

# LiveDesign Evaluation Kit

## QUICKSTART GUIDE

### Welcome to Altium Designer and LiveDesign!

The Altium Designer LiveDesign Evaluation Kit lets you experience first-hand the power that real-time communication with your design during development brings to the design process. Altium Designer is the industry's first system to unify the complete electronic product development process and allow you to harness the full potential of today's high-capacity programmable devices at both the logical and physical design level.

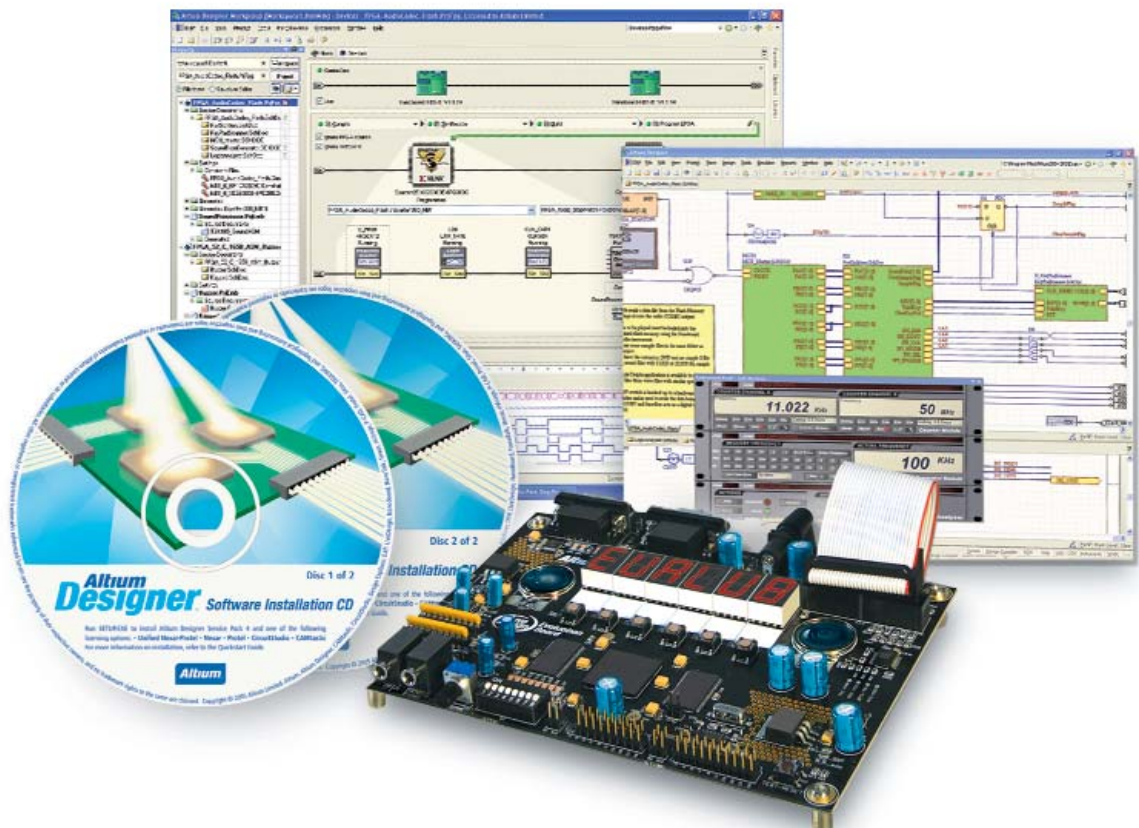
You'll be able to probe, analyze and debug your circuit just like an actual prototype. And because the implementation is performed on an FPGA, you can update your design quickly and many times over without incurring cost or time penalties.

Altium Designer and LiveDesign will have a profound impact on the way your future products are designed, providing you with benefits to the speed, design flow, quality and final product cost for the design phase.

### Start today – experience the benefits of Altium Designer and LiveDesign!

The following steps are outlined in this Quickstart Guide to start you on your evaluation of LiveDesign:

- ▶ What's in the box
- ▶ System requirements
- ▶ Requirements for FPGA design
- ▶ Installing your software
- ▶ Activating your license
- ▶ Connecting your board
- ▶ Downloading an example project
- ▶ More information



## ▼ **What's in the box**



Your Altium Designer LiveDesign Evaluation Kit includes the following components to get you started on your evaluation:

- |  |  |
|--|--|
| A LiveDesign Evaluation Board                      | E Power supply   |
| B Altium Designer Software Installation CDs        | F Power supply cord                                      |
| C Parallel cable (26 way IDC – DB25)               | G LiveDesign Evaluation Board Technical Reference Manual |
| D User header cable (20 way IDC – 20 way IDC) (x2) |  |

If any of these components are missing or appear damaged, please contact your nearest Altium Sales and Support Center or Reseller.

*Please follow standard antistatic procedures during installation.  
See the LiveDesign Evaluation Board Technical Reference Manual (G) for full details.*

## ▼ **System requirements**

For optimum performance of Altium Designer software please refer to the recommended system requirements provided below.

### **Recommended system for optimal performance**

- Windows XP (Professional or Home)
- 3 GHz Pentium 4 processor or equivalent
- 1 GByte RAM
- 2 GByte hard disk space (Install + User Files)
- Dual monitors with 1280x1024 screen resolution
- 32-bit color, 64 MB graphics card

### **Minimum system for acceptable performance**

- Windows 2000 Professional SP2
- 1.8 GHz processor
- 1 GByte RAM
- 2 GByte hard disk space (Install + User Files)
- Main monitor 1280x1024 screen resolution  
Strongly recommended: Second monitor with minimum 1024x768 screen resolution
- 32-bit color, 32 MB graphics card

## ▼ Requirements for FPGA design

To implement a design in an FPGA you will need to have a LiveDesign-enabled development board, such as the LiveDesign Evaluation Board or Altium's NanoBoard, properly connected to your computer.

To complete the device-specific processing of a design for implementation in an FPGA, Altium Designer transparently interfaces with third-party software tools supplied by the various FPGA vendors. You will require the appropriate FPGA vendor software installed and correctly licensed before you can process your design for a specific target device.

Altium Designer supports the following FPGA vendor software:

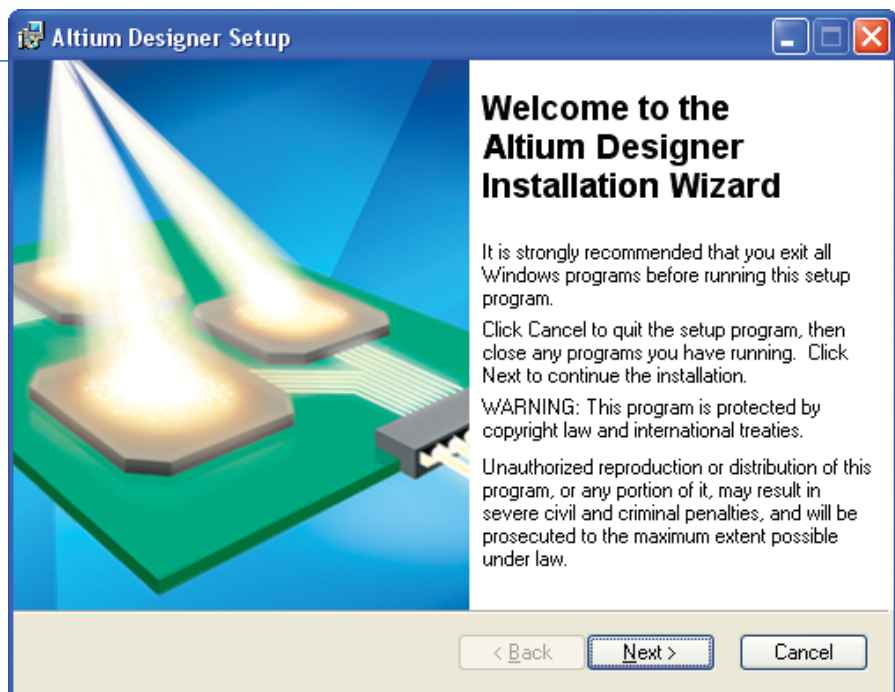


For up-to-date information on supported versions, visit [www.altium.com/vendortools](http://www.altium.com/vendortools)

## ▼ Installing your software

Altium uses a licensing system that makes the management of the licensing options within Altium Designer easy to administer and manage. Under this system, Altium collects hardware data from each machine with an Altium node-locked license installed. Depending on individual situations, up to ten pieces of hardware information may be collected and used for licensing management. The hardware information collected is not shared with any external third party or used by Altium for any reason other than license management.

### Step 1



To begin the install process, simply insert the Altium Designer Software Installation CD (Disc 1) into your computer's CD ROM drive. The Installation Wizard will automatically start and guide you through the installation process. If the Installation Wizard does not start automatically, run Setup.exe located in the Setup directory on the program CD.

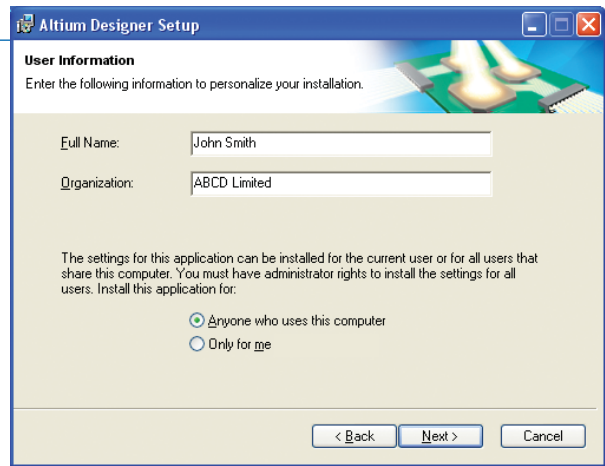
Note: If you have a previous version of Altium Designer installed on your computer, you may be prompted to uninstall it prior to continuing with the installation. If you are required to do this, use Add and Remove Programs in the Windows Control Panel. We recommend you backup any important information in the Altium Designer loadpoint prior to uninstalling.

## Step 2

A 'License Agreement' screen will now appear (not illustrated).

You must accept the license agreement in order to proceed.

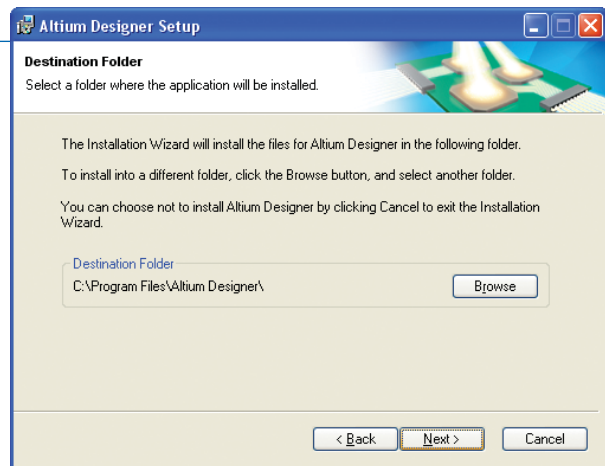
Once you have accepted, you will be asked to enter your full name and the organization name in order to personalize your installation.



## Step 3

Select the folder where you wish to install the files for Altium Designer. Click **Next** and the 'Ready to Install the Application' screen will appear (not illustrated). Click **Next** again and the installation will begin. During the installation you will be prompted to insert Disc 2. Insert and click **OK** to continue.

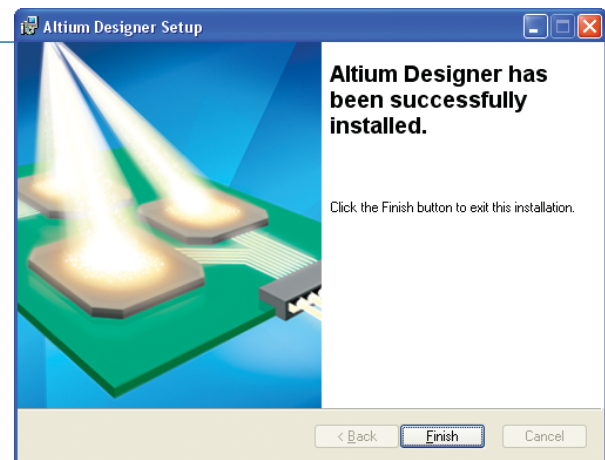
Note: The default Destination Folder will vary according to the release that you are installing.



## Step 4

The following screen will appear when Altium Designer has been successfully installed.

*Steps 5-9 will guide you through the license activation process.*



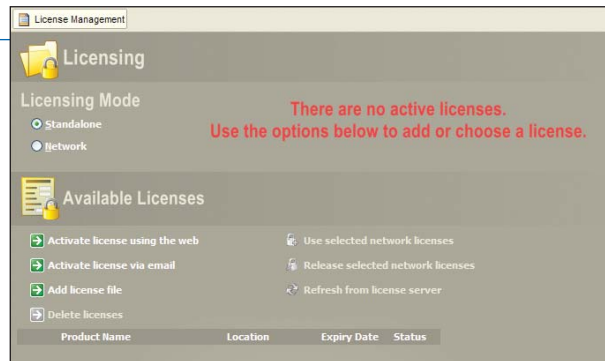
## ▼ *Activating Your License*

### Step 5

To launch the Activation Wizard, start the Altium Designer software from the Start menu. You will then be taken to the 'Licensing' screen. If this page does not appear, click on **DXP » Licensing**.

From the Available Licenses menu, select your preferred activation method, nominating either Internet or email activation. This will then lead you to the Activation Wizard Step 1 of 3 screens.

The quickest and simplest way to activate your license is by the use of the Internet. The email activation option is recommended where access to the Internet is limited. Once you have activated your license, you will have access to 30 days of unlimited use of Altium Designer.



### Step 6

You will now be asked to enter your Customer Number and product Activation Code. You will find these details in the documents received with the LiveDesign Evaluation Kit.

You will then be asked if you wish to identify a primary point of contact for the license. If you select No, you will still need to provide a valid email address to receive the license file.

## Step 7

The next step of the activation process requires you to verify the licensee details. Check the information displayed and update if necessary.

**Step 2 of 3**

Please verify the licensee details are correct and up to date.  
All fields marked with a \* character are required.

Company : ABCD Limited

First Name\* : John Last Name\* : Smith

Telephone\* : (111) 111 111 111 Fax : (111) 111 111 112

Email\* : john.smith@abcd.com

Confirm Email\* : john.smith@abcd.com

Country\* : Australia

Address\* : 111 Nowhere Drive

City\* : Notown State\* : ACT

Zip Code\* : ABCD

Back Next

## Step 8

Click on the **Finish** button in order to activate your license. Your details will be sent to Altium. The licensee details are then verified and the license file will be emailed to you.

**Step 3 of 3**


The Wizard has all the details required to activate. Click Finish to complete the process and the license file will be emailed to you.

Back Next Finish

## Step 9

You have now completed the activation of your license, giving you 30 days to evaluate the full features of Altium Designer. You will receive an email from Altium containing your license file. Follow the instructions in this email to add your license file and finalize your activation. You are now ready to use Altium Designer.

Activation Success

 **Activation Success**

Activation has been successful, an email with the license file attached has been sent to [john.smith@abcd.com](mailto:john.smith@abcd.com).

The final stage in enabling the software is to specify the location of the license file. Click the link below to do this now if the email has arrived.  
Note that you can add licenses at any time, by selecting Licensing from the DXP menu.

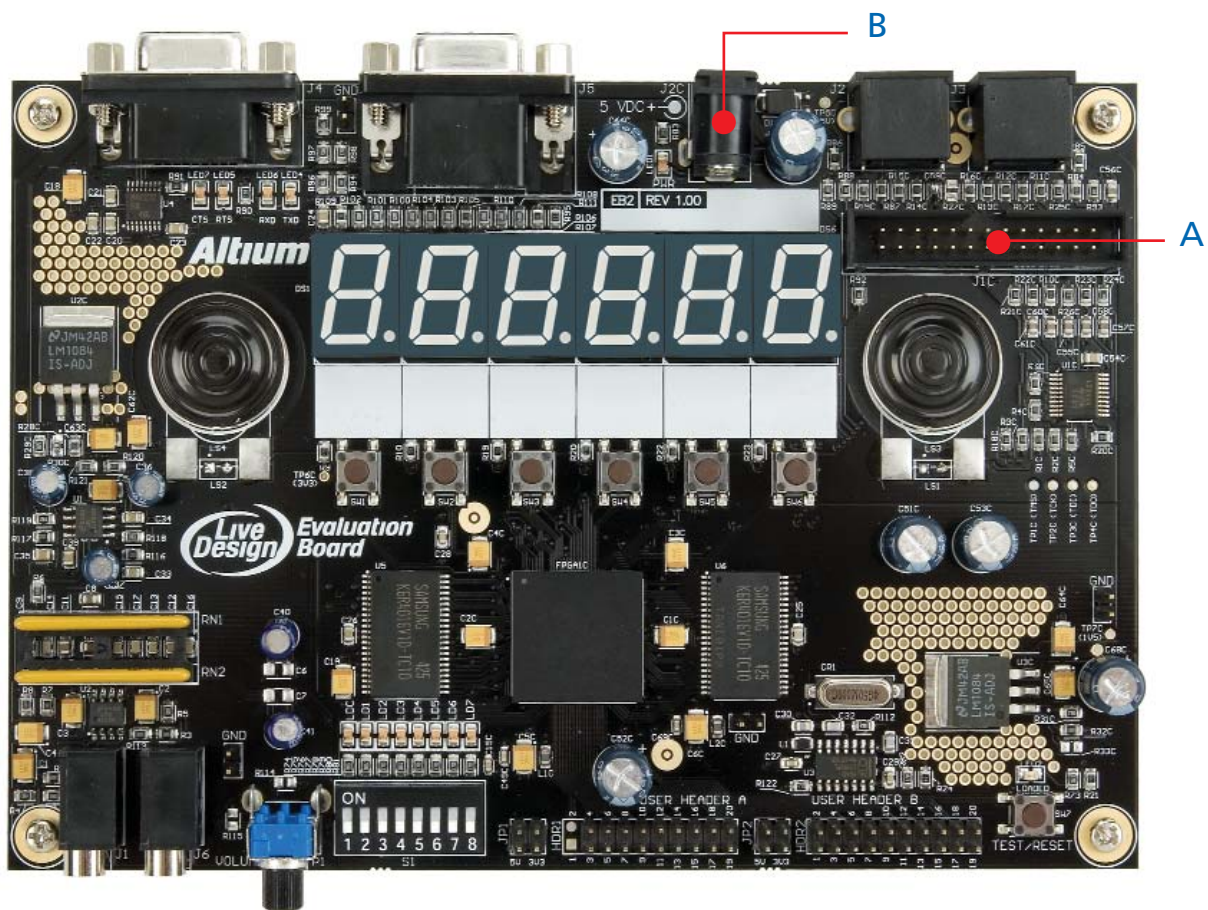
- [Add license file](#)
- [Return to licensing page](#)
- [DXP Home](#)

## ▼ Connecting your board

To start-up the LiveDesign Evaluation Board, complete the following steps:

- 1 Connect the LiveDesign Evaluation Board to your PC using the parallel cable supplied, via the parallel header socket (A).
- 2 Connect the power supply to the power supply socket (B). Connect the power supply cord to the power supply and plug it in to a standard power socket.
- 3 The LiveDesign Evaluation Board does not have a power switch. It will turn on when power is turned on at the power socket, at which point the power indicator LED will illuminate.

Refer to the LiveDesign Evaluation Board Technical Reference Manual for information regarding testing and ensuring correct setup and installation of your LiveDesign Evaluation Board.



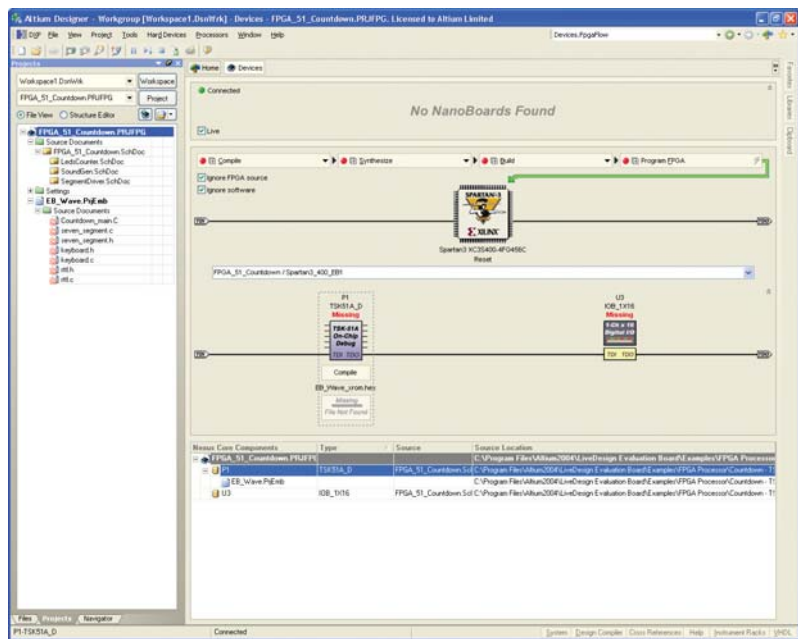
The LiveDesign Evaluation Board A = Parallel header socket B = Power supply socket

## ▼ Downloading an example project

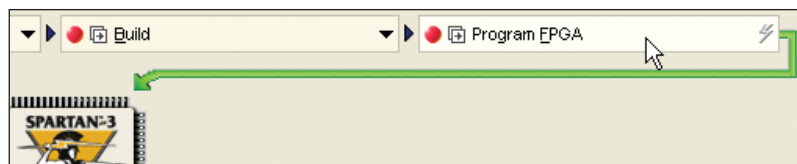
This example will take you through the steps of compiling and synthesizing a design, downloading it into the target FPGA and finally interacting with the program inside the FPGA using Digital I/O Virtual Instruments.

Before you commence this example project, ensure that you have followed the previous software and hardware installation steps in this Quickstart Guide. The LiveDesign Evaluation Board should be connected and powered up and all the appropriate software, including the relevant FPGA vendor tools, installed.

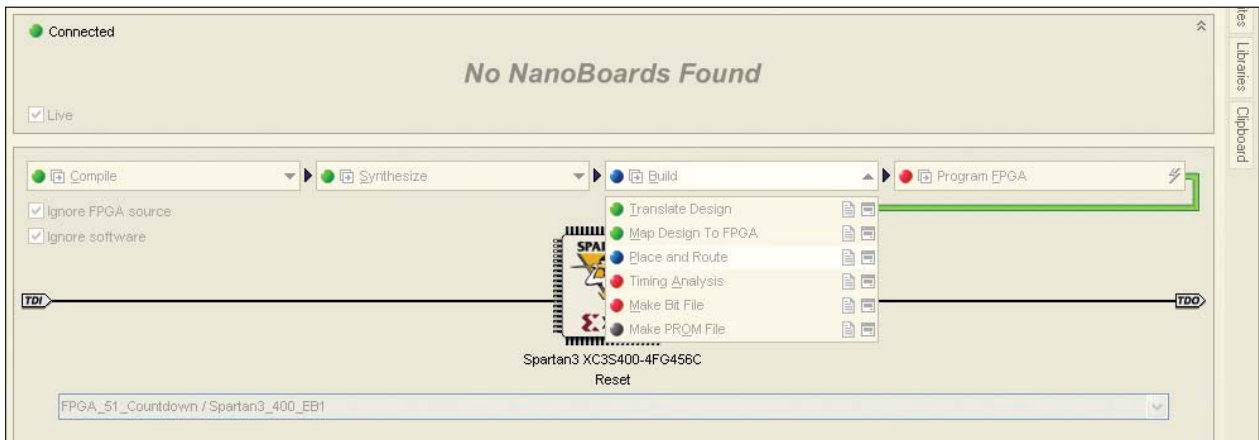
- 1 Start the Altium Designer software from the Start menu. Once the software has loaded, select **File » Open** from the menus.
- 2 Navigate to the \Examples \LiveDesign Evaluation Board \FPGA Processor\Countdown - TSK51 folder in the Altium Designer loadpoint.
- 3 In this directory, open the file **FPGA\_51\_Countdown.PrjFpg**. When the project has loaded, the Projects panel on the left side of the workspace will display the files in this project.
- 4 Select **View » Devices View** from the menus to display the Devices View window. Ensure that the Live check box is enabled and the Connected indicator is green. The project will automatically be assigned to the target FPGA device on the connected LiveDesign Evaluation Board (see image).
- 5 To process the project and download it to the LiveDesign Evaluation Board, click the **Program FPGA** button in the Devices View window.



The Devices View window. Note: Xilinx Spartan-3 device is shown. Image will be different depending on your target FPGA device.



The system will automatically compile the source project files, synthesize the design, call the vendor place and route tools to process the design for the target FPGA, and then download the design to the LiveDesign Evaluation Board. If the Confirm dialog displays, click on **Yes** to use the Altium Synthesizer. This can take several minutes depending on the speed of your computer.

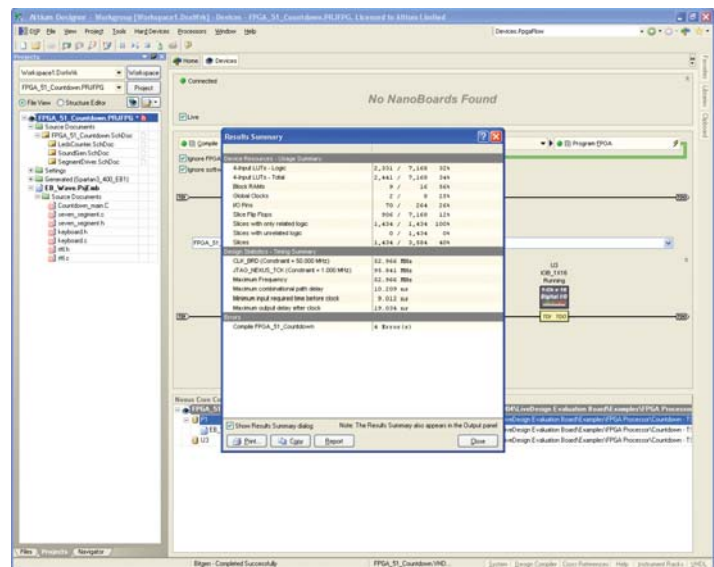


The Devices View window showing the compilation progress

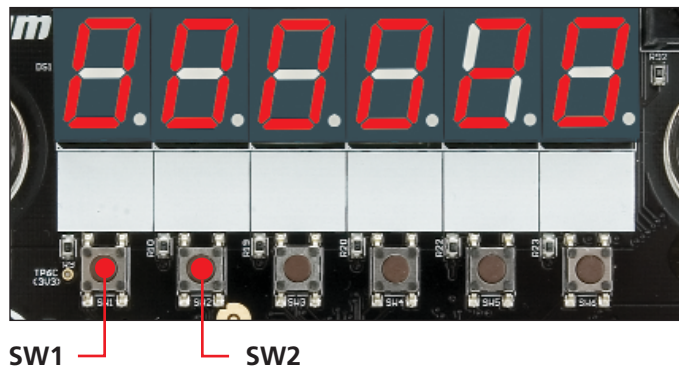
Please note: If the system fails during the Build processes, it is shown by a purple status indicator. This can be because the FPGA vendor tools are not correctly installed or have not been properly activated. Please refer to the information supplied with the FPGA vendor tools for details on installing and activating these tools. For more details, refer to the LiveDesign Evaluation Board Technical Reference Manual or visit [www.altium.com/community](http://www.altium.com/community)

Once the design has been downloaded, all of the process indicators will be green and the Results Summary dialog will be shown (see image). This indicates that the project has been successfully processed and downloaded to the LiveDesign Evaluation Board. Close the Results Summary dialog.

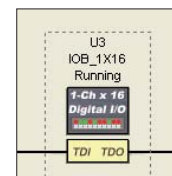
Congratulations! You have just reconfigured your LiveDesign Evaluation Board to operate as a countdown timer. If you look at the board you should see that the 7-Segment Display is showing the default counter value equal to 20.



- Press the first button (**SW1**) for the counter to commence counting down. When it reaches 0, a melody will be outputted to the Audio Speakers as well as text scrolling on the 7-Segment Display and LEDs flashing. Pressing the second button (**SW2**) will reset the counter with the initial value and stop the melody.



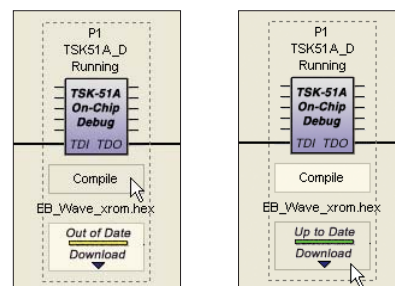
- Double-click on the **IOB\_1X16** symbol to open the virtual instruments rack, displaying the Digital I/O instrument. Note that the Digital I/O instrument input shows the current count value.



Virtual instruments rack showing Digital I/O Module

- Type in **000A** in the outputs HEX edit box (shown above) to change the default count value from 20 to 10 seconds. As you edit the values in the Digital I/O instrument, the 7-Segment Display is updated to reflect these changes. Press the first button (**SW1**) again to activate the countdown.

Setting the I/O instrument output value to zero (0000) allows the default count value to be modified in the embedded source code file (countdown\_main.c) by changing the DEFAULT\_COUNT constant. If this is done, then the embedded project needs to be re-compiled and downloaded. To do this, click on the **Compile** button below the processor symbol and then click the **Download** button below the Compile button to update the FPGA memory space.



Explore the schematics and embedded code in the FPGA\_51\_Countdown project for more information on the connection and operation of the virtual I/O modules, peripheral connections and the general operation of this project.

## ▼ **More information**

Congratulations on successfully installing your Altium Designer LiveDesign Evaluation Kit and running your first project. Now that you are ready to evaluate LiveDesign, we recommend that you refer to the following information to gain maximum benefit.

The Altium Designer Documentation Library, accessed by pressing **F1** within the application, provides an extensive range of documentation covering all areas of design. We recommend you begin with The Basics, which explain how to get started with Altium Designer and provide you with a good understanding of the software. Following this, we suggest you also read the below introductory documents which relate to FPGA design, processor cores and embedded software development:

- GU0101 FPGA Designers Quickstart Guide.pdf
- TU0116 Getting Started with FPGA Design.pdf
- TU0118 Implementing a Simple Processor-Based Design in an FPGA.pdf
- AP0103 Processing the Captured FPGA Design.pdf
- TU0123 Creating a Core Component.pdf

Note: Although references in some of these documents relate to using and downloading designs to Altium's NanoBoard-NB1 hardware development platform, the overall concepts and process flows can be applied to the LiveDesign Evaluation Board.

Over 20 example projects and reference designs have been supplied with the software that are specifically tailored for the LiveDesign Evaluation Board. Each example project demonstrates the use of various features and peripherals of the LiveDesign Evaluation Board, as well as the included processor cores and virtual instruments, showing how easy it is for you to have real-time communication and hands-on interaction with your design.

All example projects are available in the \Examples\LiveDesign Evaluation Board folder found in the Altium Designer loadpoint. Refer to the top-level schematic document of each example project for overview and general operating instructions.

Further information, support resources and additional example projects can be found at [www.altium.com/community](http://www.altium.com/community)

***For more information, visit [www.altium.com/livedesign](http://www.altium.com/livedesign)  
or contact your local Altium Sales and Support Center or Reseller.***