Background Release

Altium European Media Contact

Monika Cunnington
PRismaPR
24 Oaktree Close
Letchworth, SG6 3XY
United Kingdom
www.prismapr.com
Telephone: +44 1462 64 07 79
Email: monika@prismapr.com

Altium Media Contact:

Alan Smith
Altium Limited
Level 3, 12a Rodborough Road
Frenchs Forest, NSW 2086
Australia
www.altium.com
Telephone: +61 2 8986 4409
Email: alan.smith@altium.com.au

Electronics design decisions linked in real time to the MCAD world; guided routing; custom instruments to test designs inside FPGAs; and much more: the latest in Altium's unified electronics design solution

Sydney, Australia – electronica 2008, Munich, Germany – November 11, 2008 – The latest version of Altium's unified electronics design solution introduces over 100 new features that unify still further the different design processes.

True collaboration with the mechanical world, for the first time

Electronics generally goes into some sort of casing or enclosure, but traditionally there has been minimal linking between electronics designers and mechanical designers. Getting the electronics to fit into the case has traditionally been a matter of good luck, rather than good management.

Altium changes this by offering a real solution to the problem of matching electronics design (ECAD) to the mechanical design (MCAD) processes of enclosure design. Electronics designers, for the first time, can now link directly to the mechanical CAD world in a non-proprietary way and unify ECAD and MCAD. Altium Designer's existing 3D board design capabilities have been enhanced to allow for direct linking to external STEP models, an intelligent 3D file format supported by all major MCAD software. This means that electronics designers can bring mechanical assemblies or designs done in MCAD software into Altium Designer. Features have been added to allow full interference/

clearance checking between any objects in the design, such as components and the case that surrounds them. Updates to the source STEP models are detected by Altium Designer, providing a dynamic link between the electronic and mechanical domains.

Altium Designer now reads and writes STEP files, so electronics designers can set up dynamic linking between the PCB and mechanical design packages (depending on the MCAD system being used).

Electronics designers can interactively adjust board layout, component placement and even component package choice to suit the proposed enclosure design. They can ensure that the PCB complies with mechanical clearance constraints – tested directly against the real enclosure design – before the board is sent for prototyping or manufacture. This can significantly reduce the number of design iterations necessary to close the ECAD-MCAD loop and make electronic product design easier.

Automatic previews with Design Insight

Altium Designer has a number of features under the banner of ‘Board Insight’ for the PCB design environment. Altium now extends these Board Insight principles with the initial set of Design Insight features.

A new Document Insight mode gives designers automatic previews of documents when they hover over entries in the Project Panel or Document Bar. Project Insight allows designers to preview all documents in a project by simply hovering over the project icon in the Projects Panel. They can then select any document from the preview to open or jump to.

Connectivity Insight lets designers hover over any net in their schematic source files and get a pop-up preview of all documents in the project connected to that net, as well as a list of connection nodes. The designer can click on any connection to jump straight to it.

In combination these new Design Insight features deliver faster, easier and more intuitive project navigation.

Design Insight raises project and document navigation, and information display, to the platform level, allowing designers to make the most of Altium Designer's unified environment. Features developed under Design Insight will work across multiple document types and across design editors to provide an environment-wide solution to the problem of managing increasing design complexity across multiple projects and design disciplines.

Better design publishing management with new design data publishing features

There are a great number of different files that need to be published in various forms throughout the lifecycle of an electronic product. These include manufacturing files, design files of various types, files used for documentation, specifications, bills of materials for procurement and various other data. The process of collecting and disseminating this information can create enormous amounts of work and lead to problems if the task is not well managed.

Altium simplifies the job of generating the right outputs with an Output Job editor. This provides a centralised interface from which to define all required outputs. The information is stored at the project level, ensuring designers can recreate any necessary output files at any time.

Altium now makes generating different types of output easier with the addition of Output Media options in the Output Job editor. This lets designers consolidate multiple outputs into a single media type. For example, a designer can put the schematics, composite PCB layout and bill of materials (BoMs) into a single PDF output.

Multiple Media Outputs of the same type can be defined and named to allow outputs to be grouped and generated in a variety of ways.

Designers don't now need to generate different output files separately, and then manually consolidate them into a final output. They can define exactly the information they want to print or publish to PDF and then generate the output as a single operation.

Manufacturing discrepancies removed with new representation of internal planes

As new technologies emerge and new design techniques move into the mainstream, Altium helps designers to harness these developments.

Internal planes are becoming much more common in all areas of designs. Their use is becoming more sophisticated and therefore designers need the capabilities to work with complex plane configurations. In Altium Designer, plane support has been significantly upgraded and enhanced.

Internal planes are now modelled and displayed in 2D and 3D with 100% accuracy in the PCB editor. Also, the DRC now includes live connectivity checking of planes and detects net breaks caused by inadvertent plane splits, isolated pads and vias, and starved thermal relief connections.

The end result is that designers can create internal signal planes in their designs and work with the confidence that what they see and verify in the PCB editor will translate into a correct set of manufacturing files. This eliminates discrepancies between design and manufacturing files and gives designers a much higher degree of visibility into the final shape, connectivity and electrical integrity of their plane layers. Errors going to manufacture are reduced, and there are fewer prototype spins overall.

Improved management with new version control system integration

Altium solves the problem of comparing different versions of graphical files as they are worked on and committed to the system by building integration to version control directly into the design interface, and providing specific features to allow the direct comparison of graphical files.

The powerful differencing engine within Altium Designer can highlight the smallest of graphical changes between different versions of schematic or PCB files, even when these changes have no effect on the connectivity of the design.

Altium has enhanced version control support to allow background fetches and updates of design files from the version control library. Designers can stay focused on the task at hand without having to wait for updates to finish: a great time saver with larger projects. And designers can now update all working file copies within their project with a single command, making it much easier to manage large scale changes.

Test inside the FPGA with new custom virtual instrument component

Altium believes that the future of electronics design lies in moving the implementation of core system functionality from fixed hardware and into ‘soft’ device intelligence that is programmed, rather than manufactured, into a product. Moving hardware functionality inside devices such as FPGAs, however, means that traditional circuit test methods using physical connections to signals travelling between components are no longer viable.
In traditional FPGA design flows, simulation has often been seen as a replacement for direct testing. But simulation is extremely difficult and time consuming when applied at the system level.

To solve this problem, Altium introduces the concept of virtual instrumentation. FPGA-based components carry out the typical functions found in bench-top test instruments. The hardware portion of the instrument is connected into the design at the schematic level and downloaded to the FPGA along with the circuit under development. The instrument is then controlled using soft front panels available within Altium Designer.

Altium Designer has a new dimension to virtual instrumentation with the introduction of a new Custom Instrument. Designers can now construct their own instrument to monitor and control signals within the FPGA. They choose the number and type of inputs and outputs they want, create custom scripts (in DelphiScript) to process the signals or respond to events as required, and build a custom interface or GUI for their instrument from a palette of standard components and instrument controls.

One interesting feature of the custom instrument is that the instrument interface can be stored as part of the design that is downloaded to the FPGA. This means that designers can create full custom test or maintenance interfaces for a product and store it inside the design. Any service engineer can then plug in to the design using Altium Designer, and instantly have access to the custom instrument panel without having to have a copy of the instrument definition stored on their computer.

C-based custom FPGA logic development

Altium Designer comes standard with a large number of custom FPGA-based functionality available as royalty-free FPGA-based components. Components range from basic logic

blocks right through to sophisticated peripherals and processor cores. These components can be used to rapidly construct FPGA-hosted systems.

To extend the supplied functionality, Altium Designer supports the design of custom logic blocks. Previously this involved using the FPGA library components to create custom logic functions at the schematic level, or defining custom logic using the Verilog or VHDL Hardware Description Languages, or indeed a combination of all three.

Altium Designer now allows custom logic to be defined using the C programming language. A new type of sheet symbol, a C Code Symbol, has been added to the system to allow blocks of C code to be added within the schematic design hierarchy. The C Code Symbol refers to underlying C source code in the same way that Verilog or VHDL code is incorporated into a design. The ports on the symbol represent the parameters referenced in the underlying source code.

When the design is compiled, C code is translated to VHDL using the unified hardware/software compiler technology (or C-to-Hardware technology) and then synthesised to FPGA along with the rest of the design. Features have been added to the system to help designers generate C code templates from a defined C Code Symbol, or to generate a C Code Symbol from underlying source code.

Altium's unified hardware/software compiler is also used to generate application-specific coprocessor functionality directly from the C source code targeted for the system processor, allowing developers to accelerate code execution by running some functions directly in hardware. Altium now extends the opportunities available to embedded software developers by allowing them to write custom blocks of logic using C code and 'wire' these directly into the underlying system hardware.

Embedded developers can create application code, and participate in the creation of the hardware that will run that code, without having to learn new languages or development techniques. They can remain in the comfort of the C coding space, but extend their design reach to a much broader playing field.

Rapid design of FPGA-based systems with custom wishbone interface component

To provide an easy, modular way to rapidly build FPGA-based system designs, Altium Designer uses the non-proprietary Wishbone bus interface to ‘connect’ together the various FPGA-based peripheral component blocks available for system construction. A new custom Wishbone Interface component has been added to Altium Designer that can be easily configured to link custom peripherals to the Wishbone bus. This allows designers to extend the functionality supplied with Altium Designer and create or import custom functionality that can be easily combined with Altium Designer’s in-built components. Designers can expand beyond the peripheral devices supplied with Altium Designer, to bring in their own devices to perform whatever tasks they require. This greatly increases the scope of applications that can be developed in Altium Designer and opens the system to designers to build their own custom peripheral libraries, or indeed to develop custom peripherals that can be used by others.

Faster board routing with new interactive routing engine

Routing of connections is a major factor in determining the time taken to complete a custom board design. Board densities are on the increase, as are layer stacks and the prevalence of dense packaging technologies, making the routing job even trickier. Full automatic routing technologies provide a viable solution in a relatively limited set of scenarios. The biggest impact on routing, and therefore design time, is to be made by improving the interactive routing technologies that aid rather control the routing process.

Altium has introduced a new interactive routing engine to do this.

Basic operation modes include fast and robust push-and-shove of tracks and vias, guided routing mode for rapid trace placement with a minimum of mouse clicking, automatic hugging of existing traces when routing, and improved route auto-completion. Modes can be used independently or in combination to provide excellent control in all routing situations.

The cursor-guided routing mode, in particular, makes a huge difference to routing efficiency. It allows a designer to ‘guide’ the routing, rather than having to place and anchor each trace segment along the way. The engine intelligently places track segments

under full design rule compliance to follow as closely as possible the movement of the cursor. Routing can be undone by simply ‘backing up’ over the previous path.

Altium offers, as standard, interactive length tuning for single nets and differential pairs, full differential pair support system wide, impedance-controlled routing, multi-trace routing, pin and part swapping, automatic FPGA pin optimisation for routing, and the most attractive and intuitive interface going.

The latest release of Altium Designer is available now. Contact Altium, or go to www.altium.com/summer08 to see Altium’s unified design solution in action, and to book a live web demo.

ENDS

About Altium
Altium Limited (ASX:ALU) provides world-leading unified design solutions that break down the barriers to innovation, and help organisations easily harness the latest devices and technologies, to create their next generation of electronic products.

Altium’s solutions are unique because they unify the separate processes of electronics design, all within a single electronics design environment, working off a single data model, which links all the aspects of electronics product design into one process.

Founded in 1985, Altium has headquarters in Sydney, Australia, sales offices in the United States, Europe, Japan, China, and resellers in all other major markets. For more information, please visit www.altium.com.

Altium, Altium Designer, LiveDesign, and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners, and no trademark rights to the same are claimed.