Software Installation and Use

Installation

Quartus Prime Lite and ModelSim

You can run the software from AppsAnywhere or install it locally on your PC. Installing from AppsAnywhere is faster but requires that you be on-line to use the software. Avoid doing both – the AppsAnywhere version may conflict with the locally installed version.

To install the software locally, download the latest Quartus Prime Lite from the Intel FPGA Software Download Center. From the "Individual Files" tab download:

- Quartus Prime (includes Nios II EDS),
- Modelsim (or Questa)-Intel FPGA Edition (includes Starter Edition), and
- MAX II, MAX V device support

Quartus Prime Lite version 21.1.1 and ModelSim 20.1.1 are also available in the ELEX 2117 ShareOut folder. The BCIT ITS Knowledge Base has instructions on accessing ShareIn and ShareOut remotely.

Unfortunately, the Questa simulation software requires that you register on the Intel web site to obtain a (free) license. If you want to skip this licensing step install ModelSim (version 20.1.1) instead or run Modelsim from AppsAnywhere .

Download all three files to the same folder and run the Quartus installer. Note that the downloads total about 3 GBytes and installation requires about 15 GBytes of free disk space.

When you run the Quartus installer it should find the Modelsim (or Questa) and the device installer files in the same folder and offer to install them. Do so.

When the installation is complete the Quartus installer will show an option to install the drivers for the USB Blaster:
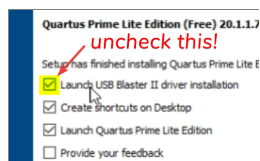

Uncheck this box! The drivers may "hang" or crash Windows if you use the ones in recent versions of Quartus with the "clone" USB Blaster in your parts kit.

Instead, download the `usb-drivers.zip` file available on the course website and extract the files. Run Device Manager and find the USB-Blaster device. Right-click on it and select "Update Driver":
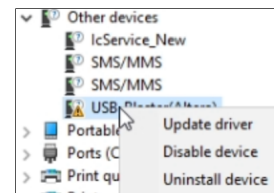
and select the folder where you extracted drivers from the course website. Finally, check that the 2009 driver version was installed:
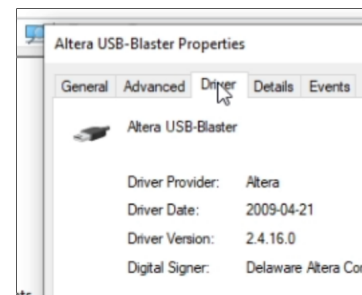

If you have problems when you run the programmer tool, see "Resolving USB-Blaster Problems" below.

There is a short video on the course website showing how to install the software.

High-Resolution Displays

The Quartus user interface may not display properly if you have a high-resolution display and you've set text scaling set to more than 100%. You can work around this by having the operating system scale the Quartus window itself. Right-click the Quartus shortcut and select **Properties** > **Compatibility** > **Change high DPI settings** > **High DPI scaling override**. Check the **Override** box and select **System (Enhanced)**:
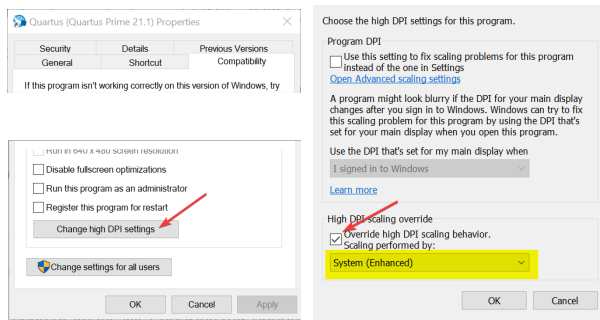
Logic Synthesis with Quartus

1. Run Quartus (on Windows select Intel FPGA ... > Quartus ...).

2. Select **File** > **New...** > **New Quartus Prime Project** > **OK**.

3. In the dialog boxes that follow: select a new folder for your lab (e.g. `C:\ELEX2117\lab1`), enter a project name (e.g. `lab1`), select an empty project, don't add any files, select the **MAX II** Family, select the specific device `EPM240T100C5`, and leave other settings at their defaults.

4. Add any existing design files using **Project** > **Add/Remove Files in Project**..., or create new ones using **File** > **New...** > **System Verilog HDL File**. One of the files must have a module with the top-level name specified above (in this example, `lab1`). The port names of this module will correspond to the names assigned to the CPLD pins.

5. After all the design files have been created and added to the project, select **Processing** > **Start Compilation**. Correct any errors and recompile as necessary.

6. Select **Assignments** > **Pin Planner** and select the correct pin in the **Location** drop-down box for each I/O pin. Note that you must compile the project before the pin names are visible in Pin Planner. Recompile the project (**Processing** > **Start Compilation**) for the assignments to take effect.

7. It's good practice to assign unused pins as inputs. Under **Assignments** > **Device...** > **Device and Pin Options...** > **Unused Pins** select "As input tri-stated with weak pull-up".

8. Connect the CPLD board's coaxial power connector to a USB port and press the power push-button so that the power LED lights up.

9. Connect the "USB Blaster" to the JTAG port on the CPLD/FPGA board and a free USB port. The POWER and ACT lights on the USB-Blaster should turn on.

10. Select **Tools** > **Programmer**, click on **Hardware Setup**..., select **USB-Blaster** from the drop-down and **Close**. **USB-Blaster** should appear next to **Hardware Setup**...

11. If necessary, click on **Add File**.., navigate to the location of the generated `.pof` file (typically in the `output_files` folder of the project folder) and select the `.pof` file.

12. Check that the **Program/Configure** checkboxes are checked and press **Start** to program the device. The progress bar should show 100%.

13. Test your design.

There is a short video on the course website showing how to use Quartus to synthesize a design and program a CPLD or FPGA.

Simulation with ModelSim or Questa

1. Run ModelSim (on Windows select Intel FPGA ... > ModelSim ...).

2. (a) If this is the first time you simulate this design, select **File** > **New** > **Project**..., select the folder where your files are located as the Project Location, enter a suitable Project Name (e.g. `lab1`) and click OK. Select Add Existing File and select the file(s) that contain the entities you want to simulate, including your design files and testbench (if any), then select Close.

   (b) If you had already created a simulation project and it's not already open, select **File** > **Open**, select Files of type: Project Files (*.mpf) and select the project file,

3. Select **Compile** > **Compile All** to compile all the files in your project into the `work` library,

4. If there are syntax errors you will need to fix the error(s), save the file and go back to step 3,

5. Otherwise select **Simulate** > **Start Simulation**; select your testbench module from the `work` library and select **OK**.

6. Drag the signals you wish to view from the 'Sim' or 'Object' windows to the 'Wave' window (use the **View** menu to open windows).

7. Select **Simulate** > **Run -All**; this will run the simulation until it's complete.

8. The Transcript window will contain output from the testbench.

9. The Wave window will show the waveforms (select the Wave window, click on '+' and **Wave** > **Zoom** > **Full**); you can use a screen capture utility (e.g. Windows Snip tool) to save the waveforms.

10. If the results are not as expected, correct the errors, run **Compile** > **Compile All**, **Simulate** > **Restart**..., click **OK** and **Simulate** > **Run -All**.

Resolving USB-Blaster Problems

If possible, check your USB-Blaster, cables and CPLD or FPGA board at a BCIT lab session by programming one of the `.pof` files from the course web site. Ask the lab instructor for help if necessary.

If your hardware works with the lab PC but not with your own computer, follow the troubleshooting guide below.

You will need:

- a Windows 10 PC with Quartus Prime installed
- two free USB ports (or one port and one USB charger)
- a USB flash drive
- (in rare cases) a known-working USB-Blaster, cables, and CPLD or FPGA board

Do the following, *in order*:

1. Press the ⊞ (Start) key and type "Core Isolation" or navigate to ⚙ (Settings) > **Update & Security** > **Windows Security** > **Device Security** and click on Core isolation details.

   Check that **Memory Integrity** is set to **Off**:

Memory integrity

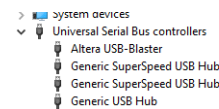Prevents attacks from inserting malicious code into high-security processes.

⊙ Off

The USB-Blaster `usblstr.sys` drivers are incompatible with Memory integrity protection and will not install if it's enabled.

2. Unplug any USB peripherals that are not required to run your computer, *including any Analog Discovery 2*, and restart your computer.

3. Check that the flash drive works in the USB port you plan to use for the USB-Blaster.[1] If other devices don't work when plugged into the ports you plan to use, shut down the computer (don't just restart it). If other USB devices still don't work, use a different USB port or computer.

4. Disconnect the USB-Blaster from the CPLD/FPGA board and plug it into the USB port. Check the green POWER LED. If it's not on, try a known-good mini-USB cable and USB-Blaster.

5. Run Windows' Device Manager and expand the USB controllers section. You should see the USB-Blaster under **Other devices**:
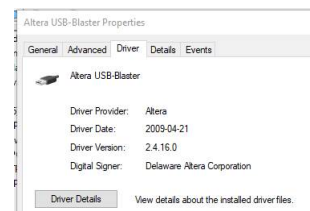
> Network adapters
> ⌄ Other devices
>   USB-Blaster(Altera)
>   Ports (COM & LPT)

or **USB Controllers**:

> System devices
> ⌄ Universal Serial Bus controllers
>   Altera USB-Blaster
>   Generic SuperSpeed USB Hub
>   Generic SuperSpeed USB Hub
>   Generic USB Hub

If not, (this is rare) try a known-good mini-USB cable and USB-Blaster.

6. Right-click on the USB-Blaster device, select Properties and click on the **Driver** tab. Check that Driver version 2.4.16.0 is installed as shown below.

Altera USB-Blaster Properties

General | Advanced | Driver | Details | Events

Altera USB-Blaster

Driver Provider: Altera
Driver Date: 2009-04-21
Driver Version: 2.4.16.0
Digital Signer: Delaware Altera Corporation

Driver Details — View details about the installed driver files.

---

[1]USB port power is turned off when you draw too much current (e.g. due to a short circuit on your breadboard).

if you have a different version then you're probably using the drivers included with Quartus rather than the ones from the course web site. Right-click on Altera USB Blaster and select **Uninstall device**. *Check the box for Delete the driver software for this device.*[2] and click **Uninstall**.
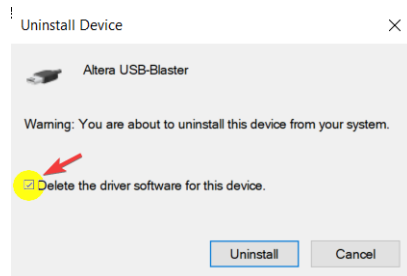
If no, or the wrong driver, was installed, install the Altera USB Blaster drivers from the course web site as described above and check the version again. Do not proceed until the correct driver version has been installed.

7. In Quartus, run **Tools** > **Programmer** and click on **Hardware Setup**. Check that you can select the USB-Blaster:
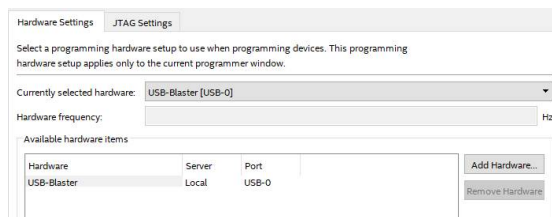
If not, close the programmer application, unplug any USB devices that may be using an FTDI serial interface IC, *including the Analog Discovery 2*, and try again again. If the Quartus programmer shows in Device manager but the Quartus programmer still cannot detect the USB-Blaster, go to step 1 (and make sure all USB devices are unplugged before restarting).

8. At this point the USB-Blaster should still be disconnected from the CPLD/FPGA board. Remove all connections to the CPLD/FPGA pins (to prevent issues due to short-circuited pins) and connect the micro-USB power connector to a USB power supply or USB port. If the

red power LED does not light, check the USB port/charger and cable. Replace the micro-USB cable or CPLD/FPGA board if the power LED does not light.

Note: The USB-Blaster supplies enough power to light the red power LED on the CPLD/FPGA board but not enough for the CPLD/FPGA to operate. Do not rely on the power LED to determine whether the CPLD/FPGA is receiving power.

9. Connect the USB-Blaster to the CPLD/FPGA board's JTAG connector using the ribbon cable. Note that the connectors are keyed.

10. Click on **Auto Detect** to check that the device is detected:
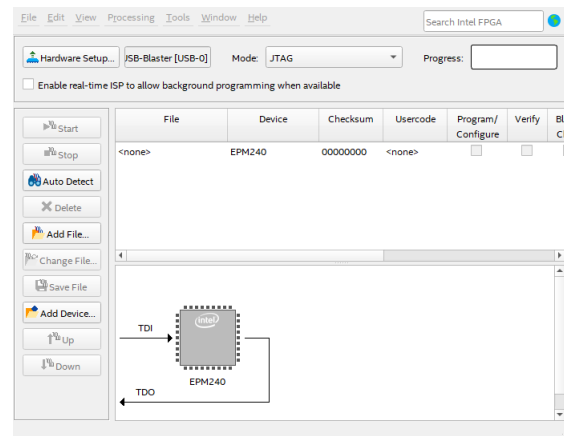
11. If the device is not detected you should see a pop-up message saying "Unable to scan device chain." Run **Tools > JTAG Chain Debugger** or Click **Yes** if the message offers to run it for you.

12. Click on **Test JTAG Chain**.

If the error is "Chain is in use." reboot the computer to restart the USB-Blaster driver.

If the error is "No device detected." double-check the connection between the USB-Blaster and the CPLD/FPGA board. Replace the USB-Blaster, ribbon cable, or CPLD/FPGA board[3].

---

[2]You must delete the newer driver so it does not get re-installed.

[3]Although the most common reason for this error is that there is no power connected to the CPLD/FPGA board.