Importing and Exporting Design Files

Altium Designer incorporates a wide variety of importer and translator technologies, allowing you to easily import designs originating from previous versions of Altium software, or alternative software packages.

> To make use of the importer/exporter technologies available in Altium Designer, you must install the relevant plugins. These can be found in the **Importers and Exporters** category of the **Plugins** view (**DXP»Plugins and Updates**).

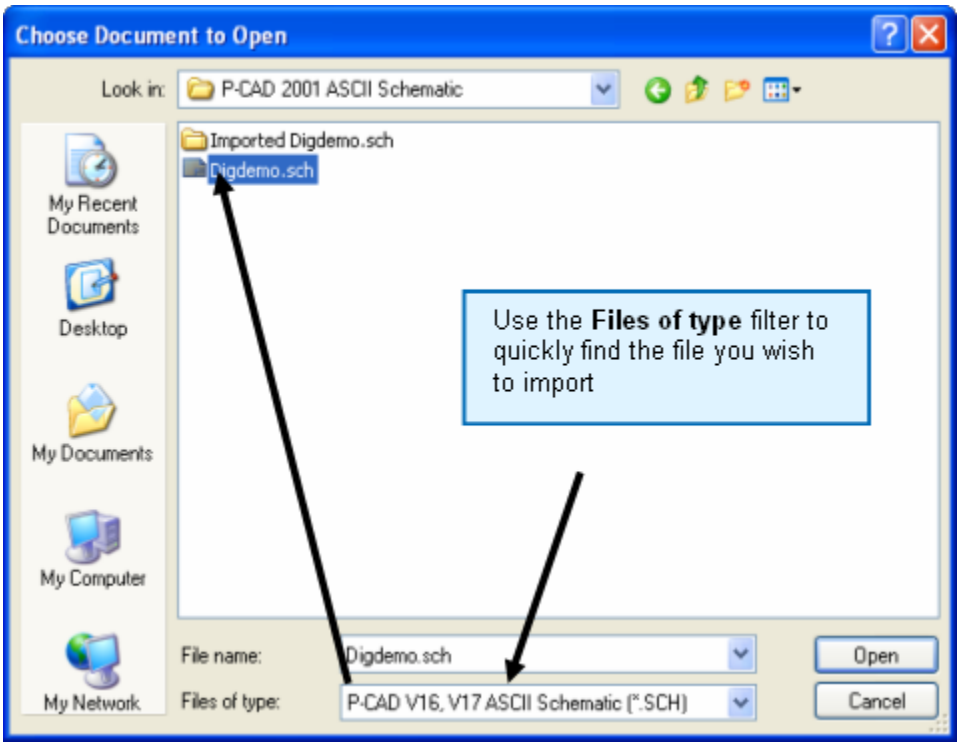
Supported Import File Formats

The following provides an overview of what file types can be imported into Altium Designer. The method of import may differ between different file types, and are detailed in the sections thereafter.

- All previous Protel/Altium Schematic files/libraries
- All previous Protel/Altium PCB files/libraries
- Protel 99SE Design Database (*.ddb)
- 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 PDIF file (*.pdf)
- Tango PCB ASCII files (*.pcb)
- CircuitMaker Schematics (*.ckt)
- CircuitMaker User Libraries (*.lib)
- CircuitMaker Device Libraries (*.lib)
- 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)
- 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)
- DxDesigner Designs and Libraries
- Expedition files (*.pcb, *.lib)
- CADSTAR Schematic Archive (*.csa)
- CADSTAR PCB Archive (*.cpa)
- CADSTAR Part Library (*.lib)
- Allegro binary PCB (*.brd)
- Allegro ASCII PCB (*.alg)
- Specctra Design file (*.dsn)
- AutoCAD DWG/DXF (*.dwg, *.dxf)

Importing via the File»Open Command

A range of file types can be imported via the *Choose Document to Open* dialog, accessed through **File»Open**. Set the **Files of type** option to the required (and supported) file format.



Importing documents using the Choose Document to Open dialog.

> Some file types cannot be imported using the **File»Open** command, and are only imported through use of the Import Wizard. It is advised to use the Import Wizard in the first instance, especially when importing both design files and their related libraries into a new Altium Designer project and its constituent documents.

Importing into the active document using the File»Import command

Certain file types can be imported directly into the active schematic or PCB document using the **File»Import** comma nd.

Schematic

- AutoCAD DWG/DXF (*.dwg, *.dxf)

PCB

Use the *Import File* dialog to specify the file type to import from one of the following:

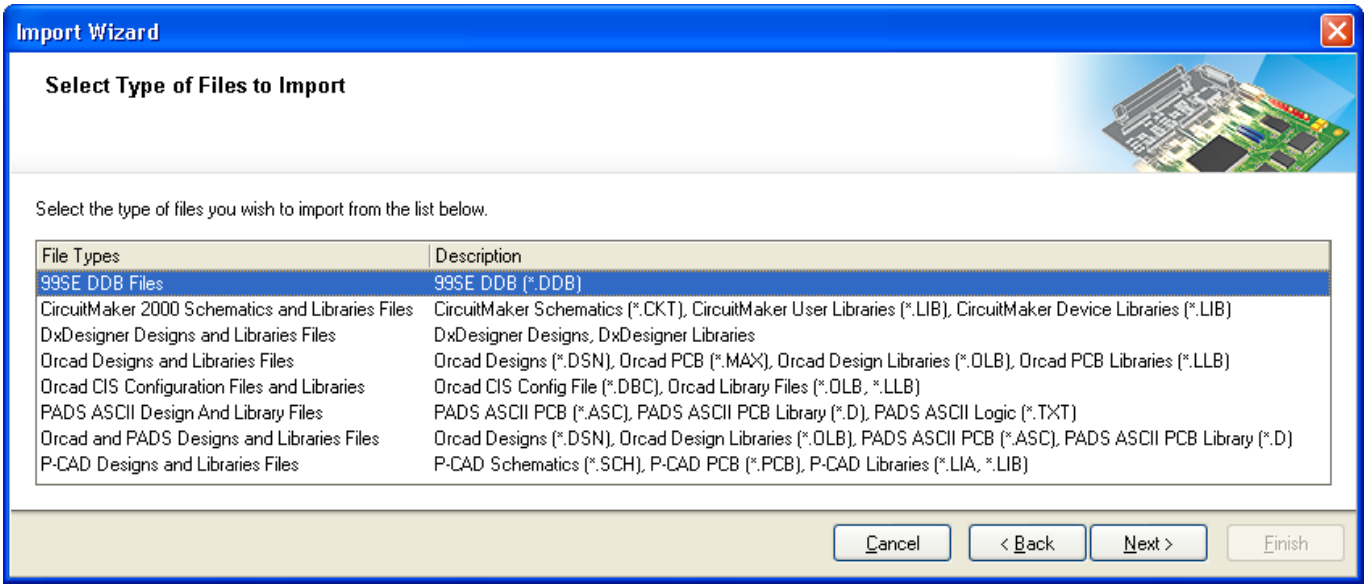
- AutoCAD (*.dxf, *.dwg)
- Gerber Batch (*.g??, *.g?)
- P-CAD PDIF files (*.pdf)
- SDRC-IDF Board files (*.brd)
- Single Gerber files (*.g*)
- Specctra Route file (*.rte)
- Tango PCB ASCII file (*.pcb)

Importing via the Import Wizard

Altium Designer also includes a unified means of importing designs from a variety of different design tools, using the **Import Wizard**. The wizard walks you through the import process, handling both the schematic and PCB parts of the project, as well as managing the relationship between them. Select **File»Import Wizard** to launch the Wizard. The **Import Wizard** can be used to import the following types of design files:

File Types	Description
Protel 99SE DDB Files	99SE DDB (*.ddb)
Cadence® Allegro® Design Files	Allegro binary PCB (*.brd), Allegro ASCII PCB (*.alg)
Zuken CADSTAR® Designs and Libraries	CADSTAR Schematic Archive (*.csa), CADSTAR PCB Archive (*.cpa), CADSTAR Part Library (*.lib)
CircuitMaker 2000 Schematics and Libraries	CircuitMaker Schematics (*.ckt), CircuitMaker User Libraries (*.lib), CircuitMaker Device Libraries (*.lib)
Mentor Graphics® DxDesigner® Designs and Libraries	DxDesigner Designs, DxDesigner Libraries
Mentor Graphics® Expedition® Designs and Libraries	Mentor Expedition files (*.pcb, *.lib). Note, this feature is currently part of the Altium Labs
Cadence® OrCAD® Designs and Libraries	OrCAD Capture Designs (*.dsn), OrCAD Layout PCB documents (*.max), OrCAD Capture Libraries (*.olb), and OrCAD PCB Libraries (*.llb)
Cadence® OrCAD® CIS Configuration Files and Libraries	OrCAD CIS Configuration file (*.dbc) and OrCAD Library files (*.llb, *.olb)
Mentor Graphics® PADS® Layout and PADS® Logic Designs and Libraries	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)
Cadence® OrCAD® and Mentor Graphics® PADS® Designs and Libraries	OrCAD Capture files (*.dsn), OrCAD Schematic Library files (*.olb), PADS Logic ASCII Design files (*.txt), PADS Layout ASCII Decal Libraries (*.d), PADS Logic ASCII CAE Decal Libraries (*.c), PADS Logic ASCII Part Type Libraries (*.p)

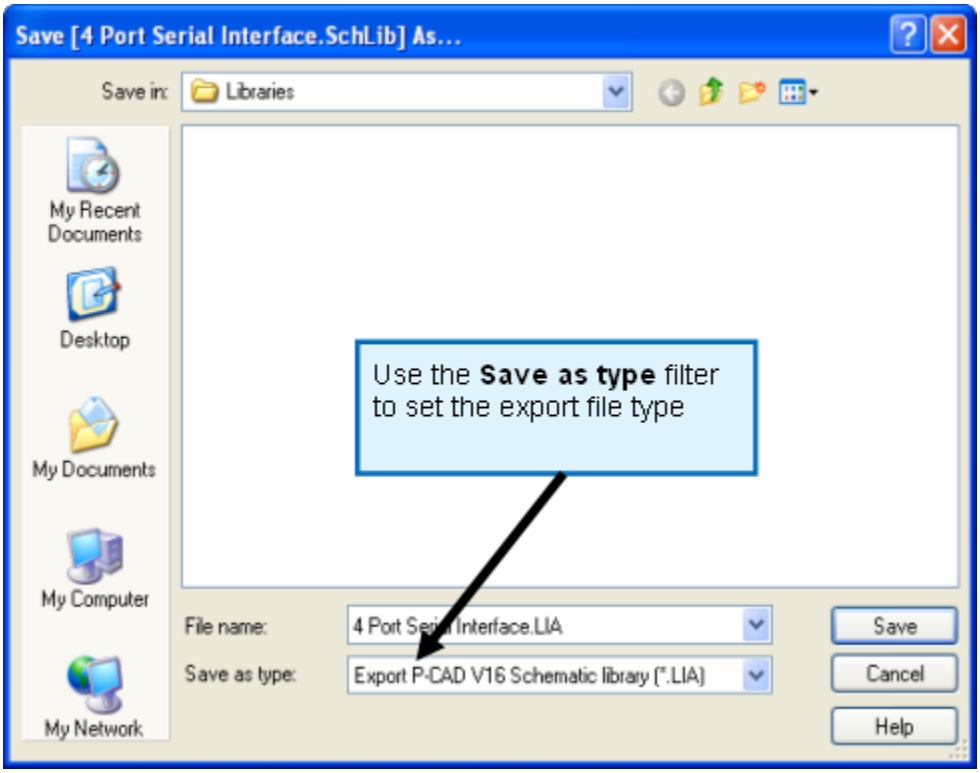
P-CAD Designs and Libraries	P-CAD V16 or V17 Binary Schematic design files (*.sch), P-CAD V16 or V17 ASCII Schematic design files (*.sch), P-CAD V15, V16, or V17 Binary PCB design files (*.pcb), P-CAD V15, V16, or V17 ASCII PCB design files (*.pcb), P-CAD V16 or V17 Binary Library files (*.lib), P-CAD V16 or V17 ASCII Library files (*.lia)



The Import Wizard supports importing from a large variety of design tools.

Exporting Design Files

Schematic and PCB design files can be exported into various formats – including previous versions of Altium's software or alternate software packages. PCB assemblies can also be exported as 3D STEP model files for use in mechanical CAD packages. Export of files is performed using the **File»Save As** command, or the **File»Save Copy As** command.



Exporting design files using the File»Save As command.

Supported Export File Formats

Schematic Editor

- Altium Schematic Binary (*.SchDoc)
- Altium Schematic ASCII (*.SchDoc)
- Altium Schematic binary 4.0 (*.sch)
- Altium Schematic ASCII (*.SchDoc)
- OrCAD SDT Schematic (*.sch)
- Altium Schematic Template (*.SchDot)
- AutoCAD DWG/DXF (*.dwg, *.dxf)

PCB Editor

- Altium PCB Binary (*.PcbDoc)
- Altium PCB binary V3.0, V4.0 (*.PCB)
- Altium PCB binary V5.0 (*.PcbDoc)
- Altium PCB ASCII (*.PcbDoc)
- Protel Netlist (*.net)
- AutoCAD DWG/DXF (*.dwg, *.dxf)
- HyperLynx (*.hyp)
- P-CAD ASCII (*.pcb)
- Protel PCB 2.8 ASCII (*.pcb)
- Protel Netlist (*.net)
- Specctra Design File (*.dsn)
- SDRC-IDF Brd Files (*.brd)
- STEP (*.step, *.stp)

- SiSoft Files (*.csv)
- Ansoft Neutral File (*.anf)

Schematic Library Editor

- Altium Schematic Binary Library (*.SchLib)
- Altium V4.0 Schematic Binary Library (*.lib)
- OrCAD Capture Schematic Library (*.olb)
- P-CAD V16 Schematic Library (*.lia)

PCB Library Editor

- Altium PCB Library (*.PcbLib)
- Altium V3.0, V4.0 Library (*.lib)
- Altium V5.0 Library (*.PcbLib)
- P-CAD V16 PCB Library (*.lia)

See Also

The Altium Designer Environment

Moving to Altium Designer from Cadence Allegro PCB Editor

Moving to Altium Designer from Mentor Graphics DxDesigner

Moving to Altium Designer From OrCAD

Moving to Altium Designer from PADS Layout and OrCAD capture

Moving to Altium Designer from Pads Logic and PADS Layout

Moving to Altium Designer From P-CAD

Moving to Altium Designer from Protel 99 SE

P-CAD to Altium Designer Terminology Guide