Version 2019.07

NEW / ENHANCED:

- Calculator:
  - Add Transformer component family to the SOFL terminal lead form
  - Fixed and Naming Convention issue with Through-hole Radial “Inductors, Polarized” – INDP
  - The Through-hole Radial Inductor was throwing an Unhandled Exception Error when moving it to FP Designer
  - Fixed the Polarity Dot issue not coming through from Through-hole component families

- FP Designer:
  - Fixed an issue with importing a .CSV text file with x/y coordinates
- Allegro/OrCAD PCB:
  - A Keepout on Top or Bottom inside of a Pad Stack will now be sent to the appropriate ROUTE KEEPOUT layer. Previously, as all pad stacks have a Placement Keepout Restrictions flag, they were going to the PLACE_BOUND layers. Its assumed that the PLACE_BOUND shape will come from the Courtyard, so this should be proper.
- Altium:
  - Fixed an issue with rounding on ultra-small parts that use non-millimeter unitsKiCad:
  - Fixed an issue that was rounding the paste mask on thermal tabs
- Pantheon:
  - Fixed issues with associated copper

Version 2019.06

NEW / ENHANCED:

- Allegro:
  - Pad Stack script generation will now turn on ‘Suppress unconnected internal pads; legacy artwork’, so the user doesn’t have to

FIXED:

- Preferences:
  - Fixed an issue with IPC-7351B Land Pattern Naming Convention for QFP and QFN component families
- Print PDF Feature:
  - Fixed an issue where the print PDF feature was only printing the page Header

- 3D STEP:
  - TO220 Vertical – Fixed issues with the 3D Model generation when D1 exceeded D
  - LCC component family STEP Model generation modified so that the pin 1 marker scales better for smaller parts with pin 1 at the edges. The diameter will now match the width of its lead.
- Pantheon:
  - Fixed a bug that was causing the Ref Des to improperly generate
  - Terminal Outlines are now suppressed and won't show in the CAD output, as the [not used] mapping wasn't working in modern Pantheon

Version 2019.05

NEW / ENHANCED:

- Drafting Symbols:
  - Added Bottom Solder Mask layer
- Altium:
  - Changed the default Target Library form "Use Existing" to "Create New"

FIXED:

- Drafting Symbols:
  - Fixed an issue when re-selecting the Drafting Outline on the Top Solder Mask layer, the Preference setting flipped back to Top Legend
- Calculator:
  - Fixed the Mounting Hole Footprint Name bug
- 3D STEP:
  - QFP – Fixed an issue that was causing the body to have errors
  - QFP w/tab – A minimum standoff of 0.02 mm is enforced to make sure the thermal tab model does not break
- Altium:
  - Fixed a problem where the Assembly Ref Des wasn't getting the proper Width value

Version 2019.04

NEW / ENHANCED:

- Calculator:

    - Custom Zero Component Orientations now saved and retrieved to FPX file
        - If you change the rotation of any calculator part, that unique rotation will be saved to FPX
        - This feature allows the user to orient every footprint per the manufacturer's tape and reel rotation saving time and money in assembly
- Updated the SM, TH and BGA FPX files that come with Library Expert Enterprise
    - Now contain over 9,000 different manufacturer Case Codes
- Zuken:
    - Added support for eCADSTAR

FIXED:

- Preferences:
    - Fixed – Terminal > Rectangular End Cap > 2.85 - 3.85 > Paste Mask, not holding updated data

- 3D STEP:
    - Fixed bugs dealing with unit conversion that would cause errors in STEP Model leads, when processing something other than millimeters
- Altium:
    - Setting the Text Width for Reference Designators to the specified value in the FPX, is now supported

Version 2019.03

FIXED/ UPDATED:

- Altium:
    - Fixed an issue where round pads were accidentally getting rotated 10 degrees
- Xpedition:
    - Fixed an issue with multiple paste mask or solder mask, should it include both polys and rectangles. Everything for a layer of a pin, should now be put into one complex pad.
    - Removed the deprecated "Direct Import" option (as it's obsolete)
- CADSTAR:
    - Adjusted the Drill Orientation so that it nomalizes to a number between 0 and 360. I.e. -90 will become 270. CADSTAR does not allow for negative rotations on the drill orientation.
    - The LASUBTYP line will now be excluded from output
- Allegro/OrCAD PCB:
    - Fixed a problem that was causing custom parts with through holes to set the thermal pad to 0 diameter
    - Fixed a problem that was causing duplication of associated copper that contained arcs

- General:
  - An updated rounding has been added for non-Millimeter values across all translators. This should hopefully round numbers that are very close to their actual number properly. Ex: 5.9998 to 6 or 5.1996 to 5.2. Millimeter values don't have the issue, as they're the native units.

Version 2019.02

FIXED/ UPDATED:

- Preferences:
  - Fixed the Land Pattern Naming Convention to add the “Reverse” (mirrored) suffix “R” to the footprint name
- Calculator:
  - SOT23 component family – Fixed the naming convention to include the pin quantity
- Library Editor:
  - Fixed the feature for “Tools > Update BOM From Library” which was disabled
  - Restricted in the free Library Expert "Pro"
- Xpedition:
  - Fixed an issue with Keep-outs that was causing them to come in as circle paths with zero line_width. They will now be set properly to circle shapes.
  - Fixed free-standing Arcs, which were coming out as circles
- Allegro/OrCAD PCB:
  - Fixed an issue with pins when no rotation was given (usually mounting holes)
  - The Device File will now exclude Mechanical Pins on both the PINORDER line and FUNCTION line, adjust the pin count downward by the number of Mechanical Pins
  - Fixed an issue with bottom layer SMD pads not generating properly
  - Fixed an issue with 17.2 S048 not being recognized as able to do donut shapes
  - Fixed an issue where equivalent Custom Pad Stacks weren’t being filtered properly
  - Fixed a visibility Issue where Parts Per Script was being disabled for 17.2.048
- CR5000/CR8000:
  - Fixed a crash bug with the Pad Naming set to CR-5000 style
  - Fixed an issue with terminal outlines
- CADSTAR:
  - Fixed an issue where Through-hole Pins were being misinterpreted as Surface Mount
  - Unique Parts:
  - Fixed an issue that was causing rectangular keep-outs to erroneously have a line width
- 3D STEP:
  - QFN – Fixed an issue that was causing Allegro/OrCAD PCB to choke
  - Custom 3D Colors: Turned the ability to edit custom 3D Colors off until the OK button calculates the Footprint
  - Color Preferences: Fixed an issue that doubled the Color Preference options when opening, closing and reopening Preferences

Version 2019.01

NEW / ENHANCED:

- Library Editor:
  - Added a new feature to auto-generate a new “Value” column when creating a new FPX file.
- OrCAD PCB Designer:
  - Added an option for the OrCAD PCB Designer Basics license. If you have this, check this box and save it as a preference. If Allegro crashes due to OrCAD PCB Designer Basics not being found, uncheck this option and save it as a preference.
- CR-5000/CR-8000:
  - As there isn’t an IPC-7351 pad naming for super-irregular pads like Paste Mask Checkerboard, all pads of that type will be generated with the CR-5000 style ‘Custom’ call out to preserve uniqueness while allowing Pads of that type to group together.

FIXED:

- Library Editor:
  - Fixed an issue when importing or copy/paste FPX files, the Datasheet link was not synced to the correct column.
  - Fixed an issue when importing multiple FPX files and some had the same Part Number the “Do this for all parts” feature was not functioning.
- 3D STEP:
  - BGA Component Family: Ball diameter will now properly display the balls at the specified “b” dimension. Previously, it used the calculated footprint pad size, which was slightly smaller.
  - Molded Body Component Family: User custom color assignments were not working correctly.
  - SOP/SOT23/SOT143/SOT223 Component Families: Polarity Marking – Fixed the body so that when the Pin 1 is placed, if pin 1 is too close to the edge, it will indent the Pin 1 designator towards the center just enough so that it keeps it away from the edge in both directions.
  - SON Component Family: Fixed an issue with the Pin 1 Indicator on small height parts. The width of said indicator will now always match the width of the lead. This will prevent collisions with the outer edge of the body on some parts.
- KiCad:
  - Fixed silkscreen ref des to be REF**, and fabrication ref des to be %R. Previously, they were reversed.
- CR-5000/CR-8000:
  - Fixed issues with custom poly-shaped pads.
  - Fixed the Circle generation so that it gets made with 360-degree arcs that come back to themselves. This is to match how Zuken does it in the latest tool.

- Fixed an issue whereupon pad stacks with a paste mask checkerboard would call out the paste mask pad a number of times equal to the number of pieces of paste mask. Now it properly just specifies the pad on that layer once.
- Fixed a bug that was generating erroneous non-named pads via the Terminal Outlines.