB fan out section (L_0).
 - more than 3x the maximum intra-pair air gap or trace width in the PCB main section (L_1).

4.4. CA Bus



- All signals use point-to-point routing topology, except CKE and RSTn.
- Refer to the EVK layout for the placement of the CKE pull-down resistor and the RSTn pull-up resistor.
- CLK must be routed in pairs on the same layer to match the propagation delay and should use VSS instead of PWR as the reference plane for optimal current return.
- The total trace length (LO + L1) is recommended to be no longer than 1200 mils to minimize attenuation.
- For each CA channel, the length matching for CA(CS) to CA(CS) ($L_p + L_0 + L_1$) should be within ± 20 ps.
- For each CA channel, the length matching for CLK to CA(CS) ($L_p + L_0 + L_1$) should be within ± 25 ps.
- For each CA channel, the length matching for CLKp to CLKn ($L_p + L_0 + L_1$) should be within ± 0.5 ps.
- For each CA channel, the length matching for CLK to DQS ($L_p + L_0 + L_1$) should be within ± 60 ps.
- The spacing (air gap between traces' edge) between single-ended signals should be:
 - more than 1x the trace width in the PCB fan out section (L0).
 - more than 2x the trace width in the PCB main section (L1).
- The spacing (air gap between traces' edge) between differential signal (CLKp / CLKn) to other signals should be:
 - more than 1x the maximum intra-pair air gap or trace width in the PCB fan out section (L0).
 - more than 3x the maximum intra-pair air gap or trace width in the PCB main section (L1).

4.5. Board 6-Layers Stack up

- The Synaptics EVK (Evaluation Kit) board uses the most common 1.6mm PCB stack-up shown below.
- It is suggested that H34 be more than 5x the thickness of H23/H45 to avoid cross talk between signals on L3 and L4.
- High-speed data signals that run on the top layer(L3) should reference L2 (VSS).
- Low-speed CA signals that run on the bottom layer (L4) should reference L5 (PWR).

Layer	Info	Thickness
TOP	Er= 3.8 Solder Mask	0.333 oz + plating
L2	Er= 4.5	3.2 (mil)
L3	Er=4.5 Core	1 oz
L3		3.937 (mil)
L3		1 oz
L4	Er=4.6	38 (mil)
L4		1 oz
L4	Er=4.5 Core	3.937 (mil)
L5	Er= 4.5	1 oz
L5		3.2 (mil)
BOT	Er= 3.8 Solder Mask	0.333 oz + plating

- For customers concerned about EMI, Synaptics has explored another layout strategy which all signals running on the inner layers (L3 and L4):
 - L3 refers to L2 (VSS) as the reference plane
 - L4 refers to L5 (PWR) as the reference plane.
- With limited routing space, some data signals may need to run on L4, which can result in worse signal integrity (SI) due to plated through-hole (PTH) vias and long return paths. To minimize the side effects of routing on L4, the thickness of H34 (the dielectric layer between L3 and L4) needs to be reduced.

5. Revision History

Revision	Description
A	Initial Release
B	Release as public document.
C	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
D	Minor update to document title.
E	Minor update to latest template.

Downloaded by Anonymous () on 7 Jan 2026 17:43:32 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: 506-001390-01 Rev E

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.