GENERAL DISCLAIMER
Integrated Device Technology, Inc. reserves the right to make changes to its products or specifications at any time, without notice, in order to improve design or performance and to supply the best possible product. IDT does not assume any responsibility for use of any circuitry described other than the circuitry embodied in an IDT product. The Company makes no representations that circuitry described herein is free from patent infringement or other rights of third parties which may result from its use. No license is granted by implication or otherwise under any patent, patent rights or other rights, of Integrated Device Technology, Inc.

CODE DISCLAIMER
Code examples provided by IDT are for illustrative purposes only and should not be relied upon for developing applications. Any use of the code examples below is completely at your own risk. IDT MAKES NO REPRESENTATIONS OR WARRANTIES OF ANY KIND CONCERNING THE NONINFRINGEMENT, QUALITY, SAFETY OR SUITABILITY OF THE CODE, EITHER EXPRESS OR IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT. FURTHER, IDT MAKES NO REPRESENTATIONS OR WARRANTIES AS TO THE TRUTH, ACCURACY OR COMPLETENESS OF ANY STATEMENTS, INFORMATION OR MATERIALS CONCERNING CODE EXAMPLES CONTAINED IN ANY IDT PUBLICATION OR PUBLIC DISCLOSURE OR THAT IS CONTAINED ON ANY IDT INTERNET SITE. IN NO EVENT WILL IDT BE LIABLE FOR ANY DIRECT, CONSEQUENTIAL, INCIDENTAL, INDIRECT, PUNITIVE OR SPECIAL DAMAGES, HOWEVER THEY MAY ARISE, AND EVEN IF IDT HAS BEEN PREVIOUSLY ADVISED ABOUT THE POSSIBILITY OF SUCH DAMAGES. The code examples also may be subject to United States export control laws and may be subject to the export or import laws of other countries and it is your responsibility to comply with any applicable laws or regulations.

LIFE SUPPORT POLICY
Integrated Device Technology’s products are not authorized for use as critical components in life support devices or systems unless a specific written agreement pertaining to such intended use is executed between the manufacturer and an officer of IDT.
1. Life support devices or systems are devices or systems which (a) are intended for surgical implant into the body or (b) support or sustain life and whose failure to perform, when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a significant injury to the user.
2. A critical component is any components of a life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

IDT, the IDT logo, and Integrated Device Technology are trademarks or registered trademarks of Integrated Device Technology, Inc.
## Contents

1. **Designing a PCB for PCIe Signals** ......................................................... 7  
   1.1 PCIe Signals .......................................................................................... 7  
   1.2 Transmission Line Terms ....................................................................... 8  
   1.3 Trace Geometry ..................................................................................... 8  
   1.4 Trace Length Matching .......................................................................... 9  
   1.5 Reference Planes .................................................................................. 9  
   1.6 Bends in PCIe Differential Pairs .............................................................. 10  
   1.7 Device Breakout Area ........................................................................... 10  
   1.8 Vias in Differential Traces ...................................................................... 11  
   1.9 Routing Guidelines Summary ............................................................... 11  
   1.10 PCIe Tx Capacitors .............................................................................. 12  
   1.11 PCIe_REFCLK .................................................................................... 14  
   1.12 Polarity Reversal ............................................................................... 15  

2. **Power Supply Filtering and Decoupling** ................................................. 17  
   2.1 Analog Power Supply Filtering ................................................................ 17  
      2.1.1 VDDA_PCIE .................................................................................... 17  
      2.1.2 VDD_PCIE .................................................................................... 18  
      2.1.3 VDDA_PLL ................................................................................... 20  
   2.2 Decoupling Capacitors ......................................................................... 22  

3. **PCI Bus Design Notes** .......................................................................... 27  
   3.1 PCI Clocking Modes ............................................................................ 27  
      3.1.1 PCI Slave Mode Clocking ............................................................... 27  
      3.1.2 PCI Master Mode Clocking ............................................................. 28  
   3.2 Bus Routing Rules ............................................................................... 29
About this Document

This section discusses the following topics:

- “Scope”
- “Document Conventions”
- “Revision History”

Scope

The PEB383 QFN Board Design Guidelines provides hardware reference information for the PEB383 QFN, such as layout guidelines and schematic review information. It is intended for hardware designers who are designing system interconnect applications with the PEB383 QFN.

The PEB383 is available in two package variants: QFP and QFN. While both variants have identical functionalities, they are of different physical packages and therefore require different attentions for board layouts. This document intends to be used for the QFN package. For designs that use the QFP variant, refer to the “IDT PEB383 QFP Board Design Guidelines”.

For the remaining of this document, “PEB383 QFN” will be simply called “PEB383”.

Document Conventions

This document uses the following conventions.

Non-differential Signal Notation

Non-differential signals are either active-low or active-high. An active-low signal has an active state of logic 0 (or the lower voltage level), and is denoted by a lowercase “n”. An active-high signal has an active state of logic 1 (or the higher voltage level), and is not denoted by a special character. The following table illustrates the non-differential signal naming convention.

<table>
<thead>
<tr>
<th>State</th>
<th>Single-line signal</th>
<th>Multi-line signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Active low</td>
<td>NAMEn</td>
<td>NAMEn[3]</td>
</tr>
<tr>
<td>Active high</td>
<td>NAME</td>
<td>NAME[3]</td>
</tr>
</tbody>
</table>
**Differential Signal Notation**

Differential signals consist of pairs of complement positive and negative signals that are measured at the same time to determine a signal’s active or inactive state (they are denoted by “_p” and “_n”, respectively). The following table illustrates the differential signal naming convention.

<table>
<thead>
<tr>
<th>State</th>
<th>Single-line signal</th>
<th>Multi-line signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inactive</td>
<td>NAME_p = 0</td>
<td>NAME_p[3] = 0</td>
</tr>
<tr>
<td></td>
<td>NAME_n = 1</td>
<td>NAME_n[3] = 1</td>
</tr>
<tr>
<td>Active</td>
<td>NAME_p = 1</td>
<td>NAME_p[3] is 1</td>
</tr>
<tr>
<td></td>
<td>NAME_n = 0</td>
<td>NAME_n[3] is 0</td>
</tr>
</tbody>
</table>

**Numeric Notation**

- Hexadecimal numbers are denoted by the prefix 0x (for example, 0x04).
- Binary numbers are denoted by the prefix 0b (for example, 0b010).

Dimensions in this document are given in British imperial units. Use this conversion ratio for Metric units:

- 1 inch = 25.4 mm
- 1 mil = 0.0254 mm

**Symbols**

- This symbol indicates a basic design concept or information considered helpful.
- This symbol indicates important configuration information or suggestions.
- This symbol indicates procedures or operating levels that may result in misuse or damage to the device.

**Revision History**

**60E2080_AN001_01, Formal, April 2010**

Corrected nominal power supply voltages for VDD_PCIE and VDDA_PLL to 1.05V.

**60E2080_AN001_01, Formal, April 2010**

This is the first version of the *PEB383 QFN Board Design Guidelines*. 
1. Designing a PCB for PCIe Signals

Topics discussed include the following:

- “PCIe Signals”
- “Transmission Line Terms”
- “Trace Geometry”
- “Trace Length Matching”
- “Reference Planes”
- “Bends in PCIe Differential Pairs”
- “Device Breakout Area”
- “Vias in Differential Traces”
- “Routing Guidelines Summary”
- “PCIe Tx Capacitors”
- “PCIe_REFCLK”
- “Polarity Reversal”

1.1 PCIe Signals

The following table lists the PEB383’s PCIe signals that are subject to the layout guidelines.

Table 1: PEB383 PCIe Signals

<table>
<thead>
<tr>
<th>Signal Name</th>
<th>Pin Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCIE_TXD_p</td>
<td>A69</td>
</tr>
<tr>
<td>PCIE_TXD_n</td>
<td>B59</td>
</tr>
<tr>
<td>PCIE_RXD_p</td>
<td>A68</td>
</tr>
<tr>
<td>PCIE_RXD_n</td>
<td>B57</td>
</tr>
<tr>
<td>PCIE_REFCLK_P</td>
<td>B55</td>
</tr>
<tr>
<td>PCIE_REFCLK_n</td>
<td>A66</td>
</tr>
</tbody>
</table>
1.2 Transmission Line Terms

- Microstrips – Traces that are adjacent to a continuous reference plane. In a PCB stackup, the microstrips are on the primary and secondary layers.
- Striplines – Traces routed in inner layers and have two reference planes. In a PCB stack-up, striplines are on internal layers.

1.3 Trace Geometry

The PCIe protocol uses differential pair routing, which requires greater care to meet the impedance targets on these signals versus other single-ended interfaces. Variations in the impedance of these signals will impact the overall jitter and loss within the system.

The main areas of concern for board trace routing of PCIe data and clock signals include:

- Spacing within the differential pair
- Coupling to an adjacent reference plane
- Thickness of the traces within the pair

Close coupling or relatively close physical spacing within the differential pairs, along with increased spacing to unrelated differential pairs and other signals, help to minimize crosstalk and EMI effects.

The width and spacing guidelines of the PCIe differential signals determines the impedance of the traces and helps to ensure that interconnect loss and jitter budgets are met. Signal integrity within the system is also impacted by whether or not appropriate spacing exists between differential pairs and adjacent traces.

Edge-coupled microstrip traces of each differential pair within the PCIe Interface should have a differential impedance of 100 Ohms. When using common PCB material, the target impedance can be achieved with trace widths of 5-mils with a 7-mil space within the pair. This 5/7/5 routing method provides the best compromise between impedance targets, loss impacts, crosstalk immunity, and routing flexibility. If stripline traces are used, they should maintain a 5/5/5 routing geometry with 5-mil trace widths combined with a 5-mil spacing within the pair.

A 20-mil edge-to-edge spacing should be used between traces of adjacent differential pairs in order to minimize negative crosstalk effects. A 20-mil spacing should also be maintained between differential pairs and non-PCIe signals. If unrelated signals have significantly higher voltage levels or edge rates when compared to PCIe, then a minimum 30-mil spacing should be used to avoid coupling issues.

Figure 1 displays an example of the width and spacing recommendations for PCIe differential pairs.

If board constraints require routing sections of a differential pair uncoupled where the intrapair spacing exceeds the 7-mil requirement, then the width of the uncoupled section of trace can be increased to 7-mils, as displayed in the figure. This applies to uncoupled sections that exceed 100 mils in distance. Using a 7-mil trace versus 5-mils within the uncoupled sections helps to maintain the correct differential impedance across the uncoupled region.
1.4 Trace Length Matching

PCIe signals have constraints with respect to trace lengths and matching in order to meet jitter and loss budgets within a system. Each inch of board trace within a differential pair can add 1ps of jitter and up to 0.35dB of loss. Typically, the PCIe specification can be met with a chip-to-chip routing length up to 15 inches. When routing differential pairs connecting the finger connector pad to a PEB383 pin on an add-in card, traces can be routed up to 3.5 inches.

Pair-to-pair length matching within the PCIe Interface is normally not required due to the large skew allowed at the receiver; however, it should be minimized in order to reduce the overall latency of the system. Intrapair skew is much more important and requires that the length of the traces within a differential pair be matched to within 5-mils of one another. Length matching within a differential pair should be performed on a segment-by-segment basis rather than only as an overall measurement.

1.5 Reference Planes

High quality reference planes within the PCIe Interface improve signal integrity, reduce EMI effects, and minimize AC common-mode noise within the differential pairs. It is recommended that the differential signal pairs reference the main ground plane within the PCB stackup, and reference only a single plane along the entire routing path. Differential traces should not be routed across or near any discontinuities in the reference plane. A 20-mil spacing should be maintained when routing near and parallel to the edge of a reference plane.
1.6 Bends in PCIe Differential Pairs

Bends or turns within the PCIe differential pairs should be kept to a minimum. The angles between traces should be greater than or equal to 135 degrees, and there should never be any 90 degree turns or bends along a trace. The inner spacing within a bend of a differential pair should equal or exceed the minimum pair-to-pair spacing of 20-mils in order to minimize the introduction of common-mode noise within the system (see Figure 1). Segments within a bend should have a length equal to or greater than 1.5x the width of the trace, and if possible the number of left- and right-hand bends should be matched as closely as possible to minimize length skew differences within a differential pair.

1.7 Device Breakout Area

Within the breakout areas of the PEB383, the PCIe signal pairs should maximize the differential routing while minimizing any discontinuities or trace length skew (see Figure 2). Length matching of the differential traces should occur as close as possible to the pad or pin while avoiding the addition of “tight” bends within the traces. The breakout areas should not exceed 250-mils in length for the PCIe Interface. Within the breakout areas, the trace routing guidelines of the differential pairs can be slightly relaxed — if absolutely necessary — to facilitate successful breakout of the signals. A width and spacing geometry of 5/5/5 can be used with a minimum spacing to other signals of 7-mils.

Figure 2: PEB383 PCIe Differential Trace Breakout (green)
1.8 **Vias in Differential Traces**

Use of vias within the differential traces of the PCIe signals can have an adverse effect on system performance. If vias must be used, they should have a pad size of less than 25-mils with a finished hole size of less than 14-mils. Vias should always be placed in pairs at the same relative location and should be symmetrical between the differential traces. A maximum of four via pairs can be used within each transmit differential pair of the PCIe link. Each receive differential pair can have a maximum of two vias along its path. A maximum of six via pairs are allowed along the entire PCIe interconnect path.

When traces change layers in a PCB, the reference plane for the differential traces also changes. It is recommended to introduce a stitching via close to the differential trace vias (see Figure 3). The stitching via connects all the ground planes together, and allows return current to flow easily from one ground plane to another.

**Figure 3: Differential Pair Vias and Stitching Via**

1.9 **Routing Guidelines Summary**

The routing guidelines should be used for board routing of the transmit and receive data. The guidelines are summarized in Table 2.

**Table 2: PCIe Interface Routing Guidelines**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Main Route System Board</th>
<th>Special Breakout Area</th>
</tr>
</thead>
<tbody>
<tr>
<td>Differential Impedance Target</td>
<td>100 ohms ±15 %</td>
<td></td>
</tr>
<tr>
<td>Single-ended Impedance Target</td>
<td>60 ohms ±15 %</td>
<td></td>
</tr>
<tr>
<td>Trace Width</td>
<td>5 mils</td>
<td>7 mils if uncoupled length ≥ 100 mils</td>
</tr>
<tr>
<td>Differential Pair (within pair) Spacing</td>
<td>7 mils Microstrips 5 mils Striplines</td>
<td>5-mil spacing is acceptable</td>
</tr>
<tr>
<td>Pair-to-Pair Spacing</td>
<td>≥ 20 mils</td>
<td>7-mil spacing is acceptable</td>
</tr>
</tbody>
</table>
1.10 PCIe Tx Capacitors

The PCIe specification requires that each lane of the link be AC coupled between its corresponding transmitter and receiver. The AC coupling capacitors for the PEB383 are discrete components located along each transmitter link on the PCB. These AC coupling capacitors allow the transmitter and receiver on a link to be biased at separate voltages. Table 3 summarizes the guidelines for implementing the PCIe AC coupling capacitors on a system board. Recommended capacitor layout and placement are provided in Figures 4 and 5.

Table 3: PCIe Interface AC Coupling Capacitor Guidelines

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Implementation Guideline</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC Coupling</td>
<td>AC coupling capacitors are required on the Tx pairs originating from the PEB383.</td>
</tr>
<tr>
<td>Capacitor Value</td>
<td>$75\text{ nF} \leq C_{AC-COUPLED} \leq 200\text{ nF}$</td>
</tr>
<tr>
<td>Capacitor Tolerance</td>
<td>Specified minimum/maximum range must be met when capacitor tolerance is considered along with effects due to temperature and voltage.</td>
</tr>
</tbody>
</table>
Table 3: PCIe Interface AC Coupling Capacitor Guidelines (Continued)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Implementation Guideline</th>
</tr>
</thead>
<tbody>
<tr>
<td>Capacitor Type</td>
<td>Size 603 ceramic capacitors are acceptable; however, size 402 capacitors are strongly encouraged. The smaller the package size, the less ESL is introduced into the topology. The same package and capacitor size should be used for each signal in a differential pair. Do not use capacitor packs for PCIe AC coupling.</td>
</tr>
<tr>
<td>Capacitor Pad Size</td>
<td>To minimize parasitic impacts, pad sizes for each capacitor should be the minimum allowed per PCB manufacturer.</td>
</tr>
<tr>
<td>Capacitor Placement</td>
<td>AC coupling capacitors should be located at the same place within the differential pair. They should not be staggered in distance from one trace of the differential pair to the other. Capacitors should be placed as close to each other as possible to avoid creating large uncoupled sections within the differential pair traces. Relative location from one differential pair to another is not important.</td>
</tr>
<tr>
<td>Capacitor Location – Chip-to-Connector Routing</td>
<td>Capacitors should be placed such that they are not located in the center point of a trace route (for example, capacitors should be placed next to the connector or 1/3 the distance between the connector and the PEB383).</td>
</tr>
<tr>
<td>Capacitor Location – Chip-to-Chip Routing</td>
<td>Capacitors should be located off-center within the interconnect (for example, placing the capacitors next to the Rx pins of one device is generally better than locating the capacitors in the midpoint of the interconnect).</td>
</tr>
</tbody>
</table>

Figure 4: PCIe AC Coupling Capacitor Symmetric Layout

Preferred Symmetric Placement

Avoid Asymmetric Placement
1.11 PCIe_REFCLK

The PEB383 must receive a 100-MHz differential reference clock on its PCIE_REFCLK_p/n input pins. This clock can be generated from an on-board oscillator if a PCIe system clock is not available. The clock generator should provide an HCSL output-level clock that meets the PCIE_REFCLK duty cycle and jitter requirement, as specified in the PEB383 User Manual. Figure 6 shows a simplified schematic of an HCSL PCIE_REFCLK circuit. The HCSL source termination is specified by the HCSL clock device vendor.
1.12 Polarity Reversal

Lane polarity reversal support is required as per the *PCI Express Specification*. This feature can be used to ease routing of differential pairs of the express link to and from the PEB383. As such, the following are recommended:

- To connect the _p to the _n of a differential pair instead of crossing the _p and _n traces through an extra PCB layer
- To connect the _p to the _n of a differential pair instead of un-crossing traces around vias or through-holes in a break-out area

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Implementation Guidelines</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCIE_REFCLK Differential Trace Impedance</td>
<td>100 Ohms</td>
</tr>
<tr>
<td>PCIE_REFCLK Length Restriction for Add-in Card</td>
<td>From finger connector to PEB383 (or any other device): 4 inches</td>
</tr>
<tr>
<td>Length Matching within Pair</td>
<td>≤ 10 mils</td>
</tr>
</tbody>
</table>
2. Power Supply Filtering and Decoupling

Topics discussed include the following:

- “Analog Power Supply Filtering”
- “Decoupling Capacitors”

2.1 Analog Power Supply Filtering

2.1.1 VDDA_PCIE

The VDDA_PCIE power pin is noise sensitive. Noise on this pin may translate into unwanted jitter on the serial bit streams. The VDDA_PCIE pin should be connected to a filtered 3.3V supply (for an example filter circuit, see Figure 7).

Based on the circuit in the figure, the VDDA_PCIE net can be implemented as a local plane located under the QFP (see Figure 8). Ideally, the 0.1uF capacitor is located under the QFP within the breakout vias. If this is not possible, the capacitor should be very close to the edge of the package. The net from the ferrite bead to the VDDA_PCIE pins can be implemented as a trace from the ferrite bead to the VDDA_PCIE pin breakout via, as long as the decoupling capacitor is located under the QFP and connected to the breakout via with short traces.

Figure 7: Supply Filter Circuit for VDDA_PCIE
2.1.2 VDD_PCIE

VDD_PCIE pins supply 1.05V to the SerDes circuit internal to the PEB383. The noise restriction on VDD_PCIE is 50mV peak-to-peak. A filter circuit, as displayed in Figure 9, can be used to reduce the noise from the main 1.05V supply. Ideally, the 0.1uF capacitors are located under the QFP within the breakout vias. If this is not possible, the capacitors should be very close to the edge of the package. Preferably, the VDD_PCIE net is implemented as a local power plane under the QFP. Figure 10 shows the VDD_PCIE net implemented with a local power plane.
Figure 10: VDD_PCIE Filter Layout Example

- Filtered VDD_PCIE Power Plane (Green Outline)
- Ferrite Bead
- 1.0uF Capacitor (Top Layer)
- 0.1uF Capacitor (Bottom Layer)
- 0.1uF Capacitors (on Top and Bottom Layers)
2.1.3 **VDDA_PLL**

The VDDA_PLL supply pin provides power to the internal clock PLL circuit inside the PEB383. Use a filter circuit similar to the one in Figure 11. Place the 0.1uF and 0.01uF capacitors in the via breakout under the QFP.

**Figure 11: Supply Filter Circuit for VDDA_PLL**

![Supply Filter Circuit for VDDA_PLL](image)
Figure 12: VDDA_PLL Filter Layout Example

- **VDDA_PLL Breakout Via**
- **VSSA_PLL Breakout Via**
- **0.01uF Capacitor**
- **0.1uF Capacitor**
- **0 Ohm Resistor to Ground**
- **Ferrite Bead**
### 2.2 Decoupling Capacitors

The PEB383 decoupling recommendation is provided in Table 5. Other components, including PCI connectors, must also be adequately decoupled.

**Table 5: Decoupling Capacitor Recommendations**

<table>
<thead>
<tr>
<th>Rail</th>
<th>Capacitor value</th>
<th>Quantity</th>
<th>Placement</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VDD</td>
<td>0.1uF</td>
<td>6</td>
<td>Ideally several are placed within the QFP breakout. Others are placed close to the QFP.</td>
<td>Ceramic (X5R or X7R) Surface mount Case size 0402 or 0603</td>
</tr>
<tr>
<td>VDD</td>
<td>1uF</td>
<td>1</td>
<td>Close to PEB383</td>
<td></td>
</tr>
<tr>
<td>VDD_PCI</td>
<td>0.1uF</td>
<td>7</td>
<td>Close to PEB383</td>
<td></td>
</tr>
<tr>
<td>VDD_PCI</td>
<td>1uF</td>
<td>7</td>
<td>Close to PEB383</td>
<td></td>
</tr>
<tr>
<td>VIO_PCI</td>
<td>0.1uF</td>
<td>3</td>
<td>Close to PEB383</td>
<td></td>
</tr>
</tbody>
</table>

The layout example in Figure 13, Figure 14 and Figure 15 show one method for inserting decoupling capacitors within the QFP breakout.
Figure 13: Top View of Bottom Layer, Decoupling Capacitors for VDD
Figure 14: Top View of Bottom Layer, Decoupling Capacitors for VDD_PCI

GREEN : VDD_PCI Decoupling Capacitors (some are located at the bottom layer)
Figure 15: Top View of Bottom Layer, Decoupling Capacitors VIO_PCI

GREEN : VIO_PCI Decoupling Capacitors (some are located at the bottom layer)
3. **PCI Bus Design Notes**

Topics discussed include the following:

- “PCI Clocking Modes”
- “Bus Routing Rules”

3.1 **PCI Clocking Modes**

The PEB383 supports two clocking modes: Master and Slave. These clocking modes are discussed in the following sections.

3.1.1 **PCI Slave Mode Clocking**

Slave mode clocking requires an external PCI clock source (see Figure 16). The clock source must connect to the PEB383 PCI_CLK, as well as to all other PCI devices on the same bus. Clock signal trace lengths should all be matched.

**Figure 16: Slave Mode Clocking**

![Slave Mode Clocking Diagram](image)

**Table 6: Slave Mode Setup**

<table>
<thead>
<tr>
<th>Signal</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>PWRUP_PLL_BYPASS</td>
<td>Don’t care</td>
</tr>
<tr>
<td>PCI_M66EN</td>
<td>Don’t care</td>
</tr>
</tbody>
</table>
3.1.2 PCI Master Mode Clocking

Master mode clocking uses the PCIe input clock to generate the PCI clock outputs (see Figure 17). PCI_CLKO[4] should be connected to PCI_CLK for PCB trace and internal delay compensation. In master mode, it is important to set PCI_M66EN at the appropriate level because this signal is used to set the output clock frequency.

Figure 17: Master Mode Clocking – PCI Clock for All Frequencies

For unused PCI_CLKOs, they can be left unconnected. To save power, unused PCI_CLKOs should be disabled in the “Clock Out Enable Function and Debug Register” register.

Table 7: Master Mode Setup

<table>
<thead>
<tr>
<th>Signal</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>PWRUP_PLL_BYPASS</td>
<td>0</td>
</tr>
</tbody>
</table>

PCI connectors or PCI devices
3.2 Bus Routing Rules

The PCB trace parameters outlined in Tables 8 and 9 are derived from the PCI Specification's System Timing Budget and the PEB383’s setup and hold timing parameters.

Table 8: PCI Clock Trace Routing Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Implementation Guideline</th>
</tr>
</thead>
</table>
| Clock length matching| Clock traces from the clock source to each PCI device should be length matched, including the trace to PEB383’s PCI_CLK. When routing the clock signal to a PCI connector, the path through the connector and through the plug-in card finger-connector to the PCI device should be taken into account when calculating the PEB383’s PCI_CLK trace length. The connector + plug-in card clock path length is 2.5 inches + connector delay. 
  a. Delay through the connector varies with the connector type. Consider adding 0.3” to 0.7” depending on the connector. |
| Trace impedance      | 60 Ohms +/- 15%                                                                           |
| Minimum spacing      | Three times the trace width from any other traces.                                        |
| Resistor             | Place 33-Ohm resistors close to PEB383’s clock outputs.                                   |

Table 9: PCI Bus Routing Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Implementation Guideline</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum bus length</td>
<td>For 66-MHz operation: 27 inches from the PEB383 to the last load on the bus, including connectors and plug-in card trace lengths.</td>
</tr>
</tbody>
</table>
| Skew with clock      | The clock traces should be matched in length. When all PCI devices see the clock rising edges at the same time, PCI bus timing budget specifies the propagation delay for the PCI bus signals between all the devices to be: 6ns for 66 MHz and 12ns for 33 MHz. However, the propagation delay must also include delays due to slow edge rates and other waveform distortion such as overshoots and undershoots. This propagation delay varies significantly depending on the bus topology and signaling level. As a general guideline the PCI bus skew with clock should be kept within:  
  • 15” for 66-MHz operation  
  • 30” for 33-MHz operation |
| Trace impedance      | 60 Ohms +/- 15%                                                                           |
**PCI Pull Ups**

As per the PCI specification, pull-up resistors are required on the signals listed in Table 10. PCI bus signals should be pulled up to the VIO rail.

**Table 10: PCI Signal Requiring Pull-up Resistor**

<table>
<thead>
<tr>
<th>Signal Name</th>
<th>Resistor value</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCI_FRAME[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_TRDY[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_IRDY[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_DEVSEL[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_STOP[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_SERR[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_PERR[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_LOCK[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_INTA[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_INTB[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_INTC[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_INTC[n]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_REQUEST[n][3:0]</td>
<td>8.2K Ohm</td>
</tr>
<tr>
<td>PCI_GNT[n][0]</td>
<td>8.2K Ohm only when an external arbiter is used</td>
</tr>
<tr>
<td>M66EN (when PEB383 is used on a system board)</td>
<td>10K Ohm</td>
</tr>
</tbody>
</table>

M66EN (when PEB383 is used on a system board) | 10K Ohm |