

Version 2024.11

Fixes & Enhancements:

- Options:
 - Components > FP Designer - removed the drafting outlines. All Drafting Outlines are now edited in the Drafting section.
 - Drafting - Every Outline is now consistent and has a "Add Outline To Footprint" switch that also turns that layer off in the viewer and the CAD tool
 - Drafting - The Centroid Crosshair and Origin are now separate entities that can be turned on/off separately
 - Terminals - Chip, update the Most Density Level for 0.75 - 1.35 from 0.05 to 0.00
 - File - Updated UI text for "New (Internal Defaults)" changed to "Use Internal Defaults"
- Calculators:
 - Through-hole - Oscillator, removed drafting outline corner radius
 - 2-Pin DFN - Added a new LED component family
- CAD Tools:
 - The main Header for Altium was updated to Renesas
- FP Designer:
 - Updated the pad stack and pad names



Version 2024.10

Fixes & Enhancements:

- Help > User Guide:
 - Several updates were made to match the current Footprint Expert release
- FP Designer:
 - Updated the pad stack and pad names for IPC-7352 version
 - Pad Stack name updated for SMD edge connectors that require a Top & Bottom pad
- Options:
 - Doubled the "Under Body Outward L Lead" Courtyard Excess for Height > 10 mm
 - Updated the Terminals > SOP > Pitch from 0.65 to 0.625 to match the IPC-7351B table
- Calculators:
 - QFN with no Thermal Pad - added longer Pin 1 and removed the "Set Pin 1" feature
 - SOT23 - User notice added when Gullwing is automatically changed to Outward L.
- CAD Tool Interfaces:
 - EAGLE changed to Autodesk Fusion/Eagle
 - eCADSTAR - added Pad Naming option back for CR-5000 and IPC-7351

- Cadence - fixed a typo for V16.5 (was 15.5)

Version 2024.09

Fixes & Enhancements:

- User Guide:
 - A new User Guide is now available with 65 new videos and HTML searchable data.
- Drawing Elements:
 - Square Keepouts had multiple issues.
 - Keepout > Top Keepout color assignment was updated to Cyan.
- 3D STEP:
 - Changing units from MM to Mils in the Through-hole Lead Extension did not work properly.
- Options:
 - Updated 3D Colors for LGA, CGA, LCC, OSCCC Pin 1 issue.
 - When opening Tools > Options, the window size was restoring down rather than retaining the same stretched size as the previous Save.
 - Updated the Option window to always open on the same screen as Footprint Expert.
 - If you moved the Option window to another screen, reopening Options reverted to that screen.
 - Updated Calculated Footprint Units so that they can be different than the master units.
 - SMD Pad Stack Rules > Thermal Tab Minimum Pattern Space was not working properly.
 - Changed the default value for Pad Place from 2 places to 3 places.
- FP Designer:
 - Added the ability to place pads at 270 degrees.
 - Changed UI text Add to Fpx to Add to FPX.
- Library Editor:
 - Selecting Utilities > Find Duplicate Entries, the Part Number is the default. If Part Number column is deleted, the Utilities > Find Duplicate Entries threw a Handled Exception Error.

Version 2024.08**Fixes & Enhancements:**

- New Free Footprint Expert Calculator.
- FP Designer:
 - Specifications panel now collapses all other panels to make scrolling easier.
- User Interface:
 - Updated the Header text in the Calculators and Options to 10pt semibold.
 - FPE references to "Library" replaced with "FPX File".
- Options:
 - DPAK - added a new Origin option for Center of Body.
 - Tools > Options will always open in the same window as the main program.
 - Console Options for Design and Library were moved to the Internal Default Options.
 - This allows the User to save Units and Library data in their Master Option file.
 - Assembly label size % getting updated with the units selection.
 - Updated the footprint name suffix from 5 to 6 characters.
- 3D STEP:
 - Custom Colors not saving to FPX file for Axial and Diode.
 - TO-220 Horizontal Mount 3D image has 'b' and 'c' terminal dimensions swapped.
- CAD Tool Translators:
 - Removed Help > About
 - Updated links to Help > User Guide
- Calculators:
 - Pin 1 polarity Line for Grid Array's now removes silkscreen where pin A1 is located.
 - Pin 1 polarity Line not following the silkscreen to Pad Gap setting in Options.
 - DPAK - changed the Solder Mask swell to Thermal Pad.
 - Corner Concave Oscillator - added a Reorder Pins feature.
 - SOP - added 0.00 pin pitch for 2-pin packages.
 - Side Concave Package - would not move to FP Designer.

Version 2024.07**Fixes & Enhancements:**

- Library Editor:
 - Library search with null manufacturer.
- CAD Tool Interfaces:
 - Xpedition and PADS Pro – fixed the thermal pad paste mask checker board pattern.
- Options:
 - Terminals > DPAK – updated Most Density Courtyard from 0.50 to 0.40.
 - Updated the default silkscreen Line Widths and Clearances to: Least = 0.12, Nominal = 0.15 and Most = 0.20.
 - Added information message to alert to Calculator user option file "may not be in V24 compatible format".
 - To fix, users must resave their personal options form the V24 program.
- Tools:
 - Grid Array Calculator – not syncing to Options.
- FP Designer:
 - Cancelling Designer Import Pins gives erroneous message.
 - Moving calculator footprints to FP Designer loses the Assembly Polarity Marker.
 - Moving footprints to FP Designer will now make the 3D STEP model polarity the same as the Assembly.
 - Non-plated through-hole pad stacks were creating 2 Bottom Solder Masks and no Top Mask.
 - When moving 2-pin polarized footprints from the calculator to FP Designer the assembly polarity changed shape.
- Calculators:
 - Oscillator, Side Concave – added an 8-pin version.
 - PLCC – added a warning message that the D2 dimension cannot exceed the D1 dimension.
 - SOT23 pad trimming for 3 and 4 pin versions needs to use 2 x pitch for pad spacing.

Version 2024.06**Fixes & Enhancements:**

- CAD Tool Interfaces:
 - Changed Translator options are not being read back.
 - All CAD tools – unable to open previously saved option files.
 - eCADSTAR – removed "Pin Naming" feature.
- Footprint Viewer Toolbar:
 - Renamed the Tool Tip for the layer dropdown menu to "Bring To Front".
- Calculators:
- Flange Mount Vertical – selecting pins threw an Unhandled Exception error.
- QFN with No Thermal – when Pin 1 is longer, the "Set New Pin 1" feature is now disabled.
- Corner Concave Oscillator – Pin 1 dot when pads too close for normal silkscreen outline.
- Molded Body – The Terminal Density Level thumbnail image was missing.
- Molded Body – updated the error message when the D1 dimension is not 0.60 smaller than the D dimension.
- Options:
 - Terminals > SM Grid Array > BGA, Non-Collapsing Ball > Add Ball Sizes 0.75, 0.65, 0.40, 0.17
 - Terminals > SM Grid Array > BGA, Non-Collapsing Ball > 0.55 Nominal Ball > Adj. % was 10% (now 15%).
 - Drafting > Silkscreen > Add Value to Footprint – not working.
 - Terminals > Through-hole – when setting up a solder mask oversize, selecting the Least Density turned the oversize off.
 - Console Options – updated the Decimal Place Accuracy for mils to be 1 mil increments and metric 0.01 increments.
 - Terminals > Gullwing, SO and QFP – Least Density > Pitch ≥ 1.00 mm updated Heel value from 0.20 to 0.40
 - When opening a new Option file, the Calculator did not readily apply the settings.
 - Terminal Options for CGA and LGA default settings were updated.

Version 2024.05**Fixes & Enhancements:**

- CAD Tool Interfaces:
 - Altium – The solder mask expansion rules were updated to overwrite Altium preferences
 - This is a check box on/off toggle in the Altium translator
 - Altium – Pads rotated at 45 degrees were not translating correctly
- 3D STEP:
 - Updated the Material Condition labels to Minimum, Nominal and Maximum
 - Exception thrown when 3D is selected as the CAD translator and then selected again
 - Changes to STEP 3D color options for Header's pin 1 not getting saved
- Calculators:
 - DFN pin 1 dot spacing too large when outline is not drawn
 - SOFL component family:
 - Pin 1 Dot placement when pins are reordered
 - Added Transformer component family
 - Keepout – the X placement coordinate resets to Zero when 'New' is selected
- FP Designer:
 - Pin Placement Wizard can exceed design limits and crash
 - CSV pin placement crashes when importing a file that is open in another program
 - Creating Bottom SMD pad stacks was missing the Bottom Paste Mask layer
- Library Editor:
 - Update BOM from Library throws an exception after update, when closing the dialog box
 - Library Batch build now works for mixed PADS-to-CAD and FPE calculator parts
 - Removed column "Footprint Date" seconds display
- Options:
 - Drafting – relocated "Add Value to Footprint" from Assembly to Silkscreen
 - Console – New Console/File option for 'Import-From' folder

Version 2024.04

Fixes & Enhancements:

- FP Designer:
 - New feature, rounded rectangle paste mask apertures for thermal pads
 - Symbol square shape turns to rectangle with certain values
 - Micro sized silkscreen gets too large when pushed to FP Designer
- Calculators:
 - Exception message box pops up when calculation button is clicked with no data entered
 - QFN – added new manufacturers recommended footprint image to reflect new longer QFN pin 1
- Options:
 - Issue with changing the thermal relief values threw an error message

Version 2024.03

Fixes & Enhancements:

- New Library Content:
 - Added link to the PCBPartz, a third party affiliate, for additional library content; requires separate paid subscription
- Options:
 - SMD terminal courtyard transposed pad and body excess
 - The orientation image for SON /PSON was updated
- Library Editor:
 - Copy/paste multiple cells not working correctly
 - Copy/paste Footprint Date was throwing exception error
 - Utilities > Find Duplicate Entries – now defaults to Part Number
- FP Designer:
 - Creating through-hole pad stacks was not producing the "Solder Mask Top" layer
 - Rounded rectangle shape produces wrong pad stack name in Pad Stack Manager
 - Importing a .csv file did not work correctly if you accidentally assigned 2 rows the same value like 2 – "X Coord." Added a popup menu that alerts the user when a column description was already assigned.
- CAD Tool Translators:
 - The Siemens PADS Standard would not output a .d ASCII file.
- Calculator:
 - Molded Body – hung up when Nominal dimensions had 0.00 tolerance
- Molded Body – hung up when Nominal dimensions had 0.00 tolerance
 - Updated silkscreen trimming feature for footprints with large thermal pads
 - BGA Footprint Name and Physical Descriptions for mirrored parts

- Chip LED – changing alphanumeric pins to numeric did not save the change to the FPX library

Version 2024.02

Fixes & Enhancements:

- Library Editor:
 - Newly loading library auto-sizes down if it's less than half the FPE window height
- CAD Tools:
 - Allegro translator mistakes the courtyard target for the CY outline
- Calculators:
 - SOD causes crash when determining to use Gullwing or Outward L
 - Double density level suffix added to batch build translations
 - Added new image for 2 pin side concave dimensional reference image
- Options:
 - Added IPC-7352 Pad Stack Naming Convention - Users have the option of switching between IPC-7351B and IPC-7352
 - Changed the Round Rectangle Minimum Corner Radius from 0.20 to 0.10 as the default
 - Fixed Typo – Leadless Flat Heel Terminal option 5.00 mm changed to 0.00
 - Fixed Typo – Gullwing DPAK Courtyard Terminal options 0.25 changed to 0.20
 - Terminal Options > SM Terminals conversion from metric to mil units weren't correct
 - Some SM Terminal Tools/Options are not editable
 - Editing SM terminal options resets to zero after leaving the cell
 - Updated the Component Options for Pin 1 Orientation, both Pin 1 image and text for QFP, QFN, SON & Chip Array
 - Added new Component families for Pin 1 Orientation:
 - SODFL, SOFL were separated into two different component families
 - Chip Array was broken into two different component families 2-Sided and 4-Sided
 - Updated the Option .opt files – Mils, IPC-7351B, IPC-7352, V23
 - Updated "Cylindrical End Cap" MELF Terminals from one set of solder joint goals to 5 different package diameters

Version 2024.01

Fixes & Enhancements:

- Options:
 - Recommended for existing users – Create a new Option file from scratch. Do not use your existing .opt file.
 - Updated the Internal Defaults to IPC-7352 Guideline
 - Removed Fab & Assy tolerances and changed all negative solder joint goals to 0.00
 - Note 1: this update might create SMD pad sizes smaller by an average of – 0.01 to 0.03 mm
 - Note 2: there will be a V23.opt file for users who want the original (IPC-7351) version
 - But you can still use your original .opt file if you don't want to update to IPC-7352
 - Note 3: if you want to use the new IPC-7352 options to upgrade to the new mathematical model for pad stack calculations, you should recreate your Master Option file from scratch by selecting “File > New (Internal Defaults)” then “File > Save As > New Name.opt”
 - Note 4: V24 installs into a separate folder to allow you to run V23 and V24 side by side to copy your existing Options to a new Option file
 - Redesigned all the Terminal Options GUI
 - Major rewrite for all Grid Array component families – BGA, CGA and LGA
 - Added the ability to enter 1 for pad size and placement
 - 3 = 0.001
 - 2 = 0.01
 - 1 = 0.10
- CAD Tools:
 - Allegro – translator dug in pad stack definition for mask layers
- FP Designer:
 - Moving a calculator footprint to FP Designer and selecting “New” pad stack created an error message
- Calculators:
 - Batch Build was only creating one footprint
 - BGA pad trimming caused a discrepancy between the actual pad size and the displayed pad size
 - Through-hole pad stack calculations were updated