

## Software Installation and Use

Revised January 31, 2021.

### Installation

#### Quartus Prime Lite and ModelSim

Download the latest “Lite” edition of Quartus Prime from: <https://fpgasoftware.intel.com/>. From the “Individual Files” tab download:

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

These three files are also available in the ELEX 2117 ShareOut folder. The BCIT ITS Knowledge Base has instructions on [accessing ShareIn and ShareOut remotely](#).

Download all three files to the same folder and run the Quartus installer. The Quartus installer will find the Modelsim and MAX device installer files and offer to install them. Do so.

Note that the downloads total about 3 GBytes and installation requires about 15 GBytes of free disk space.

If you’ve already installed Quartus version 13.0sp1 and ModelSim you may use them for this course. However, you’ll need to make minor adjustments to the instructions – they were written for version 20.1.

#### USB Blaster Drivers

The Quartus installer will also offer to install the drivers for the USB Blaster. Decline – Windows will crash if you use the drivers in recent versions of Quartus with the “clone” USB Blaster in your parts kit. Instead, install the drivers available on the course website.

If you have problems, see “Resolving USB-Blaster Problems” below.

### 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’s 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. Connect the CPLD board’s coaxial power connector to a USB port and use the pushbutton on the board to turn on the power (the power LED should go on).
8. Connect the “USB Blaster” to the CPLD and a free USB port. The **POWER** and **ACT** lights on the USB-Blaster should turn on.

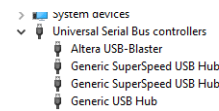
9. Select **Tools > Programmer**, click on **Hardware Setup...**, select **USB-Blaster** from the drop-down and **Close**. **USB-Blaster** should appear next to **Hardware Setup...**
  10. If necessary, click on **Add File...**, navigate to the location of the generated **.pof** file (typically in the **output\_files** folder of the project folder) and select the **.pof** file.
  11. Check that the **Program/Configure** checkboxes are checked and press **Start** to program the device. The progress bar should show 100%.
  12. Test your design.
9. The Transcript window will contain output from the testbench.
  10. 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.
  11. 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

### Simulation with ModelSim

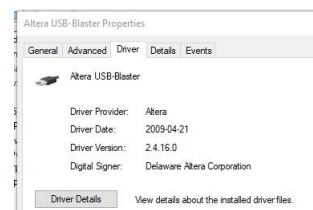
1. Run ModelSim (on Windows select Intel FPGA ... > ModelSim ...).
2. If this is the first time you simulate these files select **File > New > Project...**, select the folder where your files are located as the Project Location, enter a suitable Project Name (e.g. **lab8**) and click OK. Select Add Existing File and select the file(s) that contain the entities you want to simulate, including your design and testbench, then select Close.
3. 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,
4. Select **Compile > Compile All** to compile all the files in your project into the **work** library,
5. If there are syntax errors you will need to fix the error(s), save the file and go back to step 4,
6. Otherwise select **Simulate > Start Simulation**; select your testbench module from the **work** library and select **OK**.
7. Drag the signals you wish to view from the 'Sim' or 'Object' windows to the 'Wave' window (use the **View** menu to open windows).
8. Select **Simulate > Run -All**; this will run the simulation until it's complete.

- Check that other devices (e.g. a flash drive) work in the USB port you plan to use for the USB-Blaster. USB ports will shut down if you draw too much current (e.g. due to a short circuit on your breadboard). In this case you may need to power-cycle your computer.
- Disconnect the USB-Blaster from the CPLD and plug it into the USB port. Check the green POWER LED. If it's not on, check the USB cable and USB connector.
- Run Windows' Device Manager and expand the USB controllers section. You should see Altera USB-Blaster listed:



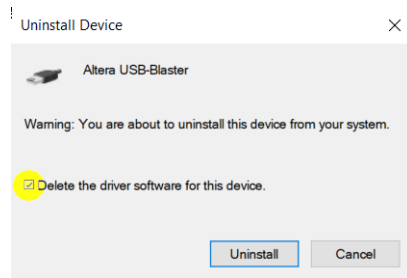
If not, it may have failed (this is rare). Try another one.

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



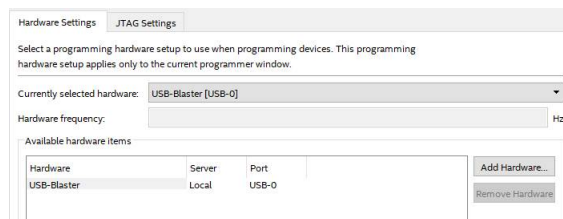
if you have a different version then you're probably using the drivers installed by Quartus rather

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.<sup>1</sup> and click **Uninstall**.



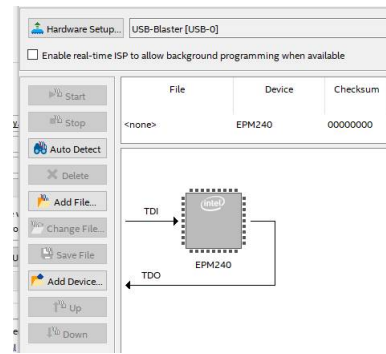
Install the Altera USB Blaster drivers from the course web site and check the version again.

- 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.

- Remove all connections to the CPLD pins (to prevent issues due to short-circuited pins) and connect the coaxial power connector to a USB power supply or port. If the red power LED next to the blue power switch is not lit, press the switch. If the power LED still does not light, check the USB port, power supply or power cable.
- Connect the USB-Blaster to CPLD board JTAG connector using the ribbon cable. Note that the connectors are keyed.
- Click on **Auto Detect** to check that the EPM240 device is detected:



- 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.
- 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 board by disconnecting the coaxial power connector. If the power LED on the CPLD board remains lit then the CPLD board or the CPLD itself may have failed, try another CPLD board (this is also rare).

<sup>1</sup>You must delete the newer driver so it does not get re-installed.