

# UltraScale Architecture PCB Design

## *User Guide*

UG583 (1.13) August 15, 2018



## Revision History

The following table shows the revision history for this document.

Date	Version	Revision
08/15/2018	1.13	<p><b>Chapter 1:</b> Added description of slew rates after <a href="#">Table 1-1</a>. Added XCVU13P-FSGA2577 to <a href="#">Table 1-5</a>. In <a href="#">Table 1-8</a>, updated type, ESL maximum, upper ESR range, and suggested part number columns, and added a row for 0.47 <math>\mu</math>F. Updated capacitor part numbers after <a href="#">Figure 1-2</a>. Added FSVH1924, FSVH2104, and FSVH2892 packages to <a href="#">Table 1-21</a>.</p> <p><b>Chapter 2:</b> Removed LPDDR3 and LPDDR4 routing guidelines from <a href="#">General Memory Routing Guidelines</a>. Added note to <a href="#">Overview</a>. Updated <a href="#">Table 2-18</a>, <a href="#">Table 2-25</a>, <a href="#">Table 2-32</a>, <a href="#">Table 2-34</a>, <a href="#">Table 2-45</a>, <a href="#">Table 2-58</a>, and <a href="#">Table 2-69</a>. Added dm[9:0] to <a href="#">Table 2-19</a> and remove note 1. Added note about ODT pins and removed ODT0/1 from FPGA Pins column in <a href="#">Table 2-36</a>, <a href="#">Table 2-37</a>, <a href="#">Table 2-46</a>, and <a href="#">Table 2-47</a>.</p> <p><b>Chapter 3:</b> Added -2LI to first sentence in <a href="#">Separate V<sub>CCINT</sub> and V<sub>CCBRAM</sub>/V<sub>CCINT_IO</sub> (-1LI, -2LE)</a>. Added FSVE1156, FSVG1517, FSVE1156, FSVG1517, FSVE1156, FSVG1517, and FSVF1760 packages to <a href="#">Table 3-1</a> and <a href="#">Table 3-2</a>. Updated description of RF signal chain after <a href="#">Table 3-6</a>. Updated guard traces bullet in <a href="#">Isolation Recommendations</a>. Added <a href="#">Figure 3-3</a>, <a href="#">Figure 3-4</a>, <a href="#">Figure 3-13</a>, <a href="#">Figure 3-14</a>, and <a href="#">Figure 3-15</a>. Updated <a href="#">SYSREF</a>. Added sentence about separate power supplies for ADC and DAC supplies to <a href="#">Power Regulation and Decoupling for ADC and DAC Supplies</a>.</p> <p><b>Chapter 4:</b> Replaced VCCO_MIO0 with VCCO_PSIO[0]. Removed bullet about 2.00 k<math>\Omega</math> from <a href="#">SPI</a>.</p>
04/10/2018	1.12.1	<p><b>Appendix A:</b> Removed <math>\pm</math> sign from all table entries.</p>
04/09/2018	1.12	<p><b>Chapter 1:</b> Added <a href="#">PCB Decoupling Capacitors for Virtex UltraScale+ High Bandwidth Memory Devices</a>.</p> <p><b>Chapter 3:</b> Added new chapter.</p> <p><b>Chapter 4:</b> In last bullet of <a href="#">SD/SDIO</a>, changed CMD3 to DAT3. Updated second paragraph in <a href="#">Unconnected V<sub>CCO</sub> Pins</a>.</p> <p><b>Appendix B:</b> Added new appendix.</p> <p><b>Appendix C:</b> Added DS926 and PG269 to <a href="#">References</a>.</p>

Date	Version	Revision
02/22/2018	1.11	<p>Reorganized content between <a href="#">Chapter 1</a>, <a href="#">Chapter 4</a>, <a href="#">Chapter 5</a>, and <a href="#">Chapter 6</a>.</p> <p><b>Chapter 1:</b> Updated first paragraph in <a href="#">Recommended PCB Capacitors per Device</a> and added bullet item to resource usage list. Added ideal value of 0.47 <math>\mu</math>F to <a href="#">Table 1-12</a>. Added <a href="#">Recommended Decoupling Capacitor Quantities for Kintex UltraScale and Virtex UltraScale Devices</a>, <a href="#">Recommended Decoupling Capacitor Quantities for Kintex UltraScale+ and Virtex UltraScale+ Devices</a>, and <a href="#">Recommended Decoupling Capacitor Quantities for Zynq UltraScale+ Devices</a>. Moved and updated <a href="#">Power Supply Consolidation Solutions for Zynq UltraScale+ MPSoCs</a> from <a href="#">Chapter 6</a>. Updated <a href="#">Table 1-20</a>.</p> <p><b>Chapter 2:</b> Rewrote <a href="#">Overview</a>. In <a href="#">Table 2-1</a>, updated descriptions of L2, L4, L13, and L15 layers. In <a href="#">General Memory Routing Guidelines</a>, removed description of maximum routing length in guideline 2, and updated guidelines 5, 6, 14, and 23. Added <a href="#">Figure 2-13</a>. Reversed order of <a href="#">PCB Guidelines for DDR3/3L SDRAM (PL and PS)</a> and <a href="#">PCB Guidelines for DDR4 SDRAM (PL and PS)</a> sections. Removed <a href="#">Table 2-11: PCB Guidelines for DDR4 SDRAM</a> and <a href="#">Table 2-22: PCB Guidelines for DDR3 SDRAM</a>. In <a href="#">Table 2-20</a>, <a href="#">Table 2-22</a>, <a href="#">Table 2-11</a>, <a href="#">Table 2-15</a>, <a href="#">Table 2-27</a> to <a href="#">Table 2-29</a>, updated L0 and L1 trace lengths, and added note 2. In <a href="#">Table 2-21</a>, <a href="#">Table 2-13</a>, and <a href="#">Table 2-26</a>, updated L0 and L1 trace lengths, and note 2. Updated skew constraints for signal group data to DQS in <a href="#">Table 2-24</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-31</a>, and <a href="#">Table 2-34</a>. Updated note after <a href="#">Table 2-25</a>. Added alert_n to <a href="#">Table 2-9</a>. Added note to <a href="#">Fly-by and Clamshell Topologies</a>. Added note to <a href="#">Table 2-10</a>. Added alert_n. Updated <a href="#">Figure 2-23</a>. In <a href="#">Table 2-12</a> and <a href="#">Table 2-14</a>, updated L0 and L1 trace lengths, and added note 1. Updated note after <a href="#">Table 2-18</a>, <a href="#">Table 2-32</a>, <a href="#">Table 2-35</a>, <a href="#">Table 2-78</a>, <a href="#">Table 2-85</a>, and <a href="#">Table 2-90</a>. Added <a href="#">Table 2-46</a>. Added Signal Segment column to <a href="#">Table 2-69</a>. Updated L1 trace length in <a href="#">Table 2-71</a>, <a href="#">Table 2-72</a>, <a href="#">Table 2-74</a> to <a href="#">Table 2-76</a>, <a href="#">Table 2-83</a>, <a href="#">Table 2-87</a>, and <a href="#">Table 2-88</a>. Added VREFCA and VREFDQ. Removed instances of <i>length constraints</i> from introductory text in <a href="#">RLDRAM 3 Memory Routing Constraints</a>, including <a href="#">Table 2-77</a>. Removed skew constraints (mil) column from <a href="#">Table 2-78</a>. Added <a href="#">PCB Guidelines for LPDDR4 Memories without ECC (PS)</a>, <a href="#">PCB Guidelines for LPDDR4 Memories with ECC (PS)</a>, and <a href="#">PCB Guidelines for LPDDR3 SDRAM (PL and PS)</a> from <a href="#">Chapter 4</a>.</p> <p><b>Chapter 4:</b> Renamed chapter title. Removed section <i>Design Example for x32 LPDDR4 with ECC</i>. In <a href="#">CAN</a>, changed PCB and package skew to <math>\pm 100</math> ps. Added bullet about series resistor to <a href="#">eMMC</a>. In <a href="#">Standard and High-Speed SDR Interfaces</a>, updated heading and changed PCB and package skew to <math>\pm 100</math> ps. Updated heading in <a href="#">HS200 (200 MHz) and High-Speed DDR Interfaces</a>. Removed TDO from <a href="#">JTAG</a>. In <a href="#">SDR Mode</a>, changed PCB and package skew to <math>\pm 100</math> ps. In <a href="#">DDR Mode (100 MHz)</a>, updated second bullet and added third bullet. In <a href="#">PS Reset (External System Reset and POR Reset)</a>, replaced VCCO_PSIO[3] with VCCO_MIO0 in first bullet, and added two new bullets. Updated first and second bullets in <a href="#">PS_INIT_B</a>, <a href="#">PS_PROG_B</a>, and <a href="#">PS_DONE</a>. Updated fifth bullet in <a href="#">QSPI</a>. Updated first bullet in <a href="#">Real-Time Clock</a>. In <a href="#">SD/SDIO</a>, updated first bullet, removed bullet about PCB and package delay skew, and added bullet about 10 k<math>\Omega</math> pull-up resistor. In <a href="#">SPI</a>, <a href="#">Triple Time Counter</a>, and <a href="#">Watchdog Timer</a>, changed PCB and package skew to <math>\pm 100</math> ps.</p>

Date	Version	Revision
02/22/2018	1.11 (Cont'd)	<p>Removed Trace B section. In <a href="#">UART</a>, changed MIO trace delay to 1.30 ns and removed bullet about matching TX line. In <a href="#">USB 2.0</a>, updated PCB and package delays, and added bullet about 30Ω series resistor. Added recommendation to tie pins to ground in <a href="#">Table 4-2</a>.</p> <p><a href="#">Chapter 6</a>: Renamed chapter title. Updated <a href="#">7. Memory Interface PCB Routing</a>, including <a href="#">Table 6-6</a>.</p> <p><a href="#">Chapter 7</a>: Added -2E speed grade to <a href="#">Table 7-1</a>. Added XCZU2EG/XCZU3EG to first sentence in <a href="#">4. -3 Speed Migration</a>. Updated note in <a href="#">5. VCU Migration</a>. Updated <a href="#">Figure 7-7</a>. Removed <a href="#">Figure 6-8: VCCINT_VCU Connection Options for CG/EG/EV Migration (PCB)</a>. Updated first two sentences in <a href="#">Example: Schematic/PCB Options to Enable VCU Migration from EV to CG/EG Devices</a>. Removed section <a href="#">7. Application Processor and Real-Time Processor Migration</a>.</p> <p><a href="#">Chapter 9</a>: Replaced “Kintex UltraScale and Virtex UltraScale FPGAs” with “UltraScale architecture-based devices.” In <a href="#">Table 9-2</a>, updated ADS and SIwave vendors, replaced Specctraquest Power Integrity with Sigrity, and removed Speed 2000, PowerSI, PowerDC.</p> <p><a href="#">Appendix A</a>: Added ± to all entries in <a href="#">Table A-1</a>, <a href="#">Table A-9</a> to <a href="#">Table A-13</a>, and <a href="#">Table A-15</a>. Updated entries in <a href="#">Table A-3</a> to <a href="#">Table A-8</a>.</p>
01/30/2017	1.10	<p>Replaced “midpoint” with “midrange” throughout.</p> <p><a href="#">Chapter 1</a>: Added <a href="#">Equation 1-1</a>. Added RLD1517 and RLF1924 packages for XQKU115 devices to <a href="#">Table 1-2</a>. Updated <math>V_{CCINT}</math> values in <a href="#">Table 1-21</a>.</p> <p><a href="#">Chapter 2</a>: Updated guideline 12 in <a href="#">General Memory Routing Guidelines</a>. Added note after <a href="#">Figure 2-33</a>, <a href="#">Figure 2-34</a>, and <a href="#">Figure 2-35</a>.</p> <p><a href="#">Chapter 6</a>: Replaced HR I/O with HD I/O in <a href="#">Table 4-1</a>. Added <a href="#">Migration between UltraScale and UltraScale+ FPGAs</a>.</p> <p><a href="#">Chapter 4</a>: Replaced HR I/O with HD I/O in <a href="#">Table 5-1</a>. Added <a href="#">V<sub>CCINT_VCU</sub> Decoupling Capacitor Recommendations</a>. Added <math>V_{CCO\_PSDDR}</math> to <a href="#">Table 5-3</a>. Added description of <math>V_{CCINT\_VCU}</math> to <a href="#">Video Codec Unit (EV Devices Only)</a>. Updated <a href="#">PCB Guidelines for DDR3 and DDR4 SDRAM</a>. Added DQ bus width to <a href="#">Figure 5-8</a>. Updated third bullet about PCB and package delay skew in <a href="#">SD/SDIO</a>. Updated list of filters for PS_MGTRAVCC and PS_MGTRAVTT in <a href="#">Table 4-2</a>.</p> <p><a href="#">Chapter 7</a>: Added new chapter.</p> <p><a href="#">Appendix A</a>: Replaced Mb/s with MHz as units for memory component rating in <a href="#">Table A-9</a> to <a href="#">Table A-15</a>.</p> <p><a href="#">Appendix C</a>: Added UG1085, WP482, UG580, UG1075, PG213, PG203, and PG212 to <a href="#">References</a>.</p>

Date	Version	Revision
11/18/2016	1.9	<p><b>Chapter 1:</b> Updated chapter title. Updated resource usage bulleted list in <a href="#">Recommended PCB Capacitors per Device</a>. Added <a href="#">Step Load Assumptions</a>, including <a href="#">Table 1-1</a>, and <a href="#">Maximum Current Draw for V<sub>CCINT</sub> in Virtex UltraScale+ Devices</a>. Added RBA676 and RFA1156 packages to <a href="#">Table 1-2</a>.</p> <p><b>Chapter 2:</b> Updated guideline 14 in <a href="#">General Memory Routing Guidelines</a>. Added <a href="#">Figure 2-12</a> and <a href="#">Figure 2-15</a>. Distinguished between fly-by and clamshell topology in <a href="#">Overview</a>. Added CS<sub>0_n</sub>, CS<sub>1_n</sub> to <a href="#">Table 2-9</a>. Added <a href="#">Fly-by and Clamshell Topologies</a> and <a href="#">Utilizing Address Mirroring to Ease Clamshell Routing</a>. Updated first sentence in <a href="#">DDR4 SDRAM Address, Command, and Control Fly-by and Clamshell Topologies</a>. Added “fly-by” to title of <a href="#">Table 2-11</a>. Added <a href="#">Figure 2-23</a>, <a href="#">Figure 2-25</a>, <a href="#">Table 2-12</a>, and <a href="#">Table 2-14</a>. Updated <a href="#">DDR4 SDRAM Data Signals Point-to-Point for Fly-by and Clamshell Configurations</a>.</p> <p><b>Chapter 5:</b> Updated recommendations in <a href="#">15. Power Supplies and Thermal Considerations</a>.</p> <p><b>Chapter 6:</b> Updated chapter title. Added notes 4 through 7 to <a href="#">Table 4-1</a>. Added XCVU9P-FSGD2104, XCVU11P-FLGD2104, and XCVU13P-FIGD2104 devices, and notes 4 through 7 to <a href="#">Table 4-2</a>. Added <a href="#">Table 6-4</a>. Removed sentence about UltraScale+ FPGAs having stricter ESD handling requirements than UltraScale FPGAs from <a href="#">10. ESD Requirements</a>.</p> <p><b>Chapter 4:</b> Updated chapter title. Updated <a href="#">Table 5-1</a>. Added <a href="#">LPDDR4 Address Copy, page 142</a>, <a href="#">LPDDR4 Address Copy, page 149</a>, and <a href="#">LPDDR3 Address Copy, page 161</a>. Removed bullet about PCB and package delay skew from <a href="#">SD/SDIO</a>.</p>
07/18/2016	1.8	<p><b>Chapter 1:</b> Added note 2 to <a href="#">Table 1-2</a> and <a href="#">Table 1-3</a>.</p> <p><b>Chapter 2:</b> Clarified guideline 5 in <a href="#">General Memory Routing Guidelines</a>. Replaced CD with CK in <a href="#">Figure 2-6</a> title. Added important note before <a href="#">Figure 2-11</a>. Replaced reset_b with reset_n throughout. Updated <a href="#">reset_n, page 77</a> and <a href="#">reset_n, page 69</a>, including removal of <a href="#">Figure 2-17: Self-Refresh Termination During FPGA Power-Down or Reconfiguration for CKE in DDR3 SDRAM</a> and <a href="#">Figure 2-22: Self-Refresh Termination during FPGA Power-Down or Reconfiguration for CKE in DDR4 SDRAM</a>. Removed note about package delay (P0) from <a href="#">Table 2-24</a>, <a href="#">Table 2-25</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-18</a>, <a href="#">Table 2-31</a>, <a href="#">Table 2-32</a>, <a href="#">Table 2-34</a>, <a href="#">Table 2-35</a>, <a href="#">Table 2-78</a>, <a href="#">Table 2-85</a>, and <a href="#">Table 2-90</a>. Added we, ras, cas, and TEN to <a href="#">Table 2-9</a>. Removed L2 column from <a href="#">Table 2-26</a>. In <a href="#">Table 2-27</a>, removed L2 column, and updated L1 parameters. Added <a href="#">Table 2-28</a>. Replaced dm with dk/qk in <a href="#">Table 2-75</a>.</p> <p><b>Chapter 5:</b> Added recommendation to use Delphi thermal model to <a href="#">15. Power Supplies and Thermal Considerations</a>.</p> <p><b>Chapter 6:</b> Removed PCI Express from <a href="#">UltraScale+ FPGA Migration Checklist</a>.</p> <p><b>Chapter 4:</b> Added Zynq UltraScale+ MPSoCs and V<sub>CCINT</sub>/V<sub>CCINT_IO</sub> at 0.47 μF to <a href="#">Table 5-1</a>. Added note 2 to <a href="#">Table 5-3</a>. Added <a href="#">Video Codec Unit (EV Devices Only)</a>. Added reset_n to <a href="#">Table 5-5</a> and <a href="#">Table 5-14</a>. Updated <a href="#">Figure 5-3</a>. In <a href="#">Table 5-11</a> and <a href="#">Table 5-13</a>, removed note about package delay (P0), updated skew constraints (mil), and removed CK to DQS (A)/(B) row. Updated L2 spacing to other group signals in <a href="#">Table 5-4</a> to <a href="#">Table 5-7</a>, and <a href="#">Table 5-11</a>. Updated <a href="#">Figure 5-4</a>. Added <a href="#">PCB Guidelines for LPDDR3 SDRAM</a>. Updated recommendations for PS_MGTRREF and PS_MGTRAVTT in <a href="#">Table 4-2</a>.</p> <p><b>Appendix A:</b> Updated second paragraph at start of appendix. Updated cells in <a href="#">Table A-1</a> to <a href="#">Table A-4</a>, added notes to all tables, and updated titles of <a href="#">Table A-2</a> and <a href="#">Table A-4</a>.</p>

Date	Version	Revision
04/26/2016	1.7	<p><b>Chapter 2:</b> In <a href="#">General Memory Routing Guidelines</a>, updated guidelines 10 and 21, and added guidelines 11 and 23. Updated paragraph after <a href="#">Table 2-4</a>. Removed DIMM from <a href="#">Figure 2-6</a>. Added introductory sentences for <a href="#">Figure 2-17</a> and <a href="#">Figure 2-28</a>. In <a href="#">Table 2-19</a>, updated descriptions of cke[1:0] and reset_b. Added heading to and updated <a href="#">reset_n</a>. Updated introductory sentence before <a href="#">Figure 2-30</a> and <a href="#">Figure 2-26</a>. Added note 3 to <a href="#">Table 2-25</a>, <a href="#">Table 2-18</a>, and <a href="#">Table 2-32</a>. In <a href="#">Table 2-9</a>, updated descriptions of cke and reset_b. Added notes about pin rules and swapping after <a href="#">Table 2-19</a> and <a href="#">Table 2-9</a>. Added <math>\pm 10\%</math> to all entries for <math>Z_{DIFF}</math> in <a href="#">Table 2-13</a> and <a href="#">Table 2-15</a>.</p> <p><b>Chapter 5:</b> Updated <a href="#">15. Power Supplies and Thermal Considerations</a>.</p> <p><b>Chapter 4:</b> Added <a href="#">V<sub>CC_PSDDR_PLL</sub> Supply</a>. Updated <a href="#">PCB Guidelines for DDR3 and DDR4 SDRAM</a>. Updated L2 column in <a href="#">Table 5-5</a>, <a href="#">Table 5-6</a>, <a href="#">Table 5-7</a>, <a href="#">Table 5-8</a>, <a href="#">Table 5-9</a>, <a href="#">Table 5-8</a>, and <a href="#">Table 5-11</a>. Updated signal names and connections in <a href="#">Figure 5-3</a>, <a href="#">Figure 5-4</a>, and <a href="#">Figure 5-8</a>. Made CKE routing pull up to <math>V_{TT}</math> in <a href="#">Figure 5-3</a>. Updated DQ width in <a href="#">Figure 5-4</a>. Updated <a href="#">Table 5-11</a> and <a href="#">Table 5-14</a>. Added channel B to ECC unit in <a href="#">Figure 5-3</a> and <a href="#">Figure 5-4</a>. Made ending via pull up to <math>V_{TT}</math> in <a href="#">Figure 5-4</a>. Added channel A label to <a href="#">Figure 5-5</a>. Updated ECC unit in <a href="#">Figure 5-6</a>. Updated trace lengths in <a href="#">Table 5-5</a> through <a href="#">Table 5-11</a>. Updated L3 <math>Z_{diff}</math> in <a href="#">Table 5-9</a>. Replaced DRAM with memory device in <a href="#">Figure 5-7</a> and <a href="#">Figure 5-8</a>. Updated L1 spacing in <a href="#">Table 5-10</a>. Updated <a href="#">Table 5-13</a>. Added <a href="#">PCB Guidelines for the PS Interface in the Zynq UltraScale+ MPSoC</a>.</p> <p><b>Appendix A:</b> Added notes to <a href="#">Table A-3</a> and <a href="#">Table A-4</a>.</p> <p><b>Appendix C:</b> Added <a href="#">Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics (DS925)</a> to <a href="#">References</a>.</p>
02/04/2016	1.6	<p><b>Chapter 1:</b> Updated paragraph after resource usage bullets in <a href="#">Recommended PCB Capacitors per Device</a>. In <a href="#">Table 1-2</a>, added SFVA784 package for XCKU035 and XCKU040, and FFVA1156 package for XCKU095. Added footnote with definition of electrical inch to <a href="#">0805 Ceramic Capacitor</a>.</p> <p><b>Chapter 2:</b> Updated <a href="#">Overview</a>. Updated note in <a href="#">Table 2-1</a>. Significant revision to the <a href="#">General Memory Routing Guidelines</a> section including adding <a href="#">Table 2-2</a> through <a href="#">Table 2-4</a> and <a href="#">Figure 2-4</a> to <a href="#">Figure 2-6</a>. Added the <a href="#">Adjusting for Different Stack-Ups</a> section. Updated cke[1:0] and reset_b descriptions in <a href="#">Table 2-19</a> and <a href="#">Table 2-9</a>. Added <a href="#">Figure 2-17</a>, <a href="#">Figure 2-28</a>, <a href="#">Figure 2-21</a>, and <a href="#">Figure 2-21</a>. Replaced strobe with dqs in <a href="#">Table 2-22</a>, <a href="#">Table 2-15</a>, and <a href="#">Table 2-29</a>. In <a href="#">Table 2-24</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-31</a>, and <a href="#">Table 2-34</a> replaced strobe with DQS and clock with CK and updated note 1. Added new rows and updated note 1 and 2 in <a href="#">Table 2-25</a>, <a href="#">Table 2-18</a>, <a href="#">Table 2-32</a>, and <a href="#">Table 2-35</a>. In <a href="#">Table 2-31</a>, updated CK to DQS constraints and added notes 2 and 5. In <a href="#">Table 2-34</a>, updated CK to DQS constraints and added note 2. Updated note 1 in <a href="#">Table 2-78</a>, <a href="#">Table 2-85</a>, and <a href="#">Table 2-90</a>. Removed <a href="#">PCB Guidelines for LPDDR3 Memories</a> section.</p> <p><b>Chapter 6:</b> In <a href="#">Table 4-1</a>, removed XCKU7P in all packages, and added XCKU3P-FFVB676, XCKU3P-FFVD900, XCKU5P-SFVB784, XCKU5P-FFVA676, and XCKU5P-FFVD900. Updated packages for XCVU9P, XCVU11P, and XCVU13P in <a href="#">Table 4-2</a>. Added A676 and A1156 packages to <a href="#">Table 6-1</a>.</p> <p><b>Chapter 4:</b> Added XCZU15EG to <a href="#">Table 5-1</a>. In <a href="#">Table 5-3</a>, replaced VCCPSGTA with PS_MGTRAVCC, VCCPSDDR with VCC_PSINTFP_DDR, and added underscores to other column headings. Moved <a href="#">PCB Guidelines for LPDDR4 Memories without ECC</a> from <a href="#">Chapter 2</a> and expanded section. Added <a href="#">PCB Guidelines for LPDDR4 Memories with ECC</a>.</p>
11/24/2015	1.5	<p>Added UltraScale+ FPGA (<a href="#">Chapter 6</a>) and Zynq UltraScale+ MPSoC information (<a href="#">Chapter 4</a>).</p>

Date	Version	Revision
09/16/2015	1.4	<p><b>Chapter 1:</b> Added V<sub>CC0</sub> (Bank 0) column to <a href="#">Table 1-2</a> and <a href="#">Table 1-3</a>.</p> <p><b>Chapter 2:</b> Added QDR-IV SRAM to chapter title and <a href="#">Overview</a>. In <a href="#">General Memory Routing Guidelines</a>, updated second, fifth, seventh, and eighth guidelines, and added third, fourth, ninth and tenth guidelines. Updated <a href="#">Figure 2-10</a>. Added reset_b to control signals in <a href="#">Table 2-19</a> and <a href="#">Table 2-9</a>. Updated L3 trace length in <a href="#">Table 2-20</a>, <a href="#">Table 2-21</a>, <a href="#">Table 2-11</a>, <a href="#">Table 2-13</a>, <a href="#">Table 2-72</a>, <a href="#">Table 2-74</a>, and <a href="#">Table 2-80</a>. Removed selected L2 breakouts from <a href="#">Figure 2-27</a>, <a href="#">Figure 2-29</a>, <a href="#">Figure 2-20</a>, <a href="#">Figure 2-24</a>, <a href="#">Figure 2-59</a>, and <a href="#">Figure 2-67</a>. Updated table title and maximum length/delay constraints column in <a href="#">Table 2-23</a>, <a href="#">Table 2-16</a>, <a href="#">Table 2-30</a>, <a href="#">Table 2-33</a>, <a href="#">Table 2-77</a>, <a href="#">Table 2-84</a>, <a href="#">Table 2-89</a>, and <a href="#">Table 2-58</a>. Added skew constraints (mil) column and added table note to <a href="#">Table 2-24</a>, <a href="#">Table 2-25</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-18</a>, <a href="#">Table 2-31</a>, <a href="#">Table 2-32</a>, <a href="#">Table 2-34</a>, <a href="#">Table 2-35</a>, <a href="#">Table 2-78</a>, <a href="#">Table 2-85</a>, <a href="#">Table 2-90</a>, and <a href="#">Table 2-59</a>. Replaced device with FPGA in <a href="#">DDR3 SDRAM Address, Command, and Control Fly-by Termination</a>. Updated <a href="#">Table 2-17</a> title. Updated <a href="#">Table 2-32</a> title and data to strobe skew constraints. Added <a href="#">Table 2-35</a>. Updated <a href="#">Figure 2-56</a>. Removed parenthetical comments from descriptions in <a href="#">Table 2-70</a> and <a href="#">Table 2-79</a>. Added “qvid to qk_p/n” row to <a href="#">Table 2-78</a>. Updated <a href="#">Figure 2-68</a>. Replaced K_P/N with K and K_B in <a href="#">QDR II+ SRAM Clock (K, K_B, and BWS) T-Branch Routing and Termination</a>. Replaced k_p/k_n with k/k_b in <a href="#">QDR II+ SRAM Clock and Data Signals (d/k/k_b) Point-to-Point Routing</a>. Replaced cq_p/cq_n with cq/cq_b in <a href="#">QDR II+ SRAM Clock and Data Signals (q/cq/cq_b) Point-to-Point Routing</a>. Added signal groups “k to k_b” and “q to q_b” to <a href="#">Table 2-85</a>. Added signal group “ck_p to ck_n” to <a href="#">Table 2-59</a>. Added <a href="#">PCB Guidelines for QDR-IV SRAM</a>.</p> <p><b>Chapter 5:</b> Replaced <i>pin</i> with <i>footprint</i> throughout. Updated <a href="#">1. Footprint Compatibility between Packages</a>. Replaced XCVU160 with XCVU190 in <a href="#">Example, page 195</a>. Added <a href="#">MGTRREF</a> and <a href="#">MGTAVTTRCAL</a>.</p> <p><b>Appendix A:</b> Added new appendix.</p>
03/09/2015	1.3	<p><b>Chapter 1:</b> In <a href="#">Table 1-2</a>, added XCKU035 and XCKU100 devices and removed note about 0402 2.2 <math>\mu</math>F capacitors. Added <a href="#">Table 1-3</a>.</p> <p><b>Chapter 2:</b> Removed LPDDR2 throughout. Updated fourth and fifth guidelines in <a href="#">General Memory Routing Guidelines</a>. Updated L0 trace length to 0.0~0.8/1.2 in Impedance, Length, and Spacing Guidelines tables. Updated L1 trace length to Address L1 + 0.25 in <a href="#">Table 2-21</a>, <a href="#">Table 2-13</a>, and <a href="#">Table 2-26</a>. Updated address/command/control signal group in <a href="#">Table 2-23</a>, <a href="#">Table 2-16</a>, <a href="#">Table 2-30</a>, and <a href="#">Table 2-33</a>. Updated skew constraints in <a href="#">Table 2-24</a>, <a href="#">Table 2-25</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-18</a>, <a href="#">Table 2-31</a>, <a href="#">Table 2-34</a>, <a href="#">Table 2-78</a>, and <a href="#">Table 2-85</a>. Added note about skew specifications to <a href="#">Table 2-24</a>, <a href="#">Table 2-25</a>, <a href="#">Table 2-17</a>, <a href="#">Table 2-18</a>, <a href="#">Table 2-34</a>, <a href="#">Table 2-78</a>, <a href="#">Table 2-85</a>, and <a href="#">Table 2-59</a>. Updated table note referring to item 5 of <a href="#">General Memory Routing Guidelines</a> throughout. Updated skew constraints for “data to strobe” signal group in <a href="#">Table 2-17</a> and <a href="#">Table 2-34</a>. Added row for differential signals to <a href="#">Table 2-78</a>. Updated all trace lengths in <a href="#">Table 2-80</a>. Updated L1 trace length to 1.0~4.0 in <a href="#">Table 2-83</a>. Updated <a href="#">Figure 2-56</a>. Updated title of <a href="#">Figure 2-59</a>. Removed “R = 50<math>\Omega</math>” label from <a href="#">Figure 2-65</a>. Added pull-up resistor to CA line in <a href="#">Figure 2-66</a>. In <a href="#">Figure 2-52</a> and <a href="#">Figure 2-53</a>, replaced V<sub>TT</sub> with V<sub>DDQ</sub> and changed pull-up resistance from 120<math>\Omega</math> to 60<math>\Omega</math>. Removed “spacing in addr/cmd/ctrl” row from <a href="#">Table 2-56</a>.</p> <p><b>Chapter 5:</b> Added new chapter.</p> <p><b>Chapter 9:</b> Updated second paragraph of <a href="#">Input Thresholds</a>. Updated second paragraph of <a href="#">V<sub>REF</sub> Stabilization Capacitors</a>.</p> <p><b>Appendix C:</b> Updated list of documents in <a href="#">References</a>.</p>

Date	Version	Revision
12/01/2014	1.2	<p><b>Chapter 1:</b> Removed recommendation asking customers to run their own power integrity simulations. In <a href="#">Table 1-2</a>, replaced “should” with “must” in notes 1 and 2 and added note 4. Removed <a href="#">Table 1-2: Virtex UltraScale Devices Power Supply Decoupling Capacitors</a>. In <a href="#">Table 1-12</a>, added row for 470 <math>\mu\text{F}</math>, and updated ESL maximum and suggested part numbers for 100 <math>\mu\text{F}</math>, 47 <math>\mu\text{F}</math>, and 4.7 <math>\mu\text{F}</math> capacitors. In <a href="#">PCB Bulk Capacitors</a>, added 470 <math>\mu\text{F}</math> to first paragraph and replaced niobium oxide with polymer aluminum in second paragraph.</p> <p><b>Chapter 2:</b> Added sentence about inner signal layer propagation time to first paragraph of <a href="#">Reference Stackup</a>. Updated Thickness column in <a href="#">Table 2-1</a>. Added second, fifth, and sixth guidelines and <a href="#">Figure 2-3</a> to <a href="#">General Memory Routing Guidelines</a>. Replaced “length matching” with “skew” throughout. Changed units in skew constraints column from mils to picoseconds and updated notes in all skew constraints tables. Replaced “routing length” with “flight times” in Important notes after skew constraints tables. Removed <a href="#">DDR3 SDRAM ODT Settings</a> and <a href="#">DDR4 SDRAM ODT Settings</a> sections. Updated L0 trace length to 0~0.6/1.2 and added table note in all impedance, length, and spacing guidelines tables. Added <a href="#">DDR3/DDR4 UDIMM/RDIMM/SODIMM Routing Guidelines (PL and PS)</a>. Replaced “depth” with “width” in <a href="#">RLDRAM 3 Memory Topology and Routing Guidelines for Clamshell and Fly-by Configurations</a>. Updated maximum PCB via count to 6 in <a href="#">Table 2-71</a> and <a href="#">Table 2-72</a>. Updated L2 trace length in <a href="#">Table 2-72</a> to <a href="#">Table 2-76</a>. Updated title of <a href="#">Figure 2-62</a>. Added <a href="#">Figure 2-63</a>. In <a href="#">Table 2-76</a>, added <math>Z_0</math> parameter and removed <math>Z_{diff}</math> and differential trace row. Updated DATA IN and DATA OUT ports in <a href="#">Figure 2-65</a>. Added <a href="#">Figure 2-66</a>. Added <a href="#">QDR II+ SRAM Topology and Routing Guidelines for T-Branch Configuration</a>. Removed “QDR II+ SRAM Clock (k and cq) Differential Point-to-Point Routing” section. Added <a href="#">QDR II+ SRAM Clock (K, K_B, and BWS) T-Branch Routing and Termination</a>. Added clock to <a href="#">QDR II+ SRAM Clock and Data Signals (d/k/k_b) Point-to-Point Routing</a> and <a href="#">QDR II+ SRAM Clock and Data Signals (q/cq/cq_b) Point-to-Point Routing</a>. Removed resistor from <a href="#">Figure 2-72</a>. Updated <a href="#">Table 2-85</a>. Removed “clock” from <a href="#">LPDDR3 Memory Address, Command, and Control Point-to-Point Routing</a>. Added Important note after <a href="#">Table 2-59</a>.</p> <p><b>Chapter 8:</b> Removed VRN from <a href="#">Unidirectional Topographies and Termination</a>.</p> <p><b>Chapter 9:</b> Replaced “inductance” with “impedance” in <a href="#">Figure 9-5</a>.</p>
08/28/2014	1.1	<p><b>Chapter 1:</b> Replaced 0603 capacitor with 0805 capacitor throughout. In <a href="#">Recommended PCB Capacitors per Device</a>, added alternate network example, Recommended icon, and <a href="#">Table 1-2</a>. Updated <a href="#">Table 1-2</a>. In <a href="#">Table 1-2</a>, updated body size and voltage rating of 4.7 <math>\mu\text{F}</math> capacitor and removed note 3 about ESR. Updated first paragraph in <a href="#">PCB Bulk Capacitors</a>.</p> <p><b>Chapter 2:</b> Added new chapter.</p> <p><b>Chapter 9:</b> In <a href="#">Noise Limits</a>, updated first paragraph and updated low-frequency variance bullet. Updated first paragraph in <a href="#">Role of Inductance</a>. Updated description of tantalum capacitors in <a href="#">Capacitor Parasitic Inductance</a>. Updated <a href="#">Capacitor Mounting Inductance</a>. Updated second trade-off bullet in <a href="#">FPGA Mounting Inductance</a>. Updated terminology in <a href="#">Noise Magnitude Measurement</a>.</p> <p><b>Appendix C:</b> Added <a href="#">UltraScale Architecture GTY Transceivers User Guide (UG578)</a> to <a href="#">References</a>.</p>
12/10/2013	1.0	Initial Xilinx release.



# Table of Contents

Revision History .....	2
<b>Chapter 1: Power Distribution System in UltraScale Devices</b>	
<b>Introduction to UltraScale Architecture</b> .....	15
<b>Introduction</b> .....	16
<b>PCB Decoupling Capacitors</b> .....	16
Recommended PCB Capacitors per Device .....	16
Recommended Decoupling Capacitor Quantities for Kintex UltraScale and Virtex UltraScale Devices .....	18
Recommended Decoupling Capacitor Quantities for Kintex UltraScale+ and Virtex UltraScale+ Devices .....	21
PCB Decoupling Capacitors for Virtex UltraScale+ High Bandwidth Memory Devices .....	23
Recommended Decoupling Capacitor Quantities for Zynq UltraScale+ Devices .....	24
V <sub>CC_PSDDR_PLL</sub> Supply .....	27
Video Codec Unit (MPSoC EV Devices Only) .....	28
<b>Capacitor Specifications</b> .....	30
<b>Capacitor Consolidation Rules</b> .....	32
PCB Capacitor Placement and Mounting Techniques .....	32
<b>Transceiver PCB Routing Guidelines</b> .....	33
<b>Power Supply Consolidation Solutions for Zynq UltraScale+ MPSoCs</b> .....	33
Power Supply Consolidation Solutions .....	35
<b>Maximum Current Draw for V<sub>CCINT</sub> in Virtex UltraScale+ Devices</b> .....	45
<b>Chapter 2: PCB Guidelines for Memory Interfaces</b>	
<b>Overview</b> .....	47
<b>Reference Stackup</b> .....	47
<b>General Memory Routing Guidelines</b> .....	50
<b>Adjusting for Different Stack-Ups</b> .....	63
Dielectric Material .....	63
Trace Width .....	64
Layer Height .....	64
Base Copper Weight .....	65
Example of Tuning Design Parameters to Meet Impedance Targets .....	65
<b>PCB Guidelines for DDR4 SDRAM (PL and PS)</b> .....	65
Overview .....	65
DDR4 SDRAM Interface Signal Description .....	66
Fly-by and Clamshell Topologies .....	67
Utilizing Address Mirroring to Ease Clamshell Routing .....	67
Topology and Routing Guidelines for DDR4 SDRAM .....	68
DDR4 SDRAM Routing Constraints .....	74

<b>PCB Guidelines for DDR3/3L SDRAM (PL and PS)</b> .....	<b>76</b>
Overview .....	76
DDR3 SDRAM Interface Signal Description .....	76
Topology and Routing Guidelines for DDR3 SDRAM .....	77
DDR3 SDRAM Routing Constraints .....	80
<b>DDR3/DDR4 UDIMM/RDIMM/SODIMM Routing Guidelines (PL and PS)</b> .....	<b>82</b>
DDR3/DDR4 DIMM Clock Point-to-Point Routing .....	82
DDR3/DDR4 DIMM Control, Command, and Address Routing .....	83
DDR3/DDR4 DIMM Data (DQ and DQS) Routing .....	85
DDR3 UDIMM/RDIMM/SODIMM Routing Constraints .....	86
DDR4 UDIMM/RDIMM/SODIMM Routing Constraints .....	87
<b>PCB Guidelines for LPDDR4 Memories without ECC (PS)</b> .....	<b>88</b>
Overview .....	88
LPDDR4 x32 without ECC Memory Interface Signals and Connections .....	89
LPDDR4 Address Copy .....	91
LPDDR4 Interface without ECC Topology and Routing Guidelines .....	92
LPDDR4 without ECC Length and Skew Constraints .....	96
<b>PCB Guidelines for LPDDR4 Memories with ECC (PS)</b> .....	<b>97</b>
Overview .....	97
LPDDR4 x32 with ECC Memory Interface Signals and Connections .....	98
LPDDR4 Address Copy .....	101
LPDDR4 with ECC Topology and Routing Guidelines .....	101
<b>PCB Guidelines for LPDDR3 SDRAM (PL and PS)</b> .....	<b>111</b>
Overview .....	111
LPDDR3 SDRAM Interface Signal Description .....	112
LPDDR3 Address Copy .....	112
VREFCA and VREFDQ .....	113
Design Example for x32 and x64 LPDDR3 SDRAM .....	113
Topology and Routing Guidelines for LPDDR3 SDRAM .....	114
LPDDR3 SDRAM Routing Constraints .....	120
<b>PCB Guidelines for RLDRAM 3 Memory</b> .....	<b>121</b>
Overview .....	121
RLDRAM 3 Memory Interface Signal Description .....	122
RLDRAM 3 Memory Topology and Routing Guidelines for Clamshell and Fly-by Configurations .....	123
RLDRAM 3 Memory Routing Constraints .....	131
<b>PCB Guidelines for QDR II+ SRAM</b> .....	<b>132</b>
Overview .....	132
QDR II+ SRAM Interface Signal Description .....	132
Design Example for Dual QDR II+ SRAM Devices .....	133
QDR II+ SRAM Topology and Routing Guidelines for Fly-by Configuration .....	134
Additional QDR II+ SRAM Routing Guidelines .....	135
QDR II+ SRAM Routing Constraints .....	138
<b>PCB Guidelines for QDR-IV SRAM</b> .....	<b>139</b>
Overview .....	139
QDR-IV SRAM Interface Signal Description .....	140
Design Example for Dual QDR-IV SRAM Devices .....	140
QDR-IV Topology and Routing Guidelines .....	140
QDR-IV SRAM Routing Constraints .....	143

## Chapter 3: PCB Guidelines for Zynq UltraScale+ RFSocS

<b>PDN Guidelines for RFSoc Digital Power Rails</b> . . . . .	<b>144</b>
Power Delivery to the Programmable Logic Voltage Rails . . . . .	144
Power Delivery to the Processing System Voltage Rails . . . . .	148
Power Delivery to the PS-GTR and GTY Transceivers . . . . .	148
<b>ADC and DAC PCB Guidelines</b> . . . . .	<b>149</b>
Analog Ground to Digital Ground Connection . . . . .	150
Choosing the Appropriate Balun . . . . .	150
Recommended Clocking Options . . . . .	152
Analog and Clock Pair Routing . . . . .	153
Sample Stackup . . . . .	168
Power Regulation and Decoupling for ADC and DAC Supplies . . . . .	169

## Chapter 4: PCB Guidelines for the PS Interface in the Zynq UltraScale+ MPSoC

Boot Mode . . . . .	175
CAN . . . . .	175
DisplayPort . . . . .	175
eMMC . . . . .	175
Ethernet MAC RGMII . . . . .	176
Ethernet MAC SGMII . . . . .	176
I2C . . . . .	176
JTAG . . . . .	176
NAND Memory Controller . . . . .	176
PCIe . . . . .	177
PS_INIT_B, PS_PROG_B, and PS_DONE . . . . .	177
PS Reference Clock . . . . .	177
PS Reset (External System Reset and POR Reset) . . . . .	177
QSPI . . . . .	177
Real-Time Clock . . . . .	178
SATA . . . . .	178
SD/SDIO . . . . .	178
SPI . . . . .	178
Trace Port Interface Unit . . . . .	179
Triple Time Counter . . . . .	179
UART . . . . .	179
USB 2.0 . . . . .	179
USB 3.0 . . . . .	179
Watchdog Timer . . . . .	179
PS-GTR Transceiver Interfaces . . . . .	179
AC Coupled Reference Clock . . . . .	184
Unused Reference Clocks . . . . .	184
Reference Clock Power . . . . .	184
Power Supply Noise . . . . .	184
PCB Design Checklist . . . . .	185

## Chapter 5: Migration between UltraScale Devices and Packages

<b>UltraScale Device Migration Checklist</b> . . . . .	<b>187</b>
1. Footprint Compatibility between Packages . . . . .	188
2. Package Dimensions . . . . .	189
3. I/O Bank and Transceiver Quad Numbers . . . . .	192
4. HP/HR Migration . . . . .	193

5. GTH/GTY Transceiver Migration	193
6. SLR Migration	194
7. Monolithic to Stacked Silicon Migration	196
8. Memory Interface Migration	197
9. DCI Cascade and Internal V <sub>REF</sub> Features	199
10. System Monitor	201
11. Decoupling Capacitors	202
12. PCI Express Migration	203
13. Integrated 100G Ethernet Migration	203
14. Interlaken Migration	204
15. Power Supplies and Thermal Considerations	204
16. Pin Flight Times across Packages	204

## Chapter 6: Migration between UltraScale and UltraScale+ FPGAs

<b>UltraScale+ FPGA Migration Checklist</b>	<b>206</b>
1. Package Migration from UltraScale to UltraScale+ FPGAs	207
2. Voltage Differences between UltraScale to UltraScale+ FPGAs	208
3. Power Supply Voltage Levels and VCCINT_IO Connection	209
4. I/O Changes from UltraScale to UltraScale+ FPGAs	210
5. Transceiver Changes from UltraScale to UltraScale+ FPGAs	211
6. Configuration	212
7. Memory Interface PCB Routing	213
8. System Monitor I2C	213
9. Block RAM	213
10. ESD Requirements	213
11. PCI Express	214
12. Migration Examples	214
13. Migration Scenarios	218

## Chapter 7: Migration between Zynq UltraScale+ MPSoCs and Packages

<b>Zynq UltraScale+ MPSoC Migration Checklist</b>	<b>220</b>
1. Footprint Compatibility between Packages	220
2. I/O Bank and Transceiver Quad Numbers	222
3. Power Supply Differences across Speed Grades and Temperature Grades	225
4. -3 Speed Migration	225
5. VCU Migration	226
6. GPU Migration	227
7. Decoupling Capacitors	227
8. PCI Express Migration	227
9. Integrated 100G Ethernet Migration	227
10. Interlaken Migration	227
11. Power Supplies and Thermal Considerations	228
12. Pin Flight Times across Packages	228

## Chapter 8: SelectIO Signaling

<b>Interface Types</b>	<b>232</b>
Single-Ended versus Differential Interfaces	232
SDR versus DDR Interfaces	233
<b>Single-Ended Signaling</b>	<b>233</b>
Modes and Attributes	233

Input Thresholds .....	234
Topographies and Termination .....	234

## Chapter 9: PCB Technology Basics

<b>Introduction .....</b>	<b>244</b>
<b>PCB Structures .....</b>	<b>244</b>
Traces .....	245
Planes .....	245
Vias .....	245
Pads and Antipads .....	245
Lands .....	246
Dimensions .....	246
<b>Basic PDS Principles .....</b>	<b>247</b>
Noise Limits .....	247
Role of Inductance .....	249
Capacitor Parasitic Inductance .....	250
PCB Current Path Inductance .....	252
Plane Inductance .....	254
Capacitor Effective Frequency .....	257
Capacitor Anti-Resonance .....	258
Capacitor Placement Background .....	259
V <sub>REF</sub> Stabilization Capacitors .....	260
Power Supply Consolidation .....	260
Unconnected V <sub>CCO</sub> Pins .....	261
Transmission Lines .....	261
Return Currents .....	262
<b>PCB Materials and Traces .....</b>	<b>262</b>
How Fast is Fast? .....	262
Dielectric Losses .....	263
Traces .....	264
Cable .....	268
<b>Simulation Methods .....</b>	<b>269</b>
<b>PDS Measurements .....</b>	<b>270</b>
Noise Magnitude Measurement .....	270
Noise Spectrum Measurements .....	272
Optimum Decoupling Network Design .....	275
<b>Troubleshooting .....</b>	<b>275</b>
Possibility 1: Excessive Noise from Other Devices on the PCB .....	275
Possibility 2: Parasitic Inductance of Planes, Vias, or Connecting Traces .....	276
Possibility 3: I/O Signals in PCB are Stronger Than Necessary .....	277
Possibility 4: I/O Signal Return Current Traveling in Sub-Optimal Paths .....	277

## Chapter 10: Design of Transitions for High-Speed Signals

<b>Excess Capacitance and Inductance .....</b>	<b>278</b>
<b>Time Domain Reflectometry .....</b>	<b>278</b>
<b>BGA Package .....</b>	<b>280</b>
<b>SMT Pads .....</b>	<b>281</b>
<b>Differential Vias .....</b>	<b>286</b>

P/N Crossover Vias . . . . .	288
SMA Connectors . . . . .	288
Backplane Connectors . . . . .	288
Microstrip/Stripline Bends . . . . .	289

## Appendix A: Memory Derating Tables

## Appendix B: Material Properties and Insertion Losses

## Appendix C: Additional Resources and Legal Notices

Xilinx Resources . . . . .	304
Solution Centers . . . . .	304
Documentation Navigator and Design Hubs . . . . .	304
References . . . . .	305
Please Read: Important Legal Notices . . . . .	306

# Power Distribution System in UltraScale Devices

---

## Introduction to UltraScale Architecture

The Xilinx® UltraScale™ architecture is the first ASIC-class architecture to enable multi-hundred gigabit-per-second levels of system performance with smart processing, while efficiently routing and processing data on-chip. UltraScale architecture-based devices address a vast spectrum of high-bandwidth, high-utilization system requirements by using industry-leading technical innovations, including next-generation routing, ASIC-like clocking, 3D-on-3D ICs, multiprocessor SoC (MPSoC) technologies, and new power reduction features. The devices share many building blocks, providing scalability across process nodes and product families to leverage system-level investment across platforms.

Virtex® UltraScale+™ devices provide the highest performance and integration capabilities in a FinFET node, including both the highest serial I/O and signal processing bandwidth, as well as the highest on-chip memory density. As the industry's most capable FPGA family, the Virtex UltraScale+ devices are ideal for applications including 1+Tb/s networking and data center and fully integrated radar/early-warning systems.

Virtex UltraScale devices provide the greatest performance and integration at 20 nm, including serial I/O bandwidth and logic capacity. As the industry's only high-end FPGA at the 20 nm process node, this family is ideal for applications including 400G networking, large scale ASIC prototyping, and emulation.

Kintex® UltraScale+ devices provide the best price/performance/watt balance in a FinFET node, delivering the most cost-effective solution for high-end capabilities, including transceiver and memory interface line rates as well as 100G connectivity cores. Our newest mid-range family is ideal for both packet processing and DSP-intensive functions and is well suited for applications including wireless MIMO technology, Nx100G networking, and data center.

Kintex UltraScale devices provide the best price/performance/watt at 20 nm and include the highest signal processing bandwidth in a mid-range device, next-generation transceivers, and low-cost packaging for an optimum blend of capability and cost-effectiveness. The family is ideal for packet processing in 100G networking and data centers

applications as well as DSP-intensive processing needed in next-generation medical imaging, 8k4k video, and heterogeneous wireless infrastructure.

Zynq® UltraScale+ MPSoC devices provide 64-bit processor scalability while combining real-time control with soft and hard engines for graphics, video, waveform, and packet processing. Integrating an Arm®-based system for advanced analytics and on-chip programmable logic for task acceleration creates unlimited possibilities for applications including 5G Wireless, next generation ADAS, and Industrial Internet-of-Things.

This user guide describes the UltraScale architecture PCB design and pin planning resources and is part of the UltraScale Architecture documentation suite available at: [www.xilinx.com/ultrascale](http://www.xilinx.com/ultrascale).

---

## Introduction

This chapter documents the power distribution system (PDS) for UltraScale devices, including decoupling capacitor selection, placement, and PCB geometries. A simple decoupling method is provided for each device. Basic PDS design principles are covered, as well as simulation and analysis methods. This chapter contains the following sections:

- [PCB Decoupling Capacitors](#)
- [Capacitor Specifications](#)
- [Capacitor Consolidation Rules](#)
- [Transceiver PCB Routing Guidelines](#)
- [Maximum Current Draw for  \$V\_{CCINT}\$  in Virtex UltraScale+ Devices](#)

---

## PCB Decoupling Capacitors

### Recommended PCB Capacitors per Device

Recommended decoupling capacitor quantities for Kintex UltraScale and Virtex UltraScale devices are listed in [Table 1-2](#) and [Table 1-3](#). Decoupling capacitor quantities for Kintex UltraScale+ and Virtex UltraScale+ devices are listed in [Table 1-4](#) and [Table 1-5](#). Decoupling capacitor quantities for Zynq UltraScale+ devices are listed in [Table 1-9](#). The optimized quantities of PCB decoupling capacitors assume that the voltage regulators have stable output voltages and meet the regulator manufacturer's minimum output capacitance requirements.

Decoupling methods other than those presented in these tables can be used, but the decoupling network should be designed to meet or exceed the performance of the simple



decoupling networks presented here. The impedance of the alternate network is recommended to be less than or equal to that of the recommended network across frequencies from 100 kHz to approximately 10 MHz.

Because device capacitance requirements vary with CLB and I/O utilization, PCB decoupling guidelines are provided on a per-device basis based on very high utilization so as to cover a majority of use cases. Resource usage consists (in part) of:

- 80% of LUTs and registers at 245 MHz and 25% toggle rate
- 80% block RAM and DSP at 491 MHz and 50% toggle rate
- 50% MMCM and 25% PLL at 500 MHz
- 25% I/O at SSTL 1.2/1.35 at 1200 MHz and 40% toggle rate
- 75% I/O at POD 1.2 at 1200 MHz DDR and 40% toggle rate

An [importable XPE file](#) that contains the above usage assumptions in a KU9P FFVE900 device can be downloaded from the Xilinx website.

### Step Load Assumptions

Different step loads are assumed for each main voltage rail. The step load is the percentage of the dynamic current that is expected to be demanded at any given switching event. [Table 1-1](#) lists the step load percentage used when calculating device capacitance requirements.

Table 1-1: Step Load for Device Capacitance

Voltage Rail	Step Load (%)
V <sub>CCINT</sub> /V <sub>CCINT_IO</sub>	25
V <sub>CCBRAM</sub>	40
V <sub>CCAUX</sub> /V <sub>CCAUX_IO</sub>	100
V <sub>CCO</sub> (HP/HR)	100

The slew rate of the switching event is dependent on the design, and can be estimated to be between 1 ns and 100 ns (or longer). Smaller current designs generally have faster current slew rates, while larger designs tend to have slower slew rates. A general rule of thumb for high-current designs can be considered to be 0.25 ns per amp (or 4 A/ns) of step current.

The [Xilinx Power Estimator \(XPE\) tool](#) is used to estimate the current on each power rail. *Kintex UltraScale FPGAs Data Sheet: DC and AC Switching Characteristics (DS892)* [Ref 1] and *Virtex UltraScale FPGAs Data Sheet: DC and AC Switching Characteristics (DS893)* [Ref 2] provide the operating range for all the various power rails. The PCB designer should ensure that the AC ripple plus the DC tolerance of the voltage regulator do not exceed the operating range.

The capacitor numbers shown in this user guide are based on the following assumptions:

$V_{CCINT}$  operating range from the data sheet = 3%;

Assumed DC tolerance = 1%;

Therefore, allowable AC ripple = 3% – 1% = 2%.

The target impedance is calculated using the 2% AC ripple along with the current estimates from XPE for the above resource utilization to arrive at the capacitor recommendations. The equation for target impedance is:

$$Z_{\text{target}} = \frac{\text{VoltageRailValue} \times \frac{\% \text{ Ripple}}{100}}{\text{StepLoadCurrent}} \quad \text{Equation 1-1}$$

$V_{CCINT}$ ,  $V_{CCAUX}$ , and  $V_{CCBRAM}$  capacitors are listed as the quantity per device, while  $V_{CCO}$  capacitors are listed as the quantity per I/O bank. Device performance at full utilization is equivalent across all devices when using these recommended networks.

Table 1-2 through Table 1-9 do not provide the decoupling networks required for the GTY or GTH transceiver power supplies. For this information, refer to the *UltraScale Architecture GTH Transceivers User Guide* (UG576) [Ref 6] or the *UltraScale Architecture GTY Transceivers User Guide* (UG578) [Ref 7].



**RECOMMENDED:** Refer to the *UltraScale Architecture Schematic Review Checklist* (XTP344) [Ref 8] for a comprehensive checklist for schematic review which complements this user guide.

## Recommended Decoupling Capacitor Quantities for Kintex UltraScale and Virtex UltraScale Devices

Table 1-2 and Table 1-3 show the recommended decoupling capacitor quantities for Kintex UltraScale and Virtex UltraScale devices.

Table 1-2: Kintex UltraScale Devices Power Supply Decoupling Capacitors

	$V_{CCINT}/V_{CCINT\_IO}$ <sup>(1)</sup>			$V_{CCBRAM}$ <sup>(2)</sup>		$V_{CCAUX}/V_{CCAUX\_IO}$ <sup>(3)</sup>		$V_{CCO}$ (Bank 0)	HRIO <sup>(4)</sup>	HPIO <sup>(4)</sup>
	680 $\mu\text{F}$ <sup>(5)</sup>	100 $\mu\text{F}$	4.7 $\mu\text{F}$	47 $\mu\text{F}$	4.7 $\mu\text{F}$	47 $\mu\text{F}$	4.7 $\mu\text{F}$	47 $\mu\text{F}$	47 $\mu\text{F}$	47 $\mu\text{F}$
XCKU025-FFVA1156	1	1	2	1	1	1	2	1	1	1
XCKU035-FBVA676	1	1	3	1	1	1	3	1	1	1
XCKU035-FBVA900	1	1	3	1	1	2	4	1	1	1
XCKU035-FFVA1156	1	1	3	1	1	2	5	1	1	1
XCKU035-SFVA784	1	1	3	1	1	2	4	1	1	1
XCKU040-FBVA676 XQKU040-RBA676	1	2	3	1	1	1	3	1	1	1
XCKU040-FBVA900	1	2	3	1	1	2	4	1	1	1

Table 1-2: Kintex UltraScale Devices Power Supply Decoupling Capacitors (Cont'd)

	$V_{CCINT}/V_{CCINT\_IO}$ <sup>(1)</sup>			$V_{CCBRAM}$ <sup>(2)</sup>		$V_{CCAUX}/V_{CCAUX\_IO}$ <sup>(3)</sup>		$V_{CCO}$ (Bank 0)	HRIO <sup>(4)</sup>	HPIO <sup>(4)</sup>
	680 $\mu$ F <sup>(5)</sup>	100 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	47 $\mu$ F	47 $\mu$ F
XCKU040-FFVA1156 XQKU040-RFA1156	1	2	3	1	1	2	5	1	1	1
XCKU040-SFVA784	1	1	3	1	1	2	4	1	1	1
XCKU060-FFVA1156 XQKU060-RFA1156	1	2	4	1	1	2	5	1	1	1
XCKU060-FFVA1517	1	2	4	1	1	3	6	1	1	1
XCKU085-FLVA1517	2	4	7	1	1	3	6	1	1	1
XCKU085-FLVB1760	2	4	7	1	1	3	6	1	1	1
XCKU085-FLVF1924	2	4	7	1	1	3	6	1	1	1
XCKU095-FFVC1517	1	2	4	1	1	2	5	1	1	1
XCKU095-FFVB1760	1	2	4	1	1	3	6	1	1	1
XCKU095-FFVB2104	1	2	4	1	1	3	6	1	1	1
XCKU095-FFVA1156 XQKU095-RFA1156	1	2	4	1	1	2	5	1	1	1
XCKU115-FLVA1517	3	5	9	1	1	3	6	1	1	1
XCKU115-FLVD1517 XQKU115-RLD1517	3	5	9	1	1	1	3	1	1	1
XCKU115-FLVB1760	3	5	9	1	1	3	7	1	1	1
XCKU115-FLVD1924	3	5	9	1	1	4	8	1	1	1
XCKU115-FLVF1924 XQKU115-RLF1924	3	5	9	1	1	3	7	1	1	1
XCKU115-FLVA2104	3	5	9	1	1	4	8	1	1	1
XCKU115-FLVB2104	3	5	9	1	1	3	7	1	1	1

**Notes:**

1.  $V_{CCINT}$  and  $V_{CCINT\_IO}$  must be tied together on the PCB.
2.  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  can be tied together if all three rails are operated at the same voltage.
3.  $V_{CCAUX}$  and  $V_{CCAUX\_IO}$  must be tied together on the PCB.
4. One 47  $\mu$ F capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
5. 470  $\mu$ F capacitors can be used in place of 680  $\mu$ F capacitors at a rate of four 470  $\mu$ F capacitors per three 680  $\mu$ F capacitors. See [Table 1-11](#) for 470  $\mu$ F capacitor specifications.
6. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. See [PCB Bulk Capacitors](#) about the need for 680  $\mu$ F and/or 100  $\mu$ F capacitors.

Table 1-3: Virtex UltraScale Devices Power Supply Decoupling Capacitors

	$V_{CCINT}/V_{CCINT\_IO}$ <sup>(1)</sup>			$V_{CCBRAM}$ <sup>(2)</sup>		$V_{CCAUX}/V_{CCAUX\_IO}$ <sup>(3)</sup>		$V_{CCO}$ (Bank 0)	HRIO <sup>(4)</sup>	HPIO <sup>(4)</sup>
	680 $\mu$ F <sup>(5)</sup>	100 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	47 $\mu$ F	47 $\mu$ F
XCVU065-FFVC1517	1	1	3	1	1	2	5	1	1	1
XCVU080-FFVC1517	1	2	3	1	1	2	5	1	1	1
XCVU080-FFVD1517	1	2	3	1	1	2	3	1	1	1
XCVU080-FFVB1760	1	2	3	1	1	3	7	1	1	1
XCVU080-FFVA2104	1	2	3	1	1	4	8	1	1	1
XCVU080-FFVB2104	1	2	3	1	1	3	7	1	1	1
XCVU095-FFVC1517	1	2	4	1	1	2	5	1	1	1
XCVU095-FFVD1517	1	2	4	1	1	2	3	1	1	1
XCVU095-FFVB1760	1	2	4	1	1	3	7	1	1	1
XCVU095-FFVA2104	1	2	4	1	1	4	8	1	1	1
XCVU095-FFVB2104	1	2	4	1	1	3	7	1	1	1
XCVU095-FFVC2104	1	2	4	1	1	2	4	1	1	1
XCVU125-FLVD1517	2	3	5	1	1	2	3	1	1	1
XCVU125-FLVB1760	2	3	5	1	1	4	7	1	1	1
XCVU125-FLVA2104	2	3	5	1	1	4	8	1	1	1
XCVU125-FLVB2104	2	3	5	1	1	4	7	1	1	1
XCVU125-FLVC2104	2	3	5	1	1	2	4	1	1	1
XCVU160-FLGB2104	2	4	7	1	1	4	7	1	1	1
XCVU160-FLGC2104	2	4	7	1	1	2	4	1	1	1
XCVU190-FLGB2104	3	4	8	1	1	4	7	1	1	1
XCVU190-FLGC2104	3	4	8	1	1	2	4	1	1	1
XCVU190-FLGA2577	3	4	8	1	1	2	4	1	1	1
XCVU440-FLGB2377	5	8	14	1	1	6	13	1	1	1
XCVU440-FLGA2892	5	8	14	1	1	7	14	1	1	1

**Notes:**

1.  $V_{CCINT}$  and  $V_{CCINT\_IO}$  must be tied together on the PCB.
2.  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  can be tied together if all three rails are operated at the same voltage.
3.  $V_{CCAUX}$  and  $V_{CCAUX\_IO}$  must be tied together on the PCB.
4. One 47  $\mu$ F capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
5. 470  $\mu$ F capacitors can be used in place of 680  $\mu$ F capacitors at a rate of four 470  $\mu$ F capacitors per three 680  $\mu$ F capacitors. See [Table 1-11](#) for 470  $\mu$ F capacitor specifications.
6. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. See [PCB Bulk Capacitors](#) about the need for 680  $\mu$ F and/or 100  $\mu$ F capacitors.

## Recommended Decoupling Capacitor Quantities for Kintex UltraScale+ and Virtex UltraScale+ Devices

Table 1-4 and Table 1-5 show the recommended decoupling capacitor quantities for Kintex UltraScale+ and Virtex UltraScale+ devices.

Table 1-4: Kintex UltraScale+ FPGAs Decoupling Capacitor Recommendations

	$V_{CCINT}/V_{CCINT\_IO}$ <sup>(1)</sup>			$V_{CCBRAM}/V_{CCINT\_IO}$ <sup>(2)</sup>		$V_{CCAUX}/V_{CCAUX\_IO}$		HDIO <sup>(4)</sup>	HPIO <sup>(4)</sup>
	680 $\mu$ F <sup>(5)</sup>	100 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	47 $\mu$ F
XCKU3P-SFVB784	1	1	2	1	1	1	2	1	1
XCKU3P-FFVA676	1	1	2	1	1	1	2	1	1
XCKU3P-FFVB676	1	1	2	1	1	1	2	1	1
XCKU3P-FFVD900	1	1	2	1	1	1	2	1	1
XCKU5P-SFVB784	1	2	3	1	1	1	2	1	1
XCKU5P-FFVA676	1	2	3	1	1	1	2	1	1
XCKU5P-FFVB676	1	2	3	1	1	1	2	1	1
XCKU5P-FFVD900	1	2	3	1	1	1	2	1	1
XCKU9P-FFVE900	1	2	4	1	1	1	2	1	1
XCKU11P-FFVD900	2	2	5	1	1	2	3	1	1
XCKU11P-FFVA1156	2	2	5	1	1	2	4	1	1
XCKU11P-FFVE1517	2	2	5	1	1	2	4	1	1
XCKU13P-FFVE900	2	3	5	1	1	2	2	1	1
XCKU15P-FFVA1156	2	3	5	1	1	2	5	1	1
XCKU15P-FFVE1517	2	3	5	1	1	2	4	1	1
XCKU15P-FFVA1760	2	3	5	1	1	2	4	1	1
XCKU15P-FFVE1760	2	3	5	1	1	2	6	1	1

### Notes:

1. Connect  $V_{CCINT}$  and  $V_{CCINT\_IO}$  together for -3, -2, and -1 speed grades.
2. Connect  $V_{CCBRAM}$  and  $V_{CCINT\_IO}$  together for -2L and -1L speed grades.
3.  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  can be tied together if all three rails are operated at the same voltage.
4. One 47  $\mu$ F capacitor is required for up to four HP/HD I/O banks when powered by the same voltage.
5. 470  $\mu$ F capacitors can be used in place of 680  $\mu$ F capacitors at a rate of four 470  $\mu$ F capacitors per three 680  $\mu$ F capacitors. See Table 1-12 for 470  $\mu$ F capacitor specifications.
6. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. See [PCB Bulk Capacitors](#) about the need for 680  $\mu$ F and/or 100  $\mu$ F capacitors.
7. See [Chapter 1, Power Distribution System in UltraScale Devices](#) for capacitor specifications.

Table 1-5: Virtex UltraScale+ FPGAs Decoupling Capacitor Recommendations

	$V_{CCINT}/V_{CCINT\_IO}^{(1)}$			$V_{CCBRAM}/V_{CCINT\_IO}^{(2)}$		$V_{CCAUX}/V_{CCAUX\_IO}$		$HRIO^{(4)}$	$HPIO^{(4)}$
	680 $\mu F^{(5)}$	100 $\mu F$	4.7 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	47 $\mu F$
XCVU3P-FFVC1517	1	2	4	1	1	3	5	1	1
XCVU5P-FLVA2104	2	4	6	1	1	4	8	1	1
XCVU5P-FLVB2104	2	4	6	1	1	4	7	1	1
XCVU5P-FLVC2104	2	4	6	1	1	2	4	1	1
XCVU7P-FLVA2104	3	5	8	1	1	4	8	1	1
XCVU7P-FLVB2104	3	5	8	1	1	4	7	1	1
XCVU7P-FLVC2104	3	5	8	1	1	2	4	1	1
XCVU9P-FLGA2104	4	7	13	1	1	4	8	1	1
XCVU9P-FLGB2104	4	7	13	1	1	4	7	1	1
XCVU9P-FLGC2104	4	7	13	1	1	2	4	1	1
XCVU9P-FSGD2104	4	7	13	1	1	3	7	1	1
XCVU9P-FLGA2577	4	7	13	1	1	2	4	1	1
XCVU11P-FLGF1924	5	8	14	1	1	3	6	1	1
XCVU11P-FLGB2104	5	8	14	1	1	3	6	1	1
XCVU11P-FLGC2104	5	8	14	1	1	2	4	1	1
XCVU11P-FLGD2104	5	8	14	1	1	3	6	1	1
XCVU11P-FLGA2577	5	8	14	1	1	2	4	1	1
XCVU13P-FHGA2104	6	10	19	1	1	4	8	1	1
XCVU13P-FHGB2104	6	10	19	1	1	4	7	1	1
XCVU13P-FHGC2104	6	10	19	1	1	2	4	1	1
XCVU13P-FIGD2104	7	11	20	1	2	2	4	1	1
XCVU13P-FLGA2577	6	10	19	1	1	2	4	1	1
XCVU13P-FSGA2577	6	10	19	1	1	2	4	1	1

**Notes:**

1. Connect  $V_{CCINT}$  and  $V_{CCINT\_IO}$  together for -3, -2, and -1 speed grades.
2. Connect  $V_{CCBRAM}$  and  $V_{CCINT\_IO}$  together for -2L and -1L speed grades.
3.  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  can be tied together if all three rails are operated at the same voltage.
4. One 47  $\mu F$  capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
5. 470  $\mu F$  capacitors can be used in place of 680  $\mu F$  capacitors at a rate of four 470  $\mu F$  capacitors per three 680  $\mu F$  capacitors. See [Table 1-12](#) for 470  $\mu F$  capacitor specifications.
6. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. See [PCB Bulk Capacitors](#) about the need for 680  $\mu F$  and/or 100  $\mu F$  capacitors.
7. See [Chapter 2, PCB Guidelines for Memory Interfaces](#) for capacitor specifications.

## PCB Decoupling Capacitors for Virtex UltraScale+ High Bandwidth Memory Devices

PCB decoupling capacitors for Virtex UltraScale+ high bandwidth memory (HBM) devices are listed in [Table 1-6](#) and [Table 1-7](#). [Table 1-6](#) lists capacitors for PL supplies, while [Table 1-7](#) lists the capacitors for the HBM supplies.

**Table 1-6: Decoupling Capacitor Recommendations for Virtex UltraScale+ HBM Devices (PL Rails)**

	V <sub>CCINT</sub>			V <sub>CCBRAM</sub> /V <sub>CCINT_IO</sub>		V <sub>CCAUX</sub> /V <sub>CCAUX_IO</sub>		HPIO
	680 μF	100 μF	4.7 μF	47 μF	4.7 μF	47 μF	4.7 μF	47 μF
XCVU31P-FSVH1924	2	3	5	1	1	1	2	1
XCVU33P-FSVH2104	2	3	5	1	1	1	2	1
XCVU35P-FSVH2104	4	6	11	1	1	2	4	1
XCVU35P-FSVH2892	4	6	11	1	1	2	4	1
XCVU37P-FSVH2892	5	9	16	1	2	3	7	1

**Notes:**

1. If V<sub>CCINT</sub>, V<sub>CCBRAM</sub>, and V<sub>CCINT\_IO</sub> are connected to the same voltage rail, combine their respective capacitors.
2. V<sub>CCAUX</sub> and V<sub>CCAUX\_IO</sub> should be tied together on the PCB.
3. One 47 μF capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
4. 470 μF capacitors can be used in place of 680 μF capacitors at a rate of four 470 μF capacitors per three 680 μF capacitors. See [Table 1-12](#) for 470 μF capacitor specifications.
5. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. 680 μF and 100 μF capacitors might not be needed depending on the regulator and switching frequency.

**Table 1-7: Decoupling Capacitor Recommendations for Virtex UltraScale+ HBM Devices (HBM Rails)**

V <sub>CC_HBM</sub>				V <sub>CC_IO_HBM</sub>				V <sub>CCAUX_HBM</sub>		
680 μF	100 μF	4.7 μF	0.47 μF	680 μF	100 μF	4.7 μF	0.47 μF	47 μF	4.7 μF	0.47 μF
1	1	2	4	1	1	2	4	1	2	4

**Notes:**

1. Place 0.47 μF underneath the FPGA, or as close to it as possible.
2. Step load assumptions for V<sub>CC\_HBM</sub> and V<sub>CC\_IO\_HBM</sub> are between 3A and 4A.

[Table 1-8](#) lists the specifications for the capacitors listed in [Table 1-6](#) and [Table 1-7](#).

**Table 1-8: PCB Capacitor Specifications**

Ideal Value	Value Range <sup>(1)</sup>	Body Size <sup>(2)</sup>	Type	ESL Maximum	ESR Range	Voltage Rating <sup>(3)</sup>	Suggested Part Number <sup>(4)</sup>
680 μF	C > 680 μF	2917/D /7343	2-Terminal Tantalum	2.1 nH	5 mΩ < ESR < 25 mΩ	2.0V	Kemet T530X687M006ATE018
470 μF	C > 470 μF	2917/D /7343	2-Terminal Polymer Aluminum	1.5 nH	3 mΩ < ESR < 25 mΩ	2.0V	Panasonic EEF-HX0D471R4

Table 1-8: PCB Capacitor Specifications (Cont'd)

Ideal Value	Value Range <sup>(1)</sup>	Body Size <sup>(2)</sup>	Type	ESL Maximum	ESR Range	Voltage Rating <sup>(3)</sup>	Suggested Part Number <sup>(4)</sup>
100 $\mu$ F	C > 100 $\mu$ F	1210	2-Terminal Ceramic	1.0 nH	1 m $\Omega$ < ESR < 10 m $\Omega$	2.0V	Murata GRM32EE70G107ME19 TDK C3216X6S0G107M160AC
47 $\mu$ F	C > 47 $\mu$ F	1210	2-Terminal Ceramic	1.0 nH	1 m $\Omega$ < ESR < 10 m $\Omega$	6.3V	Murata GRM32ER70J476ME20L TDK C3225X6S0J476M250AC
4.7 $\mu$ F	C > 4.7 $\mu$ F	0805	2-Terminal Ceramic	1.0 nH	1 m $\Omega$ < ESR < 10 m $\Omega$	6.3V	Murata GRM21BR71A475KA73 TDK C1005X5R0J475M
0.47 $\mu$ F	C > 0.47 $\mu$ F	0603	2-Terminal Ceramic	0.6 nH	1 m $\Omega$ < ESR < 20 m $\Omega$	6.3V	Murata GRM188R70J474KA01
0.47 $\mu$ F	C > 0.47 $\mu$ F	0402	2-Terminal Ceramic	0.6 nH	1 m $\Omega$ < ESR < 20 m $\Omega$	6.3V	Murata GRT155R71A474KE01
0.47 $\mu$ F	C > 0.47 $\mu$ F	0201	2-Terminal Ceramic	0.6 nH	1 m $\Omega$ < ESR < 20 m $\Omega$	6.3V	TDK C0603X6S0G474M030BC

**Notes:**

1. Values can be larger than specified.
2. Body size can be smaller than specified.
3. Voltage rating can be higher than specified.
4. When selecting capacitors, ensure they meet or exceed the temperature requirements of the system.

## Recommended Decoupling Capacitor Quantities for Zynq UltraScale+ Devices

Table 1-9 shows the recommended decoupling capacitor quantities for Zynq UltraScale+ devices.

Table 1-9: Zynq UltraScale+ MPSoC Decoupling Capacitor Recommendations

	$V_{CCINT}/V_{CCINT\_IO}$ <sup>(1)</sup>				$V_{CCBRAM}/V_{CCINT\_IO}$ <sup>(2)</sup>		$V_{CCAUX}/V_{CCAUX\_IO}$		$HDIO$ <sup>(4)</sup>	$HPIO$ <sup>(4)</sup>
	680 $\mu$ F <sup>(5)</sup>	100 $\mu$ F	4.7 $\mu$ F	0.47 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	47 $\mu$ F
<b>CG Devices</b>										
XCZU2CG-SBVA484	1	1	1	2	1	1	1	1	1	1
XCZU2CG-SFVA625	1	1	1	0	1	1	1	2	1	1
XCZU2CG-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU3CG-SFVC484	1	1	1	4	1	1	1	1	1	1
XCZU3CG-SFVA625	1	1	1	0	1	1	1	2	1	1
XCZU3CG-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU4CG-SFVC784	1	1	1	0	1	1	1	2	1	1



Table 1-9: Zynq UltraScale+ MPSoC Decoupling Capacitor Recommendations (Cont'd)

	$V_{CCINT}/V_{CCINT\_IO}^{(1)}$				$V_{CCBRAM}/V_{CCINT\_IO}^{(2)}$		$V_{CCAUX}/V_{CCAUX\_IO}$		$HDIO^{(4)}$	$HPIO^{(4)}$
	680 $\mu F^{(5)}$	100 $\mu F$	4.7 $\mu F$	0.47 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	47 $\mu F$
XCZU4CG-FBVB900	1	1	1	0	1	1	1	2	1	1
XCZU5CG-SFVC784	1	1	2	0	1	1	1	2	1	1
XCZU5CG-FBVB900	1	1	2	0	1	1	1	2	1	1
XCZU6CG-FFVC900	1	2	3	0	1	1	1	2	1	1
XCZU6CG-FBVB1156	1	2	3	0	1	1	2	3	1	1
XCZU7CG-FBVB900	1	1	3	0	1	1	1	2	1	1
XCZU7CG-FFVC1156	1	1	3	0	1	1	2	3	1	1
XCZU7CG-FFVF1517	1	1	3	0	1	1	2	4	1	1
XCZU9CG-FFVC900	1	2	4	0	1	1	1	2	1	1
XCZU9CG-FBVB1156	1	2	4	0	1	1	2	3	1	1
<b>EG Devices</b>										
XCZU2EG-SBVA484	1	1	1	2	1	1	1	1	1	1
XCZU2EG-SFVA625	1	1	1	0	1	1	1	2	1	1
XCZU2EG-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU3EG-SFVC484	1	1	1	4	1	1	1	1	1	1
XCZU3EG-SFVA625	1	1	1	0	1	1	1	2	1	1
XCZU3EG-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU4EG-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU4EG-FBVB900	1	1	1	0	1	1	1	2	1	1
XCZU5EG-SFVC784	1	1	2	0	1	1	1	2	1	1
XCZU5EG-FBVB900	1	1	2	0	1	1	1	2	1	1
XCZU6EG-FFVC900	1	2	3	0	1	1	1	2	1	1
XCZU6EG-FBVB1156	1	2	3	0	1	1	2	3	1	1
XCZU7EG-FBVB900	1	1	3	0	1	1	1	2	1	1
XCZU7EG-FFVC1156	1	1	3	0	1	1	2	3	1	1
XCZU7EG-FFVF1517	1	1	3	0	1	1	2	4	1	1
XCZU9EG-FFVC900	1	2	4	0	1	1	1	2	1	1
XCZU9EG-FBVB1156	1	2	4	0	1	1	2	3	1	1
XCZU11EG-FFVC1156	1	2	4	0	1	1	2	3	1	1
XCZU11EG-FFVB1517	1	2	4	0	1	1	2	5	1	1
XCZU11EG-FFVF1517	1	2	4	0	1	1	2	4	1	1
XCZU11EG-FFVC1760	1	2	4	0	1	1	2	5	1	1
XCZU15EG-FFVC900	2	3	5	0	1	1	1	2	1	1
XCZU15EG-FBVB1156	2	3	5	0	1	1	2	3	1	1

Table 1-9: Zynq UltraScale+ MPSoC Decoupling Capacitor Recommendations (Cont'd)

	$V_{CCINT}/V_{CCINT\_IO}^{(1)}$				$V_{CCBRAM}/V_{CCINT\_IO}^{(2)}$		$V_{CCAUX}/V_{CCAUX\_IO}$		$HDIO^{(4)}$	$HPIO^{(4)}$
	680 $\mu F^{(5)}$	100 $\mu F$	4.7 $\mu F$	0.47 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	4.7 $\mu F$	47 $\mu F$	47 $\mu F$
XCZU17EG-FFVB1517	1	2	3	0	1	1	3	6	1	1
XCZU17EG-FFVC1760	1	2	3	0	1	1	2	5	1	1
XCZU17EG-FFVD1760	1	2	3	0	1	1	1	3	1	1
XCZU17EG-FFVE1924	1	2	3	0	1	1	3	6	1	1
XCZU19EG-FFVB1517	1	2	4	0	1	1	3	6	1	1
XCZU19EG-FFVC1760	1	2	4	0	1	1	2	4	1	1
XCZU19EG-FFVD1760	1	2	4	0	1	1	1	3	1	1
XCZU19EG-FFVE1924	1	2	4	0	1	1	3	6	1	1
<b>EV Devices</b>										
XCZU4EV-SFVC784	1	1	1	0	1	1	1	2	1	1
XCZU4EV-FBVB900	1	1	1	0	1	1	1	2	1	1
XCZU5EV-SFVC784	1	1	2	0	1	1	1	2	1	1
XCZU5EV-FBVB900	1	1	2	0	1	1	1	2	1	1
XCZU7EV-FBVB900	1	1	3	0	1	1	1	2	1	1
XCZU7EV-FFVC1156	1	1	3	0	1	1	2	3	1	1
XCZU7EV-FFVF1517	1	1	3	0	1	1	2	4	1	1

**Notes:**

1. Connect  $V_{CCINT}$  and  $V_{CCINT\_IO}$  together for -3, -2, and -1 speed grades.
2. Connect  $V_{CCBRAM}$  and  $V_{CCINT\_IO}$  together for -2L and -1L speed grades.
3.  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  can be tied together if all three rails are operated at the same voltage.
4. One 47  $\mu F$  capacitor is required for up to four HP/HD I/O banks when powered by the same voltage.
5. 470  $\mu F$  capacitors can be used in place of 680  $\mu F$  capacitors at a rate of four 470  $\mu F$  capacitors per three 680  $\mu F$  capacitors. See [Table 1-12](#) for 470  $\mu F$  capacitor specifications.
6. PCB decoupling capacitors cover down to approximately 100 kHz, depending on voltage regulator design. See [PCB Bulk Capacitors](#) about the need for 680  $\mu F$  and/or 100  $\mu F$  capacitors.
7. See [Chapter 1, Power Distribution System in UltraScale Devices](#) for capacitor specifications.

Table 1-10: Zynq UltraScale+ MPSoC PS Decoupling Capacitor Recommendations

$V_{CC\_PSINTLP}$		$V_{CC\_PSINTFP}$		$V_{CC\_PSAUX}$		$V_{CC\_PSPLL}$		$PS\_MGTRAVCC$	$PS\_MGTRAVTT$	$V_{CC\_PSINTFP\_DDR}$		$V_{CCO\_PSIOX}^{(3)}$ (Each)		$V_{CC\_PSBATT}$		$V_{CCO\_PSDDR}$		
100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	4.7 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	100 $\mu F$	4.7 $\mu F$	0.47 $\mu F$
1	1	1	1	1	1	1	1	See <a href="#">note 1</a>	See <a href="#">note 1</a>	1	1	1	1	1	1	1	2	4

**Notes:**

1. For PS MGT supply decoupling, see [PS-GTR Transceiver Interfaces, page 179](#).
2. Add two 0.47  $\mu F$  capacitors to each PS voltage rail if using SBVA484 devices.
3.  $V_{CCO\_PSIO}$  banks that share the same rail can utilize one set of decoupling capacitors (1 x 100  $\mu F$ , 1 x 4.7  $\mu F$ ) for up to four connected  $V_{CCO\_PSIO}$  banks.

## V<sub>CC\_PSDDR\_PLL</sub> Supply

V<sub>CC\_PSDDR\_PLL</sub> is a 1.8V nominal supply that provides power to the PLL used for the PS DDR controller. It can be powered separately or derived from the V<sub>CC\_PSAUX</sub> supply. If powered by V<sub>CC\_PSAUX</sub>, V<sub>CC\_PSDDR\_PLL</sub> must be filtered through a 120Ω @ 100 MHz, size 0603 ferrite bead and a 10 μF or larger, size 0603 decoupling capacitor. In both cases, a 0.47 μF to 4.7 μF 0402 capacitor must be placed near the V<sub>CC\_PSDDR\_PLL</sub> BGA via.

The PCB construction of the V<sub>CC\_PSDDR\_PLL</sub> power supply must be carefully managed. The recommended connection between the 10 μF 0603 capacitor and the V<sub>CC\_PSDDR\_PLL</sub> BGA ball is a planelet with a minimum width of 80 mil (2 mm) and a length of less than 3,000 mil (76 mm). If a planelet cannot be used, a trace with a maximum impedance of 40Ω and a length of less than 2,000 mil (50.8 mm) must be used. The 0.47 μF to 4.7 μF 0402 or 0201 capacitor should be placed as close as possible to the FPGA, along with the shortest possible trace length. Figure 1-1 shows an example of the filtering and local capacitor circuit used when V<sub>CC\_PSDDR\_PLL</sub> is derived from V<sub>CC\_PSAUX</sub>.

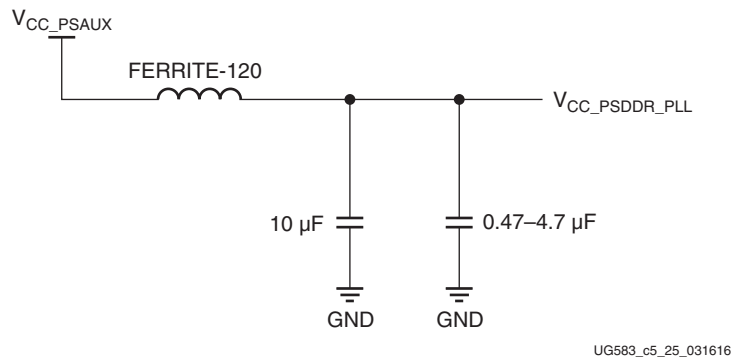


Figure 1-1: Connecting V<sub>CC\_PSDDR\_PLL</sub>

Figure 1-2 shows an example of the layout of the same filtering circuit.

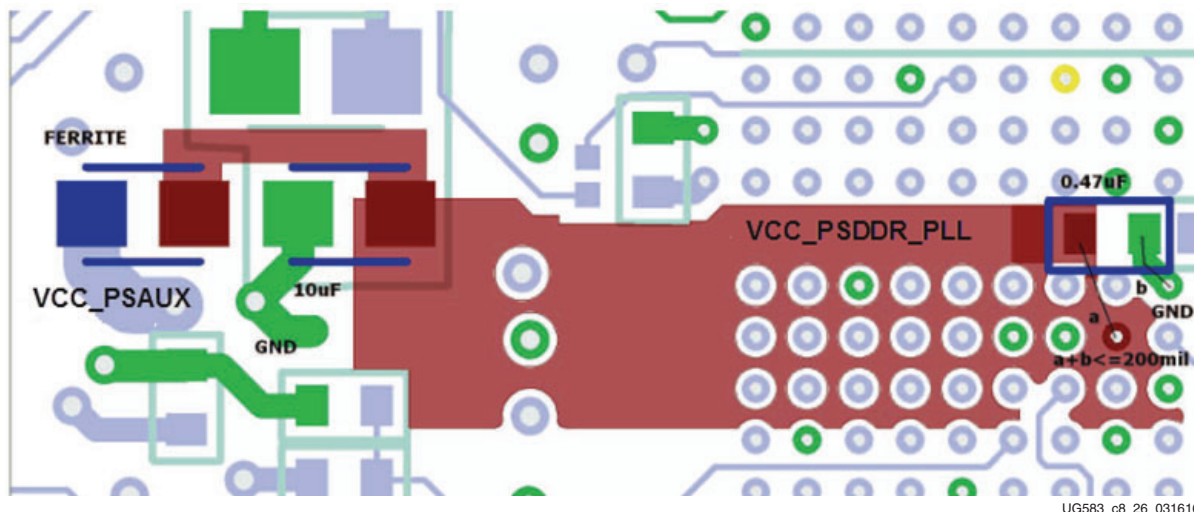


Figure 1-2: Filtering Circuit Layout

The recommended components are:

- Ferrite bead: Murata BLM18SG121TN1
- 10  $\mu$ F capacitor: Murata GXM188Z71A106MA73
- 0.47  $\mu$ F–4.7  $\mu$ F capacitor: Murata GRT155R71A474KE01

## Video Codec Unit (MPSoC EV Devices Only)

$V_{CCINT\_VCU}$  is a 0.90V rail that provides power to the video codec unit (VCU) within Zynq UltraScale+ MPSoC EV devices.  $V_{CCINT\_VCU}$  must be powered as a standalone power rail and cannot be combined with any other power rails.

If the VCU is not going to be used, the  $V_{CCINT\_VCU}$  pins can be grounded to reduce leakage current.

## $V_{CCINT\_VCU}$ Decoupling Capacitors

A series of decoupling capacitors are required on  $V_{CCINT\_VCU}$  when utilizing the VCU. Table 1-11 shows the required capacitor types and quantities for  $V_{CCINT\_VCU}$ .

Table 1-11:  $V_{CCINT\_VCU}$  Decoupling Capacitor Requirements

Parameter	10 $\mu$ F 0402	47 $\mu$ F 0603	100 $\mu$ F 0805	330 $\mu$ F 1210	470 $\mu$ F D
Quantity	3	5	1	2	3
Location	Under BGA	1 near BGA, 4 between BGA and regulator	Between BGA and regulator	Near BGA	1 near regulator, 2 between BGA and regulator
Part Number	Murata GRM155R60J106ME47	Murata GRM188C08E476ME05	Murata GRM21BC80G107ME15	Murata GRM32EC80E337ME05	Panasonic EEFGX0D471R

## $V_{CCINT\_VCU}$ Plane Design and Power Delivery

The  $V_{CCINT\_VCU}$  plane(s) or planelet(s) should be wide enough to accommodate up to 3A of current at their narrowest points, factoring in all width reductions due to vias and other keepouts. However, planes and planelets need to be drawn even wider to reduce resistive losses between the regulator and the BGA pins. While properly routed sense lines offer assistance in offsetting resistive losses, regulator efficiency is maximized when resistive loss is minimized. For these reasons, Xilinx recommends drawing  $V_{CCINT\_VCU}$  planes at 500–800 mil<sup>(1)</sup> width while outside of the BGA region, and 400 mils under the BGA before the plane gradually narrows to the  $V_{CCINT\_VCU}$  pins. Figure 1-3 shows an example  $V_{CCINT\_VCU}$  plane layout.

In addition, Xilinx recommends placing a ground plane directly above or below the plane that contains  $V_{CCINT\_VCU}$  to reduce inductance.

---

1. If there are an unusually large number of vias or other keepouts within the plane, a wider plane might need to be drawn to reduce resistive losses and to maintain the ability to provide at least 3A of current.

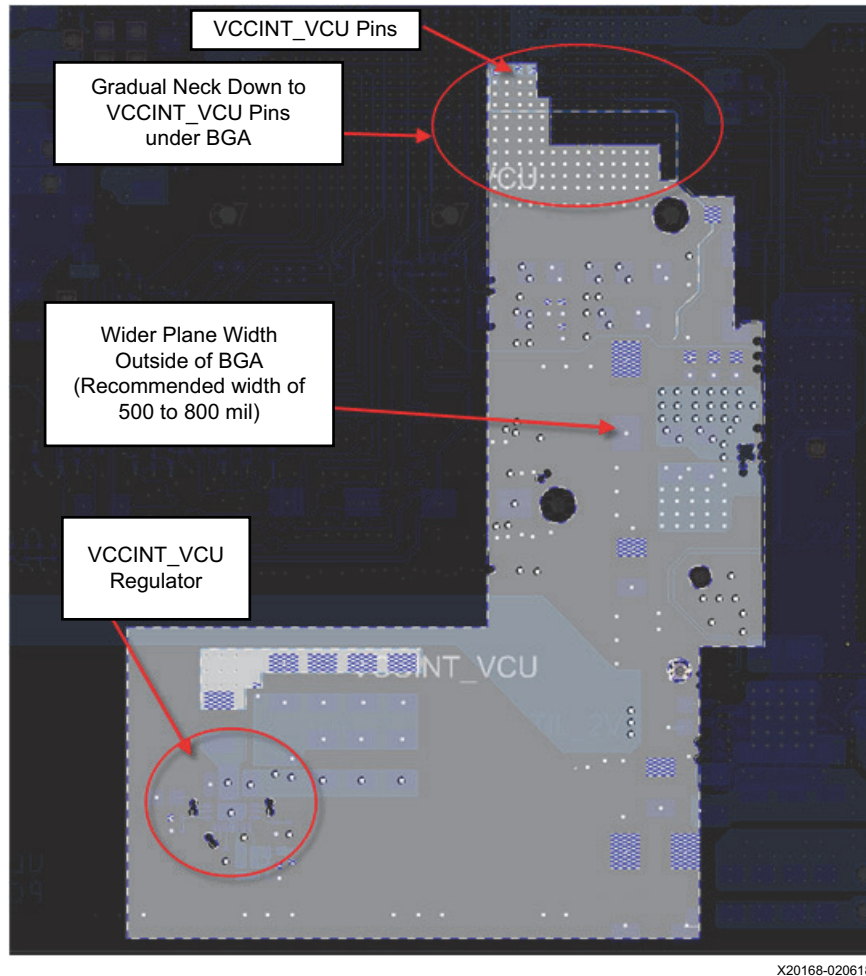


Figure 1-3: Example Plane Layout for  $V_{CCINT\_VCU}$

## Capacitor Specifications

The electrical characteristics of the capacitors in [Table 1-2](#) through [Table 1-9](#) are specified in [Table 1-12](#), and are followed by guidelines on acceptable substitutions. The equivalent series resistance (ESR) ranges specified for these capacitors can be over-ridden. However, this requires analysis of the resulting power distribution system impedance to ensure that no resonant impedance spikes result.

Table 1-12: PCB Capacitor Specifications

Ideal Value	Value Range <sup>(1)</sup>	Body Size <sup>(2)</sup>	Type	ESL Maximum	ESR Range	Voltage Rating <sup>(3)</sup>	Suggested Part Number
680 $\mu$ F	C > 680 $\mu$ F	2917/D /7343	2-Terminal Tantalum	2.1 nH	5 m $\Omega$ < ESR < 40 m $\Omega$	2.0V	T530X687M006ATE018
470 $\mu$ F	C > 470 $\mu$ F	2917/D /7343	2-Terminal Polymer Aluminum	1.5 nH	3 m $\Omega$ < ESR < 40 m $\Omega$	2.0V	EEF-GX0D471R
100 $\mu$ F	C > 100 $\mu$ F	1210	2-Terminal X7R/X7U or X5R	1.5 nH	1 m $\Omega$ < ESR < 40 m $\Omega$	2.0V	GRM32EE70G107ME19 C3216X6S0G107M160AC
47 $\mu$ F	C > 47 $\mu$ F	1210	2-Terminal Ceramic X7R or X5R	1.5 nH	1 m $\Omega$ < ESR < 40 m $\Omega$	6.3V	GRM32ER70J476ME20L C3225X6S0J476M250AC
4.7 $\mu$ F	C > 4.7 $\mu$ F	0805	2-Terminal Ceramic X7R or X5R	1.0 nH	1 m $\Omega$ < ESR < 20 m $\Omega$	6.3V	GRM21BR71A475KA73 C1005X5R0J475M
0.47 $\mu$ F	C > 0.47 $\mu$ F	0603	2-Terminal Ceramic X7R or X5R	0.5 nH	1 m $\Omega$ < ESR < 10 m $\Omega$	6.3V	GRM188R70J474KA01

**Notes:**

1. Values can be larger than specified.
2. Body size can be smaller than specified.
3. Voltage rating can be higher than specified.

### PCB Bulk Capacitors

The purpose of the bulk capacitors (D, 1210) is to cover the low-frequency range between where the voltage regulator stops working and where the on-package ceramic capacitors start working. The specified bulk capacitors (680  $\mu$ F, 470  $\mu$ F, 100  $\mu$ F, and 47  $\mu$ F) provide coverage down to approximately 100 kHz. Depending on the regulator design used, these bulk capacitors might not be necessary. Consult the regulator manufacturer to determine the frequency range of operation.

The tantalum and polymer aluminum capacitors specified in [Table 1-12](#) were selected for their values and controlled ESR values. They are also ROHS compliant. If another manufacturer's tantalum, polymer aluminum, or ceramic capacitors are used, you must ensure they meet the specifications of [Table 1-12](#) and are properly evaluated via simulation, s-parameter parasitic extraction, or bench testing.

### PCB High-Frequency Capacitors

[Table 1-12](#) shows the requirements for the 4.7  $\mu$ F capacitors in an 0805 package. Substitutions can be made for some characteristics, but not others. For details, refer to the notes in [Table 1-12](#).

## Capacitor Consolidation Rules

Sometimes a number of I/O banks are powered from the same voltage (e.g., 1.8V) and the recommended guidelines call for multiple bulk capacitors. This is also the case for  $V_{CCINT}$ ,  $V_{CCAUX}$ , and  $V_{CCBRAM}$  in the larger devices. These many smaller capacitors can be consolidated into fewer (larger value) capacitors provided the electrical characteristics of the consolidated capacitors (ESR and ESL) are equal to the electrical characteristics of the parallel combination of the recommended capacitors.

For most consolidations of  $V_{CCO}$ ,  $V_{CCINT}$ ,  $V_{CCAUX}$ , and  $V_{CCBRAM}$  capacitors, large tantalum capacitors with sufficiently low ESL and ESR are readily available.

## PCB Capacitor Placement and Mounting Techniques

### *PCB Bulk Capacitors*

Bulk capacitors (D, 1210) can be large and sometimes are difficult to place very close to the device. Fortunately, this is not a problem because the low-frequency energy covered by bulk capacitors is not sensitive to capacitor location. Bulk capacitors can be placed almost anywhere on the PCB, but the best placement is as close as possible to the device. Capacitor mounting should follow normal PCB layout practices, tending toward short and wide shapes connecting to power planes with multiple vias.

### *0805 Ceramic Capacitor*

The 0805 capacitor covers the middle frequency range. Placement has some impact on its performance. The capacitor should be placed as close as possible to the device. Any placement within two electrical inches<sup>(1)</sup> of the device's point of load is acceptable. The capacitor mounting (solder lands, traces, and vias) should be optimized for low inductance. Vias should be butted directly against the pads. Vias can be located at the ends of the pads (see [Figure 1-4B](#)), but are more optimally located at the sides of the pads (see [Figure 1-4C](#)). Via placement at the sides of the pads decreases the mounting's overall parasitic inductance by increasing the mutual inductive coupling of one via to the other. Dual vias can be placed on both sides of the pads (see [Figure 1-4D](#)) for even lower parasitic inductance. This is recommended whenever possible.

---

1. An electrical inch is the actual distance along an electrical path.



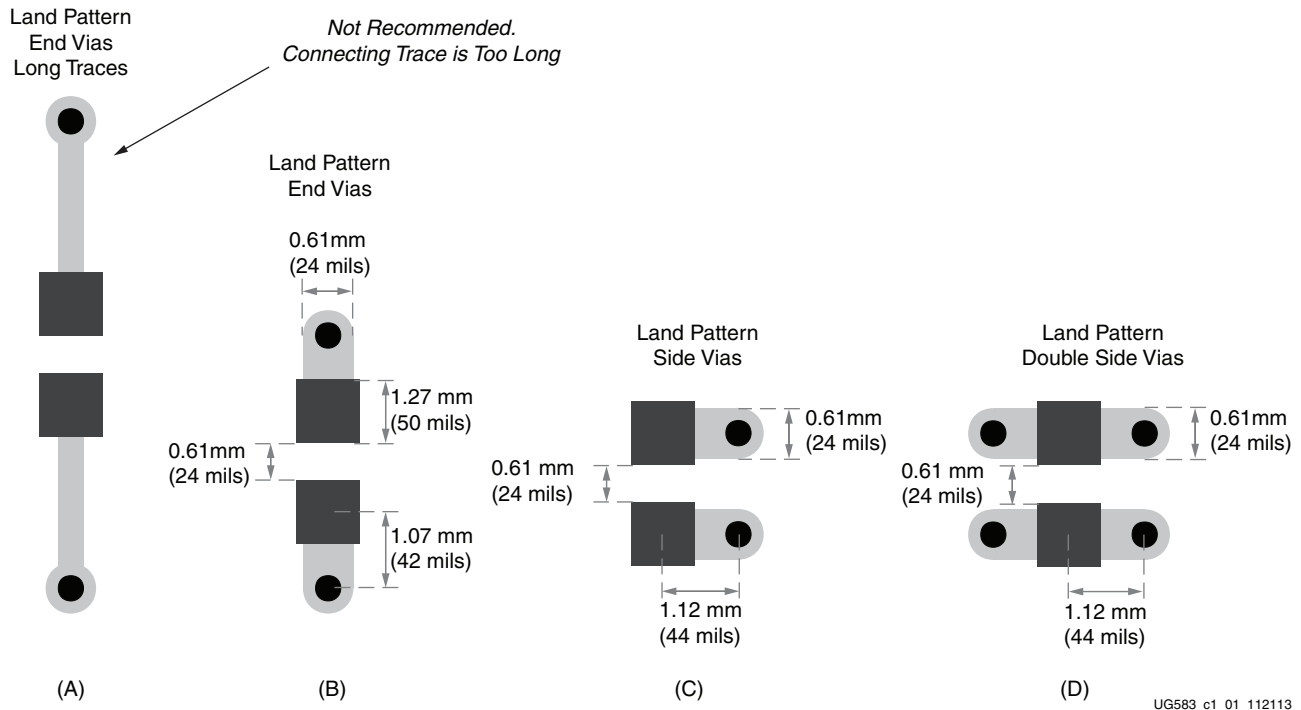


Figure 1-4: Example 0805 Capacitor Land and Mounting Geometries

## Transceiver PCB Routing Guidelines

Refer to the *UltraScale Architecture GTH Transceivers User Guide* (UG576) [Ref 6] or the *UltraScale Architecture GTY Transceivers User Guide* (UG578) [Ref 7].

## Power Supply Consolidation Solutions for Zynq UltraScale+ MPSoCs

Zynq UltraScale+ MPSoCs contain a number of power rails that correspond to the variety of features contained within the devices. A complete list of the power rails for Zynq UltraScale+ MPSoCs is listed in *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics* (DS925) [Ref 22].

For most purposes, assuming the decoupling requirements and filtering requirements are met, many of the power rails can be consolidated to reduce the total number of power regulators required to power a Zynq UltraScale+ MPSoC. In many applications, only five power regulators are required to power a Zynq UltraScale+ MPSoC. (Some of the five power regulators might be shared with other devices in the system.) However, the amount of consolidation that is possible depends on user requirements.

For example, many applications periodically need to enter ultra-low power, reduced functionality states to extend battery life or to reduce their overall power consumption. To facilitate this, Zynq UltraScale+ MPSoCs have four independent power domains that can be individually isolated: low-power domain (LPD), full-power domain (FPD), PL power domain (PLPD), and battery power domain (BPD). Applications requiring access to these ultra-low power states that cannot be achieved via power gating options in software might need independent control of the power rails associated with each power domain to enable the power associated with unused domains to reduce to zero. This results in the need for more power regulators or for the use of load switches. In this document, we refer to this use case as full power management flexibility. More detail on Zynq UltraScale+ MPSoC power domains can be found in *Zynq UltraScale+ MPSoC Technical Reference Manual* (UG1085) [Ref 23] and *Managing Power and Performance with the Zynq UltraScale+ MPSoC* (WP482) [Ref 24]. The benefits of leveraging the power domains and having full power management flexibility can be quantified using the [Xilinx Power Estimator \(XPE\) tool](#).

Other applications nearly always operate at full performance or can meet their power targets by disabling functionality via IP power-gating options (e.g., disabling processor cores or clock gating portions of the programmable logic). These applications do not need individual control of the power rails associated with each power domain. Therefore, these applications can achieve the maximum amount of power supply consolidation. For simplicity in this user guide, these applications are referred to as *always on*, and are further classified into three unique use cases. Together with the first use case (full power management flexibility), a total of four key use cases are discussed in this section:

- Always on: Cost optimized (-1 and -2 devices)
- Always on: Power/efficiency optimized (-1L and -2L devices)
- Always on: PL performance optimized (-3 devices)
- Full power management flexibility (all devices/speed grades)

The remainder of this section illustrates the power supply consolidation that can be achieved for the above use cases. In addition, [Power Management Partners](#) provides links to power delivery solutions from a range of Xilinx power delivery partners for the use cases mentioned above.

## Power Supply Consolidation Solutions

### *Always On: Optimized for Cost (-1 and -2 Devices)*

In always on, cost-optimized applications, where standard -1 or -2 speed grade devices are used, a significant amount of power rail consolidation is possible, assuming the decoupling and filtering requirements on the individual supplies as outlined in this document are met. [Table 1-13](#) broadly defines the power rail consolidation that is possible for this use case. For further clarity, [Figure 1-5](#) shows the possible power rail consolidation graphically.

As shown in [Table 1-13](#), the minimum number of regulators required to power a Zynq UltraScale+ MPSoC in this use case is five. In many cases, five or more regulators can be provided by a single integrated power management integrated circuit (PMIC). For example power delivery solutions for Xilinx products, refer to [Power Management Partners](#).

**Table 1-13: Always On: Cost-Optimized Power Rail Consolidation (-1 and -2 Devices)**

	Power Regulator	Sequence	Possible Power Rail Consolidation
Required	1	See <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics</i> (DS925) <a href="#">[Ref 22]</a>	VCCINT, VCC_PSINTFP, VCC_PSINTLP, VCC_PSINTFP_DDR, VCCINT_IO, VCCBRAM
	2		VCC_PSAUX, VCC_PSADC <sup>(1)</sup> , VCC_PSDDR_PLL <sup>(2)</sup> , VCCAUX, VCCAUX_IO, and VCCADC <sup>(1)</sup>
	3		VCC_PSPLL, VMGTAVTT (GTH), and VMGTAVTT (GTY)
	4		VCCO_PSDDR
	5		VCCO_PSIO[0:3] assuming all PS I/Os run from same voltage
Required: EV -1 and -2	9		VCCINT_VCU
User-defined	6		VPS_MGTRAVCC
	7		VPS_MGTRAVTT, VMGTVCCAUX (GTH), and VMGTVCCAUX (GTY)
	8		VMGTAVCC (GTH) and VMGTAVCC (GTY)
	10		Optional PL and PS I/O voltages

**Notes:**

1. Assuming rail is filtered as per *UltraScale Architecture System Monitor User Guide* (UG580) [\[Ref 25\]](#).
2. Assuming rail is filtered as per [VCC\\_PSDDR\\_PLL Supply, page 27](#).
3. When consolidating, ensure that all tolerances are met.

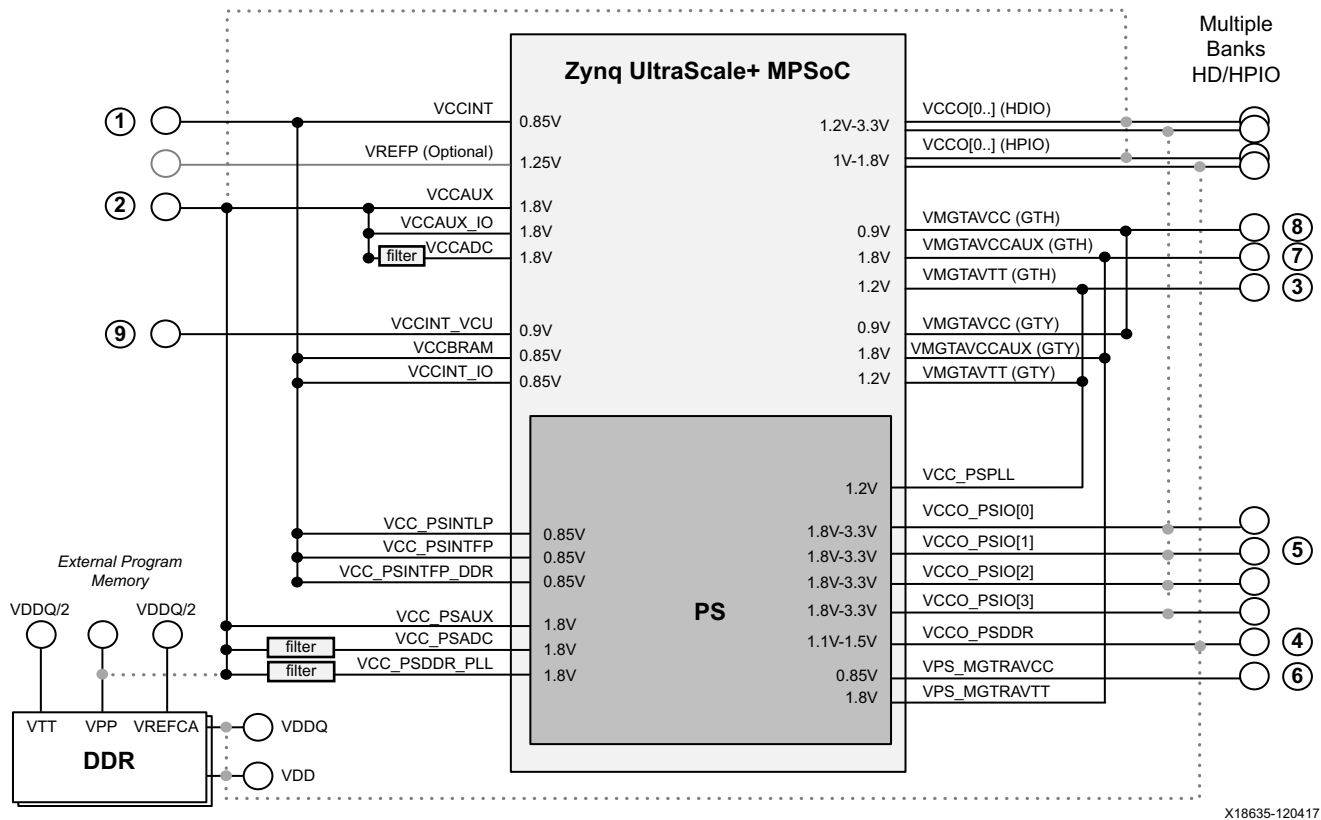


Figure 1-5: Always On: Cost-Optimized Power Rail Consolidation

**Note:** In Figure 1-5, the dashed lines are dependent on the user configuration.

Depending on user configuration, more than five regulators might be required. Table 1-14 details the number of regulators required to power Zynq UltraScale+ MPSoCs for a variety of device configurations. Figure 1-6 shows the power rail consolidation for the configuration requiring only five regulators using CG or EG devices.

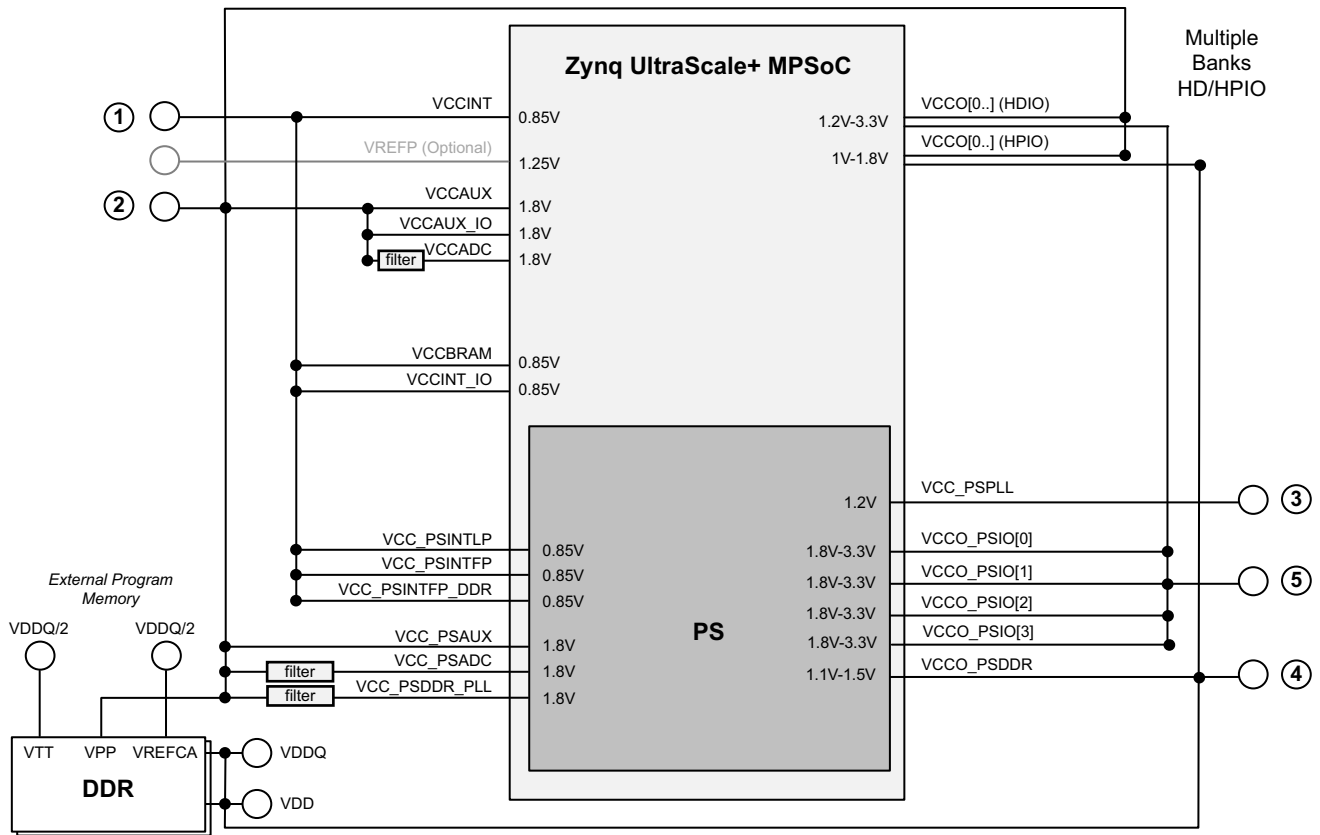
Additional power regulators are typically required for termination voltages and reference voltages of the DDR memory associated to the PS or PL.

Table 1-14: Number of Rails for “Always On: Optimized for Cost (-1 and -2 Devices)” Scenario

Configuration	Devices (Speed Grades)	Number of Power Regulators
PS and PL DDR, 1.8V I/O, and 2.5V I/O or 3.3V I/O with no MGTs	CG (-1, -2) EG (-1, -2)	Five (1, 2, 3, 4, 5)
	EV (-1, -2)	Six (1, 2, 3, 4, 5, 9)
PS and PL DDR, 1.8V I/O, 2.5V or 3.3V I/O and PS MGTs	CG (-1, -2) EG (-1, -2)	Seven (1, 2, 3, 4, 5, 6, 7)
	EV (-1, -2)	Eight (1, 2, 3, 4, 5, 6, 7, 9)

Table 1-14: Number of Rails for “Always On: Optimized for Cost (-1 and -2 Devices)” Scenario

Configuration	Devices (Speed Grades)	Number of Power Regulators
PS DDR, 1.8V I/O, 3.3V I/O, PS MGTs and PL MGTs	CG (-1, -2) EG (-1, -2)	Eight (1, 2, 3, 4, 5, 6, 7, 8)
	EV (-1, -2)	Nine (1, 2, 3, 4, 5, 6, 7, 8, 9)



X18634-120417

Figure 1-6: Always On: Cost-Optimized with Only Five Power Regulators (CG or EG Devices)

### ***Always On: Optimized for Power and/or Efficiency (-1L and -2L Devices)***

For applications requiring the lowest power dissipation and/or the highest efficiency, Xilinx offers the -1L and -2L speed grades. To achieve the highest efficiency and lowest power, the -1L and -2L devices can operate with a VCCINT voltage of 0.72V. To enable VCCINT to run at 0.72V an additional power regulator is required.

Other than the need for an additional power regulator for VCCINT, the possible power rail consolidation is similar to the previous use case. [Table 1-15](#) broadly defines what power rail consolidation is possible for this use case, assuming the decoupling and filtering requirements on the individual supplies as outlined in this document are met. For further clarity, [Figure 1-7](#) shows the possible power rail consolidation graphically.

As shown in [Table 1-15](#), the minimum number of regulators required for this use case is six.

**Table 1-15: Always On - Power/Efficiency Rail Consolidation for Low-Power Devices**

	Power Regulator	Sequence	Possible Power Rail Consolidation
Required	1	See <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics</i> (DS925) <a href="#">[Ref 22]</a>	VCCINT
	2		VCC_PSINTFP, VCC_PSINTLP, VCC_PSINTFP_DDR, VCCINT_IO, and VCCBRAM
	3		VCC_PSAUX, VCC_PSADC <sup>(1)</sup> , VCC_PSDDR_PLL <sup>(2)</sup> , VCCAUX, VCCAUX_IO and VCCADC <sup>(1)</sup>
	4		VCC_PSPLL, VMGTAVTT (GTH), and VMGTAVTT (GTY)
	5		VCCO_PSDDR
	6		VCCO_PSIO[0:3] assuming all PS I/Os run from same voltage
Required - EV devices	10		VCCINT_VCU
User-defined	7		VPS_MGTRAVCC
	8		VPS_MGTRAVTT, VMGTVCCAUX (GTH), and VMGTVCCAUX (GTY)
	9		VMGTAVCC (GTH) and VMGTAVCC (GTY)
	11		Optional PL and PS I/O voltages

**Notes:**

1. Assuming rail is filtered as per *UltraScale Architecture System Monitor User Guide* (UG580) [\[Ref 25\]](#).
2. Assuming rail is filtered as per [VCC\\_PSDDR\\_PLL Supply, page 27](#).

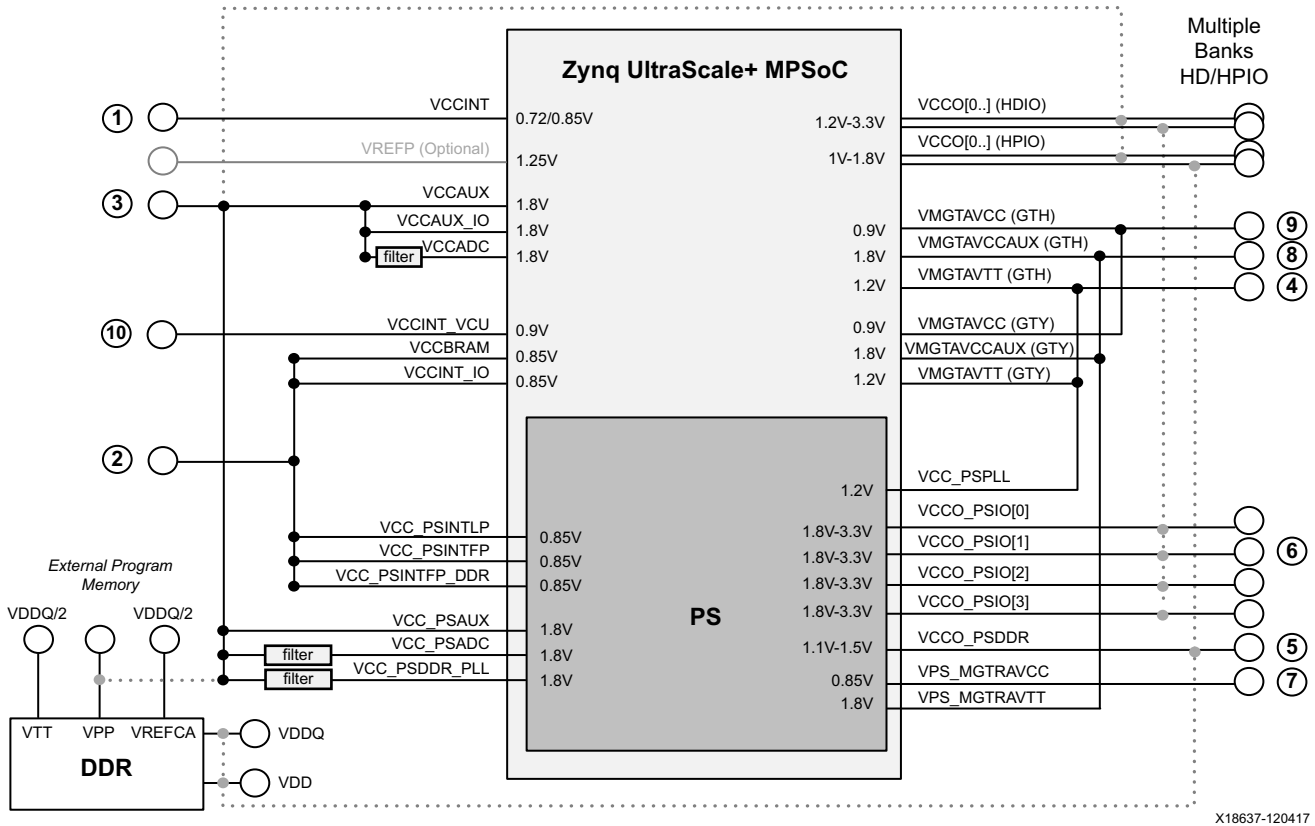


Figure 1-7: Always On: Power and/or Efficiency-Optimized Power Rail Consolidation for Low-Power Devices

**Note:** In Figure 1-7, the dashed lines are dependent on the user configuration.

Table 1-16 shows the number of power regulators for a variety of common configurations of Zynq UltraScale+ MPSoC for this use case.

Table 1-16: Number of Power Regulators Needed for “Always On: Optimized for Power and/or Efficiency (-1L and -2L Devices)” Scenario

Configuration	Number of Power Regulators
PS and PL DDR, 1.8V I/O and 2.5V or 3.3V I/O with no MGTs	Seven (1, 2, 3, 4, 5, 6, 10)
PS and PL DDR, 1.8V I/O, 2.5V I/O and 3.3V I/O with no MGTs	Eight (1, 2, 3, 4, 5, 6, 10, 11)
PS and PL DDR, 1.8V I/O, 2.5V or 3.3V I/O and PS MGTs	Nine (1, 2, 3, 4, 5, 6, 7, 8, 10)
PS DDR, 1.8V I/O, 3.3V I/O, PS MGTs and PL MGTs	Ten (1, 2, 3, 4, 5, 6, 7, 8, 9, 10)

### Always On: Optimized for PL Performance (-3 Devices)

For the highest level of PL performance, Xilinx offers Zynq UltraScale+ MPSoCs in a -3 speed grade. For the -3 speed grade device, all the core rails (VCCINT, VCCINT\_VCU, VCCBRAM, VCCINT\_IO, VCC\_PSINTLP, VCC\_PSINTFP, and VCC\_PSINTFP\_DDR) are run at a nominal voltage of 0.9V.

Table 1-17 broadly defines the power rail consolidation that is possible for this use case, assuming the decoupling and filtering requirements on the individual supplies as outlined in this document are met. For further clarity, Figure 1-8 shows the possible power rail consolidation graphically. As shown in Table 1-17, the minimum number of power regulators required to power a Zynq UltraScale+ MPSoC in this use case is five for non-EV devices, and six for EV devices.

Table 1-17: Always On: PL Performance-Optimized Rail Consolidation

	Power Regulator	Sequence	Possible Power Rail Consolidation
Required	1	See <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics</i> (DS925) [Ref 22]	VCCINT, VCC_PSINTFP, VCC_PSINTLP, VCC_PSINTFP_DDR, VCCINT_IO, VCCBRAM
	2		VCC_PSAUX, VCC_PSADC <sup>(1)</sup> , VCC_PSDDR_PLL <sup>(2)</sup> , VCCAUX, VCCAUX_IO, and VCCADC <sup>(1)</sup>
	3		VCC_PSPLL, VMGTAVTT (GTH), and VMGTAVTT (GTY)
	4		VCCO_PSDDR
	5		VCCO_PSIO[0:3] assuming all PS I/Os run from same voltage
	10		VCCINT_VCU (if used)
User-defined	6	VPS_MGTRAVTT, VMGTVCCAUX (GTH), and VMGTVCCAUX (GTY)	
	7	VMGTAVCC (GTH), and VMGTAVCC (GTY)	
	8	Additional PL and PS I/O voltages	
	9	VPS_MGTRAVCC	

**Notes:**

1. Assuming rail is filtered as per *UltraScale Architecture System Monitor User Guide* (UG580) [Ref 25].
2. Assuming rail is filtered as per [VCC\\_PSDDR\\_PLL Supply, page 27](#).



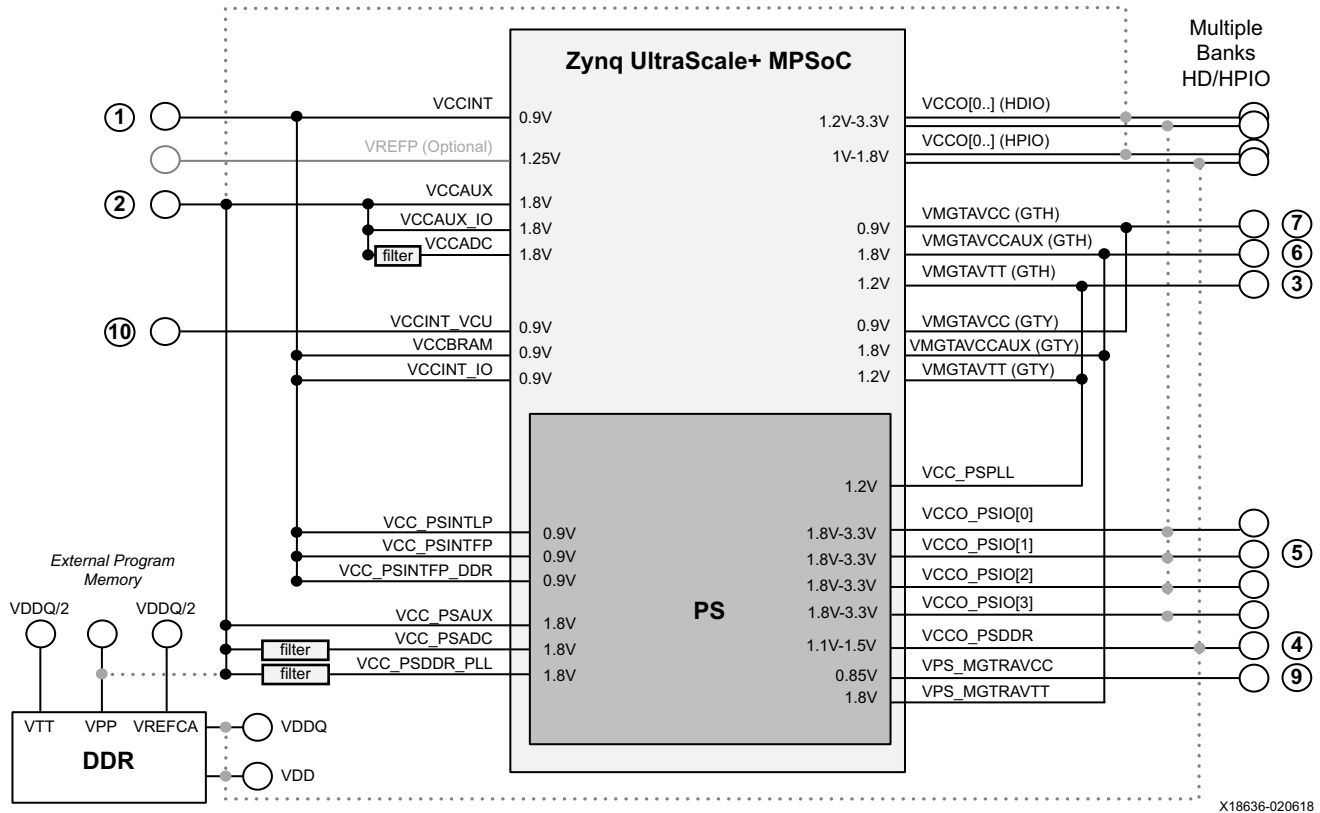


Figure 1-8: Always On: PL Performance-Optimized Rail Consolidation

**Note:** In Figure 1-8, the dashed lines are dependent on the user configuration.

Table 1-18 shows the number of regulators for common configurations of Zynq UltraScale+ MPSoC used in this use case.

Table 1-18: Number of Power Regulators Required for “Always On: Optimized for PL Performance (-3 Devices)” Scenario

Configuration	Number of Power Regulators <sup>(1)</sup>
PS and PL DDR, 1.8V I/O and 2.5V or 3.3V I/O with no MGTs	Five (1, 2, 3, 4, 5)
PS and PL DDR, 1.8V I/O, 2.5V I/O and 3.3V I/O with no MGTs	Six (1, 2, 3, 4, 5, 8)
PS and PL DDR, 1.8V I/O, 2.5V or 3.3V I/O and PS MGTs	Seven (1, 2, 3, 4, 5, 6, 9)
PS DDR, 1.8V I/O, 3.3V I/O, PS MGTs and PL MGTs	Eight (1, 2, 3, 4, 5, 6, 7, 9)

**Notes:**

1. Add regulator 10 to each scenario if using VCCINT\_VCU.

### Full Power Management Flexibility (All Speed Grades/Devices)

As mentioned earlier in this section, many applications need to enter ultra-low power, reduced functional states to maximize battery life or reduce overall power consumption. To facilitate this, Zynq UltraScale+ MPSoCs have four independent power domains: LPD, FPD, PLPD, and BPD. To enable the power associated with each of these domains to go to zero when unused, the power rails associated with each domain need to be separated, either via independent power regulators or via load switches.

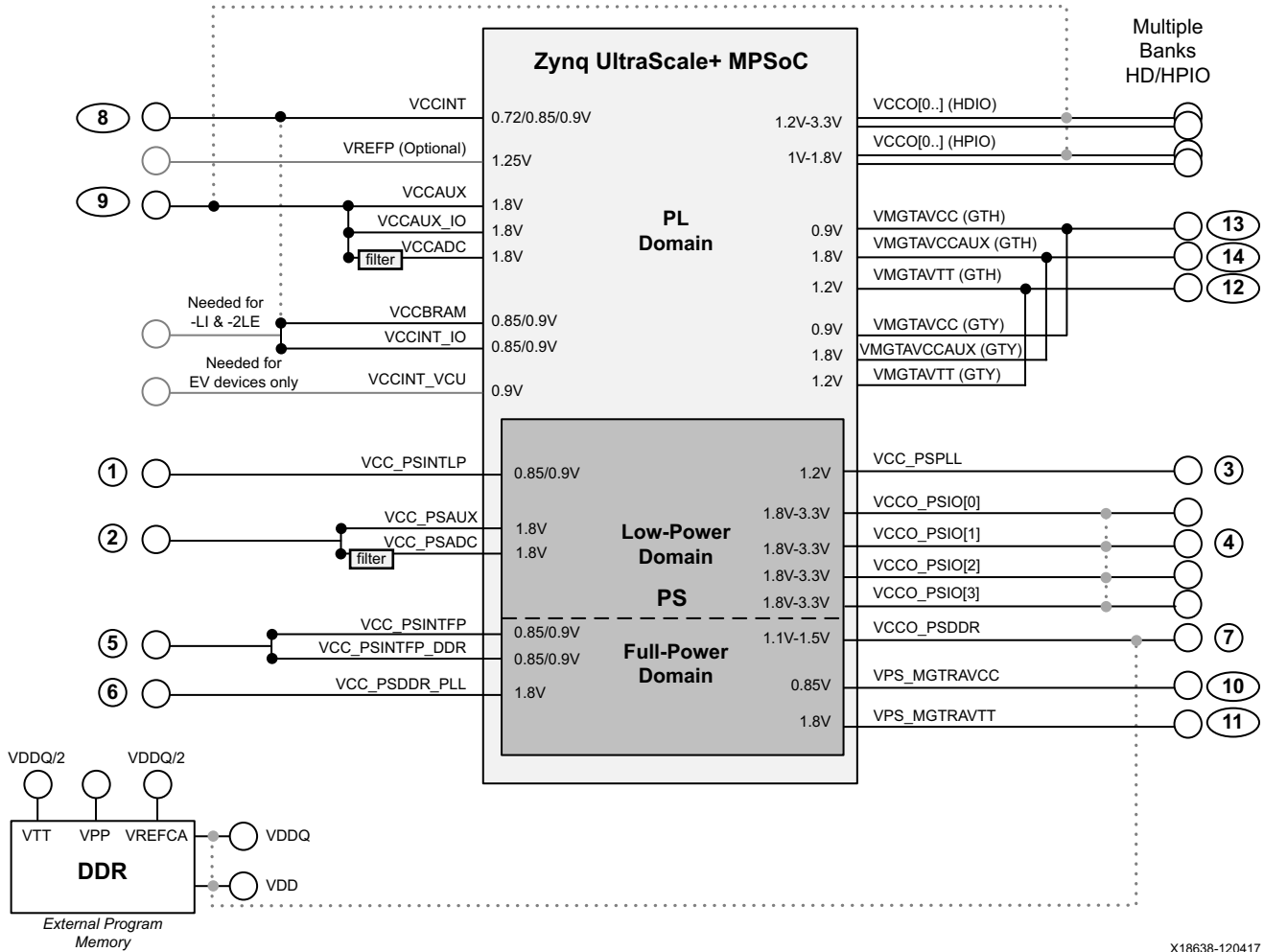
Table 1-19 broadly defines the power rail consolidation that is possible for this use case, assuming the decoupling and filtering requirements on the individual supplies as outlined in this document are met. For further clarity, Figure 1-9 shows the possible power rail consolidation graphically. As shown in Table 1-19, the minimum number of power regulators required to power a Zynq UltraScale+ MPSoC in this use case is nine. Alternatively, to achieve the same isolation, a combination of power regulator and load switch could be used, reducing the overall number of power regulators required.

Table 1-19: Full Power Management Flexibility Rail Consolidation

	Power Regulator	Sequence	Possible Power Rail Consolidation
Required	1	See Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics (DS925) [Ref 22]	VCC_PSINTLP
	2		VCC_PSAUX and VCC_PSADC <sup>(1)</sup>
	3		VCC_PSPLL
	4		VCCO_PSIO[0:3] assuming run off same voltage
	5		VCC_PSINTFP and VCC_PSINTFP_DDR
	6		VCC_PSDDR_PLL
	7		VCCO_PSDDR
	8		VCCINT <sup>(2)</sup> , VCCBRAM, and VCCINT_IO
	9		VCCAUX, VCCAUX_IO, VCCADC <sup>(1)</sup>
Required EV devices	15		VCCINT_VCU
User-defined	10		VPS_MGTRAVCC
	11		VPS_MGTRAVTT
	12		VMGTAVTT (GTH) and VMGTAVTT (GTY)
	13		VMGTAVCC (GTH) and VMGTAVCC (GTY)
	14		VMGTVCCAUX (GTH) and VMGTVCCAUX (GTY)
	16		Optional PL and PS I/O voltages

**Notes:**

1. Assuming rail is filtered as per *UltraScale Architecture System Monitor User Guide* (UG580) [Ref 25].
2. Additional voltage regulator required for -2LI or -2LE devices where user wants to run VCCINT = 0.72V



X18638-120417

Figure 1-9: Full Power Domain Flexibility Consolidation

**Note:** In Figure 1-9, the dashed lines are dependent on the user configuration.

Table 1-20 shows the number of regulators for common configurations of Zynq UltraScale+ MPSoC used in this use case. For applications that use -1LI or -2L devices, where the user wants to run VCCINT at 0.72V, an additional voltage regulator is required relative to Table 1-20. An additional regulator is required for VCCINT\_VCU when using an EV device.

Table 1-20: Number of Power Rails for Typical Configurations for Application Requiring Full Power Domain Flexibility

Configuration	Devices (Speed Grades)	Number of Power Regulators
Full power domain control, VCCINT = 0.85, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	CG (-1, -2) EG (-1, -2)	Nine (1, 2, 3, 4, 5, 6, 7, 8, 9)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	EG (-3)	Nine (1, 2, 3, 4, 5, 6, 7, 8, 9)

**Table 1-20: Number of Power Rails for Typical Configurations for Application Requiring Full Power Domain Flexibility (Cont'd)**

Configuration	Devices (Speed Grades)	Number of Power Regulators
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	EV (-3)	Ten (1, 2, 3, 4, 5, 6, 7, 8, 9, 15)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	CG (-1, -2) EG (-1, -2)	Ten (1, 2, 3, 4, 5, 6, 7, 8, 9, 16)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	EG (-3)	Ten (1, 2, 3, 4, 5, 6, 7, 8, 9, 16)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O, 1.8/2.5/3.3V PS I/O with no MGTs	EV (-3)	Eleven (1, 2, 3, 4, 5, 6, 7, 8, 9, 15, 16)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	EV (-1, -2)	Eleven (1, 2, 3, 4, 5, 6, 7, 8, 9, 15, 16)
Full power domain control, VCCINT = 0.72V/programmable, PS DDR, 1.8V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	CG (-1L, -2L) EG (-1L, -2L)	Eleven (1, 2, 3, 4, 5, 6, 7, 8, 9, 16, 17)
Full power domain control, VCCINT = 0.72V/programmable, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O and 1.8/2.5/3.3V PS I/O with no MGTs	EV (-1L, -2L)	Twelve (1, 2, 3, 4, 5, 6, 7, 8, 9, 15, 16, 17)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	CG (-1, -2) EG (-1, -2)	Fourteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EG (-3)	Fourteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EV (-3)	Fifteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EV (-1, -2)	Fifteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	CG (-1, -2) EG (-1, -2)	Fifteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 16)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EG (-3)	Fifteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 16)
Full power domain control, VCCINT = 0.9V, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EV (-3)	Sixteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16)
Full power domain control, VCCINT = 0.85V, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EV (-1, -2)	Sixteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16)
Full power domain control, VCCINT = 0.72/programmable, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	CG (-1L, -2L) EG (-1L, -2L)	Sixteen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16)
Full power domain control, VCCINT = 0.72/programmable, PS DDR, 1.8V PL I/O, DDR/2.5/3.3V PL I/O, 1.8/2.5/3.3V PS I/O, PL MGTs and PS MGTs	EV (-1L, -2L)	Seventeen (1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17)

## Maximum Current Draw for $V_{CCINT}$ in Virtex UltraScale+ Devices

UltraScale and UltraScale+ devices are designed to work over a wide temperature range, and the temperature of the device and package is directly related to the amount of current draw. This is especially relevant on the  $V_{CCINT}$  power rail, which is the rail that would typically require the most power and thus current draw of all the power rails in the device. For large Virtex UltraScale+ FPGA designs in very high-speed applications, it might be possible to exceed the maximum current that the package is rated for due to the effect of electromigration.

Table 1-21 is a guide to the maximum possible sustained current draw on the  $V_{CCINT}$  rail for Virtex UltraScale+ devices to guarantee reliable operation for ten years at a constant 100°C junction temperature.

Table 1-21: Maximum  $V_{CCINT}$  Current for Virtex UltraScale+ Devices

Package	Device	Max Possible $V_{CCINT}$ Current (A) (100°C, 10 Years)
FFVC1517	XCVU3P	40
FSVH1924	XCVU31P	68
FLGF1924	XCVU11P	102
FLGA2104/ FLVA2104	XCVU5P	120
	XCVU7P	120
	XCVU9P	99
FHGA2104	XCVU13P	92
FLGB2104/ FLVB2104	XCVU5P	93
	XCVU7P	93
	XCVU9P	93
	XCVU11P	118
FHGB2104	XCVU13P	114
FLGC2104/ FLVC2104	XCVU5P	126
	XCVU7P	126
	XCVU9P	137
	XCVU11P	130
FHGC2104	XCVU13P	118
FSGD2104	XCVU9P	217
	XCVU11P	200
FIGD2104	XCVU13P	200

Table 1-21: Maximum  $V_{CCINT}$  Current for Virtex UltraScale+ Devices (Cont'd)

Package	Device	Max Possible $V_{CCINT}$ Current (A) (100°C, 10 Years)
FSVH2104	XCVU33P	68
	XCVU35P	135
FLGA2577	XCVU9P	174
	XCVU11P	175
	XCVU13P	162
FSVH2892	XCVU35P	135
	XCVU37P	200

# PCB Guidelines for Memory Interfaces

---

## Overview

The Xilinx UltraScale architecture provides solutions for interfacing with the following memory interfaces:

- DDR4, for both programmable logic (PL) and the processing system (PS)
  - DDR3/3L (PL and PS)
  - LPDDR4 (PS)
  - LPDDR3 (PL and PS)
  - RLDRAM 3 (PL)
  - QDR-IV (PL)
  - QDR II+ (PL)
- 

## Reference Stackup

All electrical routing constraints are defined upon the reference stackup ([Table 2-1](#)). The actual stackup might be different from this reference stackup. The related constraints such as width and spacing should be adjusted accordingly to meet target impedance and crosstalk in the design guide at reference stackup. For reference, this particular stackup results in an inner signal layer propagation time of 169.5 ps/in.

Table 2-1: Reference Stackup

Layer	Thickness (mil): Copper/ Core	Description	
		Copper Weight (oz)	Layer
L1	2.5	0.5	Top
	2.9		
L2	0.6	0.5	Ground
	4.5		
L3	0.6	0.5	Signal
	4.5		
L4	0.6	0.5	Ground
	4.5		
L5	0.6	0.5	Signal
	4.5		
L6	1.2	1.0	Power/Ground
	8.0		
L7	1.2	1.0	Power/Ground
	8.0		
L8	1.2	1.0	Power/Ground
	8.0		
L9	1.2	1.0	Power/Ground
	8.0		
L10	1.2	1.0	Power/Ground
	8.0		
L11	1.2	1.0	Power/Ground
	4.5		
L12	0.6	0.5	Signal
	4.5		
L13	0.6	0.5	Ground
	4.5		
L14	0.6	0.5	Signal
	4.5		
L15	0.6	0.5	Ground
	2.9		
L16	2.5	0.5	Bottom

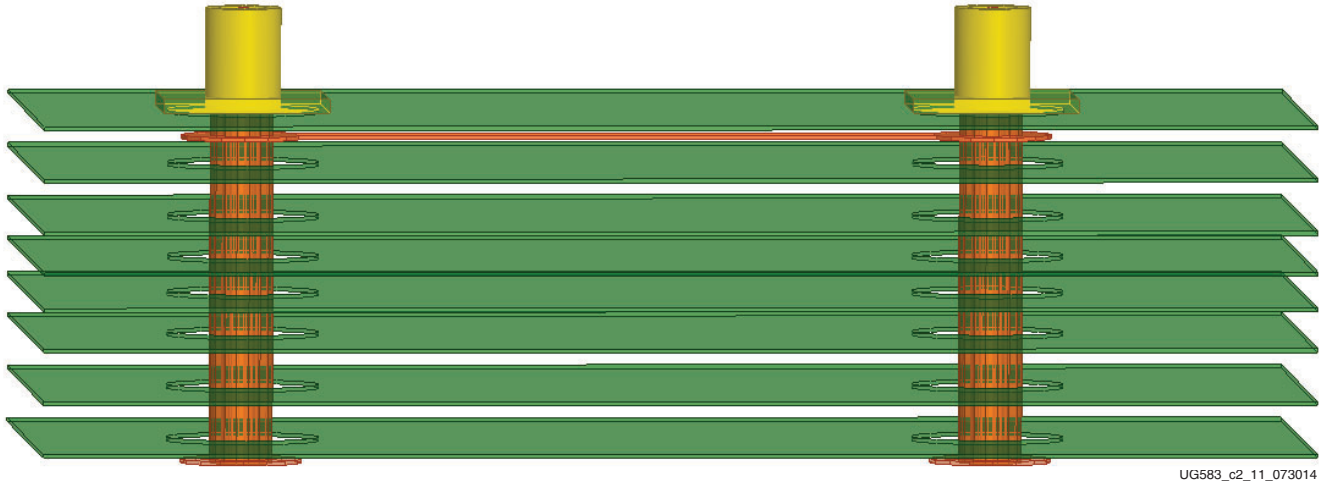
**Notes:**

1. The material for this reference stackup is Isola High-Tg FR-4, 370HR with Er = 4.0.





**IMPORTANT:** To achieve the highest memory interface performance, all the high-speed signals are recommended to be routed on the upper signal layers such as L3 and L5, as shown in [Figure 2-1](#), to minimize device pin field via crosstalk impact. Deeper signal layers could be used but signal routing spacing needs to take trade-offs into account with system-level signal integrity simulations.

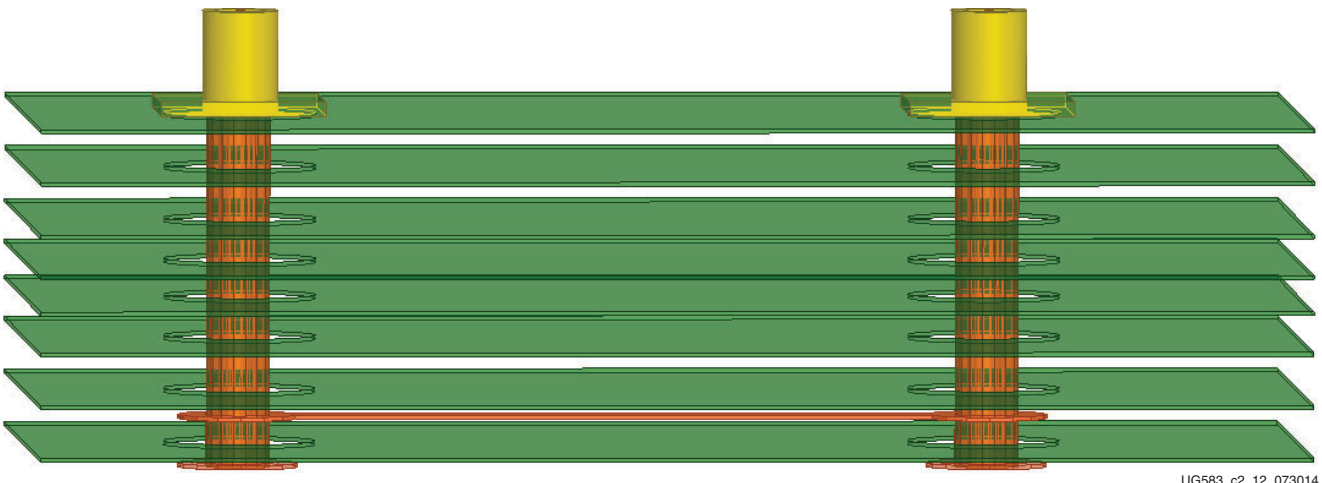


UG583\_c2\_11\_073014

Figure 2-1: Layer 3 Routing Example



**IMPORTANT:** Routing high-speed signals on lower signal layers comes with more board via coupling jitter depending on board thickness. Signal spacing in the same layer routing needs to be compromised to mitigate deep boards via crosstalk impact.



UG583\_c2\_12\_073014

Figure 2-2: Layer 14 Routing Example

To determine system timing margins in this design following the Xilinx memory simulation guidelines, system designers should run system-level memory channel simulations to confirm actual timing margin in customer-specific layout practices.

## General Memory Routing Guidelines

1. Include package delay in routing constraints when determining signal trace lengths. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values.
2. UltraScale device breakout specifications assume two signal routes between pads. If routing one signal between pads, traces can be  $39\Omega$  instead of  $50\Omega$ .



Figure 2-3: UltraScale Device Breakout Single-Ended Routing Options

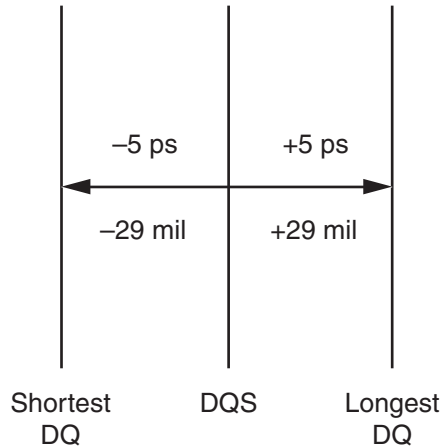
3. Signal-to-signal skew constraints in this chapter are presented in the form *X to Y*, where the Y signal is the reference point. Within the specified constraint, signal X can be shorter or longer than signal Y. If signal X is part of a bus, then the shortest and longest signals in the bus must be within the listed specification. If the Y signal is a differential clock or signal group, then Y is defined as the mid-point between shortest and longest signals in the pair/group.

Table 2-2 shows an example *DQ to DQS* signal-to-signal constraint specification.

Figure 2-4 illustrates an interpretation of the constraint. The shortest DQ in the data bus can be no more than 5 ps ahead of the DQS signal (mid-point of DQS\_P and DQS\_N), while the longest DQ in the bus can be no more than 5 ps after the DQS signal.

Table 2-2: Example Signal to Signal Skew Constraint

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
DQ to DQS	±5	±29



UG583\_c2\_104\_010816

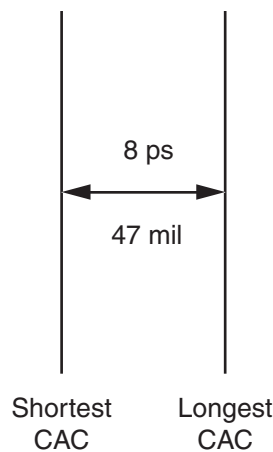
Figure 2-4: Skew Example for DQ to DQS

- Signal group skew constraints in this chapter are presented in the form  $X$ . The difference between the longest signal and the shortest signal in the  $X$  group must be no greater than the specified constraint.

Table 2-3 shows an example *Address* group constraint specification. Figure 2-5 illustrates an interpretation of the constraint. The difference in delay between the longest address signal in the bus and the shortest address in the bus can be no greater than 8 ps.

Table 2-3: Example Signal Group Skew Constraint

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Address, command, control (CAC)	8	47



ug583\_c2\_105\_121115

Figure 2-5: Skew Example for Address, Command, Control

- The CK to DQS specification for DDR3 and DDR4 component interfaces encompasses a wide range, as shown in Table 2-4. The wide range is to ensure that proper write leveling can take place at all memory devices, from the first in the chain to the last.

Table 2-4: Example CK to DQS Skew Constraint

Signal Group	Skew Constraint (ps)	Skew Constraint (mil)
CK to DQS	-149 to 1796	-879 to 10,600

The UltraScale device memory controller can internally delay the DQS line to account for negative skew, which helps because the CK line encounters more capacitive load than each individual DQS pair because the CK lines touch each memory device in the chain. This slows the CK line relative to each DQS, which only touches one memory device. The specification routes the CK and DQS lines where the skew between CK to DQS is no less than -149 ps from the UltraScale device to the first memory device in the chain and no more than 1796 ps to the furthest memory device in the chain. This is illustrated in Figure 2-6. As long as the first memory device and the last memory device in the chain are bounded by this constraint, all memory devices can be write leveled.

For example, if the DQS delay between the UltraScale device and the first memory device is 200 ps, the clock delay from the UltraScale device to the first memory device should be at least 51 ps (200 ps - 149 ps). If the DQS delay from the UltraScale device to the furthest memory device is 700 ps, the clock delay from the UltraScale device to the last memory device should be less than 2496 ps (700 ps + 1796 ps).

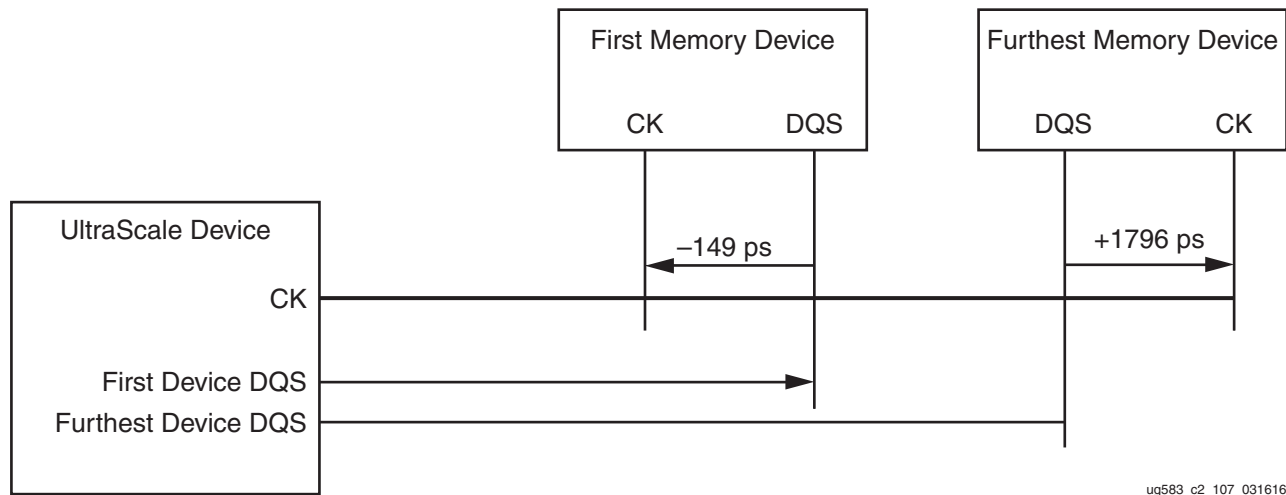
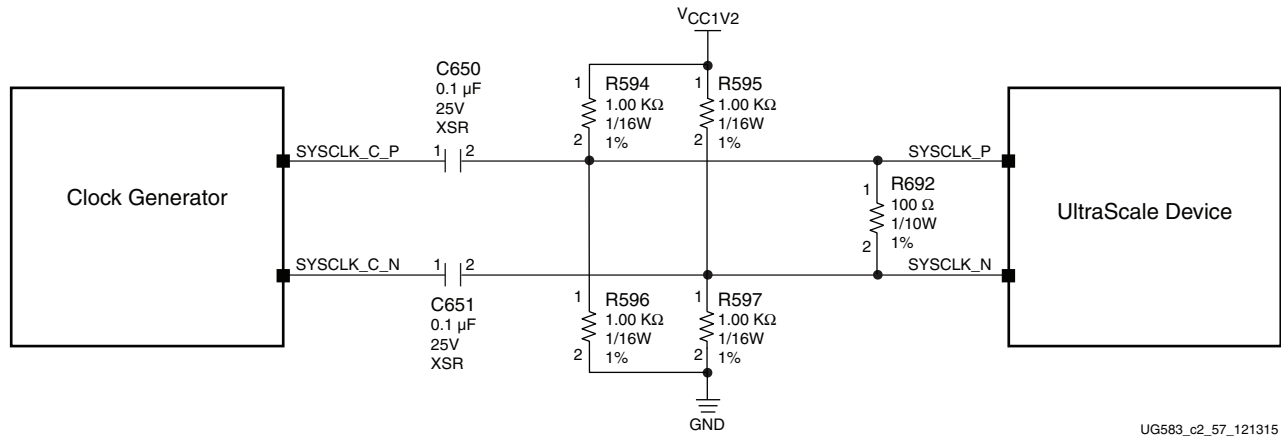


Figure 2-6: Interpretation of CK to DQS Skew Specification



**TIP:** DIMM guidelines have a much narrower CK to DQS specification because the PCB is only routed from the UltraScale device to the first memory device in the DIMM. The routing from the first memory device to the last memory device is managed by the DIMM.

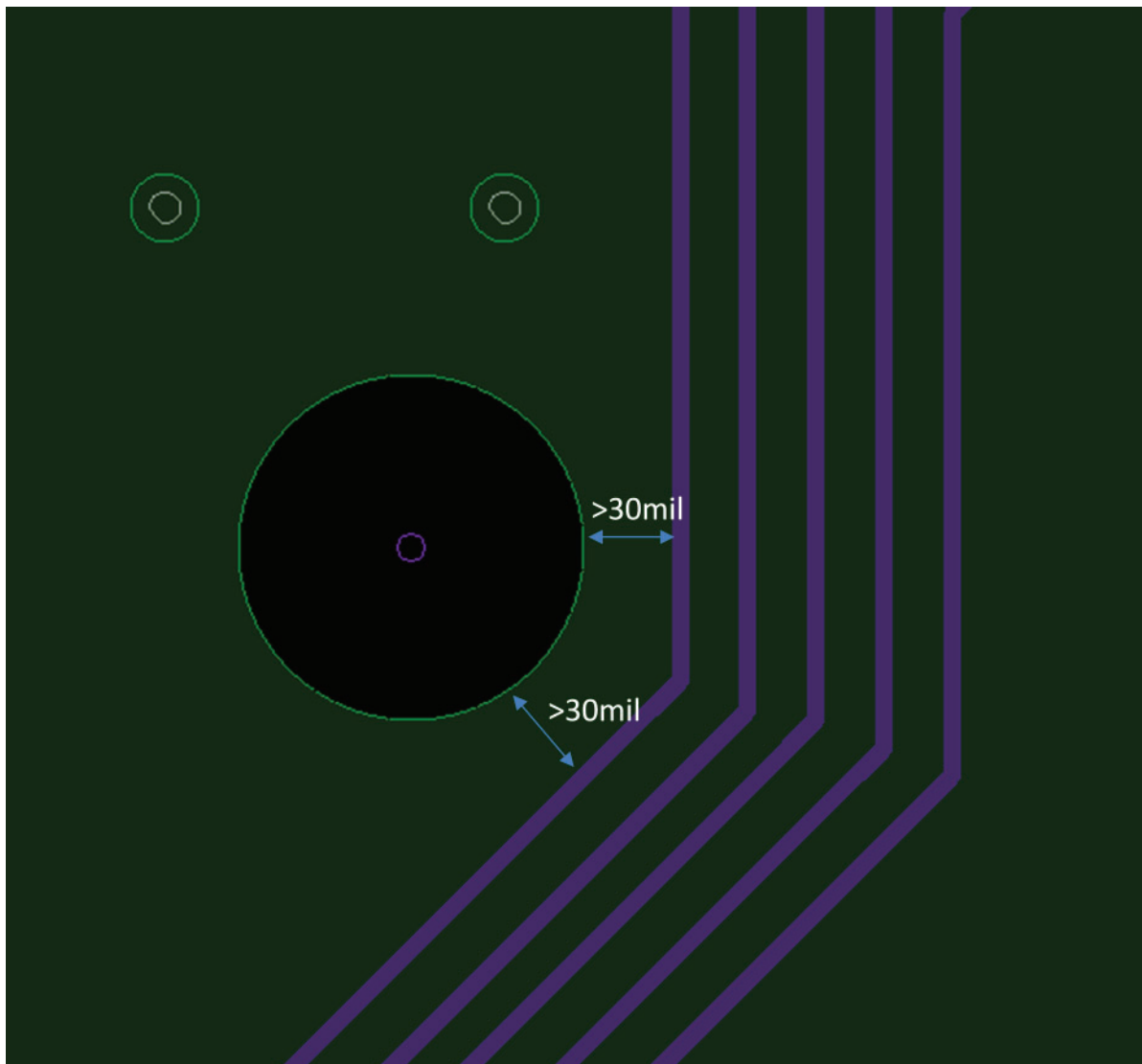
6. Skew constraints assume the fastest interface speeds. For slower speeds, certain skews can be relaxed. Refer to [Appendix A, Memory Derating Tables](#) for derating tables for the various memory interfaces.
7. Skew and length constraints assume a propagation time of 169.5 ps/in based on the stackup shown in [Table 2-1](#). Maximum trace lengths can be relaxed or tightened depending on the actual propagation time of the board. To convert, multiply the specified trace length by 169.5 ps, then divide by the actual propagation time of the board. See [Adjusting for Different Stack-Ups](#).
8. Trace widths and spacing are based on the stackup shown in [Table 2-1](#). If another stackup scheme is used, the widths and spacing can be modified to meet the impedance targets. See [Adjusting for Different Stack-Ups](#).
9. DQ and DQS signals in the same byte group should be routed in the same layer from UltraScale device to DRAM/DIMM, except in the breakout areas. Include the data mask (DM) in the byte group as applicable.
10. Do not change layers when routing from one DIMM to the next, if applicable. In addition, for DIMM routing, it is recommended to route data byte groups on the highest signal layers (closest to the DIMM connector) as much as possible, especially for byte groups located near the center of the DIMM.
11. For fly-by routing, address, command, and control signals can be routed on different layers, though it is recommended to use as few as possible. Do not route any individual signal on more than two layers to minimize inductive loops that can lead to crosstalk issues. Any signal layer switching via needs to have one ground via within a 50 mil perimeter range.
12. UltraScale device and memory drive strengths are assumed to be 40Ω. UltraScale device DCI and memory ODT are assumed to be 40Ω.
13. When utilizing the internal  $V_{REF}$ , Xilinx allows the  $V_{REF}$  pins to float in the banks that use the internal  $V_{REF}$ .  
**Note:** When internal  $V_{REF}$  is used, this pin cannot be used as an I/O.
14. If the system clock is connected to a bank that is also used for memory, terminate as shown for LVDS in [Figure 2-7](#) with the appropriate pull-up voltage. This termination circuit is necessary because of the different I/O standard of the memory bank (HSTL, SSTL, or POD).



UG583\_c2\_57\_121315

Figure 2-7: System Clock

15. Signal lines must be routed over a solid reference plane. Avoid routing over voids (Figure 2-8).



UG583\_c2\_13\_050614

Figure 2-8: Signal Routing Over Solid Reference Plane

16. Avoid routing over reference plane splits (Figure 2-9).

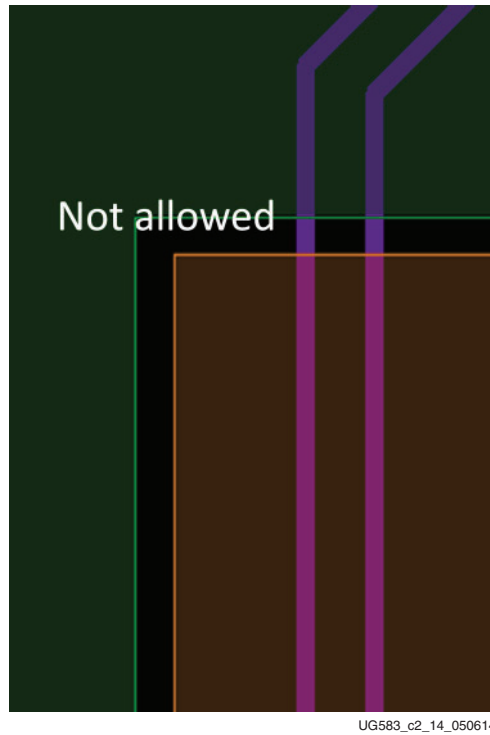


Figure 2-9: Signal Routing Over Reference Plane Split

17. Keep the routing at least 30 mils away from the reference plane and void edges with the exception of breakout regions (Figure 2-8).



- In the breakout region, route signal lines in the middle of the via void aperture. Avoid routing at the edge of via voids (Figure 2-10).

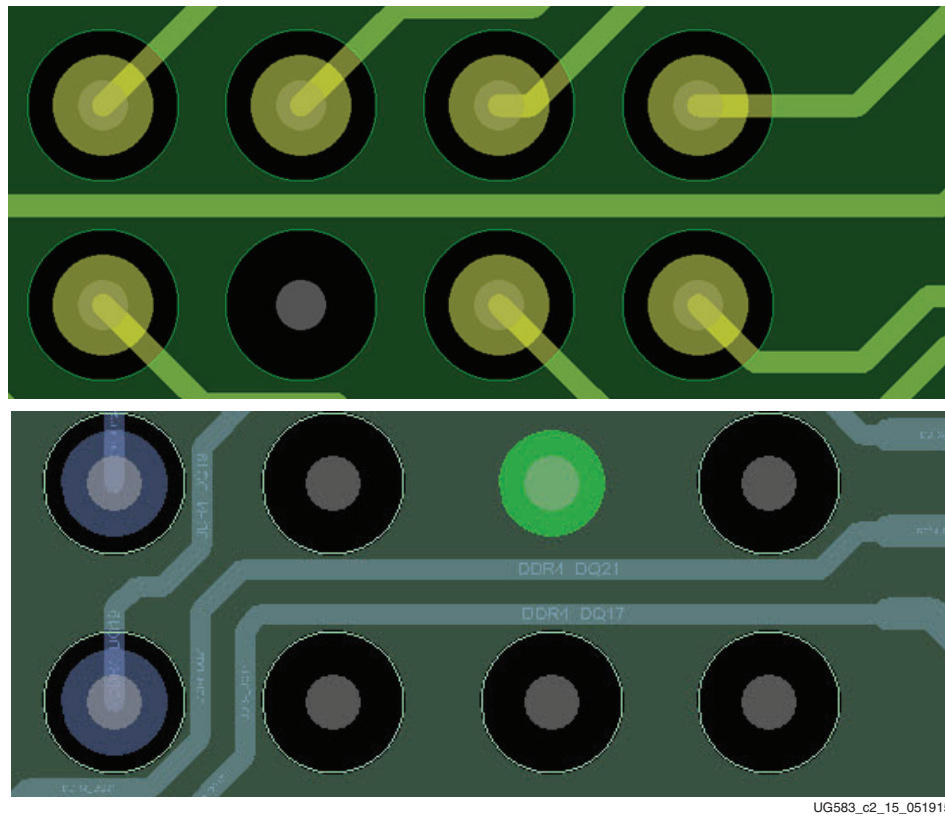
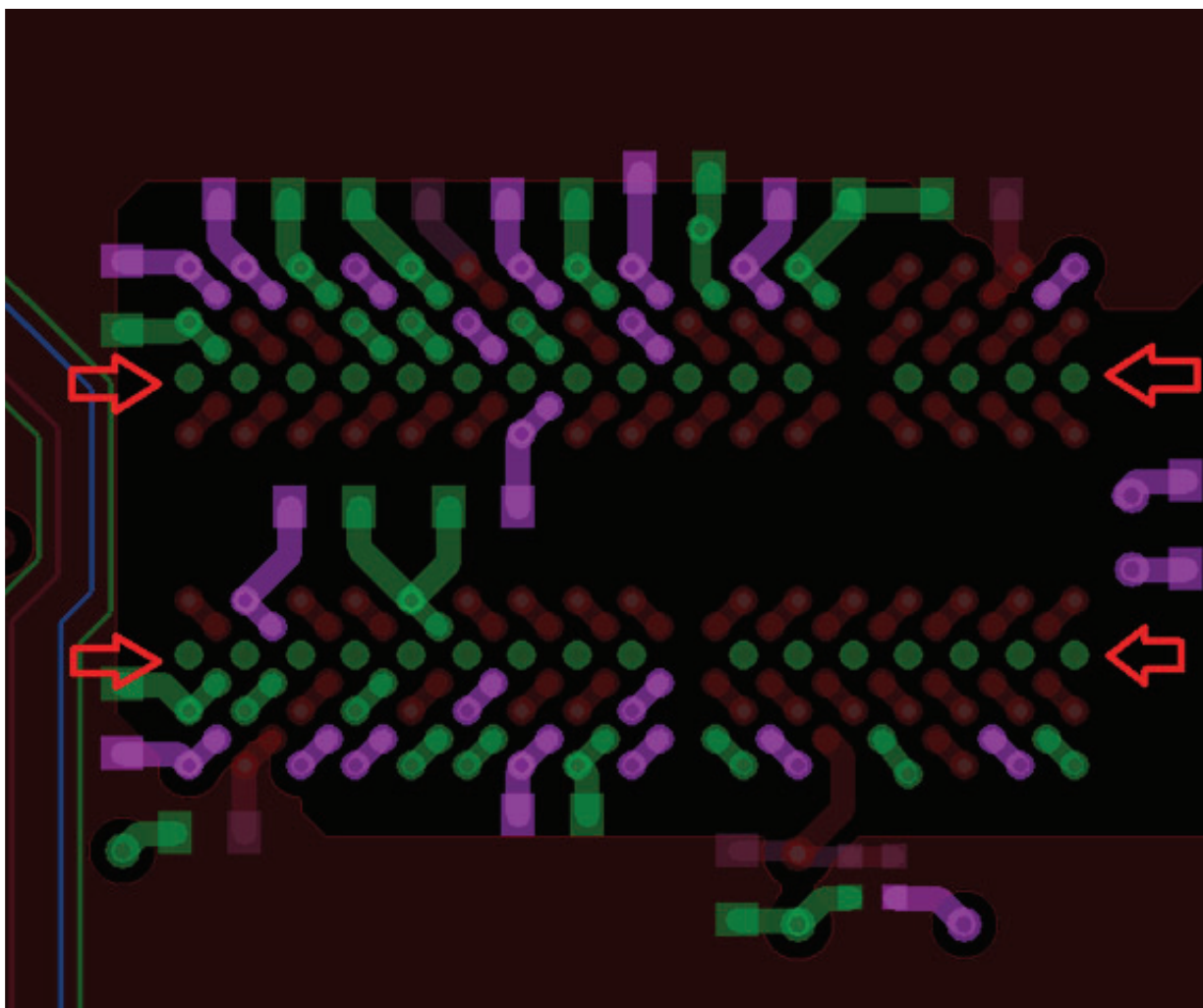


Figure 2-10: Breakout Region Routing

19. Use chevron-style routing to allow for ground stitch vias. Figure 2-11 shows recommended routing for fly-by configurations, while Figure 2-12 shows recommended routing to accommodate ground stitch vias in a more congested clamshell configuration.

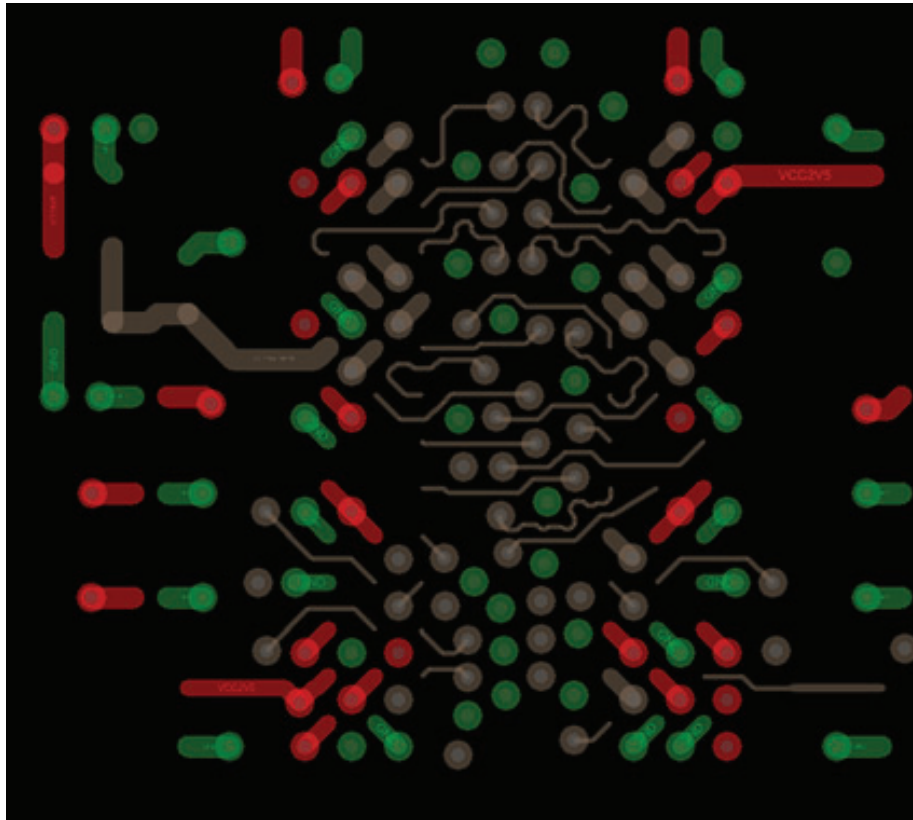


**IMPORTANT:** Crosstalk issues leading to data errors can occur with lack of proper ground stitching, especially in areas where there are fewer ground pins, such as near address pins of memory devices.



UG583\_c2\_16\_050614

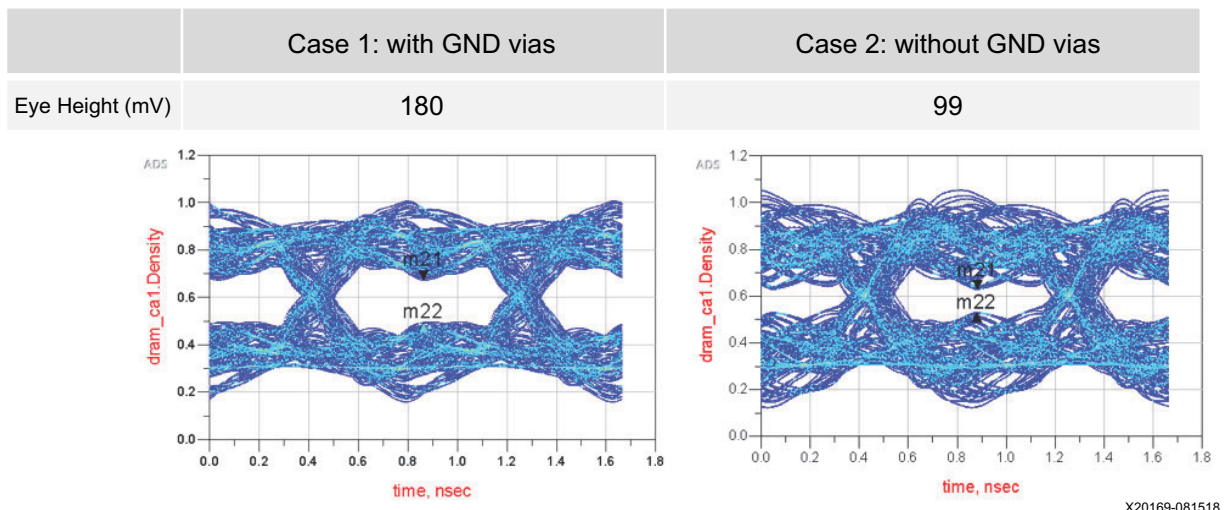
Figure 2-11: Example of Ground Stitching (Fly-by)



UG583\_c2\_59\_090816

Figure 2-12: Example of Ground Stitching (Clamshell) Red: Power, Green: Ground

Figure 2-13 shows simulated eye diagrams for a DDR4 command/address/control bit with and without ground stitching vias. The simulation on the left shows an eye height of 180 mV with ground stitch vias, while the simulation on the right shows an eye height of only 99 mV when not utilizing ground stitch vias.



X20169-081518

Figure 2-13: Simulations With and Without Ground Stitching Vias

20. Add ground vias as much as possible around the edges and inside the device (FPGA, MPSoC, memory component, DIMM) to make a better ground return path for signals and power, especially corners. Corner or edge balls are generally less populated as grounds.
21. For address/command/control  $V_{TT}$  termination, every four termination resistors should be accompanied by one 0.1  $\mu\text{F}$  capacitor, physically interleaving among resistors, as shown in Figure 2-14. Refer to the memory vendor's data sheet for specifications regarding noise limits on the address/command/control  $V_{TT}$  lines.

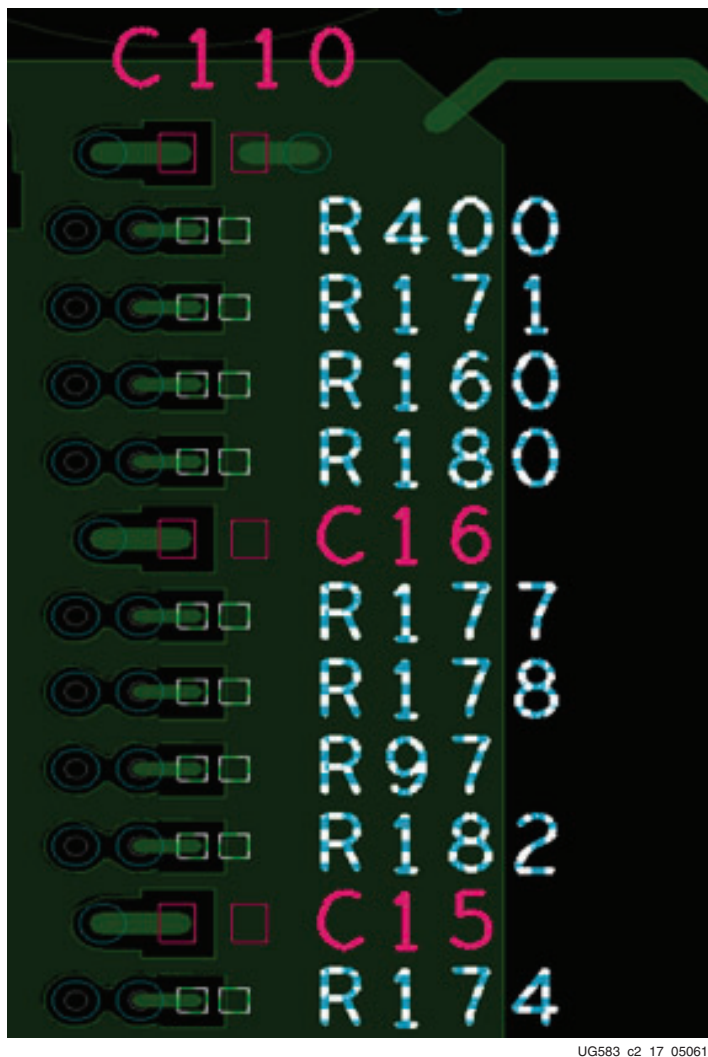
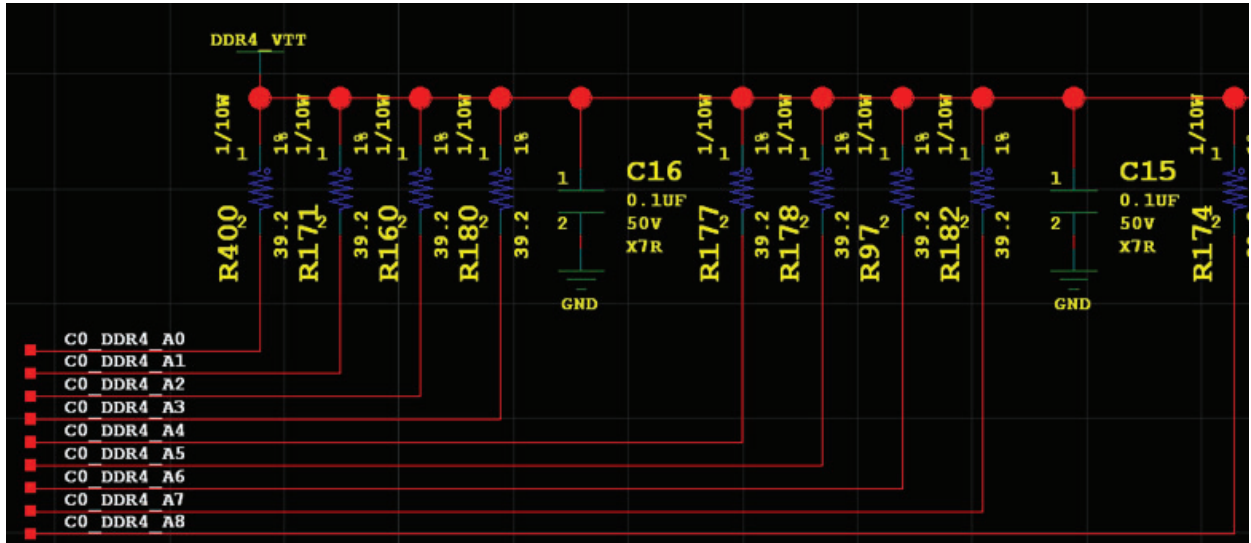


Figure 2-14: Example of  $V_{TT}$  Termination Placement

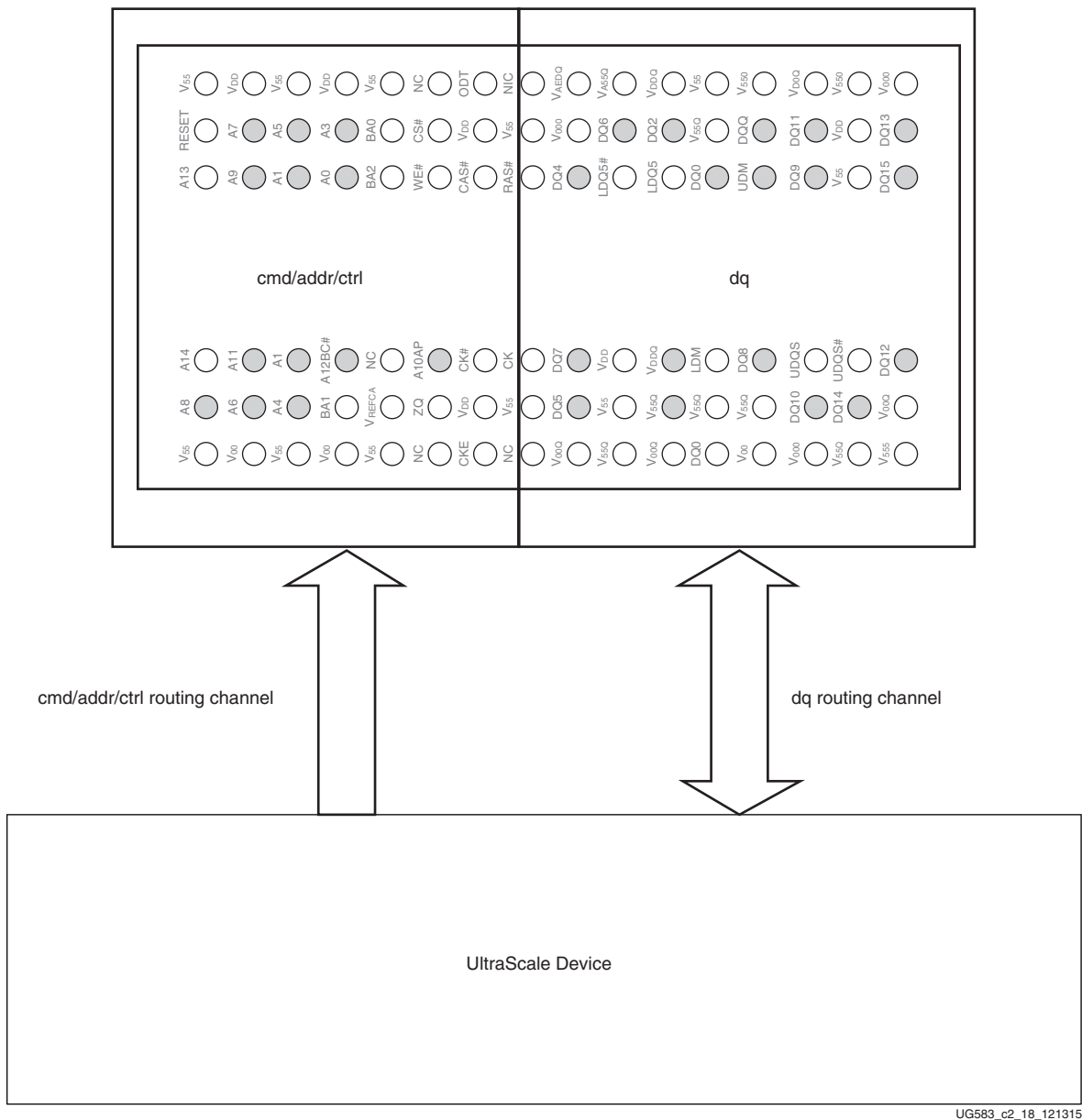


UG583\_c2\_60\_090816

Figure 2-15: Schematic Example of VTT Resistor and Capacitor Connections

22. For DIMMs, place bypass capacitors near the command/address/control pads to provide extra ground via locations. In addition, the bypass capacitors also provide a lower impedance path from power to ground, which is important because address/command/control pins are referenced to ground on the FPGA and PCB, while they are referenced to power on the DIMM.

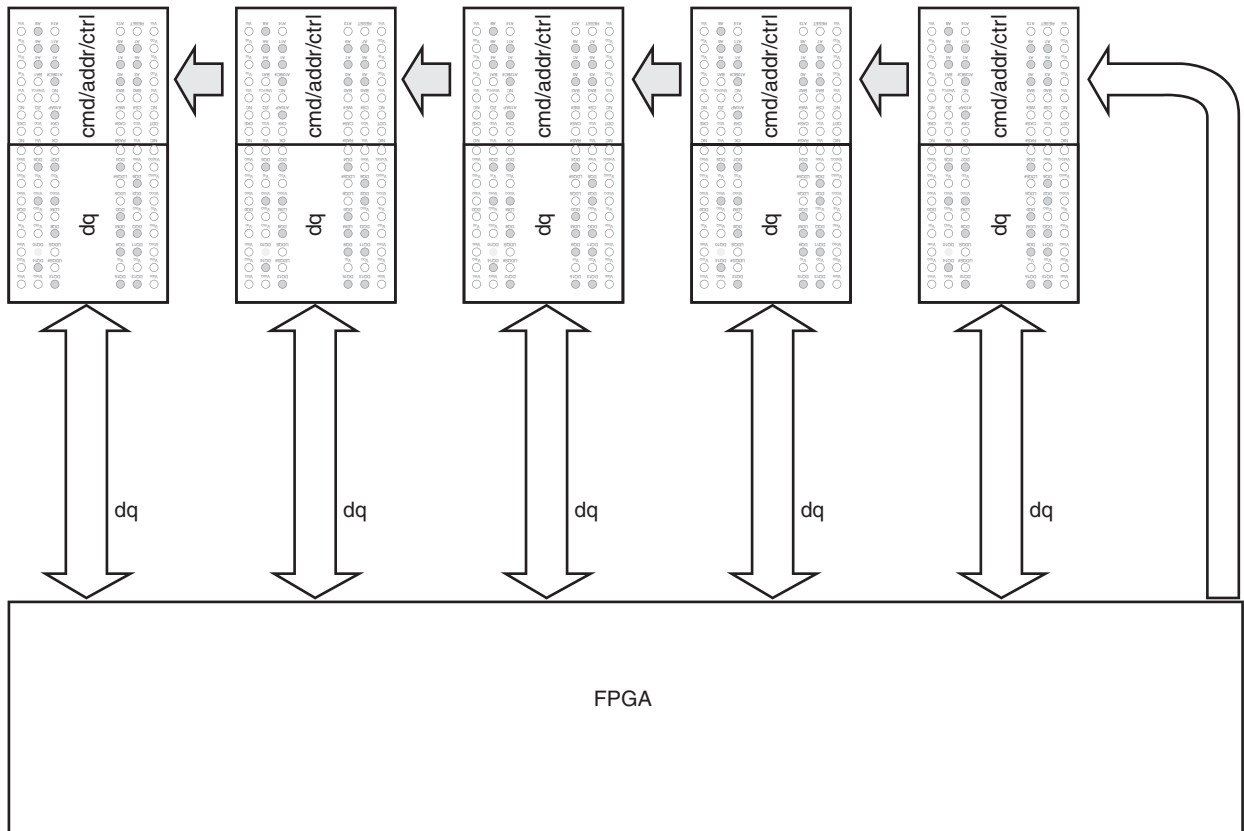
23. To optimize the signal routing, the recommendation for one component placement is shown in Figure 2-16.



UG583\_c2\_18\_121315

Figure 2-16: Component Placement Recommendations for One Component

For five components, the recommendation is shown in Figure 2-17.



UG583\_c2\_19\_073014

Figure 2-17: Component Placement Recommendations for Five Components

## Adjusting for Different Stack-Ups

The trace width, spacing, length, and skew constraints presented in this chapter are based on the reference stackup shown in Table 2-1. When not using this stackup, trace width, lengths, layer heights, spacings, and dielectric material could need adjustments to meet the impedance, length, and skew specifications.

The PCB fabrication house can adjust these factors to achieve the desired impedance and propagation delay targets. The effects of each of these items are listed in this section. In addition, a two-dimensional field solver utility allows for various combinations to be tested.

## Dielectric Material

Each dielectric material has its own relative dielectric constant ( $D_K$ ) and dissipation factor/loss tangent ( $D_F$ ) that are contributing factors to line impedance ( $Z_0$ ), signal propagation delay ( $T_{PD}$ ), and signal loss ( $\alpha$ ). As  $D_K$  increases, impedance decreases, while

the signal propagation delay and signal loss increase, and vice-versa. A typical  $D_K$  found on PCBs range from 3.4 to 4.6. The propagation delay in a given dielectric material is constant and is not affected by any other board parameters such as layer height, conductor width, or conductor spacing. Propagation delay is affected by frequency, but the effect is minimal with regards to typical memory speeds. Signal loss is also affected by frequency, with the loss increasing with increasing frequency.

Equation 2-1 shows the calculation for propagation delay ( $T_{PD}$ ), with  $D_K$  as the dielectric constant, and  $c$  as the speed of light in free space ( $2.998 \times 10^8$  m/s or  $1.180 \times 10^{10}$  in/s).

$$T_{PD} = \frac{\sqrt{D_K}}{c} \quad \text{Equation 2-1}$$

The associations in Table 2-5 show the effect of the dielectric constant ( $D_K$ ) on the impedance ( $Z_0$ ), propagation delay ( $T_{PD}$ ), and signal loss ( $\alpha$ ).

**Table 2-5: Relationship of  $D_K$  to Impedance, Propagation Delay, and Signal Loss**

$D_K \uparrow$	$Z_0 \downarrow$	$T_{PD} \uparrow$	$\alpha \uparrow$
$D_K \downarrow$	$Z_0 \uparrow$	$T_{PD} \downarrow$	$\alpha \downarrow$

## Trace Width

As the trace width ( $W$ ) increases, the impedance decreases, while the signal propagation delay remains unchanged, and vice-versa. Any adjustments in width should include adjustments in spacing ( $S$ ) to maintain immunity to crosstalk effects. Spacing factors are roughly 1.0X to 3X depending on the particular type of memory and signal. The associations in Table 2-6 show the effect of width on impedance, spacing, and propagation delay.

**Table 2-6: Relationship of Trace Width to Impedance, Required Spacing, and Propagation Delay**

$W \uparrow$	$Z_0 \downarrow$	$S \uparrow$	$T_{PD}$ (no change)
$W \downarrow$	$Z_0 \uparrow$	$S \downarrow$	$T_{PD}$ (no change)

## Layer Height

As the layer height ( $H$ ) is increased, the impedance increases, while the signal propagation delay remains unchanged, and vice-versa. When reducing or increasing layer height, consider that layer heights that are too low can be more expensive to reliably manufacture, because the PCB fabricator must avoid plane shorts. Layer heights that are too high can lead to aspect ratio violations. The associations in Table 2-7 show the effect of layer height on impedance and propagation delay.

**Table 2-7: Relationship of Layer Height to Impedance and Propagation Delay**

$H \uparrow$	$Z_0 \uparrow$	$T_{PD}$ (no change)
$H \downarrow$	$Z_0 \downarrow$	$T_{PD}$ (no change)



## Base Copper Weight

As the base copper weight (Cu oz) of the conducting lines is increased, the impedance will decrease, while the signal propagation delay remains unchanged, and vice-versa. The most common base copper weight used in PCBs is 0.5, known as half-ounce copper. The effect of changing the base copper weight is minimal, however, and is not recommended due to the potential for increased costs in relation to the small benefit typically gained.

## Example of Tuning Design Parameters to Meet Impedance Targets

Table 2-8 shows a target of  $36\Omega$  for stripline impedance in the main L1 PCB area. The trace width (W) is specified as 7.0 mils along with 8.0 mil spacing (S). The reference PCB stackup in Table 2-1 defines the layer composition and dielectric material that will result in reaching  $36\Omega$ . This consists of a 4.0 dielectric constant ( $D_K$ ), and 9.6 mil layer height (H) ( $4.5 + 0.6 + 4.5$ ). Table 2-8 shows tuning of the trace width (W) and height (H) required to meet the impedance target. The spacing (S) shows how it should change to continue to meet performance requirements, though the spacing is not a factor in the line impedance. When increasing  $D_K$ , the signal loss also increases per Table 2-5.

Table 2-8: Example of Tuning PCB Parameters to Meet Impedance Target

Target ( $Z_0$ ) $\Omega$	Material	$D_K$	W (mil)	S (mil)	H (mil)	Description
36	Isola FR-4 370H	4.00	7.0	8.0	9.6	See Table 2-1.
36	Megtron 6	3.71	7.0	8.0	9.2 ↓	Same width and spacing as reference.
36	Megtron 6	3.71	7.5 ↑	8.5 ↑	9.6	Same height as reference.
36	FR4 (standard)	4.60	7.0	8.0	10.6 ↑	Same width and spacing as reference.
36	FR4 (standard)	4.60	6.0 ↓	7.0 ↓	9.6	Same height as reference.

## PCB Guidelines for DDR4 SDRAM (PL and PS)

### Overview

This section provides electrical design guidelines for typical DDR4 SDRAM interfaces (up to nine devices) from system-level signal integrity simulations for the UltraScale architecture. A five-component DDR4 80-bit wide interface using x16 devices is used as a design example for a fly-by topology, while a nine-component example is utilized for a clamshell topology.

**Note:** For applications that utilize the Zynq UltraScale+ device Video Codec Unit (VCU) and DDR4 memory in the PL, refer to [Xilinx Answer 71209](#) for important design considerations.



**IMPORTANT:** All routing guidelines in this section must be followed to achieve the maximum data rates specified for the DDR4 SDRAM interface for typical system designs. Customers could have unique or specific designs with particular violations of some rules. In these scenarios, design or routing trade-offs have to be taken in other routing parameters to mitigate the risk. System-level channel signal integrity simulations are required to evaluate such trade-offs. It is important to read the [General Memory Routing Guidelines](#) section before continuing with this section.

## DDR4 SDRAM Interface Signal Description

The DDR4 SDRAM interface consists of clock, control, address, and data signals as shown in [Table 2-9](#).

Table 2-9: DDR4 SDRAM I/O Signal Description

Signal Name	Description
<b>Clock Signals</b>	
ck_t, ck_c	Differential clock
<b>Address and Command Signals</b>	
a[17:0]	Address inputs
bg[1:0]	Bank group inputs
ba[1:0]	Bank address inputs
we	Write enable (might be shared with address)
ras	Row address strobe (might be shared with address)
cas	Column address strobe (might be shared with address)
act_n	Activation command input
par	Command and address parity input
<b>Control Signals</b>	
cke	Clock enable
cs_n	Chip select (fly-by)
CS0_n, CS1_n	Chip select
odt	On-die termination enable
reset_n	See <a href="#">reset_n</a>
<b>Data Signals</b>	
dq[79:0]	Data input/output
dqs_t/dqs_c[9:0]	Data strobe (differential)
dm_n/dbi_n	Data mask and data bus inversion
<b>Other Signals</b>	
TEN	Test connectivity mode. CMOS level. Connect 500Ω resistor to ground at memory devices.

Table 2-9: DDR4 SDRAM I/O Signal Description (Cont'd)

Signal Name	Description
alert_n	CRC error flag, Command and Address Parity error flag. Open drain output from the DRAM requires a 50Ω pull-up to V <sub>CC0</sub> .

**Notes:**

- Actual signal list might vary based on configuration.

## Fly-by and Clamshell Topologies

Two topology types are supported for DDR4 SDRAM: fly-by, and clamshell. The fly-by topology (Figure 2-18) consists of all memory devices on one layer, usually in-line. This type of topology is generally easier to route and can offer the best signal integrity, but can take up precious board real estate.

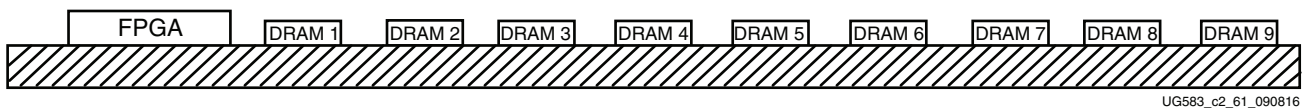


Figure 2-18: Fly-by Topology

Clamshell topology (Figure 2-19) requires more intricate routing, but is optimal for designs where board space is at a premium.



**IMPORTANT:** Clamshell is a supported DDR4 SDRAM topology in the Memory Interface Generator (MIG) and is selectable for Programmable Logic banks only. The PS in the Zynq UltraScale+ MPSoC does not have a selectable clamshell configuration option. However, the PS can be configured as clamshell if set up as a dual-rank configuration with the first rank on the top layer, and the second rank mirrored on the bottom layer. When utilizing this topology within the Vivado tools, refer to the Clamshell Topology section in UltraScale Architecture-Based FPGAs Memory IP Product Guide (PG150) [Ref 13] for additional information.

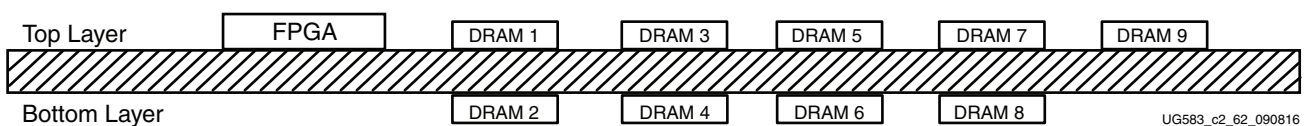


Figure 2-19: Clamshell Topology

## Utilizing Address Mirroring to Ease Clamshell Routing

While the clamshell topology has advantages such as reduced board space, the asymmetrical nature of the memory device package pin locations can lead to higher routing congestion in the areas under the memory devices between the top and bottom layers. This routing congestion could potentially lead to higher crosstalk due to the inability to include as many ground-return vias as in other topologies (see item 21 in [General Memory Routing Guidelines](#)), as well as longer stub traces on the top and bottom layers. Address mirroring

can be used to change the function of certain pins on a memory device to correspond to the pin directly above or below it. One via can be used for the signal, along with a short stub to the landing pad for each device. Address mirroring is defined in JEDEC specification JESD21-C. Twelve command/address/control pins can be mirrored for DDR4 SDRAM, according to Table 2-10.

Table 2-10: Allowable Mirror Pins for DDR4 SDRAM

Memory Controller Pin	DRAM Pin (Non-Mirrored)	DRAM Pin (Mirrored)
A3	A3	A4
A4	A4	A3
A5	A5	A6
A6	A6	A5
A7	A7	A8
A8	A8	A7
A11	A11	A13
A13	A13	A11
BA0	BA0	BA1
BA1	BA1	BA0
BG0 <sup>(1)</sup>	BG0	BG1
BG1 <sup>(1)</sup>	BG1	BG0

**Notes:**

1. BG0 and BG1 can only be mirrored when pin BG1 is present on the memory device.

## Topology and Routing Guidelines for DDR4 SDRAM

### DDR4 SDRAM Address, Command, and Control Fly-by and Clamshell Topologies

Figure 2-20 illustrates fly-by topology for address, command, and control signals. Each clock, address, command, and control pin on each SDRAM is connected to a single trace and terminated at the far end.

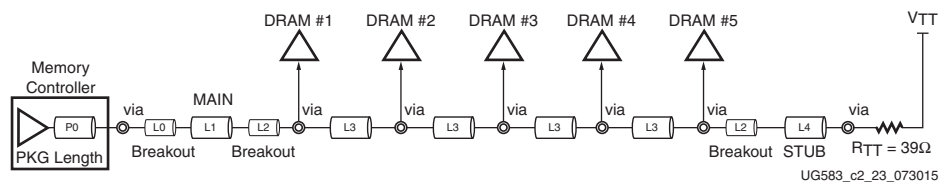


Figure 2-20: Address, Command, and Control Fly-by Termination for DDR4 SDRAM

**reset\_n**

Figure 2-21 shows the termination for reset\_n. The 4.7 kΩ to ground is to keep reset\_n Low during FPGA power-up and reconfiguration. If self-refresh is required during FPGA power-down or reconfiguration, circuitry needs to be added to ensure that reset\_n stays High during that time.

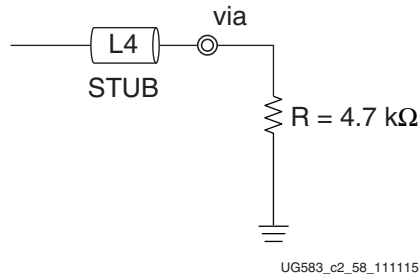


Figure 2-21: Termination for reset\_n in DDR4 DRAM

**alert\_n**

The alert\_n signal should be wired as a long fly-by connection with the FPGA at one end. Connections are from the alert\_n pin of the SDRAMs to the alert\_n pin of the FPGA. There is a pull-up resistor to V<sub>CC0</sub> at the end farthest electrically from the FPGA. Figure 2-22 shows the alert\_n termination.

**Note:** The PL memory interface does not use the alert\_n signal. Nonetheless, alert\_n from all DRAM devices should be tied together and terminated to V<sub>CC0</sub>.

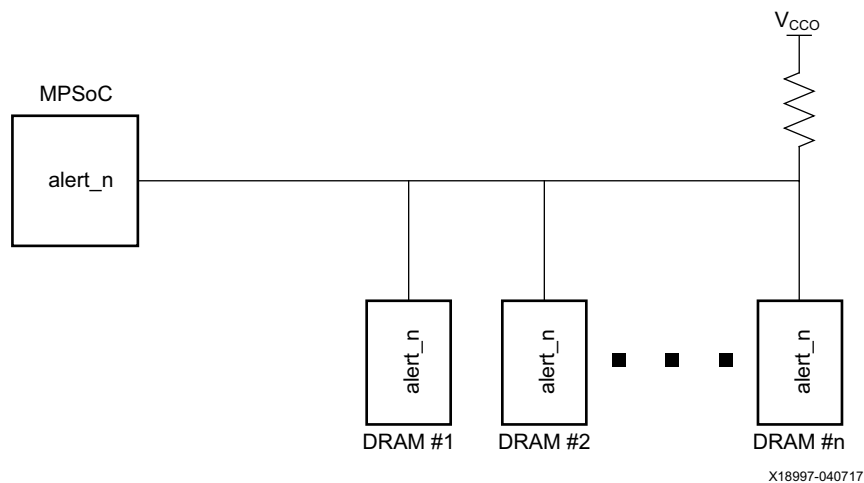


Figure 2-22: Termination for alert\_n in DDR4 SDRAM

Table 2-11 shows the DDR4 SDRAM fly-by impedance, length, and spacing guidelines for address, command, and control signals.

Table 2-11: DDR4 SDRAM Fly-by Impedance, Length, and Spacing Guidelines for Address, Command, and Control Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance Z <sub>0</sub>	50±10%	36±10%	50±10%	50±10%	39±10%	Ω
Trace width	4.0	7.0	4.0	4.0	6.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	0.0~0.1	0.35~0.75	0~1	inches
Spacing in address, command, and control signals (minimum)	4.0	8.0 <sup>(2)</sup>	4.0	8.0	8.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count	7					–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 12.0.

For DDR4 SDRAM clamshell topology, an alternating fly-by topology is recommended for control/address/command signals. The alternating layer routing properly balances the signal loads at each memory device. As depicted in [Figure 2-23](#) with the FPGA located at the top layer, the inner layer routing to top layer devices 1, 3, 5, 7, and 9 is closer to the top layer, while the inner layer routing to bottom layer devices 2, 4, 6, and 8 is closer to the bottom layer.

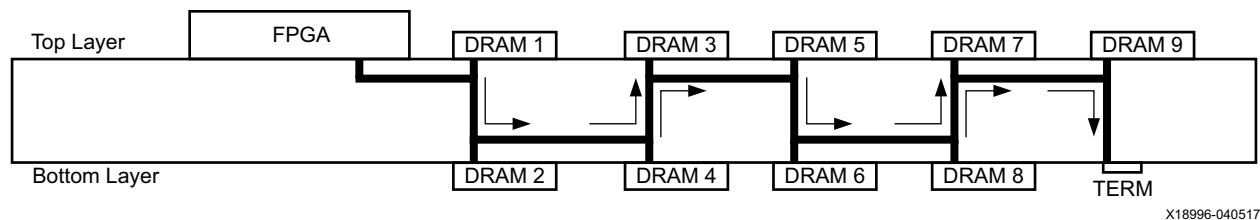


Figure 2-23: Address/Command/Control Topology for DDR4 SDRAM Clamshell

**Note:** The end-termination resistor can be located on either the top or bottom layer.

DDR4 SDRAM clamshell topology utilizes two individual chip select (CS) signals, one for the top layer DRAMs, and one for the bottom layer DRAMs. Those signals should be routed as in the standard fly-by topology as defined in [Figure 2-20](#) and [Table 2-11](#).

[Table 2-12](#) shows the DDR4 SDRAM clamshell impedance, length, and spacing guidelines for address, command, and control signals. Note the extra length of the L2 segment.

Table 2-12: DDR4 SDRAM Clamshell Impedance, Length, Width, and Spacing Guidelines for Address/Command/Control Signals

Parameter	L0 FPGA Breakout	L1	L2	L3	L4	L5	L6 V <sub>TT</sub> Stub	Units
Layer (recommended)	Upper inner	Upper inner	Lower inner	Upper inner	Top	Bottom	Bottom	
Impedance Z <sub>0</sub>	50	50	50	50	50	50	39	Ω
Length	0.0~1.5 <sup>(1)</sup>	0.0~4.0	L3+0.2	0.45~0.85	For mirrored case, L4 = L5; for non-mirrored case, make as short as possible		≤0.95	inch
Width	4.0	4.0	4.0	4.0	4.0	4.0	6.0	mil
Spacing within group	4.0	8.0 <sup>(1)</sup>	8.0	8.0	8.0	8.0	8.0	mil
Spacing to clocks	8.0	20	20	20	20	20	20	mil
Spacing to other groups	8.0	30	30	30	30	30	30	mil

**Notes:**

1. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 12.0.



**IMPORTANT:** Add as many ground vias as possible to help avoid crosstalk issues. See Item 20 in [General Memory Routing Guidelines](#).

### DDR4 SDRAM Clock Fly-By and Clamshell Termination

Inherent to fly-by topology, the timing skew between the clock and dq<sub>s</sub> signals is deskewed by the write-leveling feature on DDR4 SDRAM (Figure 2-24).

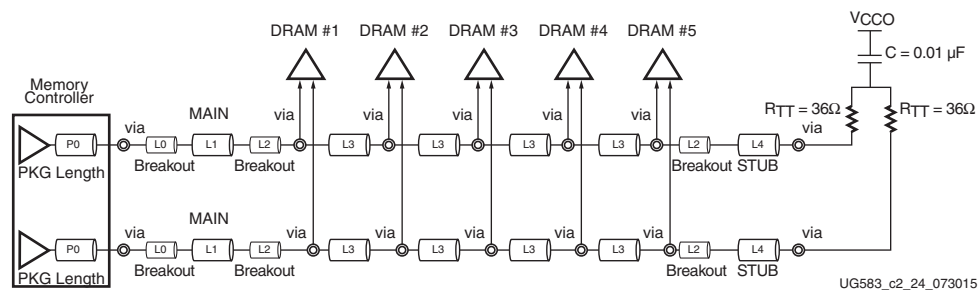


Figure 2-24: Clock Fly-by Termination for DDR4 SDRAM

Table 2-13 shows the DDR4 SDRAM impedance, length, and spacing guidelines for clock signals.

Table 2-13: DDR4 SDRAM Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance Z <sub>DIFF</sub>	86±10%	76±10%	86±10%	90±10%	76±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	4.0/5.0/4.0	6.0/6.0/6.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	0.0~0.1	0.35~0.75	0~1.0	inches
Spacing in address, command, and control signals (minimum)	8.0	20 <sup>(2)</sup>	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	mil
Maximum PCB via count per signal	7					–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).
- If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 30.

For DDR4 SDRAM clamshell topology, an alternating fly-by topology is recommended for clock signals. The alternating layer routing properly balances the signal loads at each memory device. As depicted in Table 2-27 with the FPGA located at the top layer, the inner layer routing to top layer devices 1, 3, 5, 7, and 9 is closer to the top layer, while the inner layer routing to bottom layer devices 2, 4, 6, and 8 is closer to the bottom layer.

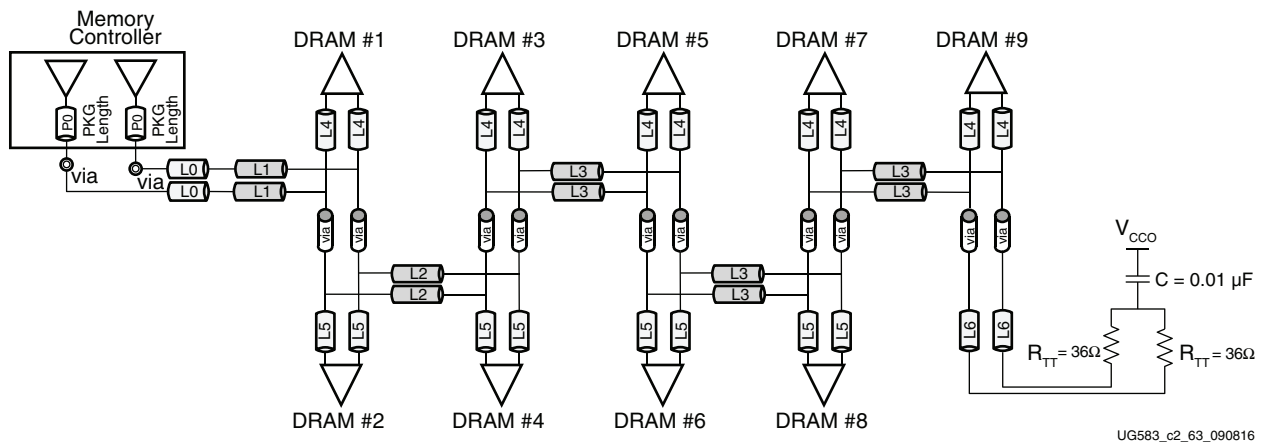


Figure 2-25: Clock Topology for DDR4 SDRAM Clamshell

**Note:** The end-termination components can be located on either the top or bottom layer.



Table 2-14 shows the DDR4 SDRAM clamshell impedance, length, and spacing guidelines for clock signals. Note the extra length of the L2 segment.

Table 2-14: DDR4 SDRAM Clamshell Impedance, Length, Width, and Spacing Guidelines for Clock Signals

Parameter	L0 FPGA Breakout	L1	L2	L3	L4	L5	L6 V <sub>TT</sub> Stub	Units
Layer (recommended)	Upper inner	Upper inner	Lower inner	Upper inner	Top	Bottom	Bottom	
Differential impedance Z <sub>0</sub>	86	76	90	90	86	86	76	Ω
Length	0.0~1.5 <sup>(1)</sup>	0.0~4.0	L3+0.2	0.45~0.85	For mirrored case, L4 = L5; for non-mirrored case, make as short as possible		≤1.0	inch
Width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/5.0/4.0	4.0/5.0/4.0	4.0/4.0/4.0	4.0/4.0/4.0	6.0/6.0/6.0	mil
Spacing to addr/cmd/ctrl	8.0	20 <sup>(1)</sup>	20	20	20	20	20	mil
Spacing to other groups	8.0	30	30	30	30	30	30	mil

**Notes:**

1. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 30.



**IMPORTANT:** Add as many ground vias as possible to help avoid crosstalk issues. See item 20 in [General Memory Routing Guidelines](#).

### DDR4 SDRAM Data Signals Point-to-Point for Fly-by and Clamshell Configurations

Figure 2-26 shows the data signals (DQ, DM, and DQS) point-to-point for DDR4 SDRAM. The guidelines are the same for fly-by and clamshell configurations.

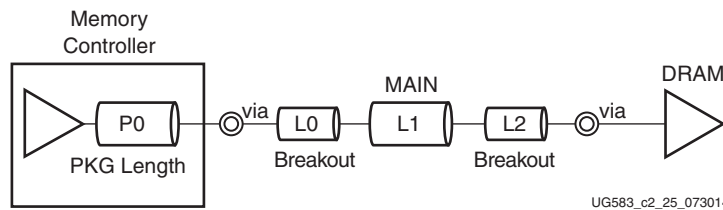


Figure 2-26: Data Signals Point-to-Point for DDR4 SDRAM

Table 2-15 shows the DDR4 SDRAM impedance, length, and spacing guidelines for data signals.

Table 2-15: DDR4 SDRAM Impedance, Length, and Spacing Guidelines for Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
dq single-ended impedance $Z_0$	50±10%	39±10%	50±10%	$\Omega$
dqs differential impedance $Z_{DIFF}$	86±10%	76±10%	86±10%	$\Omega$
Trace width (nominal)	4.0	6.0	4.0	mil
Differential trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	0.0~0.1	inches
Spacing in byte (minimum)	4.0	8.0 <sup>(2)</sup>	4.0	mil
Spacing byte to byte (minimum)	4.0	20	4.0	mil
dq to dqs spacing (minimum)	4.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 12.0.

## DDR4 SDRAM Routing Constraints

There are two constraints requirements for each signal group in the DDR4 memory interface:

- Total length/delay constraints
- Skew constraints

The total length/delay constraints are shown in [Table 2-16](#).

Table 2-16: DDR4 SDRAM Total Length/Delay Constraints

Signal Group	Reference Figure	Maximum Length/Delay Constraints
address/command/control: x1/x2/x3/x4/x5/x6/x7/x8/x9 (includes package routing P0)	<a href="#">Figure 2-20</a>	7.15/7.9/8.65/9.4/10.15/10.9/11.65/12.4/13.15 inches 1211/1339/1466/1593/1720/1847/1974/2101/2228 ps
Data Signals P0+L0+L1+L2	<a href="#">Figure 2-26</a>	6.0 inches, 1017 ps

The skew constraints are listed in [Table 2-17](#) and [Table 2-18](#).

**Table 2-17: DDR4 SDRAM Data Group Skew Constraints**

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Data <sup>(1)</sup> to DQS	±10	±58
dqs_p and dqs_n	2	12
CK to DQS	-149 to 1,796	-879 mil to 10.59 inches

**Notes:**

1. The data group includes dq and dm\_n/dbi\_n.
2. Delays are based on 169.5 ps/in. See item 8 in [General Memory Routing Guidelines](#).
3. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.

The data group skew constraints are listed in [Table 2-18](#).

**Table 2-18: DDR4 SDRAM Address, Command, and Control Skew Constraints**

Signal	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
address/command/control <sup>(1)</sup> to CK	UltraScale device to each memory device	±8 ps	±47
ck_p and ck_n	UltraScale device to each memory device	2	12

**Notes:**

1. The signal reset\_n is not required to meet the skew constraints in this table.
2. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** *FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.*

# PCB Guidelines for DDR3/3L SDRAM (PL and PS)

## Overview

This section provides electrical design guidelines for typical DDR3 SDRAM interfaces (up to nine devices) from system-level signal integrity simulations for the UltraScale architecture. A five-component DDR3 80-bit wide interface using x16 devices is used as a design example in this document.



**IMPORTANT:** All routing guidelines in this section must be followed to achieve the maximum data rates specified for the DDR3/3L SDRAM interface for typical system designs. Customers could have unique or specific designs with particular violations of some rules. In these scenarios, design or routing trade-offs have to be taken in other routing parameters to mitigate the risk. System-level channel signal integrity simulations are required to evaluate such trade-offs. It is important to read the [General Memory Routing Guidelines](#) section before continuing with this section.

## DDR3 SDRAM Interface Signal Description

The DDR3 SDRAM interface consists of clock, control, address, command, and data signals as shown in [Table 2-19](#).

Table 2-19: DDR3 SDRAM Interface Signal Description

Signal Name	Description
<b>Clock Signals</b>	
ck_p/n[1:0]	Differential clock
<b>Control Signals</b>	
cke[1:0]	Clock enable.
cs_n[1:0]	Chip select
odt[1:0]	On-die termination enable
reset_n	See <a href="#">reset_n</a>
<b>Address Signals</b>	
a[15:0]	Memory address bus
ba[2:0]	Bank address
<b>Command Signals</b>	
ras_n	Row address select
cas_n	Column address select
we_n	Write enable

Table 2-19: DDR3 SDRAM Interface Signal Description (Cont'd)

Signal Name	Description
<b>Data Signals</b>	
dq[79:0]	Data bus
dqs_p/n[9:0]	Differential data strobe
dm[9:0]	Data mask

**Notes:**

- Actual signal list might vary based on configuration.

## Topology and Routing Guidelines for DDR3 SDRAM

### DDR3 SDRAM Address, Command, and Control Fly-by Termination

With high-speed signaling in DDR3 SDRAM, fly-by topology is used for address, command, and control signals to achieve the best signal integrity. Each address, command, and control signal by itself is routed continuously in the same layer from the respective UltraScale device pin to far end termination, except in breakout areas. In other words, each individual address, command, or control signal routing is not broken into routings on multiple layers. Figure 2-27 shows the address fly-by termination for DDR3 SDRAM.

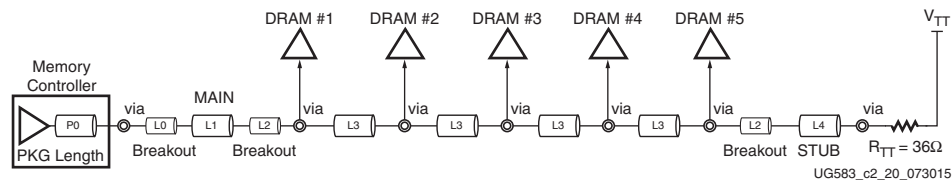


Figure 2-27: Address Fly-by Termination for DDR3 SDRAM

### reset\_n

Figure 2-28 shows the termination for reset\_n. The 4.7 kΩ resistor to ground is to keep each line Low during FPGA power-up and reconfiguration. If self-refresh is required during FPGA power-down or reconfiguration, circuitry needs to be added to ensure that reset\_n stays High during that time.

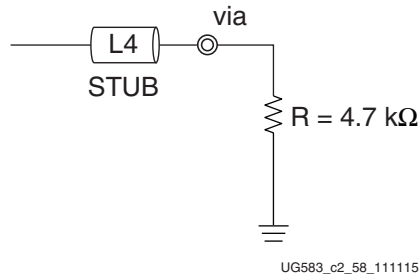


Figure 2-28: Termination for reset\_n in DDR3 DRAM

Table 2-20 shows the DDR3 SDRAM impedance, length, and spacing guidelines for address, command, and control signals.

Table 2-20: DDR3 SDRAM Impedance, Length, and Spacing Guidelines for Address, Command, and Control Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance Z <sub>0</sub>	50±10%	36±10%	50±10%	50±10%	39±10%	Ω
Trace width	4.0	7.0	4.0	4.0	6.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	0.0~0.1	0.35~0.75	0~1.0	inches
Spacing in address, command, and control signals (minimum)	4.0	8.0 <sup>(2)</sup>	4.0	8.0	8.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count	7					–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 12.0.

### DDR3 SDRAM Clock Fly-by Termination

Inherent to fly-by topology, the timing skew between the clock and dqs signals is deskewed by the write leveling feature on DDR3 SDRAM (Figure 2-29).

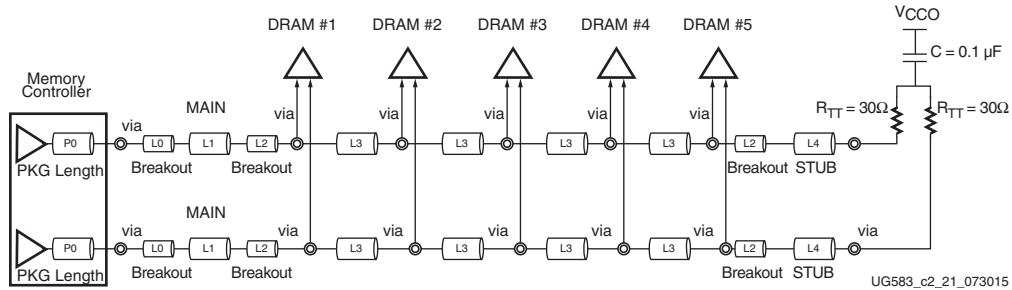


Figure 2-29: Clock Fly-by Termination for DDR3 SDRAM

Table 2-21 shows the DDR3 SDRAM impedance, length, and spacing guidelines for clock signals.

Table 2-21: DDR3 SDRAM Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance Z <sub>DIFF</sub>	86±10%	76±10%	86±10%	90±10%	76±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	4.0/5.0/4.0	6.0/6.0/6.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	0.0~0.1	0.35~0.75	0~1.0	inches
Spacing in address, command, and control signals (minimum)	8.0	20 <sup>(2)</sup>	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8	30	30	mil
Maximum PCB via count per signal	7					–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 30.

### DDR3 SDRAM Data Signals Point-to-Point

Figure 2-30 shows the data signals (DQ, DM, and DQS) point-to-point for DDR3 SDRAM.

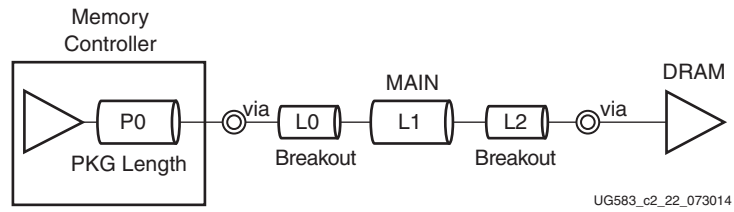


Figure 2-30: Data Signals Point-to-Point for DDR3 SDRAM

Table 2-22 shows the DDR3 SDRAM impedance, length, and spacing guidelines for data signals.

Table 2-22: DDR3 SDRAM Impedance, Length, and Spacing Guidelines for Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
dq single-ended impedance $Z_0$	50±10%	39±10%	50±10%	Ω
dqs differential impedance $Z_{DIFF}$	86±10%	76±10%	86±10%	Ω
Trace width (nominal)	4.0	6.0	4.0	mil
Differential trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	mil
Trace length (nominal)	0.0~1.5 <sup>(1)(2)</sup>	0.0~5.0	0.0~0.1	inches
Spacing in byte (minimum)	4.0	8.0 <sup>(2)</sup>	4.0	mil
Spacing byte to byte (minimum)	4.0	20	4.0	mil
dq to dqs spacing (minimum)	4.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 12.0.

### DDR3 SDRAM Routing Constraints

There are two constraints requirements for each signal group in the DDR3 memory interface:

- Total length/delay constraints
- Skew constraints



The total length/delay constraints are shown in [Table 2-23](#).

**Table 2-23: DDR3 SDRAM Total Length/Delay Constraints**

Signal Group	Reference Figure	Maximum Length/Delay Constraints
Address/command/control: x1/x2/x3/x4/x5/x6/x7/x8/x9 (includes package routing P0)	<a href="#">Figure 2-27</a>	6.15/6.9/7.65/8.4/9.15/9.9/10.65/11.4/12.15 inches 1042/1169/1296/1423/1550/1678/1805/1932/2110 ps
Data Signals P0+L0+L1+L2	<a href="#">Figure 2-30</a>	7.0 inches, 1186 ps

The skew constraints are shown in [Table 2-24](#) and [Table 2-25](#).

**Table 2-24: DDR3 SDRAM Data Group Skew Constraints**

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Data <sup>(1)</sup> to DQS	±10	±58
dqs_p and dqs_n	2	12
CK to DQS	-149 to 1,796	-879 mil to 10.6 inches

**Notes:**

1. The data group includes dq and dm.
2. Delays are based on 169.5 ps/in. See item 8 in [General Memory Routing Guidelines](#).
3. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.

The address, command, and control clock group skew constraints are listed in [Table 2-25](#).

**Table 2-25: DDR3 SDRAM Address, Command, and Control Skew Constraints**

Signal	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control <sup>(1)</sup> to CK	UltraScale device to each memory device	±8 ps	±47
ck_p and ck_n	UltraScale device to each memory device	2	12

**Notes:**

1. The signal reset\_n is not required to meet the skew constraints in this table.
2. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** *FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.*

# DDR3/DDR4 UDIMM/RDIMM/SODIMM Routing Guidelines (PL and PS)

For the case of two dual in-line memory modules (DIMMs), each DIMM has its own set of point-to-point clock and control signals, while data, command, and address route to both DIMMs in a fly-by topology.



**IMPORTANT:** The Processing System (PS) does not support multiple DIMMs.

## DDR3/DDR4 DIMM Clock Point-to-Point Routing

Figure 2-31 and Table 2-26 define the routing topology and guidelines for DDR3/DDR4 DIMM clock signals.

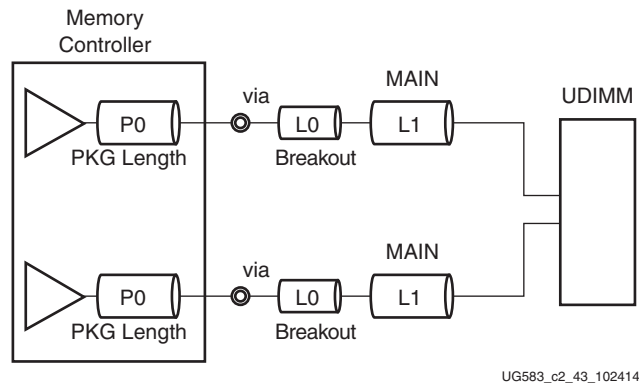


Figure 2-31: CK Point-to-Point Routing for DDR3/DDR4 DIMM Topology

Table 2-26: Impedance, Length, and Spacing Guidelines for DIMM Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	Units
Trace type	Stripline	Stripline	–
Differential impedance $Z_0$	86±10%	66±10%	Ω
Trace width/space/width	4.0/4.0/4.0	8.0/8.0/8.0	mil
Trace length	0.0~1.5 <sup>(1)</sup> (2)	0.0~4.0	inches
Spacing to address, command, and control signals (minimum)	8.0	20 <sup>(2)</sup>	mil
Spacing to other group signals (minimum)	8.0	30	mil

Table 2-26: Impedance, Length, and Spacing Guidelines for DIMM Clock Signals (Cont'd)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	Units
Maximum PCB via count	2		–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 30.

## DDR3/DDR4 DIMM Control, Command, and Address Routing

Figure 2-32, Figure 2-33, Table 2-27, and Table 2-29 define the routing topologies and routing guidelines for DDR3/DDR4 DIMM address, command, and control signals. Place four 0402 1.0  $\mu$ F capacitors in the CMD/CTRL pin area of the DIMM connector. All capacitors should connect from  $V_{CC0}$  to GND.

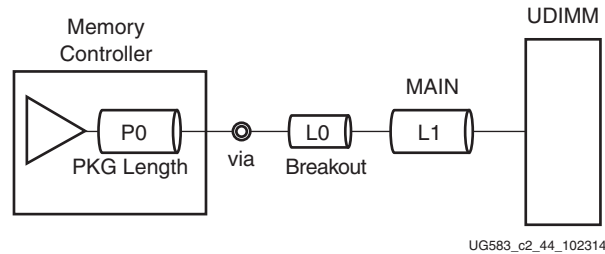


Figure 2-32: CTRL Point-to-Point Routing for DDR3/DDR4 DIMM Topology

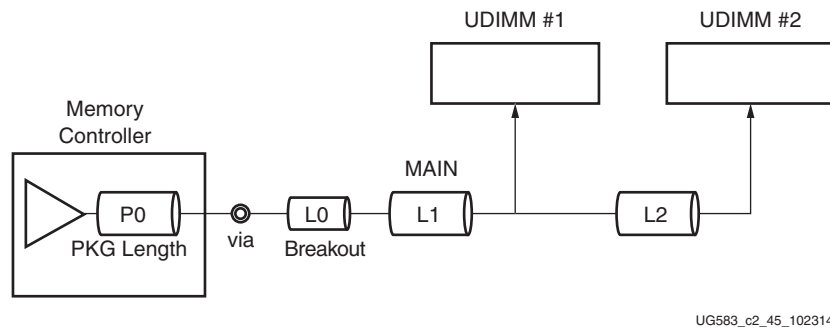


Figure 2-33: CMD/ADDR Fly-by Routing for DDR3/DDR4 DIMM Topology

**Note:** The Zynq UltraScale+ MPSoC DDR interface does not support dual-DIMM topologies.

Table 2-27: Impedance, Length, and Spacing Guidelines for One-Slot DIMM Address, Command, and Control Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	Units
Trace type	Stripline	Stripline	–
Single-ended impedance $Z_0$	$50 \pm 10\%$	$39 \pm 10\%$	$\Omega$
Trace width	4.0	6.0	mil

**Table 2-27: Impedance, Length, and Spacing Guidelines for One-Slot DIMM Address, Command, and Control Signals (Cont'd)**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	Units
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	inches
Spacing in address, command, and control signals (minimum)	4.0	12 <sup>(2)</sup>	mil
Spacing to clock signals (minimum)	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	mil
Maximum PCB via count	2		–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).
- If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 16.0.

**Table 2-28: Impedance, Length, and Spacing Guidelines for Two-Slot DIMM Address, Command, and Control Signals**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DIMM to DIMM)	Units
Trace type	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	34±10% addr/cmd 39±10% ctrl	34±10% addr/cmd 39±10% ctrl	$\Omega$
Trace width	4.0	6.9 addr/cmd 6.0 ctrl	6.9 addr/cmd 6.0 ctrl	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	<0.5	inches
Spacing in address, command, and control signals (minimum)	4.0	12.0 ctrl, 13.8 addr/cmd <sup>(2)</sup>	12.0 ctrl, 13.8 addr/cmd	mil
Spacing to clock signals (minimum)	8.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	Addr/cmd: 3 Ctrl: 2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).
- If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 18.0 ctrl, and 20.7 addr/cmd.

## DDR3/DDR4 DIMM Data (DQ and DQS) Routing

Figure 2-34, Figure 2-35, and Table 2-29 define routing topologies and guidelines for DDR3/DDR4 DIMM data routing.

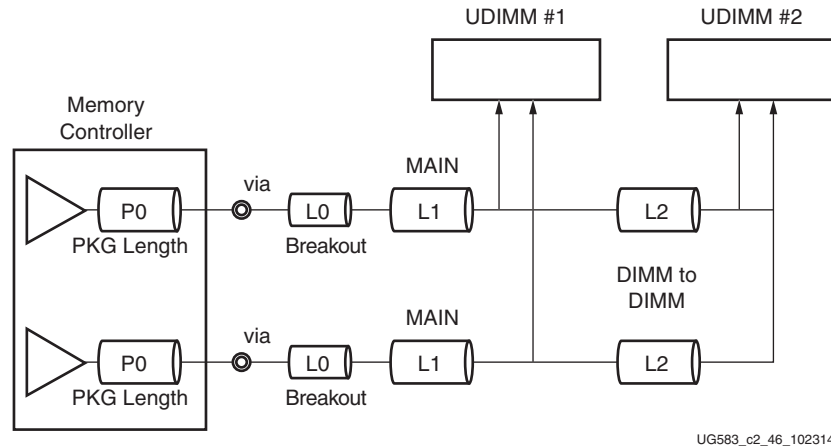


Figure 2-34: DQS Fly-by Routing for DDR3/DDR4 DIMM Topology

**Note:** The Zynq UltraScale+ MPSoC DDR interface does not support dual-DIMM topologies.

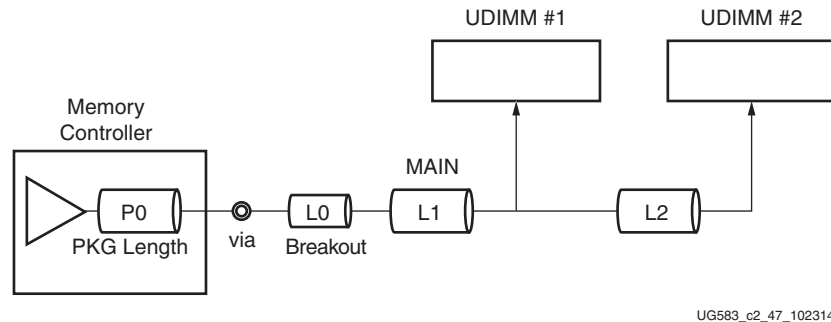


Figure 2-35: DQ Fly-by Routing for DDR3/DDR4 DIMM Topology

**Note:** The Zynq UltraScale+ MPSoC DDR interface does not support dual-DIMM topologies.

Table 2-29: Impedance, Length, and Spacing Guidelines for DIMM Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DIMM to DIMM)	Units
Trace type	Stripline	Stripline	Stripline	–
dq single-ended impedance $Z_0$	$50 \pm 10\%$	$39 \pm 10\%$	$39 \pm 10\%$	$\Omega$
dqs differential impedance $Z_{DIFF}$	86	66	66	$\Omega$
dq trace width (nominal)	4.0	6.0	6.0	mil
Dqs differential trace width/space/width	4.0/4.0/4.0	8.0/8.0/8.0	8.0/8.0/8.0	mil
Trace length	0.0~1.5 <sup>(1)(2)</sup>	0.0~4.0	$\leq 0.5$	inches

**Table 2-29: Impedance, Length, and Spacing Guidelines for DIMM Data Signals (Cont'd)**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DIMM to DIMM)	Units
Spacing in byte (minimum)	4.0	15 <sup>(2)</sup>	15	mil
Spacing byte to byte (minimum)	4.0	30	30	mil
dq to dq <sub>s</sub> spacing (minimum)	4.0	21	21	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2 for 1 DIMM, 3 for two DIMMs			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).
- If greater than 1.5 inches is needed in the breakout area, change L1 spacing to 20.0.

## DDR3 UDIMM/RDIMM/SODIMM Routing Constraints

There are two constraints requirements for each signal group in the DDR3 DIMM interface:

- Total length/delay constraints
- Skew constraints

The total length/delay constraints are shown in [Table 2-30](#).

**Table 2-30: DDR3 DIMM Total Length/Delay Constraints**

Signal Group	Reference Figure	Maximum Length/Delay Constraints
Address/command/control: x1/x2 (includes package routing P0)	<a href="#">Figure 2-33</a>	6.15/6.65 inches, 1042/1127 ps
Data Signals P0+L0+L1+L2	<a href="#">Figure 2-35</a>	7.0 inches, 1186 ps

The skew constraints are shown in [Table 2-31](#) and [Table 2-32](#).

**Table 2-31: DDR3 DIMM Data Group Skew Constraints**

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Data <sup>(1)</sup> to DQS	±10	±58
dqs <sub>p</sub> and dqs <sub>n</sub>	2	12
CK to DQS <sup>(1)</sup>	±150	±885

**Notes:**

- Clock to DQS constraints should be from the UltraScale device to the DIMM.
- The data group includes dq and dm.
- Delays are based on 169.5 ps/in. See item 8 in [General Memory Routing Guidelines](#).
- For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.

Table 2-32: DDR3 DIMM Address, Command, and Control Skew Constraints

Signal	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control <sup>(1)</sup> to CK	UltraScale device to each memory device	±8 ps	±47
ck_p and ck_n	UltraScale device to each DIMM	2	12

**Notes:**

- The signal reset\_n is not required to meet the skew constraints in this table.
- For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.

## DDR4 UDIMM/RDIMM/SODIMM Routing Constraints

There are two constraints requirements for each signal group in the DDR4 DIMM interface:

- Total length/delay constraints
- Skew constraints

The total length/delay constraints are shown in [Table 2-33](#).

Table 2-33: DDR4 DIMM Total Length/Delay Constraints

Signal Group	Reference Figure	Maximum Length/Delay Constraints
Address/command/control: x1/x2 (includes package routing P0)	<a href="#">Figure 2-33</a>	7.15/7.65 inches 1212/1296 ps
Data Signals P0+L0+L1+L2	<a href="#">Figure 2-35</a>	6.0 inches 1017 ps

The skew constraints are listed in [Table 2-34](#) and [Table 2-35](#).

Table 2-34: DDR4 DIMM Skew Constraints

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Data <sup>(1)</sup> to DQS	±10	±58
dqs_p and dqs_n	2	12

Table 2-34: DDR4 DIMM Skew Constraints (Cont'd)

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
CK to DQS <sup>(2)</sup>	±150	±885

**Notes:**

1. The data group includes dq and dm\_n/dbi\_n.
2. Clock to DQS constraints should be from the UltraScale device to the DIMM.
3. Delays are based on 169.5 ps/in. See item 8 in [General Memory Routing Guidelines](#).
4. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.

Table 2-35: DDR4 DIMM Address, Command, and Control Skew Constraints

Signal	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control <sup>(1)</sup> to CK	UltraScale device to each memory device	±8 ps	±47
ck_p and ck_n	UltraScale device to each DIMM	2	12

**Notes:**

1. The signal reset\_n is not required to meet the skew constraints in this table.
2. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.

## PCB Guidelines for LPDDR4 Memories without ECC (PS)

### Overview

This section provides electrical design guidelines for 32-bit LPDDR4 single-die package (SDP) and dual-die package (DDP) interfaces without error correcting code (ECC).



**IMPORTANT:** All guidelines in this section must be followed to achieve the maximum data rates specified for the LPDDR4 memory interface.



## LPDDR4 x32 without ECC Memory Interface Signals and Connections

### Signals and Connections for x32 Dual Channel LPDDR4 SDP without ECC

The required signals for a x32 dual channel LPDDR4 SDP without ECC interface are shown in [Table 2-36](#). The table shows which FPGA memory interface signals connect to which pins on the LPDDR4 device, along with any required termination.

Table 2-36: Signal Connection Matrix for x32 Dual Channel LPDDR4 SDP without ECC

FPGA Pins	LPDDR4 Pins	PCB Termination at Far End
<b>Clock Signals</b>		
CK0_P CK0_N	CK_t_A CK_c_A	None
CK1_P CK1_N	CK_t_B CK_c_B	None
<b>Address and Command Signals</b>		
A[5:0]	CA[5:0]_A	None
A[15:10]	CA[5:0]_B	None
<b>Control Signals (CKE, CS, and ODT<sup>(1)</sup>)</b>		
CKE0	CKE_A CKE_B	80Ω to V <sub>TT</sub> or 160Ω to V <sub>TT</sub> /160Ω to GND
CS0	CS_A CS_B	None
	ODT_CA_A	Direct connect to V <sub>DD2</sub>
	ODT_CA_B	Direct connect to V <sub>DD2</sub>
<b>Data Signals</b>		
DQ[15:0]	DQ[15:0]_A	None
DQ[31:16]	DQ[15:0]_B	None
DM0	DMI0_A	None
DM1	DMI1_A	None
DM2	DMI0_B	None
DM3	DMI1_B	None
DQS0_P/ DQS0_N	DQS0_t_A/ DQS0_c_A	None
DQS1_P/ DQS1_N	DQS1_t_A/ DQS1_c_A	None
DQS2_P/ DQS2_N	DQS0_t_B/ DQS0_c_B	None

Table 2-36: Signal Connection Matrix for x32 Dual Channel LPDDR4 SDP without ECC (Cont'd)

FPGA Pins	LPDDR4 Pins	PCB Termination at Far End
DQS3_P/ DQS3_N	DQS1_t_B/ DQS1_c_B	None
<b>Reset</b>		
RESET_N	RESET_n	4.7 kΩ to GND
-	ZQ0	240Ω to V <sub>DD2</sub>
ZQ	-	240Ω to GND

**Notes:**

1. The FPGA ODT pins should be left unconnected. The ODT pin(s) of the memory device should be directly connected as specified in this table.
2. The ZQ pin on the LPDDR4 devices should be tied to V<sub>DD2</sub> through a 240Ω resistor.
3. Actual signal list might vary based on configuration.

### Signals and Connections for x32 Dual Channel LPDDR4 DDP without ECC

The required signals for a x32 dual channel LPDDR4 DDP non-ECC interface are shown in [Table 2-37](#). The table shows which FPGA memory interface signals connect to which pins on the LPDDR4 device, along with any required termination.

Table 2-37: Signal Connection Matrix for x32 Dual Channel LPDDR4 DDP without ECC

FPGA Pins	LPDDR4 Pins	PCB Termination at Far End
<b>Clock Signals</b>		
CK0_P CK0_N	CK_t_A CK_c_A	None
CK1_P CK1_N	CK_t_B CK_c_B	None
<b>Address and Command Signals</b>		
A[5:0]	CA[5:0]_A	None
A[15:10]	CA[5:0]_B	None
<b>Control Signals (CKE, CS, and ODT<sup>(1)</sup>)</b>		
CKE0	CKE0_A CKE0_B	80Ω to V <sub>TT</sub> or 160Ω to V <sub>TT</sub> /160Ω to GND
CKE1	CKE1_A CKE1_B	80Ω to V <sub>TT</sub> or 160Ω to V <sub>TT</sub> /160Ω to GND
CS0	CS0_A CS0_B	None
CS1	CS1_A CS1_B	None

Table 2-37: Signal Connection Matrix for x32 Dual Channel LPDDR4 DDP without ECC (Cont'd)

FPGA Pins	LPDDR4 Pins	PCB Termination at Far End
	ODT_CA_A	Direct connect to V <sub>DD2</sub>
	ODT_CA_B	Direct connect to V <sub>DD2</sub>
<b>Data Signals</b>		
DQ[15:0]	DQ[15:0]_A	None
DQ[31:16]	DQ[15:0]_B	None
DM0	DMI0_A	None
DM1	DMI1_A	None
DM2	DMI0_B	None
DM3	DMI1_B	None
DQS0_P DQS0_N	DQS0_t_A DQS0_c_A	None
DQS1_P DQS1_N	DQS1_t_A DQS1_c_A	None
DQS2_P DQS2_N	DQS0_t_B DQS0_c_B	None
DQS3_P DQS3_N	DQS1_t_B DQS1_c_B	None
<b>Reset</b>		
RESET_N	RESET_n	4.7 kΩ to GND

**Notes:**

1. The FPGA ODT pins should be left unconnected. The ODT pin(s) of the memory device should be directly connected as specified in this table.
2. Each ZQ pin on the LPDDR4 devices should be individually tied to V<sub>DD2</sub> through a 240Ω resistor.
3. The ZQ pin on the FPGA should be tied to GND through a 240Ω resistor.

## LPDDR4 Address Copy

The PS DDR interface includes a feature called address copy to reduce loading on the address/command bus when operating in LPDDR4 mode. A[5:0] are replicated on A[15:10] to allow fewer loads on the CA bus (Table 2-38).

Table 2-38: LPDDR4 Address Copy

MPSoC Pin	DRAM Pin	MPSoC Pin Copy	DRAM Pin
PS_DDR_A0	CA0_A	PS_DDR_A10	CA0_B
PS_DDR_A1	CA1_A	PS_DDR_A11	CA1_B
PS_DDR_A2	CA2_A	PS_DDR_A12	CA2_B
PS_DDR_A3	CA3_A	PS_DDR_A13	CA3_B

Table 2-38: LPDDR4 Address Copy (Cont'd)

MPSoC Pin	DRAM Pin	MPSoC Pin Copy	DRAM Pin
PS_DDR_A4	CA4_A	PS_DDR_A14	CA4_B
PS_DDR_A5	CA5_A	PS_DDR_A15	CA5_B

## LPDDR4 Interface without ECC Topology and Routing Guidelines



**IMPORTANT:** To achieve the specified performance, all rules in the tables must be followed. This includes but is not limited to the trace type, width, spacing, impedance, and via count.

### LPDDR4 Interface without ECC Address/Command and Control Routing

Figure 2-36 shows the address/command routing for the LPDDR4 interface without ECC.

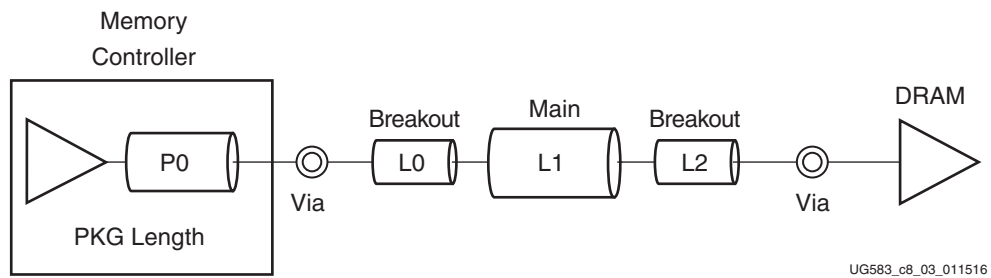


Figure 2-36: Address/Command Routing for LPDDR4 without ECC

Table 2-39 shows the impedance, length, and spacing guidelines for address, command, and control signals in the LPDDR4 interface without ECC.

Table 2-39: Impedance, Length, and Spacing Guidelines for Address and Command Signals in LPDDR4 Interface without ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	$50 \pm 10\%$	$39 \pm 10\%$	$52 \pm 10\%$	$\Omega$
Trace width	4.0	6.0	3.5	mil
Trace length	0.0~0.55/0.95 <sup>(1)</sup>	$\leq 4.0$	$\leq 0.3$	inches
Spacing in addr/cmd (minimum)	4.0	8.0	4.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	8.0	mil
Maximum PCB via count	2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 Interface without ECC Chip Select Routing**

Figure 2-37 shows the chip select (CS) routing for the LPDDR4 interface without ECC.

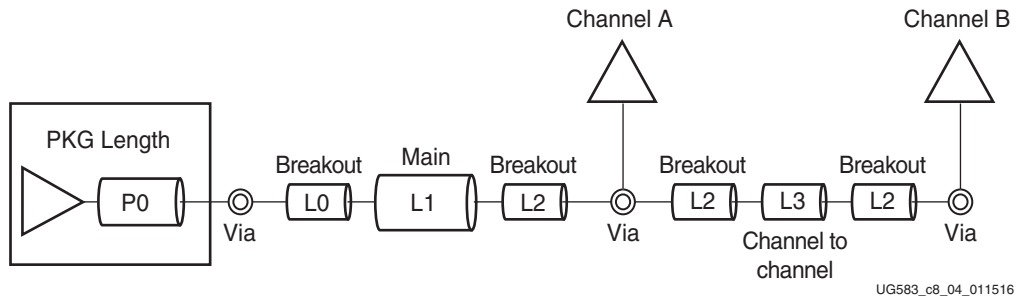


Figure 2-37: Chip Select Routing for LPDDR4 without ECC

Table 2-40 shows the impedance, length, and spacing guidelines for chip select in the LPDDR4 interface without ECC.

Table 2-40: Impedance, Length, and Spacing Guidelines for Chip Select in LPDDR4 Interface without ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	Units
Trace type	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	50±10%	$\Omega$
Trace width	4.0	6.0	3.5	4.0	mil
Trace length	0.0~0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤0.3	inches
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count	3				–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 without ECC Clock Enable Routing**

Figure 2-38 shows the clock enable (CKE) routing options for an LPDDR4 interface without ECC.

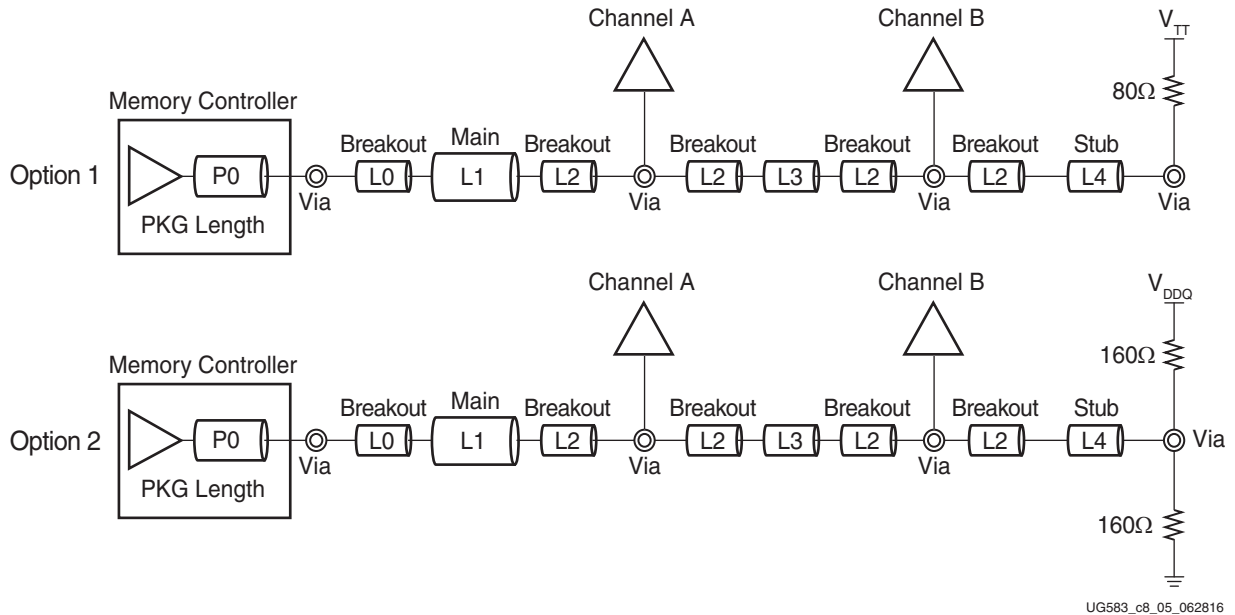


Figure 2-38: Clock Enable Routing for LPDDR4 without ECC

Table 2-41 shows the impedance, length, and spacing guidelines for clock enable in the LPDDR4 interface without ECC.

Table 2-41: Impedance, Length, and Spacing Guidelines for Clock Enable in LPDDR4 Interface without ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	50±10%	50±10%	Ω
Trace width	4.0	6.0	3.5	4.0	4.0	mil
Trace length	0.0~0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤0.3	≤0.5	inches
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	mil
Maximum PCB via count	3					–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 without ECC CK and DQS Differential Point-to-Point Routing**

Figure 2-39 shows the CK and DQS differential point-to-point routing for the LPDDR4 interface without ECC.

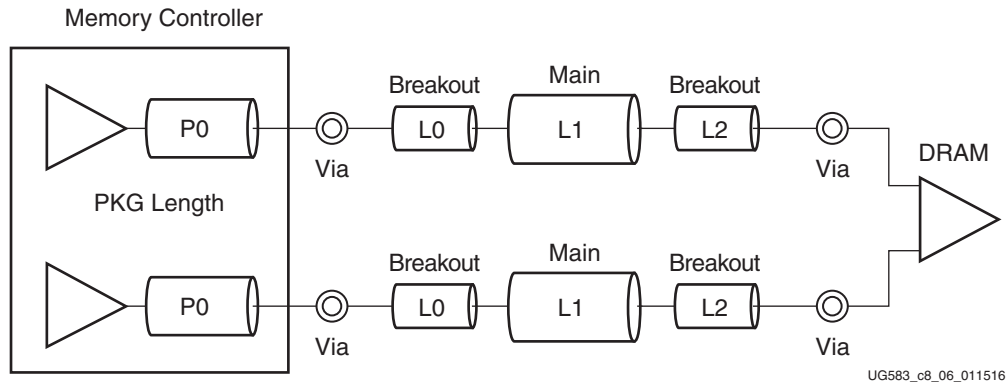


Figure 2-39: CK and DQS Routing for LPDDR4 without ECC

Table 2-42 shows the impedance, length, and spacing guidelines for CK and DQS in the LPDDR4 interface without ECC.

Table 2-42: Impedance, Length, and Spacing Guidelines for CK and DQS in LPDDR4 Interface without ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Clock differential impedance $Z_{diff}$	86±10%	76±10%	88±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.5/3.5	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	inches
Spacing in addr/cmd/ctrl (minimum)	8.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	8.0	mil
Maximum PCB via count per signal	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### LPDDR4 without ECC Data (DQ and DM) Point-to-Point Routing

Figure 2-40 shows the DQ and DM point-to-point routing for the LPDDR4 interface without ECC.

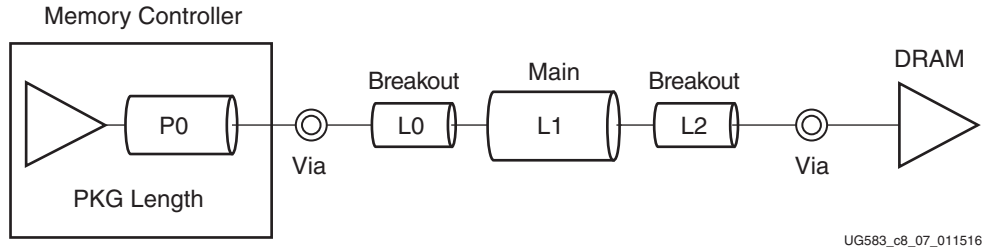


Figure 2-40: DQ and DM Routing for LPDDR4 without ECC

Table 2-43 shows the impedance, length, and spacing guidelines for DQ and DM in the LPDDR4 interface without ECC.

Table 2-43: Impedance, Length, and Spacing Guidelines for DQ and DM in LPDDR4 Interface without ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	$50 \pm 10\%$	$39 \pm 10\%$	$52 \pm 10\%$	$\Omega$
Trace width	4.0	6.0	3.5	mil
Trace length	$\leq 0.55/0.95^{(1)}$	$\leq 4.0$	$\leq 0.3$	inches
Spacing in byte (including DQS) (minimum)	4.0	8.0	4.0	mil
Spacing byte to byte (minimum)	8.0	30	8.0	mil
Spacing to other group signals (minimum)	8.0	20	8.0	mil
Maximum PCB via count	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### LPDDR4 without ECC Length and Skew Constraints

The constraints requirement for each signal group in the LPDDR4 without ECC memory interface consists of two parts:

- Maximum length constraints
- Skew constraints



The maximum length constraints are shown in [Table 2-44](#).

**Table 2-44: LPDDR4 without ECC Maximum Length Constraints**

Signal Group	Reference Figure	Maximum Length Constraints (inches)
Address/command P0+L0+L1+L2	<a href="#">Figure 2-36</a>	6.2
CS P0+L0+L1+L2+L3	<a href="#">Figure 2-37</a>	6.8
CKE P0+L0+L1+L2+L3	<a href="#">Figure 2-38</a>	7.9
Data signals P0+L0+L1+L2	<a href="#">Figure 2-40</a>	6.2

The skew constraints are listed in [Table 2-45](#).

**Table 2-45: LPDDR4 without ECC Skew Constraints**

Signal Group	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control to CK (A) Address/command/control to CK (B)	MPSoC to memory device	±8	±47
Data (DQ/DM) to DQS (A) Data (DQ/DM) to DQS (B)	MPSoC to memory device	±5	±29
DQ/DM (slowest to fastest) (A) DQ/DM (slowest to fastest) (B)	MPSoC to memory device	5	29
CK_T and CK_C (A) CK_T and CK_C (B)	MPSoC to memory device	2	11
DQS_T and DQS_C (A) DQS_T and DQS_C (B)	MPSoC to memory device	2	11

**Notes:**

- For skew specifications, refer to items 3–8 in [General Memory Routing Guidelines](#).



**IMPORTANT:** *FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.*

## PCB Guidelines for LPDDR4 Memories with ECC (PS)

### Overview

This section provides electrical design guidelines for 32-bit LPDDR4 interfaces with ECC.



**IMPORTANT:** All guidelines in this section must be followed to achieve the maximum data rates specified for the LPDDR4 interface.

## LPDDR4 x32 with ECC Memory Interface Signals and Connections

The required signals for a x32 dual channel LPDDR4 SDP with ECC interface are shown in [Table 2-46](#). The table shows how to connect the FPGA memory interface signals to the correct pins on the LPDDR4 devices, along with any required termination.

Table 2-46: Signal Connection Matrix for x32 Dual Channel LPDDR4 SDP with ECC

FPGA Pins	LPDDR4 Pins MAIN DEVICE	LPDDR4 Pins ECC DEVICE	PCB Termination at Far End
<b>Clock Signals</b>			
CK0_P CK0_N	CK_t_A CK_c_A		40Ω to GND 40Ω to GND
CK1_P CK1_N	CK_t_B CK_c_B	CK_t_A CK_c_A	40Ω to GND 40Ω to GND
<b>Address and Command Signals</b>			
A[5:0]	CA[5:0]_A		40Ω to GND
A[15:10]	CA[5:0]_B	CA[5:0]_A	40Ω to GND
<b>Control Signals (CKE, CS, and ODT<sup>(1)</sup>)</b>			
CKE0	CKE_A CKE_B	CKE_A	80Ω to VTT or 160Ω to VTT/160Ω to GND
CS0	CS_A CS_B	CS_A CS_B	40Ω to GND
	ODT_CA_A	ODT_CA_A	Direct connect to GND
	ODT_CA_B		Direct connect to GND
<b>Data Signals</b>			
DQ[15:0]	DQ[15:0]_A		None
DQ[31:16]	DQ[15:0]_B		None
DQ[71:64]		DQ[7:0]_A	None
DM0	DMI0_A		None
DM1	DMI1_A		None
DM2	DMI0_B		None
DM3	DMI1_B		None
DM8		DMI0_A	None

Table 2-46: Signal Connection Matrix for x32 Dual Channel LPDDR4 SDP with ECC (Cont'd)

FPGA Pins	LPDDR4 Pins MAIN DEVICE	LPDDR4 Pins ECC DEVICE	PCB Termination at Far End
DQS0_P DQS0_N	DQS0_t_A DQS0_c_A		None
DQS1_P DQS1_N	DQS1_t_A DQS1_c_A		None
DQS2_P DQS2_N	DQS0_t_B DQS0_c_B		None
DQS3_P DQS3_N	DQS1_t_B DQS1_c_B		None
DQS8_P DQS8_N		DQS0_t_A DQS0_c_A	None
<b>Reset</b>			
RESET_N	RESET_n	RESET_n	4.7 kΩ to GND

**Notes:**

1. The FPGA ODT pins should be left unconnected. The ODT pin(s) of the memory device should be directly connected as specified in this table.
2. Unused inputs to the LPDDR4 ECC device can be grounded through a common 100Ω resistor.
3. Each ZQ pin on the LPDDR4 devices should be individually tied to  $V_{DD2}$  through a 240Ω resistor.
4. The ZQ pin on the FPGA should be tied to GND through a 240Ω resistor.
5. Actual signal list might vary based on configuration.

The required signals for a x32 dual channel LPDDR4 DDP with ECC interface are shown in [Table 2-47](#). The table shows how to connect the FPGA memory interface signals to the correct pins on the LPDDR4 devices, along with any required termination.

Table 2-47: Signal Connection Matrix for x32 Dual Channel LPDDR4 DDP with ECC

FPGA Pins	LPDDR4 Pins Main Device	LPDDR4 Pins ECC Device	PCB Termination at Far End
<b>Clock Signals</b>			
CK0_P CK0_N	CK_t_A <sup>(2)</sup> CK_c_A <sup>(2)</sup>		40Ω to GND 40Ω to GND
CK1_P CK1_N	CK_t_B CK_c_B	CK_t_A CK_c_A	40Ω to GND 40Ω to GND
<b>Address and Command Signals</b>			
A[5:0]	CA[5:0]_A <sup>(2)</sup>		40Ω to GND
A[15:10]	CA[5:0]_B	CA[5:0]_A	40Ω to GND
<b>Control Signals (CKE, CS, and ODT)<sup>(1)</sup></b>			
CKE0	CKE0_A CKE0_B	CKE0_A	80Ω to $V_{TT}$ or 160Ω to $V_{TT}$ /160Ω to GND

Table 2-47: Signal Connection Matrix for x32 Dual Channel LPDDR4 DDP with ECC (Cont'd)

FPGA Pins	LPDDR4 Pins Main Device	LPDDR4 Pins ECC Device	PCB Termination at Far End
CKE1	CKE1_A CKE1_B	CKE1_A	80Ω to V <sub>TT</sub> or 160Ω to V <sub>TT</sub> /160Ω to GND
CS0	CS0_A CS0_B	CS0_A CS0_B	40Ω to GND
CS1	CS1_A CS1_B	CS1_A CS1_B	40Ω to GND
	ODT_CA_A	ODT_CA_A	40Ω to GND
	ODT_CA_B <sup>(2)</sup>		40Ω to GND
<b>Data Signals</b>			
DQ[15:0]	DQ[15:0]_A		None
DQ[31:16]	DQ[15:0]_B		None
DQ[71:64]		DQ[7:0]_A	None
DM0	DMI0_A		None
DM1	DMI1_A		None
DM2	DMI0_B		None
DM3	DMI1_B		None
DM8		DMI0_A	None
DQS0_P DQS0_N	DQS0_t_A DQS0_c_A		None
DQS1_P DQS1_N	DQS1_t_A DQS1_c_A		None
DQS2_P DQS2_N	DQS0_t_B DQS0_c_B		None
DQS3_P DQS3_N	DQS1_t_B DQS1_c_B		None
DQS8_P DQS8_N		DQS0_t_A DQS0_c_A	None
<b>Reset</b>			
RESET_B	RESET_n	RESET_n	4.7kΩ to GND

**Notes:**

1. The FPGA ODT pins should be left unconnected. The ODT pin(s) of the memory device should be directly connected as specified in this table.
2. This pin ends in a termination.
3. Unused inputs to the LPDDR4 ECC device can be grounded through a common 100Ω resistor.
4. Each ZQ pin on the LPDDR4 devices should be individually tied to V<sub>DD2</sub> through a 240Ω resistor.
5. The ZQ pin on the FPGA should be tied to GND through a 240Ω resistor.
6. Actual signal list might vary based on configuration.

## LPDDR4 Address Copy

The PS DDR interface includes a feature called address copy to reduce loading on the address/command bus when operating in LPDDR4 mode. A[5:0] are replicated on A[15:10] to allow fewer loads on the CA bus (Table 2-48).

Table 2-48: LPDDR4 Address Copy

MPSoC Pin	DRAM Pin	MPSoC Pin Copy	DRAM Pin
PS_DDR_A0	CA0_A	PS_DDR_A10	CA0_B
PS_DDR_A1	CA1_A	PS_DDR_A11	CA1_B
PS_DDR_A2	CA2_A	PS_DDR_A12	CA2_B
PS_DDR_A3	CA3_A	PS_DDR_A13	CA3_B
PS_DDR_A4	CA4_A	PS_DDR_A14	CA4_B
PS_DDR_A5	CA5_A	PS_DDR_A15	CA5_B

## LPDDR4 with ECC Topology and Routing Guidelines



**IMPORTANT:** To achieve the specified performance, all rules in the tables must be followed. This includes but is not limited to the trace type, width, spacing, impedance, and via count.

### LPDDR4 with ECC Address/Command and Control Routing (CA\_A)

Figure 2-41 shows the CA\_A address/command routing for the LPDDR4 interface with ECC. The  $R_{TT}$  termination exists to terminate the fly-by route.

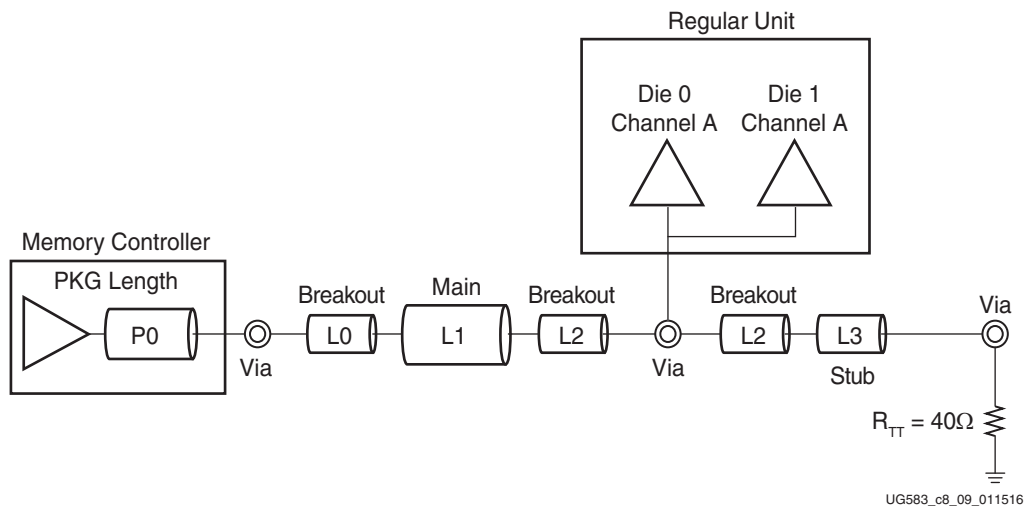


Figure 2-41: CA\_A Address/Command Routing for LPDDR4 with ECC

Table 2-49 shows the impedance, length, and spacing guidelines for CA\_A address/command signals in the LPDDR4 interface with ECC.

Table 2-49: Impedance, Length, and Spacing Guidelines for CA\_A Address/Command Signals in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	50±10%	$\Omega$
Trace width	4.0	6.0	3.5	4.0	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤0.5	inches
Spacing to addr/cmd (minimum)	4.0	6.0	4.0	6.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count	3				–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 with ECC Address/Command and Control Routing (CA\_B)**

Figure 2-42 shows the CA\_B address/command routing for the LPDDR4 interface with ECC. The  $R_{TT}$  termination exists to terminate the fly-by route.

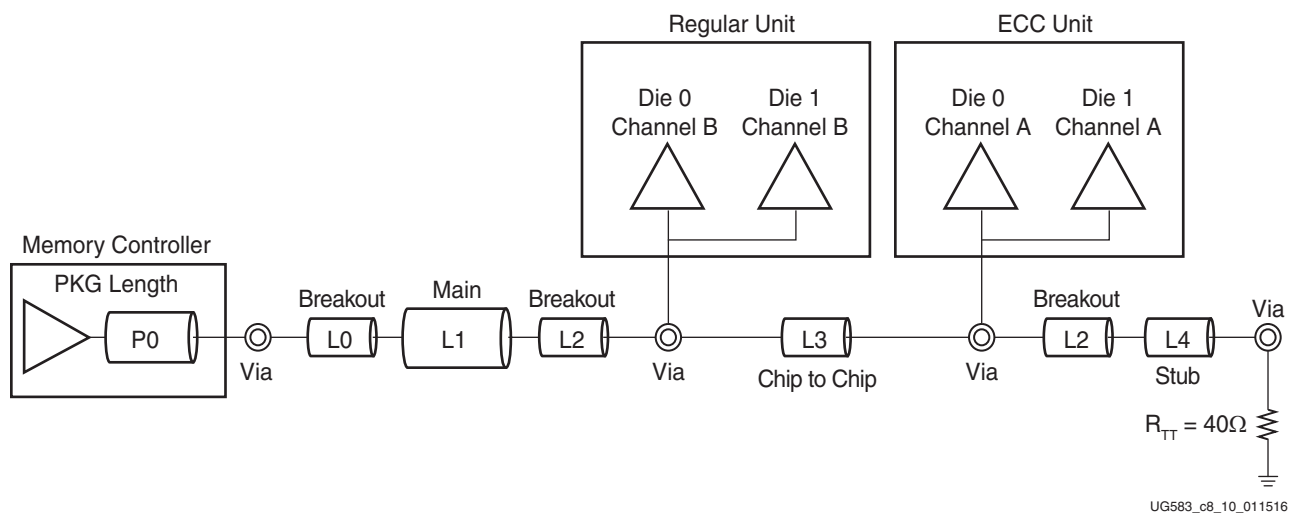


Figure 2-42: CA\_B Address/Command Routing for LPDDR4 with ECC

Table 2-50 shows the impedance, length, and spacing guidelines for CA\_B address/command signals in the LPDDR4 interface with ECC.

Table 2-50: Impedance, Length, and Spacing Guidelines for CA\_B Address/Command Signals in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Chip to Chip)	L4 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	52±10%	50±10%	$\Omega$
Trace width	4.0	6.0	3.5	3.5	4.0	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤1.5	≤0.5	inches
Spacing in addr/cmd (minimum)	4.0	6.0	4.0	6.0	6.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	mil
Maximum PCB via count	4					–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 with ECC Chip Select Routing**

Figure 2-43 shows the chip select (CS0/CS1) routing for the LPDDR4 interface with ECC. The  $R_{TT}$  termination exists to terminate the fly-by route.

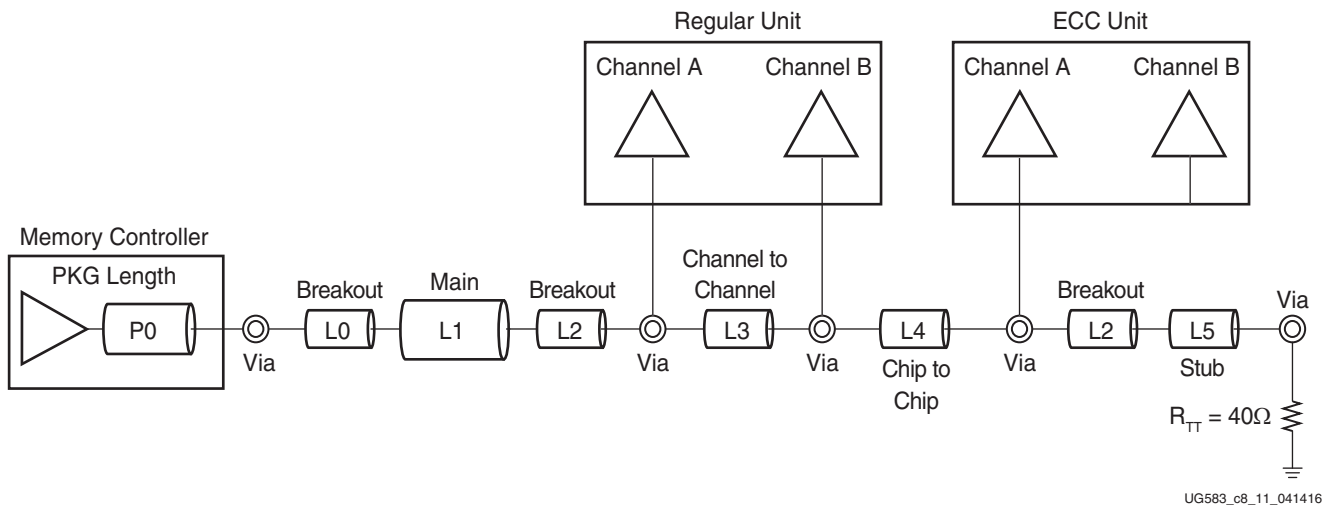


Figure 2-43: CS0/CS1 Routing for LPDDR4 with ECC

Table 2-51 shows the impedance, length, and spacing guidelines for CS0/CS1 routing in the LPDDR4 interface with ECC.

Table 2-51: Impedance, Length, and Spacing Guidelines for CS0/CS1 in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Channel to Channel)	L4 (Chip to Chip)	L5 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	52±10%	52±10%	50±10%	$\Omega$
Trace width	4.0	6.0	3.5	3.5	3.5	4.0	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤0.5	≤1.5	≤0.5	inches
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	30	mil
Maximum PCB via count	4						–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).



**LPDDR4 with ECC Clock Enable Routing**

Figure 2-44 shows the clock enable (CKE0/CKE1) routing options for the LPDDR4 interface with ECC. The  $R_{TT}$  termination exists to terminate the fly-by route.

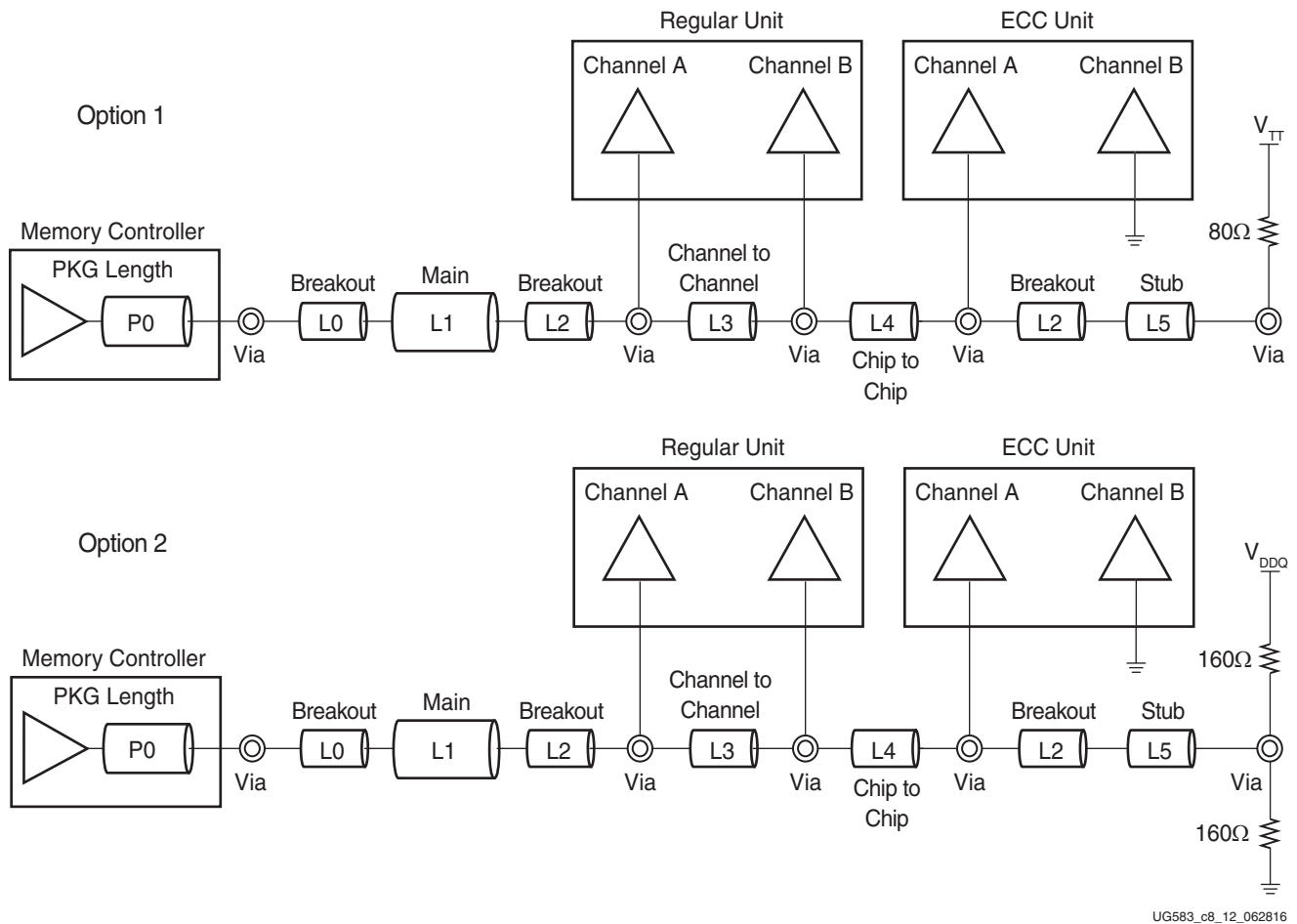


Figure 2-44: CKE0/CKE1 Routing for LPDDR4 with ECC

Table 2-52 shows the impedance, length, and spacing guidelines for CKE0/CKE1 in the LPDDR4 interface with ECC.

Table 2-52: Impedance, Length, and Spacing Guidelines for CKE0/CKE1 in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Channel to Channel)	L4 (Chip to Chip)	L5 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	52±10%	52±10%	50±10%	Ω
Trace width	4.0	6.0	3.5	3.5	3.5	4.0	mil
Trace length	0.0~0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤0.5	≤1.5	≤0.5	inches

Table 2-52: Impedance, Length, and Spacing Guidelines for CKE0/CKE1 in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Channel to Channel)	L4 (Chip to Chip)	L5 (Stub)	Units
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	30	mil
Maximum PCB via count	5						–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 with ECC CK0 Differential Point-to-Point Routing**

Figure 2-45 shows the CK0 differential point-to-point routing for the LPDDR4 interface with ECC.

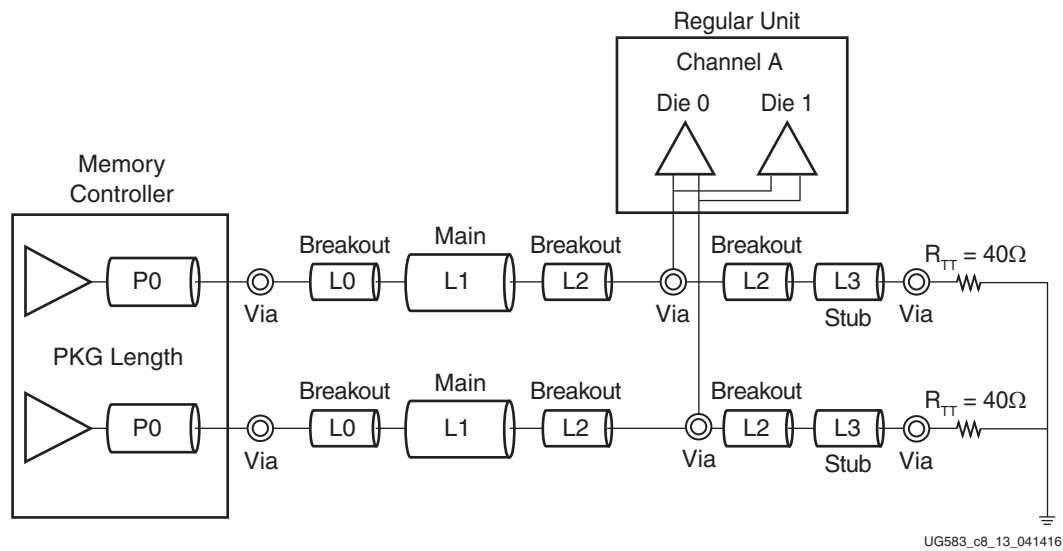


Figure 2-45: CK0 Routing for LPDDR4 with ECC

Table 2-53 shows the impedance, length, and spacing guidelines for CK0 in the LPDDR4 interface with ECC.

Table 2-53: Impedance, Length, and Spacing Guidelines for CK0 in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance Z <sub>diff</sub>	86±10%	76±10%	88±10%	93±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.5/3.5	4.0/6.0/4.0	mil

Table 2-53: Impedance, Length, and Spacing Guidelines for CK0 in LPDDR4 Interface with ECC (Cont'd)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Stub)	Units
Trace length	$\leq 0.55/0.95^{(1)}$	$\leq 4.0$	$\leq 0.3$	$\leq 0.5$	inches
Spacing in addr/cmd/ctrl (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count per signal	3				-

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

### LPDDR4 with ECC CK1 Differential Fly-by Routing

Figure 2-46 shows the CK1 differential fly-by routing for the LPDDR4 interface with ECC.

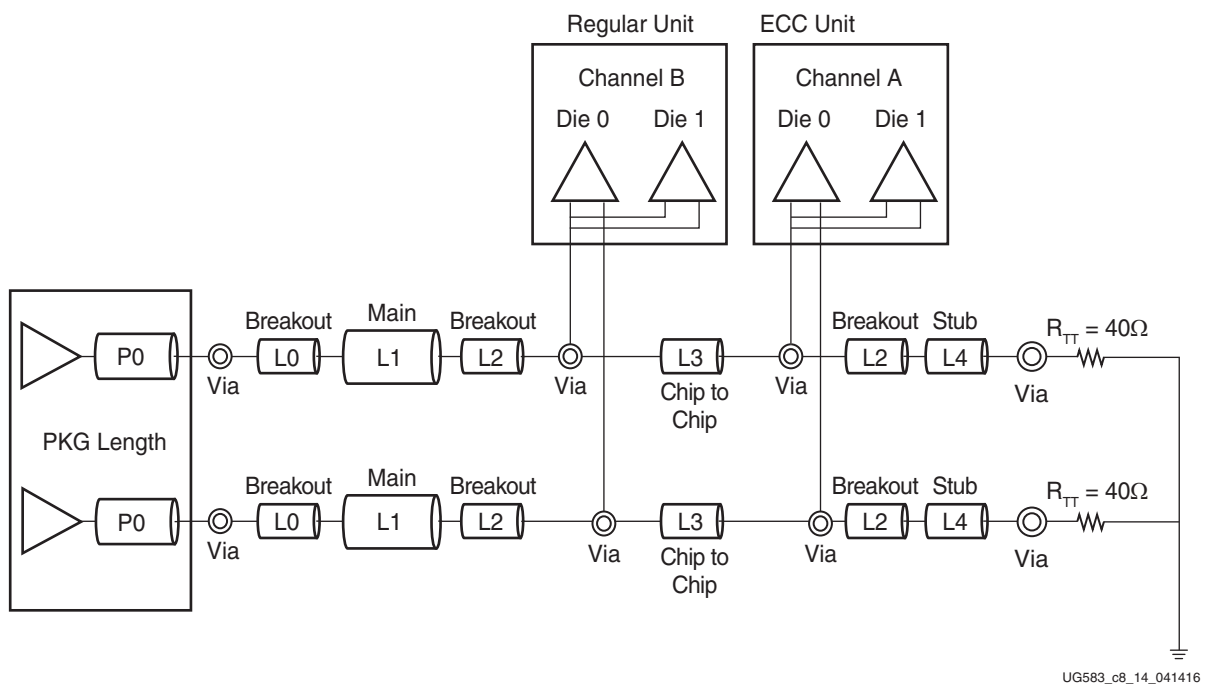


Figure 2-46: CK1 Routing for LPDDR4 with ECC

Table 2-54 shows the impedance, length, and spacing guidelines for CK1 in the LPDDR4 interface with ECC.

Table 2-54: Impedance, Length, and Spacing Guidelines for CK1 in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (Chip to Chip)	L4 (Stub)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance $Z_{diff}$	86±10%	76±10%	88±10%	93±10%	93±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.25/3.5	3.5/4.0/3.5	4.0/6.0/4.0	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	≤1.5	≤0.5	inches
Spacing in addr/cmd/ctrl (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	mil
Maximum PCB via count per signal	4					–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 with ECC Data (DQ) Point-to-Point Routing**

Figure 2-47 shows the DQ point-to-point routing for the LPDDR4 interface with ECC.

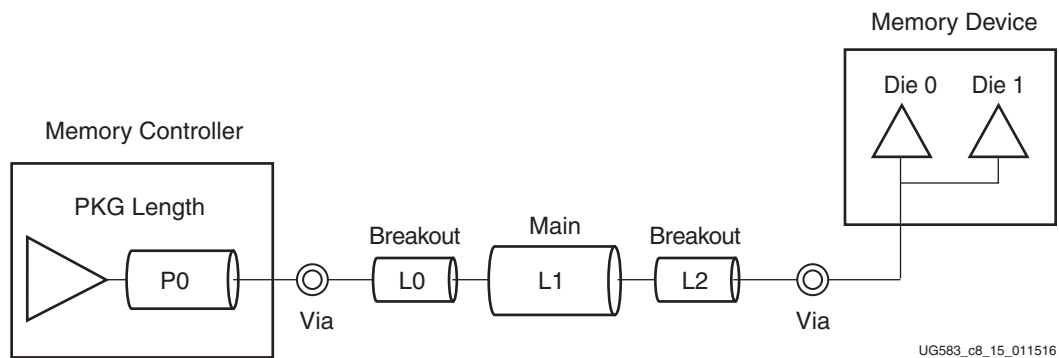


Figure 2-47: DQ Routing for LPDDR4 with ECC

Table 2-55 shows the impedance, length, and spacing guidelines for DQ in the LPDDR4 interface with ECC.

Table 2-55: Impedance, Length, and Spacing Guidelines for DQ in LPDDR4 Interface with ECC

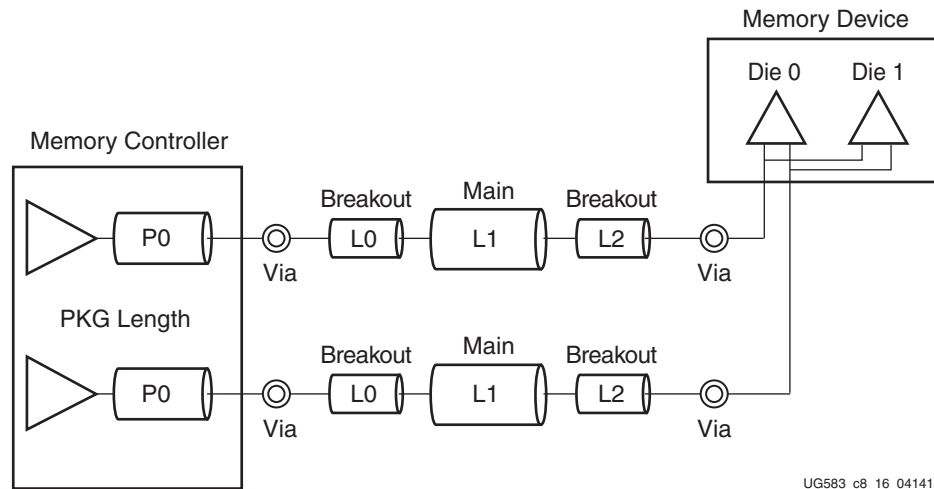
Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	$50 \pm 10\%$	$39 \pm 10\%$	$52 \pm 10\%$	$\Omega$
Trace width	4.0	6.0	3.5	mil
Trace length	$\leq 0.55/0.95^{(1)}$	$\leq 4.0$	$\leq 0.3$	inches
Spacing in byte (including DQS) (minimum)	4.0	8.0	4.0	mil
Spacing byte to byte (minimum)	8.0	20	4.0	mil
Spacing to other group signals (minimum)	8.0	30	8.0	mil
Maximum PCB via count	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**LPDDR4 with ECC DQS Differential Point-to-Point Routing**

Figure 2-48 shows the DQS point-to-point routing for the LPDDR4 interface with ECC.



UG583\_c8\_16\_041416

Figure 2-48: DQS Routing for LPDDR4 with ECC

Table 2-56 shows the impedance, length, and spacing guidelines for DQS in the LPDDR4 interface with ECC.

Table 2-56: Impedance, Length, and Spacing Guidelines for DQS in LPDDR4 Interface with ECC

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Clock differential impedance $Z_{diff}$	86±10%	76±10%	88±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.5/3.5	mil
Trace length	≤0.55/0.95 <sup>(1)</sup>	≤4.0	≤0.3	inches
Spacing in addr/cmd/ctrl (minimum)	4.0	20	4.0	mil
Spacing to other group signals (minimum)	8.0	30	8.0	mil
Maximum PCB via count per signal	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### LPDDR4 with ECC Length and Skew Constraints

The constraints requirement for each signal group in the LPDDR4 with ECC memory interface consists of two parts:

- Maximum length constraints
- Skew constraints

The maximum length constraints are shown in [Table 2-57](#).

Table 2-57: LPDDR4 with ECC Maximum Length Constraints

Signal Group	Reference Figure	Maximum Length Constraints (inches)
CA_A Address/command P0+L0+L1+L2+L3	<a href="#">Figure 2-41</a>	7.0
CA_B Address/command P0+L0+L1+L2+L3+L4	<a href="#">Figure 2-42</a>	8.5
CS0/CS1 P0+L0+L1+L2+L3+L4+L5	<a href="#">Figure 2-43</a>	9.0
CKE0/CKE1 P0+L0+L1+L2+L3+L4+L5	<a href="#">Figure 2-44</a>	9.0
Data signals P0+L0+L1+L2	<a href="#">Figure 2-47</a>	6.2

The skew constraints are listed in [Table 2-58](#).

**Table 2-58: LPDDR4 with ECC Skew Constraints**

Signal Group	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control to CK (A) <sup>(1)</sup> Address/command/control to CK (B) <sup>(1)</sup>	MPSoC to memory device	±8	±47
Data (DQ/DM) to DQS (A) Data (DQ/DM) to DQS (B)	MPSoC to memory device	±5	±29
DQ/DM (slowest to fastest) (A) DQ/DM (slowest to fastest) (B)	MPSoC to memory device	5	29
CK_T and CK_C (A) CK_T and CK_C (B)	MPSoC to memory device	2	11
DQS_T and DQS_C (A) DQS_T and DQS_C (B)	MPSoC to memory device	2	11

**Notes:**

1. Consider CS and CKE to be part of the A group.
2. For skew specifications, refer to items 3–8 in [General Memory Routing Guidelines](#).



**IMPORTANT:** *FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.*

## PCB Guidelines for LPDDR3 SDRAM (PL and PS)

### Overview

This section provides electrical design guidelines for typical LPDDR3 SDRAM interfaces for the PS architecture in Zynq UltraScale+ MPSoCs based on system-level signal integrity simulation. A 32-bit wide one 8 GB SDP SDRAM system and a 64-bit wide two 8 GB SDP SDRAM system are used as typical design examples. For both cases, clock, data, and address/command signals are point-to-point connections. For a x32 single component system, control signals are also point-to-point connections. In a x64 two-component system, they are connected in fly-by topology from the FPGA to two SDRAMs.



**IMPORTANT:** *All routing guidelines in this section must be followed to achieve the maximum data rates specified for the LPDDR3 SDRAM interface for typical system designs. Customers could have unique or specific designs with particular violations of some rules. In these scenarios, design or routing trade-offs have to be taken in other routing parameters to mitigate the risk. System-level channel signal integrity simulations are required to evaluate such trade-offs.*

## LPDDR3 SDRAM Interface Signal Description

The LPDDR3 SDRAM interface consists of clock, control, address, command, and data signals as shown in [Table 2-59](#).

**Table 2-59: LPDDR3 SDRAM Interface Signal Description**

Signal Name	Description
CK_t, CK_c	Differential clock
<b>Control Signals</b>	
CKE[1:0]	Clock enable
CS[1:0]_n	Chip select
ODT	DQ ODT control
<b>Command/Address Signals</b>	
CA[9:0]	Command/address bus
<b>Data Signals</b>	
DQ[32:0]	Data bus
DQS[3:0]_t/c	Differential data strobe
DM[3:0]	Input data mask

**Notes:**

- Actual signal list might vary based on configuration.

## LPDDR3 Address Copy

The PS DDR interface includes a feature called address copy to reduce loading on the address/command bus when operating in LPDDR3 mode. A[9:0] are replicated on ACT\_N, BG[0], BA[1:0], and A[15:10] to allow fewer loads on the CA bus ([Table 2-60](#)).

**Table 2-60: LPDDR3 Address Copy**

MPSoC Pin	DRAM Pin	MPSoC Pin Copy	DRAM Pin
PS_DDR_A0	CA0_A	PS_DDR_A10	CA0_B
PS_DDR_A1	CA1_A	PS_DDR_A11	CA1_B
PS_DDR_A2	CA2_A	PS_DDR_A12	CA2_B
PS_DDR_A3	CA3_A	PS_DDR_A13	CA3_B
PS_DDR_A4	CA4_A	PS_DDR_A14	CA4_B
PS_DDR_A5	CA5_A	PS_DDR_A15	CA5_B
PS_DDR_A6	CA6_A	PS_DDR_BA0	CA6_B
PS_DDR_A7	CA7_A	PS_DDR_BA1	CA7_B
PS_DDR_A8	CA8_A	PS_DDR_BG0	CA8_B
PS_DDR_A9	CA9_A	PS_DDR_ACT_N	CA9_B

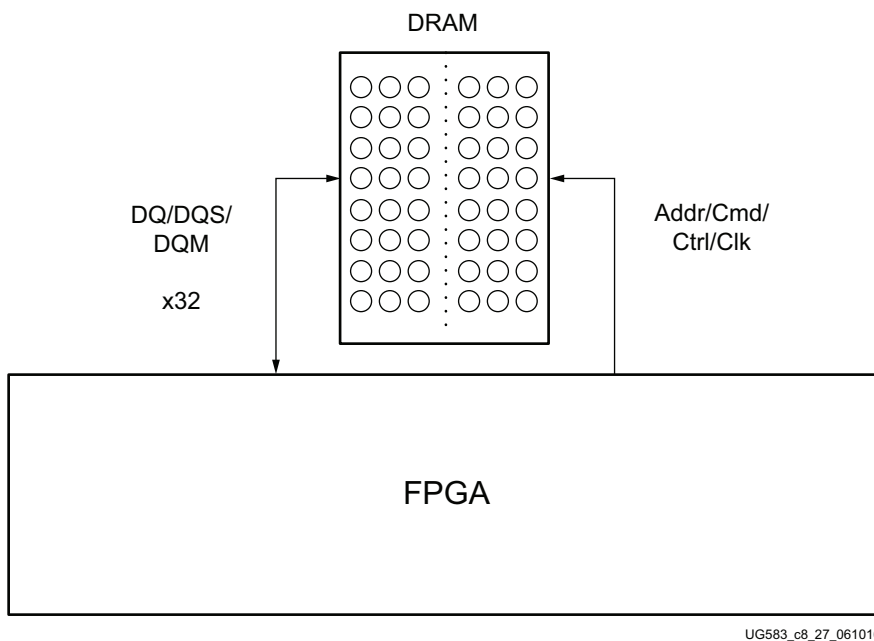


## VREFCA and VREFDQ

Both PL and PS logic use ODT for data signals, so the voltage levels for VREFCA and VREFDQ will be different. Refer to the memory vendor data sheet for specific requirements.

## Design Example for x32 and x64 LPDDR3 SDRAM

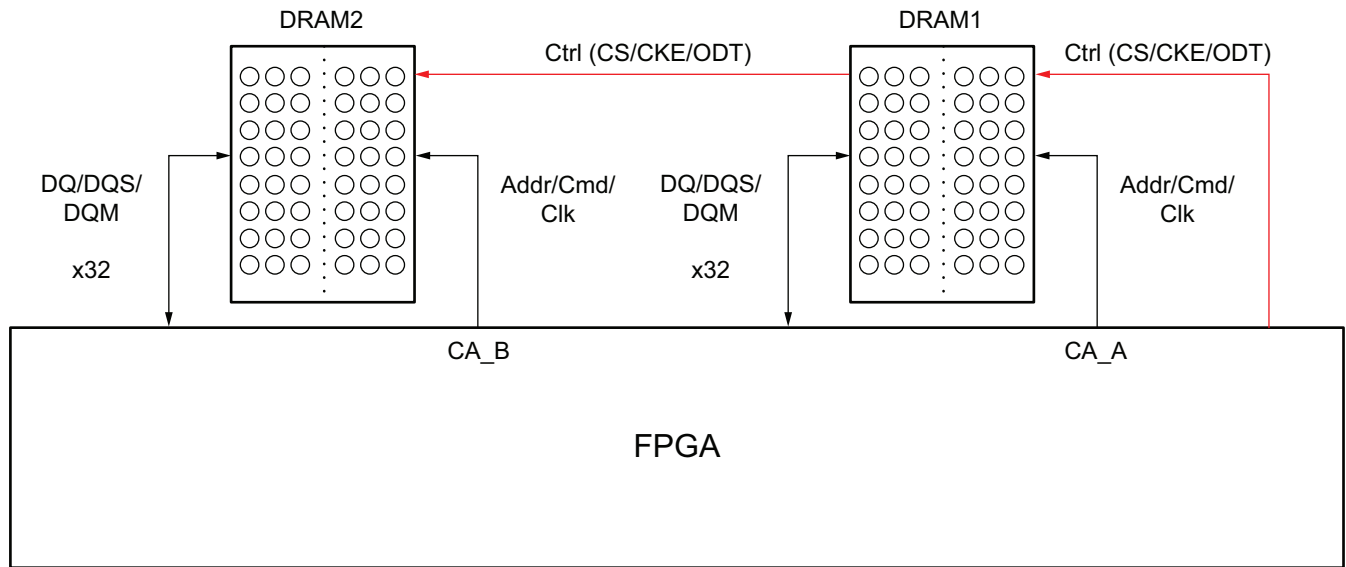
To optimize the signal routing, the recommendation for a one-component 32-bit system placement is shown in [Figure 2-49](#).



UG583\_c8\_27\_061016

Figure 2-49: Design Example for One-Component x32 LPDDR3 SDRAM

The recommendation for a two-component 64-bit system is shown in Figure 2-50.



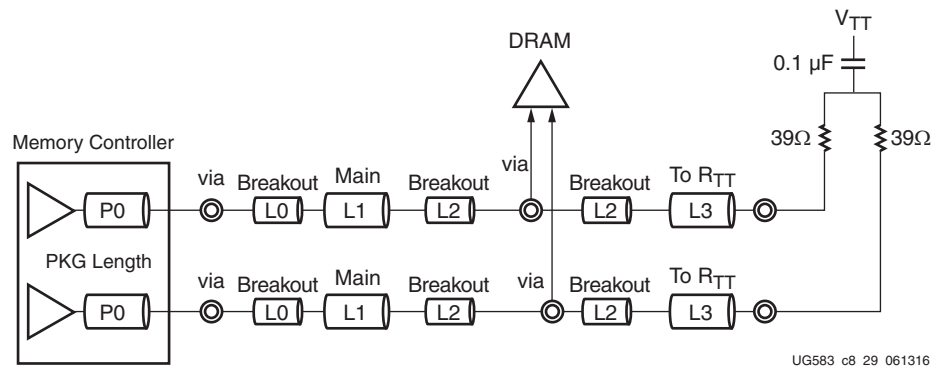
UG583\_c8\_28\_040717

Figure 2-50: Design Example for Two-Component x64 LPDDR3 SDRAM

## Topology and Routing Guidelines for LPDDR3 SDRAM

### LPDDR3 SDRAM Clk Point-to-Point Termination

Figure 2-51 shows the clock point-to-point termination for LPDDR3 SDRAM.



UG583\_c8\_29\_061316

Figure 2-51: Clk Point-to-Point Termination for LPDDR3 SDRAM

Table 2-61 and Table 2-62 show the LPDDR3 SDRAM impedance, length, and spacing guidelines for clock signals for x32 and x64 systems, respectively. For LPDDR3 SDRAM, clock signals have on-board termination. There is no on-die termination available for clock signals. For both x32 and x64 systems, clock signals are point-to-point connections to achieve the highest operation frequency.

Table 2-61: LPDDR3 x32 SDRAM System Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (OBT)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	-
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	39±10%	$\Omega$
Differential impedance $Z_{0d}$	86±10%	76±10%	88±10%	76±10%	$\Omega$
Trace width/spacing/width (nominal)	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.25/3.5	6.0/6.0/6.0	mil
Trace length	0.0~0.55	1.0~4.0	0.0~0.3	0.0~0.5	inch
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count	3				mil

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

Table 2-62: LPDDR3 x64 SDRAM System Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (OBT)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	-
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	39±10%	$\Omega$
Differential impedance $Z_{0d}$	86±10%	76±10%	88±10%	76±10%	$\Omega$
Trace width/spacing/width (nominal)	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.25/3.5	6.0/6.0/6.0	mil
Trace length	0.0~0.55	1.0~4.0	0.0~0.3	0.0~0.5	inch
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil

Table 2-62: LPDDR3 x64 SDRAM System Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (OBT)	Units
Maximum PCB via count	3				mil

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**LPDDR3 SDRAM Address, Command, and Control Point-to-Point Termination**

With high-speed signaling in LPDDR3 SDRAM, point-to-point topology is used for address, command, and control signals to achieve the best signal integrity in X32 system. Each address, command, and control signal by itself is routed continuously in the same layer from device pin to far-end termination, except in breakout areas. In other words, each individual address, command, or control signal routing is not broken into routings on multiple layers. Figure 2-52 shows the address, command, and control point-to-point termination for LPDDR3 SDRAM.

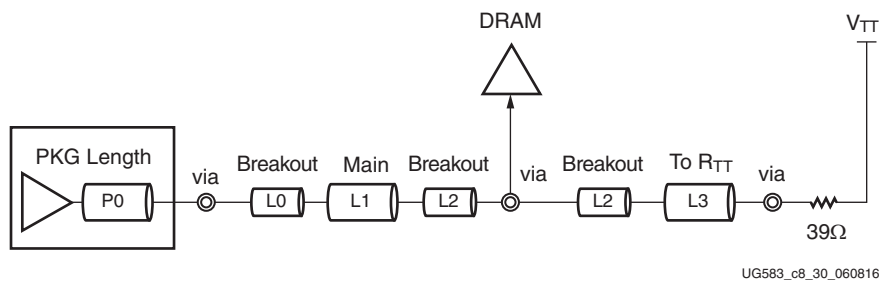


Figure 2-52: Address, Command, and Control Point-to-Point Termination for LPDDR3 SDRAM

Table 2-63 and Table 2-64 show the LPDDR3 SDRAM impedance, length, and spacing guidelines for address, command, and control signals for x32 and x64 systems, respectively. In the LPDDR3 DRAM package, the vertical pitch is only 0.65 mm. Due to limited space, the trace impedance in L2 could be up to 60Ω. Address, command, and control signals have on-board termination. There is no on-die termination available. All address, command, and control signals are point-to-point connections for a x32 single-SDRAM system. For a x64 two-SDRAM system, address and command signals are point-to-point connections while control signals including CS, CKE, and ODT are connected from the FPGA to the two SDRAMs in fly-by topology.

**Table 2-63: LPDDR3 x32 SDRAM Impedance, Length, and Spacing Guidelines for Address, Command, and Control Signal Point-to-Point Connection**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (OBT)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	-
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	39±10%	$\Omega$
Trace width (nominal)	4.0	6	3.5	6	mil
Trace length	0.0~0.55	1.0~4.0	0.0~0.2	0.0~0.5	inch
Spacing in cmd/addr/ctrl signals (minimum)	4.0	8.0	4.0	4.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count	3				mil

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**Table 2-64: LPDDR3 x64 SDRAM Impedance, Length, and Spacing Guidelines for Address and Command Signal Point-to-Point Connection**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (OBT)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	-
Single-ended impedance $Z_0$	50±10%	39±10%	52±10%	39±10%	$\Omega$
Trace width (nominal)	4.0	6	3.5	6	mil
Trace length	0.0~0.55	1.0~4.0	0.0~0.2	0.0~0.5	inch
Spacing in cmd/addr/ctrl signals (minimum)	4.0	8.0	4.0	4.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	mil
Maximum PCB via count	3				mil

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).
- Control signals (CS/CKE/ODT) are routed under different constraints.

### LPDDR3 SDRAM Control Fly-by Termination for x64 Two-SDRAM System

With high-speed signaling in LPDDR3 SDRAMs, point-to-point topology is used for address and command signals to achieve the best signal integrity. However, due to pin limits on the FPGA side, control signals are routed from the FPGA to two SDRAMs in fly-by topology in a x64 two-SDRAM system. Figure 2-53 shows the control fly-by termination for LPDDR3 SDRAM.

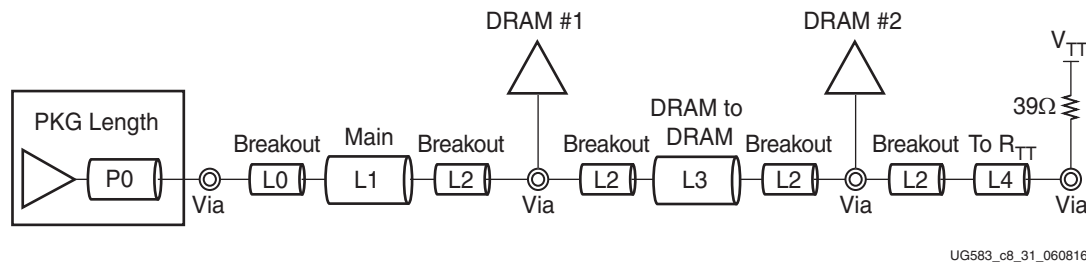


Figure 2-53: Control Fly-by Termination for LPDDR3 SDRAM

Table 2-65 shows the LPDDR3 SDRAM impedance, length, and spacing guidelines for control signals for a x64 two-SDRAM system.

Table 2-65: LPDDR3 SDRAM Impedance, Length, and Spacing Guidelines for Control Signals (CKE/CS/ODT)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (DRAM to DRAM)	L4 (OBT)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	-
Single-ended impedance $Z_0$	52±10%	39±10%	52±10%	52±10%	50±10%	Ω
Trace width (nominal)	3.5	6.0	3.5	3.5	4.0	mil
Trace length	0.0~0.55	1.2~4.0	0.0~0.3	2.0~2.4	0.0~0.5	inch
Spacing in cmd/addr/ctrl signals (minimum)	4.0	8.0	4.0	4.0	8	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	8.0	30	30	mil
Maximum PCB via count	4					mil

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).
2. The entire trace length between two SDRAM control signal balls including breakout region should be between 2.0 to 2.4 inches.
3. Clock, address, and command point-to-point traces at the second SDRAM need to have an extra 0.5 inch added when compared to the control trace to compensate for control double loading.

**LPDDR3 SDRAM Data Signals Point-to-Point**

Figure 2-54 shows the data signals point-to-point for LPDDR3 SDRAM. For DQ/DM/DQS signals, ODT is available on both the FPGA and SDRAM side.

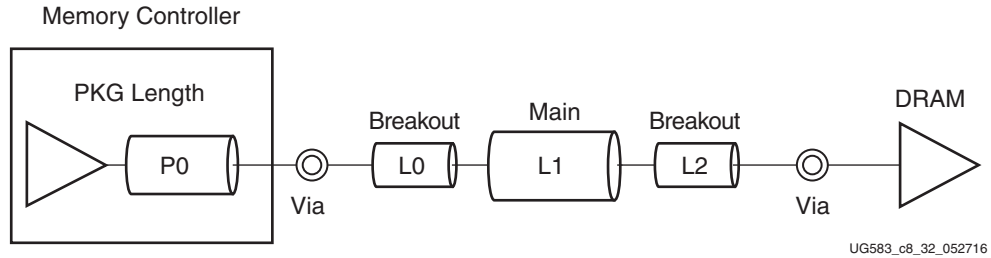


Figure 2-54: Data Signals Point-to-Point for LPDDR3 SDRAM

Table 2-66 shows the LPDDR3 SDRAM impedance, length, and spacing guidelines for data signals for both x32 and x64 systems.

Table 2-66: LPDDR3 SDRAM Impedance, Length, and Spacing Guidelines for Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	-
DQ single-ended impedance $Z_0$	50±10%	39±10%	52±10%	$\Omega$
DQS differential impedance $Z_{diff}$	86±10%	76±10%	88±10%	$\Omega$
Trace width (nominal)	4.0	6.0	3.5	mil
Differential trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	3.5/3.25/3.5	mil
Trace length (nominal)	0.0~0.55	1.0~4.0	0.0~0.3	inch
Spacing in byte (minimum)	4.0	8.0	4.0	mil
Spacing byte to byte (minimum)	4.0	20	4.0	mil
DQ to DQS spacing (minimum)	4.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	8.0	mil
Maximum PCB via count	2			mil

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

## LPDDR3 SDRAM Routing Constraints

There are two constraints requirements for each signal group in the LPDDR3 memory interface:

- Maximum length constraints
- Skew constraints

The maximum length constraints are shown in [Table 2-67](#) and [Table 2-68](#). The maximum length is counted from the FPGA, through the PCB board, to the SDRAM package.

**Table 2-67: LPDDR3 x32 SDRAM Maximum Length Constraints**

Signal Group	Reference Figure	Maximum Length Constraints (inches)
Address/command/control	<a href="#">Figure 2-52</a>	5.83
Data signals	<a href="#">Figure 2-54</a>	5.83
Clock	<a href="#">Figure 2-51</a>	5.83

**Table 2-68: LPDDR3 x64 SDRAM Maximum Length Constraints**

Signal Group	Reference Figure	Maximum Length Constraints (inches)
Address/command/control	<a href="#">Figure 2-53</a>	9.00
Data signals	<a href="#">Figure 2-54</a>	5.83
Clock	<a href="#">Figure 2-51</a>	9.00

The skew constraints are shown in [Table 2-69](#).

**Table 2-69: LPDDR3 SDRAM Memory Skew Constraints**

Signal Group	Signal Segment	Skew Constraints (ps)	Skew Constraints (mil)
Address/command/control to clock <sup>(1)</sup>	FPGA to memory device	±4	±23
Data to DQS	FPGA to memory device	±5	±29
DQS_p and DQS_n	FPGA to memory device	2	12
Clock to strobe	FPGA to memory device	-149 to 1,796	-879 to 10,600

**Notes:**

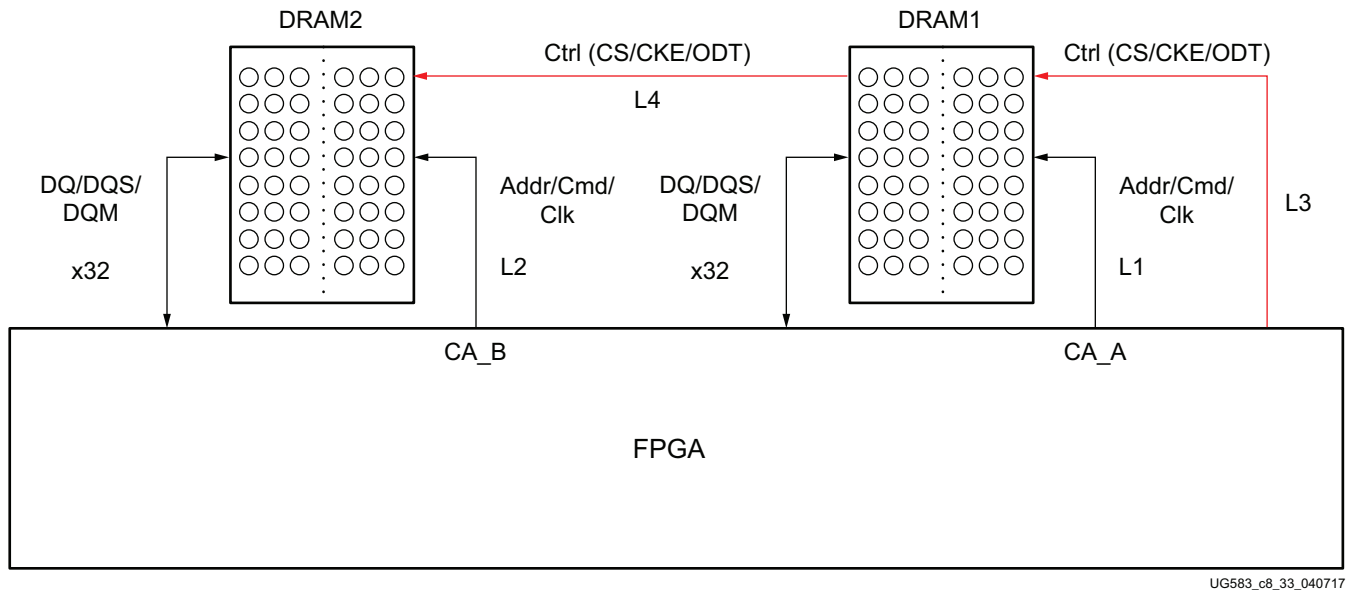
1. This does not take into account the extra 0.5 in trace on the clock, address, and command signals in a x64 system.
2. Constraints are referred to signals within the same SDRAM.
3. For skew specifications, refer to items 3–8 in [General Memory Routing Guidelines](#).



**IMPORTANT:** Package routing length must be included in both maximum length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values.



Due to the extra loading at the second DRAM in the control bus, additional trace length is needed in address, command, and clock traces at the second SDRAM in a x64 two-SDRAM system shown in Figure 2-55.



UG583\_c8\_33\_040717

Figure 2-55: LPDDR3 x64 Two-SDRAM System

The matching constraints for control, address, command, and clock signals are shown here:

- L1 = L3
- L2 = L3 + L4 + 0.5 in

## PCB Guidelines for RLDRAM 3 Memory

### Overview

This section provides electrical design guidelines for typical RLDRAM 3 memory interfaces from system-level signal integrity simulations for the UltraScale architecture. A two-component RLDRAM 3 72-bit wide interface using x36 devices is used as a typical design example in this document. Clamshell and fly-by topologies are discussed.



**IMPORTANT:** All routing guidelines in this section must be followed to achieve the maximum data rates specified for the RLD3 memory interface for typical system designs. Customers could have unique or specific designs with particular violations of some rules. In these scenarios, design or routing trade-offs have to be taken in other routing parameters to mitigate the risk. System-level channel signal integrity simulations are required to evaluate such trade-offs. It is important to read the [General Memory Routing Guidelines](#) section before continuing with this section.

## RLDRAM 3 Memory Interface Signal Description

The RLD3 memory interface consists of clock, control, address, command, and data signals as shown in [Table 2-70](#).

Table 2-70: RLD3 Memory Interface Signal Description

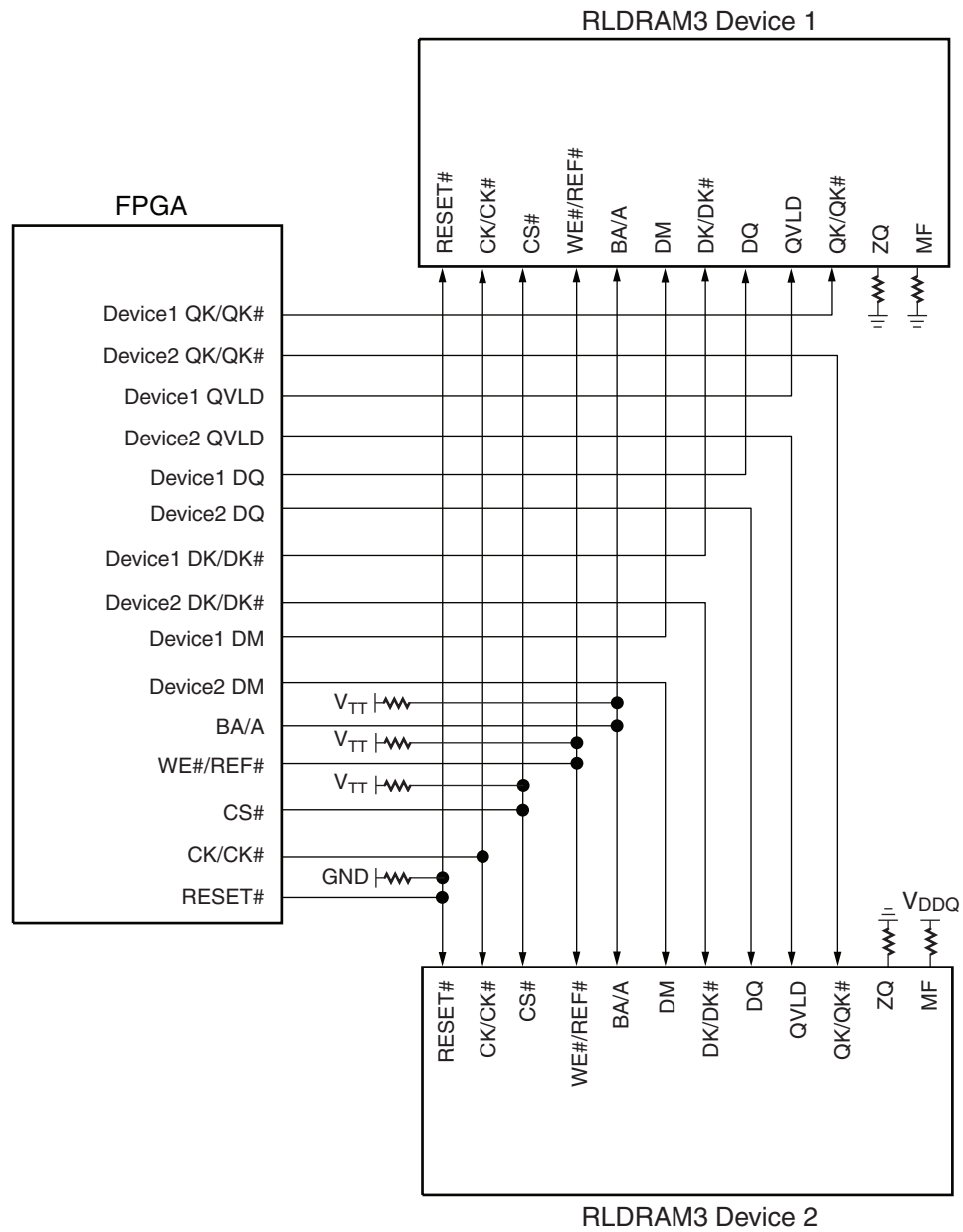
Signal Name	Description
<b>Clock Signals</b>	
ck_p/n[1:0]	Command and address clock
dk_p/n[3:0]	Write data clock
qk_p/n[7:0]	Read data clock (to device)
<b>Control Signals</b>	
reset_n	Tie low to 4.7 kΩ resistor at far-end near RLD3 device
<b>Address Signals</b>	
a[20:0]	Memory address bus
ba[3:0]	Bank address
<b>Command Signals</b>	
cs_n	Chip select - works with ref_n and we_n to define command
ref_n	Works with cs_n and we_n to define command
we_n	Works with cs_n and ref_n to define command
<b>Data Signals</b>	
dq[71:0]	Data bus
dm[3:0]	Write data mask
qvld[3:0]	Read data valid

**Notes:**

- Actual signal list might vary based on configuration.

# RLDRAM 3 Memory Topology and Routing Guidelines for Clamshell and Fly-by Configurations

## RLDRAM 3 Memory Clamshell Configuration with Width Expansion



UG583\_c2\_26\_081415

Figure 2-56: Clamshell RLDRAM 3 Memory with Width Expansion

## RLDRAM 3 Memory Fly-by Configuration with Width Expansion

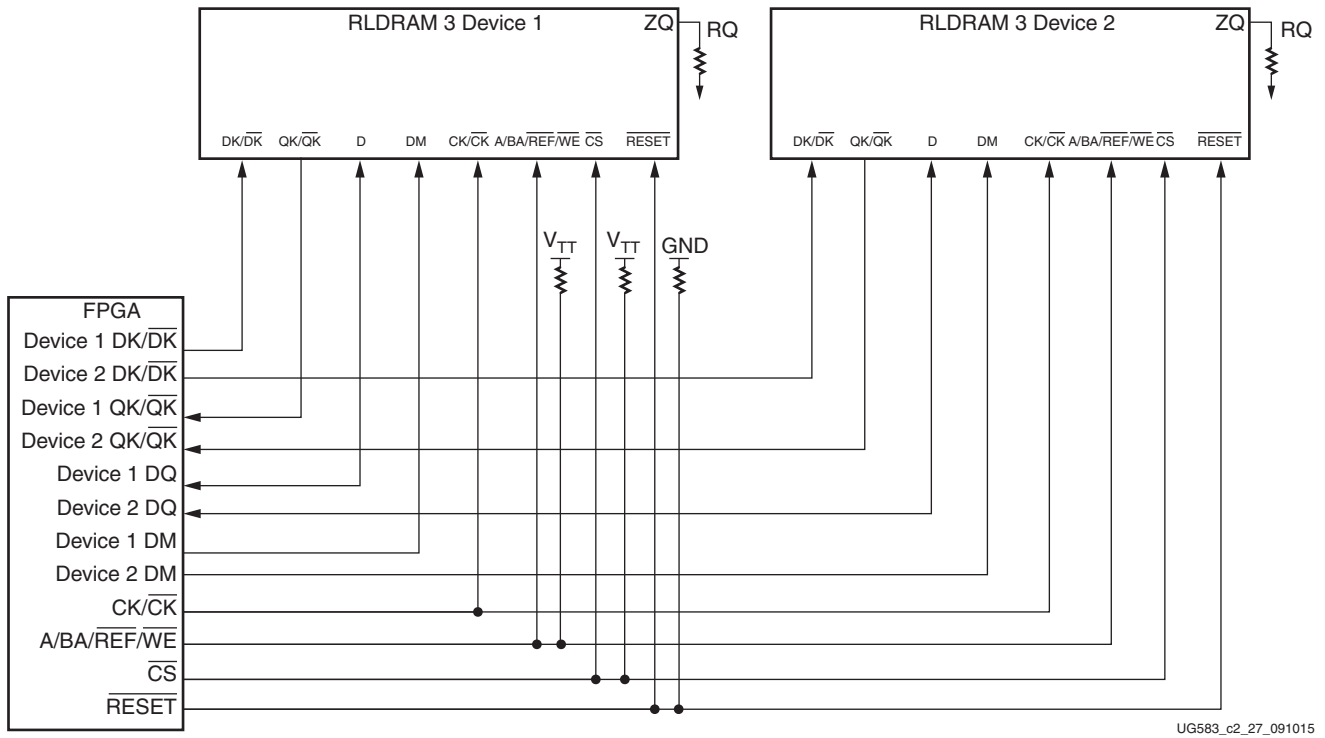


Figure 2-57: Fly-by RLDRAM 3 Memory with Width Expansion

### RLDRAM 3 Memory Address and Command Clamshell Routing and Termination

Using clamshell topology, each address and command signal is routed to opposite memory devices with matched topologies due to the mirroring ability of RLDRAM 3 devices (Figure 2-58).

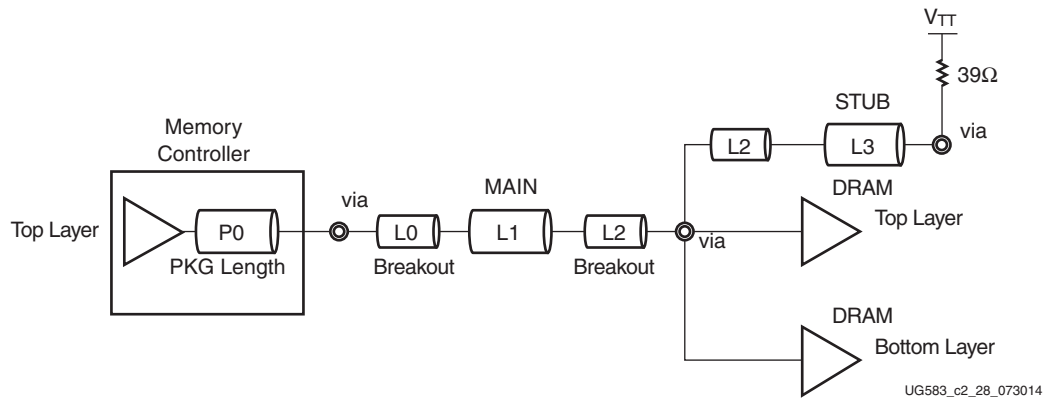


Figure 2-58: Address and Command Clamshell Routing for Clamshell-mounted RLDRAM 3 Memories

Table 2-71 shows the clamshell routing impedance, length, and spacing guidelines for address and command signals.

Table 2-71: Clamshell Routing Impedance, Length, and Spacing Guidelines for Address and Command Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance Z <sub>0</sub>	50±10%	39±10%	50±10%	39±10%	Ω
Trace width	4.0	6.0	4.0	6.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.25	0~1.0	inches
Spacing in address and command signals (minimum)	4.0	8.0	4.0	8.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	mil
Maximum PCB via count	6				–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### RLDRAM 3 Memory Address and Command Fly-by Routing and Termination

Using fly-by topology, each address and command signal is routed to drop by each memory device and connect to its corresponding pin. (Figure 2-59).

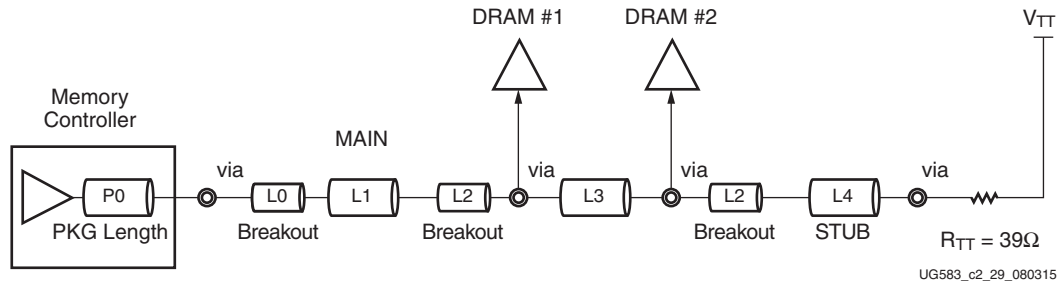


Figure 2-59: Address, Command, and Control Fly-by Termination for RLDRAM 3 Memories

Table 2-72 shows the RLDRAM 3 memory fly-by impedance, length, and spacing guidelines for address, command, and control signals.

Table 2-72: RLDRAM 3 Memory Fly-by Impedance, Length, and Spacing Guidelines for Address, Command, and Control Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance Z <sub>0</sub>	50±10%	36±10%	50±10%	50±10%	39±10%	Ω
Trace width	4.0	7.0	4.0	4.0	6.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.25	0.35~1.05	0~1.0	inches
Spacing in address, command, and control (minimum)	4.0	8.0	4.0	8.0	8.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count	6					–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**RLDRAM 3 Memory CK\_P/N Clamshell Routing**

Figure 2-60 shows the ck\_p/n clamshell routing for clamshell-mounted RLDRAM 3 memories.

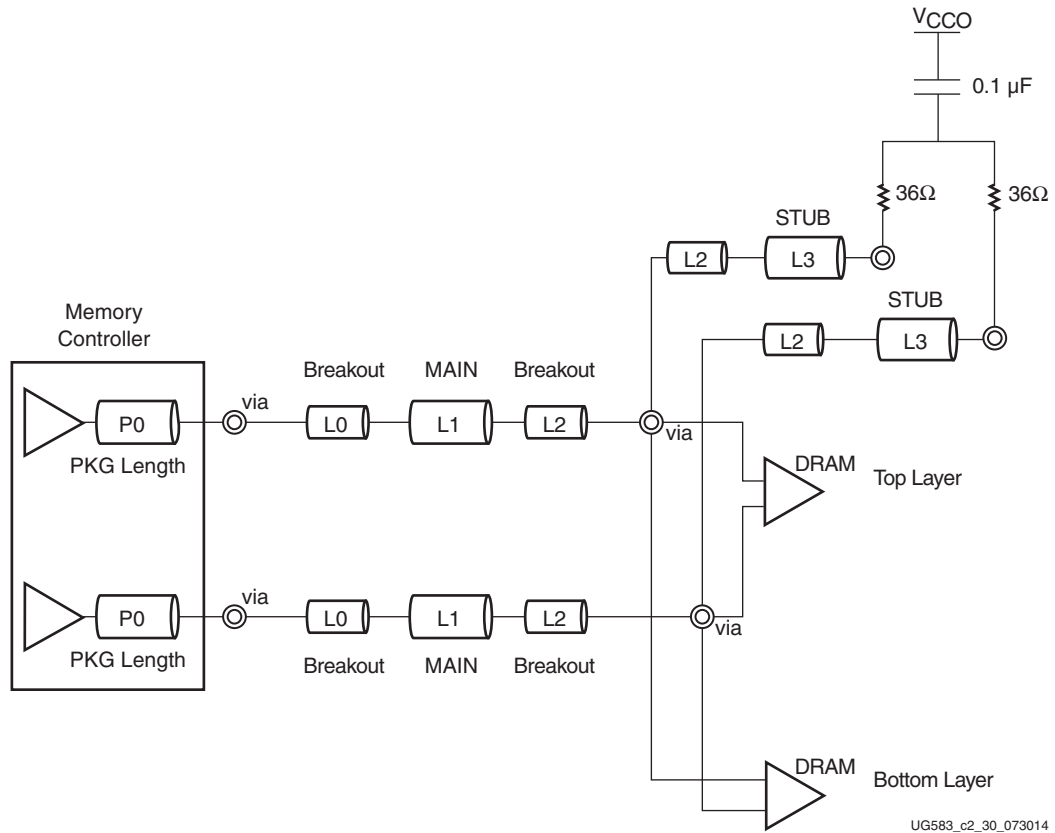


Figure 2-60: ck\_p/n Clamshell Routing for Clamshell-mounted RLDRAM 3 Memories

Table 2-73 shows the RLDRAM 3 memory clamshell impedance, length, and spacing guidelines for ck\_p/n signals.

Table 2-73: RLDRAM 3 Memory Clamshell Impedance, Length, and Spacing Guidelines for ck\_p/n Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (To Term)	Units
Trace type	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance $Z_{DIFF}$	86±10%	76±10%	86±10%	76±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	6.0/6.0/6.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~3.0	0.0~0.25	0~1.0	inches
Spacing to other group signals (minimum)	8.0	30	30	20	mil
Maximum PCB via count per signal	3				–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### RLDRAM 3 Memory CK\_P/N Fly-by Routing

Figure 2-61 shows the clock differential fly-by routing for RLDRAM 3 memory.

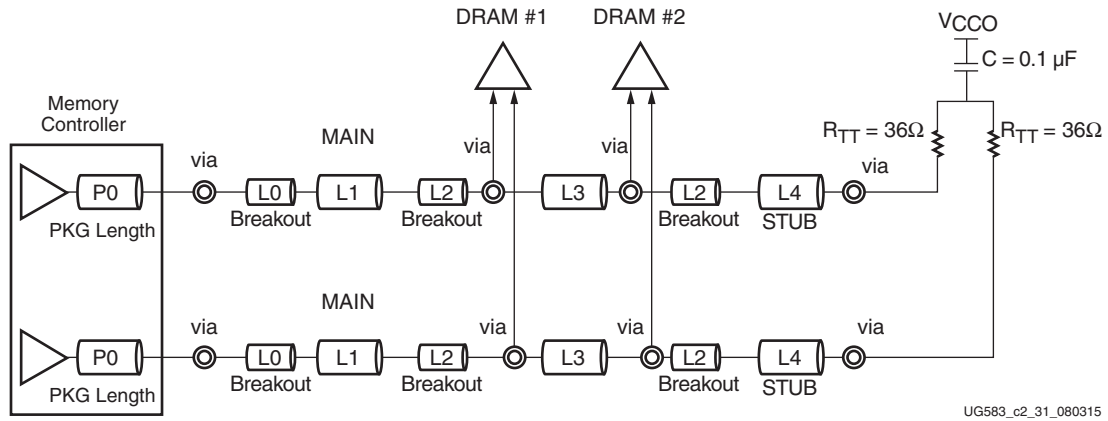


Figure 2-61: Clock Differential Fly-by Routing for RLDRAM 3 Memory

Table 2-74 shows the RLDRAM 3 memory fly-by impedance, length, and spacing guidelines for clock signals.

Table 2-74: RLDRAM 3 Memory Fly-by Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Clock differential impedance Z <sub>DIFF</sub>	86±10%	76±10%	86±10%	90±10%	76±10%	Ω
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	4.0/5.0/4.0	6.0/6.0/6.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~3.0	0.0~0.25	0.35~1.05	0~1.0	inches
Spacing in address, command, and control signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count per signal	4					–

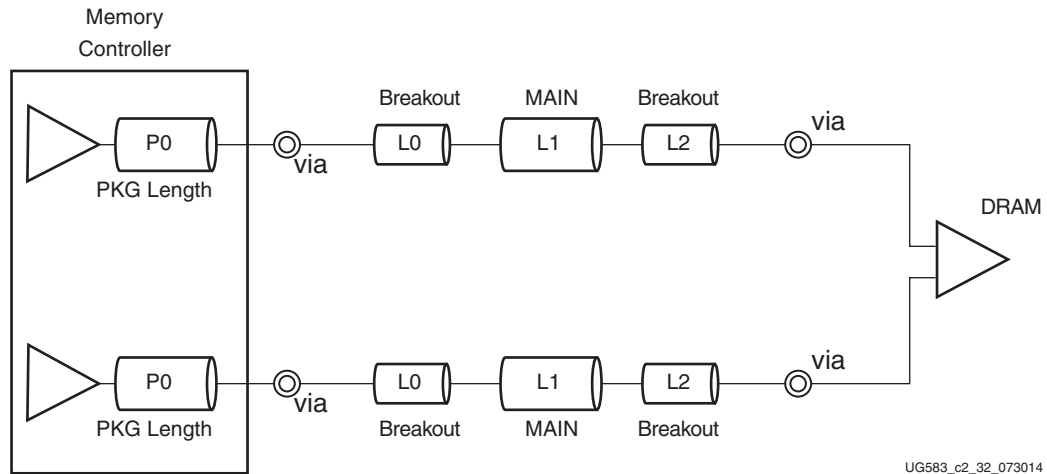
**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).



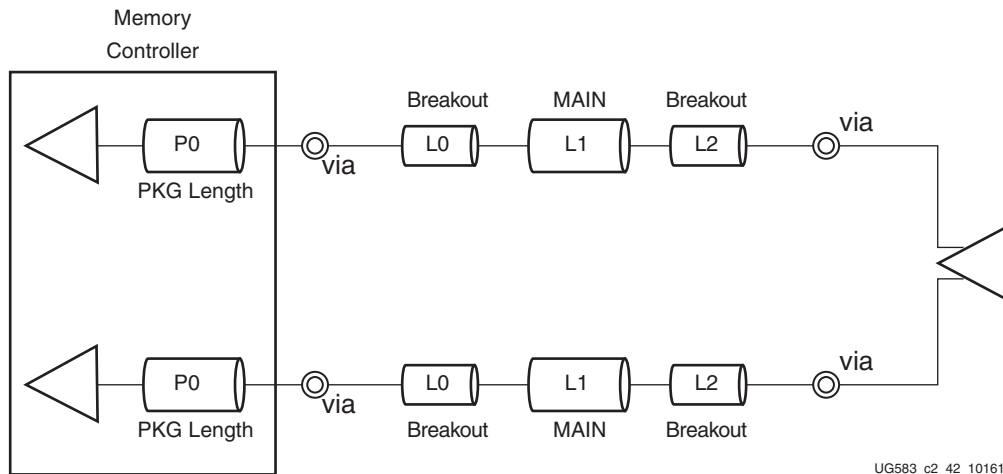
**RLDRAM 3 Memory DK\_P/N and QK\_P/N Differential Point-to-Point Routing**

Figure 2-62 and Figure 2-63 show the RLDRAM 3 memory dk\_p/n and qk\_p/n differential point-to-point routing.



UG583\_c2\_32\_073014

Figure 2-62: **dk\_p/n Point-to-Point Routing**



UG583\_c2\_42\_101614

Figure 2-63: **qk\_p/n Point-to-Point Routing**

Table 2-75 shows the RLDRAM 3 memory impedance, length, and spacing guidelines for dk and qk signals.

Table 2-75: **RLDRAM 3 Memory Impedance, Length, and Spacing Guidelines for dk and qk Signals**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
ck/qk differential impedance $Z_{DIFF}$	86±10%	76±10%	86±10%	Ω

Table 2-75: RLD RAM 3 Memory Impedance, Length, and Spacing Guidelines for dk and qk Signals (Cont'd)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace width (nominal)	4.0	6.0	4.0	mil
Differential trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	mil
Trace length (nominal)	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.25	inches
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**RLDRAM 3 Memory Data Signals Point-to-Point**

Figure 2-64 shows the RLD RAM 3 memory data signals point-to-point routing.

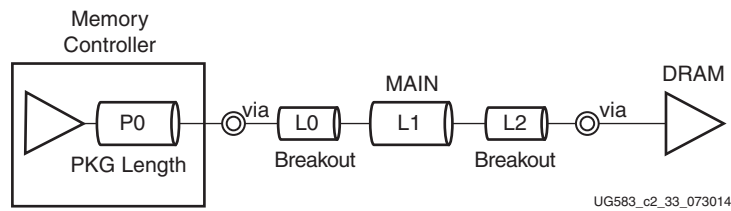


Figure 2-64: Data Signals Point-to-Point Routing

Table 2-76 shows the RLD RAM 3 memory impedance, length, and spacing guidelines for data signals.

Table 2-76: RLD RAM 3 Memory Impedance, Length, and Spacing Guidelines for Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
dq impedance $Z_0$	50±10%	39±10%	50±10%	$\Omega$
Trace width (nominal)	4.0	6.0	4.0	mil
Trace length (nominal)	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.25	inches
Spacing in byte (minimum)	4.0	8.0	4.0	mil
Spacing byte to byte (minimum)	4.0	20	4.0	mil
dq to dk/qk spacing (minimum)	4.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

## RLDRAM 3 Memory Routing Constraints

There are two constraints requirements for each signal group in the RLDRAM 3 memory interface:

- Maximum delay constraints
- Skew constraints

The maximum delay constraints are shown in [Table 2-77](#).

**Table 2-77: RLDRAM 3 Memory Maximum Delay Constraints**

Signal Group	Reference Figure	Maximum Delay Constraints (ps)
address/command	<a href="#">Figure 2-58</a> , <a href="#">Figure 2-59</a>	1017
Data signals P0+L0+L1+L2	<a href="#">Figure 2-64</a>	1017

The skew constraints are listed in [Table 2-78](#).

**Table 2-78: RLDRAM 3 Memory Skew Constraints**

Signals	Skew Constraints (ps)
dq/dm to dk_p/n	±5
dq to qk_p/n	±5
qvld to qk_p/n	±5
dk_p/n to ck_p/n	±5
addr/cmd to ck_p/n	±5
Differential signals p to n	2

**Notes:**

1. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** *FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.*

## PCB Guidelines for QDR II+ SRAM

### Overview

This section provides electrical design guidelines for typical QDR II+ SRAM interfaces from system-level signal integrity simulations for the UltraScale architecture. A two-component QDR II+ 36-bit wide interface using x18 devices is used as a typical design example in this document.



**IMPORTANT:** All guidelines in this section must be followed to achieve the maximum data rates specified for the QDR II+ SRAM interface. It is important to read the [General Memory Routing Guidelines](#) section before continuing with this section.

### QDR II+ SRAM Interface Signal Description

The QDR II+ SRAM interface consists of clock, control, address, command, and data signals as shown in [Table 2-79](#).

Table 2-79: QDR II+ SRAM Interface Signal Description

Signal Name	Description
<b>Clock Signals</b>	
k/k_b[1:0]	Write data/address/command clock
cq/cq_b[1:0]	Read data clock
<b>Control Signals</b>	
zq	Output impedance match. (Do not make a NC or connect to GND.)
odt	On-die termination select
<b>Address Signals</b>	
a[19:0]	Memory address bus
<b>Command Signals</b>	
wps_n	Write port select
rps_n	Read port select
<b>Data Signals</b>	
d[35:0]	Data input bus
bws[3:0]	Byte write select
q[35:0]	Data output bus

**Notes:**

- Actual signal list might vary based on configuration.

## Design Example for Dual QDR II+ SRAM Devices

Figure 2-65 shows a design example for QDR II+ SRAM devices with width expansion.

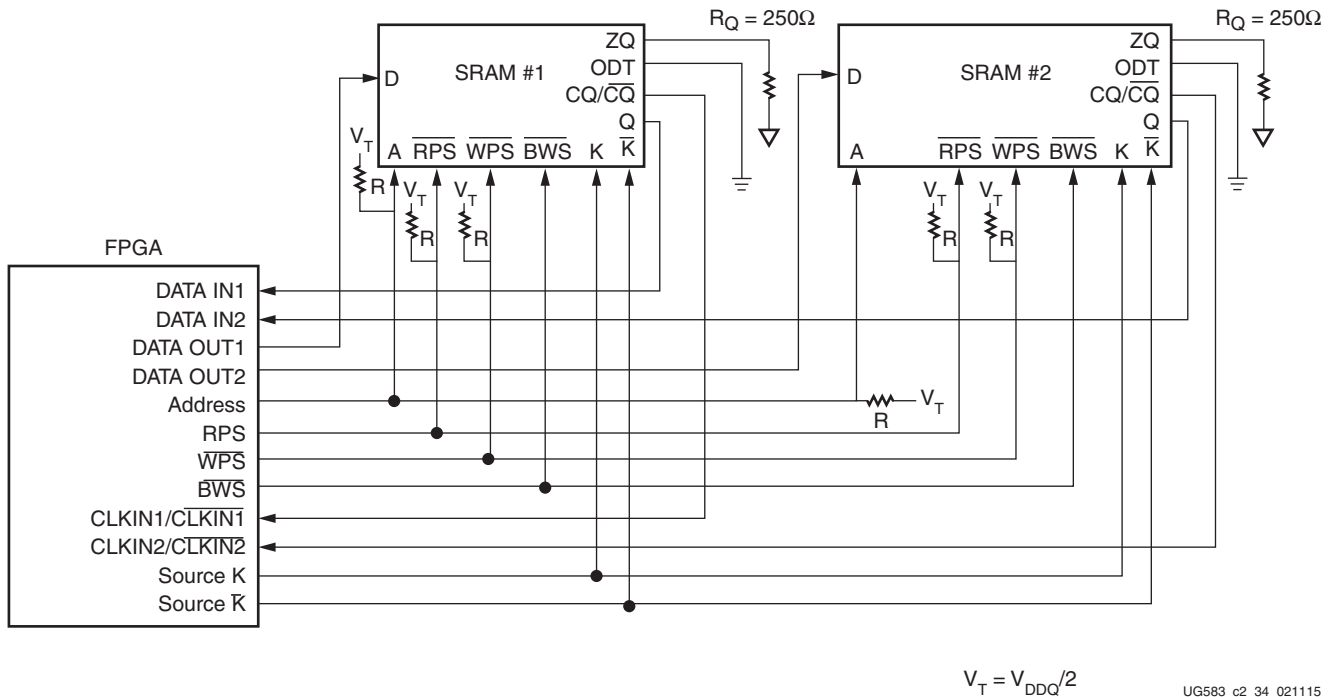


Figure 2-65: Design Example for QDR II+ SRAM Devices with Width Expansion

Figure 2-66 shows a design example for a single QDR II+ SRAM device.

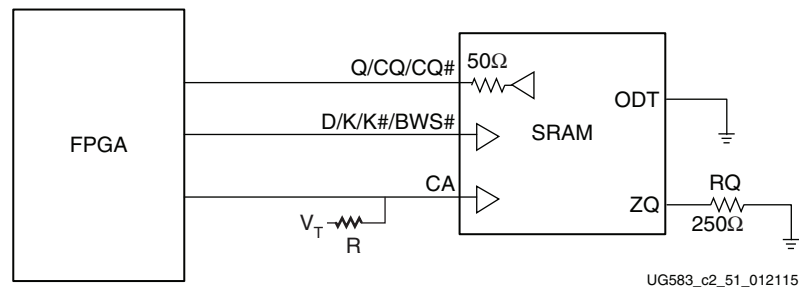


Figure 2-66: Design Example for Single QDR II+ SRAM Device

## QDR II+ SRAM Topology and Routing Guidelines for Fly-by Configuration

### QDR II+ SRAM Address and Command Fly-by Routing and Termination

Fly-by topology is used for address and command signals to achieve the best signal integrity (Figure 2-67).

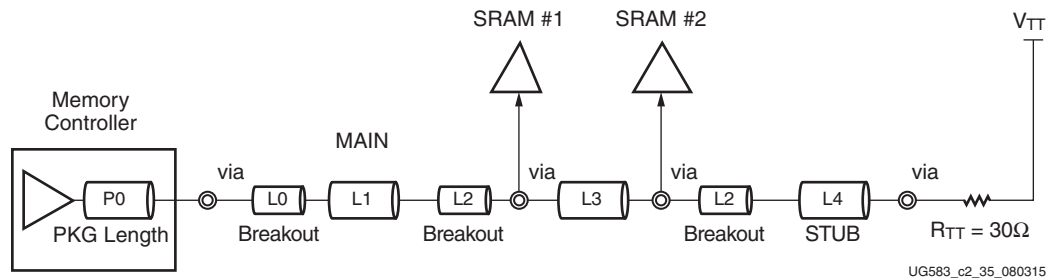


Figure 2-67: Address and Command Fly-by Routing and Termination for QDR II+ SRAM

Figure 2-68 shows the QDR II+ SRAM address and command point-to-point routing and termination.

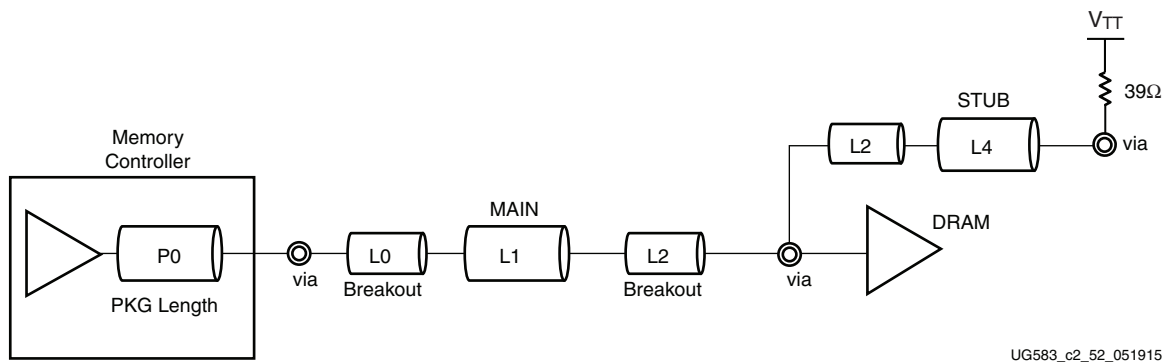


Figure 2-68: Address and Command Point-to-Point Routing and Termination for QDR II+ SRAM

Table 2-80 shows the QDR II+ SRAM impedance, length, and spacing guidelines for address and command signals.

Table 2-80: QDR II+ SRAM Impedance, Length, and Spacing Guidelines for Address and Command Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (SRAM Breakout)	L3 (Main)	L4 (To R <sub>TT</sub> )	Units
Trace type	Stripline	Stripline	Stripline	Stripline	Stripline	–
Single-ended impedance Z <sub>0</sub>	50±10%	39±10%	50±10%	50±10%	36±10%	Ω
Trace width	4.0	6.0	4.0	4.0	7.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~3.0	0.0~0.3	0.5~0.8	0.0~0.4	inches

Table 2-80: QDR II+ SRAM Impedance, Length, and Spacing Guidelines for Address and Command Signals (Cont'd)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (SRAM Breakout)	L3 (Main)	L4 (To R <sub>TT</sub> )	Units
Spacing in address, command, and control signals (minimum)	4.0	8.0	4.0	8.0	8.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	20	20	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count	4					–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

## Additional QDR II+ SRAM Routing Guidelines

### QDR II+ SRAM Topology and Routing Guidelines for T-Branch Configuration

Figure 2-69 and Table 2-81 show the QDR II+ SRAM address and command T-branch routing, termination topology, and routing guidelines.

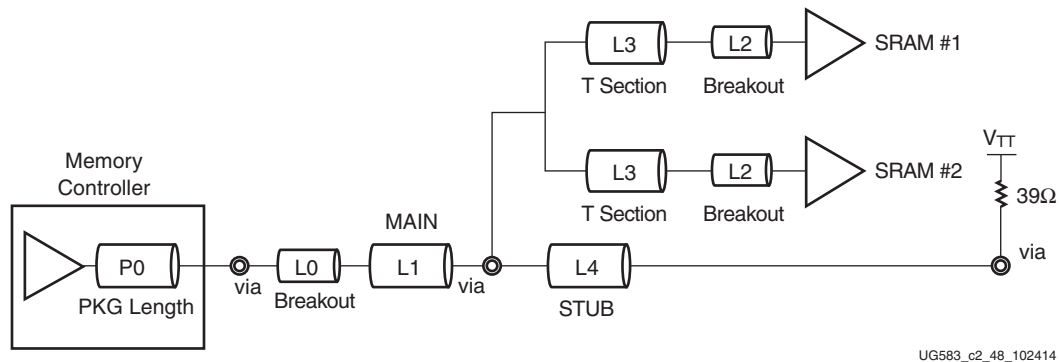


Figure 2-69: Address and Command T-Branch Routing and Termination for QDR II+ SRAM

Table 2-81: Address and Command T-Branch Routing and Termination for QDR II+ SRAM

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (DRAM to DRAM)	L4 (to RTT)	Units
Trace type	Stripline	Stripline	Microstrip	Microstrip	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	60±10%	60±10%	39±10%	Ω
Routing layer	Upper/Lower	Upper/Lower	Surface	Surface	Upper/Lower	
Trace width	4.0	6.0	4.0	4.0	6.0	mil

Table 2-81: Address and Command T-Branch Routing and Termination for QDR II+ SRAM (Cont'd)

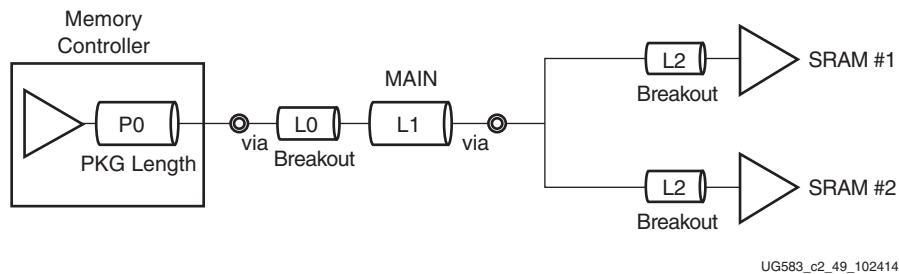
Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	L3 (DRAM to DRAM)	L4 (to RTT)	Units
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	<3.0	<0.4	<0.7	<0.5	inches
Spacing in address, command, control signals (minimum)	4.0	12.0	4.0	8.0	8.0	mil
Spacing to clock signals (minimum)	8.0	12.0	8.0	12.0	12.0	mil
Spacing to other group signals (minimum)	8.0	30	30	30	30	mil
Maximum PCB via count	3					–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

**QDR II+ SRAM Clock (K, K\_B, and BWS) T-Branch Routing and Termination**

It is preferred to have dedicated K and K\_B for each SRAM, but if that is not possible, the T-branch topology is an option. Figure 2-70 and Table 2-82 define the topology and routing guidelines for QDR II+ SRAM clock signals.



UG583\_c2\_49\_102414

Figure 2-70: Clock (K, K\_B) and BWS T-Branch Routing and Termination for QDR II+ SRAM

Table 2-82: Impedance, Length, and Spacing Guidelines for Clock Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Microstrip	–
Single-ended impedance $Z_0$	50±10%	39±10%	60±10%	$\Omega$
Routing layers	Upper/Lower	Upper/Lower	Upper/Lower	
Trace width	4.0	6.0	4.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	<4.0	<0.7	inches
Spacing in byte (minimum)	4.0	12.0	4.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil



Table 2-82: Impedance, Length, and Spacing Guidelines for Clock Signals (Cont'd)

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Maximum PCB via count	2			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

**QDR II+ SRAM Clock and Data Signals (d/k/k\_b) Point-to-Point Routing**

Figure 2-71 shows the QDR II+ SRAM clock and data signals point-to-point routing.

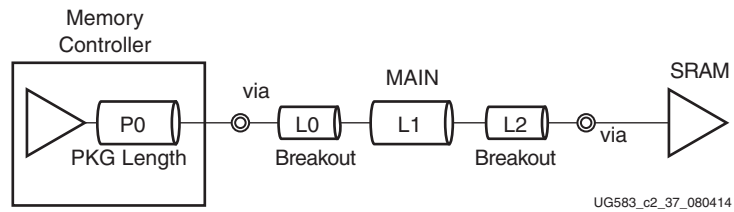


Figure 2-71: Clock and Data Signals (d/k/k\_b) Point-to-Point Routing

**QDR II+ SRAM Clock and Data Signals (q/cq/cq\_b) Point-to-Point Routing**

Figure 2-72 shows the QDR II+ SRAM clock and data signals point-to-point routing.

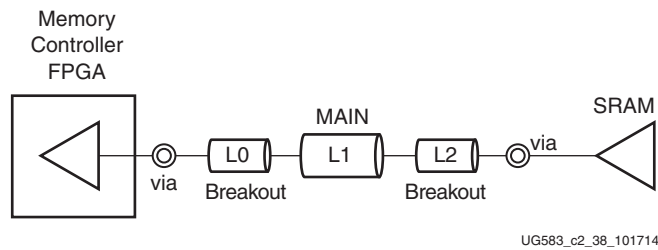


Figure 2-72: Clock and Data Signals (q/cq/cq\_b) Point-to-Point Routing

Table 2-83 shows the QDR II+ SRAM impedance, length, and spacing guidelines for clock and data signals.

Table 2-83: QDR II+ SRAM Impedance, Length, and Spacing Guidelines for Clock and Data Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (SRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
d and q single-ended impedance $Z_0$	50±10%	39±10%	50±10%	$\Omega$
Trace width (nominal)	4.0	6.0	4.0	mil
Trace length (nominal)	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.5	inches

**Table 2-83: QDR II+ SRAM Impedance, Length, and Spacing Guidelines for Clock and Data Signals (Cont'd)**

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (SRAM Breakout)	Units
Spacing in byte (minimum)	4.0	8.0	4.0	mil
Spacing byte to byte (minimum)	4.0	20	4.0	mil
CQ_P/N to other spacing	4.0	12.0	4.0	mil
K_P/N to other spacing	4.0	12.0	4.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

## QDR II+ SRAM Routing Constraints

There are two constraints requirements for each signal group in the QDR II+ SRAM interface:

- Maximum length/delay constraints
- Skew constraints

The maximum length/delay constraints are shown in [Table 2-84](#).

**Table 2-84: QDR II+ SRAM Maximum Length/Delay Constraints**

Signal Group	Reference Figure	Maximum Length/Delay Constraints
address/command	<a href="#">Figure 2-67</a>	6.0 inches 1017 ps
Data signals P0+L0+L1+L2	<a href="#">Figure 2-72</a>	6.0 inches 1017 ps



**IMPORTANT:** Address and data signals must match lengths to each respective QDR II+ SRAM device.

The skew constraints are listed in [Table 2-85](#).

**Table 2-85: QDR II+ SRAM Skew Constraints**

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
Data to clock (d to k, q to cq)	±6	±35
Data	±5	±29
T legs	±2	±12
CMD/ADDR (CA)	±6	±35

Table 2-85: QDR II+ SRAM Skew Constraints (Cont'd)

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
CMD/ADDR to CLK (CLK point-to-point and CA fly-by)	±6	±35
CMD/ADDR to CLK (CLK t-branch and CA fly-by)	±34	±200
CMD/ADDR to CLK (CLK t-branch and CA t-branch)	±6	±35
k to k_b	2	12
q to q_b	2	12

**Notes:**

1. For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.

## PCB Guidelines for QDR-IV SRAM

### Overview

This section provides electrical design guidelines for typical QDR-IV SRAM interfaces from system-level signal integrity simulations for the UltraScale architecture. A one-component QDR-IV 36-bit wide interface is used as a typical design example in this document.



**IMPORTANT:** All guidelines in this section must be followed to achieve the maximum data rates specified for the QDR-IV SRAM interface. It is important to read the [General Memory Routing Guidelines](#) section before continuing with this section.

## QDR-IV SRAM Interface Signal Description

The QDR-IV SRAM interface consists of clock, control, address, command, and data signals as shown in Table 2-86.

Table 2-86: QDR-IV SRAM Interface Signal Description

Signal Name	Description
<b>Clock Signals</b>	
CK/CK#	Address/command clock
DK/DK#	Write data clock
QK/QK#	Read data clock
<b>Control Signals</b>	
ZQ	Output impedance match
<b>Address Signals</b>	
a[19:0]	Memory address bus (20:0 if using x18 data widths)
<b>Command Signals</b>	
LD#	Synchronous load input
<b>Data Signals</b>	
DQ[35:0]	Data bus

**Notes:**

1. Actual signal list might vary based on configuration.

## Design Example for Dual QDR-IV SRAM Devices

Figure 2-73 shows a design example for dual QDR-IV SRAM devices.

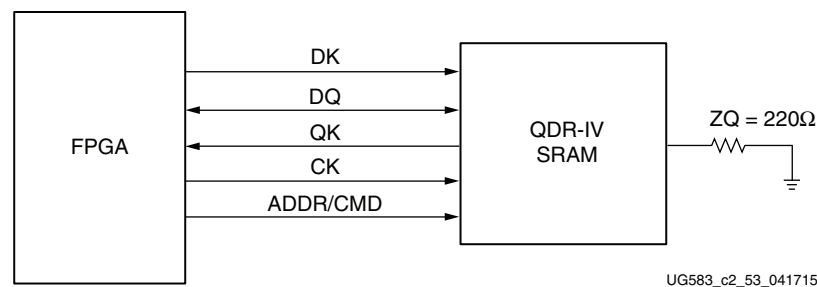


Figure 2-73: Design Example for QDR-IV SRAM Devices

## QDR-IV Topology and Routing Guidelines



**IMPORTANT:** To achieve the specified performance, all rules in the tables must be followed. This includes but is not limited to the trace type, width, spacing, impedance, and via count.

### QDR-IV SRAM DQ/Address/Command Routing

Figure 2-74 and Table 2-87 define topology and routing guidelines for QDR-IV SRAM DQ, address, and command signals.

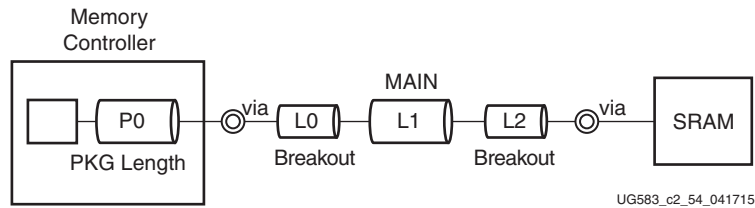


Figure 2-74: DQ/Address/Command Routing for QDR-IV SRAM

Table 2-87: QDR-IV SRAM Impedance, Length, and Spacing Guidelines for DQ/Address/Command Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Single-ended impedance $Z_0$	50±10%	39±10%	50±10%	$\Omega$
Trace width	4.0	6.0	4.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~4.0	0.0~0.1	inches
Spacing in address, command, and control signals (minimum)	4.0	8.0	4.0	mil
Spacing to clock signals (minimum)	8.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil
Maximum PCB via count	4 (addr/cmd) 2 DQ			–

**Notes:**

1. See item 2 in [General Memory Routing Guidelines](#).

### QDR-IV SRAM CK/DK/QK Routing and Termination

Figure 2-75, Figure 2-76, and Table 2-88 define topology and routing guidelines for QDR-IV SRAM clock signals.

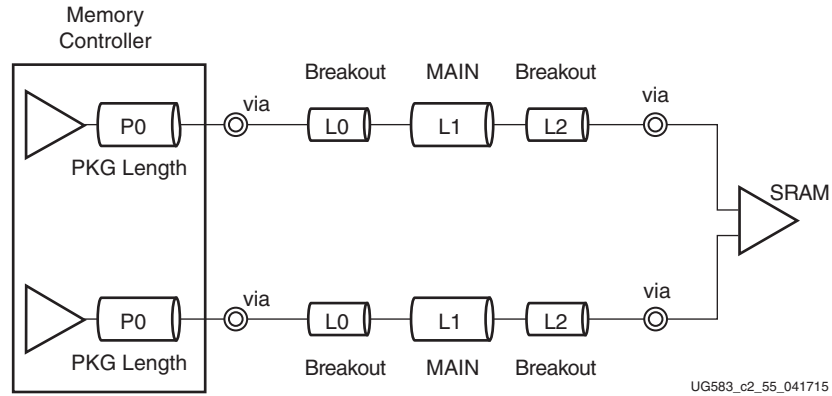


Figure 2-75: DK and CK Routing for QDR-IV SRAM

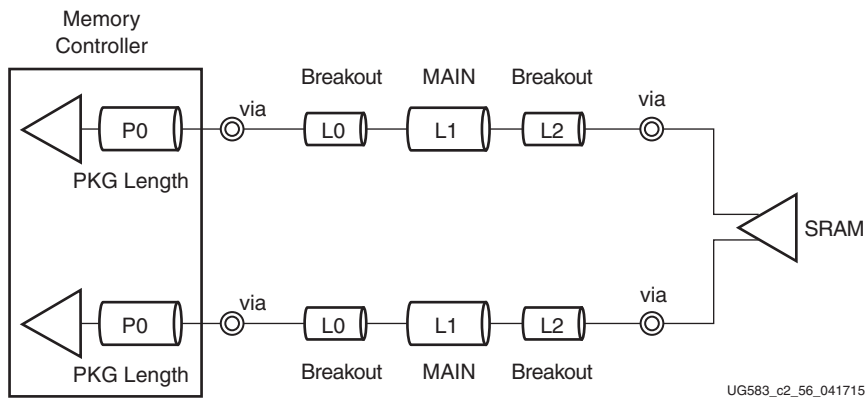


Figure 2-76: QK Routing for QDR-IV SRAM

Table 2-88: QDR-IV SRAM Impedance, Length, and Spacing Guidelines for CK/DK/QK Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Trace type	Stripline	Stripline	Stripline	–
Clock differential impedance $Z_{DIFF}$	$86 \pm 10\%$	$76 \pm 10\%$	$86 \pm 10\%$	$\Omega$
Trace width/space/width	4.0/4.0/4.0	6.0/6.0/6.0	4.0/4.0/4.0	mil
Trace length	0.0~0.8/1.2 <sup>(1)</sup>	0.0~3.0	0.0~0.1	inches
Spacing in address, command, and control signals (minimum)	8.0	20	8.0	mil
Spacing to other group signals (minimum)	8.0	30	30	mil

Table 2-88: QDR-IV SRAM Impedance, Length, and Spacing Guidelines for CK/DK/QK Signals

Parameter	L0 (Device Breakout)	L1 (Main PCB)	L2 (DRAM Breakout)	Units
Maximum PCB via count per signal	2			–

**Notes:**

- See item 2 in [General Memory Routing Guidelines](#).

## QDR-IV SRAM Routing Constraints

There are two constraints requirements for each signal group in the QDR-IV SRAM interface:

- Maximum length/delay constraints
- Skew constraints

The maximum length/delay constraints are shown in [Table 2-89](#).

Table 2-89: QDR-IV SRAM Maximum Length/Delay Constraints

Signal Group	Reference Figure	Maximum Length/Delay Constraints
Address/command P0+L0+L1+L2	<a href="#">Figure 2-74</a>	6.0 inches 1017 ps
Data Signals P0+L0+L1+L2	<a href="#">Figure 2-74</a>	6.0 inches 1017 ps

The skew constraints are listed in [Table 2-90](#).

Table 2-90: QDR-IV SRAM Skew Constraints

Signal Group	Skew Constraints (ps)	Skew Constraints (mil)
DQ to clock (DQ to DK, DQ to QK)	±5	±29
Address/command to CK	±6	±35
DQ (slowest to fastest)	2	12
Address/command (slowest to fastest)	2	12
Clock to clock# (CK/DK/QK)	2	12
CK to DK	50	295
CK to QK	50	295

**Notes:**

- For skew specifications, refer to the [General Memory Routing Guidelines](#) items 3–8.



**IMPORTANT:** FPGA package flight times must be included in both total length constraints and skew constraints. When minimum and maximum values are available for the package delay, use the midrange between the minimum and maximum values. Memory device package flight times do not need to be factored in because their variances have been accounted for in these guidelines.

# PCB Guidelines for Zynq UltraScale+ RFSocCs

---

## PDN Guidelines for RFSocC Digital Power Rails

Zynq® UltraScale+™ RFSocCs contain new digital and analog circuitry, with dedicated power rails for each block. This section focuses on the PDN guidelines for the digital power rails in RFSocCs, including the soft-decision forward error correction (SD-FEC) power rail  $V_{CCSDFEC}$ .

### Power Delivery to the Programmable Logic Voltage Rails

The guidelines in this document are based on the following use case assumptions.

- 80% of CLB LUTs and registers running at 245 MHz
- 80% of block RAMs running at 491 MHz
- 80% of UltraRAMs running at 200 MHz
- 80% of DSPs running at 491 MHz
- 100% of MMCM and PLL running at 500 MHz
- 100% of GTY transceiver usage
- 8.02A step-load current on  $V_{CCSDFEC}$

### $V_{CCINT}$ , $V_{CCINT\_IO}$ and $V_{CCBRAM}$ Tied Together (-1E, -1I, -2I, -3E)

For the -1E, -1I, -2I and -3E speed grades, Xilinx recommends connecting  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ , and  $V_{CCBRAM}$  together because all run at the same voltage (0.85V or 0.90V (-3 only)). PCB plane areas are also easier to draw, because  $V_{CCINT}/V_{CCINT\_IO}$  and  $V_{CCBRAM}$  can share the same plane design. Plane design for  $V_{CCINT}/V_{CCINT\_IO}/V_{CCBRAM}$  should be sized in such a way as to reliably provide the required amount of current/power with minimal IR drop to ensure that the voltage rails meet the data sheet specifications.



### Separate $V_{CCINT}$ and $V_{CCBRAM}/V_{CCINT\_IO}$ (-1LI, -2LE)

For the -1LI, -2LI, and -2LE speed grades,  $V_{CCINT}$  can operate at 0.72V versus 0.85V/0.90V for  $V_{CCBRAM}/V_{CCINT\_IO}$ . It is thus necessary to separate those rails onto different planes. Plane design for  $V_{CCINT}$  and  $V_{CCBRAM}/V_{CCINT\_IO}$  should be sized in such a way as to provide the required amount of current/power with minimal IR drop to ensure that the voltage rails meet the data sheet specifications.

### $V_{CCSDFEC}$

It is required to connect the  $V_{CCSDFEC}$  rail to the plane that contains  $V_{CCBRAM}$ .  $V_{CCSDFEC}$  and  $V_{CCBRAM}$  should have separate planes under the RFSoc, which are then merged when outside of the RFSoc shadow. [Figure 3-1](#) shows the recommended plane methodology. Decoupling capacitors for the  $V_{CCSDFEC}$  are described in [PCB Decoupling Recommendations](#).

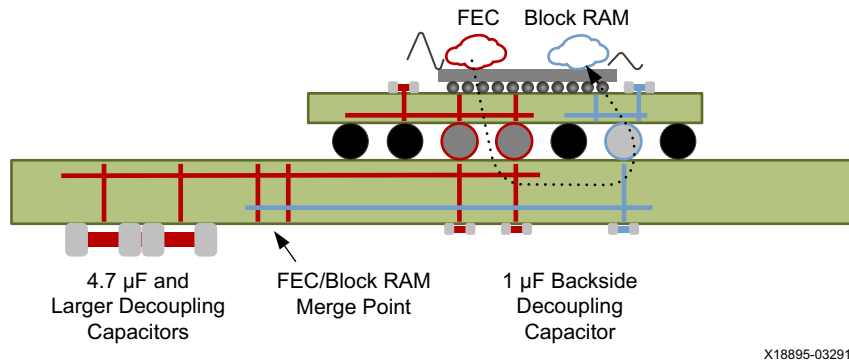


Figure 3-1: Recommended Plane Connection for  $V_{CCSDFEC}$  and  $V_{CCBRAM}$

### $V_{CCSDFEC}$ and Migration

To enable seamless migration between RFSoc devices, some specific die/package combinations like the XCZU25DR-FFVE1156, XCZU27DR-FFVE1156, XCZU25DR-FFVG1517, and XCZU27DR-FFVG1517 have pins labeled  $V_{CCSDFEC}$  despite there not being any SD-FEC circuitry in each device. To maintain compatibility with the XCZU28DR device, connect the  $V_{CCSDFEC}$  pins to  $V_{CCBRAM}$ . If SD-FEC circuitry is not going to be used, connect the  $V_{CCSDFEC}$  pins to  $V_{CCBRAM}$ . Extra decoupling capacitors are not needed in regards to  $V_{CCSDFEC}$ .

## PCB Decoupling Recommendations

Depending on the speed grade used, the decoupling guidelines differ because the  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ ,  $V_{CCBRAM}$ , and  $V_{CCSDFEC}$  (as applicable) rails are either tied together or separate.

Table 3-1 shows the PCB decoupling guidelines for the programmable logic power rails for non-L speed grade RFSocS. Table 3-2 shows the corresponding PCB decoupling guidelines for L speed grade devices. Table 3-3 shows the additional decoupling guidelines for the  $V_{CCSDFEC}$  and  $V_{CCBRAM}/V_{CCINT\_IO}$  rails when  $V_{CCSDFEC}$  is utilized.

Table 3-1: Programmable Logic Rail Decoupling Guidelines for Non-L Devices

RFSoc	$V_{CCINT}/V_{CCBRAM}/V_{CCINT\_IO}$			$V_{CCAUX}/V_{CCAUX\_IO}$		$V_{CCO}^{(1)}$	
	680 $\mu$ F	100 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F
XCZU21DR-FFVD1156	2	4	7	1	2	1	2
XCZU25DR-FFVE1156 XCZU25DR-FSVE1156	2	3	5	1	1	1	2
XCZU25DR-FFVG1517 XCZU25DR-FSVG1517	2	3	5	1	2	1	2
XCZU27DR-FFVE1156 XCZU27DR-FSVE1156	2	4	6	1	1	1	2
XCZU27DR-FFVG1517 XCZU27DR-FSVG1517	2	4	6	1	2	1	2
XCZU28DR-FFVE1156 XCZU28DR-FSVE1156	2	4	6	1	1	1	2
XCZU28DR-FFVG1517 XCZU28DR-FSVG1517	2	4	6	1	2	1	2
XCZU29DR-FFVF1760 XCZU29DR-FSVF1760	2	4	7	1	2	1	2

### Notes:

1. One 47  $\mu$ F capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
2. See Table 3-3 and Table 3-4 for extra  $V_{CCBRAM}/V_{CCINT\_IO}$  decoupling capacitors for when  $V_{CCSDFEC}$  is utilized.

Table 3-2: Programmable Logic Rail Decoupling Guidelines for L Devices

RFSoc	$V_{CCINT}$			$V_{CCBRAM}/V_{CCINT\_IO}$		$V_{CCAUX}/V_{CCAUX\_IO}$		$V_{CCO}^{(1)}$	
	680 $\mu$ F	100 $\mu$ F	4.7 $\mu$ F	4.7 $\mu$ F	0.47 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F	47 $\mu$ F	4.7 $\mu$ F
XCZU21DR-FFVD1156	2	4	6	1	6	1	2	1	2
XCZU25DR-FFVE1156 XCZU25DR-FSVE1156	1	2	4	1	4	1	2	1	2
XCZU25DR-FFVG1517 XCZU25DR-FSVG1517	1	2	4	1	4	1	2	1	2

Table 3-2: Programmable Logic Rail Decoupling Guidelines for L Devices (Cont'd)

RFSoc	V <sub>CCINT</sub>			V <sub>CCBRAM</sub> /V <sub>CCINT_IO</sub>		V <sub>CCAUX</sub> /V <sub>CCAUX_IO</sub>		V <sub>CCO</sub> <sup>(1)</sup>	
	680 μF	100 μF	4.7 μF	4.7 μF	0.47 μF	47 μF	4.7 μF	47 μF	4.7 μF
XCZU27DR-FFVE1156 XCZU27DR-FSVE1156	2	3	5	1	5	1	1	1	2
XCZU27DR-FFVG1517 XCZU27DR-FSVG1517	2	3	5	1	5	1	2	1	2
XCZU28DR-FFVE1156 XCZU28DR-FSVE1156	2	3	5	1	5	1	1	1	2
XCZU28DR-FFVG1517 XCZU28DR-FSVG1517	2	3	5	1	5	1	2	1	2
XCZU29DR-FFVF1760 XCZU29DR-FSVF1760	2	3	6	1	6	1	3	1	2

**Notes:**

1. One 47 μF capacitor is required for up to four HP/HR I/O banks when powered by the same voltage.
2. See Table 3-3 and Table 3-4 for extra V<sub>CCBRAM</sub>/V<sub>CCINT\_IO</sub> decoupling capacitors for when V<sub>CCSDFEC</sub> is utilized.

 Table 3-3: Decoupling Capacitor Quantities for V<sub>CCSDFEC</sub>, plus additional capacitors for V<sub>CCBRAM</sub>/V<sub>CCINT\_IO</sub>

V <sub>CCSDFEC</sub>			V <sub>CCBRAM</sub> /V <sub>CCINT_IO</sub>
10 μF 0805	4.7 μF 0603	1.0 μF 0402	1.0 μF 0402
3	6	8	4

**Notes:**

1. This table only applies for when V<sub>CCSDFEC</sub> is utilized.
2. V<sub>CCBRAM</sub>/V<sub>CCINT\_IO</sub> capacitors are in addition to those in Table 3-1 and Table 3-2.
3. See Table 3-4 for connection rules for these capacitors.

Table 3-4 lists the placement rules for the capacitors in Table 3-3. Refer to Figure 3-1 for visual reference.

 Table 3-4: Placement Rules for Additional V<sub>CCSDFEC</sub>/V<sub>CCBRAM</sub>/V<sub>CCINT\_IO</sub> Decoupling Capacitors

Capacitor	Rule
10 μF 0805	All three next to FPGA footprint near V <sub>CCSDFEC</sub> and V <sub>CCBRAM</sub> /V <sub>CCINT_IO</sub> merge point.
4.7 μF 0603	All six next to FPGA footprint near V <sub>CCSDFEC</sub> and V <sub>CCBRAM</sub> /V <sub>CCINT_IO</sub> merge point.
1.0 μF 0402	<ul style="list-style-type: none"> <li>• Eight connected to V<sub>CCSDFEC</sub> plane under FPGA shadow.</li> <li>• Two connected under V<sub>CCBRAM</sub> ball area between via plating holes.</li> <li>• Two connected under V<sub>CCINT_IO</sub> ball area between via plating holes.</li> </ul>

## Power Delivery to the Processing System Voltage Rails

Xilinx recommends following the processing system decoupling guidelines shown in [Table 1-10](#).

## Power Delivery to the PS-GTR and GTY Transceivers

PCB guidelines for the PS-GTR transceivers can be found in [PS-GTR Transceiver Interfaces](#), page 179. The guidelines include decoupling capacitors and a design checklist.

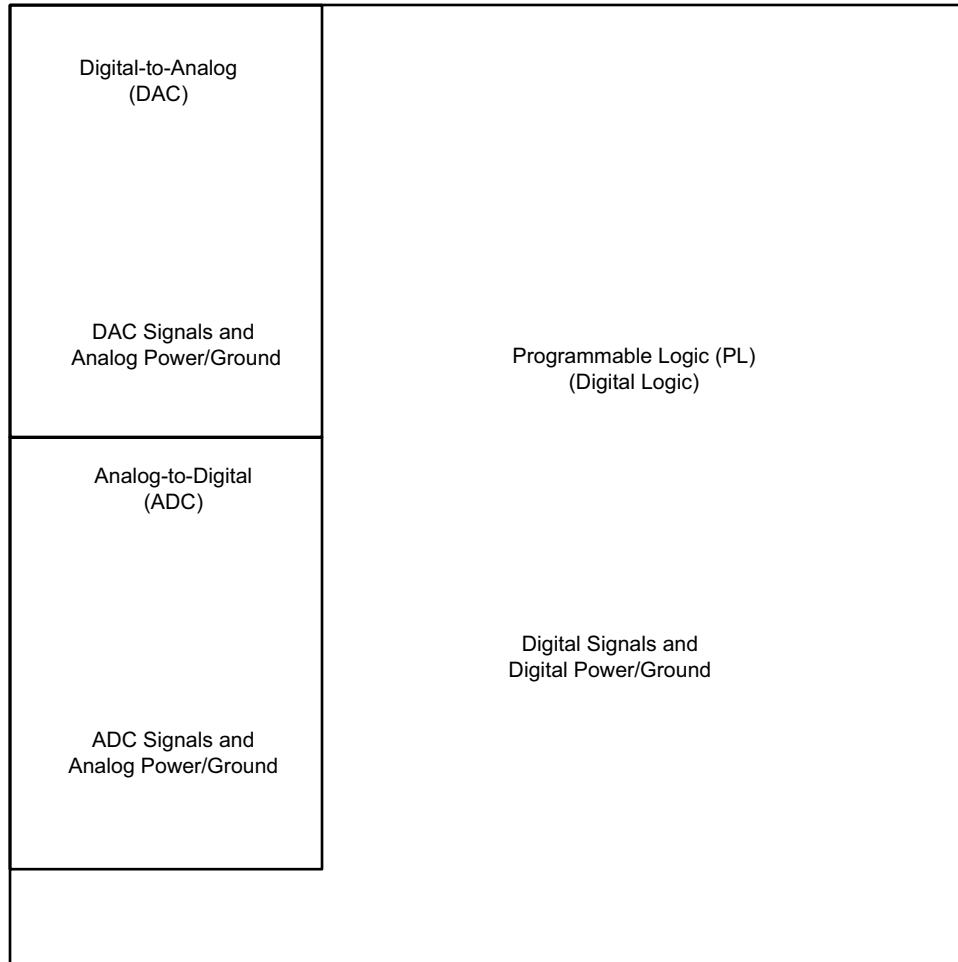
PCB design guidelines for GTY transceivers can be found in the Board Design Guidelines chapter of *UltraScale Architecture GTY Transceivers User Guide* (UG578) [\[Ref 7\]](#). The guidelines include decoupling capacitors and a design checklist.

## ADC and DAC PCB Guidelines

This section provides routing guidelines for the Zynq® UltraScale+™ RFSocS with analog-to-digital (ADC) and digital-to-analog (DAC) functionality.

### Zynq UltraScale+ RFSoc Device Organization and PCB Design Overview

To assist with keeping sensitive analog circuitry as separate as possible from digital circuitry, the Zynq UltraScale+ RFSoc is sectioned on the die such that the Analog-to-Digital (ADC) and Digital-to-Analog (DAC) portions are partitioned away from each other and from digital logic. Figure 3-2 shows the floorplan of the XCZU28DRFFVE1156 RFSoc with the ADC and DAC signals and pins located separately from the digital programmable logic (PL).



X18569-121216

Figure 3-2: XCZU28DR-FFVE1156 Floorplan

The overall goal of a successful PCB design is to keep the analog portions as isolated as possible from both outside electromagnetic (EM) interference and potential on-chip EM interference by the digital logic and analog circuits.

## Analog Ground to Digital Ground Connection

While the analog and digital grounds are isolated on the Zynq UltraScale+ RFSoc, eventually they must connect to each other on the PCB. The recommended method to ensure that the analog ground planes on the Zynq UltraScale+ RFSoc connect to the digital ground plane with as little noise transfer as possible is to use one common ground plane.

When connecting analog ground and digital ground as one common plane, adhere to the following guidelines:

- Do not add any slots in the ground plane.
- Place digital voltage regulator modules (VRMs) and capacitors near the digital power rails.
- Place RF VRMs and capacitors near RF power rails.
- Add ground stitching around groups of RF signals to 118 mil (3.0 mm) pitch. See [Figure 3-9](#), [Figure 3-11](#) and [Figure 3-12](#).

## Choosing the Appropriate Balun

A balun is typically used to interface to the differential input/output of the data converter for converting to a single-ended signal. The bandwidth of the chosen balun should be greater than the band of interest for the user application. This ensures that the measured RFSoc bandwidth is not limited by the balun bandwidth. [Table 3-2](#) lists the 3 dB bandwidth for each of the data converters and the sampling clocks.

**Table 3-5: Data Converter Bandwidth**

RFSoc Signal Type	Bandwidth (GHz)
ADC input	0–4
DAC output	0–4
ADC clock input	4
DAC clock input	6.4

The Zynq UltraScale+ RFSoc is designed to ensure that all the data converter inputs and outputs along with the clock inputs see a 100Ω DC resistance under normal operating conditions. The purpose of the balun is to convert the single-ended signal with a typical characteristic impedance of 50Ω/75Ω to a differential 100Ω system to meet the RFSoc requirements. Customers can either opt for a wire-bound or a stripline balun depending on the application requirements. The wire-bound baluns are generally used to generate differential signals for low-frequency applications on the order of hundreds of kHz to hundreds of MHz while the RF stripline baluns are more suited for GHz applications. The RF

stripline baluns tend to exhibit a band-pass filter characteristic and thus care should be taken to pick a balun with the correct bandwidth requirements that matches the application requirements.

Customers should pay close attention to the balun specification recommendations in [Table 3-7](#) over the application bandwidth of interest when picking a balun. Three parts that can be considered are:

- Anaren BD1631J50100AHF (approximate bandwidth 1.5 GHz – 3.5 GHz)
- Anaren BD3150N50100AHF (wireless applications)
- Mini Circuit TC1-1-13M-75X (cable applications)

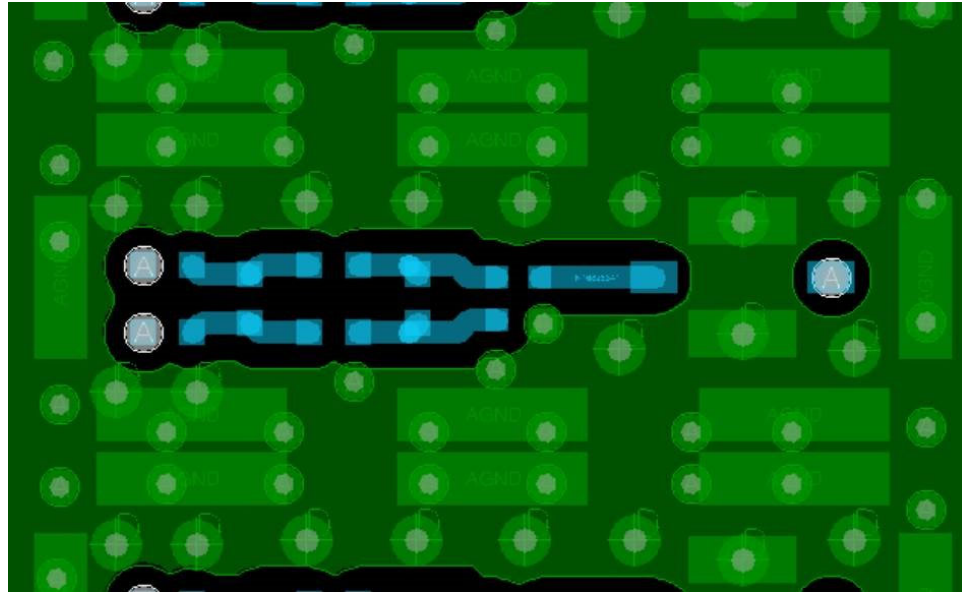
**Table 3-6: Balun Specification Recommendations**

Specification	Typical Result
Impedance ratio	2 or 1
Bandwidth	Application specific
Insertion loss	-1 dB or better
Return loss	-15 dB or better
Common-mode rejection ratio (CMRR)	>30 dB <sup>(1)</sup>
Amplitude imbalance	<~0.5 dB
Phase imbalance	<~1.5°

**Notes:**

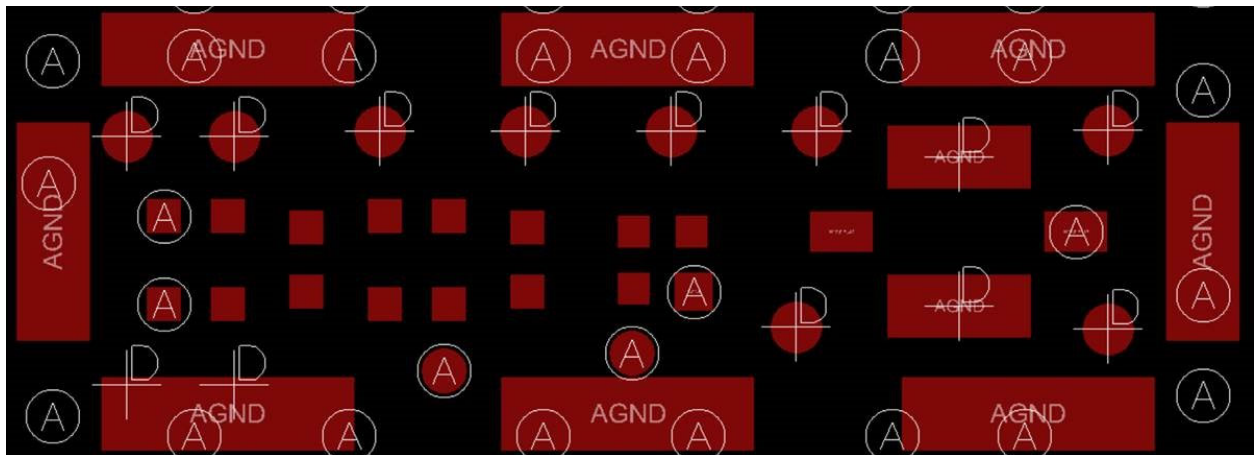
1. CMRR can be relaxed if the system is designed to avoid second harmonic distortion (HD2) by frequency planning.

The RF signal chain for each DAC and ADC, which consists of differential DC blocks, differential Pi attenuators, baluns, and filters, fits well on a 4 mm pitch including the RF shields which cover the RF signal chain components. The RF shield should be electrically attached to ground vias in the PCB at 3 mm spacing (same spacing as the guard stitching vias for signal traces). Failure to adhere to this recommendation results in violating crosstalk guidelines. While the attachment pitch is nominally 3 mm, each attachment point is allowed a tolerance of  $\pm 1$  mm from its nominal position. To help facilitate attaching the RF shields to the PCB in a tight space, Xilinx recommends coating the inside surface of the shield with liquid photo-imageable (LPI) solder mask. This avoids potential shorting to components located inside the RF shield. If soldering the RF shield to the PCB, Xilinx recommends plating the shield with either bright tin, solder plating, or other plating material that can bond to the solder. The ground vias to which the RF shield attaches should be through-hole vias connected to all inner ground plane layers. An example of this layout can be seen in [Figure 3-3](#).



X21264-073018

Figure 3-3: Balun Isolation with GND Stitching and GND Plane



X21265-073018

Figure 3-4: RF Shield Solder Mask

## Recommended Clocking Options

The ADC and the DAC both have options to take in a direct RF sampling clock or a lower frequency reference clock to drive the internal PLLs. The built-in internal PLLs offer excellent phase noise and spurious performance and are likely to satisfy the requirements from the user application. The detailed performance of the PLL can be found in the *Zynq UltraScale+ RFSoc Data Sheet: DC and AC Switching Characteristics (DS926)* [Ref 5] along with requirements for the reference clocks. Using the internal PLLs offers the most economical and highly integrated solution. In the case where very specific sampling clocking

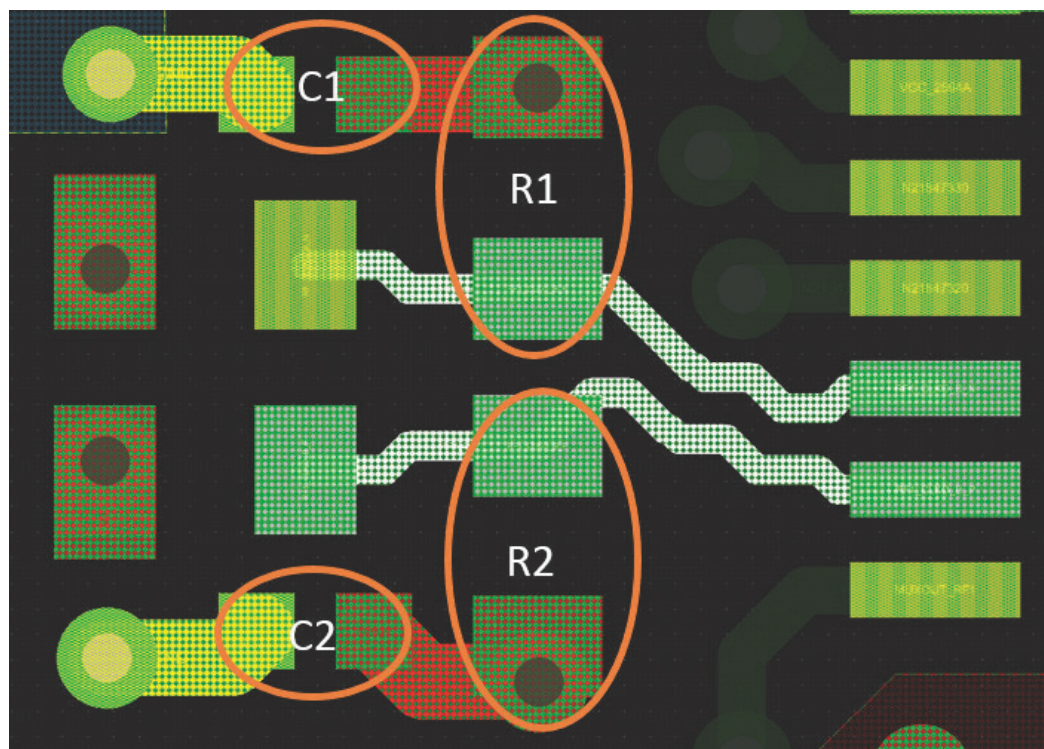


performance is required, an external RF PLL clock can be provided. Devices such as the Texas Instruments LMX259x family can be considered.

### ***RF PLL Placement and Routing***

It is recommended to place the RF PLLs close to the RFSoc device. The DAC RF PLLs should be placed next to the DAC side of the RFSoc while the ADC RF PLLs should be placed next to the ADC side of the RFSoc device. Simulation analysis should be performed to ensure that the clock input amplitude at the RFSoc device meets the specifications in the data sheet.

For PLLs that require pull-up resistors, ensure the resistor and decoupling capacitor layouts are symmetrical with respect to P and N. It is recommended to put the pull-up resistors and the decoupling capacitors for the buffer on the same layer as the RF PLL, typically layer one. [Figure 3-5](#) shows a snapshot of a typical layout implementation.



X20580-032618

Figure 3-5: PLL Layout

### **Analog and Clock Pair Routing**

Keeping the analog and clock signals as free as possible from any noise sources is essential, whether they be from the outside environment, or on the PCB itself. The following sections show various specifications and recommendations that result in optimal board and system performance.

Table 3-7 and Table 3-8 summarize the signal integrity specifications for DAC/ADC and clock pairs (including reference clocks). Refer to the sections below for more details regarding each type of specification.

**Table 3-7: Signal Integrity Specifications for DAC/ADC Pairs**

Signal Integrity Metrics	Limit (dB)	Measurement Frequency (GHz)
Differential NEXT @ RFSoc	-75	4
Differential Insertion Loss	-1.5	4
Differential Return Loss	-23 using simulation; -18 after taking into account $\pm 10\%$ variation to nominal trace impedance due to manufacturing tolerance	4
Common Mode NEXT @ RFSoc	-65	4
Common Mode Return Loss	-15	4
P & N Balance	Total = 2000 fs; 1000 fs using simulation, with an additional 1.0 ps to account for manufacturing tolerances	4

**Notes:**

1. An impedance variation of  $\pm 10\%$  itself equals -20 dB.

**Table 3-8: Signal Integrity Specifications for DAC/ADC Clocks and Reference Clocks**

Signal Integrity Metrics	Limit (dB)	Measurement Frequency (GHz)
Differential NEXT @ RFSoc	-80 (ADC to DAC; DAC to ADC)	6.4
Differential NEXT @ RFSoc	-70 (ADC to ADC; DAC to DAC)	
Differential Insertion Loss	-1.5	6.4
Differential Return Loss	-23 using simulation; -18 after taking into account $\pm 10\%$ variation to nominal trace impedance due to manufacturing tolerance	6.4
Common Mode NEXT @ RFSoc	-70	6.4
Common Mode Return Loss	-15	6.4
P & N Balance	Total = 2000 fs; 1000 fs using simulation, with an additional 1.0 ps to account for manufacturing tolerances	6.4

**Notes:**

1. An impedance variation of  $\pm 10\%$  itself equals -20 dB.

### Trace Routing Impedance Specification

Analog pairs should be loosely coupled and routed differentially to  $100\Omega \pm 10\%$ .

### AC/DC Coupling Guidelines

In applications that use AC coupling for interfacing the data converters, DAC outputs should be AC coupled using a 100 nF capacitor in package size 0402 or smaller. ADC can be AC coupled, if necessary, and should be 100 nF in package size 0402 or smaller. Figure 3-17 illustrates a representative placement of the AC coupling capacitors in a typical ADC interfacing circuit.

DC coupling is also supported for the ADC and DAC. For DAC DC coupling, there are two constraints. First, the DAC requires the correct effective termination resistance of 100Ω differential. Second, the DAC outputs should be biased at the correct DC common mode point. The DC common mode point is 1.9V for 20 mA mode (2.5V) and 2.1V for 32 mA mode (3.0V) operation.

For ADC DC coupling, the ADC VCM output should be coupled with a 100 nF capacitor in package size 0402 or smaller. In addition, the input common-mode level should be taken from the ADC VCM output signal.

A typical ADC DC coupling configuration is shown in Figure 3-6.

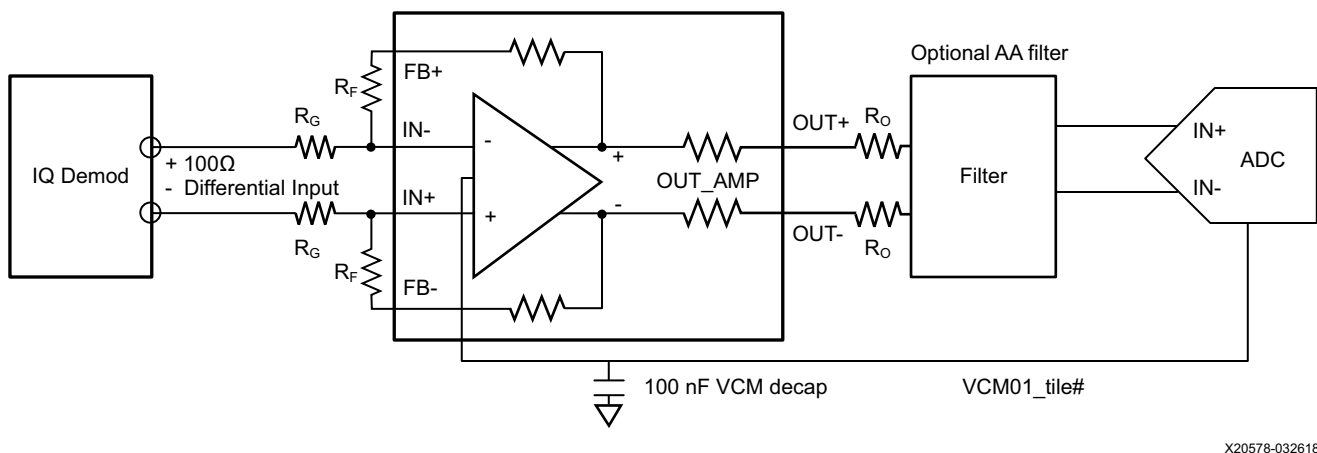


Figure 3-6: Typical ADC DC Coupling Configuration

**Note:** The VCM output buffer is only enabled in DC coupled mode.

### Isolation Recommendations

Incorporating all the following PCB design techniques results in the highest isolation between traces on the PCB. If significant deviation from the guidelines is planned, Xilinx recommends performing 3D EM analysis of the board structure to verify that performance at the package launch traces is acceptable for the application.

**Note:** Separation between traces is not entirely sufficient to achieve the isolation recommendations listed below. The effect of crosstalk rolls off as  $1/(1+(s/h)^2)$ , where  $s$  is trace separation, and  $h$  is

dielectric thickness. This function does not have a steep roll-off, and because space is limited, other methods must be used to achieve the recommended isolation.

Table 3-9 lists the isolation recommendations for ADC and DAC pairs.

Table 3-9: Isolation Recommendations for ADC & DAC Pairs

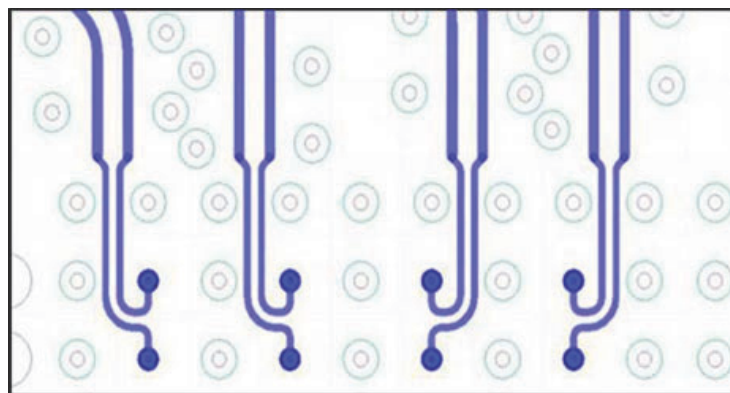
ADC & DAC Pair	Isolation (0–4 GHz)
ADC input pair to ADC input pair	–75 dB (differential)/–65 dB (common mode)
DAC output pair to DAC output pair	–75 dB (differential)/–65 dB (common mode)
DAC output pair to ADC Input pair	–80 dB (differential)/–70 dB (common mode)
ADC clock to DAC clock/DAC clock to ADC clock	–80 dB (differential)/–70 dB (common mode)
ADC clock to ADC clock/DAC clock to DAC clock	–70 dB (differential)/–70 dB (common mode) <sup>(1)</sup>
ADC/DAC clock to ADC/DAC input/output	–80 dB (differential)/–65 dB (common mode) <sup>(1)</sup>

**Notes:**

- 80 dB isolation is not required if the clocks are synchronous and at the same frequency.

A number of methods can be used to achieve the best isolation:

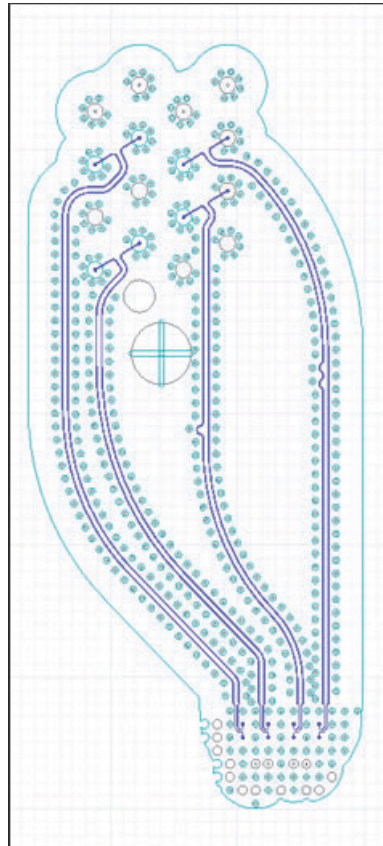
- **Micro vias, 3 dB isolation improvement:** Micro vias are preferred over back drilling due to any residual stubbing and the presence of a resonance cavity.
- **Shorter vias, 1 dB isolation improvement per 5 mil (0.13 mm) length:** Vias are a source of crosstalk, which can be minimized by keeping lengths as short as possible.
- **Differential breakout, 3 dB isolation improvement:** Differential breakout, as shown in Figure 3-7, is preferred over single-ended breakout. Single-ended breakout crosstalk is highest within 78 mil (2.0 mm) from the package edge and should be avoided, if possible.



X19470-070617

Figure 3-7: Differential Breakout Example

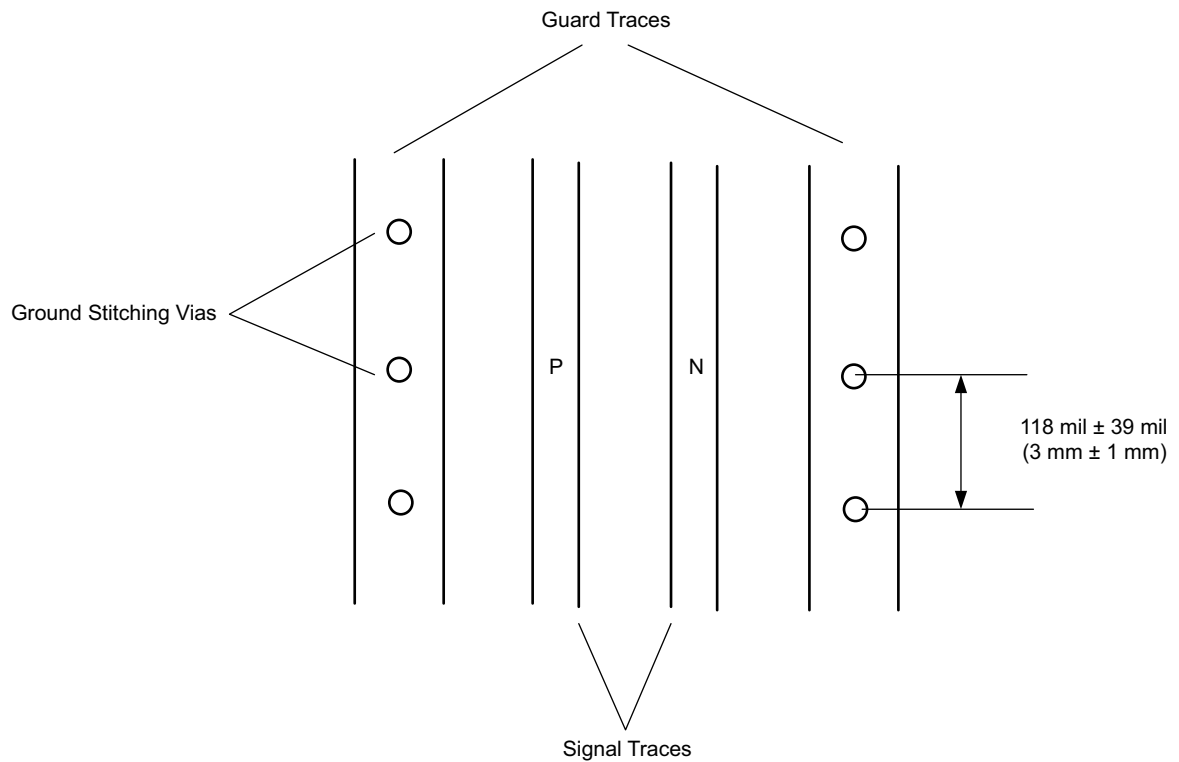
- Guard traces, 10 dB isolation improvement:** Separate each P/N pair by a ground guard-band trace that is 2X the layer height. Ensure that the above and below ground planes extend at least 4H away from the edge of each signal trace. Stitch the guard-band trace to ground every 118 mil  $\pm$  39 mil (3.0 mm  $\pm$  1 mm). All ground stitching should be implemented with through-hole vias. [Figure 3-8](#) shows an example of guard-band stitching around P/N pairs.



X19471-070617

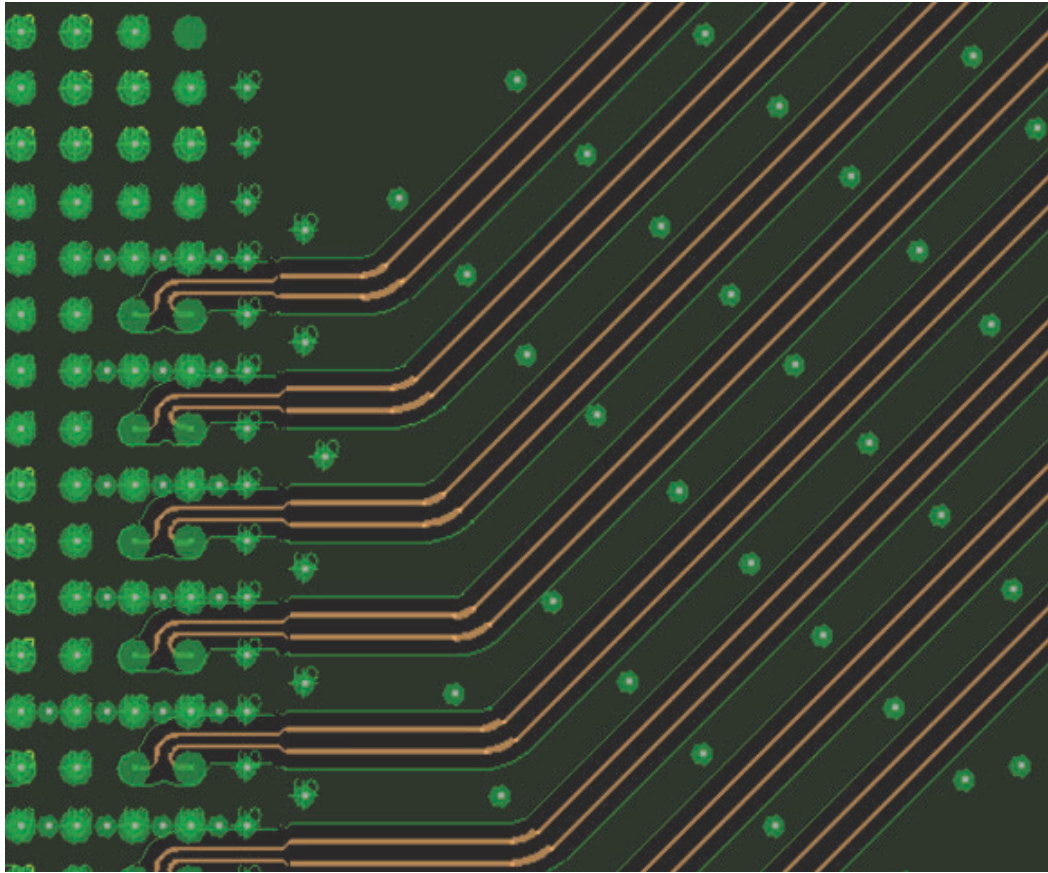
Figure 3-8: Example of Ground Stitching around Signal Pairs

Figure 3-9 shows closer detail of guard traces and ground stitching vias. Ensure to continue ground stitching around the pins of the RFSoc, as shown in Figure 3-10.



X19904-121217

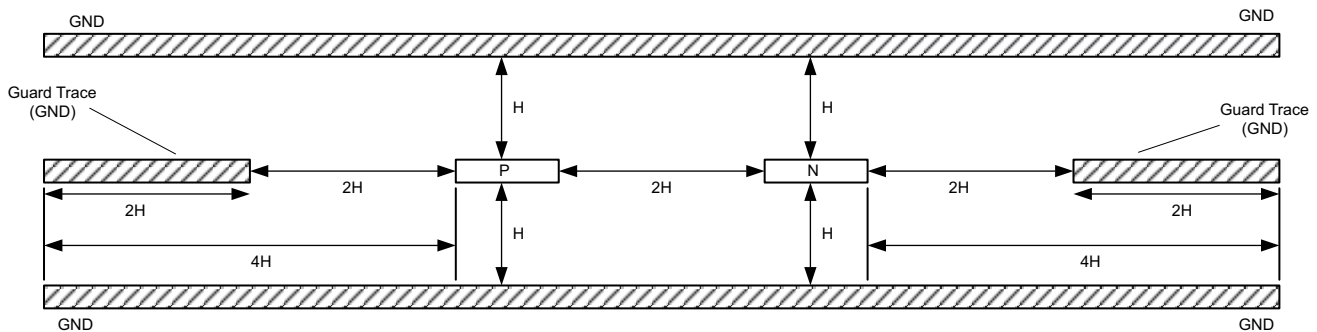
Figure 3-9: Close View of Guard Traces and Ground Stitching



X20581-032618

Figure 3-10: Ground Stitching around RFSoc Pins

Figure 3-11 shows a cross-section view of the guard traces.



X19902-092917

Figure 3-11: Cross-Section of Guard Traces

- Via-In-Pad, Plated Over (VIPPO), 5 dB isolation improvement:** Plated via-in-pad is preferred over routing a dog-bone trace to a via, because a dog-bone on layer one is subject to more crosstalk from PCB and external sources.

## P and N Skew Specifications

The total P to N skew should be no longer than 2000 fs (2.0 ps). This skew specification is broken down into two components:

- $\pm 1000$  fs (1.0 ps) using simulation and taking into account RFSoc package delays and PCB trace length mismatch.
- $\pm 1000$  fs (1.0 ps) due to manufacturing variations.

Simulations should include a complete channel s-parameter analysis that includes the RFSoc package and PCB S-parameter models. The PCB should be deskewed using the simulation analysis to meet the 1000 fs specification. You can request the package S-parameter models (Table 3-10) for the ADC/DAC channels by talking to your I/O/RF specialist.

Table 3-10: Available S-Parameter Models for ADC/DAC Channels

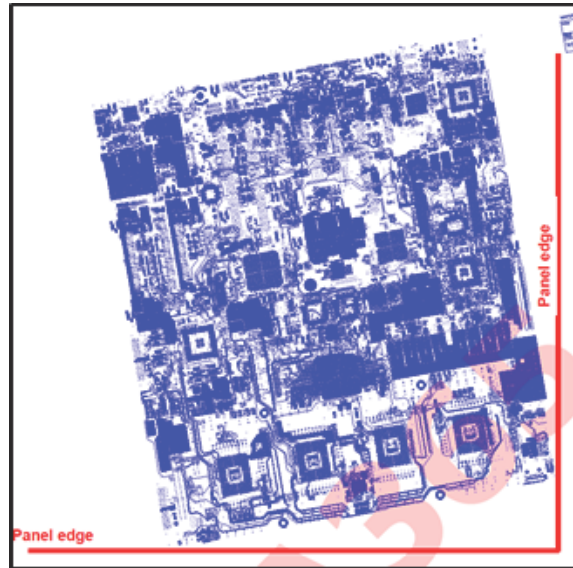
Touchstone Files	Description
ADC_On_Die_Correlated_Model.s2p	Hardware correlated ADC on-die model
DAC_On_Die_Correlated_Model.s2p	Hardware correlated DAC on-die model
ADC_FFVE1156_Package.s48p	FFVE1156 package model-ADC Channel & Ref clock channels for ZU25DR/ZU27DR/ZU28DR
DAC_FFVE1156_Package.s44p	FFVE1156 package model-ADC Channel & Ref clock channels for ZU25DR/ZU27DR/ZU28DR
ADC_FFVG1517_Package.s32p	FFVG1517 package model-ADC Channels for ZU25DR/ZU27DR/ZU28DR
ADC_clk_FFVG1517_Package.s16p	FFVG1517 package model-ADC Ref clock channels for ZU25DR/ZU27DR/ZU28DR
DAC_FFVG1517_Package.s32p	FFVG1517 package model-DAC Channels for ZU25DR/ZU27DR/ZU28DR
DAC_clk_FFVG1517_Package.s8p	FFVG1517 package model-DAC Ref clock channels for ZU25DR/ZU27DR/ZU28DR
ADC_full_FFVF1760_Package.s80p	FFVF1760 package model-ADC & Ref clock channels for ZU29DR
DAC_FFVF1760_Package.s64p	FFVF1760 package model-DAC Channels for ZU29DR
DAC_clk_FFVF1760_Package.s16p	FFVF1760 package model-DAC Ref clock channels for ZU29DR

Manufacturing variations can be minimized via the following methods:

- **1086 Mechanically Spread (MS) Glass, 700–900 fs skew per inch:** MS glass is spread in both directions to achieve greater homogeneity.
- **NE glass, 400–600 fs per inch:** Replacing E-glass with NE glass can provide for further skew reduction.



- **Image Rotation, 500–100 fs skew per inch:** Rotating the image on the PCB panel, as shown in Figure 3-12, can help offset panel x-y non-homogeneity. The recommended rotation is 12 degrees.

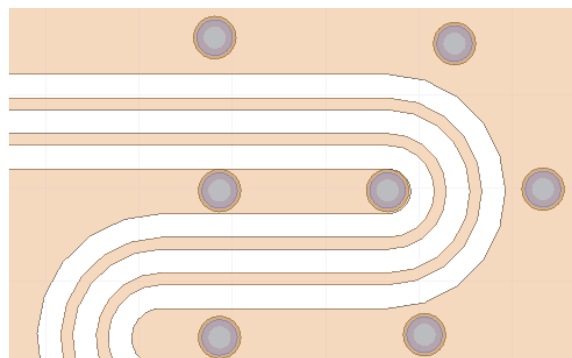


X19472-070617

Figure 3-12: Rotated Panel to Compensate for Glass Non-Homogeneity

- **Trace Length Matching, ~150 fs:** Matching the P and N trace length to 1 mil (0.0254 mm) can help to achieve approximately 150 fs of skew. Trace length matching should be done as close as possible to any AC coupling capacitors.

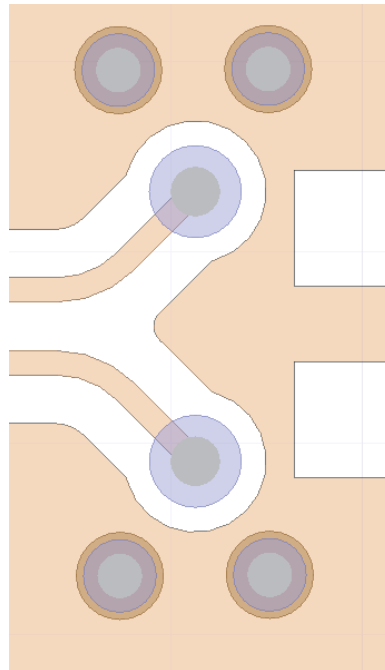
When routing a 180-degree bend of a P/N pair, the minimum bend radius must be 3H. This distance allows for a guard trace and ground stitching vias to be placed between the two segments of the inner trace, before and after the bend. Ground stitching vias must be placed on the guard trace in this inner region with the final ground via placed at the end of the guard trace near the signal trace bend. Figure 3-13 shows a properly routed P/N trace pair 180-degree bend with guard trace and ground stitching vias.



X21203-071918

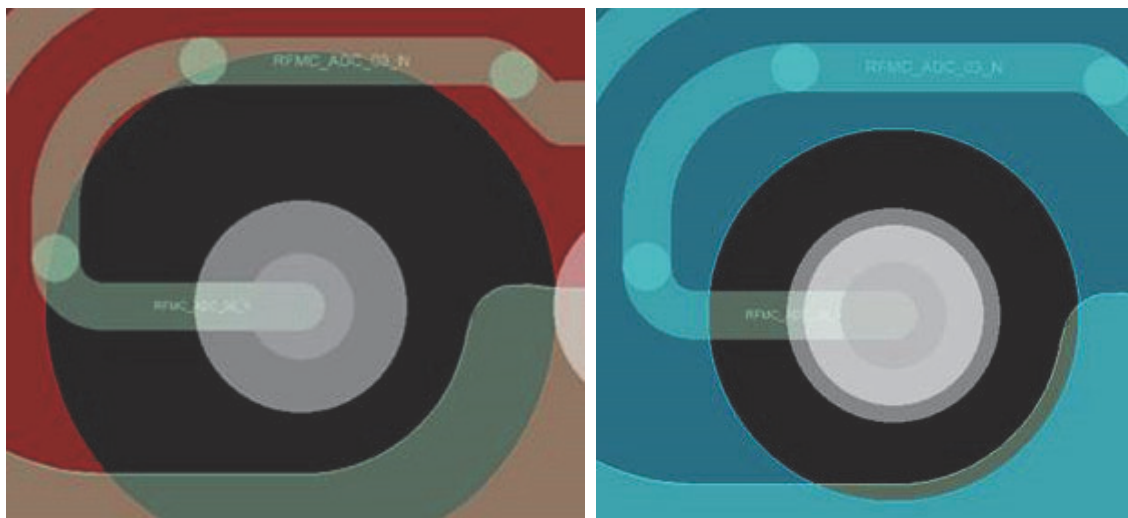
Figure 3-13: P/N Trace Pair with 180-Degree Bend

When P/N traces transition layers, two ground vias must be added per signal for impedance control, noise isolation, and P/N skew control. The ground vias must be within 1 mm of the signal vias. [Figure 3-14](#) illustrates the via configuration for a signal layer transition.



**Figure 3-14: Via Configuration for Signal Layer Transition**

P/N signal traces should be routed a minimum of  $0.5H$  away from any anti-pad or cutout in the ground planes above or below the signal traces. The left side of [Figure 3-15](#) illustrates an incorrectly routed signal trace. The right side of [Figure 3-15](#) illustrates the correct routing of a signal trace.



X21205-071918

Figure 3-15: Signal Trace Routing

Signal guard traces must be continuous with no breaks and must have proper ground stitching. An improperly implemented guard trace could result in additional P/N skew as well as increased noise susceptibility.

### Inter-Pair Skew

The skew between separate ADC pairs should not extend beyond  $\pm 50$  ps. The skew between DAC pairs should not extend beyond  $\pm 50$  ps. Include the package traces when routing.

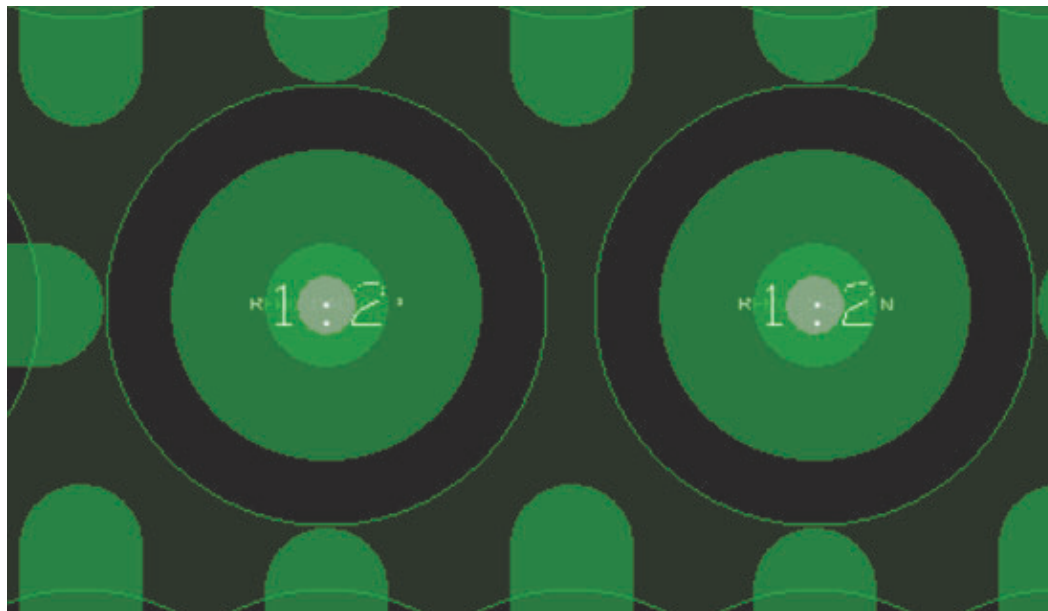
### Insertion Loss Recommendations

Insertion loss is recommended to be better than  $-1.5$  dB from the BGA ball to its balun (or similar interfacing component). Most board materials are capable of achieving better than  $-1.5$  dB of insertion loss at frequencies up to around 6 GHz. Insertion loss mitigation does come with a potentially significant cost multiplier, so care should be taken in choosing the proper material. See [Appendix B, Material Properties and Insertion Losses](#) for insertion loss values based on trace widths and frequencies.

### Return Loss Recommendations

Return loss is recommended to be better than  $-20$  dB across the bandwidth of interest (e.g., 0–4 GHz). Return loss can be mitigated with the following methods:

- **Circular ground antipad on layer one, 5 dB improvement:** A circular antipad on layer one should be used for P/N pairs, as shown in [Figure 3-16](#). The microvia drill diameter should be 5 mil, with a 25 mil pad diameter and 35 mil anti-pad diameter.



X20582-032618

Figure 3-16: Rectangular Antipad

- **Micro via, 7 dB:** Back drilling can leave residual metal and a resulting resonance cavity.

**Note:** Xilinx does not recommend using ground flooding on signal layers as this might cause the board to warp, causing strain on soldered components. Xilinx recommends adding a  $-2$  dB differential resistive Pi attenuator to optimize the RL of the device. This Pi network can be placed anywhere on the PCB between the package BGA balls and the balun. See [Figure 3-17](#).

### ***Pi Network for Improved Return Loss***

Xilinx ADCs and DACs have a nominal on-die  $100\Omega$  differential termination, which generally provides sufficient return loss performance for most RF applications. However, for applications requiring better than  $-10$  dB return loss up to 4 GHz, Xilinx recommends placing  $-2$  dB balanced Pi pad attenuators on the PCB. These Pi pad attenuators can be placed anywhere on the PCB between the ADC/DAC BGA balls of the package and the baluns. A representative  $-2$  dB Pi pad attenuator for a differential ADC/DAC is shown in [Figure 3-17](#).

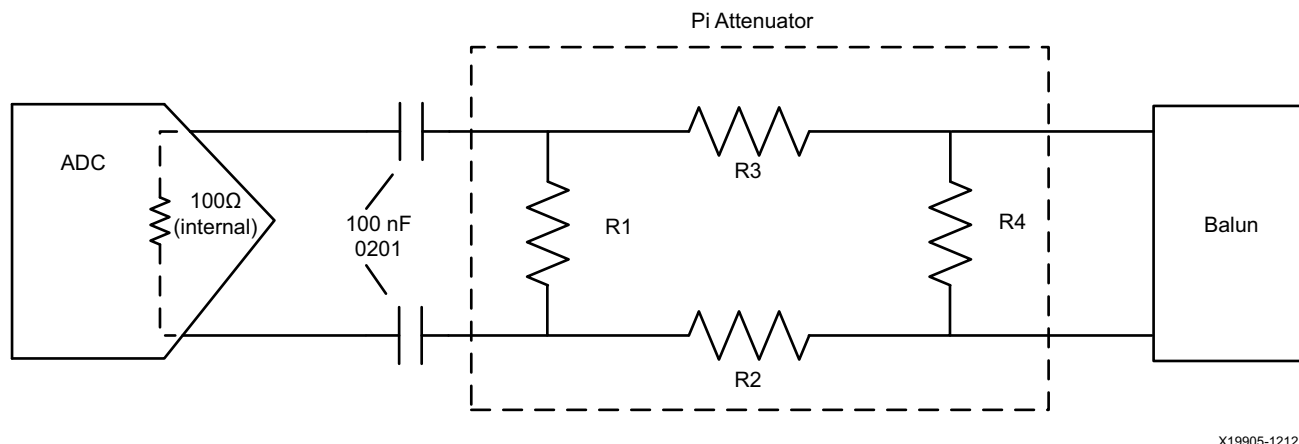


Figure 3-17: -2 dB Pi Attenuation Network

The component descriptions are listed below with recommended part numbers:

- R2 = R3 = 11.5Ω (Panasonic part number [ERJ-1GEF11R5C](#))
- R1 = R4 = 866Ω (Panasonic part number [ERJ-1GEF8660C](#))

Xilinx recommends the following characteristics for the discrete resistive components:

- 0201 body size or smaller to minimize parasitic inductance.
- ±1% resistor tolerance or better to minimize insertion loss variation
- 100 ppm/°C temperature coefficient or better to minimize variation of the resistance value over temperature

For purposes of PCB routing, there should be a minimum of four ground vias for each and every signal via. The length of the vias and trace routing between components should be minimized as much as possible. Placing the Pi pad attenuators underneath cans minimizes crosstalk between them.

### **SYSREF**

In each Zynq UltraScale+ RFSoc there is one dedicated input SYSREF pin pair located in DAC tile 0. This SYSREF signal is used for multi-tile and multi-chip channel synchronization. This differential input pair is referred to as the Analog\_SYSREF. To fully implement the synchronization features in the RFSoc, an additional SYSREF signal is needed to drive into the PL fabric through a pair of PL GPIO pins. This pair is referred to as the PL\_SYSREF. It is possible to use the multi-tile sync feature even if not all available tiles in the device are being used. In such an event, DAC tile 0 should be one of the active tiles. ADC tile 0 should also be one of the active tiles if multi-tile sync for the ADCs is a requirement in the application. All DAC tiles are synchronized to DAC tile 0, and all ADC tiles are synchronized to ADC tile 0. That is why tile 0 is required to be active for the respective data converter types.

Xilinx recommends that the user prioritize the lower indexed tiles over the higher ones if not all tiles are to be enabled. For example, if only two tiles are used, tile 0 and tile 1 should be enabled, while tile 2 and 3 are disabled. Analog\_SYSREF can also be used for synchronizing the phase of the digital up converter (DUC) numerically controlled oscillators (NCOs) and digital down converter (DDC) NCOs, respectively, in addition to synchronizing the overall delay of each tile (done with the MTS function). The NCO synchronization is a separate process from the MTS synchronization and only applicable if the fine complex mixer is used in the application. Refer to the Multi-Converter Synchronization section in *Zynq UltraScale+ RFSoc RF Data Converter LogiCORE IP Product Guide* (PG269) [Ref 17] for the detailed theory of operations and connection diagram.

The Analog\_SYSREF is internally terminated differentially at 100Ω. It can be used in AC coupled mode or DC coupled mode. For applications that require NCO synchronization across multiple devices, DC coupled mode is required along with the ability to generate the SYSREF signal in a pulse mode. Refer to *Zynq UltraScale+ RFSoc RF Data Converter LogiCORE IP Product Guide* (PG269) for additional guidance. Table 3-11 provides further requirements.

Table 3-11: Analog\_SYSREF Requirements

Parameter	Value
Coupling	AC or DC coupling
Signal Levels	Greater than 500 mVppd (peak-to-peak differential swing)
Termination	100Ω differential
Jitter	< 1 ps
Frequency	< 10 MHz
Phase	Phase relationship to the ADC/DAC sampling clocks should be deterministic from reset to reset.

If synchronizing ADC and DAC tiles with SYSREF, the frequency must be an integer sub-multiple of Equation 3-1.

$$GCD\left(\frac{DAC_{SampleRate}}{16}, \frac{ADC_{SampleRate}}{16}\right) \quad \text{Equation 3-1}$$

An example calculation for SYSREF is shown below.

$$ADC_{SampleRate} = 3.93216 \text{ GHz} \quad \text{Equation 3-2}$$

$$DAC_{SampleRate} = 4.91520 \text{ GHz} \quad \text{Equation 3-3}$$

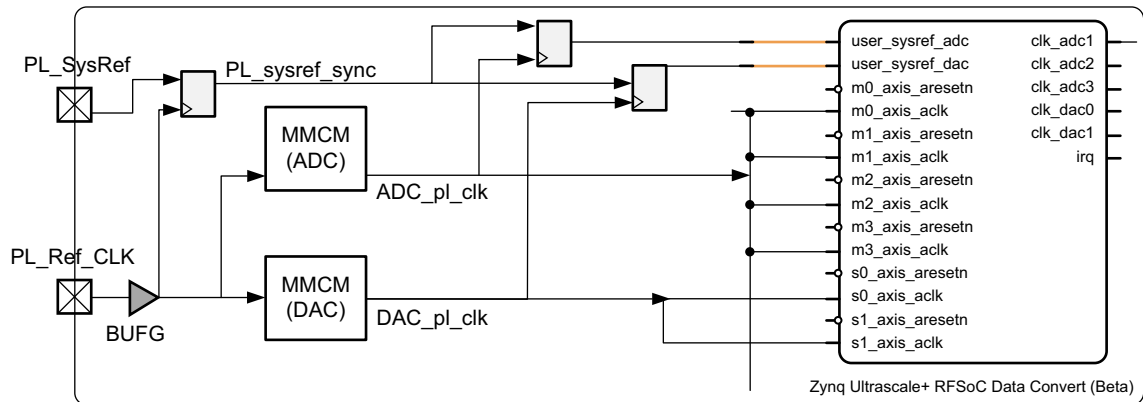
$$GCD\left(\frac{ADC_{SampleRate}}{16}, \frac{DAC_{SampleRate}}{16}\right) = 61.44 \text{ MHz} \quad \text{Equation 3-4}$$

$$\text{Integer Sub - Multiple AND } < 10 \text{ MHz} = 7.68 \text{ MHz} \quad \text{Equation 3-5}$$

For full functionality of the synchronization features, the PL\_SYSREF should meet the following requirements:

- PL\_SYSREF should be the same frequency as the Analog\_SYSREF.

- PL\_SYSREF must be a sub-multiple of the PL frequencies that interface to the DAC and ADC FIFOs.
- PL\_SYSREF and PL clock must meet setup and hold to ensure the PL clock can be used to deterministically capture the rising edge of the PL\_SYSREF.
- Analog\_SYSREF and PL\_SYSREF should be phase aligned at the package pins of the RFSoc.
- The same clock quality (if not the same clock) should be used for the PL\_SYSREF as the Analog\_SYSREF. Refer to the PL GPIO electrical requirements for coupling, voltage swing, and termination.
- If the MMCM is used to generate the final PL clocks for the DAC FIFO interface and ADC FIFO interface, the 0-delay (0 phase) feature of the MMCM should be used in addition to using flip-flops to synchronize the PL\_SYSREF capture as shown in the example below.



X20579-032618

Figure 3-18: PL\_SYSREF Capture

### Ground Plane Cutout Under Passive Components

All passive components, including resistors for the Pi attenuator and AC coupling capacitors, should have a rectangular cutout in the ground layer below/above them. The cutout dimensions should be the same as the dimensions of the component lands.

### Calibration Resistors

Two separate pins exist for ADC and DAC calibration:

- ADC External Resistor: Tie to GND through a 2.49 kΩ resistor as close to the RFSoc pin as possible.
- DAC External Resistor: Tie to GND through a 2.49 kΩ resistor as close to the RFSoc pin as possible.

## Sample Stackup

Table 3-12 shows a sample stackup for the top five layers of the board. The trace dimensions in Table 3-12 are approximate trace geometries based on the sample stackup using a typical dielectric thickness between the signal/ground layers to achieve a 100Ω differential impedance. Customers are requested to run their own SI analysis to arrive at these numbers based on their specific stackup information taking manufacturing tolerances into account.

It is good design practice to use the top routing layers to route the ADC/DAC clock pairs. You can use layer 2 for routing the ADC/DAC channels and layer 4 for routing the clock pairs to attain the -80 dB NEXT isolation that Xilinx recommends between the DAC to ADC clock pairs.

Table 3-12: Sample Stackup for Top Five Layers

Layer	Material <sup>(1)</sup>	Net(s)	Trace Specifications
1 (top)		Ground	
	Pre-Preg		
2		ADC/DAC Signals	5 mil (0.13 mm) trace (P) 10 mil (0.25 mm) gap 5 mil (0.13 mm) trace (N) 10 mil (0.25 mm) gap to guard trace 10 mil (0.25 mm) guard trace Ground stitch every 118 mil (3.0 mm)
	Core		
3		Ground	
	Pre-Preg		
4		ADC/DAC Clock Pairs	5 mil (0.13 mm) trace (P) 10 mil (0.25 mm) gap 5 mil (0.13 mm) trace (N) 10 mil (0.25 mm) gap to guard trace 10 mil (0.25 mm) guard trace Ground stitch every 118 mil (3.0 mm)
	Core		
5	Pre-Preg	Ground	

**Notes:**

- Vendor: Isola Group.  
Product Name: I-Speed.  
Construction/Glass Style: 2 x 1067 Spread LDk.  
Thickness: 5 mil (0.127 mm).  
Resin Content: 74.5%.  
Dielectric constant (DK) 3.26 (@10 GHz).  
Loss Tangent/Dissipation Factor (DF): 0.0055 (@10 GHz).



### Unused ADC & DAC Pins

- All unused ADC I/Os should be pulled to ground.
- All unused DAC outputs can be left floated (there is an internal pull-down on the outputs).
- All unused clock and SYSREF pins can be floated.

## Power Regulation and Decoupling for ADC and DAC Supplies

The recommended power delivery methodology for the various ADC and DAC power supplies are as shown in [Figure 3-19](#). VCCINT\_AMS can be supplied directly from a switching regulator, while ADC\_AVCC, ADC\_AVCCAUX, DAC\_AVCC, DAC\_AVCCAUX, and DAC\_AVTT are recommended to be supplied by a low noise, high power supply rejection ratio (PSRR) LDO that feeds from a common switching regulator. Xilinx recommends that separate power supplies be used for the ADC and DAC supplies as shown in [Figure 3-19](#). Capacitor specifications are noted in [Table 3-9](#), [Table 3-16](#), and [Table 3-18](#).



---

**IMPORTANT:** *The guidelines provided as part of this document are for reference only. However, Xilinx recommends you do a detailed simulation analysis to validate the various specifications prior to releasing the board for fabrication. Xilinx can provide the necessary ADC/DAC S-parameter models for the die and package trace extracted using a full 3D field solver to facilitate this analysis. You can use these models in any EDA tool like ADS from Keysight to perform a detailed frequency-domain analysis to analyze insertion loss, return loss, crosstalk performance, and PN balance, respectively. You can request the die package models by talking to your local I/O/RF specialist.*

---

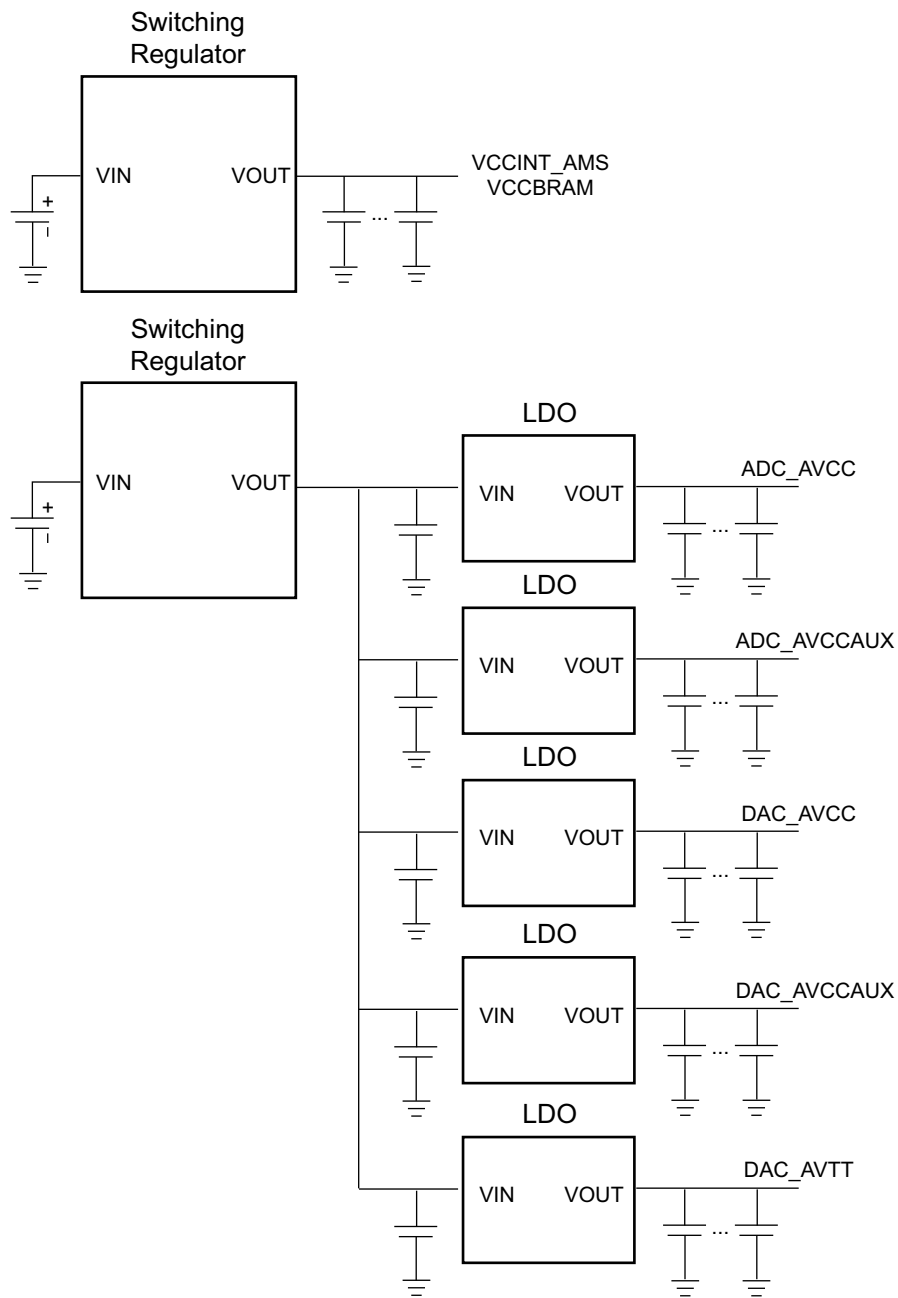


Figure 3-19: Power Regulation for Analog Supplies

### Recommended Linear Regulators

Table 3-16 lists regulators that Xilinx recommends using on the ADC and DAC RFSoc rails.

Table 3-13: Recommended Linear Regulators

Vendor	Devices
Intersil/Renesas	ISL80101-A ISL80102 ISL80103

### ADC and DAC Voltage Supply Specifications

The ADC and DAC voltage supply specifications are listed in [Table 3-19](#).

Table 3-14: ADC and DAC Voltage Supply Specifications

Supply	Nominal Voltage (V)	Tolerance (%) <sup>(1)</sup>	Maximum Current (A)	Frequency Range (MHz)	Maximum Supply Ripple (mVpp) <sup>(2)</sup>
ADC_AVCC	0.925	±3	XPE*	0.1-15	0.25
ADC_AVCCAUX	1.8	±3	XPE*	0.1-15	11.03
DAC_AVCC	0.925	±3	XPE*	0.1-15	0.40
DAC_AVCCAUX	1.8	±3	XPE*	0.1-15	2.00
DAC_AVTT	2.5/3.0 <sup>(3)</sup>	±3	XPE*	0.1-15	8.94
VCCINT_AMS	0.85/0.90	±3	XPE*	0.1-15	20.00

**Notes:**

1. The tolerance percentage is for the switching regulator that feeds the LDO.
2. Output of the LDO.
3. DAC\_AVTT should be set to 2.5V if used in 20 mA mode, and 3.0V if used in 32 mA mode.

### Impedance Targets for ADC and DAC Power Supply Rails

The decoupling capacitor guidelines specified below are based on impedance targets that each ADC/DAC should meet at various frequencies. The impedance targets are listed in [Table 3-15](#).

Table 3-15: Impedance Targets for ADC/DAC Power Rails

Rail Name	Frequency (MHz)	Impedance Target (mΩ)
DAC_AVCC	0.5–15	15
ADC_AVCC	0.5–15	15
DAC_AVCCAUX	0.5–15	65
ADC_AVCCAUX	0.5–15	65
DAC_AVTT	0.5–15	120

**Notes:**

1. Regulator is assumed to regulate below 500 kHz.

## Decoupling Capacitor Specifications for ADC and DAC Power Supply Rails

Table 3-16 shows the minimum capacitor specifications for ADC and DAC power rails. The capacitor quantities are outlined in the sections below.

Table 3-16: Minimum Capacitor Specifications for ADC & DAC Power Rails

Value ( $\mu\text{F}$ )	Case Size	ESR ( $\text{m}\Omega$ )	Capacitor Tolerance	Voltage Rating	Temperature Rating	Example
10	1206	6	$\pm 20\%$	4V	X5R	Murata GRM31CR71C106MA12
4.7	0402	8	$\pm 20\%$	4V	X5R	Murata GRM155C80J475MEAA#
2.2	0402	7	$\pm 20\%$	4V	X5R	Murata GRT155C81C225KE13#
1.0	0402	10	$\pm 20\%$	4V	X5R	Murata GRM155R70J105MA12#
0.68	0402	10	$\pm 20\%$	4V	X5R	Murata GRM155R61A684KE15#
0.47	0402	20	$\pm 20\%$	4V	X5R	Murata GRM152D80G474ME15#

### VCCINT\_AMS Decoupling and Sharing with VCCBRAM/VCCINT\_IO

A switching regulator is suitable for this VCCINT\_AMS, with decoupling capacitor quantities as shown in Table 3-17.

**Note:** VCCINT\_AMS can connect to the same plane as VCCINT, VCCBRAM, and VCCINT\_IO so long as all voltage levels are the same and the noise specifications for each individual rail can be adhered to.

Table 3-17: Capacitor Quantities for VCCINT\_AMS

	10 $\mu\text{F}$ near RFSoc	4.7 $\mu\text{F}$ near RFSoc	2.2 $\mu\text{F}$ at Underside of RFSoc	1.0 $\mu\text{F}$ at Underside of RFSoc
VCCINT_AMS	2	4	2	2

**Notes:**

- See Table 3-16 for capacitor specifications.

### Decoupling for ADC\_AVCC, ADC\_AVCCAUX, DAC\_AVCC, DAC\_AVCCAUX, and DAC\_AVTT

The capacitor quantities for the ADC and DAC power rails are shown in Table 3-18.

Table 3-18: Capacitor Quantities for ADC and DAC Rails

	4.7 $\mu\text{F}$ at Output of Regulator	2.2 $\mu\text{F}$ at Output of Regulator	4.7 $\mu\text{F}$ at Underside of BGA	2.2 $\mu\text{F}$ at Underside of BGA	0.68 $\mu\text{F}$ at Underside of BGA	0.47 $\mu\text{F}$ at Underside of BGA
ADC_AVCC	2	2	2	3	0	0
ADC_AVCCAUX	2	2	0	2	1	1
DAC_AVCC	2	2	2	3	0	0
DAC_AVCCAUX	2	2	0	2	1	1

Table 3-18: Capacitor Quantities for ADC and DAC Rails (Cont'd)

	4.7 $\mu$ F at Output of Regulator	2.2 $\mu$ F at Output of Regulator	4.7 $\mu$ F at Underside of BGA	2.2 $\mu$ F at Underside of BGA	0.68 $\mu$ F at Underside of BGA	0.47 $\mu$ F at Underside of BGA
DAC_AVTT	2	2	0	2	1	1

**Notes:**

1. See Table 3-16 for capacitor specifications.

**Decoupling Capacitor Antipad to Reduce Insertion and Return Loss**

It is recommended to place a rectangular antipad around each ADC and DAC decoupling capacitor on the top and bottom layer, as shown in Figure 3-20.

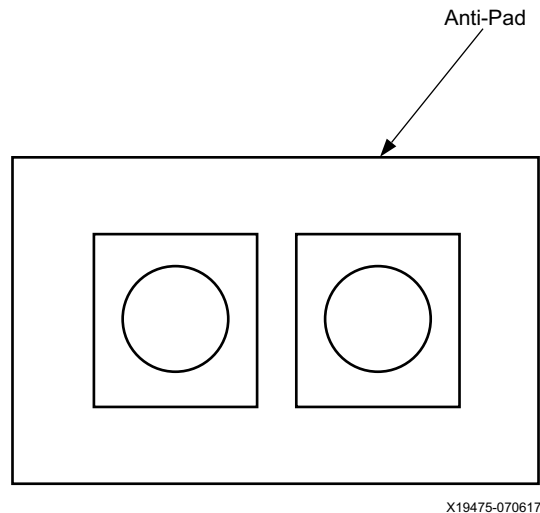


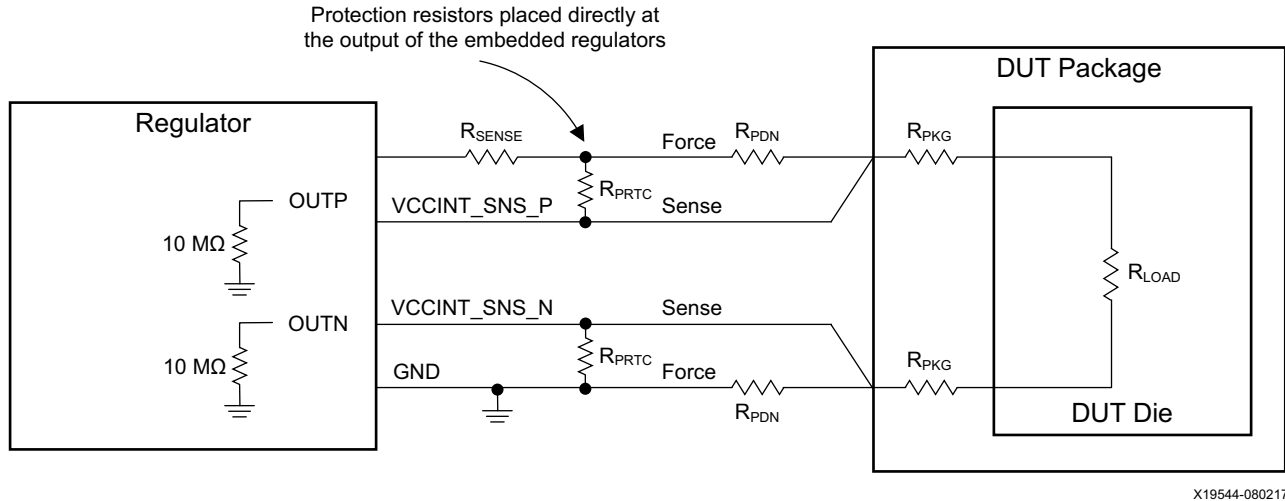
Figure 3-20: Decoupling Capacitor Antipad

**Voltage Sensing**

To compensate for PCB IR losses, all of the ADC and DAC voltage rails need to properly utilize the sense features of their respective regulators, as listed below:

- ADC\_AVCC
- DAC\_AVCC
- ADC\_AVCCAUX
- DAC\_AVCCAUX
- DAC\_AVTT
- VCCINT\_AMS

Sense line connections should not use a dedicated spy hole pin on the RFSoc, because all power supply pins should be fully utilized and connected to their respective planes. Instead, a "Force" solution should be utilized, as shown in Figure 3-21.



X19544-080217

Figure 3-21: Recommended Sense Line Circuitry

Notes related to Figure 3-21:

- $R_{PDN}$ : Power distribution network resistance (PCB parasitics).
- $R_{SENSE}$ : Discrete 2 mΩ precision sense resistor.
- $R_{PRTC}$ : Discrete 10Ω regulator protection resistor.

### Unused ADC and DAC Power Pins

- The DAC VTERM supply can be tied to the VCCAUX 1.8V supply when the DAC is not enabled.
- The power supply solution should be ensured to be capable of providing the leakage current to the unused tiles.
- If any ADC or DAC tiles are not used, they should be tied to the same supplies as the respective used tiles. The leakage current of each unused tile should be factored when determining regulator output current. Leakage current has an exponential effect at around 100°C. Table 3-19 lists static leakage current per tile for the ADC and DAC at 100°C.

Table 3-19: Leakage Current Values for ADC and DAC Tiles

Tile	VCCINT_AMS (mA)	ADC/DAC_AVCC (mA)	ADC/DAC_AVCCAUX (mA)	DAC_AVTT (mA)
ADC	10	115	15	N/A
DAC	120	40	10	16

# PCB Guidelines for the PS Interface in the Zynq UltraScale+ MPSoC

This chapter lists layout guidelines specific to the PS MIO and PS-GTR interfaces in the Zynq UltraScale+ MPSoC.

## Boot Mode

- Connect the boot mode pins to a 4.7 k $\Omega$  or lower pull-up resistor to  $V_{CCO\_PSIO[3]}$  or pull-down to ground depending on the desired setting.
- If multiple switchable boot modes are desired, connect one pull-up/down resistor to the mode pin and place a jumper on the other side of the resistor to select between pull-up or pull-down.
- An easily switchable boot mode configuration is recommended for debug ease-of-use.

## CAN

- PCB and package skew between the TX/RX and clock should be within  $\pm 100$  ps.
- A level shifter must be implemented if using a CAN PHY that operates at a voltage higher than  $V_{CCO\_PSIO}$ , e.g., 5.0V.

## DisplayPort

DisplayPort is a PS-GTR transceiver interface, and rules for PS-GTR transceiver connections can be found in [PS-GTR Transceiver Interfaces, page 179](#). In addition, the DP\_OE, DP\_AUX\_IN, and DP\_AUX\_OUT signals need to be connected to a bidirectional LVDS buffer like the Fairchild FIN1019MTC. For additional details on connectivity, refer to the VESA DisplayPort Standard Version 1, Section 3.4, AUX Channel [\[Ref 20\]](#).

## eMMC

- Pull-up resistors are required on data[0-7] per JEDEC specification JESD84-B451 [\[Ref 21\]](#).
- A 30 $\Omega$  series resistor should be placed on the CLK, CMD, and DATA lines, as close to the MIO pins as possible.

### ***Standard and High-Speed SDR Interfaces***

PCB and package skew between data/cmd and clock should be within  $\pm 100$  ps.

### ***HS200 (200 MHz) and High-Speed DDR Interfaces***

PCB and package skew between data/cmd and clock should be within  $\pm 50$  ps.

## **Ethernet MAC RGMII**

Delay skew for TXD/RXD[3:0] and CTL to clock delay should be within  $\pm 50$  ps including package time.

## **Ethernet MAC SGMII**

Ethernet SGMII is a PS-GTR transceiver interface, and rules for PS-GTR transceiver connections can be found in [PS-GTR Transceiver Interfaces, page 179](#).

## **I2C**

- Place 4.7 k $\Omega$  pull-up resistors at the far end of the SCL and SDA lines, furthest from the Zynq UltraScale+ MPSoC.
- A level-shifter/repeater might be required depending on the particular multiplexers used.

## **JTAG**

Place 4.7 k $\Omega$  pull-up resistors on the TMS, TCK, and TDI lines.

## **NAND Memory Controller**

### ***SDR Mode***

- PCB and package skew between ALE/CE/CLE/IO[7:0] and WE/RE should be within  $\pm 100$  ps.
- Place 4.7 k $\Omega$  pull-up resistors on CE and RB near the NAND device.

### ***DDR Mode (100 MHz)***

- PCB and package skew between ALE/CE/CLE and CLK should be within  $\pm 50$  ps.
- To operate in mode 0 to mode 4, PCB and package skew between CLK and DQS should be within  $\pm 50$  ps.



- To operate in mode 5 at 100 MHz, PCB and package skew between CLK and DQS should be a minimum of 400 ps + 50 ps. CLK should be skewed with respect to DQS by 400 ps.
- PCB and package skew between DQ and DQS should be within  $\pm 25$  ps.
- Place 4.7 k $\Omega$  pull-up resistors on CE and RB near the NAND device.

## PCIe

PCIe is a PS-GTR transceiver interface, and rules for PS-GTR transceiver connections can be found in [PS-GTR Transceiver Interfaces](#), page 179.

## PS\_INIT\_B, PS\_PROG\_B, and PS\_DONE

- Connect PS\_INIT\_B to a 4.7 k $\Omega$  pull-up resistor to V<sub>CCO\_PSIO[3]</sub>.
  - PS\_INIT\_B is open drain and should not be driven during logic built-in self test (LBIST).
- Connect PS\_PROG\_B to a 4.7 k $\Omega$  pull-up resistor to V<sub>CCO\_PSIO[3]</sub>.
  - PS\_PROG\_B is open drain and should not be driven during LBIST.
- Connect PS\_DONE to a 4.7 k $\Omega$  pull-up resistor to V<sub>CCO\_PSIO[3]</sub>.

## PS Reference Clock

PS\_REF\_CLK is a single-ended LVCMOS signal.

## PS Reset (External System Reset and POR Reset)

- Connect PS\_SRST\_B to a 4.7 k $\Omega$  pull-up resistor to V<sub>CCO\_PSIO[0]</sub> near the Zynq UltraScale+ MPSoC.
- Connect PS\_POR\_B to a 4.7 k $\Omega$  pull-up resistor to V<sub>CCO\_PSIO[0]</sub> near the Zynq UltraScale+ MPSoC.
- Refer to the “Power Supply Sequencing” section in *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics* (DS925) [Ref 22] for rules regarding PS\_POR\_B during the power-on sequence.

## QSPI

- The clock, data, and SS lines are recommended to have matched lengths to facilitate meeting setup and hold times.
- PCB and package delay skew for I/O[3:0] and SS lines relative to CLK should be within  $\pm 50$  ps.

- Keeping the clock and data lines equal provides greater immunity to undesirable setup and hold time effects.
- It is highly recommended to perform a signal integrity analysis on the clock line at the near (close to the Zynq UltraScale+ MPSoC) and far ends.
- For optimum performance, limit trace delays to less than 175 ps.
- Place 4.7 k $\Omega$  pull-up resistors on the HOLD, WP, and CS lines.
- Keep MIO[6] unconnected for higher FQSPICLK1 or FQSPICLK2 operating frequencies (>40 MHz). This allows the loopback feature to work properly.

## Real-Time Clock

- See the *PS RTC Crystal Requirements* table in *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics (DS925)* for full specifications.
- If the real-time clock is not being used, connect PADI to ground and leave PADO floating.

## SATA

SATA is a PS-GTR transceiver interface, and rules for PS-GTR transceiver connections can be found in [PS-GTR Transceiver Interfaces, page 179](#).

## SD/SDIO

- A 30 $\Omega$  series resistor should be placed on the CLK, CMD, and DATA lines, as close to the MIO pins as possible.
- A level shifter might be required depending on the particular voltages used on the Zynq UltraScale+ MPSoC and SD chip.
- Asynchronous signals CDn and WPn have no timing relationship to CLK.
- The CDn and WPn lines should both be pulled up with their own 4.7 k $\Omega$  resistors to the MIO I/O voltage. When using Micro-SD, WPn and CDn can be no connects.
- A 10 k $\Omega$  pull-up resistor should be added to DAT3 on the SD card side of the level shifter.

## SPI

- PCB and package skew between the master-output, slave-input (MOSI)/master-input, slave-output (MISO) and clock should be within  $\pm 100$  ps.
- Place a 4.7 k $\Omega$  pull-up resistor on the SS pin near the serial peripheral interface (SPI) device.

## Trace Port Interface Unit

- When operating the trace port interface unit (TPIU) in MIO mode, the trace clock output should be delayed by approximately one half clock period. This can be done on the PCB, or by the debugging device (ARM\_DSTREAM, Lauterbach, or Agilent).

## Triple Time Counter

- PCB and package skews between WAVE\_OUT and CLK should be within  $\pm 100$  ps.

## UART

- Keep MIO trace delays below 1.30 ns.

## USB 2.0

### *ULPI Interface (60 MHz)*

- PCB and package delays should be kept to 1.30 ns or below.
- PCB and package delay skews for DATA[7:0]/DIR/NXT/STP and CLK should be within  $\pm 100$  ps.
- For optimum signal integrity, add a  $30\Omega$  series resistor to the DATA and STP lines near the Zynq UltraScale+ MPSoC.

## USB 3.0

USB 3.0 is a PS-GTR transceiver interface, and rules for PS-GTR transceiver connections can be found in [PS-GTR Transceiver Interfaces, page 179](#).

## Watchdog Timer

PCB and package skews between WAVE\_OUT and CLK should be within  $\pm 100$  ps.

## PS-GTR Transceiver Interfaces

PS-GTR transceivers are analog circuits that require special attention when being designed for and implemented on a printed circuit board. Besides an understanding of the functionality of the device pins, a design that performs optimally requires attention to issues such as device interfacing, transmission line impedance and routing, power supply design filtering and distribution, component select, and PCB layout and stackup design.

These PS interfaces utilize PS-GTR transceivers:

- DisplayPort

- Ethernet SGMII
- PCIe
- SATA
- USB3.0

### Pin Description and Design Guidelines

Table 4-1 and Figure 4-1 detail the PCB guidelines for PS-GTR transceiver interfaces.

Table 4-1: PS-GTR Transceiver Quad Pin Descriptions

Pins	Direction	Descriptions
PS_MGTREFCLK[3:0]P PS_MGTREFCLK[3:0]N	In (Pad)	Differential clock input pin pair for the reference clock of the PS-GTR transceiver.
PS_MGTRRXP[3:0] PS_MGTRRXN[3:0]	In (Pad)	RXP and RXN are the differential input pairs for each of the PS-GTR receivers.
PS_MGTRTXP[3:0] PS_MGTRTXN[3:0]	Out (Pad)	TXP and TXN are the differential output pairs for each of the PS-GTR transmitters.
PS_MGTRREF	In (Pad)	Calibration input pin for the termination resistor calibration circuit.
PS_MGTRAVCC	In (Pad)	MGTRAVCC is the receiver, transmitter, and clock core power supply. The nominal voltage 0.85 VDC.
PS_MGTRAVTT	In (Pad)	MGTRAVTT is the I/O supply for the transmitter and receiver. The nominal voltage is 1.8 VDC.

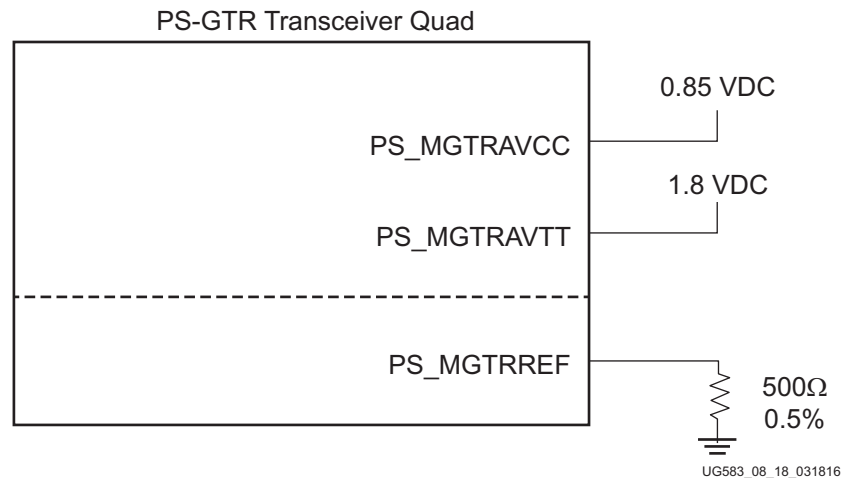


Figure 4-1: PS-GTR External Power Supply and Calibration Resistor Connections

Note relevant to Figure 4-1:

- The voltage values are nominal. See the *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics* (DS925) [Ref 22] for values and tolerances.

## Reference Clock

### Overview

This section focuses on the selection of the reference clock source or oscillator. An oscillator is characterized by:

- Frequency range
- Output voltage swing
- Jitter (deterministic, random, peak-to-peak)
- Rise and fall times
- Supply voltage and current
- Noise specification
- Duty cycle and duty-cycle tolerance
- Frequency stability

These characteristics are selection criteria when choosing an oscillator for PS-GTR transceiver design. Figure 4-2 illustrates the convention for the single-ended clock input voltage swing, peak-to-peak, as used in the PS-GTR transceiver portion of the *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics* (DS925) [Ref 22]. This figure is provided to show the contrast to the differential clock input voltage swing calculation shown in Figure 4-3.

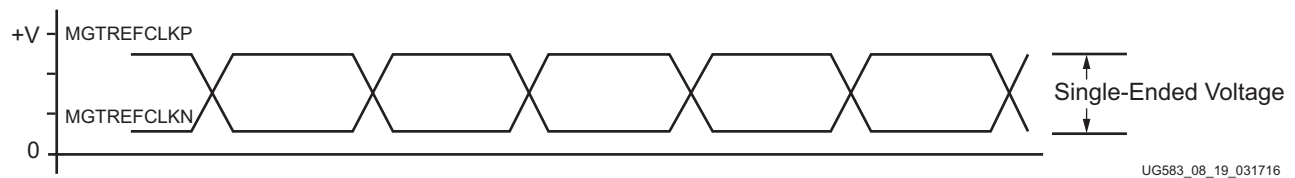


Figure 4-2: Single-Ended Clock Input Voltage Swing, Peak-to-Peak

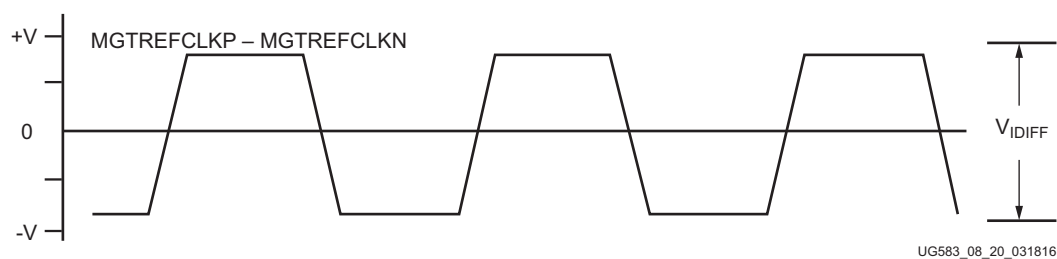
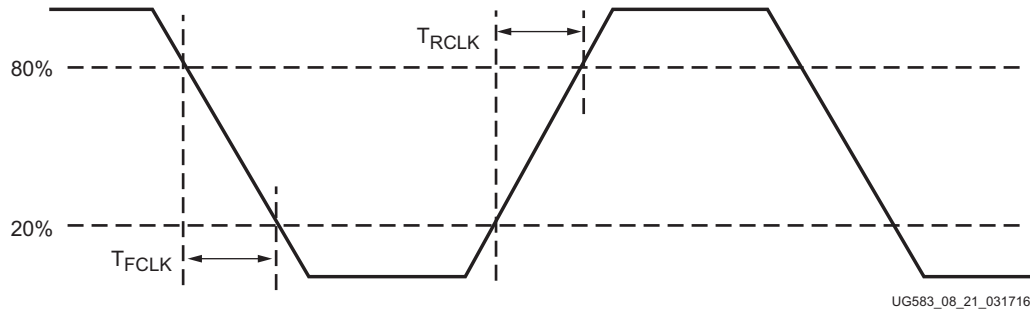


Figure 4-3: Differential Clock Input Voltage Swing, Peak-to-Peak

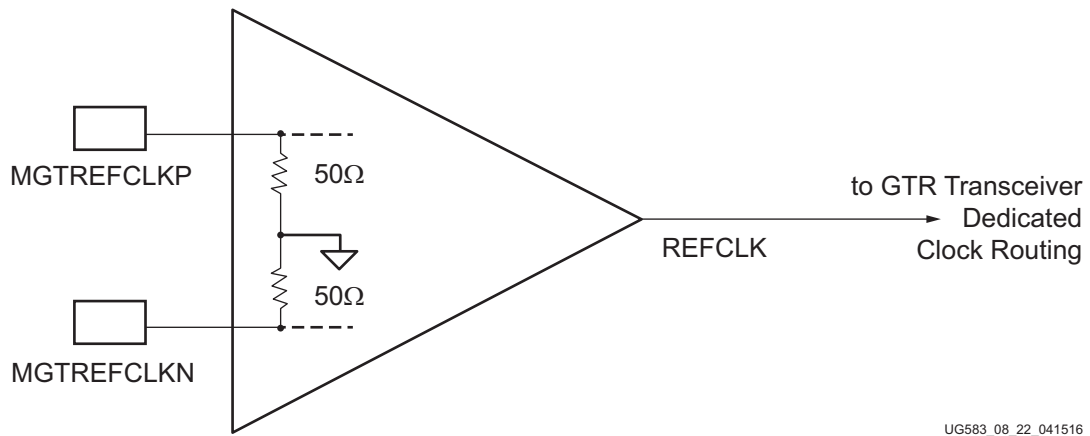
Figure 4-4 shows the rise and fall time convention of the reference clock.



UG583\_08\_21\_031716

Figure 4-4: Rise and Fall Times

Figure 4-5 illustrates the internal details of the MGTREFCLK input buffer. The dedicated differential reference clock input pin pair MGTREFCLKP/MGTREFCLKN is internally terminated with a 100Ω differential impedance. The common mode voltage of this differential reference clock input pair is ground or nominal 0.0V. See the *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics (DS925)* for exact specifications.



UG583\_08\_22\_041516

Figure 4-5: MGTREFCLK Input Buffer Details

## Reference Clock Interface

### LVDS

Figure 4-6 shows how an LVDS, current mode logic (CML), or high-speed current steering logic (HCSL) oscillator is connected to a reference clock input of the PS-GTR transceiver.

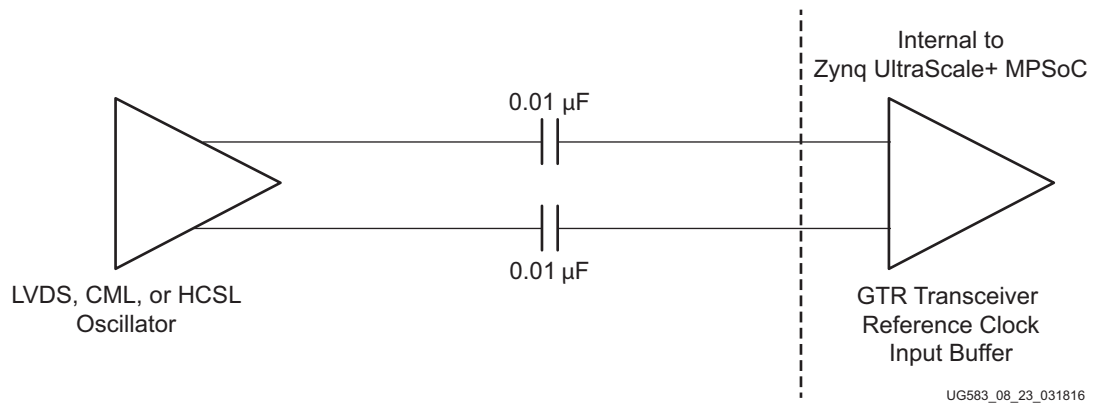


Figure 4-6: Interfacing an LVDS, CML, or HCSL Oscillator to the Zynq UltraScale+ MPSoC PS-GTR Transceiver Reference Clock Input

### LVPECL

Figure 4-7 shows how a low-voltage positive emitter-coupled logic (LVPECL) oscillator is connected to a reference clock input of the PS-GTR transceiver.

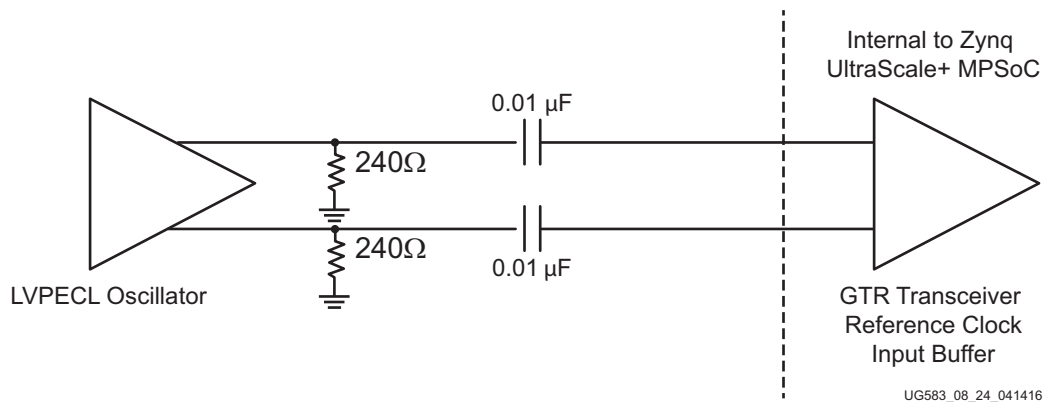


Figure 4-7: Interfacing an LVPECL Oscillator to the Zynq UltraScale+ MPSoC PS-GTR Transceiver Reference Clock Input

## AC Coupled Reference Clock

AC coupling of the oscillator reference clock output to the PS-GTR transceiver reference clock input serves multiple purposes:

- Blocking a DC current between the oscillator and the PS-GTR transceiver reference clock dedicated reference clock input pins (which also reduces power consumption of both parts)
- Common-mode voltage independence
- The AC coupling capacitor forms a high-pass filter with the on-chip termination that attenuates a wander of the reference clock

To minimize noise and power consumption, external AC coupling capacitors between the sourcing oscillator and the PS-GTR transceiver dedicated reference clock input pins are required.

## Unused Reference Clocks

If the reference clock input is not used, the reference clock input pins can be left unconnected.

## Reference Clock Power

The PS-GTR transceiver reference clock input circuit is powered by PS\_MGTRAVCC. Excessive noise on this supply has a negative impact on the performance of any PS-GTR transceivers that use the reference clock from this circuit.

## Power Supply Noise

The PS-GTR transceiver power supplies PS\_MGTRAVCC and PS\_MGTRAVTT must not have noise greater than 10 mV<sub>pp</sub> from 10 kHz to 80 MHz. Any noise on these power supplies that exceeds this requirement might affect the performance of the transceiver. If the noise on the power supply is less than or equal to this requirement, the performance of the transceiver is not negatively impacted.



## PCB Design Checklist

Table 4-2 is a checklist of items that can be used to design and review any Zynq UltraScale+ MPSoC PS-GTR transceiver schematic and layout.

Table 4-2: PCB Design Checklist for PS-GTR

Pins	Recommendations
PS_MGTREFCLK0P PS_MGTREFCLK0N PS_MGTREFCLK1P PS_MGTREFCLK1N PS_MGTREFCLK2P PS_MGTREFCLK2N PS_MGTREFCLK3P PS_MGTREFCLK3N	<ul style="list-style-type: none"> <li>Use AC coupling capacitors for connection to oscillator.</li> <li>For AC coupling capacitors, see <a href="#">Reference Clock Interface, page 183</a>. The recommended value for LVDS is 10 nF.</li> <li>Reference clock traces should be provided enough clearance to eliminate crosstalk from adjacent signals.</li> <li>Reference clock oscillator output must comply with the minimum/maximum input amplitude requirements for these input pins. See the PS-GTR Transceiver Clock DC Input Level Specifications table in <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics (DS925)</i> [Ref 22].</li> <li>If the reference clock input is not used, leave the associated pin pair unconnected or tie to ground.</li> <li>Traces should be routed to 50Ω single-ended/100Ω differential.</li> </ul>
PS_MGTRRX0/ PS_MGTRRXN0 PS_MGTRRX1/ PS_MGTRRXN1 PS_MGTRRX2/ PS_MGTRRXN2 PS_MGTRRX3/ PS_MGTRRXN3	<ul style="list-style-type: none"> <li>Use AC coupling capacitors for connection to the transmitter. The recommended value for AC coupling capacitors is 100 nF.</li> <li>Receiver data traces should be provided enough clearance to eliminate crosstalk from adjacent signals.</li> <li>If a receiver is not used, leave the associated pin pair unconnected or tie to ground.</li> <li>Trace match P/N to 10% (or less) of the UI.</li> <li>Traces should be routed to 50Ω single-ended/100Ω differential.</li> </ul>
PS_MGTRTX0/ PS_MGTRTXN0 PS_MGTRTX1/ PS_MGTRTXN1 PS_MGTRTX2/ PS_MGTRTXN2 PS_MGTRTX3/ PS_MGTRTXN3	<ul style="list-style-type: none"> <li>The transmitter should be AC coupled to the receiver. The recommended value for the AC coupling capacitors is 100 nF.</li> <li>Transmitter data traces should be provided enough clearance to eliminate crosstalk from adjacent signals.</li> <li>If a transmitter is not used, leave the associated pin pair unconnected or tie to ground.</li> <li>Trace match P/N to 10% (or less) of the UI.</li> <li>Traces should be routed to 50Ω single-ended/100Ω differential.</li> </ul>
PS_MGTRREF	<ul style="list-style-type: none"> <li>Connect to a 500Ω 0.5% resistor to this pin with the other terminal of the resistor connected to ground.</li> <li>Can be left unconnected or tied to ground if the transceivers will not be used.</li> </ul>

Table 4-2: PCB Design Checklist for PS-GTR (Cont'd)

Pins	Recommendations
PS_MGTRAVCC	<ul style="list-style-type: none"> <li>• The nominal voltage is 0.85 VDC.</li> <li>• See the <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics</i> (DS925) [Ref 22] for power supply voltage tolerances.</li> <li>• Not sharing this supply with other non-transceiver loads reduces the chance of exceeding the noise requirement for the power supply.</li> <li>• The following filter is recommended:                             <ul style="list-style-type: none"> <li>◦ One of 4.7 <math>\mu</math>F 10% ceramic</li> </ul> </li> <li>• For optimal performance, power supply noise must be less than 10 mV<sub>pp</sub>. If the noise on the power supply is less than or equal to this requirement, the performance of the transceiver will not be negatively impacted.</li> <li>• If all of the PS-GTR transceivers in the Quad are not used, the associated power pins can be left unconnected or tied to ground.</li> <li>• For power consumption, refer to the <a href="#">Xilinx Power Estimator (XPE)</a>.</li> </ul>
PS_MGTRAVTT	<ul style="list-style-type: none"> <li>• The nominal voltage is 1.8 VDC.</li> <li>• See <i>Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics</i> (DS925) [Ref 22] for power supply voltage tolerances.</li> <li>• Not sharing this supply with other non-transceiver loads reduces the chance of exceeding the noise requirement for the power supply.</li> <li>• The following filter is recommended:                             <ul style="list-style-type: none"> <li>◦ One of 4.7 <math>\mu</math>F 10% ceramic</li> </ul> </li> <li>• If all of the PS-GTR transceivers in the Quad are not used, the associated power pins can be left unconnected or tied to ground.</li> <li>• For power consumption, refer to the Zynq UltraScale+ MPSoC XPower Estimator at <a href="#">Xilinx Power Estimator (XPE)</a>.</li> </ul>

# Migration between UltraScale Devices and Packages

When migrating a design from one UltraScale device to another, special attention must be paid to potential differences between the two devices. Refer to the checklist below, and the subsequent sections that explain each item in the checklist.

---

## UltraScale Device Migration Checklist

1. Review footprint compatibility, including banks that are bonded vs. unbonded.
2. Review package dimensions, including height and overhang.
3. Review I/O and transceiver Quad numbering and locations.
4. Review HPIO to HRIO and HRIO to HPIO migration.
5. Review GTH to GTY and GTY to GTH transceiver migration.
6. Review the super logic region (SLR) boundaries.
7. Review monolithic to stacked silicon migration as it pertains to configuration banks.
8. Review memory interface locations to maintain bank grouping.
9. Review DCI cascade and internal  $V_{REF}$  requirements.
10. Review system monitor requirements and pin differences.
11. Review decoupling capacitor requirements.
12. Review PCI Express® requirements and block locations.
13. Review integrated 100G Ethernet requirements and block locations.
14. Review Interlaken requirements and block locations.
15. Review power supply and thermal requirements.
16. Review pin flight time differences between devices.

## 1. Footprint Compatibility between Packages

Any two packages with the same footprint identifier code are footprint compatible. The footprint identifier code consists of a package designator code and BGA pin count information (such as A2104, B2104, or C2104). A customer that wants pin compatibility across any two devices in a given package should select a footprint compatible package such that they can be designed to be electrically compatible. If migrating from a bigger device to a smaller device within the same package, some I/O and transceiver banks and their pins might be unbonded or not exist on the smaller device. Refer to the Footprint Compatibility, I/O Bank Migration, and Transceiver Quad Migration tables in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) [Ref 9] to determine which UltraScale devices are footprint compatible, as well as the extent to which the various banks might become unbonded or not exist.

### Examples

Figure 5-1 shows a snippet of the Footprint Compatibility table from *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575). The XCKU035 in the A1156<sup>(1)</sup> package can only be compatible with the XCKU040 and XCKU060 in the A1156 package, but not with any of the devices in the A1517. A design created with the XCKU035 in the A1156 package, for example, can be footprint compatible with any bigger A1156 device.

A1156	XCKU035	XCKU040	XCKU060
A1517	XCKU060	XCKU085	XCKU115

UG583\_c3\_14\_022515

Figure 5-1: Portion of Footprint Compatibility Table

1. For purposes of understanding footprint compatibility and migration, only the last letter in the package name is relevant. For example, the FFVA2104 and FLVA2104 are essentially the same as far as footprint compatibility and migration due to the last letter "A" being the same.

If migrating from a bigger device to a smaller device within the same package, there exists the potential for some I/O and transceiver banks to not be bonded out or present on the smaller device. [Figure 5-2](#) shows a snippet of the Transceiver Quad Migration table from *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification (UG575)*. Transceiver banks 120, 121, 122, 220, 221, and 222 are present on the XCVU160 and XCVU190 C2104 devices, but those transceiver banks are not available on the XCVU080, XCVU095, and XCVU125 C2104 devices.

Power Supply Group:	LS	LS	LS	LC	LC	LC	LC	LC	LN	LN	LN	LN	LN	RS	RS	RS	RC	RC	RC	RC	RC	RN	RN	RN	RN	RN
FFVC2104	XCVU080			124	125	126	127	128	129	130	131						224	225	226	227	228	229	230	231		
	XCVU095																									
FLVC2104	XCVU125			124	125	126	127	128	129	130	131	132	133				224	225	226	227	228	229	230	231	232	233
				RCAL	RCAL					RCAL									RCAL				RCAL			
FLGC2104	XCVU160	120		124	125	126	127	128	129	130	131	132	133	220	221	222	224	225	226	227	228	229	230	231	232	233
	XCVU190	RCAL	121	122	RCAL					RCAL				RCAL					RCAL				RCAL			

UG583\_c3\_15\_022515

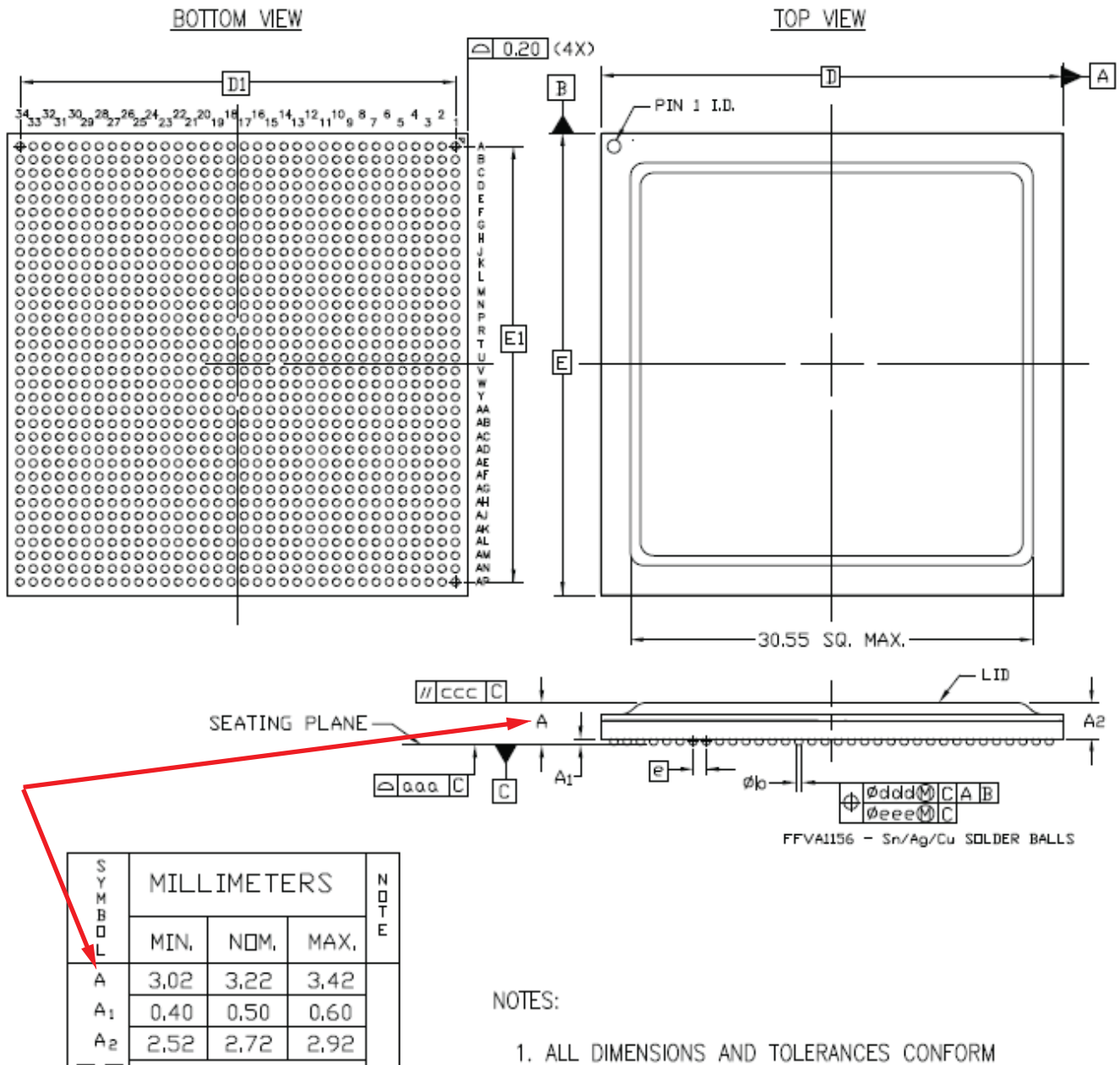
Figure 5-2: Transceiver Quad Availability within C2104 Packages

## 2. Package Dimensions

Package dimensions for UltraScale devices can vary from one package to another, so care must be taken in system design to ensure that changes in length, width, or height do not interfere with other components in the system. Some packages are also “overhang” packages in that their lengths and widths extend further beyond the pin array than other devices with similar pin counts. If migrating from a smaller package to a bigger package, ensure that an appropriate keep-out area is in place so that no capacitors or other components interfere with the bigger outline. Refer to the Mechanical Drawings chapter in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification (UG575)* [Ref 9] for the dimensions of the various UltraScale device packages.

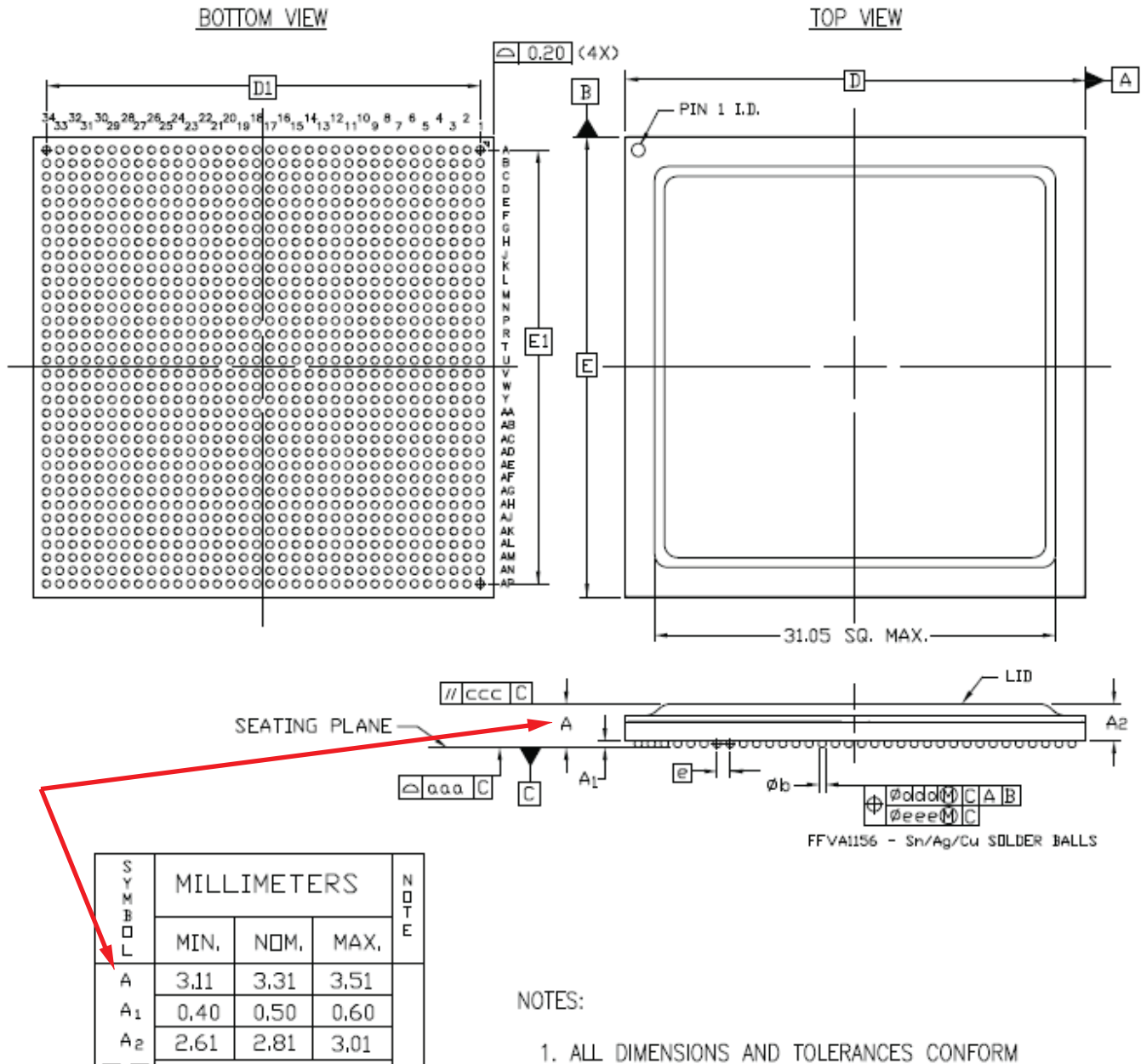
### Example

According to the Mechanical Drawings chapter in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification (UG575)*, the FFVA1156 package used for the XCKU035 and XCKU040 ([Figure 5-3](#)) has a smaller height than the FFVA1156 package used for the XCKU060 ([Figure 5-4](#)).



UG583\_c3\_16\_022515

Figure 5-3: Portion of Package Drawing for FFVA1156 (XCKU035 and XCKU040)



UG583\_c3\_17\_022515

Figure 5-4: Portion of Package Drawing for FFVA1156 (XCKU060)

### 3. I/O Bank and Transceiver Quad Numbers

Certain UltraScale devices that are footprint compatible in a package might have different I/O bank and transceiver Quad numbers associated with the same package pins. Depending on where these banks are physically located on the die, certain multi-bank interfaces (such as memory or transceiver) could be impacted in regards to migrating from one device to the other. If a bank number change also involves the bank moving from one column on the die to another, a multi-bank interface that requires both banks to be in the same column could be impacted.

Refer to the I/O Bank Migration and Transceiver Quad Migration tables in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) [Ref 9] for the bank and transceiver Quad numbering differences. To determine bank locations on a die, refer to the Die Level Bank Numbering Overview section of *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575).

The example below shows a successful migration path despite both bank number changes and banks being in different columns between the two different devices.

#### Example: Planning Ahead for Bank Number Changes

As shown in [Figure 5-5](#), banks 69, 70, and 71 in the XCVU095-FFVD1517 connect to the same pins as banks 71, 72, and 73 in the XCVU125-FLVD1517. [Figure 5-6](#) confirms that the multi-bank interface remains intact despite the differing locations on each die. All three banks remain contiguous on each respective die. This would result in a successful migration for interfaces that utilize these three banks, with pin flight times being the only potential differences.

FFVD1517	XCVU080 XCVU095	Right								
		84	94	65	66	67	69	70	71	
FLVD1517	XCVU125	Right-South						Right-North		
		84	94	65	66	67	71	72	73	
	XCKU115	84	94	65	66	67	71	72	73	

UG583\_c3\_18\_022515

Figure 5-5: Portion of I/O Bank Migration Table



UG583\_c3\_19\_022515

Figure 5-6: XCVU095 (L) and XCVU125 (R) Bank Locations

## 4. HP/HR Migration

In a small number of cases, an HR bank in one device might become an HP bank in an otherwise footprint-compatible package, and vice-versa. If migration is desired at the initial design stage, it is advisable to use only 1.8V I/O standards so that any move from HR to HP banks is not affected. If this is not possible, avoid HR banks that can become HP banks. Refer to the I/O Bank Migration table in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) to determine which banks could be affected, or refer to [Table 5-1](#).

Table 5-1: Interchanged HP and HR Banks

Package	Bank(s)	HP	HR
A2104	70	XCVU125	XCKU115

In addition, there are functional differences between HP and HR I/Os, such as whether DCI is available (HP only) and I/O standard availability. Refer to the Supported Features in the HR and HP I/O Banks table in *UltraScale Architecture SelectIO Resources User Guide* (UG571) [[Ref 10](#)] to determine the capabilities and I/O standards available with each type of I/O.

When it is not possible to avoid migrating from an HR bank to an HP bank, the new HP I/Os might need to be level translated due to their maximum 1.8V output. Refer to *Interfacing 7 Series FPGAs High-Performance I/O Banks with 2.5V and 3.3V I/O Standards* (XAPP520) [[Ref 11](#)], which gives many methodologies for interfacing HP I/Os with higher voltage I/Os.

## 5. GTH/GTY Transceiver Migration

GTH and GTY transceivers are designed to support pin migration and have identical differential signal and reference clock bond-out. GTY transceivers offer higher performance

than GTH transceivers, and thus have different operating characteristics. For example, GTY transceivers are capable of supporting line rates up to 30.5 Gb/s, while GTH transceivers can support up to 16.375 Gb/s. If moving to/from GTH/GTY transceivers, these differences and any others must be accounted for. Refer to *Virtex UltraScale FPGAs Data Sheet: DC and AC Switching Characteristics* (DS893) [Ref 2], *UltraScale Architecture GTH Transceivers User Guide* (UG576) [Ref 6], and *UltraScale Architecture GTY Transceivers User Guide* (UG578) [Ref 7] to learn more about the capabilities of each type of transceiver.

To determine if a migration path will result in any swapping between GTH and GTY transceivers (or vice versa), refer to the Transceiver Quad Migration table in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) [Ref 9].

**Example**

Figure 5-7 shows a snippet of the Transceiver Quad Migration table from *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575). Transceiver banks 126, 127, 128, 131, 132, and 133 are shown as GTH transceivers in the XCKU115 A2104 device, while the corresponding transceiver banks are GTY transceivers in the XCVU080, XCVU095, and XCVU125 A2104 devices. Note how the transceiver numbers are different, indicating that these are different transceiver Quads that bond to the same pins (see 3. I/O Bank and Transceiver Quad Numbers, page 192).

Power Supply Group:		LS	LS	LS	LN	LN	LN	RS	RS	RS	RS	RN	RN	RN
FFVA2104	XCVU080	<b>125</b>	<b>126</b>	<b>127</b>	<b>128</b>	<b>129</b>	<b>130</b>	224	225	226	227	228	229	230
	XCVU095	<b>125</b>	<b>126</b>	<b>127</b>	<b>128</b>	<b>129</b>	<b>130</b>	224	225	226	227	228	229	230
FLVA2104	XCVU125	<b>125</b>	<b>126</b>	<b>127</b>	<b>130</b>	<b>131</b>	<b>132</b>	224	225	226	227	231	232	233
	XCKU115	126	127	128	131	132	133	224	225	226	227	231	232	233

UG583\_c3\_20\_022515

Figure 5-7: Snippet of Transceiver Quad Migration Table (GTY in bold)

**MGTRREF and MGTAVTTRCAL**

Some MGTRREF and MGTAVTTRCAL pins might have slightly different labels in different devices within the same package footprint. For example, pin A39 in the B1760 package is either labeled MGTAVTTRCAL\_L or MGTAVTTRCAL\_LN depending on the device. The functionality of the pin does not change, but the change in label should be noted.

**6. SLR Migration**

Some UltraScale devices are implemented with stacked silicon interconnect (SSI) technology, and certain implementations can only be contained within one SLR. SLRs are divided by a passive silicon interposer. Migrations that span multiple SLRs are not recommended. For example, DDR4 interfaces running at the maximum data rate must be

contained within one SLR. See 8. Memory Interface Migration, page 197 for a specific example of where the interposer can break an existing implementation.

The bank drawings in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) identify SLRs connected by interposers.

**Example**

Figure 5-8 shows three SLRs for the XCVU190 device, each separated from the next by a silicon interposer.

GTY Quad 133 X0Y56–X0Y59	CMAC X0Y8	Bank 53 HP I/O	PLL[28:29] CMT MMCM[14]	Bank 73 HP I/O	PLL[58:59] CMT MMCM[29]	PCIe X0Y5	GTH Quad 233 X1Y56–X1Y59
GTY Quad 132 X0Y52–X0Y55	CMAC X0Y7	Bank 52 HP I/O	PLL[26:27] CMT MMCM[13]	Bank 72 HP I/O	PLL[56:57] CMT MMCM[28]	ILKN X1Y8	GTH Quad 232 X1Y52–X1Y55
GTY Quad 131 X0Y48–X0Y51	ILKN X0Y7	Bank 51 HP I/O	PLL[24:25] CMT MMCM[12]	Bank 71 HP I/O	PLL[54:55] CMT MMCM[27]	SYSMON CFG	GTH Quad 231 X1Y48–X1Y51
GTY Quad 130 X0Y44–X0Y47	CMAC X0Y6	Bank 50 HP I/O	PLL[22:23] CMT MMCM[11]	Bank 70 HP I/O	PLL[52:53] CMT MMCM[26]	CFG	GTH Quad 230 X1Y44–X1Y47
GTY Quad 129 X0Y40–X0Y43	ILKN X0Y6	Bank 49 HP I/O	PLL[20:21] CMT MMCM[10]	Bank 69 HP I/O	PLL[50:51] CMT MMCM[25]	PCIe X0Y4	GTH Quad 229 X1Y40–X1Y43
SLR Crossing							
GTY Quad 128 X0Y36–X0Y39	CMAC X0Y5	Bank 48 HP I/O	PLL[18:19] CMT MMCM[09]	Bank 68 HP I/O	PLL[48:49] CMT MMCM[24]	PCIe X0Y3	GTH Quad 228 X1Y36–X1Y39
GTY Quad 127 X0Y32–X0Y35	CMAC X0Y4	Bank 47 HP I/O	PLL[16:17] CMT MMCM[08]	Bank 67 HP I/O	PLL[46:47] CMT MMCM[23]	ILKN X1Y5	GTH Quad 227 X1Y32–X1Y35
GTY Quad 126 X0Y28–X0Y31	ILKN X0Y4	Bank 46 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 66 HP I/O	PLL[44:45] CMT MMCM[22]	SYSMON CFG	GTH Quad 226 X1Y28–X1Y31
GTY Quad 125 X0Y24–X0Y27	CMAC X0Y3	Bank 45 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 65 HP I/O	PLL[42:43] CMT MMCM[21]	CFG	GTH Quad 225 X1Y24–X1Y27
GTY Quad 124 X0Y20–X0Y23	ILKN X0Y3	Bank 44 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 84/94 HR I/O	PLL[40:41] CMT MMCM[20]	PCIe X0Y2 (tandem)	GTH Quad 224 X1Y20–X1Y23
SLR Crossing							
GTY Quad 123 X0Y16–X0Y19	CMAC X0Y2	Bank 43 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 63 HP I/O	PLL[38:39] CMT MMCM[19]	PCIe X0Y1	GTH Quad 223 X1Y16–X1Y19
GTY Quad 122 X0Y12–X0Y15	CMAC X0Y1	Bank 42 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 62 HP I/O	PLL[36:37] CMT MMCM[18]	ILKN X1Y2	GTH Quad 222 X1Y12–X1Y15
GTY Quad 121 X0Y8–X0Y11	ILKN X0Y1	Bank 41 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 61 HP I/O	PLL[34:35] CMT MMCM[17]	SYSMON CFG	GTH Quad 221 X1Y8–X1Y11
GTY Quad 120 X0Y4–X0Y7	CMAC X0Y0	Bank 40 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 60 HP I/O	PLL[32:33] CMT MMCM[16]	CFG	GTH Quad 220 X1Y4–X1Y7
GTY Quad 119 X0Y0–X0Y3	ILKN X0Y0	Bank 39 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 59 HP I/O	PLL[30:31] CMT MMCM[15]	PCIe X0Y0	GTH Quad 219 X1Y0–X1Y3

UG583\_c3\_21\_022515

Figure 5-8: Bank Drawing for the XCVU190 Showing SLRs

## 7. Monolithic to Stacked Silicon Migration

All UltraScale devices have pins dedicated to configuration functions contained in I/O bank 0. There are also I/O pins in bank 65 (multifunction configuration bank) known as multifunction or multipurpose pins that can be used for configuration, but then convert to normal I/O pins after configuration is complete. Additionally, in SSI devices, pins in bank 60 and bank 70 (as present) have restrictions during configuration similar to multifunction pins. However, pins in these banks do not have any configuration functions. See the Configuration Details chapter in *UltraScale Architecture Configuration User Guide* (UG570) [Ref 12] to learn about how the behavior of these pins might affect your design.

### Example: Migration from Monolithic XCKU060 to SSI XCKU115 in A1517 Package

Figure 5-9 shows the bank drawing for the monolithic XCKU060 device. The multifunction configuration bank (65) is circled. Figure 5-10 shows the bank drawing for the SSI XCKU115 device. The multifunction configuration bank (65) is circled along with bank 70. While bank 65 is the actual multifunction configuration bank, bank 70 shares the same restrictions during configuration.

GTH Quad 128 X0Y16–X0Y19	Bank 48 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 68 HP I/O	PLL[22:23] CMT MMCM[11]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTH Quad 127 X0Y12–X0Y15	Bank 47 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 67 HP I/O	PLL[20:21] CMT MMCM[10]	CMAC X0Y0	GTH Quad 227 X1Y12–X1Y15
GTH Quad 126 X0Y8–X0Y11	Bank 46 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 66 HP I/O	PLL[18:19] CMT MMCM[09]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
Bank 25 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 45 HP I/O	Bank 65 HR I/O	PLL[16:17] CMT MMCM[08]	CFG	GTH Quad 225 X1Y4–X1Y7
Bank 24 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 44 HP I/O	Bank 64 HR I/O	PLL[14:15] CMT MMCM[07]	PCIe X0Y0 (tandem)	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_22\_022515

Figure 5-9: Bank Drawing for XCKU060 Showing Multifunction Configuration Bank 65

GTH Quad 133 X0Y36–X0Y39	Bank 53 HP I/O	PLL[26:27] CMT MMCM[13]	Bank 73 HP I/O	PLL[46:47] CMT MMCM[23]	PCIe X0Y3	GTH Quad 233 X1Y36–X1Y39
GTH Quad 132 X0Y32–X0Y35	Bank 52 HP I/O	PLL[24:25] CMT MMCM[12]	Bank 72 HP I/O	PLL[44:45] CMT MMCM[22]	CMAC X0Y1	GTH Quad 232 X1Y32–X1Y35
GTH Quad 131 X0Y28–X0Y31	Bank 51 HP I/O	PLL[22:23] CMT MMCM[11]	Bank 71 HP I/O	PLL[42:43] CMT MMCM[21]	SYSMON CFG	GTH Quad 231 X1Y28–X1Y31
Bank 30 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 50 HP I/O	Bank 70 HR I/O	PLL[40:41] CMT MMCM[20]	CFG	GTH Quad 230 X1Y24–X1Y27
Bank 29 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 49 HP I/O	Bank 69 HR I/O	PLL[38:39] CMT MMCM[19]	PCIe X0Y2	GTH Quad 229 X1Y20–X1Y23
SLR Crossing						
GTH Quad 128 X0Y16–X0Y19	Bank 48 HP I/O	PLL[16:17] CMT MMCM[08]	Bank 68 HP I/O	PLL[36:37] CMT MMCM[18]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTH Quad 127 X0Y12–X0Y15	Bank 47 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 67 HP I/O	PLL[34:35] CMT MMCM[17]	CMAC X0Y0	GTH Quad 227 X1Y12–X1Y15
GTH Quad 126 X0Y8–X0Y11	Bank 46 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 66 HP I/O	PLL[32:33] CMT MMCM[16]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
Bank 25 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 45 HP I/O	Bank 65 HR I/O	PLL[30:31] CMT MMCM[15]	CFG	GTH Quad 225 X1Y4–X1Y7
Bank 24 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 44 HP I/O	Bank 64 HR I/O	PLL[28:29] CMT MMCM[14]	PCIe X0Y0 (tandem)	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_23\_022515

Figure 5-10: Bank Drawing for XCKU115 Showing Multifunction Configuration Bank 65, along with Bank 70

## 8. Memory Interface Migration

When migrating a memory interface from one device to another, it is important to verify that all pinout and banking restrictions remain valid. For example, most memory interfaces require the use of two or three consecutive banks in one column. In addition, memory interfaces in UltraScale devices should not cross SLR boundaries. Refer to *UltraScale Architecture FPGAs Memory IP Product Guide* (PG150) [Ref 13] for a more comprehensive guide regarding the proper generation and verification of UltraScale memory interfaces.

### Example

A memory interface in the XCVU095 that spans banks 47, 48, and 49 (Figure 5-11) would NOT migrate over to the XCVU125 because banks 48 and 49 are separated by an SLR in the XCVU125 (Figure 5-12). A better option for the memory interface would be banks 44, 45, and 46, because both banks remain contiguous on both devices.

GTY Quad 131 X0Y28–X0Y31	CMAC X0Y3	Bank 51 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 71 HP I/O	PLL[30:31] CMT MMCM[15]	PCIe X0Y3	GTH Quad 231 X1Y28–X1Y31
GTY Quad 130 X0Y24–X0Y27	ILKN X0Y4	Bank 50 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 70 HP I/O	PLL[28:29] CMT MMCM[14]	ILKN X1Y4	GTH Quad 230 X1Y24–X1Y27
GTY Quad 129 X0Y20–X0Y23	CMAC X0Y2	Bank 49 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 69 HP I/O	PLL[26:27] CMT MMCM[13]	PCIe X0Y2	GTH Quad 229 X1Y20–X1Y23
GTY Quad 128 X0Y16–X0Y19	ILKN X0Y3	Bank 48 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 68 HP I/O	PLL[24:25] CMT MMCM[12]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTY Quad 127 X0Y12–X0Y15	CMAC X0Y1	Bank 47 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 67 HP I/O	PLL[22:23] CMT MMCM[11]	ILKN X1Y2	GTH Quad 227 X1Y12–X1Y15
GTY Quad 126 X0Y8–X0Y11	ILKN X0Y1	Bank 46 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 66 HP I/O	PLL[20:21] CMT MMCM[10]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
GTY Quad 125 X0Y4–X0Y7	CMAC X0Y0	Bank 45 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 65 HP I/O	PLL[18:19] CMT MMCM[09]	CFG	GTH Quad 225 X1Y4–X1Y7
GTY Quad 124 X0Y0–X0Y3	ILKN X0Y0	Bank 44 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 84/94 HR I/O	PLL[16:17] CMT MMCM[08]	PCIe X0Y0 (tandem)	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_24\_022515

Figure 5-11: Bank Drawing of XCVU095 Showing Consecutive Banks 47, 48, and 49

GTY Quad 133 X0Y36–X0Y39	CMAC X0Y5	Bank 53 HP I/O	PLL[18:19] CMT MMCM[09]	Bank 73 HP I/O	PLL[38:39] CMT MMCM[19]	PCIe X0Y3	GTH Quad 233 X1Y36–X1Y39
GTY Quad 132 X0Y32–X0Y35	CMAC X0Y4	Bank 52 HP I/O	PLL[16:17] CMT MMCM[08]	Bank 72 HP I/O	PLL[36:37] CMT MMCM[18]	ILKN X1Y5	GTH Quad 232 X1Y32–X1Y35
GTY Quad 131 X0Y28–X0Y31	ILKN X0Y4	Bank 51 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 71 HP I/O	PLL[34:35] CMT MMCM[17]	SYSMON CFG	GTH Quad 231 X1Y28–X1Y31
GTY Quad 130 X0Y24–X0Y27	CMAC X0Y3	Bank 50 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 70 HP I/O	PLL[32:33] CMT MMCM[16]	CFG	GTH Quad 230 X1Y24–X1Y27
GTY Quad 129 X0Y20–X0Y23	ILKN X0Y3	Bank 49 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 69 HR I/O	PLL[30:31] CMT MMCM[15]	PCIe X0Y2	GTH Quad 229 X1Y20–X1Y23
SLR Crossing							
GTY Quad 128 X0Y16–X0Y19	CMAC X0Y2	Bank 48 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 68 HP I/O	PLL[28:29] CMT MMCM[14]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTY Quad 127 X0Y12–X0Y15	CMAC X0Y1	Bank 47 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 67 HP I/O	PLL[26:27] CMT MMCM[13]	ILKN X1Y2	GTH Quad 227 X1Y12–X1Y15
GTY Quad 126 X0Y8–X0Y11	ILKN X0Y1	Bank 46 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 66 HP I/O	PLL[24:25] CMT MMCM[12]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
GTY Quad 125 X0Y4–X0Y7	CMAC X0Y0	Bank 45 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 65 HP I/O	PLL[22:23] CMT MMCM[11]	CFG	GTH Quad 225 X1Y4–X1Y7
GTY Quad 124 X0Y0–X0Y3	ILKN X0Y0	Bank 44 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 84/94 HR I/O	PLL[20:21] CMT MMCM[10]	PCIe X0Y0	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_25\_022515

Figure 5-12: Bank Drawing of XCVU125 Showing SLR between Banks 47, 48, and 49

## 9. DCI Cascade and Internal V<sub>REF</sub> Features

DCI Cascade and Internal V<sub>REF</sub> are features that span multiple banks in an HPIO column. When migrating, if a bank (or banks) moves to another column or if it crosses an SLR/interposer boundary, it is no longer allowed as part of the original DCI cascade or internal V<sub>REF</sub> group. Each SLR contains its own DCI and V<sub>REF</sub> circuitry. See the DCI Cascading and Internal V<sub>REF</sub> sections in *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10] to learn more about these features.

### Example

Figure 5-13 highlights an HPIO column in the XCVU095. DCI cascade or internal V<sub>REF</sub> can be used in this column among some or all of these banks. If migrating to the XCVU125 device, the interposer does not allow DCI cascade or internal V<sub>REF</sub> to cross between banks 48 and 49, as shown in Figure 5-14.

GTY Quad 131 X0Y28–X0Y31	CMAC X0Y3	Bank 51 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 71 HP I/O	PLL[30:31] CMT MMCM[15]	PCIe X0Y3	GTH Quad 231 X1Y28–X1Y31
GTY Quad 130 X0Y24–X0Y27	ILKN X0Y4	Bank 50 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 70 HP I/O	PLL[28:29] CMT MMCM[14]	ILKN X1Y4	GTH Quad 230 X1Y24–X1Y27
GTY Quad 129 X0Y20–X0Y23	CMAC X0Y2	Bank 49 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 69 HP I/O	PLL[26:27] CMT MMCM[13]	PCIe X0Y2	GTH Quad 229 X1Y20–X1Y23
GTY Quad 128 X0Y16–X0Y19	ILKN X0Y3	Bank 48 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 68 HP I/O	PLL[24:25] CMT MMCM[12]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTY Quad 127 X0Y12–X0Y15	CMAC X0Y1	Bank 47 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 67 HP I/O	PLL[22:23] CMT MMCM[11]	ILKN X1Y2	GTH Quad 227 X1Y12–X1Y15
GTY Quad 126 X0Y8–X0Y11	ILKN X0Y1	Bank 46 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 66 HP I/O	PLL[20:21] CMT MMCM[10]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
GTY Quad 125 X0Y4–X0Y7	CMAC X0Y0	Bank 45 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 65 HP I/O	PLL[18:19] CMT MMCM[09]	CFG	GTH Quad 225 X1Y4–X1Y7
GTY Quad 124 X0Y0–X0Y3	ILKN X0Y0	Bank 44 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 84/94 HR I/O	PLL[16:17] CMT MMCM[08]	PCIe X0Y0 (tandem)	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_26\_022515

Figure 5-13: XCVU095 Showing HPIO Column

GTY Quad 133 X0Y36–X0Y39	CMAC X0Y5	Bank 53 HP I/O	PLL[18:19] CMT MMCM[09]	Bank 73 HP I/O	PLL[38:39] CMT MMCM[19]	PCIe X0Y3	GTH Quad 233 X1Y36–X1Y39
GTY Quad 132 X0Y32–X0Y35	CMAC X0Y4	Bank 52 HP I/O	PLL[16:17] CMT MMCM[08]	Bank 72 HP I/O	PLL[36:37] CMT MMCM[18]	ILKN X1Y5	GTH Quad 232 X1Y32–X1Y35
GTY Quad 131 X0Y28–X0Y31	ILKN X0Y4	Bank 51 HP I/O	PLL[14:15] CMT MMCM[07]	Bank 71 HP I/O	PLL[34:35] CMT MMCM[17]	SYSMON CFG	GTH Quad 231 X1Y28–X1Y31
GTY Quad 130 X0Y24–X0Y27	CMAC X0Y3	Bank 50 HP I/O	PLL[12:13] CMT MMCM[06]	Bank 70 HP I/O	PLL[32:33] CMT MMCM[16]	CFG	GTH Quad 230 X1Y24–X1Y27
GTY Quad 129 X0Y20–X0Y23	ILKN X0Y3	Bank 49 HP I/O	PLL[10:11] CMT MMCM[05]	Bank 69 HR I/O	PLL[30:31] CMT MMCM[15]	PCIe X0Y2	GTH Quad 229 X1Y20–X1Y23
SLR Crossing							
GTY Quad 128 X0Y16–X0Y19	CMAC X0Y2	Bank 48 HP I/O	PLL[08:09] CMT MMCM[04]	Bank 68 HP I/O	PLL[28:29] CMT MMCM[14]	PCIe X0Y1	GTH Quad 228 X1Y16–X1Y19
GTY Quad 127 X0Y12–X0Y15	CMAC X0Y1	Bank 47 HP I/O	PLL[06:07] CMT MMCM[03]	Bank 67 HP I/O	PLL[26:27] CMT MMCM[13]	ILKN X1Y2	GTH Quad 227 X1Y12–X1Y15
GTY Quad 126 X0Y8–X0Y11	ILKN X0Y1	Bank 46 HP I/O	PLL[04:05] CMT MMCM[02]	Bank 66 HP I/O	PLL[24:25] CMT MMCM[12]	SYSMON CFG	GTH Quad 226 X1Y8–X1Y11
GTY Quad 125 X0Y4–X0Y7	CMAC X0Y0	Bank 45 HP I/O	PLL[02:03] CMT MMCM[01]	Bank 65 HP I/O	PLL[22:23] CMT MMCM[11]	CFG	GTH Quad 225 X1Y4–X1Y7
GTY Quad 124 X0Y0–X0Y3	ILKN X0Y0	Bank 44 HP I/O	PLL[00:01] CMT MMCM[00]	Bank 84/94 HR I/O	PLL[20:21] CMT MMCM[10]	PCIe X0Y0	GTH Quad 224 X1Y0–X1Y3

UG583\_c3\_27\_022515

Figure 5-14: XCVU125 Showing HPIO Column



## 10. System Monitor

Each I/O bank provides access to up to sixteen of the system monitor differential auxiliary analog input pairs. Some banks do not contain full access, so choose banks carefully so that any desired system monitor analog input pairs are available for device migration.

Refer to [Table 5-2](#) for a summary of UltraScale banks that do not contain full access to the sixteen differential auxiliary analog input pairs. If a particular device/bank is not listed, then full access should be assumed for that bank.

**Table 5-2: Unavailable Analog Input Pairs**

Package	Limited System Monitor Analog Access (Bank <sup>(1)</sup> )
D1517	XCKU115 (67) XCVU095 (67) XCVU125 (67)
B1760	XCKU115 (53) XCVU095 (51) XCVU125 (53)
B2104	XCKU115 (68) XCVU095 (68) XCVU125 (68) XCVU190 (68)
A2577	XCVU190 (66 <sup>(2)</sup> , 68)

**Notes:**

1. No access to pairs 0, 1, 2, 3, 8, 9, 10, 11 unless noted.
2. No access to any pairs.

In addition, analog auxiliary pair access can be found within each device's package file. Refer to the Package Files chapter in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* (UG575) [[Ref 9](#)] to obtain links to the UltraScale device package files. Analog auxiliary pins are in the form `_ADxP_` in the pin name.

### Example

Figure 5-15 shows a portion of a package file. The pin name portions within the red boxes denote which analog auxiliary pair access those particular pins have access to.

```
Device/Package xcku035fbva676 6/16/2014 16:20:56
```

Pin	Pin Name	Memory Byte Group	Bank	I/O Type	Super Logic Region	No-Connect
T11	DXN	NA	NA	NA	NA	NA
N12	VCCADC	NA	NA	NA	NA	NA
N11	GNDADC	NA	NA	NA	NA	NA
T12	DXP	NA	NA	NA	NA	NA
R12	VREFP	NA	NA	NA	NA	NA
P11	VREFN	NA	NA	NA	NA	NA
P12	VP	NA	NA	NA	NA	NA
R11	VN	NA	NA	NA	NA	NA
K7	M0_0	NA	0	CONFIG	NA	NA
H7	M1_0	NA	0	CONFIG	NA	NA
G7	INIT_B_0	NA	0	CONFIG	NA	NA
L7	M2_0	NA	0	CONFIG	NA	NA
Y7	CFGBVS_0	NA	0	CONFIG	NA	NA
J8	FUDC_B_0	NA	0	CONFIG	NA	NA
F7	POR_OVERRIDE	NA	NA	NA	NA	NA
V7	DONE_0	NA	0	CONFIG	NA	NA
M7	PROGRAM_B_0	NA	0	CONFIG	NA	NA
L9	TDO_0	NA	0	CONFIG	NA	NA
N9	TDI_0	NA	0	CONFIG	NA	NA
N7	RDWR_FCS_B_0	NA	0	CONFIG	NA	NA
R7	D02_0	NA	0	CONFIG	NA	NA
U7	D00_MOSI_0	NA	0	CONFIG	NA	NA
P7	D03_0	NA	0	CONFIG	NA	NA
T7	D01_DIN_0	NA	0	CONFIG	NA	NA
P9	TMS_0	NA	0	CONFIG	NA	NA
U9	CCLK_0	NA	0	CONFIG	NA	NA
R9	TCK_0	NA	0	CONFIG	NA	NA
AA7	VBATT	NA	NA	NA	NA	NA
U20	IO_L24P_T3U_N10_44	3U	44	HP	NA	NA
U21	IO_L24N_T3U_N11_44	3U	44	HP	NA	NA
V21	IO_T3U_N12_44	3U	44	HP	NA	NA
P20	IO_L23P_T3U_N8_44	3U	44	HP	NA	NA
R20	IO_L23N_T3U_N9_44	3U	44	HP	NA	NA
T19	IO_L22P_T3U_N6_DBC_AD0P_44	3U	44	HP	NA	NA
T20	IO_L22N_T3U_N7_DBC_AD0N_44	3U	44	HP	NA	NA
P18	IO_L21P_T3L_N4_AD0P_44	3L	44	HP	NA	NA
P19	IO_L21N_T3L_N5_AD0N_44	3L	44	HP	NA	NA
P21	IO_L20P_T3L_N2_AD1P_44	3L	44	HP	NA	NA
R21	IO_L20N_T3L_N3_AD1N_44	3L	44	HP	NA	NA

UG583\_c3\_28\_022515

Figure 5-15: Package File Showing Analog Pair Access

## 11. Decoupling Capacitors

If the design is not expected to change when migrating to the new device, the same decoupling scheme can be utilized. If adding logic, Xilinx recommends using a decoupling scheme appropriate for the new device and design. Refer to [Chapter 1, Power Distribution System in UltraScale Devices](#) for decoupling capacitor guidelines for the various UltraScale devices.

### Example

Figure 5-16 shows a portion of the Kintex UltraScale decoupling recommendations. If beginning a design in the XCKU040-FFVA1156 and migrating to the XCKU060-FFVA1156, use the XCKU040 capacitor scheme if no design changes are anticipated in the migration. If more logic is added during the migration to the XCKU060, Xilinx recommends using the decoupling scheme for that device.

Table 1-1: Kintex UltraScale Devices Power Supply Decoupling Capacitors

	V <sub>CCINT</sub> /V <sub>CCINT_IO</sub> <sup>(1)</sup>			V <sub>CCBRAM</sub>		V <sub>CCAUX</sub> /V <sub>CCAUX_IO</sub> <sup>(2)</sup>		HRIO <sup>(3)</sup>	HPIO <sup>(3)</sup>
	680 μF <sup>(4)</sup>	100 μF	4.7 μF	47 μF	4.7 μF	47 μF	4.7 μF	47 μF	47 μF
XCKU035-FBVA676	1	1	3	1	1	1	3	1	1
XCKU035-FBVA900	1	1	3	1	1	2	4	1	1
XCKU035-FFVA1156	1	1	3	1	1	2	5	1	1
XCKU040-FBVA676	1	2	3	1	1	1	3	1	1
XCKU040-FBVA900	1	2	3	1	1	2	4	1	1
XCKU040-FFVA1156	1	2	3	1	1	2	5	1	1
XCKU060-FFVA1156	1	2	4	1	1	2	5	1	1
XCKU060-FFVA1517	1	2	4	1	1	3	6	1	1

UG583\_c3\_28\_022615

Figure 5-16: Portion of Kintex Decoupling Capacitor Recommendations

## 12. PCI Express Migration

PCI Express lane assignments might differ between devices. In addition, the block that is used as tandem boot might also be different from device to device. Refer to the Core Pinouts appendix in *UltraScale Architecture Gen3 Integrated Block for PCI Express LogiCORE IP Product Guide* (PG156) [Ref 14] for UltraScale device core pinouts.

## 13. Integrated 100G Ethernet Migration

The 100G Ethernet IP core is a dedicated block that provides 100 Gb/s MAC and PCS logic capability. The block consists of a number of dedicated locations that are located at different points on different devices. There are specific rules for connecting these dedicated blocks to specific transceiver locations, which might impact migration to another device. In addition, the CAUI-4 and switchable CAUI-10/CAUI-4 modes require GTY transceivers.

Refer to *UltraScale Architecture Integrated Block for 100G Ethernet LogiCORE IP Product Guide* (PG165) [Ref 15] for a more comprehensive guide regarding designing with the 100G Ethernet core, with special attention to the Transceiver Selection Rules section.

## 14. Interlaken Migration

The Xilinx UltraScale architecture integrated IP core for Interlaken is a highly configurable integrated IP core that can support an overall bandwidth up to 150 Gb/s for protocol logic transmission. The core connects to serial transceivers at defined rates up to 12.5 Gb/s with GTH transceivers, and up to 25.78125 Gb/s with GTY transceivers. The block consists of a number of dedicated locations that are located at different points on different devices. There are specific rules for connecting these dedicated blocks to specific transceiver locations, which might impact migration to another device. Refer to *UltraScale Architecture Integrated IP Core for Interlaken LogiCORE IP Product Guide* (PG169) [Ref 16] for a more comprehensive guide regarding the proper generation and verification of Interlaken interfaces in UltraScale devices with special attention to the Transceiver Interface section.

## 15. Power Supplies and Thermal Considerations

Migration across devices, especially from smaller to bigger devices, can result in higher power consumption requiring improved thermal management and power delivery. Consult the Xilinx Power Estimator (XPE) tool and select the largest device being considered in order to estimate power consumption and thermal requirements.



---

**RECOMMENDED:** *Xilinx recommends use of the Delphi thermal model during thermal modeling of a package. Xilinx does not recommend using a two-resistor model for thermal simulation and design due to lack of precision and accuracy. If more accuracy is desired than the Delphi model, Xilinx can provide a detailed model representation of the package by request. However, this might consume more simulation memory and run time in its use. The user of the thermal model needs to include a consideration of thermal sensor accuracy, thermal interface material parameters, and manufacture variation on the thermal solution. Also, other examples of manufacture variations include the tolerance in airflow from a fan, the tolerance on performance of the heat pipe and vapor chamber, and the manufacture variation of the attachment of fins to the heat-sink base and the flatness of the surface.*

---

## 16. Pin Flight Times across Packages

Even if two devices are footprint compatible from a design and PCB standpoint, FPGA package flight times will be different across different devices, so they will need to be accounted for. Flight time information can be found within the Vivado tools in the Package Pins tab during the I/O planning stage or after synthesis. The ideal strategy to account for pin flight time differences is to deskew the printed circuit board when migrating to the new device. If this is not possible, it is recommended to lay out the printed circuit board with the final device in mind to maximize system performance for the long term. System performance might have to be derated when using the initial device. As a last option, choosing the mid-point of the range of flight times and routing the board based on that value can act as a compromise, though maximum system performance might not be achievable with this method.

**Example**

Pin flight time information can be obtained via the Package Pins tab within the Vivado tools, both in the I/O planning stage and after synthesis (Figure 5-17).

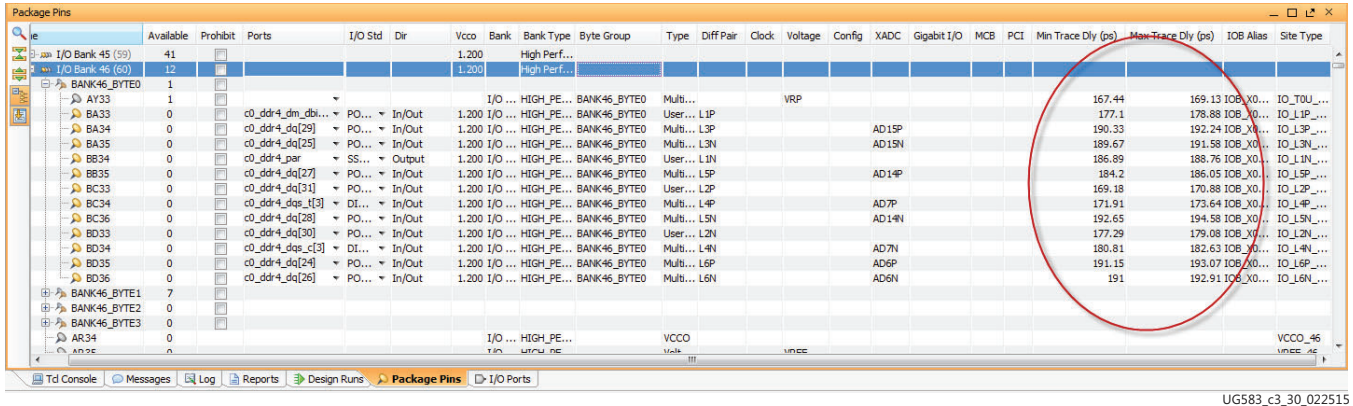


Figure 5-17: Vivado Example Showing Pin Flight Times

# Migration between UltraScale and UltraScale+ FPGAs

When migrating a design from one UltraScale FPGA to another, special attention must be paid to potential differences between the two FPGAs. [Chapter 5, Migration between UltraScale Devices and Packages](#) contains a comprehensive guide towards general migration concerns within the UltraScale families. This section addresses additional concerns specific to migration from UltraScale to UltraScale+ FPGAs, as well as from UltraScale+ to UltraScale+ FPGAs.

The underlying architecture of UltraScale+ FPGAs is the same as that of UltraScale FPGAs, meaning that most IP that contains CLBs, routing resources, DSP slices, and block RAM is the same. *UltraScale Architecture and Product Overview* (DS890) [Ref 19] contains a complete listing of the available resources found in UltraScale and UltraScale+ FPGAs.

The following checklist and sections contain the necessary items to review for optimal UltraScale FPGAs migration.

---

## UltraScale+ FPGA Migration Checklist

Review [Chapter 5, Migration between UltraScale Devices and Packages](#) for general migration concerns within the UltraScale family.

1. Review the package migration table from UltraScale to UltraScale+ FPGAs.
2. Review power supply voltage differences between UltraScale and UltraScale+ FPGAs.
3. Review  $V_{CCINT\_IO}$  connection requirements.
4. Review I/O differences between UltraScale to UltraScale+ FPGAs.
5. Review transceiver differences between UltraScale to UltraScale+ FPGAs.
6. Review configuration details that pertain to UltraScale+ FPGAs.
7. Review memory interface PCB routing guidelines.
8. Review new SMBAlert signal for UltraScale+ FPGAs.
9. Review block RAM/UltraRAM capacities between UltraScale and UltraScale+ FPGAs.
10. Review ESD requirements pertaining to UltraScale+ FPGAs.

11. Review migration examples between UltraScale and UltraScale+ FPGAs.
12. Review migration scenarios to better understand the migration process.

## 1. Package Migration from UltraScale to UltraScale+ FPGAs

The packages listed in [Table 6-1](#) support UltraScale to UltraScale+ FPGA migration.

*Table 6-1: UltraScale to UltraScale+ FPGA Migration Paths by Package*

Package	UltraScale FPGA	UltraScale+ FPGA
A676	KU035 KU040	KU3P <sup>(1)</sup> KU5P <sup>(1)</sup>
A1156	KU025 KU035 KU040 KU060 KU095	KU11P <sup>(2)</sup> KU15P <sup>(2)</sup>
C1517	KU095 <sup>(3)</sup> VU065 <sup>(3)</sup> VU080 <sup>(3)</sup> VU095 <sup>(3)</sup>	VU3P <sup>(3)</sup>
F1924	KU085 <sup>(3)</sup> KU115	VU11P <sup>(4)</sup>
A2104	KU115 VU080 <sup>(3)</sup> VU095 <sup>(3)</sup> VU125	VU5P VU7P VU9P VU13P
B2104	KU095 <sup>(3)</sup> KU115 VU080 <sup>(3)</sup> VU095 <sup>(3)</sup> VU125 VU160 VU190	VU5P VU7P VU9P VU11P VU13P
C2104	VU095 <sup>(3)</sup> VU125 VU160 VU190	VU5P VU7P VU9P VU11P VU13P

**Table 6-1: UltraScale to UltraScale+ FPGA Migration Paths by Package (Cont'd)**

Package	UltraScale FPGA	UltraScale+ FPGA
A2577	VU190	VU9P VU11P VU13P

**Notes:**

1. KU3P and KU5P GTY transceivers up to 16.3 Gb/s in A676 package.
2. KU11P and KU15P GTY transceivers up to 16.3 Gb/s in A1156 package.
3. Monolithic device, otherwise SSI device.
4. VU11P GTY transceivers up to 16.3 Gb/s in F1924 package.

## 2. Voltage Differences between UltraScale to UltraScale+ FPGAs

Pay special attention to the voltage differences between  $V_{CCINT}$ ,  $V_{CCINT\_IO}$ ,  $V_{CCBRAM}$ , and  $AVCC$  between UltraScale and UltraScale+ FPGAs (Table 6-2).

**Table 6-2: Voltages Differences between UltraScale and UltraScale+ FPGAs**

Voltage	UltraScale FPGA (V)	UltraScale+ FPGA (V)
AVCC	1.0	0.9
AVTT	1.2	1.2
AVCCPLL	1.8	1.8
VCCINT (-3)	1.0	0.90
VCCINT (-2, -1)	0.95	0.85
VCCINT (-2L)	N/A	0.72 or 0.85
VCCINT (-1L)	0.90	0.72 or 0.85
VCCINT_IO (-3)	1.0	0.90
VCCINT_IO (-2, -1)	0.95	0.85
VCCINT_IO (-2L)	N/A	0.85
VCCINT_IO (-1L)	0.90	0.85
VCCBRAM (-3)	1.0	0.90
VCCBRAM (-2, -1)	0.95	0.85
VCCBRAM (-2L)	N/A	0.85
VCCBRAM (-1L)	0.95	0.85
VCCAUX	1.80	1.80
VCCAUX_IO	1.80	1.80



### 3. Power Supply Voltage Levels and VCCINT\_IO Connection

For UltraScale and UltraScale+ FPGAs in the -3, -2, or -1 speed grades, it is required to connect VCCINT\_IO to VCCINT. However, for UltraScale+ -2L and -1L FPGAs, it is required to connect VCCINT\_IO to VCCBRAM.

To support this requirement, the system shown in Table 6-3 is recommended in conjunction with Figure 6-1. The VCCBRAM plane must be sized appropriately to support the additional load for VCCINT\_IO when using -2L or -1L devices.

It is possible to connect VCCINT, VCCINT\_IO, and VCCBRAM together when using -2 or -1 devices because the respective voltages are the same. It is also possible to connect VCCINT, VCCINT\_IO, and VCCBRAM together when using -2L or -1L devices if VCCINT is always operated at 0.85V.

Table 6-3: VCCINT/VCCINT\_IO/VCCBRAM Connection Matrix

	-3, -2, -1	-2L, -1L
VCCINT_IO connection	VCCINT	VCCBRAM
VCCINT sense line connection	Averaged with VCCINT_IO	Single connection to VCCINT
VCCBRAM sense line connection	Single connection to VCCBRAM	Averaged with VCCINT_IO
R1	Do not populate	0Ω 0603
R2	0Ω 0603	Do not populate
R3	10Ω 0603	10Ω 0603
R4	10Ω 0603	Do not populate
R5	Do not populate	10Ω 0603
R6	10Ω 0603	10Ω 0603

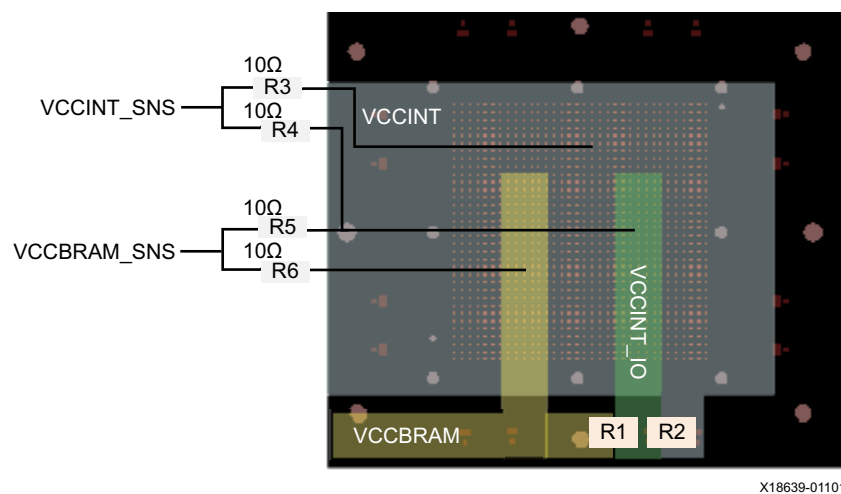


Figure 6-1: Reference Layout to Support VCCINT/VCCINT\_IO/VCCBRAM Connection Matrix

## 4. I/O Changes from UltraScale to UltraScale+ FPGAs

HRIO banks in UltraScale FPGAs become HPIO banks in Virtex UltraScale+ FPGAs.



**IMPORTANT:** *There are a number of cases in which two “half” banks in an UltraScale device correspond to one “full” bank in an UltraScale+ device, and vice versa. When migrating from a device with half banks to a device with full banks, ensure that the voltage level on all  $V_{CC0}$  pins for the half banks match the intended voltage of the corresponding full bank. Refer to [Table 6-4](#) and to the I/O bank migration table in *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification (UG575)* [Ref 9] to determine if this situation applies to your intended migration path.*

**Table 6-4: Corresponding Half-Banks and Full-Banks**

Package	Half Banks	Full Bank
A676	XCKU3P XCKU5P (Banks 84/85)	XCKU035 XCKU040 (Bank 64)
A1156	XCKU11P (Banks 88/89)	XCKU025 XCKU035 XCKU040 XCKU060 XCKU095 (Bank 64)
A1156	XCKU15P (Banks 90/91)	XCKU025 XCKU035 XCKU040 XCKU060 XCKU095 (Bank 64)
C1517	XCVU065 XCVU080 XCVU095 (Banks 84/94)	XCVU3P (Bank 64)
A2104	XCKU115 XCVU080 XCVU095 XCVU125 (Banks 84/94)	XCVU5P XCVU7P XCVU9P XCVU13P (Bank 64)

Table 6-4: Corresponding Half-Banks and Full-Banks (Cont'd)

Package	Half Banks	Full Bank
B2104	XCKU095	XCVU5P
	XCKU115	XCVU7P
	XCVU080	XCVU9P
	XCVU095	XCVU11P
	XCVU125	XCVU13P
	XCVU160	(Bank 64)
	XCVU190 (Banks 84/94)	
C2104	XCVU095	XCVU5P
	XCVU125	XCVU7P
	(Banks 84/94)	XCVU9P
		XCVU11P
		XCVU13P (Bank 64)

## 5. Transceiver Changes from UltraScale to UltraScale+ FPGAs

- All GTH transceivers in UltraScale FPGAs become GTY transceivers in Virtex UltraScale+ FPGAs.
- Some GTY transceiver voltage levels change from UltraScale to UltraScale+ FPGAs, as noted in [Table 6-2](#).
- While still maintaining footprint compatibility, some devices might have one or more extra RCAL and RREF pins than smaller dies in the same package. If planning to migrate from a smaller die to a bigger die in the same package, Xilinx recommends reserving any extra RCAL and RREF pins for later use, because they are NC pins in the smaller devices.
- Special exceptions for the F1924 package are:
  - VU11P-FLVF1924 GTY transceivers run up to 16.3 Gb/s versus the standard GTY transceiver speed of 32.75 Gb/s.
  - Two ground pins (G9 and G10) in the UltraScale KU085/115 F1924 device later give way to becoming RREF (G9) and RCAL (G10) pins in the UltraScale+ VU11P F1924 device. The recommended way to design for this case would be by utilizing the schematic shown in [Figure 6-2](#) and [Table 6-5](#). When using the KU085/115 F1924, the G9 and G10 pins are grounded, and the resistors can be left unpopulated. When the RCAL and RREF pins are present when using the VU11P, both resistors should be populated.

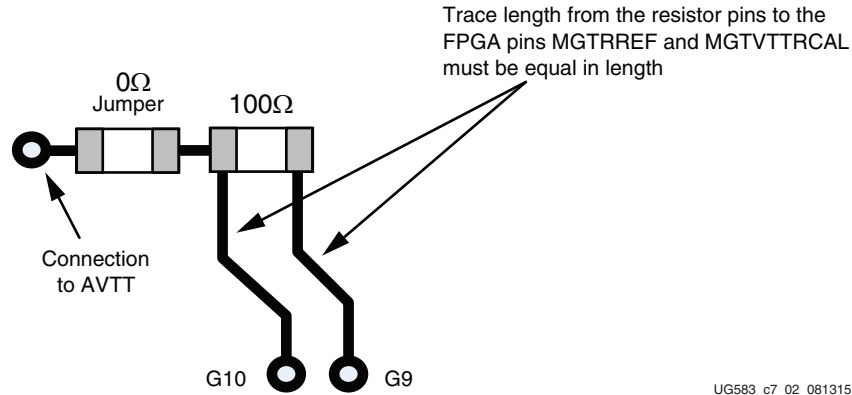


Figure 6-2: Schematic for Reserving Future RCAL and RREF Pins for F1924

Table 6-5: Reserving RCAL and RREF Pins for F1924

F1924	G9	G10	Resistors
KU085/KU115	GND	GND	Do not populate
VU11P	MGTRREF	MGTAVTTRCAL	Populate

## 6. Configuration

- UltraScale+ FPGAs do not support 3.3V operation on bank 0:
  - Refer to *Interfacing 7 Series FPGAs High-Performance I/O Banks with 2.5V and 3.3V I/O Standards* (XAPP520) [Ref 11] for methods of level translation between high-performance I/O banks and 2.5V/3.3V I/O standards.
- UltraScale+ FPGAs do not support master SelectMAP or master serial configuration modes:
  - Slave SelectMAP with an externally sourced clock can be used in place of master SelectMAP.
  - SPIx1 or slave serial (with an externally sourced clock) can be used in place of master serial mode.

## 7. Memory Interface PCB Routing

Table 6-6 illustrates the expected PL DDR4 SDRAM performance for UltraScale+ FPGAs in the presence of migration for a single-rank component.

**Table 6-6: DDR4 SDRAM Performance in UltraScale+ FPGAs in the Presence of Migration for Single-Rank Component**

DDR4 Data Rate in UltraScale FPGAs	V <sub>CCINT</sub> = 0.9V	UltraScale+ V <sub>CCINT</sub> = 0.85V		UltraScale+ V <sub>CCINT</sub> = 0.72V
	-3 (Max = 2667 Mb/s)	-2I, -2E, -2LE (Max = 2667 Mb/s)	-1E, -1I, (Max = 2400 Mb/s)	-2LE (Max = 2400 Mb/s)
-3 @ 2400	2400	2400	2133	2133
-3 @ 2133	2133	2133	2133	2133
-3 @ 1866	1866	1866	1866	1866
-2E, -2I @ 2400	2400	2400	2133	2133
-2E, -2I @ 2133	2133	2133	2133	2133
-2E, -2I @ 1866	1866	1866	1866	1866
-1, -1I @ 2133	2133	2133	2133	2133
-1, -1I @ 1866	1866	1866	1866	1866

**Notes:**

1. This table applies only to PL memory interfaces. PS interfaces do not derate in performance due to speed grade.

## 8. System Monitor I2C

The SMBAlarm pin is new to UltraScale+ FPGAs. If this feature is used in a migration from UltraScale to UltraScale+ FPGAs, the corresponding UltraScale FPGA pin must be reserved.

## 9. Block RAM

Typically, as device densities get larger, the amount of block RAM also keeps increasing. However, in certain UltraScale+ devices, the amount of block RAM might be lesser than in devices of smaller density. This is due to the addition of UltraRAM resources to UltraScale+ FPGAs and MPSoCs. With UltraRAM, there is an increase in overall bit count compared to UltraScale FPGAs but with fewer block RAM resources available. Xilinx advises coding the memory to use features present in both block RAM and UltraRAM, and the tools target UltraRAM as applicable.

## 10. ESD Requirements

UltraScale FPGA ESD handling requirements are based on JEDEC JEP155, available at [www.jedec.org/standards-documents/docs/jep-155](http://www.jedec.org/standards-documents/docs/jep-155), as well as Industry Council ESD-CDM Target Levels, JEP157, available at [www.esdindustrycouncil.org/ic/en](http://www.esdindustrycouncil.org/ic/en).

## 11. PCI Express

Some UltraScale+ FPGAs only contain one PCI reset pin (PERSTN0). PERSTN1 is not available.

## 12. Migration Examples

This section describes two examples of differing complexity to illustrate the procedures for assessing migration.

### Example One: VU125 to VU7P in C2104 Package

Figure 6-3 shows a die-level floorplan for a design utilizing the VU125 in the C2104 package. This design includes several memory interfaces, a 100G Ethernet interface, an Interlaken 12x12.5G interface, and a Tandem PCIe interface. The bank(s) that each interface utilizes are shown color-coordinated, along with the relevant block locations.

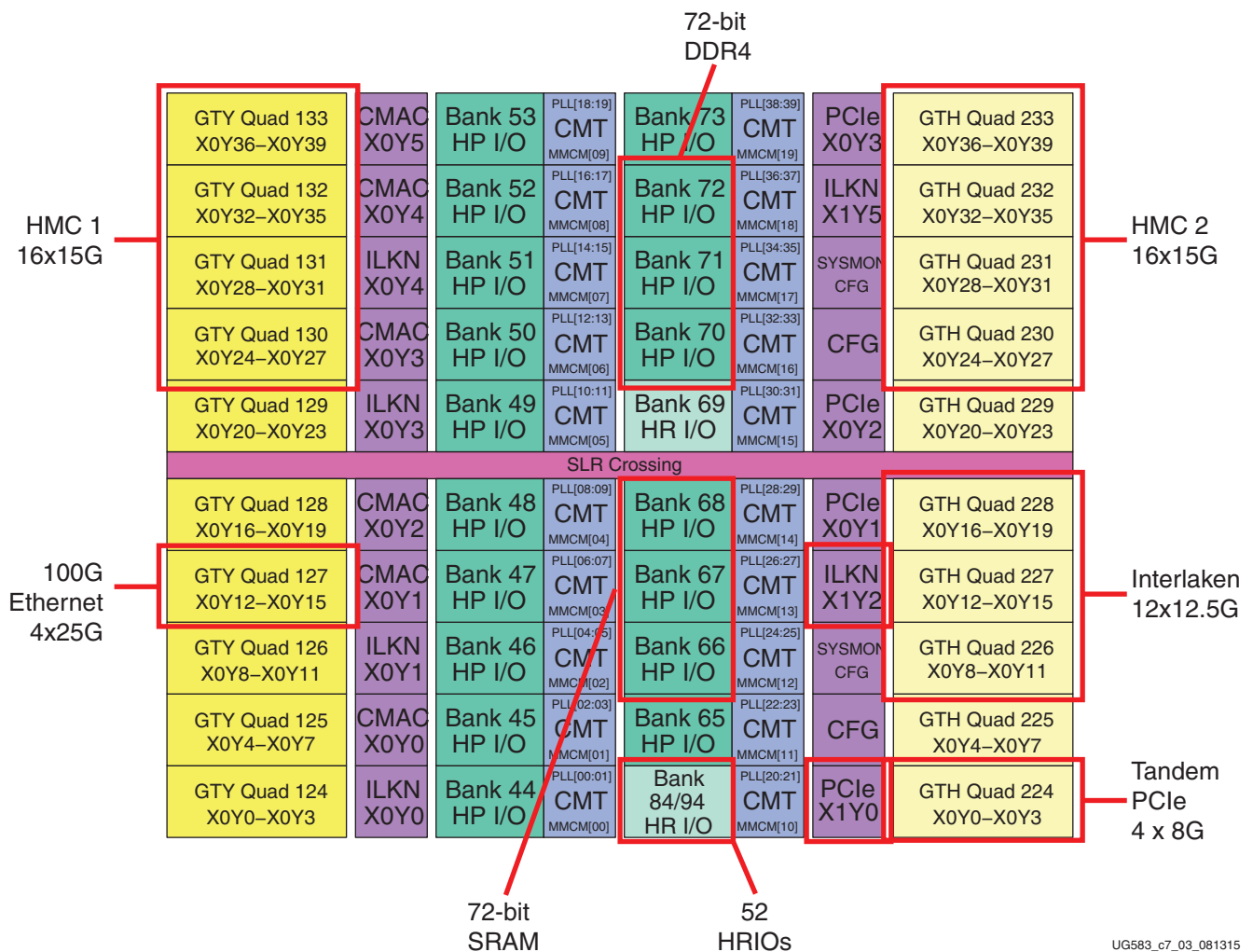


Figure 6-3: Example VU125 Design Floorplan

Figure 6-4 shows how the various interfaces can migrate into a VU7P device that is also in the C2104 package. All interfaces can and must map to the same banks (and corresponding pins), as well as adhering to all block location requirements such as MAC, Interlaken, and PCIe banks needing to be within one row of their respective locations on the die. This particular example is reasonably straightforward in that the two device floorplans are very close to equal, with similar bank and block locations along with similar SLR boundaries.

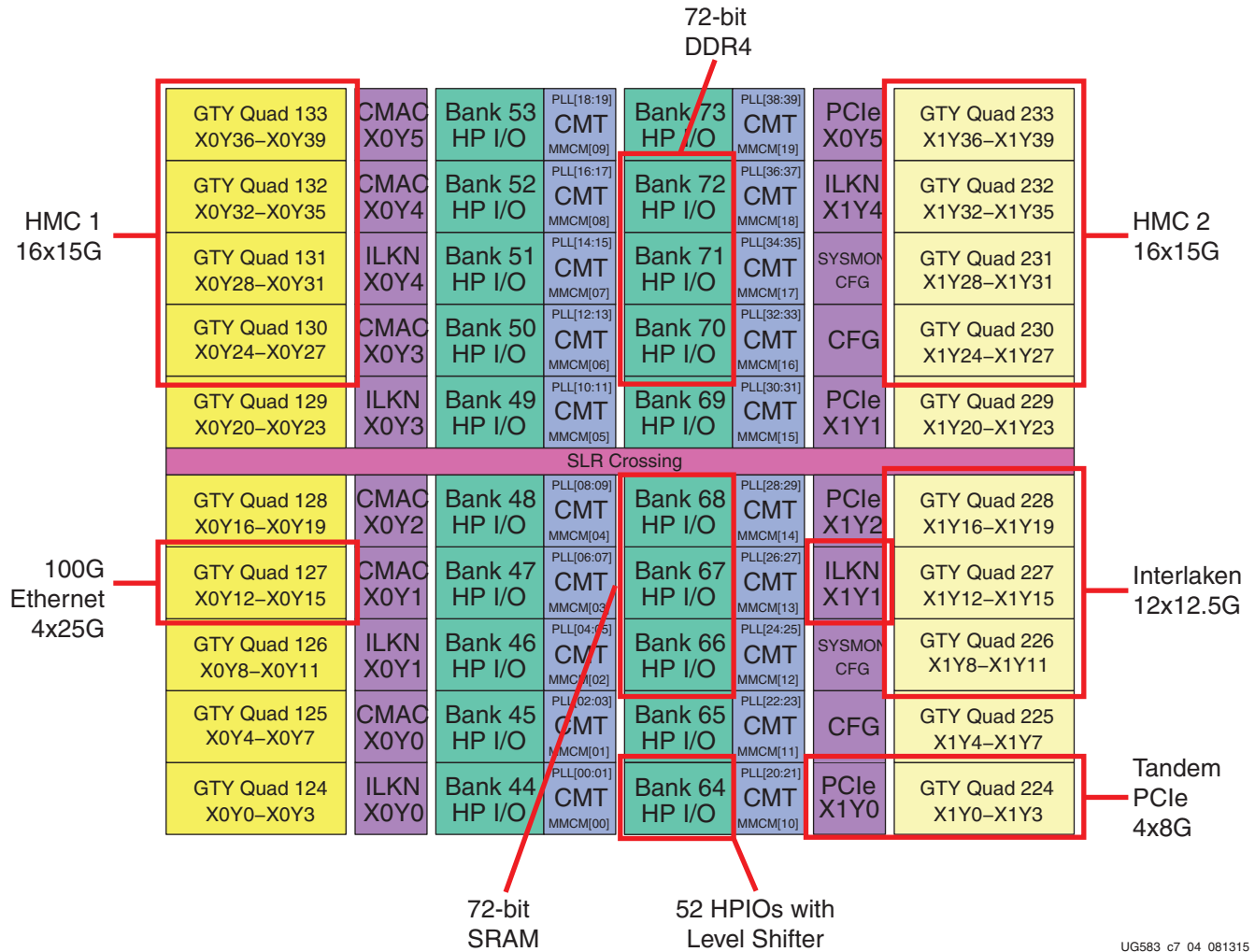


Figure 6-4: VU7P Floorplan for Migration from VU125

UG583\_c7\_04\_081315

**Example Two - VU190 to VU13P in A2577 Package**

Figure 6-5 shows a die-level floorplan for a design utilizing the VU190 device in the A2577 package. This design is similar to the design in [Example One: VU125 to VU7P in C2104 Package](#) in that it includes several memory interfaces, a 100G Ethernet interface, an Interlaken 12x12.5G interface, and a Tandem PCIe interface.

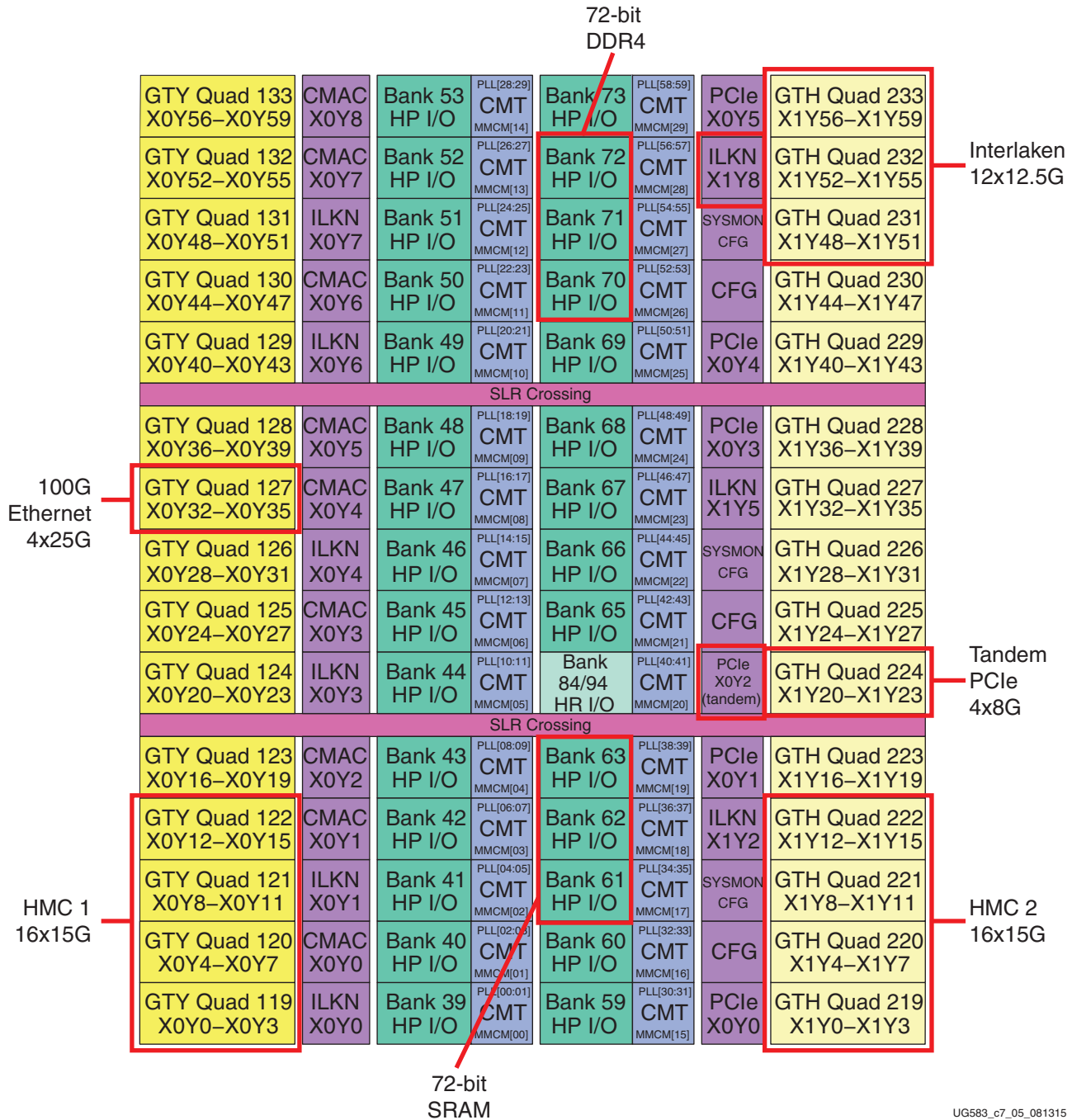


Figure 6-5: Example VU190 Design Floorplan

UG583\_c7\_05\_081315



Figure 6-6 shows how this design could migrate into a VU13P device. Careful bank selections for the various interfaces are necessary due to the notable differences in the device floorplans. The key differences in the device floorplans are such that there are more available banks in the VU13P, the SLRs cross different bank boundaries, and the block locations are noticeably different in the two devices relative to their neighboring banks.

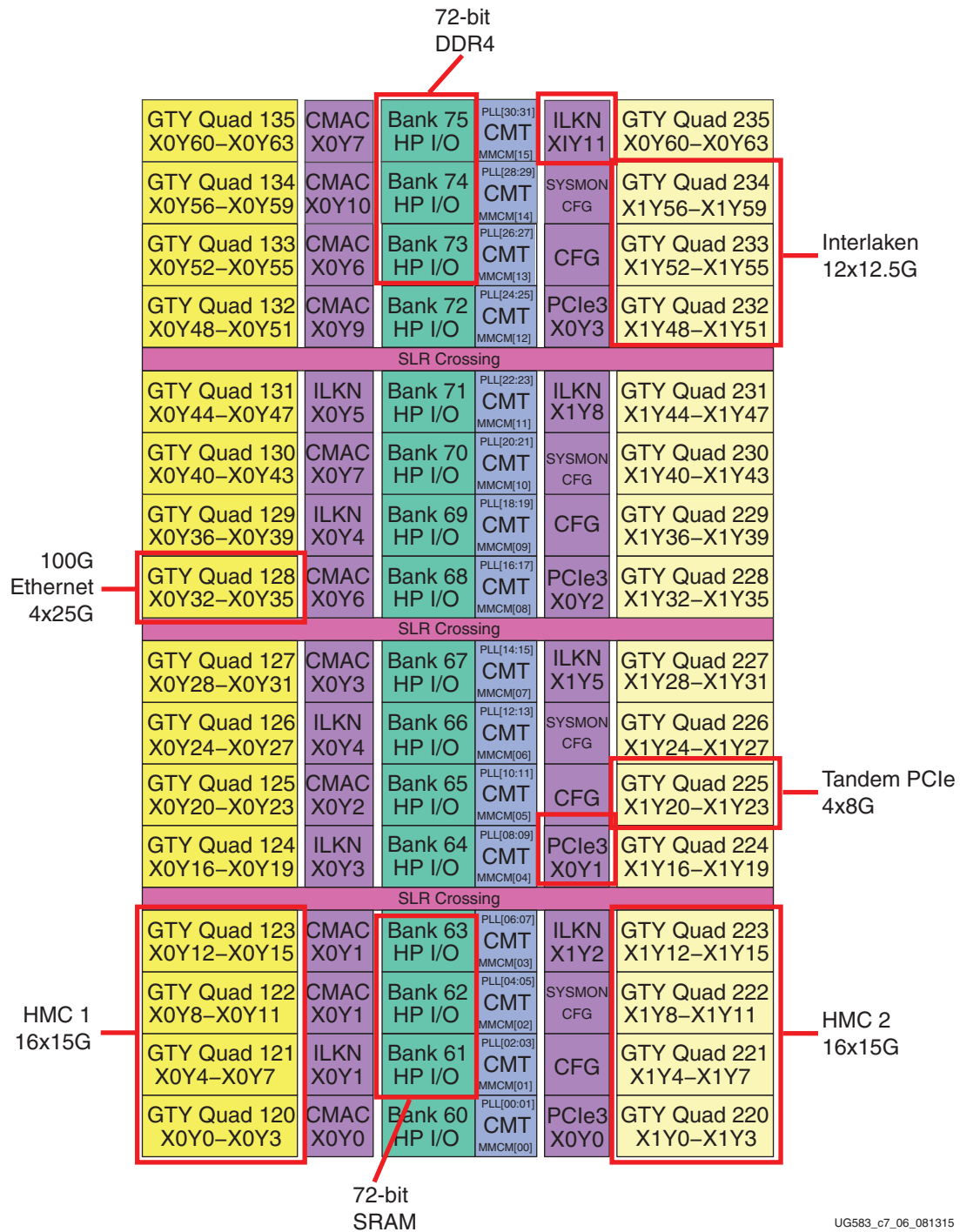


Figure 6-6: VU13P Floorplan for Migration from VU190

UG583\_c7\_06\_081315

## 13. Migration Scenarios

Not all migration scenarios have the same level of complexity in the 80 paths shown in [Table 6-1](#). [Table 6-7](#) outlines various scenarios in various degrees of complexity.

Migration from a Kintex or Virtex UltraScale FPGA into a Virtex UltraScale+ FPGA built with a five row tall SLR is seamless compared to migrating into a device made up of SLRs that are four rows tall. However, all migration paths are supported as long as the customer plans upfront on the extra considerations needed to make this switch successful.

An example of an easy and seamless migration scenario is from the VU095 to the V3P where both devices are monolithic. Similarly, migrating from the KU115, VU125, VU160, or VU190 into the VU5P, VU7P, or VU9P is also seamless and straightforward because they are based on SLRs that are five rows tall. Extra consideration is needed to migrate to the VU11P or VU13P because these devices consist of SLRs that are four rows tall.

**Table 6-7: UltraScale Device Migration Scenarios**

Package	UltraScale FPGAs	UltraScale+ FPGAs	Package Migration	Comment
C1517	KU095, VU065, VU080, VU095	VU3P	Supported	Easy and seamless because all the devices are monolithic.
F1924	KU085, KU115	VU11P	Supported	Extra consideration needed because migration path is from devices with 2 SLRs (5 rows) to 3 SLRs (4 rows) tall.
A2104	VU080, VU095	VU5P, VU7P, VU9P, VU13P	Supported	Extra consideration needed because migration path is from monolithic device to SLR devices.
A2104	KU115, VU125	VU5P, VU7P, VU9P	Supported	Easy and seamless because all the devices are made up of 5 row tall SLRs.
A2104	KU115, VU125	VU13P	Supported	Extra consideration needed because migration path is from devices with 2 SLRs (5 rows) to 3 SLRs (4 rows) tall.
B2104	KU095, VU080, VU095	VU5P, VU7P, VU9P, VU11P, VU13P	Supported	Extra consideration needed because migration path is from monolithic device to SLR devices.
B2104	KU115, VU125, VU160, VU190	VU5P, VU7P, VU9P	Supported	Easy and seamless because all the devices are made up of 5 row tall SLRs.
B2104	KU115, VU125, VU160, VU190	VU11P, VU13P	Supported	Extra consideration needed because migration path is from devices with 2/3 SLRs (5 rows) to 3/4 SLRs (4 rows) tall.

Table 6-7: UltraScale Device Migration Scenarios (Cont'd)

Package	UltraScale FPGAs	UltraScale+ FPGAs	Package Migration	Comment
C2104	VU095	VU5P, VU7P, VU9P, VU11P, VU13P	Supported	Extra consideration needed because migration path is from a monolithic device to SLR devices.
C2104	VU125, VU160, VU190	VU5P, VU7P, VU9P	Supported	Easy and seamless because all the devices are made up of 5 row tall SLRs.
C2104	VU125, VU160, VU190	VU11P, VU13P	Supported	Extra consideration needed because migration path is from devices with 2/3 SLRs (5 rows) to 3/4 SLRs (4 rows) tall.
A2577	VU190	VU9P	Supported	Easy and seamless because both the devices are made up of 3 to 5 row tall SLRs.
A2577	VU190	VU11P, VU13P	Supported	Extra consideration needed because migration path is from device with 3 SLRs (5 rows) to 3/4 SLRs (4 rows) tall.

# Migration between Zynq UltraScale+ MPSoCs and Packages

When migrating a design from one Zynq UltraScale+ MPSoC to another, special attention must be paid to potential differences between the two devices. Refer to the checklist below, and the subsequent sections that explain each item.

---

## Zynq UltraScale+ MPSoC Migration Checklist

1. Review footprint compatibility, including banks that are bonded vs. unbonded.
2. Review I/O and transceiver Quad numbering and locations.
3. Review power supply differences across speed and temperature grades.
4. Review -3 speed grade migration.
5. Review video codec unit (VCU) migration.
6. Review graphics processing unit (GPU) migration.
7. Review decoupling capacitor requirements.
8. Review PCI Express® requirements and block locations.
9. Review integrated 100G Ethernet requirements and block locations.
10. Review Interlaken requirements and block locations.
11. Review power supply and thermal requirements.
12. Review pin flight time differences between devices.

### 1. Footprint Compatibility between Packages

Any two packages with the same footprint identifier code are footprint compatible. The footprint identifier code consists of a package designator code and BGA pin count information (such as B1156 or C1156, for example). A customer that wants pin compatibility across any two devices in a given package should select a footprint compatible package such that they can be designed to be electrically compatible. If migrating from a bigger device to a smaller device within the same package, some I/O and transceiver banks and their pins might be unbonded or not exist on the smaller device. Refer to the Footprint

Compatibility, I/O Bank Migration, and Transceiver Quad Migration tables in *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide* (UG1075) [Ref 26] to determine which Zynq UltraScale+ MPSoCs are footprint compatible, as well as the extent to which the various banks might become unbonded or not exist.

**Example 1**

Figure 7-1 shows a snippet of the Footprint Compatibility table from *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide* (UG1075). The XCZU2CG/EG SBVA484 devices are footprint compatible with the XCZU3CG/EG A484 devices, but the XCZU2CG/EG A484 devices are NOT compatible with any of the A625 devices, even though they share the same die.

Table 1-5: Footprint Compatibility

Packages	Footprint Compatible Devices		
SBVA484	XCZU2CG and XCZU2EG	XCZU3CG and XCZU3EG	
SFVA625	XCZU2CG and XCZU2EG	XCZU3CG and XCZU3EG	

X18640-011017

Figure 7-1: Example of Footprint Compatibility

**Example 2**

Figure 7-2 shows a snippet of the I/O Bank Migration table from *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide* (UG1075). I/O banks 72, 73, and 74 are present on the XCZU17 and ZCZU19 B1517 devices, but those I/O banks are not available on the XCZU11 B1517 device.

Table 1-6: I/O Bank Migration (HD Banks are Shaded) (Cont'd)

Package	Device	Package to Device I/O Mapping <sup>(1)</sup>																										Unbonded I/O Banks
		A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z	
FFVB1517	XCZU11			65	64	66										88	89	90				71	70	69	68	67		91
	XCZU17			65	64	66										90	91	93	74	73	72	71	70	69	68	67		94
	XCZU19			65	64	66										90	91	93	74	73	72	71	70	69	68	67		94

X18641-011017

Figure 7-2: Example I/O Bank Migration within B1517 Devices

**Example 3**

Figure 7-3 shows a snippet of the Transceiver Quad Migration table from *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide* (UG1075). Transceiver Quads

230 and 231 are present on the XCZU11 F1517 device, but those transceiver Quads are not available on the XCZU7 F1517 device.

Power Supply Group		RS					RN		
FFVF1517	XCZU7	223	224	225	226	227	228		
	XCZU11	224	225	226	227	228	229	230	231
							131, 130, 129, 128, 127		

X18642-022218

Figure 7-3: Example Transceiver Quad Migration within F1517 Devices

## 2. I/O Bank and Transceiver Quad Numbers

Certain Zynq UltraScale+ MPSoCs that are footprint compatible in a package might have different I/O bank and transceiver Quad numbers associated with the same package pins. Depending on where these banks are physically located on the die, certain multi-bank interfaces (such as memory or transceiver) could be impacted in regards to migrating from one device to the other. If a bank number change also involves the bank moving from one column on the die to another, a multi-bank interface that requires both banks to be in the same column could be impacted.

Refer to the I/O Bank Migration and Transceiver Quad Migration tables in *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide (UG1075)* for the bank and transceiver Quad numbering differences. To cross-reference bank locations on a die, refer to the Die Level Bank Numbering Overview section of *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide (UG1075)*.

The example below shows a successful migration path despite both bank number changes and banks being in different columns between the two different devices.

**Example: Planning Ahead for Bank Number and Location Changes**

As shown in Figure 7-4, banks 88, 89, and 90 in the XCZU11-FFVB1517 device connect to the same pins as banks 90, 91, and 93 in the XCZU17-FFVB1517 and XCZU19-FFVB1517 devices. Figure 7-5 and Figure 7-6 show how the banks are arranged on each particular device. For the XCZU11, the banks are contiguous and could support a three-bank interface that requires the banks to be contiguous. However, the three banks that connect to the same pins on the XCZU17 and ZXCUI9 are not contiguous, and this might result in timing differences and possible design rule issues for interfaces that require the banks to be contiguous, such as memory interfaces.

Table 1-6: I/O Bank Migration (HD Banks are Shaded) (Cont'd)

Package	Device	Package to Device I/O Mapping <sup>(1)</sup>																										Unbonded I/O Banks	
		A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z		
FFVB1517	XCZU11			65	64	66											88	89	90				71	70	69	68	67		91
	XCZU17			65	64	66											90	91	93	74	73	72	71	70	69	68	67		94
	XCZU19			65	64	66											90	91	93	74	73	72	71	70	69	68	67		94

X18643-081518

Figure 7-4: I/O Bank Migration for B1517

GTU Quad 131 X0Y16-X0Y19	PCIE4 X0Y4	HP I/O Bank 71 T	HD I/O Bank 91	GTH Quad 231 X0Y28-X0Y31
GTU Quad 130 X0Y12-X0Y15	CMAC X0Y1	HP I/O Bank 70 U	HD I/O Bank 90 P	GTH Quad 230 X0Y24-X0Y27
GTU Quad 129 X0Y8-X0Y11 (RCAL)	ILKN X0Y0	HP I/O Bank 69 V	HD I/O Bank 89 O	GTH Quad 229 X0Y20-X0Y23
GTU Quad 128 X0Y4-X0Y7	PCIE4 X0Y2	HP I/O Bank 68 W	HD I/O Bank 88 N	GTH Quad 228 X0Y16-X0Y19
GTU Quad 127 X0Y0-X0Y3	CMAC X0Y0	HP I/O Bank 67 X	PCIE4 X1Y1	GTH Quad 227 X0Y12-X0Y15 D [R]
PS GTR 505	PS MIO 502	HP I/O Bank 66 E	SYSMON Configuration	GTH Quad 226 X0Y8-X0Y11 C [R] (RCAL)
PS DDR 504	PS MIO 501	HP I/O Bank 65 C	Configuration	GTH Quad 225 X0Y4-X0Y7 B [R]
PS CONFIG 503	PS MIO 500	HP I/O Bank 64 D	PCIE4 X1Y0 (tandem)	GTH Quad 224 X0Y0-X0Y3 A [R]

X18627-010917

Figure 7-5: XCZU11 Bank Locations for B1517

GTU Quad 134 X0Y28-X0Y31	CMAC X0Y3	HP I/O Bank 74 Q	HD I/O Bank 94	GTH Quad 234 X0Y40-X0Y43
GTU Quad 133 X0Y24-X0Y27	ILKN X0Y3	HP I/O Bank 73 R	HD I/O Bank 93 P	GTH Quad 233 X0Y36-X0Y39
GTU Quad 132 X0Y20-X0Y23	CMAC X0Y2	HP I/O Bank 72 S	ILKN X1Y2	GTH Quad 232 X0Y32-X0Y35
GTU Quad 131 X0Y16-X0Y19	PCIE4 X0Y4	HP I/O Bank 71 T	HD I/O Bank 91 O	GTH Quad 231 X0Y28-X0Y31
GTU Quad 130 X0Y12-X0Y15	CMAC X0Y1	HP I/O Bank 70 U	HD I/O Bank 90 N	GTH Quad 230 X0Y24-X0Y27
GTU Quad 129 X0Y8-X0Y11 (RCAL)	ILKN X0Y0	HP I/O Bank 69 V	ILKN X1Y1	GTH Quad 229 X0Y20-X0Y23
GTU Quad 128 X0Y4-X0Y7	PCIE4 X0Y2	HP I/O Bank 68 W	PCIE4 X1Y3	GTH Quad 228 X0Y16-X0Y19
GTU Quad 127 X0Y0-X0Y3	CMAC X0Y0	HP I/O Bank 67 X	PCIE4 X1Y1	GTH Quad 227 X0Y12-X0Y15 D [R]
PS GTR 505	PS MIO 502	HP I/O Bank 66 E	SYSMON Configuration	GTH Quad 226 X0Y8-X0Y11 C [R] (RCAL)
PS DDR 504	PS MIO 501	HP I/O Bank 65 C	Configuration	GTH Quad 225 X0Y4-X0Y7 B [R]
PS CONFIG 503	PS MIO 500	HP I/O Bank 64 D	PCIE4 X1Y0 (tandem)	GTH Quad 224 X0Y0-X0Y3 A [R]

X18628-010917

Figure 7-6: ZCZU17 and XCZU19 Bank Locations for B1517



### 3. Power Supply Differences across Speed Grades and Temperature Grades

For programmable logic (PL), VCCINT, VCCINT\_IO, and VCCBRAM have voltage limits that can differ across the speed and temperature grades as shown in [Table 7-1](#). For the processing system (PS), VCC\_PSINTFP, VCC\_PSINTPL, VCC\_PSINTFP\_DDR, and VPS\_MGTRAVCC can have voltage limits that differ across the speed and temperature grades as shown in [Table 7-2](#).

Table 7-1: MPSoC PL Voltage Matrix by Speed/Temperature Grade

Speed/ Temperature Grade	VCCINT (V)	VCCINT_IO (V)	VCCBRAM (V)
-1E	0.85	0.85	0.85
-2E	0.85	0.85	0.85
-2LE	0.72 or 0.85	0.85	0.85
-3E	0.90	0.90	0.90
-1I	0.85	0.85	0.85
-1LI	0.72 or 0.85	0.85	0.85
-2I	0.85	0.85	0.85

**Notes:**

1. VCCINT\_IO must connect to VCCBRAM.

Table 7-2: MPSoC PS Voltage Matrix by Speed/Temperature Grade

Speed/Temperature Grade	VCC_PSINTFP (V)	VCC_PSINTLP (V)	VCC_PSINTFP_DDR (V)	VPS_MGTRAVCC(V)
-1E	0.85	0.85	0.85	0.85
-2LE	0.85	0.85	0.85	0.85
-3E	0.90	0.90	0.90	0.90
-1I	0.85	0.85	0.85	0.85
-1LI	0.85	0.85	0.85	0.85
-2I	0.85	0.85	0.85	0.85

**Notes:**

1. VCC\_PSINTFP\_DDR must connect to VCC\_PSINTFP.

### 4. -3 Speed Migration

CG devices and the XCZU2EG/XCZU3EG are not available in the -3 speed grade. CG devices can only migrate to EG or EV devices in non -3 speed grades to maintain similar performance. Likewise, EG and EV devices can only migrate to a CG device in non -3 speed grades to maintain similar performance. EG to EV devices can migrate to any speed grades, as well as EV to EG.

## 5. VCU Migration

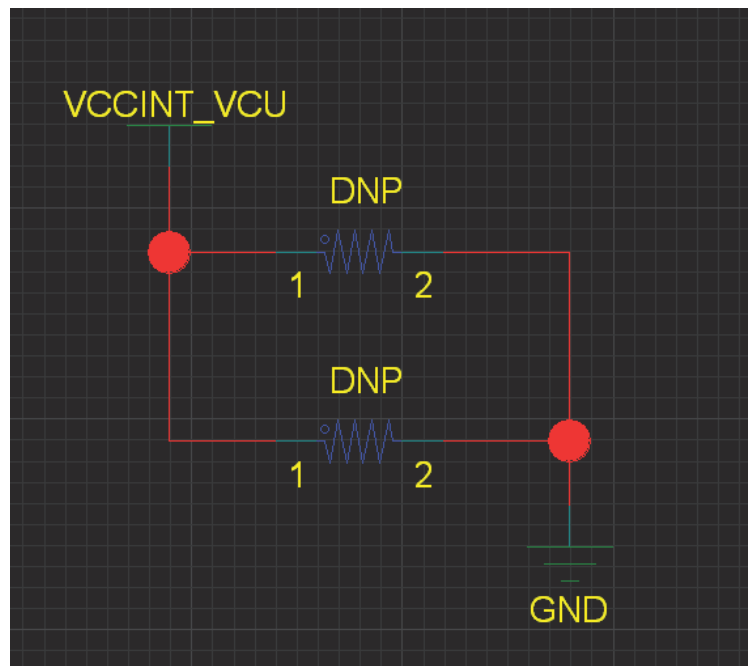
The VCU is only available in EV devices. EV devices have a number of VCCINT\_VCU power pins that correspond to ground pins on the CG and EG devices. If moving from a CG or EG to an EV device, the corresponding pins (VCCINT\_VCU) should remain grounded if the VCU is not used.



**IMPORTANT:** A feasible migration path only exists for moving from an EV device to a CG/EG device. To go to an EV device from a CG/EG device, the PCB would need to be modified substantially to add the regulator for VCCINT\_VCU.

### Example: Schematic/PCB Options to Enable VCU Migration from EV to CG/EG Devices

Figure 7-7 shows a schematic representation of a connection option to connect VCCINT\_VCU to ground when moving from an EV device to a CG or EG device. The migration path would be to DNP the VCCINT\_VCU regulator and populate the two previously DNP resistors to ground the VCCINT\_VCU planelet. To connect to ground (CG/EG devices), populate the resistors on the side with zero ohms. Two resistors are used to minimize inductance and increase current carrying ability.



X18629-020718

Figure 7-7: VCCINT\_VCU Connection Options for CG/EG/EV Migration (Schematic)

## 6. GPU Migration

The GPU is only available in EG and EV devices. If migrating from an EG or EV device to a CG device, the GPU should not be used.

## 7. Decoupling Capacitors

If the design is not expected to change when migrating to the new device, the same decoupling scheme can be utilized. If adding logic, Xilinx recommends using a decoupling scheme appropriate for the new device and design. Refer to [Table 1-9](#) for decoupling capacitor guidelines for the various Zynq UltraScale+ MPSoCs.

## 8. PCI Express Migration

PCI Express lane assignments might differ between devices. In addition, the block that is used as tandem boot might also be different from device to device. Refer to the GT Locations appendix in *UltraScale+ Devices Integrated Block for PCI Express LogiCORE IP Product Guide* (PG213) [\[Ref 27\]](#) for GT locations.

## 9. Integrated 100G Ethernet Migration

The 100G Ethernet IP core is a dedicated block that provides 100 Gb/s MAC and PCS logic capability. The block consists of a number of dedicated locations that are located at different points on different devices. There are specific rules for connecting these dedicated blocks to specific transceiver locations, which might impact migration to another device. In addition, the CAUI-4 and switchable CAUI-10/CAUI-4 modes require GTY transceivers.

Refer to *UltraScale+ Devices Integrated 100G Ethernet Subsystem Product Guide* (PG203) [\[Ref 28\]](#) for a more comprehensive guide regarding designing with the 100G Ethernet core, with special attention to the Transceiver Selection Rules section.

## 10. Interlaken Migration

The Xilinx UltraScale architecture integrated IP core for Interlaken is a highly configurable integrated IP core that can support an overall bandwidth up to 150 Gb/s for protocol logic transmission. The core connects to serial transceivers at defined rates up to 12.5 Gb/s with GTH transceivers, and up to 25.78125 Gb/s with GTY transceivers. The block consists of a number of dedicated locations that are located at different points on different devices. There are specific rules for connecting these dedicated blocks to specific transceiver locations, which might impact migration to another device. Refer to *Interlaken 150G LogiCORE IP Product Guide* (PG212) [\[Ref 29\]](#) for a more comprehensive guide regarding the proper generation and verification of Interlaken interfaces.

## 11. Power Supplies and Thermal Considerations

Migration across devices, especially from smaller to bigger devices, can result in higher power consumption requiring improved thermal management and power delivery. Consult the Xilinx Power Estimator (XPE) tool and select the largest device being considered in order to estimate power consumption and thermal requirements.



---

**RECOMMENDED:** Xilinx recommends use of the Delphi thermal model during thermal modeling of a package. The Delphi thermal model includes consideration of the thermal interface material parameters and the manufacture variation on the thermal solution. Examples of manufacture variations include the tolerance in airflow from a fan, the tolerance on performance of the heat pipe and vapor chamber, and the manufacture variation of the attachment of fins to the heat sink base and the flatness of the surface. Further information regarding thermal considerations can be found in the Thermal Management Strategy chapter of Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide (UG1075) [Ref 26].

---

## 12. Pin Flight Times across Packages

Even if two devices are footprint compatible from a design and PCB standpoint, FPGA package flight times will be different across different devices, so they will need to be accounted for. Flight time information can be found within the Vivado® tools during the I/O planning stage or after synthesis. The ideal strategy to account for pin flight time differences is to deskew the printed circuit board when migrating to the new device. If this is not possible, Xilinx recommends laying out the printed circuit board with the final device in mind to maximize system performance for the long term. System performance might have to be derated when using the initial device. As a last option, choosing the midpoint of the range of flight times and routing the board based on that value can act as a compromise, though maximum system performance might not be achievable with this method. Pin flight time information can be obtained via the Package Pins tab within the Vivado tools, both in the I/O planning stage and after synthesis.

### **Example 1: Obtaining Pin Flight Times During I/O Planning**

Open the Vivado tools and follow the following steps in order to create an I/O planning project:

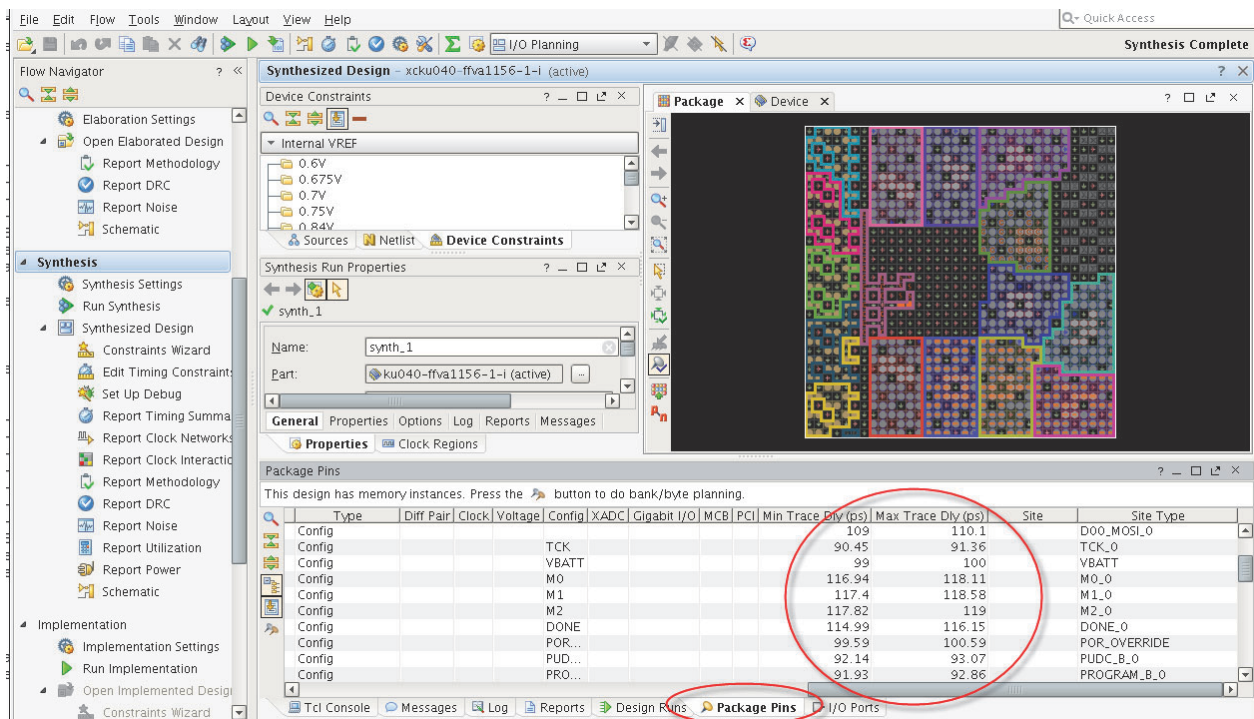
1. Double-click **Create New Project**, then click **Next**.
2. Choose a project name and location, then click **Next**.
3. Specify **I/O Planning Project**, then click **Next**.
4. Click **Do not import I/O ports at this time**, then click **Next**.
5. Select the device for which you wish to desire pin delays, then click **Next**.
6. Click **Finish**.

7. When the project loads, click **Export I/O Ports** from the Project Manager pane.
8. Make sure **CSV** is checked, select a filename and location, then click **OK**.

The resultant CSV file lists all I/Os and their associated pin delays under the Min Trace and Max Trace columns.

**Example 2: Obtaining Pin Flight Times after Synthesis**

After synthesis has completed, open the synthesized design (**Synthesis > Open Synthesized Design**), then click the **Package Pins** tab, as shown in Figure 7-8. Pin delays are found in the Min Trace Dly and Max Trace Dly columns.



X18631-010917

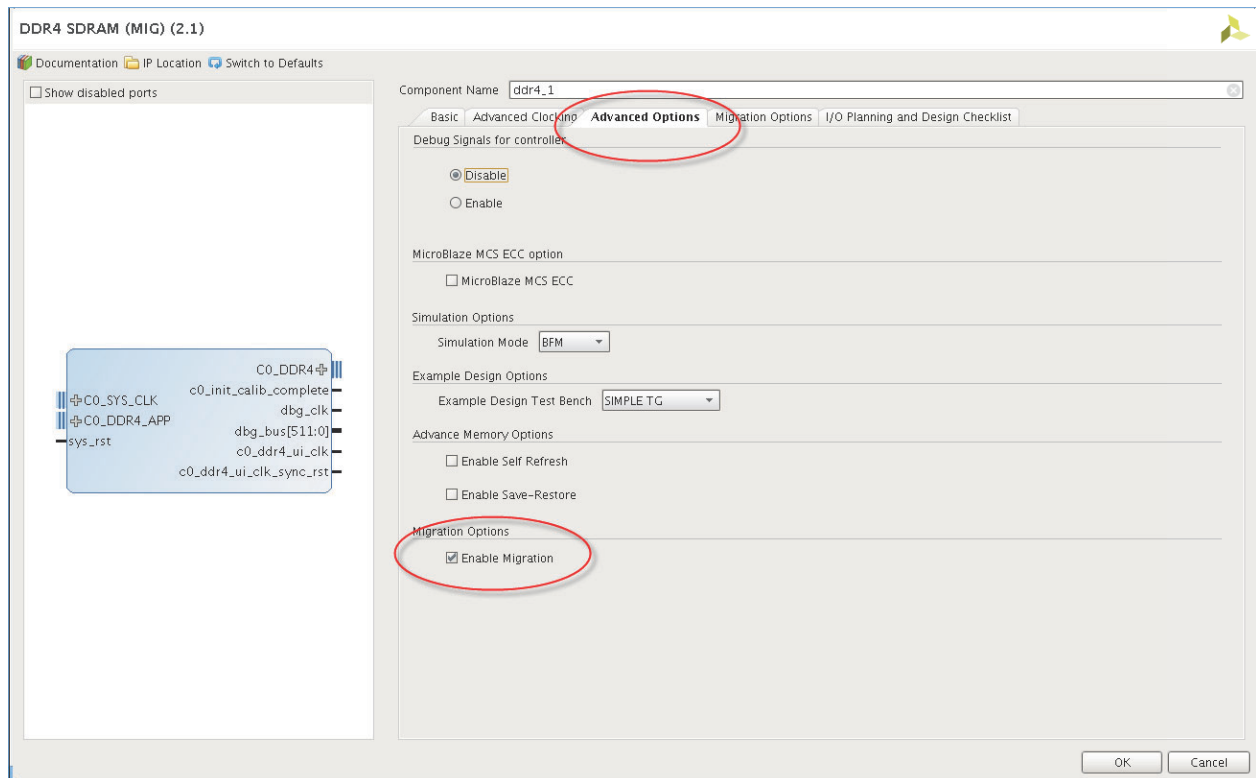
Figure 7-8: Pin Delays shown in the Vivado Tools

Alternatively, the Tcl command `write_csv {filename.csv}` can be used in the Tcl Console tab to export I/O information to a file that includes the pin delays.

### Using the Enable Migration Feature for DDR4 Memory Designs (Vivado Tools 2016.3)

A new feature for DDR4 memories in the Memory Interface Generator (MIG) tool allows users to prepare for migration from one device to another by specifying the differences in address/command/control pin delays from the initial device to the migrated device.

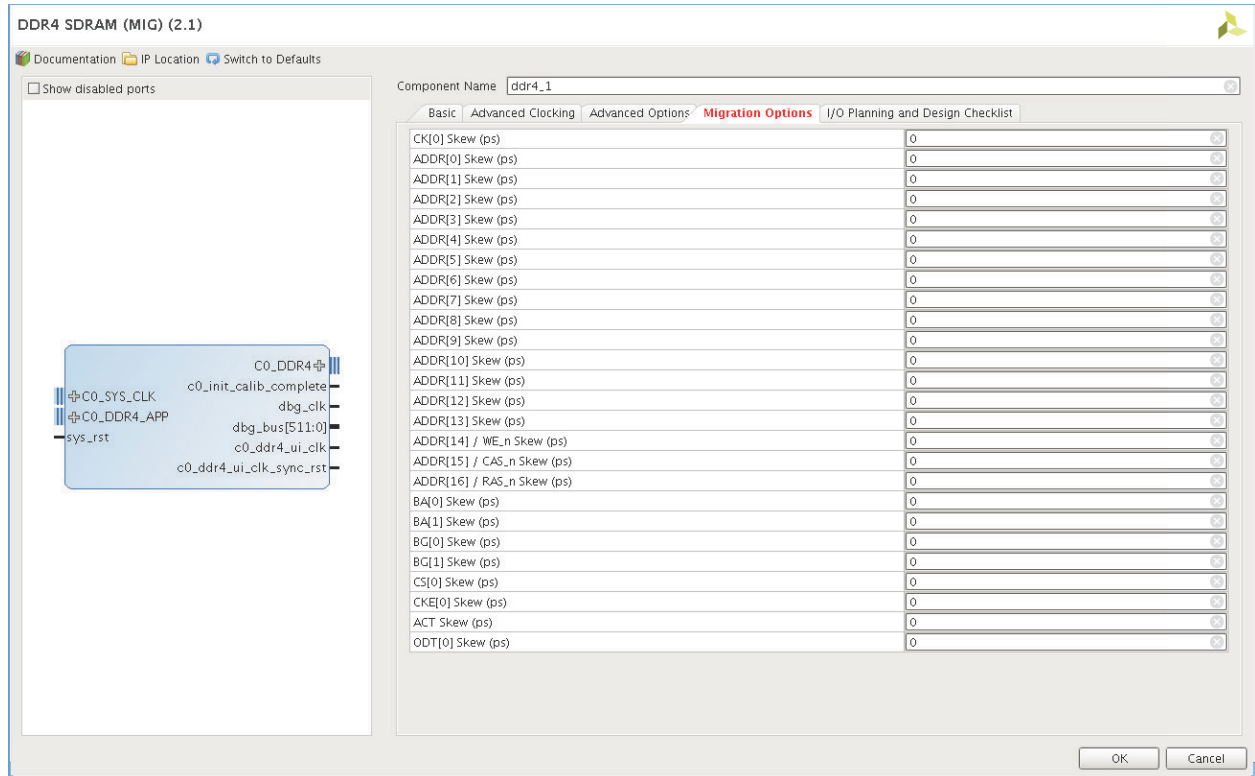
To enable the DDR4 migration feature in the MIG tool, select the **Enable Migration** checkbox in the Advanced Options tab, as shown in Figure 7-9.



X18632-010917

Figure 7-9: Selecting the Enable Migration Feature for DDR4 Designs

After enabling migration, select the **Migration Options** tab and enter the relevant skew values as outlined in *UltraScale Architecture FPGAs Memory IP Product Guide* (PG150) [Ref 13] between the two devices (Figure 7-10). The pin delays can be found via [Example 1: Obtaining Pin Flight Times During I/O Planning](#) or [Example 2: Obtaining Pin Flight Times after Synthesis](#). The delay values entered should be positive and between 0 and 75 ps.



X18633-010917

Figure 7-10: Entering the Pin Delay Differences for Migration

# SelectIO Signaling

The UltraScale architecture SelectIO resources are the general-purpose I/O and its various settings. With numerous I/O standards and hundreds of variants within these standards, these SelectIO resources offer a flexible array of choices for designing I/O interfaces.

This chapter provides some strategies for choosing I/O standard, topography, and termination, and offers guidance on simulation and measurement for more detailed decision making and verification. In many cases, higher-level aspects of the system (other device choices or standards support) define the I/O interfaces to be used. In cases where such constraints are not defined, it is up to the system designer to choose I/O interface standards and optimize them according to the purpose of the system.

This chapter contains the following sections:

- [Interface Types](#)
- [Single-Ended Signaling](#)

---

## Interface Types

To better address the specifics of the various interface types, it is necessary to first break interfaces into categories. Two relevant divisions are made:

- [Single-Ended versus Differential Interfaces](#)
- [SDR versus DDR Interfaces](#)

## Single-Ended versus Differential Interfaces

Traditional digital logic uses single-ended signaling – a convention that transmits a signal and assumes a GND common to the driver and receiver. In single-ended interfaces, a signal's assertion (whether it is High or Low) is based on its voltage level relative to a fixed voltage threshold that is referenced to GND. When the voltage of the signal is higher than the  $V_{IH}$  threshold, the state is considered High. When the voltage of the signal is lower than the  $V_{IL}$  threshold, the state is considered Low. TTL is one common example of a single-ended I/O standard.



To reach higher interface speeds and increase noise margin, some single-ended I/O standards rely on a precise dedicated local reference voltage other than GND. HSTL and SSTL are examples of I/O standards that rely on a  $V_{REF}$  to resolve logic levels.  $V_{REF}$  can be thought of as a fixed comparator input.

Higher-performance interfaces typically make use of differential signaling – a convention that transmits two complementary signals referenced to one another. In differential interfaces, a signal's assertion (whether it is High or Low) is based on the relative voltage levels of the two complementary signals. When the voltage of the P signal is higher than the voltage of the N signal, the state is considered High. When the voltage of the N signal is higher than the voltage of the P signal, the state is considered Low. Typically the P and N signals have similar swing, and have a common-mode voltage above GND (although this is not always the case). LVDS is one common example of a differential I/O standard.

## SDR versus DDR Interfaces

The difference between Single Data Rate (SDR) and Double Data Rate (DDR) interfaces has to do with the relationship of the data signals of a bus to the clock signal of that bus. In SDR systems, data is only registered at the input flip-flops of a receiving device on either the rising *or* the falling edge of the clock. One full clock period is equivalent to one bit time. In DDR systems, data is registered at the input flip-flops of a receiving device on both the rising *and* falling edges of the clock. One full clock period is equivalent to two bit times. The distinction of SDR and DDR has nothing to do with whether the I/O standard carrying the signals is single-ended or differential. A single-ended interface can be SDR or DDR, and a differential interface can also be SDR or DDR.

---

## Single-Ended Signaling

A variety of single-ended I/O standards are available in the UltraScale architecture I/O. For a complete list of supported I/O standards and detailed information about each one, refer to the *SelectIO Resources* chapter of the *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10]. Tables at the end of this chapter summarize for each supported I/O standard which ones support DRIVE and SLEW attributes, bidirectional buffers, and the DCI options. It also describes which I/O standards are supported in the high-performance (HP) and high-range (HR) I/O banks.

## Modes and Attributes

Some I/O standards can be used only in unidirectional mode, while some can be used in bidirectional mode or unidirectional mode.

Some I/O standards have attributes to control drive strength and slew rate, as well as the presence of weak pull-up or pull-down and weak-keeper circuits (not intended for use as parallel termination). Drive strength and slew rate can be used to tune an interface for

adequate speed while not overdriving the signals. Weak pull-ups, weak pull-downs, and weak keepers can be used to ensure a known or steady level on a floating or 3-stated signal. The *SelectIO Resources* chapter of the *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10] describes which standards support these attributes. Refer to this user guide for more information.

LVC MOS, when set to 6 mA DRIVE and FAST slew, has an approximate output impedance close to  $50\Omega$ , allowing it to be used as a crude approximation of a controlled-impedance driver. The impedance match of the weak driver to the transmission line is only approximate and varies with voltage and temperature. LVDCI and HSLVDCI, true controlled-impedance drivers, are adaptive, maintain a much closer impedance match, and remain constant over voltage and temperature.

## Input Thresholds

The input circuitry of the single-ended standards fall into two categories: those with fixed input thresholds and those with input thresholds set by the  $V_{REF}$  voltage. The use of  $V_{REF}$  has three advantages:

- It allows for tighter control of input threshold levels
- It removes dependence on die GND for the threshold reference
- It allows for input thresholds to be closer together, which reduces the need for a large voltage swing of the signal at the input receiver

Two 1.8V I/O standards that illustrate this are LVC MOS18 and SSTL18 Class 1. The thresholds for 1.8V LVC MOS are set at 0.63V and 1.17V (necessitating that the signal at the receiver swing a full 540 mV at minimum to make a logic transition). The thresholds for SSTL18 Class 1 are set at  $V_{REF} - 0.125V$  and  $V_{REF} + 0.125V$ , or for a nominal  $V_{REF}$  of 0.9V, set at 0.775V and 1.025V (necessitating that the signal at the receiver only swing 250 mV at minimum to make a logic transition). This smaller required swing allows for higher frequency of operation in the overall link. A smaller swing at the driver means reduced DC power is required with less transient current. In the UltraScale architecture-based devices, the reference voltage can either be provided using the dedicated  $V_{REF}$  pins, or optionally generated internally using the Internal  $V_{REF}$  feature. See the *SelectIO Resources* chapter of the *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10] for more details on Internal  $V_{REF}$ . For more information on  $V_{REF}$  decoupling and decoupling of all other supplies, see [Chapter 1, Power Distribution System in UltraScale Devices](#).

## Topographies and Termination

Topography generally refers to the arrangement of drivers, receivers, interconnect and terminations in an interface. The techniques used in unidirectional topographies are different from those used in bidirectional topographies, so these are treated separately.

The SelectIO standards can be used in countless topographies depending on the requirements of the system. SelectIO drivers and receivers adhering to a standard (SSTL, LVCMOS, etc.) either can be used according to the letter of the standard (published by a standards body such as EIA/TIA or JEDEC) or they can be mixed and matched with drivers or receivers from another standard or hybrid I/O. An I/O standard specification might define something as limited as the  $V_{IL}$  and  $V_{IH}$  of the receiver, or it might define every aspect of the interface, including driver impedance and slew rate, PCB trace length and topography, value and position of passive termination, the maximum input capacitance of a receiving device, and even the maximum number of receivers.

It is up to the designer to apply the standard in question to the system in which it is working. There are many decisions to make with respect to topographies and termination, which determine the signal integrity of the interface. It is of utmost importance that the signal integrity of each interface be verified through both simulation and measurement.

Termination generally refers to impedance-matching or impedance-compensating devices that are used to maintain signal integrity in an interface. While many types of elements can be used as *terminators* (such as, resistors, capacitors, diodes), this discussion is limited to resistive termination. In general, capacitor and diode termination techniques are more complicated.

### Unidirectional Topographies and Termination

The two basic subsets of unidirectional topographies are point-to-point and multi-drop. A point-to-point topography has one driver and one receiver, while a multi-drop topography has one driver and many receivers. Whether or not a topography is point-to-point or multi-drop defines important aspects of the interface that determine which termination strategies are appropriate and which are not.

#### Unidirectional Point-to-Point Topographies

The simplest unidirectional topography is point-to-point. That is, there is one driver and one receiver. Termination, if present, can consist of parallel termination at the receiver (Figure 8-1), series termination at the driver (Figure 8-2), or a controlled-impedance driver (Figure 8-3 and Figure 8-4). Always use IBIS simulation to determine the optimal resistor values,  $V_{TT}$  voltage level, and VRP reference resistors for these terminations.

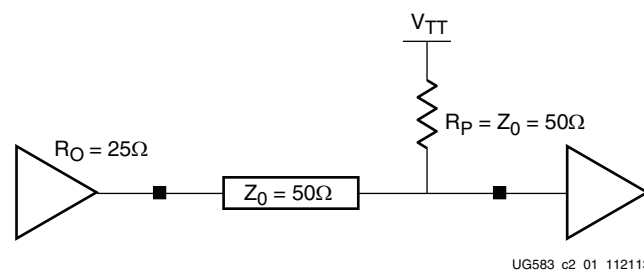


Figure 8-1: Parallel-Terminated Unidirectional, Point-to-Point Topography

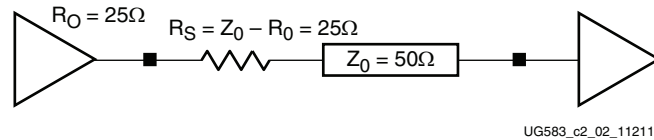


Figure 8-2: **Series-Terminated Unidirectional, Point-to-Point Topography**

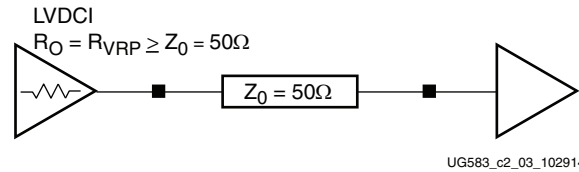


Figure 8-3: **DCI-Controlled Impedance Driver Unidirectional, Point-to-Point Topography**

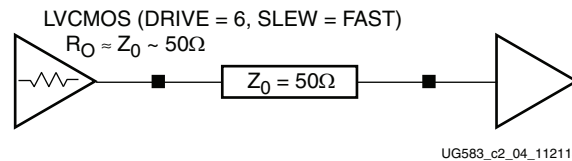


Figure 8-4: **“Weak Driver” Unidirectional, Point-to-Point Topography**

In general, parallel resistive termination ( $R_p$ ) has a value equal to the characteristic impedance ( $Z_0$ ) of the transmission line it is terminating. Series resistive terminations ( $R_s$ ) have a value equal to the characteristic impedance of the transmission line ( $Z_0$ ) minus the output impedance of the driver ( $R_O$ ) to which they are connected. Controlled-impedance drivers are tuned such that the driver output impedance ( $R_O$ ) is equal to the characteristic impedance ( $Z_0$ ) of the transmission line it is terminating.

Assuming transmission lines with  $50\Omega$  characteristic impedance and a driver output impedance ( $R_O$ ) of  $25\Omega$ , a  $25\Omega$  series termination (Figure 8-2) or a  $50\Omega$  parallel termination (Figure 8-1) is appropriate. Controlled-impedance drivers, whether implemented with DCI or with weak LVCMOS drivers, should be sized to have an output impedance ( $R_O$ ) of  $50\Omega$ . This corresponds to a VRP resistor equal to  $50\Omega$  for DCI. Weak LVCMOS drivers of 6 mA to 8 mA drive strength have an output impedance approximately equal to  $50\Omega$  (Figure 8-3).

Typically, parallel terminations have best performance when  $V_{TT}$  (the voltage source connected to the parallel termination resistor) is equal to half of the signaling voltage. For 2.5V signals ( $V_{CC0} = 2.5V$ ),  $V_{TT}$  is ideally 1.25V. In cases where this voltage is not available, it is possible to use a Thevenin parallel termination. Thevenin parallel termination consists of a voltage divider with a parallel equivalent resistance ( $R_{PEQ}$ ) equal to the characteristic impedance of the transmission line ( $50\Omega$  in most cases). The divided voltage point is designed to be at  $V_{TT}$ . Figure 8-5 illustrates a Thevenin parallel termination powered from 2.5V  $V_{CC0}$ , made up of two  $100\Omega$  resistors, resulting in a  $V_{TT}$  of 1.25V and a parallel equivalent resistance ( $R_{PEQ}$ ) of  $50\Omega$ .

Parallel termination can be less desirable than series termination or controlled-impedance drivers because it dissipates more power. This trade-off must be weighed against other trade-offs to determine the optimum termination topography for an interface.

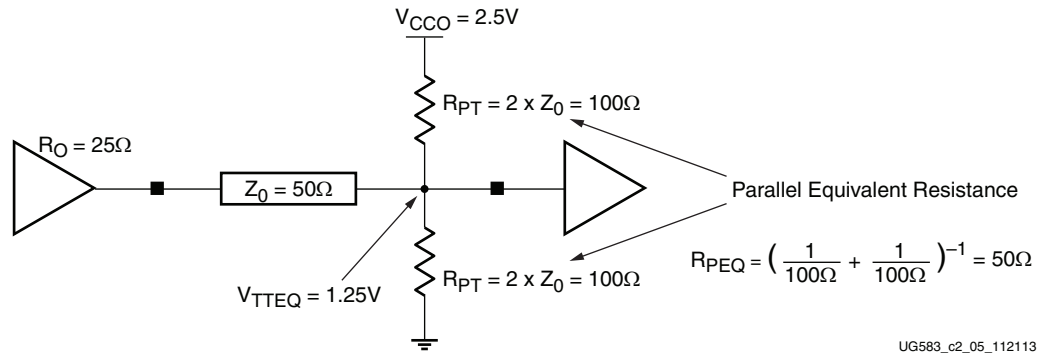


Figure 8-5: Thevenin Parallel Termination

Table 8-1 lists example I/O interface types that can be used with the unidirectional point-to-point topography.

Table 8-1: Example I/O Interface Type for Unidirectional Point-to-Point Topographies

LVTTTL
LVC MOS
LVDCI
SSTL Class I
HSTL Class I

LVTTTL and LVC MOS do not specify any canonical termination method. Series termination at the driver or parallel termination at the receiver are both appropriate considerations.

LVDCI implicitly uses controlled-impedance driver termination. No form of termination is needed at the receiver.

Every I/O standard can have different requirements for termination techniques. In some cases the specification for the I/O standard can rigidly define the termination topography. Other standards might not have any hard requirements, but rather might simply provide examples of termination topologies. An example of a standard with specific termination requirements is HSTL. HSTL Class I is a unidirectional I/O standard that recommends a parallel termination at the receiver. In the case of HSTL Class I, the termination voltage  $V_{TT}$  is defined as half of the supply voltage  $V_{CC}$ . The designer can ultimately elect either not to use termination at all or to use a different termination, such as series termination at the driver. There are a number of reasons why this selection might be advantageous in a given system. It is up to the designer to verify through simulation and measurement that the signal integrity at the receiver is adequate.

The SSTL standards tend to not have rigid requirements for termination topology. Rather, the JEDEC specifications provide example termination techniques that tend to be the commonly used topographies. The *SelectIO Resources* chapter of the *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10] provides example termination techniques for each of the I/O standards, including the SSTL standards, for the purpose of providing a good starting point for consideration. Similar to HSTL, it is ultimately up to the designer to verify through simulation and measurement that the signal integrity at the receiver is adequate.

### Unidirectional Multi-Drop Topographies

In more complex topographies, a single driver can drive multiple receivers. The receivers represent loads that must be fed by individual transmission line stubs. From a signal integrity standpoint, the best topography to use in this case is a single long transmission line with the driver at one end and parallel termination at the other, with receivers connected to the main trace by short stubs in between. This type of topography is often referred to as a *flyby multi-drop* topography.

There are two critical aspects of this topography. The first is the presence of a single parallel termination at the far end of the transmission line. Series termination at the driver or a controlled impedance driver must *never* be used. Parallel termination is the only applicable termination type for this topography. The second critical aspect is the length of the connecting stubs at each receiver. These must remain short: no more than a fraction of a signal rise time in length. With a typical signal rise time of 600 ps, a stub no longer than  $700 \text{ ps}/4 = 150 \text{ ps}$ , or 0.9 inches (22.86 mm) should be used. As the stubs become longer, they present a larger impedance discontinuity to the signal travelling down the transmission line, and can support significant reflections. These impedance discontinuities corrupt the signal. With increasing numbers of loads and increasing length of stubs, the signal is corrupted to the point where it is no longer usable.

Star topographies are not recommended. The constraints involved in designing a star topography with good signal integrity are beyond the scope of this document.

As stated in [Unidirectional Point-to-Point Topographies](#), ideal parallel resistive termination has a value equal to the characteristic impedance of the transmission line it is terminating. The best performance is achieved when  $V_{TT}$  is equal to half of the signaling voltage, and when this voltage is not available, a Thevenin parallel termination is recommended, as defined in the previous section.

[Figure 8-6](#) illustrates a Thevenin parallel termination powered from  $V_{CC0}$ , made up of two  $100\Omega$  resistors, resulting in a  $V_{TT}$  of  $V_{CC0}/2$  and a parallel equivalent resistance of  $50\Omega$ . This figure shows a topography with one driver (an LVCMOS driver) and four receivers. The driver is on the left side, the receivers are spaced at interim points across the  $50\Omega$  transmission line, and the Thevenin parallel termination of two  $100\Omega$  resistors is on the right side.

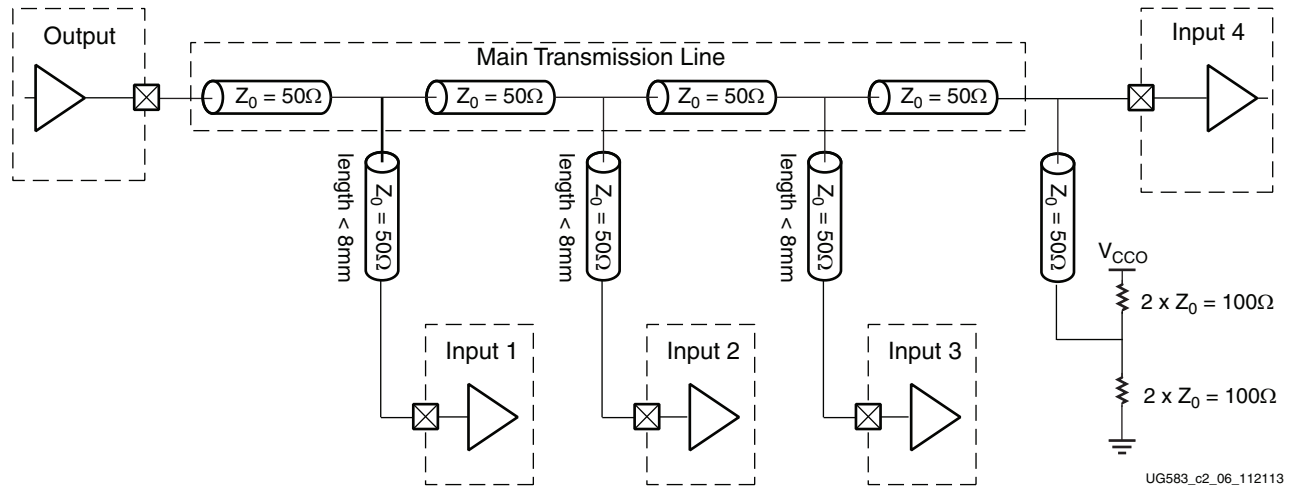


Figure 8-6: Basic Multi-Drop Topography

The main transmission line should be kept as short as possible. Lengths up to 20 inches or more are practical for most I/O standards as long as precise trace impedance is maintained and crosstalk sources are avoided. The lengths of interim segments of the main transmission line need not be equal. Their relative lengths can be arbitrary. Receivers at different points along the main transmission line receive the signal with varying amounts of delay, but all signal rise times are similar.

*Stubs stretching from the main transmission line to the individual receivers must be kept as short as possible.* The longer these stubs become, the more corrupted the received waveforms are. Simulation and measurement are required to assess signal integrity at the individual receivers.

Table 8-2 lists example I/O interface types that can be used with the unidirectional multi-drop topography.

Table 8-2: Example I/O Interface Types for Unidirectional Multi-Drop I/O Topographies

LVTTTL
LVC MOS
HSTL
SSTL

LVTTTL and LVC MOS do not specify any canonical termination method. Parallel termination at the end of the long t-line is an appropriate termination method.

### Bidirectional Topography and Termination

The two basic subsets of bidirectional topographies are point-to-point and multi-point. A point-to-point topography has two transceivers (driver and receiver sharing one device pin), while a multi-point topography can have many transceivers. Whether or not a

topography is point-to-point or multi-point defines important aspects of the interface that determine which termination strategies are appropriate and which are not.

**Bidirectional Point-to-Point Topographies**

The simplest bidirectional topography is point to point. That is, there are two transceivers connected by a transmission line. Because bidirectional interfaces need to operate equally well in both directions, symmetry of the topography is desirable. While asymmetrical topographies can be designed with reasonably good signal integrity, the easiest way to ensure good signal integrity is to keep the topography symmetrical. Thus any termination used on one side of the link should also be used on the other side of the link. Series termination (Figure 8-8) is rarely appropriate for bidirectional interfaces as incoming signals are attenuated by the series resistor of the receiving transceiver. Parallel termination (Figure 8-7) almost always achieves better signal levels at both receivers. Controlled-impedance drivers, whether crudely controlled in the form of a weak LVCMOS driver or adaptively controlled in the form LVDCI or HSLVDCI, also can have good results as shown in Figure 8-9 (implemented with a low-drive strength LVCMOS driver). Always use IBIS simulation to determine the optimal termination resistor value,  $V_{TT}$  voltage level and VRP reference resistor values for these terminations.

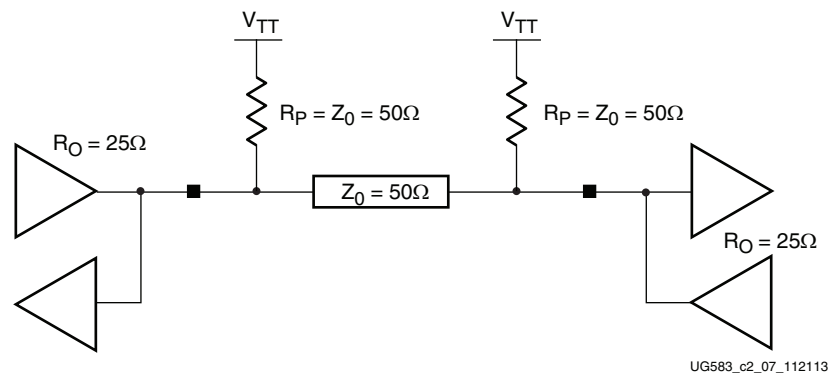


Figure 8-7: Parallel Terminated Bidirectional Point-to-Point Topography

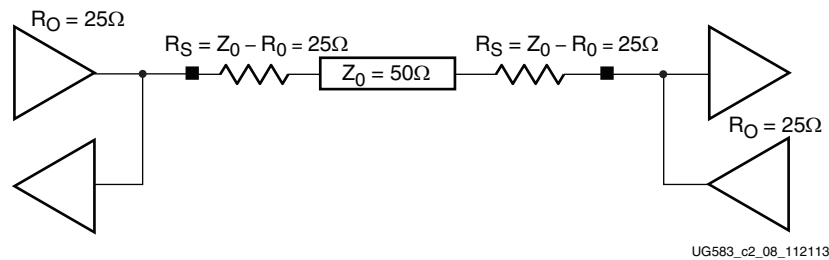


Figure 8-8: Series Terminated Bidirectional Point-to-Point Topography: Not Recommended



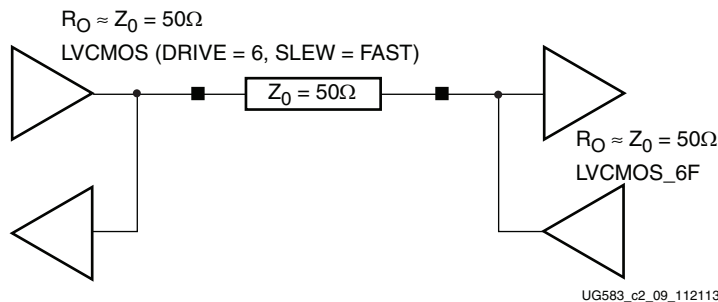


Figure 8-9: “Weak Driver” Bidirectional Point-to-Point Topography

In general, parallel resistive termination ( $R_p$ ) has a value equal to the characteristic impedance  $Z_0$  of the transmission line it is terminating. Some interfaces, such as DDR2 memory interfaces, use 75Ω termination resistors instead of 50Ω in an effort to open the data eye. In this case, the trade-off is eye height against a small amount of signal reflection from the impedance discontinuity. Controlled-impedance drivers are typically tuned such that the driver output impedance ( $R_O$ ) is equal to the characteristic impedance ( $Z_0$ ) of the transmission line it is terminating.

Assuming transmission lines with 50Ω characteristic impedance and a driver output impedance of 25Ω, 50Ω parallel terminations are appropriate (Figure 8-7). Controlled-impedance drivers, whether implemented with DCI or with weak LVC MOS drivers, should be sized to have an output impedance ( $R_O$ ) of 50Ω. An example of the use of a controlled-impedance driver would be the LVDCI\_15 I/O standard. Weak LVC MOS drivers of 6 mA to 8 mA drive strength have an output impedance approximately equal to 50Ω (Figure 8-9).

Parallel terminations have the best performance when  $V_{TT}$  (the voltage source connected to the parallel termination resistor) is equal to half of the signaling voltage, since this is typically the center voltage of the data eye. For 2.5V signals ( $V_{CCO} = 2.5V$ ),  $V_{TT}$  is ideally 1.25V. In cases where this voltage is not available, it is advisable to use a Thevenin parallel termination. Thevenin parallel termination consists of a voltage divider with a parallel resistance equal to the characteristic impedance of the transmission line (50Ω in most cases). The divided voltage point is designed to be at  $V_{TT}$ . Figure 8-10 illustrates a Thevenin parallel termination powered from 2.5V  $V_{CCO}$ , made up of two 100Ω resistors, resulting in a  $V_{TT}$  of 1.25V and a parallel equivalent resistance ( $R_{PEQ}$ ) of 50Ω.

Parallel termination can be less desirable than series termination or controlled-impedance drivers because it dissipates more power. This trade-off must be weighed against other trade-offs to determine the optimum termination topography for an interface.

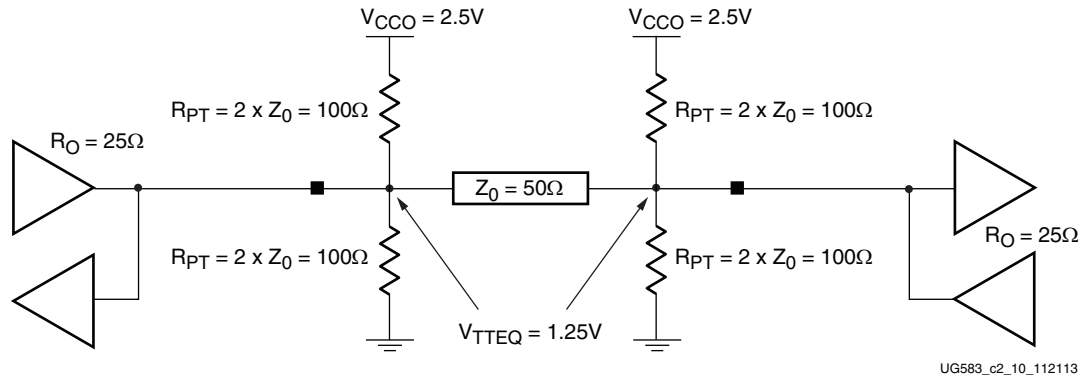


Figure 8-10: Thevenin Parallel Termination (Bidirectional Point-to-Point Topography)

Table 8-3 lists example I/O interface types that can be used with the bidirectional point-to-point topography.

Table 8-3: Example I/O Interface Types for Bidirectional Point-to-Point I/O Topographies

LVTTL
LVC MOS
LVDCI
HSLVDCI
SSTL15
SSTL15 DCI
SSTL18 CLASS II
SSTL18 CLASS II DCI
HSTL CLASS II
HSTL CLASS II DCI

LVTTL and LVC MOS do not specify any canonical termination method. Series termination is not recommended for bidirectional interfaces. Parallel termination and weak drivers, however, are both appropriate.

LVDCI and HSLVDCI both implicitly use controlled-impedance driver termination.

HSTL Class II specifies parallel termination at both transceivers. The termination voltage  $V_{TT}$  is defined as half of the supply voltage  $V_{CCO}$ . The designer can elect either not to use termination at all or to use a different termination. It is up to the designer to verify through simulation and measurement that the signal integrity at the receiver is adequate.

The JEDEC specifications for SSTL provide examples of both series termination and parallel termination. The termination voltage  $V_{TT}$  is defined as half of the supply voltage  $V_{CCO}$ . While the specification document provides examples depicting series termination at the drivers, it is important to note that the purpose of this is to attempt to match the impedance of the driver with that of the transmission line. Because the UltraScale

architecture SSTL drivers target to have output impedances close to 40–50Ω, better signal integrity can be achieved without any external source-series termination. When possible, it is a better starting point to consider the use of the 3-state DCI I/O standards (“T\_DCI”), which provide internal parallel termination resistors that are only present when the output buffer is in 3-state. It is up to the designer to carefully choose the I/O standard(s) at the Xilinx device, drive strengths, and on-die termination (ODT) options at the other device(s) in the interface (usually DRAM ICs) and termination topography though careful simulation and measurement. See the *UltraScale Architecture SelectIO Resources User Guide* (UG571) [Ref 10] for more details on the available I/O standards and options.

### ***Bidirectional Multi-Point Topographies***

In more complex topographies, any transceiver in a multi-point bus can transmit to all other transceivers. Usually these topographies can only run at very slow clock rates because they only support very slow signal rise times (10 ns to 50 ns). While useful in some situations, the drawbacks usually outweigh the benefits. The constraints involved in designing these topographies with good signal integrity are beyond the scope of this document.

# PCB Technology Basics

---

## Introduction

Printed circuit boards (PCBs) are electrical systems, with electrical properties as complicated as the discrete components and devices mounted to them. The PCB designer has complete control over many aspects of the PCB; however, current technology places constraints and limits on the geometries and resulting electrical properties. The following information is provided as a guide to the freedoms, limitations, and techniques for PCB designs using FPGAs.

This chapter contains the following sections:

- [PCB Structures](#)
  - [Transmission Lines](#)
  - [Return Currents](#)
- 

## PCB Structures

PCB technology has not changed significantly in the last few decades. An insulator substrate material (usually FR4, an epoxy/glass composite) with copper plating on both sides has portions of copper etched away to form conductive paths. Layers of plated and etched substrates are glued together in a stack with additional insulator substrates between the etched substrates. Holes are drilled through the stack. Conductive plating is applied to these holes, selectively forming conductive connections between the etched copper of different layers.

While there are advancements in PCB technology, such as material properties, the number of stacked layers used, geometries, and drilling techniques (allowing holes that penetrate only a portion of the stackup), the basic structures of PCBs have not changed. The structures formed through the PCB technology are abstracted to a set of physical/electrical structures: traces, planes (or planelets), vias, and pads.

## Traces

A trace is a physical strip of metal (usually copper) making an electrical connection between two or more points on an X-Y coordinate of a PCB. The trace carries signals between these points.

## Planes

A plane is an uninterrupted area of metal covering the entire PCB layer. A planelet, a variation of a plane, is an uninterrupted area of metal covering only a portion of a PCB layer. Typically, a number of planelets exist in one PCB layer. Planes and planelets distribute power to a number of points on a PCB. They are very important in the transmission of signals along traces because they are the return current transmission medium.

## Vias

A via is a piece of metal making an electrical connection between two or more points in the Z space of a PCB. Vias carry signals or power between layers of a PCB. In current plated-through-hole (PTH) technology, a via is formed by plating the inner surface of a hole drilled through the PCB. In current microvia technology (also known as High Density Interconnect or HDI), a via is formed with a laser by ablating the substrate material and deforming the conductive plating. These microvias cannot penetrate more than one or two layers, however, they can be stacked or stair-stepped to form vias traversing the full board thickness.

## Pads and Antipads

Because PTH vias are conductive over the whole length of the via, a method is needed to selectively make electrical connections to traces, planes, and planelets of the various layers of a PCB. This is the function of pads and antipads.

Pads are small areas of copper in prescribed shapes. Antipads are small areas in prescribed shapes where copper is removed. Pads are used both with vias and as exposed outer-layer copper for mounting of surface-mount components. Antipads are used mainly with vias.

For traces, pads are used to make the electrical connection between the via and the trace or plane shape on a given layer. For a via to make a solid connection to a trace on a PCB layer, a pad must be present for mechanical stability. The size of the pad must meet drill tolerance/registration restrictions.

Antipads are used in planes. Because plane and planelet copper is otherwise uninterrupted, any via traveling through the copper makes an electrical connection to it. Where vias are not intended to make an electrical connection to the planes or planelets passed through, an antipad removes copper in the area of the layer where the via penetrates.

## Lands

For the purposes of soldering surface mount components, pads on outer layers are typically referred to as lands or solder lands. Making electrical connections to these lands usually requires vias. Due to manufacturing constraints of PTH technology, it is rarely possible to place a via inside the area of the land. Instead, this technology uses a short section of trace connecting to a surface pad. The minimum length of the connecting trace is determined by minimum dimension specifications from the PCB manufacturer. Microvia technology is not constrained, and vias can be placed directly in the area of a solder land.

## Dimensions

The major factors defining the dimensions of the PCB are PCB manufacturing limits, FPGA package geometries, and system compliance. Other factors such as Design For Manufacturing (DFM) and reliability impose further limits, but because these are application specific, they are not documented in this user guide.

The dimensions of the FPGA package, in combination with PCB manufacturing limits, define most of the geometric aspects of the PCB structures described in this section ([PCB Structures](#)), both directly and indirectly. This significantly constrains the PCB designer. The package ball pitch (1.0 mm for FF packages) defines the land pad layout. The minimum surface feature sizes of current PCB technology define the via arrangement in the area under the device. Minimum via diameters and *keep-out areas* around those vias are defined by the PCB manufacturer. These diameters limit the amount of space available in-between vias for routing of signals in and out of the via array underneath the device. These diameters define the maximum trace width in these *breakout* traces. PCB manufacturing limits constrain the minimum trace width and minimum spacing.

The total number of PCB layers necessary to accommodate an FPGA is defined by the number of signal layers and the number of plane layers.

- The number of signal layers is defined by the number of I/O signal traces routed in and out of an FPGA package (usually following the total User I/O count of the package).
- The number of plane layers is defined by the number of power and ground plane layers necessary to bring power to the FPGA and to provide references and isolation for signal layers.

Most PCBs for large FPGAs range from 12 to 22 layers.

System compliance often defines the total thickness of the board. Along with the number of board layers, this defines the maximum layer thickness, and therefore, the spacing in the Z direction of signal and plane layers to other signal and plane layers. Z-direction spacing of signal trace layers to other signal trace layers affects crosstalk. Z-direction spacing of signal trace layers to reference plane layers affects signal trace impedance. Z-direction spacing of plane layers to other plane layers affects power system parasitic inductance.

Z-direction spacing of signal trace layers to reference plane layers (defined by total board thickness and number of board layers) is a defining factor in trace impedance. Trace width (defined by FPGA package ball pitch and PCB via manufacturing constraints) is another factor in trace impedance. A designer often has little control over trace impedance in area of the via array beneath the FPGA. When traces escape the via array, their width can change to the width of the target impedance (usually 50Ω single-ended).

Decoupling capacitor placement and discrete termination resistor placement are other areas of trade-off optimization. DFM constraints often define a keep-out area around the perimeter of the FPGA (device footprint) where no discrete components can be placed. The purpose of the keep-out area is to allow room for assembly and rework where necessary. For this reason, the area just outside the keep-out area is one where components compete for placement. It is up to the PCB designer to determine the high priority components. Decoupling capacitor placement constraints are described in [Chapter 1, Power Distribution System in UltraScale Devices](#). Termination resistor placement constraints must be determined through signal integrity simulation, using IBIS or SPICE.

---

## Basic PDS Principles

The purpose of the PDS and the properties of its components are discussed in this section. The important aspects of capacitor placement, capacitor mounting, PCB geometry, and PCB stackup recommendations are also described.

### Noise Limits

In the same way that devices in a system have a requirement for the amount of current consumed by the power system, there is also a requirement for the cleanliness of the power. This cleanliness requirement specifies a maximum amount of noise present on the power supply, often referred to as ripple voltage ( $V_{\text{RIPPLE}}$ ). Most digital devices, including all UltraScale architecture-based devices, require that  $V_{\text{CC}}$  supplies not fluctuate more than the specifications documented in the device data sheet.

The power consumed by a digital device varies over time and this variance occurs on all frequency scales, creating a need for a wide-band PDS to maintain voltage stability.

- Low-frequency variance of power consumption is usually the result of devices or large portions of devices being enabled or disabled or event occurrences like data traffic or processing. This variance occurs in time frames from milliseconds to days.
- High-frequency variance of power consumption is the result of individual switching events inside a device. This occurs on the scale of the clock frequency and the first few harmonics of the clock frequency up to about 5 GHz.

Because the voltage level of  $V_{CC}$  for a device is fixed, changing power demands are manifested as changing current demand. The PDS must accommodate these variances of current draw with as little change as possible in the power-supply voltage.

When the current draw in a device changes, the PDS cannot respond to that change instantaneously. As a consequence, the voltage at the device changes for a brief period before the PDS responds. Two main causes for this PDS lag correspond to the two major PDS components: the voltage regulator and decoupling capacitors.

The first major component of the PDS is the voltage regulator. The voltage regulator observes its output voltage and adjusts the amount of current it is supplying to keep the output voltage constant. Most common voltage regulators make this adjustment in milliseconds to microseconds. Voltage regulators effectively maintain the output voltage for events at all frequencies from DC to a few hundred kHz, depending on the regulator (some are effective at regulating in the low MHz). For transient events that occur at frequencies above this range, there is a time lag before the voltage regulator responds to the new current demand level.

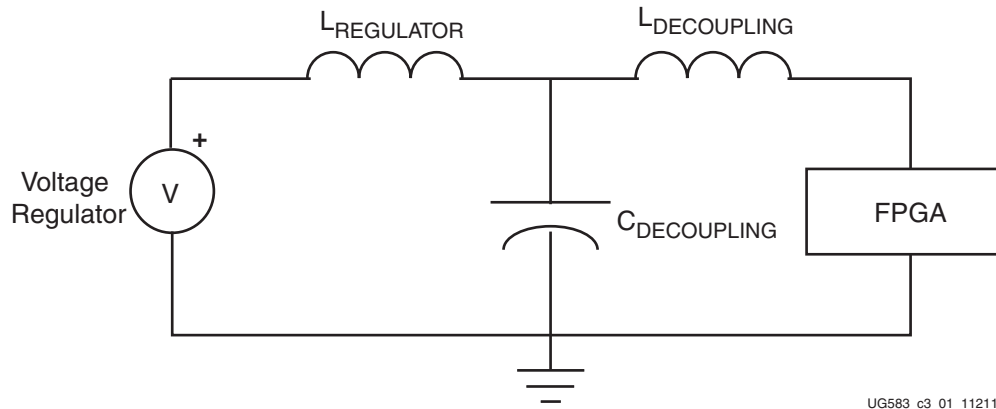
For example, if the device's current demand increases in a few hundred picoseconds, the voltage at the device sags by some amount until the voltage regulator can adjust to the new, higher level of required current. This lag can last from microseconds to milliseconds. A second component is needed to substitute for the regulator during this time, preventing the voltage from sagging.

This second major PDS component is the *decoupling* capacitor (also known as a bypass capacitor). The decoupling capacitor works as the device's local energy storage. The capacitor cannot provide DC power because it stores only a small amount of energy (voltage regulator provides DC power). This local energy storage should respond very quickly to changing current demands. The capacitors effectively maintain power-supply voltage at frequencies from hundreds of kHz to hundreds of MHz (in the milliseconds to nanoseconds range). Discrete decoupling capacitors are not useful for events occurring above or below this range.

For example, if current demand in the device increases in a few picoseconds, the voltage at the device sags by some amount until the capacitors can supply extra charge to the device. If current demand in the device maintains this new level for many milliseconds, the voltage-regulator circuit, operating in parallel with the decoupling capacitors, replaces the capacitors by changing its output to supply this new level of current.



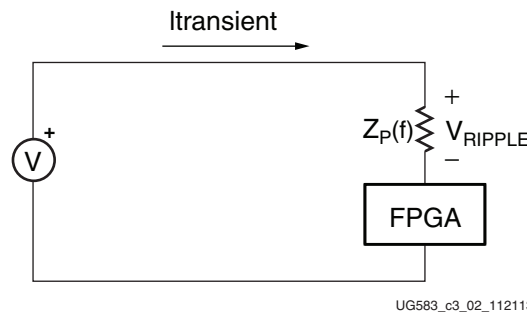
Figure 9-1 shows the major PDS components: the voltage regulator, the decoupling capacitors, and the active device being powered (FPGA).



UG583\_c3\_01\_112113

Figure 9-1: Simplified PDS Circuit

Figure 9-2 shows a simplified PDS circuit with all reactive components represented by a frequency-dependent resistor.



UG583\_c3\_02\_112113

Figure 9-2: Further Simplified PDS Circuit

## Role of Inductance

Inductance is the electrical property of conductors by which a changing magnetic field creates an electromagnetic force or voltage. This field opposes the change of the current in the current path. Inductance is the reason why capacitors cannot respond instantaneously to transient currents or to changes that occur at frequencies higher than their effective range.

Inductance can be thought of as the momentum of charge. Charge moving through a conductor represents some amount of current. If the level of current changes, the charge moves at a different rate. Because momentum (stored magnetic-field energy) is associated with this charge, some amount of time and energy is required to slow down or speed up the charge flow. The greater the inductance, the greater the resistance to change, and the longer the time required for the current level to change. A voltage develops across the inductance as this change occurs.

The PDS, made up of a regulator and multiple stages of decoupling capacitors, accommodates the device current demand and responds to current transients as quickly as necessary to maintain the voltage within the specified limits. When these current demands are not met, the voltage across the device's power supply changes. This is observed as noise. Inductance in the current path of the capacitors should be minimized, because it retards the ability of decoupling capacitors to quickly respond to changing current demands.

Inductances occur between the FPGA device and capacitors and between the capacitors and the voltage regulator (see [Figure 9-2](#)). These inductances occur as parasitics in the capacitors and in all PCB current paths. It is important that each of these parasitics be minimized.

## Capacitor Parasitic Inductance

The capacitance value is often considered to be a capacitor's most important characteristic. In power system applications, the parasitic inductance (ESL) has the same or greater importance. Capacitor package dimensions (body size) determine the amount of parasitic inductance. Physically small capacitors usually have lower parasitic inductance than physically large capacitors.

Requirements for choosing decoupling capacitors:

- For a specific capacitance value, choose the smallest package available.  
- or -
- For a specific package size (essentially a fixed inductance value), choose the highest capacitance value available in that package.

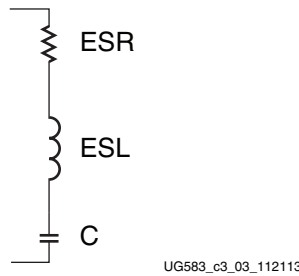
Surface-mount chip capacitors are the smallest capacitors available and are a good choice for discrete decoupling capacitors:

- For values from 100  $\mu\text{F}$  to very small values such as 0.01  $\mu\text{F}$ , ceramic X7R or X5R type capacitors are usually used. These capacitors have a low parasitic inductance and a low ESR, with an acceptable temperature characteristic.
- For larger values, such as 47  $\mu\text{F}$  to 1000  $\mu\text{F}$ , tantalum capacitors are usually used. These capacitors have a low parasitic inductance and a medium ESR, giving them a low Q factor and consequently a very wide range of effective frequencies.

If tantalum capacitors are not available or cannot be used, low-ESR, low-inductance electrolytic capacitors can be used, provided they have comparable ESR and ESL values. Other new technologies with similar characteristics are also available (Os-Con, POSCAP, and Polymer-Electrolytic SMT). Electrolytic or tantalum capacitors are generally part of the voltage regulator design. They are considered as part of the voltage regulator control loop and should be co-designed with the voltage regulator or control firmware setting. The ESR

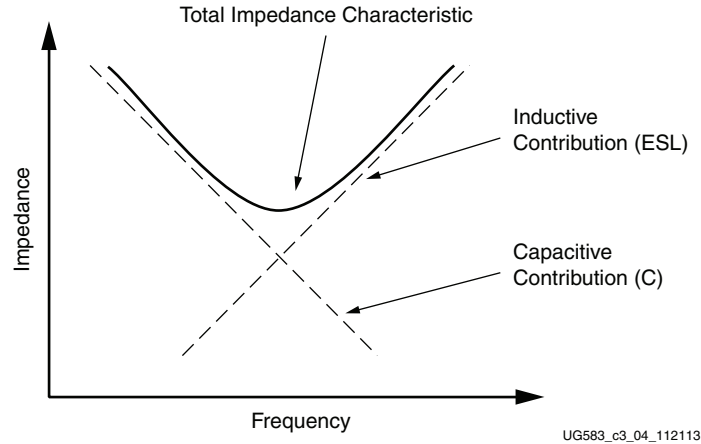
of these capacitors sets the limit of switching regulator ripple and switching noise along with regulator FET and output inductor design.

A *real* capacitor of any type then not only has capacitance characteristics but also inductance and resistance characteristics. [Figure 9-3](#) shows the parasitic model of a real capacitor. A real capacitor should be treated as an *RLC circuit* (a circuit consisting of a resistor (R), an inductor (L), and a capacitor (C), connected in series).



**Figure 9-3: Parasitics of a Real, Non-Ideal Capacitor**

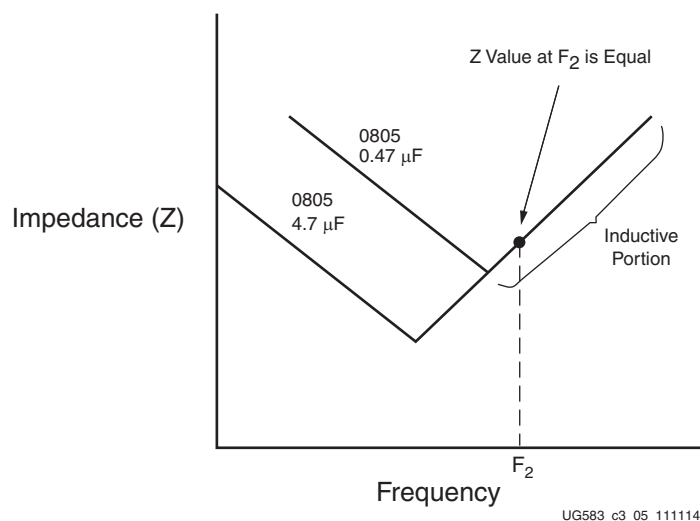
[Figure 9-4](#) shows a *real* capacitor’s impedance characteristic. Overlaid on this plot are curves corresponding to the capacitor’s capacitance and parasitic inductance (ESL). These two curves combine to form the RLC circuit’s total impedance characteristic, softened or sharpened by the capacitor’s ESR.



**Figure 9-4: Contribution of Parasitics to Total Impedance Characteristics**

As capacitive value is increased, the capacitive curve moves down and left. As parasitic inductance is decreased, the inductive curve moves down and right. Because parasitic inductance for capacitors in a specific package is fixed, the inductance curve for capacitors in a specific package remains fixed.

As different capacitor values are selected in the same package, the capacitive curve moves up and down against the fixed inductance curve, as shown in [Figure 9-5](#).



**Figure 9-5: Effective Frequency Example**

The low-frequency capacitor impedance can be reduced by increasing the value of the capacitor; the high-frequency impedance can be reduced by decreasing the inductance of the capacitor. While it might be possible to specify a higher capacitance value in the fixed package, it is not possible to lower the inductance of the capacitor (in the fixed package) without putting more capacitors in parallel. Using multiple capacitors in parallel divides the parasitic inductance, and at the same time, multiplies the capacitance value. This lowers both the high and low frequency impedance at the same time.

## PCB Current Path Inductance

The parasitic inductance of current paths in the PCB have three distinct sources:

- Capacitor mounting
- PCB power and ground planes
- FPGA mounting

### **Capacitor Mounting Inductance**

Capacitor mounting refers to the capacitor's solder lands on the PCB, the trace (if any) between the land and via, and the via.

The vias, traces, and capacitor mounting pads of a 2-terminal capacitor contribute inductance between 300 pH to 4 nH depending on the specific pad routing, via structure and PCB stackup.

Because the current path's inductance is proportional to the loop area the current traverses, it is important to minimize this loop size. The loop consists of the path through one power plane, up through one via, through the connecting trace to the land, through the capacitor, through the other land and connecting trace, down through the other via, and into the other plane, as shown in Figure 9-6.

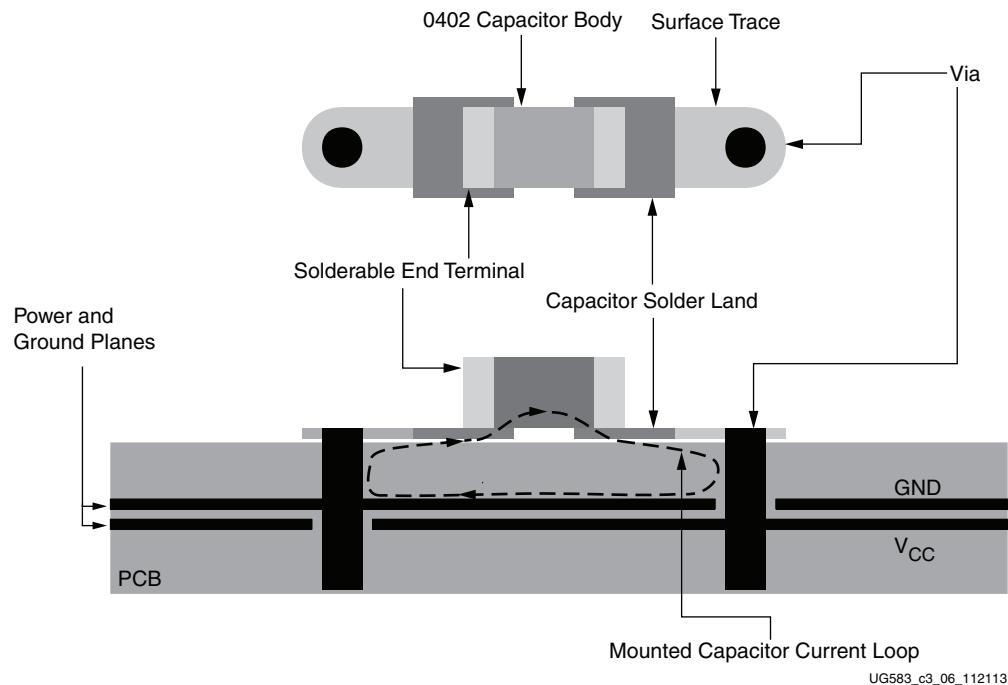


Figure 9-6: Example Cutaway View of PCB with Capacitor Mounting

A connecting trace length has a large impact on the mounting's parasitic inductance and if used, should be as short and wide as possible. When possible, a connecting trace should **not** be used and the via should butt up against the land. Placing vias to the side of the capacitor lands or doubling the number of vias, further reduces the mounting's parasitic inductance.

Some PCB manufacturing processes allow via-in-pad geometries, an option for reducing parasitic inductance. Using multiple vias per land is important with ultra-low inductance capacitors, such as reverse aspect ratio capacitors that place wide terminals on the sides of the capacitor body instead of the ends.

PCB layout engineers often try to squeeze more parts into a small area by sharing vias among multiple capacitors. *This technique should not be used under any circumstances.* PDS improvement is very small when a second capacitor is connected to an existing capacitor's vias. For a larger improvement, optimize the total number of capacitors and improve the mounting via inductance path.

The capacitor mounting (lands, traces, and vias) typically contributes about the same amount or more inductance than the capacitor's own parasitic self-inductance. If the mounting via structure is not optimized, the capacitor might not be effective at all to PDS.

## Plane Inductance

Some inductance is associated with the PCB power and ground planes. The geometry of these planes determines their inductance.

Current spreads out as it flows from one point to another (due to a property similar to *skin effect*) in the power and ground planes. Inductance in planes can be described as *spreading inductance* and is specified in units of henries per square. The square is dimensionless; the shape of a section of a plane, not the size, determines the amount of inductance.

Spreading inductance acts like any other inductance and resists changes to the amount of current in a power plane (the conductor). The inductance retards the capacitor's ability to respond to a device's transient currents and should be reduced as much as possible. Because the designer's control over the X-Y shape of the plane can be limited, the only controllable factor is the spreading inductance value. This is determined by the thickness of the dielectric separating a power plane from its associated ground plane.

For high-frequency power distribution systems, power and ground planes work in pairs, with their inductances coexisting dependently with each other. The spacing between the power and ground planes determines the pair's spreading inductance. The closer the spacing (the thinner the dielectric), the lower the spreading inductance. Approximate values of spreading inductance for different thicknesses of FR4 dielectric are shown in [Table 9-1](#).

**Table 9-1: Capacitance and Spreading Inductance Values for Different Thicknesses of FR4 Power-Ground Plane Sandwiches**

Dielectric Thickness		Inductance (pH/square)	Capacitance	
(micron)	(mil)		(pF/in <sup>2</sup> )	(pF/cm <sup>2</sup> )
102	4	130	225	35
51	2	65	450	70
25	1	32	900	140

Decreased spreading inductance corresponds to closer spacing of  $V_{CC}$  and GND planes. When possible, place the  $V_{CC}$  planes directly adjacent to the GND planes in the PCB stackup. Facing  $V_{CC}$  and GND planes are sometimes referred to as *sandwiches*. While the use of  $V_{CC}$  – GND sandwiches was not necessary in the past for previous technologies (lead frames, wire bond packages), the speeds involved and the sheer amount of power required for fast, dense devices often demand it.

However, because of the presence of substrate decoupling capacitors in UltraScale architecture-based devices, there is a limit to the amount of fast transient current demanded from PCB decoupling capacitors. This means that there is little benefit from dielectric thicknesses below  $50\mu$  (2 mil). Dielectric thickness of  $50\mu$  or  $75\mu$  between  $V_{CC}$  and GND layers is sufficient for UltraScale architecture-based devices.

Besides offering a low-inductance current path, power-ground sandwiches also offer some high-frequency decoupling capacitance. As the plane area increases and as the separation

between power and ground planes decreases, the value of this capacitance increases. Capacitance per square inch is shown in [Table 9-1](#). However, the amount of capacitance arising from these PCB power-ground plane pairs is generally inconsequential, given the substrate decoupling capacitors present in UltraScale architecture-based devices.

### **FPGA Mounting Inductance**

The PCB solder lands and vias that connect the FPGA power pins ( $V_{CC}$  and GND) contribute an amount of parasitic inductance to the overall power circuit. For existing PCB technology, the solder land geometry and the dogbone geometry are mostly fixed, and parasitic inductance of these geometries does not vary. Via parasitic inductance is a function of the via length and the proximity of the opposing current paths to one another.

The relevant via length is the portion of the via that carries transient current between the FPGA solder land and the associated  $V_{CC}$  or GND plane. Any remaining via (between the power plane and the PCB backside) does not affect the parasitic inductance of the via (the shorter the via between the solder lands and the power plane, the smaller the parasitic inductance). Parasitic via inductance in the FPGA mounting is reduced by keeping the relevant  $V_{CC}$  and GND planes as close to the FPGA as possible (close to the top of the PCB stackup).

Device pinout arrangement determines the proximity of opposing current paths to one another. Inductance is associated with any two opposing currents (for example, current flowing in a  $V_{CC}$  and GND via pair). A high degree of mutual inductive coupling between the two opposing paths reduces the loop's total inductance. Therefore, when given a choice,  $V_{CC}$  and GND vias should be as close together as possible.

The via field under an FPGA has many  $V_{CC}$  and GND vias, and the total inductance is a function of the proximity of one via to another:

- For core  $V_{CC}$  supplies ( $V_{CCINT}$  and  $V_{CCAUX}$ ), opposing current is between the  $V_{CC}$  and GND pins.
- For I/O  $V_{CC}$  supplies ( $V_{CCO}$ ), opposing current is between any I/O and its return current path, whether carried by a  $V_{CCO}$  or GND pin.

To reduce parasitic inductance:

- Core  $V_{CC}$  pins such as  $V_{CCINT}$  and  $V_{CCAUX}$  are placed in a checkerboard arrangement in the pinout.
- $V_{CCO}$  and GND pins are distributed among the I/O pins.

Every I/O pin in the Kintex UltraScale and Virtex UltraScale FPGA pinouts is adjacent to a return-current pin.

FPGA pinout arrangement determines the PCB via arrangement. The PCB designer cannot control the proximity of opposing current paths but has control over the trade-offs between the capacitor's mounting inductance and FPGA's mounting inductance:

- Both mounting inductances are reduced by placing power planes close to the PCB stackup's top half and placing the capacitors on the top surface (reducing the capacitor's via length).
- If power planes are placed in the PCB stackup's bottom half, the capacitors are recommended to be mounted on the PCB backside. In this case, FPGA mounting vias are already long, and making the capacitor vias long (by coming down from the top surface) is a bad practice. A better practice is to take advantage of the short distance between the underside of the PCB and the power plane of interest, mounting capacitors on the underside.

### ***PCB Stackup and Layer Order***

$V_{CC}$  and ground plane placement in the PCB stackup (the layer order) has a significant impact on the parasitic inductances of power current paths. Layer order must be considered early in the design process:

- High-priority supplies should be placed closer to the FPGA (in the PCB stackup's top half)
- Low-priority supplies should be placed farther from the FPGA (in the PCB stackup's bottom half)

Power supplies with high transient current should have the associated  $V_{CC}$  planes close to the top surface (FPGA side) of the PCB stackup. This decreases the vertical distance ( $V_{CC}$  and GND via length) that currents travel before reaching the associated  $V_{CC}$  and GND planes. To reduce spreading inductance, every  $V_{CC}$  plane should have an adjacent GND plane in the PCB stackup. The skin effect causes high-frequency currents to couple tightly, and the GND plane adjacent to a specific  $V_{CC}$  plane tends to carry the majority of the current complementary to that in the  $V_{CC}$  plane. Thus, adjacent  $V_{CC}$  and GND planes are treated as a pair.

Not all  $V_{CC}$  and GND plane pairs reside in the PCB stackup's top half because manufacturing constraints typically require a symmetrical PCB stackup around the center (with respect to dielectric thicknesses and etched copper areas). The PCB designer chooses the priority of the  $V_{CC}$  and GND plane pairs: high priority pairs carry high transient currents and are placed high in the stackup, while low priority pairs carry lower transient currents (or can tolerate more noise) and are placed in the lower part of the stackup.



## Capacitor Effective Frequency

Every capacitor has a narrow frequency band where it is most effective as a decoupling capacitor. This band is centered at the capacitor's self-resonant frequency  $F_{RSELF}$ . The effective frequency bands of some capacitors are wider than others. A capacitor's ESR determines the capacitor's quality (Q) factor, and the Q factor can determine the width of the effective frequency band:

- Tantalum capacitors generally have a very *wide* effective band.
- Ceramic chip capacitors with a lower ESR, generally have a very *narrow* effective frequency band.

An ideal capacitor only has a capacitive characteristic, whereas *real* non-ideal capacitors also have a parasitic inductance (ESL) and a parasitic resistance (ESR). These parasitics work in series to form an RLC circuit (Figure 9-4). The RLC circuit's resonant frequency is the capacitor's self-resonant frequency.

To determine the RLC circuit's resonant frequency, use Equation 9-1:

$$F = \frac{1}{2\pi\sqrt{LC}} \quad \text{Equation 9-1}$$

Another method of determining the self-resonant frequency is to find the minimum point in the impedance curve of the equivalent RLC circuit. The impedance curve can be computed or generated in SPICE using a frequency sweep. See the [Simulation Methods, page 269](#) section for other ways to compute an impedance curve.

It is important to distinguish between the capacitor's self-resonant frequency and the mounted capacitor's effective resonant frequency when the capacitor is part of the system,  $F_{RIS}$ . This corresponds to the resonant frequency of the capacitor with its parasitic inductance, plus the inductance of the vias, planes, and connecting traces between the capacitor and the FPGA.

The capacitor's self-resonant frequency,  $F_{RSELF}$ , (capacitor data sheet value) is much higher than its effective mounted resonant frequency in the system,  $F_{RIS}$ . Because the mounted capacitor's performance is most important, the mounted resonant frequency is used when evaluating a capacitor as part of the greater PDS.

Mounted parasitic inductance is a combination of the capacitor's own parasitic inductance and the inductance of: PCB lands, connecting traces, vias, and power planes. Vias traverse a full PCB stackup to the device when capacitors are mounted on the PCB backside. For a board with a finished thickness of 1.524 mm (60 mils), these vias contribute approximately 300 pH to 1,500 pH, (the capacitor's mounting parasitic inductance,  $L_{MOUNT}$ ) depending on the spacing between vias. Wider-spaced vias and vias in thicker boards have higher inductance.

To determine the capacitor's total parasitic inductance in the system,  $L_{IS}$ , the capacitor's parasitic inductance,  $L_{SELF}$ , is added to the mounting's parasitic inductance,  $L_{MOUNT}$ :

$$L_{IS} = L_{SELF} + L_{MOUNT} \quad \text{Equation 9-2}$$

For example, using X7R Ceramic Chip capacitor in 0402 body size:

$$C = 0.01 \mu\text{F} \text{ (selected by user)}$$

$$L_{SELF} = 0.9 \text{ nH} \text{ (capacitor data sheet parameter)}$$

$$F_{RSELF} = 53 \text{ MHz} \text{ (capacitor data sheet parameter)}$$

$$L_{MOUNT} = 0.8 \text{ nH} \text{ (based on PCB mounting geometry)}$$

To determine the effective in-system parasitic inductance ( $L_{IS}$ ), add the via parasitics:

$$\begin{aligned} L_{IS} &= L_{SELF} + L_{MOUNT} = 0.9 \text{ nH} + 0.8 \text{ nH} \\ L_{IS} &= 1.7 \text{ nH} \end{aligned} \quad \text{Equation 9-3}$$

The values from the example are used to determine the mounted capacitor resonant frequency ( $F_{RIS}$ ). Using [Equation 9-4](#):

$$F_{RIS} = \frac{1}{2\pi\sqrt{L_{IS}C}} \quad \text{Equation 9-4}$$

$$F_{RIS} = \frac{1}{2\pi\sqrt{(1.7 \times 10^{-9} \text{ H}) \cdot (0.01 \times 10^{-6} \text{ F})}} = 38 \times 10^6 \text{ Hz} \quad \text{Equation 9-5}$$

$F_{RSELF}$  is 53 MHz, but  $F_{RIS}$  is lower at 38 MHz. The addition of mounting inductances shifts the effective-frequency band down.

A decoupling capacitor is most effective at the narrow-frequency band around its resonant frequency, and thus, the resonant frequency must be reviewed when choosing a capacitor collection to build up a decoupling network. This being said, capacitors can be effective at frequencies considerably higher and lower than their resonant frequency. Recall that capacitors of differing values in the same package share the same inductance curve. As shown in [Figure 9-5](#), for any given frequency along the inductive portion of the curve, the capacitors are equally effective.

## Capacitor Anti-Resonance

One problem associated with combinations of capacitors in a PDS of an FPGA is anti-resonant spikes in the PDS aggregate impedance. The cause for these spikes is a bad combination of energy storage elements in the PDS (intrinsic capacitances, discrete capacitors, parasitic inductances, and power and ground planes).

Anti-resonance can arise between any two consecutive stages of a power distribution system, such as between the high-frequency PCB capacitors and the PCB plane capacitance. The inter-plane capacitance of the power and ground planes generally has a high-Q factor. If the high-frequency PCB capacitors also are high-Q, the crossover point between the high-frequency discrete capacitors and the plane capacitance might exhibit a high-impedance anti-resonance peak. If the FPGA has a high transient current demand at this frequency (as a stimulus), a large noise voltage can occur.

To correct this type of problem, the characteristics of the high-frequency discrete capacitors or the characteristics of the  $V_{CC}$  and ground planes must be changed, or FPGA activity shifted to a different frequency away from the resonance.

## Capacitor Placement Background

To perform the decoupling function, capacitors should be close to the device being decoupled.

Increased spacing between the FPGA and decoupling capacitor increases the current flow distance in the power and ground planes, and it often increases the current path's inductance between the device and the capacitor.

The inductance of this current path (the loop followed by current as it travels from the  $V_{CC}$  side of the capacitor to the  $V_{CC}$  pin[s] of the FPGA, and from the GND pin[s] of the FPGA to the GND side of the capacitor[s]), is proportional to the loop area. Inductance is decreased by decreasing the loop area.

Shortening the distance between the device and the decoupling capacitor reduces the inductance, resulting in a less impeded transient current flow. Because of typical PCB dimensions, this lateral plane travel tends to be less important than the phase relationship between the FPGA noise source and the mounted capacitor.

The phase relationship between the FPGA's noise source and the mounted capacitor determines the capacitor's effectiveness. For a capacitor to be effective in providing transient current at a certain frequency (for example, the capacitor's resonant frequency), the phase relationship, based on the distance travelled by the current from the FPGA to the capacitor, must be within a fraction of the corresponding period.

The capacitor's placement determines the length of the transmission line interconnect (in this case, the power and ground plane pair) between the capacitor and FPGA. The propagation delay of this interconnect is the key factor.

FPGA noise falls into certain frequency bands, and different sizes of decoupling capacitors take care of different frequency bands. Thus, capacitor placement requirements are determined by each capacitor's effective frequency.

When the FPGA initiates a current demand change, it causes a small local disturbance in the PDS voltage (a point in the power and ground planes). Before it can counteract this, the decoupling capacitor must first sense a voltage difference.

A finite time delay (Equation 9-6) occurs between the start of the disturbance at the FPGA power pins and the point when the capacitor senses the disturbance.

$$\text{Time Delay} = \frac{\text{Distance from the FPGA power pins to the capacitor}}{\text{Signal propagation speed through FR4 dielectric}} \quad \text{Equation 9-6}$$

The dielectric is the substrate of the PCB where the power planes are embedded.

Another delay of the same duration occurs when the compensation current from the capacitor flows to the FPGA. For any transient current demand in the FPGA, a round-trip delay occurs before any relief is seen at the FPGA.

- Negligible energy is transferred to the FPGA with placement distances greater than one quarter of a demand frequency's wavelength.
- Energy transferred to the FPGA increases from 0% at one-quarter of a wavelength to 100% at zero distance.
- Energy is transferred efficiently from the capacitor to the FPGA when capacitor placement is at a fraction of a quarter wavelength of the FPGA power pins. This fraction should be small because the capacitor is also effective at some frequencies (shorter wavelengths) above its resonant frequency.

One-tenth of a quarter wavelength is a good target for most practical applications and leads to placing a capacitor within one-fortieth of a wavelength of the power pins it is decoupling. The wavelength corresponds to the capacitor's mounted resonant frequency,  $f_{RIS}$ .

When using large numbers of external termination resistors or passive power filtering for transceivers, priority should be given to these over the decoupling capacitors. Moving away from the device in concentric rings, the termination resistors and transceiver supply filtering should be closest to the device, followed by the smallest-value decoupling capacitors, then the larger-value decoupling capacitors.

## V<sub>REF</sub> Stabilization Capacitors

In V<sub>REF</sub> supply stabilization, one capacitor per pin is placed as close as possible to the V<sub>REF</sub> pin. The capacitors used are in the 0.022 μF – 0.47 μF range. The V<sub>REF</sub> capacitor's primary function is to reduce the V<sub>REF</sub> node impedance, which in turn reduces crosstalk coupling. Since no low-frequency energy is needed, larger capacitors are not necessary.

This only applies when Internal V<sub>REF</sub> is not used. Internal V<sub>REF</sub> is an UltraScale architecture feature wherein the reference voltage rail is generated internally. See *UltraScale Architecture SelectIO User Guide* (UG571) [Ref 10] for more details on Internal V<sub>REF</sub>.

## Power Supply Consolidation

Powering 1.8V V<sub>CCO</sub> and V<sub>CCAUX</sub> from a common PCB plane is allowed in Kintex UltraScale and Virtex UltraScale FPGA designs. However, careful consideration must be given to power

supply noise—in particular, any noise on the  $V_{CCO}$  rail should not violate the recommended operating condition range for the  $V_{CCAUX}$  supply. See the data sheet for these requirements.

## Unconnected $V_{CCO}$ Pins

In some cases, one or more I/O banks in an FPGA are not used (for example, when an FPGA has far more I/O pins than the design requires). In these cases, it might be desirable to leave the bank's associated  $V_{CCO}$  pins unconnected, as it can free up some PCB layout constraints (less voiding of power and ground planes from via antipads, less obstacles to signals entering and exiting the pinout array, more copper area available for other planelets in the otherwise used plane layer).

Leaving the  $V_{CCO}$  pins of unused I/O banks floating reduces the level of ESD protection on these pins and the I/O pins in the bank. For maximum ESD protection in an unused bank, all  $V_{CCO}$  pins in that bank should be connected together to the same potential, whether that be ground, a valid  $V_{CCO}$  voltage, or a floating plane. I/O pins are also recommended to be connected to the same potential as  $V_{CCO}$ , or they can be left floating.

## Transmission Lines

The combination of a signal trace and a reference plane forms a transmission line. All I/O signals in a PCB system travel through transmission lines.

For single-ended I/O interfaces, both the signal trace and the reference plane are necessary to transmit a signal from one place to another on the PCB. For differential I/O interfaces, the transmission line is formed by the combination of two traces and a reference plane. While the presence of a reference plane is not strictly necessary in the case of differential signals, it is necessary for practical implementation of differential traces in PCBs.

Good signal integrity in a PCB system is dependent on having transmission lines with controlled impedance. Impedance is determined by the geometry of the traces and the dielectric constant of the material in the space around the signal trace and between the signal trace and the reference plane.

The dielectric constant of the material in the vicinity of the trace and reference plane is a property of the PCB laminate materials, and in the case of surface traces, a property of the air or fluid surrounding the board. PCB laminate is typically a variant of FR4, though it can also be an exotic material.

While the dielectric constant of the laminate varies from board to board, it is fairly constant within one board. Therefore, the relative impedance of transmission lines in a PCB is defined most strongly by the trace geometries and tolerances. Impedance variance can occur based on the presence or absence of glass in a local portion of the laminate weave, but this rarely poses issues except in high-speed (>6 Gb/s) interfaces.

## Return Currents

An often neglected aspect of transmission lines and their signal integrity is return current. It is incorrect to assume that a signal trace by itself forms a transmission line. Currents flowing in a signal trace have an equal and opposite complimentary current flowing in the reference plane beneath them. The relationship of the trace voltage and trace current to reference plane voltage and reference plane current defines the characteristic impedance of the transmission line formed by the trace and reference plane. While interruption of reference plane continuity beneath a trace is not as dramatic in effect as severing the signal trace, the performance of the transmission line and any devices sharing the reference plane is affected.

It is important to pay attention to reference plane continuity and return current paths. Interruptions of reference plane continuity, such as holes, slots, or isolation splits, cause significant impedance discontinuities in the signal traces. They can also be a significant source of crosstalk and contributor to Power Distribution System (PDS) noise. The importance of return current paths cannot be underestimated.

---

## PCB Materials and Traces

The choice of transmission media, whether PCB materials or cable type, can have a large impact on system performance. Although any transmission medium is lossy at gigahertz frequencies, this chapter provides some guidelines on managing signal attenuation so as to obtain optimal performance for a given application.

### How Fast is Fast?

Signal edges contain frequency components called harmonics. Each harmonic is a multiple of the signal frequency and has significant amplitude up to a frequency determined by [Equation 9-7](#):

$$f \approx \frac{0.35}{T} \quad \text{Equation 9-7}$$

Where:

$f$  = Frequency in GHz

$T$  = The smaller of signal rise ( $T_r$ ) or fall ( $T_f$ ) time in ns

Because dielectric losses in a PCB are frequency dependent, a bandwidth of concern must be determined to find the total loss of the PCB. Frequencies must start at the operation frequency and extend to the frequency in [Equation 9-7](#). For example, a 10 Gb/s signal with a 10 ps rise time has a bandwidth from 10 GHz to 35 GHz.

## Dielectric Losses

The amount of signal energy lost into the dielectric is a function of the material's characteristics. Some parameters used to describe the material include relative permittivity  $\epsilon_r$  (also known as the dielectric constant) and loss tangent. Skin effect is also a contributor to energy loss at line speeds in the gigahertz range.

### *Relative Permittivity*

Relative permittivity is a measure of the effect of the dielectric on the capacitance of a conductor. The higher the relative permittivity, the slower a signal travels on a trace and the lower the impedance of a given trace geometry. A lower  $\epsilon_r$  is almost always preferred.

Although the relative permittivity varies with frequency in all materials, FR4 exhibits wide variations in  $\epsilon_r$  with frequency. Because  $\epsilon_r$  affects impedance directly, FR4 traces can have a spread of impedance values with increasing frequency. While this spread can be insignificant at 1.125 Gb/s, it can be a concern at 10 Gb/s operation.

### *Loss Tangent*

Loss tangent is a measure of how much electromagnetic energy is lost to the dielectric as it propagates down a transmission line. A lower loss tangent allows more energy to reach its destination with less signal attenuation.

As frequency increases, the magnitude of energy loss increases as well, causing the highest frequency harmonics in the signal edge to suffer the most attenuation. This appears as a degradation in the rise and fall times.

### *Skin Effect and Resistive Losses*

The skin effect is the tendency for current to flow preferentially near the outer surface of a conductor. This is mainly due to the magnetic fields in higher frequency signals pushing current flow in the perpendicular direction towards the perimeter of the conductor.

As current density near the surface increases, the effective cross-sectional area through which current flows decreases. Resistance increases because the effective cross-sectional area of the conductor is now smaller. Because this skin effect is more pronounced as frequency increases, resistive losses increase with signaling rates.

Resistive losses have a similar effect on the signal as loss tangent. Rise and fall times increase due to the decreased amplitude of the higher harmonics, with the highest frequency harmonics being most affected. In the case of 10 Gb/s signals, even the fundamental frequency can be attenuated to some degree when using FR4.

For example, an 8 mil wide trace at 1 MHz has a resistance on the order of 0.06 $\Omega$ /inch, while the same trace at 10 Gb/s has a resistance of just over 1 $\Omega$ /inch. Given a 10 inch trace and

1.6V voltage swing, a voltage drop of 160 mV occurs from resistive losses of the fundamental frequency, not including the losses in the harmonics and dielectric loss.

### ***Choosing the Substrate Material***

The goal in material selection is to optimize both performance and cost for a particular application.

FR4, the most common PCB substrate material, provides good performance with careful system design. For long trace lengths or high signaling rates, a more expensive substrate material with lower dielectric loss must be used.

Substrates, such as Nelco, have lower dielectric loss and exhibit significantly less attenuation in the gigahertz range, thus increasing the maximum bandwidth of PCBs. At 3.125 Gb/s, the advantages of Nelco over FR4 are added voltage swing margin and longer trace lengths. At 10 Gb/s, a low-loss dielectric like Nelco is necessary unless high-speed traces are kept very short.

The choice of substrate material depends on the total length of the high-speed trace and also the signaling rate.

What-if analysis can be done in HSPICE simulation to evaluate various substrate materials. By varying the dielectric constant, loss tangent, and other parameters of the PCB substrate material. The impact on eye quality can be simulated to justify the use of higher cost materials. The impact of other parameters such as copper thickness can also be explored.

## **Traces**

### ***Trace Geometry***

For any trace, its characteristic impedance is dependent on its stackup geometry as well as the trace geometry. In the case of differential traces, the inductive and capacitive coupling between the tightly coupled pair also determines the characteristic impedance of the traces.

The impedance of a trace is determined by its inductive and capacitive coupling to nearby conductors. For example, these conductors can be planes, vias, pads, connectors, and other traces, including the other closely coupled trace in a differential pair. The substrate properties, conductor properties, flux linkage area, and distance to a nearby conductor determine the amount of coupling and hence, the contribution to the final impedance.

2D field solvers are necessary in resolving these complex interactions and contribute to the calculation of the final impedance of the trace. They are also a useful tool to verify existing trace geometries.

Wider traces create a larger cross-sectional area for current to flow and reduce resistive losses in high-speed interfaces. Use the widest traces that space constraints allow. Because



trace width tolerances are expressed in absolute terms, a wider trace also minimizes the percentage variation of the manufactured trace, resulting in tighter impedance control along the length of the transmission line.

Sometimes, striplines are preferred over microstrips because the reference planes on both sides of the trace provide radiation shielding. Microstrips are shielded on only one side (by the reference plane) because they run on the top-most or bottom-most layers, leaving the other side exposed to the environment.

For best results, the use of a 2D or 3D field solver is recommended for verification.

### Trace Characteristic Impedance Design for High-Speed Transceivers

Because the transceivers use differential signaling, the most useful trace configurations are differential edge-coupled stripline and differential microstrip. While some backplanes use the differential broadside-coupled stripline configuration, it is not recommended for 10 Gb/s operation, because the P and N vias are asymmetrical and introduce common-mode non-idealities.

With few exceptions, 50Ω characteristic impedance ( $Z_0$ ) is used for transmission lines in the channel. In general, when the width/spacing (W/S) ratio is greater than 0.4 (8 mil wide traces with 20 mil separation), coupling between the P and N signals affects the trace impedance. In this case, the differential traces must be designed to have an odd mode impedance ( $Z_{0O}$ ) of 50Ω, resulting in a differential impedance ( $Z_{DIFF}$ ) of 100Ω, because  $Z_{DIFF} = 2 \times Z_{0O}$ .

The same W/S ratio also must be less than 0.8, otherwise strong coupling between the traces requires narrower, lossier traces for a  $Z_{0O}$  of 50Ω. To clarify, with  $Z_{0O}$  at 50Ω, an even mode impedance ( $Z_{0E}$ ) of 60Ω or below is desired.

Figure 9-7 through Figure 9-10 show example cross sections of differential structures.

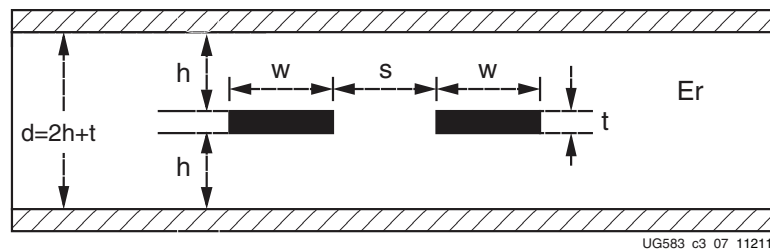


Figure 9-7: Differential Edge-Coupled Centered Stripline

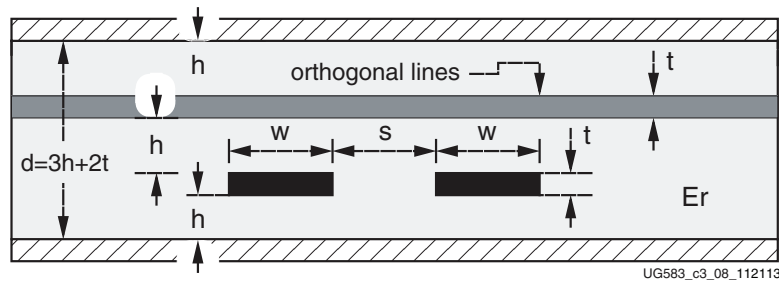


Figure 9-8: Differential Edge-Coupled Offset Stripline

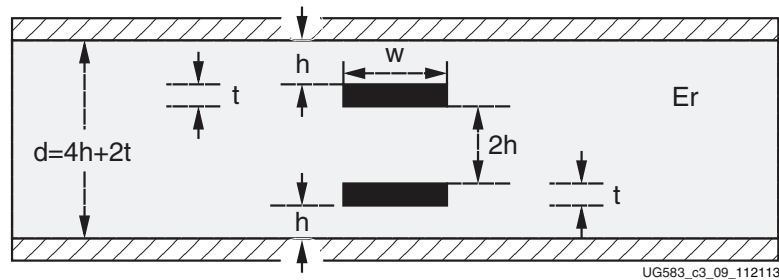


Figure 9-9: Centered Broadside-Coupled Stripline

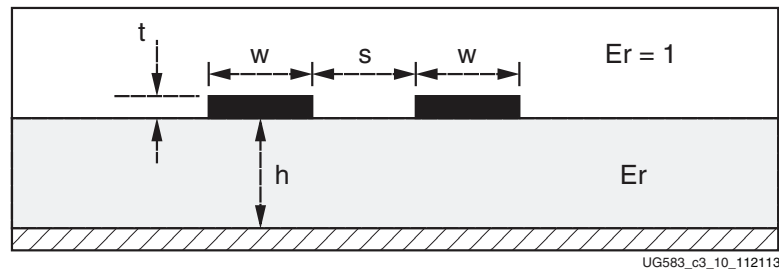


Figure 9-10: Differential Microstrip

A good PCB manufacturer understands controlled impedance and allows fine adjustments for line widths to produce a  $Z_{00}$  of  $50\Omega$ . The PCB manufacturer also provides the parameters necessary for the specific PCB layout. Some parameters can be calculated or simulated from the guideline outlined in the example. Although  $\pm 10\%$  tolerance on  $Z_{00}$  is typical and can provide adequate performance, the additional cost of a tighter tolerance results in better channel performance.

### Trace Routing

High-speed serial differential traces are routed with the highest priority to ensure that the optimal path is available to these critical traces. This reduces the need for bends and vias and minimizes the potential for impedance transitions. Traces must be kept straight, short, and with as few layer changes as possible. The impact of vias is discussed in [Differential Vias, page 286](#).

Routing of high-speed traces must be avoided near other traces or other potential sources of noise. Traces on neighboring signal planes should run perpendicular to minimize crosstalk.

Striplines are to be used whenever possible, as are the uppermost and lowermost stripline layers to minimize via stubs. When the stackup is being planned, these layers must be placed as close to the top and bottom layers whenever possible.

Design constraints might require microstrips for the BGA exit path or from via to connector launch or SMT pads. In such cases, the microstrip trace must be kept as short as possible.

Mitered 45-degree bends are recommended (as opposed to 90-degree bends). At a 90-degree bend, the effective width of the trace changes, causing an impedance discontinuity due to the capacitive coupling of the additional conductor area to the reference plane.

The two traces of a differential pair must be length-matched to eliminate skew. Skew creates mismatches in the common mode and reduces the differential voltage swing as a result.

### ***Plane Splits***

Ground planes should be used as reference planes for signals, as opposed to noisier power planes. Each reference plane should be contiguous for the length of the trace, because routing over plane splits creates an impedance discontinuity. In this case, the impedance of the trace changes because its coupling to the reference plane is changed abruptly at the plane split.

### ***Return Currents***

Routing over plane splits also creates issues with the return current. High-speed signals travel near the surface of the trace due to the skin effect mentioned in [Dielectric Losses, page 263](#). Meanwhile, the return current also travels near the surface of the tightly coupled reference plane.

Because of the tight coupling, the return current has the tendency to travel close to the original signal-carrying trace. At the plane split, the return current can no longer follow the same path parallel to the trace, but must instead find an alternative route.

A plane split causes a suboptimal current return path and increases the current loop area, thereby increasing the inductance of the trace at the plane split, changing the impedance of the trace.

### ***Simulating Lossy Transmission Lines***

Due to the different modeling implementations used by various circuit simulators (frequency-domain versus time-domain techniques), it is important to check that the

models accurately reflect actual losses. One method is to compare the models against known published configurations.

## Cable

Cables are controlled-impedance transmission lines due to the constant physical dimensions of conductor and dielectric along the length of the cable. The highest quality cable shows little variation in these dimensions and also has a wide bandwidth with low loss at high frequencies.

### *Connectors*

The connectors attached to cables should exhibit low parasitic inductance, low-parasitic capacitance, and low crosstalk for high bandwidth operation.

### *Skew Between Conductors*

When selecting a cable, look for a specification of the skew between the conductors in a cable. If the conductors are not length matched, the skew appears in the common mode and directly reduces the eye height.

## Simulation Methods

Simulation methods, ranging from very simple to very complex, exist to predict the PDS characteristics. An accurate simulation result is difficult to achieve without using a fairly sophisticated simulator and taking a significant amount of time.

Basic lumped RLC simulation is one of the simplest simulation methods. Though it does not account for the distributed behavior of a PDS, it is a useful tool for selecting and verifying that combinations of decoupling capacitor values will not lead to large anti-resonances. Lumped RLC simulation is a good method for establishing equivalence of decoupling networks, such as evaluating an alternative to the capacitors of [Table 1-12](#).

Lumped RLC simulation is performed either in a version of SPICE or other circuit simulator, or by using a mathematical tool like MathCAD or Microsoft Excel. Istvan Novak publishes a free Excel spreadsheet for lumped RLC simulation (among other useful tools for PDS simulation) on his website under **Tool Download**:

<http://www.electrical-integrity.com>

[Table 9-2](#) also lists a few EDA tool vendors for PDS design and simulation. These tools span a wide range of sophistication levels.

**Table 9-2: EDA Tools for PDS Design and Simulation**

Tool	Vendor	Website URL
ADS	Keysight Technologies	<a href="http://www.keysight.com">http://www.keysight.com</a>
SIwave, HFSS	Ansys	<a href="http://www.ansys.com">http://www.ansys.com</a>
Sigrity	Cadence	<a href="http://www.cadence.com">http://www.cadence.com</a>
Hyperlynx PI	Mentor	<a href="http://www.mentor.com">http://www.mentor.com</a>

## PDS Measurements

Measurements can be used to determine whether a PDS is adequate. PDS noise measurements are a unique task, and many specialized techniques have been developed. This section describes the noise magnitude and noise spectrum measurements.

### Noise Magnitude Measurement

Noise measurement must be performed with a high-bandwidth oscilloscope (minimum 3 GHz oscilloscope and 1.5 GHz probe or direct coaxial connection) on a design running realistic test patterns. The measurement is taken at the device's power pins (referred to as a *spyhole measurement*) or at an unused I/O driven High or Low.

$V_{CCINT}$  and  $V_{CCAUX}$  can only be measured at the PCB backside vias.  $V_{CCO}$  can also be measured this way, but more accurate results are obtained by measuring static (fixed logic level) signals at unused I/Os in the bank of interest.

When making the noise measurement on the PCB backside, the via parasitics in the path between the measuring point and FPGA must be considered. Any voltage drop occurring in this path is not accounted for in the oscilloscope measurement.

PCB backside via measurements also have a potential problem: decoupling capacitors are often mounted directly underneath the device, meaning the capacitor lands connect directly to the  $V_{CC}$  and GND vias with surface traces. These capacitors confuse the measurement by acting like a short circuit for the high-frequency AC current. To make sure the measurements are not shorted by the capacitors, remove the capacitor at the measurement site (keep all others to reflect the real system behavior).

When measuring  $V_{CCO}$  noise, the measurement can be taken at an I/O pin configured as a driver to logic 1 or logic 0. In most cases, the same I/O standard should be used for this measurement as for the other signals in the bank. Measuring a static logic 0 shows the crosstalk (via field, PCB routing, package routing) induced on the victim. Measuring a static logic 1 shows all the same crosstalk components as well as the noise present on the  $V_{CCO}$  net for the I/O bank. By subtracting (coherently in time) the noise measured on static logic 0 from the noise measured on static logic 1, the noise on  $V_{CCO}$  at the die can be viewed. For an accurate result, the static logic 0 and static logic 1 noise must be measured at the same I/O location. This means storing the time-domain waveform information from both logic states and performing the subtraction operation on the two waveforms in a post-process math computation tool such as MATLAB or Excel.

### Oscilloscope Measurement Methods

There are two basic ways of using the oscilloscope to view power system noise, each for a different purpose. The first surveys all possible noise events, while the second is useful for focusing on individual noise sources.

- Place the oscilloscope in infinite persistence mode to acquire all noise over a long time period (many seconds or minutes). If the design operates in many different modes, using different resources in different amounts, these various conditions and modes should be in operation while the oscilloscope is acquiring the noise measurement.
- Place the oscilloscope in averaging mode and trigger on a known aggressor event. This can show the amount of noise correlated with the aggressor event (any events asynchronous to the aggressor are removed through averaging).

Power system noise measurements should be made at a few different FPGA locations to ensure that any local noise phenomena are captured.

Figure 9-11 shows an averaged noise measurement taken at the  $V_{CC0}$  pins of a sample design. In this case, the trigger was the clock for an I/O bus interface sending a 1-0-1-0 pattern at 250 Mb/s.

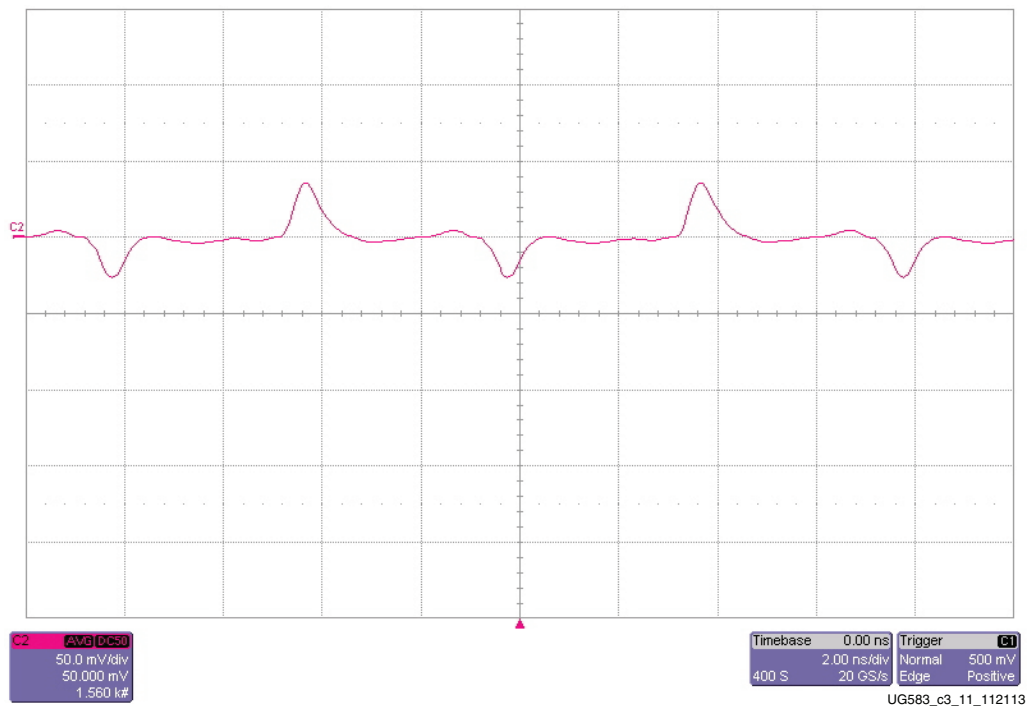


Figure 9-11: Averaged Measurement of  $V_{CC0}$  Supply with Multiple I/O Sending Patterns at 250 Mb/s

Figure 9-12 shows an infinite persistence noise measurement of the same design with a wider variety of I/O activity. Because the infinite persistence measurement catches **all** noise events over a long period, both correlated and non-correlated with the primary aggressor, all power system excursions are shown.

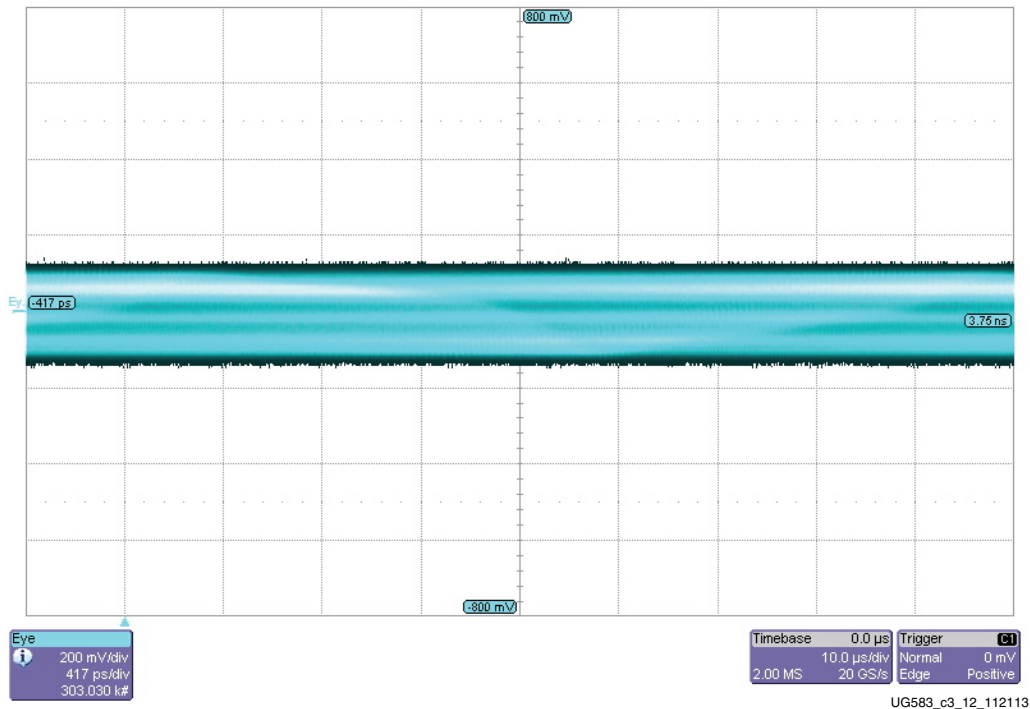


Figure 9-12: Infinite Persistence Measurement of Same Supply

The measurement shown in Figure 9-11 and Figure 9-12 represents the peak-to-peak noise. If the peak-to-peak noise is outside the specified acceptable voltage range (data sheet value,  $V_{CC} \pm 5\%$ ), the decoupling network is inadequate or a problem exists in the PCB layout.

## Noise Spectrum Measurements

Having the necessary information to improve the decoupling network requires additional measurements. To determine the frequencies where the noise resides, noise power spectrum measurement is necessary. A spectrum analyzer or a high-bandwidth oscilloscope coupled with FFT math functionality can accomplish this.

The FFT math function can be built into the oscilloscope, however, many of these functions do not have resolution sufficient to give a clear picture of the noise spectrum. Alternatively, a long sequence of time-domain data can be captured from an oscilloscope and converted to frequency domain using MATLAB or other post-processing software supporting FFT. This method has the advantage of showing as much resolution as you are willing to process. If neither math capacity is available, the noise frequency content can be approximated by



visually examining the time-domain waveform and estimating the individual periodicities present in the noise.

A spectrum analyzer is a frequency-domain instrument, showing the frequency content of a voltage signal at its inputs. Using a spectrum analyzer, you see the exact frequencies where the PDS is inadequate.

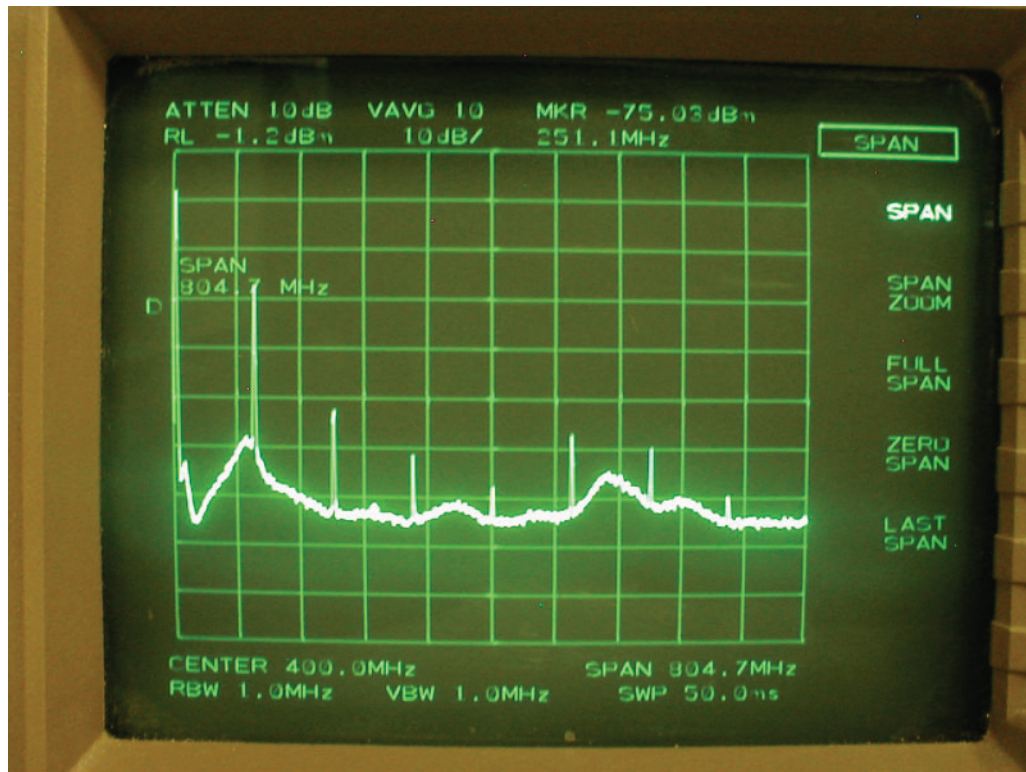
Excessive noise at a certain frequency indicates a frequency where the PDS impedance is too high for the device's transient current demands. Using this information, the designer can modify the PDS to accommodate the transient current at the specific frequency. This is accomplished by either adding capacitors with effective frequencies close to the noise frequency or otherwise lowering the PDS impedance at the critical frequency.

The noise spectrum measurement should be taken in the same manner as the peak-to-peak noise measurement, directly underneath the device, or at a static I/O driven High or Low. A spectrum analyzer takes its measurements using a 50Ω cable instead of an active probe.

- A good method attaches the measurement cable through a coaxial connector tapped into the power and ground planes close to the device. This is *not* available in most cases.
- Another method attaches the measurement cable at the lands of a decoupling capacitor in the vicinity of the device that has been removed. The cable's center conductor and shield are soldered directly to the capacitor lands. Alternatively, a probe station with 50Ω RF probes can be used to touch the decoupling capacitor lands.

To protect the spectrum analyzer's sensitive front-end circuitry, add a DC blocking capacitor or attenuator in line. This isolates the spectrum analyzer from the device supply voltage.

Figure 9-13 is an example of a noise spectrum measurement of the  $V_{CC0}$  power-supply noise, with multiple I/O sending patterns at 100 MHz.



UG583\_c3\_13\_112113

Figure 9-13: Screenshot of Spectrum Analyzer Measurement of  $V_{CC0}$

## Optimum Decoupling Network Design

If a highly optimized PDS is needed, measurements and simulations of a prototype system can inform the PDS design. Using knowledge of the noise spectrum generated by the prototype system along with knowledge of the system's power system impedance, the unique transient current of the design can be determined and accommodated.

To measure the noise spectrum of the design under operating conditions, use either a spectrum analyzer or an oscilloscope with FFT. The power system impedance can be determined either through direct measurement or simulation, or a combination of these two as there are often many variables and unknowns.

Both the noise spectrum and the impedance are functions of frequency. By examining the quotient of these per frequency point, transient current as a function of frequency is computed (Equation 9-8):

$$I(f) = \frac{V(f) \text{ From Spectrum Analyzer}}{Z(f) \text{ From Network Analyzer}} \quad \text{Equation 9-8}$$

Using the data sheet's maximum voltage ripple value, the impedance value needed at all frequencies can be determined. This yields a target impedance as a function of frequency. A specially designed capacitor network can accommodate the specific design's transient current.

---

## Troubleshooting

In some cases the proper design work is done up-front, but noise problems still exist. This next section describes possible issues and suggested resolution methods.

### Possibility 1: Excessive Noise from Other Devices on the PCB

Sometimes ground and/or power planes are shared among many devices, and noise from an inadequately decoupled device affects the PDS at other devices. Common causes of this noise are:

- RAM interfaces with inherently high-transient current demands resulting either from temporary periodic contention or high-current drivers
- Large ASICs

When unacceptable amounts of noise are measured locally at these devices, the local PDS and the component decoupling networks should be analyzed.

## Possibility 2: Parasitic Inductance of Planes, Vias, or Connecting Traces

Sometimes the decoupling network capacitance is adequate, but there is too much inductance in the path from the capacitors to the FPGA.

Possible causes are:

- Wrong decoupling capacitor connecting-trace geometry or solder-land geometry
- The path from the capacitors to the FPGA is too long
  - and/or -
- A current path in the power vias traverses an exceptionally thick PCB stackup

For inadequate connecting trace geometry and capacitor land geometry, review the loop inductance of the current path. If the vias for a decoupling capacitor are spaced a few millimeters from the capacitor solder lands on the board, the current loop area is greater than necessary.

To reduce the current loop area, vias should be placed directly against capacitor solder lands. *Never* connect vias to the lands with a section of trace.

Other improvements of geometry are via-in-pad (via under the solder land), not shown, and via-beside-pad (vias straddle the lands instead of being placed at the ends of the lands). Double vias also improve connecting trace geometry and capacitor land geometry.

Exceptionally thick boards (> 3.2 mm or 127 mils) have vias with higher parasitic inductance.

To reduce the parasitic inductance, move critical  $V_{CC}/GND$  plane sandwiches close to the top surface where the FPGA is located, and place the capacitors on the top surface where the FPGA is located.

### Possibility 3: I/O Signals in PCB are Stronger Than Necessary

If noise in the  $V_{CCO}$  PDS is still too high after refining the PDS, the I/O interface slew rate and/or drive strength can be reduced. This applies to both outputs from the FPGA and inputs to the FPGA. In severe cases, excessive overshoot on inputs to the FPGA can reverse-bias the IOB clamp diodes, injecting current into the  $V_{CCO}$  PDS.

If large amounts of noise are present on  $V_{CCO}$ , the drive strength of these interfaces should be decreased, or different termination should be used (on input or output paths).

### Possibility 4: I/O Signal Return Current Traveling in Sub-Optimal Paths

I/O signal return currents can also cause excessive noise in the PDS. For every signal transmitted by a device into the PCB (and eventually into another device), there is an equal and opposite current flowing from the PCB into the device's power/ground system. If a low-impedance return current path is not available, a less optimal, higher impedance path is used. When I/O signal return currents flow over a less optimal path, voltage changes are induced in the PDS, and the signal can be corrupted by crosstalk. This can be improved by ensuring every signal has a closely spaced and fully intact return path.

Methods to correct a sub-optimal return current path:

- Restrict signals to fewer routing layers with verified continuous return current paths.
- Provide low-impedance paths for AC currents to travel between reference planes (high-frequency decoupling capacitors at PCB locations where layer transitions occur).

# Design of Transitions for High-Speed Signals

Each transition in the channel must be designed to minimize any negative impact on the link performance. This chapter addresses the interface at either end of a transmission line.

Transmission lines have defined and controlled characteristic impedance along their length. However, the three-dimensional structures that they interface do not have easily defined or constant impedance along the signal path. Software tools such as 3D field solvers are necessary for computing the impedance that a 10 Gb/s signal sees as it passes through these structures, while 2D field solvers are sufficient for computing transmission line characteristic impedance.

PCB designers can use the analyses and examples in this chapter to assist the design of such a channel. Cases not covered in this chapter might need further simulation and analysis.

---

## Excess Capacitance and Inductance

Most differential transitions are overly capacitive. The P and N paths couple to each other, increasing capacitance. Many transitions have a frequency response identical to that of a lumped capacitor over a wide frequency band.

By design, adding inductance cancels this excess capacitance in many cases except when impacted by density concerns and physical limitations. While techniques such as blind vias, solder balls on a larger pitch, and very small via pads reduce capacitance, they are not always feasible in a design.

Time domain reflectometry (TDR) techniques, either through simulation or measurement, allow the designer to identify excess capacitance or excess inductance in a transition.

---

## Time Domain Reflectometry

To make TDR measurements, a step input is applied to the interconnect. The location and magnitude of the excess capacitance or inductance that the voltage step experiences as it traverses the interconnect can be determined through observing the reflected signal.

A shunt capacitance (see Figure 10-1) causes a momentary dip in the impedance, while a series inductance (see Figure 10-2) causes an impedance discontinuity in the opposite direction.  $T_d$  is the propagation delay through the first transmission line segment on the left. The reflected wave due to the impedance discontinuity takes  $2 * T_d$  to return to the TDR port. If the signal propagation speed through the transmission line is known, the location of the excess capacitance or inductance along the channel can be calculated.

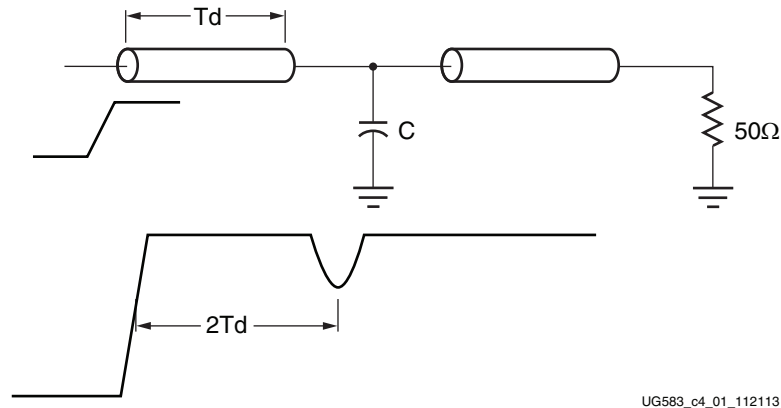


Figure 10-1: TDR Signature of Shunt Capacitance

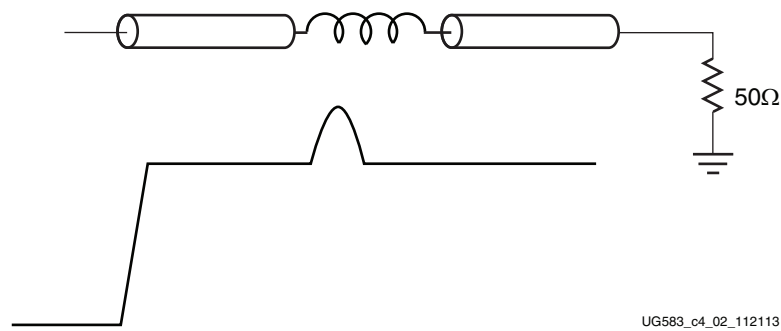


Figure 10-2: TDR Signature of Series Inductance

The magnitude of this excess capacitance (C) or inductance (L) can also be extracted from the TDR waveform by integrating the normalized area of the transition’s TDR response. The respective equations for capacitance and inductance are:

$$C = -\frac{2}{Z_0} \int_{t_1}^{t_2} \frac{V_{tdr}(t) - V_{step}}{V_{step}} dt \tag{Equation 10-1}$$

$$L = 2Z_0 \int_{t_1}^{t_2} \frac{V_{tdr}(t) - V_{step}}{V_{step}} dt \tag{Equation 10-2}$$

Figure 10-3 shows the integration of the normalized TDR area.

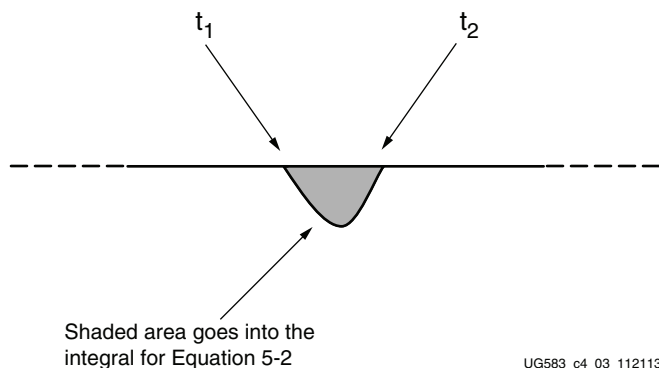


Figure 10-3: Integration of Normalized TDR Area

The results using these equations are not sensitive to rise time variation and are valid for simulated TDR measurements provided that the leading and trailing transmission lines are very close to  $50\Omega$ . However, for actual measurements, accuracy is very dependent on  $Z_0$ .

---

## BGA Package

Each signal path within the BGA package is carefully designed to optimize signal integrity. Traces supporting single-ended I/O are nominally designed for  $50\Omega$  trace impedance. Traces supporting high-speed SERDES I/O are designed for nominally  $100\Omega$  differential impedance. Special care is taken in the design of signal paths to optimize discontinuities such as solder balls and substrate vias to minimize their effect on signal integrity. A 3D full-wave electromagnetic solver and a vector network analyzer are used to model and measure package performance.



## SMT Pads

For applications that require AC coupling between transmitter and receiver, SMT pads are introduced in the channel to allow coupling capacitors to be mounted. Standard SMT pads have excess capacitance due to plate capacitance to a nearby reference plane. In the [Figure 10-4](#) example, a 5 mil trace with a  $Z_0$  of  $50\Omega$  transitions to an 0402 SMT pad that is 28 mils wide, all over 3 mils of FR4.

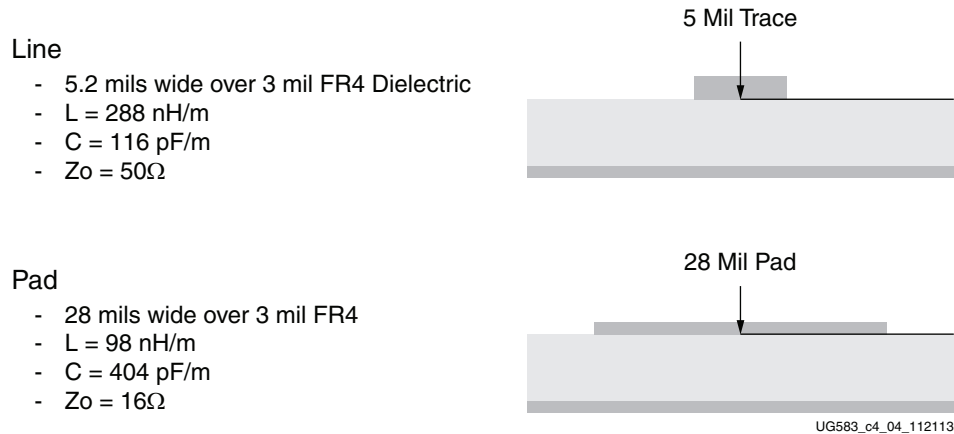


Figure 10-4: 2D Field-Solver Analysis of 5 Mil Trace and 28 Mil Pad

Using a 2D field solver on these dimensions yields a  $Z_0$  of  $50\Omega$  for the 5 mil trace. The  $Z_0$  for the 0402 pad is  $16\Omega$  because the pad has too much capacitance and too little inductance, resulting in an impedance of less than  $50\Omega$ . Performance of this transition can be optimized in one of two ways.

The first method makes the trace the same width as the pad and moves the ground plane deeper into the stackup to maintain the  $Z_0$  of the transition at  $50\Omega$ . This method does not require any special analysis, but there might be some error due to the fringing capacitance of the SMT capacitor body. Trace density is limited because traces are now 28 mils wide.

The second method, shown in [Figure 10-5](#), clears the ground plane underneath the pad, which removes much of the excess capacitance caused by the plate capacitance between the pad and the ground plane. This technique allows for greater trace density than the first method, but requires 3D field-solver analysis or measurement along with several board iterations to get the desired performance.



Figure 10-5: Transition Optimization

The 2D field-solver example shows that close to  $50\Omega$  can be achieved if the ground plane under the pad footprint is cleared out. A 3D field solver is then used to verify this result to a greater degree of accuracy.

Figure 10-6 shows the ground plane cleared away exactly as it was for the 2D simulation. Using frequency domain analysis within HFSS, there is a 20 dB (10x) improvement in return loss using this technique.

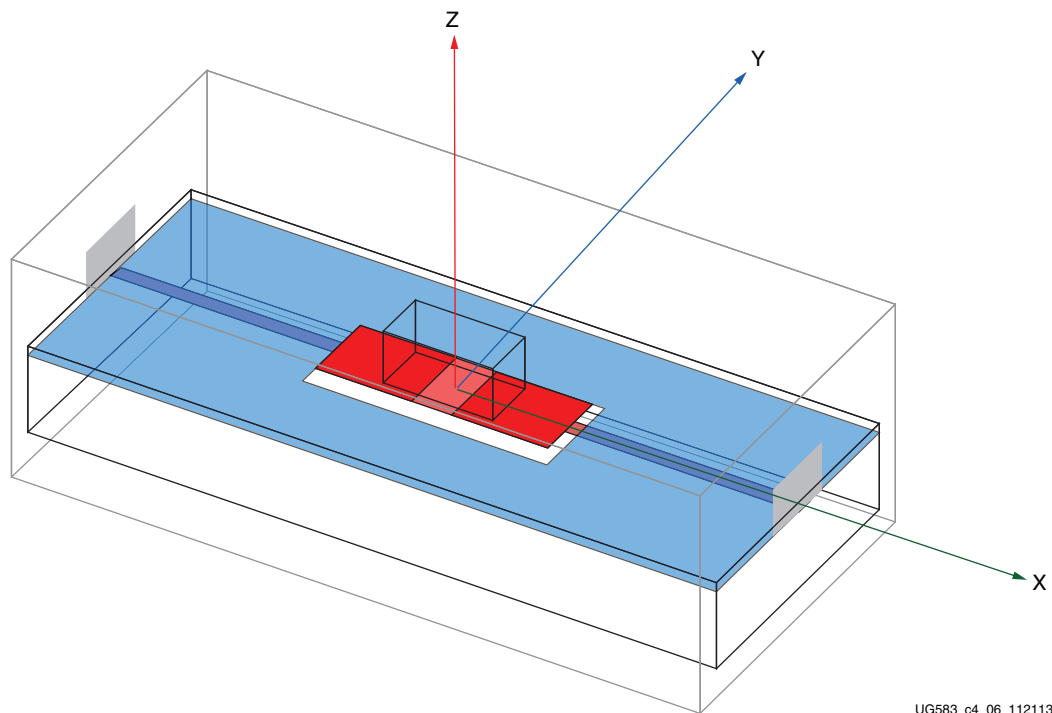


Figure 10-6: Ansoft HFSS Model of Pad Clear-Out

Figure 10-7 shows the return loss comparison between 0402 pad structures with linear scale.

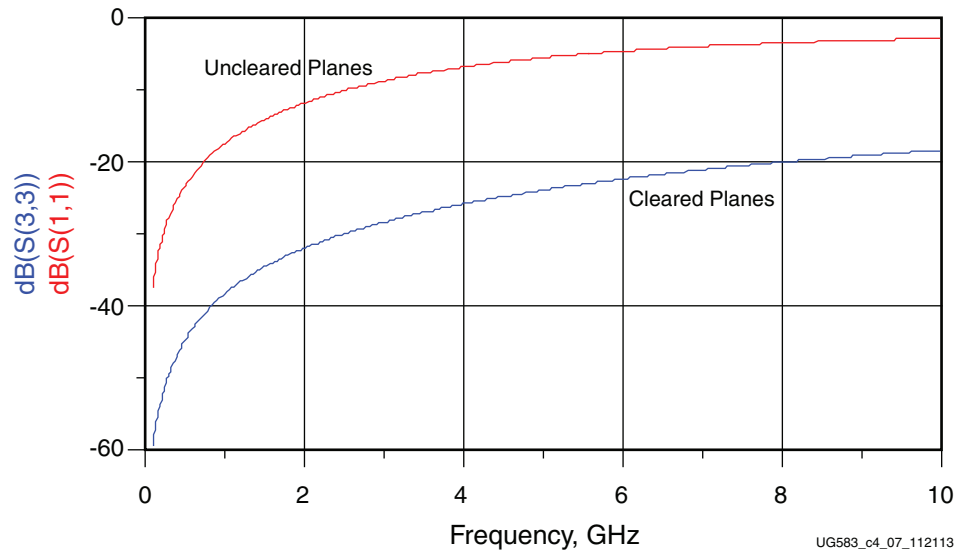


Figure 10-7: Return Loss Comparison Between 0402 Pad Structures

The approximately  $-40$  dB/decade slope in Figure 10-8 shows good fit to the frequency response of a lumped capacitor.

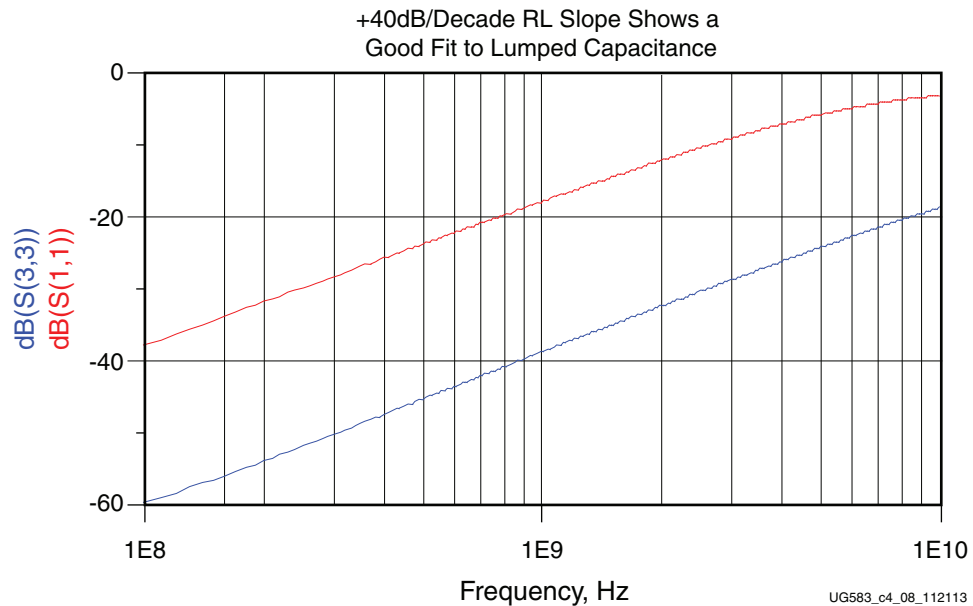


Figure 10-8: Return Loss Comparison Between 0402 Pad Structures on Log (Frequency) Scale

Next, using simulated measurements on the same transition modeled in HFSS, the time-domain performance of this transition can be measured by doing a TDR on the S-parameter results from the earlier frequency domain analysis.

In Figure 10-9 and Figure 10-10, the red curve with the large capacitive dip corresponds to the SMT pad without the ground plane cleared from underneath. The blue curve shows that clearing out the ground plane removes much of the excess capacitance. This improvement can be quantified using Equation 10-1 and Equation 10-2.

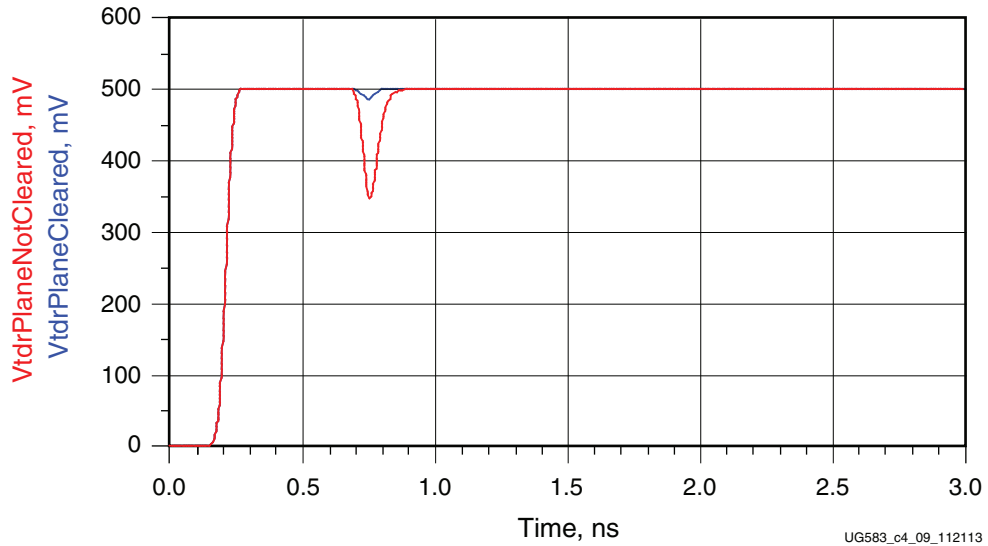


Figure 10-9: TDR Results Comparing 0402 Pad Structures

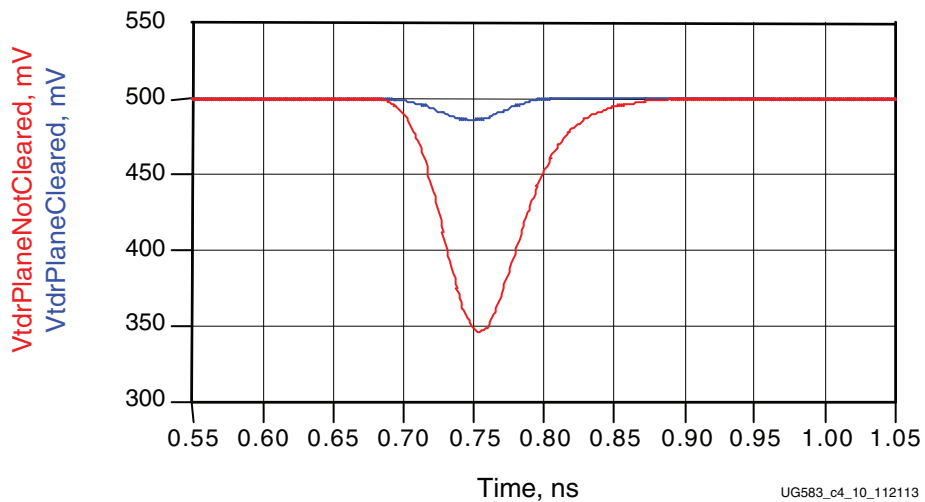


Figure 10-10: TDR Results Comparing 0402 Pad Structures

As shown from Figure 10-11 and Figure 10-12, clearing the ground plane under SMT pads yields a significant improvement in the performance of an SMT pad transition. Excess capacitance is reduced by 15x, and return loss is improved by 20 dB.

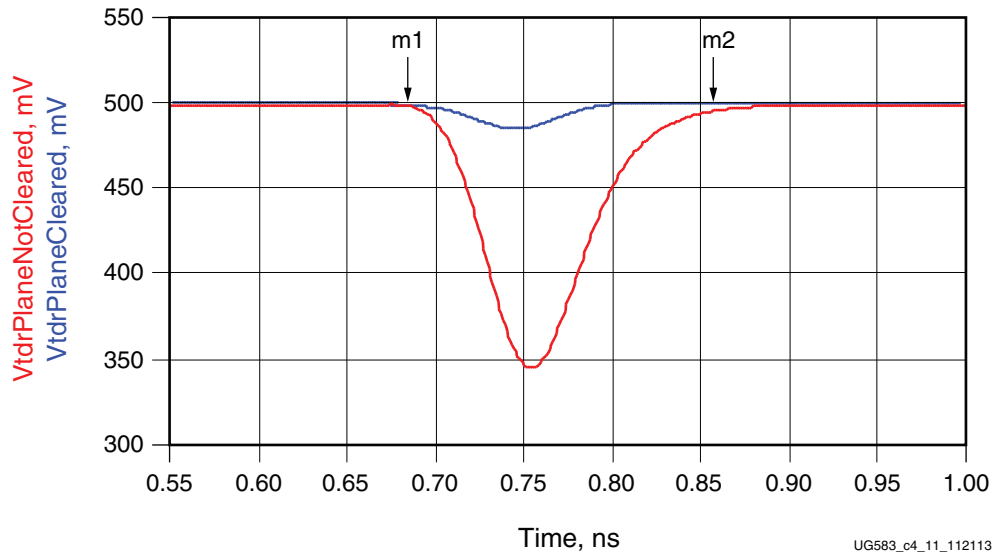


Figure 10-11: 840 fF Excess Capacitance with Ground Plane Intact

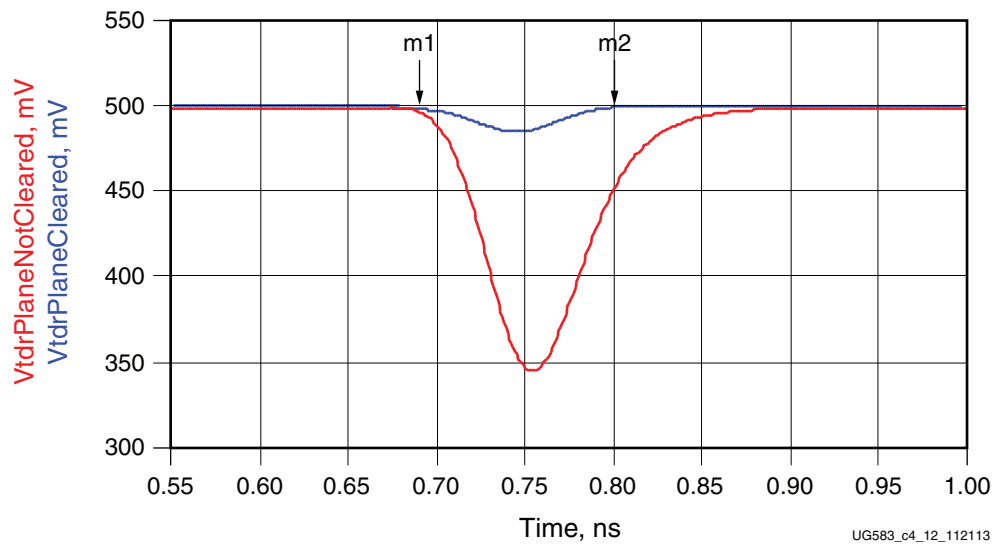


Figure 10-12: 57 fF Excess Capacitance with Ground Plane Intact

## Differential Vias

The most common transition is the differential via where the signal pair must transition from an upper stripline layer or top microstrip to a lower stripline layer or bottom microstrip.

Figure 10-13 shows a Ground-Signal-Signal-Ground (GSSG) type differential via. Ground vias are connected to each ground plane in the stackup, while signal layers only contain pads for the entry and exit layers.

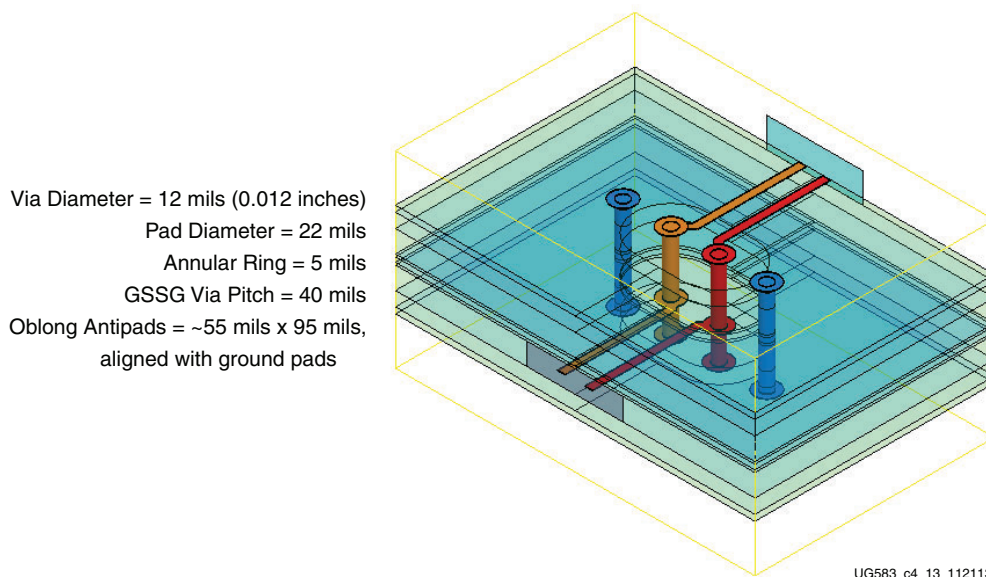


Figure 10-13: Differential Via Design Example

A key advantage of a GSSG via is that it allows for the signal's return current to flow in the ground via near the corresponding signal via, reducing excess inductance. The signal path is also symmetrical between the P and N halves of the differential signal, which is critical in controlling common-mode artifacts due to P/N imbalance.

The larger oblong antipads reduce excess fringing capacitance between the via body and the surrounding planes edges. Unused pads are also removed.

A good starting point is to use the dimensions shown in Figure 10-13 as an example differential via design for an 80 mil board. To accommodate density constraints or the lack thereof, the dimensions can be scaled accordingly to preserve the ratios of each dimension relative to the others. Such scaling preserves the impedance performance of the differential via while allowing variation in overall size to better suit specific applications. These final dimensions are limited by manufacturability and density constraints.

While the via length can be varied by a small amount to suit boards that are thicker or thinner than the 80 mil example, changing the ratio of the via length relative to other dimensions affects the via's impedance. For this and other configurations of differential vias, it is best to simulate a model using 3D field-solver tools to ensure that performance targets are met.

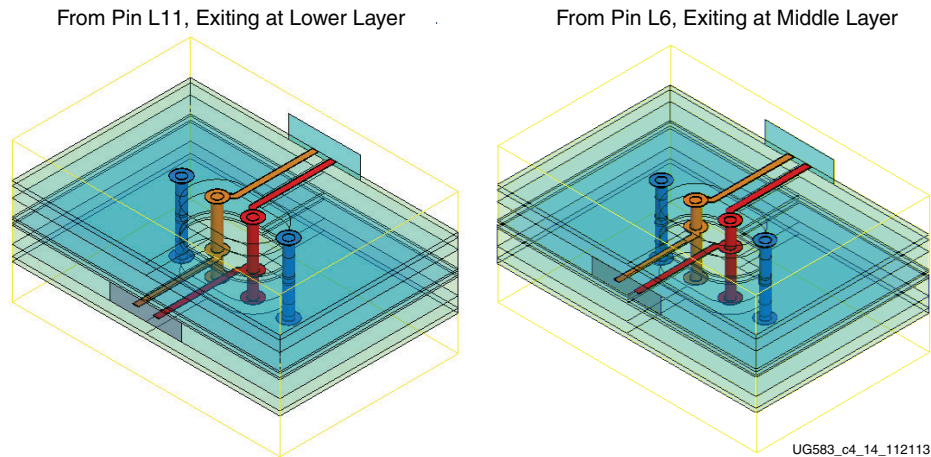


Figure 10-14: Differential GSSG Via in 16-Layer PCB from Pins L11 and L6

As a general rule, the P and N paths need to be kept at equal lengths through a transition. Where possible, via stub length should be kept to a minimum by traversing the signal through the entire length of the vias. The analysis shown in Figure 10-15 compares the S-parameter return loss for common-mode (SCC11) and differential (SDD11) responses.

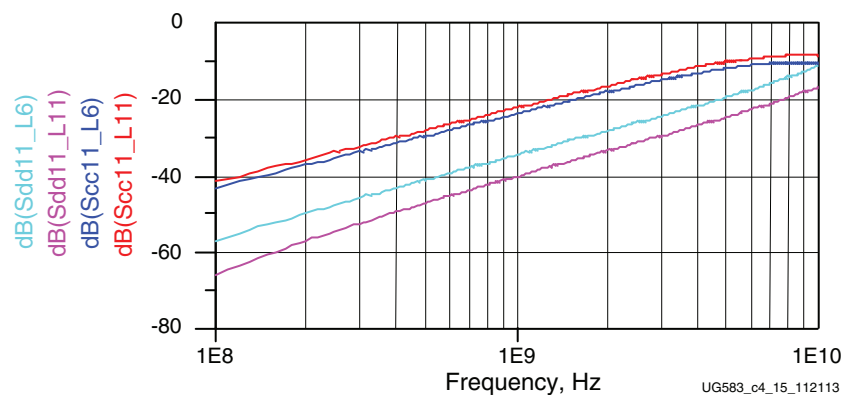


Figure 10-15: Simulated Return Loss Comparing Differential and Common-Mode Losses for L11 and L6 GSSG Vias

From the graph in Figure 10-15, the common-mode response is 20 dB worse in terms of return loss. The much worse common-mode response relative to the differential response is the reason why it is a good idea to reduce P/N skew as much as possible before entering a transition. The 60/40 rule of thumb is 40 dB of return loss at 1 GHz, which implies 60 fF of excess capacitance. Because excess capacitance is a single pole response, simple

extrapolation rules can be used. For example, a shift to 34 dB return loss doubles the excess capacitance. Due to the excellent performance characteristics of GSSG vias, even long via stubs only double the differential via's capacitance at the most.

---

## P/N Crossover Vias

Some transceivers offer the ability to independently switch the polarity of the transmit and receive signal pairs. This functionality eliminates the need to cross over the P/N signals at the board level, which in turn significantly enhances signal integrity. If possible, P/N crossover vias are to be avoided and the polarity switch of the transceiver should be used.

---

## SMA Connectors

Well-designed SMA connectors can reduce debugging time and allow a high-performance channel to be designed correctly on the first pass. SMA connectors that perform well at 10 Gb/s need to be simulated, designed, and manufactured to meet this performance target. Vendors can also offer design services that ensure that the connector works well on a specific board. Assembly guidelines are crucial in ensuring that the process of mating the connector to the board is well-controlled to give the specified performance.

Xilinx uses precision SMA connectors from Rosenberger and other precision connector manufacturers because of their excellent performance and because of the points listed in the previous paragraph.

---

## Backplane Connectors

There are numerous signal integrity issues associated with backplane connectors including:

- P/N signal skew
- Crosstalk
- Stubs due to connector pins

Some connector manufacturers offer not only S parameters, models, and layout guidelines for their connectors but also design support, seminars, and tutorials.



## Microstrip/Stripline Bends

A bend in a PCB trace is a transition. When routing differential traces through a 90° corner, the outer trace is longer than the inner trace, which introduces P/N imbalance. Even within a single trace, signal current has the tendency to hug the inside track of a corner, further reducing the actual delay through a bend.

To minimize skew between the P and N paths, 90° turns in microstrips or striplines are routed as two 45° bends to give mitered corners. The addition of a jog-out also allows the trace lengths to be matched. Figure 10-16 shows example bends in traces.

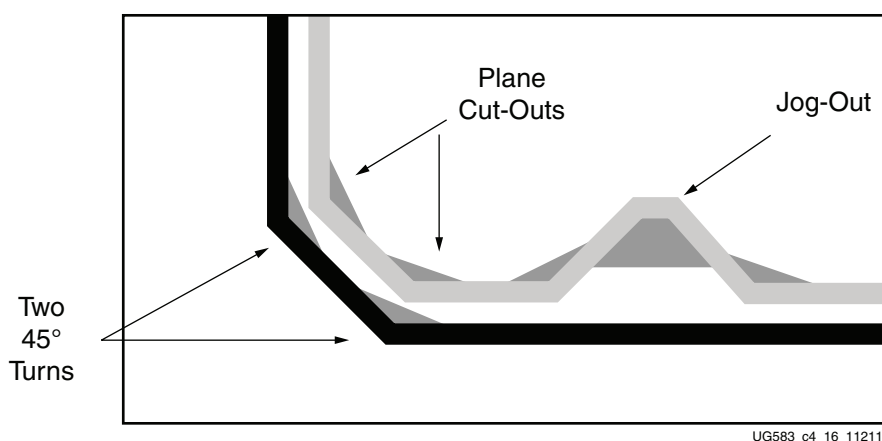


Figure 10-16: Example Design for 90 Degree Bends in Traces

Turns add capacitance because the trace at a 90° corner is 41% wider. That difference is reduced to 8% with a 45° turn. The addition of plane cutouts to a depth of 30 mils act to reduce this amount of excess capacitance. The trace was not widened to maintain 50Ω with the plane cutouts in place.

When this mitered bend is simulated with the jog-out and plane cutouts, excess capacitance is reduced and P/N length and phase matching is improved. Without jog-outs, the P/N length mismatch is 16 mils. Given FR4 material, the 16 mil difference translates to a phase mismatch of 4.8° at 5 GHz, or 2.68 ps (0.0268 UI) at 10 Gb/s.

Figure 10-17 through Figure 10-19 show that phase mismatch is reduced to  $0.75^\circ$  with jog-outs and  $0.3^\circ$  with jog-outs and plane cutouts. The combination of jog-outs and plane cutouts yields simulation results that show the excess capacitance of the structure is reduced to 65 fF.

Designers are tempted to widen lines to compensate for the characteristic impedance increase as the lines are separated and couple less strongly. However, even without widening the lines, the combined capacitance of the corners and jog-outs is still overly capacitive, and therefore the uncoupled section of the jog-out must not be widened.

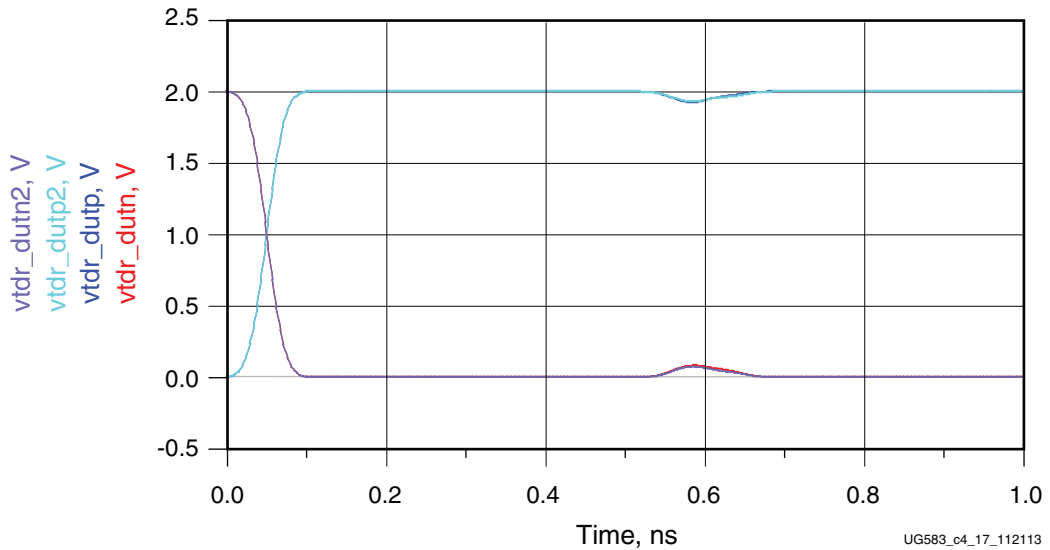


Figure 10-17: Simulated TDR of 45 Degree Bends with Jog-Outs

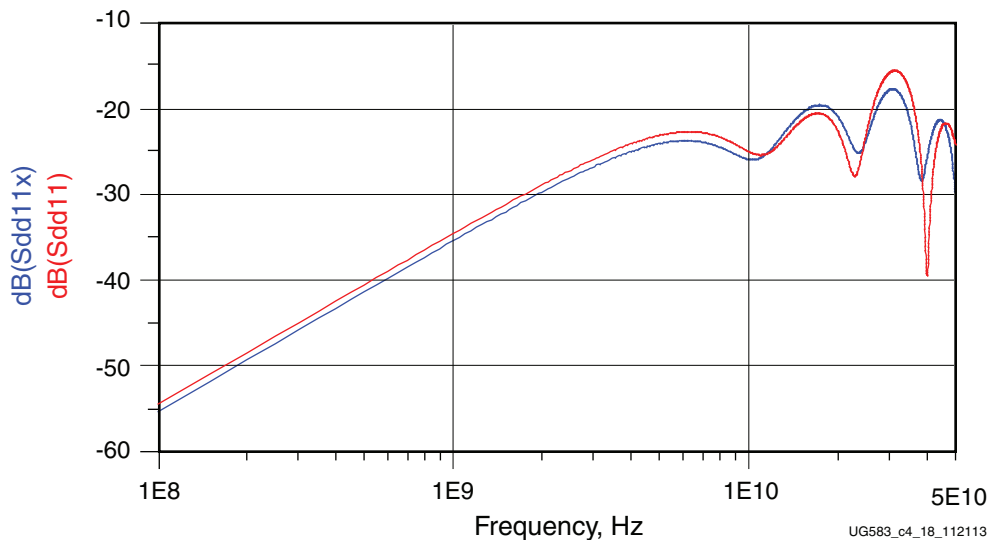


Figure 10-18: Simulated Return Loss of 45 Degree Bends with Jog-Outs

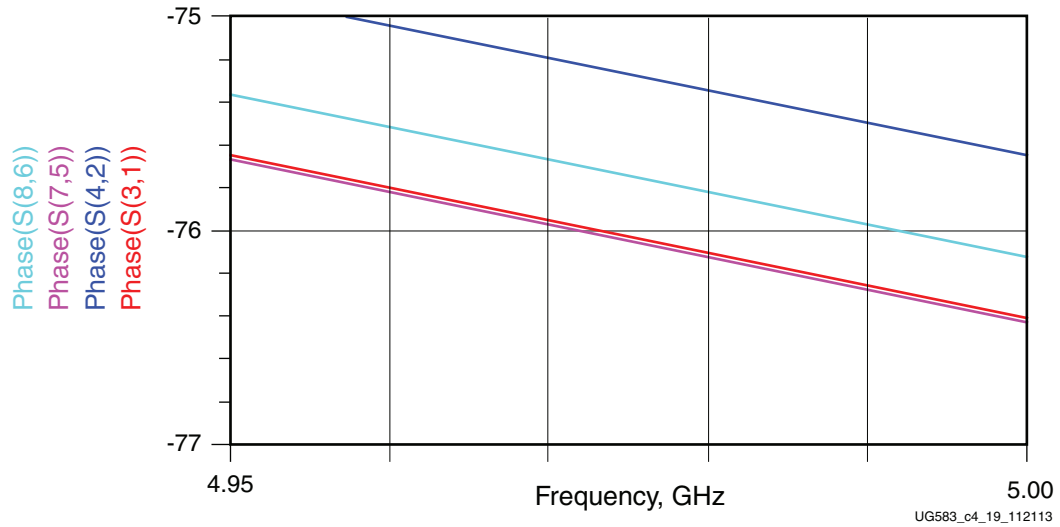


Figure 10-19: Simulated Phase Response of 45 Degree Bends with Jog-Outs

For wide traces, curved routing can also be helpful as shown in Figure 10-20.

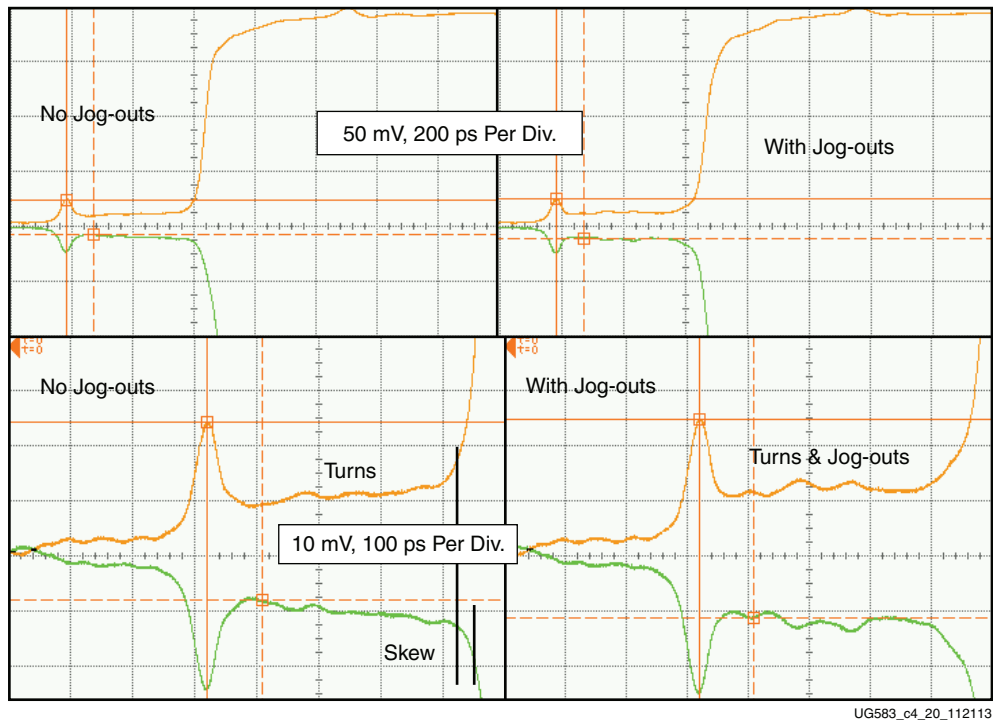


Figure 10-20: Measured TDR of 45 Degree Bends with and without Jog-Outs

# Memory Derating Tables

The skew numbers presented in [Chapter 2, PCB Guidelines for Memory Interfaces](#) were calculated to operate the memory interfaces at their maximum data rates. Some of those skew limits can be relaxed if the respective memory interface is not intended to be operated at the maximum data rate. The tables in this appendix represent the extent to which the respective skew numbers can be relaxed based on the speed rating of the FPGA, the memory component rating, and the actual speed at which the system is operated.

For example, for DDR3 DQ to DQS ([Table A-1](#)) using an FPGA rated at 2133 Mb/s and a memory component rated at 1333 Mb/s while operating at 1333 Mb/s, the skew can be relaxed from 5 ps to 84 ps.

**Note:** The shaded numbers in [Table A-1](#) to [Table A-15](#) represent the maximum skew (plus or minus) allowed for the given FPGA speed rating and memory component rating when operated at that speed.

Table A-1: **DDR3 Data to DQS Skew Limit**

FPGA Rating (Mb/s)		Memory Component Rating (Mb/s)					
Rated	Actual	2133	1866	1600	1333	1066	800
2133	2133	10	N/A	N/A	N/A	N/A	N/A
	1866	39	26	N/A	N/A	N/A	N/A
	1600	83	70	52	N/A	N/A	N/A
	1333	146	133	114	87	N/A	N/A
	1066	150	150	150	150	146	N/A
	800	150	150	150	150	150	150
1866	1866	18	10	N/A	N/A	N/A	N/A
	1600	63	50	31	N/A	N/A	N/A
	1333	125	112	94	66	N/A	N/A
	1066	150	150	150	150	125	N/A
	800	150	150	150	150	150	150
1600	1600	36	23	10	N/A	N/A	N/A
	1333	99	86	68	40	N/A	N/A
	1066	150	150	150	134	99	N/A
	800	150	150	150	150	150	150

Table A-1: DDR3 Data to DQS Skew Limit (Cont'd)

FPGA Rating (Mb/s)		Memory Component Rating (Mb/s)					
Rated	Actual	2133	1866	1600	1333	1066	800
1333	1333	64	51	33	10	N/A	N/A
	1066	150	145	126	99	64	N/A
	800	150	150	150	150	150	150
1066	1066	99	86	68	40	10	N/A
	800	150	150	150	150	150	99
800	800	150	148	130	102	67	10

**Notes:**

- See Table 2-24 and Table 2-31 for the original specifications associated with this table.

 Table A-2: DDR3 Address/Command/Control Bus Skew Limit<sup>(1)</sup>

FPGA Rating (Mb/s)		Memory Component Rating (Mb/s)					
Rated	Actual	2133	1866	1600	1333	1066	800
2133	2133	8	N/A	N/A	N/A	N/A	N/A
	1866	8	8	N/A	N/A	N/A	N/A
	1600	32	32	32	N/A	N/A	N/A
	1333	77	77	77	77	N/A	N/A
	1066	119	119	119	119	119	N/A
	800	150	150	150	150	150	150
1866	1866	8	8	N/A	N/A	N/A	N/A
	1600	32	32	32	N/A	N/A	N/A
	1333	77	77	77	77	N/A	N/A
	1066	119	119	119	119	119	N/A
	800	150	150	150	150	150	150
1600	1600	32	28	8	N/A	N/A	N/A
	1333	77	77	77	77	N/A	N/A
	1066	119	119	119	119	119	N/A
	800	150	150	150	150	150	150
1333	1333	58	48	28	8	N/A	N/A
	1066	119	119	119	119	119	N/A
	800	150	150	150	150	150	150
1066	1066	119	119	101	81	8	N/A
	800	150	150	150	150	150	150

Table A-2: DDR3 Address/Command/Control Bus Skew Limit<sup>(1)</sup> (Cont'd)

FPGA Rating (Mb/s)		Memory Component Rating (Mb/s)					
Rated	Actual	2133	1866	1600	1333	1066	800
800	800	150	150	150	150	150	150

**Notes:**

1. This specification is for the address/command/control bus skew only. The separate CK-to-address specification must be met as specified.
2. See [Table 2-25](#) and [Table 2-32](#) for the original specifications associated with this table.

Table A-3: DDR4 Data to DQS Skew Limit

FPGA Rating (Mb/s)	Running At (Mb/s)	Memory Component Rating (Mb/s)						
		3200	2933	2666	2400	2133	1866	1600
2666	2666	18	16	10	N/A	N/A	N/A	N/A
	2400	38	37	31	24	N/A	N/A	N/A
	2133	64	63	57	50	46	N/A	N/A
	1866	98	96	90	83	79	66	N/A
	1600	142	141	135	128	124	110	92
2400	2400	25	23	17	10	N/A	N/A	N/A
	2133	51	49	43	36	32	N/A	N/A
	1866	84	82	77	70	65	52	N/A
	1600	129	127	121	114	110	97	79
2133	2133	29	27	21	14	10	N/A	N/A
	1866	62	61	55	48	44	30	N/A
	1600	107	105	99	92	88	75	57
1866	1866	42	40	35	28	23	10	N/A
	1600	87	85	79	72	68	55	37
1600	1600	60	58	52	45	41	28	10

**Notes:**

1. See [Table 2-17](#) and [Table 2-34](#) for the original specifications associated with this table.

 Table A-4: DDR4 Address/Command/Control Bus Skew Limit<sup>(1)</sup>

FPGA Rating (Mb/s)	Running At (Mb/s)	Memory Component Rating (Mb/s)						
		3200	2933	2666	2400	2133	1866	1600
2666	2666	23	15	8	N/A	N/A	N/A	N/A
	2400	65	57	50	43	N/A	N/A	N/A
	2133	117	109	102	95	77	N/A	N/A
	1866	150	150	150	150	144	124	N/A
	1600	150	150	150	150	150	150	150

Table A-4: DDR4 Address/Command/Control Bus Skew Limit<sup>(1)</sup> (Cont'd)

FPGA Rating (Mb/s)	Running At (Mb/s)	Memory Component Rating (Mb/s)						
		3200	2933	2666	2400	2133	1866	1600
2400	2400	30	22	15	8	N/A	N/A	N/A
	2133	82	74	67	60	42	N/A	N/A
	1866	149	141	134	127	109	89	N/A
	1600	150	150	150	150	150	150	150
2133	2133	48	40	33	26	8	N/A	N/A
	1866	115	107	100	93	75	55	N/A
	1600	150	150	150	150	150	144	129
1866	1866	68	60	53	46	28	8	N/A
	1600	150	149	142	135	117	97	82
1600	1600	83	75	68	61	43	23	8

**Notes:**

1. This specification is for the address/command/control bus skew only. The separate CK-to-address specification must be met as specified.
2. See [Table 2-18](#) and [Table 2-35](#) for the original specifications associated with this table.

Table A-5: RLD RAM 3 DQ/DM to DK\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)			
Rated	Running at	1200	1066	933	800
1200	1200	5	N/A	N/A	N/A
	1066	31	26	N/A	N/A
	933	65	60	45	N/A
	800	109	104	89	64
1066	1066	10	5	N/A	N/A
	933	43	38	23	N/A
	800	88	83	68	43
933	933	25	20	5	N/A
	800	70	65	50	25
800	800	50	45	30	5

**Notes:**

1. See [Table 2-78](#) for the original specifications associated with this table.

Table A-6: RLDRAM 3 DQ to QK\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)			
Rated	Running at	1200	1066	933	800
1200	1200	5	N/A	N/A	N/A
	1066	31	6	N/A	N/A
	933	65	40	9	N/A
	800	109	84	54	13
1066	1066	30	5	N/A	N/A
	933	63	38	8	N/A
	800	108	83	53	11
933	933	60	35	5	N/A
	800	105	80	50	8
800	800	105	77	46	5

**Notes:**

1. See [Table 2-78](#) for the original specifications associated with this table.

Table A-7: RLDRAM 3 DK\_P/N to CK\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)			
Rated	Running at	1200	1066	933	800
1200	1200	5	N/A	N/A	N/A
	1066	57	29	N/A	N/A
	933	124	96	60	N/A
	800	150	150	149	101
1066	1066	33	5	N/A	N/A
	933	100	72	36	N/A
	800	150	150	125	77
933	933	69	41	5	N/A
	800	150	130	94	46
800	800	118	89	53	5

**Notes:**

1. See [Table 2-78](#) for the original specifications associated with this table.



Table A-8: RLDRAM 3 ADDR/CMD to CK\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)			
Rated	Running at	1200	1066	933	800
1200	1200	5	N/A	N/A	N/A
	1066	57	42	N/A	N/A
	933	124	109	74	N/A
	800	150	150	150	128
1066	1066	20	5	N/A	N/A
	933	87	72	37	N/A
	800	150	150	126	91
933	933	55	40	5	N/A
	800	144	129	94	59
800	800	90	75	40	5

**Notes:**

- See [Table 2-85](#) for the original specifications associated with this table.

Table A-9: QDR II+ D to K\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	6	N/A	N/A
	600	27.72	27.72	N/A
	550	65.6	65.6	65.6
600	633	N/A	N/A	N/A
	600	6	6	N/A
	550	43.88	43.88	43.88
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	6	6	6

**Notes:**

- See [Table 2-85](#) for the original specifications associated with this table.

Table A-10: QDR II+ Q to CQ\_P/N Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	6	N/A	N/A
	600	27.72	27.72	N/A
	550	65.6	65.6	5.6

Table A-10: QDR II+ Q to CQ\_P/N Skew Limit (Cont'd)

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
600	633	N/A	N/A	N/A
	600	6	6	N/A
	550	103.88	73.88	43.88
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	66	36	6

**Notes:**

1. See [Table 2-85](#) for the original specifications associated with this table.

Table A-11: QDR II+ Data Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	5	N/A	N/A
	600	42.88	42.88	N/A
	550	64.6	64.6	64.6
600	633	N/A	N/A	N/A
	600	5	5	N/A
	550	42.88	42.88	42.88
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	5	5	5

**Notes:**

1. See [Table 2-85](#) for the original specifications associated with this table.

Table A-12: QDR II+ ADDR/CMD Skew Limit

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	6	N/A	N/A
	600	49.44	49.44	N/A
	550	125.2	125.2	125.2
600	633	N/A	N/A	N/A
	600	6	6	N/A
	550	81.76	81.76	81.76

Table A-12: QDR II+ ADDR/CMD Skew Limit (Cont'd)

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	6	6	6

**Notes:**

- See [Table 2-85](#) for the original specifications associated with this table.

Table A-13: QDR II+ ADDR/CMD to CLK Skew Limit for Clock Point-to-Point and CA Fly-by

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	6	N/A	N/A
	600	49.44	49.44	N/A
	550	125.2	125.2	125.2
600	633	N/A	N/A	N/A
	600	6	6	N/A
	550	81.76	81.76	81.76
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	6	6	6

**Notes:**

- See [Table 2-85](#) for the original specifications associated with this table.

Table A-14: QDR II+ ADDR/CMD to CLK Skew Limit for Clock T-Branch and CA Fly-by

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	34	N/A	N/A
	600	77.44	77.44	N/A
	550	153.2	153.2	153.2
600	633	N/A	N/A	N/A
	600	34	34	N/A
	550	109.76	109.76	109.76
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	34	34	34

**Notes:**

- See [Table 2-85](#) for the original specifications associated with this table.

Table A-15: QDR II+ ADDR/CMD to CLK Skew Limit for Clock T-Branch and CA T-Branch

FPGA Rating (MHz)		Memory Component Rating (MHz)		
Rated	Actual	633	600	550
633	633	6	N/A	N/A
	600	49.44	49.44	N/A
	550	125.2	125.2	125.2
600	633	N/A	N/A	N/A
	600	6	6	N/A
	550	81.76	81.76	81.76
550	633	N/A	N/A	N/A
	600	N/A	N/A	N/A
	550	6	6	6

**Notes:**

1. See [Table 2-85](#) for the original specifications associated with this table.

# Material Properties and Insertion Losses

This appendix lists various material types and their associated characteristics along with relative cost multipliers to assist with choosing the best printed circuit board material.

**Table B-1: Material Parameters and Cost Multipliers**

	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
DK	3.48	3.49	3.25	3.65	4.4	4.4
DF	0.0057	0.006	0.0074	0.0115	0.017	0.017
Surface Roughness	0.5 $\mu\text{m}$	2 $\mu\text{m}$	2 $\mu\text{m}$	2 $\mu\text{m}$	5 $\mu\text{m}$	8 $\mu\text{m}$
Copper Conductivity	$4 \times 10^7$ S/m	$4 \times 10^7$ S/m	$4 \times 10^7$ S/m	$4 \times 10^7$ S/m	$4 \times 10^7$ S/m	$4 \times 10^7$ S/m
Cost	3.8X	3.5X	2.3X	1.7X	1.2X	1X

**Table B-2: Insertion Loss @ 2.4 GHz**

IL @ 2.4GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 4 mil	-0.318	-0.469	-0.474	-0.528	-0.658	-0.692
W = 6 mil	-0.246	-0.362	-0.367	-0.422	-0.539	-0.565
W = 8 mil	-0.211	-0.302	-0.308	-0.359	-0.467	-0.488
W = 10 mil	-0.184	-0.263	-0.272	-0.323	-0.427	-0.444
W = 12 mil	-0.167	-0.233	-0.246	-0.293	-0.399	-0.414

**Notes:**

- 1 mil = 0.0254 mm

**Table B-3: Insertion Loss @ 4.0 GHz**

IL @ 4 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 4 mil	-0.452	-0.667	-0.679	-0.765	-0.954	-1.002
W = 6 mil	-0.354	-0.518	-0.529	-0.616	-0.794	-0.829
W = 8 mil	-0.306	-0.435	-0.446	-0.531	-0.696	-0.724

Table B-3: Insertion Loss @ 4.0 GHz (Cont'd)

IL @ 4 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 10 mil	-0.270	-0.381	-0.397	-0.481	-0.641	-0.665
W = 12 mil	-0.246	-0.340	-0.362	-0.440	-0.603	-0.624

**Notes:**

- 1 mil = 0.0254 mm

Table B-4: Insertion Loss @ 5.0 GHz

IL @ 5 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 4 mil	-0.531	-0.774	-0.788	-0.896	-1.126	-1.178
W = 6 mil	-0.417	-0.604	-0.618	-0.728	-0.942	-0.981
W = 8 mil	-0.362	-0.509	-0.525	-0.628	-0.831	-0.861
W = 10 mil	-0.321	-0.447	-0.467	-0.572	-0.768	-0.794
W = 12 mil	-0.294	-0.400	-0.427	-0.524	-0.724	-0.747

**Notes:**

- 1 mil = 0.0254 mm

Table B-5: Insertion Loss @ 6.0 GHz

IL @ 6 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 4 mil	-0.607	-0.872	-0.890	-1.018	-1.290	-1.356
W = 6 mil	-0.479	-0.683	-0.699	-0.832	-1.085	-1.134
W = 8 mil	-0.417	-0.577	-0.596	-0.720	-0.961	-0.999
W = 10 mil	-0.370	-0.508	-0.532	-0.658	-0.891	-0.924
W = 12 mil	-0.339	-0.455	-0.487	-0.604	-0.842	-0.871

**Notes:**

- 1 mil = 0.0254 mm

Table B-6: Insertion Loss @ 12.9 GHz

IL @ 6 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 4 mil	-1.099	-1.440	-1.485	-1.754	-2.307	-2.694
W = 6 mil	-0.882	-1.147	-1.191	-1.466	-1.996	-2.275
W = 8 mil	-0.775	-0.983	-1.032	-1.294	-1.792	-2.020
W = 10 mil	-0.695	-0.878	-0.932	-1.198	-1.683	-1.877

Table B-6: Insertion Loss @ 12.9 GHz (Cont'd)

IL @ 6 GHz (dB/in)	Rogers Corporation (Rogers_4350)	Panasonic (MEGTRON 6)	Isola Group (I-Speed)	Isola Group (FR408HRIS)	Isola Group (Standard FR4_LP)	Isola Group (Standard FR4_LP)
W = 12 mil	-0.642	-0.795	-0.863	-1.115	-1.607	-1.776

**Notes:**

- 1 mil = 0.0254 mm

# Additional Resources and Legal Notices

---

## Xilinx Resources

For support resources such as Answers, Documentation, Downloads, and Forums, see [Xilinx Support](#).

For a glossary of technical terms used in Xilinx documentation, see the [Xilinx Glossary](#).

---

## Solution Centers

See the [Xilinx Solution Centers](#) for support on devices, software tools, and intellectual property at all stages of the design cycle. Topics include design assistance, advisories, and troubleshooting tips.

---

## Documentation Navigator and Design Hubs

Xilinx® Documentation Navigator provides access to Xilinx documents, videos, and support resources, which you can filter and search to find information. To open the Xilinx Documentation Navigator (DocNav):

- From the Vivado® IDE, select **Help > Documentation and Tutorials**.
- On Windows, select **Start > All Programs > Xilinx Design Tools > DocNav**.
- At the Linux command prompt, enter `docnav`.

Xilinx Design Hubs provide links to documentation organized by design tasks and other topics, which you can use to learn key concepts and address frequently asked questions. To access the Design Hubs:

- In the Xilinx Documentation Navigator, click the **Design Hubs View** tab.
- On the Xilinx website, see the [Design Hubs](#) page.



**Note:** For more information on Documentation Navigator, see the [Documentation Navigator](#) page on the Xilinx website.

---

## References

1. *Kintex UltraScale FPGAs Data Sheet: DC and AC Switching Characteristics* ([DS892](#))
2. *Virtex UltraScale FPGAs Data Sheet: DC and AC Switching Characteristics* ([DS893](#))
3. *Kintex UltraScale+ FPGAs Data Sheet: DC and AC Switching Characteristics* ([DS922](#))
4. *Virtex UltraScale+ FPGAs Data Sheet: DC and AC Switching Characteristics* ([DS923](#))
5. *Zynq UltraScale+ RFSoc Data Sheet: DC and AC Switching Characteristics* ([DS926](#))
6. *UltraScale Architecture GTH Transceivers User Guide* ([UG576](#))
7. *UltraScale Architecture GTY Transceivers User Guide* ([UG578](#))
8. *UltraScale Architecture Schematic Review Checklist* ([XTP344](#))
9. *UltraScale and UltraScale+ FPGAs Packaging and Pinouts Product Specification* ([UG575](#))
10. *UltraScale Architecture SelectIO Resources User Guide* ([UG571](#))
11. *Interfacing 7 Series FPGAs High-Performance I/O Banks with 2.5V and 3.3V I/O Standards* ([XAPP520](#))
12. *UltraScale Architecture Configuration User Guide* ([UG570](#))
13. *UltraScale Architecture FPGAs Memory IP Product Guide* ([PG150](#))
14. *UltraScale Architecture Gen3 Integrated Block for PCI Express LogiCORE IP Product Guide* ([PG156](#))
15. *UltraScale Architecture Integrated Block for 100G Ethernet LogiCORE IP Product Guide* ([PG165](#))
16. *UltraScale Architecture Integrated IP Core for Interlaken LogiCORE IP Product Guide* ([PG169](#))
17. *Zynq UltraScale+ RFSoc RF Data Converter LogiCORE IP Product Guide* ([PG269](#))
18. *UltraScale+ FPGA and Zynq UltraScale+ MPSoC Schematic Review Checklist* ([XTP427](#))
19. *UltraScale Architecture and Product Overview* ([DS890](#))
20. VESA DisplayPort Standard Version 1  
[www.vesa.org](http://www.vesa.org)
21. JESD84-B451, *Embedded Multi-media Card (eMMC), Electrical Standard (5.1)*  
[www.jedec.org/standards-documents/results/jesd84-b451](http://www.jedec.org/standards-documents/results/jesd84-b451)
22. *Zynq UltraScale+ MPSoC Data Sheet: DC and AC Switching Characteristics* ([DS925](#))

23. *Zynq UltraScale+ MPSoC Technical Reference Manual* ([UG1085](#))
24. *Managing Power and Performance with the Zynq UltraScale+ MPSoC* ([WP482](#))
25. *UltraScale Architecture System Monitor User Guide* ([UG580](#))
26. *Zynq UltraScale+ MPSoC Packaging and Pinouts Product Specification User Guide* ([UG1075](#))
27. *UltraScale+ Devices Integrated Block for PCI Express LogiCORE IP Product Guide* ([PG213](#))
28. *UltraScale+ Devices Integrated 100G Ethernet Subsystem Product Guide* ([PG203](#))
29. *Interlaken 150G LogiCORE IP Product Guide* ([PG212](#))

---

## Please Read: Important Legal Notices

The information disclosed to you hereunder (the "Materials") is provided solely for the selection and use of Xilinx products. To the maximum extent permitted by applicable law: (1) Materials are made available "AS IS" and with all faults, Xilinx hereby DISCLAIMS ALL WARRANTIES AND CONDITIONS, EXPRESS, IMPLIED, OR STATUTORY, INCLUDING BUT NOT LIMITED TO WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, OR FITNESS FOR ANY PARTICULAR PURPOSE; and (2) Xilinx shall not be liable (whether in contract or tort, including negligence, or under any other theory of liability) for any loss or damage of any kind or nature related to, arising under, or in connection with, the Materials (including your use of the Materials), including for any direct, indirect, special, incidental, or consequential loss or damage (including loss of data, profits, goodwill, or any type of loss or damage suffered as a result of any action brought by a third party) even if such damage or loss was reasonably foreseeable or Xilinx had been advised of the possibility of the same. Xilinx assumes no obligation to correct any errors contained in the Materials or to notify you of updates to the Materials or to product specifications. You may not reproduce, modify, distribute, or publicly display the Materials without prior written consent. Certain products are subject to the terms and conditions of Xilinx's limited warranty, please refer to Xilinx's Terms of Sale which can be viewed at <https://www.xilinx.com/legal.htm#tos>; IP cores may be subject to warranty and support terms contained in a license issued to you by Xilinx. Xilinx products are not designed or intended to be fail-safe or for use in any application requiring fail-safe performance; you assume sole risk and liability for use of Xilinx products in such critical applications, please refer to Xilinx's Terms of Sale which can be viewed at <https://www.xilinx.com/legal.htm#tos>.

### **AUTOMOTIVE APPLICATIONS DISCLAIMER**

AUTOMOTIVE PRODUCTS (IDENTIFIED AS "XA" IN THE PART NUMBER) ARE NOT WARRANTED FOR USE IN THE DEPLOYMENT OF AIRBAGS OR FOR USE IN APPLICATIONS THAT AFFECT CONTROL OF A VEHICLE ("SAFETY APPLICATION") UNLESS THERE IS A SAFETY CONCEPT OR REDUNDANCY FEATURE CONSISTENT WITH THE ISO 26262 AUTOMOTIVE SAFETY STANDARD ("SAFETY DESIGN"). CUSTOMER SHALL, PRIOR TO USING OR DISTRIBUTING ANY SYSTEMS THAT INCORPORATE PRODUCTS, THOROUGHLY TEST SUCH SYSTEMS FOR SAFETY PURPOSES. USE OF PRODUCTS IN A SAFETY APPLICATION WITHOUT A SAFETY DESIGN IS FULLY AT THE RISK OF CUSTOMER, SUBJECT ONLY TO APPLICABLE LAWS AND REGULATIONS GOVERNING LIMITATIONS ON PRODUCT LIABILITY.

© Copyright 2013–2018 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, ISE, Kintex, Spartan, Virtex, Vivado, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. PCI, PCIe, and PCI Express are trademarks of PCI-SIG and used under license. All other trademarks are the property of their respective owners.