

Version 2018.06

NEW / ENHANCED:

- Proteus:
 - Added the ability to output to PADS Layout Part Type .p and Decal .d files



FIXED:

- Calculator:
 - Fixed an issue in the PQFN component family for parts under 1.60 mm in length, the incorrect terminal type was used
 - Fixed 2 issues in the PSON component family:
 - When Pin 1 was longer than the other pins, it caused the solder mask to swell for violating the Minimum Solder Mask Spacing Rule
 - For parts under 1.60 mm in length, the incorrect terminal type was used
- Altium:
 - Fixed an issue with generating pads that are both rotated and offset
 - Script Naming will now allow you to use a period "." in the name of your footprint and still have it generate that way in Altium
- EAGLE:
 - Fixed a problem that could cause the first point to be called out twice for a polygon. This was causing an error to be thrown in Eagle, which will now go away.
- Pantheon:
 - Adjusted the Attributes to remove @coord which was not being recognized
- 3D STEP:
 - Fixed a conversion issue with off-unit (anything but mm) model output. This issue was primarily with TO-92 parts.

Version 2018.05

NEW / ENHANCED:

- Global License:
 - Added new Global License option for worldwide use, beyond the traditional Network (site-based) - [contact us](#) to convert your license from Network to Global
- Library Expert Pro:
 - The notification to select a CAD tool will only appear when there is no CAD tool selected

FIXED:

- Calculator:

- Updated the Axial Lead component family with the latest IPC-2221 / IPC-J-STD-001 lead bend standards
- Allegro:
 - Fixed a bug that was causing Route Keep-outs to not translate properly
- PADS variants/CADSTAR/Allegro:
 - Solder Mask will be excluded in a pad stack when an area of Solder Mask is used to draw a unique overall pad shape
- CADSTAR:
 - Whenever a Paste Mask or Solder Mask Pad is excluded from a pad stack, the code will now put in a 0 size reassignment pad in order to exclude it from the CPA Pad stack. This should prevent issues like paste mask being generated internally to CADSTAR, when there is a drawn paste mask checkerboard.

Version 2018.04

NEW / ENHANCED:

- Preferences:
 - Expanded the Footprint Name Density Level suffix to 5-digits instead of 3; also allowed the use of lower case characters
 - Multi-part default file name “New_Parts” can now be controlled in the Files tab in Console Preferences
 - Rules: added a Thermal Pad Paste Mask Minimum web thickness
- CADSTAR:
 - Added V2018 as an option
 - V2018 will now properly have a CPA version callout of 20, rather than 19
- Circuit Studio:
 - Added this CAD tool interface to all versions of Library Expert

FIXED:

- FP Designer – Round Packages
 - Fixed an issue with saving Square Body and then retrieving and updating to Round Body would not properly save as Round Body
 - Fixed an issue where the silkscreen Legend was not properly communicating with Legend Clearance Preferences
- Calculator - Molded Body Component Family:
 - The Export using Multi-part was producing a different (incorrect) Body Outline Length than the Single-part output
 - Physical Description was updated to produce the Body Length per the “D1” dimension (rather than the “D” dimension)
- Library Editor:
 - FPX Creation from PADS p/d files - fixed an issue that was causing Donut Pads in a pad stack to not convert properly
- 3D STEP:
 - Molded Body Component Family - fixed an issue that was causing the body and leads of the part not to reflect its true D1 width

- Chip Array, Convex Component Family- fixed an issue with the Terminal Leads coming out as no width when the “b1” dimensions were not used

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