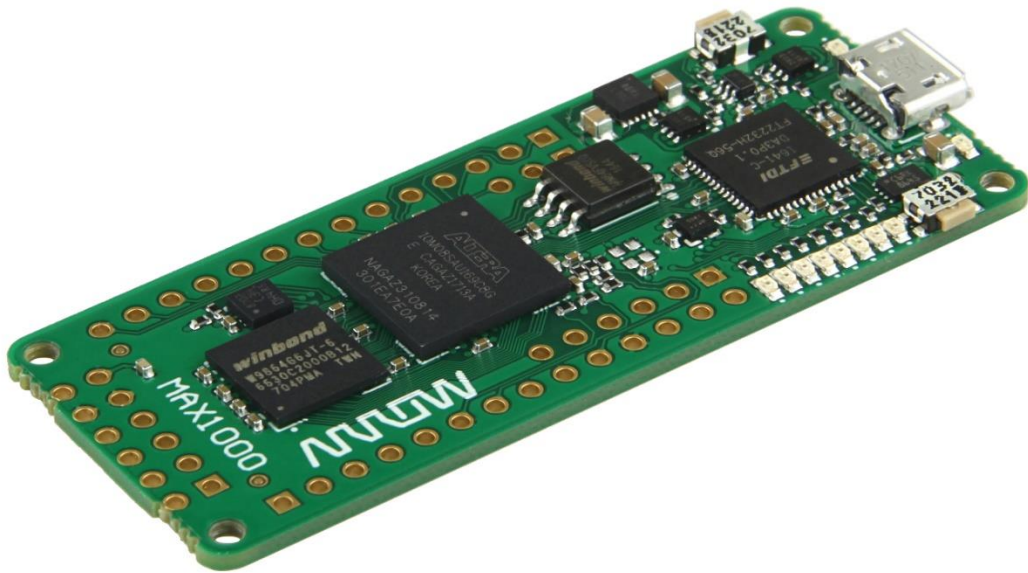


MAX1000

User Guide



Please read the legal disclaimer at the end of this document.

Revision 1.0



Table of Contents

Chapter 1 - MAX1000 IoT / Maker Board	5
1.1 About Arrow MAX1000 Board	5
1.2 Useful Links	6
1.3 Getting Help	6
Chapter 2 - Introduction to the MAX1000 Board	7
2.1 Layout and Components	7
2.2 Block Diagram	8
Chapter 3 - Connections and Peripherals of the MAX1000 Board	10
3.1 Board Status Elements	10
3.2 Clock Circuitry	10
3.3 Peripherals Connected to the FPGA	11
3.3.1 LEDs	11
3.3.2 Push Buttons	12
3.3.3 Accelerometer	13
3.3.4 SDRAM Memory	14
3.3.5 Flash Memory	15
3.3.6 User I/O	16
3.3.7 Arduino MKR Connectors	16
3.3.8 PMOD Connector	18
3.3.9 Communication and Configuration	19
3.3.10 Power Tree	21
Chapter 4 - Software and Driver Installation	22
4.1 Installing Quartus Prime Software (Windows)	22
4.2 Installing Arrow USB Programmer2 (Windows)	24
4.3 Installing Quartus Prime Software (Linux)	26
4.4 Installing Arrow USB Programmer2 (Linux)	28
Chapter 5 - New Project with MAX1000	30
5.1 Creating a new Blinky Project with MAX1000	30
5.2 Building a Blinky Project with MAX1000	35
5.2.1 Block Diagram	35
5.2.2 Components of the Design	35
5.2.3 Catalog IP	36
5.2.4 Create and Configure the PLL	37



5.2.5	Create and Configure the Counter	41
5.2.6	Create and Configure the Multiplexer	44
5.2.7	Adding the Components to the Schematic	47
5.2.8	Connecting the Components	49
5.2.9	Analysis and Synthesis	55
5.2.10	Adding Timing Constraints	55
5.2.11	Pinning Assignments	58
5.2.12	Compiling the Design	61
5.2.13	Reading the Compilation Report.....	63
Chapter 6 - Configuring the MAX1000		64
6.1	Configure the FPGA in JTAG Mode	64
6.2	Internal Configuration	69
Chapter 7 - Common Issues and Fixes.....		71
Chapter 8 - Appendix		72
7.1	Revision History	72
7.2	Legal Disclaimer	73



Table of Figures

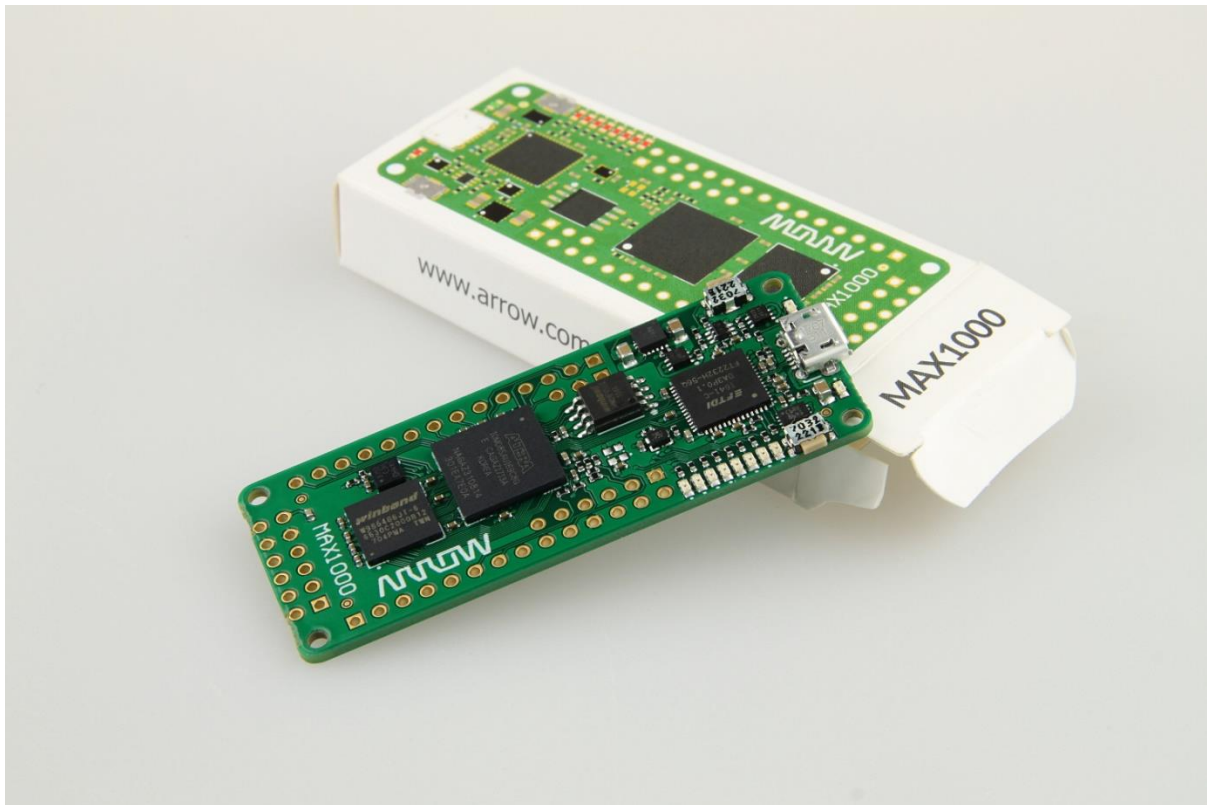
Figure 1 - MAX1000 Board (top view).....	7
Figure 2 - MAX1000 Block Diagram	8
Figure 3 – Position of Indication LEDs.....	10
Figure 4 – MAX1000 Clock Tree	10
Figure 5 – LED Connections.....	11
Figure 6 – Button Debouncing	12
Figure 7 – Button Connections.....	12
Figure 8 – Accelerometer Connections.....	13
Figure 9 – SDRAM Connections	14
Figure 10 – Flash Connections	15
Figure 11 - User I/O Connections.....	16
Figure 12 - Arduino MKR Header Connections	16
Figure 13 – PMOD Header Connections	18
Figure 14 – UART Connections.....	19
Figure 15 – JTAG Connections.....	20
Figure 16 – Power Tree Connections	21



Chapter 1 - MAX1000 IoT / Maker Board

1.1 About Arrow MAX1000 Board

The MAX1000 is a customizable IoT / Maker Board ready for evaluation, development and/or use in a product. It is built around the Intel MAX10 FPGA, which is the industry's first single chip, non-volatile programmable logic device (PLDs) to integrate the optimal set of system components. Users can now leverage the power of tremendous re-configurability paired with a high-performance, low-power FPGA system. Providing internally stored dual images with self-configuration, comprehensive design protection features, integrated ADCs and hardware to implement the Nios II 32-bit microcontroller IP, MAX10 devices are ideal solution for system management, protocol bridging, communication control planes, industrial, automotive and consumer applications. The MAX1000 is equipped with an Arrow USB Programmer2, SDRAM, flash memory, accelerometer sensor and PMOD/ARDUINO MKR connectors making it a fully featured plug and play solution without any additional costs.



The MAX1000 board contains all the tools needed to use the board in conjunction with a computer that runs a 64-bit Linux / Microsoft Windows 7 operating system or later.



1.2 Useful Links

A set of useful links that can be used to get relevant information about the MAX1000 or the MAX10 FPGA.

- [MAX1000 at Arrow Shop](#)
- [MAX1000 at Trenz Electronic Shop](#)
- [Intel MAX10 Webpage](#)
- [MAX1000 Wiki Page](#)

1.3 Getting Help

Here are the addresses where you can get help if you encounter any problems:

- **Arrow Electronics**

- In Person

- Arrow EMEA

- + 49 (0) 6102 5030 0

- Online

- <https://arrow.com>

- **Trenz Electronic GmbH**

- <https://www.trenz-electronic.de/en/>

Chapter 2 - Introduction to the MAX1000 Board

2.1 Layout and Components

Figure 1 shows a top view of the board. It depicts the layout of the board and indicates the location of the various connectors and key components.

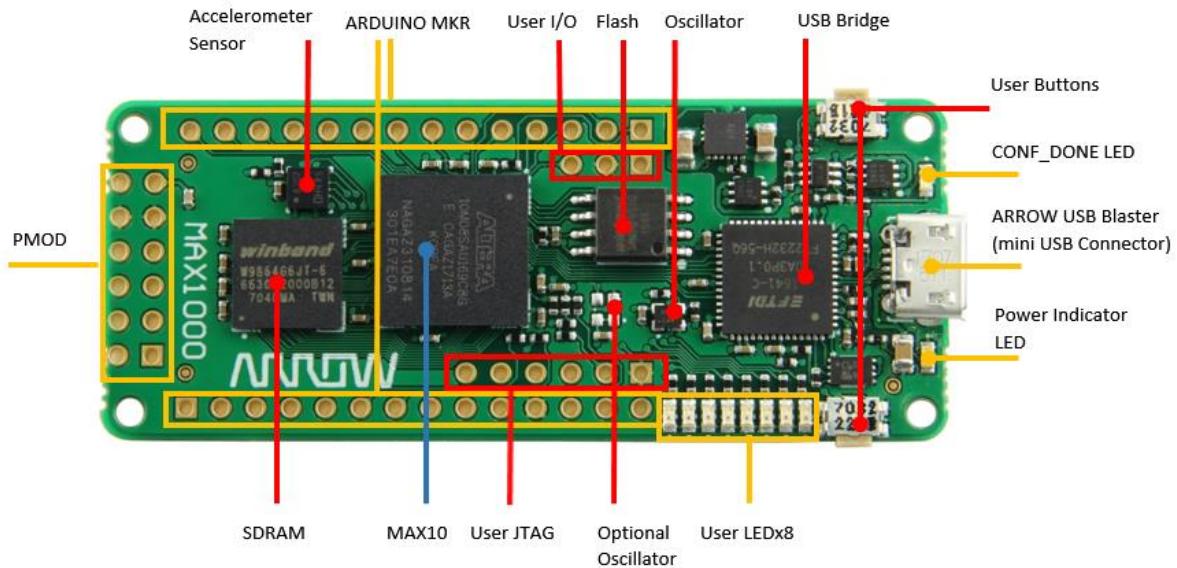


Figure 1 - MAX1000 Board (top view)

The following are available on the MAX1000 board:

- Intel MAX®10 10M08SAU169C8G device
- Arrow USB Programmer2 on-board for programming; JTAG Mode
- 64MBit SDRAM (16-bit data bus)
- 64Mbit Flash Memory
- One 12MHz MEMS Oscillator
- One optional MEMS Oscillator of preferred frequency
- Eight red user LEDs
- Two board indication LEDs
- Two user buttons
- One 3-axis accelerometer
- One 12-pin PMOD header
- One Arduino MKR header
- One User JTAG header
- One User I/O header

2.2 Block Diagram

Figure 2 represents the block diagram of the board. All the connections are established through the MAX 10 FPGA device to provide maximum flexibility for users. Users can configure the FPGA to implement any system design.

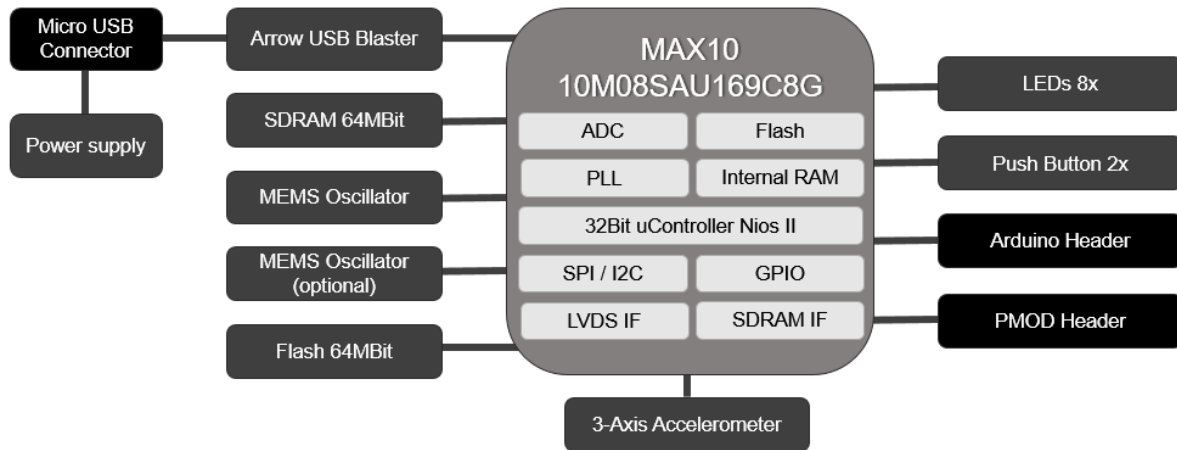


Figure 2 - MAX1000 Block Diagram

FPGA Device

- Available MAX10 Devices for the MAX1000

Resources	Device			
	10M02	10M04	10M08	10M16
Logic Elements (LE) (K)	2	4	8	16
M9K Memory (K)	108	189	378	549
User Flash (K)	96	1248	1376	2368
Internal Configuration Image	1	2	2	2
ADC	-	1	1	1
PLLs	2	2	2	4

Configuration and Debug

- On-board Arrow USB Programmer2 (mini-USB type B connector)

Memory Devices

- 64MBit to 128 MBit external flash memory
- 64MBit to 256 MBit external SDRAM memory



Headers

- PMOD Header
- Arduino MKR Header
- User JTAG Header
- User I/O Header

Buttons and Indicators

- 2 x side-buttons
- 8 x red user LEDs
- 2 x board indication LEDs

Sensors

- One 3-axis accelerometer

Power

- Recommended external supply voltage range: +5.0 V (nominal)
- Recommended I/O signal voltage range: 0/0 to +3.3 V

Chapter 3 - Connections and Peripherals of the MAX1000 Board

3.1 Board Status Elements

In addition to the 8 LEDs that the FPGA can control, there are 2 additional LEDs which can indicate the status of the board.

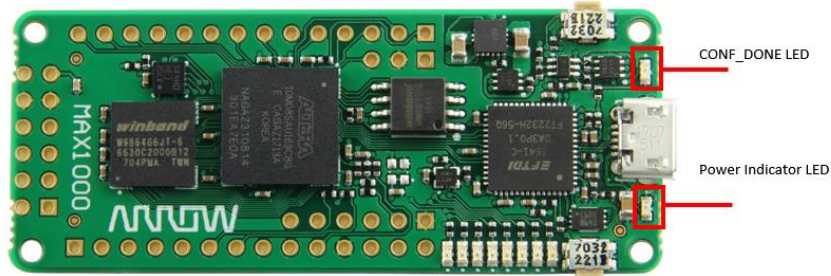


Figure 3 – Position of Indication LEDs

Board Reference	LED Name	Description
D1	3.3V	On when 3.3V power is active
D10	CONF_DONE	On when configuration data was loaded to MAX10 device without error

3.2 Clock Circuitry

All the external clocks of the system can be seen in Figure 4. The default clock (CLK12M) is at 12 MHz and is connected and driving the FPGA’s user logic and the Arrow USB Programmer2. There is an optional slot of another clock (CLK_X) to add another preferred clock source to the FPGA. Both clocks are driving PLL1/PLL3, which are able to drive the ADC clock.

For more information on clocks and PLLs of the MAX10, please refer to this [document](#).

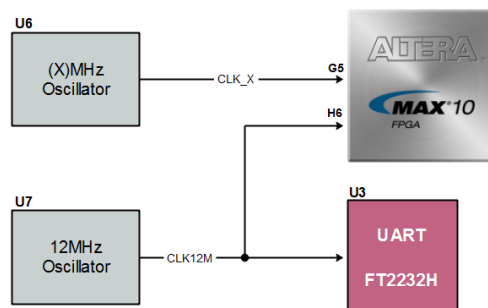


Figure 4 – MAX1000 Clock Tree

Board Reference	FPGA Pin No.	Description	I/O Standard
CLK12M	PIN_H6	12MHz clock input	3.3 V
CLK_X	PIN_G5	Optional clock input	3.3 V

3.3 Peripherals Connected to the FPGA

3.3.1 LEDs

There are eight red user-controllable LEDs connected to the FPGA. Each LED is driven directly and individually by the MAX10 FPGA; driving its associated pin to a high logic level for on or low logic level for off.

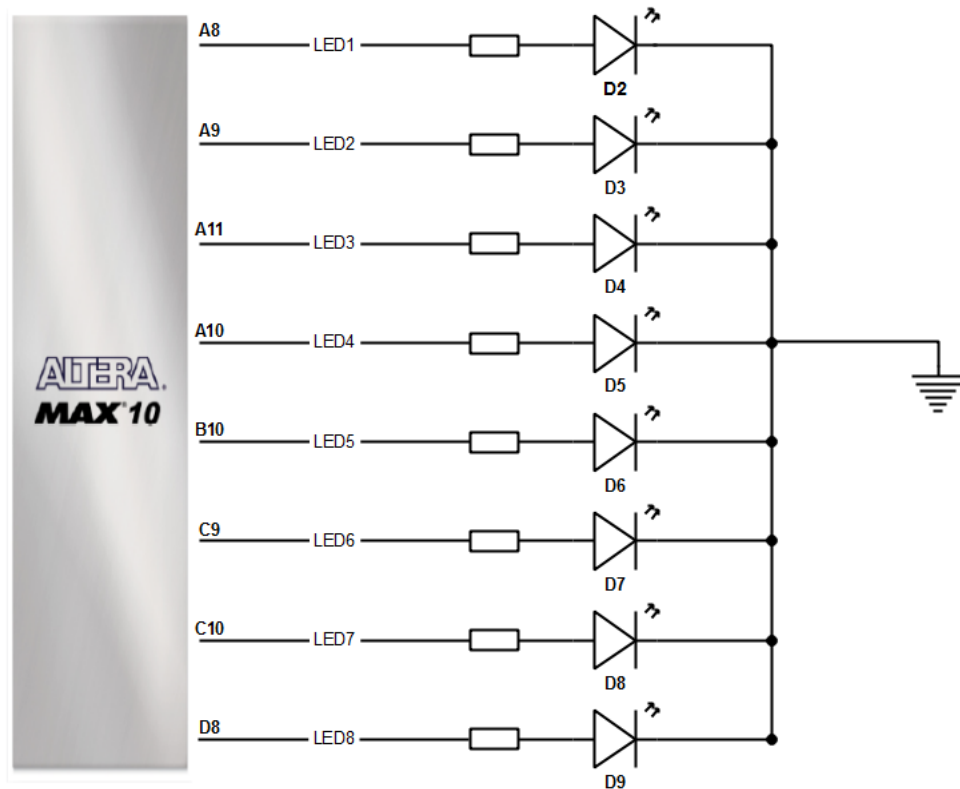


Figure 5 – LED Connections

Board Reference	FPGA Pin No.	I/O Standard
LED1	PIN_A8	3.3 V
LED2	PIN_A9	3.3 V
LED3	PIN_A11	3.3 V
LED4	PIN_A10	3.3 V
LED5	PIN_B10	3.3 V
LED6	PIN_C9	3.3 V
LED7	PIN_C10	3.3 V
LED8	PIN_D8	3.3 V

3.3.2 Push Buttons

The board has two push buttons connected to the FPGA. MAX10 devices support Schmitt trigger input on all I/O pins. A Schmitt trigger feature introduces hysteresis to the input signal for improved noise immunity, especially for signal with slow edge rate and acts as switch debouncer. Push buttons drive their associated pins low logic level when pressed and high logic level when released.

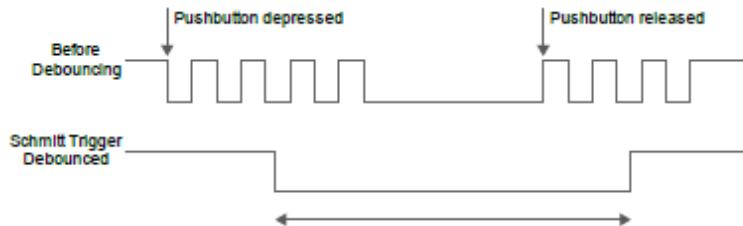


Figure 6 – Button Debouncing

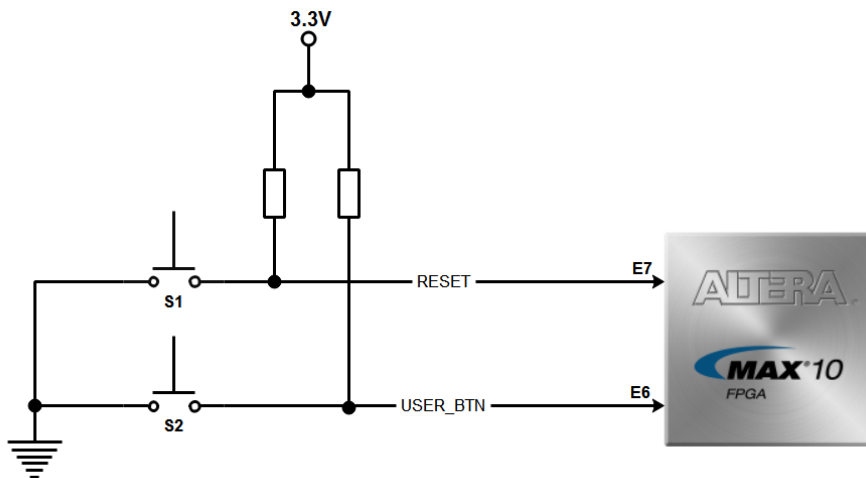


Figure 7 – Button Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
RESET	PIN_E7	Dual-function Pin: GPIO or nCONFIG	3.3 V Schmitt Triger
USER_BTN	PIN_E6	User button	3.3 V Schmitt Triger

3.3.3 Accelerometer

The board comes with a digital accelerometer (LIS3DH), commonly known as the G-Sensor. This G-Sensor is a small, thin, ultra-low power consumption, 3-axis accelerometer with digital I2C/SPI serial interface, standard output. The LIS3DH has user-selectable full scales of +/-2g, +/-4g, +/-8g, +/-16g and it is capable of measuring accelerations with output data rates from 1 Hz to 5 kHz. The supplied power to the board (coming either from micro-USB connection or user Vin) can be monitored through the ADC channel 3 of the accelerometer.

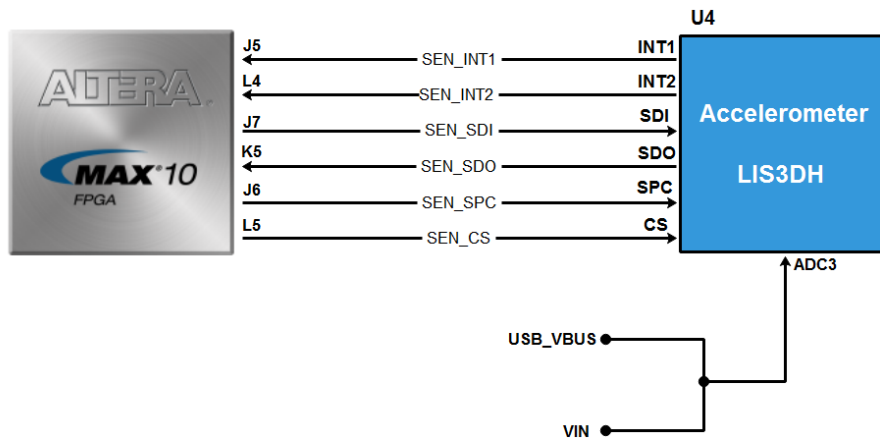


Figure 8 – Accelerometer Connections

Board Reference	FPGA Pin No.	Description*	I/O Standard
SEN_INT1	PIN_J5	Interrupt 1	3.3 V
SEN_INT2	PIN_L4	Interrupt 2	3.3 V
SEN_SDI	PIN_J7	Data In (MOSI)	3.3 V
SEN_SDO	PIN_K5	Data Out (MISO)	3.3 V
SEN_SPC	PIN_J6	Clock	3.3 V
SEN_CS	PIN_L5	Chip Select	3.3 V

*For SPI connection

3.3.4 SDRAM Memory

The MAX1000 board supports 64 MBit (default version) or up to 256 MBit (customized version) SDRAM which can operate up to 166 MHz clock frequency. Below are the connections and pinning of the SDRAM used in the MAX1000.

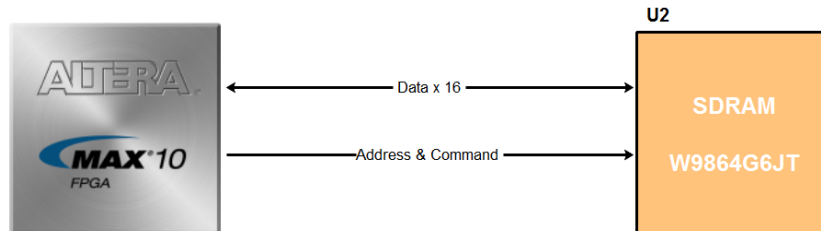


Figure 9 – SDRAM Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
A0	PIN_K6	SDRAM Address [0]	3.3 V
A1	PIN_M5	SDRAM Address [1]	3.3 V
A2	PIN_N5	SDRAM Address [2]	3.3 V
A3	PIN_J8	SDRAM Address [3]	3.3 V
A4	PIN_N10	SDRAM Address [4]	3.3 V
A5	PIN_M11	SDRAM Address [5]	3.3 V
A6	PIN_N9	SDRAM Address [6]	3.3 V
A7	PIN_L10	SDRAM Address [7]	3.3 V
A8	PIN_M13	SDRAM Address [8]	3.3 V
A9	PIN_N8	SDRAM Address [9]	3.3 V
A10	PIN_N4	SDRAM Address [10]	3.3 V
A11	PIN_M10	SDRAM Address [11]	3.3 V
A12	PIN_L11	SDRAM Address [12]	3.3 V
A13	PIN_M12	SDRAM Address [13]	3.3 V
BA0	PIN_N6	SDRAM Bank Address [0]	3.3 V
BA1	PIN_K8	SDRAM Bank Address [1]	3.3 V
CLK	PIN_M9	SDRAM Input Clock	3.3 V
CKE	PIN_M8	SDRAM Clock Enable	3.3 V
RAS	PIN_M7	SDRAM Row Address Strobe	3.3 V
CAS	PIN_N7	SDRAM Column Address Strobe	3.3 V
WE	PIN_K7	SDRAM Write Enable	3.3 V
CS	PIN_M4	SDRAM Chip Select	3.3 V

Board Reference	FPGA Pin No.	Description	I/O Standard
DQ0	PIN_D11	SDRAM Data[0]	3.3 V
DQ1	PIN_G10	SDRAM Data [1]	3.3 V
DQ2	PIN_F10	SDRAM Data [2]	3.3 V
DQ3	PIN_F9	SDRAM Data [3]	3.3 V
DQ4	PIN_E10	SDRAM Data [4]	3.3 V
DQ5	PIN_D9	SDRAM Data [5]	3.3 V
DQ6	PIN_G9	SDRAM Data [6]	3.3 V
DQ7	PIN_F8	SDRAM Data [7]	3.3 V
DQ8	PIN_F13	SDRAM Data [8]	3.3 V
DQ9	PIN_E12	SDRAM Data [9]	3.3 V
DQ10	PIN_E13	SDRAM Data [10]	3.3 V
DQ11	PIN_D12	SDRAM Data [11]	3.3 V
DQ12	PIN_C12	SDRAM Data [12]	3.3 V
DQ13	PIN_B12	SDRAM Data [13]	3.3 V
DQ14	PIN_B13	SDRAM Data [14]	3.3 V
DQ15	PIN_A12	SDRAM Data [15]	3.3 V
DQM0	PIN_E9	SDRAM Lower Data Mask	3.3 V
DQM1	PIN_F12	SDRAM Upper Data Mask	3.3 V

3.3.5 Flash Memory

The MAX1000 board supports up to 128 MBit of serial flash memory that can be used for user data and programming non-volatile storage. Single SPI interface is supported that can reach up to 104 MHz SPI performance. Device offers unique advanced security features that could benefit greatly end node security applications.

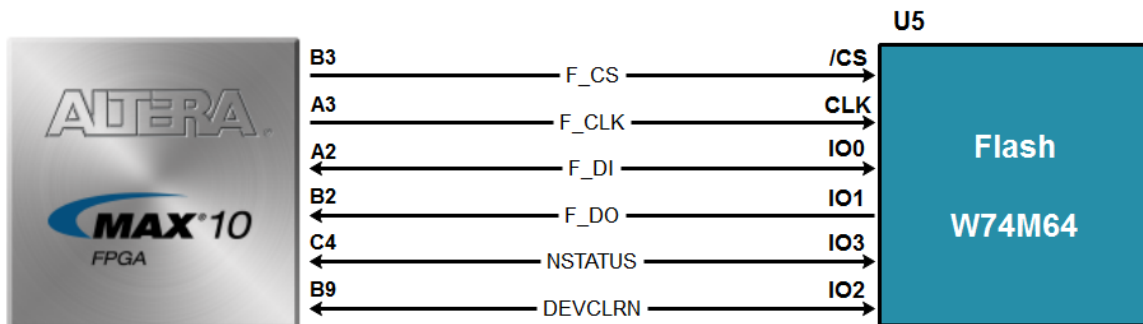


Figure 10 – Flash Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
F_CS	PIN_B3	Flash Chip Select	3.3 V
F_CLK	PIN_A3	Flash Clock Input	3.3 V
F_DI	PIN_A2	Flash Data Input Input (IO0)	3.3 V
F_DO	PIN_B2	Flash Data Input Output (IO1)	3.3 V
NSTATUS	PIN_C4	NSTATUS	3.3 V
DEVCLR_N	PIN_B9	DEVCLR_N	3.3 V

3.3.6 User I/O

The MAX1000 board has two pins that can be connected to the board and take analogue inputs. Those two pins are connected to the ADC block of the MAX10 and can measure up to 3.6V. This 12-bit SAR ADC can reach 1 MSPS and has single ended capabilities.

For more information on the ADCs of the MAX10, please refer to this [document](#).

Note: If you use bank 1A for ADC, you cannot use the bank for GPIO.

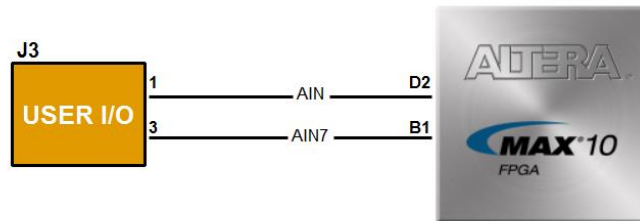


Figure 11 - User I/O Connections

Board Reference	FPGA Pin No.	Description
AIN	PIN_D2	Dedicated analogue input pin
AIN7	PIN_B1	Dual function pin

3.3.7 Arduino MKR Connectors

The MAX1000 board offers connectivity to Arduino MKR compatible shields that could also alternatively be used as GPIOs. The MKR connectors offer up to 7 analogue inputs and 15 digital I/Os. There is also an option to use or not use the 4.7k Ohm pull-up resistor lines for communication interfaces.

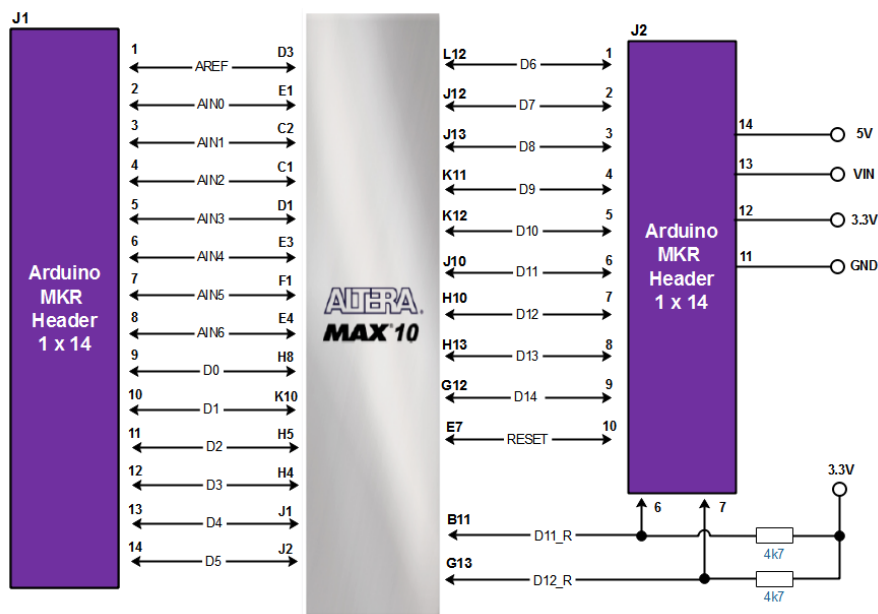


Figure 12 - Arduino MKR Header Connections



Note: If you use bank 1A for ADC, you cannot use the bank for GPIO (includes all AIN[0..7] pins)

Board Reference	FPGA Pin No.	Description	MKR Header/Pin	I/O Standard
AREF	PIN_D3	Analogue Reference	J1 / 1	N/A
AIN0	PIN_E1	Analogue In Channel 8 of ADC or GPIO	J1 / 2	3.3 V
AIN1	PIN_C2	Analogue In Channel 2 of ADC or GPIO	J1 / 3	3.3 V
AIN2	PIN_C1	Analogue In Channel 5 of ADC or GPIO	J1 / 4	3.3 V
AIN3	PIN_D1	Analogue In Channel 1 of ADC or GPIO	J1 / 5	3.3 V
AIN4	PIN_E3	Analogue In Channel 3 of ADC or GPIO	J1 / 6	3.3 V
AIN5	PIN_F1	Analogue In Channel 7 of ADC or GPIO	J1 / 7	3.3 V
AIN6	PIN_E4	Analogue In Channel 4 of ADC or GPIO	J1 / 8	3.3 V
D0	PIN_H8	Digital In[0]	J1 / 9	3.3 V
D1	PIN_K10	Digital In[1]	J1 / 10	3.3 V
D2	PIN_H5	Digital In[2]	J1 / 11	3.3 V
D3	PIN_H4	Digital In[3]	J1 / 12	3.3 V
D4	PIN_J1	Digital In[4]	J1 / 13	3.3 V
D5	PIN_J2	Digital In[5]	J1 / 14	3.3 V
D6	PIN_L12	Digital In[6]	J2 / 1	3.3 V
D7	PIN_J12	Digital In[7]	J2 / 2	3.3 V
D8	PIN_J13	Digital In[8]	J2 / 3	3.3 V
D9	PIN_K11	Digital In[9]	J2 / 4	3.3 V
D10	PIN_K12	Digital In[10]	J2 / 5	3.3 V
D11	PIN_J10	Digital In[11]	J2 / 6	3.3 V
D12	PIN_H10	Digital In[12]	J2 / 7	3.3 V
D13	PIN_H13	Digital In[13]	J2 / 8	3.3 V
D14	PIN_G12	Digital In[14]	J2 / 9	3.3 V
D11_R	PIN_B11	Digital In[11] with pull-up resistor	J2 / 6	3.3 V
D12_R	PIN_G13	Digital In[12] with pull-up resistor	J2 / 7	3.3 V
RESET	N/A	Connected to system reset of the board	J2 / 10	N/A
GND	N/A	Ground output to the connector	J2 / 11	N/A
3.3V	N/A	3.3V power to the connector	J2 / 12	N/A
VIN	N/A	User power into to the MAX1000	J2 / 13	N/A
5V	N/A	5V power to the connector	J2 / 14	N/A

**Can only choose one, hence same name pinning*

3.3.8 PMOD Connector

The MAX1000 board offers connectivity to PMOD compatible connectors (2x6-pin or 1x12-pin), making it possible to add a big variety of sensors or ICs to the system. Below is the connection schematic and pinning information.

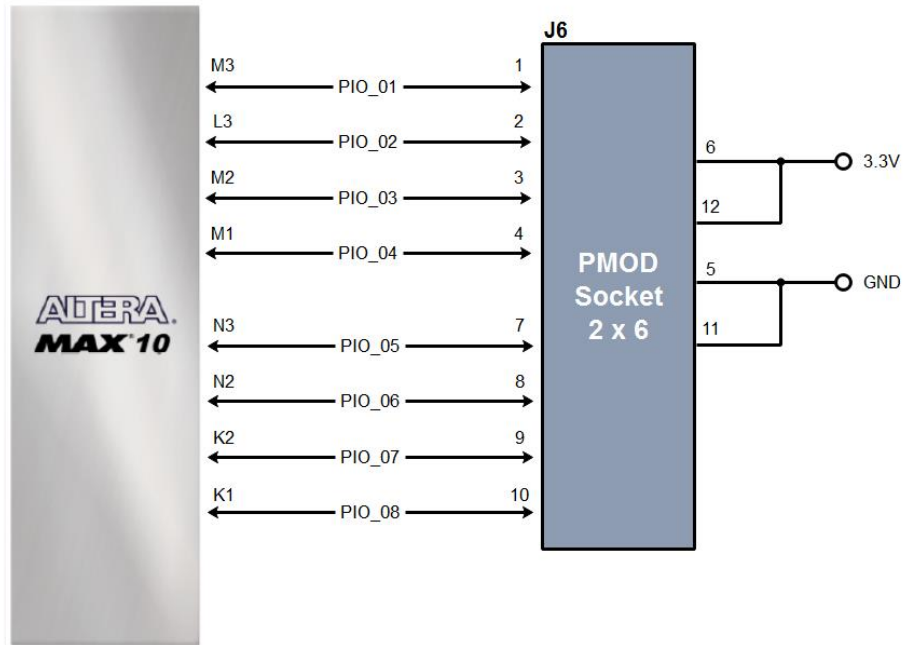


Figure 13 – PMOD Header Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
PIO_01	PIN_M3	PMOD Pin 1	3.3 V
PIO_02	PIN_L3	PMOD Pin 2	3.3 V
PIO_03	PIN_M2	PMOD Pin 3	3.3 V
PIO_04	PIN_M1	PMOD Pin 4	3.3 V
PIO_05	PIN_N3	PMOD Pin 5	3.3 V
PIO_06	PIN_N2	PMOD Pin 6	3.3 V
PIO_07	PIN_K2	PMOD Pin 7	3.3 V
PIO_08	PIN_K1	PMOD Pin 8	3.3 V
GND	N/A	Ground	N/A
3.3V	N/A	3.3 V Power to PMOD	3.3 V

3.3.9 Communication and Configuration

The MAX1000 board uses a single chip to perform configuration of the device and USB to UART communications, having each described below.

3.3.9.1 UART Communication

UART to USB communication supports USB 2.0 High Speed (up to 480 Mb/s) independently of other protocols used in the chip like JTAG. Below is the connection schematic and pinning information.

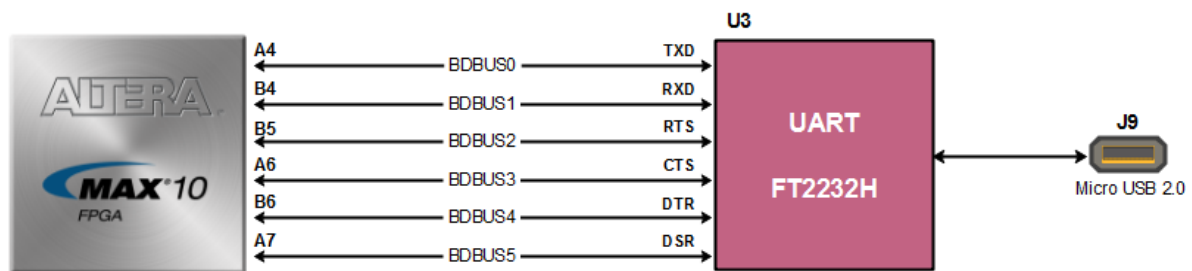


Figure 14 – UART Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
BDBUS0	PIN_A4	Transmitter output of FT2232H (Tx)	3.3 V
BDBUS1	PIN_B4	Receiver input of FT2232H (Rx)	3.3 V
BDBUS2	PIN_B5	Ready To Send handshake output (RTS)	3.3 V
BDBUS3	PIN_A6	Clear To Send handshake input (CTS)	3.3 V
BDBUS4	PIN_B6	Data Transmit Ready (DTR)	3.3 V
BDBUS5	PIN_A7	Data Set Ready (DSR)	3.3 V

3.3.9.2 JTAG Chain Configuration

There are two types of configuration methods supported by MAX1000:

1. **JTAG Configuration:** configuration using JTAG ports. JTAG configuration scheme allows you to directly configure the device core through JTAG pins (TDI, TDO, TMS and TCK pins). The Quartus Prime software automatically generates a .sof that can be downloaded to the MAX10 with a download cable through the Quartus Prime Programmer.
2. **Internal Configuration:** configuration using internal flash. Before internal configuration, you need to program the configuration data .pof into the configuration flash memory (CFM) which provides non-volatile storage for the bit stream. The information is retained within CFM even if the MAX1000 is turned off. When the board is powered on, the configuration data in the CFM is automatically loaded into the MAX10 FPGA.



The FPGA device can be configured through JTAG interface on MAX1000, but the JTAG chain must form a closed loop, which allows Quartus Prime programmer to detect the FPGA device.

MAX1000 offers two ways of configuring your board.

- 1) Through the on-board Arrow USB Programmer2
- 2) Pins for connecting user’s preferred JTAG interface

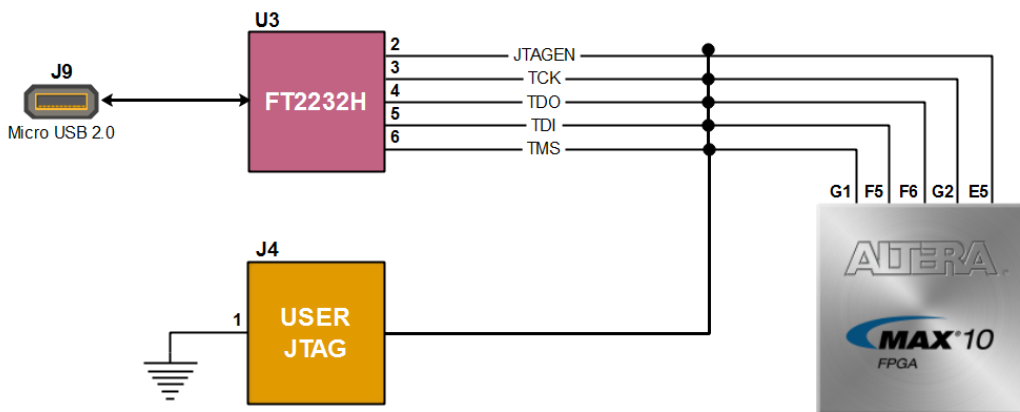


Figure 15 – JTAG Connections

Board Reference	FPGA Pin No.	Description	I/O Standard
JTAGEN	PIN_E5	Dual Function: JTAG Pin Sharing	3.3 V
TCK	PIN_G2	Test Interface Clock	3.3 V
TDO	PIN_F6	Test Data Out	3.3 V
TDI	PIN_F5	Test Data In	3.3 V
TMS	PIN_G1	Test Mode Select	3.3 V

For detailed information about how to configure the MAX10 device, please refer to Chapter 6.

3.3.10 Power Tree

The MAX1000 is powered by Enpirion’s buck regulator, which provides high efficiency up to 1A with integrated magnetics, switches, control, and compensation. As seen from the diagram below, the board can be powered either by a micro-USB connection, or by user input voltage from the Arduino MKR header (takes precedence over the USB bus). All devices are powered by 3.3V voltage line and the 5V and 3.3V lines are fed back to the Arduino header to power that connection if needed.

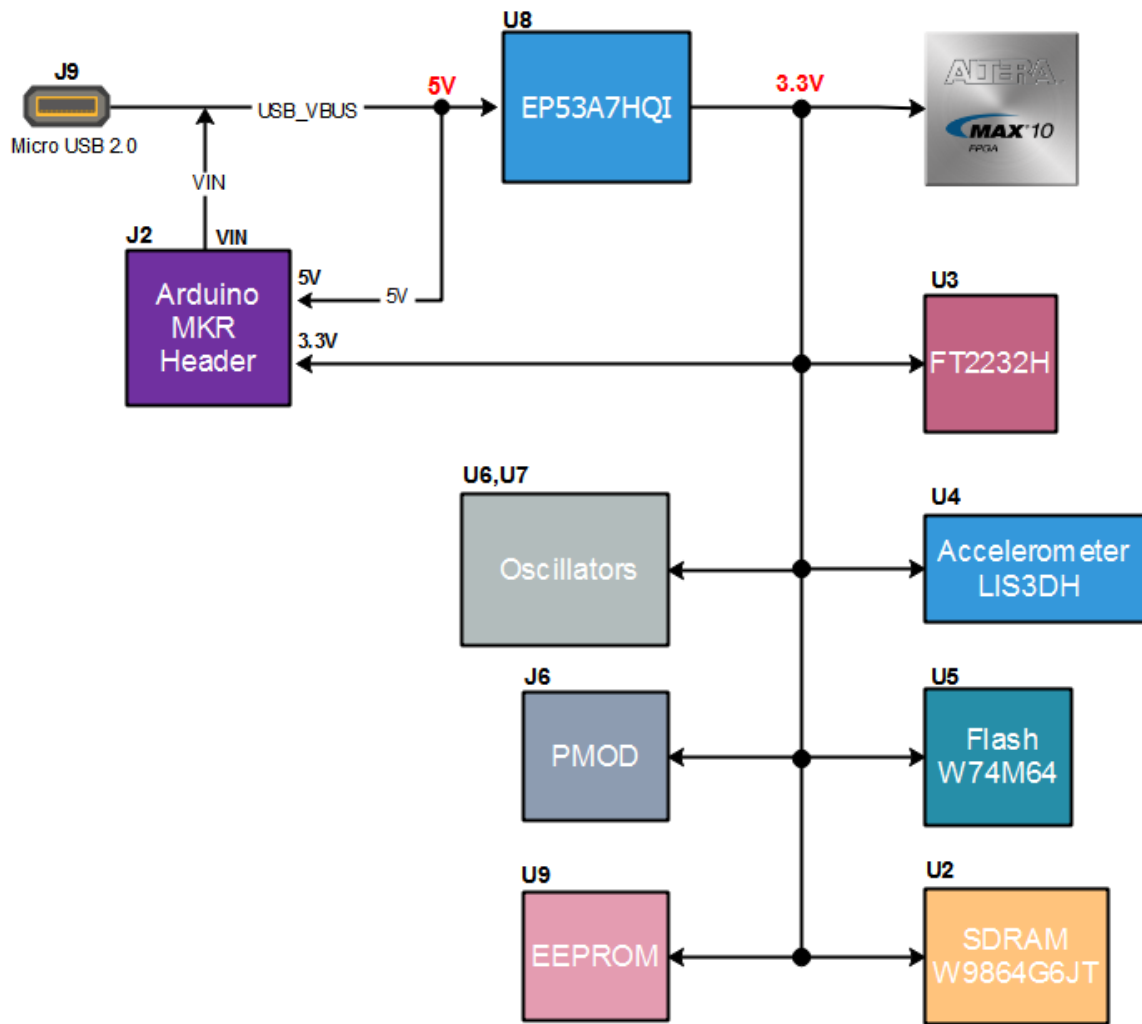


Figure 16 – Power Tree Connections

Chapter 4 - Software and Driver Installation

Firstly it is required to create your [myAltera Account](#) if you don't own one already. It is required to download the software. Below are guides for installing the software and drivers for Windows and Linux operating systems.

4.1 Installing Quartus Prime Software (Windows)

4.1.1 Go to the Intel Download Centre: [Link](#).

4.1.2 Select **Windows** as the operating system (highlighted in red).

4.1.3 Select Release **17.0**, or your preferred version (highlighted in red).

4.1.4 Select the following files from the "Individual Files" tab to be downloaded (highlighted in yellow):

- Quartus Prime Lite Edition (Free)
- ModelSim-Altera Edition (includes Starter Edition)
- MAX10 FPGA device support

Quartus Prime Lite Edition
Release date: May, 2017
Latest Release: v17.0

Select edition: Lite
Select release: 17.0

Operating System: Windows (selected), Linux

Download Method: Akamai DLM3 Download Manager (selected), Direct Download

✓ The Quartus Prime software version 17.0 supports the following device families: Arria II, Cyclone 10 LP, Cyclone IV, Cyclone V, MAX II, MAX V, and MAX 10 FPGA. [More](#)

Combined Files | Individual Files | Additional Software | Updates

Download and install instructions: [More](#)
[Read Intel FPGA Software v17.0 Installation FAQ](#)
[Quick Start Guide](#)

Select All

Quartus Prime Lite Edition (Free)
 Quartus Prime (includes Nios II EDS)
Size: 1.7 GB MD5: F95E47F859713C3C6DD59A94A9FC5E43
 ModelSim-Intel FPGA Edition (includes Starter Edition)
Size: 1.1 GB MD5: B3E4A6C66187D198FF976FFB3566D967

Devices
You must install device support for at least one device family to use the Quartus Prime software

Arria II device support
Size: 499.6 MB MD5: BDFCC96CB847AADEA46B8FF4E5E1A0B6

Cyclone IV device support
Size: 466.6 MB MD5: F36179518284561D5FAE66CF584748BC

Cyclone 10 LP device support
Size: 266.1 MB MD5: D0726B51AEFF72E88BCD629A2C2A23

Cyclone V device support
Size: 1.1 GB MD5: 978CC235B50DA9BCC95C709EDADE4503

MAX II, MAX V device support
Size: 11.4 MB MD5: CD9DC70DB8AFF4616DCFA78BFD98BD74

MAX 10 FPGA device support
Size: 325.1 MB MD5: A5499E644D470EC0609AF1B4E69CEB83

[Updates Available](#)

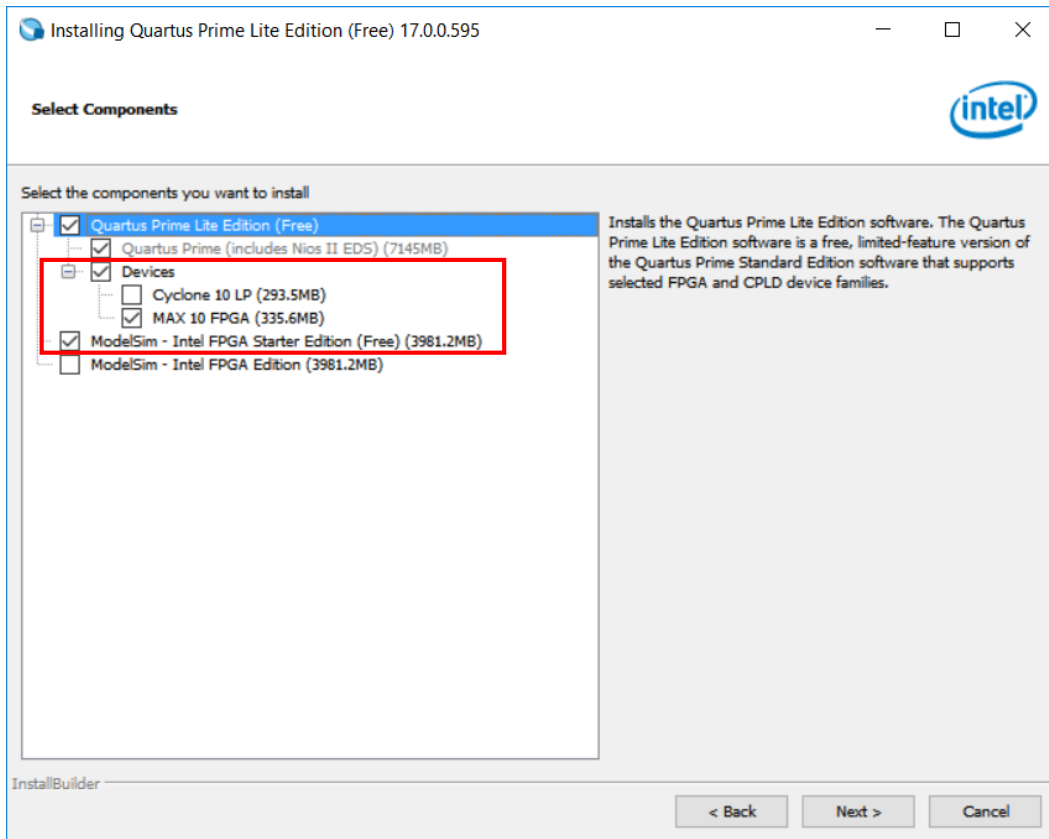
[Download Selected Files](#)

4.1.5 Click "Download Selected Files" to begin the download and save them in the same folder.

Note: In case of browser issues using Akamai DLM3 Manager, you can alternative download the above files using the “Direct Download” (highlighted in green).

4.1.6 After the download is finished, run the Quartus Prime installer.

4.1.7 When prompted to select the components, the installer will detect automatically the MAX10 device support and ModelSim packages when they are in the same folder. Make sure these components are selected:



4.1.9 Finish the installation of the Quartus Lite and proceeded to the next section to install Arrow USB Programmer2 to be able to connect to the MAX1000 board.

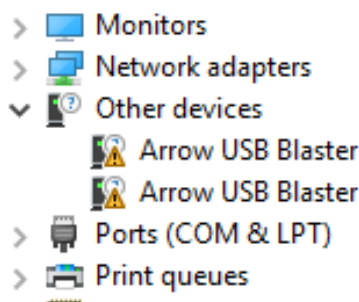
4.2 Installing Arrow USB Programmer2 (Windows)

The MAX1000 board uses version 2 of the Arrow USB Programmer2 programming solution, that is an FTDI FT232H Hi-Speed USB controller plus a programmer DLL. Since this FTDI USB controller is a very common standard device, usually no specific drivers are needed to make the MAX1000 work.

4.2.1 Download the appropriate version of Arrow USB Programmer2 for MAX1000 from Trenez Electronic Wiki page or alternatively this direct [link](#).

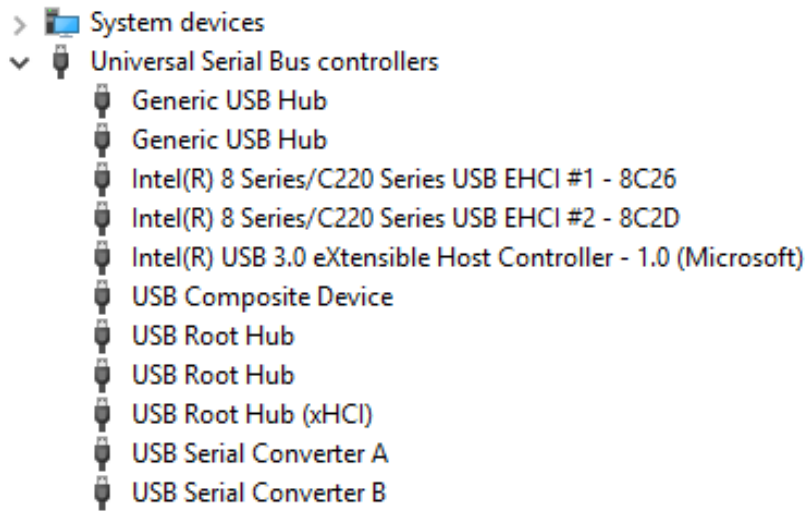
4.2.2 After downloading the file, run the installer to install the Arrow USB Programmer2. The setup executable installs the programmer DLL and adds some keys to the registry of the PC.

4.2.3 After connecting the MAX1000 board to the PC, two unknown devices might appear in the “Other devices” section of device manager of the PC.

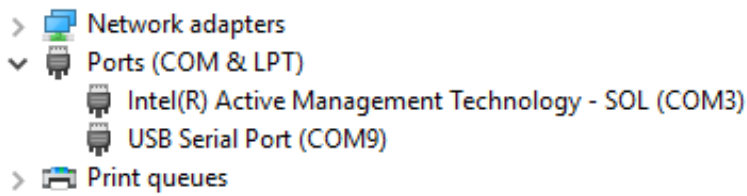




Windows usually automatically finds the appropriate drivers for these devices. After some time, the “Other devices” section should be empty. Instead, two USB Serial Converters should be listed in the section “USB Serial Bus controllers”:



Furthermore, a USB Serial Port should be listed in the “Ports (COM & LPT)” section.



Note: The number of the port will most probably be different from the one shown here.

In case Windows does not automatically find the appropriate drivers go to <http://www.ftdichip.com/Drivers/D2XX.htm> to download the setup executable to install the required drivers.

4.3 Installing Quartus Prime Software (Linux)

- 4.3.1 Go to the Intel Download Centre: [Link](#).
- 4.3.2 Select **Linux** as the operating system (highlighted in red).
- 4.3.3 Select Release **17.0**, or your preferred version (highlighted in red).
- 4.3.4 Select the following files to be downloaded from the “Individual Files” tab (highlighted in yellow):
 - Quartus Prime (Includes Nios II EDS)
 - ModelSim-Intel FPGA Edition (includes Starter Edition)
 - MAX10 FPGA device support

Quartus Prime Lite Edition
Release date: May, 2017
Latest Release: v17.0

Intel® Quartus® Prime
Design Software

Select edition: Lite
Select release: 17.0

Operating System: Windows, Linux

Download Method: Akamai DLM3 Download Manager, Direct Download

✓ The Quartus Prime software version 17.0 supports the following device families: Arria II, Cyclone 10 LP, Cyclone IV, Cyclone V, MAX II, MAX V, and MAX 10 FPGA. [More](#)

Combined Files | **Individual Files** | Additional Software | Updates

Download and install instructions: [More](#)
[Read Intel FPGA Software v17.0 Installation FAQ](#)
[Quick Start Guide](#)

Quartus Prime Lite Edition (Free)
Quartus Prime (includes Nios II EDS)
Size: 2.0 GB MD5: 454350830D7B0072396B5F0DED77E18B [Updates Available](#)

ModelSim-Intel FPGA Edition (includes Starter Edition)
Size: 1.1 GB MD5: 0B5F39AFFB77E90700C30F96DAB1A0C
** Require 32-bit libraries, see [installation manual](#).

Devices
You must install device support for at least one device family to use the Quartus Prime software.

Arria II device support
Size: 499.6 MB MD5: BDFCC96CB847AADEA46B8FF4E5E1A0B6

Cyclone IV device support
Size: 466.6 MB MD5: F36179518284561D5FAE66CF5B4748BC

Cyclone 10 LP device support
Size: 266.1 MB MD5: D0726B51AEFF72E88BCCD629A2C2A23

Cyclone V device support
Size: 1.1 GB MD5: 978CC235B50DA9BCC95C709EDADE4503

MAX II, MAX V device support
Size: 11.4 MB MD5: CD9DC70DB8AFF4616DCFA7BBFD98BD74

MAX 10 FPGA device support
Size: 325.1 MB MD5: A5499E644D470EC0609AF1B4E69CEB83

4.3.5 Download the selected files and store them into the same folder.

For example, the Downloads folder under the home of the current user **~/Downloads:**
Quartus Prime Lite edition: ~/Downloads/QuartusLiteSetup-17.0.0.595-linux.run
ModelSim-Intel FPGA Edition: ~/Downloads/ModelSimSetup-17.0.0.595-linux.run
MAX10 device support package for Quartus: ~/Downloads/max10-17.0.0.595.qdz

Run Intel Quartus installer. Open a terminal and navigate through the folder where the installers are downloaded:

```
$ cd ~/Downloads
```

Make sure the installer has execution permissions. If you have a different version of Quartus Software, please match the name of the file:

```
$ chmod +x QuartusLiteSetup-17.0.0.595-linux.run
```

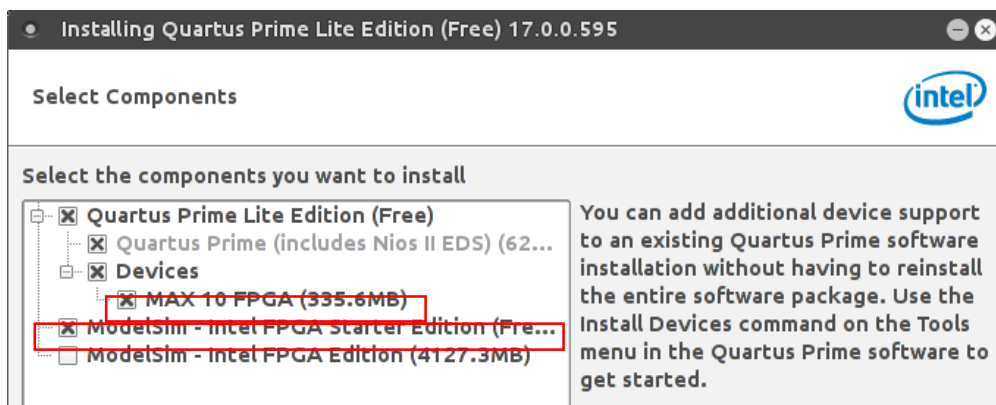
Run the installer:

```
$ ./QuartusLiteSetup-17.0.0.595-linux.run
```

4.3.6 Follow the steps through the installation assistant. When prompted the destination folder, you can choose a path within your home directory, so you don't need additional permissions:



4.3.7 The installer will detect automatically the MAX10 device support and ModelSim packages when they are located in the same folder. Make sure these components are selected:





4.3.8 Continue through the installation assistant until the process has finished. Now you can run Quartus Lite.

To connect with the MAX1000 board you need to install additional Arrow USB Programmer2 drivers as explained in the next section.

4.4 Installing Arrow USB Programmer2 (Linux)

4.4.1 Download the Arrow USB Programmer2 Driver for Linux from Trenz Electronic Wiki page or alternatively this direct [link](#).

The screenshot shows the Arrow website's navigation menu with 'Download' selected. The breadcrumb trail is: Download > Trenz_Electronic > Software > Drivers > Arrow_USB_Programmer > Arrow_USB_Programmer_2.1. A sidebar menu on the left lists various product categories, with 'Trenz_Electronic' selected. The main content area is titled 'Arrow USB Programmer 2.1 Libraries' and includes a 'Notes' section with a bullet point: 'If you did not find the necessary documents, please send a request mail to Trenz Electri...'. Below this is a 'Files' section listing two zip files: 'Arrow_USB_Programmer_2.1_linux64.zip' (Size 64,86 KB / Modified 12.12.2017 - 20:36:33) and 'Arrow_USB_Programmer_2.1_win64.zip' (Size 790,13 KB / Modified 12.12.2017 - 20:37:07).

4.4.2 You can download the files in a temporal folder such as the Downloads folder under the home of the current user **~/Downloads**:

- Udev USB rules: `~/Downloads/51-usbProgrammer2.rules`
- Default configuration file: `~/Downloads/arrow_usb_Programmer2.conf`:
- Plugin for Quartus Prime: `~/Downloads/libjtag_hw_arrow.so`:

4.4.3 Open a terminal and navigate through your Downloads folder:

```
$ cd ~/Downloads
```

4.4.4 Install udev USB rules:

```
$ sudo cp 51-usbProgrammer2.rules /etc/udev/rules.d/
```



4.4.5 Make sure your linux user belongs to the plugdev group:

```
$ sudo useradd -G plugdev <your linux username>
```

4.4.6 Install default configuration file:

```
$ sudo cp arrow_usb_Programmer2.conf /etc/
```

4.4.7 Install the Quartus Prime plugin:

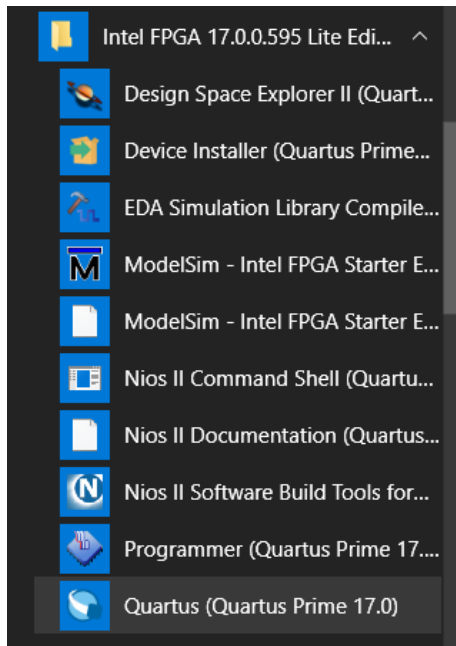
Use the directory quartus/linux64 under the installation path selected on step 4.3.6.

```
$ cp libjtag_hw_arrow.so <Quartus Path>/quartus/linux64
```

Chapter 5 - New Project with MAX1000

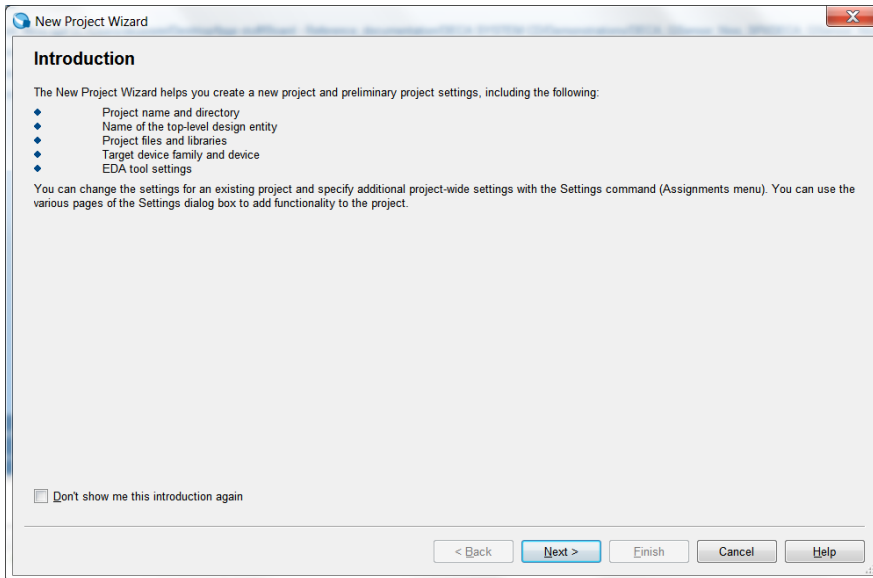
5.1 Creating a new Blinky Project with MAX1000

5.1.1 Launch Quartus Prime Lite Edition from the Start Menu.



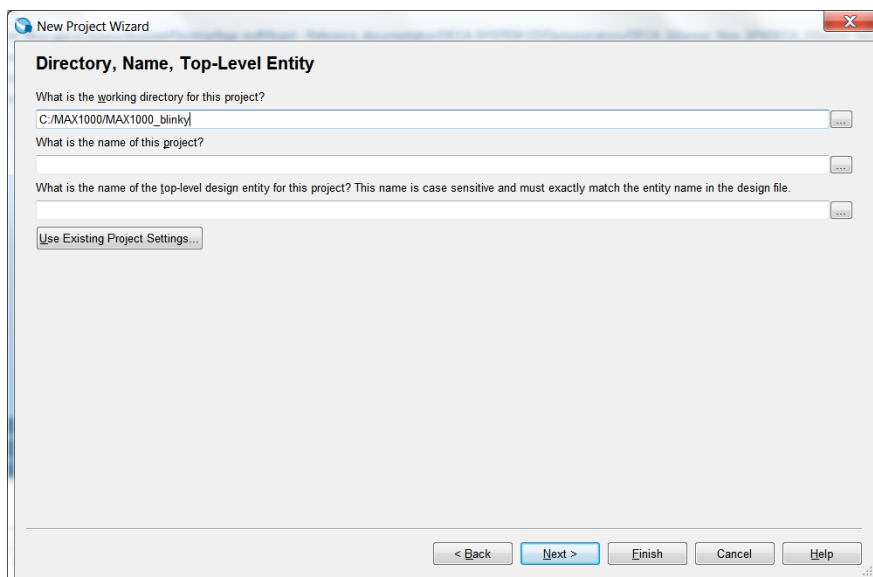
5.1.2 In the Quartus Prime tool, create a new project: **File -> New Project Wizard.**

The New Project Wizard walks you through the project settings, such as the name, directories, files, directories, device family and other settings. These settings can be changed later if needed.

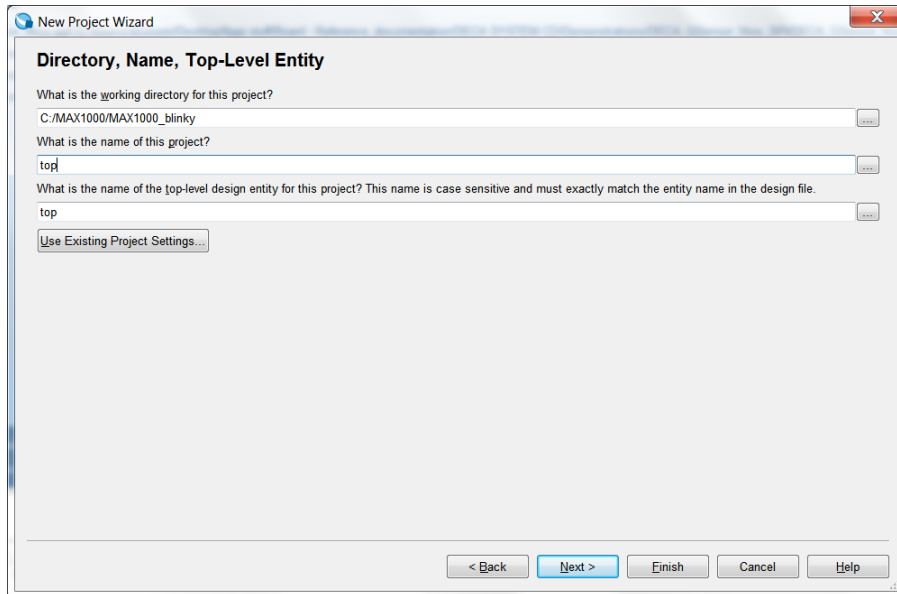


5.1.3 Click “Next”.

5.1.4 Browse in the project directory and choose a preferred location for the new project. Then create new folder named MAX1000_blinky. This will be the folder containing all the project files.



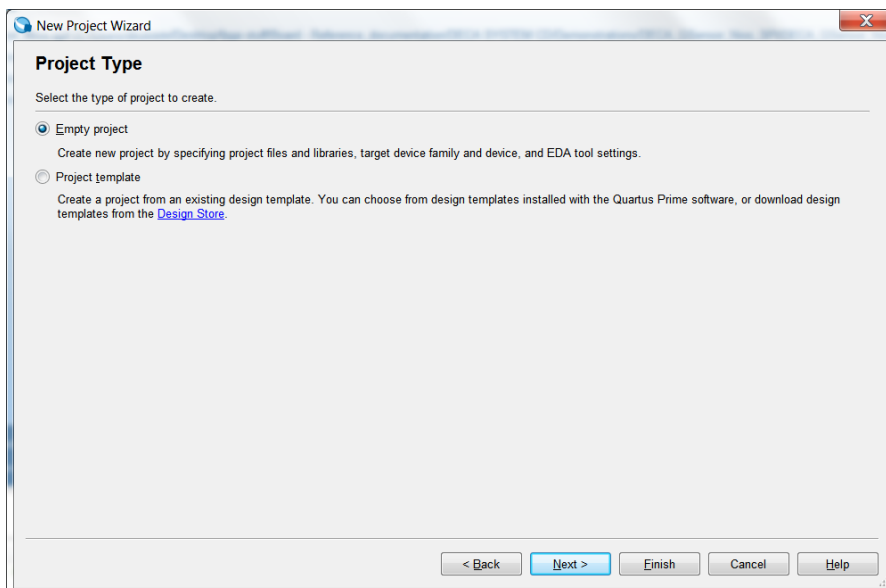
5.1.5 Enter the project name: “top”.



5.1.6 Click “Next”.

5.1.7 Project Type

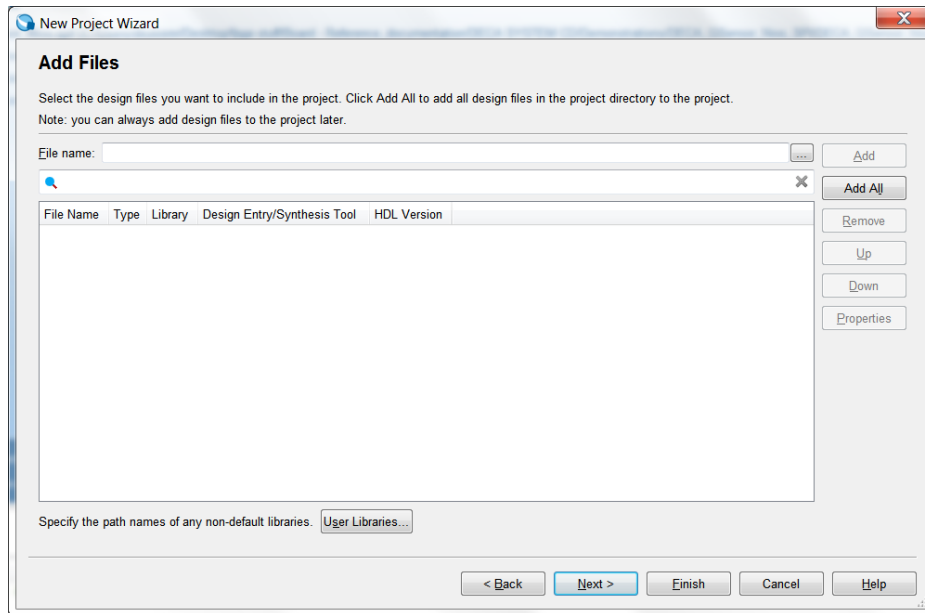
In this page you choose the Project Type. In this tutorial, a new project will be created, and thus the default settings of empty project should be selected.



5.1.8 Click “Next”.

5.1.9 Add Project Files

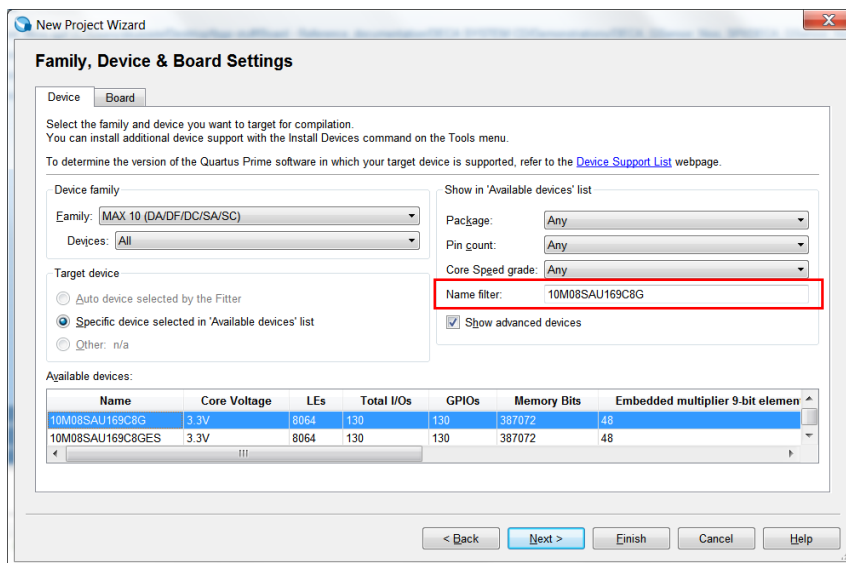
The Add File window will appear. For this tutorial, new design files will be created so no files will be added. For other designs, files could be added here.



5.1.10 Click “Next”.

5.1.11 Select the Device Part Number of the MAX1000 Board

In the Family and Device Settings, use the pull down menu to select the family as MAX10. Then in the Name Filter enter **10M08SAU169C8G**.

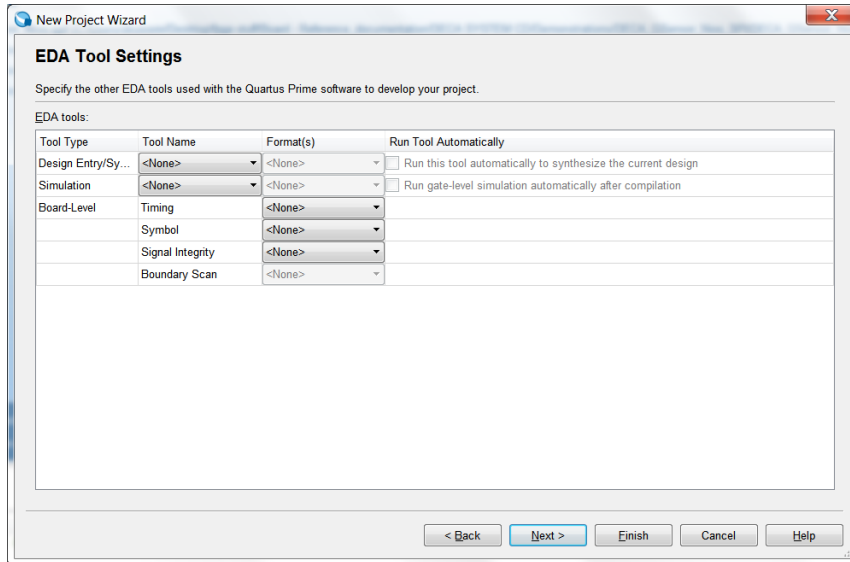


Rather than entering the exact part number, the pull-down menus can be used to select the correct family, package, pin count, and speed grade. Quartus Prime will use these settings to compile the design, and also provide the programming file that you will use later to program the device.

5.1.12 Click “Next”.

5.1.13 EDA Tool Settings

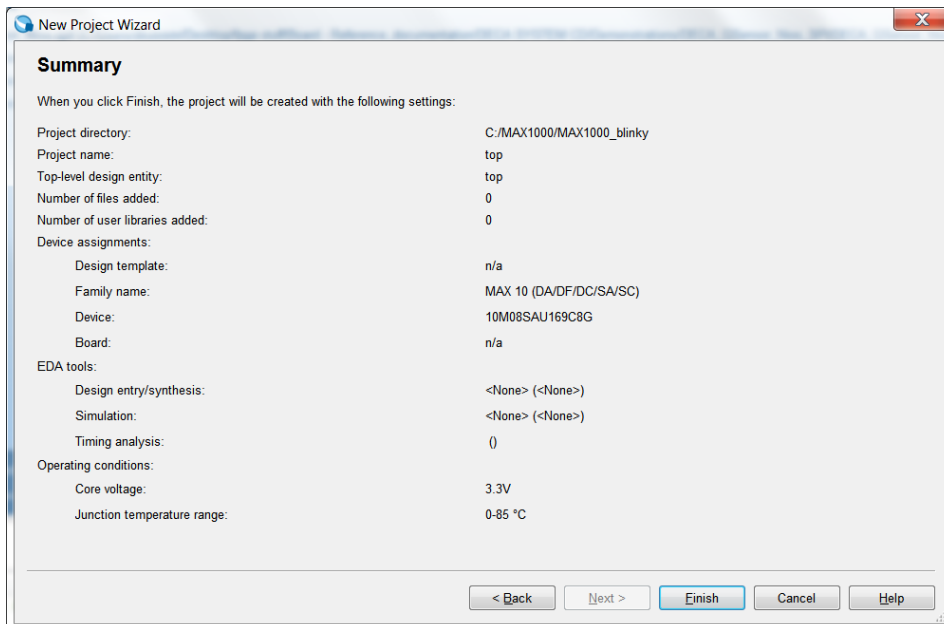
In the EDA tool Settings window, disable any EDA tools, if there are any present. EDA tools are third party tools that work with Quartus Prime for design entry, simulation, verification and board-level timing. For this tutorial, no EDA software will be used, as only Quartus Prime will be used.



5.1.14 Click “Next”.

5.1.15 Project Summary Page

This is the Summary Page that showing the settings Quartus Prime will use for this Project. Those settings can be changed if required at a later time.



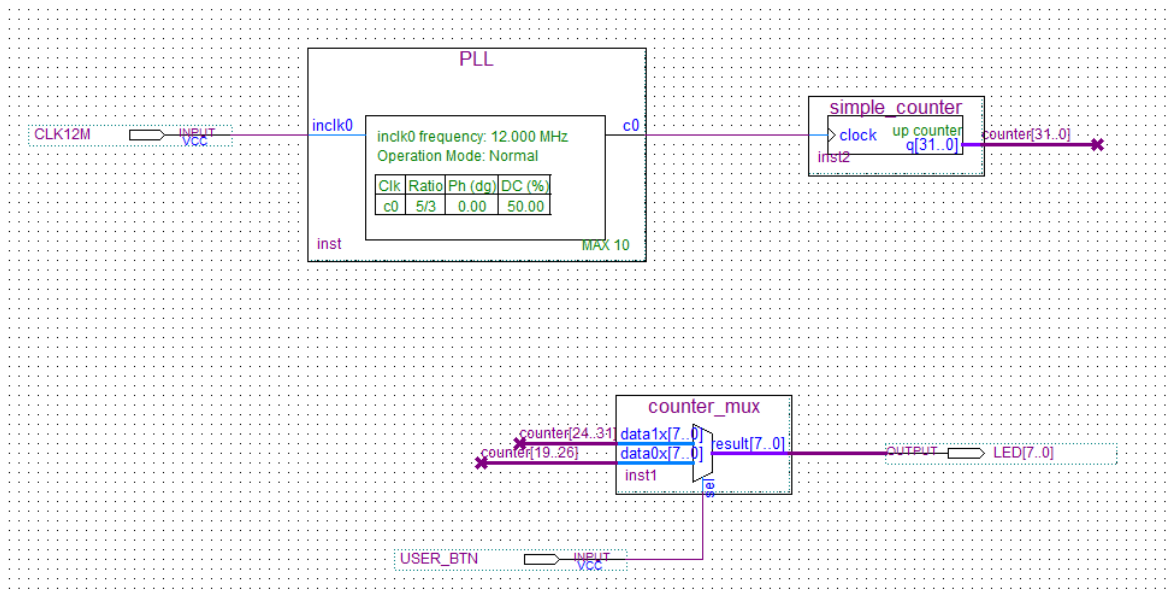
5.1.16 Click “Finish”.

5.2 Building a Blinky Project with MAX1000

Overview: In this section you will create the components to a design, make connections, set the pins and compile a project. The goal is to go through the design process of a simple blinky project, where the toggle speed of the LEDs could be controlled by one of the pushbuttons of the MAX1000.

5.2.1 Block Diagram

The final system that will be built with the following steps will look as follows when complete:

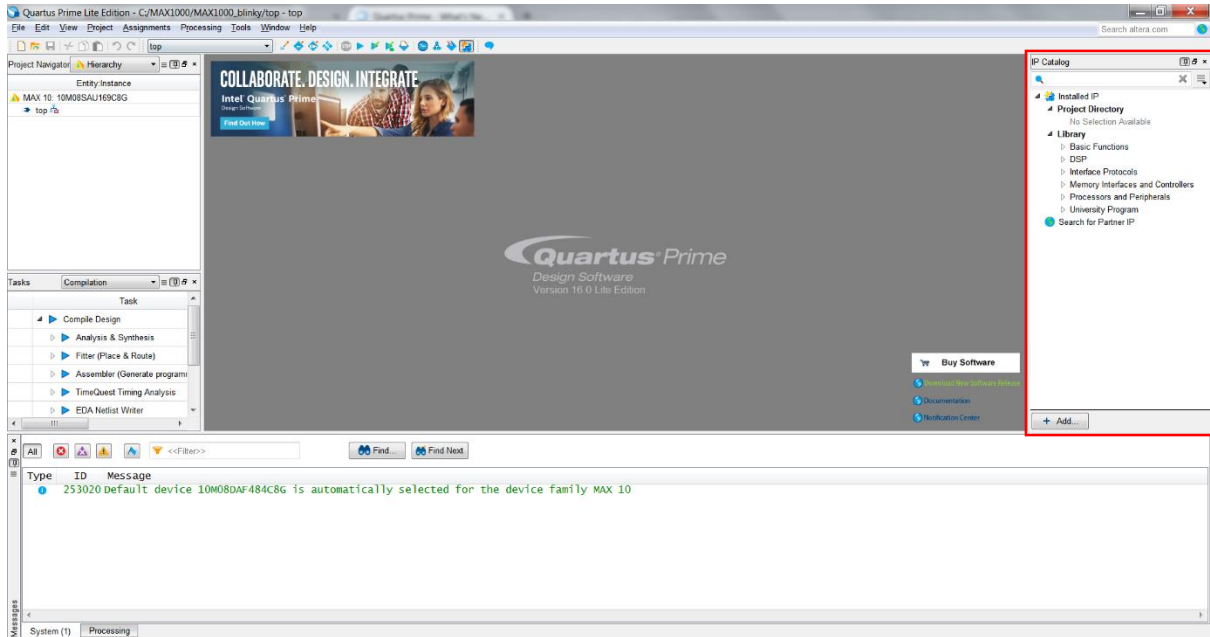


5.2.2 Components of the Design

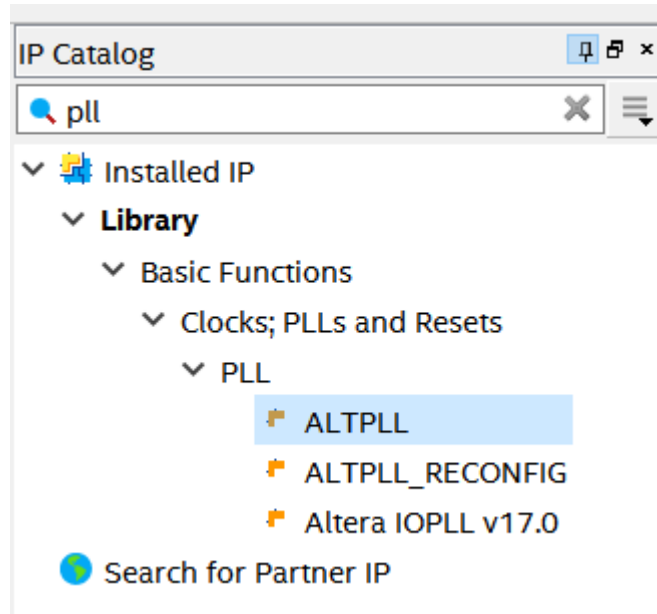
There are three components in the system: a PLL, a counter and a mux. The components, in the following steps, will be built separately and then connected together. A user push button on the board controls the mux. The mux in turn control which of the counter outputs (slow counting or fast counting) will be shown on the LEDs. There are different ways to create components, such as RTL or schematic. In this lab, schematics will be used. There are also different ways for entering schematics such as Qsys and IP Catalog. This lab will focus on the IP Catalog.

5.2.3 Catalog IP

The IP Catalog allows you to create and modify design files with custom variations. The IP Catalog window is open by default when you open Quartus Prime. If it's not present, you can open it by going to the tab **Tool** → **IP Catalog**.

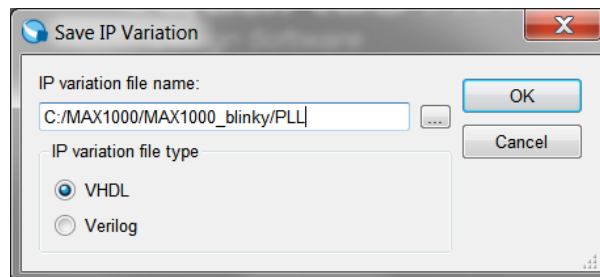


5.2.3.1 In the Search bar of the IP Catalog, type “pll” and select **ALTPLL** which stands for Altera Phase Locked Loop.



5.2.3.2 Click “Add”.

When the Save IP Variation window appears, enter the file name variation as PLL and select VHDL (Verilog can be used as well). Both Verilog and VHDL schematics will be created.



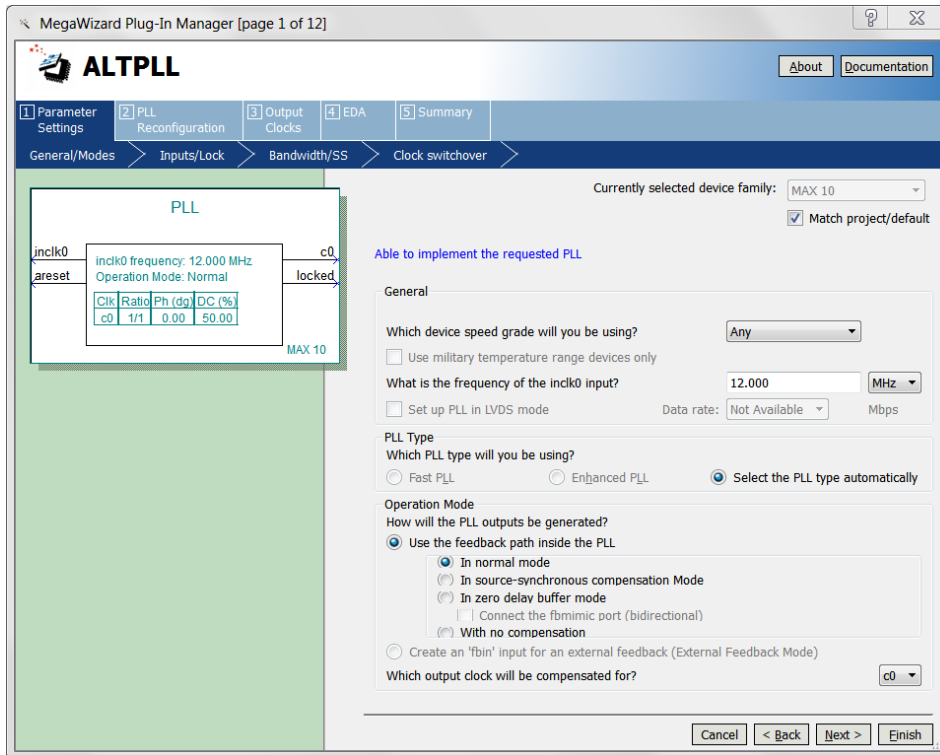
5.2.3.3 Click “OK”.

5.2.4 Create and Configure the PLL

The next step is to configure the PLL component that we just named.

5.2.4.1 Enter the PLL reference clock frequency to match the clock input on the MAX1000 Board. Since we have a 12 MHz coming into the FPGA, the inclk input will be 12 MHz.

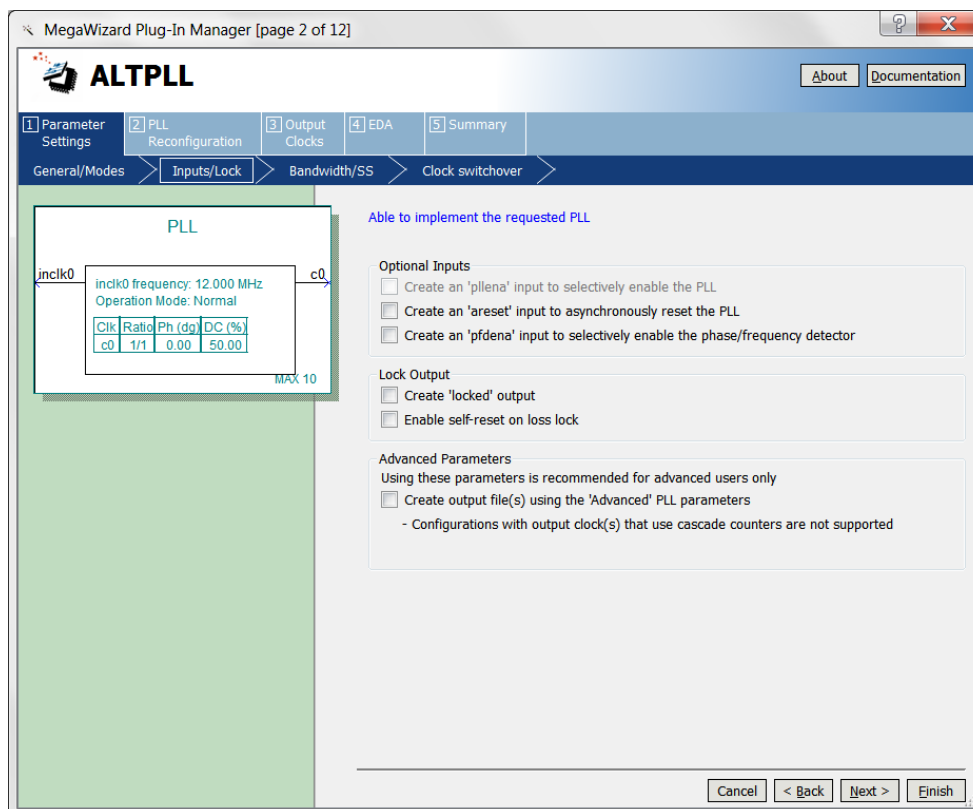
The setting should look like this:



5.2.4.2 Click “Next”.

5.2.4.3 Simplify the PLL, by disabling areset and locked outputs.

The setting should look like this:

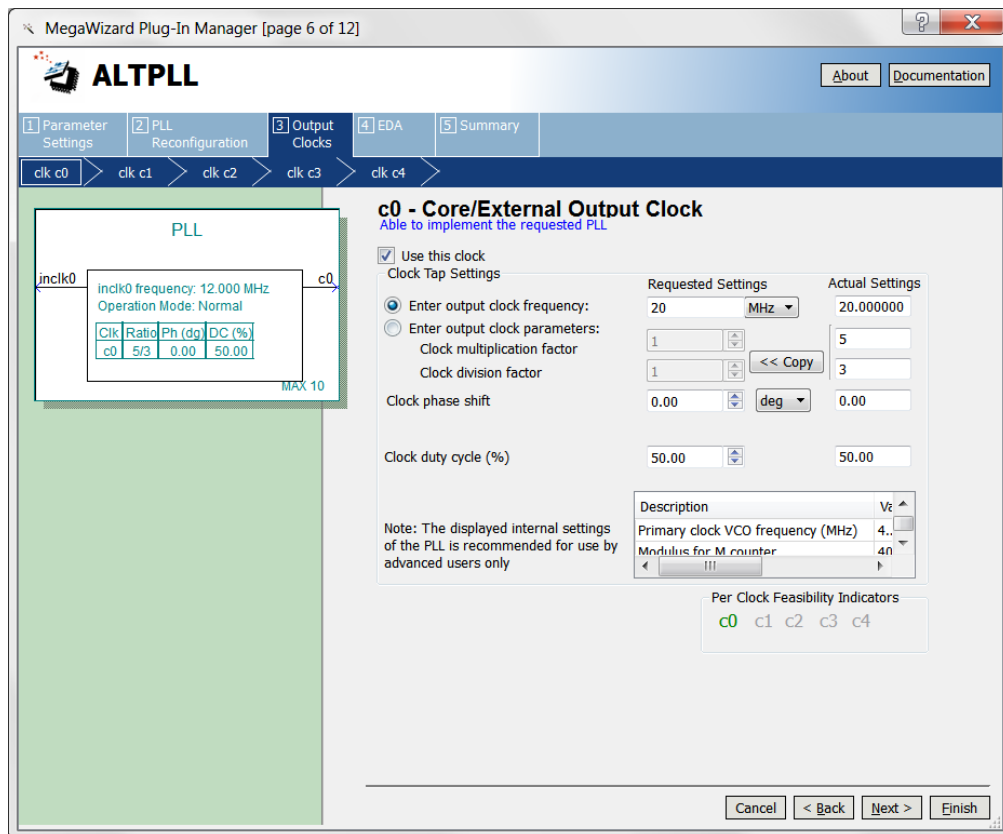


5.2.4.4 Click “Next”.

5.2.4.5 Continue to select Next to go through the various options (e.g. Pages 3 to Pages 5), but leaving the default options as they are. The page numbers can be seen on the top of the window.

5.2.4.6 On page 6, (c0-Core/External Output Clock) select “Enter output clock frequency” and set the requested setting to 20 MHz, leave the rest as default. For simplification, there is one input to the PLL (12 MHz), and one output of the PLL (20 MHz)

The setting should look like this:

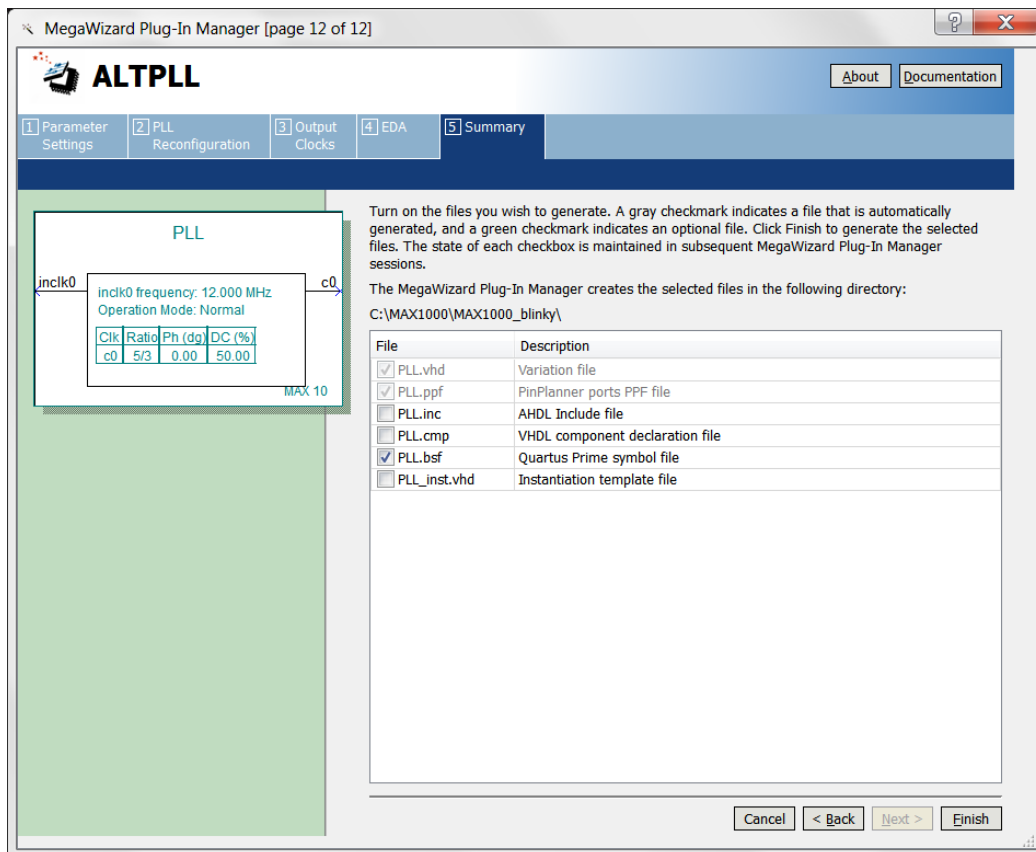


5.2.4.7 Click “Next”.

5.2.4.8 Click “Next” for pages 7 to 11, as default settings are to be used.

5.2.4.9 On page 12 there is a list of output files that will be generated. Since the design will be done in a schematic, you will need to select PLL.bsf checkbox. The .bsf file provides a symbol that can be used in the schematic design we will be creating later.

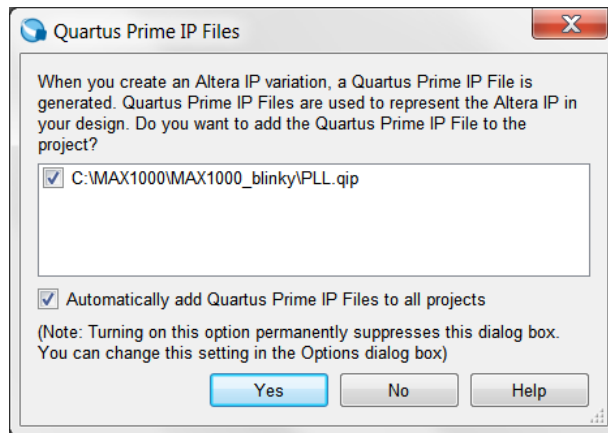
The setting should look like this:



5.2.4.10 Click “Finish”.

The PLL (1st component) will now be created.

5.2.4.11 If this is the first time that you are using this version of Quartus Prime, you might see a pop-up Window for Quartus Prime IP Files, that asks if the tool should add IP files automatically after generating them.



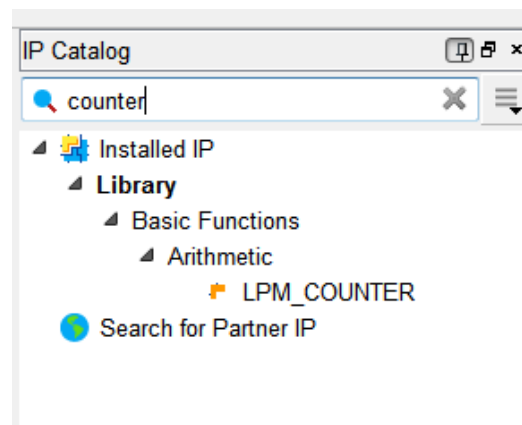
5.2.4.12 Select “Automatically add Quartus Prime IP Files to all projects.”

5.2.4.13 Click “Yes” to allow all of the IP to automatically be added to the project, and so that this message will not be seen for other designs.

5.2.5 Create and Configure the Counter

The next step is to create the counter which will drive the LEDs on the MAX1000 board.

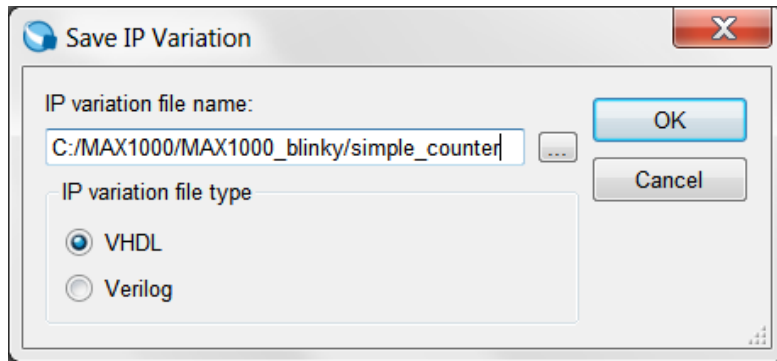
5.2.5.1 To create this counter, select the IP Catalog and expand the **Basics** → **Arithmetic** and select the LPM_COUNTER or type “counter” in the search field.



Note: LPM stands for Library of Parameterized Modules

5.2.5.2 Click “Add”.

5.2.5.3 When the Save IP Variation pop up appears, enter “simple_counter” and select VHDL as below:



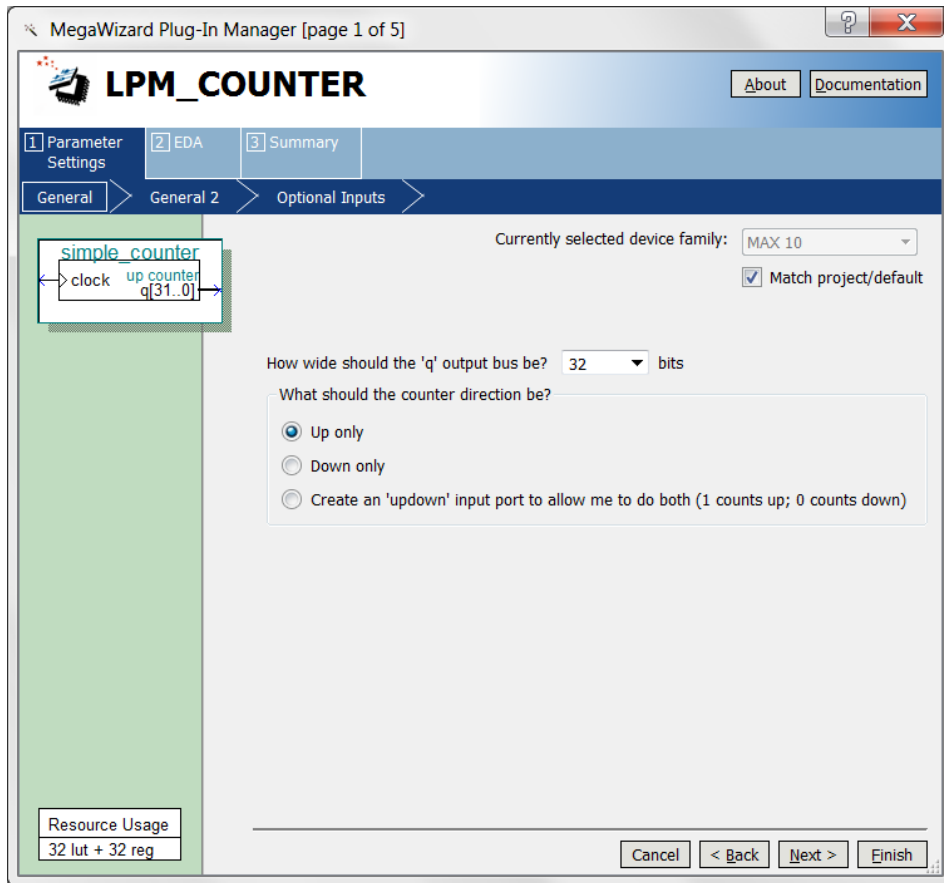
5.2.5.4 Click “OK”.

5.2.5.5 The next step is to increase the size of the counter to a number of bits large enough to divide down the clock so we can see the LEDs toggling.

5.2.5.6 Change this number to 32.

5.2.5.7 Let the counter to be Up only, so the LEDs will show the counters counting up.

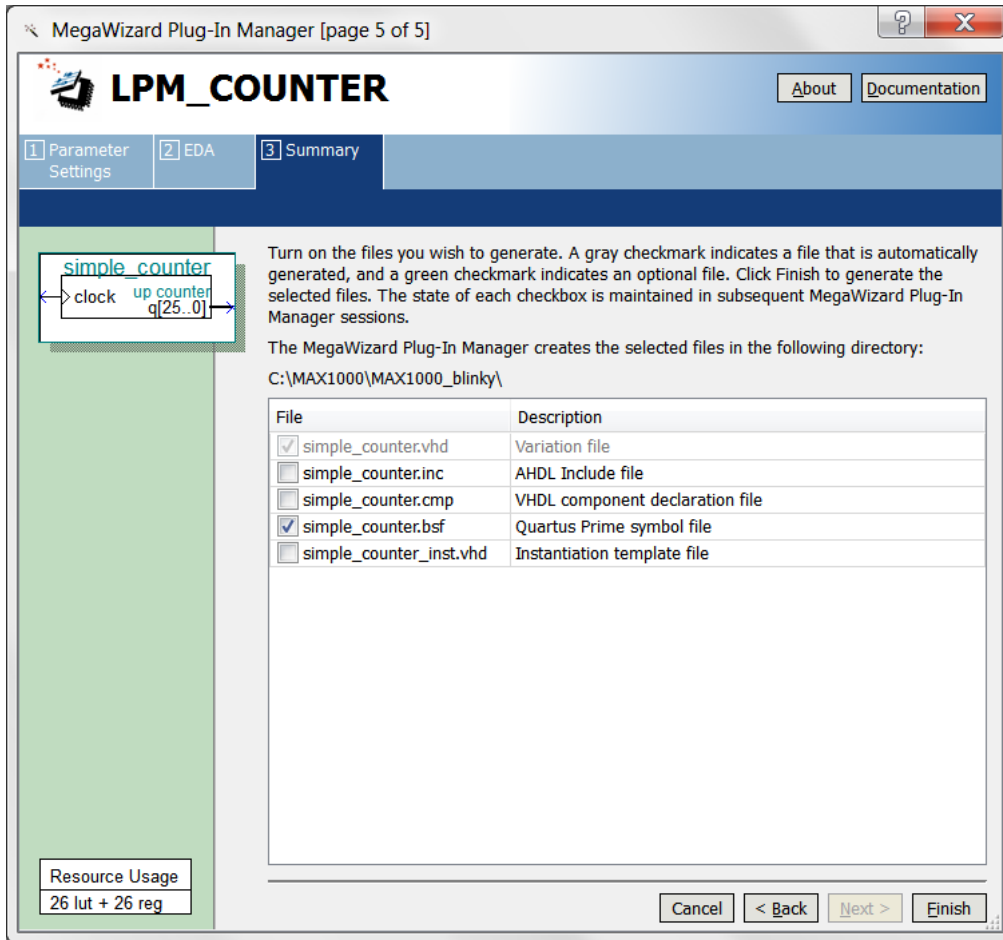
The setting should look like this:



5.2.5.8 Select “Next” until reaching Page 5.

Select simple_counter.bsfc checkbox to generate a symbol for our schematic design.

The screen should look like this now:



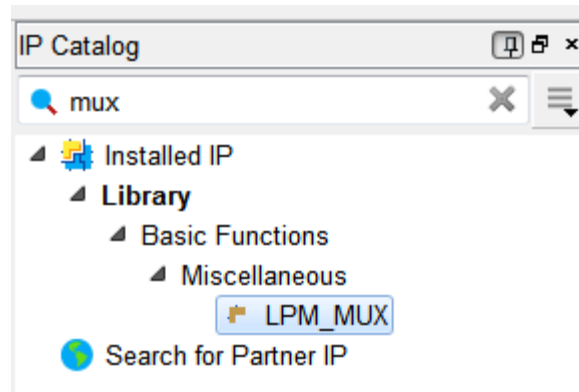
5.2.5.9 Click “Finish”.

The counter is now created.

5.2.6 Create and Configure the Multiplexer

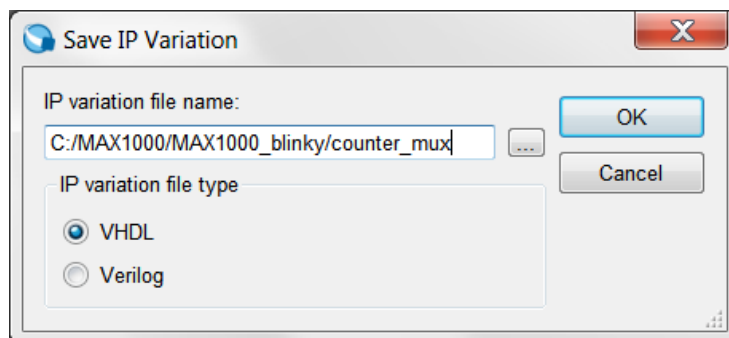
The next step is to create a mux component. This mux will be used along with a push button on the MAX1000 board to control the speed of the counter, where the counter outputs will be seen on the LEDs.

5.2.6.1 To create this mux, select IP Catalog and expand **Basic Functions** → **Miscellaneous** and select LPM_MUX or type mux in the search field.



5.2.6.2 Click “Add”.

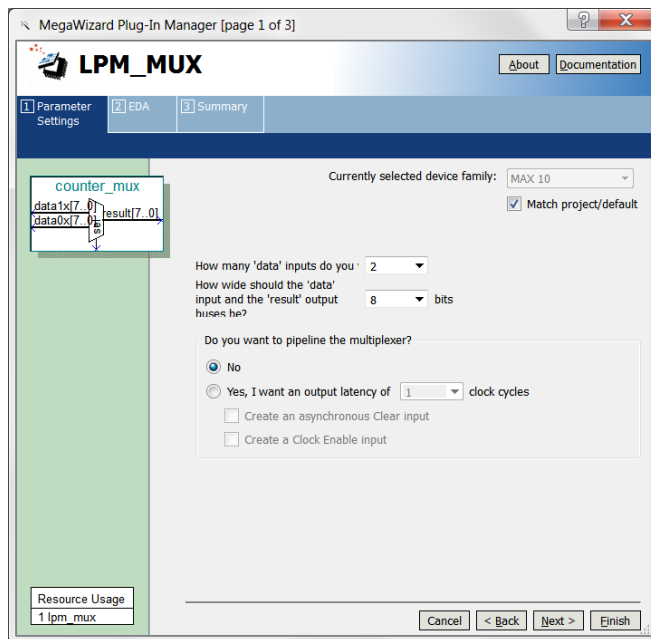
5.2.6.3 In the Save IP Variation, enter the name of the counter_mux and the file type to be VHDL.



5.2.6.4 Click “OK”.

5.2.6.5 Select 2 data inputs and the width of the input and output buses to be 8 bits. The reason for 8 bits is that there are 8 LEDs to be toggled (showing count values).

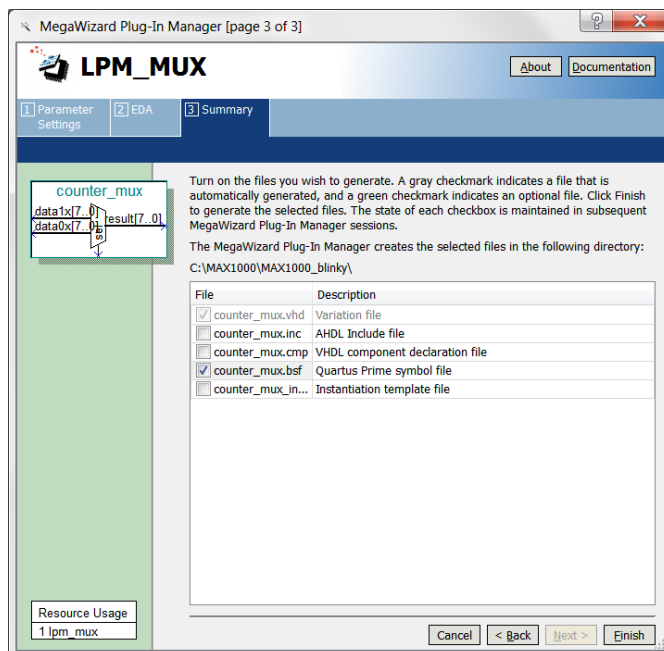
The screen should look like this now:



5.2.6.6 Click “Next” until Page 3.

5.2.6.7 Select counter_mux.bsf checkbox to generate a symbol for our schematic design.

The view of this now looks like:

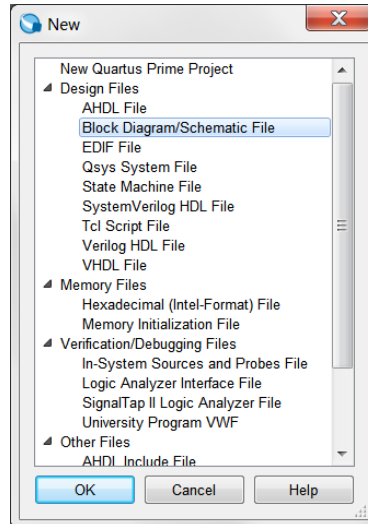


5.2.6.8 Click “Finish”.

5.2.7 Adding the Components to the Schematic

The next step would be to connect all three components together.

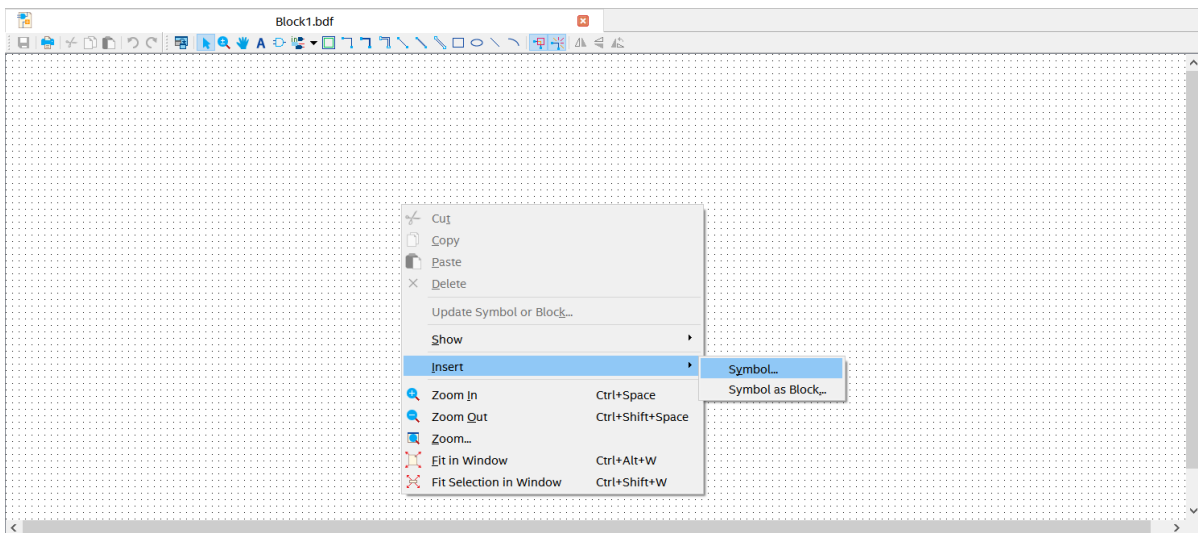
5.2.7.1 To do so, select File menu, then select New and select Block Diagram/Schematic File.



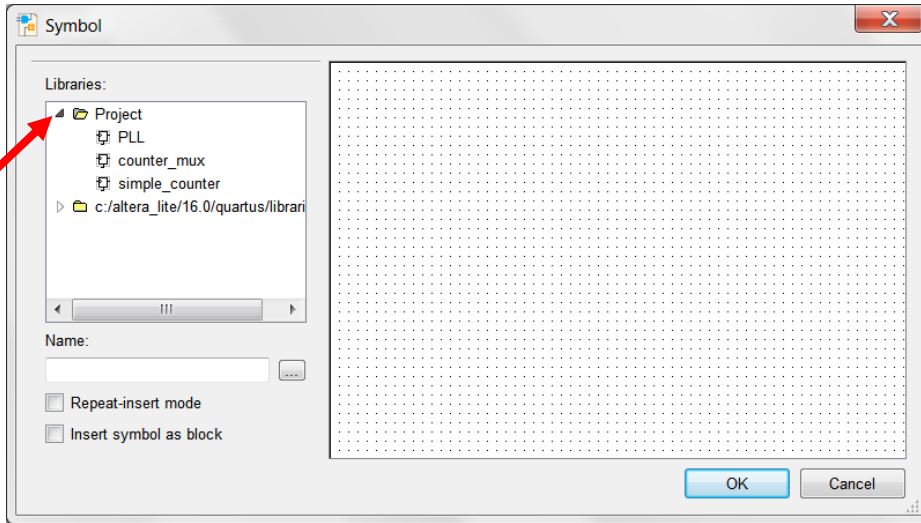
5.2.7.2 Click “OK”.

A new schematic will be created, where the components can be added.

5.2.7.3 Right click on the schematic page, and select **Insert** → **Symbol...** as seen below.



5.2.7.4 In the new window, expand “Project” and the three components that were created can now be seen.



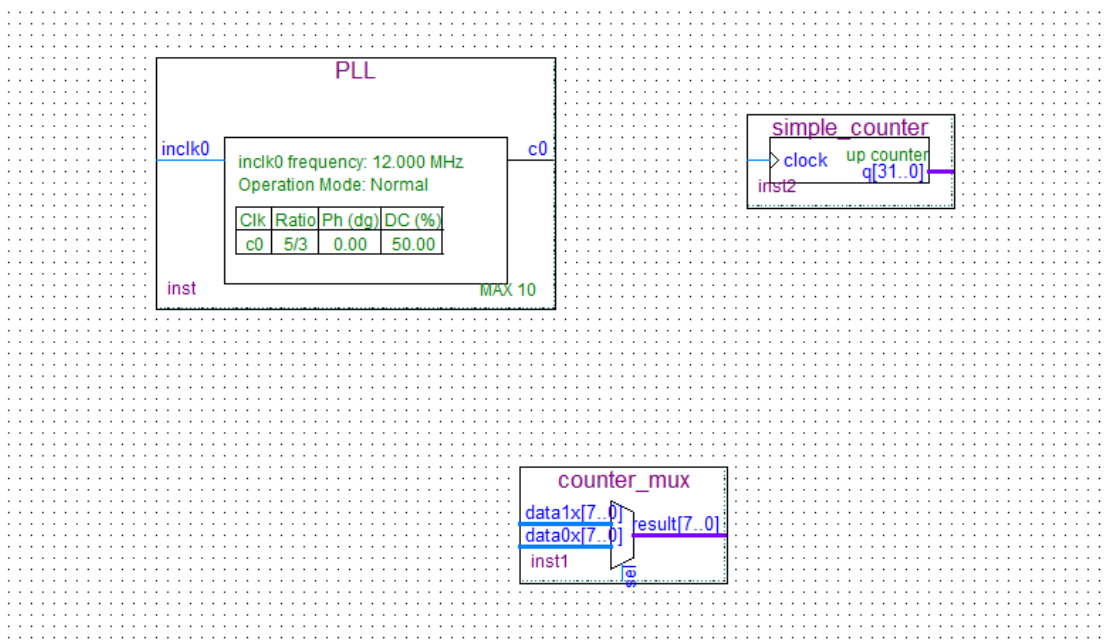
5.2.7.5 Select “PLL”.

5.2.7.6 Click “OK”.

The PLL component can be added now by left clicking on the schematic page.

5.2.7.7 Just like in the steps from 5.7.2.3 to 5.2.7.6, do the same for counter_mux and simple_counter to add them to the schematic page. The order of adding the components does not matter, as the connections between them will happen in the following steps.

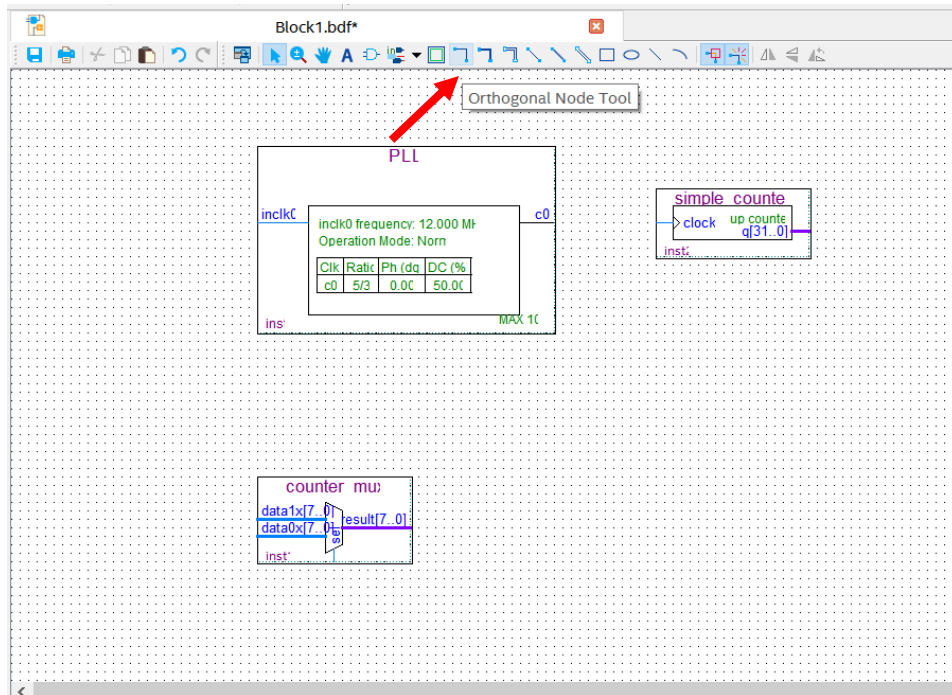
5.2.7.8 After adding three components, your schematic should look similar to the following. To place them similarly, simply drag the components to the appropriate locations.



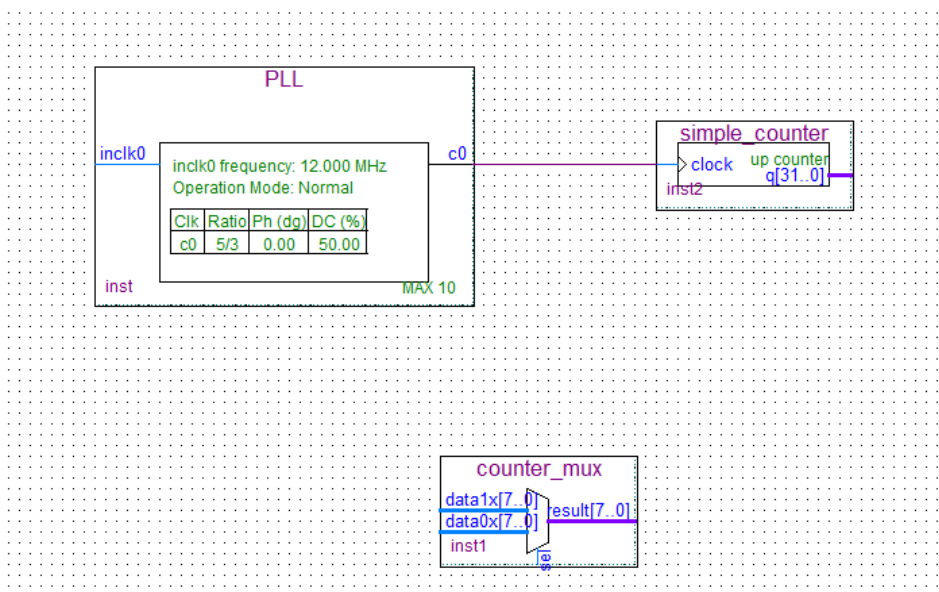
5.2.8 Connecting the Components

Next step is to make the proper connections between the three components we just added to the schematic.

5.2.8.1 Select the “Node Tool”.

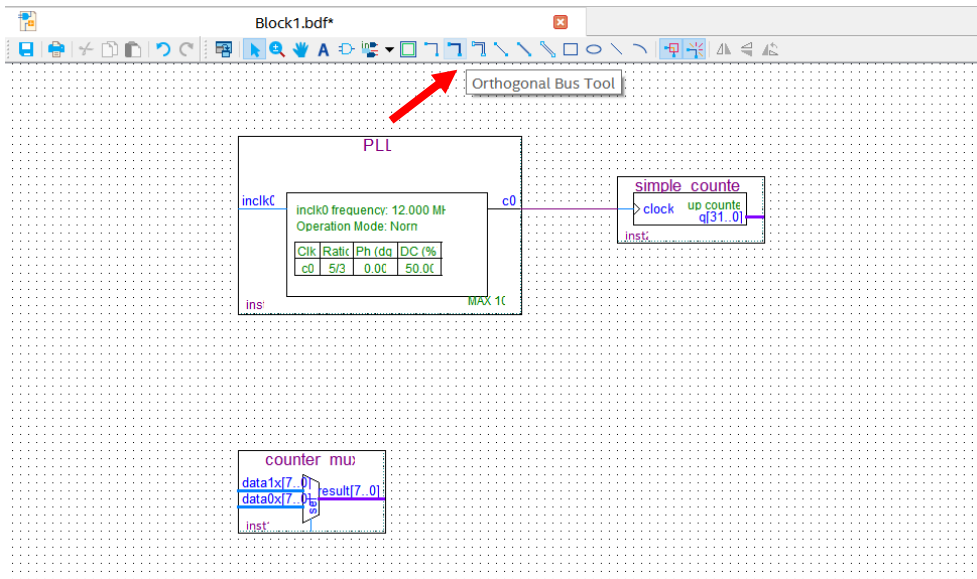


5.2.8.2 Connect the c0 of the PLL to the simple_counter as shown below:

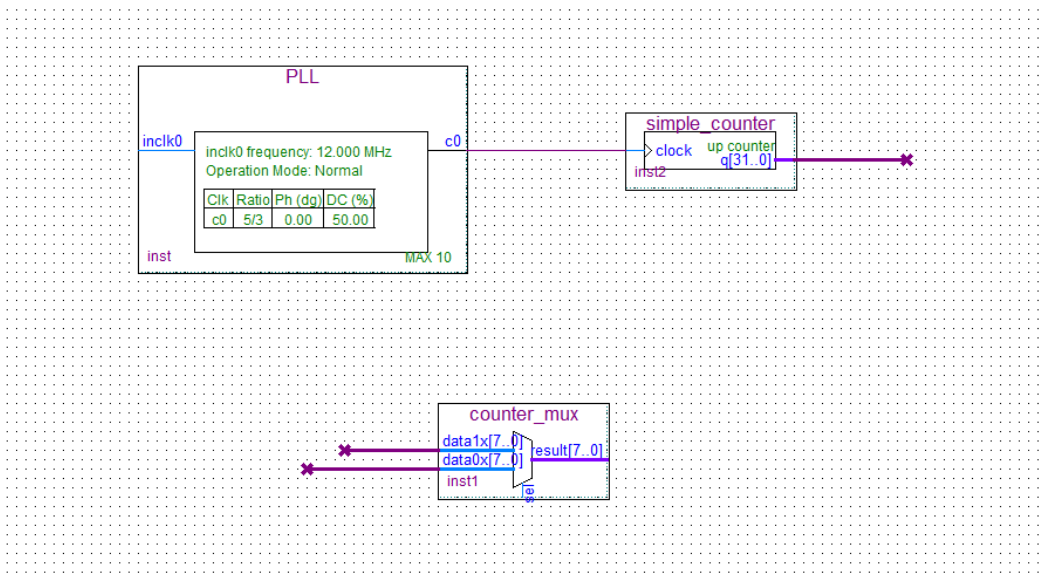


This will mean that a single signal (c0) is connected to the simple_counter (clock).

5.2.8.3 Select the “Bus Tool”.



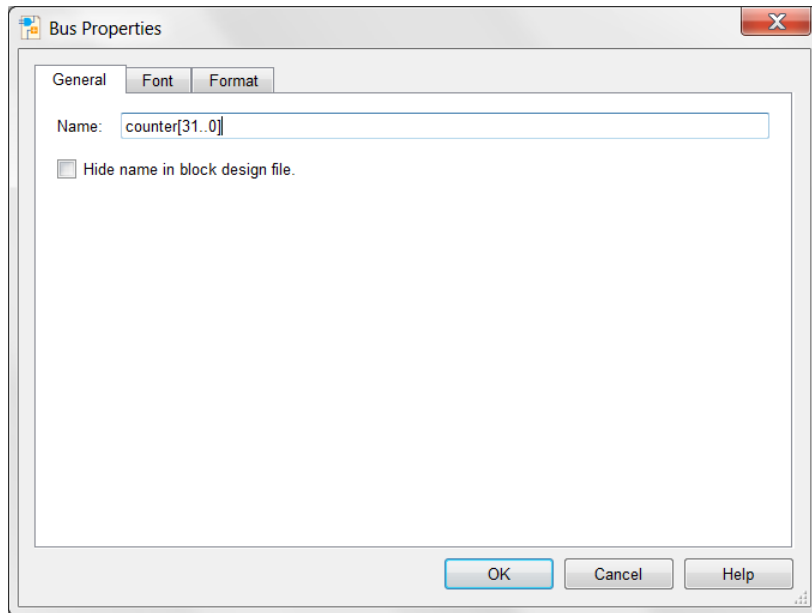
5.2.8.4 Using the bus tool create a connection coming out of the simple_counter and one connection for each of the inputs of the counter_mux as show below.



5.2.8.5 Right click on the output bus of the simple counter that you just created and select “Properties”.

Set the name of the bus to: **counter[31..0]**

The view of the “Bus Properties” should look like this:

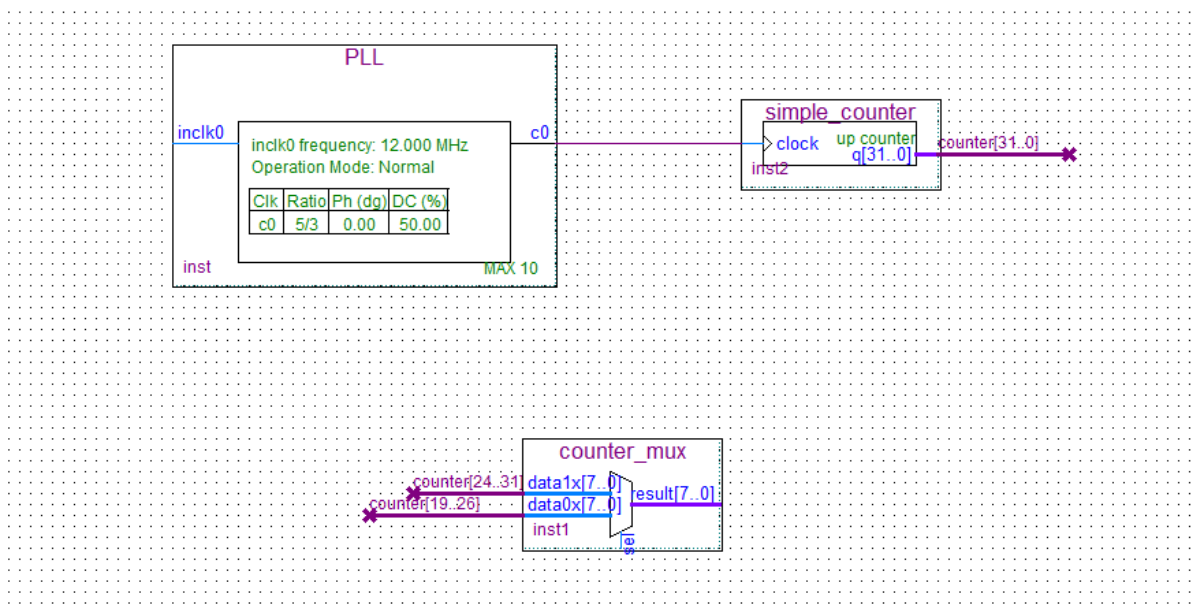


5.2.8.6 Do the same for input buses of the mux:

Name the top bus input: **data1x[7..0] → counter[24..31]**

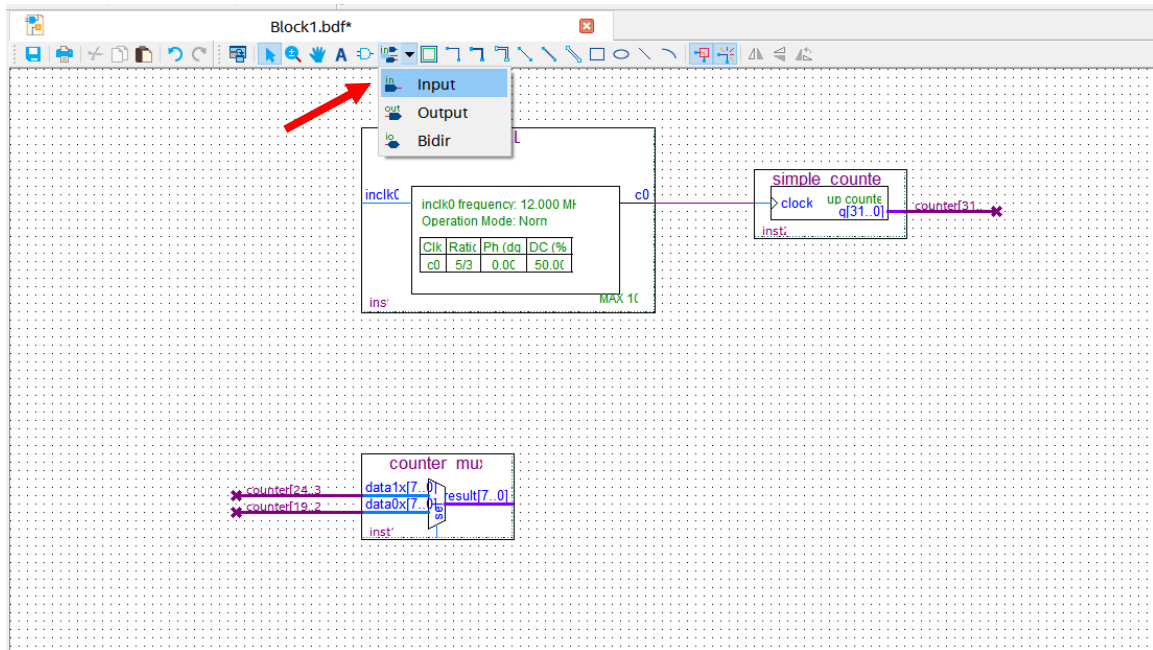
Name the bottom bus input: **data0x[7..0] → counter[19..26]**

Schematic should look like this:



5.2.8.7 Add inputs to the schematic

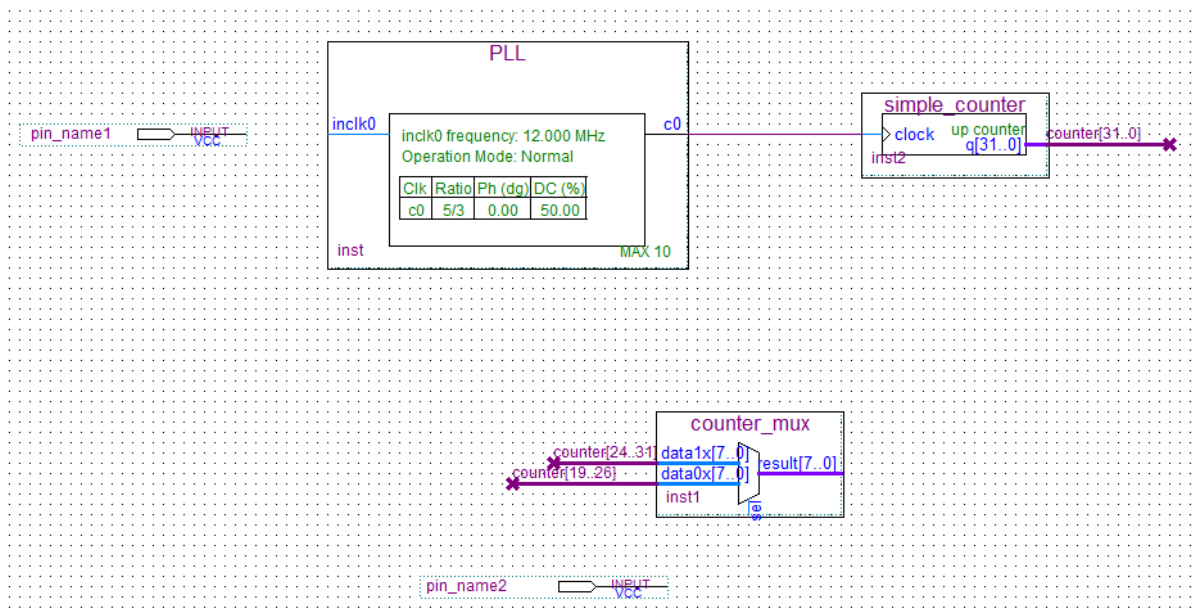
Click on the “Pin Tool” as show below and select “Input”.



5.2.8.8 Add one input pin for inclk0 of the PLL.

Add one input pin for sel of counter_mux.

Your schematic should look like this:



5.2.8.9 Rename the pin_name1 to **CLK12M** by double clicking its current name. This is going to be the clock signal coming into the FPGA.

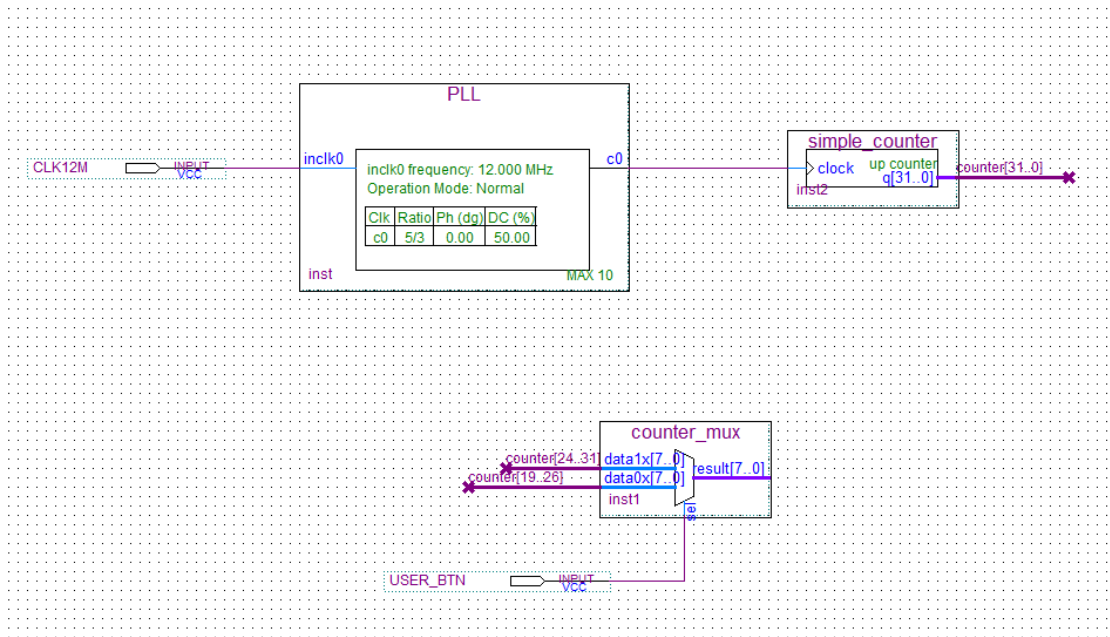
5.2.8.10 Rename the pin_name2 to **USER_BTN** by double clicking its current name. This is going to be the user button of the MAX1000 board to select the mux.

5.2.8.11 Using the **“Node Tool”** connect:

CLK12M → **inclk0** (of the PLL component)

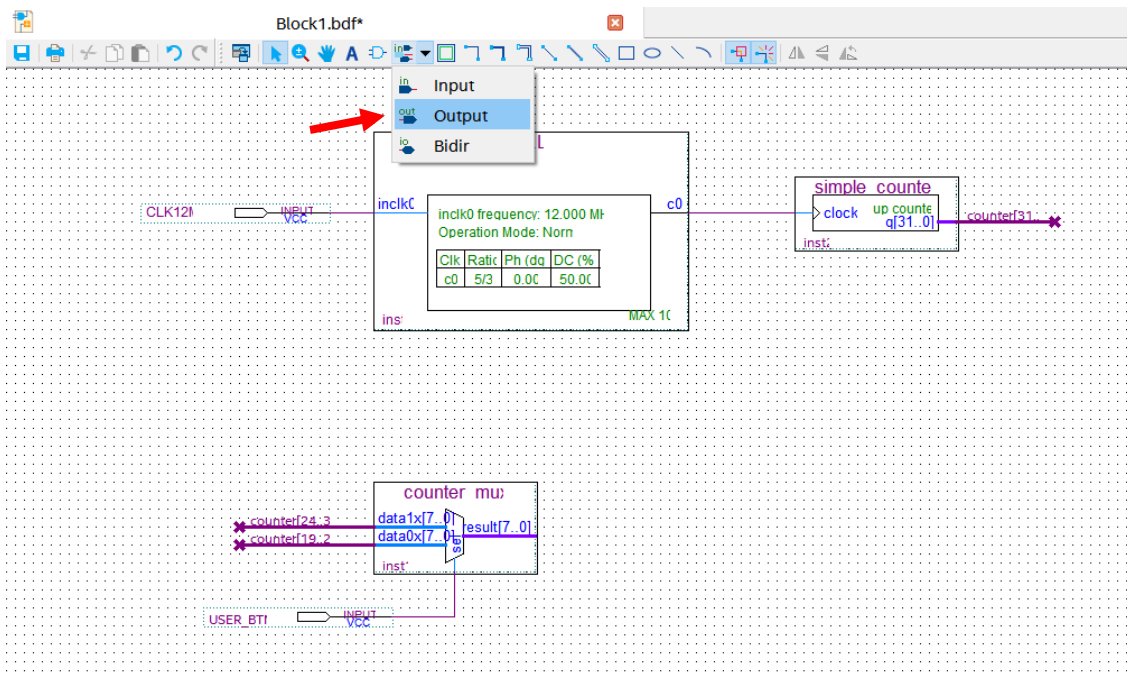
USER_BTN → **sel** (of the counter_mux component)

Your schematic should look like this now:



5.2.8.12 Add the outputs to the schematic

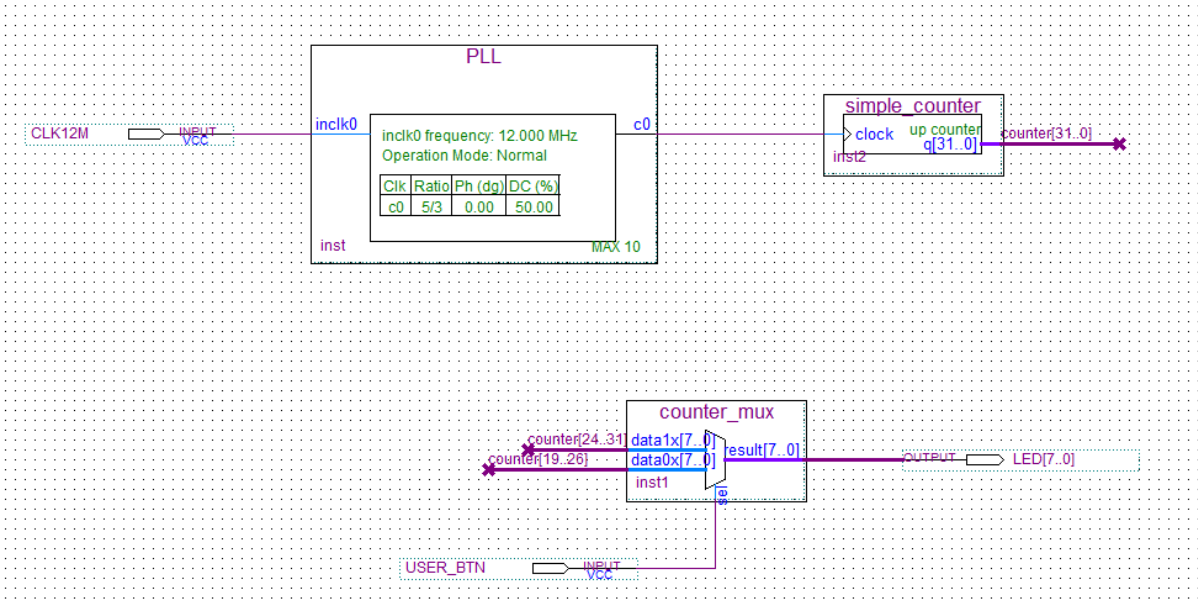
Click on the **“Pin Tool”** as before, but this time select **“Output”**.



- 5.2.8.13 Add one output pin for the LEDs.
- 5.2.8.14 Rename the pin to **LED[7..0]**.
- 5.2.8.15 Using the “**Bus Tool**”, make the connection between counter_mux component and output pin:

result[7..0] → LED[7..0]

The final schematic should look like the following:



Looking at the schematic, even though the buses are not connected together by wires, the names of counter tell Quartus Prime to connect the signals together. Overall, the user button will toggle between displaying higher 8 bits of the counter and 8 lower bits of the counter. The signals of the counter that are not connected will not be used by Quartus Prime.

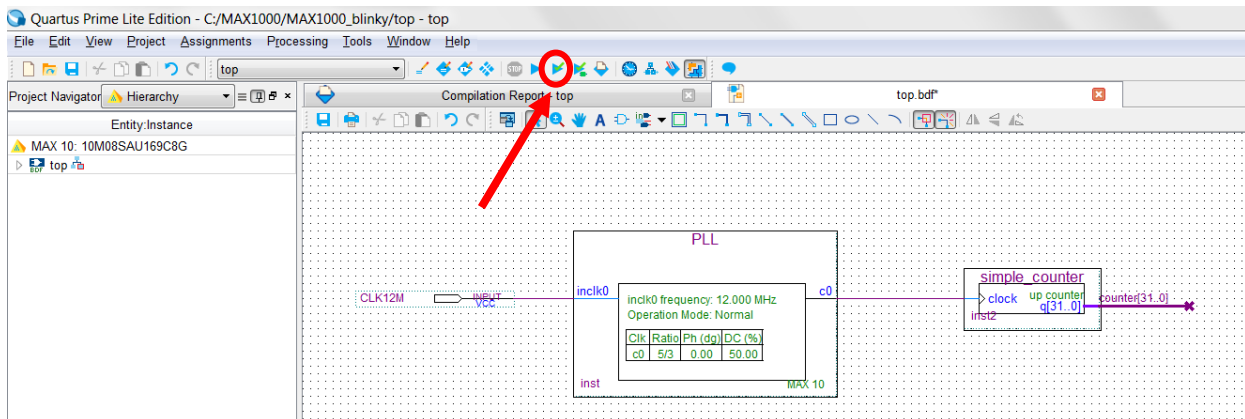
- 5.2.8.16 Save your design
- Open the File Menu and select “**Save**”. Save it as **top.bdf**

5.2.9 Analysis and Synthesis

The next step is to run Analysis and Synthesis to ensure that there is no errors in the design.

To run Analysis and synthesis open **Processing** → **Start** → **Analysis and Synthesis**

or from clicking the button on the top toolbar as seen below:

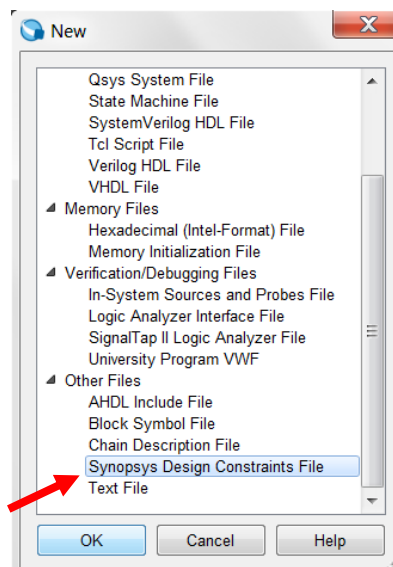


There should be no errors. If there are errors, they should be fixed before continuing and Analysis and Synthesis run again.

5.2.10 Adding Timing Constraints

Timing Constraints tell the Quartus what are the timing requirements for this design. Timing Constraints are required in every CPLD/FPGA design.

To add the timing constraints, select **File** → **New** and under the “Other File” section, select “Synopsys Design Constraints File” and select “OK”.





Type or copy the following lines into this new file:

```
#create input clock which is 12MHz
create_clock -name CLK12M -period 83.333 [get_ports {CLK12M}]

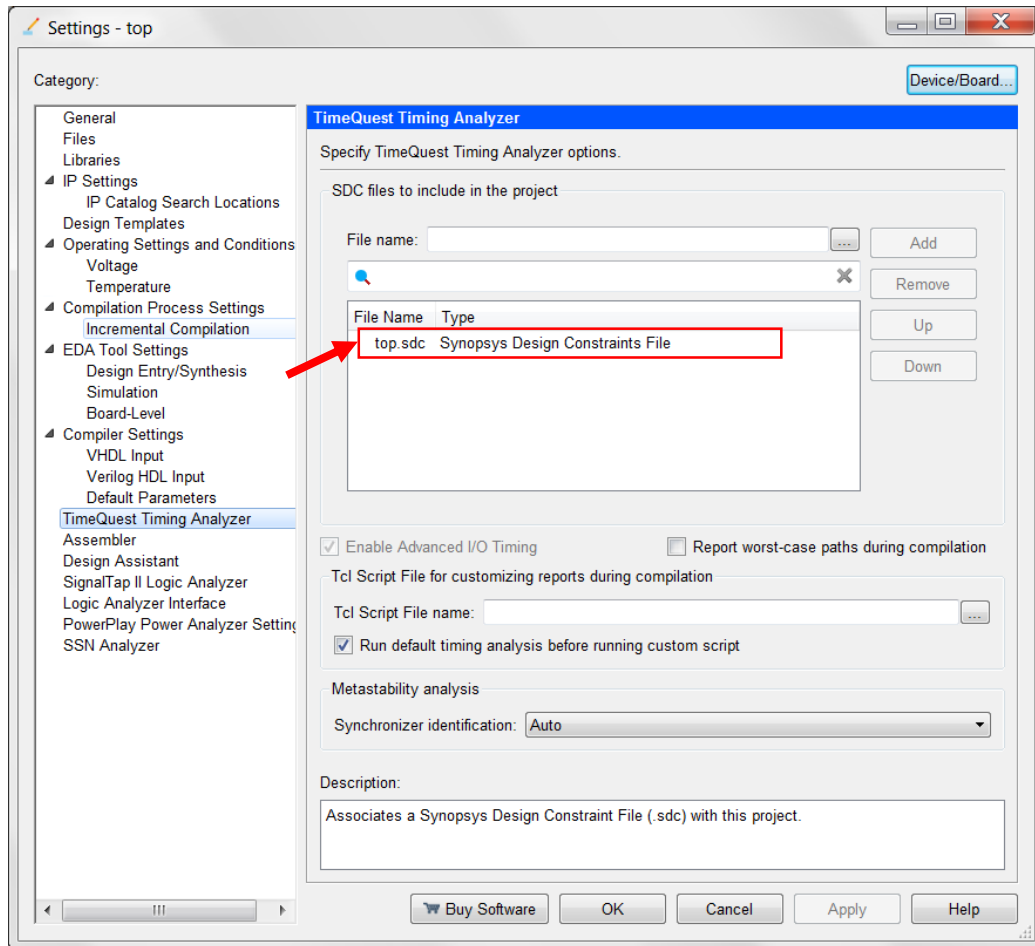
#derive PLL clocks
derive_pll_clocks

#derive clock uncertainty
derive_clock_uncertainty

#set false path
set_false_path -from [get_ports {USER_BTN}]
set_false_path -from * -to [get_ports {LED*}]
```

- 5.2.10.1 The first line “create_clock” tells Quartus Prime that the clock, CLK12M is 83.333 ns (12 MHz). It also assigns the CLK12M to a pin (port) in the .sdc format.
- 5.2.10.2 The second line “derive_pll_clocks” tells the software to look if there are any PLLs, and if so, automatically derive the clock multiplication/division of the outputs of the PLL even if they are used internally within the CPLD/FPGA.
- 5.2.10.3 The third line “derive_clock_uncertainty” tells the software to automatically determine the internal clock uncertainty. No clock is ideal, and thus there will be some internal jitter within the FPGA associated with it.
- 5.2.10.4 The fourth and fifth line “set_false_path” tells the software to not do any timing optimization to the stated paths/pins. The I/Os of this design are trivial, so they can be ignored in the Timing Analysis.
- 5.2.10.5 Use **File** → **Save** to save it as **top.sdc**.

5.2.10.6 Ensure that the file is added to the Project: **Assignments** → **Settings** and select “Timequest Timing Analyzer”. The top.sdc should have been already added by default. If it is not, it will need to be added manually.



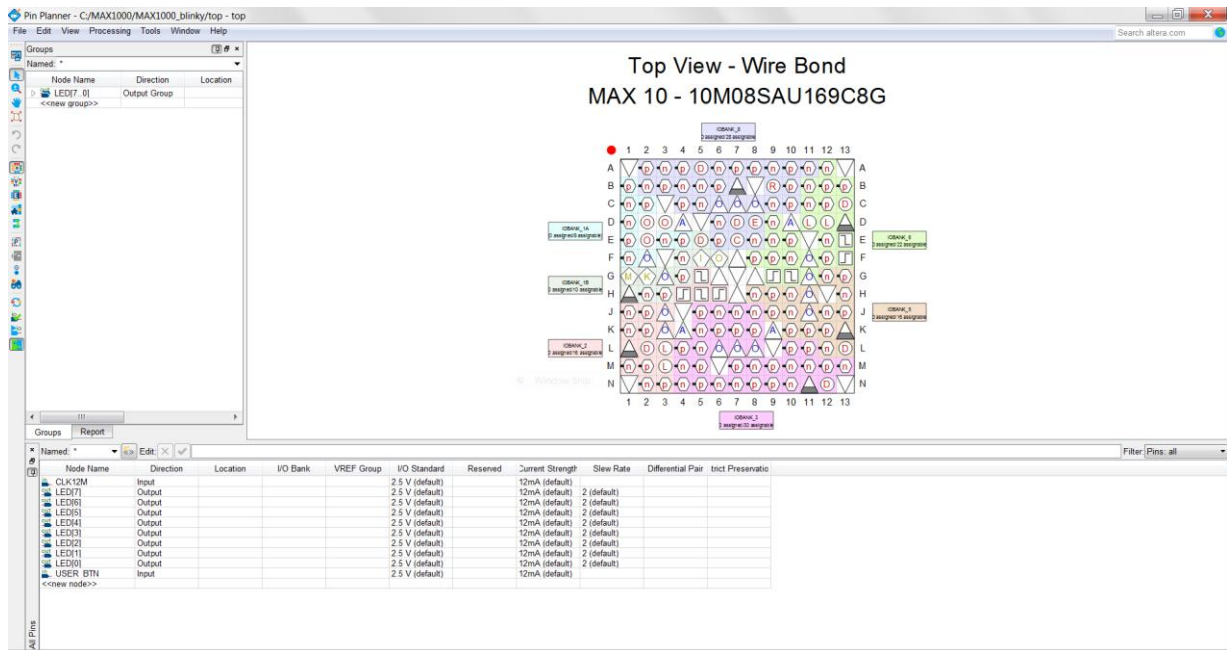
5.2.11 Pinning Assignments

Before the design can be downloaded to the FPGA, pin assignments that match the hardware on the board are needed. There are different ways to do this such as the Pin Planner, Assignment Editor, and text files.

The following steps will show one of these ways, the Pin Planner. Since there are only 10 pins that need to be assigned, the Pin Planner can be used. If many pins are needed, other ways can be used such as the Quartus Assignment Editor, or by importing constraints from a text file or spreadsheet.

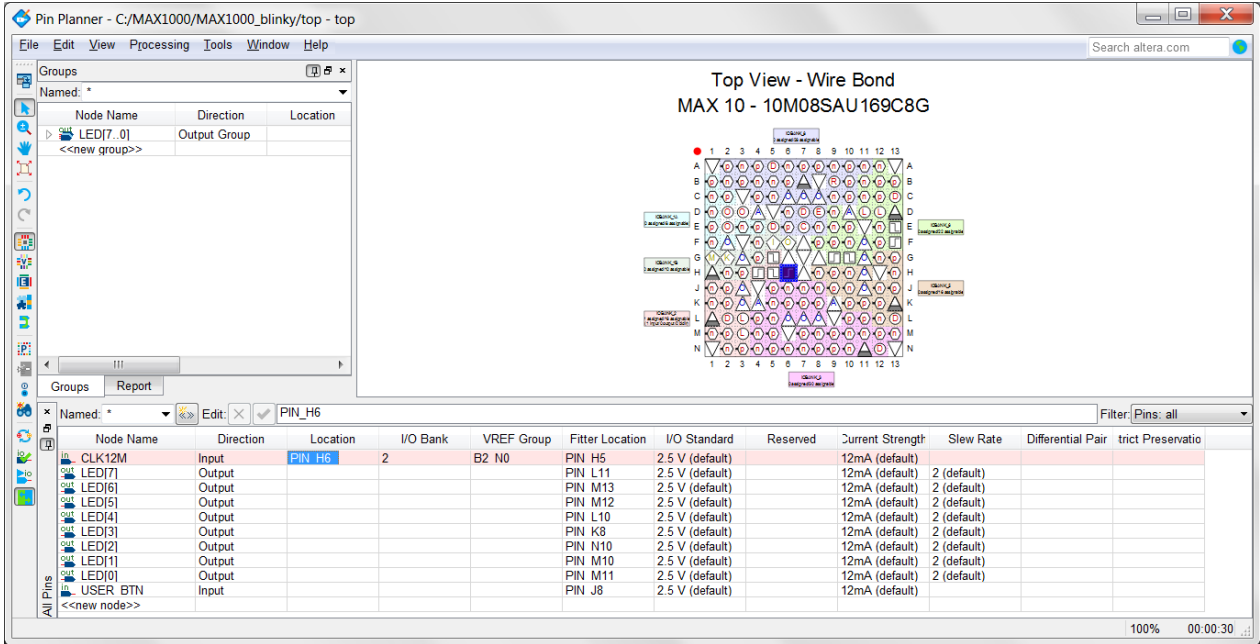
5.2.11.1 Open the Pin Planner: **Assignments** → **Pin Planner**.

A new window will open as seen below:



5.2.11.2 To make pin assignments, select the CLK12M (node name) on the bottom portion and drag and drop it to pin H6 of the Top View of the FPGA or alternatively set the Location field of the CLK12M to **PIN_H6**.

The window should now look like this:

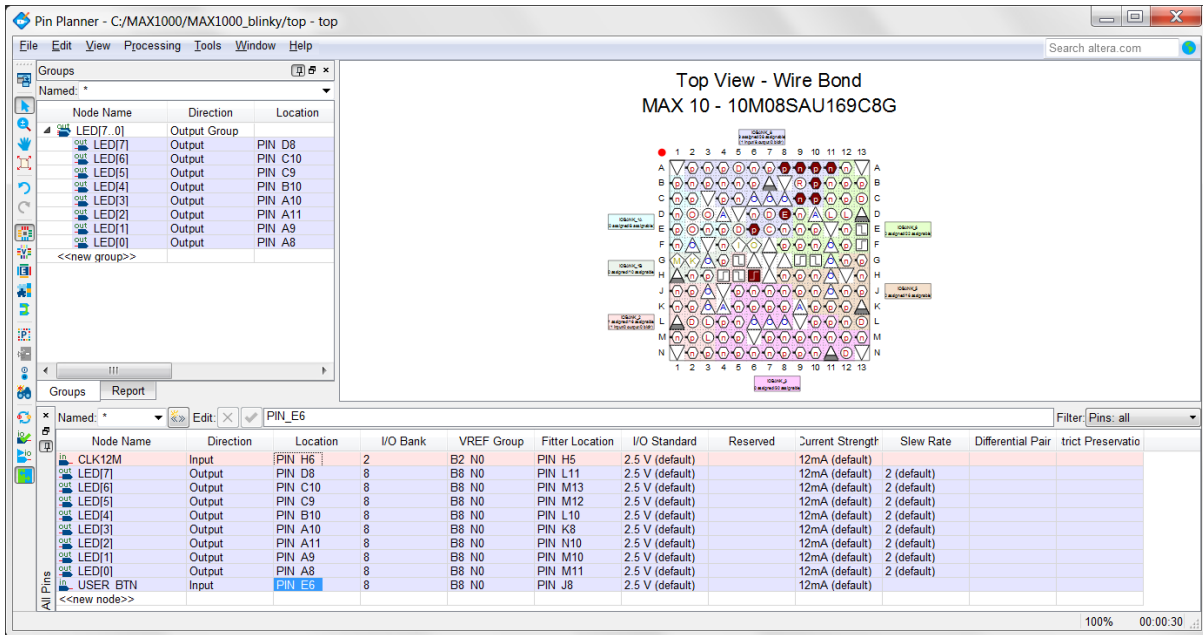


Note that the Location of the CLK12M is now set to Location PIN_H6 (as seen in blue colour in the top view of the FPGA).

5.2.11.3 The other pins need to be assigned as well. Just like previously set all the pins to their appropriate locations using the table below, by either drag and drop or writing manually the location.

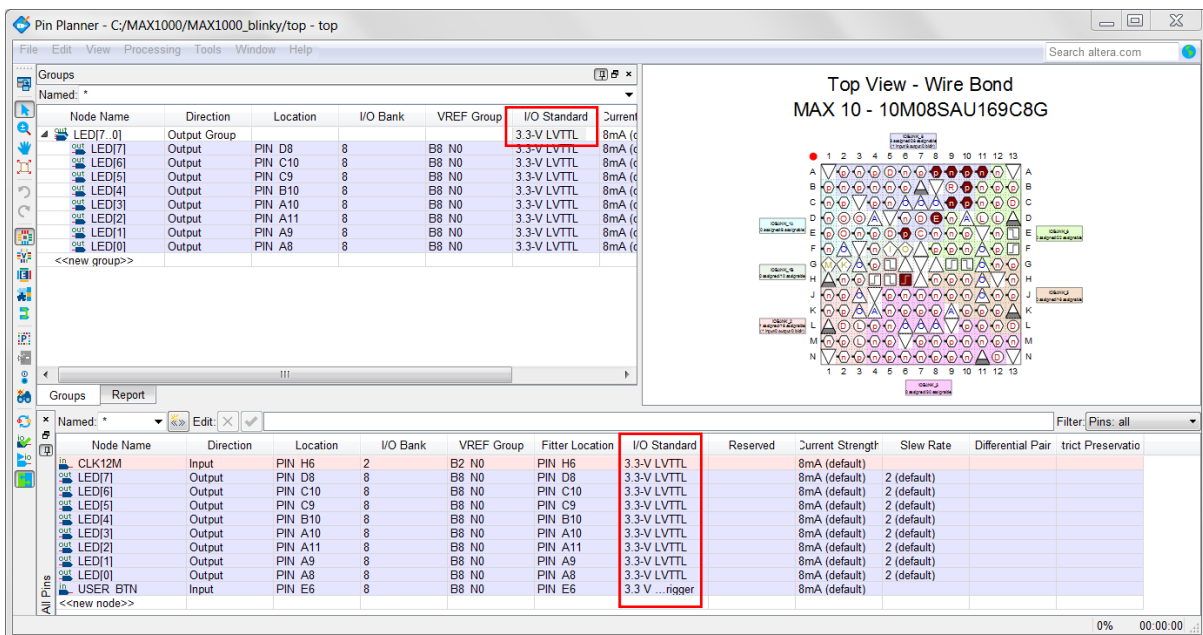
Node Name	Pin Location
LED[7]	PIN_D8
LED[6]	PIN_C10
LED[5]	PIN_C9
LED[4]	PIN_B10
LED[3]	PIN_A10
LED[2]	PIN_A11
LED[1]	PIN_A9
LED[0]	PIN_A8
USER_BTN	PIN_E6

5.2.11.4 Now the Pin Planner should look like this after assigning all the pin locations.



The specific pins are now selected, but the I/O standards now need to be set as well. The button, LEDs, and clock pins are the same I/O standard for MAX1000 since all banks and peripherals are powered by 3.3V. The USER_BTN is a 3.3 V Schmitt Trigger, the LEDs and clock pins are 3.3-V LVTTTL. These I/O standards can be set in the Pin Planner, by selecting the I/O Standard. Select the I/O standard either from the “All Pins” tab or the “Groups” tab, and change the 2.5V (default) to the specific I/O standard.

5.2.11.5 The Pin Planner should now look like this:

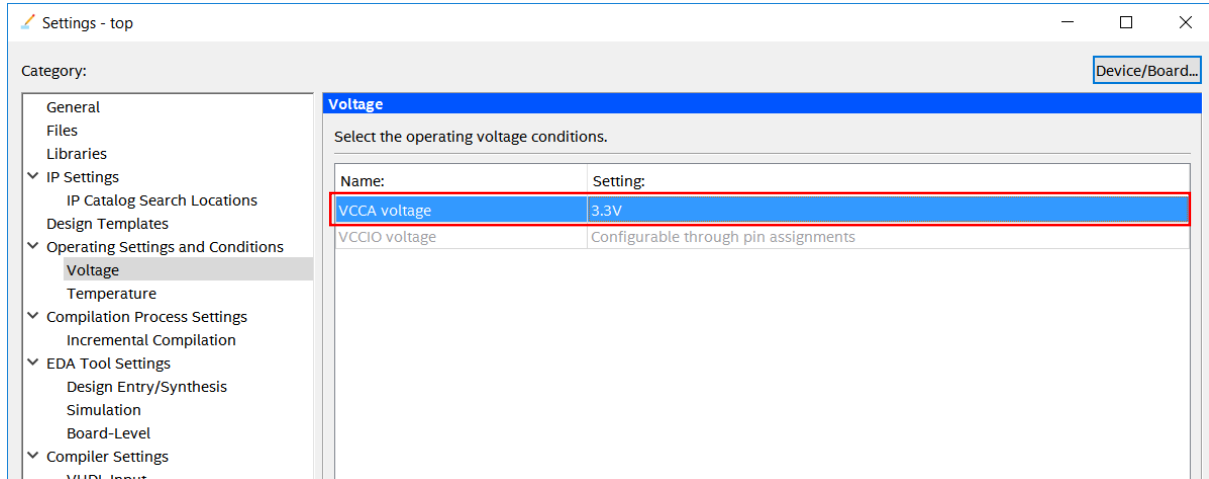


5.2.11.6 Close the Pin Planner. The settings are automatically saved.

5.2.12 Compiling the Design

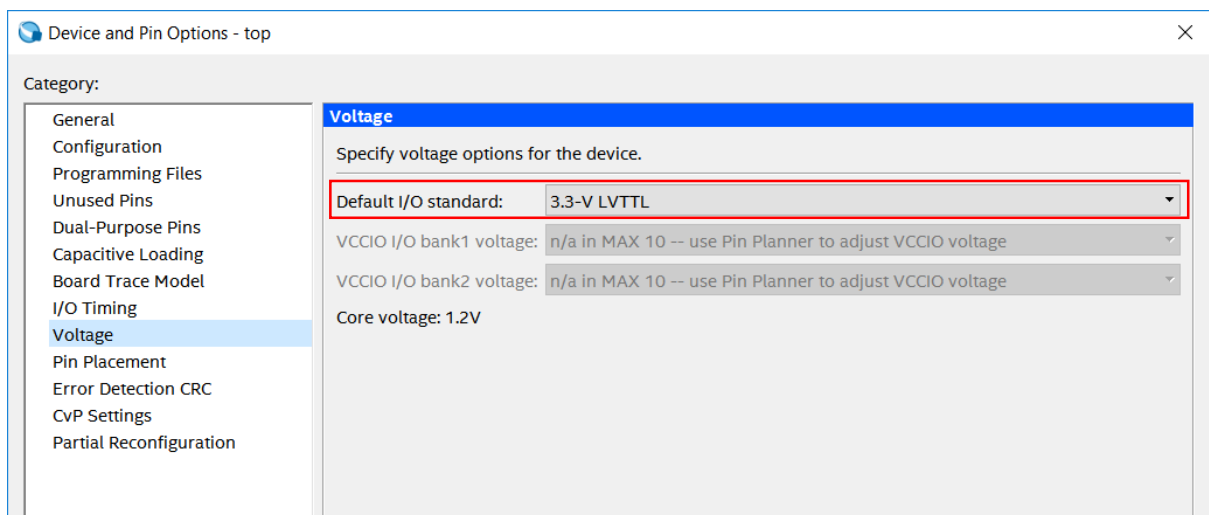
5.2.12.1 First, make sure that the operating conditions of the device are set correctly.

Open **Settings** → **Operating Settings and Conditions** → **Voltage** and set VCCA voltage to “**3.3V**” and press “Apply” and “OK”.



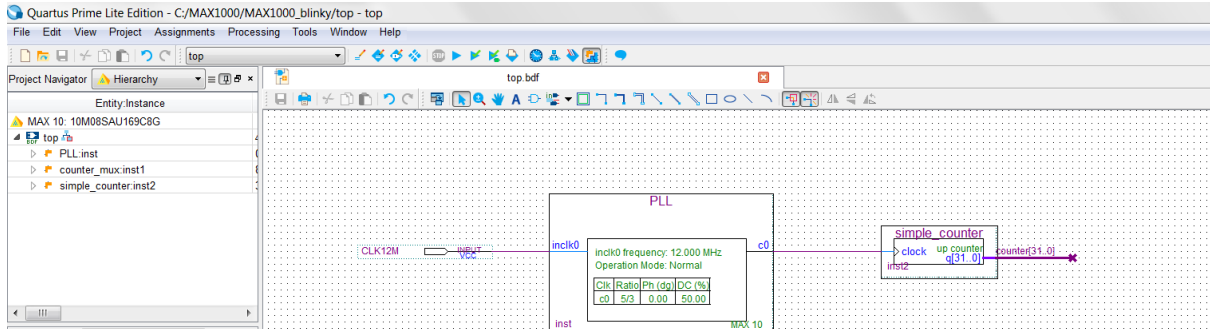
5.2.12.2 You can also set the default I/O Standard which can eliminate some design warning and save you time from setting the standard for some pins manually.

Open **Assignments** → **Device** → **Device and Pin Options** → **Voltage** and set Default I/O Standard to “**3.3-V LVTTTL**” and press “OK” to all the windows.



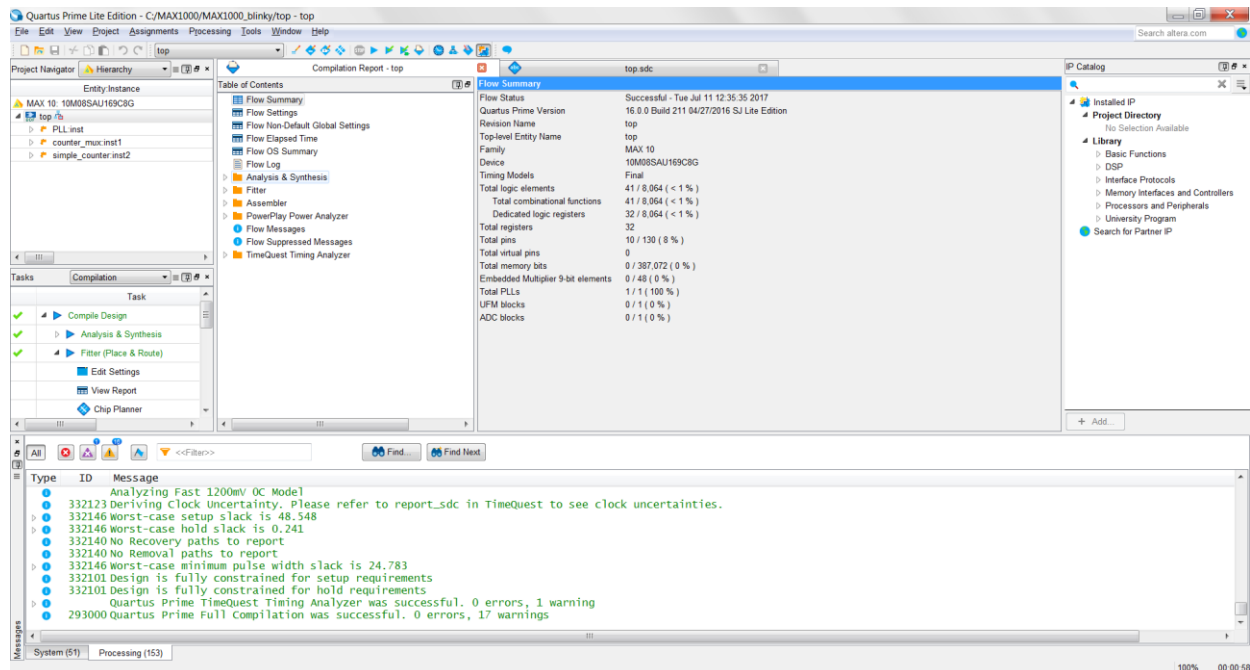
5.2.12.3 The next step is to compile and complete the design. This step will verify that there are no errors, create internal databases, and create programming files that will be used in the next steps.

To compile the design, select **Processing** → **Start Compilation** or through the button as show below:



If there are errors, they will need to be resolved and re-compiled before the design can be programmed to the board. When Compiling finishes and there are no errors, there will be a message at the bottom of the window that states: Full Compilation was successful and a 100% indication along with the compile time in the right bottom corner.

5.2.13 Reading the Compilation Report



After successfully compiling the design, a Compilation Report should appear as shown above:

This report is very useful with a lot of information about the design. Last message state that the design was fully constrained, Timing Analysis and compilation successful, but there is more to it:

- In the Flow Summary, it can be seen how many logic elements the whole design took, along with total PLLs, registers, pins, etc.
- In Analysis and Synthesis, more detailed information about the resources used can be seen in Resource Usage Summary, as well how many LEs were used for each component in Resource Utilization by Entity.
- In the Fitter, more detailed information about the pins and their banks can be seen. For example if we used the RESET_BTN (PIN_E7), we could have issues since it is configured as a secondary function, which can be seen in **Fitter** → **Resource Section** → **Dual Purpose** and **Dedicated Pins**, which would require to disable in the settings.
- TimeQuest Timing Analyzer shows various timing information concerning the design, as well as if the design has met the timing requirements. In this case timing requirements were met, but in other cases that requirements might not be met, could be solved by going over the information provided in the reports inside this folder. Most notable reports in this folder are: the maximum frequency the design can achieve, setup and hold slack, unconstrained paths in case they were missed, etc.

Chapter 6 - Configuring the MAX1000

After successfully compiling your project, there should new files generated. In case of MAX10 devices, two type of files are generated due to the architecture of the device, .sof and .pof. In other device architectures, only the .sof file is generated automatically.

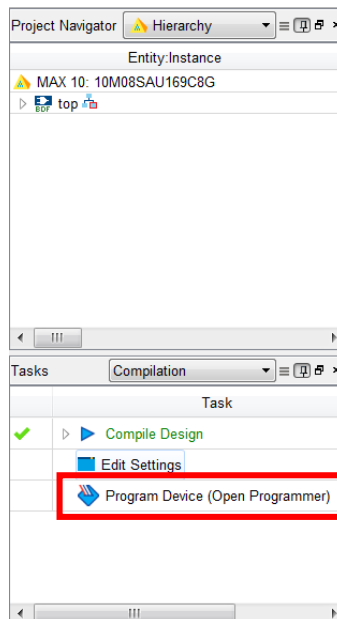
Note: *Difference between .pof and .sof files*

sof: is for the direct volatile configuration, which programs the FPGA fabric directly (SRAM cells). Configuration is lost on power off.

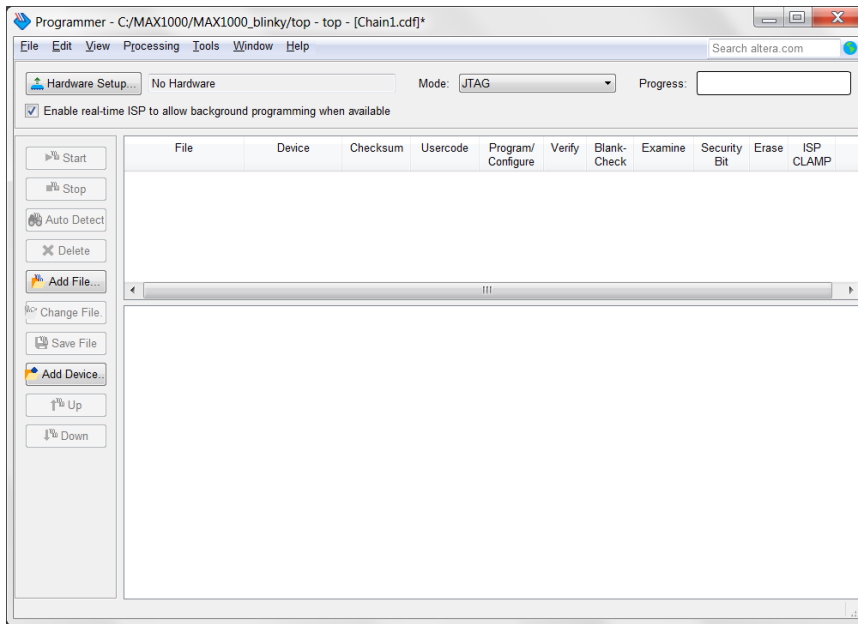
pof: is for non-volatile programming, which program the either an external configuration devices or in MAX10 case, itself as it is a non-volatile FPGA.

6.1 Configure the FPGA in JTAG Mode

6.1.1 Open the Quartus Prime Programmer from **Tools** → **Programmer** or double-click on Program Device (Open Programmer) from the Tasks pane. Since the MAX1000 is not connected yet, the Programmer should show a blank configuration.



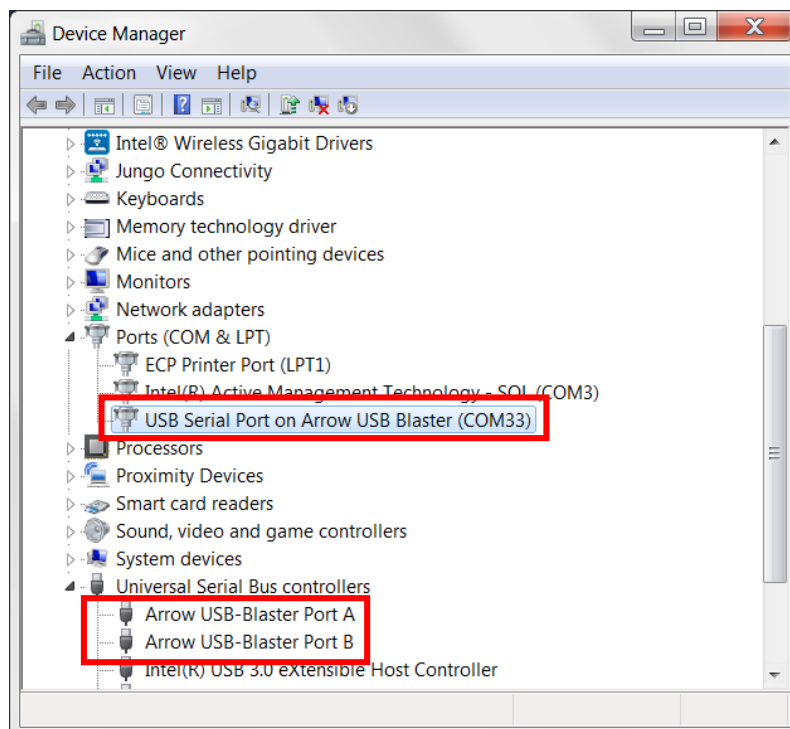
6.1.2 After opening the program this should be the view of the new window:



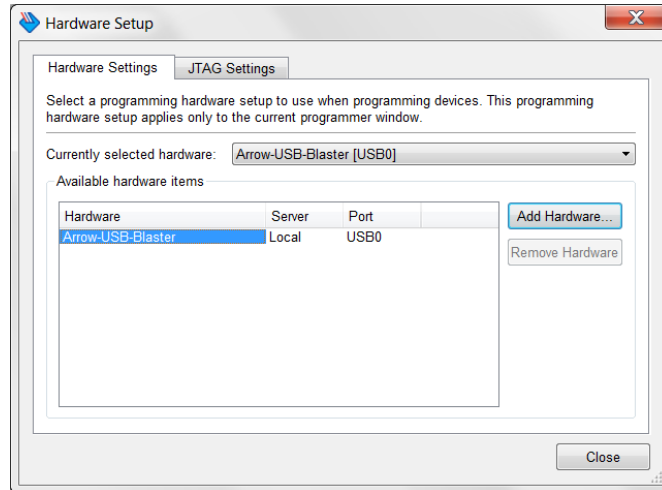
6.1.3 Connect your MAX1000 board to your PC using a USB cable. The Arrow USB Programmer2 driver software should already be installed, if not please refer to Chapter 4.2 (Windows based operating system) or 4.4 (Linux based operating system) for installing the drivers.

After plugging in the MAX1000, the green LED should also light up indicating power on, along with a brief flash of the red LED CONF_DONE.

In the Window's Device Manager should display the following entries that are highlighted in red.

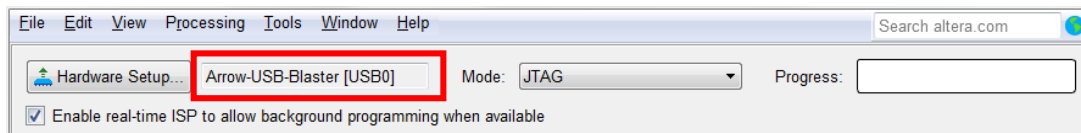


- 6.1.4 In the Programmer window, click Hardware Setup and double-click the ARROW-USB-PROGRAMMER2 entry in the Hardware Settings tab (you should only be able to see this Arrow USB Programmer2 if MAX1000 is connected to the PC). The “Currently selected hardware” drop-down should now show ARROW-USB-PROGRAMMER2 [USB-0]. Depending on your PC, the USB port number may vary.

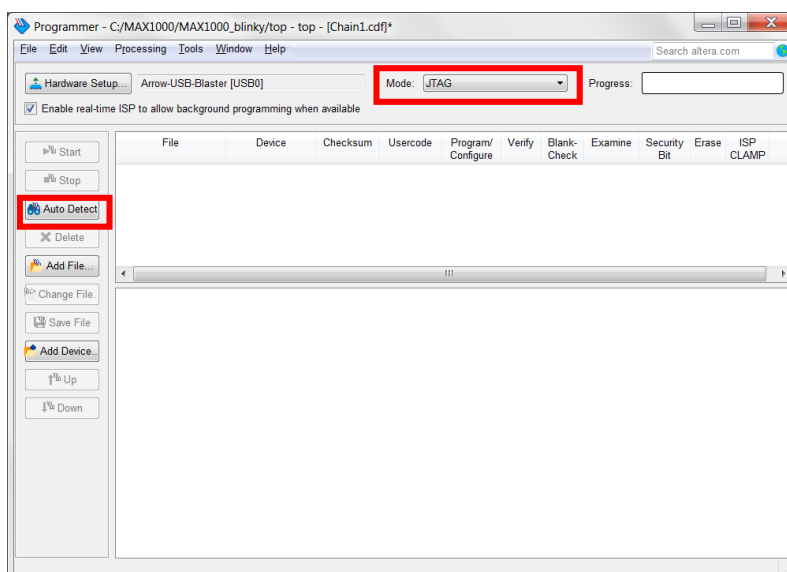


- 6.1.5 Click “Close”.

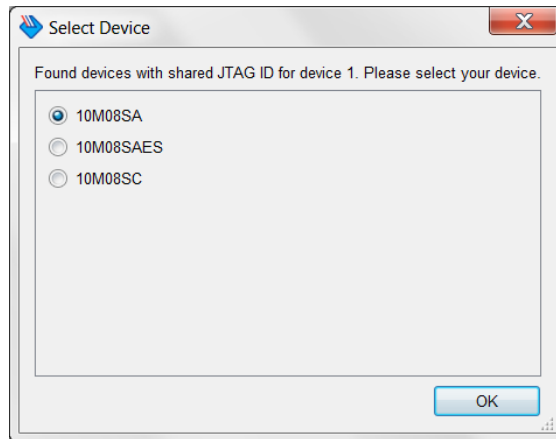
- 6.1.6 The Programmer window should now have a Hardware Setup such as:



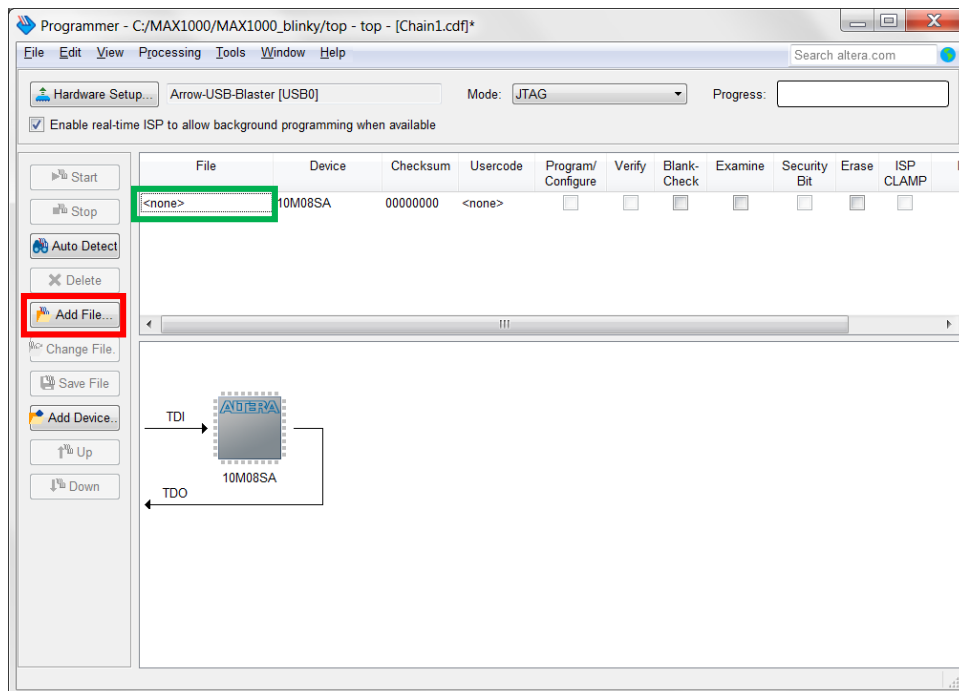
- 6.1.7 Click “Auto Detect” on the left side of the Programme window and make sure the Mode is JTAG.



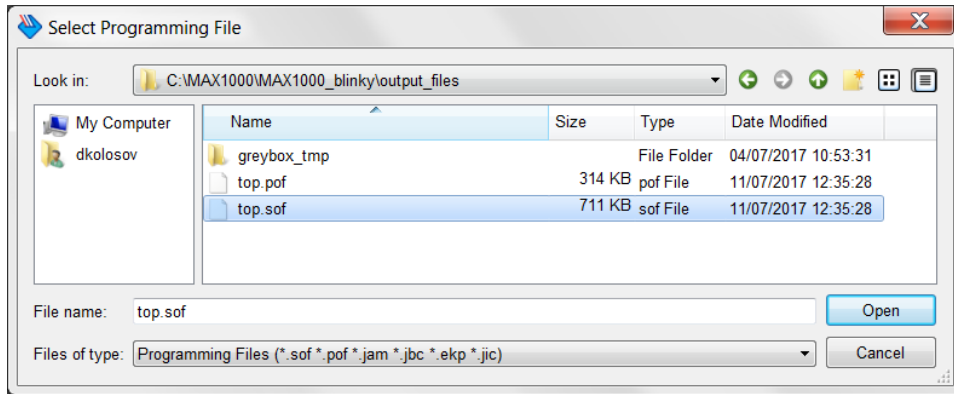
6.1.8 Select your device and click OK. In this case its “10M08SA”



6.1.9 To choose the programming file, you can either click “Add File..” on the right side of the Programmer window (highlighted in red) or double-click <none> below the File tab (highlighted in green).

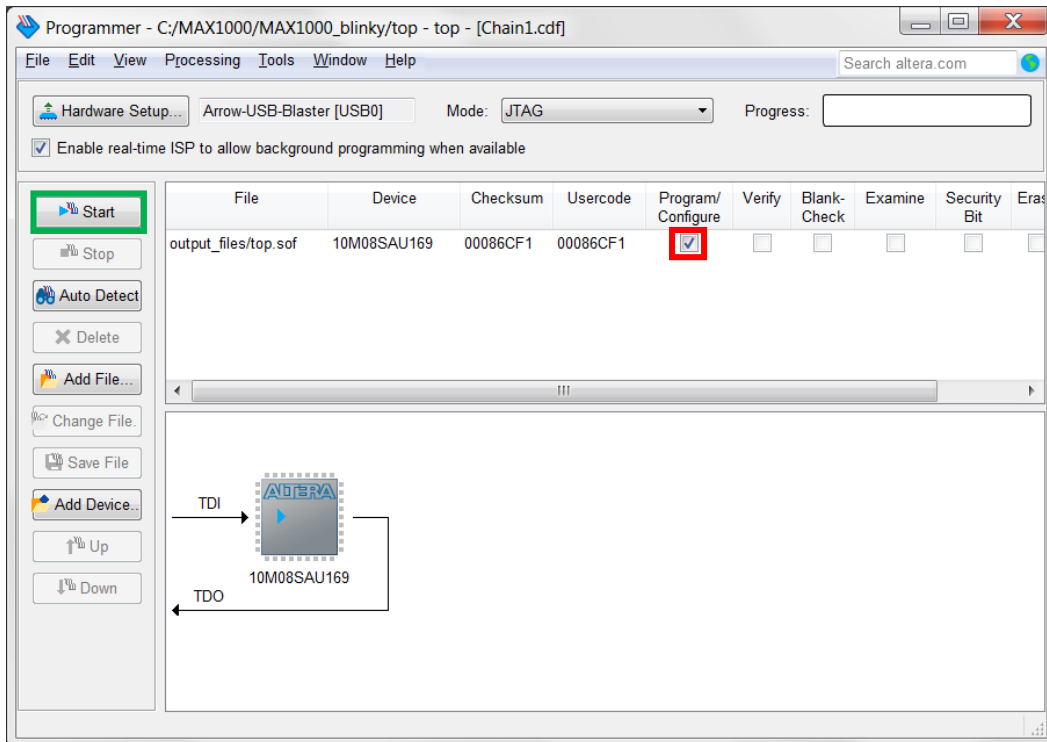


6.1.10 In either case, navigate to: <project_directory>/output_files/ in your compilation directory. Select and open the top.sof file.



6.1.11 Click “Open”.

6.1.12 The Programmer window should look like this now:

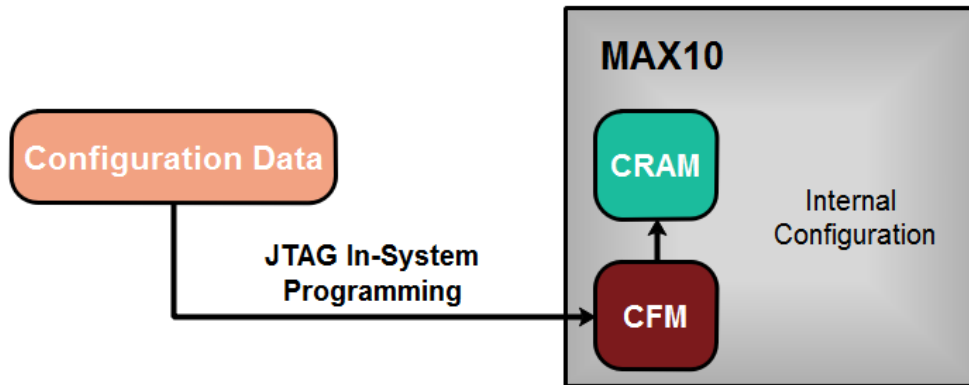


6.1.13 Make sure the Program/Configure checkbox is checked (highlighted in red above) and click “Start” (highlighted in green above) to program the MAX1000. You should see the CONF_DONE LED toggle briefly to indicate that the configuration is complete and the Progress bar should reach 100% (Successful). The design is now programmed to the FPGA.

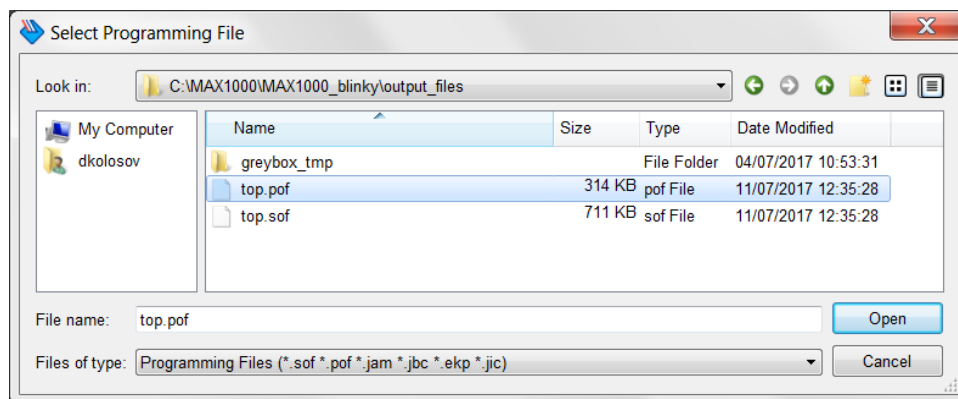
Note: that turning off and then on the FPGA will result into losing its configuration.

6.2 Internal Configuration

The configuration data to be written to CFM will be part of the programmer object file (.pof). This configuration data is automatically loaded from the CFM into the MAX10 device when the board is powered up.

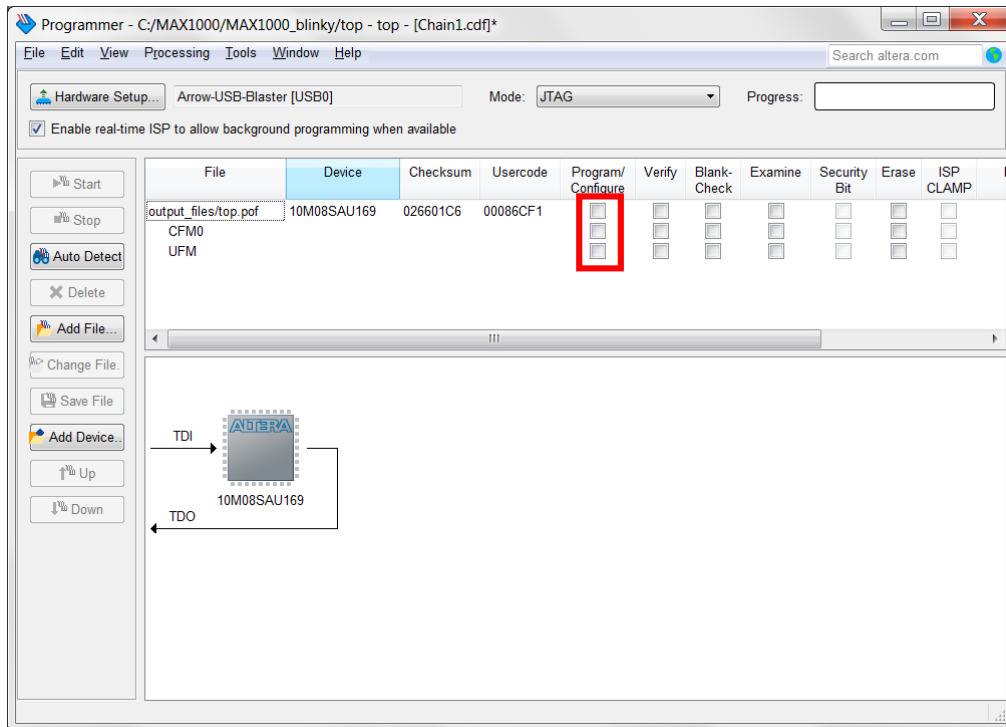


6.2.1 Going back to step 6.1.10, instead of choosing the .sof file, choose the .pof file.



6.2.2 Click "Open".

6.2.3 The window should look like this:



Select all the checkboxes under the Program/Configure for CFM0 and UFM (highlighted in red above).

6.2.4 Click “Start” to configure MAX10. The programming could take a while.

6.2.5 When the programming is finished, the MAX1000 should be able to keep its configuration data even after powered off.

6.3 Testing the Design

Does not matter which way the MAX1000 was configured, the results should be the same for both methods, with the only difference being if configuration is retained after power off.

On the board by default, the LEDs should now toggle in a slow counting sequence.

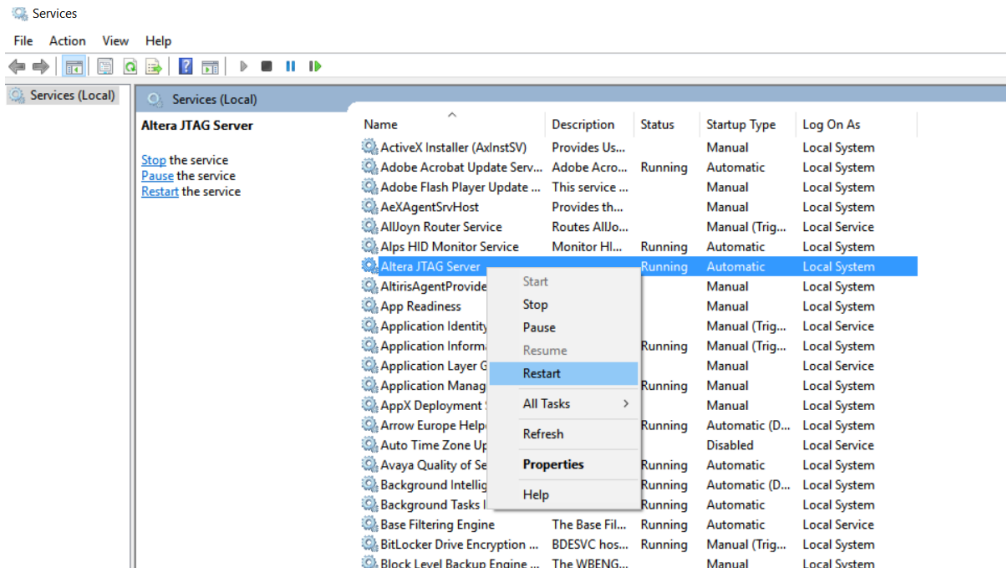
Push and hold the USER_BTN to see that the LEDs will now toggle in a very fast counting sequence. USER_BTN is on the side of the LEDs.

Releasing the USER_BTN, will make the LEDs toggle at a slower rate as before.

Chapter 7 - Common Issues and Fixes

- 1) **Issue:** In some rare cases when using Windows 10 operating system, the programmer DLL is not properly loaded/unloaded, causing the Quartus Programmer to not detect the Arrow USB Programmer2.

Solution: Restart the Altera JTAG Server using the Services application of Windows.





Chapter 8 - Appendix

7.1 Revision History

Version	Change Log	Date of Change
V1.0	Initial Version	11/07/2017





7.2 Legal Disclaimer

ARROW ELECTRONICS

EVALUATION BOARD LICENSE AGREEMENT

By using this evaluation board or kit (together with all related software, firmware, components, and documentation provided by Arrow, "Evaluation Board"), You ("You") are agreeing to be bound by the terms and conditions of this Evaluation Board License Agreement ("Agreement"). Do not use the Evaluation Board until You have read and agreed to this Agreement. Your use of the Evaluation Board constitutes Your acceptance of this Agreement.

PURPOSE

The purpose of this evaluation board is solely intended for evaluation purposes. Any use of the Board beyond these purposes is on your own risk. Furthermore, according the applicable law, the offering Arrow entity explicitly does not warrant, guarantee or provide any remedies to you with regard to the board.

LICENSE

Arrow grants You a non-exclusive, limited right to use the enclosed Evaluation Board offering limited features only for Your evaluation and testing purposes in a research and development setting. Usage in a live environment is prohibited. The Evaluation Board shall not be, in any case, directly or indirectly assembled as a part in any production of Yours as it is solely developed to serve evaluation purposes and has no direct function and is not a finished product.

EVALUATION BOARD STATUS

The Evaluation Board offers limited features allowing You only to evaluate and test purposes. The Evaluation Board is not intended for consumer or household use. You are not authorized to use the Evaluation Board in any production system, and it may not be offered for sale or lease, or sold, leased or otherwise distributed for commercial purposes.

OWNERSHIP AND COPYRIGHT

Title to the Evaluation Board remains with Arrow and/or its licensors. This Agreement does not involve any transfer of intellectual property rights ("IPR) for evaluation board. You may not remove any copyright or other proprietary rights notices without prior written authorization from Arrow or it licensors.

RESTRICTIONS AND WARNINGS

Before You handle or use the Evaluation Board, You shall comply with all such warnings and other instructions and employ reasonable safety precautions in using the Evaluation Board. Failure to do so may result in death, personal injury, or property damage.

You shall not use the Evaluation Board in any safety critical or functional safety testing, including but not limited to testing of life supporting, military or nuclear applications. Arrow expressly disclaims any responsibility for such usage which shall be made at Your sole risk.

WARRANTY

Arrow warrants that it has the right to provide the evaluation board to you. This warranty is provided by Arrow in lieu of all other warranties, written or oral, statutory, express or implied, including any warranty as to merchantability, non-infringement, fitness for any particular purpose, or uninterrupted or error-free operation, all of which are expressly disclaimed. The evaluation board is provided "as is" without any other rights or warranties, directly or indirectly.

You warrant to Arrow that the evaluation board is used only by electronics experts who understand the dangers of handling and using such items, you assume all responsibility and liability for any improper or unsafe handling or use of the evaluation board by you, your employees, affiliates, contractors, and designees.



LIMITATION OF LIABILITIES

In no event shall Arrow be liable to you, whether in contract, tort (including negligence), strict liability, or any other legal theory, for any direct, indirect, special, consequential, incidental, punitive, or exemplary damages with respect to any matters relating to this agreement. In no event shall arrow's liability arising out of this agreement in the aggregate exceed the amount paid by you under this agreement for the purchase of the evaluation board.

IDENTIFICATION

You shall, at Your expense, defend Arrow and its Affiliates and Licensors against a claim or action brought by a third party for infringement or misappropriation of any patent, copyright, trade secret or other intellectual property right of a third party to the extent resulting from (1) Your combination of the Evaluation Board with any other component, system, software, or firmware, (2) Your modification of the Evaluation Board, or (3) Your use of the Evaluation Board in a manner not permitted under this Agreement. You shall indemnify Arrow and its Affiliates and Licensors against and pay any resulting costs and damages finally awarded against Arrow and its Affiliates and Licensors or agreed to in any settlement, provided that You have sole control of the defense and settlement of the claim or action, and Arrow cooperates in the defense and furnishes all related evidence under its control at Your expense. Arrow will be entitled to participate in the defense of such claim or action and to employ counsel at its own expense.

RECYCLING

The Evaluation Board is not to be disposed as an urban waste. At the end of its life cycle, differentiated waste collection must be followed, as stated in the directive 2002/96/EC. In all the countries belonging to the European Union (EU Dir. 2002/96/EC) and those following differentiated recycling, the Evaluation Board is subject to differentiated recycling at the end of its life cycle, therefore: It is forbidden to dispose the Evaluation Board as an undifferentiated waste or with other domestic wastes. Consult the local authorities for more information on the proper disposal channels. An incorrect Evaluation Board disposal may cause damage to the environment and is punishable by the law.