Transceiver Signal Integrity Development Kit, Stratix IV GX Edition User Guide

101 Innovation Drive
San Jose, CA 95134
www.altera.com

P30-38445-00

Document Version: 2.0
Document Date: December 2009

UG-01057-2.0

Contents

Chapter 1. About This Kit
Introduction 1–1

Kit Features 1–1

Chapter 2. Getting Started
Introduction 2–1
Before You Begin 2–1
Check the Kit Contents 2–1
Inspect the Board 2–1
Hardware Requirements 2–2
Software Requirements 2–2
Quartus II Programmer System Requirements 2–2
References 2–2

Chapter 3. Software Installation
Introduction 3–1
Installing the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM 3–1
Installing the Quartus II Software 3–2
Installing the USB-Blaster Driver 3–2

Chapter 4. Development Board Setup
Introduction 4–1
Powering Up the Board 4–1

Chapter 5. Board Update Portal
Introduction 5–1
Board Update Portal 5–1
Connecting to the Board Update Portal Web Page 5–1
Using the Board Update Portal to Update User Designs 5–2

Chapter 6. Stratix IV GX Transceiver Signal Integrity Demonstration
Introduction 6–1
Test Designs 6–1
Configuring the FPGA Using Quartus II Programmer 6–1
LCD Information 6–2
Running the Demonstration Application and Test Designs 6–2
Demonstration Application Description 6–3
Analog Settings 6–3
Resets 6–4
Help 6–4
Power Down 6–4
Serial Loopback 6–4
Autolink Setup 6–5
Autolink Start and Stop 6–5
Data Patterns 6–5
Link Statistics Tab 6–6
PCI Express Tab 6–6
Power and Temperature Tab 6–6
Channel Reconfig Tab 6–7
EyeQ Tab 6–7

Appendix A. Programming the Flash Device
Overview A–1
Parallel Flash Loader A–1
Board Update Portal CFI Flash Memory Map A–1
Custom User Design A–2
Creating Flash Files Using the Nios II EDS A–2
Nios II EDS Flash Programming Instructions A–3
Restoring the Factory Design to the Flash Device A–4
Nios II EDS Recovery Instructions A–4

Additional Information About–1
Revision History About–1
How to Contact Altera About–1
Typographic Conventions About–2

1. About This Kit

Introduction

The Transceiver Signal Integrity Development Kit, Stratix® IV GX Edition provides everything you need for the signal integrity evaluation and interoperability of Stratix IV GX transceivers on the Altera® Stratix IV GX EP4SGX230 device. The kit includes a full-featured FPGA development board, hardware and software evaluation tools, documentation, and accessories needed to begin development.

With this signal integrity development kit, you can do the following:

- Evaluate transceiver performance at data rate ranging from 600 Mbps to 8.5 Gbps.
- Generate and check pseudo-random binary sequence (PRBS) patterns through an easy-to-use demonstration application (does not require the Quartus® II software).
- Understand the effects of changing differential output voltage (V_{OD}), preemphasis, and equalization settings.
- Perform jitter analysis.
- Verify physical medium attachment (PMA) compliance to PCI Express (Gen 1 and Gen 2), Serial RapidIO®, gigabit Ethernet, 10 gigabit Ethernet XAUI, CEI-6G, high definition serial digital interface (HD-SDI), Fibre Channel 1G/4G/8G, and other major standards.

Kit Features

This section briefly describes the following Stratix IV GX transceiver signal integrity development kit features:

- **Stratix IV GX EP4SGX230 Signal Integrity Transceiver Development Board**—a development platform that allows you to develop and prototype hardware designs running on the Stratix IV GX FPGA (ordering code: DK-SI-4SGX230N).

  For detailed information about board components and interfaces, refer to the *Transceiver Signal Integrity Development Kit, Stratix IV GX Edition Reference Manual*.

- **Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM**—This CD-ROM includes the following:
  - Schematic and board design files
  - Design examples for the Board Update Portal Embedded Nios® II webserver and the Golden Top Level Project
  - Device data sheets
  - Stratix IV GX Transceiver Signal Integrity Demonstration software
  - Quartus II Stand-Alone Programmer software
  - Stratix IV GX signal integrity development kit application and device driver
  - Complete documentation:
    - *Transceiver Signal Integrity Development Kit, Stratix IV GX Edition Getting Started User Guide* (this document)—Describes how to use the kit.
    - *Transceiver Signal Integrity Development Kit, Stratix IV GX Edition Reference Manual*—Provides specific information about the board components and interfaces, steps for using the board, and pin-outs and signal specifications.
    - **Readme.txt**—Contains special instructions and refers to the kit documentation.
- **Power Supply and Cable**—The following items are included in the development kit:
  - USB cable
  - Ethernet CAT-5/RJ-45 cable
  - Power supply and AC adapters for North America, Japan, Europe, and the United Kingdom

2. Getting Started

Introduction

This user guide familiarizes you with the contents of the kit and guides you through the Stratix IV GX transceiver signal integrity board setup. Using this user guide, you can do the following:

- Inspect the contents of the kit
- Install the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM
- Set up, power up, and verify correct operation of the signal integrity board
- Configure the Stratix IV GX FPGA
- Run the signal integrity software and use the test designs

For complete information about the signal integrity board, refer to the *Transceiver Signal Integrity Development Kit, Stratix IV GX Edition Reference Manual*.

Before You Begin

Before using the kit or installing the software, check the kit contents and inspect the board to verify that you received all of the items listed in this section. If any of the items are missing, contact Altera before you proceed.

Check the Kit Contents

Refer to "Kit Features" on page 1–1 for the contents of your kit.

To ensure that you have the most up-to-date information about this product, go to the Altera website at www.altera.com/products/devkits/kit-dev_platforms.jsp.

Inspect the Board

To inspect the board, perform the following steps:

1. Place the board on an anti-static surface and inspect it to ensure that it has not been damaged during shipment.

   CAUTION: Without proper anti-static handling, the Stratix IV GX transceiver signal integrity board can be damaged.

2. Verify that all components are on the board and appear intact.

In typical applications with the Stratix IV GX transceiver signal integrity board, a heat sink is not necessary. However, under extreme conditions the board may require additional cooling to stay within operating temperature guidelines. You may wish to perform power consumption and thermal modeling to determine whether your application requires additional cooling.

For more information about power consumption and thermal modeling, refer to *AN 358: Thermal Management for FPGAs*.

Hardware Requirements

The kit provides all the hardware you need to use the board.

Software Requirements

The kit requires the following software:

- Windows XP operating system
- Quartus II Programmer

Quartus II Programmer System Requirements

The Quartus II Programmer has some minimum system requirements.

For Quartus II Programmer system requirements, refer to the Altera website at: www.altera.com/products/software/products/quartus2web/sof-quarwebmain.html.

References

For other related information, refer to the following websites:

- For additional daughter cards available for purchase:

www.altera.com/products/devkits/kit-daughter_boards.jsp

- For the Stratix IV device documentation:

www.altera.com/literature/lit-stratix-iv.jsp

- For the eStore if you want to purchase devices:

www.altera.com/buy/devices/buy-devices.html

- For Stratix IV GX OrCAD symbols:

www.altera.com/support/software/download/pcb/pcb-pcb_index.html

- For Nios II 32-bit embedded processor solutions:

www.altera.com/technology/embedded/emb-index.html

3. Software Installation

Introduction

The instructions in this section explain how to install the following software:

- Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM.
- Quartus II software—you do not need the Quartus II software for development kit evaluation; however, Altera recommends you obtain the Quartus II software to create new designs.
- USB-Blaster driver.

Before starting the installation, verify that you have complied with the conditions described in "Software Requirements" on page 2–2.

Installing the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM

To install the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM, perform the following steps:

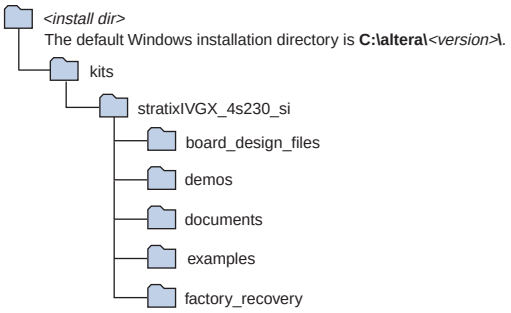
1. Insert the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM into the CD-ROM drive.

   The CD-ROM should start an auto-install process. If it does not, browse to the CD-ROM drive and double-click on the **setup.exe** file.

2. Follow the on-screen instructions to complete the installation process.

The installation program creates the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition directory structure shown in Figure 3–1.

Figure 3–1. Stratix IV GX Transceiver Signal Integrity Kit Installed Directory Structure



Note to Figure 3–1:

(1) Early-release (engineering silicon) versions might have slightly different directory names.

Table 3–1 lists the file directory names and a description of their contents.

Table 3–1. Installed Directory Contents

Directory Name	Description of Contents
board_design_files	Contains schematic, layout, assembly, and bill of material board design files. Use these files as a starting point for a new prototype board design.
demos	Contains demonstration applications that may change from release to release.
documents	Contains the development kit documentation.
examples	Contains the sample design files for the Stratix IV GX transceiver signal integrity development kit.
factory_recovery	Contains the original data programmed onto the board before shipment. Use this data to put the board into the original condition.

Installing the Quartus II Software

The Quartus II software and the Nios II Embedded Design Suite (EDS) are the primary FPGA development tools used to create the reference designs in this development kit. Although running this kit and the included example designs requires only the Quartus II Programmer, to create new designs you can download and install the Altera software tools by performing the following steps:

1. Go to Altera's Download Center.
2. Follow the online instructions to complete the installation process.

If you have difficulty installing the Quartus II software, refer to the *Quartus II Installation & Licensing for Windows and Linux Workstations*.

Installing the USB-Blaster Driver

The Stratix IV GX transceiver signal integrity board includes integrated USB-Blaster circuitry for FPGA programming. However, for the host computer and signal integrity board to communicate, you must install the USB-Blaster driver on the host computer.

To download the USB-Blaster driver, go to the Altera support site at www.altera.com/support/software/drivers/dri-index.html.
To install it, go to www.altera.com/support/software/drivers/usb-blaster/dri-usb-blaster-xp.html.

4. Development Board Setup

Introduction

The instructions in this chapter explain how to power up the signal integrity development board.

Powering Up the Board

To power up the board, perform the following steps:

1. Verify that the MAX/JTAG jumper J26 is OFF, and that the mini-DIP switch SW2 settings located adjacent to jumper J62 match the settings in the following table:

Switch Pin	SW2.1	SW2.2	SW2.3	SW2.4
Label	S0	S1	S2	S3
Position	up or 0	down or 1	down or 1	down or 1

Maintain these settings as they control the speed of the board oscillator. For more information, refer to the *Transceiver Signal Integrity Development Kit, Stratix IV GX Edition Reference Manual*.

2. Connect the power cable to the board and plug the other end into a power outlet.
3. Connect the DC adapter (+16 V, 3.75 A) to the DC power jack (J1).

CAUTION: Use only the supplied 16-V power supply. Power regulation circuitry on the board could be damaged by supplies greater than 16 V.

4. Ensure the POWER switch (SW1) is in the ON position. When power is supplied to the board, LED D3 turns on indicating that the board has power.

After the board powers up, the on-board flash memory, which ships preprogrammed with the factory design, automatically configures the Stratix IV GX device. The FACTORY LED illuminates, signaling that the Stratix IV GX device is configured with the preprogrammed factory design.

5. Board Update Portal

Introduction

This chapter describes the Board Update Portal which allows you to upload new designs and provides access to useful and relevant information about the kit.

Board Update Portal

This development kit is shipped with an example design stored in the factory portion of the flash memory on the board. Whenever jumper J62 is set to LOAD FACTORY, the Stratix IV GX FPGA is automatically configured with the Board Update Portal example design. The example design is an embedded webserver, which serves the Board Update Portal web page. The web page allows you to upload new FPGA designs to the designated user flash memory on your board, and also provides links to useful information, on the Altera website at www.altera.com, including links to kit-specific and design resources.

After the Board Update Portal is used to successfully update a user design, jumper J62 can be set to LOAD USER and the user design configures upon reset or power up. This cycle can be repeated for different user designs as long as the factory Board Update Portal is preserved. If the Board Update Portal is corrupted or deleted from the flash memory, refer to "Restoring the Factory Design to the Flash Device" on page A–4 to restore the board to factory condition.

The source for the Board Update Portal design can be found in the **examples** directory installed from the CD-ROM included in the development kit. It consists of a Nios II embedded processor, an Ethernet MAC, and an HTML webserver. When the board is connected to the network, the Nios II processor obtains an IP address and allows the browser access to its HTML web page.

This section provides instructions on how to connect to the Board Update Portal web page.

Before you proceed, ensure that you have the following:

- A PC with a connection to a working Ethernet port on a DHCP enabled network.
- A separate working Ethernet port connected to the same network for the board.
- The Ethernet and power cables that are included in the kit.

Connecting to the Board Update Portal Web Page

1. With the board powered down, make sure jumper J62 is in the LOAD FACTORY position.
2. Attach the Ethernet cable from the board to the LAN.
3. Power up the board. The board connects to the LAN's gateway router, and obtains an IP address. The LCD on the board displays the IP address.

4. Launch a web browser on a PC that is connected to the same network, and enter the IP address from the LCD into the browser address bar. The Board Update Portal web page appears in the browser page.
5. Click **Signal Integrity Development Kit** to view the latest version of the development kit software (the software version also appears on the CD-ROM).

   If you download new software, double-click the downloaded **.exe** file to begin the installation process.

   Visit the Board Update Portal web page to check for additional new designs that are not included on the CD-ROM and documentation updates.

If the Board Update Portal cannot connect for some reason, then go to www.altera.com/products/devkits/altera/kit-signal_integrity_sivgx.html to ensure that the board has the latest kit software.

Using the Board Update Portal to Update User Designs

The Board Update Portal allows remote update of new FPGA configurations to the user portion of the flash memory. Perform the following steps to update the user memory on your board with a design downloaded from the Altera website.

1. Type the IP address displayed on the LCD into the web browser on the PC or use the browser's **Back** button to return to the main Board Update Portal web page.
2. Click the **Browse** button next to the **Hardware File Name** field and browse to the new **.flash** file that was downloaded. If there is a software component to the design, then include it in the same manner in the **Software File Name** field, otherwise leave the field empty.
3. Click **Upload**. The progress bar indicates the percent complete.
4. After the upload process is complete, the FPGA can be configured with the new image. To do this, change jumper J62 to LOAD USER. To enable reconfiguration of the FPGA, power up the board again or press RESET SW8. The design running in flash is the selected **.flash** file.

As long as the factory image is not overwritten, the Board Update Portal can be used to update new user images in the same manner. If the factory memory image is overwritten, it can be restored by following the instructions in "Restoring the Factory Design to the Flash Device" on page A–4.

6. Stratix IV GX Transceiver Signal Integrity Demonstration

Introduction

The kit installs a demonstration application and test designs. The application provides an easy-to-use interface where you can select various transceiver settings and observe the result. Before you run the application and test designs, connect the USB cable to the board and navigate to the Stratix IV GX Transceiver Signal Integrity Demonstration application as explained in "Installing the Transceiver Signal Integrity Development Kit, Stratix IV GX Edition CD-ROM" on page 3–1.

Test Designs

Altera provides a set of test designs for the evaluation of the Stratix IV GX device transceiver performance and board features. The **.sof** file names and data rates for each test design are listed in Table 6–1. Before you run the application, use the Quartus II Programmer to configure the Stratix IV GX device with one of the **.sof** files.

Table 6–1. Data Rates for Test Designs

File Name	Transceiver Block 0 Channel 0 Data Rate (Gbps)	Transceiver Block 1 Channel 1, Channel 2 Data Rate (Gbps)	Transceiver Block 1 Channel 3, Channel 4 Data Rate (Gbps)	Transceiver Block 2 Channel 7 Data Rate (Gbps)	Clocking scheme
signal_integrity_demo1.sof	soc_clk*20 *(1)*	3.125	PCI Express (Gen1 or Gen2)	soc_clk*20 *(1)*	socket clock
signal_integrity_demo2.sof	soc_clk*20 *(1)*	soc_clk*20 *(1)*	soc_clk*20 *(1)*	soc_clk*20 *(1)*	socket clock
signal_integrity_demo3.sof	Refer to "Channel Reconfig Tab" on page 6–7 for the available transceiver channel data rates and functional modes.				socket clock or external SMA clock (J19 and J20)

Note to Table 6–1:

(1) The clock input is expected from the socketed oscillator (Y3) (soc_clk) on the board. If you are targeting –2 speed grade device on the board, you can use 425 MHz. For –3 speed grade, the maximum data rate limit supported is 6.5 Gbps. Therefore, you must use 312.5 MHz (312.5*20 < 6.5 Gbps) and not the 425-MHz clock crystal.

Configuring the FPGA Using Quartus II Programmer

It is sometimes necessary to use the Quartus II Programmer to configure the FPGA with specific **.sof** files, such as the designs in Table 6–1. Before configuring the FPGA, ensure that the Quartus II Programmer and the USB-Blaster driver are installed on the host computer and the development board is powered up.

To configure the Stratix IV GX FPGA, perform the following steps:

1. Start the Quartus II Programmer.
2. Click **Add File** and select the path to the desired **.sof**.
3. Turn on the **Program/Configure** option for the added file.

4. Click **Start** to download the selected file to the FPGA. The FPGA is configured when the progress bar reaches 100%.

To determine if the appropriate test design **.sof** is programmed, check the LCD for the test design number. Refer to "LCD Information" for more information.

LCD Information

The LCD shows the following information:

- The Stratix IV GX device junction temperature in Celsius.
- The power in watts for the different transceivers (VCCA_L/R, VCCT, VCCR, VCCH_GXB, VCCL_GXB) and core (VCC) voltage supply rails. Turn the rotary switch SW16 to observe the different voltage supply values.
  To learn more about the switch position for displaying the various supply rail values on the LCD, click the **Help** button in the demonstration application.
- The sof/pof number that is programmed is indicated by 'pof' followed by the number.

The power values shown for the VCCA_L/R and the VCCH_GXB assume that the jumper settings are set to 3.0 V and 2.5 V, respectively. Click the **Help** button to see the required jumper settings.

Running the Demonstration Application and Test Designs

The demonstration application communicates with the set of test designs provided with the kit. You can change the various transceiver parameters that are described in this section.

To run the application, make sure that the board is powered up properly and there is a USB cable attached, then go to the *<install dir>***\kits\stratixIVGX_4sgx230_si\demos** directory and double-click on **stratixIVGX_si_demo.exe** file. Ensure that the Stratix IV FPGA is programmed with the **.sof** specified in Table 6–1 on page 6–1.

To enable the application to communicate with the board, click the **Connect** button. To close the application, click the **Disconnect** button (Figure 6–1).

Because the demonstration application communicates with the board using the same interface as the Quartus II Programmer or SignalTap® II Embedded Logic Analyzer, you can run only one of these applications at a time.

The application will not run unless the USB cable is attached, power is applied, and the correct **.sof** is programmed to the Stratix IV GX FPGA as specified in Table 6–1 on page 6–1.

CAUTION

For operating system stability, keep the USB cable connected and the board powered ON when running the demonstration application.

Demonstration Application Description

The demonstration application provides an easy-to-use interface (Figure 6–1) to change transceiver parameters and observe the performance. You can customize your board design by choosing transceiver settings from the categories described in this section.

Figure 6–1. Demonstration Application Control Panel Window

Analog Settings

You can use the application to dynamically control transceiver PMA settings for the different transceiver blocks. Select the VCCHTX value based on the jumper (J11) settings. Similarly, ensure that the VCCA_L/R supply jumper (J6) is connected according the settings shown in Table 6–2.

Table 6–2. Jumper Header Connections *(Note 1)*

Jumper Header Connection VCCHTX (Volt)		Jumper Header Connection VCCA_L/R (Volt)	
Pins 1 and 2	1.4 V	Pins 2 and 3	3.0 V

Note to Table 6–2:

(1) Before you power up the board, specify the VCCHTX and the VCCA jumper settings.

The following list defines the analog setting parameters in the control panel window:

Parameter	Description
VOD	differential output driver voltage.
EQ	equalization.
Gain	DC gain.
PE	preemphasis/deemphasis. **pre**, **1stpost**, and **2ndpost** settings represent different taps.

Resets

The following list describes the available resets:

Reset	Description
System	Reset for the transceiver.
Error	Reset for all the error counters to zero.

After the **System Reset** is asserted, the **DataChk Status** may show **unsynced** for some channels due to the asynchronous nature of the reset. Asserting the **Data Patrst** synchronizes the error checker to the transmitted data.

Help

The **Help** button displays the image of the Stratix IV signal integrity board and also highlights the channel locations and their data rates based on the **.sof** loaded.

Power Down

Turn on **Powerdown** to power down the transceiver block.

Serial Loopback

Serial loopback is available for all the channels and can be controlled during run time. After the serial loopback status in the interface changes, the **DataChk Status** field may show **unsynced** for some channels due to the asynchronous nature of the serial loopback signal. The **Data Patrst** should be asserted in this case to synchronize the error checker with the transmitted data.

In the **signal_integrity_demo1.sof**, internal serial loopback is not enabled for the two channels configured in PCI Express (PIPE) mode. Therefore, you must connect an external SMA cable between the transmit output and the receive input to loopback the data.

Autolink Setup

This feature provides the following options:

Parameter	Description
Manual	You can manually change the preemphasis and equalization to observe the setting that provides an error free link. For more information about **DataChk Status**, refer to "Link Statistics Tab" on page 6–6.
PEandEQ	When you select this option and the **start** button, the hardware finds a preemphasis and equalization setting that meets the bit-error rate (BER) of 10–12.
EQonly	When you select this option and the **start** button, the hardware finds an equalization setting that meets the BER of 10–12.
PEonly	When you select this option and the **start** button, the hardware finds the preemphasis settings (**pre** tap and **1stpost** tap) to obtain an error free link.

The receiver data checker must be active to determine whether the settings attempted by the hardware are successful. Therefore, you must connect the transmitter output to the receiver input through external cable.

If you are evaluating the characteristic of a third party upstream transmitter connected to the Stratix IV GX receiver, you must supply the appropriate data pattern as selected in the application, for the data checker to determine the BER for a given setting.

Autolink Start and Stop

Selecting the **start** option enables the hardware to perform the operation specified in **Autolink setup**. After this setup is complete, the **stop** field gets highlighted in green or red. Set the **stop** field and record the converged settings. The following explains the color coding of the **stop** field:

Color Code	Description
green	the hardware successfully finds the settings that give an error free link.
red	if the hardware is not able to find the settings, it provides the setting that yielded the least number of errors.
yellow	the hardware is finding the PMA settings.

The **DataChk Status** field also shows whether the hardware is **in progress**, **done** (successful), or **no setting** (failure) status.

Data Patterns

The application supports PRBS15i, PRBS7, PRBS23, CJTPAT, compliance pattern (only for specific designs and channels), high frequency, and low frequency patterns. No synchronization patterns are sent prior to sending the PRBS pattern. Therefore, you can use a third party receiver to recognize the PRBS/CJTPAT data.

Data verifiers are not available for the high frequency (1010..) and low frequency (5'1s and 5'0s) patterns.

Link Statistics Tab

The **Link Statistics** settings include the following options:

Parameter	Description
What statistic to display?	Displays the BER, number of bits received, number of errors received, and the error slope based on the selection from the list. The error slope shows the error trend (increase or decrease). You can use this statistic to determine whether the PMA control settings must be increased or decreased to get an error free link.
Inject Error	Injects errors in the channels. Every time this button is asserted, one-bit error is introduced.
Data Patrst	Reset for the data pattern generators and checkers.
Data Rate	Based on the test design selected, the application displays the serial data rate of the transceiver channels.
GXB Encoding	Displays whether the data sent by the test design is 8B/10B encoded.
DataChk Status	The **DataChk Status** field displays the following: ■ **synced** status displayed in green indicates that the error checker has received the predefined header byte and no errors are detected. ■ **unsynced** status displayed in gray indicates that the error checker has not received the selected pattern. ■ If the **DataChk Status** field shows **unsynced**, check whether the transmitter of the channel showing **unsynced** is connected to the receiver channel by external cable or by internal serial loopback. However, when sending the high frequency data pattern, the **DataChk Status** shows **unsynced**. This is because the error checker is not provided for high frequency pattern. ■ **error** status indicates that the error checker is detecting errors in the received pattern.
CDRLock Mode	Shows whether the transceiver Clock Recovery Unit (CRU) is locked to the reference clock or to the data. When the transceiver locks to the incoming data, this field displays **data** indicating that the receive PLL has recovered the clock from the incoming data.
Freeze Display	When you click the **Freeze display** button, the display field does not change and the counting continues. When you click the **Unfreeze display**, the current running values are shown.

PCI Express Tab

This tab is only available when **signal_integrity_demo1.sof** is loaded. It provides the options to control **PCI Express (PIPE)** parameters for channels 3 and 4 in the middle transceiver block. Turning on **Switch to Gen2 data rate** enables the hardware to change channels 3 and 4 to **Gen2 data rate** and makes the following options available:

Parameter	Description
Txdeemphasis	0: –6 db; 1: –3.5 db
TxMargin	0–7

If you have selected the **Switch to Gen2 data rate** option, set the **pre** tap and **2ndpost** tap to 0 to meet the Gen 2 specification.

Power and Temperature Tab

The application displays the Stratix IV device junction temperature. It also shows the power or current values for the six supply rails.

You can also observe the power and temperature values on the LCD. For more information, refer to "LCD Information" on page 6–2.

The power values shown for the VCCA_L/R and the VCCH_GXB assumes that the jumper settings are set to 3.0 V and 1.4 V, respectively. Click the **Help** button to see the required jumper settings.

Channel Reconfig Tab

This tab is only available when **signal_integrity_demo3.sof** is loaded. You can dynamically select the input reference clock from the socket clock input or an external SMA clock input (J19 and J20). To receive the input clock from the external SMA, turn on **switch to SMA clock**.

The serial data rate of each transceiver channel can be 16 times or 20 times the clock rate. The **Change Data Rate** controls configure each transceiver block with one of the following data rates:

5 Gbps—Configures the transceiver channel to run at 16 times the input reference clock. If input reference clock is 312.5 MHz, the transceiver runs at 5 Gbps.

6.25 Gbps—Configures the transceiver channel to run at 20 times the input reference clock. If input reference clock is 312.5 MHz, the transceiver runs at 6.25 Gbps.

Reverse serial 5G (Post CDR)—Configures the transceiver channel in reverse serial loop back mode. In this configuration, the output of the RX CDR that is configured to track serial data input at 16 times the input reference clock is looped to the transmitter serializer. The serializer is clocked by the recovered clock generated by the RX CDR. Data checkers are not available for this option.

Reverse serial 6.25G (Post CDR)—Configures the transceiver channel in reverse serial loop back mode. In this configuration, the output of the RX CDR that is configured to track serial data input at 20 times the input reference clock is looped to the transmitter serializer. The serializer is clocked by the recovered clock generated by the RX CDR. Data checkers are not available for this option.

Reverse serial (Pre CDR)—Configures the transceiver channel in reverse serial (Pre CDR) mode. The received serial data input is looped to the transmit buffer before it passes through the RX CDR. Data checkers are not available for this option.

EyeQ Tab

This tab is only available when **signal_integrity_demo3.sof** is loaded. The options set the phase step to move the sampling point of the recovered data. For more information about this feature, refer to "EyeQ" in the *Stratix IV Dynamic Reconfiguration* chapter in volume 2 of the *Stratix IV Device Handbook*.

The following EyeQ features can be used in channels configured up to 6 Gbps:

Manual—Sets the phase step from 0 to 31. Once the GUI and the hardware computes 3*10E12 bits, the GUI displays the bit error rate (BER) for the phase step, plots the value, and records the BER information for each phase step in a **.csv** format. The **block**<*block number*>**stats.csv** file is saved in the same folder from which the application is launched.

Automatic—Automatically sequences through the phase steps 0-31, plots the value, and records the BER information for each phase step in a **.csv** format.

For proper GUI operation, close all the **.csv** files previously created by the GUI before launching the GUI.

To estimate the eye opening, use the following equation:

Eye opening (as a % of unit interval) = *N* * 1/32 * 100 +/- 3%,

where *N* is the number of phase steps with no errors.

A. Programming the Flash Device

Overview

There is a Common Flash Interface (CFI) type flash memory device on the Stratix IV GX transceiver signal integrity board. When you first receive the kit, the CFI flash device arrives programmed with a default factory FPGA configuration for running the Board Update Portal example design and a default user configuration for running the primary transceiver signal integrity demonstration. There are several other factory software files written to the CFI flash device to support the running of the Board Update Portal. These software files were created using the Nios II EDS just as the hardware design was created using the Quartus II Design application.

For more information about Altera development tools, refer to www.altera.com/products/software/sfw-index.jsp.

Parallel Flash Loader

The development kit features a MAX II configuration design in the *<install dir>*\kits\stratixIVGX_4sgx230_si\examples\maxII_epm1270_si directory that includes a PFL megafunction. The Parallel Flash Loader (PFL) on the MAX II device is used to configure from CFI flash when the RESET SW8 is pressed or the board is powered up.

For more information about the PFL megafunction, refer to *AN 386: Using the Parallel Flash Loader with the Quartus II Software*.

Board Update Portal CFI Flash Memory Map

Table A–1 shows the default memory contents of the 512-Mb (64-MB) Intel PC48F4400P0VB00 CFI flash device. For the Board Update Portal to run correctly and update designs in the user memory, this memory map must not be altered.

Table A–1. Byte Address Flash Memory Map (Part 1 of 2)

Block Description	Size	Address Range
Unused	32 KB	0x03FF8000 - 0x03FFFFFF
Unused	32 KB	0x03FF0000 - 0x03FF7FFF
Unused	32 KB	0x03FE8000 - 0x03FEFFFF
Unused	32 KB	0x03FE0000 - 0x03FE7FFF
User software	24,320 KB	0x02820000 - 0x03FDFFFF
Factory software	8,192 KB	0x02020000 - 0x0281FFFF
zipfs (html, web content)	8,192 KB	0x01820000 - 0x0201FFFF
User hardware	12,288 KB	0x00C20000 - 0x0181FFFF
Factory hardware	12,288 KB	0x00020000 - 0x00C1FFFF
PFL option bits	32 KB	0x00018000 - 0x0001FFFF

Table A–1. Byte Address Flash Memory Map (Part 2 of 2)

Block Description	Size	Address Range
Reserved	32 KB	0x00010000 - 0x00017FFF
Ethernet option bits	32 KB	0x00008000 - 0x0000FFFF
User design reset vector	32 KB	0x00000000 - 0x00007FFF

Using the Board Update Portal or Nios II EDS tools, you can update the user flash configuration with designs provided on Altera's website for demonstration and evaluation. For more information, refer to "Board Update Portal" on page 5–1.

CAUTION

Altera advises against overwriting the FACTORY HW or SW images unless you are expert with the Altera tools or overwriting the factory design is deliberate. If you unintentionally overwrite the FACTORY HW or SW images, refer to "Restoring the Factory Design to the Flash Device" on page A–4.

Custom User Design

As you develop your own project using the Altera tools, you may wish to program the flash device so that, upon power up, it loads the FPGA with your own user design. With the Nios II EDS tool **sof2flash**, your Quartus II compiled **.sof** can be converted to a **.flash** file. The **.flash** file can then be written to the user hardware location of the flash memory using either the Board Update Portal or the Nios II EDS **nios2-flash-programmer** from a Nios II command-line shell.

If you used the Nios II EDS to derive your software design, you can use the Nios II EDS tools to convert your compiled and linked software Executable and Linkable Format File (**.elf**) design to **.flash** files. After your design files are in the **.flash** format, then both hardware and software files can be written using the Board Update Portal or using the Nios II EDS **nios2-flash-programmer**.

For more information about Nios II EDS software tools and practices, refer to www.altera.com/products/ip/processors/nios2/tools/ni2-development_tools.html

The following sections describe how to program the flash device using first the Board Update Portal, then using only the Nios II EDS command-line tools.

Creating Flash Files Using the Nios II EDS

If you have hardware developed using the Quartus II application, and software developed using the Nios II EDS, follow these instructions:

1. On the Windows Start Menu, go to **All Programs**> **altera**> **Nios II EDS**> **Nios II Command Shell**.
2. In the Nios II Command Shell, navigate to the directory where your design files are located and type the following Nios II EDS commands:
   - For **.sof** Quartus II hardware files:

```
sof2flash --input=yourfile_hw.sof --output=yourfile_hw.flash --offset=0xC20000 ↵
```

   - For **.elf** Nios II software files:

```
elf2flash --base=0x0A000000 --end=0x0BFFFFFF --reset=0x0A020000 --input=yourfile_sw.elf
--output=yourfile_sw.flash
--boot=$SOPC_KIT_NIOS2/components/altera_nios2/boot_loader_cfi.srec ↵
```

These steps result in a set of **.flash** files suitable for Board Update Portal writing to the user flash memory. For more information, refer to "Board Update Portal" on page 5–1.

If your own design uses additional software files such as files used by the runtime program or image data, you must first convert the files to **.flash** format and concatenate them into one **your_sw.flash** file before using the Board Update Portal to upload them.

The Board Update Portal standard **.flash** format conventionally uses either <*filename*>**_hw.flash** for hardware design files or <*filename*>**_sw.flash** for software design files.

If you have a hardware **.sof** developed using the Quartus II application that operates without a software design file, you should still use the Nios II EDS tool **sof2flash** to convert the **.sof** to **.flash** file, as described earlier for the hardware files. When the Board Update Portal is implemented in this case, only the **yourfile_hw.flash file** is selected without the **yourfile_sw.flash** file.

You can program the **.flash** files using the Nios II EDS **nios2-flash-programmer** instead of the Board Update Portal as described in the next section.

Nios II EDS Flash Programming Instructions

To program the **.flash** files or any compatible S-Record File (**.srec**) to the board using **nios2-flash-programmer**, perform the following steps:

Firstly, configure the FPGA with a compatible **.sof** for flash-programming:

1. Move jumper J62 from LOAD FACTORY to LOAD USER.
2. With the board powered up and the USB cable attached, as described in "Configuring the FPGA Using Quartus II Programmer" on page 6–1, start the Quartus II Programmer.
3. Click **Add File** and select the following path:
   <*install dir*>**\kits\stratixIVGX_4sgx230_si\factory_recovery\s4gx230_si_bup.sof**
4. Turn on the **Program/Configure** option for the added file.
5. Click **Start** to download the selected configuration file to FPGA. The FPGA is configured when the progress bar reaches 100%, after which the user LEDs flash repeatedly on and off indicating that the flash device is ready for programming.

Secondly, run the Nios II EDS **nios2-flash-programmer**:

1. On the Windows Start Menu, go to **All Programs> altera> Nios II EDS> Nios II Command Shell**.
2. In the Nios II Command Shell, navigate to the <*install dir*>**\kits\stratixIVGX_4sgx230_si\factory_recovery** directory and type the following Nios II EDS command:

```
nios2-flash-programmer  --base=0x08000000  yourfile_hw.flash ↵
```

and after programming completes and if there is a software component, type the following Nios II EDS command:

```
nios2-flash-programmer  --base=0x0A000000  yourfile_sw.flash ↵
```

This step results in writing the **.flash** file or files to their appropriate user locations.

3. Click the Quartus II Programmer **Start** button to download the configuration file to the FPGA. The FPGA is configured when the progress bar reaches 100%, after which the user LEDs flash repeatedly on and off indicating that the flash is written.
4. Repower the board so that the new user design runs on the board.

For more information about the **nios2-flash-programmer** utility, refer to the *Nios II Flash Programmer User Guide*.

Restoring the Factory Design to the Flash Device

To restore the development board to factory condition, the contents of the flash device must be rewritten and the board repowered. Make sure you have the Nios II EDS installed, and perform the instructions in this section.

Nios II EDS Recovery Instructions

1. Set the board switches to the factory default settings described in "Powering Up the Board" on page 4–1.
2. Start the Quartus II Programmer.
3. Click **Add File** and select the following path: <*install dir*>**\kits\stratixIVGX_4sgx230_si\factory_recovery\s4gx230_si_bup .sof**
4. Turn on the **Program/Configure** option for the added file.
5. Click **Start** to download the selected configuration file to FPGA. The FPGA is configured when the progress bar reaches 100%, after which the user LEDs flash repeatedly on and off indicating that the flash device is ready for programming.
6. On the Windows Start Menu, go to **All Programs**> **altera**> **Nios II EDS**> **Nios II Command Shell**.
7. In the Nios II Command Shell, navigate to the <*install dir*>**\kits\stratixIVGX_4sgx230_si\factory_recovery** directory and type the following Nios II EDS commands:

```
nios2-flash-programmer --base=0x08000000 restore_0_s4gx230_si_bup.flash ↵

nios2-flash-programmer --base=0x0A000000 restore_1_s4gx230_si_bup.flash ↵
```

Because the device is a dual-die CFI device, the device base address is referred to differently even though they are one physical device. For more information, refer to Table A–1 on page A–1.

8. After all flash programming completes, cycle POWER switch SW1 OFF then ON again.
9. Using the Quartus II Programmer, click **Add File** and select *<install dir>***kits\stratixIVGX_4sgx230_si\factory_recovery\s4gx230_si_bup.sof**.
10. Turn on the **Program/Configure** option for the added file.
11. Click **Start** to download the selected configuration file to FPGA. The FPGA is configured when the progress bar reaches 100%, after which the user LEDs flash repeatedly on and off indicating that the flash device is now restored to factory condition.
12. Cycle POWER switch SW1 OFF then ON to load and run the restored factory design.
13. The restore script cannot restore the board's MAC address automatically. In the Nios II command shell, type the following Nios II EDS command:

    ```
    nios2-terminal ↵
    ```

    and follow the instructions in the terminal window to generate a unique MAC address.

To ensure that you have the most up-to-date factory restore files and information about this product, go to the Altera website at www.altera.com/products/devkits/kit-dev_platforms.jsp.

Revision History

The following table displays the revision history for this user guide.

Date	Version	Changes Made
December 2009	2.0	■ Added a third test design (**signal_integrity_demo3.sof**) and two new tabs to control it. **signal_integrity_demo3.sof** is only available for the production silicon version of the board. ■ Moved power and temperature information to a separate tab. ■ Changed directory path name from **stratixIVGX_4sgx230es_si** to **stratixIVGX_4sgx230_si**. ■ Changed flash proggramming **.sof** name from **stratixIVGX_4sgx230es_si_flash_prog.sof** to **s4gx230_si_bup.sof**.
February 2009	1.0	Initial release.

How to Contact Altera

For the most up-to-date information about Altera products, refer to the following table.

Contact *(Note 1)*	Contact Method	Address
Technical support	Website	www.altera.com/support
Technical training	Website	www.altera.com/training
	Email	custrain@altera.com
Product literature	Website	www.altera.com/literature
Non-technical support (General)	Email	nacomp@altera.com
(Software Licensing)	Email	authorization@altera.com

Note to Table:

(1) You can also contact your local Altera sales office or sales representative.

Typographic Conventions

This document uses the typographic conventions shown in the following table.

Visual Cue	Meaning
Bold Type with Initial Capital Letters	Indicates command names, dialog box titles, dialog box options, and other GUI labels. For example, **Save As** dialog box.
bold type	Indicates directory names, project names, disk drive names, file names, file name extensions, and software utility names. For example, **\qdesigns** directory, **d:** drive, and **chiptrip.gdf** file.
Italic Type with Initial Capital Letters	Indicates document titles. For example, *AN 519: Stratix IV Design Guidelines.*
Italic type	Indicates variables. For example, *n* + 1. Variable names are enclosed in angle brackets (< >). For example, *<file name>* and *<project name>***.pof** file.
Initial Capital Letters	Indicates keyboard keys and menu names. For example, Delete key and the Options menu.
"Subheading Title"	Quotation marks indicate references to sections within a document and titles of Quartus II Help topics. For example, "Typographic Conventions."
`Courier type`	Indicates signal, port, register, bit, block, and primitive names. For example, `data1`, `tdi`, and `input`. Active-low signals are denoted by suffix `n`. For example, `resetn`. Indicates command line commands and anything that must be typed exactly as it appears. For example, `c:\qdesigns\tutorial\chiptrip.gdf`. Also indicates sections of an actual file, such as a Report File, references to parts of files (for example, the AHDL keyword `SUBDESIGN`), and logic function names (for example, `TRI`).
1., 2., 3., and a., b., c., and so on.	Numbered steps indicate a list of items when the sequence of the items is important, such as the steps listed in a procedure.
■ ■	Bullets indicate a list of items when the sequence of the items is not important.
	The hand points to information that requires special attention.
	A caution calls attention to a condition or possible situation that can damage or destroy the product or your work.
	A warning calls attention to a condition or possible situation that can cause you injury.
	The angled arrow instructs you to press Enter.
	The feet direct you to more information about a particular topic.