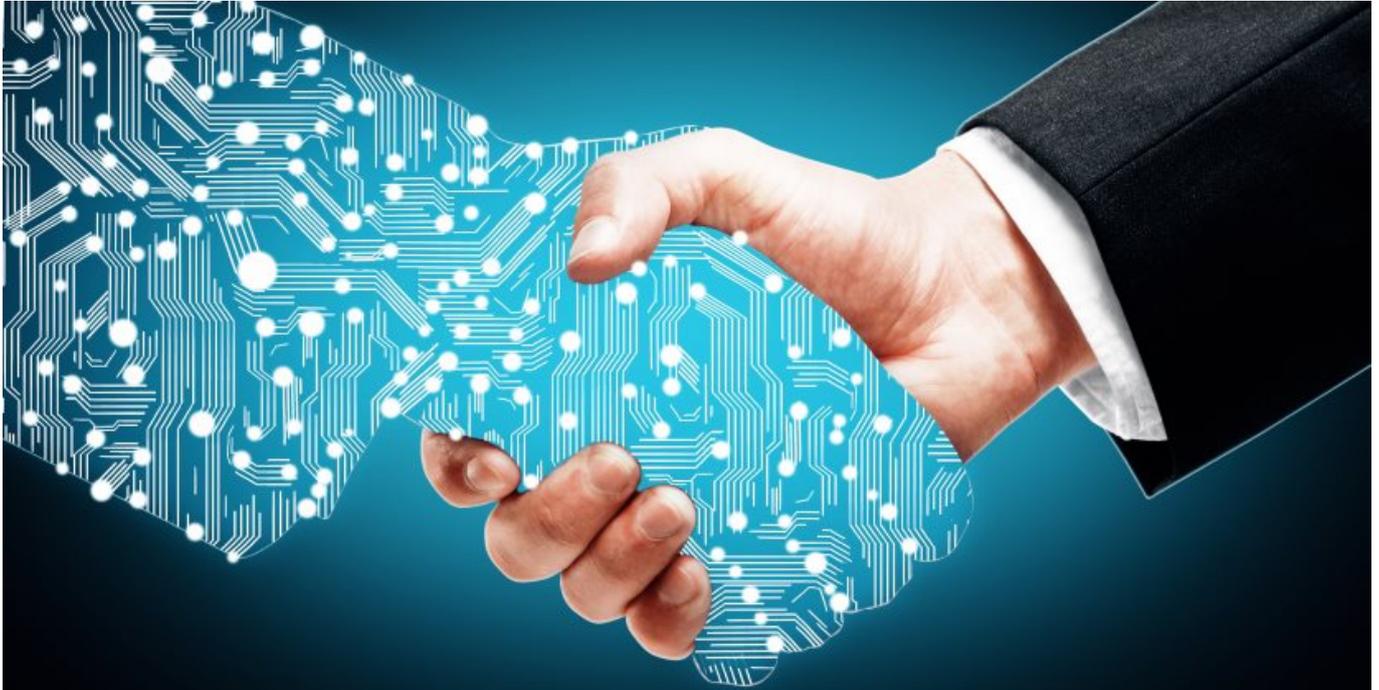


## Using Altium Documentation

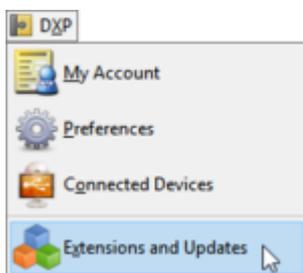
Modified by Jason Howie on May 21, 2018



Altium Designer is probably just one of many engineering design environments that you work with, or need to interface to.

Perhaps you need to interface to another design and analysis platform to perform a power simulation, or perhaps you need to import an older design that was developed in another design environment. Whatever the reason, it's likely that one day you'll need to be able to interface to another design tool.

## Interfacing to another Design Tool



Interfacing to another design environment is all about transferring design data. That can either be done by reading/writing the design data from/to a different file format, or directly transferring data via software API's.

In Altium Designer, transferring design data, either into a file or directly into another design tool, is done via an [Extension](#). All extensions are installed via the **DXP » Extensions and Updates**

command. This command opens the **Extensions** view, which, as well as being used to add/remove/update the software and the available extensions, is also the home of license management.

The following types of extensions are supported:

- Platform extension - smaller extensions, such as loading older format files, are delivered through the standard software installer, click  on the Extensions and Updates page to access them.
- Software extension - larger and more sophisticated extensions, including those developed by 3rd parties. These are accessed via the **Purchased** tab of the Extensions and Updates page.
- Hardware Device extensions - a specialized extension to support a specific physical device family, for use in soft design.

Learn more about [Extending Altium Designer](#)

There are also System extensions - this type of extension bring new design capabilities into the Altium Designer environment.

## Performing an Import or Export

Import actions are performed via the **File » Import** command, or the **File » Open** command. Alternatively, many of the importers can be accessed via the [Import Wizard](#). Using the Wizard allows related files, such as design files and their libraries, to be batch imported.

Export actions are performed via the **File » Export** menu, in the appropriate editor.

## Design Extensions

These extensions add functionality to Altium Designer.

Name	Comments
AutoCAD® DWG/DXF (*.dwg, *.dxf)	Import & export DXF and DWG format files, up to AutoCAD 2013. #
Specetra® Autorouter	Import & export Specetra DSN & RTE format design files. #
BQR fiXtress™	Use BQR fiXtress to help you to design reliable electronics systems without expensive re-spins or physical prototypes.
Webench® Power Designer	Accurately design and simulate your power supply directly in Altium Designer with Texas Instrument's WEBENCH Power Designer.
Aldec® Active-HDL Simulator	Accurately simulate your next FPGA design directly in Altium Designer with Aldec's VHDL and Verilog simulation tool.
JTAG Maps™	The free JTAG Maps extension helps hardware engineers assess the potential of using Boundary scan testing in their designs and prepares the JTAG test data upfront.

Name	Comments
XJTAG® DFT	The free XJTAG DFT Assistant extension allows the board designer to assess and improve the testability of their PCBs throughout the design process.
Solidworks® PCB Connector	Take the guesswork out of your collaboration process with an integrated link between Altium Designer and your MCAD environment in SOLIDWORKS®.

# Platform extension. The others are Software extensions.

Learn more about the available design extensions in the [Altium Products](#) area.

## Extensions to interface to other EDA Design Tools

The following interfaces are delivered either as a platform extension or a software extension. These extensions add the following file import/export functionality to Altium Designer.

Name	Comments
Protel / Altium	<b>Import</b> All previous Protel/Altium Schematic files/libraries All previous Protel/Altium PCB files/libraries Protel 99SE Design Database (*.ddb) CircuitMaker 2000 Schematics (*.ckt) CircuitMaker 2000 User Libraries (*.lib) CircuitMaker 2000 Device Libraries (*.lib) CircuitMaker (current), CircuitStudio, PCBWorks PCB files <b>Export</b> Protel 2.8 ASCII
P-CAD®	<b>Import</b> Tango PCB ASCII files (*.pcb) P-CAD V16 or V17 Binary Schematic design files (*.sch) P-CAD V16 or V17 ASCII Schematic design files (*.sch) P-CAD V15, V16, or V17 Binary PCB design files (*.pcb) P-CAD V15, V16, or V17 ASCII PCB design files (*.pcb) P-CAD V16 or V17 Binary Library files (*.lib) P-CAD V16 or V17 ASCII Library files (*.lia) P-CAD PDFIF file (*.pdf) <b>Export</b> P-CAD ASCII PCB
CadSoft® Eagle®	<b>Import</b> ( <a href="#">XML format v6.4 or later</a> ) Eagle Schematic (*.sch) Eagle PCB (*.pcb) Eagle Library (*.lbr)

Name	Comments
Cadence® OrCAD®	<b>Import</b> ( <a href="#">up to OrCAD version 16.xx</a> ) OrCAD Capture Designs (*.dsn) OrCAD Capture Libraries (*.olb) OrCAD Layout PCB documents (*.max) OrCAD PCB Libraries (*.llb) OrCAD CIS Configuration file (*.dbc) OrCAD SDT (*.sch) <b>Export</b> OrCAD SDT Schematic (*.sch) OrCAD v7 Capture Design (*.sch)
Cadence® Allegro®	<b>Import</b> Allegro binary PCB (*.brd) Allegro ASCII PCB (*.alg)
Mentor Graphics® PADS®	<b>Import</b> PADS Layout ASCII Design files (*.asc) PADS Layout ASCII Decal Libraries (*.d) PADS Logic ASCII Design files (*.txt) PADS Logic ASCII CAE Decal Libraries (*.c) PADS Logic ASCII Part Type Libraries (*.p) <b>Export</b> <a href="#">PADS Logic 5</a>
Mentor Graphics® DxDesigner®	<b>Import</b> DxDesigner Designs and Libraries
Mentor Graphics® Expedition®	<b>Import</b> Expedition files (*.pcb, *.lib)
Zuken® CADSTAR®	<b>Import</b> CADSTAR Schematic Archive (*.csa) CADSTAR PCB Archive (*.cpa) CADSTAR Part Library (*.lib)



## Getting Help

If you're reading this page because you're new to Altium Designer, here's some tips to help you get started:

- If you're moving from another design environment, you can [start Exploring Altium Designer here](#), or dive straight into the [go-to-whoa introductory tutorial](#).
- Use the resources available in this documentation space ([www.altium.com/documentation](http://www.altium.com/documentation)), where you can either search via the field above, or browse using the navigation tree on the left.
- Whatever type of document you are editing, you can press **F1** over an object, editor, panel, menu entry or button to access reference information about that item.
- Press **Shift+F1** while running a command, for a list of shortcuts you can use in that command.
- Explore the [video library](#), there's lots of them to watch, each short video details the exact



steps to complete a task.

- Attend a [webinar](#) or another of Altium's [learning events](#).
- [Join a discussion forum](#) , where you can share with and learn from your industry peers.

---

**Source URL:**

[https://www.altium.com/documentation/display/ADES/Altium+Designer+-+\(\(Interfacing+to+other+Design+Tools\)\)](https://www.altium.com/documentation/display/ADES/Altium+Designer+-+((Interfacing+to+other+Design+Tools)))