



Library Expert 2014

- 70 new functions
- 117 FIXED / ENHANCED

Library Expert 2014.11 Released!!	1
Library Expert 2014.10 Released!!	2
Library Expert 2014.09 Released!!	2
Library Expert 2014.08 Released!!	3
Library Expert 2014.07 Released!!	3
Library Expert 2014.06 Released!!	4
Library Expert 2014.05 Released!!	5
Library Expert 2014.04 Released!!	5
Library Expert 2014.03 Released!!	6
Library Expert 2014.02 Released!!	7
Library Expert 2014.01 Released!!	7

Library Expert 2014.11 Released!!

FIXED / ENHANCED:

- Expedition - an issue in the CAD tool export generator that crashed the program
- PADS - a problem when changing the decal name to a custom name in single part mode
- FP Designer:
 - An issue in dealing with pins on the bottom layer
 - Pad Stack Manager
 - Ability to reassign pin numbers to pad stacks
 - Custom through-hole pad stack creation
 - Thermal Pad Paste Mask (ability to define the checker board size and gap)
- Calculator:

- SOT Component Family – resolved Gang Mask issue related to hidden/missing pins
- Library Editor:
 - Fixed regression issue with Tools > Regenerate > Name & Description
-

Library Expert 2014.10 Released!!

NEW:

- The .10 release is now available to all Lite Users as the default download
 - You must Opt In to our newsletter to receive V2015 releases - [opt in here](#)
- Calculator:
 - Added two new component families “Resistor” & “Fuse” to the Side Concave (2, 4 pins) calculator
 - Added a new component family “Fuse” to the MELF calculator
- 3D STEP
 - Added 3D STEP models for the new Calculator component families

FIXED / ENHANCED:

- Code Update - removed new code that was causing Library Editor issues
- Calculator:
 - Turning off all layers in the Layer Display dropdown threw an Exception Error
- EAGLE:
 - Issue that was including Paste Mask and removing Solder Mask for local fiducials to be included should none exist
- Altium/PADS:
 - Issue that was causing the .ASC files not to import. This bug fix will also add back in the Family (BGA, etc.) if importing ASCII into PADS
 - Issue that was causing a batch build to drop part type names from the parts
 - Issue when dealing with missing pins
- P-CAD/Design Spark/Pulsonix:
 - A layer issue that was preventing the file from loading in PCAD
- Library Editor:
 - Issue when deleting rows in an FPX file
 - Issue when pasting an item from one file into another that contained that same item
 - Sample.fpx was updated
 - Issue with editing the Manufacturer cell

Library Expert 2014.09 Released!!

FIXED / ENHANCED:

- Allegro/OrCAD PCB:
 - The all color UI visibility settings in the script have been disabled per feedback. Newer versions of Allegro/OrCAD PCB had issues, older ones didn't care if it wasn't there, so it's been disabled.
- PADS Layout:
 - Removed Family Name from the Part Type in the .ASC output for V9.x
 - Removed Alphanumeric Names from the Decal in the .ASC output. They're in the Part Type and PADS gets confused if they're in both sections.
 - In the Attributes section of the .ASC file, it now properly calls out the Part Type Name, rather than the Decal Name
- OrCAD Layout & SoloPCB:
 - .MIN format – Pad Stacks now properly display their Non-Plated value
- Eagle:
 - All Paste Mask and Solder Mask is now generated to and matches the exact values specified in FPX. If you want the Solder Mask to expand, you'll need to specify it.
- Calculator:
 - LGA & CGA were not displaying the Environment
 - Display Grid was updated so that whatever the last grid setting used would be kept upon save
 - The graphic image for all pull-back lead component families was updated
- Library Editor:
 - Removed the Description auto-complete function

- Enhanced the Manufacturer Name auto-complete function. Right Mouse Click on the Manufacturer cell and a mfr. list will appear. Type some characters.
- When a footprint name was missing on a "Save FPX" the FPX format changed to V2015. To convert back to the 2012 format use File > Save As.
- General:
 - Issue that was causing the program to save in 2015 format even though there was no custom data
 - Issue that was causing duplication errors when dealing with a Footprint Name that started with the same string as another Footprint Name

Library Expert 2014.08 Released!!

FIXED / ENHANCED:

- Library Editor:
 - Library Editor "Sort Column Data" feature was broken
 - Updating FPX file Rows throws error messages
 - Editing "Physical Description" column caused problems
- General:
 - An issue in dealing with placeholder pins (i.e. removed/hidden) that was crashing the tool
 - Trailing placeholder pins (i.e. removed/hidden) are now clipped after the model has been made
- FP Designer:
 - Enhanced the pad stack manager

Library Expert 2014.07 Released!!

NEW:

- FP Designer: New custom Pad Stack Manager
 - Chamfer or radius any SMD pad corner
 - Create custom through-hole pad stacks for outer and inner layers, plane thermal relief and anti-pad
 - Offset slotted holes
 - Assign pad stacks to pin numbers
- New FPXC to CAD Translator:
 - This feature was removed and FPXC was converted to FPX. Now all Unique footprints are stored in our new V2015 FPX format.
 - New "Sample Data - V2015.fpx" file for unique packages now comes with installation
- Altium:
 - PADS ASCII files for Altium now produces "Plated" surface mount pads to avoid DRC errors

FIXED / ENHANCED:

- Calculator:
 - TO DPAK issue when saving to FPX and retrieving from Library Editor removed a pin
 - TO220 Horizontal issue when saving to FPX and retrieving from Library produced the Vertical dimensional picture
 - Rename pins when Mirror was turned on threw an unhandled exception error
- User Preferences:
 - Ability to turn off Local Fiducials
 - In the new Chip micro-miniature parts, the Courtyard Excess was not retaining values that were changed
- Library Editor:
 - The auto-generation of Physical Description and Manufacturer Name was causing problems when the cells were edited
- 3D STEP:
 - The Chip Family Types Capacitor, Fuse, Thermistor, and Varistor all use the same base 3D model. They differ by default body color(Orange, Green, Blue, and Black.. respectively). That model has also been updated to a nicer form.
 - The Chip Family Type Capacitor, Non-Polarized has been renamed to Capacitor
 - The Chip Family Type Capacitor, Polarized now has the 3D model of Chip, Wire Rectangular. Chip, Wire Rectangular has been deprecated. Use Chip, Polarized for any Chip, Wire Rectangular applications.
- CAD Tool Interfaces:

- Eagle:
 - The paste mask flag as it concerns through-hole parts
 - Through-hole pad has a drill size bigger than the pad, it's now turned into a Hole. This should make it easier to see.
 - P-CAD
 - Problem with the default Pad Stack. Files should import as usual again
- FP Designer:
 - Was not producing a "Body Outline" for IDF 3D extrusion
- PADS to CAD Translator:
 - Problem in the detection of Squares and Rectangles

Library Expert 2014.06 Released!!

NEW:

- Calculator
 - Ability to set tab stops for entering dimensions in either nom/tolerance or min/max
 - New Component Families
 - Side Concave Inductors
 - DFN 2-Pin Fuse
 - Updated Toe Solder Joint Goals for Chip component less than 1.6 mm in length per the upcoming IPC-7351C
 - 0402 Toe is now set to 0.15 mm
 - 0201 Toe is now set to 0.12 mm
 - 01005 Toe is now set to 0.10 mm
 - Updated Periphery Solder Joint Goals for DFN component less than 1.6 mm in length per upcoming IPC-7351C
 - Periphery Goal for these micro-miniature bottom only terminal packages is now set to -0.04 mm
- FP Designer
 - Offset slotted holes
 - Side Chamfered Rectangle Pads that are new in the Pad Stack Designer now export to CAD tools properly
 - Issue related to the offsetting of pads. It should now translate properly the new offset functionality in the Pad Stack Designer.
- New FPXC to CAD Translator
 - Added 3D STEP model output from FPXC files from the translator as an extruded block
 - PADS to CAD was changed to FPXC to CAD. Now converts our new neutral library format to any CAD tool.
 - FPXC to CAD now has a new Viewer to visually see the footprint features prior to importing it into their library
 - FPXC to CAD was relocated from the Main Toolbar to the "Utility Tools" drop down
- 3D STEP
 - Added models for new family Inductor, Side Concave 2 and 4 pin. Also added a model for new Family DFN 2-pin Fuse.
- Parts on Demand (POD)
 - Added 17,000+ new parts
- Allegro/OrCAD PCB
 - The ability to turn off the truncation of land pattern names is now available. Only use this if your version of Allegro or OrCAD PCB supports land pattern names longer than 30 characters.
 - The grid change line in the pad stack creation is no longer applicable and useless(it previously caused an irrelevant error in the .jrl file that was ignored and did nothing). As such, it has been deprecated.
- PCAD, Pulsonix, and DesignSpark
 - These 3 formats now only accept numeric layers and should be more understandable, as the LIA format they're based on was never designed to actually accept custom strings as layer names. The LayerNumRef literal that was used in the Layer String is now automatically inserted any time a layer number is called out in the output.
- PADS
 - VX is now supported as a PADS output
 - PADS ASCII: When using a Part Type Name that is different from the Part Decal Name, the program will now properly use the Part Type Name rather than the Part Decal Name when calling out the Part in the .ASC
- Altium
 - PADS ASCII files for Altium will be generated without use of PADS arcs. Instead, it will describe the arc as a series of points in the polygon. This should cause the D-Shapes and other such pads that use associated copper to render properly.

Library Expert 2014.05 Released!!

NEW:

- Calculator:
 - Chip Array, 2 Side Flat/Concave
 - Added an 18 pin version
 - Added "IC" component family
 - TO (DPAK):
 - Ability to enter a negative value in "L1" cell for the Thermal Tab extension for DPAK's with recessed Thermal Tab
 - New Chip component families:
 - Added Varistor
 - Added Thermistor
 - Added new through-hole component family:
 - Radial Lead Inductors
- Parts on Demand (POD):
 - Introduced a new "Library Creation Service". Library Expert users can request parts not in POD and PCB Libraries, Inc. will add them.
 - This library service is limited to 10 new parts per week per Library Expert seat
 - Added 37,500 new parts
- 3D STEP:
 - A model for Radial Inductor has been added
- General:
 - Double Click on any object auto-opens the Properties Dialog Window

FIXED / ENHANCED:

- Expedition:
 - Fixed various output issues having to do with the translation of Polygons
 - Various naming issues when using the PADS Importer
- PADS to CAD Translator:
 - Irregular pad shape issue
 - Various issues when using the PADS Importer
- Calculator:
 - Concave, 2-Sided:
 - Added an 18 pin version
 - TO (DPAK):
 - Rename Pin for Thermal Pad now works correctly
 - Molded Body & Chip:
 - Parts with Polarity now have silkscreen wrapped around pin 1
 - Headers:
 - The Physical Description for single row headers
 - Removed polarity from a single pin Header
 - SOT23-3:
 - Pads were being trimmed if they were wide when using mfr. recommended pattern
 - DFN 3 & 4-Pin:
 - Changed the default Reference part pin assignments to match M0266C-AA
- FP Designer:
 - 3-Teir Environment for through-hole now adds a _A, _B or _C to the suffix of the footprint name
- User Preferences:
 - Component Families:
 - Oscillator L-Bend lead form graphical data was missing
- Default Preferences Editor:
 - The Default Preferences editor had the "Display" Tab and it was removed as the Color Display is a local User preference
 - The Local Fiducials were not properly saving user changes
- Librarian:
 - Program now produces a message when closing the program if you have unsaved library editing
- PADS to CAD Translator:
 - Irregular pad shape issue

Library Expert 2014.04 Released!!

NEW:

- FP Designer:
 - Now auto-generates the Physical Description
- User Preferences:
 - Added new button to restore User Preference default values
- General:
 - Improved object selection. No longer have to change layer to the same as the object to be selected.
 - Improved selected object highlighting
 - If multiple objects get selected (possible if objects are stacked) use F7 key to loop thru selected objects
 - The keyboard key "Enter" now tabs between cells in every Dialog Window and acts as an Save & Close when cursor is in last cell
- GUI Toolbars:
 - Added "Function Text" below the toolbar icons
 - Added a new toolbar button for PADS to CAD

FIXED / ENHANCED:

- Allegro/OrCAD PCB:
 - Issue that was causing the Library Name to be disabled when using multipart mode and you changed to that translator from a separate translator
 - Issue that would crash the program if pins were completely removed from the part, rather than simply hidden
 - Problem that was preventing donut shapes from converting properly
- PADS:
 - Turning "Map Alphanumeric Pins" off when generating a p/d pair will now properly send any alphanumeric pins to the Part Decal
 - Hidden or Deleted pins now import properly
- CR-5000:
 - Polarity Marker Dots are now properly handled, and the output has been cleaned up
- FP Designer:
 - Adjusted the size limits for small Die Patterns getting error message "Design Limits Exceeded"
 - Chamfered and rounded corner pad shapes are now back
- New PADS Translator / Filter:
 - Issues found in importing files generated from older versions of the PADS V2004
- General:
 - Any Placeholder Pins at the end of a pin list will now be clipped

Library Expert 2014.03 Released!!

NEW:

- PADS Layout Translator Feature:
 - Added a new "p/d ASCII Filter" that allows the user to translate PADS p/d files to any CAD tool format –
 - Change drafting line widths
 - Change silkscreen and assembly Ref Des height
 - Change layer assignments
 - Adjust or remove Polarity Dot
 - Adjust or remove drafting items

FIXED / ENHANCED:

- PADS:
 - Parts in which pins have been removed and the numbers were then reordered now translate properly
 - Issue preventing the proper line width for rectangles from being used
- Eagle:
 - Added an option to allow the 3D Height to be divided by 1000 and used as the 3D Line Width. This is applicable when using the Eagle ULP for post processing to allow it to strip the height into a tool like SolidWorks.
- 3D STEP:

- Fixed an issue that was preventing Rotate Axis from displaying when using the standalone 3D STEP translator
- Various models for families have been updated so they display properly in the CREO line of tools. These families are: SOTFL, SOL, SOT23, SOT143, and SOT223
- PADS p/d Translator:
 - Fixed a problem that was causing Part Type Gates to be read improperly by the parser.
- General:
 - All medium Font Translator Form UI issues

Library Expert 2014.02 Released!!

NEW:

- Librarian Feature:
 - Automatically load the user default FPX library defined in User Preferences (this is in addition to the command line library load and open)
- Email Notifications of new version releases (not via this forum) begin with 2014.03. If you have not subscribed to these upon registration, or if you registered before this option was available, subscribe to release alerts [here](#). Please be sure to confirm your opt-in via the email you will receive!

FIXED / ENHANCED:

- PADS Layout:
 - .d ASCII files will now properly use degrees, rather than tenths of a degree, when outputting to a format older than V9.0
- CADInt:
 - Units issue that was preventing parts with copper finger shapes from importing properly
- Altium:
 - Issue that was preventing the Polarity dot from being converted from a label to a shape
- 3D STEP:
 - SODFL – Lead generation and placement, and cleaned up the 3D model body
- General:
 - 3D part of the Translator Window to be visible (although not active), if you didn't have a 3D License

Library Expert 2014.01 Released!!

NEW:

- **"Parts on Demand" (POD) database**
 - Added new toolbar icon to link your default web-browser.
 - 226,000+ parts available to Library Expert customers, many more to come!
- **SOL Component Family:**
 - Add component manufacturer recommended pattern
- **PADS Layout .p and .d file translator to any other CAD tool format**
 - Translate PADS library parts to any other CAD tool.
 - The ability to set your polarity marker DOT as an Attribute/Text combination is now supported. Should you chose not use this, the Polarity Marker DOT will come in as normal drafting shapes. All other CAD Translators - Polarity Marker DOT will be generated as regular drafting shapes.
- **PCB Library Expert Viewer**
 - Allow all your co-workers or customers to freely browse your CAD library.
 - Quickly locate Footprint Names.
 - Have other people Quality Control your work.
- **General**
 - Specify in the command line what specific Preference File to use when the Library Expert opens. This allows users to create multiple shortcuts that point to different Preference Files.
 - User Preferences now has the ability to turn on/off the courtyard and centroid marker independently to allow you to turn off the courtyard without turning off the centroid marker and vice-versa.
- **FP Designer**
 - Ability to freely switch pad stack pin assignments.
- **STEP**
 - New 3D group box for parts generated with a Y-up or Z-up orientation.
 - Note: The Original Library Expert default orientation was Y-up and the Z-up orientation was added for CAD tool preference; Altium Designer and CADSTAR prefer Z-up orientation.

FIXED / ENHANCED:

- **Allegro/OrCAD PCB**
 - Line width was ending up greater than the diameter for a circle due to rounding. This was then preventing the circle from displaying properly.
 - Footprints will be created with pad rotations limited to 0 and 90 degrees. 180 and 270 will be converted to 0 and 90, respectively. This should incur no change to the part itself, it should resolve a problem with ODB having issues with rotations ≥ 180 .
 - In order to map a Layer (subclass in Allegro terms) to a different Class, you can now specify the full layer name in a class/subclass format. I.e. 'Manufacturing/No_Probe_Top'. If no class is specified (i.e. no '/'), Package Geometry is the implied default.
 - Issue causing the Run Scripts in Background option to be ignored for complex pad stacks.
 - When Parts per Script is not 1 and exceeds the number of parts being generated, the -1 will be properly excluded.
- **CADSTAR**
 - Version 15 is now supported.
 - Extraneous layers have been removed from the Layers and Layer Stack-up.
 - Bottom 3D Model Outline will now properly get set to Non-Electrical layer 2, rather than 1.
- **PADS Layout**
 - On a multipart operation, Library Name will now be Decal Name and the Part Type Name will have the same name as the Decal Name. This is primarily for cosmetic and read ability reasons and doesn't change anything functionally.
 - Lowered the default Line Width for solid Associated Copper to 0.01 from 0.05 to allow for finer features.
 - Problem that was preventing the minimum Pad Width from being used, resulting in a negative Pad Width.
- **Eagle**
 - The NOCREAM and NOSTOP from being set properly based on whether a pin contained either Paste Mask or Solder Mask, respectively.
 - NoCream only is fixed to only be used on surface mount pins.
- **Expedition**
 - Fixed a conversion issue when specifying parts in inches.
- **CADInt**
 - Alphanumeric pins are excluded as its PADS Importer cannot handle them.
- **Altium**
 - This translator now defaults to outputting PADS V9.3.
 - The default for Altium is now PADS Layout V9.3. This can be seen by going 'Restore Entries to Defaults' and then saved by going 'Save Entries as Preferences'. It's highly suggested if you are using Altium V13 or V14, that you output PADS Layout V9.3 ASCII for import purposes.
- **General**
 - Parts whose first pin was hidden, that was causing the program to crash.
 - All non-PADS based translators that use numeric layers will have a default maximum of 255 layers, rather than 100. PADS will work as normal with 30 and 250 being the max depending on if Extended Layers is selected.
 - Vertex Generation for Rectangles and Squares that occurred when the Lower Left Corner Length was more than half the actual Length.
 - Updated the "IPC-7351B_DefaultPreferences.dat" for the Zero Component Orientations.
 - When STEP is selected as the translator, when you change to any other translator the window will properly resize.
- **STEP**
 - Added an option to include or exclude the environment suffix (L, M, or N) in the generated STEP File Name.