

Version 2018.03

NEW / ENHANCED:

- Library Editor:
 - Added a new dialog box editor for the library date column.
- 3D STEP:
 - LCC component package now has wraparound terminal leads (instead of Bottom Only).
 - LCC component package now has polarity marking in 8 different locations.
 - FP Designer - added an option for Polarity Marker in any corner or center edge. There are 9 options.
- Xpedition:
 - Added Physical Description to the Cell Description field.



FIXED:

- Calculator:
 - Fixed an issue with the Vertical Mirror feature. When switching between component families disabled Mirror Vertical.
 - Vertical Headers: fixed an issue where a particular pin pattern turned all the pad shapes to square.
 - Fixed an issue with the DPAK component family Paste Mask reduction pattern.
- Preferences:
 - Solder mask defined pad oversize now displays properly in the viewer and CAD tool
- Altium:
 - Fixed an issue with bottom layer pads that was causing the program to crash.
 - Fixed script output so that it now allows spaces in the name.
- PADS Layout:
 - Fixed an issue that was causing rotated paste mask pads not to offset properly.
 - Fixed an issue that was causing Square and Rectangular rotated pads not to translate with the proper rotation.
- Xpedition:
 - Pad names will now include 'mm' if millimeter. This will separate them from pads in Mils.
- Allegro/OrCAD PCB:
 - Fixed an issue with specialized through-hole pad stacks that could cause the program to hang.
- General:
 - When checking if a pad stack is unique for programs that split-out pad stacks, the polys that were introduced will be ignored, if they're the Terminal Outline.

Version 2018.02

NEW / ENHANCED:

- Preferences:
 - The default LGA pad shape was updated from “Rounded Square” to “Square”
- Library Editor:
 - Added a new SAMPLE BOM.xlsx file for FPX to BOM compare demo
- FP Designer:
 - Doubled the size for pad stack creation. Was limited to a 25.4 mm pad size.
- Library Expert Pro:
 - Added a notification in the CAD tool translator dialog box on how to change CAD tool formats

FIXED:

- FP Designer:
 - Fixed an issue with Silkscreen Legend Outline. An outline that had no cuts was coming out as a non-closed polygon.
- 3D STEP:
 - The 3D STEP model for MELF Diodes was not being created correctly. The polarity band was cutting through the body.
- Xpedition:
 - Fixed an issue when generating “Unique” parts that were originally created in PADS Layout and inserted into FPX
- PADS Layout:
 - Fixed an issue where in some installations, Extended Layers was the default

Version 2018.01

NEW / ENHANCED:

- Library Expert Lite:
 - LE Lite is now LE Pro to allow users to import FPX files from Parts on Demand (POD)
 - FP Designer has been turned on to allow the user to build custom parts
- Altium Nexus:
 - Added Altium Nexus as new CAD tool interface
- Drafting Shapes:
 - Added Solder Mask Top layer (for custom solder mask removal)
- Symbol Expert:
 - Beta version is planned for March 2018, and will share FPX data files with Library Expert. In preparation, the master FPX file format was updated. Version 2017 FPX format files will auto-update on Save FPX,

so if you have earlier FPX file versions, you will need to step them through version 2017 first.

- Added a Footprint Column to save unique custom patterns (invisible in Library Editor)
- Added Schematic Symbol column to save and retrieve Symbol Data (invisible in Library Editor)
- Added a new Value column for schematic symbols (visible in Library Editor)
- Sample FPX:
 - New Sample FPX has schematic symbols added for Symbol Expert

FIXED:

- Preferences:
 - Fixed an issue relating to Console Preferences. When editing Console Preferences and saving them, Library Expert updated the Master Preference file.
 - Updated the Rule value for Minimum Solder Mask Web from 0.08 to 0.075
 - Fixed an issue with the TO Cylindrical rotation option not working correctly
- PADS Layout:
 - Fixed an issue with Extended Layers in the CAD tool interface
- 3D STEP:
 - Fixed that Chip Antenna polarity marker