

# 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.

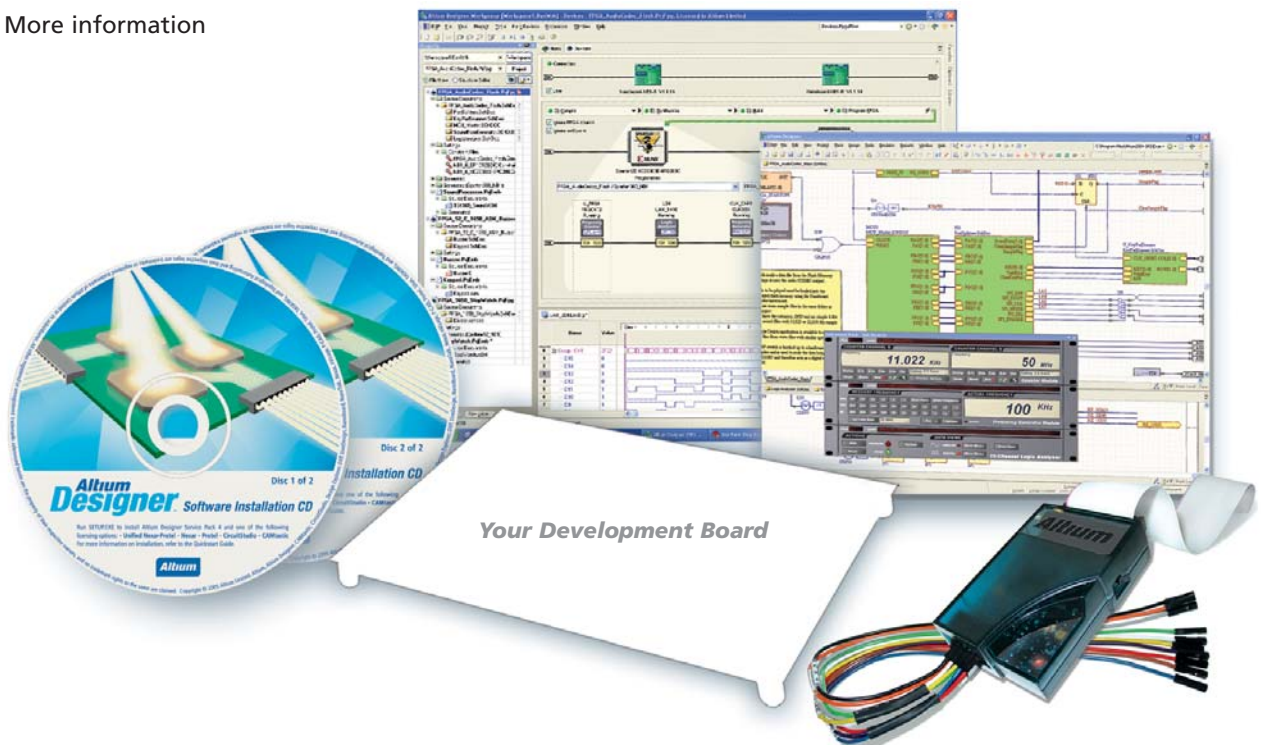
You'll be able to probe, analyze and debug your circuit just like an actual prototype. And because the implementation is performed on an FPGA, you can update your design quickly and many times over without incurring cost or time penalties.

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.

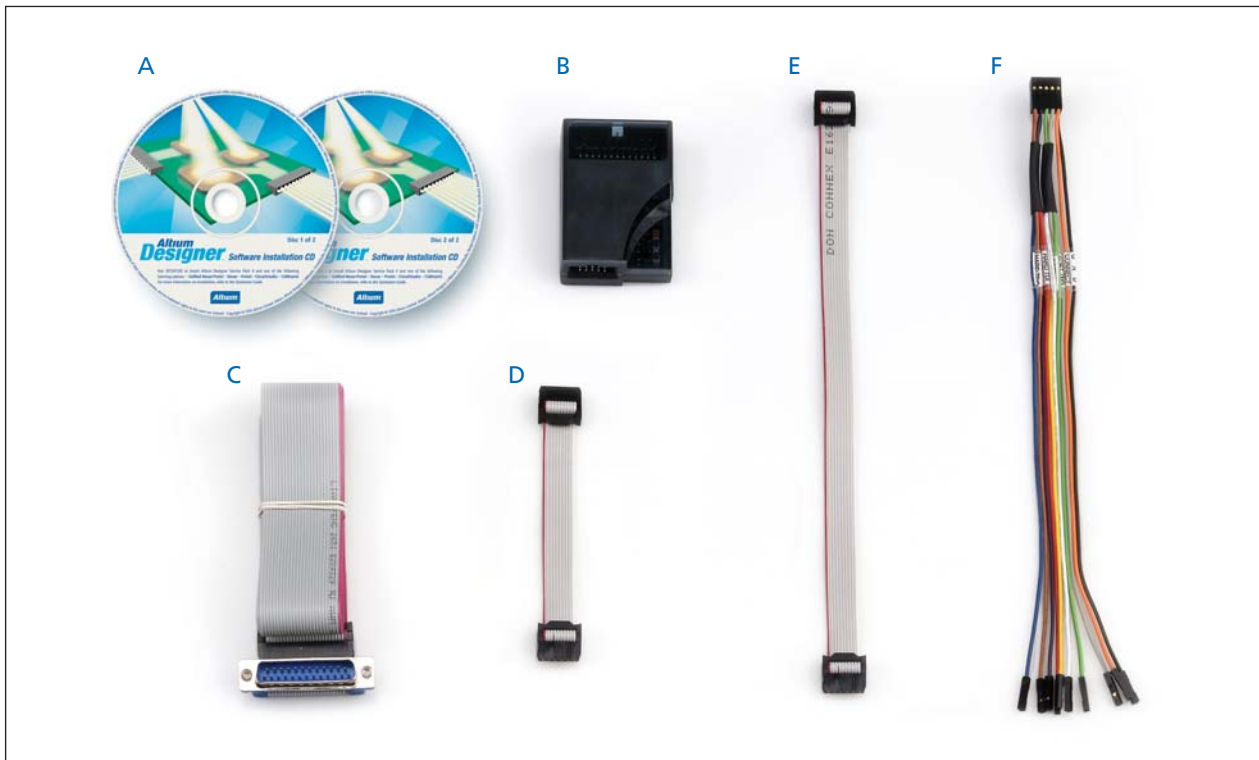
### Start today – experience the benefits of Altium Designer and LiveDesign!

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
- ▶ Creating a constraint file for your board
- ▶ Online third-party board support
- ▶ More information



## ▼ **What's in the box**



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

- |  |  |
|--|--|
| <b>A</b> Altium Designer Software Installation CDs | <b>D</b> JTAG ribbon cable (short 10 way IDC – 10 way IDC)     |
| <b>B</b> Universal JTAG Interface                  | <b>E</b> JTAG ribbon cable (long 10 way IDC – 10 way IDC)      |
| <b>C</b> Parallel cable (26 way IDC – DB25)        | <b>F</b> JTAG flying lead cable (10 way IDC – 10 flying leads) |

*Please follow standard antistatic procedures during installation.*

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

## ▼ **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 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:

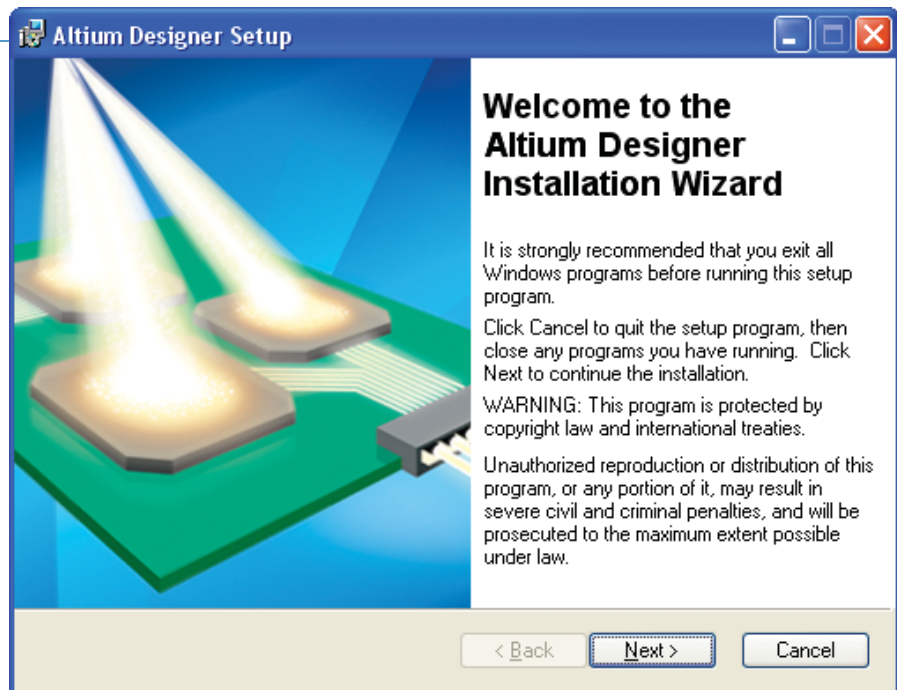


For up-to-date information on supported versions, visit [www.altium.com/vendortools](http://www.altium.com/vendortools)

## ▼ 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.

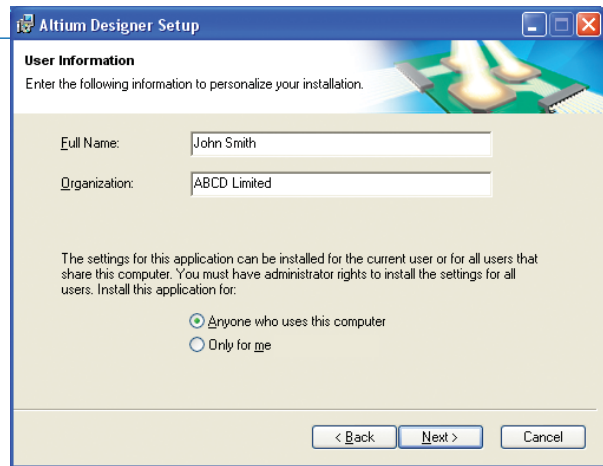
Note: If you have a previous version of Altium Designer installed on your computer, you may be prompted to uninstall it prior to continuing with the installation. If you are required to do this, use Add and Remove Programs in the Windows Control Panel. We recommend you backup any important information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

## Step 2

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

You must accept the license agreement in order to proceed.

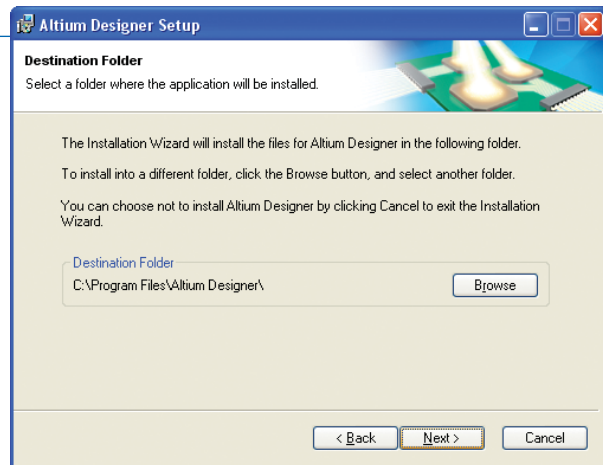
Once you have accepted, you will be asked to enter your full name and organization name in order to personalize your installation.



## 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.

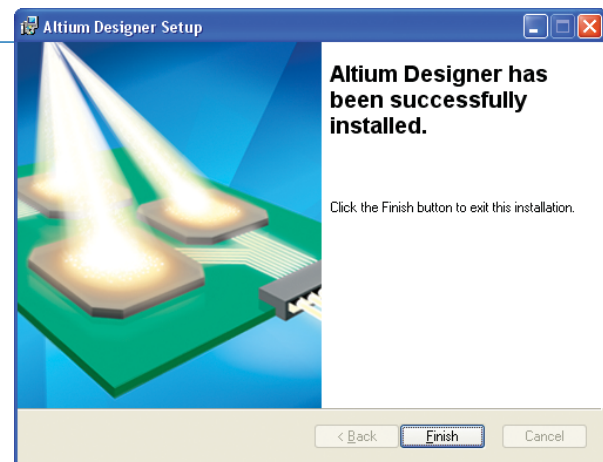
Note: The default Destination Folder will vary according to the release that you are installing.



## 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.*



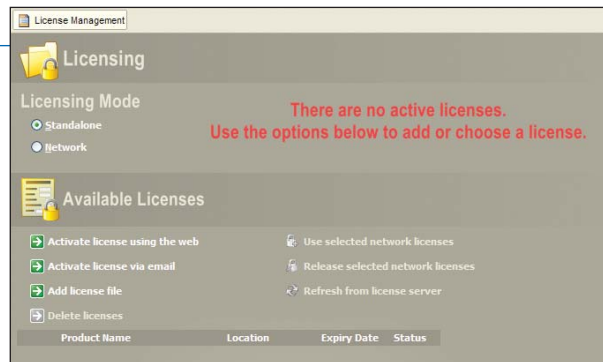
## ▼ **Activating your license**

### 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.



### 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		

## 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.

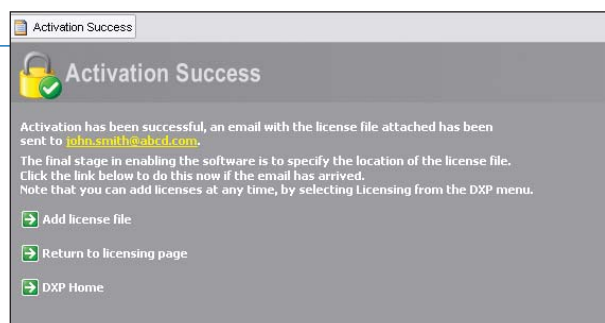
**Step 3 of 3**

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

## Step 9

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, giving you 30 days to evaluate the full features of Altium Designer.

*You are now ready to use Altium Designer.*



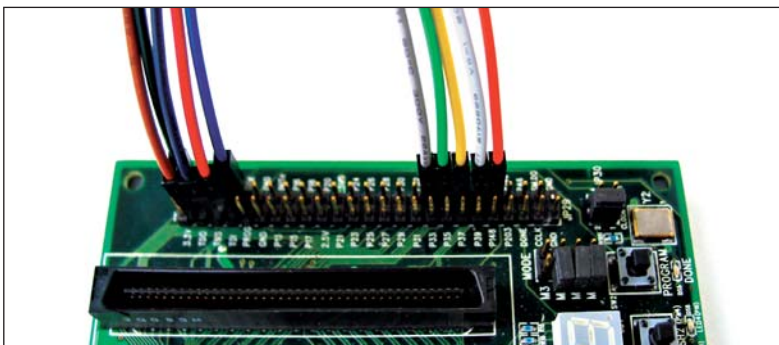
## ▼ **Connecting your board**

- 1 Connect the Universal JTAG Interface to your PC using the parallel cable supplied, via the parallel header socket (A).
- 2 Connect the flying lead cable to the 10 pin JTAG interface header socket (B). Note: A 10 way – 10 way ribbon cable is supplied for third-party boards with a 10 pin compatible header interface.



*The Universal JTAG Interface  
A = Parallel header socket  
B = Interface header socket*

- 3 Referring to the third-party development board's silkscreen or technical documentation, locate and connect the five flying leads marked GND, HARD\_TDI, HARD\_TDO, HARD\_TCK and HARD\_TMS to the corresponding JTAG header pins on the target board.
- 4 Connect the four flying leads marked SOFT\_TDI, SOFT\_TDO, SOFT\_TCK and SOFT\_TMS to any free I/O pins of the FPGA on the target board. The flying lead marked CONNECT is floating and can remain unconnected.



*Hard and soft JTAG chains  
connected to a target board*

## ▼ *Creating a constraint file for your board*

This section will take you through the steps of creating the constraint file for your target board and adding it to an FPGA project.

A constraint file is a text file containing a list of statements or lines known as constraint groups, each of which targets one or more objects and contains one or more constraints.

### 1 Creating the file

Start the Altium Designer software from the Start menu. Once the software has loaded, select **File » New » Other » Constraint File** from the menus.

Constraints can be defined by typing them in directly, or by using the **Design » Add/Modify Constraint** menu options.

### 2 Defining the target board

Within the constraint file editor, type in the following constraint group:

```
Record=Constraint | TargetKind=PCB | TargetId=My Third Party Development Board
```

The TargetId specifies the name of the PCB you are targeting. This can be any text string to identify your board.

### 3 Defining the target device

Within the constraint file editor, either type in the following constraint group or select **Design » Add/Modify Constraint » Part...** menu command and choose the device from the 'Choose Physical Device' dialog.

```
Record=Constraint | TargetKind=Part | TargetId=EP1C12Q240C6
```

The TargetId above is specifying an Altera Cyclone device. Substitute this with the actual device on your board. Note that this device must be supported by the system. To check that your device is supported, select **View » Devices View**. Then select **Tools » Browse Physical Devices** and click on the vendor's name in the Vendors list to display supported devices.

### 4 Defining the Nexus JTAG soft chain

Within the constraint file editor, either type in the following constraint groups or choose **Design » Add/Modify Constraint » Port...** menu command for each new pin.

```
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXUS_TDI | FPGA_PINNUM=P108
```

```
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXUS_TDO | FPGA_PINNUM=P109
```

```
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXUS_TCK | FPGA_PINNUM=P45
```

```
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXUS_TMS | FPGA_PINNUM=P115
```

The FPGA\_PINNUM field indicates the pin number of I/O pins on the FPGA that are connected to the SOFT\_TDI, SOFT\_TDO, SOFT\_TCK and SOFT\_TMS flying leads.

## 5 Defining the other FPGA pins

Within the constraint file editor, create new constraint groups specifying all the remaining connected FPGA pins on the board. The pins may be grouped together to target a particular peripheral or feature of the board.

Record=Constraint | TargetKind=Port | TargetId=  
[Port Name] | FPGA\_PINNUM=[pin#]\*

For example, a 4 pin DIP Switch could be defined as follows:

Record=Constraint | TargetKind=Port | TargetId=SW[3..0]  
| FPGA\_PINNUM=P29,P28,P27,P26

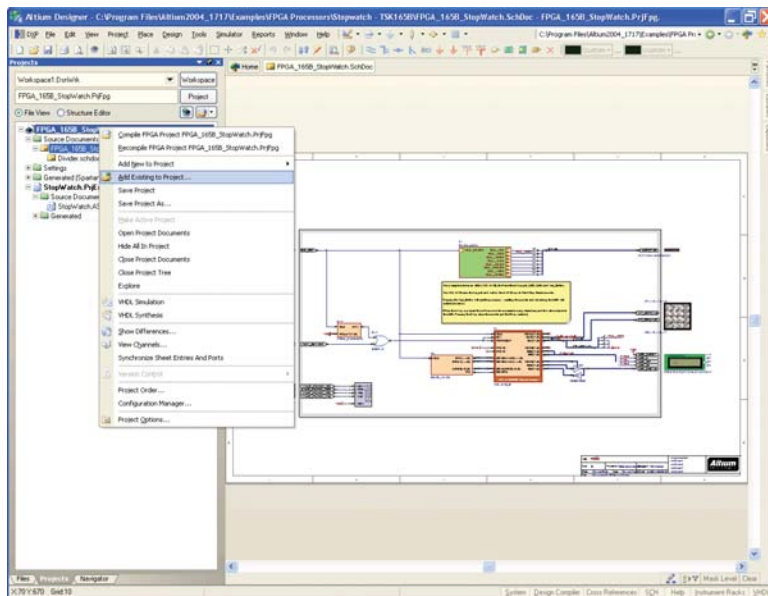
The TargetId above is specifying the schematic port name that will be used in the design to connect to the specified pin.

\* Note that for QFP packaged FPGAs, valid pin numbers may begin with the letter P, immediately followed by the pin number, i.e. P22 means Pin 22. For FPGAs in PGA or BGA packages, the pin number is a grid reference, i.e. A22 is the pin on Row A, Column 22, and P22 is Row P, Column 22.

```
.....
:Constraint File
: Device : Altera Cyclone EP1C12Q240C6
: Board : My Third Party Development Board
: Project : Any
:
: Created 26 October 2004
: Altium Limited
.....
Record=FileHeader | ID=00P Constraints v1.0
.....
Record=Constraint | TargetKind=Port | TargetId=My Third Party Development Board
Record=Constraint | TargetKind=Port | TargetId=EP1C12Q240C6
.....
: NEXOS JTAG Soft-Device Chain Connections
.....
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXOS_TDI | FPGA_PINNUM=P100
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXOS_TDO | FPGA_PINNUM=P109
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXOS_TCK | FPGA_PINNUM=P45
Record=Constraint | TargetKind=Port | TargetId=JTAG_NEXOS_TMS | FPGA_PINNUM=P115
.....
: DIP Switch
Record=Constraint | TargetKind=Port | TargetId=SW[3..0] | FPGA_PINNUM=P29,P28,P27,P26
.....
: LEDs
Record=Constraint | TargetKind=Port | TargetId=LEDS[3..0] | FPGA_PINNUM=P74,P75,P76,P77
.....
: Speaker
Record=Constraint | TargetKind=Port | TargetId=SPKAEPR | FPGA_PINNUM=P110
.....
```

## 6 Adding the constraint file to a project

Once your constraint file is created and saved, you may add it to an existing FPGA project by right-clicking on the FPGA Project in the Projects Panel, and then choosing the **Add Existing To Project** command.



For more information about creating and using constraint files refer to the documents below. These PDF files can be opened from within the Altium Designer Documentation Library, accessed by pressing **F1** within the application, and browsing to the **FPGA Hardware Design » Working with Constraints** section.

- AR0124 Design Portability, Configurations and Constraints.pdf
- AP0104 Re-targeting the design to the Production Board.pdf
- TR0103 Constraint Sheet File Reference.pdf

## ▼ **Online third-party board support**

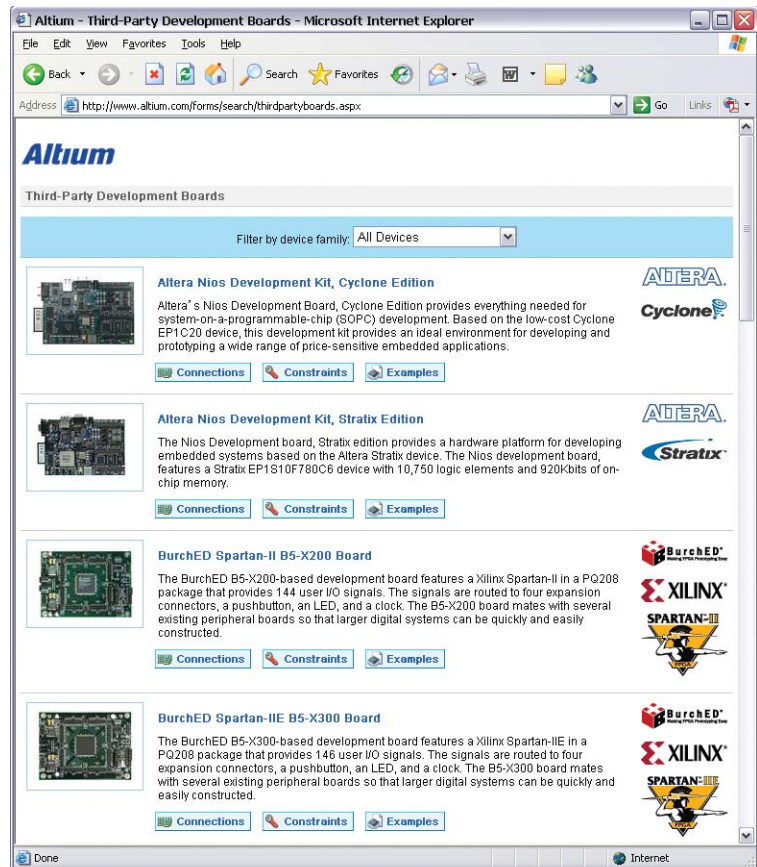
Altium's website contains a wealth of information on third-party development boards, with instructions on getting these boards up and running with Altium Designer.

Visit [www.altium.com/community](http://www.altium.com/community) and click through to Third-Party Development Board Support to obtain general instructions on:

- Connecting to boards
- Building a constraint file
- Building an example project

Altium also provides support for an extensive range of third-party development boards. This support can include instructions for connecting the board (including images), constraint files and example projects that demonstrate the use of processor cores and virtual instruments. To assist in finding a particular board, you can search by details such as manufacturer, device or board name.

This site is continuously being updated to provide relevant and up-to-date information on third-party development boards available. If a specific board is not listed, or level of support is limited, you can submit a request for future support.



## ▼ **More information**

Congratulations on successfully installing your Altium Designer LiveDesign Evaluation Kit. Now that you are ready to evaluate LiveDesign, we recommend 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 that once you have read the introductory documents, which will show you how to get started with Altium Designer and give you a good understanding of the software, that you also read the below 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
- GU0107 Using the TSK51x TSK52x Embedded Tools.pdf
- GU0108 Using the TSK165x Embedded Tools.pdf
- GU0109 Using the TSK80x Embedded Tools.pdf
- GU0110 Using the PowerPC Embedded Tools.pdf
- GU0111 Using the TSK3000 Embedded Tools.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 your third-party development board.

As well as the specific third-party development board resources and example projects provided on Altium's website, there are over 40 FPGA projects and reference designs included with this system. All example projects are available in the \Examples folder found in the Altium Designer installation directory. Like the documents mentioned above, these example projects have been configured to be used with the Nanoboard-NB1. Using the constraint file you created for your board, you should be able to retarget and reuse various parts of the designs.

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.**