# Getting Started with the Virtex-7 FPGA VC707 Evaluation Kit

**UG848 (v1.1) March 1, 2013**

**0402902-02**

# Revision History

The following table shows the revision history for this document.

| Date | Version | Revision |
|---|---|---|
| 07/31/12 | 1.0 | Initial Xilinx release. |
| 03/01/13 | 1.1 | Add Vivado Design Suite to VC707 Evaluation Kit Contents. Removed references to USB flash drive throughout Chapter 1, Getting Started with the Virtex-7 FPGA VC707 Evaluation Kit. Updated Step 1 and removed note in Extract the AMS Design Files, page 17. |

# *Table of Contents*

XILINX®

Chapter 1

# Getting Started with the Virtex-7 FPGA VC707 Evaluation Kit

## Introduction

This document describes how to use the materials provided in the VC707 Evaluation Kit to set up the VC707 board and a host computer to run three reference designs, which test and demonstrate some of the key features of the XC7VX485T FPGA and the VC707 board:

- Built-in self test (BIST)
- Integrated bit error rate test (IBERT)
- Analog mixed signal (AMS) card demonstration

Three additional reference designs that are compatible with the VC707 board are also summarized in this guide.

- Multiboot
- Memory interface generator (MIG)
- Integrated endpoint block for PCI EXPRESS®, and LogiCORE™ IP Ethernet

***Note:*** These design summaries are for use as a quick start method for users who are familiar with Xilinx tools, technology, and reference designs. Additional instructions and background information are available at http://www.xilinx.com/vc707.

## VC707 Evaluation Kit Contents

The VC707 Evaluation Kit includes:

- VC707 board with the Virtex®-7 XC7VX485T FPGA
- ISE® Design Suite: Logic Edition (full seat, device-locked to the XC7VX485T FPGA)
- Vivado™ Design Suite Installation DVD
- Printed entitlement voucher: provides entitlement of the Vivado Design Suite Logic Edition device-locked to the XC7VX485T FPGA. Follow the printed instructions on the voucher to redeem your software entitlement.
- AMS101 evaluation card
- USB cable, standard-A plug to mini-B plug
- USB cable, standard-A plug to micro-B plug
- HDMI™ cable, type-A plug to type-A plug
- Power Supply: 100 VAC–240 VAC input, 12 VDC 5.0A output
- Power cords to support three main plug types

- Getting Started Guide

# Host Computer Requirements

The example designs described in this document require an Intel processor based computer running Windows 7 or Windows XP operating system. The computer must have two USB ports and an Ethernet interface.

***Note:*** The Windows 7 operating system is used in the setup instructions and examples.

# Preliminary Setup

Complete the tasks in this section before running the reference designs.

## Install ISE Software

Install the latest version of the Xilinx ISE Design Suite on the host computer.

## Install the USB UART Drivers

Download and install the Silicon Laboratories CP210x VCP drivers on the host computer. The drivers are available for download at no cost from www.silabs.com/Support Documents/Software/CP210x_VCP_Win_XP_S2K3_Vista_7.exe.

## Configure the Host Computer COM Port

The BIST design uses a terminal program to communicate between the host computer and the VC707 board. To configure the host computer COM port for this purpose:

1. Connect the VC707 board to the host computer and power supply as shown in Figure 1-1.

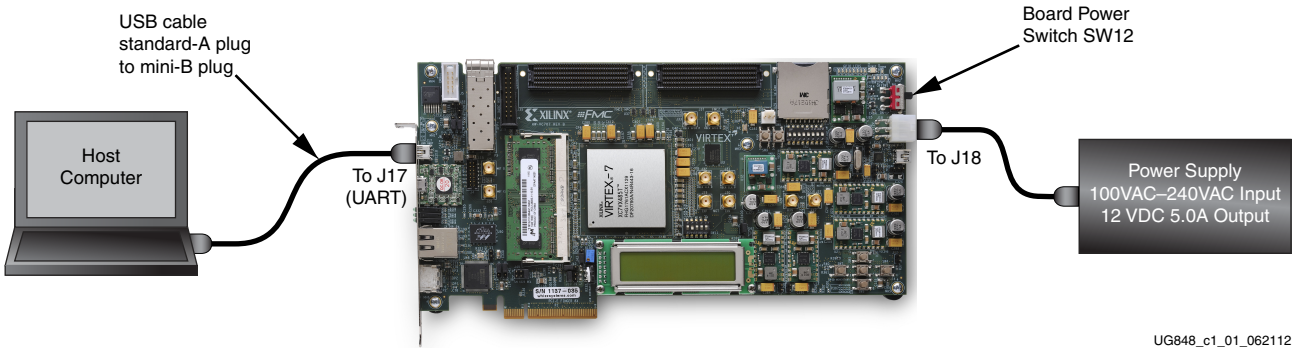



*Figure 1-1:* **Host Computer COM Port Configuration**

2. Turn Board power on (SW12).

3. Open the host computer Device Manager (Figure 1-2). In the Windows task bar, Click **Start**, click **Control Panel**, and then click **Device Manager**.

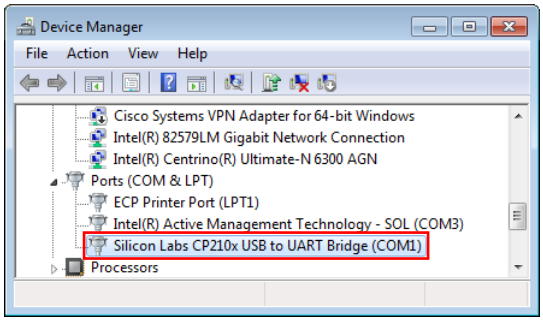



*Figure 1-2:* **Device Manager**

4. Open UART properties. Expand **Ports (COM & LPT)**, right-click **Silicon Labs CP210x USB to UART Bridge**, and then click **Properties**.
5. In the properties window, select the **Port Settings** tab, verify the settings match the values shown in Figure 1-3 and then click **Advanced**.

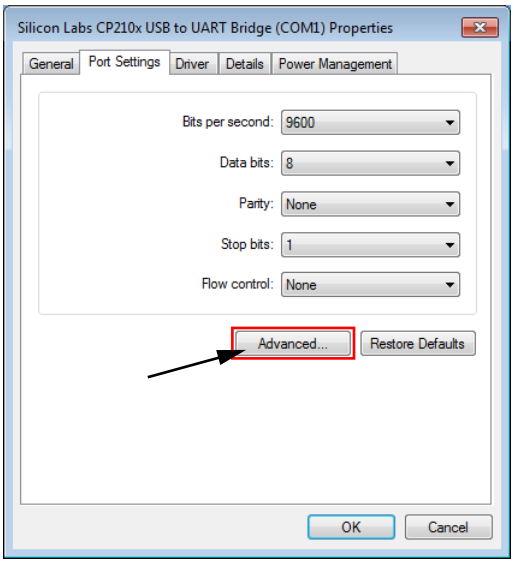



*Figure 1-3:* **Port Settings**

6. Select an unused COM Port Number and then click **OK**. Figure 1-4 shows **COM1** as the selected COM port number.

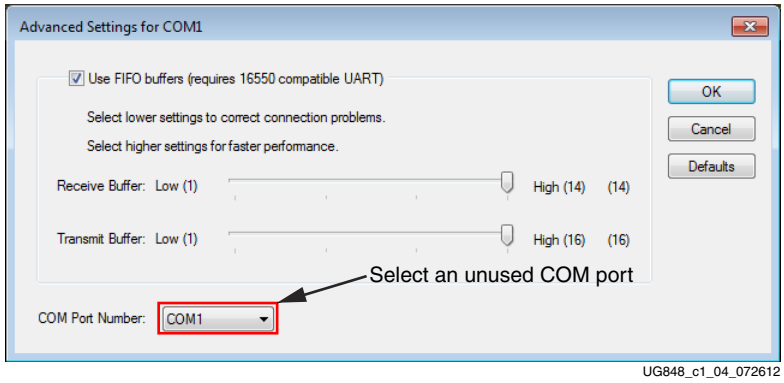



*Figure 1-4:* **Advanced Settings**

7. Click **OK** in the properties window (Figure 1-3, page 7), and then close the Device Manager and the Control Panel.

## Install the Terminal Program

Download and install the TeraTerm Pro terminal program on the host computer. TeraTerm Pro is available for download at no cost from http://www.ayera.com/teraterm/.

To communicate with the VC707 board, configure the New Connection and Serial Port settings as shown in Figure 1-5. These settings must match the host computer COM port settings shown in Figure 1-3, page 7 and Figure 1-4.

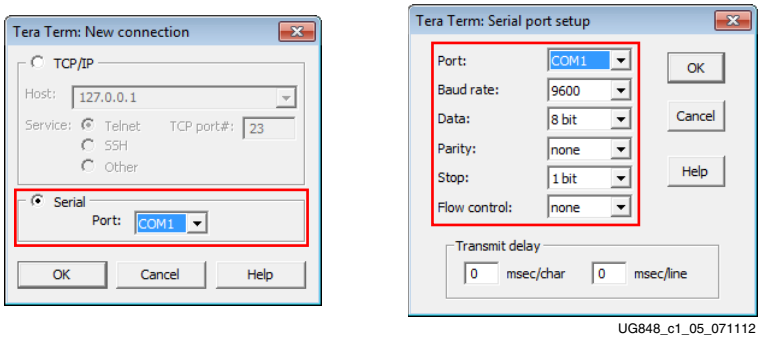



*Figure 1-5:* **TeraTerm Pro Settings**

Preliminary setup is complete and the reference designs can now be run.

## Verify Jumpers are in Default Positions

Verify the jumpers on the VC707 board are positioned as shown in Table 1-1.

*Table 1-1:* **Default Jumper Positions**

| Jumper Connector | Function | Jumper Position |
|---|---|---|
| J6 | SFP Enable | None |
| J7 | EPHY Interface Mode: None = GMII/MII to copper | None |
| J9 | XADC GND ferrite filter bypass jumper | None |
| J10 | XADC GND-to-XADC_AGND jumper | 1–2 |
| J11 | TI Controller U42 Addr 52 Reset jumper | None |
| J12 | TI Controller U43 Addr 53 Reset jumper | None |
| J13 | USB Mini-B Connector J2 VBUS | None |
| J14 | USB SMBC U8 CLKOUT selector | None |
| J38 | SFP RX Rate: 1-2 = Full BW Rate, 2-3 = Low BW Rate | 1–2 |
| J39 | SFP TX Rate: 1-2 = Full BW Rate, 2-3 = Low BW Rate | 1–2 |
| J40 | EPHY Interface Mode: 1-2 = GMII/MII to copper | 1–2 |
| J41 | EPHY Interface Mode: 1-2 = GMII/MII to copper | 1–2 |
| J42 | XADC external 1.2V or internal VREFP selector | 1–2 |
| J43 | XADC VCC Select Header | 2–3 |
| J44 | USB Mini-B Connector J2 GND jumper | None |
| J45 | USB SMBC U8 VBUS | 1–2 |
| J49 | PCIe® Bus Width Select Header | 5–6 |
| J50 | TI Controller U64 Addr 54 Reset jumper | None |
| J51 | FMC_VADJ_ON_B jumper | 1–2 |
| J52 | FPGA U1 INIT_B-to-PROG_B jumper | None |
| J53 | XADC VCC5V0-to-XADC_VCC5V0 jumper | 1–2 |
| J54 | XADC REF3012 U35 $V_{IN}$ Select | 1–2 |

# Built-In Self Test

The BIST tests several XC7VX485T FPGA and VC707 board features. The BIST interface is a menu of tests displayed by a terminal program running on the host computer.

## Project Files

Project files for the BIST are located in `vc707_BIST_rdf0157_<ISE revision>.zip` online as RDF0157 at: http://www.xilinx.com/vc707.

The BIST is pre-loaded in the device and the project files are not required to run this demonstration.

## Run BIST

1. Complete the tasks under Preliminary Setup, page 6.
2. Connect the VC707 board to the host computer and power supply as shown in Figure 1-6.

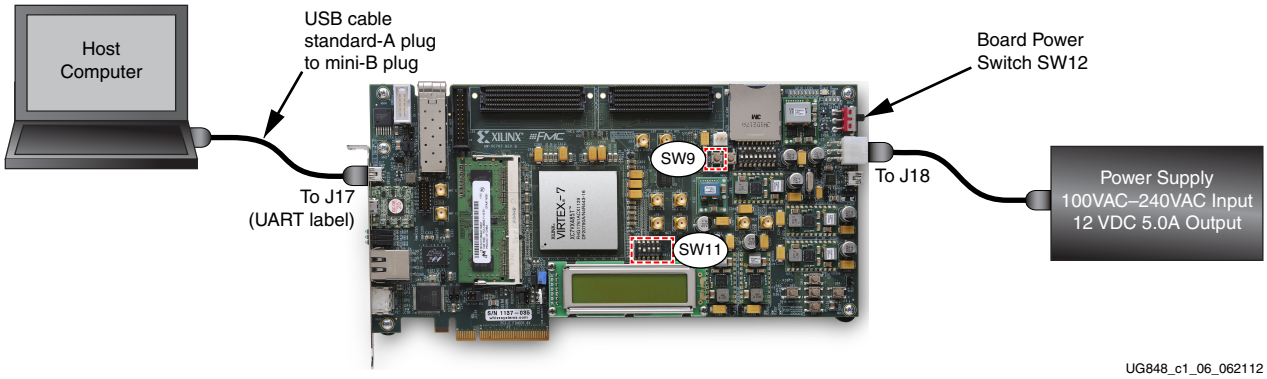



*Figure 1-6:* **BIST Board Connections**

3. Turn board power on (SW12).
4. Set DIP switch SW11 as shown in Figure 1-7.

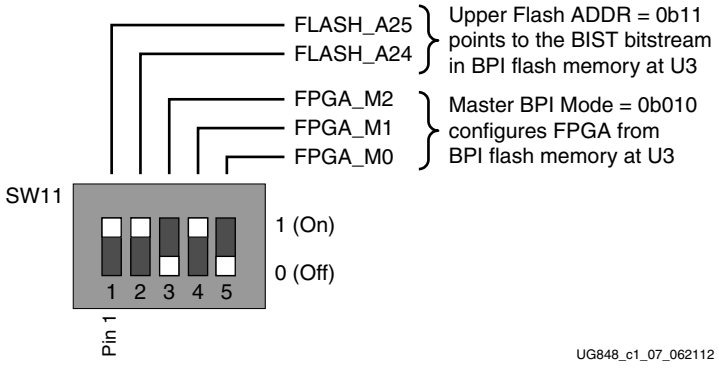



*Figure 1-7:* **SW11 BIST Settings**

5. Press and release the Program button SW9 (Figure 1-6). The BIST bitstream configures the FPGA and then runs BIST. The terminal program displays the BIST menu shown in Figure 1-8.

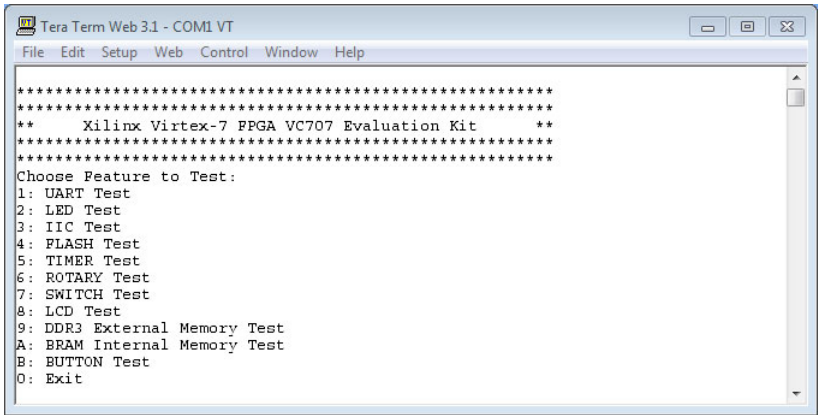



*Figure 1-8:* **BIST Menu**

6. To run a test, type the test number, and press the **`Enter`** key. Press any key to end the test and return to the menu.

# IBERT Demonstration

The IBERT demonstration shows the operation of the GTX transceivers running in near-end PCS loopback mode using the IBERT console window provided by the ChipScope™ Pro Analyzer.

## Extract the Project Files

The `.cpj` project and `.bit` files for the IBERT demonstration are located in `vc707_IBERT_rdf0158_<ISE version>.zip` available online at: http://www.xilinx.com/vc707

To extract the project files, locate the file `vc707_IBERT_rdf0158_<ISE version>.zip` and unzip the files to a working directory on the host computer.

## Set Up the Hardware

1. Complete the tasks under Preliminary Setup, page 6.

2. Connect the VC707 board to the host computer and power supply as shown in Figure 1-9.

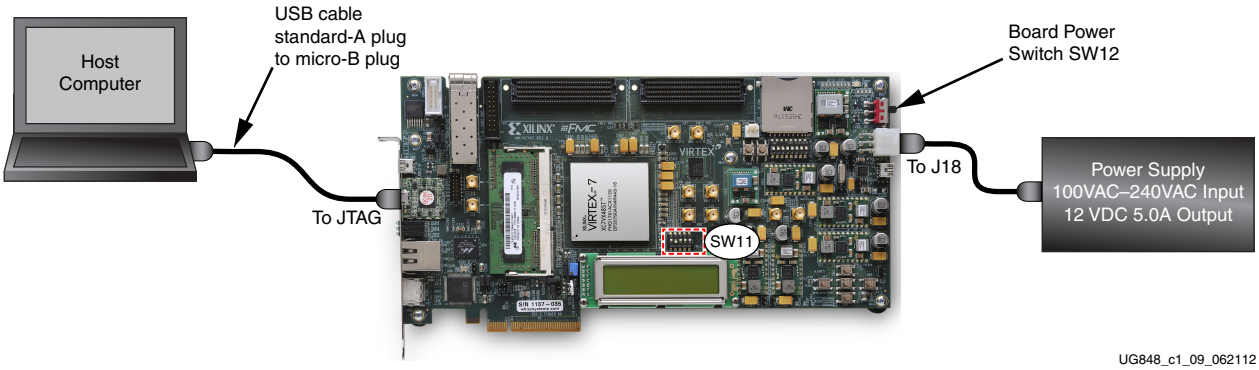



*Figure 1-9:* **IBERT Board Connections**

3. Turn board power on (SW12).
4. Set DIP switch SW11 as shown in Figure 1-10.

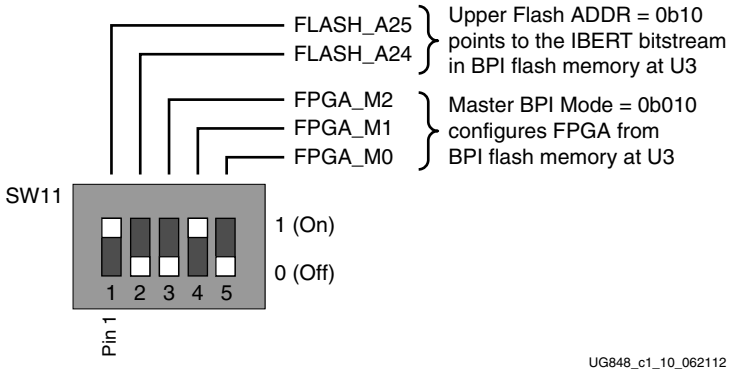



*Figure 1-10:* **SW11 IBERT Settings**

5. Press and release the Program button (SW9). The IBERT bitstream configures the FPGA and then runs the IBERT design.

## Set up ChipScope Pro Analyzer

1. Open the ChipScope Pro Analyzer.
   Click **Start** → **All Programs** → **Xilinx Design Tools** → **ISE Design Suite XX.X** → **ChipScopePro** → ...

   If the host computer system is 64-bits, open **ChipScope 64-bit** → **Analyzer**.
   If the host computer system is 32-bits, open **ChipScope 32-bit** → **Analyzer**.
2. Click **JTAG Chain** → **Digilent USB JTAG Cable** (Figure 1-11).

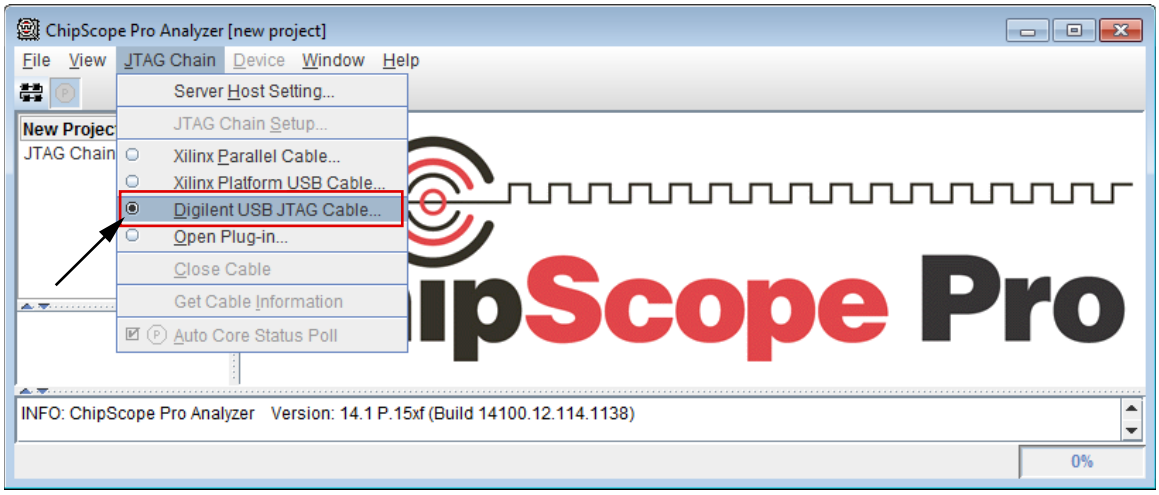



*Figure 1-11:* **Select Digilent USB JTAG Cable**

3. Set the JTAG cable for 10 MHz operation and click **OK** (Figure 1-12).

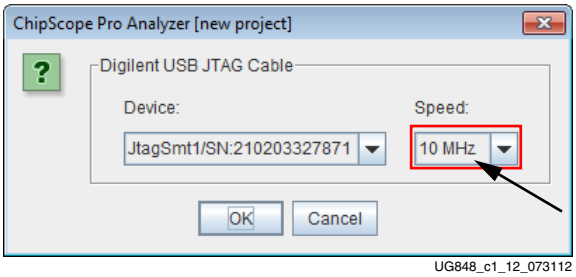



*Figure 1-12:* **Set Cable Speed**

4. Click **OK** (Figure 1-13).

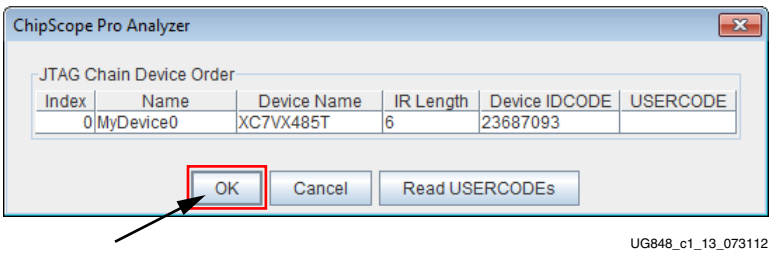



*Figure 1-13:* **JTAG Chain Device Order**

## Load Bitstream File

1. Click **Device** and click **DEV:0 MyDevice0 (XC7VX485T)** → **Configure** and click **Select New File** (Figure 1-14).

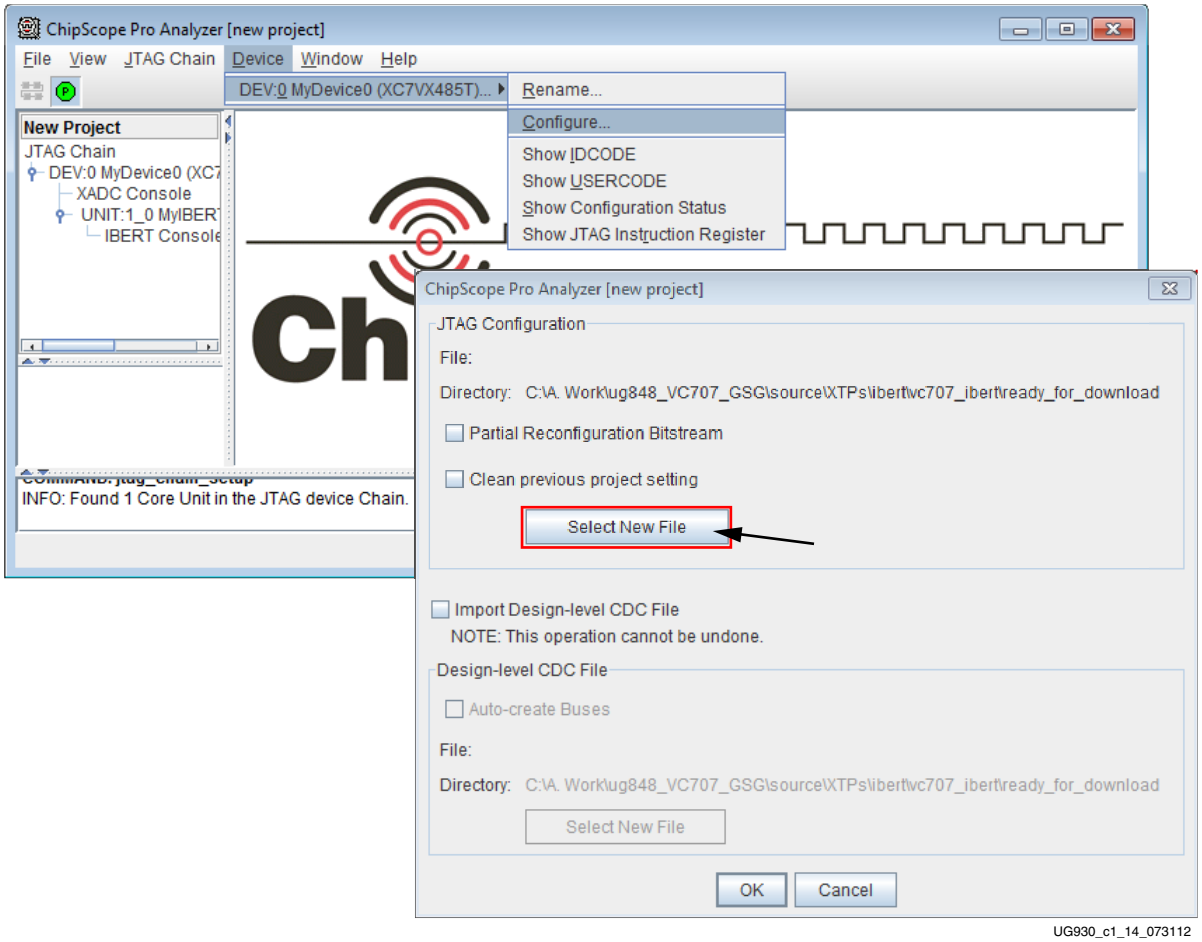



*Figure 1-14:* **Select New File**

2. Select `example_ibert_bank_113.bit`, click **Open**, and then click **OK** (Figure 1-15).

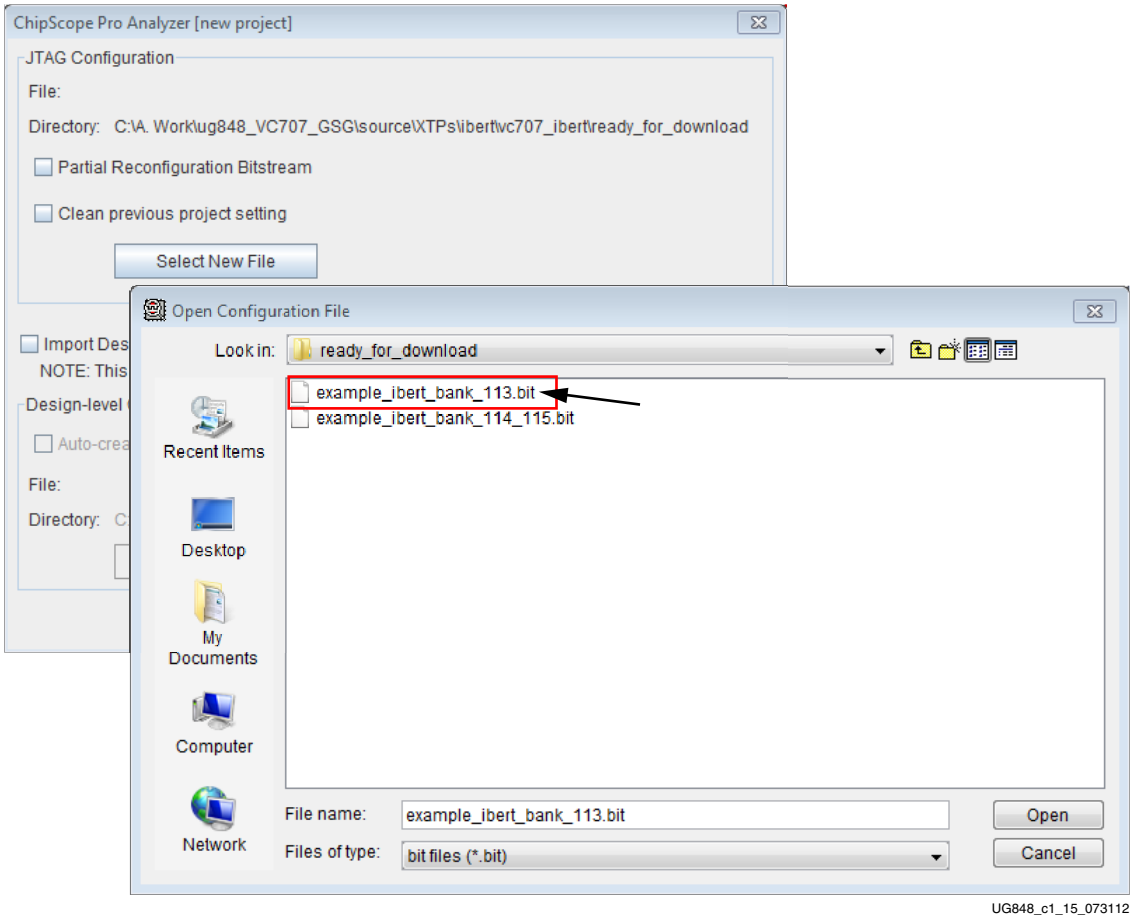



*Figure 1-15:* **Select Bitstream File**

## Examine GTX Transceiver Operation

1. In the project panel, double-click **IBERT Console** (Figure 1-16).


*Figure 1-16:* **Project Panel - IBERT Console (GTX)**

2. The status and test settings are displayed on the **MGT/IBERT Settings** tab in the IBERT Console shown in Figure 1-17. Click the BERT **Reset** button for each GTX transceiver and verify that there are no bit errors.

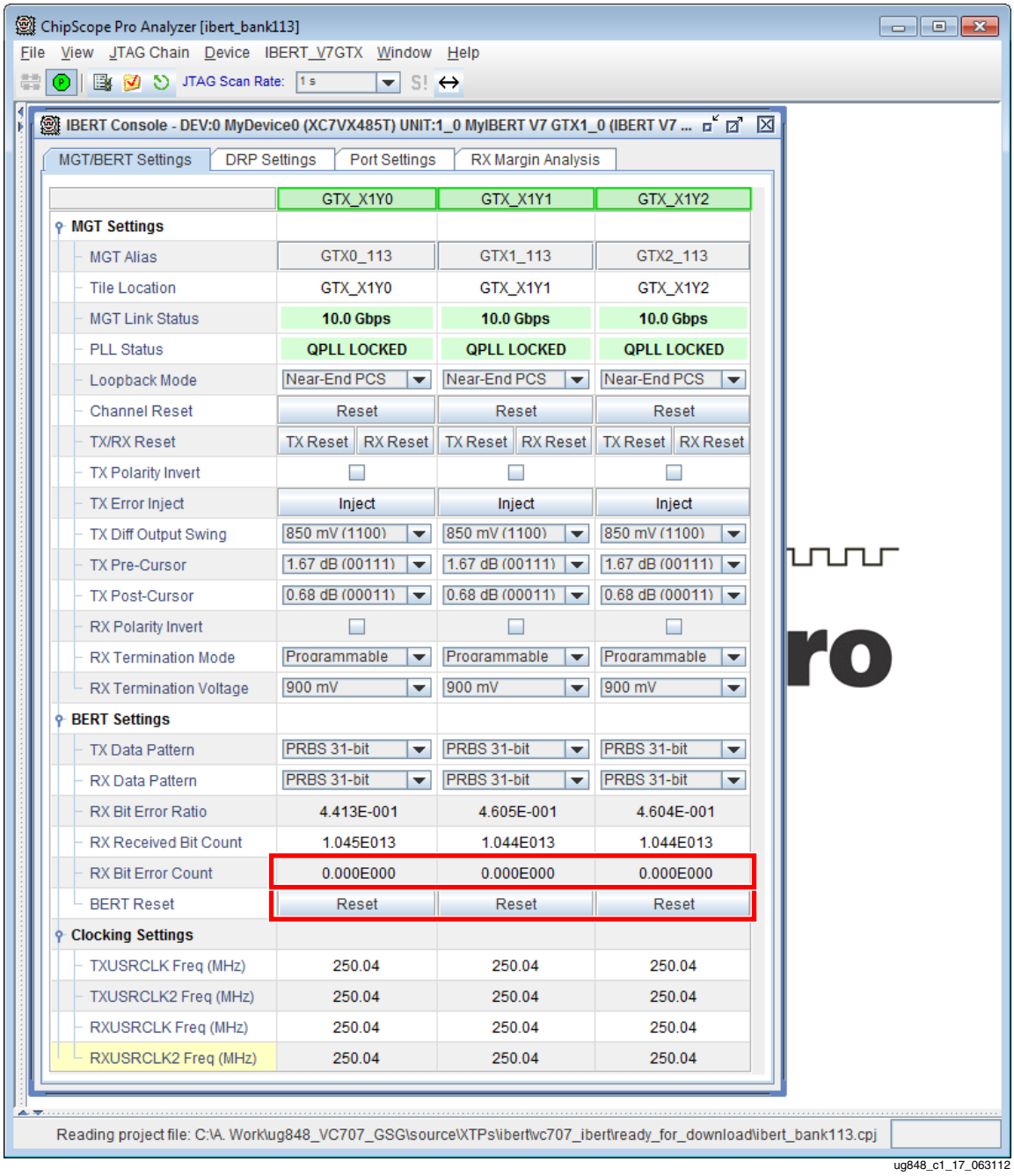



*Figure 1-17:* **GTX IBERT Console**

# AMS 101 Card Demonstration

The XC7VX485T FPGA features dual one Mega-sample per second (MS/s), 12-bit, analog-to-digital converters (XADC) built into the FPGA. The AMS card demonstration uses the AMS 101 card (Figure 1-19, page 19) to generate an analog signal and the AMS evaluator tool (Figure 1-20, page 20) to view and control the signal.

## Install the LabView Run-time Engine

Install the National Instruments LabVIEW Run-Time Engine 2011 Installer using either Method A, Method B (For 32-Bit Systems), or Method C (For 64-Bit Systems).

### Method A

1. Locate the file `AMS_Eval_Demo_Files_<ISE_Version>.zip` and unzip the `AMS101_Installer.zip` folder.
2. Install the LabVIEW Run-Time Engine. Open `AMS101_Installer\Volume` and double-click `setup.exe`.
3. Reboot the host computer to complete the installation.

### Method B (For 32-Bit Systems)

Download and install the National Instruments LabVIEW Run-Time Engine (32-Bit) on the host computer. The drivers are available for download at no cost from http://joule.ni.com/nidu/cds/view/p/id/2534/lang/en (LabVIEW 32-bit Run-Time Engine).

### Method C (For 64-Bit Systems)

Download and install the National Instruments LabVIEW Run-Time Engine (64-Bit) on the host computer. The drivers are available for download at no cost from http://joule.ni.com/nidu/cds/view/p/id/2536/lang/en (LabVIEW 64-bit Run-Time Engine).

***Note:*** The procedures for 32-bit and 64-bit host computers are identical except for the run-time engine that is downloaded.

## Extract the AMS Design Files

1. Go to http://www.xilinx.com/vc707 and navigate to the Docs & Designs tab. Download the zip file `AMS_Eval_Demo_Files_<ISE_Version>.zip`
2. Locate the file `AMS_Eval_Demo_Files_<ISE_Version>.zip` and unzip the files to a working directory on the host computer.
3. Open the ChipScope Pro Analyzer in the ISE design tools.
4. Click **Open_cable**.
5. Select **Device**, choose **Configure**, and click **Select New File**.
6. Open `AMS_VC707_Bitstream.bit` from the working directory.

## Set Up the Hardware

1. Complete the tasks under Preliminary Setup, page 6.
2. On the AMS101 card (Figure 1-19), place jumpers across pins 1–2 on J3 and J5.

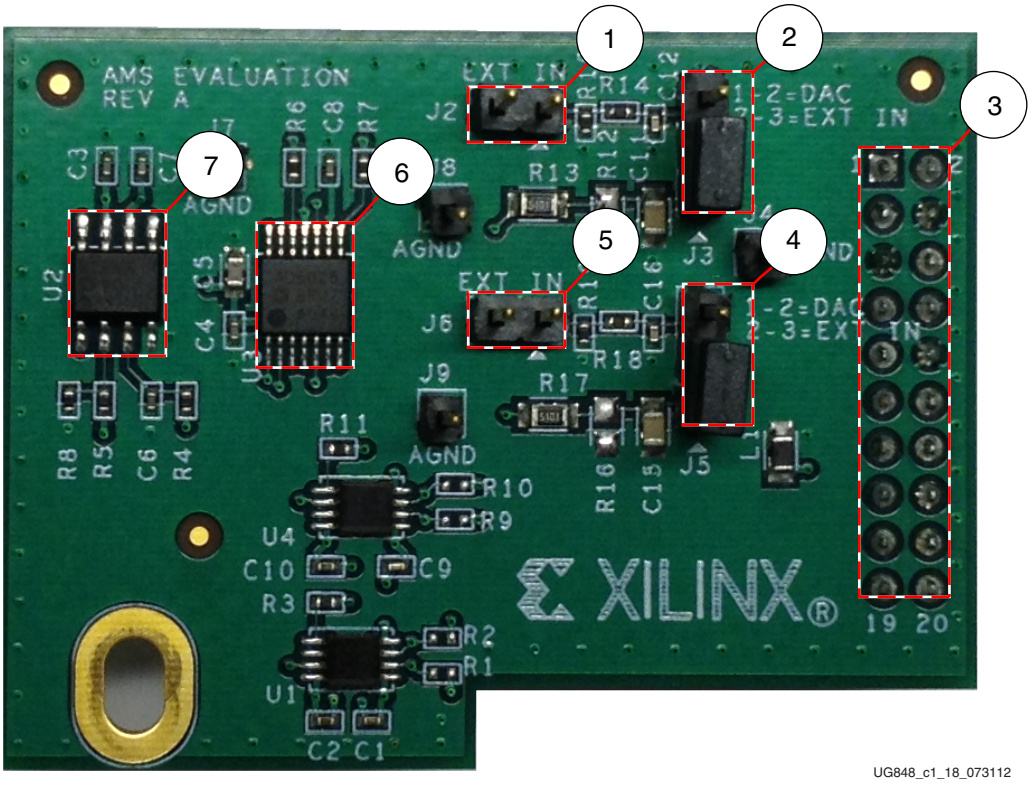



*Figure 1-18:* **AMS101 Evaluation Card**

*Table 1-2:* **AMS101 Evaluation Card Jumper and Component Notes**

| Callout | Reference Designator | Notes |
|---|---|---|
| 1 | J2 | External signal source input to $V_P$ positive analog input. |
| 2 | J3 | Jumper on pins 1–2 selects DAC signal source.<br>Jumper on pins 2–3 selects external input source on J2. |
| 3 | | 20-pin connector to XADC header J35 on the VC707 board. |
| 4 | J5 | Jumper on pins 1–2 selects DAC signal source.<br>Jumper on pins 2–3 selects external input source on J6. |
| 5 | J6 | External signal source to $V_N$ negative analog input. |
| 6 | U3 | 16-bit DAC. Sets analog test voltage. |
| 7 | U2 | Reference buffer for DAC. |

3. Plug the AMS 101 Card into the XADC header J35 on the VC707 board as shown in Figure 1-19.
4. Connect the VC707 board to the host computer and power supply as shown in Figure 1-19.

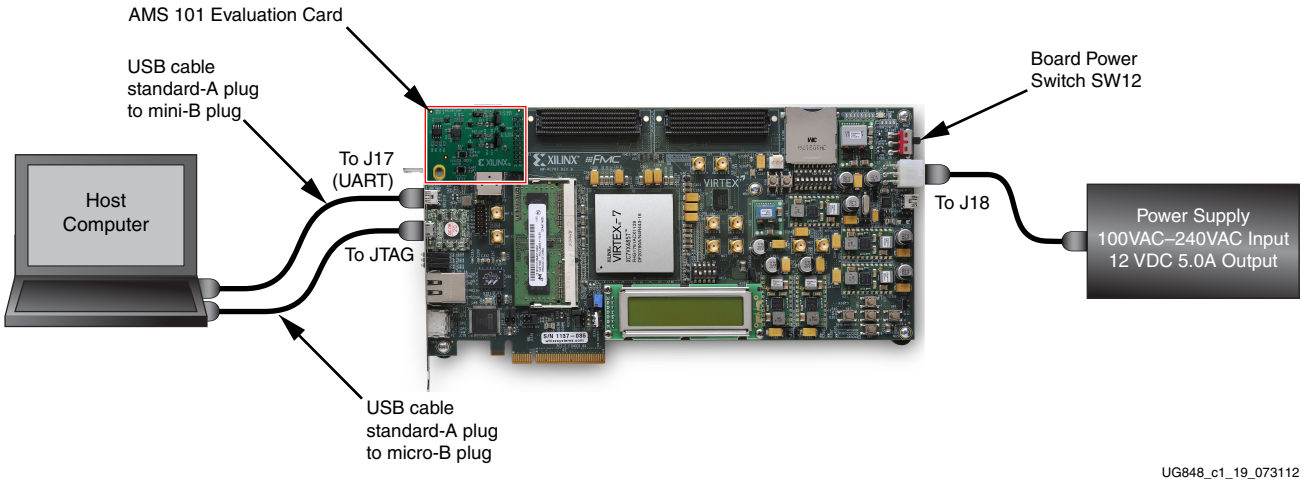



*Figure 1-19:* **BIST Board Connections**

5. Turn board power on (SW12).

## Examine Analog Mixed Signal Features

The AMS evaluator tool (Figure 1-20) is useful for examining analog signals in the time and frequency domains, displaying linearity, viewing the XADC register settings, and monitoring the internal FPGA temperature sensor and supply voltages. The AMS evaluator tool also provides user-controllable decimation on the XADC output data to enhance the signal-to-noise ratio (SNR) performance.

To run the AMS evaluator tool executable file:

1. Open the `AMS_Eval_Demo_Files_<ISE_Version>` directory and double-click `AMS101 Evaluator GUI V1.0.exe`. See Figure 1-20.

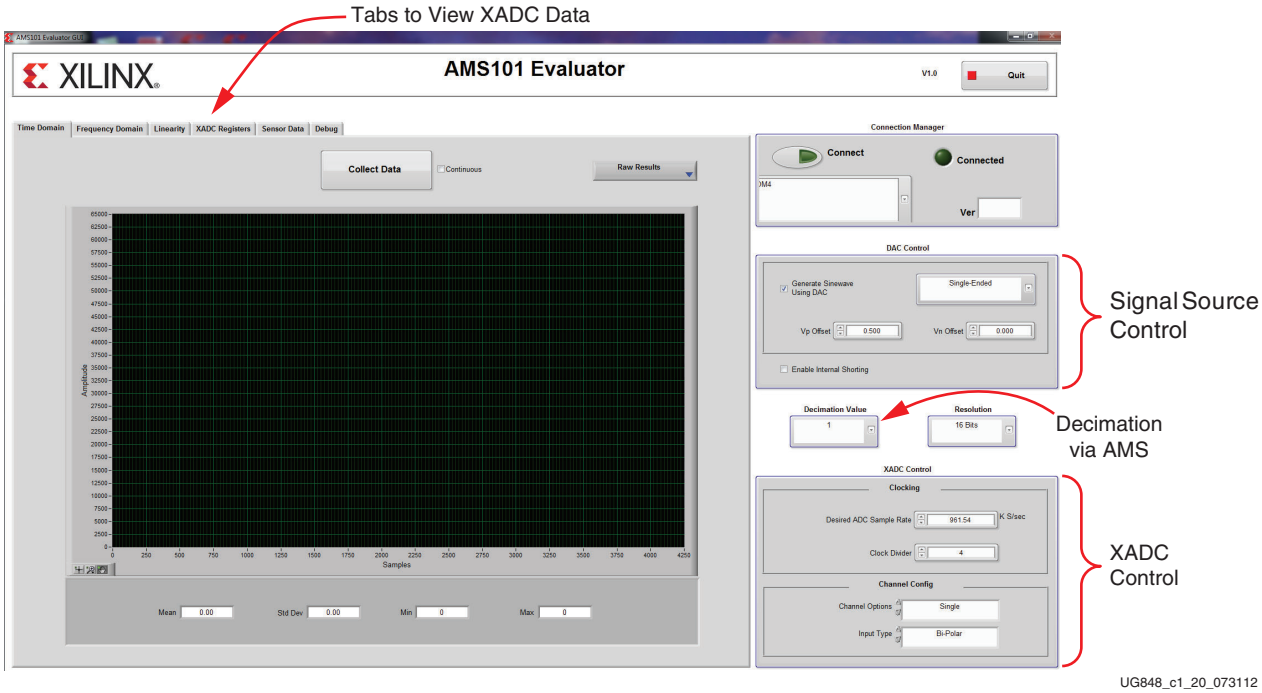



*Figure 1-20:* **AMS101 Evaluator Tool**

For an extensive explanation of the AMS101 evaluation card see UG886, *AMS101 Evaluation Card User Guide.*

# Multiboot Design

Files for the multiboot are located in `vc707_multiboot_rdf0159_<ISE revision>.zip` online as RDF0159 at: http://www.xilinx.com/vc707.

To run the multiboot reference design, set the configuration mode and flash address switch SW11 to `00001`, which is Master SPI configuration from the linear flash. Connect the USB-JTAG and power cables as shown in Figure 1-9, page 12. Set the user switches (SW2) to `00000000`. Unzip the design files to your C: drive.

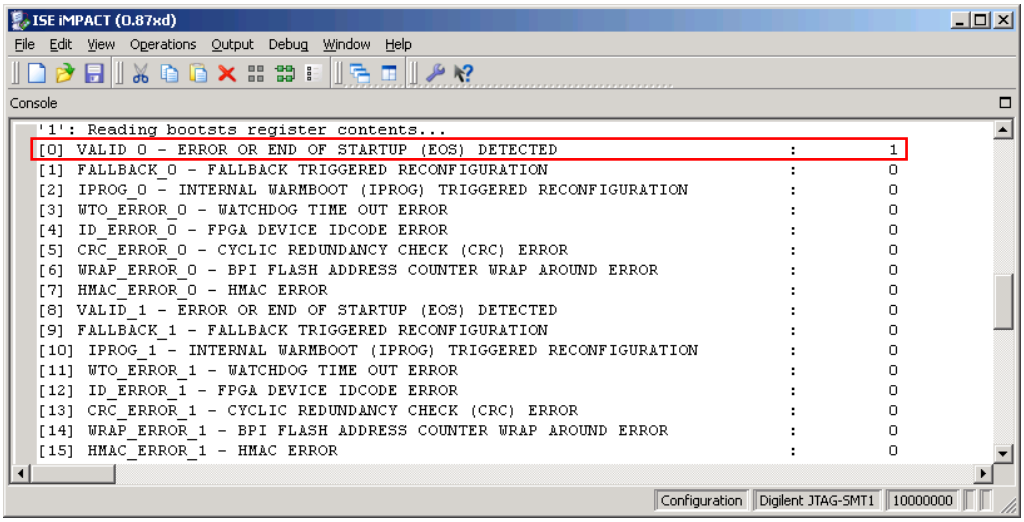
1. Program the design into the SPI flash using these command line instructions:

   ```
   cd C:\kc705_multiboot\SPI_Flash
   impact -batch kc705_program_spi.cmd
   ```

   A cycling LED pattern appears on the user LEDs.

2. Run the iMPACT tool and select options to configure the device over JTAG.

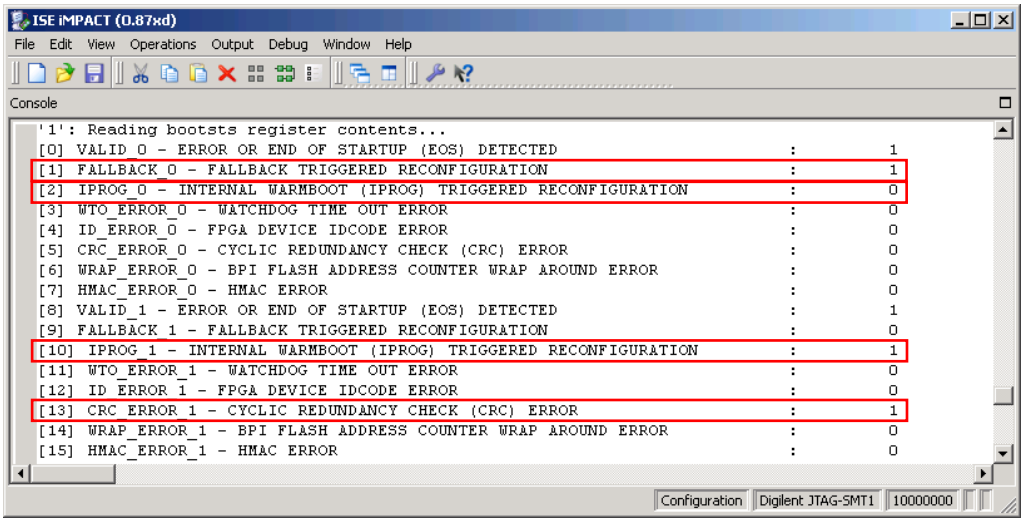
3. Read the device status and verify in the console display that only VALID_0 is set (Figure 1-21).



UG848_c1_21_073112

*Figure 1-21:* **Console Display**

4. Set SW2 to 10000000 to force the multiboot reboot.
5. When the DONE LED (DS10) goes out, set switch SW2 to 00000000. The LED pattern changes and the device status shows that both IPROG_0 and VALID_1 are set.
6. Press and release the Program button (SW9) to reboot back to the golden bitstream.
7. Repeat these steps, changing SW2 to 00010000 and verify that the device status IPROG_0 is no longer active, and that FALLBACK_0, IPROG_1, and CRC_ERROR_1 are active (Figure 1-22).



UG848_c1_22_073112

*Figure 1-22:* **Console Display**

# MIG Design

Files for the MIG design are located in `vc707_MIG_rdf0160_<ISE revision>.zip` online as RDF0160 at: http://www.xilinx.com/vc707.

To use the MIG, first unzip the design files to your C: drive:

1. Open the ChipScope Pro Analyzer.
2. Open the cable (the top-left button) and configure the device `DEV:0 MyDevice0 (XC7VX485T)` with `<Design path>\mig_7series_v1_3\example_design \par\example_top.bit`.
3. After configuration, verify this LED behavior:
   a. After bitstream loads, LED 0 (right most LED) will be lit, and LED1 will be blinking. If LED 1 is not blinking, check the external OSC.
   b. After about 30 seconds, LED 3 will light and stay on. Wait until LED 3 lights.
   c. If an error occurs, LED 0 will go out and LED 2 will light.
   d. Press SW8 to reset.
4. In ChipScope, open the project `<Design Path>\ready_for_download\vc707B_mig.cpj` and view **Waveform**.
5. Click **Trigger** to view the data waveforms and validate the design (Figure 1-23).

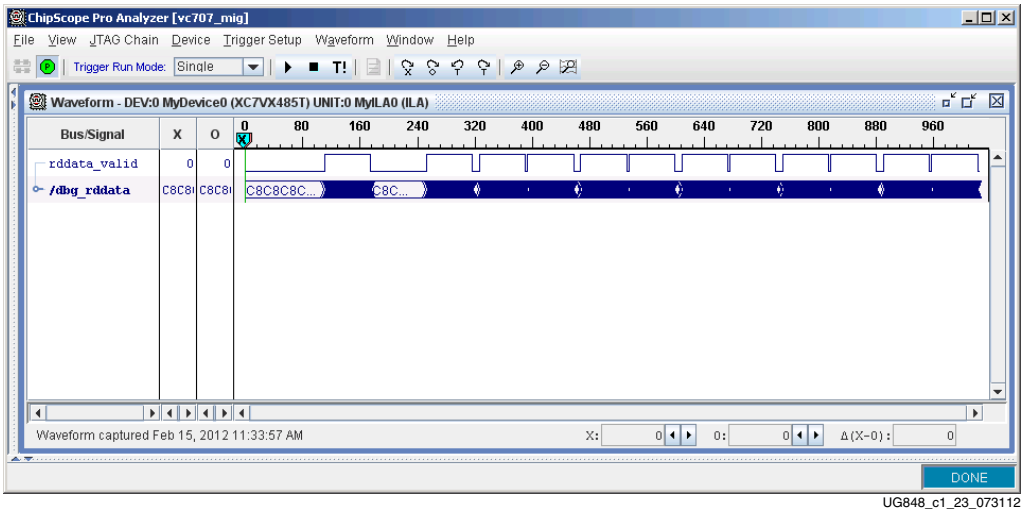



*Figure 1-23:* **ChipScope Display**

# Integrated Endpoint Block for PCI Express

Files for this design are located in `vc707_PCIe_rdf0161_<ISE revision>.zip` online as RDF0161 at: http://www.xilinx.com/vc707.

To run the PCIe reference design, set up the VC707 board as described in Preliminary Setup, page 6. Set the Configuration Mode and Flash address DIP switch SW11 to Master BPI mode `00010`. Unzip the files to your C: drive.

Start the iMPACT tool and program the Master BPI flash with the PCIe design:

1. Perform a boundary scan and initialize the chain.

2. Add BPI flash part number 28F00AG18F, data width 16, and address bits 25:24. Right-click on FLASH to select **Program**. Choose the file: `<design path>\ready_for_download\kc705_pcie_x8_gen1.mcs`.
3. Program with the **Erase before Programming** option.
4. Plug the VC707 board into a PCIe slot in a PC using the PC Power adapter (*not* the PCIe connector from the PC power supply).
5. Power on the PC and start up PciTree.
6. Set the number of configuration registers to 64, and Refresh Dump.
7. Locate the Xilinx device and navigate the linked list in configuration space to locate the PCIe® Capabilities Structure (Figure 1-24).

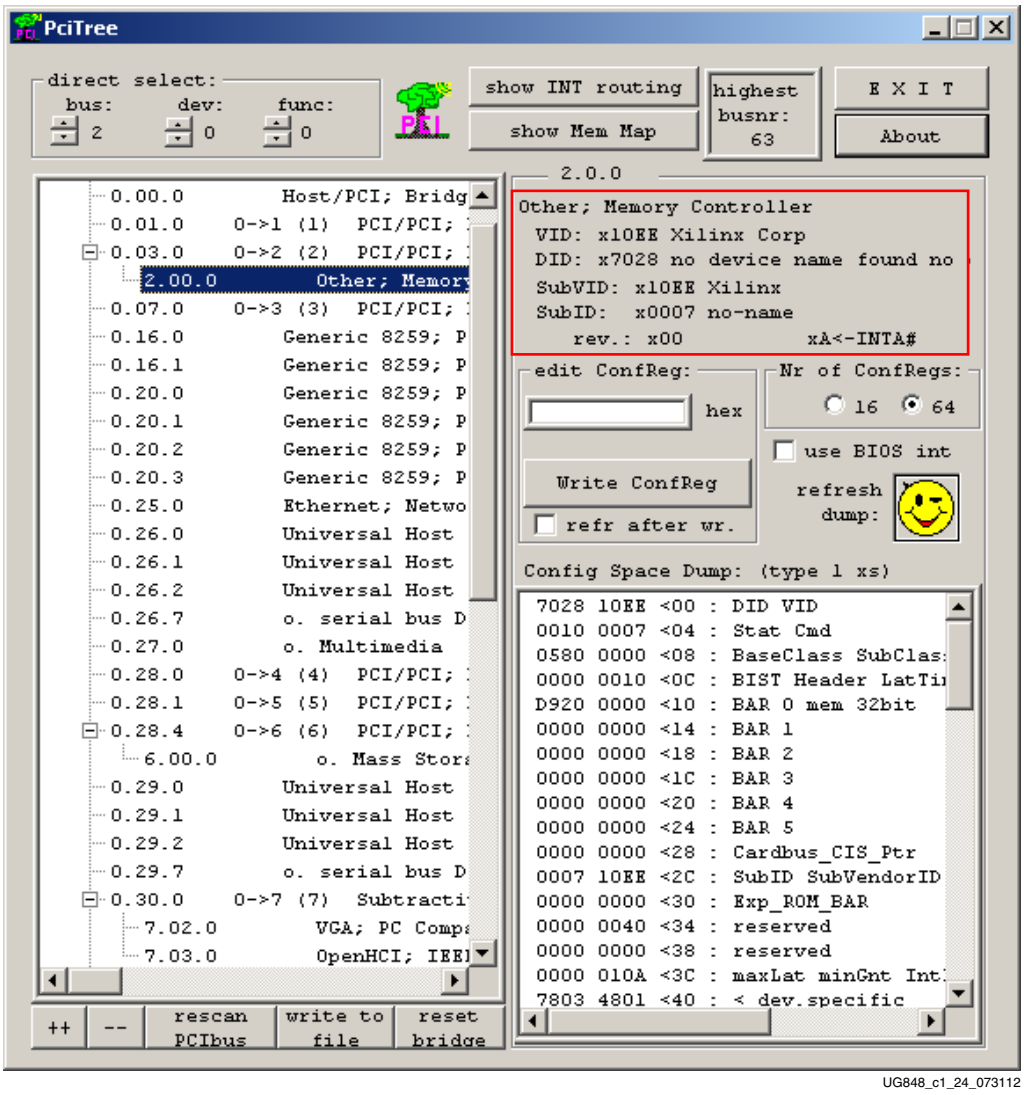



*Figure 1-24:* **PciTree Display**

# LogiCORE IP Ethernet SGMII Design

Files for this design are located in `vc707_ethernet_rdf0165_<ISE revision>.zip` online as RDF0165 at: http://www.xilinx.com/vc707.

The Ethernet 1000BASE-X PCS/PMA or SGMII core provides the functionality to implement the 1000BASE-X PCS and PMA sublayers or can provide a GMII to SGMII/SGMII to GMII bridge when used with a device-specific transceiver. The core interfaces to a device-specific transceiver. It provides some of the PCS layer functionality, such as 8B/10B encoding/decoding, the PMA SerDes, and clock recovery.

1. Install Wireshark Protocol Analyzer version 1.6.1 (http://www.wireshark.org/).
2. Set up VC707 board as in Figure 1-9, page 12. Add an Ethernet cable connected between the VC707 and host PC and power on the board. Unzip the design files to your C: drive.
3. Start ChipScope Pro Analyzer, and initialize the chain.
4. Configure the XC7VX485T device with `<Design Path>\ready_for_download\routed.bit`.
5. Set SW2 to `00000000`. This turns off the packet generator.
6. Open Wireshark and configure the capture options as follows:
   a. Select the PC's Gigabit Ethernet connection from the drop-down menu.
   b. Set the capture filter as in Figure 1-25.

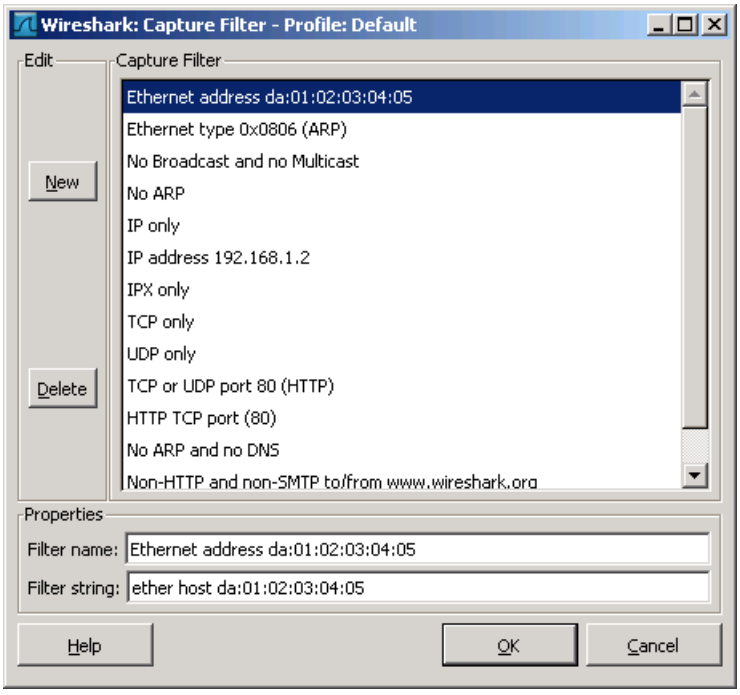



*Figure 1-25:* **Ethernet Core**

   c. Click **Start**.
7. Set SW2 to `01000000` for a moment, then back to `00000000`. This briefly turns on the packet generator.

8. Check the Wireshark window for a display of the packets transmitted. Figure 1-26.

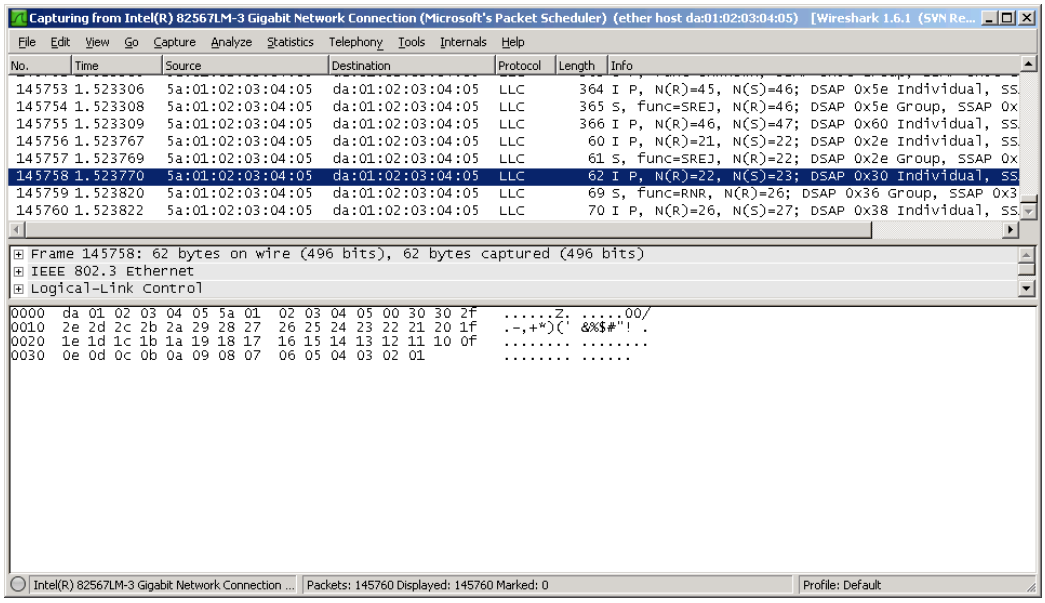



*Figure 1-26:* **Ethernet Core**

# Next Steps

1. Download UG885, *VC707 Evaluation Board for the Virtex-7 FPGA User Guide*.
2. Review and run the reference designs available at www.xilinx.com/vc707.

# Additional Information

VC707 board reference design files, user guides, schematics, and bill of materials, can be downloaded from:

http://www.xilinx.com/products/boards-and-kits/VC707

Other documents associated with Xilinx devices, design tools, intellectual property, boards, and kits are available at the Xilinx documentation website at:

http://www.xilinx.com/support/documentation/index.htm

Instructions for restoring the BPI flash memory with the factory-loaded reference designs are provided in XTP145, *VC707 Restoring Flash Contents*.

For an extensive explanation of the AMS101 evaluation card, see UG886, *AMS101 Evaluation Card User Guide.*

XILINX®

# Appendix A

# Warranty

THIS LIMITED WARRANTY applies solely to standard hardware development boards and standard hardware programming cables manufactured by or on behalf of Xilinx ("Development Systems"). Subject to the limitations herein, Xilinx warrants that Development Systems, when delivered by Xilinx or its authorized distributor, for ninety (90) days following the delivery date, will be free from defects in material and workmanship and will substantially conform to Xilinx publicly available specifications for such products in effect at the time of delivery. This limited warranty excludes:
(i) engineering samples or beta versions of Development Systems (which are provided "AS IS" without warranty); (ii) design defects or errors known as "errata";
(iii) Development Systems procured through unauthorized third parties; and
(iv) Development Systems that have been subject to misuse, mishandling, accident, alteration, neglect, unauthorized repair or installation. Furthermore, this limited warranty shall not apply to the use of covered products in an application or environment that is not within Xilinx specifications or in the event of any act, error, neglect or default of Customer. For any breach by Xilinx of this limited warranty, the exclusive remedy of Customer and the sole liability of Xilinx shall be, at the option of Xilinx, to replace or repair the affected products, or to refund to Customer the price of the affected products. The availability of replacement products is subject to product discontinuation policies at Xilinx. Customer may not return product without first obtaining a customer return material authorization (RMA) number from Xilinx.

THE WARRANTIES SET FORTH HEREIN ARE EXCLUSIVE. XILINX DISCLAIMS ALL OTHER WARRANTIES, WHETHER EXPRESS, IMPLIED OR STATUTORY, INCLUDING, WITHOUT LIMITATION, ANY WARRANTY OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT, AND ANY WARRANTY THAT MAY ARISE FROM COURSE OF DEALING, COURSE OF PERFORMANCE, OR USAGE OF TRADE. (2008.10)

Do not throw Xilinx products marked with the "crossed out wheeled bin" in the trash. Directive 2002/96/EC on waste electrical and electronic equipment (WEEE) requires the separate collection of WEEE. Your cooperation is essential in ensuring the proper management of WEEE and the protection of the environment and human health from potential effects arising from the presence of hazardous substances in WEEE. Return the marked products to Xilinx for proper disposal. Further information and instructions for free-of-charge return available at: http:\\www.xilinx.com\ehs\weee.htm.