This document provides a brief introduction and instructions to install and run the MachXO2-4000HC Control Development Kit on Windows 7/Vista/XP/2000 operating systems. Please refer to the complete documentation at www.latticesemi.com/mxo2-control-kit.

1 Check Kit Contents

The MachXO2-4000HC Control Development Kit includes the following items:

- MachXO2-4000HC Control Evaluation Board pre-loaded with Control SoC Demo
- USB cable
- AC adapter
- *QuickSTART Guide*

Detailed information about the evaluation board is provided in the *MachXO2-4000HC Control Development Kit User's Guide* at www.latticesemi.com/mxo2-control-kit.

Note: Static electricity can shorten the lifespan of electronic components. Please handle the kit components carefully.

2 Download Windows Hardware Drivers

For the Windows 7 OS, skip this step.

For pre-Windows 7 OS, you will need to obtain the necessary hardware drivers from the Lattice web site:

1. Browse to www.latticesemi.com/mxo2-control-kit and locate the hardware device drivers for the USB interface.
2. Download the zip file to your system and unzip it to a location on your PC.

For Linux, the USB interface drivers for the evaluation board are included in Linux kernel v.2.4.20 or later, including distributions compatible with Lattice Diamond® 2.0 or later (Red Hat Enterprise v.3, v.4 or Novell SUSE Enterprise v.10).

3 Assemble and Connect to the MachXO2-4000HC Control Evaluation Board

In this step, power the board and connect it to your PC using the USB cable provided.

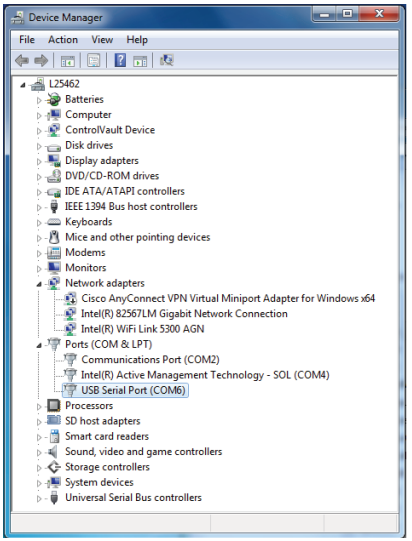
1. Install jumper J6, shown in the Layout Diagram on page 4. When the jumper connects 2-3, the JTAG signals are routed to the MachXO2 device. When the jumper connects 1-2, JTAG signals are routed to the ispPAC®-POWR1014A device. For now, connect 2-3 with the jumper.
2. Connect the AC adapter from a wall outlet to the power socket. After a connection is made, a red power LED (D12) will light indicating the board is powered on.
3. Connect the USB cable provided from a USB port on your PC to the board's USB interface socket on the bottom of the board as shown in the layout diagram on the last page of this document.

Note: Only perform steps 4-7 for pre-Windows 7 OS (Windows 7 installs the FTDI driver automatically).

4. When you are prompted "Windows may connect to Windows Update" select **No, not this time** from available options and click **Next** to proceed with the installation. Choose the **Install from specific location (Advanced)** option and click **Next**.
5. Select **Search for the best driver in these locations** and click the **Browse** button to navigate to the Windows driver folder created in Step 2 of this *QuickSTART Guide*. Select the **CDM 2.04.06 WHQL Certified** folder and click **OK**.
6. Click **Next**. A screen will display as Windows copies the required driver files. Windows will display a message indicating that the installation was successful.
7. Click **Finish** to install the USB driver.

4 Set Up a Terminal Program

You will use a terminal program to communicate with the evaluation board. The following instructions describe the Windows HyperTerminal program which is found on many Windows PCs. You may use another terminal program using the COM and port settings below. Tera Term is recommended for Windows 7. For Linux, Minicom is a good alternative.



1. From the Windows **Start** menu, select **Run** (for Windows 7, **Search programs and files**).
2. Type **mmc devmgmt.msc**. The “Device Manager” dialog appears.
3. Expand the **Ports (COM & LPT)** entry and note the COM port number for the “USB Serial Port”.
4. Start your desired terminal program. For Windows XP, HyperTerminal is pre-installed. From the **Start** menu, select **Programs > Accessories > Communications > HyperTerminal**.
5. Select the COM port identified in Step 3 from the “Connect using:” or “Port...” list.
6. Set the following port settings:

   Bits per second: 115200Data bits: 8
   Parity: None
   Stop bits: 1
   Flow control: None

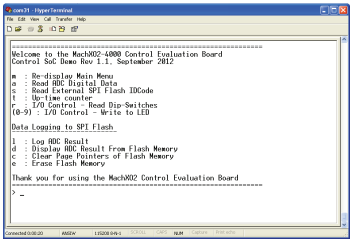
5 Run the Control SoC Demo

These instructions highlight the voltage monitoring and data logging function of the Control SoC Demo.

1. On the evaluation board, press the **S1** push-button (GSR). The Control SoC Demo main menu appears in HyperTerminal.
   Note: Press ‘m’ to redisplay the main menu if it scrolls off screen. Refer to the MachXO2-4000HC Control Development Kit User’s Guide for main menu command usage.
2. In the terminal window, press ‘**l**’ (lower case ‘L’ character). A data point from the voltage monitoring ADC will be saved in page 0 of the SPI Flash device on the board. Repeat logging a few more times.
3. In the terminal window, press ‘**d**’. The first logged data point will be read on page 0 of the SPI Flash device and be displayed on the HyperTerminal. You can display additionally logged data points until the message “No More Data” is displayed.
4. To display the logged data again, press ‘**c**’. This will clear the page address pointers and data will be read beginning with page 0 of the SPI Flash.
5. Press ‘**e**’ to erase the content of the SPI Flash device.
6. To vary the input voltage to the soft ADC core, connect a voltage source to pin 2 of the J9 header. The accuracy of the ADC is centered around 1.65V. The maximum voltage applied should not exceed 3.0V.

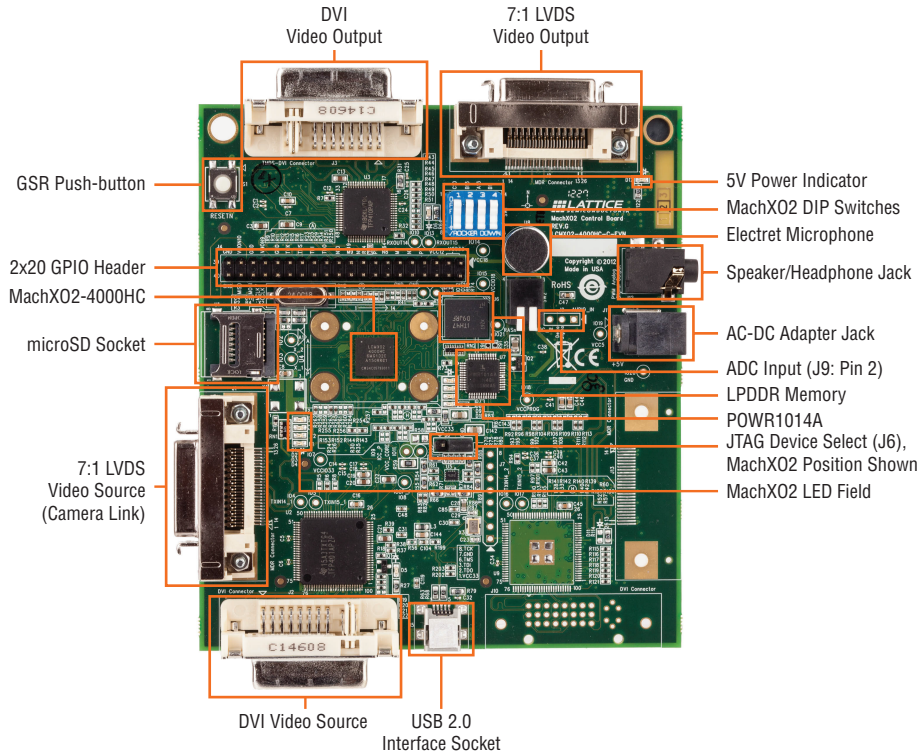
6 Done!

Congratulations! You have successfully connected and demonstrated the MachXO2-4000HC Control Development Kit. Please refer to the *MachXO2-4000HC Control Development Kit User's Guide* available on the Lattice web site at www.latticesemi.com/mxo2-control-kit for the following:



- Running advanced demos
- Details on additional evaluation board features and operation
- Programming the MachXO2-4000HC Control Evaluation Board with the USB cable
- Modifying and generating the demo bitstreams from the Diamond project source files
- Schematics

MachXO2-4000HC Control Evaluation Board Layout Diagram



Technical Support

1-800-LATTICE(528-8423)
+1-503-268-8001
techsupport@latticesemi.com

QS017
September 2012