

Version 2016.14

NEW / ENHANCED:

- Library Editor:
 - When scrolling left to right, the Footprint Name column will stay frozen so that it always appears in the menu

FIXED:

- Calculator:
 - TO-220 – the “d” dimension in the component dimension panel was missing
 - Fixed an issue with Calculator to FP Designer for through-hole parts SIP & DIP
- Allegro/OrCAD PCB:
 - Fixed an issue with slotted holes
- Pulsonix:
 - Translator did not function properly



Version 2016.13

NEW / ENHANCED:

- Calculator:
 - Added Inductor, Polarized to the Molded Body component family
- 3D STEP:
 - Added Molded Body – Inductor, Polarized
- Altium:
 - Added an option for Stroke Text Font

FIXED:

- Preferences:
 - Updated the IPC-7351C Proposal.dat file. Fixed a typo in the Underbody Outward L “Terminal Preferences” for the Heel.
 - Fixed a typo in the L, N & M solder joint goals for Side Concave with Pitch > 1.27 mm
 - If you created any Side Concave library parts with a pin pitch > 1.27 mm we recommend that you recreate it
- Library Editor:
- Parts downloaded from POD can now translate Physical Description and Footprint Names from millimeter units to mil units using “Tools > Regenerate Library”
- Calculator:
 - Moving some parts to FP Designer were throwing exception errors
 - Fixed an issue related to Polarity Marking on some occasions not saving to FPX
 - Fixed an issue with footprint rotation turning off the “Build Part” option
 - QFN – if you had a part loaded with Thermal Tab, selecting the Clear icon did not clear the Chamfer or Radius data from the buffer
 - Fixed an issue regarding mil unit Pin Pitch rounding to whole integer when using Build Library (Batch Build)
 - Fixed an issue with SOT23 Gang Mask
 - Added Chip, Molded Body and MELF to the list of parts that can go from Calculator to FP Designer
 - Fixed an issue with SOT143 producing double Legend outlines

- Added Diodes to SOFL component family?
- FP Designer:
 - Legend trimming issue, caused by basic flaw in the trimming algorithm, allowed Legend on pads; issue only occurred when Legend Line Width and Gap Space were different values
- Allegro/OrCAD PCB:
 - When using version 15.7, the program will shorten any pad stack names over 18 characters, to 18 characters, which is the maximum length for pad stack name that that tool supports
- PADS Layout Translator:
 - Fixed an issue with the program creating an XML file on each CAD library output
- Documentation:
 - Updated IPC-7351B and IPC-7351C Proposal Excel spreadsheets in the Documents folder, Version 1.4

Updated Library Expert Surface Mount Families PDF file

Version 2016.12

NEW / ENHANCED:

- Library Editor:
 - Added a new column to all FPX files at the end called “Date” when selecting “New FPX”
 - When you add a new part to FPX, the Date is auto-populated if the Date column is present
 - When you update an existing part and update FPX the Date will reflect the new Date & Time
 - Existing FPX files must have the Date column manually added if you want this feature
 - If you create a New FPX and don't want the Date column, RMB > Delete > Column
 - Updated SM, BGA, TH and Sample FPX files with a Date / Time column
- Altium:
 - Added a new translator option for outputting Stroke Text Font

FIXED:

- Calculator:
 - Updated Footprint Naming convention for parts with Thermal Tabs. The L X W was backwards.
 - To update your FPX file Footprint Names select – “Tools > Regenerate Library > Names”
 - Added the DPAK Thermal Tab data to the Footprint Name
 - SON with Thermal Tab produced an incorrect Thermal Pad when building the footprint in Batch Build (instead of Build Part)
- FP Designer:
 - When adding a FP Designer part with a Thermal Pad to FPX, the program was changing the 50% Paste Mask Reduction to 100%
 - When naming a Footprint with Case Code, the program now replaces periods with an underscore _ and lower case letters with UPPER case
- CAD Tool Translators:
 - Not outputting the Triangle Polarity Marker
- Allegro/OrCAD PCB:
 - Fixed an issue with D-Shape pad pin text coming out upside down when the Pad was rotated

- D-shaped Pads now have their pin number rotated properly for 180 and 270 degrees to match what is done with the symmetrical pads. This will cause a D-Shaped Pad Stack to be created for each rotation.
- When you map Component Outline to PLACE_BOUND_TOP, Poly Component Outlines will now properly be built as shapes rather than lines

Version 2016.11

NEW / ENHANCED:

- Library Editor:
 - Can now update multiple rows from a single component change
 - If you have the same component dimensions for multiple part numbers
 - Can now Right Mouse Button Copy & Paste into multiple cells from a single text

FIXED:

- Calculator & FP Designer:
 - Positioning of rotated symbols and keep-outs.
 - Self-correction for circle symbols where user data has been incorrectly entered
 - Self-correction for removal of duplicate symbols that occupy the same space
 - Dimensional values for Nominal dimensions are passed from Calculator to FP Designer
 - Fixed a Legend Polarity Dot touching Pin 1 issue for 2-pin parts
 - Fixed an issue with pullback leads. Entering the min/max dimension did not generate the necessary nominal dimension.
- Library Editor:
 - Fixed an issue with Find/Replace not holding the text when using it multiple times
 - When new text is manually or copy/paste entered into cell use Return key to hold it
 - Fixed an issue with Find/Replace All/Match Case
- FP Designer:
 - Pin step numbering and pin properties not reflecting unit selection
 - Set the minimum pin increment can now be zero for vias in FP designer
 - Fixed an issue with offset slotted hole solder mask
 - Fixed the drawing symbols and keep-outs shifting around with rotational changes
 - Fixed an issue where the program was not loading the Physical Description until the user selected the Finish tab
 - Fixed an issue with rotating round parts
- Preferences:
 - Changes to 'SOD' and 'Side flat, concave, convex – pitch = 0' user preference changes weren't being saved
- Polarity Markers:
 - Unable to place a symbol under certain conditions
- 3D STEP:
 - 3D Model output now has a checkbox option 'Use .STEP'. If this box is checked, you will get .STEP extensions for 3D model files. If it is not checked, you will get a .STP extension. The default is on but can be set using the Save Entries As Preferences.
- Allegro/OrCAD PCB:
 - Between builds S002 and S005, Cadence slightly modified the syntax of one of the commands used by the 17.2 scripting. A change has been made to reflect this, but it will BREAK all builds before S005. You *must* upgrade to S005 or later of Allegro/OrCAD PCB after you implement this patch. This will also fix the No DEFAULT INTERNAL issue that people on S005 were having.
 - Added a command which turns off the line merge capabilities of Allegro during Graphic placement, and another that turns it back on when done. This should prevent

component outlines from accidentally merging with terminal outlines and causing graphical issues.

- Expedition:
 - Expedition should now be able to detect whether your part is Mixed, Surface Mount, or Through-hole and set the mount type appropriately

Version 2016.10

New/Enhanced:

- LEAP (Library Expert with Advanced Preferences):
 - New LEAP library content subscriptions available here – www.pcblibraries.com/LEAP
 - Adds hundreds of thousands of new parts not on POD
 - Includes Schematic Symbols, Footprints and customized 3D STEP models
- Documents on – www.pcblibraries.com/downloads
 - Updated – Surface Mount Component Families.pdf
 - Added BGA and CGA IPC-7351C (Proposed) Solder Joint Goal Tables v1.2.xlsx
 - Updated – IPC-7351B Naming Convention for Mounting Holes

Fixed:

- Calculator:
 - Non-plated Hole footprint name now has a “A” when Annular Ring Pad Shape is used
 - Updated the indicator light in the Viewer to enhance the description of the various colors
 - Fixed all parts with Thermal Tabs with disappearing radius corners when outputting Terminals
 - Fixed PSON Mfr. Footprint ‘S’ dimension display value
- Preferences:
 - Added new 3D STEP families for through-hole Radial Disk Resistor and Varistor
 - CGA and LGA Terminal Leads were updated in the IPC-7351B version
- FP Designer:
 - Fixed an issue when the hole size exceeded 12.70 mm the program now throws a Warning message
 - Calculator QFN was dropping the Legend line if it was less than 5 mm. Rule changed to 2 mm.
 - Fixed an issue with Quad packages in Calculator and moved to FP Designer created dual pad stacks with the same name
 - Fixed the pin text so long pin names will not overlap
- 3D STEP:
 - Added through-hole Radial Disk Resistor and Varistor
 - Fixed Side Concave 2, 4-Pin Oscillator for micro-miniature version
- PADS to CAD:
 - Fixed a rotation problem concerning free-floating pieces of paste mask
 - Fixed an issue that was causing the corner radius on rounded rectangles to only be half what they should be when importing PADS files
- PADS:
 - Fixed an issue that was causing D-Shaped pads to translate to PADS improperly
 - Fixed an issue with chamfered cornered pads with associated copper
- Expedition:
 - Fixed a translation problem with paste mask checkerboard when it’s on a pin that is both rotated and offset from the origin
 - Fixed a problem that was causing the layers not to fully display
 - Fixed a problem that was preventing the Component Outline from being translated into a Placement Outline
- Allegro:
 - The 17.2 pad stack scripts will now set the unit precision manually to 4 decimal places. This mirrors what was done previously and allows it to not rely on the default precision.

- CR-5000:
 - m callouts for solder mask pads will split out properly to be used for the pad name when using IPC-7351 naming
 - Adjusted the naming so that the corners callout stays with its copper pad name, and not the paste mask callout of the pad name
 - Fixed a bug that was causing paste mask checkerboard to also be generated outside of the pad stack, causing an error
- OrCAD Layout:
 - Terminal Shapes will be redirected to Layer 16 (Comment Layer)
- Windows:
 - Fixed form format in windows with 125% resolution bug causing Calculator Form Layout issues

Version 2016.09

NEW / ENHANCED:

- Calculator to FP Designer:
 - This new feature allows the user to transfer one way any Calculator part to FP Designer to use features in both modules
- Calculator:
 - Updated the Land Pattern Naming Convention for 2-pin Chip, Molded Body and SOD parts to match IPC-7351C
 - Upgraded 2-pin parts for the silkscreen to map to the real component body outline
- Library Editor:
 - Updated the SM, TH and BGA FPX files in the www.pcblibraries.com/downloads webpage
 - To update your FPX file: In Library Editor select Tools > Regenerate Library > Names
- Help:
 - Added “Check for Update”
- CAD Tool Translators:
 - Added a ‘User Guide’ button in bolded red text to help users with technical issues

FIXED:

- Polarity Marking:
 - Fixed an issue with Polarity Marking attaching to the centroid marker
- Library Editor:
 - Fixed an issue with BGA’s saving Local Fiducial data in FPX file
- 3D STEP:
 - Fixed a general bug where if you changed the default orientation of a part And moved the origin away from Center of Extents, it would cause the model to build improperly
- Allegro/OrCAD PCB:
 - Fixed a problem where it would crash if you excluded the labels in FP Designer. Keep in mind, if you do this Allegro will throw a warning about not having a ref des.
- PADS:
 - Fixed problems that were causing labels to be suppressed when outputting a PADS ASCII file
- KiCad:
 - Keep-outs, as they aren’t translatable to KiCad, are now filtered properly
- Pulsonix/PCAD/DesignSpark:
 - Fixed a problem that was causing component outlines for certain families to be suppressed

Version 2016.08

NEW / ENHANCED:

- Documentation:
 - What's New in IPC-7351C? has been updated – www.pcblibraries.com/downloads
 - Help > Topics > User Interface has been updated
 - Added new Excel spreadsheet documentation to compare IPC-7351B with the proposed IPC-7351C
 - Added in this installation folder:
C:\Program Files (x86)\PCB Libraries\Library Expert 2016\Documents
- Preferences:
 - Add Incremental Pin Pitch for Gull Wing and Concave and Body Length for Chips
 - Now you have more options for solder joint goals
 - There is a new Preference file – PCB Libraries.dat that has all new solder joint goal values
 - If you need a webcast for an explanation of this new technology contact us
 - Additional information about this update is here - http://www.pcblibraries.com/forum/new-2016-solder-joint-goals_topic1921.html
 - Added a new Preference > Rule option for Cathode/Anode default pin names – Alphanumeric or Numeric
- Altium:
 - A layer tabs update has been installed just before the full update at the end of component creation to help post-process scripts
 - At the start of library creation the cursor will now become an hourglass, at the end it changes back to an arrow
 - As part of the STEP update, updated the scripting so that it can handle mils and inches natively. As there are no functions that I know of for Microns, millimeter conversions of the Micron values (basically what was done before) are used in that instance.
- FP Designer:
 - Pad Stack Manager:
 - Any edited pad layers with a zero pad size are deleted
 - Any pad stacks with all pad layers with a zero pad size will be deleted unless there's a hole
- 3D STEP:
 - Added a new option for micro-miniature SOFL's

FIXED:

- Preferences:
 - Changing the Pad Stack Name to Mils did not save to Console Preferences
- FP Designer:
 - Creating hole sizes in Mil Units sometimes rounded of incorrectly
- Calculator:
 - Fixed an issue with 4-Pin Concave Chip Array pad size calculation
 - Fixed an issue where the calculator Footprint tab was dropping the default values
 - The scroll bar in the component dimension panel now defaults to the UP position
 - Fixed a Legend issue with Corner Concave Oscillators
 - Fixed the Pin 1 default location for Receptacle Headers to upper left
- 3D STEP:
 - Fixed an issue with changing units between mils and mm
 - Fixed issues in dealing with non-millimeter unit types
 - Fixed a graphic display issue for the color selection images when the screen resolution was set to 125%
- CR-5000:

- Fixed an issue where terminal outlines were causing the pad stacks to build improperly
- Allegro/OrCAD PCB:
 - Fixed an issue with pads with associated copper being rotated in the wrong direction during a PADS -> CAD Conversion

Version 2016.07

NEW / ENHANCED:

- Documentation:
 - Updated Surface Mount and Through-hole Component Families. Download here – www.pcblibraries.com/downloads
- Calculator:
 - Added keep-outs for contour courtyards
 - When you Mirror a BGA to see the Bottom Side, the indicator light will turn Yellow and throw a warning
 - Molded Body Mfr. Recommended Footprint “L1” dimension would not allow user to edit the value
 - Through-hole Oscillator Physical Description was OSC and fixed to Oscillator
 - Fixed the Triangle Polarity Marker to Fill correctly
 - Headers – added a new origin location, Center of Pins
 - Added Axial Lead Non-polarized (bidirectional) Diodes
 - Dip Socket – fixed the auto-generated polarity line marker
- Preferences:
 - Drafting > Courtyard > Origin preferences were split into crosshair and target
- 3D STEP:
 - Added support for Center of Pins that is used by the Right Angle Post Header
 - Added support for the non-polarized Axial Diode
- Altium:
 - Added a Secondary Reference Designator has been added that can be used for Assembly. You can change the height of this designator by setting the Assembly Ref Des Height preference and you can turn it off by changing the Assembly Ref Des Height to 0.00.
 - Added an option ‘Use Mask Expansion Rules’. Disabling this option will prevent the default expansion rules from being used for Solder Mask and Paste Mask should the expansion be 0. This will provide a WYSIWYG 1:1 building of the Mask to the Pad if no expansion is used.
- Xpedition:
 - Fixed an issue with oval slotted holes not translating properly
- Allegro/OrCAD PCB:
 - Fixed an issue that was causing parts with Keep-out Bottom to crash
- CR-5000:
 - Added Terminal Outlines support. It gets its own layer, with a default of SYMBOL-A1 (Assembly). Layer is user definable.

FIXED:

- Pulsonix:
 - Fixed an issue that was causing Polys attached to a pad to not translate properly
- Pantheon:

Fixed an issue with oval slotted holes. A diameter the length of the short side is used for a circular drill. Turning it into a full oval slot is currently a post-process by the user

Version 2016.06

NEW / ENHANCED:

- Calculator:
 - Added another “Add to FPX” toolbar icon close to the OK button
 - Changed the Reset button to Clear and the Reference button to Demo
 - The letter “R” was added to the Footprint Naming convention for Reversed Order Pins
 - Removed the Mirror option for Through-hole component families
- FP Designer:
 - Added a new feature that replaces special characters in Part number with underscore or hyphen when using the Part Number to generate the Footprint Name
 - Updated and relocated the “Done” button to the “OK” button to match the Calculator UI
 - Added a Sample Part
- Help:
 - Updated the IPC-7351B and C Land Pattern Naming Convention documents
 - Updated the “Important FPX Guidelines” document
- Allegro:
 - Add support for 17.2

FIXED:

- General:

Fixed a bug introduced in V2016.05 that would cause wrong corner pad radius values to be generated

Version 2016.05

NEW / ENHANCED:

- Preferences:
 - Added IPC-7351C footprint naming convention
Library Expert now supports IPC-7351B and IPC-7351C Land Pattern Naming Conventions
 - Added back the Fabrication and Assembly Tolerances
Note: It is highly recommended that you create a new Preference file
- Calculator:
 - Through-hole Headers – updated the pin pattern option graphics
 - Updated the GUI markers for Required and Optional entries
 - Updated reference graphics for required and optional dimensions
 - Enlarged indicator light for Red, Green and Yellow status
 - Changed Transistor Outline (TO, DPAK) to DPAK
 - Changed SOTFL to SOFL and added the IC component family
 - Removed all A1 dimensions from Through-hole dimensional thumbnail images
 - Increased the Window Splitter line width
- Altium:
 - Added freeform Keep-outs
- PADS:
 - When two pins with same pin name are used, PADS will now properly use the generated FPX alphanumeric in its output
 - The default line width for associated copper was updated from 0.01 mm to 0.05 mm
- General:
 - Increased the pad size of the internal pad that is generated for pads with associated copper

FIXED:

- Altium:
 - Fixed an issue with poly shaped pads
- Allegro/OrCAD PCB:
 - Fixed a problem that was causing Excess to not save as a preference
- PADS:
 - Fixed an import issue when dealing with a .p and .d that had two different sets of alphanumeric pins
 - Fixed an import issue when there was no inner pad or drill, but an anti-pad and thermal were in the pad stack
- 3D STEP:
 - Fixed a problem with the QFP that was causing a face to display improperly

Fixed an issue with SOIC that was causing it to display improperly in some viewers

Version 2016.04

NEW / ENHANCED:

- Calculator:
 - 3-pin DFN – Added the IC component family
- Preferences:
 - Updated the “Drafting > Legend > Auto-generate Pin 1 Indicator Line” to include 2-pin parts
 - Updated all the wireframe models for editing color assignments
 - Updated solder mask size resolution to 3 decimal places
- FP Designer:
 - Updated the Finish Tab to automatically add spaces to the Case Code if it is used to generate the Footprint Name
 - Removed the period. as an illegal character for a footprint name
- 3D STEP:
 - Updated all the wireframe models for editing color assignments
- CR-8000:
 - Added a new option for CR-8000. The only difference between this and CR-5000 is that radius values will be built properly per spec. It's a known bug in Zuken CR-5000 that the radius of a curve must be negative to display properly. This was fixed in Zuken CR-8000 and the output for this translator has now been adjusted to match, while allowing the CR-5000 that is correct to remain the same.
- CADSTAR:
 - Added support for CADSTAR 17
- Allegro/OrCAD PCB:
 - Added keep-outs
 - If using a Courtyard No Probe Area, the No Probe Area will be circular for Radial Electrolytic, Radial Inductor, and Radial LED Land Patterns

FIXED:

- 3D STEP:
 - Fixed an issue that was causing XTAL's to not expand their height properly
 - When the material condition is set to Maximum or Minimum in the 3D window, if a Maximum or Minimum value does not exist for a parameter, the Nominal value will now be used
- CADSTAR:
 - Terminal Outlines will be mapped to the Component Outline (Top 3D Model Outline by default) layer
- Expedition:
 - Fixed an issue with thermal tab paste mask, when the thermal tab is offset from the origin

Version 2016.03

NEW / ENHANCED:

- General:
 - New installation certificate to prevent Windows 10's SmartScreen false alarm alert
- Main Toolbar:
 - Relocated the "Parts on Demand" icon from the "Library Editor" toolbar to the "Main" toolbar
- Calculator:
 - Update PTH TO-220 to add Alpha-numeric pin assignments
- Preferences:
 - Updated the Components tree to add PSON and PQFN
- FP Designer:
 - Updated the Finish Tab to allow the user to enter Case Code and make that the Footprint Name
 - Updated Non-plated hole pad shape from "Donut" to "None"
 - Updated the pad stack name when the inner layer pad size is different than the outer layers
- Tools > PADS to CAD:
 - When you enter either a part type or a part decal filename, the program will check if the other file is available, and automatically fill in both boxes
 - Now reads free-form text properly
 - Through-hole parts will now have proper naming that uses the h value to denote hole size
- Expedition:
 - Adjusted the ASCII Keep-out translation to support Placement Keep-outs along with Plane Obstructs. Trace and Via are now separated should one be excluded.
 - If the Top Pad of a Pad Stack is Complex, while the regular Pad Stack name is not, the two values will be concatenated together, spaces will be removed, and COMPLEX will be shortened to CMPX. This should kill any potential naming collisions with regular pads in Expedition.
Ex: s100s86_CMPX_12.2X19.5_A237.9_S1
 - Fixed a problem in the ASCII output that was causing MILS not to work. Microns and Inches should also work properly now.
 - Mixed (having Parts and Pad Stacks of more than one Unit type) ASCII is now fully supported
 - Added a Translation Target Type of ASCII per customer request. This will allow the output of plain vanilla ASCII files before they're Encrypted. Useful if you have Expedition 2007 or earlier. Use the Encrypted ASCII selection for anything later.
 - Fixed various conversion issues
- Altium:
 - Pin Shapes on Paste Mask Bottom now import properly
 - The Altium Translator now supports the ability to declare a layer from eMechanical1 to eMechanical50. Any mapping value over 50 will get set to layer eMechanical50.

FIXED:

- 3D STEP:
 - Fixed unit conversion issues for the BGA, CGA, LGA, and DIP families
 - XTALDFN – Fixed an issue in the 3D Model calculation that was generating improper data
 - Fixed an issue with QFN/PQFN rounded and rounded/chamfer thermal tabs that was causing it to display slightly incorrect leads
- Preferences:
 - Terminals > Through-hole > Hole Over Square Lead – was confused with Hole Over Round Lead
 - Drafting > Assembly > Ref des > Zero – changing the value to zero caused Unhandled Exception Errors

- Files tab > Number of Backups – causing problems when the FPX file was on a network drive
- Allegro/OrCAD PCB:
 - Parts with a circular courtyard diameter over 0.50 mm (to exclude the origin crosshair) now import properly as shapes, rather than circles. This will allow the package height to import properly for circular parts.
 - Circular parts now also mirror the PLACE_BOUND_TOP courtyard to DFA_BOUND_TOP
- EAGLE:
 - Fixed an issue that was suppressing attached copper in certain circumstances

Version 2016.02

NEW:

- Calculator:
 - Added Radial Disk Resistors and Varistors
 - A new attribute was added for the part number on the assembly layer
- Altium:
 - Add Reference Designator to user definable layer in CAD output menu
- Xpedition:
 - Added Top, Bottom and All Layer Keep-outs

FIXED / ENHANCED:

- General / Calculator:
 - Updated Sample.fpx
 - Fixed an issue with filled Polarity Markers were translated as Copper and not 2D Lines
 - The Part Type (Part Number) text object will now be placed on the assembly layer for all CAD tools
- FP Designer:
 - Fixed a legend trimming bug
- Preferences:
 - Updated the default color assignments for Keep-outs
- Allegro/OrCAD PCB:
 - Fixed an issue dealing with naming, when you have multiple pad stacks that are nearly the same
 - Fixed an issue when dealing with values to 4 decimal places getting rounded to 3 decimal places
- EAGLE:
 - At a Solder Mask or Paste Mask expansion value of 0, the Eagle rules will now take over and the Solder Mask and Paste Mask will be generated internally by Eagle. The exception for this is a Thermal Tab. All thermal tabs will have their Solder Mask and Paste Mask generated manually
- KiCad:
 - Fixed a missing bracket "(" before the "descr" keyword in the *.kicad_mod file
 - Added the ability to have a Part Type value as VAL**
- PADS p/d Files:
 - Updated PADS p/d files – www.pcblibraries.com/downloads
- Library Expert Lite:
 - Fixed the Oscillator a pin limit from 0 to 6

Version 2016.01

NEW:

- IPC-7351C Solder Joint Goals:
 - Changed all Negative solder joint goal values to zero 0
 - Changed Fabrication Tolerance from 0.05 to zero 0
 - Changed Assembly Tolerance from 0.025 to zero 0
- Toolbar icons:
 - Completely overhauled the Toolbar icons
- FPX File Backup:
 - Added a new feature in Preferences that allows the user to set the number of .BAK files
- KiCad:
 - New CAD tool interface

FIXED / ENHANCED:

- Calculator and FP Designer:
 - Add keep-outs to “Bottom Layer” for through-hole parts
- Calculator:
 - DFN silkscreen outline had open corners when Preferences > Drafting > Indicate Polarity was turned off
 - DFN pin shape complies with the preference to add line for pin 1
 - BGA & QFP fiducial preferences now are added according to preferences for the fiducial threshold in both the calculator and from a FPX library
 - Thermal pad on DPAK with Rounded Corners bug
 - A message was added to decline the Hide command for Thermal Tabs
- Preferences:
 - Window does not reset on Save
 - Removed the Close X control in the upper right corner
 - When editing Preference > Terminals and selecting the X to close the Save Settings option did not work
 - Replaced GUI text for Drafting > Silkscreen to Drafting > Legend
 - PTH preference ratios are better defined. Added Hole over Round Lead and Hole over Square Lead.
 - Fixed an issue when the Courtyard Line Width was set to zero the Header component family assembly outline reacted unfavorably
- CAD Tool Output:
 - Fixed issues involved with validation of text boxes for certain translators in the translator window. All text boxes should now have proper validation.
 - All GUI text Silkscreen was changed to Legend
- Xpedition:
 - Encrypted ASCII will now have the parts be a mount type of Mixed. This matches the functionality of the Direct Import.
- Allegro:
 - The plus + polarity marker was coming through as a 4
 - Radial LED's are now working correctly
 - Rectangular Complex Pad Stacks will now have a 4-digit callout that represents the shape of each corner. This should prevent naming collisions.
- Target 3001!:
 - Fixed a problem that was causing a preference to be saved erroneously and the program to crash