Version 2022.15

Fixes & Enhancements:

- Calculators:
    - User interface updated – Unrequired calculator dimensional cell background color changed from gray to light blue
    - All Surface Mount non-polarized diodes – updated the footprint naming convention from DIONP to DION
    - SOD – when pad trimming under component is turned on the Toe goal was also trimmed
    - SODFL – Non-polarized diodes had assembly and silkscreen polarity markings
    - DPAK – Flat Lug Tab redefined from ‘Thermal’ to ‘Mechanical’.
- FP Designer:
    - Pad stack manager not enabling ‘Save’ buttons after adding layers
    - Rotating rectangle footprints the pads rotated but not the drafting outlines
- 3D STEP:
    - When the CAD output directory was set to “Same As Translator Directory” the 3D STEP model would not auto-import
- Drafting Elements:
    - Selecting a Round shape Keepout threw a Handled Exception Error

Version 2022.14

Fixes & Enhancements:

-
- Calculators:
    - DFN component families – assembly polarity marker appeared on non-polarized parts
    - Reselection required in order to query even though pin or shape or label or symbol indicates it is selected
- Options:
    - Changing Density Levels did not affect the silkscreen line width
- FP Designer:
    - FP Designer parts with offset pins batch building from library without offset
    - Pad Stack Designer slotted holes were rotated 90 degrees from pad
    - Pad Stack Designer not creating Side Radiused surface mount pads

Version 2022.13

Fixes & Enhancements:

-
- Calculators:
    - New – added a 6-pin version of the Side Concave Package
    - Mounting Hole – removed “A” (for annular) from mounting hole name
    - DFN 4-pin:
        - Pad to Pad Clearance bug not following Options rule
        - Option to expand silkscreen outline
    - Axial Lead – Body Diameter uses height (A) instead of diameter (E) dimension
    - SOL – Width uses Body width (E1) instead of Lead Span (E) dimension
    - Corrected Mfr. Dimensions' roundoffs for place and pad size round offs
    - Corrected paste mask percent-to-excess calculation

  - Round-To function for TH pad stacks resolved for non-millimeter units
  - Courtyard relation to pads with zero excess
  - Symbol & Keepout rectangle shape orientations now rotate properly with the footprint
- FP Designer:
  - New – added pin orientation to FP Designer pin placement import .csv file
  - Pad Stack Designer – hole size round off and non-plated hole mask pad stack name
  - Move Origin by Coordinates not recognizing other than mm units
  - Ref Des for Silkscreen and Assembly label orientations synced
  - Zero values in the pin-place text boxes
- Options:
  - Options/Pad Place Units not saving
- CAD Tool Translators:
  - Altium – added feature that automatically opens the output folder if the user wants to
- Library Editor:
  - Exception thrown when a library is loaded when there are no data grid rows showing

Version 2022.12

Fixes & Enhancements:

-
- Library Editor:
  - Import efficiency increased 10X for importing an FPX file into another FPX file; now shows a progress bar
  - Utilities > Update BOM From Library can open multiple dialog boxes and was reduced it to one
  - Utilities > Find Duplicates – improved the duplicate entries to display all duplicates rather than one at a time
- FP Designer:
  - When rotating footprints with Drafting lines, was not working correctly
  - Pad Stack Manager Rounded Rectangular pad shape was unable to turn off
  - When moving a Vertical Header to FP Designer, none of the values came across
  - Added the ability to rotate square pad stacks 45 degrees
  - Improved solder mask defined pad stacks for chamfered pad stacks
- Options:
  - New individual orientation options for SOFL & SODFL
  - Pad Size and Pad Place Roundoff in footprint calculators not complying with options selection
  - Changing the Console Options to mil units caused the percentage features to be disabled
- Calculators:
  - New features:
    - Ability to select the ESC keyboard button to close Property dialog boxes

    - Ability to move through-hole Oscillators to FP designer
    - Updating an existing Row in the FPX library will no display the update message if there is no change
  - Inserting a Keep Out on All Layers was disabled. Calculators and FP Designer
    - Layer Display Controls – unselecting “Display All” does not turn off keep outs
    - DPAK component family – Silkscreen Pin 1 Line indicator was throwing an exception error
- CAD Tools:
  - KiCad – generating 'Right Angle Shrouded' header footprint didn’t obey the 'Autogenerate Pin 1 Indicator Line' rule
  - Cadence – courtyard excess rounded incorrectly for pad to courtyard for PLACE_BOUND feature

Version 2022.11

Updates & Bugs Fixed:

- Library Editor:
  - The Physical Description pin quantity was producing incorrect values when pins were Deleted or Hidden
  - Utilities > Regenerate Library > Physical Descriptions did not work correctly for Mounting Holes
  - Headers – Physical Description was changed from # Pins to # Pin
- FP Designer:
  - Improved auto silkscreen trimming and pad placement
  - Pad Stack Editor – when selecting SMD Pad Shape “Both”, the Chamfer size was limited to 50% of the pad width
- Calculators:
  - All Through-hole Radial Rectangular component families – missing Assembly Outline
  - Updated the Solder Mask corner radius for thermal pads that have a non-solder mask defined pad
  - Drafting Shape for Round – Unchecking the filled box did not work

Version 2022.10

Updates & Bugs Fixed:

- Library Editor:
  - Library sort didn’t work right after renaming column
- CAD Interfaces:
  - KiCad: crashes building PADS-to-CAD parts
- 3D STEP Output:
  - STEP file was not saving to selected output directory in some cases for most CAD tools
- Options:
  - Terminals > Surface Mount > Inward L Flat Ribbon – had duplicate terminal option
  - Components > Through-hole > Mounting Holes had an unnecessary dropdown for Origin
  - Turning off Drafting > Courtyard > Add Outline to Footprint did not work as intended
  - Components > Surface Mount > SODFL/SOFL pin 1, 2 and more than 2 pins added
- FP Designer:
  - Creating a Thermal Pad with a paste mask pattern, the Pad Stack Manger threw Exception errors when changing the Shape
- Calculators:
  - QFN – the silkscreen outline was not adhering to the Clearance Option
  - LCC – The pad length calculations need to be adjusted
  - QFN with Thermal – Rotating the footprint so Pin 1 is Upper Left but the thermal chamfer was in Lower Left
  - Removed the “SMD Pad Stack Rules” panel from all Through-hole calculators

Version 2022.09

Updates & Bugs Fixed:

- License:
    - Updated the HASP license to allow Footprint Expert to run as a Viewer if the HL or SL is are not found on startup or connection with key is lost
- FP Designer:
    - Batch build FP Designer parts with an offset origin that caused the pins to move
- Options:
    - Drafting > Component Outlines > turning off the "Output to CAD" turned the outline off in the viewer
    - FP Designer – Pad Stack Designer for Through-hole hole size calculations not synced with Terminal Options
    - Chip courtyard not sizing correctly for micro-mini parts
- Calculators:
    - Removed the "SMD Pad Stack Rules" panel from all SMD Grid Array calculators.

Version 2022.08

Updates & Bugs Fixed:

- Options:
    - Drafting Outlines – Assembly polarity indicator option changed to on as Default
    - Silkscreen polarity dot default options changed to most = 0.60, nom = 0.50, least = 0.40
    - BGA and CGA default percent values changed to reflect IPC-7351B pad size calculation
    - Options 'Ask to Save' message pops up redundantly on program close
    - Terminals > Grid Array > BGA – updated the pad reduction percentages to match IPC-7351B pad sizes
- Calculators:
    - DPAK – orientation changed to match dimension reference drawing
    - 2-Pin Packages SMD & PTH – updated the assembly polarity marking to lower left to be consistent with semiconductor assembly polarity marking
    - BGA – silkscreen outline not closing; dot not relocating correctly when rotated
    - BGA – when using the min/max dimensions only and mapping the Assembly Outline to the Nominal body caused errors
    - Problem with silkscreen for AE Caps and DFN2's when terminal widths get too large or silkscreen maps to minimum body
    - DFN3 and DPAK Tabs now treated as signal pins instead of thermal tabs
    - DFN silkscreen over pads in some parts with wide lead spacing
- Library Editor:
    - Added trap for illegal ASCII characters in library edits, helps prevent FPX files from getting corrupted
    - Library 'Find' function doesn't work after translating a part
    - Date formatted info not loading correctly from library sometimes
    - Library sort – pasted cells go to the end
    - Importing libraries with missing footprint dates not working
- FP Designer:
    - Rotation, Mirror and Polarity Dot functions not working correctly
    - Pin import function not functioning correctly
- CAD Tool Interfaces:
    - Xpedition menu not showing the Component Outline layer selection option
    - Allegro and OrCAD PCB 3D files not saving to the correct folder
- Miscellaneous:

    - Programs crashes when opening with FPX double-click

Version 2022.07

<u>Updates & Bugs Fixed:</u>

- Calculators:
    - BGA – fixed an issue with loading a non-collapsing BGA from an FPX library
    - DPAK – added 7 and 9-pin versions
    - Side Concave Package – fixed silkscreen and assembly polarity indicators
    - DFN 4 – fixed polarity indicators
    - All Component Families – when all pins are hidden program threw a Handled Exception error
- Miscellaneous:
    - Program wouldn’t run in a read only directory
- Library Editor:
    - Updated the column sort feature for the Date column
- FP Designer:
    - Moving origin didn’t work in Mil units
- Proteus:
    - Fixed an issue with the CAD tool interface not working

Version 2022.06

<u>Updates & Bugs Fixed:</u>

- Calculators:
    - Corner Concave Oscillator did not have automatic silkscreen & assembly pin 1 polarity markings
- FP Designer:
    - Resizing thermal pad paste mask checker-board pattern in the Pad Stack Manager caused corruption
- Options:
    - New Feature – relocated the automatic Pin 1 polarity dot to the left of Pin 1 for all footprints that have pads outside the package body
    - Turning off component outline and terminals still produced them in the exported footprint
- License:
    - Fixed issue for FIPS enabled computers
- Library Editor:
    - Selecting the header columns did not sort the data correctly
- 3D STEP:
    - Users could not rename 3D STEP files in the translator UI

Version 2022.05

<u>Updates & Bugs Fixed:</u>

- FPX File Converter:
    - Not updating FP Designer outline on/off selections
    - Courtyard excess for FP Designer footprints were incorrect
- Options:
    - Drafting – Text for terminal and body outline text mismatch

    - Updated Console Options – put the Component Outline to the top of the layer list
- Library Editor:
    - Library not adding copied parts to the end of the list
- Calculators:
    - SOT143 – pin reordering did not allow for fat pin on any pin except #1
    - 4-Sided Chip Array – mfr. recommended footprint pattern, the G1 was not applied correctly
    - 2-Sided Chip Array – courtyard was not resizing to include silkscreen outline
    - SOT23 – corrected the Gang Mask feature
    - SOP's & QFP's with thermal tabs – fixed pads under body not trimming correctly
    - Right Angle Shrouded Header – silkscreen not conforming to body outline
    - Through-hole Right Angle Post Header – fixed and issue with the Pin 1 polarity dot and line
- FP Designer:
    - Pad Stack Manager paste mask percent % not displaying value
    - Triangle symbol size now matches V2020
    - The "Add to Library" button was not enabling after editing drafting symbols
    - Courtyard contoured outline not forming to some rotated pad stacks
    - Export .CSV file defaulted to Tab delimiter, and it was changed to Comma delimiter
- CAD Interfaces:
    -
    - Xpedition - Create and Close and circle back and Create did not work
    - OrCAD PCB - Designer Basics user option issue with not saving
    - PADS Layout -
        - Multipart (Batch Build) Part Type file was not generated correctly
    - KiCad -
        - Not putting Batch Built files in library folder
        - Fixed a Reference Designator line width issue
        - Added Rounded Rectangle pad shape

Version 2022.04

Updates & Bugs Fixed:

- SMD Calculators:
    - SOD, SODFL, Side Concave Package 2-Pin – updated the assembly outline polarity marker to a chamfer
    - QFP, CQFP – Setting Pin 1 to a different pin did not relocate the Pin 1 dot correctly
- Options:
    - Component > Surface Mount > SOD – changing the pad shape to Rectangle did not work correctly
    - Drafting > Silkscreen – added a new feature for minimum Pin 1 Dot Size
- 3D STEP:
    - Headers – changing the Pin 1 color did not work correctly
- CAD Interfaces:
    - Allegro/OrCAD
        - Outputting a footprint created a subfolder. Added an option for "No Library Name"
    - KiCad:
        - Fixed Thermal Pad checker-board pattern paster mask reduction.
        - Over/Under sized solder & paste mask was not translating correctly
    - PADS Standard:
        - Added a feature for Right Reading reference designator
        - Fixed an issue where the translator didn't recognize any PADS VX versions
    - Fixed a license issue

Version 2022.03

Updates & Bugs Fixed:

- Added ability to measure between objects
- FP Designer:
  - Reordering pins in the Calculator disables pins when moving to FP Designer
  - Grid space and show settings are now saving to registry
  - Added a "Save and Close" button to the Pad Stack Manager
  - When manually placing pads on a grid, added the ESC key to exit the add pin process
- Options:
  - Changed UI text Pad Stack Rules to SMD Pad Stack Rules
- Library Editor:
  - Deleting columns in a FPX file threw a Handled Exception Error when adding new parts
- CAD Interfaces:
  - Altium:
    - The IPC Density Level Suffix was not added to the end of the Footprint Name
  - Allegro/OrCAD PCB:
    - The Save My Settings feature wouldn’t work for Place 3D STEP Models via Script
- 3D STEP:
  - Fixed an Exception thrown for molded body TH and certain other parts due to step color error
  - LED’s – Chip, Molded Body, Side Concave: changing the LED color changed the pin 2 color
- Calculators:
  - SOP, QFP, Side Concave, SON, SOJ:
    - The automatic Pin 1 Dot Size was not adhering to the rule setup for all the other component families
  - SOT-23:
    - Changing the pin names to Alpha characters were not saving to FPX
  - Chip:
    - LED did not have Alphanumeric pin names
  - Grid Array Packages – BGA, PGA, LGA, CGA
    - The silkscreen outline was hatched and changed to full outline
- Added:
  - Ability to open a user option file (.opt) based on a command line entry. How to use:
    - Right-click on the 'Footprint Expert' shortcut and select 'Properties'
    - In the 'Target' option, after the Footprint Expert path (ending with ".exe"), add a space and one of the following
    - If the file is located in the program directory, enter a quote (") followed by the name of the option file, quote (") or
    - Quote (") followed by the full path to the desired option file, quote (")
  - Similarly, a library file can be automatically loaded use the same approach:
    - In the 'Target' option, after the Footprint Expert path (ending with ".exe"), add a space and one of the following
    - If the file is located in the program directory / Libraries directory, enter a quote (") followed by the name of the library file, quote (") or
    - Quote (") followed by the full path to the desired library file, quote (")
  - Both can be used as long as separated by a 'space’

Version 2022.02

Updates & Bugs Fixed:

- Options:
    - Added a new Drafting > Silkscreen > All Density Levels
    - Allow Expanded Outline – this is for mini chip packages
- Calculator Component Families:
    - DPAK:
        - Manufacturer Recommended Pattern dimension for pad size Y not working
        - Added Hide/Delete pins feature back
        - Fixed Footprint Name for Hidden and Deleted pins
        - Updated the Physical Description
    - Side Concave 2, 4 pin – fixed an issue with the silkscreen outline
    - SMD Chips – when moving from Calculator to FP Designer sometimes the courtyard would become irregular shape
    - SMD Oscillators – Silkscreen Outline was not translating to the CAD tool
    - All Families – when moving from Calculator to FP Designer the Silkscreen Outline would change Line Width
    - SMD CAPAE & Crystal – fixed a mathematical bug when using D1 & D2 to calculate L
- Library Editor:
    - Updated the Physical Description to allow users to edit it and save it and output to CAD tool
- CAD Interfaces:
    - Altium
        - When creating Footprint Only (no 3D STEP) the import was not working correctly
        - Fixed an issue with Multipart Density Level not being produced in Footprint Name
        - 3D STEP model name is now independent from the Footprint name
- 3D STEP:
    - Removed MELF and SOD Non-polarized Diodes Polarity Marker

Version 2022.01

Updates & Bugs Fixed:

    - Console Options:
        - IMPORTANT!: If you installed the V2022.01 pre-release you should manually delete ConsoleOptions.opt in this folder **C:\Program Files (x86)\PCB Libraries\Footprint Expert 2022**
        - When you open Footprint Expert it will auto-generate a new ConsoleOptions.opt that you can edit and save
    - Redesigned the Decimal Place Accuracy Units interface
    - Add new decimal place accuracy for Pad Size and Pad Place
- Options:
    - Options > Drafting > Silkscreen Line Width:
        - The Silkscreen Line Widths were not applying the assigned Density Level line width
    - Options > Components > BGA
        - If the “Add Fiducials if Pin Pitch” is any value other than 0.00, Batch Build auto-generates Fiducials outside the range specified
- FPX File Converter:
    - 3-pin DFN component family was swapping the G & G1 dimensions
- 3D STEP:
    - Radial Electrolytic Capacitor component family:
        - Colorizing Pin 1 was reversed in the 3D STEP output
    - SMD Aluminum Electrolytic Capacitor:
        - Chamfered silkscreen outline was not matching the 3D STEP model chamfer

  - CFP and CQFP component families
    - Models were updated to remove the radius edges
  - All Component Families:
    - When changing a 3D model color to a Custom color, the color text name now indicates “Custom Color”
- Calculators:
  - DPAK component family:
    - Fixed an issue where you could not Hide or Delete pins after a pin rename
  - Chip component family:
    - Added a new silkscreen outline feature for small chips that could not produce a line without violating the Option rules
  - CQFP component family with large pin pitch:
    - Pin 1 polarity line was placing it on the right side of Pin 1
- Library Utilities:
  - The Utilities > Update BOM From Library is the Cloud License version did not work
- CAD Interfaces:
  - Xpedition:
    - Added .asc file name when outputting ASCII version
  - PADS Standard:
    - Changed Version 10 to Version VX
  - General:
    - Added a Create and Close button to reduce mouse clicks
  - Altium:
    - Outputting 3D STEP added a Density Level Suffix in the .pas file to the 3D model name
    - Updated the Bottom Side Assembly Outline in the translator to be optional
    - Added a new feature that auto-deletes the PCBComponent_1 when creating a New Library
    - Now saves then reopens a library when a new part is added to enable properties
  - 3D STEP:
    - Translator output was not allowing saving a .STEP file to a separate directory folder