

Version 2021.21

Updates & Bugs Fixed:

- Altium CAD Tool Interface:
 - Added the Bottom Side Silkscreen layer
 - Added Layer Pairs for all mechanical layers
- FP Designer:
 - The import .csv file feature wasn't working correctly
 - The Pad Stack Designer was not applying Solder Mask Option to non-plated through-hole pad stacks
- Calculators:
 - 4-Sided Chip Array – fixed an issue with the Terminal Outlines
 - All DFN component families – fixed a Silkscreen Polarity Dot issue



Version 2021.20

Updates & Bugs Fixed:

- CAD Tool Interfaces:
 - Proteus – Removed the PADS ASCII option
- FP Designer:
 - 3D model was not creating a polarity marker
- Options:
 - Changed the Pad Stack Rule option for Maximum Corner Radius to 0.20 mm
- Calculators:
 - Chip – fixed an issue with contour courtyard not working properly
 - DFN's – fixed a courtyard calculation issue
 - Convex Chip Array – fixed a pad stack calculation issue
 - Add new component families:
 - Chip – Crystal & Choke
 - Chip Array 2-Side Flat/Concave – Oscillator & Choke
 - Chip Array 2-Side Convex – Oscillator & Choke
 - Chip Array 4-Side Flat/Concave – Choke
 - Dual Flat No-Lead (DFN 2 Pin) – Antenna, Crystal, Choke, Thermistor, Varistor
 - Dual Flat No-Lead (DFN 3 Pin) – Choke, Ferrite Bead, Thermistor, Varistor
 - Dual Flat No-Lead (DFN 4 Pin) – Choke, Ferrite Bead, Thermistor, Varistor
 - Molded Body – Choke, Thermistor, Varistor
 - Side Concave Package (2, 4 Pins) – Choke, Ferrite Bead, Filter, Thermistor, Varistor

Version 2021.19

Updates & Bugs Fixed:

- Options:
 - The option for Drafting > Courtyard > Expand Courtyard to Include Silkscreen – was not working for all component families
- CAD Tool Interfaces:
 - eCADSTAR – fixed the 3D STEP file was not included in the ZIP file
 - Altium – Removing the eMechanical layer assignments for Altium will turn off that feature
 - KiCad – added Rounded Rectangle pad shape
- Calculators:
 - SON and Chip Array 4-Sided – the “G” calculation was slightly incorrect

Version 2021.18

Updates & Bugs Fixed:

- CAD Tool Translators:
 - Fixed an issue with multiple CAD tools. When the Build Library path was too long the program created an alternate folder
 - Fixed an issue with KiCad not getting Assembly Outlines on Non-Plated holes
 - Fixed an issue with Saving My Settings causing an Unhandled Exception Error
- Options:
 - Added a new Rounded Rectangle feature for through-hole square pads in V2021.17 that caused regression for all SMD and PTH rounded rectangle pads
 - Changing the 'Density level (suffix)' Option, the “Courtyard to Body Excess” and “Courtyard to Pad Excess” didn't update
- FP Designer:
 - Added Cole switches to the ManufactureList.txt
 - Fixed a Silkscreen on Pad bug in V2021.17 that caused regression for all FP Designer footprint silkscreen outlines
- Calculators:
 - The master options value for Paste Mask is for Annular Ring and it was set to oversize instead
 - QFP w/Thermal Add Fiducials changed the thermal pad pin number
 - Axial Lead family – the silkscreen did not trim properly when the pads were a close pitch
 - Chip component family – the body width dimension was missing the leading zero when using the IPC-7351B naming convention
 - BGA Courtyard was 90 degrees out of rotation when opening a rectangle BGA

Version 2021.17

Updates & Bugs Fixed:

- Drafting Symbols:
 - Removed the scrollbar when selecting a Shape
- Library Editor:
 - Updated Utilities > Update Manufacturer List and Physical Description List
- FP Designer:
 - Fixed a Silkscreen outline issue where the silkscreen did not apply the Silkscreen to Pad Clearance
- Options:
 - Changing the Density Level disabled the Courtyard settings
 - When opening an FPX file with a double click, the program loaded the Default Options. Fixed
 - Added Paste Mask Reduction by +/- Value or % Percentage
 - Through-hole – added the ability to create rounded square pad shape

Version 2021.16

Updates & Bugs Fixed:

- Options:
 - Added new feature to TO-220 Component for mounting hole to be plated or non-plated.
- Calculators:
 - DFN component family - Fixed an issue with the Local Options not being able to change the pad shape to Rounded Rectangle.
 - TO-220 Flange Mount family –
 - Fixed an issue in Options that prevented users from changing Pin 1 from Square to Round shape.
 - Updated the mounting hole inner layer plane connection from a thermal relief to a direct connection.
 - 2,4-Pin Side Concave family –
 - Fixed an issue with the V2021 FPX File Translator and 3D STEP model.
 - The LED component family Pin names were permanently set to numeric only and updated for Alphanumeric or Numeric.
 - Fixed an issue with the zero component orientation of the 2-Pin version to be Pin 1 Left.
 - SMD Crystal – Fixed an issue where the program was throwing a Handled Exception error message.
- CAD Tool Interfaces:
 - Intercept Pantheon – made improvements to the interface code.
 - Zuken eCADSTAR – updated the 3D STEP import feature.

- Added Tech Tips – when the layer name is too long to fit in the cell, mouse over will display the entire name.
- Fixed an issue with Save My Settings for multiple CAD tools.
- Added the 3D STEP folder option back to allow to save 3D STEP files in a separate folder.
- FP Designer:
 - Added a new feature for creating Mounting Holes with a direct connection to the GND plane. No Thermal Relief.
- Batch Build:
 - Fixed an issue when changing the Density Level in the Batch Build Options always defaulted to Nominal.

Version 2021.15

Updates & Bugs Fixed:

- CAD Tool Translator UI:
 - Updated all 25 CAD tool translator user interfaces and 3D STEP.
- Improved the interface to Intercept Pantheon
- Upgraded the FP Designer feature to streamline the creation of custom footprints.
 - Added solder and paste mask in the Pad Stack Designer to match the Option rules.
- Calculators:
 - 2-Pin DFN – Fixed a placement courtyard issue.
 - SOFL – Fixed a solder mask issue for fine pitch packages.
 - Corner Concave Oscillator – when the D1 and E1 dimensions had no tolerance, the 3D model terminal leads were incorrect.

Version 2021.14

Updates & Bugs Fixed:

- Tools > Options:
 - The Drafting Courtyard Outline – unchecking the Contoured Outline option threw Handled Exception Errors.
- FPX File Converter:
 - Fixed an issue with FPX footprints that had missing Package Type and Data and would not translate.
 - Fixed an issue when the old FPX file for FP Designer parts has the origin anywhere but the center.
 - Fixed an issue when FP Designer parts has a Pin 1 chamfer on the assembly drawing layer was not translating properly.
- FP Designer:
 - When creating a FP Designer footprint, the default shape is now set to Rectangle and when the user enters the package dimensions and

- selects the OK button, the Pad Stack Designer automatically opens, and the Body panel collapses.
- Editing the Drafting values did not work.
- Fixed an issue where the hole size was rounding up or down twice.
- The Courtyard Outline was not following the Courtyard to Pad Excess value set in Options. Slightly off.
- Fixed an issue when Batch Building footprint with the origin on Pin 1.
- When finalizing a footprint in the Specification panel, the component manufacturer name was not appearing in the cell after searching and selecting it.
- Library Editor:
 - Fixed an issue with processing multiple (batch build) unique parts created in PADS and imported into FPX.
- Footprint Expert Pro:
 - The print feature was turned back on.
- Calculators:
 - Fixed an issue when the QFP with Thermal Pad added local fiducials, the thermal pin name would change.
 - When the contour courtyard was turned off, the rectangle courtyard violated the excess rules.
 - Fixed an issue for through-hole parts where the origin was on Pin 1, the CAD output was incorrect.
 - SIP component family – the zero-component orientation had different words that did not match in the Global & Local Options.
 - When the package orientation is set to Pin 1 Up and the footprint was moved to FP Designer, the body dimensions swapped.

Version 2021.13

Updates & Bugs Fixed:

- Tools:
 - Using the flip 3D STEP model feature disabled all the other feature in the Tools menu.
- Calculators:
 - Corner Concave Oscillator – The silkscreen Outline was not mapping to the Maximum package dimension.
 - When selecting the Pad Stack icon and then selecting the Close button, threw an unhandled exception error.
 - Flange Mount, Vertical component family – Changed Center of Extents to Center of Body.
- Layer Display:
 - When turning off layer and then select All, the layers did not reappear.
- Library Utilities:
 - The “Update BOM from Library” feature was disabled.
- FPX File Translator:
 - V2018 format Collapsing Ball BGA sometimes translates to Non-Collapsing Ball.

- FP Designer:
 - Creating new footprints was improved. Rectangle is the new default and next panels auto-open.
 - When opening an FP Designer part, the user was forced to select the Finish button. Finish was changed to Ready.
 - Creating round footprints generated a rectangular 3D model.
- Batch Build Update:
 - Creating a Single Row produced a Footprint Name in the CAD tool translator New_Parts and not the actual Footprint Name.

Version 2021.12

Updates & Bugs Fixed:

- Library Editor:
 - Adding new FPX files to an existing library added the part to the top of the list (rather than the bottom).
 - Updating a FP Designer part row in an existing library created a duplicate row and did not update the existing row.
- Calculators:
 - The Density Level Suffix was not generated in the footprint name.
- FP Designer:
 - The silkscreen outline was not producing the correct line width Density Level set in Console Options.
 - Pin reassignment did not work correctly.
 - After adding new pins manually to an existing FP Designer part, the Delete button was disabled.
 - Added a new button to reassign pad stack pin numbers.
 - When adding pad stacks manually, added the ESC button to stop adding pins.

Version 2021.11

Updates & Bugs Fixed:

- Calculator Component Families:
 - BGA – Fix a typo in the pad stack manager the paste and solder mask were displaying text for a Rectangle shape.
 - BGA – the notification text for the Optional “e” dimension was missing.
 - Quad packages – when changing the location of Pin 1, the autogenerated pin 1 silkscreen dot would relocate incorrectly.
 - A case code or part number with a dash “ – “ was creating exception errors. It was tagged as an illegal character.
 - When selecting the Pad Stack Manager, the Pad Stack Designer opened instead.
- CAD Tools:

- Altium – when using the Autogenerated Pin 1 Dot, the dot size in Altium was 50% less than the Option setting.
- Options:
 - Drafting > Silkscreen > Clearance to Pads was creating an incorrect Gap from the end of the line to the pad.
- General:
 - Updated the pad stack name to match the pad size after rounding.
- Batch Build:
 - Sometimes batch building FP Designer parts threw a warning message: Get Silkscreen Options

Version 2021.10

Updates & Bugs Fixed:

- Calculator Component Families:
 - BGA – when changing the paste mask reduction, the calculator produced the incorrect result.
 - Quad and Grid Array packages – when entering the D & E pin qty the total was an inaccurate value until the Calculate button was selected.
 - SON – using the mfr. recommended pattern was causing pads to react differently under certain dimensions.
 - SOD – moving the Calculator SOD to FP Designer caused several missing cell data information.
 - All component families with Thermal Tabs – Tabs on Assembly produced 2 outlines.
 - All SMD component families – if the mfr. recommended pattern is being used it should remove the Density Level from the footprint name.
 - All PTH component families – there is only one Density Level for through-hole footprints and the footprint name should not indicate “Nominal Density”

Version 2021.09

Updates & Bugs Fixed:

- Drafting Options:
 - Added a new Right-Angle Triangle. This is mostly used for Pin 1 corner Polarity Marking.
- Console Options:
 - IPC-7351B Naming Convention – in the SOIC component family, the program was adding an extra X after the pin pitch.
 - Master FPX File Name – when the master FPX file name was longer than 20 characters, the program was throwing an Unhandled Exception Error.

- Pad Stack Rules – Gang Solder Mask Shape had a bug issue that affected solder mask shape on SMD pads.
- Calculator Component Families:
 - All Grid Array's (BGA, CGA & LGA) – setting the local Calculator solder mask in “Pad Stack Rules” panel does not work. A warning message was inserted prompting the user to set the solder mask swell in the “Terminal Density Level” panel.
 - All Through-hole Headers – When moving Headers to FP Designer and save to FPX, reopening the footprint dropped the Body Length & Width.
 - MELF:
 - When creating a Diode MELF, the local “Pad Stack Rules > Miscellaneous > Cathode/Anode Pin Names” was disabled.
 - When moving the MELF to FP Designer and save to FPX threw an error message.
 - Chip, MELF, Molded, SOD & SODFL Diodes - fixed Numeric/Alpha pin names and pin swap options on the right mouse button.
 - SMD Crystal – Physical Description had Body Length listed twice and no Body Width.
 - PQFN & PQFN w/Thermal – footprint name had wrong body length and width.
 - SOP w/Thermal – the Physical Description did not indicate w/Thermal Tab.
 - DIP and DIP Socket:
 - The Physical Description had the Length and Width swapped.
 - The Silkscreen Outline was not translating to any CAD tool.
 - When moving the DIP to FP Designer and save to FPX, reopening the footprint dropped the Body Width.
 - Flange Mount (TO) Horizontal – the Physical Description was missing the Body Length and the Pin Qty was incorrect.
 - Flange Mount (TO) Vertical – the Footprint Name and Physical Description had the Body Length and Width swapped.
 - All Through-hole families:
 - Pads and Holes on Assembly did not work properly.
 - Courtyard to Pad Excess was 0.45 and updated to 0.25.
- CAD Tools:
 - KiCad – through-hole pad stacks had holes on assembly going to the fabrication layer.
- Options:
 - Drafting > Silkscreen – changing the Reference Designator line width % was resetting back to 10% default when closing and reopening the program.

Version 2021.08

Updates & Bugs Fixed:

- Added the ability to edit the pad stack names in all Calculator created footprints.
- Added a new feature for converting Excel spreadsheets into FPX files.
- CAD Tools:
 - Allegro/OrCAD PCB – fixed an issue with the “Pad on Assembly” outline was going to the Silkscreen Layer.
- Calculator Component Families:
 - Grid Array’s (BGA, LGA, CGA, PGA) – fixed an issue with rectangular shape packages.
 - TO Flange Mount Horizontal – the “d” dimension did not allow a value of 0.00.
 - Header, Right Angle – fixed an issue with the Pin Pattern option.
 - BGA – fixed an issue with Non-collapsing ball saved to FPX and reopened as Collapsing.
 - Several 2-Pin Component Families – when the component tolerance is 0.00 and the Map Outline to Body > Nominal the outline would flatline.
 - All – Pad on Assembly was disabled.
 - LCC – under certain conditions with dimensions and tolerances, pin 1 would inadvertently relocate from the Left Center to the Lower Left.
 - 4-Pin DFN – when using the mfr. recommended pattern, pins 3 & 4 did not follow the mfr. dimensional values.
 - Header, Right Angle Post and Shrouded – the footprint name had 0 for package height and the Physical Description had 0.00 for Height.
- FP Designer:
 - Pad Stack Manager – updated the feature for renaming pad stacks to include spaces in the name.
 - Footprint Name – selecting Case Code or Part Number did not register as a change.
- Options:
 - There were updates and the only way to fix them is to recreate your Option file.

Version 2021.07

Updates & Bugs Fixed:

- Updated all the CAD tool interface UI menus.
- Updated the V2021 FPX File Converter.
- Console Options:
 - Mil Units – working strictly in Mil units slightly altered the pin pitch in Gull Wing, J-Lead and Bottom Terminal footprints. Translating metric to mil units worked OK.
- Print Feature:
 - Updated the Footprint Dimension letters in the table to match the letters in the image.
- FP Designer:
 - Through-hole pad stacks with a solder mask swell was not working correctly.
 - The Placement Courtyard was not going around Non-plated Holes correctly.
- Calculator Component Families:
 - Options – setting the “Pads & Tabs on Assembly” would not turn off when selecting “Clear and Reset Options”.
 - SOT23 – fixed a typo. The 0.95 pitch was displaying the wrong pitch range in the Terminal Density Level UI.
 - SON with Thermal Tab – translating these parts to V2021 had the X1/Y1 dimensions swapped.
 - SIP – added the ability to enter a negative value.
 - PLCC – corrected the Pin 1 location for 20-pin versions. Pin 1 was offset and relocated to the Center.
 - SOJ – fixed an issue for the minimum pad to pad clearance.
 - TO-220 Flange Mount – Updated the component family from TO-220 to TO because it can be used for all Flange Mounted packages.
 - TO – Updated the footprint naming convention to add both the pin pitch and pin span.
 - Header, Right Angle Post – fixed an issue when the E dimension was missing and the Calculate button threw an Exception Error.
 - Radial, LED – updated the 3D Model pin colours. During testing the pins were set to green and yellow and not converted back.
 - DPAK – the feature for “Pads on Assembly” was disabled.
- FPX File Converter:
 - Fixed an Unhandled Exception error when the Cancel button was selected.

Version 2021.06

Updates & Bugs Fixed:

- Updated the FPX file Converter – 2021 FPX File Converter.exe
- Replaced all UI text “Legend” to “Silkscreen”

- Console Options – added Pad Size Roundoff for Terminal > Through-hole pad stacks
- 2021 FPX Converter:
 - Molded Body component family – fixed an issue for when pin 1 is different size than pin 2
 - Fixed an issue with converting FP Designer parts with slotted holes
 - Fixed an issue with converting Round Body footprints
- Tools > Options > Drafting > Courtyard > Origin:
 - The checkbox on/off settings for Origin were not mapping to the Calculators
- FP Designer:
 - Auto-trim Silkscreen – fixed an issue for microminiature components
 - Converting 2018 format FPX files with round Body Shapes – the V2021 converter was changing the Body diameter to 0.00
- Library Editor:
 - Added AEconversion as a component manufacturer
- Calculators:
 - Pad Stack Rules > Thermal Tabs > Defined Solder Mask – Was not evoking the “Recalculate” icon to accept the update
 - TO-92 – fixed a silkscreen outline issue
- Option File - issue with loading (defaulted to Internal Settings) - Cloud License only:
 - Fixed an issue where the main Launch Footprint Expert as open the last saved .opt file (rather than “InternalDefault.opt”)
 - To fix this bug, you must Uninstall V2021 Footprint Expert and reinstall V2021.06 (back up your personal files)
 - No need to delete any folders as they contain your personal files
 - Note: the uninstall will not delete your personal Options or FPX file, but you will have to reload your Options and FPX back into the new V2021.06 installation

Version 2021.05

Updates & Bugs Fixed:

- Library Editor:
 - Updated the 3D Model Name column to add the file extension .STEP
- User Options > Drafting:
 - Changed the term “Legend” to “Silkscreen”.
- Console Options:
 - Updated the Layer order assignments.
- FP Designer:
 - Fixed an issue with translating older FPX files that had “None” for the Body Outline.
 - Cleaned up some UI text in the Pad Stack Designer.
- Library Editor:
 - Batch Build did not work on multiple parts.

- Updated some TH component family Physical Descriptions to make them consistent with SM. DIP, SIP, OSC, DIPS were updated.
- Calculators:
 - PSON – The mfr. Recommended pattern had a G1 dimension that was not intended. Removed.
 - PSON – The Silkscreen Legend was not translating to CAD tools.
 - Vertical Header – Assembly polarity marker did not move with relocating Pin 1.
 - Vertical Header – Manually inserting package dimensions turned off the Pin Pattern feature.
 - TO-220 Horizontal – Batch Build does not import 3D STEP.
 - Axial Lead – The Silkscreen Legend was not translating to CAD tools.
 - SMD Crystal – The Silkscreen Legend on the lower part was not displaying.
 - Chip Array Side Concave – The Silkscreen Legend was not translating to CAD tools.

Version 2021.04

Updates & Bugs Fixed:

- Tools > Options:
 - Terminals > Rectangular End Cap > Courtyard Excess for Pad was not applying the excess to the sides.
 - Console > Mil Units – major upgrade to everything relating to building footprints in Mil Units.
 - De-selecting “Add origin crosshair to footprint” removes the courtyard instead of the crosshair.
 - De-selecting “Add origin target to footprint” removes the crosshair instead of the target.
 - Pad Stack Rules Options – The SMD Corner Rounding Radius Size % was not translating over to the calculator.
- FP Designer:
 - Fixed an issue for creating Slotted Holes. The pad size did not reflect the Options.
 - Fixed an issue for Drafting Line lengths. The length value was not reflective in the drawn line.
 - Fixed an issue when selecting Corner Radius in the pad stack Manager, the default location was blank.
 - After selecting Designer and then selecting any other icon button threw an unhandled exception error.
 - When creating a new pad stack in the Pad Stack Designer, the Pad Stack Manager had value fields that were not supposed to be there.
 - Legend issue fixed – when the footprint is too small for an accurate Legend Outline, FP Designer auto-generates a fake legend outline around the pads.
- Calculators:
 - Tabbing through the dimension cells was not working properly.

- Naming Convention used the Nominal Package Height rather than the maximum height.
- PSON component family was completely disabled.
- Library Editor:
 - Rename a footprint in the library editor Footprint Name cell, the CAD tool output inserted the original name in the CAD tool translator.
 - Batch Build does not produce the Density Level Suffix in the Footprint Name L, N or M.

Version 2021.03

NOTE: The V2021.03 was available online for a few days, but a new V2021.03 build was released at the time of this post. If you already downloaded V2021.03 prior to this post, it's recommended you re-download the newer build.

NEW or Enhanced Features:

- Updated UI. Relocated the Open & Close FPX Library buttons from the right side to the left side of the screen.
- Updated all program splash screens to include the IEC Logo and IEC 61188-7 for Zero Component Orientations.
- Updated all Assembly and Silkscreen Legend Ref Des to rotation 0 or 90 degrees.
- Drill size roundoff feature added in Options > Through-hole > Terminals.

Updates & Bugs Fixed:

- Tools > Axial Lead Calculator – fixed the pin pitch bug.
- SMD Side Concave Package component family – the footprint name was missing the package width for 2-pin parts.
- SMD Side Concave Package component family – the physical description was updated.
- FP Designer Pad Stack Designer – removed the UI text “Hole Over Pad” for SMD pad stacks.
- FP Designer Pad Stack Designer – Fixed the PTH terminal lead size. The X & Y values were swapped.
- Move Calculator footprint to FP Designer – fixed an issue to allow pad stack editing.
- PTH SIP component family – fixed an issue where the silkscreen legend was not translating to CAD tools.
- Print Feature broken. Fixed an issue with the Enterprise version to allow printed documentation.
- All component families – the maximum material condition 3D STEP model was coming out distorted.
- All PTH component families – mfr. recommended pattern was disabled.
- Fixed an issue with Drafting Shapes and Keep-outs vanishing after the Calculate button is selected.

- Fixed an issue with Drafting Shapes. When you leave the Properties dialog box and reopen it, the move arrows were disabled.
- All component families – when the Mfr. Recommended Pattern was selected, changing a package dimension was not removing the Mfr. Recommended Pattern.
- Assembly Pin 1 Polarity – removed the arc and replaced with chamfered lines.

Version 2021.02

NEW or Enhanced Features:

- New Options > Drafting > Courtyard feature to Exclude Body and Pad Excess and ignore pads.
- Drafting Shapes – Changed the default orientation for the Diode symbol to Cathode on the Left.
- Updated the IPC-7351C Option file. Removed all negative solder joint goals and manufacturing tolerances.
- Aluminum Capacitor component family – Footprint Name updated. The Lead Span was removed from old 2018 CAPAE name.
- Capacitor Polarized, Axial Lead component family – Footprint Name updated. The V2018 axial body physical dimension wrong.
- Fuse, Radial Lead component family – Footprint Name updated. The old name the letter O added to denote offset leads on radial parts.
- LED, Radial Disk component family – Footprint Name updated. The old name was using the terminal thickness instead of the width.

Updates & Bugs Fixed:

- Options > Pad Stack Rules > Diode Pin Names – fixed an issue for Numeric pin names.
- Fixed an FP Designer Courtyard issue when translating older FPX files to V2021 format.
- Fixed an FP Designer issue for Round footprints the diameter was coming in at 0.00 when translating old FPX files to V2021 format.
- Drafting Symbols – when changing the line width value, then selecting “Apply or Done” the original 0.12 value came back.
- Courtyard Excess – when changing the courtyard excess to 0.00 the courtyard hugs the pads and body, except Chip Components. Axial Lead component family – Correction made to Footprint Name.
- BGA component family – Correction made to Footprint Name. V2021 was not pulling the body style from the FPX file plus 2021 BGA pins in D & E axis were swapped.
- Molded Body component family – Correction made to Footprint Name. V2021 footprint name was missing optional L1 dimension.
- Header, Right Angle component family – Correction made to Footprint Name. V2021 right-angle-header height was missing.

- Header, Vertical component family – Correction made to Footprint Name. R and S added to name prefix to denote receptacle or shrouded; height missing in vertical configuration; H replaced X before height.
- 2-pin Side Concave component family – Correction made to Footprint Name. V2021 doesn't need pin count for 2 pin parts; pitch removed as a factor in the side-concave name.
- QFN Component Family – Correction made to Footprint Name. Thermal Tab missing from pin quantity in parts with removed or hidden pins.
- TO, Flange Mount component family – Correction made to Footprint Name. Replaced H with X before height.
- SOIC component family – Correction made to Footprint Name, Physical Description and 3D STEP model.

Version 2021.01

NEW or Enhanced Features:

- **Complete code overhaul!!**
 - Improved efficiency & user experience
 - New User Interface, better suited for high resolution displays
- New Options (was Preferences)
 - Up to 6 place unit resolution
 - Totally redesigned interface to make it easier to define options
- The Free Pro version has all Enterprise features turned on
- CAD outputs migrated to **Enterprise** (Perpetual & Subscription) and **POD Builder** editions only:
 - **Allegro, PADS Professional, and Xpedition** - Upgrade to Enterprise, software license for FREE, just pay maintenance/subscription (80% discount)
 - Following will be affected in April - **Altium, OrCAD PCB, Siemens EDA** (Mentor), **Zuken**
 - Switch to the [Subscription Plan](#) to get **full Enterprise features enabled**
- Updated FPX file format
 - Added schematic symbol data
 - Added additional data columns
 - New V2018 to V2021 FPX file Converter program
 - Automatically adds all new columns
 - Checks for duplicate part numbers