SmartFusion 2 and IGLOO 2 FPGA Board and Layout Design Guidelines

AN4153

Introduction (Ask a Question)

This application note provides board-level design guidelines for SmartFusion® 2 and IGLOO® 2 devices. The 2. Layout Guidelines for SmartFusion 2 and IGLOO 2-Based Board Design section describes the PCB design. These guidelines must be treated as a supplement to standard board-level design practices.

Good board design practices are required to obtain expected performance from both PCB and SmartFusion 2/IGLOO 2 devices. High quality and reliable results depend on minimizing noise levels, preserving signal integrity, meeting impedance and power requirements.

This document assumes that the reader has a good understanding of the SmartFusion 2/IGLOO 2 devices, is experienced in digital and analog board design, and knows about the electrical characteristics of systems.

Important: The standard uses “Master” and “Slave.” The equivalent Microchip terminology used in this document is **Initiator** and **Target**, respectively.

Table of Contents

1. Design Considerations (Ask a Question)

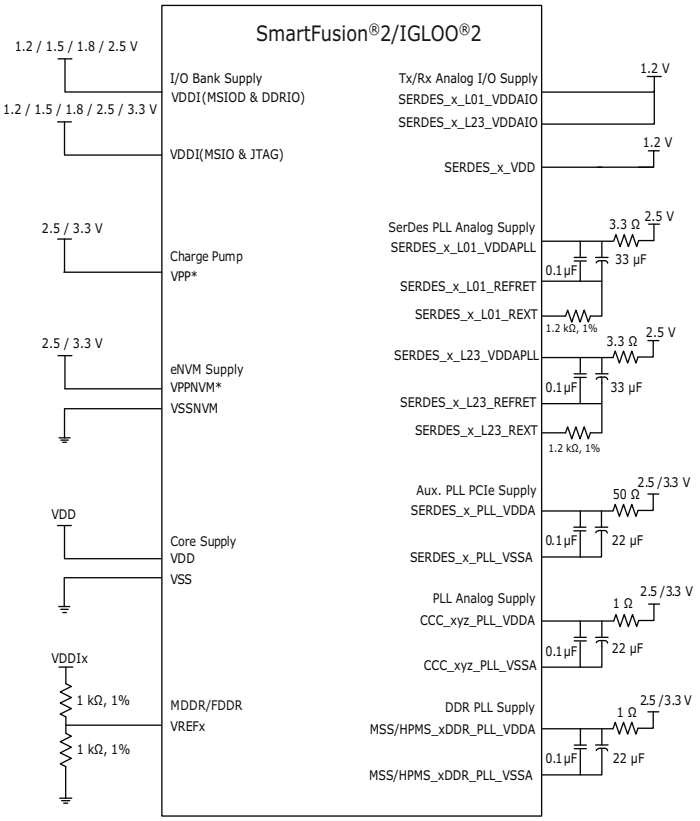
The SmartFusion2/IGLOO2 devices support various high-speed interfaces using both Double Data Rate Input/Output (DDRIO) and SerDes I/O. DDRIO is a multi-standard I/O optimized for low-power DDR, DDR2, and DDR3 performance. SerDes I/O are dedicated to high-speed serial communication protocols. The SerDes I/O supports protocols such as PCI Express 2.0, 10 Gbps attachment unit interface (XAUI), Serial Gigabit Media Independent Interface (SGMII), JESD204B, and user-defined high-speed serial protocol implementation in fabric.

Routing high-speed serial data over a PCB is a challenge as losses, dispersion, and crosstalk effects increase with speed. Channel losses and crosstalk decrease the signal-to-noise ratio and limit the data rate on the channel.

1.1 Power Supplies (Ask a Question)

The following figure shows the typical power supply requirements for SmartFusion2/IGLOO2 devices. For more information about decoupling capacitors associated with individual power supplies, see Table 1-2.

Figure 1-1. Power Supplies



Important: The power supply settings for all the PLLs (Auxiliary PLL PCIe Supply, SerDes, MDDR, FDDR, and CCC) must be the same in Libero® SoC and on the board. The PLLs (Auxiliary PLL PCIe Supply, SerDes, MDDR, FDDR, and CCC) can be connected to a 2.5V or 3.3V supply.

For M2S090T(S), M2GL090T(S), M2S150T(S), and M2GL150T(S) devices, the VPP and VPPNVM must be connected to a 3.3V supply.

For the CCC_xyz_PLL supplies, xy refers to the location of the PLL in the device (NE/ NW/ SW) and z refers to the number associated with the PLL (0 or 1).

The PLL RC values shown in the preceding figure are applicable to all variants of SmartFusion2/ IGLOO2 devices.

When the power rails are in use, see Figure 1-1. For unused cases, see Figure 1-3. The power supply needed for IGLOO2 and SmartFusion2 FPGAs is Core VDD (Fabric voltage). Serdes_VDD is tied to VDD internally. Therefore, the same regulator must be used to drive SERDES_x_VDD, SERDES_x_Lyz_VDDAIO, and Core VDD together.

For the device-package combinations listed in the following table, the SERDES_x_VDD pins are shorted with VDD pins inside the package substrate to free the package pins.

Table 1-1. Device-Package Combinations Without SERDES_x_VDD Pin

Device	Package
M2S025T, M2GL025T	FCS325
M2S050T(S), M2GL050T(S)	FCS325
M2S060T(S), M2GL060T(S)	FCS325
M2S90T(S), M2GL90T(S)	FCS325
M2S10T(S), M2GL010T(S)	VF256
M2S025TS, M2S025T, M2GL025TS, M2GL025T	VF256
M2S150T(S), M2GL150T(S)	FCV484
M2S150TS, M2S150T, M2GL150TS, M2GL150T	FCS536

For detailed pin descriptions, see DS0115: SmartFusion2 Pin Descriptions Datasheet or DS0124: IGLOO2 Pin Descriptions Datasheet.

1.1.1 Power Supply Decoupling (Ask a Question)

To reduce any potential fluctuation on the power supply lines, decoupling capacitors, bypass capacitors, and other power supply filtering techniques must be used.

- For values ranging from 1 nF to 100 µF, use X7R or X5R (dielectric material) type capacitors.
- For values ranging from 100 µF to 1000 µF, use tantalum capacitors.

The following table lists the recommended number of PCB decoupling capacitors for an M2S050T/ M2GL050T-FG896 device.

Table 1-2. Power Supply Decoupling Capacitors

Pin Name	Number of Pins	Ceramic Caps					Tantalum Caps			
		0.01 µF	0.1 µF	10 µF	33 µF	22 µF	47 µF	100 µF	220 µF[1]	330 µF
VDD	24	12	12	—	—	—	—	1	3	1
VDD10	29	14	14	—	—	—	2	—	—	—
VDDI1	4	2	2	1	—	—	—	—	—	—
VDDI2	4	2	2	1	—	—	—	—	—	—
VDDI3	5	2	3	1	—	—	—	—	—	—
VDDI4	3	2	1	1	—	—	—	—	—	—
VDDI5	29	14	14	—	—	—	2	—	—	—
VDDI6	1	1	—	1	—	—	—	—	—	—
VDDI7	6	3	3	1	—	—	—	—	—	—
VDDI8	5	2	3	1	—	—	—	—	—	—

.....continued										
Pin Name	Number of Pins	Ceramic Caps					Tantalum Caps			
VDDI9	1	1	1	—	—	—	—	—	—	—
VPP	4	2	2	1	—	—	—	—	—	—
VREF0	3	2	1	1	—	—	—	—	—	—
VREF5	3	2	1	1	—	—	—	—	—	—
VPPNVM	1	1	—	1	—	—	—	—	—	—
SERDES_0_VDD	2	1	1	1[2]	—	—	—	—	—	—
SERDES_1_VDD	2	1	1	1[2]	—	—	—	—	—	—
SERDES_0_L01_VDDAIO	1	1[2]	1[2]	1[3]	—	—	—	—	—	—
SERDES_0_L23_VDDAIO	1	1[2]	1[2]	1[3]	—	—	—	—	—	—
SERDES_1_L01_VDDAIO	1	1[2]	1[2]	1[3]	—	—	—	—	—	—
SERDES_1_L23_VDDAIO	1	1[2]	1[2]	1[3]	—	—	—	—	—	—
CCC_NE0_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
CCC_NE1_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
CCC_NW0_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
CCC_NW1_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
CCC_SW0_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
CCC_SW1_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
MSS_FDDR_PLL_VDDA	1	—	1	—	—	1	—	—	—	—
MSS_PLL_MDDR_VDDA	1	—	1	—	—	1	—	—	—	—
PLL_SERDES_0_VDDA	1	—	1	—	—	1	—	—	—	—
PLL_SERDES_1_VDDA	1	—	1	—	—	1	—	—	—	—
SERDES_0_L01_VDDAPLL	1	—	1	—	1	—	—	—	—	—
SERDES_0_L23_VDDAPLL	1	—	1	—	1	—	—	—	—	—
SERDES_1_L01_VDDAPLL	1	—	1	—	1	—	—	—	—	—
SERDES_1_L23_VDDAPLL	1	—	1	—	1	—	—	—	—	—

Notes:

1. 220 µF is used to limit surge current for the VDD supply.
2. Single ceramic decoupling capacitor is required for both the pins at the device.
3. Single ceramic decoupling capacitor is required for four pins at the device.

The following table lists the recommended decoupling capacitors for the SmartFusion2/IGLOO2 devices. For placement and routing details, see Layout Guidelines for SmartFusion2- and IGLOO2-Based Board Design.

Table 1-3. Recommended Capacitors

Part Number	Manufacturer	Description
GRM155R71C103KA01D	Murata	Ceramic 0.01 µF, 16V, 10%, X7R, 0402
GRM155R71C104KA88D	Murata	Ceramic 0.1 µF, 16V, 10%, X7R, 0402
GRM188R60J106ME47D	Murata	Ceramic 10 µF, 6.3V, X5R, 0603
T491B475M016AT	KEMET	Tantalum 4.7 µF, 16V, 20%, 1411
T491B226M016AT	KEMET	Tantalum 22 µF, 16V, 20%, 1411

.....continued		
Part Number	**Manufacturer**	**Description**
T491B476M010AT	KEMET	Tantalum 47 µF, 10V, 20%SMD
T520V107M010ATE050	KEMET	Tantalum 100 µF, 10V, 20%, 2917
TPSD337K010R0050	AVX	Tantalum 330 µF, 10V, 10%, 2917

1.1.2 Power Supply Sequencing (Ask a Question)

On detection of a power-up event, the POR circuit sends the power-on reset signal to the system controller and reset controller in the SmartFusion2/IGLOO2 devices. The power-on reset circuitry in SmartFusion2/IGLOO2 devices require the VDD and VPP supplies to ramp monotonically from 0V to the minimum recommended operating voltage within a predefined time. There is no sequencing requirement on VDD and VPP. Four ramp rate options are available during design generation: 50 µs, 1 ms, 10 ms, and 100 ms. Each selection represents the maximum ramp rate to apply to VDD and VPP. The ramp rates can be configured by using the Libero software.

The SERDES_VDD pins are shorted to VDD on silicon die; therefore, Microchip recommends using the same regulator to power up the VDD, SERDES_VDD and SERDES_VDDAIO pins. These three voltage supplies must be powered at the same voltage and must be ramped up and ramped down at the same time.

For information about the power-up to functional time sequence, see DS0128: IGLOO2 and SmartFusion2 Datasheet.

SmartFusion2 and IGLOO2 FPGAs do not have internal brown-out detection and protection circuitry. As described in section 1.12. Brownout Detection (BOD) of this document, VDD of the FPGA must be protected from any brown-out conditions. You must follow one of the following recommendations during the power-down of the SmartFusion2/IGLOO2 FPGAs:

- The DEVRSTn pin of the FPGA is asserted (low) once VDD drops below the recommended operating levels.
- Or VPP rail is powered down before VDD. The VPP voltage level must drop to less than 1V before VDD can start its power-down.

If neither of the previous recommendations are followed during power-down, the FPGA device might experience malfunction that might result in partial or full erasure of user data in the eNVM.

1.2 I/O Glitch (Ask a Question)

Glitches were observed in SmartFusion2/IGOOL2 devices during power-up and power-down cases in different scenarios. The following sections describe the glitch observations.

1.2.1 I/O Glitch During Power-Up (Ask a Question)

I/O Glitches can occur in some power-up sequences, and they can be ignored if good design practices are used.

To mitigate the I/O glitch:

1. Use any one of the following power-up sequence:
   a. Ensure VDD/VDDIOx powers-up before VPP (VDD/VDDIOx → VPP). During this time, DEVRST_N should also be de-asserted along with VPP.
   b. Assert DEVRST_N (Keep signal Low) until all the power rails are up. Only after all power rails are up de-assert DEVRST_N.
2. If power sequencing is not possible, add a 10 KΩ resistor to ground on all critical signal outputs like clocks and resets.

1.2.2 I/O Glitch During Power-Down (Ask a Question)

I/O glitches can occur in some power-down sequences, and they can be ignored if good design practices are used. If the glitch on the I/Os during power-down of the FPGA cannot be ignored, the following might be used to mitigate the I/O glitch:

1. Add a 1 KΩ resistor to ground on all critical signal outputs such as clocks and resets and other glitch sensitive I/Os.
2. If possible, place the device in F*F mode prior to power-down.

1.2.3 I/O Glitch in a Blank Device (Ask a Question)

I/O glitch was observed on bank 2 of a blank device before programming. On a blank device, the I/Os are placed in the Flash*Freeze state (tristate with weak pull-ups). When the programming starts, the I/Os transition to the boundary scan mode. On I/O bank 2, there is a race condition between exiting the Flash*Freeze mode and the entering boundary scan mode. During this transition, the outputs on bank 2 briefly drive high until the boundary scan mode is enabled. This transition results in an I/O glitch.

To prevent this glitch, use the JTAG command to adjust the I/O drive strength to zero before programming starts.

1.2.3.1 Application Impact Due to Glitch (Ask a Question)

Application Impact: There is no reliability impact because the duration of the I/O glitch does not exceed the datasheet overshoot specifications. The glitch amplitude tracks the VDDI bank voltage and rises slightly above the VDDI. For example, at 3.3V of VDDI, the glitch rises above 3.3V by approximately 0.3V for 5 ns. The glitch amplitude is directly proportional to the VDDI value.

Resolution: Regenerate the bitstream using Libero v11.8 SP1.

1.2.4 I/O Glitch at Auto-Update During POR (Ask a Question)

I/O glitches are observed during POR with designs where the auto-update option is enabled. As the device powers up along with the DEVRSTb which is then in the state of being de-asserted, the I/O Glitch occurs at the beginning and end of the auto-update process.

The glitch is observed only on BANK 0, 1, and 2 of M2S005, M2S010, and M2S025 devices. The glitch is independent of the type of packages being used. This glitch is only in positive pulse (0 to 1 and then back to 0), therefore, only signals that are at logic low state will observe this glitch. The width of the glitch pulse is measured to be less than 10 ns.

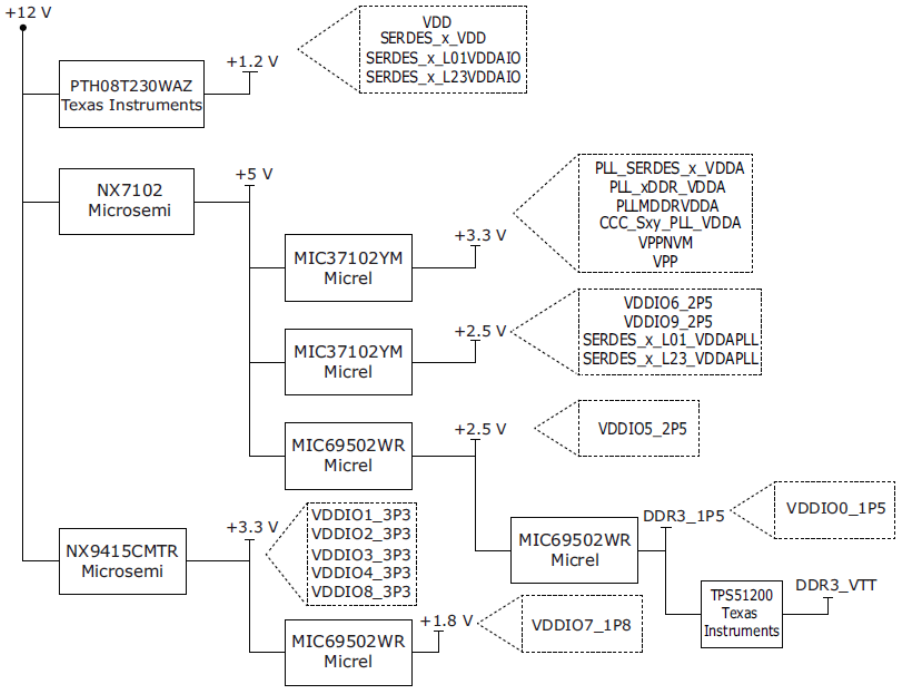
To avoid this glitch:

- Place critical signal like clocks and resets on IO BANK 4, 5, 6, and 7 across the dies M2S005, M2S010, and M2S025 devices.
- Use In-Application Programming (IAP) option instead of auto-update.
- No banks on M2S060, M2S090, M2S0150 experience any glitches.

1.2.5 Power Supply Flow (Ask a Question)

SmartFusion2/IGLOO2 FPGA devices require multiple power supplies. The following figure shows a topology for generating the required power supplies from a single 12V source. This example power supply topology is based on SmartFusion2 M2S050T-FG896 device with two SerDes channels (SERDES0 and SERDES1) and a DDR3 interface.

Figure 1-2. Example Power Supply Topology



1.2.6 Unused Pin Configurations (Ask a Question)

In cases where certain interfaces are not used, the associated pins need to be configured properly. For example, the pins of an unused crystal oscillator can be left floating (DNC) and must not be grounded. If a PLL is not used or bypassed, and only the divider circuitry is used, then the PLL's pins can be powered without RC filter circuitry.

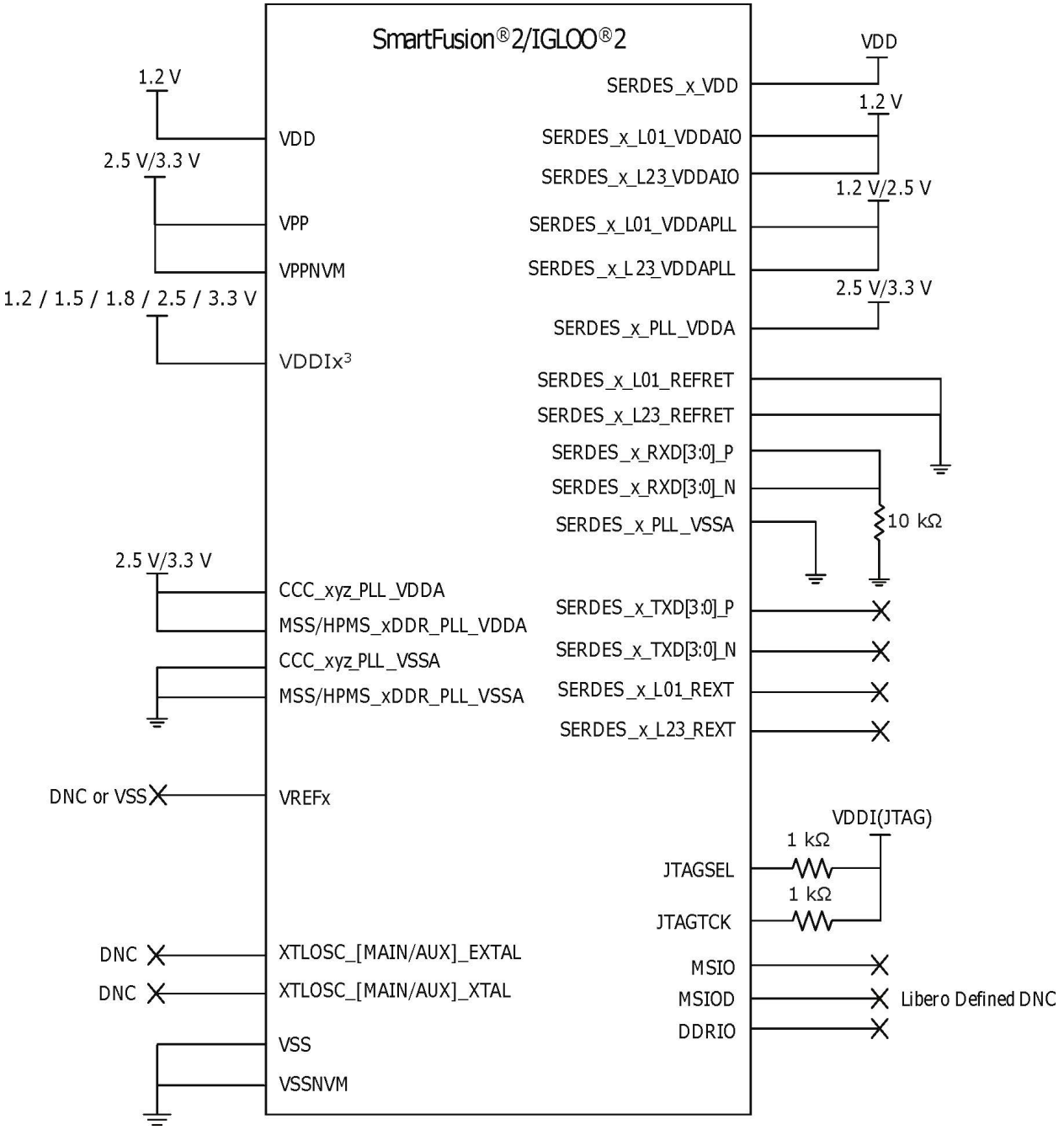
For SmartFusion2/IGLOO2 devices with multiple SerDes blocks, designers should tie off unused SerDes blocks, as shown in the following figure.

For banks configured as LPDDR or single-ended I/O (and MDDR or FDDR functionalities are not being used), VREFx can be left floating (DNC) even though the corresponding bank supply is still powered.

To allow a SmartFusion2/IGLOO2 device, to exit from reset, some of the bank supplies (VDDIx) must always be powered, even if associated bank I/Os are unused (as shown in Table 1-6 and Table 1-9).

For details on bank locations for all the devices, see DS0115: SmartFusion2 Pin Descriptions Datasheet or DS0124: IGLOO2 Pin Descriptions Datasheet.

Figure 1-3. Recommendations for Unused Pin Configurations[1, 2]



Important:

1. For M2S090T(S), M2GL090T(S), M2S150T(S), and M2GL150T(S) devices, the VPP and VPPNVM must be connected to a 3.3V supply.
2. SERDES_RXD pin connections are changed to VSS through a 10 kΩ resistor to reduce the latch-up risk. This change does not affect the old board design functionality.
3. There are some exceptions. For recommendations on unused VDDI supplies, see the following tables. The SmartFusion2/IGLOO2 devices have multiple bank supplies. In cases where specific banks are not used, connect them as listed in the following tables. If there is no recommendation provided for a device-bank supply combination, it means that the bank is not pinned out.

Table 1-4. Recommendation for Unused Bank Supplies for FC1152, FG896, FG676, FCS536, and FCV484 Packages

Bank Supply Names	FC1152	FG896	FG676		FCS536	FCV484
	M2S150T/ M2GL050T	M2S050T/ M2GL050T	M2S090T/ M2GL090T	M2S060T/ M2GL060T	M2S150T/ M2GL150T	M2S150T/ M2GL150
VDDI0	—	Connect to VSS through a 10 kΩ resistor			—	—
VDDI1	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI1	Connect to VSS through a 10 kΩ resistor.			
VDDI2	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Connect to VSS through a 10 kΩ resistor	
VDDI3	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI3	Must connect to VDDI3	Must connect to VDDI3	Connect to VSS through a 10 kΩ resistor	
VDDI4	Connect to VSS through a 10 kΩ resistor		—	Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor	
VDDI5	Connect to VSS through a 10 kΩ resistor					
VDDI6	Must connect to VDDI6	Connect to VSS through a 10 kΩ resistor			Must connect to VDDI6	Must connect to VDDI6
VDDI7	Must connect to VDDI7	Connect to VSS through a 10 kΩ resistor			Must connect to VDDI7	Must connect to VDDI7
VDDI8	Connect to VSS through a 10 kΩ resistor					
VDDI9	Connect to VSS through a 10 kΩ resistor		—	Connect to VSS through a 10 kΩ resistor		
VDDI10	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI11	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI12	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI13	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI14	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI15	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	—

.....continued						
Bank Supply Names	FC1152	FG896	FG676		FCS536	FCV484
	M2S150T/ M2GL050T	M2S050T/ M2GL050T	M2S090T/ M2GL090T	M2S060T/ M2GL060T	M2S150T/ M2GL150T	M2S150T/ M2GL150
VDDI16	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI17	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	
VDDI18	Connect to VSS through a 10 kΩ resistor	—	—	—	Connect to VSS through a 10 kΩ resistor	—

Important:

- The unused VDDI# pins must be connected to VSS through a 10 kΩ resistor. The VDDI# pins can be grouped and connected to one 10 kΩ resistor or a 10 kΩ resistor can be used for each VDDI# bank, it completely depends on the board layout. In the earlier version of the document, the unused VDDI# pins were recommended to be configured as DNC, and this cannot create functionality issue. Microchip recommends connecting to Ground through a 10 kΩ resistor to improve the board reliability.
- These I/O bank supplies are mandatory power supplies and must be powered up, even if associated bank I/Os are in unused condition.

Table 1-5. Recommendation for Unused Bank Supplies for FG484 Package

Bank Supply Names	FG484					
	M2S090T/ M2GL090T	M2S060T/ M2GL060T	M2S050T/ M2GL050T	M2S025T/ M2GL025T	M2S010T/ M2GL010T	M2S005/ M2GL005
VDDI0	—	—	Connect to VSS through a 10 kΩ resistor			
VDDI1	Connect to VSS through a 10 kΩ resistor		Must connect to VDDI1	Must connect to VDDI1	Must connect to VDDI1	Must connect to VDDI1
VDDI2	Must connect to VDDI2	Must connect to VDDI2	—	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2
VDDI3	Must connect to VDDI3	—	Must connect to VDDI3	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI3	Connect to VSS through a 10 kΩ resistor
VDDI4	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI4
VDDI5	Connect to VSS through a 10 kΩ resistor					
VDDI6	Connect to VSS through a 10 kΩ resistor					
VDDI7	Connect to VSS through a 10 kΩ resistor					—
VDDI8	Connect to VSS through a 10 kΩ resistor			—	—	—
VDDI9	—	Connect to VSS through a 10 kΩ resistor	—	—	—	—

Important:

- The unused VDDI# pins must be connected to VSS through a 10 kΩ resistor. The VDDI# pins can be grouped and connected to one 10 kΩ resistor or a 10 kΩ resistor can be used for each VDDI# bank, it completely depends on the board layout. In the earlier version of the document, the unused VDDI# pins were recommended to be configured as DNC, and this cannot create functionality issue. Microchip recommends connecting to Ground through a 10 kΩ resistor to improve the board reliability.
- These I/O bank supplies are mandatory power supplies and must be powered up, even if associated bank I/Os are in unused condition.

Table 1-6. Recommendation for Unused Bank Supplies for VF400 and FCS325 Packages

Bank Supply Names	VF400						FCS325		
	M2S060T/ M2GL060T	M2S050T/ M2GL050T	M2S025T/ M2GL025T	M2S010T/ M2GL010T	M2S005/ M2GL005	M2S090T/ M2GL090T	M2S060T/ M2GL060T	M2S050T/ M2GL050T	M2S025T/ M2GL025T
VDDI0	—	Connect to VSS through a 10 kΩ resistor				—	—	Connect to VSS through a 10 kΩ resistor	
VDDI1	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI1	Must connect to VDDI1	Must connect to VDDI1	Must connect to VDDI1	Connect to VSS through a 10 kΩ resistor		Must connect to VDDI1	Must connect to VDDI1
VDDI2	Must connect to VDDI2	—	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	—
VDDI3	—	Must connect to VDDI3	Connect to VSS through a 10 kΩ resistor			Must connect to VDDI3	Must connect to VDDI3	Must connect to VDDI3	Connect to VSS through a 10 kΩ resistor
VDDI4	Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor			Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI4	Connect to VSS through a 10 kΩ resistor	Must connect to VDDI4
VDDI5	Connect to VSS through a 10 kΩ resistor								
VDDI6	Connect to VSS through a 10 kΩ resistor								
VDDI7	Connect to VSS through a 10 kΩ resistor				—	Connect to VSS through a 10 kΩ resistor			
VDDI8	Connect to VSS through a 10 kΩ resistor		—	—	—	Connect to VSS through a 10 kΩ resistor			
VDDI9	Connect to VSS through a 10 kΩ resistor	—	—	—	—	—	Connect to VSS through a 10 kΩ resistor	—	—

Important:

- The unused VDDI# pins must be connected to VSS through a 10 kΩ resistor. The VDDI# pins can be grouped and connected to one 10 kΩ resistor or a 10 kΩ resistor can be used for each VDDI# bank, it completely depends on the board layout. In the earlier version of the document, the unused VDDI# pins were recommended to be configured as DNC, and this cannot create functionality issue. Microchip recommends connecting to Ground through a 10 kΩ resistor to improve the board reliability.
- These I/O bank supplies are mandatory power supplies and must be powered up, even if associated bank I/Os are in unused condition.

Table 1-7. Recommendation for Unused Bank Supplies for VF256 and TQ144 Packages

Bank Supply Names	VF256			TQ144	
	M2S025T/ M2GL025T	M2S010T/ M2GL010T	M2S005S/ M2GL005S	M2S010S/ M2GL010S	M2S005S/ M2GL005S
VDDI0	Connect to VSS through a 10 kΩ resistor				
VDDI1	Must connect to VDDI1	Must connect to VDDI1	Must connect to VDDI1	—	—
VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2	Must connect to VDDI2
VDDI3	Connect to VSS through a 10 kΩ resistor				
VDDI4	Must connect to VDDI4	Must connect to VDDI4	Must connect to VDDI4	Must connect to VDDI4	Must connect to VDDI4
VDDI5	Connect to VSS through a 10 kΩ resistor			—	Connect to VSS through a 10 kΩ resistor
VDDI6	Connect to VSS through a 10 kΩ resistor				
VDDI7	Connect to VSS through a 10 kΩ resistor		—	Connect to VSS through a 10 kΩ resistor	—

Important:

- The unused VDDI# pins must be connected to VSS through a 10 kΩ resistor. The VDDI# pins can be grouped and connected to one 10 kΩ resistor or a 10 kΩ resistor can be used for each VDDI# bank, it completely depends on the board layout. In the earlier version of the document, the unused VDDI# pins were recommended to be configured as DNC, and this cannot create functionality issue. Microchip recommends connecting to Ground through a 10 kΩ resistor to improve the board reliability.
- These I/O bank supplies are mandatory power supplies and must be powered up, even if associated bank I/Os are in unused condition.

1.3 Limiting VDD Surge Current (Ask a Question)

After device power-up, certain user-initiated functions can result in a surge current on VDD. This section describes how to minimize this surge current in SmartFusion2/IGLOO2 devices. Minimizing this surge current ensures that VDD voltage is maintained within the recommended operating range. This additional surge current does not occur during device power-up and has no effect on device reliability. If VDD voltage drops below the minimum recommended operating voltage, the FPGA may experience brownout. For more information about brownout and brownout prevention, see 1.12. Brownout Detection (BOD).

1.3.1 Device Reset Induced VDD Surge Current (Ask a Question)

After device power-up, if the application asserts the DEVRST_N pin, a surge current on VDD may be observed. This section describes how to minimize additional surge current during the device reset operation. This additional surge current does not occur during device power-up; it is applicable only when DEVRST_N is asserted.

SmartFusion 2/IGLOO 2 device reset can be activated either directly through an external DEVRST_N pin or indirectly through the tamper macro IP. When the device reset is asserted, the system controller immediately puts the FPGA core in the inactive state, which induces a temporary current demand on VDD. This surge current is for a very short duration and is normally handled by bulk decoupling capacitors on the power plane in a typical system. In cases where Microchip-recommended board design guidelines cannot be implemented for decoupling capacitors for VDD

(due to limited board spacing or other reasons), higher-than-expected surge current may occur during device reset.

The following table provides characterized surge current data for VDD during DEVRST_N assertion. This data represents the worst-case condition with no decoupling capacitors on the board.

Table 1-8. Surge Current on VDD during DEVRST_N Assertion (No Decoupling Capacitors on Board)

Device	Width of Surge at 50% of Pulse (µS)	Surge Current on VDD			Units
		0 °C to 85 °C	–40 °C to 100 °C	–55 °C to 125 °C	
M2S005/M2GL005	2	0.5	0.6	0.6	A
M2S010/M2GL010	3	0.9	0.9	0.9	A
M2S025/M2GL025	6	1.7	1.7	1.7	A
M2S050/M2GL050	12	3.2	3.2	3.2	A
M2S060/M2GL060	12	3.2	3.2	3.2	A
M2S090/M2GL090	22	4.4	4.6	4.6	A
M2S150/M2GL150	42	7.0	7.3	7.3	A

The surge current data in the preceding table does not represent a typical system. To illustrate this, surge current during device reset was measured at room temperature separately for the M2S090 security evaluation kit and the M2S150 advanced development kit. These kits have decoupling capacitors according to the Microchip recommended board design guidelines. The following table lists the surge currents observed on the M2S090 security evaluation kit and the M2S150 advanced development kit. The surge current values were found to be within acceptable limits.

Table 1-9. M2S090 and M2S150 Surge Current During DEVRST_N Assertion (With Decoupling Capacitors on Board)

Kit	Width of Surge at 50% of Pulse	Surge Current
M2S090 Security Evaluation Kit	5 µs	150 mA
M2S150 Advanced Development Kit	40 µs	1.5A

Note:

1. The amount of bulk capacitance placed for VDD was 1–100 µF, 3–220 µF, and 1–330 µF.

1.3.2 System Controller Suspend Mode Exit Induced VDD Surge Current (Ask a Question)

SmartFusion2 and IGLOO2 devices support system controller suspend mode for safety critical applications. Customers using this feature can force the system controller to exit suspend mode to support system controller features including device reprogramming. Exiting suspend mode is accomplished by driving the JTAG TRSTn pin high. Exiting the device from system controller suspend mode, after device is fully operational, temporarily disables the FPGA core which results in a current surge on VDD. This surge current is for a very short duration and is normally handled by bulk decoupling capacitors on the VDD power plane in a typical system. The magnitude of this surge current is similar to the surge resulting from a DEVRST_N assertion (Table 1-8).

The surge current can be kept within acceptable limits by adhering to VDD decoupling capacitors requirements per Microchip-recommended board design guidelines as shown in Table 1-9.

1.3.3 Digest Check Inducted VDD Surge Current (Ask a Question)

The digest check system service performs on-chip NVM data integrity check on SmartFusion2 devices. The use of system services by digest check may cause additional surge current on VDD. For more information on digest check service, see the UG0450: SmartFusion2 and IGLOO2 System Controller User Guide

The following table provides surge current data recorded for VDD when system services were being used by the digest check service. To limit surge current during digest check, follow the Microchip recommended board design guidelines.

Table 1-10. Surge Current on VDD During Digest Check Using System Services (No Decoupling Capacitors on Board)

Device	Width of Surge at 50% of Pulse (µS)	Surge Current on VDD			Units
		0 °C to 85 °C	–40 °C to 100 °C	–55 °C to 125 °C Units	
M2S005/M2GL005	12	0.2	0.2	0.2	A
M2S010/M2GL010	12	0.5	0.5	0.5	A
M2S025/M2GL025	13	0.6	0.6	0.6	A
M2S050/M2GL050	13	0.9	0.9	0.9	A
M2S060/M2GL060	13	0.9	0.9	0.9	A
M2S090/M2GL090	20	1.0	1.0	1.0	A
M2S150/M2GL150	26	1.0	1.0	1.0	A

1.4 Clocks (Ask a Question)

SmartFusion2 devices have two on-chip RC oscillators and up to two crystal oscillators for generating clocks for the on-chip resources and logic in the FPGA fabric, as listed in the following table.

RC Oscillators:

- 1 MHz RC oscillator
- 50 MHz RC oscillator

Crystal Oscillators:

- Main crystal oscillator
- Auxiliary (RTC) crystal oscillator

All IGLOO2 devices and the M2S050 SmartFusion2 device have a main crystal oscillator. They do not have an auxiliary (RTC) crystal oscillator.

Table 1-11. Clock Circuit

Resource		SmartFusion2 SoC Part Number					IGLOO2 Part Number				
		M2S005	M2S010	M2S025	M2S050	M2S150	M2GL005	M2GL010	M2GL025	M2GL050	M2GL150
	RC Oscillators	2	2	2	2	2	2	2	2	2	2
On-Chip Oscillators	Crystal Oscillators	2	2	2	1	2	1	1	1	1	1

1.4.1 Main Crystal Oscillator (Ask a Question)

The main crystal oscillator works with an external crystal, ceramic resonator, or a resistor-capacitor network to generate a high-precision clock in the range of 32 kHz to 20 MHz and is connected via the pins XTLOSC_[MAIN/AUX]_EXTAL and XTLOSC_[MAIN/AUX]_XTAL.

The following table lists the output frequency range of the main crystal oscillator with different possible sources.

Table 1-12. Crystal Oscillator Output Frequency Range

Source	Output Frequency Range
Crystal	32 kHz to 20 MHz
Ceramic Resonator	500 kHz to 4 MHz
R C Circuit	32 kHz to 4 MHz

The main crystal oscillator is operated in medium gain mode when a ceramic resonator is connected between the XTLOSC_[MAIN/AUX]_EXTAL and XTLOSC_[MAIN/AUX]_XTAL pins.

When a crystal is used, the load capacitance is determined by the external capacitors C1 and C2, internal capacitance, and stray capacitance (CS).

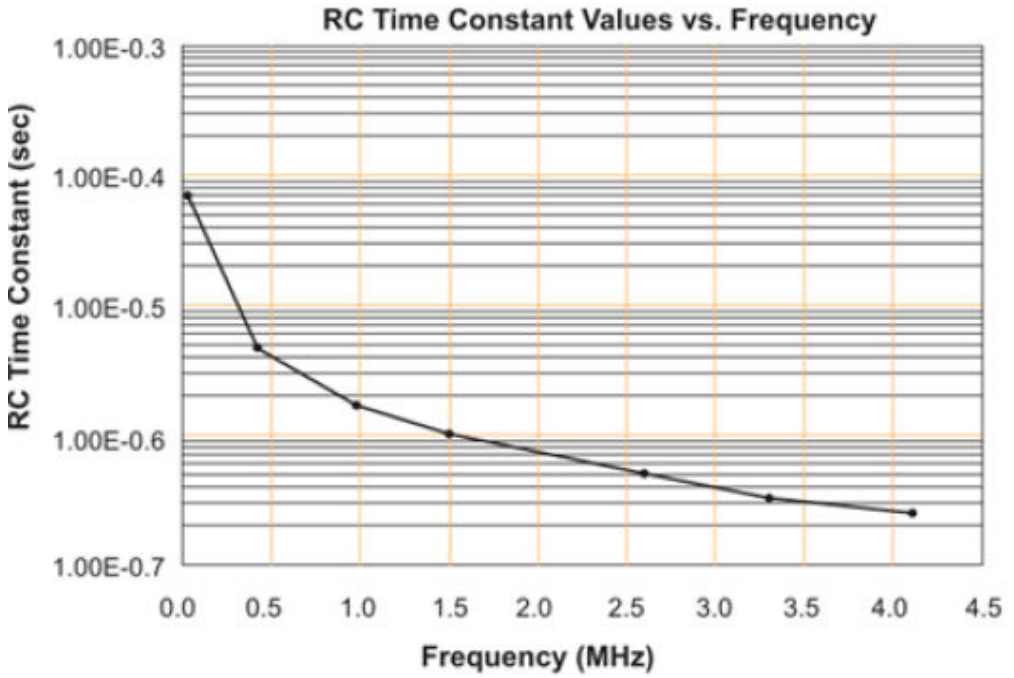
The oscillator and the load capacitance should be used as per the recommendation of the manufacturer.

Table 1-13. Suggested Crystal Oscillator

CRYSTAL 32.768 kHz 12.5 pF SMD	Citizen	CM519-32.768KEZF-UT

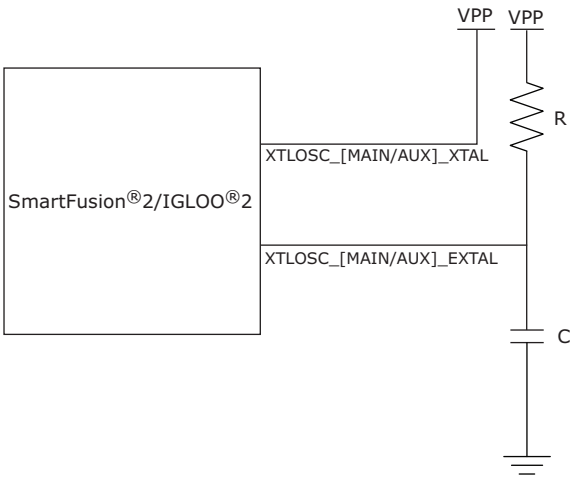
The frequency generated by an RC network is determined by the RC time constant of the selected components, as shown in the following figure.

Figure 1-4. RC Time Constant



The R and C components are connected to the XTLOSC_[MAIN/AUX]_EXTAL pin, with the XTLOSC_[MAIN/AUX]_XTAL pin connected to the power pin VPP, as shown in the following figure.

Figure 1-5. RC Oscillator



The operating mode of the main crystal oscillator is configured by the oscillator's macro available in Libero SoC.

1.4.2 Auxiliary (RTC) Crystal Oscillator (Ask a Question)

The SmartFusion 2 devices, except M2S050, have an auxiliary crystal oscillator dedicated to real-time clocking as an alternative source for the 32 kHz clock. The RTC can take its 32 kHz clock source from the auxiliary crystal oscillator when the main crystal oscillator is being used.

Similar to the main crystal oscillator, the auxiliary crystal oscillator can work with an external crystal, ceramic resonator, or an RC circuit to generate a high-precision clock in the range of 32 kHz to 20 MHz. There are two I/O pads for connecting the external frequency source to the auxiliary crystal oscillator: XTLOSC_AUX_EXTAL and XTLOSC_AUX_XTAL. The output frequency range, operating modes, and characteristics for the auxiliary crystal oscillator are the same as those for the main crystal oscillator.

For detailed information, see UG0449: SmartFusion2 and IGLOO2 Clocking Resources User Guide.

Important: Auxiliary (RTC) crystal oscillator is not available in the IGLOO 2 device.

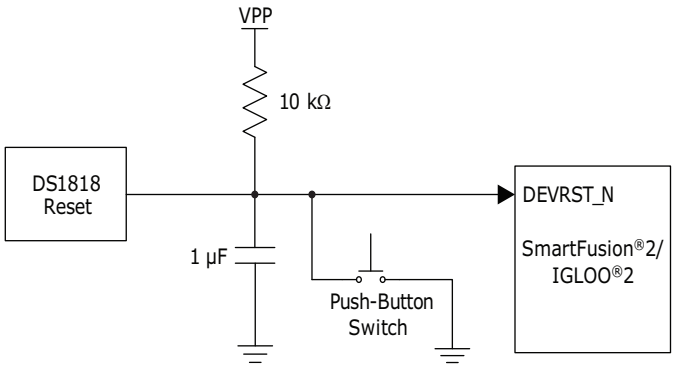
1.5 Reset Circuit (Ask a Question)

SmartFusion 2/IGLOO 2 devices have a dedicated asynchronous Schmitt-trigger reset input pin (DEVRST_N) with a maximum ramp time not more than 1 µs. This active-low signal must be asserted only when the device is unresponsive due to some unforeseen circumstances. It is not recommended to assert this pin during a programming (including eNVM) operation, as it may cause severe consequences including corruption of the device configuration. Asserting this signal tristates all user I/O and resets the system. Deasserting DEVRST_N enables the system controller to begin its startup sequence.

The following figure shows an example of a reset circuit using the Maxim DS1818 reset device, which maintains reset for 150 ms after the 3.3V supply returns to an in-tolerance condition. Adding a capacitor to ground on DEVRST_N avoids high-frequency noise and unwanted glitches that could reset the device.

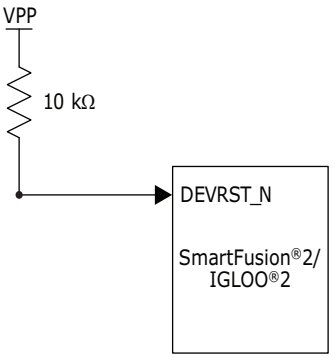
Important: Use DEVRST_N only for IAP or auto update. Do not use DEVRST_N for user logic reset.

Figure 1-6. Reset Circuit



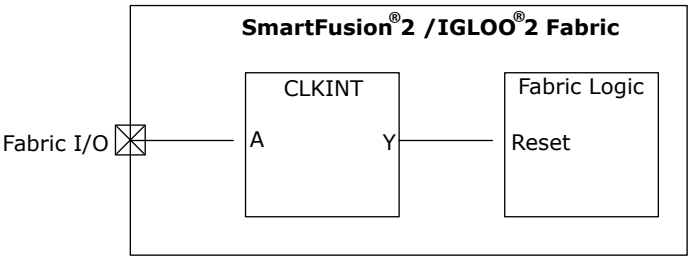
If the reset device is not used, DEVRST_N must be pulled up to VPP through a 10 kΩ resistor, as shown in the following figure.

Figure 1-7. Without Reset Circuit



If the user logic needs to be reset, any FPGA I/O can be used as an asynchronous reset for the user logic, as shown in the following figure.

Figure 1-8. Fabric Logic Reset



Use the fabric logic reset for CM3 Reset, fabric logic reset, MSS reset (including all peripherals), FDDR reset, and SerDes reset. For more information about fabric reset, see SmartFusion2 MSS DDR Controller Configuration Guide.

1.6 Device Programming (Ask a Question)

A SmartFusion 2/IGLOO 2 device are programmed through one of two dedicated interfaces: JTAG or SPI. These two interfaces support the following programming modes:

- Auto-programming (initiator) mode
- In-system programming:
  - JTAG programming mode
  - SPI Target programming mode
- In-application update:
  - Arm® Cortex®-M3 update mode (only for SmartFusion 2 devices)
  - Auto update mode

For detailed information about programming the device, see the UG0451: SmartFusion2 and IGLOO2 Programming User Guide.

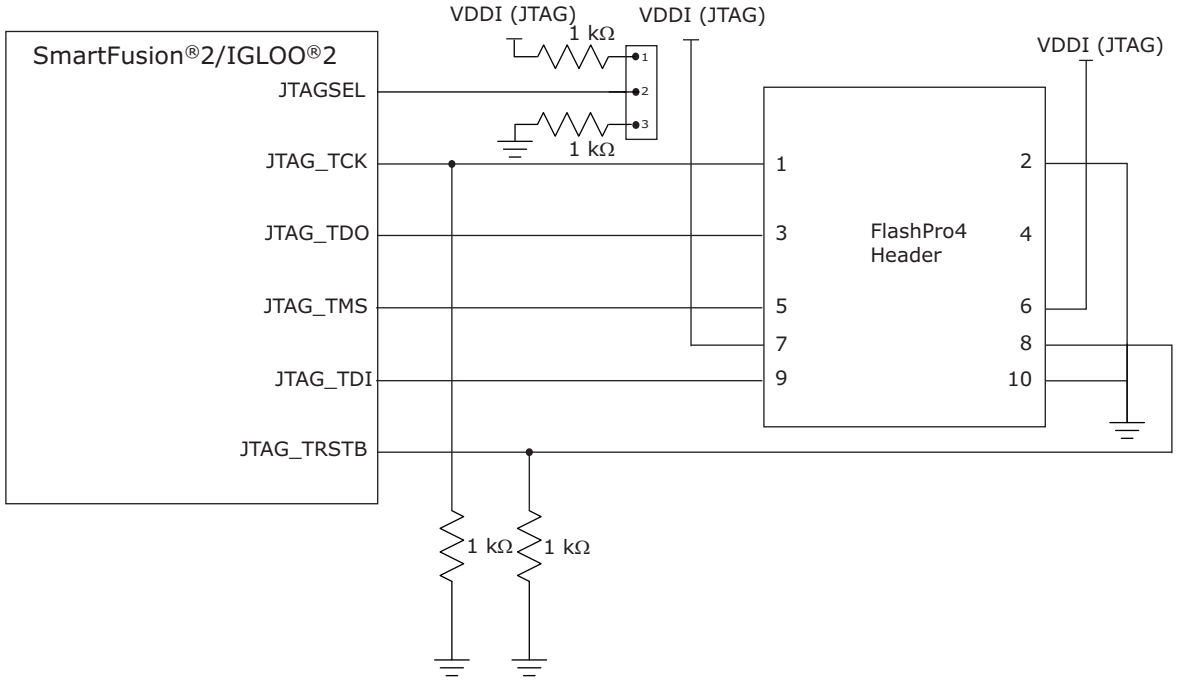
1.6.1 JTAG Programming (Ask a Question)

The JTAG interface is used for device programming and testing or for debugging the Arm Cortex-M3 firmware, as listed in the following table. These functions are enabled depending on the state of the JTAGSEL input. When the device reset is asserted, JTAG I/Os are still enabled but cannot be used as the TAP controller in the reset. JTAG I/O are powered by VDDI in the I/O bank where they reside. JTAG pins must be connected as shown in the following figure.

Table 1-14. JTAG Pins

Pin Names	Direction	Weak Pull-up	Description
JTAG_TMS	Input	Yes	JTAG test mode select.
JTAG_TRSTB	Input	Yes	JTAG test reset. Must be held low during device operation.
JTAG_TDI	Input	Yes	JTAG test data in.
JTAG_TCK	Input	No	JTAG test clock. Microchip recommends that TCK be tied to VSS or VDDI through a resistor on the board when unused per IEEE® 1532 requirements. This prevents totempole current on the input buffer.
JTAG_TDO	Output	No	JTAG test data out.
JTAGSEL	Input	Connect the JTAGSEL pin to an external pull-up resistor. The default configuration should enable the FPGA fabric TAP.	JTAG controller selection. Depending on the state of the JTAGSEL pin, an external JTAG controller connects to either the FPGA fabric TAP (high) or the Arm Cortex-M3 JTAG debug interface (low). For SmartFusion2-based designs, this signal must be held high or low through jumper settings. For IGLOO2-based designs, this signal must be held high through a pull-up resistor.

Figure 1-9. JTAG Programming



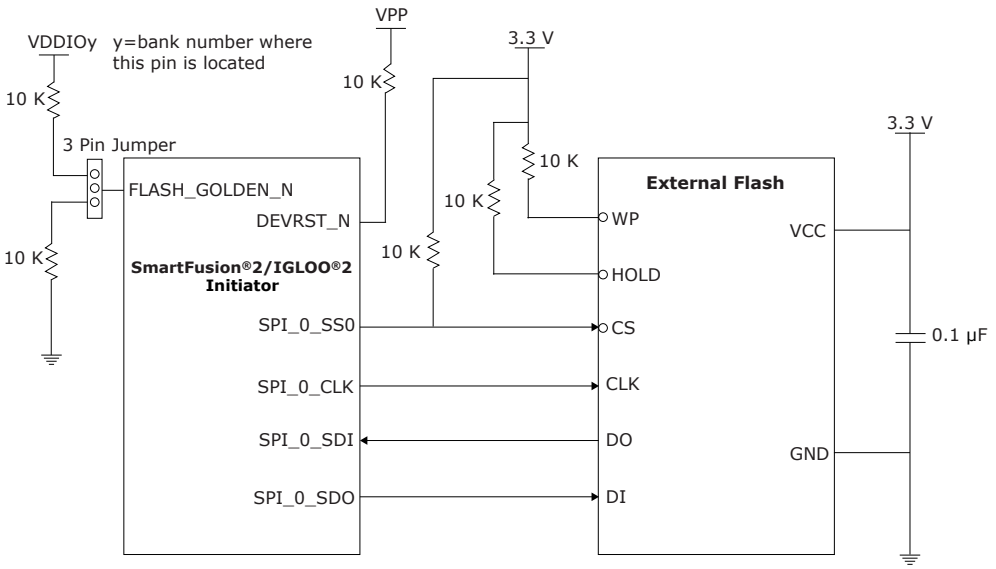
1.6.2 SPI Initiator Programming (Ask a Question)

The SmartFusion2/IGLOO2 devices have dedicated pins for programming the device and probing the fabric I/O.

The embedded system controller contains a dedicated SPI block for programming, which can operate in initiator or target mode. In initiator mode, the SmartFusion2/IGLOO2 devices interface with the external SPI flash from which programming data is downloaded. In target mode, the SPI block communicates with a remote device that initiates download of programming data to the device.

The following figure shows the board-level connectivity for SPI initiator mode programming in SmartFusion2 and IGLOO2 devices.

Figure 1-10. SPI Initiator Mode Programming



The following table lists the dedicated pins used for programming the device and probing the fabric I/O.

Table 1-15. Dedicated Pins

Pin Names	Direction	Description
SPI_0_SDI	Input	Serial data input
SPI_0_SDO	Output	Serial data input
SPI_0_CLK	Output	Serial clock. It is a serial programmable bit rate clock out signal.
SPI_0_SSO[1]	Output	Target select
FLASH_GOLDEN_N[1]	Input	If pulled low, the SPI_0 port is put into initiator mode, which indicates that the device is to be reprogrammed from an image in the external SPI flash attached to the SPI_0 interface.
NC	—	No connect. Indicates the pin is not connected to circuitry within the device. NC pins can be driven to any voltage or can be left floating with no effect on the operation of the device.
DNC	—	Do not connect. Must not be connected to any signals on the PCB. DNC pins must be left unconnected.

.....continued		
Pin Names	**Direction**	**Description**
PROBE_A	—	The two live probe I/O pins are dual-purpose: • Live probe functionality • User I/O If probe I/Os are reserved in Libero SoC, they will be configured as tristated outputs. It is recommended to add an external 10k pull-up resistor to each of these I/Os. If probe I/Os are unreserved in Libero SoC, they will be configured as a general purpose user I/O. If probe I/Os are unused in design, they will be configured as disabled input buffer or an output buffer tristated with a weak pull-up. The 10 kΩ external resistor power supply must be the same as the I/O bank power supply (VDDI). Ensure to power up the Bank where the Live Probe signals are assigned. That Bank may be different across die/package combination.
PROBE_B	—	

Note:

1. Active Low Signal.

For more information about programming, see UG0451: SmartFusion2 and IGLOO2 Programming User Guide.

Related Links

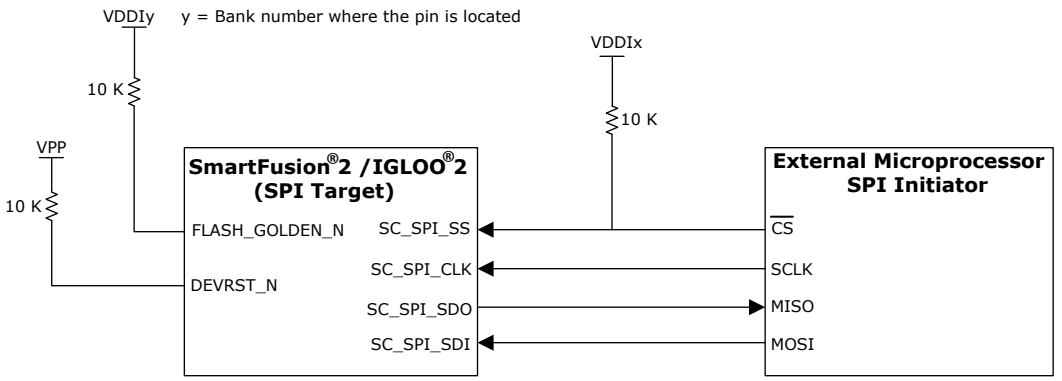
1. Active Low Signal.

1. Active Low Signal.

1.6.3 SPI Target Programming (Ask a Question)

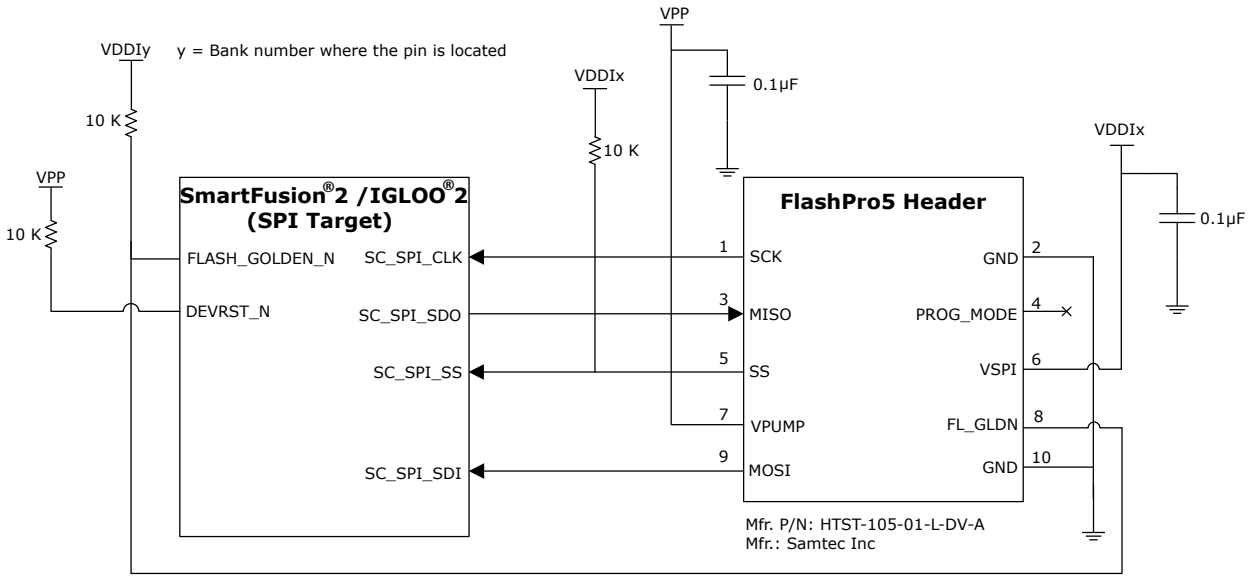
The following figure shows the SmartFusion2/IGLOO2 SPI target programming configuration when an external processor is the initiator.

Figure 1-11. SPI Target Programming by External Microprocessor



The following figure shows the SmartFusion2/IGLOO2 SPI target programming configuration when an external programmer is the initiator.

Figure 1-12. SPI Target Programming by External Programmer



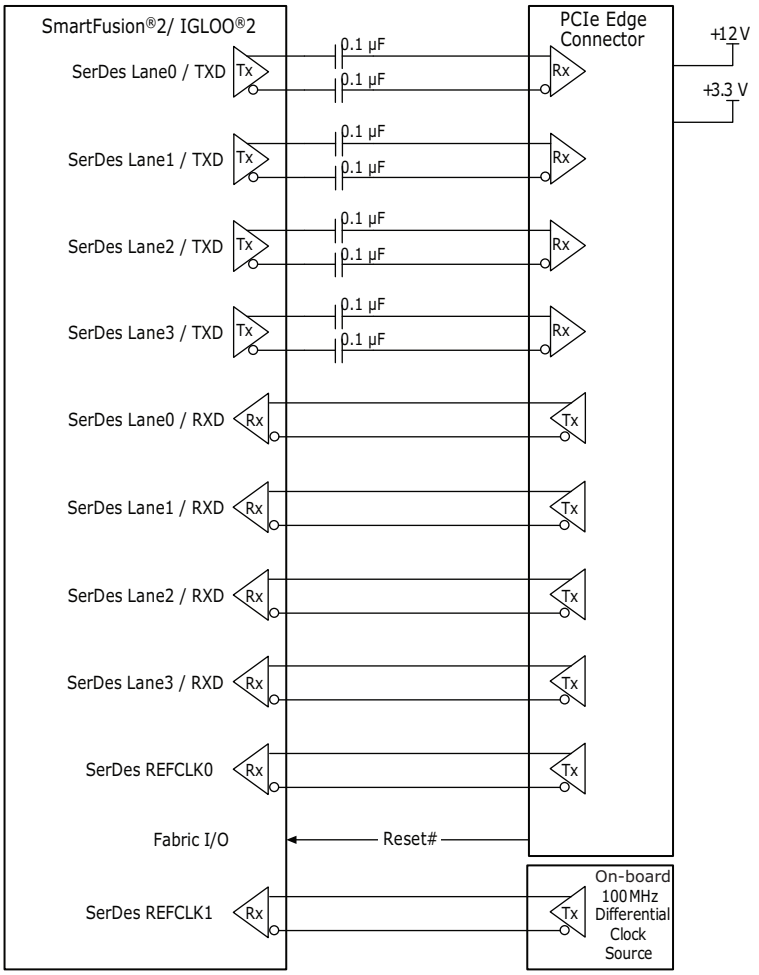
1.7 SerDes (Ask a Question)

SmartFusion2/IGLOO2 SerDes I/O reside in dedicated I/O banks. The number of SerDes I/O depends on the device size and pin count.

1.7.1 PCI Express (PCIe) (Ask a Question)

PCIe is a point-to-point serial differential low-voltage interconnect supporting up to four channels. Each lane consists of two pairs of differential signals: a transmit pair, TXP/TXN, and receive pair, RXP/RXN. The following figure shows the connectivity between the SmartFusion2/IGLOO2 SerDes interface and the PCIe edge connector.

Figure 1-13. SerDes Schematics



1.7.2 AC Coupling (Ask a Question)

Each transmit channel of a PCIe lane must be AC coupled to allow link detection.

For non-PCIe applications, the SmartFusion2/IGLOO2 device requires the receive inputs to be AC coupled to prevent common-mode mismatches between devices. Suitable values (for example, 0.1 µF) for AC-coupling capacitors must be used to maximize link signal quality and must conform to DS0128: IGLOO2 and SmartFusion2 Datasheet electrical specifications.

1.7.3 SerDes Reference Clock Requirements (Ask a Question)

The selection of the reference clock source or clock oscillator is driven by many parameters such as frequency range, output voltage swing, jitter (deterministic, random, and peak-to-peak), rise and fall times, supply voltage and current, noise specification, duty cycle and duty cycle tolerance, and frequency stability.

For SerDes reference clock pins, the internal ODT option must be enabled, and therefore, external termination is not required.

Following are the requirements for the SerDes reference clock:

- Must be within the range of 100 MHz to 160 MHz.
- Must be within the tolerance range of the I/O standard.
- The input clock for PCIe is typically a 100 MHz reference clock provided by the host slot for an end point device through the PCIe connector of the motherboard. If two components connected

through the PCIe bus use the same 100 MHz clock source, it is called common clock mode. In any other case, the PCIe device is in separated clock mode where one component either does not use a 100 MHz reference clock or uses a 100 MHz reference clock that does not have the same source and phase as the one used by the connected component.

See the *PCI Express Base specification Rev 2.1* for detailed PHY specifications. Also see the *PCIe Addin Card Electro-Mechanical (CEM) specifications*.

1.7.4 PLL Filter (Ask a Question)

To obtain a reasonable level of long-term jitter, it is vital to supply the PLL with analog-grade power. An RC or RLC filter is used, where C is composed of multiple devices to obtain a wide spectrum of noise absorption. Although the circuit is simple, its effectiveness depends on specific board layout requirements. See Figure 1-1 for a typical power supply connection.

- The DC series resistance of this filter must be limited. Limit the voltage drop across this device to less than 5% under worst-case conditions.
- Place a main ceramic or tantalum capacitor (see Figure 1-1), in the filter design to obtain good low-frequency cut-off. At least one low equivalent series inductance (ESL) and low ESR capacitor in parallel (~0.1 µF ceramic capacitor in 0402 package) enables the filter to maintain its attenuation through moderately high frequencies.
- The package ball grid array (BGA) pattern allows the placement of 0402 or 0201 components across the SERDES_x_Lyz_VDDAPLL and SERDES_x_Lyz_REFRET pins on the backside of the board.
- For the SerDes block, SERDES_x_Lyz_REFRET serves as the local on-chip ground return path for SERDES_x_Lyz_VDDAPLL. Therefore, the external board ground must not get shorted with SERDES_x_Lyz_REFRET under any circumstances.
- High-quality series inductors must not be used without a series resistor when there is a high-gain series resonator. Avoid using inductive chokes in any supply path unless care is taken to manage resonance.

See Figure 1-1 for SerDes analog power connections. A high-precision 1.2 KΩ, 1% resistor in either a 0402 or 0201 package is required for the external reference resistor connected between SERDES_x_Lyz_REXT and SERDES_x_Lyz_REFRET.

1.8 LPDDR, DDR2, and DDR3 (Ask a Question)

DDRIO is a multi-standard I/O buffer optimized for LPDDR, DDR2, and DDR3 performance. SmartFusion2/IGLOO2 devices include two DDR subsystems: the fabric DDR controllers (FDDR) and microcontroller subsystem (MSS) DDR (MDDR) controllers. All DDRIO can be configured as differential I/O or two single-ended I/O. DDRIO can be connected to the respective DDR sub-system PHYs or can be used as user I/O.

For more information on FDDR and MDDR, see the SmartFusion2 FPGA Fabric DDR Controller Configuration Guide and SmartFusion2 MSS DDR Controller Configuration Guide

The following table lists the differences between LPDDR, DDR2, and DDR3.

Table 1-16. LPDDR/DDR2/DDR3 Parameters

Parameter	LPDDR	DDR2	DDR3
VDDQ	1.8 V	1.8 V	1.5 V
VTT, VREF	—	0.9 V	0.75 V
Clock, address, and command (CAC) layout	Asymmetrical tree branch	Symmetrical tree branch	Daisy chained (fly-by)
Data strobe	Single-ended	Differential	Differential
ODT	None	Static	Dynamic

.....continued

Parameter	LPDDR	DDR2	DDR3
Match Addr/CMD/Ctrl to clock tightly	Yes	Yes	Yes
Match DQ/DM/DQS tightly	Yes	Yes	Yes
Match DQS to clock loosely	Yes	Yes	Not required
Interface	LVCMOS_18 or SSTL18 for LPDDR1	SSTL_18	SSTL_15
Impedance Calibration	LVCMOS18 - Not required SSTL18 - Required	150_1%	240_1%

A major difference between DDR2 and DDR3 SDRAM is the use of data leveling. To improve signal integrity and support higher frequency operations, a fly-by termination scheme is used with the clocks, command, and address bus signals. Fly-by termination reduces simultaneous switching noise by deliberately causing flight-time skew between the data strobes at every DDR3 chip. Fly-by termination requires controllers to compensate for this skew by adjusting the timing per byte lane. To obtain length matching, short TMATCH_OUT to TMATCH_IN with the shortest loop.

For more information about DDR memories, see the following documents:

- JESD209B-JEDEC STANDARD—Low Power Double Data Rate (LPDDR) SDRAM Standard
- JESD79-2F-JEDEC STANDARD—DDR2 SDRAM Specification
- JESD79-3F-JEDEC STANDARD—DDR3 SDRAM Standard

1.8.1 MDDR/FDDR Impedance Calibration (Ask a Question)

The MDDR and FDDR have a DDRIO calibration block. DDRIO can use fixed impedance calibration for different drive strengths, and these values can be programmed using the Libero SoC software for the selected I/O standard.

Before initiating DDRIO impedance calibration, either of the following must be performed.

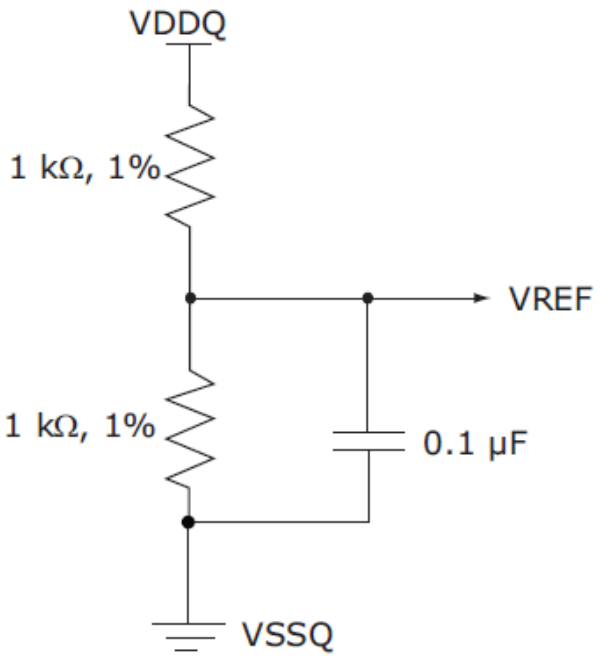
- Power sequencing, where the DDRIO bank VDDIx supply must be up and stable before VDD core supply.
- DDRIO re-calibration through the APB interface after DDRIO-VDDIx and VDD are up and stable.

For more information on impedance calibration, see the UG0445: SmartFusion2 SoC FPGA and IGLOO2 FPGA Fabric User Guide.

1.8.2 VREF Power (Ask a Question)

VREF is a low-power reference voltage equal to half of VDDQ. It must also be equal to VTT ± 40 mV. The following figure shows the VREF generation circuit.

Figure 1-14. VREF Generation



The following are the guidelines for connecting VREF power:

- For light loads (less than four DDR components), connect VDDQ to VSSQ through a simple resistor divider composed of two equivalent 1% 1 kΩ resistors.
- Generate a local VREF at every device, rather than generating a single VREF with one divider and routing it from the controller to the memory devices.
- Decouple at each device or connector to minimize noise.

Important: Use discrete resistors, not a resistor pack, to generate VREF.

1.8.3 VTT Power (Ask a Question)

VTT is memory bus termination voltage. To maintain noise margins, VTT must be equal to VDDQ/2, with an accuracy of ± 3%. VTT terminates command and address signals to VDDQ/2 using a parallel resistor (RT) tied to a low impedance source.

VTT is not used to terminate any DDR clock pairs. Rather, the xDDR_CLK and xDDR_CLK_N termination consists of a parallel 100-121 Ω resistor between the two lines.

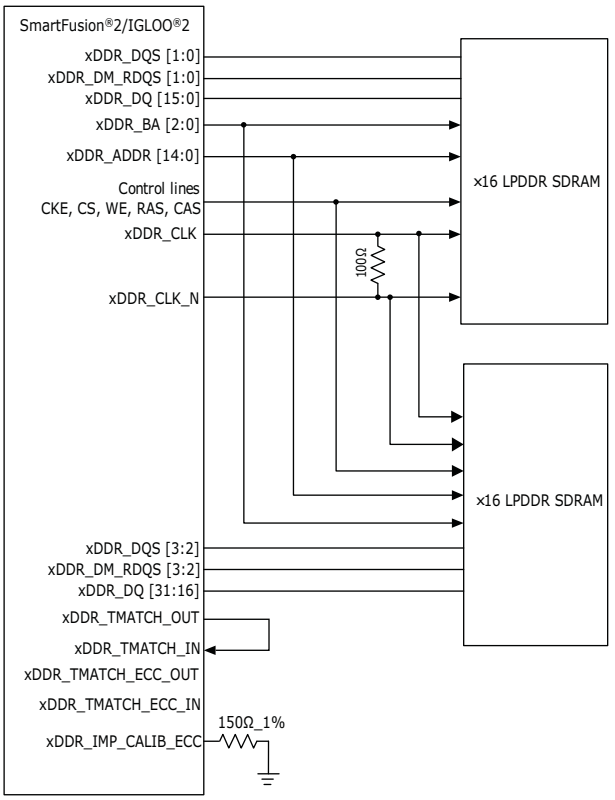
- VTT islands require a 10 µF capacitor.
- Since each data line is connected to VTT with relatively low impedance, this supply must be extremely stable. Any noise on this supply directly affects the data lines.
- Sufficient bulk and bypass capacitance must be provided to keep this supply at VDDQ/2. VREF power must not be derived from VTT, but must be derived from VDDQ with a 1% or better resistor divider.

1.8.4 LPDDR and DDR2 Design (Ask a Question)

The designer must be familiar with the specification and the basic electrical operation of the LPDDR/DDR2 interface. Data bus, data strobe, and data mask (byte enable) signals are point-to-point,

whereas all other address, control, and clock signals are not point-to-point. The following figures show the connectivity of the SmartFusion2/IGLOO2 LPDDR interface and a 32-bit DDR2 interface respectively.

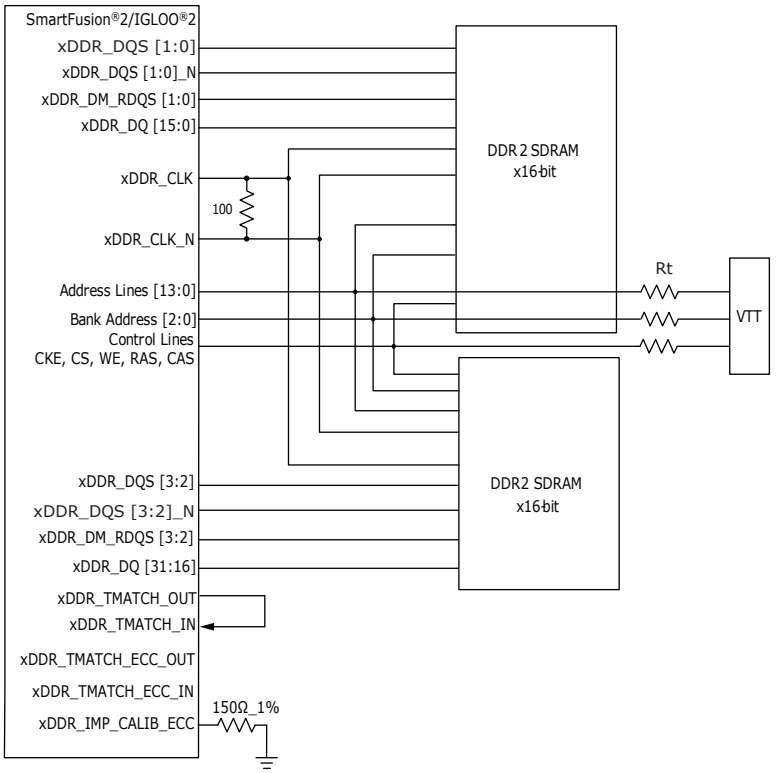
Figure 1-15. LPDDR Interface



Notes:

1. Impedance calibration is optional for LPDDR operating in LVCMOS mode and is required for LPDDR1 operating in SSTL18 mode.
2. For a 4-bit or 8-bit DRAM, all DQ pins are interchangeable. All 4-bit and 8-bit DQ pins are interchangeable in LPDDR, DDR2, and DDR3 memories. For a 16-bit DRAM, DQ0 through DQ7 are interchangeable. Also, DQ8 through DQ15 are interchangeable. However, DQ0-7 pins or signals must not be interchanged with the DQ8-15 pins or signals.
3. Short ECC_TMATCH_ OUT and ECC_TMATCH_ IN when using ECC bits.

Figure 1-16. DDR2 Interface



With short traces, the address, control, and command signals might not require both parallel (RT) and series (RS) termination. In a worst-case scenario, a small series resistor (RS) of about 10 Ω or less is required. This series termination is not used for impedance matching, but for dampening the signals.

Notes:

1. To get length matching, short the TMATCH_OUT to TMATCH_IN with the shortest loop.
2. Short ECC_TMATCH_ OUT and ECC_TMATCH_ IN when using ECC bits.

1.8.5 DDR3 Guidelines (Ask a Question)

The following are the guidelines for connecting to the DDR3 memory:

- DDR3 data nets have dynamic on-die termination (ODT) built into the controller and SDRAM. The configurations are 40 Ω, 60 Ω, and 140 Ω. VTT pull-up is not necessary.
- Characteristic impedance: Zo is typically 50 Ω, and Zdiff (differential) is 100 Ω.

DDR3 interfacing with SmartFusion2/IGLOO2 devices for 8-bit and 16-bit interfaces is shown in the following figures.

Figure 1-17. 8-Bit DDR3 Interface

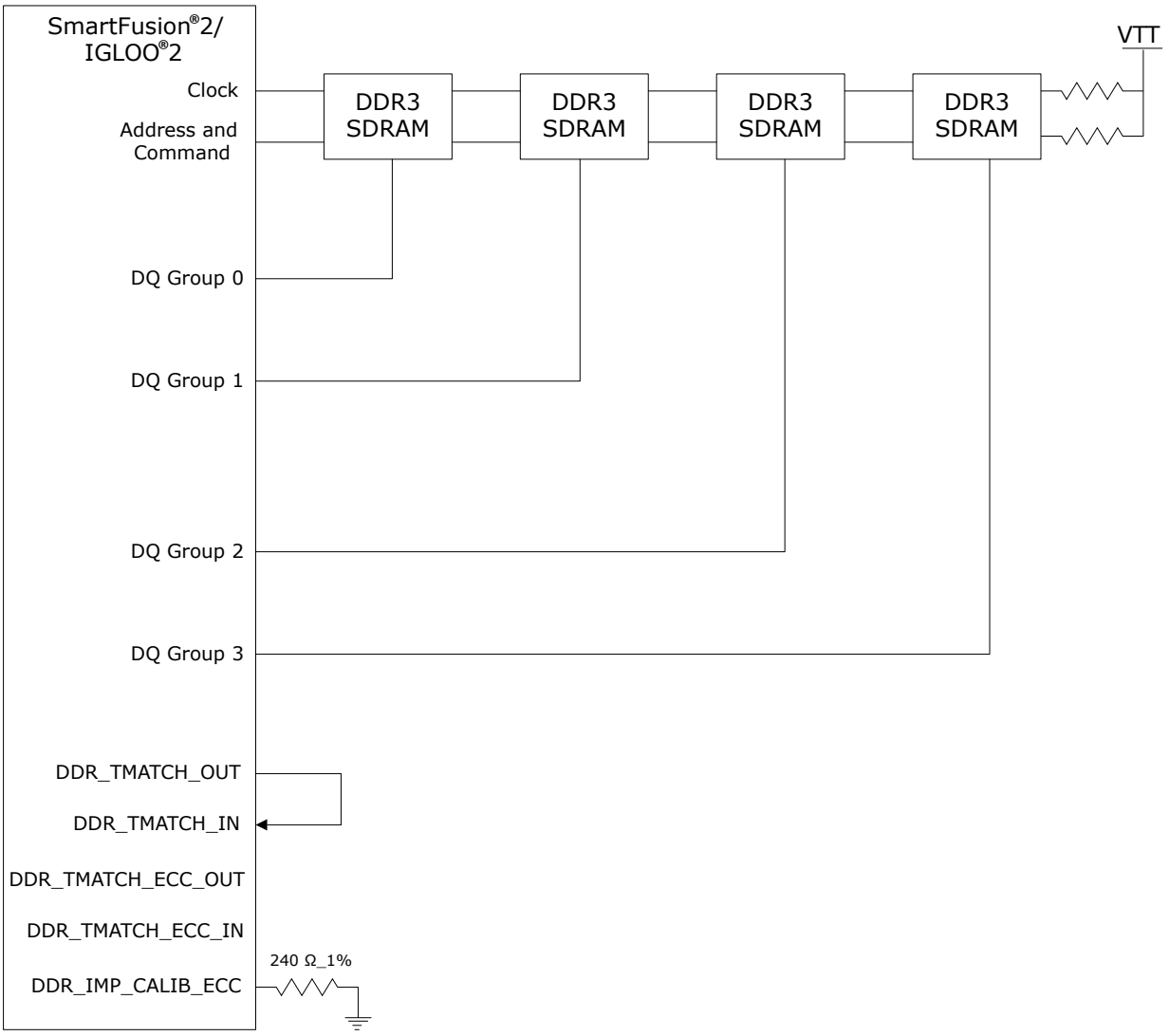
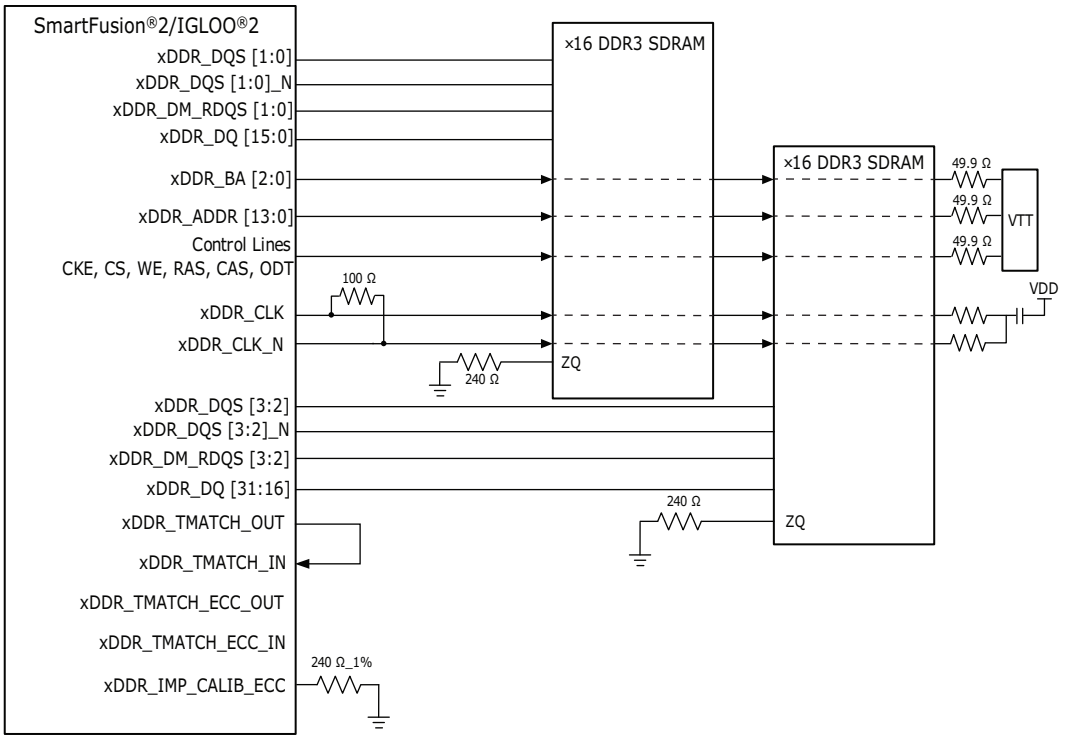


Figure 1-18. 16-Bit DDR3 Interface



Important: Short ECC_TMATCH_ OUT and ECC_TMATCH_ IN when using ECC bits.

1.9 User I/O and Clock Pins (Ask a Question)

The following table lists the unused I/O and clock pins in a SmartFusion2/IGLOO2 device.

Table 1-17. Recommendations for Unused I/O and Clock Pins

I/O	Unused Condition	Remarks
MSIO	Libero-Defined DNC[1]	Internal weak pull-up is available
MSIOD		
DDRIO		
Programming SPI pins		
Crystal oscillator pins	Must be left floating and must not connect to ground (VSS)	Internal weak nominal 50 kΩ pull-up to VPP

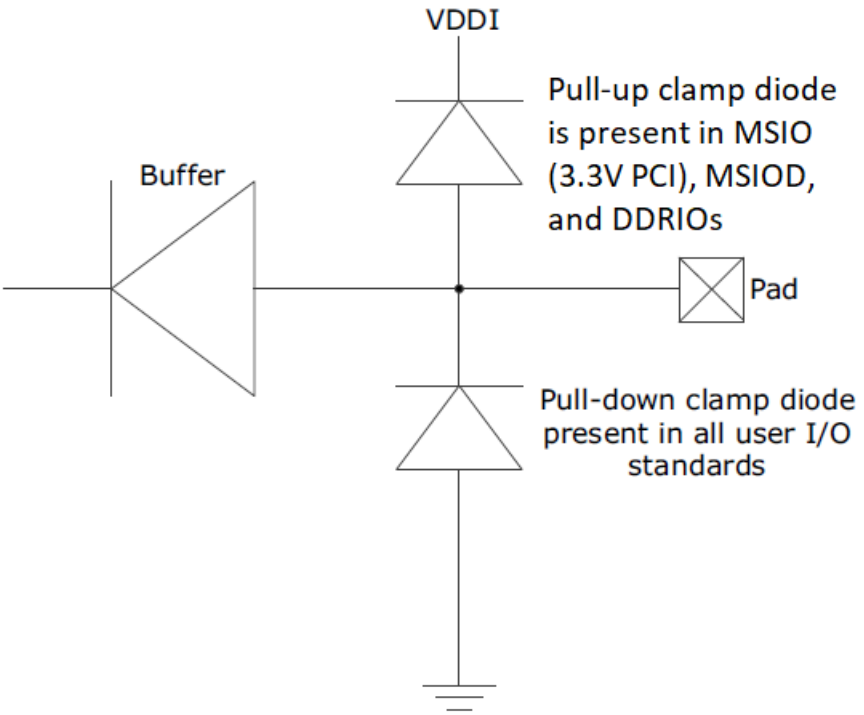
Note:

1. Libero configures unused user I/O (MSIO, MSIOD, and DDRIO) as: input buffer disabled, output buffer tristated with weak pull-up.

1.9.1 Internal Clamp Diode Circuitry (Ask a Question)

All I/O banks—MSIO, MSIOD, and DDRIO—have a clamp diode. The PCI 3.3V I/O standard requires a pull-up clamp diode which is available only in MSIO banks. If you use non-PCI I/O standards in MSIO banks, the pull-up diode clamp is disabled and is available for hot-swapping, which requires the pull-up diode clamp to be disabled. You cannot disable the pull-up clamp diode for the MSIOD and DDRIO banks for any I/O standard supported by these banks.

Figure 1-19. Internal Clamp Diode Control Circuitry



The following table lists the I/O banks that have clamp diode.

Table 1-18. I/O Banks with Clamp Diode

I/O Bank	I/O Standard	Clamp Diode
MSIO	PCI	Present
MSIO	All non-PCI standards	Not present
MSIOD	All I/O standards	Present
DDR	All I/O standards	Present

For more information about hot swapping and cold sparing applications, see AC396: SmartFusion2 and IGLOO2 in Hot Swapping and Cold Sparing Application Note.

1.10 Obtaining a Two-Rail Design for Non-SerDes Applications (Ask a Question)

SmartFusion 2/IGLOO 2 devices need multiple power supplies for functional operation, programming, and high-speed serial interfaces. It is possible to design an application with only two voltage rails using SmartFusion2/IGLOO2 devices.

I/O banks in SmartFusion2 and IGLOO2 devices support a wide range of I/O standards. I/O bank supplies can operate at 1.2V, 1.5V, 1.8V, 2.5V, or 3.3V. To obtain a two-voltage-rail design, the core voltage must be connected to 1.2V, and the mandatory I/O bank supplies and VPP supplies can be connected to 2.5V or 3.3V.

1.10.1 Operating Voltage Rails (Ask a Question)

SmartFusion2/IGLOO2 devices need 1.2V for the core supply and either 2.5V or 3.3V for I/O and analog supplies. The following table lists operating voltage requirements for the devices.

Table 1-19. Operating Voltage Rails

Pin Name	Description	Operating Voltage
VDD	DC core supply voltage	1.2V

.....continued		
Pin Name	**Description**	**Operating Voltage**
VDDIx[1]	I/O bank supply	1.2V, 1.5V, 1.8V, 2.5V, or 3.3V
SERDES_x_VDD	PCIe/PCS supply	1.2V
SERDES_x_L[01/23]_VDDAIO	Tx/Rx analog I/O voltage. Low-voltage power for lanes 0, 1, 2, and 3 of the SerDes interface	1.2V
VPP[2]	Power supply for charge pump	2.5V or 3.3V
VPPNVM[1]	Analog sense-circuit supply for the embedded nonvolatile memory (eNVM).	2.5V or 3.3V
CCC_xyz_PLL_VDDA	Analog power pad for CCC PLL	2.5V or 3.3V
MSS/HPMS_xDDR_PLL_VDDA	Analog power pad for xDDR PLL	2.5V or 3.3V
SERDES_x_PLL_VDDA	High supply voltage for SerDes PLL	2.5V or 3.3V
SERDES_x_L[01/23]_VDDAPLL	Analog power for SerDes PLL of lanes 0, 1, 2, and 3	2.5V

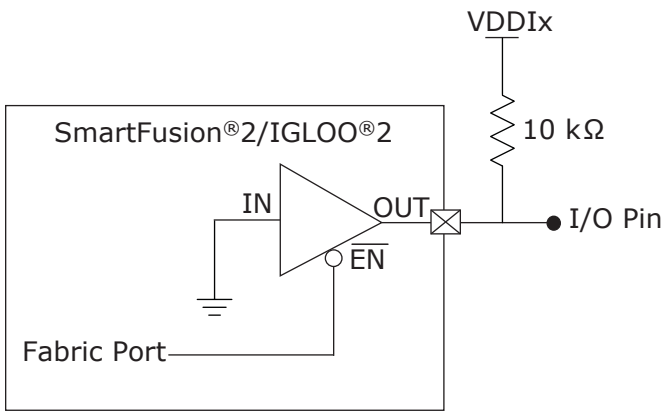
Notes:

1. The 3.3 V supply can be connected to MSIO VDDIx bank only.
2. For M2S090T(S), M2S150T(S) devices, VPP, and VPPNVM must be connected to 3.3 V.

1.11 Configuring Pins in Open Drain (Ask a Question)

To configure fabric pins in open-drain mode, the input port of the tristate buffer must be tied low, and the enable port of the buffer must be driven from the user logic via the fabric port, as shown in the following figure. For more information about the bank supplies, see Table 1-4, Table 1-7, and Table 1-17.

Figure 1-20. Configuring Pins in Open Drain



The following table lists the truth table for configuring pins in open-drain mode.

Table 1-20. Truth Table

Buffer Enable Port	Buffer In Port	Buffer Out Port
0 (low)	0 (low)	0 (low)
1 (high)	0 (low)	$VDDI_X$

1.12 Brownout Detection (BOD) (Ask a Question)

SmartFusion2/IGLOO2 functionality is guaranteed only if VDD is above the recommended level specified in the datasheet. Brownout occurs when VDD drops below the minimum recommended operating voltage. As a result, it is not possible to ensure proper or predictable device operation. The design might continue to malfunction even after the supply is brought back to the

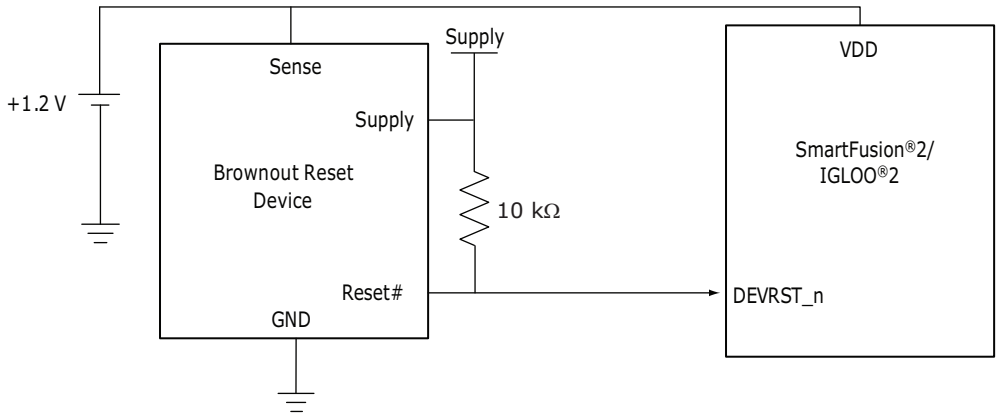
recommended values, as parts of the device might have lost functionality during brownout. The VDD supply must be protected by a brownout detection circuit.

To recover from VDD brownout, the device must either be power-cycled, or an external brownout detection circuit must be used to reset the device for correct operation. The recommended guideline for the threshold voltage of brownout detection is mentioned in DS0128: IGLOO2 and SmartFusion2 Datasheet. The brownout detection circuit must be designed such that when the VDD falls below the recommended voltage mentioned in the datasheet, the device is held in power-down mode via the DEVRST_N pin.

Important: Brownout detection must be implemented through a standalone circuit or included as part of power management circuitry.

SmartFusion2/IGLOO2 devices do not have a built-in brownout detection circuitry, but an external brownout detection circuitry can be implemented as shown in the following figure.

Figure 1-21. BOD Circuit Implementation



The BOD device must have an open-drain output to connect to VPP through a 10 kΩ resistor externally. During power-on, the brownout reset keeps the device powered down until the supply voltage reaches the threshold value. Thereafter, the brownout reset device monitors VDD and keeps RESET# output active as long as VDD remains below the threshold voltage. An internal timer delays the return of the output to the inactive state (high) to ensure proper system reset.

The delay time is in milliseconds and starts after VDD has risen above the threshold voltage. When the supply voltage drops below the threshold voltage, the output becomes active (low) again.

1.13 Simultaneous Switching Noise (Ask a Question)

When multiple output drivers switch simultaneously, they induce a voltage drop in the chip or package power distribution. The simultaneous switching momentarily raises the ground voltage within the device relative to the system ground. This apparent shift in the ground potential to a non-zero value is known as simultaneous switching noise (SSN) or ground bounce.

For SSO guidelines for SmartFusion2 and IGLOO2 I/Os, see UG0445: SmartFusion2 SoC FPGA and IGLOO2 FPGA Fabric User Guide.

2. Layout Guidelines for SmartFusion 2 and IGLOO 2-Based Board Design (Ask a Question)

This section provides guidelines for the hardware board layout that incorporates SmartFusion 2 SoC FPGA or IGLOO 2 FPGA devices. Good board layout practices are required to achieve the expected performance from the printed circuit boards (PCB) and SmartFusion 2/IGLOO 2 devices. These are essential to achieve high quality and reliable results such as low-noise levels, signal integrity, impedance, and power requirements. The guidelines mentioned in this document act as a supplement to the standard board-level layout practices.

Understanding of the SmartFusion 2/IGLOO 2 chip, experience in digital and analog board layout, and knowledge of transmission line theory and signal integrity is needed. For more information about the recommended guidelines for designing SmartFusion 2/IGLOO 2-based boards, see 1. Design Considerations.

Important: The target impedance calculated in this document is with respect to the development board. The simulations show the impedance that meets the target impedance of the development board. The target impedance depends on the logic implemented on SmartFusion 2/IGLOO 2. Therefore, calculate the target impedance of the board.

2.1 Power Supply (Ask a Question)

In power supply design, the target impedance of the power planes must be known. The target impedance varies based on the design. This helps in planning the requirement of the number of decoupling capacitors based on the target impedance. The number of decoupling capacitors varies based on the design.

Complex FPGA designs have increasing amounts of current transients switching across the power bus. Simultaneously switching outputs (SSO) contribute a major share of instantaneous current issues. Decoupling is necessary to prevent the instantaneous currents. Decoupling is only effective when inductance is minimized. Low inductance decoupling provides localized high frequency energy to decouple noise from the switching currents of the device power bus. This is most effective when capacitors are in close proximity to the device. Some of these high-frequency de-coupling capacitors must be placed directly under the FPGA or on single side. These capacitors must be placed close to the power and ground pins of the device and routed with thick trace.

To calculate the number of decoupling capacitors, it is important to know the target impedance of the power plane. Target impedance is calculated as follows:

$$Z_{Max} = \%\ \text{Ripple} \times \frac{V_{supply}}{I_{trans}}$$

Where,

V_{supply}: Supply voltage of the power plane.

% Ripple: Percentage of ripples that is allowed on the power plane. See DS0128: IGLOO2 and SmartFusion2 Datasheet for more information about ripple in Recommended Operating Conditions table.

I_{trans}: Transient current drawn on the power plane. The transient current is half of the maximum current. Maximum current is taken from the power calculator sheet.

Z_{max}: Target impedance of the plane.

Subsequent sections of this document, display simulation results based on target impedance calculated using preceding equations. Microchip strongly recommends calculating the target impedance and performing simulations for the impedance profile of the power plane. These simulations help in optimizing the decoupling capacitors to reduce the production cost and have the optimal placement. The plane shapes in this document are according to the UG0557: SmartFusion2 SoC FPGA Advanced Development Kit User Guide. This might vary depending on the design. For simulation topology, see Appendix D: Power Integrity Simulation Topology.

SmartFusion2/IGLOO2 power supplies are classified as:

- Core power supply
- I/O power supply
- Serializer/deserializer (SerDes) power supply
- Double data rate (DDR) power supply
- Phase-locked loop (PLL) power supply

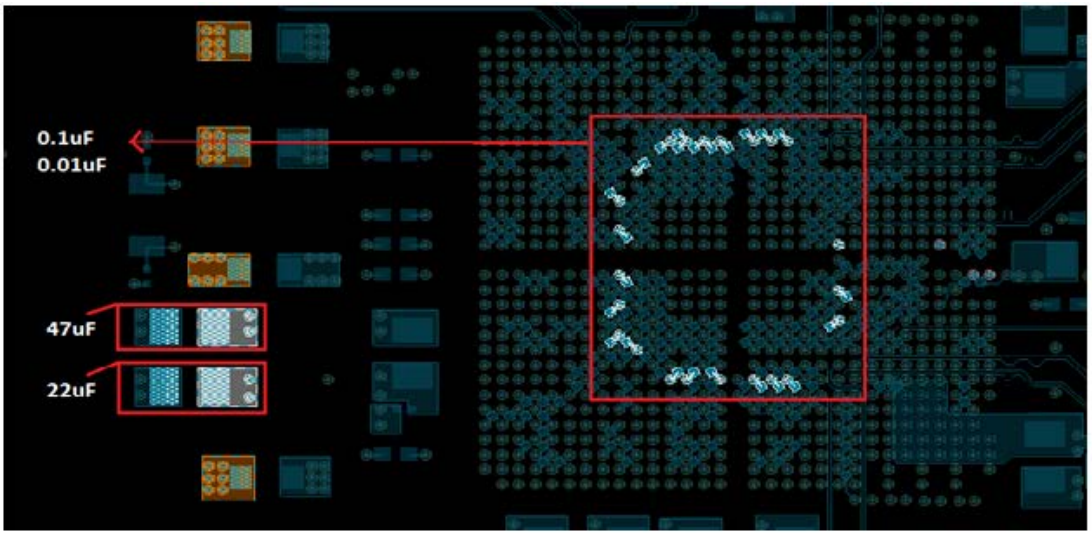
2.2 Core Supply (VDD) (Ask a Question)

The core power supply must have a low-noise and low-ripple voltages, as per datasheet. Proper care must be taken while designing the power supply (VDD) for core. Proper placement of decoupling capacitors and plane geometry greatly influences the power supply distribution going into SmartFusion2/IGLOO2 device.

2.2.1 Component Placement (Ask a Question)

- The bulk capacitors (330 µF and 100 µF) must be placed near by the SmartFusion2/IGLOO2 device.
- The bypass capacitors (47 µF and 22 µF) must be placed near or if possible, on the periphery of the device. The placement on the SmartFusion2 Development Kit board is shown in the following figure.

Figure 2-1. Placement of Capacitors for VDD Plane



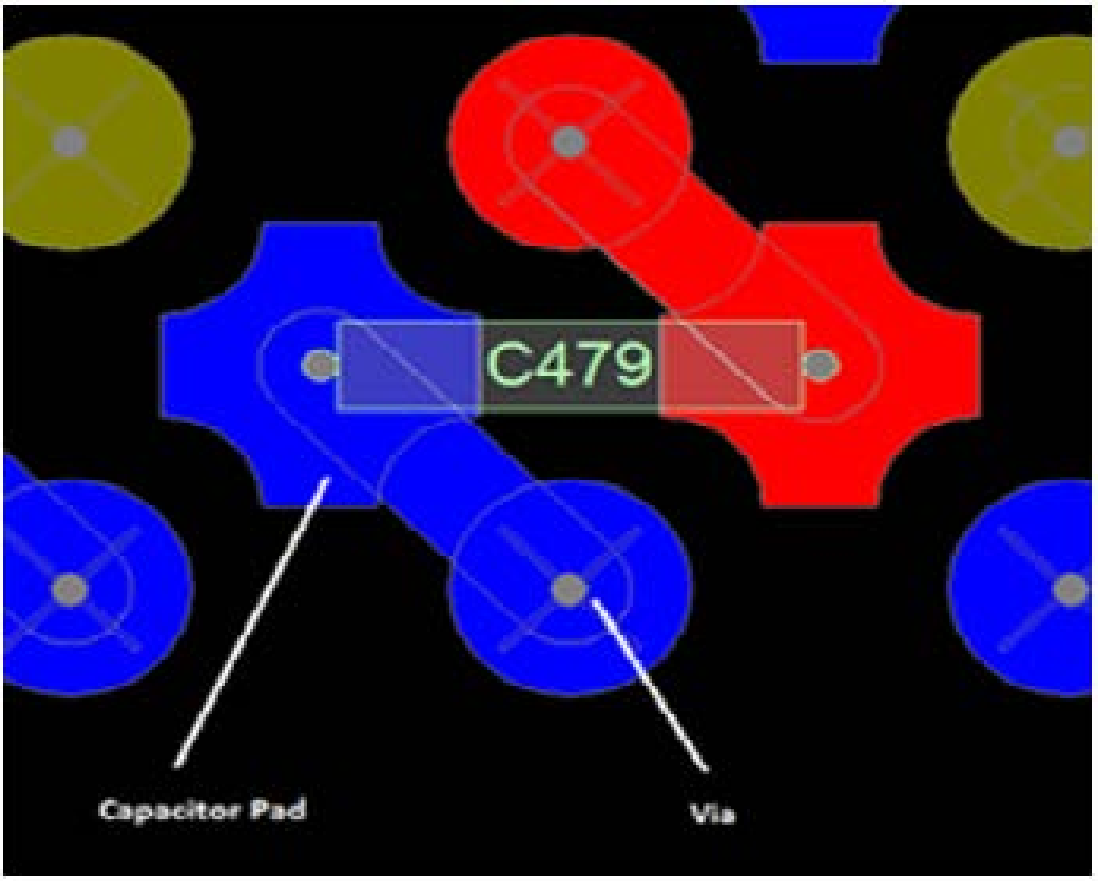
- All decoupling capacitors (0.1 µF and 0.01 µF) must be 0402 or of a smaller package size, as they must be mounted on the back side of the board. They must fit between the adjacent vias of ball grid array (BGA) package pins. These decoupling capacitors are selected to have a low impedance over operating frequency and temperature range. Capacitor pad to via trace must be as small as possible. The following figure shows how these capacitors need to be mounted. Keep

the capacitor pad directly on the corresponding vias. The capacitors must not share ground vias. Each decoupling capacitor must have its own via connection to the PCB ground plane.

- The Decoupling capacitor and the Smart Fusion2/IGOOL2 device can be placed side-by-side. If placed side-by-side, route the power with thick traces.

Important: Microchip does not guarantee on noise on power rails. User must run the power simulation.

Figure 2-2. Capacitor Placement under BGA Vias



2.2.2 Plane Layout (Ask a Question)

Use the VDD plane, as shown in the following figure.

Important: The plane can be routed in multiple ways to have dedicated and low-impedance plane.

Figure 2-3. VDD Plane

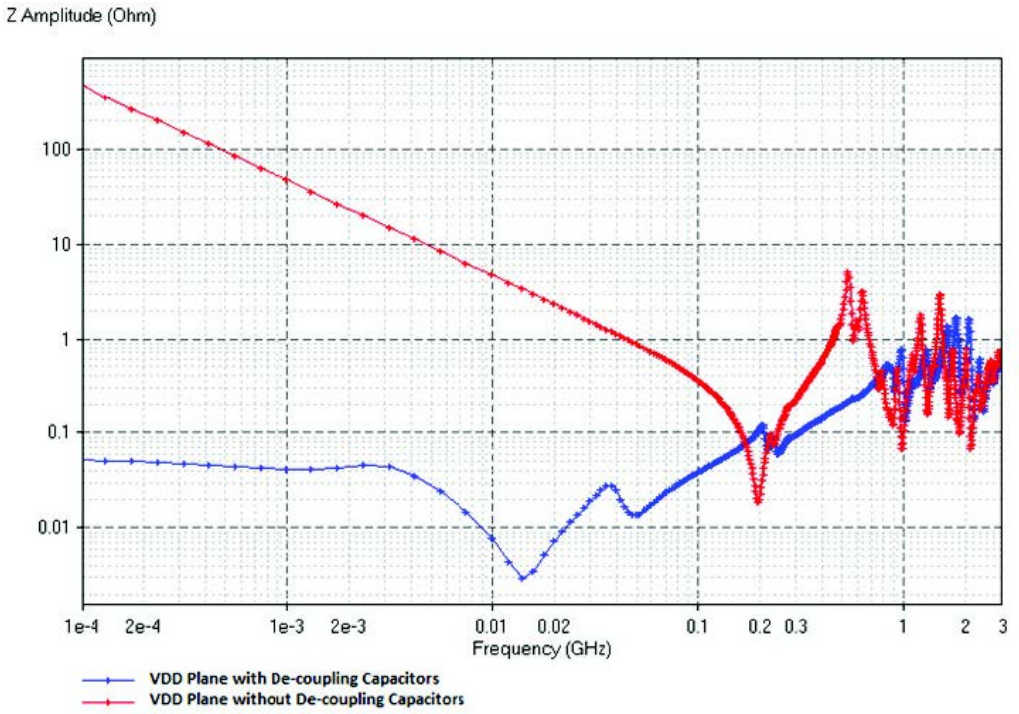
2.2.3 Simulations (Ask a Question)

The effect of the decoupling capacitors can be visualized through the power integrity simulations. The target impedance of the VDD is calculated as 40 mΩ, based on the following values:

- V_{SUPPLY} = 1.2 V
- I_{trans} = 1.5 A
- Ripple = 5%

The following figure shows the impedance profile of the VDD plane of the SmartFusion2 Development Kit. It shows that the capacitors used are adequate to improve the impedance profile over the bandwidth. Good coupling between the planes can be achieved by having power and ground plane in adjacent layers. Once all the capacitors (0.1 µF and 0.01 µF) are placed, the impedance of the VDD plane impedance profile improves over the frequency range. The simulation results shown in this document are done in Sigrity PowerSI tool. For more information on simulation, see the *Sigrity PowerSI Tutorial*.

Figure 2-4. Impedance Profile of VDD Plane with Respect to Frequency



2.3 SerDes (Ask a Question)

PCB designers overlook the requirement of isolating the noise generated by the digital components with the SerDes high-speed designs. Provide a low-noise supply for the sensitive analog portions of the SerDes devices. Noise due to various power supply voltages can be coupled into the analog portion of the chip and may produce unwanted fluctuations in the sensitive stages of the device. The performance of SerDes depends on robust layout techniques. This section discusses the layout guidelines for power supply for the SerDes and the SerDes PLL.

2.3.1 Component Placement (Ask a Question)

2.3.1.1 SerDes Core Power (SERDES_x_VDD) (Ask a Question)

- All decoupling capacitors (0.1 µF and 0.01 µF) are placed on the pad adjacent to the BGA via of the corresponding pin, as shown in Figure 2-2. The capacitor pad to via trace must be as small as possible. At least one 0.1 µF and one 0.01 µF capacitors must be placed for each SerDes bank.
- The bypass capacitor (10 µF) must be placed at the edge of the integrated circuit (IC).

2.3.1.2 SerDes I/O Power (SERDES_x_VDDAIO) (Ask a Question)

- All decoupling capacitors (0.1 µF and 0.01 µF) are placed on the pad adjacent to the BGA via of the corresponding pin, as shown in Figure 2-1. At least one of the capacitors (0.1 µF and 0.01 µF) must be placed for each SerDes bank. The capacitor pad to via trace must be small.
- The bypass capacitor (10 µF) must be placed at the edge of the IC.

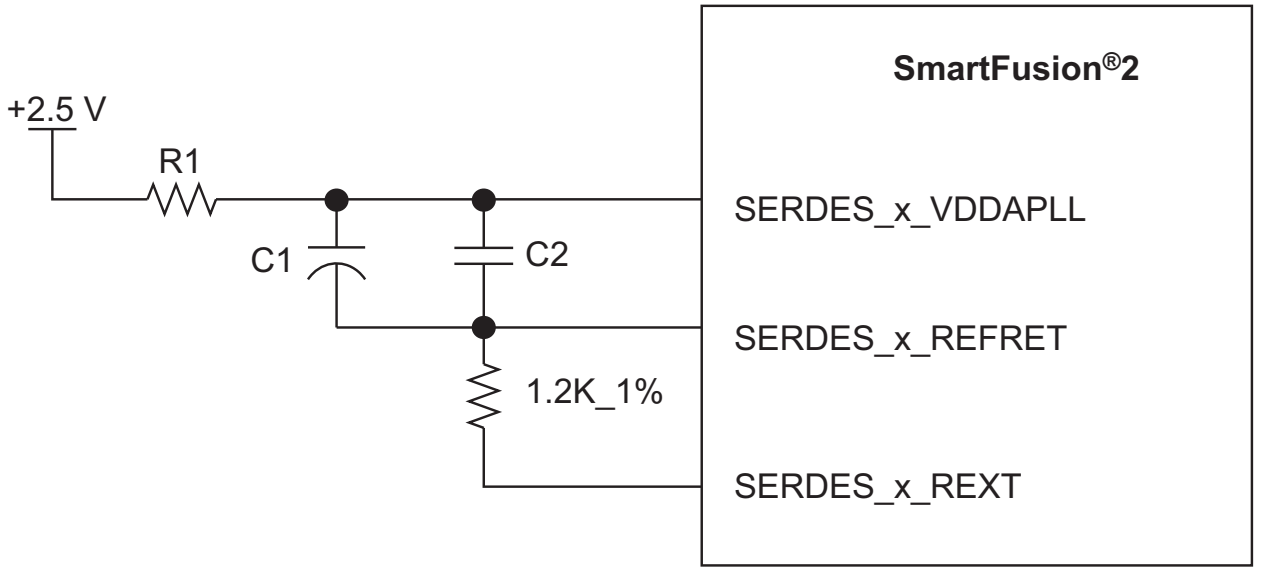
2.3.1.3 SerDes PLL (Ask a Question)

The following are the two power supply nodes for SerDes.

1. SERDES_x_VDDAPLL
2. SERDES_x_PLL_VDDA

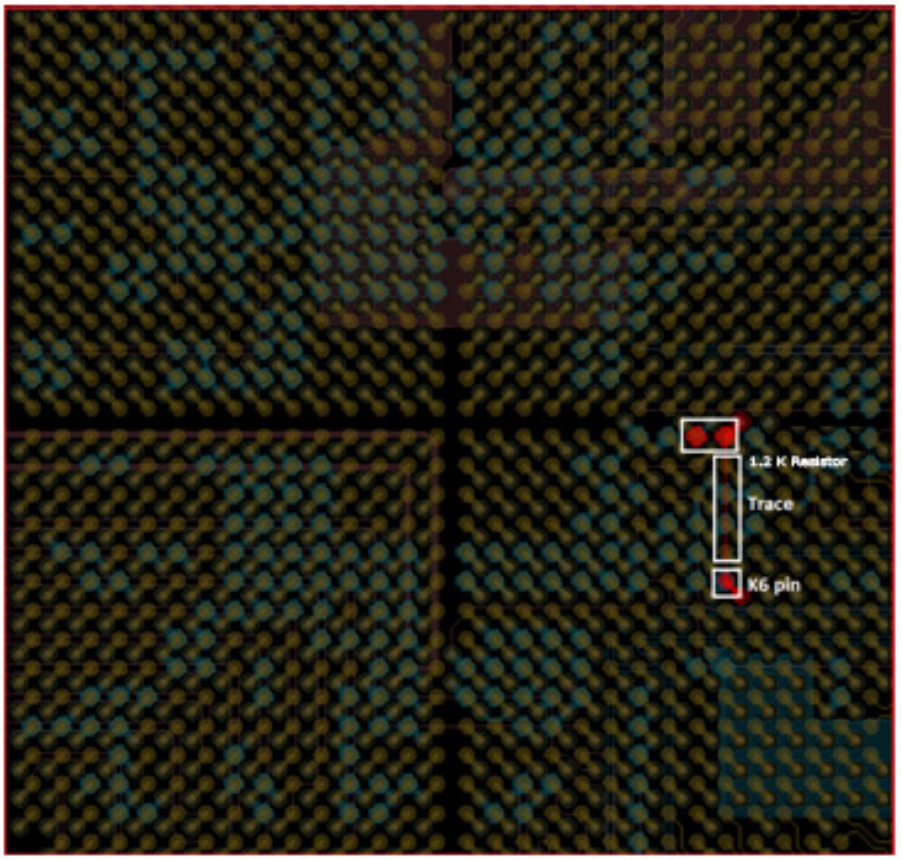
These supplies need separate filter circuits. Filter circuit for SERDES_x_VDDAPLL and a typical filter circuit for SERDES_x_PLL_VDDA is shown in the following figure.

Figure 2-5. Filter Circuit for SerDes PLL Power Supply



- C1 and R1 must be placed near the device.
- C2 must to be placed under the BGA via. The capacitor pad to via trace must be as small as possible.
- Apart from this, a precision resistor (1.2 K) is placed between the SERDES_x_REXT and SERDES_x_REFRET pins. This resistor must be placed near the BGA via of SERDES_x_REXT pin. Any aggressive signal traces must be kept away from this resistor to avoid unwanted noise from coupling into this critical circuit. A sample placement is shown in the following figure.

Figure 2-6. Component Between 1.2 K Resistor and K6 Pin



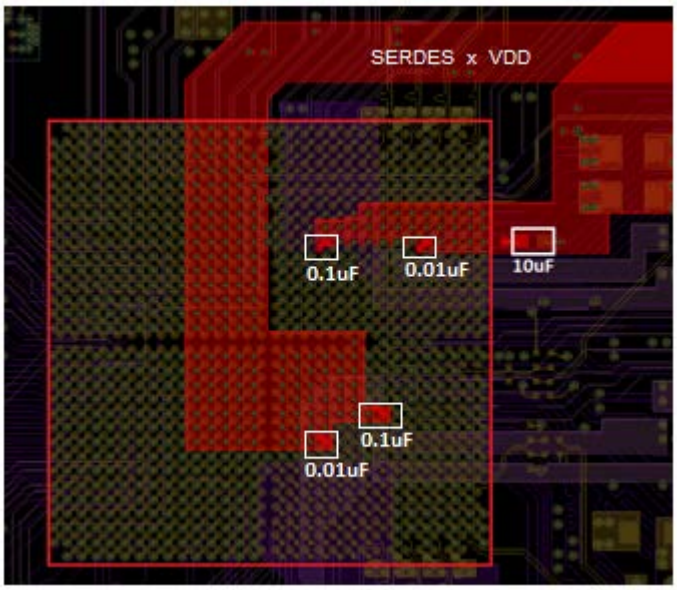
For more information about R1, C1, and C2, see 1. Design Considerations.

2.3.2 Plane Layout (Ask a Question)

2.3.2.1 SerDes Core Power (SERDES_x_VDD) (Ask a Question)

Even though SERDES0 and SERDES1 cores share the same power supply, separate planes must be made while connecting to corresponding SerDes blocks, as shown in the following figure. This reduces the noise coupling between SERDES0 and SERDES1 blocks.

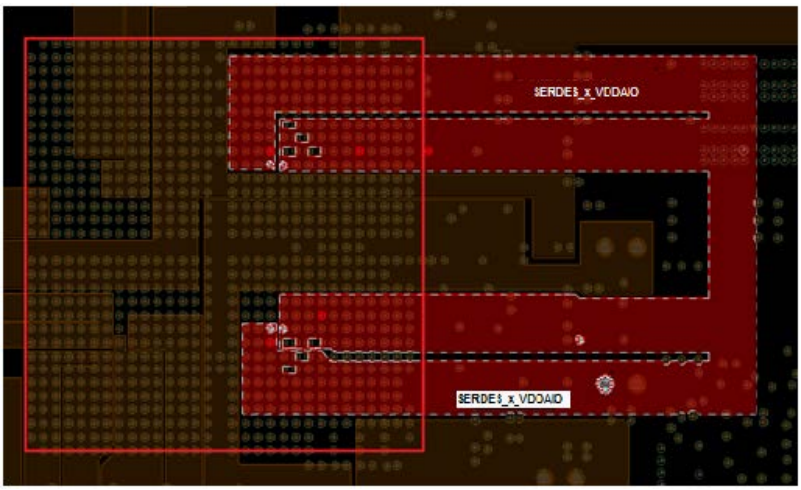
Figure 2-7. Layout for SERDES_x_VDD Plane



2.3.2.2 SerDes I/O Power (SERDES_x_VDDAIO) (Ask a Question)

Even though SERDES0 and SERDES1 I/Os share the same power supply, make separate planes while connecting to the corresponding pins, as shown in the following figure. Each plane is separated as SERDES_0_L01_VDDAIO, SERDES_0_L23_VDDAIO, SERDES_1_L01_VDDAIO, and SERDES_1_L01_VDDAIO, as shown in the following figure. This reduces the noise coupling between the differential lanes.

Figure 2-8. Layout of SERDES_x_VDDAIO Plane

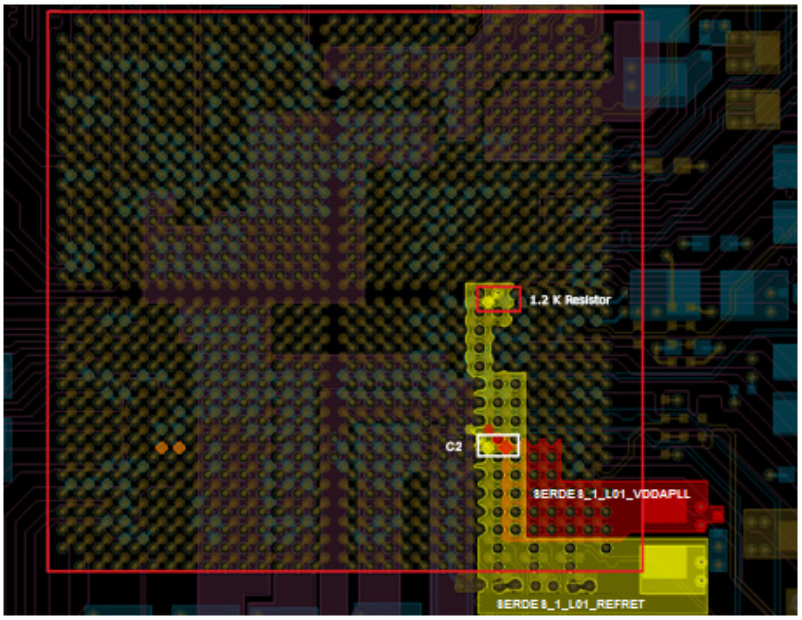


2.3.2.3 SerDes PLL (Ask a Question)

- Plane routing for SERDES_1_L01_VDDAPLL and SERDES_1_L01_REFRET is shown in the following figure.
- SERDES_1_L01_VDDAPLL and SERDES_1_L01_REFRET should not be routed as traces. A small trace width causes poor noise performance due to the high inductive behavior of the trace. Even though the current requirement is low, these supply traces should be routed as small planes, as shown in the following figure.

- The connections of 1.2 kΩ resistor and SERDES_1_L01_REXT of SmartFusion2/IGLOO2 must not be routed as a thick plane. It must be routed as a signal trace to meet minimum capacitance requirement of the SERDES_1_L01_REXT pin. The length of the trace should be short. The following figure shows the sample layout.
- Same layout guidelines should be followed for the remaining SerDes PLL power supplies.

Figure 2-9. Layout of SERDES_1_L01_VDDAPLL and SERDES_1_L01_REFRET



2.3.3 Simulations (Ask a Question)

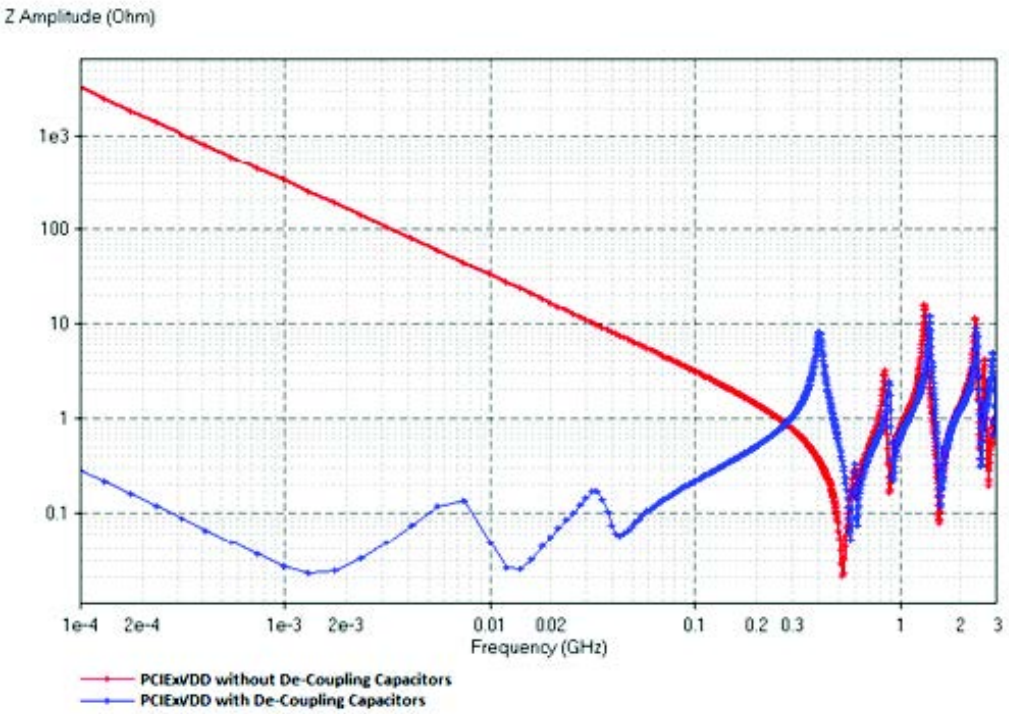
2.3.3.1 SerDes Core Power (SERDES_x_VDD) (Ask a Question)

The target impedance of the SERDES_x_VDD pin is calculated as 300 mΩ, based on the following values (see Power Supply):

- V_{SUPPLY} = 1.2 V
- I_{trans} = 200 mA
- Ripple = 5%

The following figure shows the impedance of the plane (SERDES_x_VDD) improved by the decoupling capacitors. The impedance of the plane is kept under 0.2 Ω till 100 MHz.

Figure 2-10. Impedance Profile of SERDES_x_VDD Plane Over Frequency Range



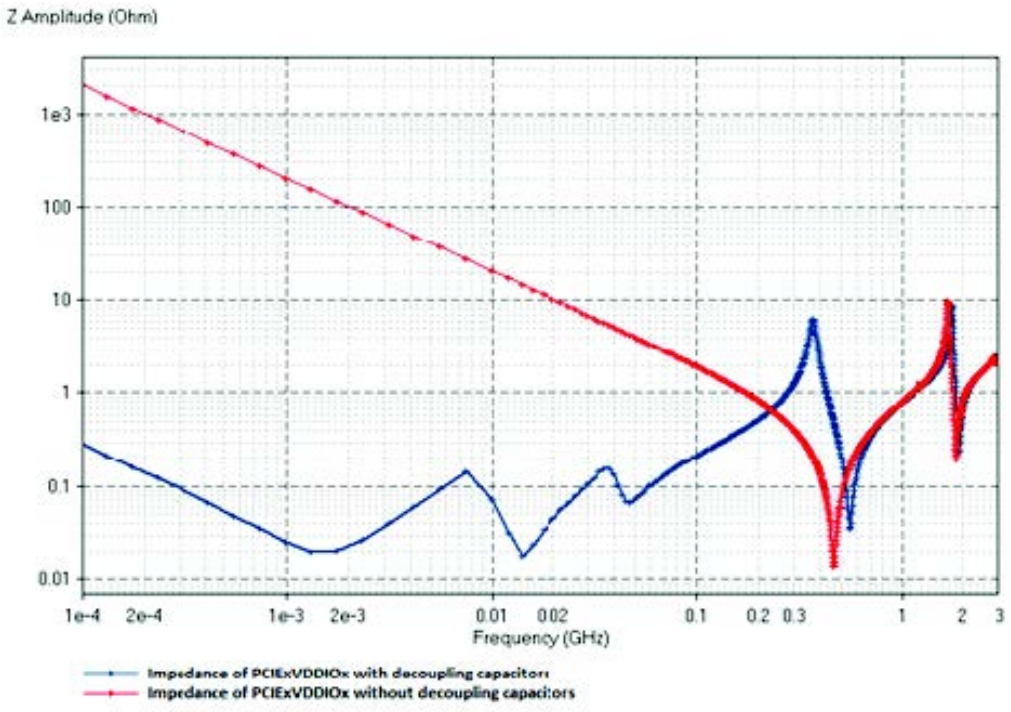
2.3.3.2 SerDes I/O Power (SERDES_x_VDDAIO) (Ask a Question)

The target impedance of the SERDES_x_VDDAIO pin is calculated as 240 mΩ, based on the following values (see Power Supply):

- V_{SUPPLY} = 1.2 V
- I_{trans}= 250 mA
- Ripple = 5%

The following figure shows the impedance of the plane (SERDES_x_VDDAIO) improved by the decoupling capacitors. The impedance of the plane is kept under 0.2 Ω till 100 MHz.

Figure 2-11. Impedance Profile of SERDES_x_VDDAIO Plane Over Frequency Range



2.4 DDR (Ask a Question)

Some of the variants support the fabric DDR (FDDR) and microcontroller subsystem DDR (MDDR) and some variants support only FDDR in SmartFusion2. See datasheet to know the bank on which the DDR is supported on each particular device. The layout guidelines of the respective VDDIO should be followed. It also needs VREF voltage for an internal reference. Noise on VREF impacts the read performance of SmartFusion2/IGLOO2 devices. VREF lines must not be routed near the aggressive nets or switching power supplies. The VDDIO guidelines should be followed for DDR bank VDDIO. This section explains the guidelines to be used for VREF.

2.4.1 Component Placement (Ask a Question)

2.4.1.1 VREF (Ask a Question)

- The bypass capacitor (10 µF) must be placed near, or at the edge of the device.
- All decoupling capacitors (0.1 µF and 0.01 µF) must be 0402 or of a smaller package size as they are needed to be mounted on the reverse side of the board. They must be fit between the adjacent vias of the BGA package pins. These decoupling capacitors are selected to have a low impedance over the operating frequency and temperature range.
- The capacitor pad to via trace must be small. Figure 2-1 shows how these capacitors are mounted. Keep the capacitor pad directly on the corresponding vias.

2.4.1.2 VDDIO (Ask a Question)

- The bypass capacitors (47 µF and 22 µF) must be placed near, or at the edge of the device.
- All decoupling capacitors (0.1 µF and 0.01 µF) must be 0402 or of a smaller package size as they are required to be mounted on the reverse side of the board. They must fit between the adjacent vias of the BGA package pins. These decoupling capacitors are selected to have a low impedance over the operating frequency and temperature range.

- The capacitor pad to via trace must be small. Figure 2-1 shows how these capacitors are mounted. The capacitors can also be mounted directly on the pad on the vias.

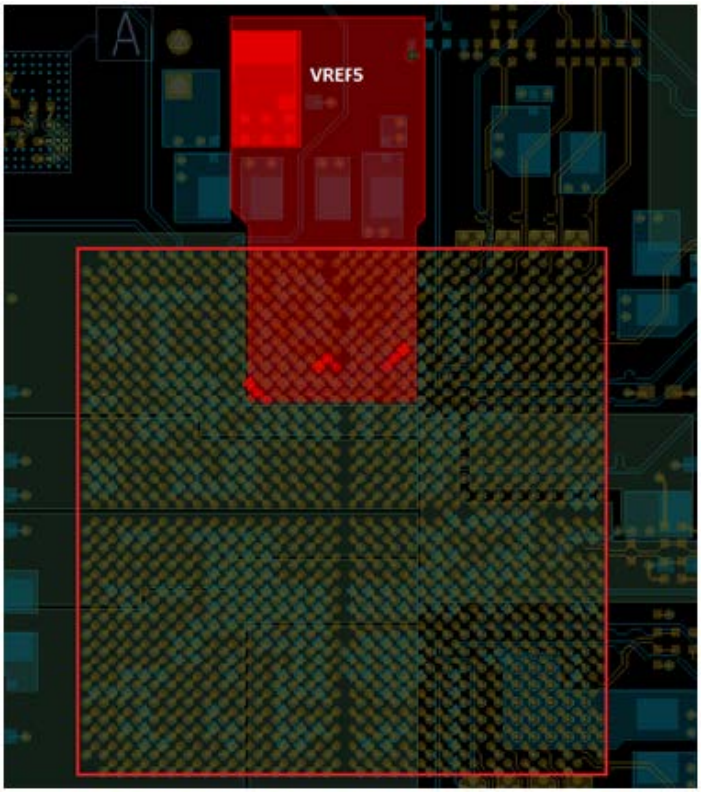
2.4.2 Plane Layout (Ask a Question)

2.4.2.1 VREF (Ask a Question)

Noise on VREF impacts the read performance of SmartFusion2/IGLOO2 devices. The VREF lines must be routed with no aggressive net or switching power supply nearby. Even the current is low, VREF must not be routed as trace as it is very susceptible to noise.

The following figure shows the VREF5 used for MDDR.

Figure 2-12. Layout of VREF5



2.4.2.2 VDDIO (Ask a Question)

The shape of the plane does not have a specific requirement. The width of the plane must be sufficient to carry the required current. The following figures show the sample layout for VDDIO0 and VDDIO5 planes.

Figure 2-13. Layout of VDDIO0 Plane

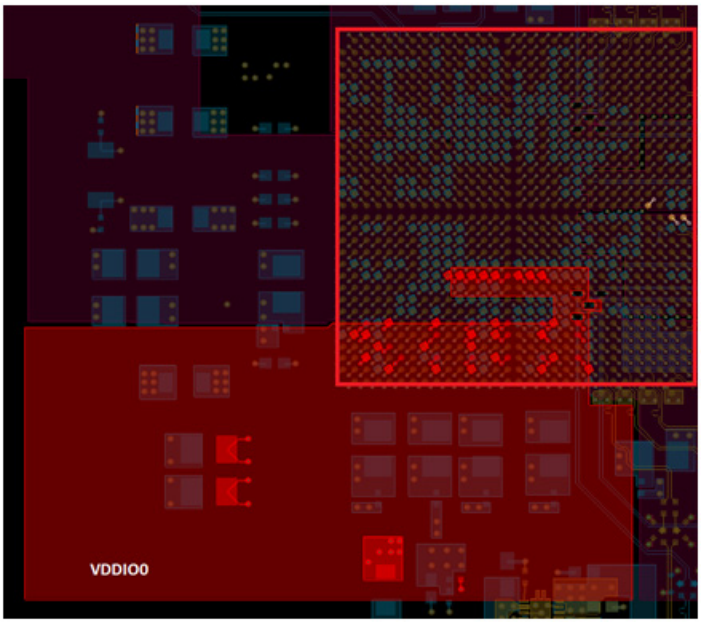
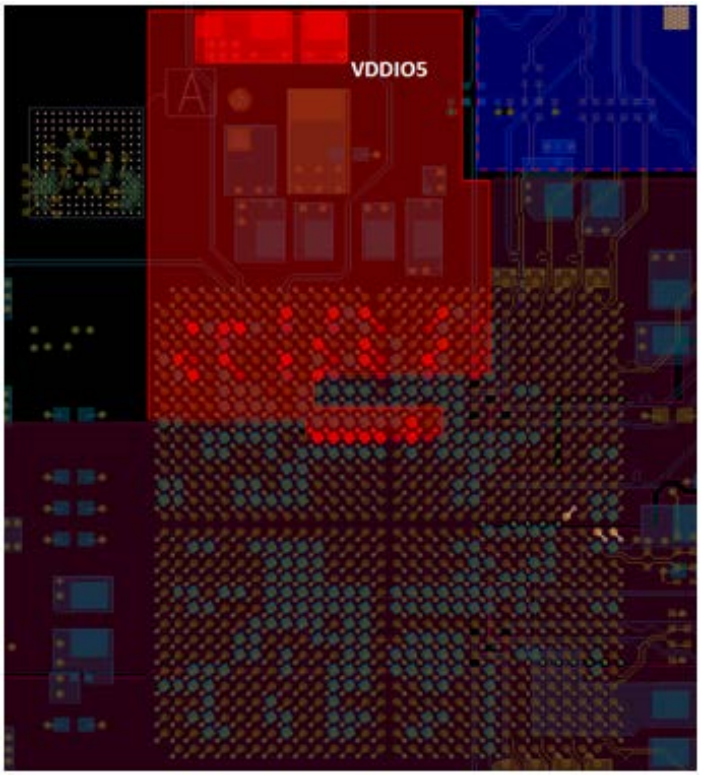


Figure 2-14. Layout of VDDIO5 Plane



2.4.3 Simulations (Ask a Question)

The target impedance of the DDR VDDIO is calculated as 240 mΩ, based on the values (see Power Supply):

- V_{SUPPLY} = 1.5 V
- I_{trans} = 250 mA

- Ripple = 5%

The impedance profile of the DDR VDDIO plane over frequency range is shown in the following figures. The impedance improves with the decoupling capacitors provided. The target impedance of 0.3 Ω has been achieved till 500 MHz.

Figure 2-15. Impedance Profile of VDDIO0 Plane Over Frequency Range

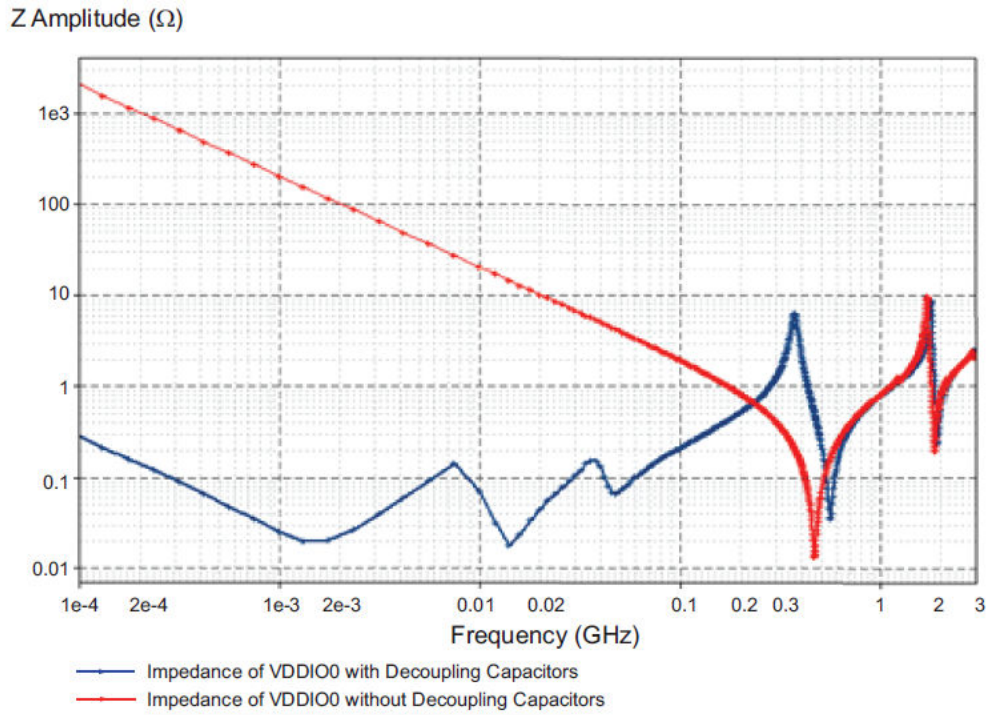
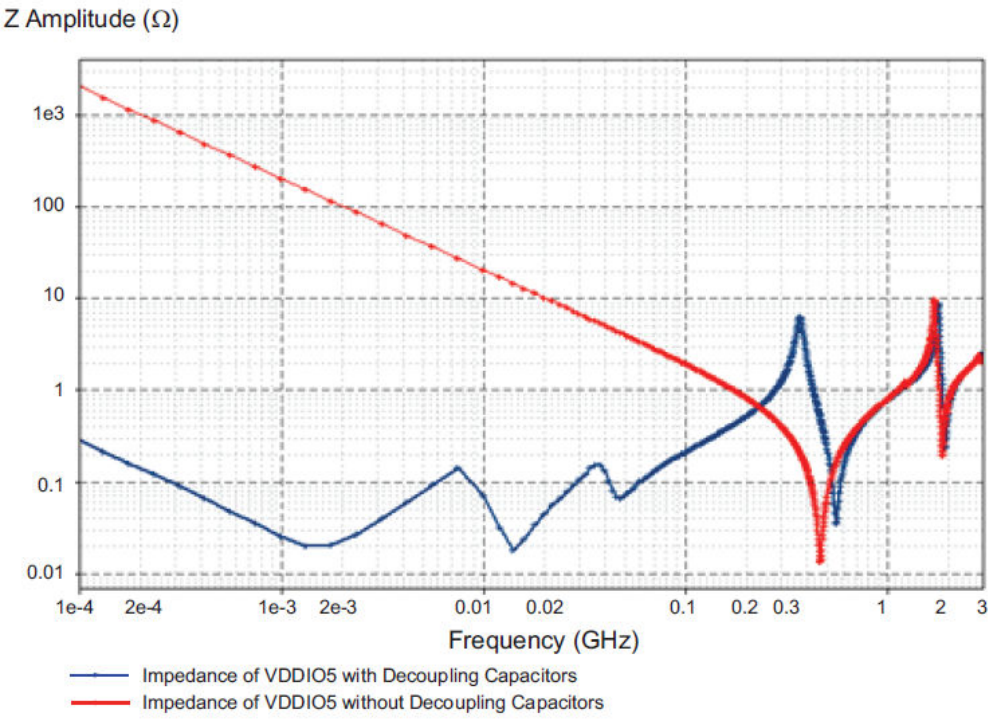


Figure 2-16. Impedance Profile of VDDIO5 Plane Over Frequency Range

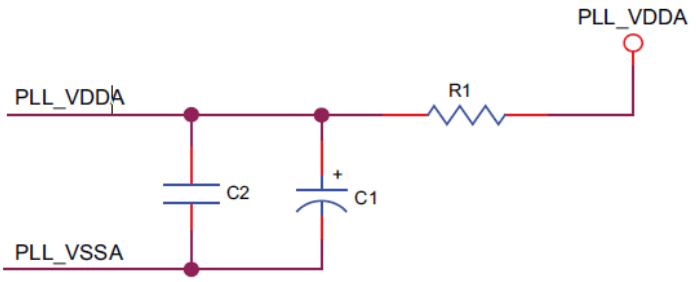


2.5 PLL (Ask a Question)

To achieve a reasonable level of long term jitter, deliver an analog grade power supply to the PLL. An R-C or R-L-C filter is used with the C being composed of multiple devices to achieve a wide spectrum of noise absorption. Even the circuit is simple, there are specific board layout requirements. Board layout around the high-frequency capacitor and the path to the pads are critical. It is vital that the quiet ground and power are treated like analog signals. The entire VDDPLL and PLLVSSA wiring path must not be coupled with any signal aggressors, such as any high-swing and high slew rate signals such as TTL, CMOS, or SSTL signals used in DDR buses, and so on.

The circuit for the power supply filter is shown in the following figure.

Figure 2-17. Filter Circuit for PLL

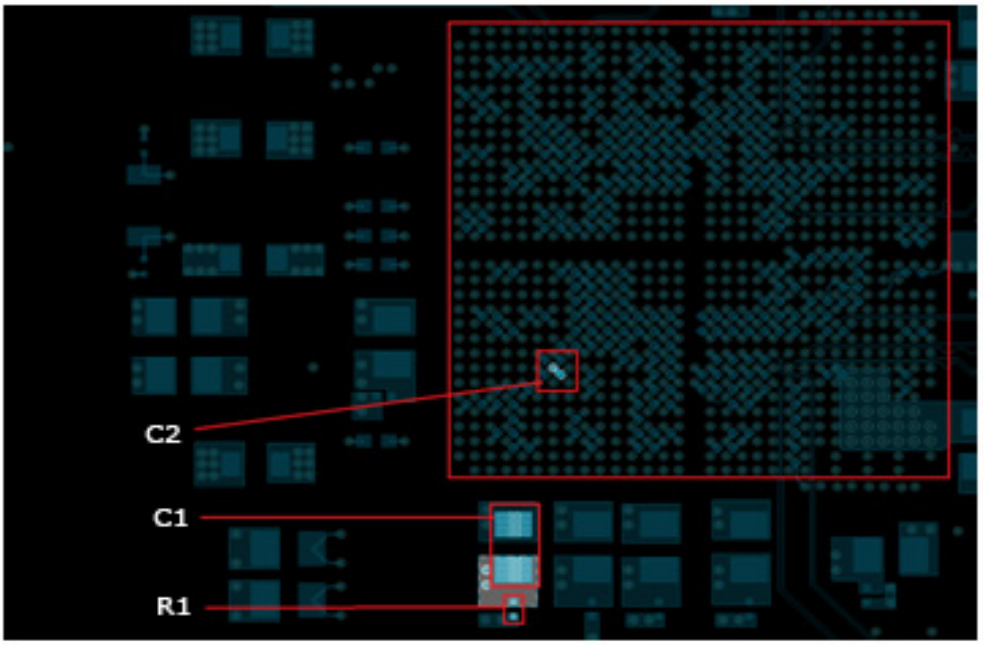


For more information about R1, C1, and C2, see 1. Design Considerations.

2.5.1 Component Placement (Ask a Question)

- The capacitor (C1) and series resistor (R1) must be placed near the device as close as possible to C2 device. A sample placement is shown in the following figure.
- The decoupling capacitor (C2) must be placed near the BGA via. The capacitor pad to via trace must be small. For more information about R1, C1, and C2, see 1. Design Considerations.

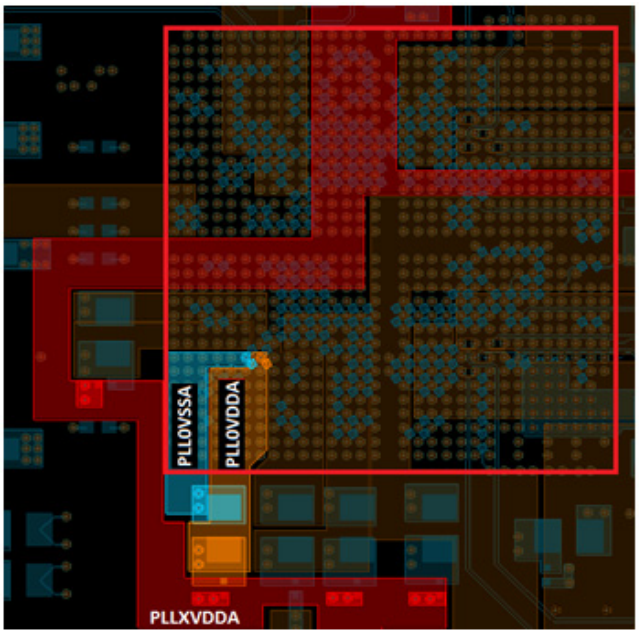
Figure 2-18. Placement of Capacitors for PLL Filter Circuit



2.5.2 Plane Layout (Ask a Question)

- Plane routing for PLL0VDDA and PLL0VSSA is shown in the following figure. These are with respect to the schematic, as shown in Figure 2-17.
- The capacitor (22 µF) and series resistor must be placed close to the device to the 0.1 µF capacitor. A sample placement is shown in the following figure.

Figure 2-19. Routing for PLL Filter Circuit



- PLL0VDDA and PLL0VSSA must not be routed with a small trace width as it increases the inductance resulting in ripples. These supply traces must be routed as plane (as shown in Figure 2-19), even though the current requirements are small.
- Same layout guidelines must be followed for DDR PLL power supplies. For more information about PCIe PLL guidelines, see SerDes.

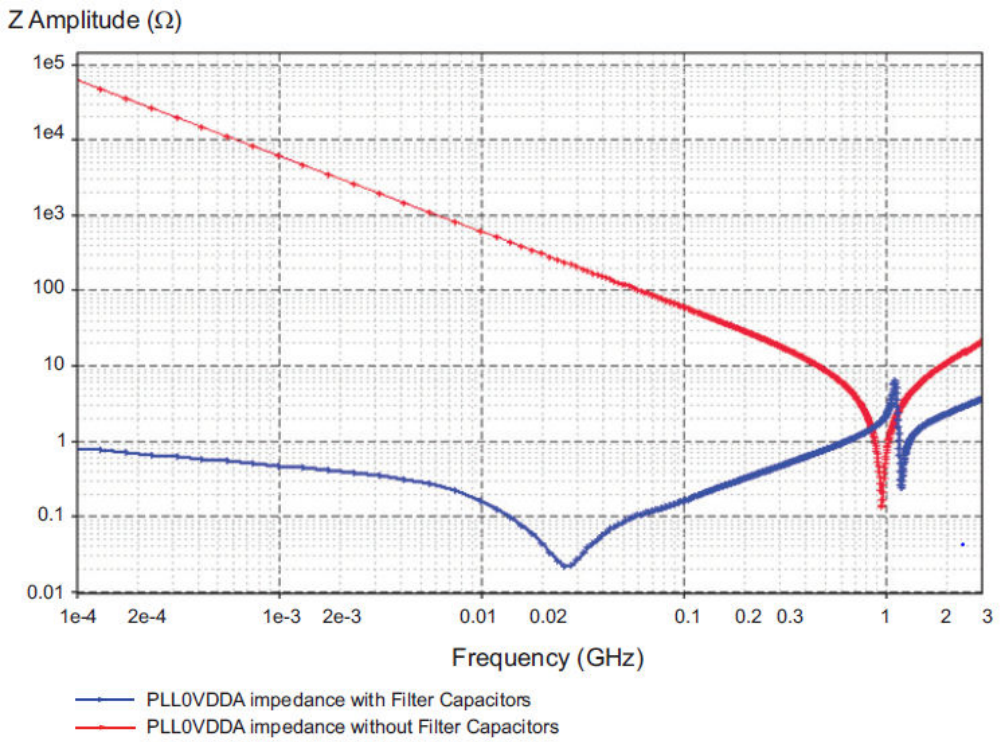
2.5.3 Simulations (Ask a Question)

The target impedance of the PLL0VDDA plane is calculated as 16.5 Ω based on the values (see Power Supply):

- V_{SUPPLY} = 3.3 V
- I_{trans} = 10 mA
- Ripple = 5%

The impedance of the place (Z) must be 16.5 Ω or less. For more information about ripples and its values, see DS0128: IGLOO2 and SmartFusion2 Datasheet. Plane impedance with and without filter circuit is shown in the following figure.

Figure 2-20. PLL0VDDA Plane Impedance



2.6 I/O Power Supply (Ask a Question)

2.6.1 Component Placement (Ask a Question)

- The bypass capacitors (47 µF and 22 µF) must be placed near, or if possible, at the edge of the device.
- All decoupling capacitors (0.1 µF and 0.01 µF) must be 0402 or of a smaller package size as they are required to be mounted under BGA package. They must be fit between the adjacent vias of BGA package pins. These decoupling capacitors are selected to have a low impedance over operating frequency and temperature range.

The capacitor pad to via trace must be small. Figure 2-1 shows how these capacitors are mounted. The capacitors can also be mounted directly on the pad on the vias. The decoupling capacitors must not be shared via connections.

2.6.2 Plane Layout (Ask a Question)

The shape of the plane does not have a specific requirement. The width of the plane must be sufficient to carry the required current.

2.6.3 Simulations (Ask a Question)

The target impedance of the VDDIO1 plane is calculated as 330 mΩ based on the following values (see Power Supply):

- V_{SUPPLY} = 3.3 V
- I_{trans} = 500 mA
- Ripple = 5%

The following figures show the impedance of the planes (VDDIO1 and VDDIO2). The impedance of the plane is improved by decoupling capacitors and is kept under 0.2 Ω till 100 MHz.

Figure 2-21. Impedance Profile of VDDIO1 Plane Over Frequency Range

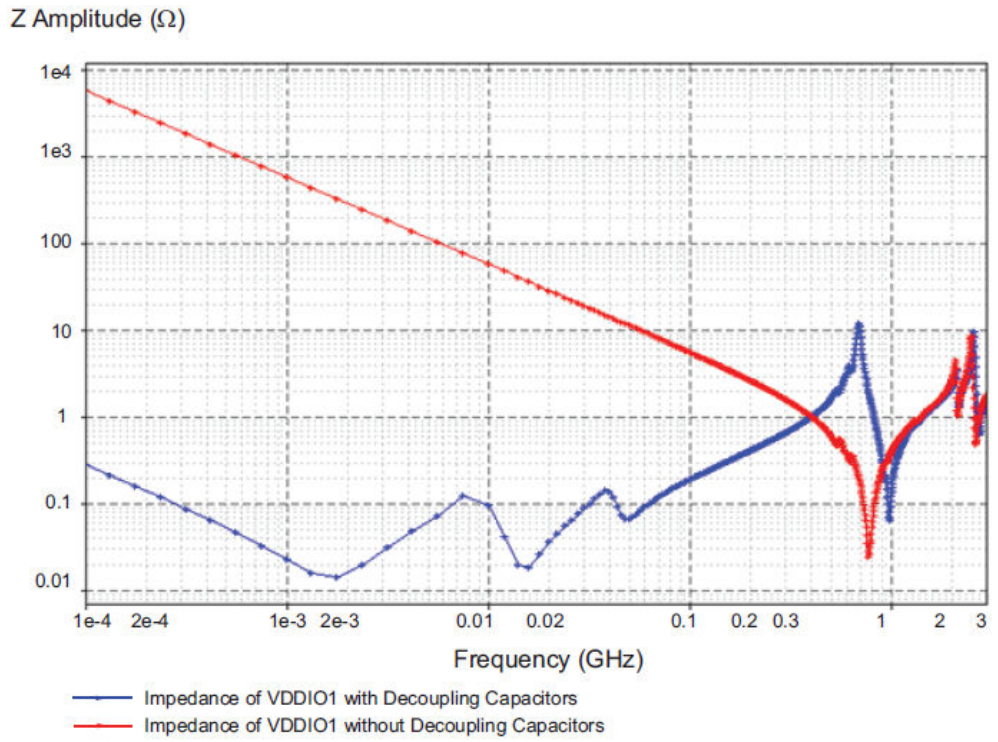
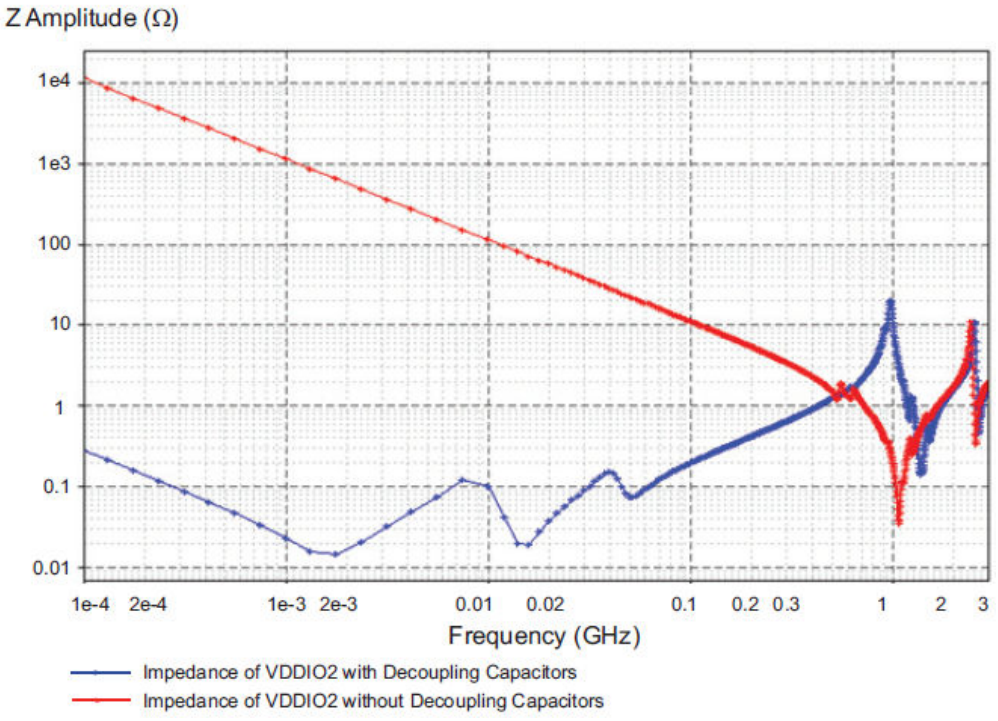


Figure 2-22. Impedance Profile of VDDIO2 Plane Over Frequency Range



2.7 Programming Power Supply (VPP or VCCENVM) (Ask a Question)

VPP is used as an input for the internal charge pump that generates the required voltage to program flash. VCCENVM is an embedded non-volatile memory (eNVM) supply.

2.7.1 Component Placement (Ask a Question)

- The bypass capacitors (47 µF and 22 µF) must be placed near, or at the edge of the device.
- All decoupling capacitors (0.1 µF and 0.01 µF) must be 0402 or of a smaller package size as they are needed to be mounted on the reverse side of the board. They must be fit between the adjacent vias of BGA package pins. These decoupling capacitors are carefully selected to have low impedance over the operating frequency and temperature range.
- The capacitor pad to via trace must be small. Figure 2-1 shows how these capacitors are mounted. The capacitor can also be mounted directly on the pad on the vias.

2.7.2 Plane Layout (Ask a Question)

The shape of the plane does not have a specific requirement. The width of the plane must be sufficient to carry the required current.

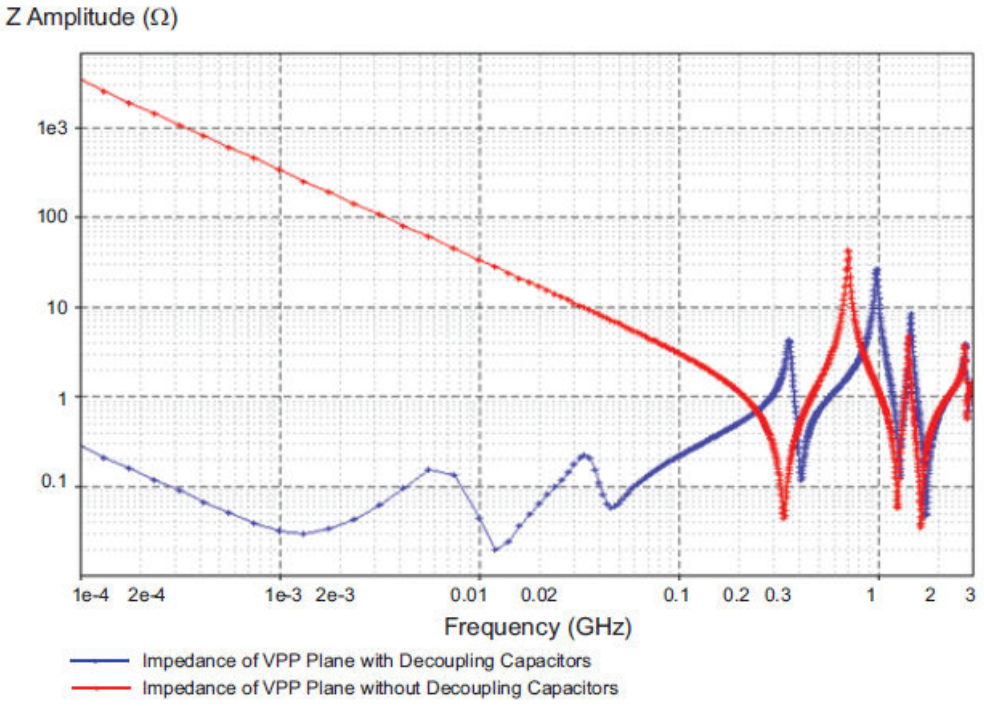
2.7.3 Simulations (Ask a Question)

The target impedance of the VPP is calculated as 3.3Ω, based on the values (see Power Supply):

- V_{SUPPLY} = 3.3V
- I_{trans} = 50 mA
- Ripple = 5%

The simulation result (as shown in the following figure) shows that it meets the required impedance levels.

Figure 2-23. Impedance Profile of VPP Plane Over Frequency Range



2.8 High-Speed Serial Link (SerDes) (Ask a Question)

2.8.1 Layout Considerations (Ask a Question)

2.8.1.1 Differential Traces (Ask a Question)

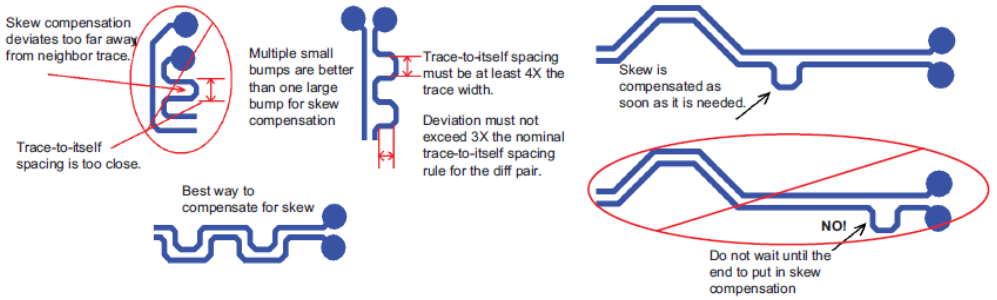
A well designed differential trace not must have the following qualities:

- Mismatch in impedance
- Insertion loss and return loss
- Skew within the differential traces

The following points need to be considered while routing the high-speed differential traces to meet the preceding qualities.

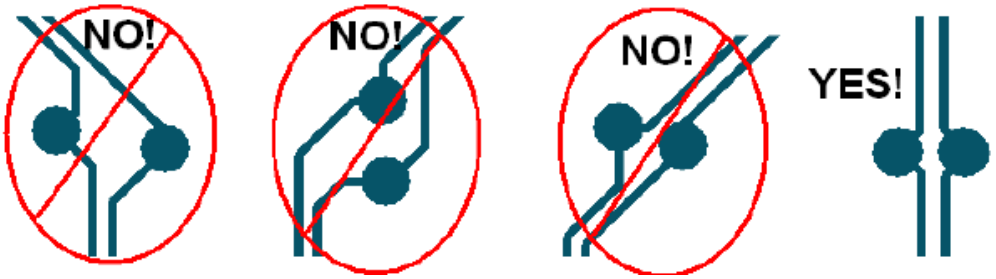
- The traces must be routed with tight length matching (skew) within the differential traces. Asymmetry in length causes conversion of differential signals in Common mode signals. The differential pair must be routed such that the skew within differential pairs is less than 5 mils. The length match must be used by matching techniques, as shown in the following figure.

Figure 2-24. Skew Matching



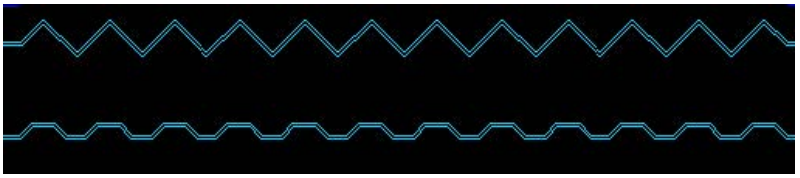
- The length of differential lanes must be matched within the TX and RX group. This is applicable only to specific protocols like XAUI and so on.
- Route differential pairs symmetrically into and out of structures, as shown in the following figure.

Figure 2-25. Example of Asymmetric and Symmetric Differential Pairs Structure



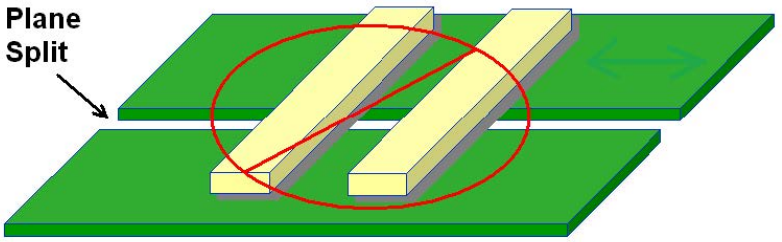
- Skin effect dominates as the speed increases. To reduce the skin effect, width of the trace has to be increased (loosely coupled differential traces). Increase in trace width causes increase in dielectric losses. To reduce the dielectric loss, use low Dissipation Factor (Df) PCB materials like Nelco 4000-13. This is approximately double the cost of FR4 PCB material, but can provide increased eye-opening performance when longer trace interconnections are required. Remember to maintain 100 Ω differential impedance. Need to consider this if the data rate is 5 Gbps and above.
- Far end cross talk is eliminated by using stripline routing. However, routing in stripline causes more dielectric loss and more variation in the impedance. Cross talk affects only when there is a high density routing. It is better to route as microstrip, if there is enough space between differential pairs (> 4 times the width of the conductor) to reduce dielectric loss. Simulations are recommended to see the best possible routing.
- 2116 or 2113 glass weaving PCB materials must be used to avoid the variations in the impedances. Zig-zag routing must be used instead of straight line routing to avoid glass weaving effect on impedance variations, as shown in the following figure. Instruct the fabrication vendor to use these PCB materials before manufacturing.

Figure 2-26. Zig-Zag Routing



- These traces must be kept away from the aggressive nets or clock traces. For example, on M2S050T devices, the SerDes and DDR traces must not be adjacent to each other.
- Separation between the coupled differential trace pairs must be 1x. Spacing between channels must be > 3x separation. Trace stubs must be avoided. The stub length must not exceed 40 mils for 5 Gbps data rate.
- The trace lengths must be kept as small as possible.
- Use low roughness, that is, smooth copper. As the speed increases insertion loss due to the copper, then roughness increases. The attenuation due to skin effect is increased proportional to the square root of frequency. The roughness courses this loss proportional to frequency. Instruct the PCB fabrication house to use smooth copper, if the frequency exceeds 2 Gbps.
- Split reference planes must be avoided. Ground planes must be used for reference for all the SerDes lanes.

Figure 2-27. Ground Planes for Reference

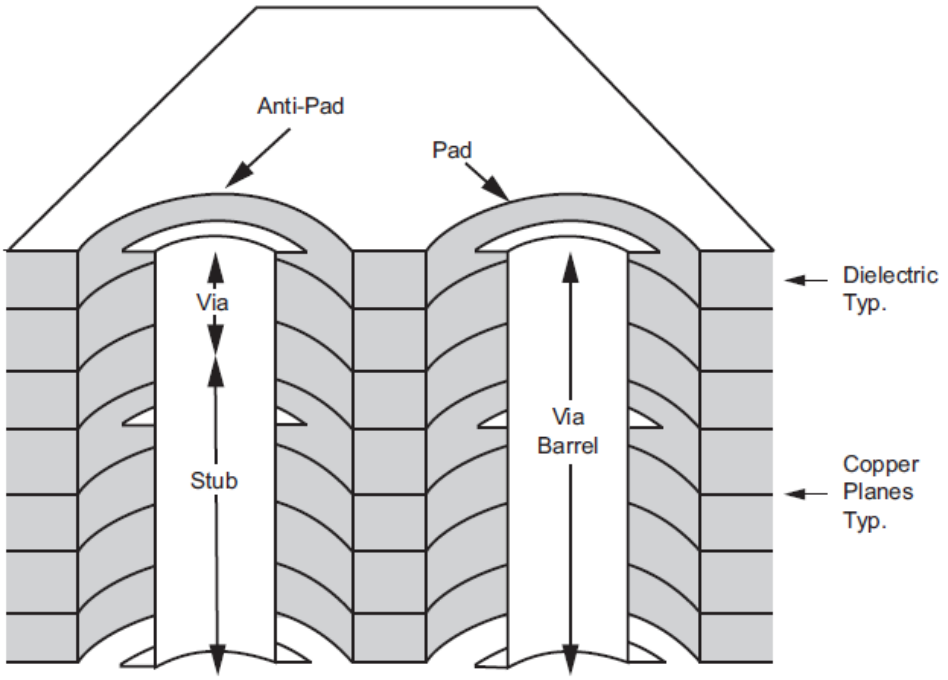


2.8.2 Via (Ask a Question)

The following are the via placement steps:

- The target impedance of vias are designed by adjusting the pad clearance (anti-pad size). Field solver must be used to optimize the via according to the stack-up.

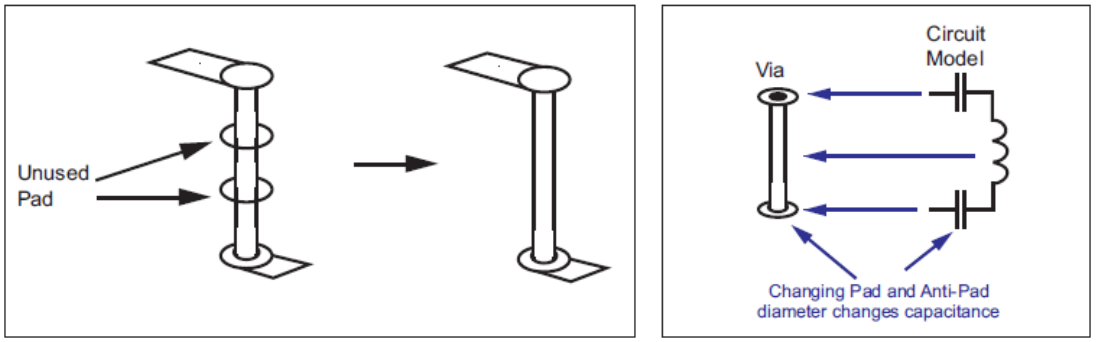
Figure 2-28. Via Illustration[1]



Important: Simonovich, E. Bogatin, Y. Cao, Method of Modeling Differential Vias, White Paper, Jan 2011

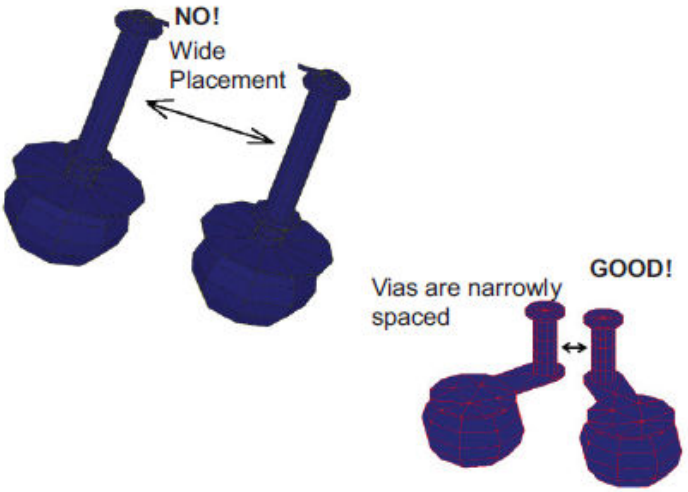
- Number of vias on different traces must be avoided or minimized. SerDes signals must be routed completely on a single layer with the exception of via transitions from component layer to the routing layer (3-via maximum).
- The length of via stub must be minimized by back drilling the vias, or by routing the signals from near top to near bottom layer, or else blind or buried vias can be used. Using blind-vias or back drilling is a good method to eliminate via stubs and reduce reflections.
- The stub length must be kept below 100 mils, if the data rate is 2.5 Gbps and 40 mils for 5 Gbps.
- If feasible, non-functional pads must be removed. Non-functional pads on via are the pads where no trace is connected. This reduces the via capacitance and stub effect of pads.

Figure 2-29. Non-Functional Pads of Via



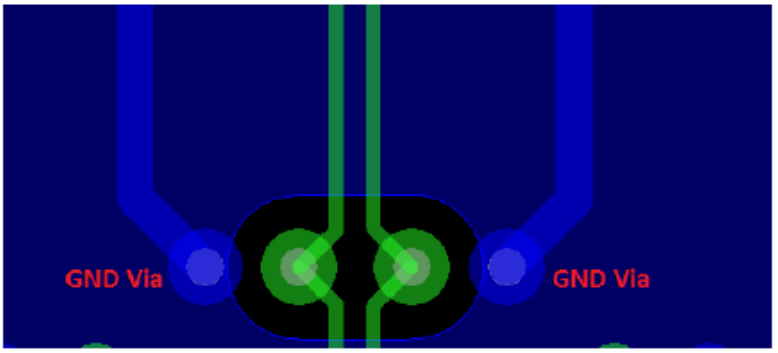
- The tight via to via pitch is practical to reduce the cross talk effect, as shown in the following figure.

Figure 2-30. Via-to-Via Pitch



- Symmetrical ground vias (return vias) must be used to reduce discontinuity for Common mode signal component, as shown in the following figure. Common mode of part of the signal requires continuous return path RX to TX and GND. Return vias help in maintaining the continuity.

Figure 2-31. GND Via or Return Via

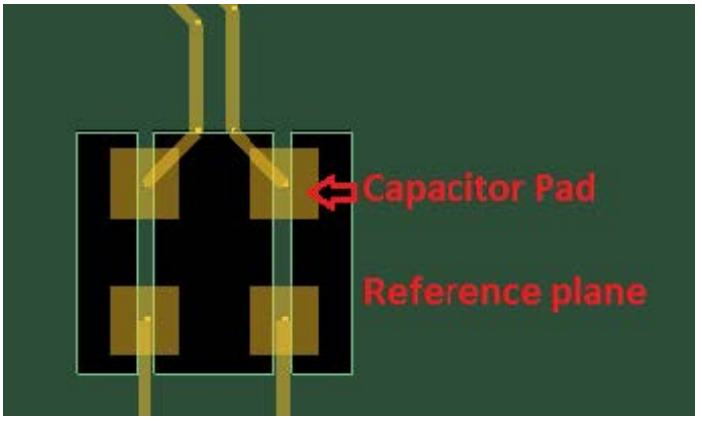


2.8.3 DC Blocking Capacitors (Ask a Question)

The plane under the pads of DC blocking capacitors must be removed (as shown in the following figure) to match the impedance of the pad to 50Ω.

Important: This has to be performed only on immediate reference plane and not on all planes.

Figure 2-32. Capacitor Pad Reference Plane



2.8.4 Connectors (Ask a Question)

The plane keep-out clearance should be optimized from the pin to get 50 Ω impedance when through hole SMAs or connectors are used. This reduces the reflection loss.

2.9 Considerations for Simulation (Ask a Question)

Simulations must confirm the quality of the received signal. The following files are needed to simulate the serial channel:

- IBIS: AMI files for SmartFusion2/IGLOO2 and any other devices that are connected to SerDes
- Package: files (optional). S-parameter of package improves the accuracy instead of using package parameters in the IBIS file
- Board traces model file including via models
- Connector models, if required

The following steps describe how to run the serial channel simulations:

2.9.1 Step 1: Gathering the Required Files (Ask a Question)

2.9.1.1 IBIS-AMI Models (Ask a Question)

The IBIS-AMI models of SmartFusion2/IGLOO2 and the IBIS-AMI models of IC interfaced with SmartFusion2/IGLOO2 can be downloaded from the Microchip website:

- www.microchip.com/en-us/products/fpgas-and-plds/system-on-chip-fpgas/smartfusion-2-fpgas#ibis
- www.microchip.com/en-us/products/fpgas-and-plds/fpgas/igloo-2-fpgas#ibis

2.9.1.2 Package Models (Ask a Question)

The package models (S-parameter models) of SmartFusion2/IGLOO2 can be downloaded from the Microchip website:

- www.microchip.com/en-us/products/fpgas-and-plds/system-on-chip-fpgas/smartfusion-2-fpgas#ibis
- www.microchip.com/en-us/products/fpgas-and-plds/fpgas/igloo-2-fpgas#ibis

Accuracy of simulation improves with S-parameter model of package file instead of using package models available in the IBIS file. If S-parameter models for package are used, the package details in IBIS must be commented.

2.9.1.3 PCB Trace Models (Ask a Question)

The PCB file must be converted into a compatible format of simulator software. For example, the `.HYP` file format of PCB is required to be simulated in Hyperlynx, and `.SPD` file format of PCB is

required to be simulated in Sigrity. Once the PCB file is loaded in the simulation tool, the stack-up that matches the PCB stack-up must be checked. The dielectric constant, Dk and dissipation factor, and Df of PCB material must be defined. The tool extracts incorrect models, if the points are not defined properly.

The SerDes traces must be identified and ports on both the sides of the traces need to be assigned. The S-parameter models of traces should be extracted. The following tools can be used to extract S-parameter models of PCB traces:

- Agilents ADS
- Mentors Hyperlynx
- Sigritys PowerSI

Important: It is not mandatory to use the mentioned tools, many other tools are available in the market which can extract S-parameter models.

2.9.2 Step 2: Creating Simulation Topology (Ask a Question)

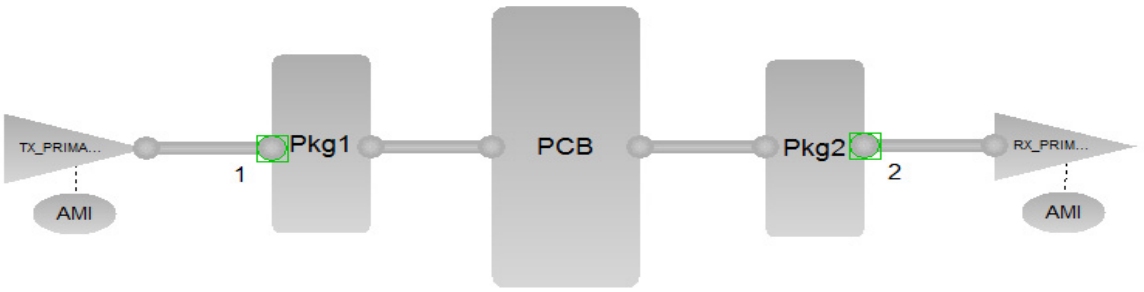
The typical topology, as shown in the following figure, shows the blocks involved in the serial link analysis. These blocks are taken from the Sigrity tool. All simulations related to SerDes are done on Sigritys SystemSI tool in this document. Topology is same in any tool. This can be done in any tool that supports the serial link analysis.

The typical topology for SLA simulation is listed as follows:

- AMI: AMI models of TX and RX
- TX_PRIMARY: IBIS model of TX I/O
- Pkg1 and Pkg2: Package model of TX and RX I/O
- PCB: S-parameter model of SmartFusion2 Development Kit SerDes Traces
- RX_PRIMARY: S-parameter model of either the connector or the IBIS model of the receiver IC device

Once all the model files are imported into the topology, the default configuration in the AMI model must be left to calculate the appropriate coefficients by the tool and then to run the simulations.

Figure 2-33. Typical Topology for SLA Simulation



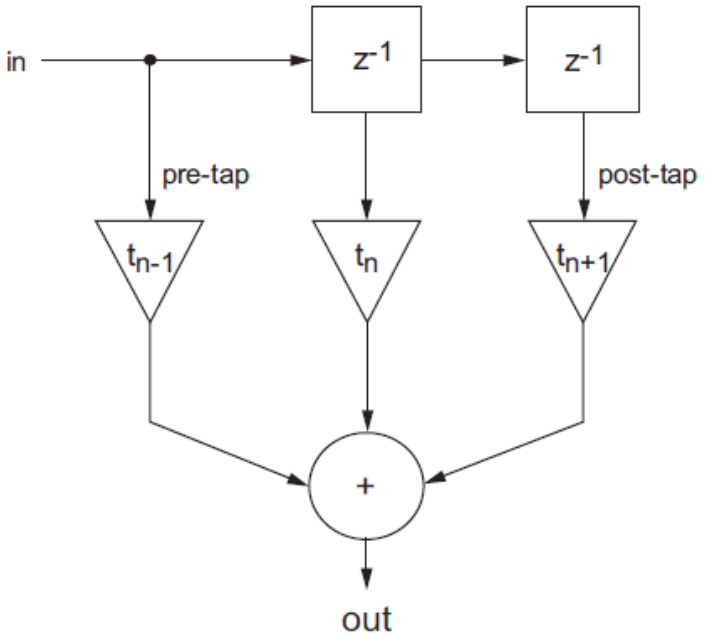
2.9.3 Step 3: Configuration of AMI Model (Ask a Question)

The following configurations on the AMI model are needed before simulating the serial channel.

2.9.3.1 TX AMI Model (Ask a Question)

The following figure shows the 3-tap Feed Forward equalizer structure for TX. The output of the TX is given by the transfer function tn-1 + tnZ-1 + tn+1Z-2. The TX output depends on the value of tap coefficients.

Figure 2-34. Block Diagram of the 3-tap Feed Forward Equalizer



The following are the details of coefficients.

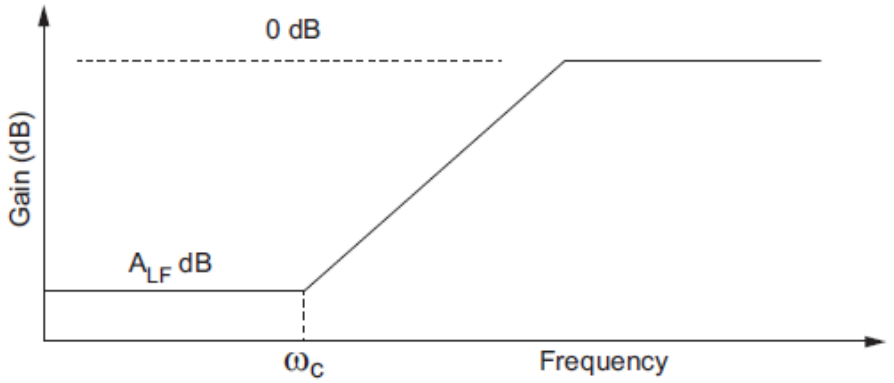
- t0: Pre-cursor tap setting. The range is from –0.4 to –0.01, default value is –0.01.
- t1: Main tap. The range is from 0.1 to 1, default value is 1.
- t2: Post-cursor tap. The range is from –0.5 to –0.01, default value is –0.01.
- TapsFromFile: Explicit feed forward equalizer (FFE) coefficients can be set through this file. If a file is used, it overrides the manual tap settings and automatic generation.
- TapsToFile: Output FFE tap coefficients to this file when automatic generation coefficients is used.

2.9.3.2 RX AMI Model (Ask a Question)

SerDes supports the programmable single pole continuous time linear equalization (CTLE) at the receiver. The linear equalization comprises of amplifying higher frequency components that have been more severely attenuated by the Interconnect, or attenuating the lower frequency components to a greater degree than the higher frequency components.

The low-frequency attenuation level and the low-frequency flat-band bandwidth are programmable, as shown in the following figure.

Figure 2-35. Continuous Time Linear Equalization Response



Both ALF and ωc (f0) can be set to maximize the signal quality of the receiver for achieving the highest possible bit error rate (BER).

- Alf: Low-frequency dB loss of the filter. The range is from 0 to 50, default value is 6.
- f0: High pass cut-off frequency. The range is from 1e6 to 5e10, default value is 1e9.

2.9.4 Step 4: Results (Ask a Question)

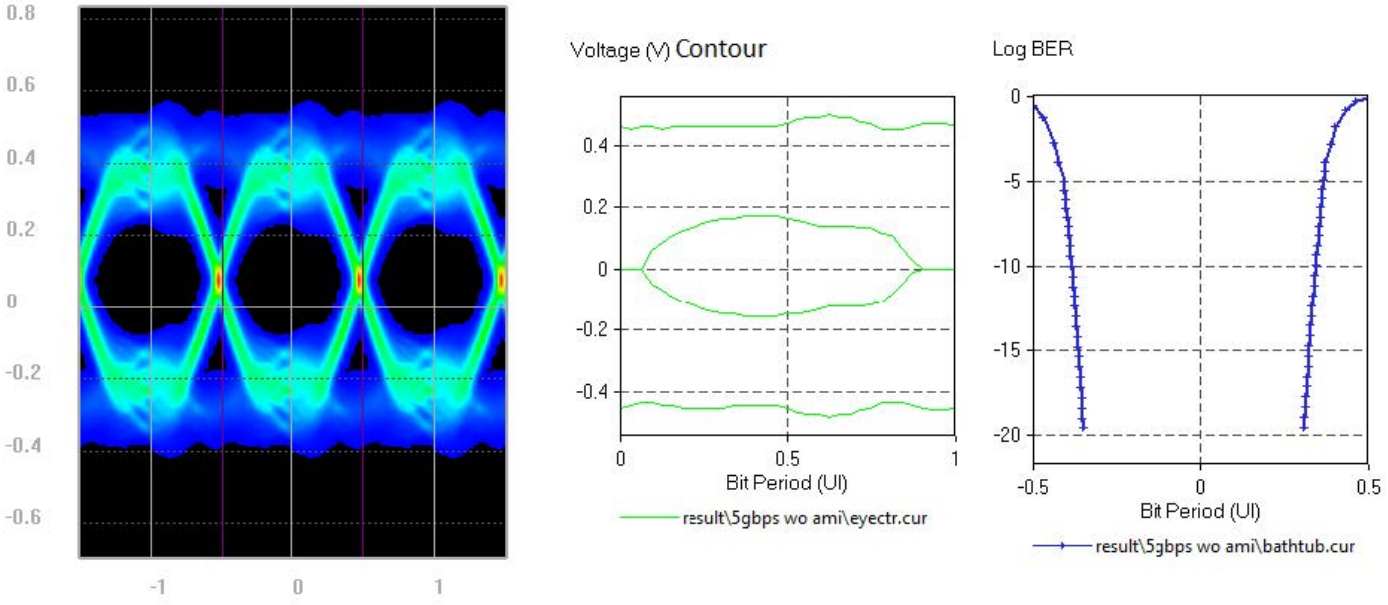
Qualification of simulation results is based on the eye-height, eye-width, and BER curves. Check the eye-height and eye-width at target BER of 10e-12. These results are found in the report generated by the simulation tool. For example, the Sigrity tool gives the following information at RX:

At BER of 10e-12, running at 5 Gbps bit rate

- The eye-width is 0.68 UI (Unit Interval)
- The eye-height is 213 mV

This simulation is on the SmartFusion2 Development Kit using the Sigrity tool and the waveforms are shown in the following figure. The simulation result shows that it meets the PCIe 2.0 requirements.

Figure 2-36. Expected Results from Simulations (Eye Diagram, Eye Contour, and Bath Tub Curve)



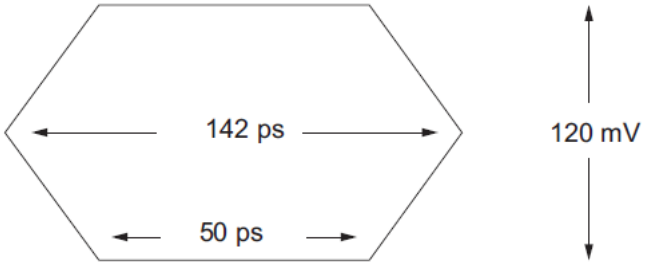
The following table lists the specifications of the received signal for PCIe.

Table 2-1. Specifications of the Received Signal for PCIe

Bit Rate	Min Height of the Eye at RX	Min Width of the Eye at RX
2.5 Gbps	175 mV	0.6 UI
5 Gbps	120 mV	0.6 UI

To know if the system is meeting the requirements, the eye mask can be imported. Specifications of eye mask depend on the application the system is using. For example, PCIe 1.0, PCIe 2.0, XAUI, and SGMII. A typical eye mask for PCIe 2.0 is shown in the following figure.

Figure 2-37. Eye Mask for PCIe 2.0



For more information about IBIS-AMI model, see AC292- IBIS Models- Background and Usage Application Note.

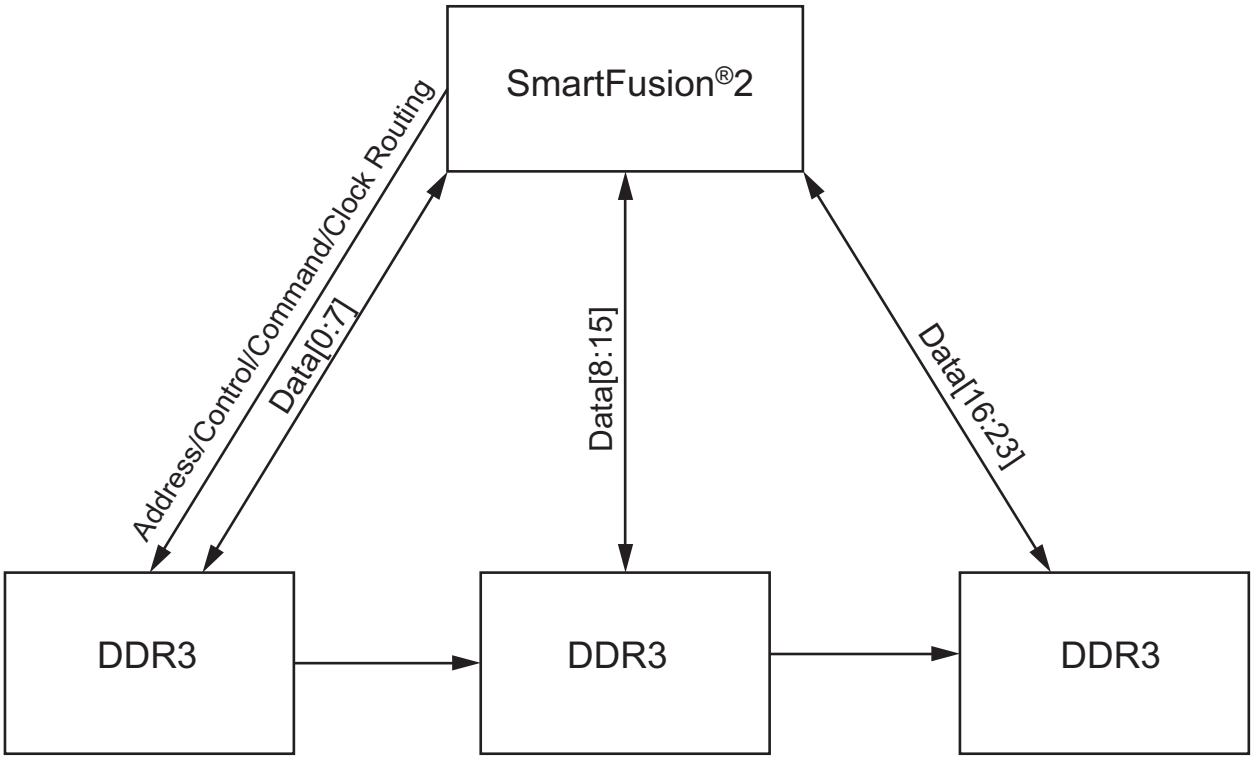
2.10 DDR3 Layout Guidelines (Ask a Question)

2.10.1 Placement (Ask a Question)

Ensure that the placement for the DDR3 memories looks like L (shape), where, memories are at the bottom of the L and controllers are on the top of the L. This gives enough space to route the DQ signals with less number of layers.

Important: This is not mandatory to follow the suggested placement. However, the placement also depends on the board constraints. The maximum trace length of any signal in the placement must not be more than 7 inches.

Figure 2-38. DDR3 Memories



The termination resistors are not required for the DQ and DQS signals as these signals have on-chip ODTs. The termination resistors are placed at the end of the address, command, control, and clock signals as these signals use fly-by topology. VTT plane/island is thick enough to handle the current required by termination resistors; at least 150 mil trace is required. The sense pin of VTT regulator must be connected at the center of the VTT island.

2.10.2 Routing (Ask a Question)

Reliability of DDR interface depends on the quality of the layout. There are many layout guidelines available from memory vendors. The following recommendations can also be used for routing the DDR3 signals. The following DDR3 signals are grouped.

- Data
- Address/Command
- Control
- Clocks
- Power

The following table lists the signals that come under a particular group:

Table 2-2. Grouping of DDR3 Signals

Group	Signals
Data	DQ[0:7], DQ[8:15], DQ[16:23], DQ[24:31] and DQS[0:3], DM[0:3]
Address/command	A[0:15], BA[0:2], RAS#, CAS#, and WE#
Control	CS#, CKE, and ODT
Clock	CK and CK#

2.10.2.1 Data Group Signal Routing (Ask a Question)

- The data signals must not be over the split planes.
- The reference plane for data signals must be GND plane and must be contiguous between memory and SmartFusion2/IGLOO2.
- Traces must not be routed at the edge of the reference plane and over via anti pads.
- When routing the data signals, the longest signals must be routed first, this allows to adjust the length for the short length signals, when routing data signals.
- Serpentine routing must be used to adjust the data group signals to meet this requirement.
- The DQS signal must be routed along with associated data byte lane on the same critical layer with the same via count. Using more than three vias in the connection between the FPGA controller and memory device must be avoided.
- The impedance for the data traces depends on the stack-up and the trace width. There are options to select the impedance based on the stack-up and trace width.
  - 40 Ω impedance, which requires wide traces (~7 to 8 mils). This gives the less cross talk and less spacing between the traces (~2x). Spacing between non-DDR signals and DDR signals must be ~4x.
  - 50 Ω impedance, which requires smaller trace width (~4 to 6 mils). This requires more spacing between the traces (~3x). Spacing between non DDR signals and DDR signals must be ~4x.
- All data lanes must be matched to within 0.5 inch.
- Within the data lane, each trace must be matched to within ±10mils of its respective data strobe
- The DQS and DQS# need to be matched within ± 5mils.
- Differential impedance must be between 75 to 100 Ω.
- Differential traces adjacent to noisy signals or clock chips must be avoided.
- Spacing between differential lines must be 5 to 8 mils.

2.10.2.2 Address, Control, Command, and Clock Routing (Ask a Question)

- These signals must be routed in the fly-by topology and terminated with appropriate termination resistor at the end of the signals. The resistor termination must not have a stub longer than 600 mil.
- The impedance for the trace depends on the stack-up and trace width. There are options to select the impedance based on the stack-up and trace width:
  - 40 Ω impedance that needs wide traces (~7 to 8 mils). This gives the less cross talk and less spacing between the traces (~2x). Spacing between non DDR signals and DDR signals must be ~4x.
  - 50 Ω impedance that requires smaller trace width (~4 to 6 mils). This needs more spacing between the traces (~3x). Spacing between non DDR signals and DDR signals must be ~4w to avoid crosstalk issues.
  - Address and control signals can be referenced to a power plane if a ground plane is not available. The power plane must be related to the memory interface. However, a ground reference is preferred. Address and control signals must be kept on a different routing layer from DQ, DQS, and DM to isolate crosstalk between the signals.

2.10.2.3 Clock (Ask a Question)

- Clock signals are routed differentially, and the length matches between traces should be ±5 mils.
- It should be referenced to ground plane.
- The space between clock and other signals must be 25 mils.

- One clock signal is routed per rank of the Dual Inline Memory Module (DIMM), that is, one clock for single-ranked DIMM, two clock signals for the dual ranked DIMM. For non-DIMM systems, the differential terminations used by the CK/CK# pair must be located as close as possible to the memory.
- If more than one CS is used, the same clock to DQS skew must be applied to all CS.
- Address/control signals and the associated CK and CK# differential FPGA clock should be routed with trace matching ±100 mil.

Notes: The following guidelines are applicable for DDR2, DDR3, and LPDDR:

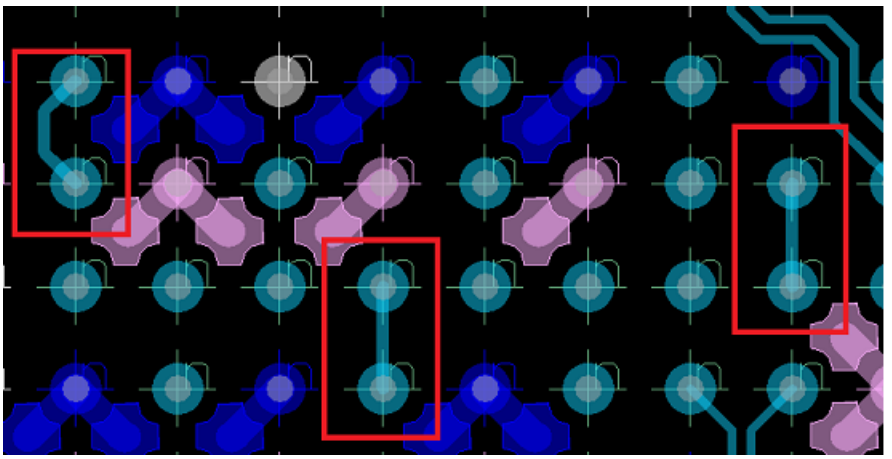
- Short the MDDR_TMATCH_0_IN and MDDR_TMATCH_0_OUT pins under BGA using short trace.
- Short the MDDR_TMATCH_1_IN and MDDR_TMATCH_1_OUT pins under BGA using short trace.
- Short the MDDR_TMATCH_ECC_IN and MDDR_TMATCH_ECC_OUT pins under BGA using short trace.
- Short the FDDR_TMATCH_0_IN and FDDR_TMATCH_0_OUT pins under BGA using short trace.
- Short the FDDR_TMATCH_1_IN and FDDR_TMATCH_1_OUT pins under BGA using short trace.
- Short the FDDR_TMATCH_ECC_IN and FDDR_TMATCH_ECC_OUT pins under BGA using short trace.

For more information about DDR2 and LPDDR memory layout guidelines, see the Micron Memory Layout Guidelines in the following documents, available on the *Micron website*:

- *TN-47-20: Hardware Tips for Point-to-Point System Design from Micron*
- *TN-46-19: Hardware Tips for Point-to-Point System Design from Micron*

The following figure shows an example layout.

Figure 2-39. TMATCH Signals (Example Layout)



2.10.3 Simulation Considerations (Ask a Question)

Simulations ensure that the DDR and controller meet timing requirements and also ensure that the quality of the received waveform in terms of undershoot, overshoot and jitter and so on.

The following files are required for the DDR3 simulation:

- IBIS file of SmartFusion2/IGLOO2
- IBIS file of DDR3 memory
- PCB files of SmartFusion2/IGLOO2 board and DIMM, if used
- Connector models if DIMM is used

The following sections describe how to run the serial channel simulations.

2.10.3.1 Step 1: Gathering the Required Files (Ask a Question)

2.10.3.1.1 IBIS Models (Ask a Question)

To download the IBIS models of SmartFusion2/IGLOO2 and the IBIS-AMI models of DDR3 memory that interfaces with SmartFusion2/IGLOO2, see the following web pages on the Microchip website:

- www.microchip.com/en-us/products/fpgas-and-plds/system-on-chip-fpgas/smartfusion-2-fpgas#ibis
- www.microchip.com/en-us/products/fpgas-and-plds/fpgas/igloo-2-fpgas#ibis

2.10.3.1.2 PCB Trace Models (Ask a Question)

The PCB file needs to be converted into a compatible format of simulator software. For example, .HYP file format of PCB is needed to simulate in Hyperlynx and SPD file format of PCB is required to simulate in Sigrity. Once the PCB file is loaded in the simulation tool, check the stack-up that matches the PCB stack-up and define the dielectric constant, Dk and dissipation factor, and Df of PCB material. The tool extracts wrong models, if the preceding points are not defined properly. Some tools run the simulations on PCB file itself like Hyperlynx and some tools need S-parameter files of DDR3 traces to continue the simulations. To extract S-parameter models of PCB traces assign the ports on both sides of the traces and extract the S-parameter models of traces.

The following tools can be used to extract S-parameter models of PCB traces:

- Agilents ADS
- Mentors Hyperlynx
- Sigritys PowerSI

Important: It is not mandatory to use the preceding tools, there are many tools available in the market which can extract S-parameter models.

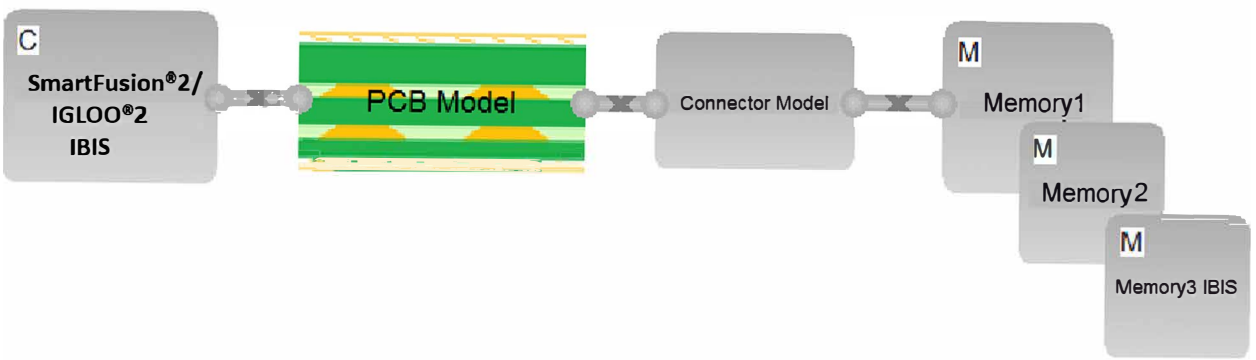
2.10.3.2 Step 2: Creating Simulation Topology (Ask a Question)

The following blocks are extracted from the Sigrity tool. Topology is the same in any tools. The simulation can be carried out in any tool, which supports DDR3 simulation.

The following figure shows the typical topology blocks involved in the DDR3 simulations.

- SmartFusion2/IGLOO2 IBIS: IBIS model of SmartFusion2/IGLOO2
- PCB: S-parameter model of PCB file, connector models and DIMM PCB models
- Connector model: Spice models of connector
- Memory IBIS: IBIS models of DDR3 memory

Figure 2-40. DDR3 Simulation Topology



2.10.3.3 Step 3: Simulation Setup (Ask a Question)

- Assign IBIS models to SmartFusion2/IGLOO2 and memory
- Assign connector model if used
- Assign the models for on board termination resistors
- Identify the DDR3 nets and classify according to data, control, and address bus
- Keep the appropriate ODT for SF2 and memory
- Keep the 40 to 60 Ω ODT for data and 80 Ω to 120 Ω for DQS
- Set the maximum frequency at which the system will operate (For SF2, it is 333 MHz).

2.10.3.4 Step 4: Results (Ask a Question)

Observe the following results:

- Setup and hold time between data signals and the respective DQS over all corners.
- Setup and hold time between Control/Command/Address signals and the clock over all corners.
- Overshoot and undershoot of all signals with respect to JEDEC specifications over all corners, and also DC threshold multi crossing that is due to the excessive ringing.

The simulation tool generates the report where all the details are available. For example, Hyperlynx generates the set of excel sheets which contain all setup and hold margin, overshoot, and undershoot information for all corners. It also generates driver and receiver waveforms for all the nets.

The following figure shows the file list where all the information regarding the simulation are stored.

Figure 2-41. List of Reports Generated by Hyperlynx

Name	Date modified	Type	Size
DDR_report_address_allcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	132 KB
DDR_report_address_violations_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	11 KB
DDR_report_address_worstcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	11 KB
DDR_report_data_allcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	104 KB
DDR_report_data_violations_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	6 KB
DDR_report_data_worstcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	11 KB
DDR_report_SI_measurements_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	11 KB
DDR_report_skew_allcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	51 KB
DDR_report_skew_violations_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	6 KB
DDR_report_skew_worstcases_Typ.xls	5/30/2013 12:37 PM	Microsoft Office E...	11 KB

The reports give setup and hold time for each net and also signal integrity details where overshoot and undershoots are mentioned. The following figure shows the example of report for DQ0 net. It also shows that the DQ0 has enough setup and holds time margins.

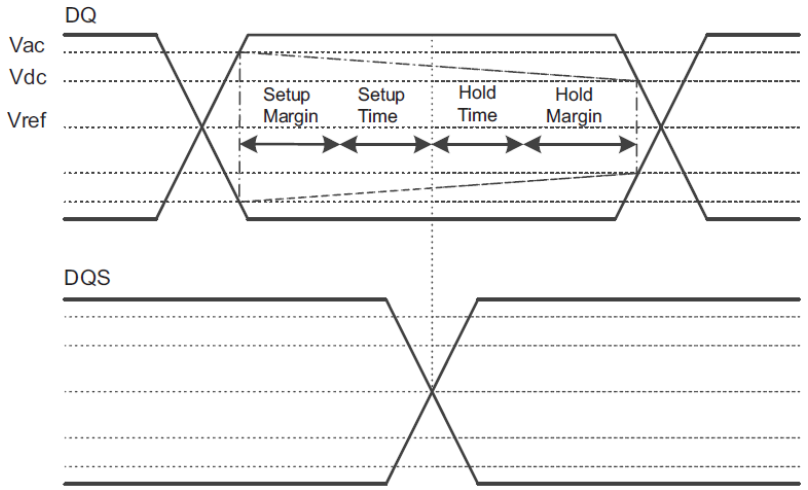
Figure 2-42. Setup and Time Margins of DQ0

	A	O	U
1	Net Name	Setup Time Margin, tDS(margin) = tDS(min) - tDS(req), [ps]	Hold Time Margin, tDH(margin) = tDH(min) - tDH(req), [ps]
2	DDR2_DQ0	N/A	333.3
3	DDR2_DQ0	267.6	N/A
4	DDR2_DQ0	N/A	316.6

If any of the net is violating the setup and holding time margins, the length of the net must be changed accordingly. If there is any high peak overshoot or undershoot, it might be because of the high value termination resistor. Adjust the value of ODT and re-iterate the simulation.

The following figure shows how to look at setup and hold time margins for DQ and DQS signals. The same is applicable to margin between the Command/Control/Address and CLK signals.

Figure 2-43. Setup and Time Margins for DQ and DQS Signals



2.11 References (Ask a Question)

- Power Distribution Network (PDN) by Eric Bogatin
- Sigrity PowerSI Tutorial

3. PCB Inspection Guidelines (Ask a Question)

As electronic components become more complex and dense, in smaller packages, the need for X-ray and Real-time X-ray inspection is increasing for use in failure analysis and in production. Real-time X-ray technology can be used to determine if there is a short between two or more pins in a device. FPGAs are dense devices that uses a high count of pins in a typical BGA package with fine pitches.

Set a limit of 15 rads using a zinc filter tray of exposure from a real-time X-ray source to avoid damaging the programming circuitry of IGLOO2 and SmartFusion2 FPGAs. At higher doses than 15 rads there is a higher risk that the device fails to program, device ages and could potentially have retention issues.

Tube voltage, power, current, duration, distance, and filter (shielding) all impact real-time X-ray radiation dose. In testing, a Quadra 7 Real-Time X-ray Inspection system from Nordson DAGE is used.

4. Creating Schematic Symbols Using Cadence OrCAD Capture CIS for SmartFusion2 and IGLOO2 Designs (Ask a Question)

Creating symbols manually for application can be a time consuming task and is error prone. Cadence OrCAD Capture is a popular schematic design entry tool for system-level design.

This section describes the recommended procedure to create schematic symbols. The schematic symbols can be created using two methods:

- Using the Microchip published pin assignment tables (PAT) in spreadsheet format. Pin Information is imported into OrCAD Capture CIS tool. This is a two step process as explained in Creating Schematic Symbols using Pin Assignment Tables (PAT). Use this method for creating schematic symbols using Microchip defined pins.
- Using the data export feature of Libero SoC software. Pin information can be exported to a spreadsheet and then use OrCAD Design Capture CIS tool to create symbols. This is a three step process as explained in Creating Schematic Symbols with User Defined Pin Names. Use this method for creating symbols using user defined pins for the fabric I/Os.

4.1 Creating Schematic Symbols using Pin Assignment Tables (PAT) (Ask a Question)

4.1.1 Preparing the PAT Layout File for Import into OrCAD Capture (Ask a Question)

1. Download the PAT files from the following path in the Microchip website:
   - www.microchip.com/en-us/products/fpgas-and-plds/system-on-chip-fpgas/smartfusion-2-fpgas#Documentation
   - www.microchip.com/en-us/products/fpgas-and-plds/fpgas/igloo-2-fpgas#Documentation
2. Open the *Pin_Assignment_Table_Public.xlsx file.

Figure 4-1. Example PAT Spreadsheet - Initial View

PKG.PIN	M2S050TS/M2S050T-FG896 Pin Name	Bank No	Direction	Total			Pin Parasitics (Applicable to Usable DIGITAL I/Os ONLY)						Unused Condition
				MSIOD (2.5V max)	MSIO (3.3V max)	DDRIO (2.5V max)	Net Length	SelfR(Ohm	SelfL(nH)	SelfC(pF)	SelfG(uMho)	Delay(pS)	
A10	DDRIO90PB0/MDDR_DQS_ECC	0	I/O	N	N	Y	14.7524	0.69979	9.75479	1.7336	0	130.042	Libero Defined* DNC
A11	DDRIO88PB0/MDDR_DQ_ECC3	0	I/O	N	N	Y	14.1504	0.712139	9.25784	1.62799	0	122.767	Libero Defined* DNC
A12	DDRIO86PB0/MDDR_DQ0	0	I/O	N	N	Y	13.4834	0.669683	9.24447	1.57524	0	120.674	Libero Defined* DNC
A13	DDRIO84PB0/MDDR_DQS0	0	I/O	N	N	Y	13.4861	0.673458	8.70731	1.57518	0	117.114	Libero Defined* DNC
A14	DDRIO83NB0/MDDR_DQ4	0	I/O	N	N	Y	13.2354	0.675006	8.99641	1.52207	0	117.018	Libero Defined* DNC
A15	DDRIO80PB0/MDDR_DQ8	0	I/O	N	N	Y	13.3326	0.66684	9.24054	1.54588	0	119.519	Libero Defined* DNC
A16	DDRIO78PB0/GB8/CCC_NE0_CLKI3/MD	0	I/O	N	N	Y	13.4351	0.694428	8.85242	1.57645	0	118.133	Libero Defined* DNC
A17	DDRIO76PB0/GB12/CCC_NE1_CLKI2/M	0	I/O	N	N	Y	13.3008	0.664554	9.24915	1.55802	0	120.043	Libero Defined* DNC
A18	DDRIO74PB0/MDDR_DQ16	0	I/O	N	N	Y	13.292	0.675689	8.72718	1.54978	0	116.298	Libero Defined* DNC
A19	DDRIO72PB0/MDDR_DQS2	0	I/O	N	N	Y	14.1135	0.678681	9.28264	1.62298	0	122.742	Libero Defined* DNC
A2	SERDES_1_TXD0_N		O	N	N	N							DNC
A20	DDRIO71NB0/MDDR_DQ20	0	I/O	N	N	Y	14.1485	0.684028	9.43141	1.64465	0	124.545	Libero Defined* DNC
A21	DDRIO68PB0/MDDR_DQ24	0	I/O	N	N	Y	14.677	0.725941	9.61066	1.69539	0	127.647	Libero Defined* DNC
A22	DDRIO66NB0/MDDR_DQS3_N	0	I/O	N	N	Y	15.0811	0.702886	9.71092	1.74809	0	130.29	Libero Defined* DNC
A23	DDRIO64PB0/MDDR_DQ28	0	I/O	N	N	Y	15.016	0.718421	9.5998	1.72399	0	128.647	Libero Defined* DNC
A24	DDRIO60PB0/MDDR_RESET_N	0	I/O	N	N	Y	15.0875	0.704002	9.88004	1.72551	0	130.568	Libero Defined* DNC
A25	DDRIO59PB0/MDDR_CLK	0	I/O	N	N	Y	15.502	0.722614	9.44108	1.80314	0	130.475	Libero Defined* DNC
A26	DDRIO57PB0/MDDR_BA2	0	I/O	N	N	Y	15.7885	0.706085	9.86659	1.77692	0	132.409	Libero Defined* DNC
A27	DDRIO55PB0/MDDR_ADDR3	0	I/O	N	N	Y	17.2149	0.75936	10.0156	1.95462	0	145.397	Libero Defined* DNC
A28	DDRIO55NB0/MDDR_ADDR4	0	I/O	N	N	Y	17.2833	0.763441	10.7336	1.95027	0	144.684	Libero Defined* DNC
A29	VSS			N	N	N							VSS

3. Go to the sheet that has the device name.
4. Retain the following columns and delete the remaining columns:
   - PGK.PIN
   - <Device> Pin Name
   - Direction
5. Insert a row at the beginning of the worksheet. See the following figure.

Figure 4-2. Example PAT Spreadsheet - Editing Stage

	A	B	C	D	E	F	G	H
1								
2	A10	DDRIO90PB0/MDDR_DQS_ECC	I/O					
3	A11	DDRIO88PB0/MDDR_DQ_ECC3	I/O					
4	A12	DDRIO86PB0/MDDR_DQ0	I/O					
5	A13	DDRIO84PB0/MDDR_DQS0	I/O					
6	A14	DDRIO83NB0/MDDR_DQ4	I/O					
7	A15	DDRIO80PB0/MDDR_DQ8	I/O					
8	A16	DDRIO78PB0/GB8/CCC_NE0_CLKI3/MDDR_DQS1	I/O					
9	A17	DDRIO76PB0/GB12/CCC_NE1_CLKI2/MDDR_DQ12	I/O					
10	A18	DDRIO74PB0/MDDR_DQ16	I/O					
11	A19	DDRIO72PB0/MDDR_DQS2	I/O					
12	A2	SERDES_1_TXD0_N	O					
13	A20	DDRIO71NB0/MDDR_DQ20	I/O					
14	A21	DDRIO68PB0/MDDR_DQ24	I/O					
15	A22	DDRIO66NB0/MDDR_DQS3_N	I/O					
16	A23	DDRIO64PB0/MDDR_DQ28	I/O					
17	A24	DDRIO60PB0/MDDR_RESET_N	I/O					
18	A25	DDRIO59PB0/MDDR_CLK	I/O					
19	A26	DDRIO57PB0/MDDR_BA2	I/O					
20	A27	DDRIO55PB0/MDDR_ADDR3	I/O					
21	A28	DDRIO55NB0/MDDR_ADDR4	I/O					
22	A29	VSS						
23	A3	VSS						
24	A4	SERDES_1_TXD1_N	O					
25	A5	VSS						
26	A6	SERDES_1_TXD2_N	O					
27	A7	VSS						
28	A8	SERDES_1_TXD3_N	O					
29	A9	DDRIO91PB0/GB0/CCC_NW0_CLKI3	I/O					
30	AA1	MSIOD134NB7	I/O					
31	AA10	CCC_SW0_PLL_VSSA						
32	AA11	SERDES_0_L01_VDDAIO						
33	AA12	SERDES_0_L23_VDDAIO						
34	AA13	VDDI5						
35	AA14	VDDI5						
36	AA15	VDDI5						
37	AA16	VDDI5						
38	AA17	VDDI5						
39	AA18	VDDI5						
40	AA19	VDDI5						
41	AA2	MSIOD134PB7	I/O					

6. Add the following headings for the columns. See Figure 4-3:
   - Number
   - Name
   - Type
   - Pin Visibility
   - Shape
   - Pin Group
   - Position
   - Section

   For Type, Shape, Position, and Section columns, enter the information manually to avoid warnings from the OrCAD Capture tool.
7. For **Type** column, choose and type one of the following options:
   - Replace I with Input

- Replace O with Output
- Replace I/O with Bidirectional

Other pin types for the remaining pins:

- 3-State
- Open Collector
- Open Emitter
- Passive (Unused pins like DNC or NC)
- Power (Supply and ground pins)

8. Leave the **Pin Visibility** column blank. The check boxes are automatically populated in the **New Part Creation** dialog in the OrCAD Capture tool. See Figure 4-6.
9. In the **Shape** column, enter one of the following shapes according to the requirement:
   - Clock
   - Dot
   - Dot-Clock
   - Line
   - Short Clock
   - Short Dot
   - Short Dot clock
   - Short
   - Zero Length

   The default shape for most of the FPGA symbol pins is LINE.
10. Leave the **Pin Group** column blank.
11. In the **Position** column, enter one of the following positions according to the requirement:
    - Bottom
    - Left
    - Right
    - Top
12. In the **Section** column, enter either a number or an alphabet based on the selection made for the Part Numbering option. OrCAD Capture supports two Part Numbering options, that is 1,2,3,4, and so on., for Numeric option and A, B, C, D, and so on., for Alphabetical option. See Figure 4-6.
13. **Save** the Excel file with an appropriate name.

Figure 4-3. Example PAT Spreadsheet - Final Stage

	A	B	C	D	E	F	G	H
1	Number	Name	Type	Pin Visibility	Shape	Pin Group	Position	Section
2	A1	VSS	Power		Line		Right	I
3	A10	DDRIO75NB0/MDDR_DQ15	Bidirectional		Line		Right	A
4	A11	VSS	Power		Line		Right	L
5	A12	DDRIO83PB0/MDDR_DM_RDQS0	Bidirectional		Line		Right	A
6	A13	DDRIO83NB0/MDDR_DQ4	Bidirectional		Line		Right	A
7	A14	DDRIO82PB0/MDDR_DQ5	Bidirectional		Line		Right	A
8	A15	DDRIO84PB0/MDDR_DQS0	Bidirectional		Line		Right	A
9	A16	VSS	Power		Line		Right	L
10	A17	DDRIO88NB0/MDDR_DQ_ECC2	Bidirectional		Line		Right	A
11	A18	DDRIO88PB0/MDDR_DQ_ECC3	Bidirectional		Line		Right	A
12	A19	DDRIO90PB0/MDDR_DQS_ECC	Bidirectional		Line		Right	A
13	A2	DDRIO53NB0/MDDR_ADDR7	Bidirectional		Line		Left	A
14	A20	DDRIO92PB0/MDDR_TMATCH_ECC_OUT	Bidirectional		Line		Right	A
15	A21	VSS	Power		Line		Right	L
16	A3	DDRIO53PB0/MDDR_ODT	Bidirectional		Line		Left	A
17	A4	DDRIO57NB0/MDDR_ADDR0	Bidirectional		Line		Left	A
18	A5	DDRIO55PB0/MDDR_ADDR3	Bidirectional		Line		Left	A
19	A6	VSS	Power		Line		Right	L
20	A7	DDRIO59NB0/MDDR_CLK_N	Bidirectional		Line		Left	A
21	A8	DDRIO62PB0/MDDR_RAS_N	Bidirectional		Line		Left	A
22	A9	DDRIO62NB0/MDDR_WE_N	Bidirectional		Line		Left	A
23	AA1	VSS	Power		Line		Right	L
24	AA10	DDRIO147NB5/CCC_SW0_CLKI2	Bidirectional		Line		Right	F
25	AA11	DDRIO148NB5/PROBE_B	Bidirectional		Line		Right	F
26	AA12	VSS	Power		Line		Right	L
27	AA13	SERDES_0_TXD2_N	Output		Line		Right	J
28	AA14	VSS	Power		Line		Right	L
29	AA15	SERDES_0_RXD2_P	Input		Line		Left	J
30	AA16	VSS	Power		Line		Right	L
31	AA17	SERDES_0_TXD0_P	Output		Line		Right	J
32	AA18	VSS	Power		Line		Right	L
33	AA19	SERDES_0_RXD0_N	Input		Line		Left	J
34	AA2	MSIO2PB3/USB_STP_B	Bidirectional		Line		Right	D
35	AA20	VSS	Power		Line		Right	L
36	AA21	MSIOD145NB6/SERDES_0_REFCLK0_N	Bidirectional		Line		Right	G
37	AA3	JTAG_TMS/M3_TMS/M3_SWDIO	Input		Line		Left	E
38	AA4	XTLOSC_MAIN_XTAL	Clock		Line		Left	J
39	AA5	NC	Passive		Line		Right	L
40	AA6	DDRIO186PB5/FDDR_ODT	Bidirectional		Line		Right	F
41	AA7	DDRIO186NB5/FDDR_ADDR7	Bidirectional		Line		Right	F

Recommendations for arranging pins in the **Section** column:

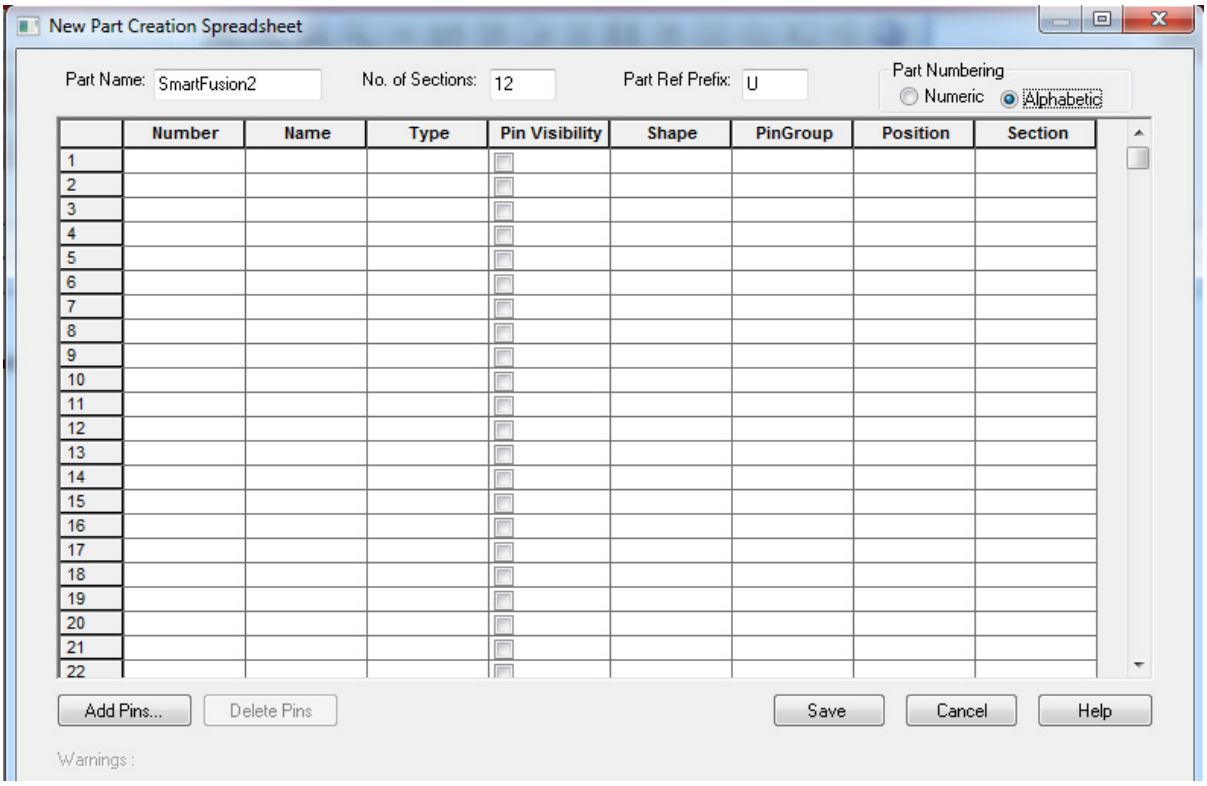
- Arrange individual bank pins in separate sections
- Arrange all power supply pins in one section
- Arrange all ground pins in one section
- All passive pins can be in one section
- Arrange the remaining pins like Clock, JTAG, and SerDes in one section.

4.1.2 Generating a OrCAD Capture Schematic Symbol (Ask a Question)

1. Invoke Cadence OrCAD Capture CIS. 2. . 3. 4.
2. Go to File > New > Library, then select the `*.olb` file.

3. Go to Design > New Part from Spreadsheet... The New Part Creation Spreadsheet dialog appears as shown in the following figure.
4. In the New Part Creation Spreadsheet dialog, specify the following:
   - Part Name
   - Number of sections
   - Part Ref Prefix - choose Alphabetic

Figure 4-4. New Part Creation Spreadsheet Dialog



5. From the Example PAT Spreadsheet, select and copy all the cells, excluding the column headers as shown in the following figure.

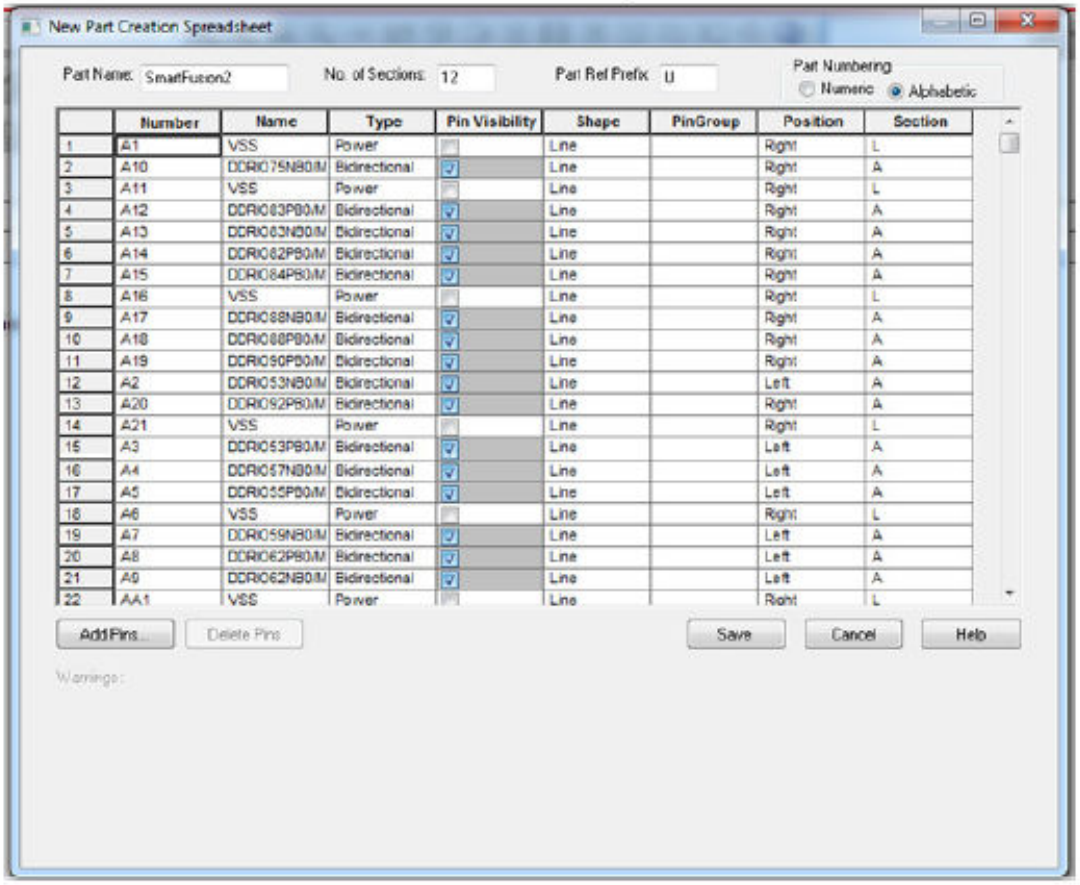
Figure 4-5. Example PAT Spreadsheet - Final Stage

	A	B	C	D	E	F	G	H
1	Number	Name	Type	Pin Visibility	Shape	Pin Group	Position	Section
2	A1	VSS	Power		Line		Right	L
3	A10	DDRIO75NB0/MDDR_DQ15	Bidirectional		Line		Right	A
4	A11	VSS	Power		Line		Right	L
5	A12	DDRIO83PB0/MDDR_DM_RDQS0	Bidirectional		Line		Right	A
6	A13	DDRIO83NB0/MDDR_DQ4	Bidirectional		Line		Right	A
7	A14	DDRIO82PB0/MDDR_DQ5	Bidirectional		Line		Right	A
8	A15	DDRIO84PB0/MDDR_DQS0	Bidirectional		Line		Right	A
9	A16	VSS	Power		Line		Right	L
10	A17	DDRIO88NB0/MDDR_DQ_ECC2	Bidirectional		Line		Right	A
11	A18	DDRIO88PB0/MDDR_DQ_ECC3	Bidirectional		Line		Right	A
12	A19	DDRIO90PB0/MDDR_DQS_ECC	Bidirectional		Line		Right	A
13	A2	DDRIO53NB0/MDDR_ADDR7	Bidirectional		Line		Left	A
14	A20	DDRIO92PB0/MDDR_TMATCH_ECC_OUT	Bidirectional		Line		Right	A
15	A21	VSS	Power		Line		Right	L
16	A3	DDRIO53PB0/MDDR_ODT	Bidirectional		Line		Left	A
17	A4	DDRIO57NB0/MDDR_ADDR0	Bidirectional		Line		Left	A
18	A5	DDRIO55PB0/MDDR_ADDR3	Bidirectional		Line		Left	A
19	A6	VSS	Power		Line		Right	L
20	A7	DDRIO59NB0/MDDR_CLK_N	Bidirectional		Line		Left	A
21	A8	DDRIO62PB0/MDDR_RAS_N	Bidirectional		Line		Left	A
22	A9	DDRIO62NB0/MDDR_WE_N	Bidirectional		Line		Left	A
23	AA1	VSS	Power		Line		Right	L
24	AA10	DDRIO147NB5/CCC_SW0_CLKI2	Bidirectional		Line		Right	F
25	AA11	DDRIO148NB5/PROBE_B	Bidirectional		Line		Right	F
26	AA12	VSS	Power		Line		Right	L
27	AA13	SERDES_0_TXD2_N	Output		Line		Right	J
28	AA14	VSS	Power		Line		Right	L
29	AA15	SERDES_0_RXD2_P	Input		Line		Left	J
30	AA16	VSS	Power		Line		Right	L
31	AA17	SERDES_0_TXD0_P	Output		Line		Right	J
32	AA18	VSS	Power		Line		Right	L
33	AA19	SERDES_0_RXD0_N	Input		Line		Left	J
34	AA2	MSIO2PB3/USB_STP_B	Bidirectional		Line		Right	D
35	AA20	VSS	Power		Line		Right	L
36	AA21	MSIOD145NB6/SERDES_0_REFCLK0_N	Bidirectional		Line		Right	G
37	AA3	JTAG_TMS/M3_TMS/M3_SWDIO	Input		Line		Left	E
38	AA4	XTLOSC_MAIN_XTAL	Clock		Line		Left	J
39	AA5	NC	Passive		Line		Right	L
40	AA6	DDRIO186PB5/FDDR_ODT	Bidirectional		Line		Right	F
41	AA7	DDRIO186NB5/FDDR_ADDR7	Bidirectional		Line		Right	F

6. Select the top left cell of **New Part Creation Spreadsheet** dialog and paste the copied data. Check if all the columns match between the **Example PAT Spreadsheet** and **New Part Creation Spreadsheet** dialog as shown in the following figure.

Important: In the Pin Visibility column, select all the check boxes. Some of the check boxes for the power pins might not be selected. If you want those pins to be visible, ensure that they are selected.

Figure 4-6. New Part Creation Spreadsheet Dialog with Data



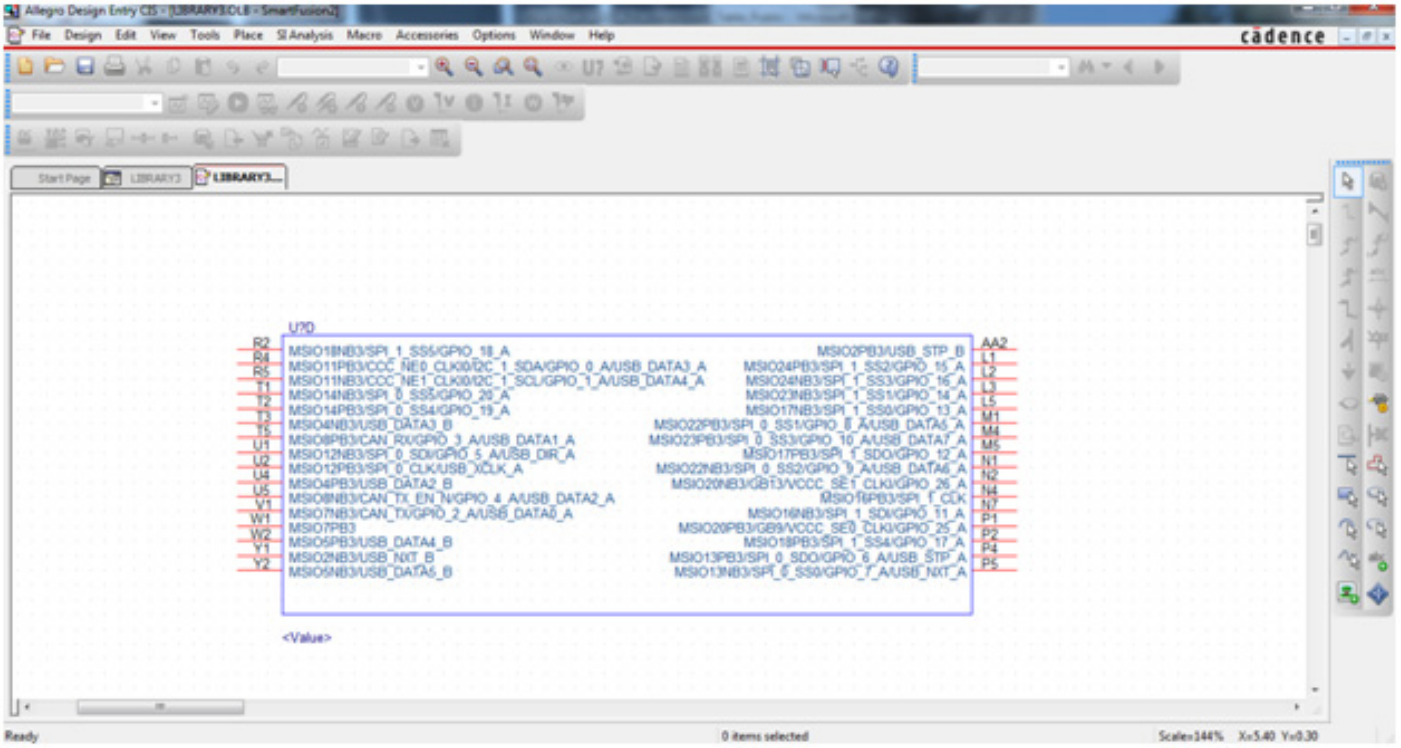
7. Click **Save**

Important: When you click save, the Design Rule Check (DRC) operation is triggered. If there are any errors reported during the DRC, modify the Example PAT Spreadsheet to fix those errors. If there are just warnings and if you want to ignore them, click **Continue** to proceed with generating the Part.

Figure 4-7. New Part Created in the Library

8. Double-click the part that is created to display the first section as shown in the following figure.

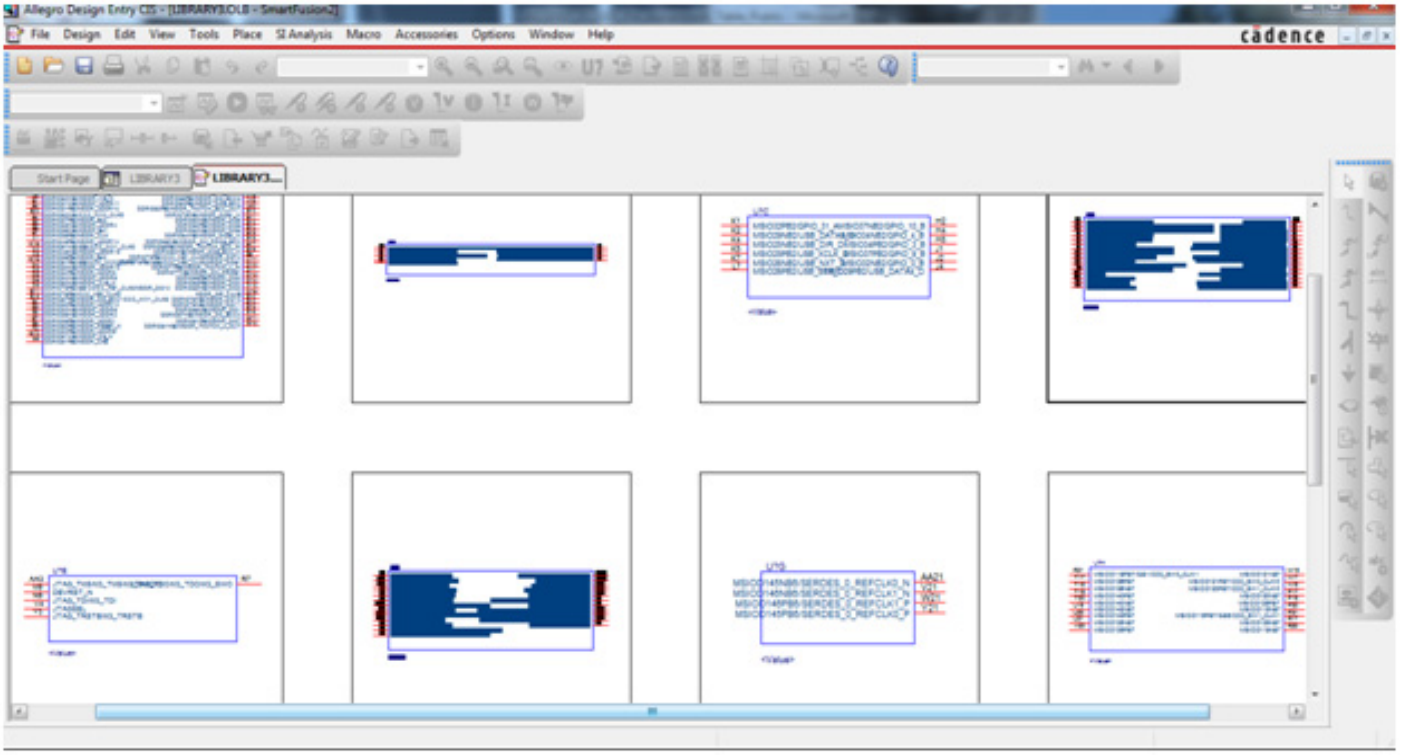
Figure 4-8. Schematic Symbol - First Section



9. You can navigate to all the sections that are created.
   - To go to the next part, go to **View > Next Part** or press Ctrl + N.
   - To go to the previous part, go to **View > Previous Part** or press Ctrl + B.
   - To see all the blocks of the schematic symbol, go to **View > Package**.

   The schematic symbol is ready to use. The following figure shows all the blocks of the schematic symbol that are generated from the Example PAT spreadsheet.

Figure 4-9. Package View of the Schematic Symbol

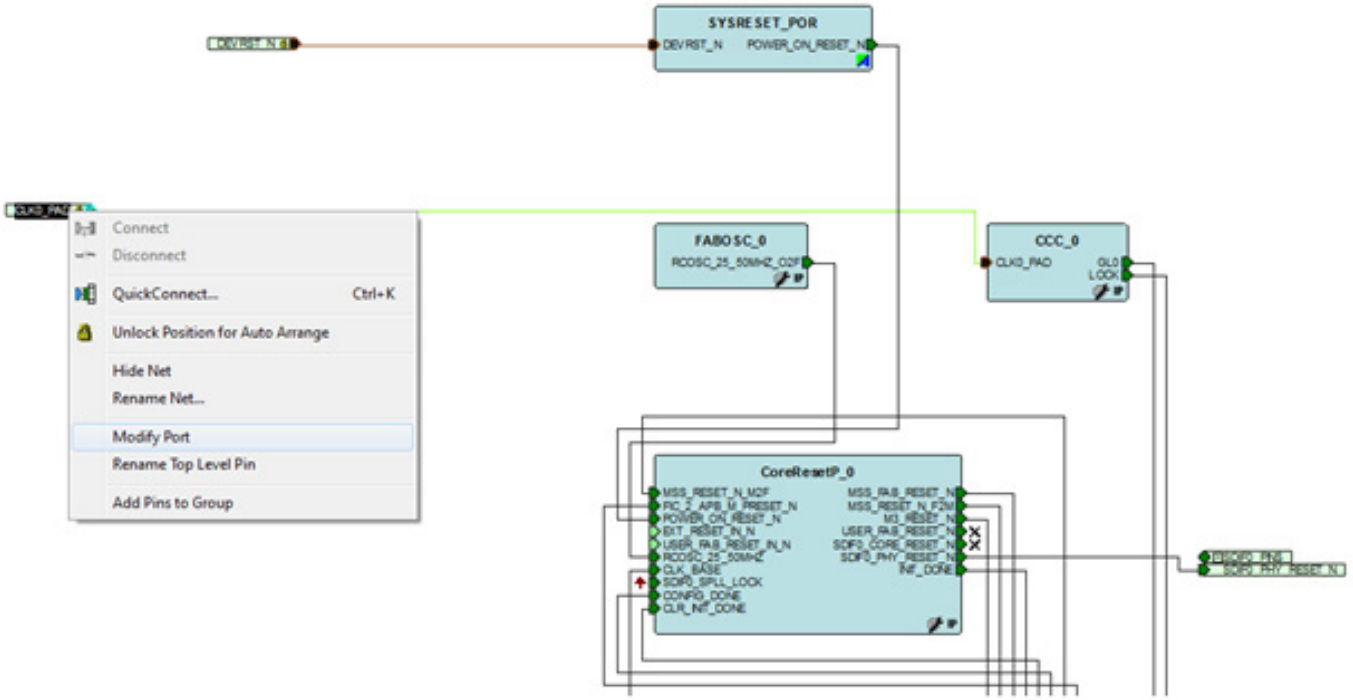


4.2 Creating Schematic Symbols with User Defined Pin Names (Ask a Question)

4.2.1 Exporting Pin Information from the Libero Design (Ask a Question)

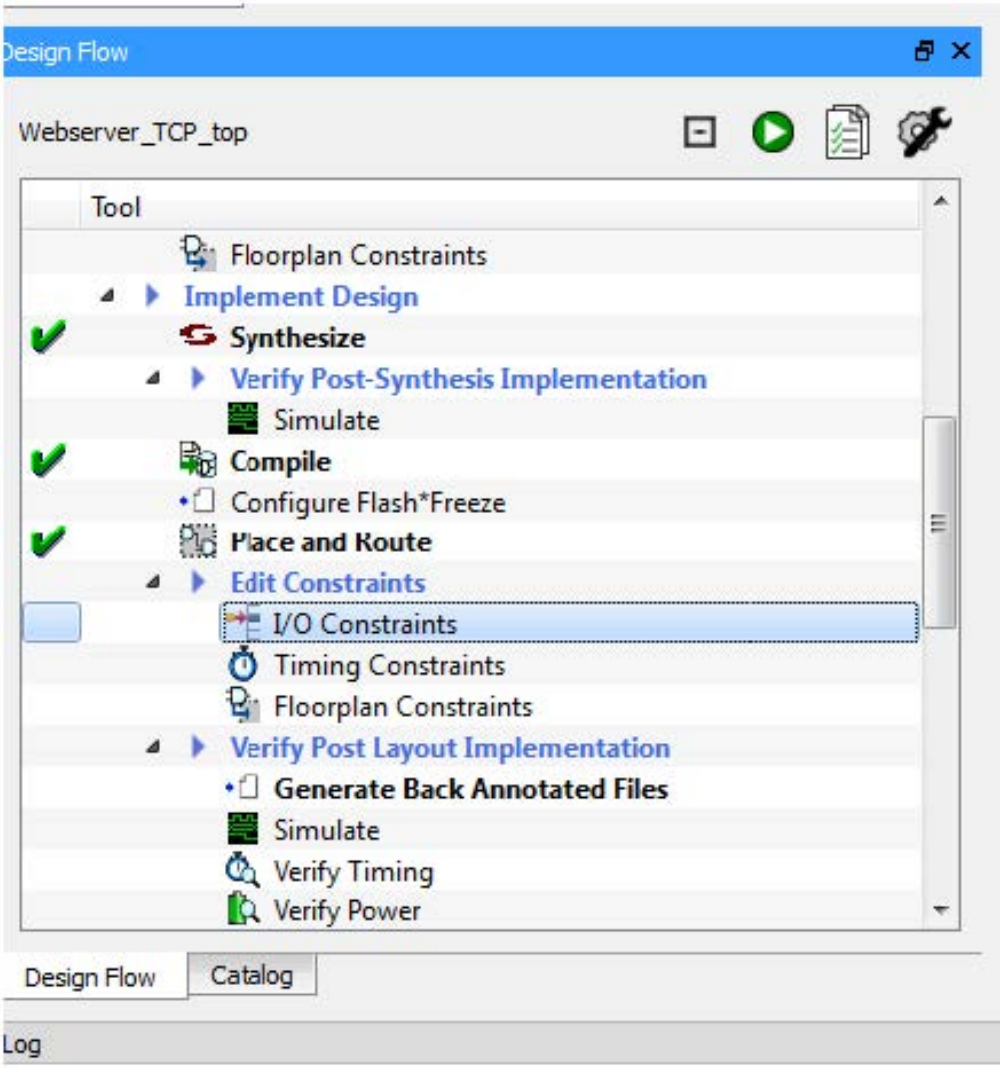
1. Launch Libero and open project. For more information about how to use the System Builder wizard in the Libero design, see
   - coredocs.s3.amazonaws.com/Actel/Tool/SysBuilder/sf2_system_builder_ug_1.pdf
   - coredocs.s3.amazonaws.com/Libero/Tool/SysBuilder/igl2_system_builder_ug_1.pdf
2. Right click port to change the name and click **Modify Port** as shown in the following figure. Change the name according to the requirement.

Figure 4-10. Modifying Port Names



3. Check and verify all the pin names in the design. To verify the pin names, double click **I/O Constraints** in the **Design Flow** tab as shown in the following figure.

Figure 4-11. I/O Constraints



The **I/O Editor** dialog lists the port names with the updated pin names as shown in the following figure. The pin names that are not modified follow the Microchip pin naming convention.

Important: For the I/O Editor dialog to open, the design must be synthesized and compiled.

Figure 4-12. I/O Editor

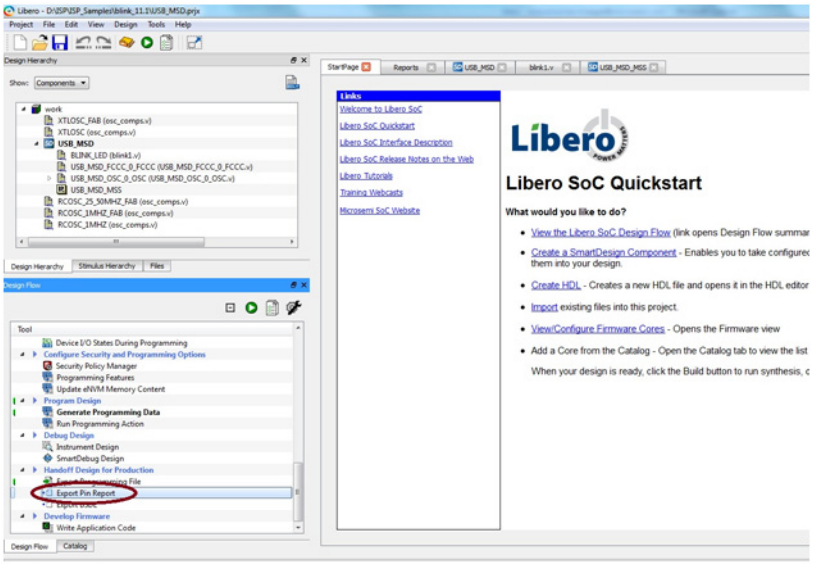
I/O Editor

Port Name ▲	Direction	I/O Standard	Pin Number	Locked	Bank Name	I/O state in Flash*Freeze mode
CLK0_PAD	Input	LVCMOS25	U7	☑	Bank7	TRISTATE
DEVRST_N	Input		AC27	☑	--	--
G4M_RCVRD_CLK1	Output	LVCMOS33	P9	☑	Bank8	TRISTATE
G4M_RCVRD_CLK2	Output	LVCMOS33	R9	☑	Bank8	TRISTATE
G4M_SYNCE_IN	Input	LVCMOS33	P3	☑	Bank8	TRISTATE
G4M_SYNCE_OUT	Output	LVCMOS25	L23	☑	Bank1	TRISTATE
GPIO_8_M2F	Output	LVCMOS25	A18	☑	Bank0	TRISTATE
GPIO_9_M2F	Output	LVCMOS25	B18	☑	Bank0	TRISTATE
GPIO_10_M2F	Output	LVCMOS25	D18	☑	Bank0	TRISTATE
GPIO_11_M2F	Output	LVCMOS25	E18	☑	Bank0	TRISTATE
GPIO_12_M2F	Output	LVCMOS25	A20	☑	Bank0	TRISTATE
GPIO_13_M2F	Output	LVCMOS25	D20	☑	Bank0	TRISTATE
GPIO_14_M2F	Output	LVCMOS25	E20	☑	Bank0	TRISTATE
GPIO_15_M2F	Output	LVCMOS25	B20	☑	Bank0	TRISTATE
MMUART_1_RXD_F2M	Input	LVCMOS25	R29	☑	Bank3	TRISTATE
MMUART_1_TXD_M2F	Output	LVCMOS25	R24	☑	Bank3	TRISTATE
PHY_MDC	Output	LVCMOS33	N5	☑	Bank8	TRISTATE
PHY_MDIO	Inout	LVCMOS33	R7	☑	Bank8	TRISTATE
PHY_RST	Output	LVCMOS33	N4	☑	Bank8	TRISTATE
REFCLK1_P	Input	LVDS	AF1	☑	Bank6	TRISTATE
REFCLK1_N	Input	LVDS	AE1	☑	Bank6	TRISTATE
RXD0_N	Input		AB9	☑	--	--
RXD0_P	Input		AB8	☑	--	--
RXD1_N	Input		AC10	☑	--	--
RXD1_P	Input		AC9	☑	--	--
RXD2_N	Input		AB11	☑	--	--
RXD2_P	Input		AB10	☑	--	--
RXD3_N	Input		AD11	☑	--	--

Ports | Package Pins | Package Viewer

4. To export the pin names, double-click **Export Pin Report** in the **Design Flow** tab. This report has the file extension `.rpt`.

Figure 4-13. Exporting Pin Information from Libero

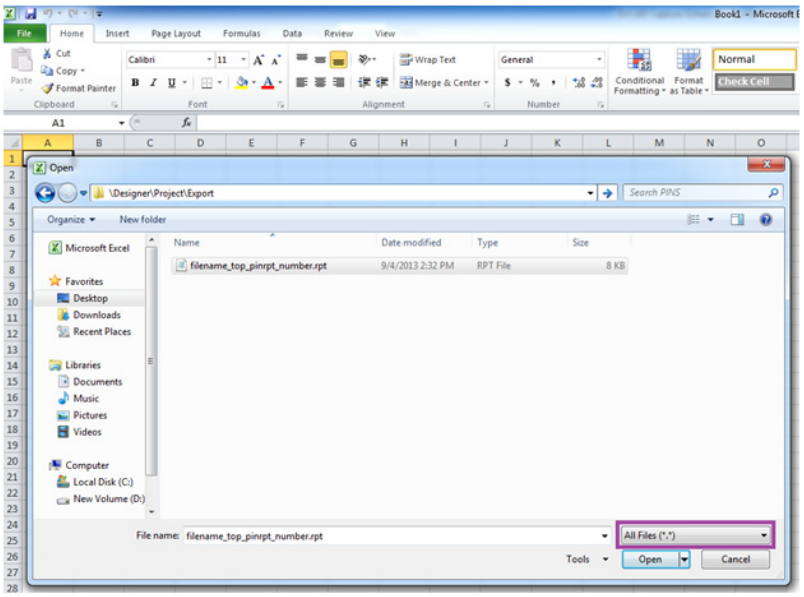


The report is stored in the project directory. The path is: `<Libero Project Directory>\designer\project\export\<project Name>_top_pinrpt_number.rpt`.

4.2.2 Preparing the Pin List for Import into OrCAD Capture CIS (Ask a Question)

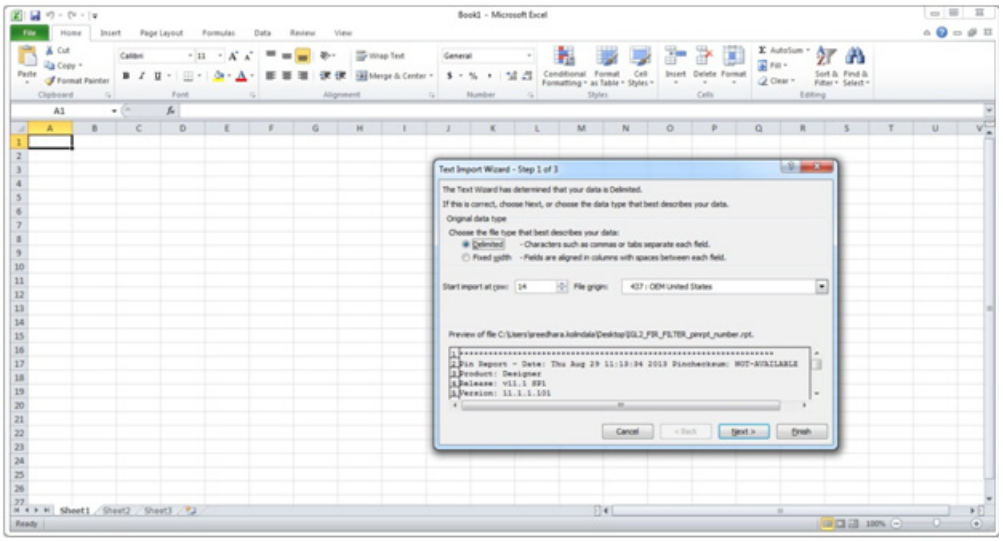
1. Launch Microsoft Excel, and open the `*.rpt` file that has the exported pin information. Before opening, ensure that **All Files (*.*)** is the file type as shown in the following figure.

Figure 4-14. Importing Pin Names to the Spreadsheet



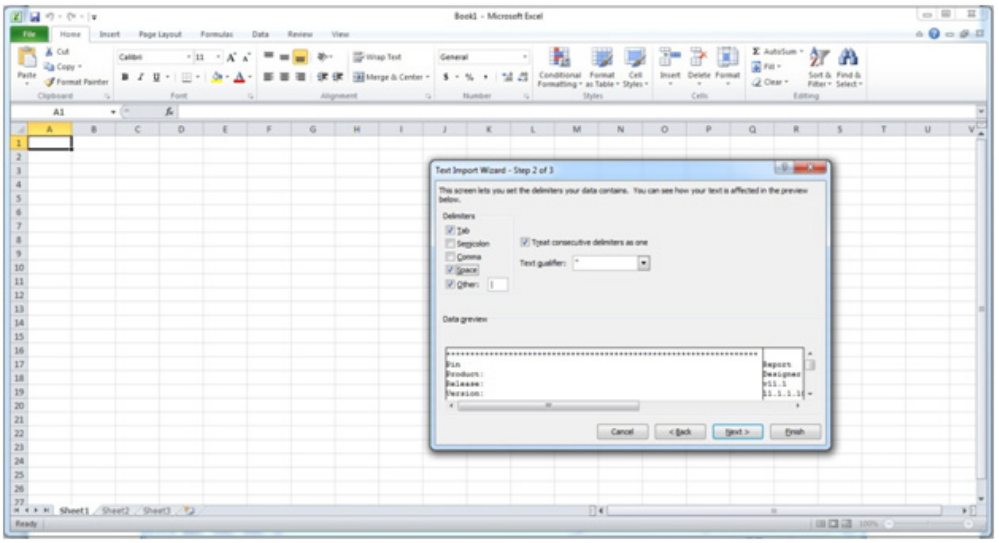
2. In the **Text Import Wizard - Step 1** dialog, select **Delimited**, start import at page 14 and click **Next**. The first 14 rows of the spreadsheet have data unrelated to the pin information. The following figure shows the **Text Import Wizard** with the **Delimited** option selected and the **Start Import at Page** option having the value 14.

Figure 4-15. Importing Pin Names to the Spreadsheet—Step 1



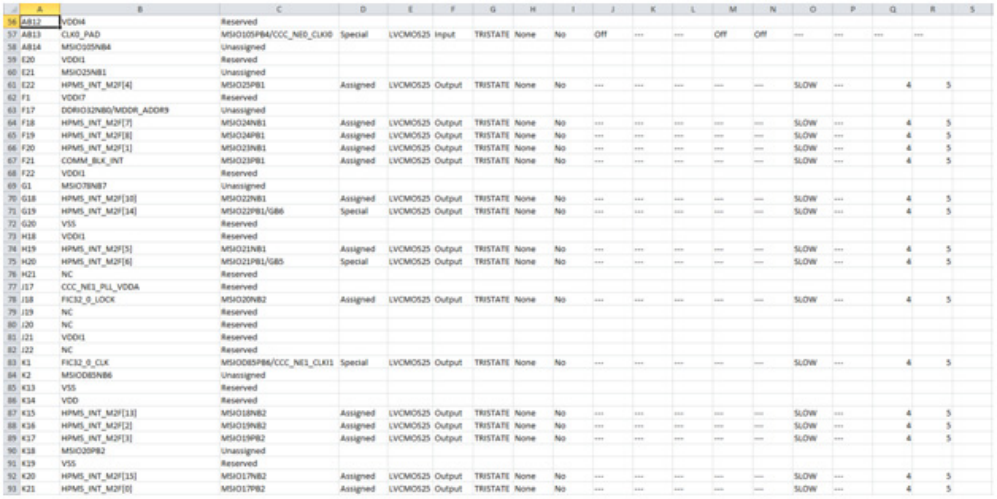
3. In the **Text Import Wizard - Step 2** dialog, select the following as **Delimiters** and click **Next**:
   - Tab
   - Space
   - I as other

Figure 4-16. Importing Pin Names to the Spreadsheet—Step 2



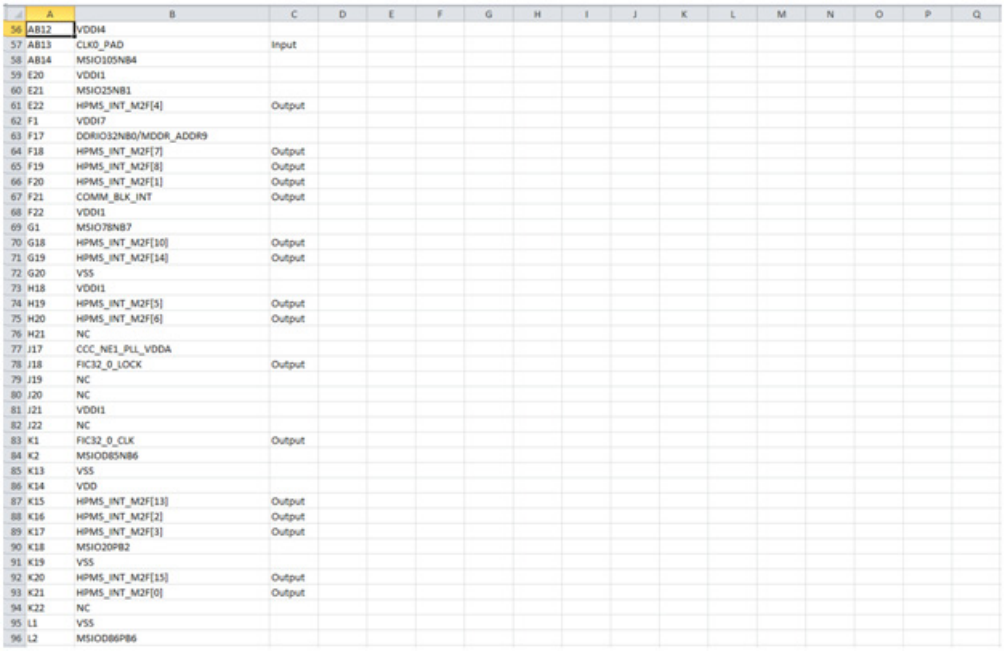
4. Click **Finish** to import the data in separate columns.

Figure 4-17. Importing Pin Names to the Spreadsheet—Final Step



5. Retain the columns A, B, F and delete the remaining columns as they are not required for generating schematic symbols.

Figure 4-18. Spreadsheet with the Pin Names Imported



	A	B	C
56	AB12	VDDI4	
57	AB13	CLK0_PAD	Input
58	AB14	MSIO105NB4	
59	E20	VDDI1	
60	E21	MSIO25NB1	
61	E22	HPMS_INT_M2F[4]	Output
62	F1	VDDI7	
63	F17	DDRIO32NB0/MDDR_ADDR9	
64	F18	HPMS_INT_M2F[7]	Output
65	F19	HPMS_INT_M2F[8]	Output
66	F20	HPMS_INT_M2F[1]	Output
67	F21	COMM_BLK_INT	Output
68	F22	VDDI1	
69	G1	MSIO78NB7	
70	G18	HPMS_INT_M2F[10]	Output
71	G19	HPMS_INT_M2F[14]	Output
72	G20	VSS	
73	H18	VDDI1	
74	H19	HPMS_INT_M2F[5]	Output
75	H20	HPMS_INT_M2F[6]	Output
76	H21	NC	
77	J17	CCC_NE1_PLL_VDDA	
78	J18	FIC32_0_LOCK	Output
79	J19	NC	
80	J20	NC	
81	J21	VDDI1	
82	J22	NC	
83	K1	FIC32_0_CLK	Output
84	K2	MSIOD85NB6	
85	K13	VSS	
86	K14	VDD	
87	K15	HPMS_INT_M2F[13]	Output
88	K16	HPMS_INT_M2F[2]	Output
89	K17	HPMS_INT_M2F[3]	Output
90	K18	MSIO20PB2	
91	K19	VSS	
92	K20	HPMS_INT_M2F[15]	Output
93	K21	HPMS_INT_M2F[0]	Output
94	K22	NC	
95	L1	VSS	
96	L2	MSIOD86PB6	

6. Add the following headings for the columns. See Figure 4-19.
   - Number
   - Name
   - Type
   - Pin Visibility
   - Shape
   - Pin Group
   - Position

- Section

By default **Number**, **Name** and **Type** columns are populated from the report. For **Type**, **Shape**, **Position**, and **Section** columns, add information manually to avoid warnings from the OrCAD Capture tool.

7. For Type column, choose and type one of the following options:
   - 3-State
   - Bidirectional
   - Input
   - Open Collector
   - Open Emitter
   - Output
   - Passive (Unused pins like DNC or NC)
   - Power (Supply and ground pins)
8. Leave the **Pin Visibility** column blank. The check boxes are automatically populated in the **New Part Creation** dialog.
9. In the **Shape** column, enter one of the following shapes according to the requirement:
   - Clock
   - Dot
   - Dot-Clock
   - Line
   - Short Clock
   - Short Dot
   - Short Dot Clock
   - Short
   - Zero Length

   The default shape for most of the FPGA symbol pins is the `LINE`.
10. Leave the **Pin Group** column blank.
11. In the **Position** column, enter one of the following positions according to the requirement:
    - Bottom
    - Left
    - Right
    - Top
12. In the **Section** column, enter either a number or an alphabet based on the selection made for the Part Numbering option. OrCAD Capture supports two Part Numbering options, that is, 1, 2, 3, 4,... for **Numeric** option and A, B, C, D... for **Alphabetical** option.
13. **Save** the Excel file with an appropriate name.

The following figure shows the final pin assignment spreadsheet.

Figure 4-19. Final Example Spreadsheet to be Imported into OrCAD Capture

1	Number	Name	Type	Pin Visibility	Shape	Pin Group	Position	Section
2	AB13	CLK0_PAD	Input		Clock		Right	A
3	E22	HPMS_INT_M2F[4]	Output		Line		Right	A
4	F18	HPMS_INT_M2F[7]	Output		Line		Right	A
5	F19	HPMS_INT_M2F[8]	Output		Line		Right	A
6	F20	HPMS_INT_M2F[1]	Output		Line		Right	A
7	F21	COMM_BLK_INT	Output		Line		Right	A
8	G18	HPMS_INT_M2F[10]	Output		Line		Right	A
9	G19	HPMS_INT_M2F[14]	Output		Line		Right	A
10	H19	HPMS_INT_M2F[5]	Output		Line		Right	A
11	H20	HPMS_INT_M2F[6]	Output		Line		Right	A
12	J18	FIC32_0_LOCK	Output		Line		Right	A
13	K1	FIC32_0_CLK	Output		Line		Left	A
14	K15	HPMS_INT_M2F[13]	Output		Line		Left	A
15	K16	HPMS_INT_M2F[2]	Output		Line		Left	A
16	K17	HPMS_INT_M2F[3]	Output		Line		Left	A
17	K18	MSIO20PB2	Bidirectional		Line		Left	A
18	K19	VSS	Power		Short		Right	C
19	K20	HPMS_INT_M2F[15]	Output		Line		Left	A
20	K21	HPMS_INT_M2F[0]	Output		Line		Left	A
21	K22	NC	Passive		Line		Left	D
22	L6	VDDI6	Power		Short		Right	B
23	L9	VDD	Power		Short		Left	B
24	L18	HPMS_INT_M2F[9]	Output		Line		Left	A
25	L19	HPMS_FAB_RESET_N	Output		Line		Left	A

4.2.3 Generating a Capture Schematic Symbol (Ask a Question)

See Generating a OrCAD Capture Schematic Symbol.

5. Board Design and Layout Checklist (Ask a Question)

This section provides a set of checks for designing hardware using Microchip SmartFusion2 and IGLOO2 FPGAs. The checklists provided in this section are a high-level summary to assist the design engineers in the design process.

5.1 Prerequisites (Ask a Question)

Ensure to read following sections:

- 1. Design Considerations
- 2. Layout Guidelines for SmartFusion 2 and IGLOO 2-Based Board Design

The SmartFusion2 and IGLOO2 families consists of FPGAs ranging from densities of 6K to 100K Logic Elements (LE).

Important: This checklist is intended as a guideline only.

5.2 Design Checklist (Ask a Question)

The following table lists the checks that design engineers must take care of while designing the system.

Table 5-1. Design Checklist

S.No.	Checklist	Yes/No
Prerequisites		
1	Read datasheet and pin description user guides: • DS0128: IGLOO2 and SmartFusion2 Datasheet • DS0124: IGLOO2 Pin Descriptions Datasheet • DS0115: SmartFusion2 Pin Descriptions Datasheet	—
2	Check for available designs and development tools.	—
3	See the board-level schematics of the UG0594: SmartFusion2 Security Evaluation Kit User Guide or UG0557: SmartFusion2 SoC FPGA Advanced Development Kit User Guide.	—
Design Specifications		
4	Draw the high-level design with architectural block diagram including all the basic interfaces.	—
5	Specify all the I/O interfaces for all banks in the FPGA.	—
6	Create a detailed functional verification test plan.	—
7	Check for IP software that impacts the system design.	—
Device Selection		
8	Check for available device variants of the SmartFusion2 or IGLOO2 FPGA. Select a device based on the I/O pin count, transceivers, Microcontroller Subsystem (MSS) peripherals, Phase-Locked Loops (PLLs), and speed grade.	—

.....continued

S.No.	Checklist	Yes/No
9	Check device errata: • SmartFusion2 Errata • SmartFusion2 M2S050 (T,TS) ER0195 Errata • IGLOO2 Errata	—
10	Compare the design requirements with the available interfaces and number of I/Os.	—
11	Estimate the required logic utilization, memory, number of I/O pins, and density.	—
12	**Vertical Migration** If desiring pin compatibility within the same package, choose the largest density device for easy vertical migration without any pin conflict in case of future development. For more information, see the application notes available under Schematics/PCB section.	—
13	Identify the clocking requirements.	—
14	Verify that the number of differential channels is adequate.	—
Design		
15	**Power Analysis** Perform power analysis and check the results against the power budget. (Microchip Power Calculator can be used to analyze the power consumption. Estimate the dynamic and static power consumption, and ensure that the design does not violate the power budget.)	—
16	**Noise Margin Analysis** Analyze the dynamic drive capability of output drivers to ensure that the drivers are not loaded beyond the limits (V_{OH}, V_{OL}, V_{IH}, and V_{IL}). **Loading Analysis** Analyze the dynamic drive capability of output drivers to confirm that the drivers are not loaded beyond the limits (C_L)	—
17	**Programming and Debugging Scheme** Check for the programming modes and the procedure to program the device. For programming or debugging through JTAG, add a 10-pin vertical header (2.54 mm pitch). For more information about programming, see UG0451: SmartFusion2 and IGLOO2 Programming User Guide.	—
Power Supply		
18	**Reference Documentation** • DS0128: IGLOO2 and SmartFusion2 Datasheet, Operating Conditions section • DS0115: SmartFusion2 Pin Descriptions Datasheet and DS0124: IGLOO2 Pin Descriptions Datasheet. • Figure 1-1 (for more detailed connectivity)	—

.....continued

S.No.	Checklist	Yes/No
19	**Voltage Rails** The design can be created with just two voltage rails. See Obtaining a Two-Rail Design for Non-SerDes Applications.	—
20	**VDD: Core Supply** VDD operates at 1.2V.	—
21	**VPP: Programming Supply** Charge pump and eNVM can operate at 2.5V or 3.3V.	—
22	**VDDI Bank Supplies** Connect VDDI pins to support the I/O standards of each bank. Ensure I/O power pin compatibility with I/O standards. Check for the banks that must be powered even when unused. See Table 1-4 to Table 1-7. For recommendations on unused bank supplies, see Table 1-17. The recommendations vary from device to device.	—
23	**SerDes Power Supplies** • SERDES VDD—VDD • SERDES VDDAIO—1.2V • SERDES VDDAPLL to REFRET through Resistor-Capacitor (RC) filter circuitry (2.5V) • SERDES PLL VDDA to PLL VSSA through RC filter circuitry—2.5V or 3.3V **VDDA PLL Filter** • REFRET for the SerDes serves as the local on-chip ground return path for VDDAPLL. Therefore, external board ground must not short with REFRET under any circumstances. • A high precision 1.21K_1% Ù resistor is required to connect between REXT and REFRET. For detailed information about power supplies, see Figure 1-1.	—
24	If SerDes transceiver is not used, the pins need to be connected as follows: • SERDES VDD—VDD • SERDES VDDAIO—1.2V • SERDES VDDAPLL—1.2V or 2.5V • SERDES PLL VDDA—2.5V or 3.3V • SERDES PLL VSSA—Ground	—
25	**V_{REF} Power Supply** Design V_{REF} pins to be noise free. V_{REF} must be equal to half of VDDQ. See Figure 1-14.	—
26	**Other Supplies** CCC PLL VDDA to PLL VSSA through RC filter circuitry—2.5V/ 3.3V. DDR PLL VDDA to PLL VSSA through RC filter circuitry—2.5V/3.3 V. All PLL VDDA supplies must be tied to same supply source (either 2.5V or 3.3V). Using the Libero SoC software, a single supply can be selected globally.	—

.....continued

S.No.	Checklist	Yes/No
27	**Decoupling Capacitors** Perform Power Integrity (PI) analysis through the PI tool, and analyze the decoupling capacitor values and placement on the PCB.	—
28	**Unused Condition** For unused conditions of power supply pins, see the corresponding pin assignment table available on the following pages: • SmartFusion2 SoC FPGA Documentation • IGLOO2 FGPA Documentation • Also see Figure 1-3.	—
29	**Brownout Detection (BOD) Circuit** Ensure that brownout detection is implemented standalone or included as part of power management circuitry. See Brownout Detection (BOD).	—
Clocks		
30	**Crystal Oscillators (External)** • Main crystal oscillator • Auxiliary (RTC) crystal oscillator (not available in the M2S050T) **RC Oscillators (Internal)** • 1-MHz RC oscillator • 50-MHz RC oscillator IGLOO2 devices have only main crystal oscillator without auxiliary (RTC) crystal oscillator. For more information about crystal oscillators, see Table 1-12. **FPGA Fabric Clock Sources** The input clock frequency range for fabric clock conditioning circuits (FABCCC) depends on the usage of PLL for output clock generation: • If the PLL is used, the PLL reference clock frequency must be between 1 MHz and 200 MHz. • If the PLL is bypassed, the FABCCC input clock frequency can be up to 400 MHz. All CCC pins support external oscillators (differential or single ended).	—
31	Global buffer (GB) can be driven through dedicated global I/Os, CCC, or fabric (regular I/Os) routing. The global network is composed of GBs to distribute low-skew clock signals or high-fanout nets. Dedicated global I/Os drive the GBs directly and are the primary source for connecting external clock inputs (to minimize the delay) to the internal global clock network. For more information, see UG0449: SmartFusion2 and IGLOO2 Clocking Resources User Guide.	—
Reset		

.....continued

S.No.	Checklist	Yes/No
32	**DEVRST_N—Input** The DEVRST_N pin must have a 10 KΩ pull-up resistor. The pin must not be left floating. If a push-button switch is used to generate reset, check for switch de-bounce. For more information about DEVRST_N, see Power Supply Flow.	—
JTAG		
33	In the JTAG interface, the VDDI bank supply must be powered up for programming. The following is a list of pins available for different activities: • JTAGSEL: Low (pull-down) > Arm Cortex-M3 JTAG; High (pull-up) > FPGA fabric JTAG. • TMS: Internal weak pull-up resistor. • TCK: External pin. must be pulled-down through 1 KΩ resistor. There is no internal pull-up resistor for this pin. • TDI: Internal weak pull-up resistor. • TDO: No internal pull-up resistor. • TRSTB: Internal weak pull-up resistor. An FP4 or FP5 header (10 pin – 2.54 mm pitch) can be used to connect to FlashPro4 or FlashPro5. For more information about JTAG, see Figure 1-9 and Table 1-14.	—
Programming		
34	The device can be programmed either through JTAG interface or Serial Peripheral Interface (SPI). **FLASH_GOLDEN_N** If pulled low, it indicates that the device is to be re-programmed from an image in the external SPI flash attached through the SPI interface. If pulled high, the SPI is put in target mode. Add a 10kΩ external pull-up resistor to VDDI. Some devices do not support the FLASH_GOLDEN_N pin. Check the PPAT spreadsheets available on the following Microchip web pages: • SmartFusion2 SoC FPGA Documentation • IGLOO2 FGPA Documentation For more information about dedicated pins including Flash_GOLDEN_N, see Table 1-15.	—
Configuring Pins in Open Drain Using Tri-state Buffer		
35	To configure fabric pins in open-drain mode, the tristate buffer input pin must always be grounded, and the I/O pin of the FPGA must be connected to the active-low enable pin of the buffer. For more information, see Figure 1-20.	—
SerDes Pins		

.....continued

S.No.	Checklist	Yes/No
36	Dedicated I/O are available for the SerDes high-speed serial interface, which supports the PCIe, SGMII, XAUI, and JESD204B protocols. **SERDES Clock:** 100 MHz to 160 MHz LVDS source. The SerDes reference clock pins have internal On-Die Termination (ODT) settings. These settings can be enabled through the Libero software. The reference clock source (differential clock oscillator) is selected based on many parameters such as frequency range, output voltage swing, jitter (deterministic, random, and peak-to-peak), rise and fall times, supply voltage and current, noise specification, duty cycle, duty cycle tolerance, and frequency stability. An example clock source can be the CCLD-033-LVDS clock oscillator. SerDes clock requirements for different protocols are as follows: • PCIe: 100 MHz • XAUI: 156.25 MHz • SGMII: 125 MHz • EPCS: 125 MHz **SerDes TXD:** The transmit pair must alone have AC-coupling capacitors near the SmartFusion2/ IGLOO2 device. AC-coupling capacitors of 75 nF–200 nF are required for link detection. If the SerDes unit is unused, these pins must remain floating (DNC). **SerDes RXD:** The receive pair must have AC-coupling capacitors near the endpoint device. If the SerDes unit unused, these pins must always be connected to ground. For more information about SerDes, see SerDes.	—
DDR Interface		
37	DDR Interface Short DDR TMATCH IN to DDR TMATCH OUT. Short DDR TMATCH ECC IN to DDR TMATCH ECC OUT. VDDI bank supply must be powered as per the application: • For LPDDR—VDDI must be 1.8V • For DDR2—VDDI must be 1.8V • For DDR3—VDDI must be 1.5V DDR impedance calibration pin must be pulled down with the following resistors: • For LPDDR—150Ω • For DDR2—150Ω • For DDR3—240Ω Though calibration is not required, it is recommended to use corresponding resistor placeholder to connect the pin to the ground with or without a resistor. All data and data strobe signals have internal ODT settings, which can be enabled through the Libero software.	—
Hot-swapping and Cold-Sparing		

.....continued

S.No.	Checklist	Yes/No
38	All user I/Os have internal clamp diode control circuitry for protection. MSIO pins (except PCI 3.3V standard) support the hot-swapping and coldsparing operations. MSIOD and DDRIO pins do not support hot swapping and cold-sparing operations	—
General Guidelines		
39	For all MSIO, MSIOD, and DDRIO, a weak internal pull-up resistor is available. In unused condition, these pins can be left floating.	—
40	MSIOD and DDRIO support a maximum of 2.5V. MSIO supports maximum of 3.3V.	—
41	There is one MSI special pin (MSIO) available that can be used as input only. This pin is differentially paired with FLASH_GOLDEN_N, which is always input only. For more information, see the following documents: • DS0124: IGLOO2 Pin Descriptions Datasheet • DS0115: SmartFusion2 Pin Descriptions Datasheet	—
42	One internal signal can be allocated for probing (for example, towards the oscilloscope feature). The two live probe I/O cells are dual-purpose. They can be used for the live probe functionality or used as user I/Os (MSIO).	—
43	MSS peripherals (SPI, I^2C, USB, and UART) are available.	—
44	Provide pull-up resistors for all open-collector or open-drain pins, even if a pin is not used.	—
45	Provide separate pull-down resistors for all used open-emitter or open-source pins.	—
46	Enable internal pull-up/pull-down resistor option for all tristate nets through the Libero tool.	—
47	Ensure that all the critical signals on the board are terminated properly.	—
48	Terminate the unused interface signals properly to avoid metastability and electromagnetic interference (EMI)/Electromagnetic Compatibility (EMC) problems.	—
49	Provide a sufficient number of ground pins for board-to-board connectors to ensure Signal Integrity (SI) across connectors. Dense board-to-board connectors may cause severe cross-talk problems. The severity of crosstalk depends on the frequency of the signal and the spacing between signal pins on the connectors (the number of ground pins may be obtained after performing SI analysis). The severity can be reduced by providing ground pins between signal pins.	—
50	Use proper voltage-level translator devices for interfacing higher-operating voltage devices with lower-operating-voltage devices.	—

.....continued

S.No.	Checklist	Yes/No
51	Perform timing analysis of all components, taking into consideration the delays introduced by buffers in the data, address, or control paths.	—
52	Perform signal integrity analysis (pre-layout and post-layout) for all critical interfaces and all types of I/Os using input/output Buffer Information Specification (IBIS).	—
53	Analyze the design for SSN problems: • Use differential I/O standards and lower-voltage standards for high switching I/Os. • Reduce the number of simultaneously switching output pins within each bank. • Reduce the number of pins that switch voltage levels at the same time. • Use lower drive strengths for high switching I/Os. The default drive strength setting might be higher than the design requirement. • Spread output pins across multiple banks if possible. • If bank usage is substantially below 100%, spread the switching I/Os evenly throughout the bank to reduce the number of aggressors in a given area to reduce SSN. • Separate simultaneously switching pins from input pins that are susceptible to SSN.	—
54	Place important clock and asynchronous control signals near ground signals and away from large switching buses.	—
55	**I/O Pin Assignment** Use a spreadsheet to capture the list of design I/Os. Microchip provides detailed pinout information that can be downloaded from the website and customized to store the pinout information for specific designs. Pinout details for various packages with different densities are available on the following pages: • SmartFusion2 SoC FPGA Documentation • IGLOO2 FGPA Documentation	—
56	Check if there are any incompatible I/O standards combined in the same bank.	—
57	Check if there are two interfaces with different voltage standards in the same bank.	—
58	See the bank location diagrams in the DS0124: IGLOO2 Pin Descriptions Datasheet/ DS0115: SmartFusion2 Pin Descriptions Datasheet documents to assess the preliminary placement of major components on PCB.	—

5.3 Layout Checklist (Ask a Question)

The following table lists the layout checks.

Table 5-2. Layout Checklist

Sl No.	Description	Yes/No
Power		
1.	Are 0402 or lesser size capacitors used for all decaps (less than value)?	
2.	Are power supply filters implemented on SERDES_x_VDDAPL, and SERDES_x_PLL_VDDA as shown in the Figure 2-5 and Figure 2-17, respectively?	
3.	Is precision 1.2 K resistor between SERDES_x_REFRET and SERDES_x_REXT used?	
4.	Are placement and layout guidelines followed for 1.2 K resistor?	
5.	Is the target impedance met on all power planes?	
6.	Are VREF planes for DDRx reference supply isolated from the noisy planes?	
7.	Are enough number of decoupling capacitors used for DDRx core and VTT supply? For more information about DDRx core and VTT supply, see 1. Design Considerations.	
8.	Is one 0.1 µF cap for two VTT termination resistors used for DDRx?	
9.	Is enough plane width provided for VTT plane?	
DDR3		
10.	Are length match recommendations followed according to the DDR3 guidelines?	
SerDes		
11.	Are length match recommendations followed according to the SerDes guidelines?	
12.	Are the DC blocking capacitors used for SerDes TX and if required on RX lines?	
13.	Is tight controlled impedance maintained along the SerDes traces?	
14.	Are differential vias well designed to match SerDes trace impedance?	
15.	Are DC blocking capacitor pads designed to match SerDes trace impedance?	
Dielectric Material		
16.	Is proper PCB material selected for critical layers?	

6. Appendix A: Special Layout Guidelines—Crystal Oscillator (Ask a Question)

The placement of the crystal needs to be close to the SmartFusion2 or IGLOO2 device. Two capacitors are placed symmetrically around the crystal so that the lengths from the crystal pad to capacitor are equal, as shown in the following figure. Two traces from crystal to SmartFusion2/ IGLOO2 devices should have equal lengths.

Figure 6-1. Layout of the Crystal Oscillator

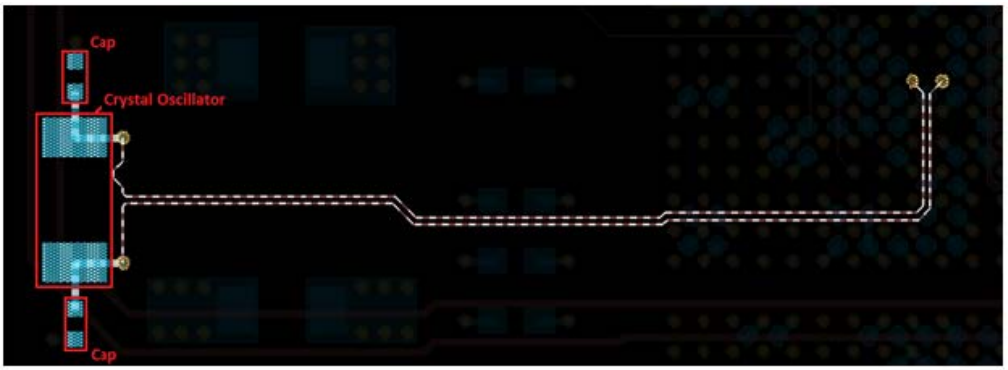
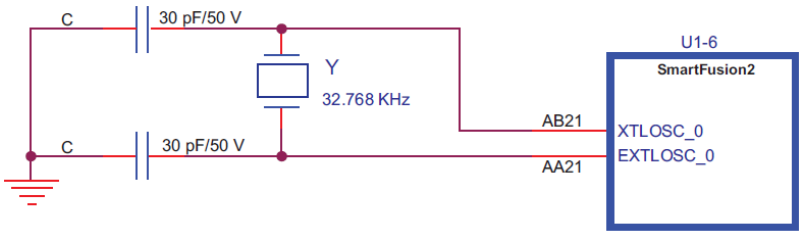


Figure 6-2. Schematics of Crystal Oscillator



7. Appendix B: Stack-Up (Ask a Question)

A good stack-up leads towards better performance. The number of layers in the stack-up is dependent factors such as form factor of the board, number of signals to be routed, and power requirements. Therefore, the designer chooses how many layers the board requires. The SmartFusion2 Development Kit/IGLOO2 Evaluation Kit has 16-layer stack-up, as shown in Figure 7-1.

Important: All the guidelines in this document are with respect to 16-layer board stack-up.

Utilizing upper power layers must be used for high priority supplies. High-switching current supplies must be placed vertically closer to the devices to decrease the distance the currents need to travel through vias. Ground planes are placed adjacent to the high transient current power planes to reduce inductance and couple the high-frequency noise.

It is good to have power and ground layers on side-by-side layer. The benefits of this inter-plane capacitance provide better decoupling at high frequencies. The effect of via on power pins is reduced by having a power plane near the device.

Signal integrity depends on how well the traces have controlled impedance, hence it is always recommended to have controlled impedance.

All critical high-speed signals like DDR and PCIe signals need to have ground reference. All signal layers must be separated from each other by ground or power planes. This minimizes crosstalk and provides balanced and clean transmission lines with properly controlled characteristic impedance between devices and other board components.

Best performance is obtained when using dedicated ground plane layers that are continuous across the entire board area. Power planes can provide adequate reference, however, the power planes must be related to the signals they serve to reference.

Important: Refrain from using unrelated power planes as a signal reference.

Slots must not interrupt the planes, or else they can possibly force current to find an alternate return path. This undesired return path could cause a localized bounce on the power or ground plane that can possibly be capacitive coupled to all signals adjacent to the planes.

Figure 7-1. Stack-up Used in Development Board

Lamination Stack-up: L#/Type:	Description:	Thickness and Tolerance: Cu+:	Lamination/PrePreg:		Base Material Requirements: Type:	Description:
1 Mix 2 Pin	Core 0.0040 Q/H	.00035 .00060	.0040			NP 4000-13EP
	Pre-Preg (1 x 2113)		.0034			NP 4000-13EP
3 Mix 4 Pin	Core 0.0035 H/H	.00060 .00060	.0035			NP 4000-13EP
	Pre-Preg (1 x 2113)		.0034	+/- 0.0003		NP 4000-13EP
5 Mix 6 Pin	Core 0.0035 H/H	.00060 .00060	.0035			NP 4000-13EP
	Pre-Preg (1 x 1080)		.0022	+/- 0.0002		NP 4000-13EP
7 Mix 8 Pin	Core 0.0030 1/H	.00120 .00060	.0030			NP 4000-13EP
	Pre-Preg (1 x 1080)		.0026	+/- 0.0003		NP 4000-13EP
9 Mix 10 Pin	Core 0.0030 H/1	.00060 .00120	.0030			NP 4000-13EP
	Pre-Preg (1 x 1080)		.0023	+/- 0.0002		NP 4000-13EP
11 Mix 12 Pin	Core 0.0035 H/H	.00060 .00060	.0035			NP 4000-13EP
	Pre-Preg (1 x 2113)		.0034	+/- 0.0003		NP 4000-13EP
13 Mix 14 Pin	Core 0.0035 H/H	.00060 .00060	.0035			NP 4000-13EP
	Pre-Preg (1 x 2113)		.0034	+/- 0.0003		NP 4000-13EP
15 Mix 16 Pin	Core 0.0040 H/Q	.00060 .00035	.0040			NP 4000-13EP

Target Post-Lam Thickness: 0.0600 +/- 0.0030

Copper Oz Legend: H = 1/2 Oz T = 3/8 Oz Q = 1/4 Oz S = 1/16 Oz

Stack-up Notes:

0.004 Q/H CORES MUST BE MADE OF (1 x 2116 PREG)
0.0035 H/H CORES MUST BE MADE OF (1 x 2113 PREG)
0.003 1/H CORES MUST BE MADE OF (1 x 1080 PREG)

8. Appendix C: Dielectric Material (Ask a Question)

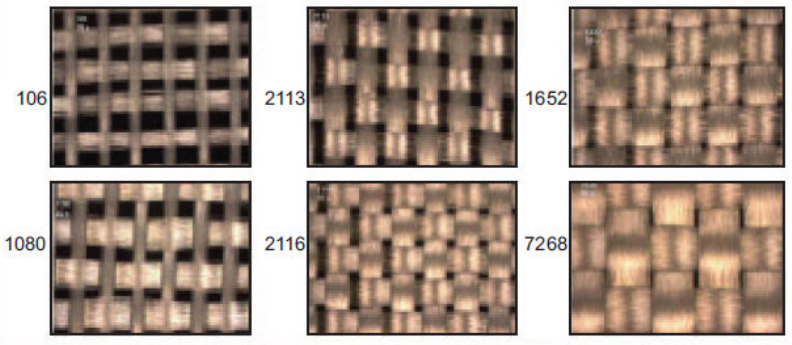
The impedance of the traces depends on the geometry of the traces and the dielectric material used. The skew of the signal depends on the dielectric constant and loss of signal strength depends on the loss tangent of the material. The SmartFusion2 Development Kit board uses Nelco 4000-13 dielectric material. However, selection of the material is made based on the speed and length of the high-speed traces. Simulations are recommended on high-speed serial links to converge on the type of the material used.

If the total trace length is less than 20 inches with a speed at or below 3.125 Gbps, FR-4 may be acceptable. Another design option is to use low-loss dielectric PCB material, such as Rogers 4350, GETEK, or ARLON. It can provide increased eye-opening performance when longer trace interconnections are required. If longer traces or faster speed are required, consider using a high-speed material such as ROGERS 3450.

While designing for gigabit serial links, the weaving structure of PCB dielectric material should be taken into consideration. A PCB dielectric substrate is constructed from woven fiberglass fabrics strengthened and bound together with epoxy resin.

A typical weaving is shown in the following figure.

Figure 8-1. Fiberglass Weaving[1]

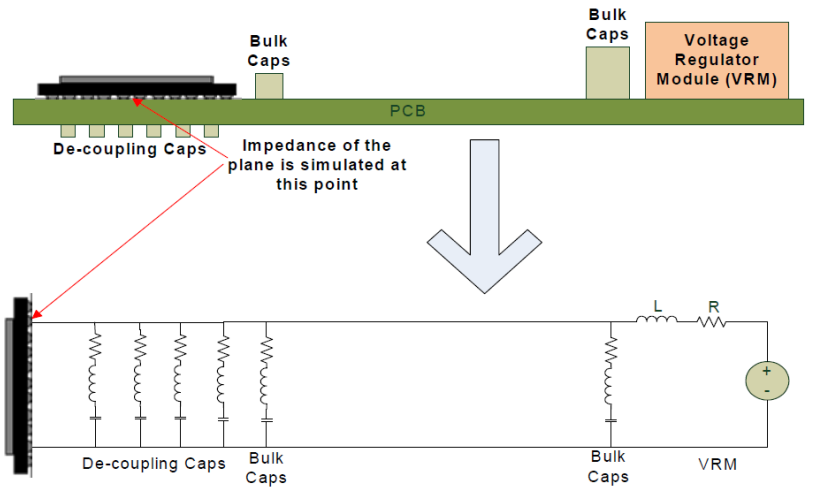


Depending on the density of weaving, the PCB materials are numbered as 106, 1080, 2113, 2116, 1652, and 7268. Trace routed on the PCB is non-homogeneity in dielectric constant due to weaving. This causes discontinuities in the trace impedance, which results in improper eye-opening at the receiving end. For further reading, refer to the Solving PCB Fiber Weave Issues.

9. Appendix D: Power Integrity Simulation Topology (Ask a Question)

The following figure shows the topology that is considered for simulating the power plane for power integrity analysis.

Figure 9-1. Power Integrity Simulation Topology



Important: Package parameters of SmartFusion2 are not considered for simulations.

10. Appendix E: X-Ray Inspection (Ask a Question)

Real time X-ray inspection is being adapted by some customers to assist in the detection of manufacturing defects. SmartFusion2/IGLOO2 family of devices are sensitive to X-ray exposure. Microchip recommends limiting X-ray exposures to a maximum of 15 Rads using a Zinc tray.

11. Revision History (Ask a Question)

The revision history describes the changes that were implemented in the document. The changes are listed by revision, starting with the most current publication.

Revision	Date	Description
H	08/2024	The following is a summary of changes made in this revision: • Removed the figure titled "Crystal Oscillator" from 1.4.1. Main Crystal Oscillator. • Removed the location pointers for Notes 1 and 2 from all the tables in 1.2.6. Unused Pin Configurations.
G	11/2023	Revised the figure and associated text in section 1.9.1. Internal Clamp Diode Circuitry.
F	10/2023	Added Table 1-18.
E	07/2022	The following is a summary of changes made in this revision. • Updated note to include SERDES_x_PLL_VDDA power supply in 1.1. Power Supplies. • Replaced all Microsemi links with Microchip links. • Added a footnote in Table 1-4, Table 1-5, Table 1-6, and Table 1-7.
D	06/2022	The following is a summary of changes made in this revision. • Updated 1.1.2. Power Supply Sequencing. • Updated 1.2.2. I/O Glitch During Power-Down.
C	01/2022	The following is a summary of changes made in this revision. • Updated Power Supply Sequencing. • Updated I/O Glitch During Power-Up and 1.2.2. I/O Glitch During Power-Down.
B	11/2021	The following is a summary of changes made in this revision. • Replaced "slew rate" with "ramp time" in 1.5. Reset Circuit for the DEVRST_N introduction. • Reprashed the table titles as "Unused Bank Supplies" in 1.2.6. Unused Pin Configurations for more clarity. • Updated the footnote of tables Table 1-4 to Table 1-7 for clarity.
A	09/2021	The following is a summary of changes made in this revision. • Document name prefix was changed from AC393 to AN4153. • Document number was changed from 51900393 to DS00004153. • Added more information in 1.2.4. I/O Glitch at Auto-Update During POR. • Updated termination information for probe I/Os, see Table 1-15. • Reorganized and updated the earlier "section 2.4 Limiting Surge Current During Device Reset" to 1.3. Limiting VDD Surge Current. • Corrected table numbering from Table 14 to Table 4 in 1.12. Brownout Detection (BOD). • Added 10. Appendix E: X-Ray Inspection.

.....continued

Revision	Date	Description
15.0	10/2020	The following is a summary of changes made in this revision. • Renamed Table 1-4 to Table 1-7 as "Requirements for Bank Supplies". • Information about I/O glitch at auto-update during POR was added, see 1.2.4. I/O Glitch at Auto-Update During POR. • Updated the note in 1.1. Power Supplies to include Aux. PLL PCIe Supply in the power supply settings description for PLLs. • Internal pull-up clamp diode is not present in MSIO. Hence, removed MSIO from Figure 1-19. • In Table 5-1 (Design Checklist → Power Supplies → VDDI Bank Supplies), added reference to Table 1-4 to Table 1-7 for verifying the requirements for VDDI Bank supplies. • Updated the pull-down resistor requirement to 150 Ω for the xDDR_IMP_CALIB_ECC pin of LPDDR and DDR2 interface. See Figure 1-15 and Figure 1-16.
14.0	—	The following is a summary of changes made in revision 14.0 of this document. • Information about I/O glitches during power-up, power-down, and on blank devices was updated. For more information, see I/O Glitch. • A figure was added to illustrate SPI initiator mode programming. For more information, see SPI Master Programming • Information about simultaneous switching noise support was added. For more information, see Simultaneous Switching Noise. • The design checklist from CL0034: SmartFusion2/IGLOO2 Hardware Board Design Checklist was merged into this document. • List of device-package combinations that do not have SERDES_x_VDD pins was added. For more information, see Table 1-1. • Design checklist was added in Board Design and Layout Checklist. • Information about the de-coupling capacitor and SmartFusion2/IGLOO2 placement was added in Component Placement. • Figures were updated in LPDDR and DDR2 Design and DDR3 Guidelines.
13.0	—	The following is a summary of changes made in revision 13.0 of this document. • Updated Figure 1-3, Figure 1-15, Figure 1-16, Figure 1-17, and Figure 1-18. • Updated Table 1-4, Table 1-5, Table 1-6, Table 1-7, and Table 1-16. • AC408: Creating Schematic Symbols using Cadence OrCAD Capture CIS for SmartFusion2 and IGLOO2 Designs is merged with this document.
12.0	—	The following is a summary of changes made in revision 12.0 of this document. • Recommended bank supplies are updated for the FG484 package. See Table 1-5. • Recommended bank supplies are updated for VF400 and FCS325 Packages. See Table 1-6. • Recommended bank supplies are updated for VF256 and TQ144 Packages. See Table 1-7. • Added a note about DQ pins that all 4- and 8-bit pins are interchangeable in LPDDR, DDR2, and DDR3 memories. See Figure 1-15. • Added that the SERDES_x_L[01/23]_VDDAPLL pin supports only 2.5 V, and removed 1.2 V references from all occurrences. For more information, see figure 1 and Table 1-7. • AC394: Layout Guidelines for SmartFusion2- and IGLOO2-Based Board Design was added as a part of Board Design guidelines itself.

.....continued

Revision	Date	Description
11.0	—	The following is a summary of changes made in revision 11.0 of this document. • The filter circuit for SERDES_x_VDD was removed. Even if it was used in the board design previously, it does not affect the functionality of the board. See Figure 1-1. • Information about VDDI2 was updated. See Table 1-6 and Table 1-7. • Information about reset circuit was updated. see Reset Circuit. • Changed the document to the new template.
10.0	—	The following is a summary of the changes made in revision 10.0 of this document. • Updated Power Supplies (SAR 77745 and SAR 79670). • Updated Table 1-4 (SAR 78887). • Updated SPI Master Programming (SAR 75910). • Updated SerDes (SAR 78504). • Updated User I/O and Clock Pins (SAR 61314).
9.0	—	The following is a summary of the changes made in revision 9.0 of this document. • Updated Figure 1-1 (SAR 72533). • Added CCC_PLL_VDDA and MSS_xDDR_PLL_VDDA details under Power Supplies (SAR 72533). • Deleted the RC Values for Filter Circuitry table (SAR 72533). • Added the M2S060T/M2GL060T device column in Table 1-7(SAR 70484).
8.0	—	The following is a summary of the changes made in revision 8.0 of this document. • Updated Figure 1-1 (SAR 66682). • Updated the RC Values for Filter Circuitry table (SAR 66682 and SAR 65367). • Updated Table 1-4 (SAR 70545). • Updated Table 1-7 (SAR 67599). • Updated Table 1-16 to replace pin SC_SPI_SS with SC_SPI_SDO. • Updated PLL Filter (SAR 60798). • Updated Figure 1-15 (SAR 65438, SAR 69743 and SAR 69580). • Updated Figure 1-16 (SAR 65438). • Updated Figure 1-18 (SAR 65438). • Added Figure 1-19 (SAR 64377).
7.0	—	The following is a summary of the changes made in revision 7.0 of this document. • Updated Figure 1-3 and Figure 1-1 (SAR 62858). • Updated 1.1.2. Power Supply Sequencing(SAR 64117). • Updated 1.2.5. Power Supply Flow. • Added Table 1-7. • Updated Table 1-4, Table 1-20, and Table 1-11 (SAR 62858). • Updated AC Coupling. • Updated Reset Circuit. • Updated Figure 1-15, Figure 1-16, Figure 1-17, and Figure 1-18 (SAR 65438). • Replaced all instances of VQ144 with TQ144 Package. • Removed all instances of and references to M2S100 and M2GL100 device (SAR 62858).

.....continued

Revision	Date	Description
6.0	—	The following is a summary of the changes made in revision 6.0 of this document. • Updated Design Considerations (SAR 58055). • Updated Power Supplies (SAR 52580). • Updated Power Supply Sequencing (SAR 59593 and SAR 57004). • Updated Figure 1-3 (SAR 52580). • Added M2S090T/M2GL090T-FCS325 information for power supplies in Table 1-4 (SAR 58241). • Added a foot note to Table 1-16(SAR 59563). • Updated SerDes Reference Clock Requirements (SAR 60213). • Updated Table 1-15 (SAR 58085). • Updated Figure 1-21 (SAR 56598). • Added a foot note to Table 1-16 (SAR 59563). • Updates were made to maintain the style and consistency of the document.
5.0	—	The following is a summary of the changes made in revision 5.0 of this document. • Updated Power Supply Sequencing, Figure 1-1, Figure 1-3, and Table 1-4 (SAR 52580). • Updated main crystal oscillator pins naming convention (SAR 53177). • Updated Table 1-20 (SAR 55659). • Updated Figure 1-15, Figure 1-16, Figure 1-17, and Figure 1-18(SAR 53161). • Updated Table 1-4(SAR 53348).
4.0	—	The following is a summary of the changes made in revision 4.0 of this document. • Corrected the ramp rate description to 50 µs in the Power Supply Sequencing (SAR 50245) (SAR 50844). • Updated Figure 1-9 (SAR 50725). • Added the 1.11. Configuring Pins in Open Drain section.
3.0	—	The following is a summary of the changes made in revision 3.0 of this document. • Updated the content for IGLOO2 devices (SAR 48630). • Updated Power Supply Sequencing. • Updated DDR3 Guidelines. • Updated the Temperature Sensing section.
2.0	—	The following is a summary of the changes made in revision 2.0 of this document. • Modified the 1.2.6. Unused Pin Configurations (SAR 47904). • Updated 1.10.1. Operating Voltage Rails (SAR 47548). • Updated the 1.12. Brownout Detection (BOD) (SAR 47904). • Added Figure 1-21 (SAR 47904).
1.0	—	The following is a summary of the changes made in revision 1.0 of this document. • Added the Power Supply Sequencing and Power-on Reset section (SAR 47223). • Updated Figure 1-3 (previously figure 2) and added figure 4 (SAR 47223). • Updated Table 1-4 and Table 1-19, and added Table 1-20 (SAR 47223).

Microchip FPGA Support

Microchip FPGA products group backs its products with various support services, including Customer Service, Customer Technical Support Center, a website, and worldwide sales offices. Customers are suggested to visit Microchip online resources prior to contacting support as it is very likely that their queries have been already answered.

Contact Technical Support Center through the website at www.microchip.com/support. Mention the FPGA Device Part number, select appropriate case category, and upload design files while creating a technical support case.

Contact Customer Service for non-technical product support, such as product pricing, product upgrades, update information, order status, and authorization.

- From North America, call **800.262.1060**
- From the rest of the world, call **650.318.4460**
- Fax, from anywhere in the world, **650.318.8044**

Microchip Information

The Microchip Website

Microchip provides online support via our website at www.microchip.com/. This website is used to make files and information easily available to customers. Some of the content available includes:

- **Product Support** – Data sheets and errata, application notes and sample programs, design resources, user's guides and hardware support documents, latest software releases and archived software
- **General Technical Support** – Frequently Asked Questions (FAQs), technical support requests, online discussion groups, Microchip design partner program member listing
- **Business of Microchip** – Product selector and ordering guides, latest Microchip press releases, listing of seminars and events, listings of Microchip sales offices, distributors and factory representatives

Product Change Notification Service

Microchip's product change notification service helps keep customers current on Microchip products. Subscribers will receive email notification whenever there are changes, updates, revisions or errata related to a specified product family or development tool of interest.

To register, go to www.microchip.com/pcn and follow the registration instructions.

Customer Support

Users of Microchip products can receive assistance through several channels:

- Distributor or Representative
- Local Sales Office
- Embedded Solutions Engineer (ESE)
- Technical Support

Customers should contact their distributor, representative or ESE for support. Local sales offices are also available to help customers. A listing of sales offices and locations is included in this document.

Technical support is available through the website at: www.microchip.com/support

Microchip Devices Code Protection Feature

Note the following details of the code protection feature on Microchip products:

- Microchip products meet the specifications contained in their particular Microchip Data Sheet.
- Microchip believes that its family of products is secure when used in the intended manner, within operating specifications, and under normal conditions.
- Microchip values and aggressively protects its intellectual property rights. Attempts to breach the code protection features of Microchip product is strictly prohibited and may violate the Digital Millennium Copyright Act.
- Neither Microchip nor any other semiconductor manufacturer can guarantee the security of its code. Code protection does not mean that we are guaranteeing the product is “unbreakable”. Code protection is constantly evolving. Microchip is committed to continuously improving the code protection features of our products.

Legal Notice

This publication and the information herein may be used only with Microchip products, including to design, test, and integrate Microchip products with your application. Use of this information in any other manner violates these terms. Information regarding device applications is provided only for your convenience and may be superseded by updates. It is your responsibility to ensure that your application meets with your specifications. Contact your local Microchip sales office for additional support or, obtain additional support at www.microchip.com/en-us/support/design-help/client-support-services.

THIS INFORMATION IS PROVIDED BY MICROCHIP “AS IS”. MICROCHIP MAKES NO REPRESENTATIONS OR WARRANTIES OF ANY KIND WHETHER EXPRESS OR IMPLIED, WRITTEN OR ORAL, STATUTORY OR OTHERWISE, RELATED TO THE INFORMATION INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY, AND FITNESS FOR A PARTICULAR PURPOSE, OR WARRANTIES RELATED TO ITS CONDITION, QUALITY, OR PERFORMANCE.

IN NO EVENT WILL MICROCHIP BE LIABLE FOR ANY INDIRECT, SPECIAL, PUNITIVE, INCIDENTAL, OR CONSEQUENTIAL LOSS, DAMAGE, COST, OR EXPENSE OF ANY KIND WHATSOEVER RELATED TO THE INFORMATION OR ITS USE, HOWEVER CAUSED, EVEN IF MICROCHIP HAS BEEN ADVISED OF THE POSSIBILITY OR THE DAMAGES ARE FORESEEABLE. TO THE FULLEST EXTENT ALLOWED BY LAW, MICROCHIP’S TOTAL LIABILITY ON ALL CLAIMS IN ANY WAY RELATED TO THE INFORMATION OR ITS USE WILL NOT EXCEED THE AMOUNT OF FEES, IF ANY, THAT YOU HAVE PAID DIRECTLY TO MICROCHIP FOR THE INFORMATION.

Use of Microchip devices in life support and/or safety applications is entirely at the buyer’s risk, and the buyer agrees to defend, indemnify and hold harmless Microchip from any and all damages, claims, suits, or expenses resulting from such use. No licenses are conveyed, implicitly or otherwise, under any Microchip intellectual property rights unless otherwise stated.

Trademarks

The Microchip name and logo, the Microchip logo, Adaptec, AVR, AVR logo, AVR Freaks, BesTime, BitCloud, CryptoMemory, CryptoRF, dsPIC, flexPWR, HELDO, IGLOO, JukeBlox, KeeLoq, Kleer, LANCheck, LinkMD, maXStylus, maXTouch, MediaLB, megaAVR, Microsemi, Microsemi logo, MOST, MOST logo, MPLAB, OptoLyzer, PIC, picoPower, PICSTART, PIC32 logo, PolarFire, Prochip Designer, QTouch, SAM-BA, SenGenuity, SpyNIC, SST, SST Logo, SuperFlash, Symmetricom, SyncServer, Tachyon, TimeSource, tinyAVR, UNI/O, Vectron, and XMEGA are registered trademarks of Microchip Technology Incorporated in the U.S.A. and other countries.

AgileSwitch, ClockWorks, The Embedded Control Solutions Company, EtherSynch, Flashtec, Hyper Speed Control, HyperLight Load, Libero, motorBench, mTouch, Powermite 3, Precision Edge, ProASIC, ProASIC Plus, ProASIC Plus logo, Quiet-Wire, SmartFusion, SyncWorld, TimeCesium, TimeHub, TimePictra, TimeProvider, and ZL are registered trademarks of Microchip Technology Incorporated in the U.S.A.

Adjacent Key Suppression, AKS, Analog-for-the-Digital Age, Any Capacitor, AnyIn, AnyOut, Augmented Switching, BlueSky, BodyCom, Clockstudio, CodeGuard, CryptoAuthentication, CryptoAutomotive, CryptoCompanion, CryptoController, dsPICDEM, dsPICDEM.net, Dynamic

ISBN: 978-1-6683-0055-8

Quality Management System

For information regarding Microchip's Quality Management Systems, please visit www.microchip.com/quality.

Worldwide Sales and Service

AMERICAS	ASIA/PACIFIC	ASIA/PACIFIC	EUROPE
Corporate Office 2355 West Chandler Blvd. Chandler, AZ 85224-6199 Tel: 480-792-7200 Fax: 480-792-7277 Technical Support: www.microchip.com/support Web Address: www.microchip.com	**Australia - Sydney** Tel: 61-2-9868-6733	**India - Bangalore** Tel: 91-80-3090-4444	**Austria - Wels** Tel: 43-7242-2244-39 Fax: 43-7242-2244-393
Atlanta Duluth, GA Tel: 678-957-9614 Fax: 678-957-1455	**China - Beijing** Tel: 86-10-8569-7000	**India - New Delhi** Tel: 91-11-4160-8631	**Denmark - Copenhagen** Tel: 45-4485-5910 Fax: 45-4485-2829
Austin, TX Tel: 512-257-3370	**China - Chengdu** Tel: 86-28-8665-5511	**India - Pune** Tel: 91-20-4121-0141	**Finland - Espoo** Tel: 358-9-4520-820
Boston Westborough, MA Tel: 774-760-0087 Fax: 774-760-0088	**China - Chongqing** Tel: 86-23-8980-9588	**Japan - Osaka** Tel: 81-6-6152-7160	**France - Paris** Tel: 33-1-69-53-63-20 Fax: 33-1-69-30-90-79
Chicago Itasca, IL Tel: 630-285-0071 Fax: 630-285-0075	**China - Dongguan** Tel: 86-769-8702-9880	**Japan - Tokyo** Tel: 81-3-6880- 3770	**Germany - Garching** Tel: 49-8931-9700
Dallas Addison, TX Tel: 972-818-7423 Fax: 972-818-2924	**China - Guangzhou** Tel: 86-20-8755-8029	**Korea - Daegu** Tel: 82-53-744-4301	**Germany - Haan** Tel: 49-2129-3766400
Detroit Novi, MI Tel: 248-848-4000	**China - Hangzhou** Tel: 86-571-8792-8115	**Korea - Seoul** Tel: 82-2-554-7200	**Germany - Heilbronn** Tel: 49-7131-72400
Houston, TX Tel: 281-894-5983	**China - Hong Kong SAR** Tel: 852-2943-5100	**Malaysia - Kuala Lumpur** Tel: 60-3-7651-7906	**Germany - Karlsruhe** Tel: 49-721-625370
Indianapolis Noblesville, IN Tel: 317-773-8323 Fax: 317-773-5453 Tel: 317-536-2380	**China - Nanjing** Tel: 86-25-8473-2460	**Malaysia - Penang** Tel: 60-4-227-8870	**Germany - Munich** Tel: 49-89-627-144-0 Fax: 49-89-627-144-44
Los Angeles Mission Viejo, CA Tel: 949-462-9523 Fax: 949-462-9608 Tel: 951-273-7800	**China - Qingdao** Tel: 86-532-8502-7355	**Philippines - Manila** Tel: 63-2-634-9065	**Germany - Rosenheim** Tel: 49-8031-354-560
Raleigh, NC Tel: 919-844-7510	**China - Shanghai** Tel: 86-21-3326-8000	**Singapore** Tel: 65-6334-8870	**Israel - Hod Hasharon** Tel: 972-9-775-5100
New York, NY Tel: 631-435-6000	**China - Shenyang** Tel: 86-24-2334-2829	**Taiwan - Hsin Chu** Tel: 886-3-577-8366	**Italy - Milan** Tel: 39-0331-742611 Fax: 39-0331-466781
San Jose, CA Tel: 408-735-9110 Tel: 408-436-4270	**China - Shenzhen** Tel: 86-755-8864-2200	**Taiwan - Kaohsiung** Tel: 886-7-213-7830	**Italy - Padova** Tel: 39-049-7625286
Canada - Toronto Tel: 905-695-1980 Fax: 905-695-2078	**China - Suzhou** Tel: 86-186-6233-1526	**Taiwan - Taipei** Tel: 886-2-2508-8600	**Netherlands - Drunen** Tel: 31-416-690399 Fax: 31-416-690340
	China - Wuhan Tel: 86-27-5980-5300	**Thailand - Bangkok** Tel: 66-2-694-1351	**Norway - Trondheim** Tel: 47-72884388
	China - Xian Tel: 86-29-8833-7252	**Vietnam - Ho Chi Minh** Tel: 84-28-5448-2100	**Poland - Warsaw** Tel: 48-22-3325737
	China - Xiamen Tel: 86-592-2388138		**Romania - Bucharest** Tel: 40-21-407-87-50
	China - Zhuhai Tel: 86-756-3210040		**Spain - Madrid** Tel: 34-91-708-08-90 Fax: 34-91-708-08-91
			Sweden - Gothenberg Tel: 46-31-704-60-40
			Sweden - Stockholm Tel: 46-8-5090-4654
			UK - Wokingham Tel: 44-118-921-5800 Fax: 44-118-921-5820