



Application Note

Astra Machina Micro Evaluation Platform Kit General Design Layout Guidelines

Abstract: This application note provides PCB layout guidelines for the Astra™ Machina Micro Evaluation Platform Kit, with a focus on 6-layer HDI board design, high-speed signal integrity, and power supply decoupling. It covers best practices for differential pair routing, trace impedance control, length matching, via placement, and fanout strategies for FCCSP and WLCSP packages. These guidelines help ensure stable performance and design robustness across the platform.

Contents

1.	PCB Layout Guidelines.....	3
2.	Power Supply Guidelines.....	4
3.	6L HDI PCB Design Rules	5
3.1.	SMD Pad - FCCSP.....	5
3.2.	SMD Pad - WLCSP.....	5
3.3.	Via.....	5
3.4.	Spacing.....	5
4.	Chip Fanout.....	7
4.1.	FCCSP-122pin Fanout.....	7
5.	Revision History	8

Downloaded by Anonymous () on 21 Jun 2026 04:02:24 UTC

1. PCB Layout Guidelines

- Use $100\Omega \pm 10\%$ differential trace impedance for CSI. For USB2.0, $90\Omega \pm 10\%$ differential trace impedance is required. Other single-ended signals should maintain a characteristic trace impedance of $50\Omega \pm 10\%$.
- Match signal lengths within the same interface to less than 0.5mm (≈ 0.02 in). For high-speed differential pairs, maintain length matching within 0.25 mm (≈ 0.01 in) or less.
- Match the skew between any data lane and clock lane within ± 10 ps on both the package and PCB.
- Ensure a continuous solid ground plane beneath signal traces to provide a consistent return path. Avoid routing traces across different reference planes or over anti-pads.
- Minimize vias on differential traces. To keep symmetry within a differential pair, must place the same number of vias on each trace. Place ground vias close to the cluster of signal vias to ensure the shortest possible return path..
- To reduce crosstalk, take care of signal traces which are routed close to the data differential pairs. The minimum recommended spacing is 3x trace width for low-speed non-periodic signals, and 7x trace width for high-speed periodic signals. A continuous ground plane below the differential traces is required.

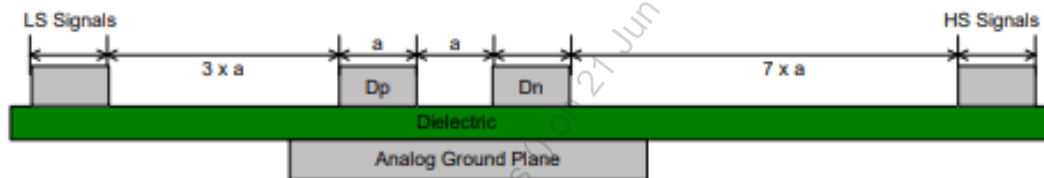


Figure 1. Recommended Spacing for Differential Pairs to Minimize Crosstalk from Adjacent Signals

- Avoid routing TX and RX pairs side-by-side on the same signal layer to reduce crosstalk.
- Total crosstalk from all aggressors must be accounted for and should remain below the -3 dB threshold up to the Nyquist frequency.

2. Power Supply Guidelines

- Allocating decoupling capacitors underneath main MCU is recommended to reduce ground bounce. 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 small-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 shorten return path and maximize their effectiveness. When connecting from the package pin to the bypass capacitors, use as wide as possible plane / trace to reduce inductive and resistive losses. Typical capacitor placement can be under the package (opposite side of the board) or on the same side but closed. 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 and ground planes should be continuous and provide a solid return path—not fragmented or broken into isolated sections. Special attention should be given to voids caused by vias. If vias disrupt the plane continuity, the layout must be adjusted to compensate for the loss of copper area and ensure sufficient effective plane width is maintained.

3. 6L HDI PCB Design Rules

3.1. SMD Pad - FCCSP

- Minimum pin pitch = 0.386 mm (15.2 mil)
- Footprint pad / Paste mask of pad = 12 mil / 9 mil (CSP)
- Solder mask of pad = 8 mil

3.2. SMD Pad – WLCSP

- Minimum pin pitch = 0.41 mm (16.1875 mil)
- Footprint pad / Paste mask of pad = 8 mil / 8 mil (CSP)
- Solder mask of pad = 7 mil

3.3. Via

- Minimum through hole pad / Drill size = 16 / 8 mil
- Minimum blind via pad / Drill size = 10 / 4 mil (1-2 Layer, 5-6 Layer)
- Minimum buried via pad / Drill size = 12 / 6 mil (2-5 Layer)

3.4. Spacing

- Minimum trace-to-trace = 3 mil
- Minimum trace-to-via = 3 mil
- Minimum trace-to-pad = 2.5 mil

The example stack-up/trace width and spacing of 6L HDI PCB is shown in [Figure 2](#) and [Table 1](#).

Stack-up					
	Type	Material	Parameters	Er	T (mil)
L1	Copper		0.33_PT_1OZ		0.036
	PP	S1000-2MB	1080(69)	3.99	0.0725
L2	Copper		0.33_PT_1OZ		0.035
	PP	S1000-2MB	3313(57)	4.32	0.0913
L3	Core	S1000-2M	1OZ	4.46	0.032
L4			1.1(With copper)		1.03
	PP	S1000-2MB	3313(57)	4.32	0.0919
L5	Copper		0.33_PT_1OZ		0.035
	PP	S1000-2MB	1080(69)	3.99	0.0721
L6	Copper		0.33_PT_1OZ		0.036

Press PCB Thickness : 1.52mm
 Finished PCB Thickness : 1.6 (+/-0.16) mm

Figure 2. Example Stack-up/trace width

Table 1. Stack-up/trace width and spacing of 6L HDI PCB

Number	Type	Control layer	Reference layer	Adjust Line Width (mil)	Adjust Spacing (mil)	Design Impedance (ohm)
1	Single-End	L1	L2	3.6	—	55
2	Single-End	L6	L5	3.6	—	55
3	Single-End	L3	L2/L5	3.4	—	55
4	Single-End	L4	L2/L5	3.4	—	55
5	Differential	L1	L2	3.2	3	90
6	Differential	L6	L5	3.2	3	90
7	Differential	L1	L2	3	4.2	100
8	Differential	L6	L5	3	4.2	100

4. Chip Fanout

- Both SR1xx FCCSP and WLCSP packages support full function fanout using a 6-layer HDI PCB design. A 4-layer non-HDI PCB design may also be used with the FCCSP-122 package, though with limited functionality.

Note: These guidelines provide recommended practices for optimal fanout layout and are not a set of strict requirements.

4.1. FCCSP-122pin Fanout

SR1xx MCU platforms using the FCCSP package can be implemented on a non-HDI PCB if interfaces from inner balls are not required. The impact interfaces are SDIO1, SWIRE, and part of xSPI for pSRAM. The example in [Figure 3](#) demonstrates a fanout layout for this configuration.

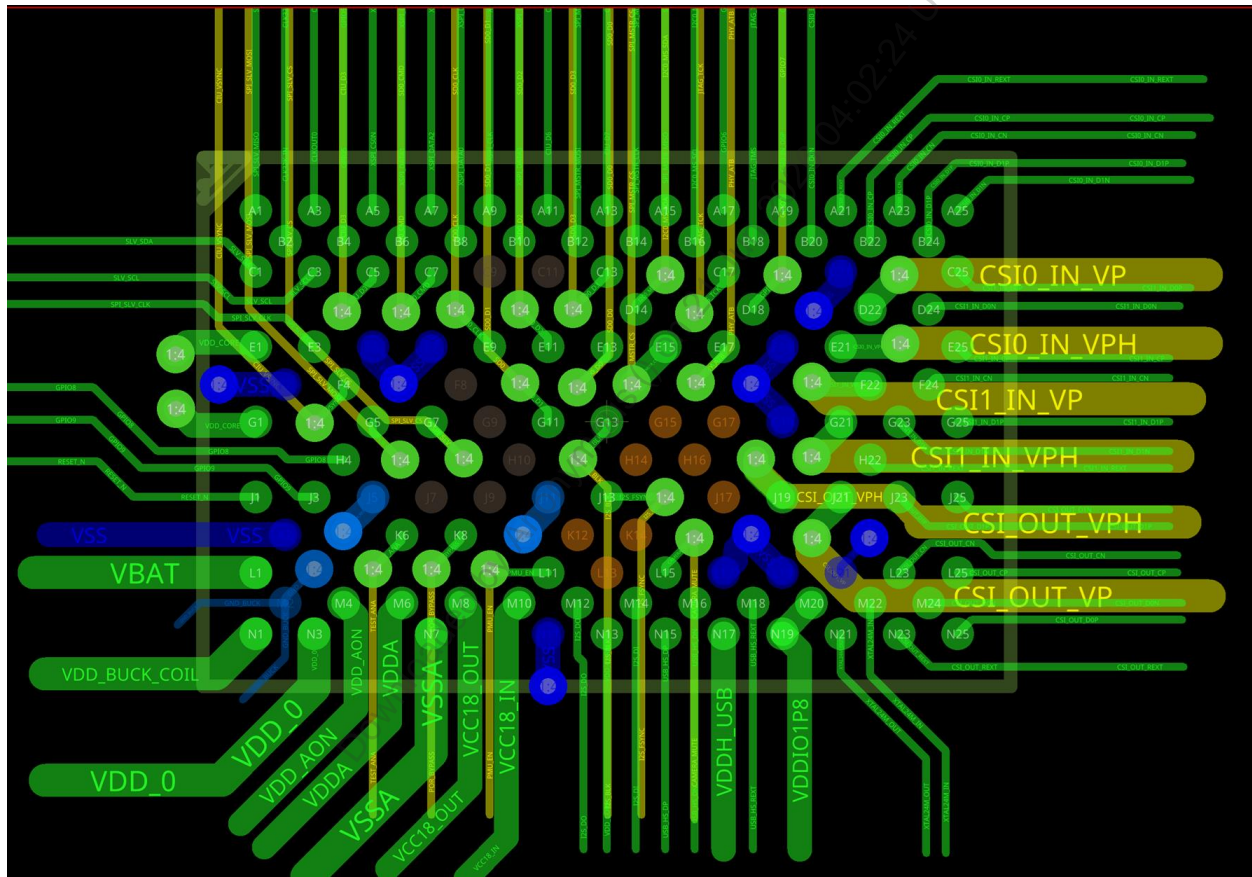


Figure 3. Impact interfaces: SDIO1, SWIRE, and part of xSPI for pSRAM

5. Revision History

Revision	Description
A	Initial Release

Downloaded by Anonymous () on 21 Jun 2026 04:02:24 UTC



Copyright

Copyright © 2025 Synaptics Incorporated. All Rights Reserved.

Trademarks

Synaptics, the Synaptics logo, and Astra Machina 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: 506-001622-01 Rev A

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.