

Version 2018.08

NEW / ENHANCED:

- Altium:
 - Added the ability to create and focus a temporary component before building the specified footprints. This temporary component will delete and re-focus after footprints are built.
 - Rather than Create the Component directly, a new function called CreateAComponent has been added. This function will check if there is a currently existing component of that name and remove it and all of its primitives. If there isn't, it will simply create the part normally. The footprint will then be built as specified, essentially taking its place. This should allow for a better handling of the updating of parts in Altium during script generation.
- OrCAD PCB/Allegro V17.2.048:
 - An extra version has been added called 17.2.048. 048 has added an extra UI window, and 048 specific code was added to bypass it. Not everyone has the Allegro 048 patch, so for the time being I'm leaving the original translator alone while adding this one for 048 and later. Simply use the translator that matches the level of patch for that tool you have installed.
- 3D STEP
 - Updated the SOP component family to have scalable polarity markers for micro-miniature packages
 - The 3D Model for DPAK will now have a full thermal tab going into a cavity of the part, taking into account the D2 dimension; added the ability to generate a model without side walls for the thermal tab, for parts where the body of the DPAK in width, is less than or equal to the thermal tab width
 - All vertical and right-angle headers were updated to be compatible with SolidWorks V2018
 - All STEP models which have a Thermal Tab have been simplified to remove the use of cavities for the tab to sit in, and had bugs fixed to display properly in Solid Works 2018. The tab will now rest between the model body and the board.
- Yearly Maintenance Reminder:
 - Added feature to the "Help > Installed Options"
 - ♣ Check for – Software Updates
 - ♣ Check for – Maintenance Renewal



FIXED:

- Calculator:
 - 3-Pin DFN: the larger pad in a was redefined from a Thermal Pad to a Bottom Only Terminal
- Xpedition:
 - Round Keep-outs now Translate properly
- OrCAD PCB/Allegro V16.6:

- Fixed a solder mask regression issue that was introduced in V2018.05; Library Expert was not outputting solder mask in versions 05, 06 and 07
- Altium Designer:
 - Fixed an issue with mixed parts where the layers differ. Ex: No top pad, but a rounded rectangular pad on the middle and the bottom. Previously, it would only use the top layer as a 'Simple' pad.
- 3D STEP:
 - Fixed a bug with the LGA model
 - Fixed the CGA, CAPAE, and TO-220H models

Version 2018.07

NEW / ENHANCED:

- FP Designer:
 - Update the Print feature to add the package dimensions and pad stack data
- Pulsonix:
 - Implemented the keepoutType keyword, which appears to be specific to Pulsonix PLX. A type of Comp is added for every Component Keepout, which will come from the Courtyard of the Part

FIXED:

- P-CAD .LIA (covers DesignSpark, all P-CAD versions, and Pulsonix):
 - Fixed an issue where the plated value was being set to false for through-hole pad stacks.
 - Adjusted the code so that the Placement Courtyard gets translated as a PolyKeepout in the output. This should allow tools like Pulsonix to pull the keepout.
 - Fixed an issue with Polys with Arcs. They should now be fully described as points.
- Pulsonix:
 - Adjusted the Courtyard Keepout to set its LayerNumRef to Courtyard_Top so that it hopefully picks up the keepout rules. This is only for Pulsonix and not the other PCAD variants.
- CR-5000/CR-8000:
 - Added the start of functionality to allow for the tangent arc nomenclature for calling out Pad Outlines. Right now, it's restricted to square rounded pads for testing purposes.

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