

Version 2017.22

UPDATED:

- Allegro/OrCAD PCB:
 - Fixed a problem that was preventing Pad Stack Outlines on the Assembly Layer from generating
- CADSTAR:
 - Output to Version 18 format now supported
- PADS:
 - Fixed an issue that was causing slotted holes to come out with a 360-degree rotation rather than 0
 - Fixed an issue that was causing parts that had a hole but no Inner Layer to crash
- General:
 - Fixed a bug that was causing Rectangular or Square Keep-outs on the Bottom layer to be generated as Copper rather than a Keep-out
 - Parts that have multiple pins of the same name translate properly again



Version 2017.21

UPDATED:

- NEW Name: changed name from "**Professional**" to "**Enterprise**", which much better suits the powerful features that have been added in the last couple years, and how the Library Expert is implemented to provide enterprise-level CAD library support to some of the largest companies in the world.
- Preferences:
 - Fixed an issue with the Drafting Legend placement round-off
- Allegro/OrCAD PCB
 - Fixed issue preventing Pad Stack Outlines on Assembly Layer from generating
- CADSTAR
 - Output to version 2018 now supported
- PADS Layout:
 - The Translator now handles unintentionally rotated pads due to Length > Height, with an offset
 - Fixed issue where slotted holes had a 360-degree rotation rather than 0
 - Fixed issue that was causing parts that had a hole but no Inner Layer to crash
- Xpedition:
 - Fixed an issue that was causing Pad Offsets not to translate
- General:
 - When importing paste mask via the PADS Importer, any shape that is off center of the pin is now properly filtered to associated copper, so that it becomes part of the checkerboard pattern
 - Fixed an issue where pad stacks with offsets would not be generated properly in pins that were rotated. The rotation of the pin will now be properly taken into account when generating the offset.
 - Fixed a bug that was causing Rectangular or Square Keep-outs on the Bottom Layer to be generated as Copper rather than a Keep-out

Version 2017.20

UPDATED:

- Calculator:
 - Fixed an issue when Pin 1 Pad Shape was different than the other pad shapes, the Terminal Lead Outline was suppressed in the output
 - SOFL – Fixed a problem with the generation of the 3D Model for SOFL that was causing the leads to calculate improperly
 - Fixed an issue when selecting Shapes and leaving them Selected and opening another part, the highlight of the selection appears in the new part
 - Fixed an issue with BGA & LGA for single pin parts throwing an erroneous error message and not producing the pad
 - Fixed an issue with the SOTFL Transistor component family, the thumbnail image with the packages dimensions was suppressed
- Polarity Marking:
 - Fixed an issue when changing a Round Polarity Marker to Triangle the Rotation button wasn't turning on
- Preferences:
 - Fixed an issue in the LGA and CGA component Families, when entering a new Terminal Lead Size, the program threw and Handled Exception Error
 - Fixed an issue for setting the SON & QFN Thermal Pad Paste Mask to 100%, Library Expert was still using a "Pattern" pad, rather than a solid pad
- FP Designer:
 - Fixed an issue with Non-plated through-hole pad stacks that require a copper annular ring with a ID larger than the hole size
- CAD Tool Interfaces:
 - Fixed an issue when the Footprint Name had an illegal character, Library Expert crashed. Now it throws a warning message.
- Allegro/OrCAD PCB:
 - Cleaned up the plating code for 17.2 pad stacks and fixed a problem where plated mounting holes were being set to non-plated
 - Fixed an issue with the last update that caused Allegro to stop making regular .pad Pad Stacks for rounded rectangle pad shape
 - Four additional scripting commands were added just as pin generation goes to start. These commands are settings which are used to put the scripting engine into a known expected state for FPX, most of which should already be the default, but these are added in case the user has changed the defaults. In order, these are:
 - Set the coordinate system to Rectangular
 - Increment pin numbers in the horizontal direction by 1
 - Increment pin numbers in the vertical direction by 1
 - Set the initial pin rotation to 0

- EAGLE:
 - Fixed an issue with thermal tabs with a chamfer, where custom stop was not being generated
- KiCad:
 - Fixed a problem where if you regenerated fresh preferences, it would use 3D_TOP, rather than THREED_TOP and cause an exception to be thrown
 - Fixed an issue for Terminal Outlines coming out as stacked on multiple occurrences

Version 2017.19

NEW / ENHANCED:

- Calculator:
 - All parts with Thermal Pads – if a Thermal Pad is less than 1.00 mm width, it will now get 100% Paste Mask Coverage

FIXED:

- Calculator:
 - Non-Plated Mounting Holes – fixed an issue with the Keep-out size generation to follow the Hole Size instead of the pad size
- Unique Parts:
 - Fixed an issue with footprints with associated copper on slotted holes. This affected all CAD tool translators.
- POD Builder:
 - Fixed an issue that was creating Handled Exception Errors for certain parts
- Library Editor:
 - The automatic Manufacturer Name feature was fixed to only activate in the Manufacturer column
- Drafting Items:
 - Fixed an issue when placing Drafting Outlines on the Bottom Assembly layer, reopening the properties menu displayed the incorrect layer assignment
- PADS Layout:
 - BGA component family – fixed an issue where unpopulated pins showed up in the Part Type
- Xpedition:
 - Fixed a problem with non-plated slotted holes that used circular pads smaller than the hole size
- Allegro/OrCAD PCB:
 - Fixed an issue with offsetting top pads

- Chamfered Rectangular pads with a corner length greater than half the edge are now converted to custom pads. A rectangle pad stack in Cadence cannot have a corner length that exceeds more than half the width.
- Custom Pads which are offset from the center of the Pad Stack are now removed from the pad stack and converted to free floating copper. This is the *only* known way of offsetting a shape symbol pad as Allegro appears to be bugged and will not save offsets for Shape Symbol Pads.
- Fixed a problem with negative rotation arcs that end at 0
- Fixed a problem that was causing an error when the build was run more than once
- Attached copper shapes with an offset will now rotate properly with their underlying pad stack
- The translator will now differentiate when you have two pad stacks that are exactly the same, but differ in offset of attached copper
- KiCad:
 - Fixed an issue where the terminal outlines were getting duplicated. They should now be generated only once.
 - The two Ref Des texts will now swap their layer. The primary reference text being on the Fabrication layer, and the extra user reference being on the Silkscreen layer.
 - Filled Paste Mask shapes for thermal tab checker-board patterns will no longer translate as unfilled lines. Instead, they will come in as Pads named "" on the F.Paste layer.

Version 2017.18

NEW / ENHANCED:

- FP Designer:
 - Accounted for condition where the ManufacturerList.txt or PhysicalDescription.txt is missing
- KiCad:
 - Added the ability to output both the component outline and terminal outlines; these graphics get placed on the Dwgs.user layer

FIXED:

- Calculator:
 - Mounting Hole component family – fixed an issue with non-plated hole solder mask not mapping to the Hole Size
 - Molded Body component family – fixed an issue when using Tools > Regenerate Library > Descriptions the Length value was 0.00
 - SOD & SODFL component families – fixed an issue with the Footprint Naming Convention: Tools > Regenerate Library > Names fixed the problem

- SOD & SODFL component families – fixed an issue with the Physical Description: Tools > Regenerate Library > Descriptions fixed the problem
- LCC component family – fixed an issue when L and L1 dimensions are the same value, pin 1 relocates from the middle to the corner and the Footprint Name should change from LCC to LCCS
- FP Designer:
 - Fixed an issue in the Pad Stack Designer for chamfered SMD pads with solder mask swell; the chamfer on the solder mask was not 100% accurate
 - Fixed an issue when moving parts from the Calculator that had mfr. recommended patterns to FP Designer, the program created 2 pad stacks the same size
- Preferences:
 - Fixed an issue with "Save As" a new Preference file when Library Expert Pro is installed in a new folder
- Xpedition:
 - Fixed issue with arcs in associated copper polygons
 - Fixed issue with Mounting Hole Satellite Vias having a direct connection to the GND plane
- KiCad:
 - The Reference Designator on Assembly Top will now translate properly

Version 2017.17

NEW / ENHANCED:

- Added new user definable Top and Bottom Placement Keep-out Layers to the Interface menu

FIXED:

- Calculator:
 - Molded Body component family – Add D1 dimension
 - Chip Array component families – Add Ferrite Bead
 - Side Concave 2/4-pin component family – Add Capacitor
 - Fixed a silkscreen Legend mapping to body issue in Chips, Molded Body, SOD & SODFL
 - Updated the graphic thumbnail image for QFP with Thermal Pad was missing the A1 dimension
- Preferences:
 - Updated Through-hole Pad Size round-off to equal the Hole Size round-off
 - Fixed an issue where changing Preference files required closing and reopening the program before activating the new Preference file
 - Fixed an issue in the DFN component family for Diode and LED alphanumeric pin names
- FP Designer:
 - Fixed the Rounded Rectangle % to consider the smallest pad dimension instead of relying on the Y dimension only

- Altium:
 - Fixed an error that was suppressing Bottom-side PTH Placement Keep-outs that were attached to pins
- Allegro / OrCAD PCB:
 - Lines have now been added to the batch file to delete pre-existing .PAD or .DRA files; this should allow for regeneration without manual deletion
 - Fixed an issue where irregular chamfered pads were inverting their corners
- Xpedition:
 - Fixed an issue with PTH Thermal Relief patterns not coming through translator

Version 2017.16

NOTE!!!

- Some Drafting issues were broken in V2017.15. We advise that you discontinue the use of that release and update to V2017.16 ASAP.

FIXED:

- Calculator:
 - Fixed an issue with the Courtyard Excess mapping to the Nominal package body and changed to Maximum package body
 - Axial Lead component family – fixed an issue with the Contour Courtyard
 - Fixed an issue with missing Component Outlines on all Grid Array component families
 - Fixed an issue where the Component Body Outline got changed from Nominal to Maximum mapping
 - Note: the default Component Body Outline should be Nominal
- 3D STEP:
 - Removed Secondary Body color and updated the graphic image for CFP and CQFP
- PADS to FPX:
 - Fixed an issue that was causing the FPX not to display the Associated Copper Thermal Tab
 - Fixed an issue with the PADS p/d import feature relating to outer and inner layers
 - Updated Viewer so that large parts do not get clipped

Version 2017.15

NEW / ENHANCED:

- Library Editor:
 - Right Mouse Button on any cell and select – Search the Web > Google
 - This new feature auto-generates the Google search engine string for the Part Number; the primary use for this feature is to quickly locate the Datasheet link
 - FP Designer: Export > Pins to .CSV file; this new feature creates a report of all pin names and their X, Y coordinate
- Added PADS Professional CAD tool format - <https://www.pads.com/professional/>
- Added “Filter” to these component family Calculators and 3D STEP models: DFN2, DFN3, DFN4 and SOFL

FIXED:

- Calculator:
 - Fixed a Courtyard issue when relocating the Origin to Pin 1
 - Updated the thumbnail graphic image for the Through-hole Radial Round LED
 - SIP component family – Fixed a regression issue that was introduced updating the DIP Socket
 - Updated the thumbnail graphic image for the Through-hole Radial Electrolytic Capacitor
- Altium:
 - FP Designer parts with bottom side SMD pads were not translating correctly
 - Fixed an issue with slotted holes with rectangle pad shape not rotating correctly
 - Adjusted the STEP model generation to set the State Identifier
 - Adjusted the output to be able to use multiple pieces of Solder Mask in a Pin
 - Used for Solder Mask Defined Thermal Pads
- PADS to CAD:
 - Fixed an issue where Top Layer Pieces weren't mapping properly
 - Fixed an issue where Multiple Top Layer Pieces in a single Footprint weren't converting properly
- Xpedition:
 - Updated the Import instructions - [Here](#)
- 3D STEP:
 - SIP component family – the 3D STEP did not account for the “E1” dimension offset
 - Radial Electrolytic – rotate model 180 degrees so that the Polarity Strip is on Pin 2

Version 2017.14

NEW / ENHANCED:

- All parts with Thermal Tabs
 - Added a new Preference Rule for Solder Mask Defined Thermal Pads
 - This new process saves fabrication costs and reduces assembly issues with BTC packages:
https://www.pcblibraries.com/forum/ipc7093a-btc-qfn-solder-mask-defined-thermal-pad_topic2154.html
 - Turn on the new feature in Preferences > Rules > Thermal Tab Pattern Defined Solder Mask
see new video:
<http://www.pcblibraries.com/products/fpx/userguide/default.asp?ch=7.7>
- FP Designer:
 - Added new Pad Stack Manager feature to “Pattern” Solder Mask for Thermal Pads

FIXED:

- Main Toolbar:
 - Fixed an issue when Restore Down the main toolbar icons did not wrap
- Calculator:
 - SODFL component family:
 - Micro-miniature versions only produced Legend outlines on one side
 - 3D STEP Model will now take into account if the pins have been Mirrored and adjust the Pin 1 marker appropriately
 - SOFL component family:
 - The pin qty. in the footprint name was stuck on 6 regardless if it was 3, 4, 5, or 8 pins
 - DIP Socket component family:
 - Fixed an issue that prevented the use of round terminal leads
 - Column Grid Array (CGA) component family:
 - Updated the solder joint goal periphery from 0.00 to 0.10 mm to match IPC-7351C
- FP Designer:
 - Round parts: Silkscreen Legend was updated from rotation 90 degrees to 0
- 3D STEP:
 - Molded Body component family will now shrink the Physical Body (not including Leads), if there isn't enough room to place the leads at their minimum size
- Allegro:
 - Fixed an issue that was causing bottom-side pad stacks not to translate properly
 - Fixed an issue with excluding the courtyard crosshair
- EAGLE:
 - Fixed an issue with chamfered pads and associated copper
- KiCad:
 - Fixed an issue with standalone Legend arcs not coming over
- PADS Layout

- Fixed an issue with FP Designer via pad stacks. The Flood Over Thermal Relief Inner Diameter was the same value as the Drill Size.

Version 2017.13

NEW / ENHANCED:

- Updated all Terminal Lead-Form images in the Calculator and Preferences

FIXED:

- Calculator:
 - Fixed an issue with the Pin Rename feature: once you rename pins you were unable to rename them again
 - Updated footprint names for:
 - Oscillator Side Concave
 - Molded Body
 - Oscillator J & L lead
 - SOFL Diode, IC & Transistor
 - DIOSOFL
 - IC SOFL
 - TRXSOFL
- Preferences:
 - When changing Pin 1 pad shape in one location, did not change all locations
- 3D STEP:
 - Fixed an issue with Molded Body 3D model scaling
- License selection / default:
 - Under certain conditions, a previously expired evaluation license interfered with the full Pro license
- Documentation:
 - Updated - IPC-7351C (Proposed) Solder Joint Goal Tables v1.5.xlsx
 - Added the new Terminal Lead-Form images
 - Updated – Surface Mount Component Families.pdf
 - Added Terminal Lead-Form images to pages 36 & 37
 - Fixed 6 typos – IPC-7351C Land Pattern Naming Convention.pdf
 - Molded Body - PKGM + Lead Span X Body Width X Height + L Lead Length X Width
 - Oscillator J-Lead - OSCJ + Pin Qty. + P Pitch _ Body Length X Lead Span X Height + L Lead Width
 - Oscillator L-Lead - OSCL + Pin Qty. + P Pitch _ Body Length X Lead Span X Height + L Lead Length X Width
 - Diode SOFL is now DIOSOFL
 - Transistor SOFL is now TRXSOFL
 - IC SOFL is now IC SOFL

Version 2017.12

NEW / ENHANCED:

- All versions of Library Expert Lite, Viewer, IPC, Builder and Pro:
 - Added Part Number Search for component availability using the **Octopart** search engine
- **Library Expert Lite** now fully supports **3D STEP** export:
 - User color control
 - Vertical Axis control
 - Nominal or Maximum Material condition control
 - Through-hole lead depth control

FIXED:

- Preferences:
 - Added the SOFL to Component Families
- Library Editor:
 - SOFL component family – retrieving SOFL data from FPX into the Calculator did not work
- 3D STEP:
 - Fixed an issue with SOFL not producing a 3D STEP file
 - Added new color assignments for the Chip Filter component family
- Xpedition:
 - Fixed a rotation issue with complex paste mask
- PADS Layout:
 - Fixed a problem with the importer that was inverting the corners of chamfered pads
 - Provided the ability to import donut shapes properly
 - PADS p/d Viewer – Provided the ability to display donut shapes

Version 2017.11

ALERT!! *If you are using a custom Preference file, you must manually correct this setting –*

- *Preferences > Console > Least Density > Terminals > Gullwing > SOD > Density Level > Side > From 0.20 to 0.05*
- *Go back to Console and change the Density Level back to your default setting*
- *Save your Preference file*

NEW / ENHANCED:

- Calculator & 3D STEP:
 - Added 'Filter' component family to Chip and Molded Body
- Preferences:
 - Added a new feature in Terminals > Through-hole > Bottom Side Keepout option



FIXED:

- Calculator:
 - Fixed a SOFL component family issue. Moving SOFL parts to FP Designer set the body width to the Lead Span.
 - SOL component family – updated the Terminal > Image picture
- Library Editor:
 - Fixed a problem with PADS-to-FPX viewer and builder
- Preferences:
 - Terminals > L Bend – updated the Image picture
- FP Designer:
 - Pad Stack Manager: added Bottom Side keep-outs to pad stacks
- Allegro/OrCAD PCB:
 - Fixed a problem with slotted hole rotations
 - Fixed an issue with offset pad stack data
 - Moved the Labels to the origin 0, 0
- PADS Layout:
 - Fixed offset slotted holes rotated in 180 and 270 degree rotations when the CAD tool can only handle 0 and 90 degree rotations
- General:
 - Bottom Keep-outs should be working again for PADS Layout and Allegro/OrCAD PCB

Version 2017.10

ALERT! If you are using a personal FPX file you must **MANUALLY** update 4 solder joint goal settings for IPC-7351C > Preferences > Terminals. If you are using the default preferences, then there is no need to update anything.

Preferences:

- LEAST Density Level, Side Lead (Concave), Pitch ≤ 0.40 mm had incorrect values for HEEL and SIDE
 - HEEL & SIDE value was **-0.70** mm and changed to **-0.07** mm
- LEAST Density Level, Side Lead (Concave), Pitch > 0.50 and ≤ 0.65 mm had incorrect value for SIDE
 - SIDE value was **-0.70** mm and changed to **-0.07** mm

- MOST Density Level, Outward L Lead, Pitch > 0.40 and <=0.50 mm had incorrect value for HEEL
 - HEEL value was 20.00 mm and changed to 0.20 mm
- MOST Density Level, Rectangular End Cap, Size <= 0.50 mm, TOE
 - TOE value was 0.60 mm and changed to 0.06 mm

NEW / ENHANCED:

- Library Expert License:
 - New 12-month maintenance license is now available! Purchases of new licenses now get all updates, regardless of software version, for a full year

FIXED:

- Unique Part Viewer - updated the unique part viewer
- Fixed a graphics display issue for UHD 4K monitors
- Calculator:
 - Saving component dimensions to FPX did not get the "Type" inserted and unable to retrieve the data
 - Axial Lead: Fixed an exception error when clicking in the Pitch cell
 - SOL terminal density levels not responding to pitch dimension in Preferences
 - Fixed a typo in SOL terminal mislabeled as Gull Wing
 - Fixed an issue with SOL solder joint goals were not properly set to IPC-7351C
 - Removed Contour Courtyard from Chip, MELF and Side Concave 2, 4-Pin component families
- Library Editor:
 - Fixed an issue when importing Excel spreadsheet data the Logical Description was not coming in
- Preferences:
 - LEAST Density Level, Side Lead (Concave), Pitch <=0.40 mm had incorrect values for Heel and SIDE
 - HEEL & SIDE value was -0.70 mm and changed to -0.07
 - LEAST Density Level, Side Lead (Concave), Pitch > 0.50 and <=0.65 mm had incorrect value for SIDE
 - SIDE value was -0.70 mm and changed to -0.07
 - MOST Density Level, Outward L Lead, Pitch > 0.40 and <=0.50 mm had incorrect value for HEEL
 - HEEL value was 20.00 mm and changed to 0.20 mm
 - MOST Density Level, Rectangular End Cap, Size <= 0.50 mm, TOE
 - TOE value was 0.60 mm and changed to 0.06 mm

Version 2017.09

NEW / ENHANCED:

- V2017 FPX File Format:
 - Going forward, all saved FPX files will be upgraded to 2017 format
 - Your V2012 FPX file will automatically be upgraded when you save your FPX file to V2017 format
 - V2017 FPX files are not backward compatible with older versions of Library Expert
 - Highly recommended that all team members upgrade to V2017.09 at the same time
- Library Editor:
 - Updated FPX file format to accommodate unique/complex packages
 - This makes for easier viewing and creation
 - See [this video](#) for more details
- Parts on Demand:
 - Added 455 new Unique packages to POD
- Calculator:
 - Added “Bead” to the Molded Body component family
 - Added the supporting 3D STEP model

FIXED:

- Calculator:
 - SOD component family – fixed auto-generated Assembly Polarity Marker
 - Fixed an issue with Chip part courtyards when using 3 place dimensions
- Library Editor:
 - Fixed an issue with deleting columns at the end of a FPX file and then selecting Datasheet
 - Fixed an issue with Sorting Column data when importing new FPX files into an existing FPX file
- Altium:
 - Fixed an issue with FP Designer parts with SMD pads on Top & Bottom sides
- PADS VX:
 - Fixed an issue for 3D STEP with LGA component family
 - Fixed the sizing of the “Type” to map to the Legend Height
- Xpedition:
 - Irregular Rotated Paste Mask on a thermal tab now rotate properly

Version 2017.08

NEW / ENHANCED:

- Help & User Guide:
 - Created a new video for the Free BOM Builder Service - [Click Here!](#)
- Calculator:

- Added alphanumeric pin assignment text in the component dimension panel for BGA, LGA, CGA and PGA; New UI Text – Increments: alpha on D axis; numeric on E axis
- Library Editor:
 - New “Tools > Check Manufacturer Names” feature: Checks names to standardize and ensure the spelling matches POD
 - Added new feature for importing Excel spreadsheet data
 - Pasting from Excel can now be by column and can be in any order (row 1 column headers must match the fpx header)
 - Pasted columns can be by manufacturer (required), part number, case code, logical description, data sheet
 - When changing the manufacturer name an option was added to apply the change to all similar manufacturer names

FIXED:

- Calculator:
 - 2-pin packages – fixed the auto-generated Polarity Dot to adhere to Legend to Pad Preferences
 - Updated the Mounting Hole calculator
- Library Editor:
 - Tools > Regenerate Library > Names – SOD was regenerating name changes prefix to CAP
 - Copy / Paste cells did not update text case
 - Saving an Excel spreadsheet generated FPX to file alternating blank rows are added

Version 2017.07

NEW / ENHANCED:

- Calculator:
 - Chip – added a new component family for Antenna – ANTC
 - When selecting the Red indicator light in the viewer a new message appears and the OK button updates the Data
 - Added alpha-numeric pin text in the thumbnail pictures for BGA, LGA, CGA and PGA
- 3D STEP:
 - Low Quality version is now available

FIXED:

- Calculator:
 - TO-92 – Added the missing A1 dimension to the thumbnail picture
 - Side Concave Package (2, 4 pins) – Synced pin names C & A with Preferences 1 & 2
 - DPAK – Fixed an issue when adding polarity marking and mirror footprint
- FP Designer:
 - Fixed an issue when adding multiple corner radius pad stacks
 - 3D STEP has been adjusted so that the 3D model will offset to a checkdone-list of Extents depending on whether you included them or not. The first that it finds, it will use. In order, It will center to the Component Outline Extents, then the Assembly Extents, then the Courtyard Extents, then the Extents of anything available. This should solve any offsetting problems.
- 3D STEP:
 - Updated PTH Radial Disk
 - Connected Model for Resistor, Varistor, and Thermistor families
 - Connected Model for Thermistor family
 - SOT23, SOT143, SOT223
 - Fixed a problem with the backside negative Z-axis gull-wing lead
 - SOT223
 - Put in a new body. The old one had an issue with PADS 3D.
- CADSTAR:
 - Fixed an issue when dealing with multiple alphanumeric pin numbers (i.e. 2A, 2B, etc.)
- Allegro/OrCAD PCB:
 - When the Anti-pad Size is smaller than the Pad Size, the translator will force it to be larger by 0.005 mm. This is done to bypass the needless warnings from Allegro. It shouldn't have any effect on the manufacturing.
 - Fixed an issue that was generating duplicate pad-stacks for symmetrical pads at different orientations as found on a Chip Resistor, etc.
 - Added an extra line when setting the unit's precision

Version 2017.06

NEW / ENHANCED:

- Help:
 - Added 70 new educational videos that cover every feature in Library Expert
 - Introduced BOM Builder program. Exchange your component list spreadsheet for POD parts.
- Calculator:
 - Added 3 new component families
 - Radial Molded, Fuse
 - SOD, Non-polarized
 - MELF, Non-polarized Diode

FIXED:

- Calculator:
 - Fixed the Courtyard Excess. When swelling solder mask, the Courtyard Excess was determined by the Mask and not the Pad.
 - All Header component families, the upper left “Pin Pattern” option was disabled
 - QFN component family - Fixed an issue with Rectangular Lead shape, changing pad shape did not work correctly
 - Fixed a Pad-to-Pad rule violation that trimmed the pad but not the solder mask
- FP Designer:
 - The Finish tab was not counting the Thermal Pad in the total pin quantity
- Preferences:
 - Added a note to the Preferences > Rules > Minimum Spacing Pad-to-Pad “for SMD use only (not PTH)”
- 3D STEP:
 - Fixed an issue with offset body outline in FP Designer
 - Removed the Polarity Marking for 2-Pin DFN Non-polarized Diodes
 - Updated SOT23, removed the Polarity Dot because no mfr. package has one
 - Updated PTH Radial Disk with Offset Leads - fixed a bug with the new Radial Disk Pin Lead that was not offsetting the leads properly per their Z (in 3D Mechanical Coordinates) location
- Allegro/OrCAD PCB V17.2:
 - Solved an issue when dealing with irregular pad stacks with corners
 - Fixed an issue where Chamfered Pads in FP Designer were not rotating correctly
- P-CAD:
 - The error about Capture files has been fixed

Version 2017.04

NEW / ENHANCED:

- Preferences:
 - Added Oblong pad shape to these component families:
 - SOD
 - SON w/rectangle leads with and without Thermal Pad
 - QFN w/rectangle leads with and without Thermal Pad
- FP Designer:
 - Removed a restriction of 50% pad width for Chamfered pad shape, now unlimited
- Help > Topics:
 - Added 70 new educational training videos to the program Help > Topics

FIXED:

- Calculator:
 - In the calculator dimension panel the Drafting > Courtyard > on/off switch was disabled
- Calculator to FP Designer:
 - Side Concave Package (2, 4-pins) swapped D & E dimensions going to FP Designer
- Preferences:
 - Fixed an issue with SOT Pin Pitch < 0.40 mm
- Print Feature:
 - Fixed an issue where the print feature was disabled
- Allegro/OrCAD PCB:
 - Fixed an issue with Drafting Line Arcs not coming out as intended
 - Fixed an issue with 16.6 and earlier that was causing the suppression of the inner layers of a through-hole pad stack. V17.2 scripts are unaffected
 - Fixed a rotation issue with irregular polys when using PADS to CAD
- PADS Layout:
 - Parts from FP Designer with pads that had corner radius on 2 sides crashed Library Expert

Version 2017.03

NEW / ENHANCED:

- Library Editor:
 - When updating a FPX Row, added option to throw a warning message if the new footprint name is different
- General:
 - Help file included with installer and no longer needs to be downloaded

FIXED:

- Calculator:
 - MELF 3D STEP model color assignment image (preferences) was clipped
 - Fixed an issue where the Legend and Assembly Ref Des Label was stuck on 270 degrees
 - Now the Labels are rotated correctly reflecting the rotation and length of the part
 - Label orientation value not displaying correctly in the query properties dialog form
 - When moving a rotated Calculator part to FP Designer, the Legend in FP Designer did not rotate

- FP Designer:
 - Pad Stacks with Offset Pads was not working as intended
 - Removed code for offsetting slotted holes but kept offset pads
 - If you have a FPX files with offset slotted holes, the pad stacks need to be recreated
 - The center of the slotted hole is the new pad stack origin
 - Fixed an issue with vias being added to a thermal pad and having the code recognize them as vias
 - Fix issue with thermal vias getting deleted after selecting the OK button
 - Added “via” as a new pad stack type
 - Updated via pad stack to remove thermal pattern for direct connection to plane
- Preferences:
 - When changing units to Mils, the Library Editor “Tools > Regenerate > Names” was not working correctly
 - When changing units to Mils, selecting FP Designer defaulting to Millimeters instead of Mils
 - Changing the Legend Label line width changed the Assembly Label line width (not the Legend)
- PADS to CAD:
 - Fixed an issue when importing older PADS V2004 & V2007 format p/d files
- Altium:
 - Updated progress bar in the CAD tool output menu
 - When creating 3D STEP from FP Designer now sets to center origin
- Xpedition:
 - Translation of square thermal pads during PADS to CAD
 - Pad Stack Naming when using multiple complex pad stacks of the same name
 - Addressed issues dealing with Rotation of pad stacks

Version 2017.02

NEW / ENHANCED:

- Library Expert Lite
 - Removed pin quantity limits
- Calculator:
 - Axial Lead Pitch is now auto-calculated
 - Added Thermistor component family to Radial Disk and Radial Disk w/Offset Leads
 - Updated Footprint Name for TO92. Added a dash – after TO92- to conform to the standard.
- FP Designer:
 - Add new “Via” tab in Pad Stack Designer to insert a Via matrix to Thermal Pads

- Now supports Contour Courtyards
 - Added "Via" as a terminal type
- Calculator and FP Designer Drafting Icon - changed text "Polarity Symbol" to "Drafting Symbols"
 - Added Legend Bottom
 - Added Assembly Drawing Bottom
- Preferences:
 - Courtyard feature to define separate excess distances for package body and pad
 - When changing Pad Shape, a notification appears to ask if you want to Change All
 - Added controls for Contour Courtyard minimum Gap of the contour bend
 - Solder Joint Goals now default to IPC-7351C but the user can switch back to IPC-7351B
 - When working in Mil Units, the user can now select fractions of a mil option

FIXED:

- FP Designer:
 - Fixed an issue when a part has multiple pins with the same pin name
 - No more hole offsets in pad stacks (because some pad tools can only handle pad offsets). This will effect some FP Designer parts if they were made with hole offsets.
 - Through-hole parts with pin rotations that weren't 0, 90, 180 or 270 had hole-slot rotations that didn't match the pad rotation
 - Automatic courtyard perimeter wasn't conforming to pad stacks with pad offsets or were associated with pins with orientations that weren't at 0, 90, 180 or 270.
- KiCAD Interface:
 - It was disabled in Library Expert Lite