Version 2023.14

<u>Fixes & Enhancements:</u>

- Options:
  - New Option file for IPC-7351B was added to the installer
  - Terminals – New "Flat Lug Lead" added
  - DPAK tab terminal redefined as new Flat Lug Lead
  - Added an information icon to the DPAK Paste Mask shape selection options
  - IPC-7351B Footprint Name prefix for polarized capacitor changed from CAPPM to CAPMP

- FP Designer:
  - Paste Mask for Through-hole (Pin-in-Paste Technology) was not working
  - Pad stack names for rounded rectangles on only one side missing the corner locations (r, l, t, b)
  - Pad stack through-hole calculator Units not working correctly.

Version 2023.13

<u>Fixes & Enhancements:</u>

- Batch Build:
  - Create new name in selected units not working
- Options:
  - "Rectangular End Cap" terminal definition changed to "Rectangular or Square End" per IPC-7352
  - "Side Lead" definition changed to "Leadless" per IPC-7352
  - J-Lead terminal option for Most Density Level toe changed from 0.01 to 0.10
  - IPC-7352.opt – Changed Inward Flat Ribbon L Lead Height > 4.20 mm Least Density Side from -0.05 to 0.00
- 3D STEP:
  - Chip and Molded body 3D polarity stripe can appear toward #2 pin in small package parts
- FP Designer:
  - Move Origin by X & Y using non-millimeter units isn’t working
  - Assembly outline polarity not responding to Mirror or Rotate footprint commands
- Library Editor:
  - Code added to disallow extended ascii in all text either edited or pasted
- Calculators:
  - DFN 3 and 4 pin – The IPC footprint naming convention had the pin qty listed twice
  - Corner Concave Package polarity dot is not following pad-silkscreen rule
  - Molded Body:
    - Added Thyristor component family and 3D STEP model
    - Added D – D1 nominal not to exceed 0.60 mm error flag
- User Interface:
  - Resize options now allowed on all dialog boxes
  - Viewer mouse wheel functions re-defined – (no key) pan up/down, +shift - pan right left, +control - zoom in/out
- CAD Tool Interfaces:
- Altium:
  - Added option to include footprint name in the ‘save-to’ directory path
- KiCad:
  - Pad non 0-90-180-270 orientations now allowed

- Rectangular thermal tab pattern defined solder mask shapes not filled
- Thermal tab rectangular shapes corners are always rounded

Version 2023.11 + 2023.12

Fixes & Enhancements:

- Calculators:
    - Calculator now will not trim pads for any part with manufacturer's dimensions and will display (in Footprint Warnings) which trimming functions were needed
    - SMD Thermal Pads – Paste Mask checkerboard pattern created a non-symmetrical aperture when rounded corner pad shape was used
    - Through-hole pad stacks – a minor adjustment was made to the PTH pad stack calculations for Thermal Relief patterns and Anti-pads
- FP Designer:
    - Via pad stack calculator was not producing the correct Anti-pad size
- CAD Tool Interfaces:
    - KiCad – versions 6 & 7 were not creating Rounded Rectangle or Chamfered Corner pad shape
    - Eagle – rectangular through-hole pads were converted to square
    - Allegro – Allegro import error 'Form editprop1'
    - CAD tool options are now saved to individual files (this allows multiple setups to be saved)

Version 2023.10

Fixes & Enhancements:

- Calculators:
    - Chip Array
        - 4-sided Concave/Flat – added Filter component family
        - 4-sided Concave/Flat – added the ability to change Pin 1 location
        - 2-sided Concave/Flat – added Filter component family
    - Molded Body & MELF – inserting Nominal height and tolerance generated the wrong height in the footprint name
    - Corner Concave Oscillator – added a provision for an expanded silkscreen outline
- Options:
    - DPAK and DFN 3-Pin – added a feature to change the thermal pad paste mask from Solid to Pattern with 3 options
        - Automatic is a solid shape if the tab terminal extends beyond the body and has a toe; a pattern if the tab is only under the body
        - Simple is always a solid shape
        - Pattern is always a checkerboard
- CAD Tool Interfaces:
    - Zuken CR-5000/8000 D-Shaped pads
    - KiCad now supports version 7
    - Allegro version 22.1 version selection not working
    - Altium – Save My Settings was not saving changes to the Mechanical Layer "Assembly Bottom"

Version 2023.09

Fixes & Enhancements:

- CAD Tool Interfaces:
  - Siemens PADS Standard – when in the translator menu, changing output from Decal to Decal and Part Type crashed the program
- Cadence Allegro/OrCAD PCB:
  - Translator bug that was corrupting pad stacks with arc corners
  - Device type label mini class subtype changed from “DEVICE TYPE” to “COMPONENT VALUE” with text of “VALUE”
  - Symbol type for fiducials and mounting holes changed from “Package Symbol” to Mechanical Symbol”
  - Added “Footprint Properties Lock” option
  - Added label size to text_block converter
  - Application of “OrCAD designer basics” checkbox function restored
- FP Designer:
  - The Component Outline also turned off when the Silkscreen Outline was unchecked
  - Duplication of mechanical pins in FP Designer
  - Pad Stack Designer – in certain cases, the through-hole pad stacks were not following the minimum annular ring setting in Options
  - Silkscreen and Assembly outlines not following Option Settings
- Calculators:
  - Axial Lead – Added missing ‘optional’ indicator note to Axial GUI
  - HC49 Through-hole Crystal – updated the Physical Description. Swapped “T” and “W” references

Version 2023.08

Fixes & Enhancements:

- Options:
  - Drafting > Silkscreen - added option to offset silkscreen outline outward of the body ‘Map-To’ dimension by half the line width which will insure no silkscreen under the body (for Surface Mount only). This feature is set by a checkbox in ‘Drafting/Silkscreen Options/All Density Levels’ “Offset outline away from SM body”. The default is checked/enabled. Uncheck to align silkscreen outline center with the ‘Map-To’ selection.
  - Added “Enable Edits” menu item to the Tools/Options dialog box. Expedites access to the file open and save as menu items.
  - Options not saving drafting/silkscreen/option for offset line offset
  - Tools/Options can’t be closed if it was dragged to a second windows screen and that screen is subsequently closed
- Library Editor:
  - Deleting a row when the cursor was in the edit cell mode threw an Unhandled Exception Error
- Calculators:
  - LGA, TH LED, Headers – the default terminal lead shape was updated from Round to Square
  - TO Flange Mount Horizontal – silkscreen outline-to-tab spacing was incorrectly calculated
  - TO Flange Mount Vertical – relocated the Pin 1 polarity dot and removed stray silkscreen on pads
  - SOP – pin 1 polarity line was getting located in the wrong place

  - BGA – changing the silkscreen “Map Outline to Body” from Maximum to Nominal nothing changed
  - DFN 3-Pin – changing the Silkscreen “Map Outline to Body” from Maximum to Nominal changed the Assembly Outline
  - Corner Concave Oscillator – Pin 1 dot location is 1 line width too far from the pad
  - Rounding for the calculator pad placement properties display and CAD translator location output were not synced
  - Silkscreen not following pad clearance rule for end pads on SOP when D dimension isn’t sufficiently larger than the row span
  - Silkscreen on CHPAX not following pad clearance rule for end pads
  - Corner Concave Oscillator (OSCCC) component type changed to Corner Concave Package (CCP). New Crystal family added to corner concave type. Libraries are unaffected. Footprint name for OSCCC is still the same, XTALCC now an option.
- CAD Tool Interfaces:
  - Cadence OrCAD PCB and Allegro – V22.1 version option added to interface
  - KiCad – Keepouts are enabled for a layer called “Cmts.User” (comments). This layer option was added in response to a user recommendation and cannot be reassigned or disabled.
- FP Designer:
  - Origin Options were disabled except for Defined Coordinates
  - Silkscreen outline offset added to FP Designer

Version 2023.07

<u>Fixes & Enhancements:</u>

- Options:
  - Updated the IPC-7352.opt file – The default Thermal Tab Paste Mask Reduction was changed from 50% to 60%
- New Library Files:
  - Added 5 FPX files to the Library folder that contain over 18,600 component mfr. Case Codes:
  - BGA, SM Discrete, TH Discrete, Semiconductors, Connectors
- Calculators:
  - Updated – Enlarged the graphic dimensional images so that the image fits in the box
  - DPAK – L1 dimension can now be a negative value and it autogenerates a Thermal Pad
  - QFN – Assembly Outline was not mapping to the Nominal dimension when set “Map to Nominal” in Options
  - LCC – Changing a package dimension threw an unhandled exception error
  - Flange Mount (Horizontal) – the silkscreen to Flat Lug Lead clearance was not following Option settings
- Library Editor:
  - When selecting a single footprint row, selecting the Batch Build option added a duplicate Density Level suffix
- FP Designer:
  - Silkscreen to Pad Clearance Option was not being applied to through-hole pads
  - Editing package dimensions was not updating the Physical Description
  - The 3D model was not generating a Polarity Dot
- CAD Tool Interfaces:
  - KiCad – translator didn’t produce rounded-corner shapes for surface mount Square pad

Version 2023.06

<u>Fixes & Enhancements:</u>

- Options:
  - Silkscreen Clearance was only generating 50% of the set value for some component families (QFN & QFP)
  - Added new feature in – Drafting > Assembly > “Add Value to Footprint” check box
- Calculators:
  - New silkscreen feature for all surface mount component families:
    - Instead of using the center of the silkscreen outline to draw it, the program now uses the inside edge of the outline to ensure the silkscreen is always visible after assembly and it will never be located under a component package
  - TO-92 – silkscreen trimming was not working correctly
  - DFN – added alphanumeric pin name option
  - SOFL – bug causing interference between polarity dot and silkscreen outline
  - QFN – bug that put polarity line in the wrong place
- Library Editor:
  - Sorting and column width bug in library manager
- FP Designer:
  - Specifications were not loading Physical Description correctly
- CAD Tools:
  - Kicad – Translation Non-plated hole descriptor
  - Kicad – Library folder name creator

Version 2023.05

<u>Fixes & Enhancements:</u>

- Help:
  - Added user option to deactivate a cloud license and force Viewer mode (in “Help” dropdown menu)
- FP Designer:
  - Adding footprints to FPX library produced a double (duplicate) Physical Description
  - When a footprint is moved from a Calculator to FP Designer, the Auto-Assembly and Silkscreen Polarity will be transferred
- Library Editor:
  - The Physical Description for footprints with Thermal Pads now includes the Thermal Pad in the Pin Qty
  - The footprint Viewer was expanded to enlarge the footprint image on the computer screen
- Calculator:
  - DFN 3-Pin – Pin 1 polarity dot was not following Option settings
- Options:
  - SMD Pad Stack Rules > Thermal Tabs > Thermal Tab Paste Mask (%) – was changed from 50% to 60%
  - Terminals > Surface Mount > Flat No-Lead (Side) > Heel – was changed from 0.00 to -0.04 for all density levels
  - SMD Pad Stack Rules > Thermal Tabs > Minimum Pattern Space – could not change the value

  - Drafting > Silkscreen – Added the ability to turn off Silkscreen by setting the line width to 0.00
- 3D STEP:
  - Chip Crystal would not generate a 3D STEP model

Version 2023.04

Fixes & Enhancements:

- FP Designer:
  - Updated all non-plated holes to "Mechanical" pins
  - Fixed an issue with the non-plated hole with an annular ring pad
  - Removed annular size options from the non-plated pad stack Keepout
  - Pin Placement Wizard getting non-metric calculated values wrong
  - Non-plated annular pad stack selection should result in a 'Mechanical' pin application
  - Keep outs not required for Annular non-plated pad stacks
- Calculators:
  - Right Angle Headers – Added a default Pin Pattern image needs to show something when it's an option
  - DIP Socket: Pin 1 Polarity Dot was in the wrong location
  - SODFL: Assembly label orientation was wrong
- Options:
  - Add new feature for silkscreen outlines to allow the user to have a full body or hatched silkscreen
    - All Right-Angle Headers – Post, Receptacle, Shrouded
    - All Grid Array packages – BGA, CGA, LGA, PGA
    - DIP Socket
    - SIP
    - Axial
    - Radial Rectangle – Dipped, Disk, Molded, LED
- Library Editor:
  - Comma added after "(_GA)," in grid array Physical Descriptions
  - Updating a Footprint to library updates multiple lines under certain conditions

Version 2023.03

Fixes & Enhancements:

- Calculators:
  - New Feature: Right Mouse Button no longer required to toggle between feature selection
  - Molded Body LED: not following Option settings for alpha pin names
  - Mounting Holes: when saving mounting hole data to the FPX library, retrieving the data grayed out the “E1” dimension
  - Flange Mount Vertical: updated the Pin 1 Polarity Dot location
  - Corner Concave Oscillator: the silkscreen outlines for the “D” and “E” dimensions were swapped
  - Dip Socket: round lead option not working
- 3D STEP:
  - Chip, Molded and Side Concave LED and default terminal color options are cross connected
- Library Editor:

- When opening a new FPX library, the library name does not immediately change
- Drafting Symbols/Shapes:
    - When entering data in the menu, the Line Width defaulted back to the original setting when any other cell was edited
    - When selecting the “New” button, the Centroid X dimension automatically changed to 0.00
- FP Designer:
    - Added new feature for adding a single pad stack
    - Updated the Radial Pin Placement feature
    - When adding a new footprint to the FPX library, the “Save to Library” button turned off
- CAD Translators:
    - Altium: Added new code to streamline the Altium translator
- Footprint Expert Pro:
    - Fixed the banner popup ads to not show when the program in minimized

Version 2023.02

Fixes & Enhancements:

- Resolved FIPS issue
- Options:
    - Console Options was not restoring the component outline color if it was changed and restored
- FP Designer:
    - Added a new feature – the silkscreen and assembly outlines now map to the outside of the package body outline
    - Assign Pins – Pin reorder was not working 100%
- Physical Description Text File:
    - Updated the Physical Descriptions for all footprints with Thermal Pads
    - This impacted physical descriptions for footprints created in FP Designer
- Calculators:
    - Chip and MELF Assembly Ref Des was using the wrong package dimension to calculate the height
    - Resolved issues with Footprint Names:
        - DFN 2, 3, 4 – duplicating pin quantity
        - DPAK missing a tab dimension
        - SOJ & SOL, and OSCJ & OSCL & Crystal was using lead span E instead of body dimension of E1

Version 2023.01

Fixes & Enhancements:

- Console Options:
    - Added IPC-7352 Naming Convention
- Calculators:
    - 2-Pin DFN – updated the assembly outline polarity marker from upper left to lower left
    - Physical Description – updated QFN, PQFN, SON, PSON, QFP & SOP to add a comma after the closed parentheses
- Options:

- Pad Stack Rules – Corner Rounding > Corner Radius Limit – changing the value to 0.00 now results in a rectangle pad
- Library Editor:
    - These FPX columns were removed – Symbol Name, Pin Quantity, Value, Tolerance, Voltage, STEP File Name
        - You can manually remove these columns from your master FPX file
    - New Feature – Middle mouse wheel zoom now zooms wherever the cursor is
- FPX File Converter:
    - Updated to the new V23 format both internally and the separate .exe version
- FP Designer:
    - Enhanced the Import .CSV file feature
- CAD Tool Translators:
    - Altium – fixed an issue with pad place round-offs for 0.65 mm Pin Pitch parts
    - Allegro – 3D STEP file was not placed in the correct directory folder
- Footprint Expert Viewer:
    - This free edition of the Footprint Expert was discontinued, but the Enterprise version runs as a Viewer when no license is found
- POD Builder:
    - This free edition of the Footprint Expert was discontinued