

Vergleichstabelle Altium Designer vs PCB Library Expert




Footprint Wizard

PCB Library Expert

	Footprint Wizard	PCB Library Expert
Typical time for updates	mostly same since original	every 1-2 weeks
IPC Compliance	yes, unknown revision	IPC-7351 - 2017
Request a new part, regardless how complex	-	\$5
Parts on Demand (1 million+ parts)	-	✓
Non-standard unique packages	-	✓
Batch Build per User Preferences - unlimited part libraries	-	✓
Footprint and 3D Model auto-generated simultaneously (Altium and 3D modules required)	-	✓
On Line DRC checking	-	✓
Automatic pad trimming to meet DRC rules	-	✓
Footprint rotation	-	✓
Build/modify footprint in a single menu	-	✓
Visually see component and footprint	-	✓
Mirror to see bottom during construction	-	✓
Print	-	✓
Automate manufacturer recommended patterns	-	✓
View library manager and calculator simultaneously	-	✓
Automated rounded rectangle pad shape	-	✓
Support for connectors	-	✓
Selectively delete pads with a click	-	✓
Rename/renumber pins	-	✓
Set, manage, and modify User / Default Preferences	-	✓
Solder Joint Statistical Analysis	-	✓
See/verify the Footprint before you add to Library	-	✓
Query Padstack Results prior to creation	-	✓
PTH Padstack Calculator	-	✓
Through-hole component families	-	✓
Paste Mask reduction percentage for Thermal Pads	-	✓
Auto-trim pads under low profile packages	-	✓
Auto-Silkscreen trimming around pads	-	✓
Auto-generation of Description and Height attributes	-	✓

Other (more detailed) noteworthy differences: Altium Designer's footprint wizard has 6 dialog windows to create a BGA, and 11 to create a QFN, the Library Expert simplifies the steps for all components to *one* simple window.

The Library Expert supports Plated Through-hole Axial, Radial, Headers, DIP, SIP, Sockets, Oscillators, Connectors, Surface Mount Crystals, Side Concave or Corner Concave or J-Lead or L-Bend Oscillators, Side Concave of Flat Lead Chip Arrays, Land Grid Arrays, Column Grid Arrays, Dual Flat No-Leads (DFN), SOTFL, SODFL, SOD, DPAK remove, PQFN and the Altium Wizard does not. Also, the Library Expert allows you to get both Footprint and the 3D model in Altium so you can see the component superimposed on the footprint for verification/analysis. The Library Expert offers other very powerful PCB library management features such as interactive graphic editing to delete or hide pins and rename pins.