

Version 2017.13

NEW / ENHANCED:

- Updated all Terminal Lead-Form images in the Calculator and Preferences



FIXED:

- Calculator:
 - Fixed an issue with the Pin Rename feature: once you rename pins you were unable to rename them again
 - Updated footprint names for:
 - Oscillator Side Concave
 - Molded Body
 - Oscillator J & L lead
 - SOFL Diode, IC & Transistor
 - DIOSOFL
 - ICSOFL
 - TRXSOFL
- Preferences:
 - When changing Pin 1 pad shape in one location, did not change all locations
- 3D STEP:
 - Fixed an issue with Molded Body 3D model scaling
- License selection / default:
 - Under certain conditions, a previously expired evaluation license interfered with the full Pro license
- Documentation:
 - Updated - IPC-7351C (Proposed) Solder Joint Goal Tables v1.5.xlsx
 - Added the new Terminal Lead-Form images
 - Updated – Surface Mount Component Families.pdf
 - Added Terminal Lead-Form images to pages 36 & 37
 - Fixed 6 typos – IPC-7351C Land Pattern Naming Convention.pdf
 - Molded Body - PKGM + Lead Span X Body Width X Height + L Lead Length X Width
 - Oscillator J-Lead - OSCJ + Pin Qty. + P Pitch _ Body Length X Lead Span X Height + L Lead Width
 - Oscillator L-Lead - OSCL + Pin Qty. + P Pitch _ Body Length X Lead Span X Height + L Lead Length X Width
 - Diode SOFL is now DIOSOFL
 - Transistor SOFL is now TRXSOFL
 - IC SOFL is now ICSOFL

Version 2017.12

NEW / ENHANCED:

- All versions of Library Expert Lite, Viewer, IPC, Builder and Pro:
 - Added Part Number Search for component availability using the **Octopart** search engine
- **Library Expert Lite** now fully supports **3D STEP** export:
 - User color control
 - Vertical Axis control
 - Nominal or Maximum Material condition control
 - Through-hole lead depth control

FIXED:

- Preferences:
 - Added the SOFL to Component Families
- Library Editor:
 - SOFL component family – retrieving SOFL data from FPX into the Calculator did not work
- 3D STEP:
 - Fixed an issue with SOFL not producing a 3D STEP file
 - Added new color assignments for the Chip Filter component family
- Xpedition:
 - Fixed a rotation issue with complex paste mask
- PADS Layout:
 - Fixed a problem with the importer that was inverting the corners of chamfered pads
 - Provided the ability to import donut shapes properly
 - PADS p/d Viewer – Provided the ability to display donut shapes

Version 2017.11

ALERT!! If you are using a custom Preference file, you must manually correct this setting –

- Preferences > Console > Least Density > Terminals > Gullwing > SOD > Density Level > Side > From 0.20 to 0.05
- Go back to Console and change the Density Level back to your default setting
- Save your Preference file

NEW / ENHANCED:

- Calculator & 3D STEP:
 - Added 'Filter' component family to Chip and Molded Body
- Preferences:
 - Added a new feature in Terminals > Through-hole > Bottom Side Keepout option



FIXED:

- Calculator:
 - Fixed a SOFL component family issue. Moving SOFL parts to FP Designer set the body width to the Lead Span.
 - SOL component family – updated the Terminal > Image picture
- Library Editor:
 - Fixed a problem with PADS-to-FPX viewer and builder
- Preferences:
 - Terminals > L Bend – updated the Image picture
- FP Designer:
 - Pad Stack Manager: added Bottom Side keep-outs to pad stacks
- Allegro/OrCAD PCB:
 - Fixed a problem with slotted hole rotations
 - Fixed an issue with offset pad stack data
 - Moved the Labels to the origin 0, 0
- PADS Layout:
 - Fixed offset slotted holes rotated in 180 and 270 degree rotations when the CAD tool can only handle 0 and 90 degree rotations
- General:
 - Bottom Keep-outs should be working again for PADS Layout and Allegro/OrCAD PCB

Version 2017.10

ALERT! If you are using a personal FPX file you must **MANUALLY** update 4 solder joint goal settings for IPC-7351C > Preferences > Terminals. If you are using the default preferences, then there is no need to update anything.

Preferences:

- LEAST Density Level, Side Lead (Concave), Pitch ≤ 0.40 mm had incorrect values for HEEL and SIDE
 - HEEL & SIDE value was **-0.70** mm and changed to **-0.07** mm
- LEAST Density Level, Side Lead (Concave), Pitch > 0.50 and ≤ 0.65 mm had incorrect value for SIDE
 - SIDE value was **-0.70** mm and changed to **-0.07** mm
- MOST Density Level, Outward L Lead, Pitch > 0.40 and ≤ 0.50 mm had incorrect value for HEEL
 - HEEL value was **20.00** mm and changed to **0.20** mm
- MOST Density Level, Rectangular End Cap, Size ≤ 0.50 mm, TOE
 - TOE value was **0.60** mm and changed to **0.06** mm

NEW / ENHANCED:

- Library Expert License:
 - New 12-month maintenance license is now available! Purchases of new licenses now get all updates, regardless of software version, for a full year

FIXED:

- Unique Part Viewer - updated the unique part viewer
- Fixed a graphics display issue for UHD 4K monitors
- Calculator:
 - Saving component dimensions to FPX did not get the "Type" inserted and unable to retrieve the data
 - Axial Lead: Fixed an exception error when clicking in the Pitch cell
 - SOL terminal density levels not responding to pitch dimension in Preferences
 - Fixed a typo in SOL terminal mislabeled as Gull Wing
 - Fixed an issue with SOL solder joint goals were not properly set to IPC-7351C
 - Removed Contour Courtyard from Chip, MELF and Side Concave 2, 4-Pin component families
- Library Editor:
 - Fixed an issue when importing Excel spreadsheet data the Logical Description was not coming in
- Preferences:
 - LEAST Density Level, Side Lead (Concave), Pitch ≤ 0.40 mm had incorrect values for Heel and SIDE
 - HEEL & SIDE value was **-0.70** mm and changed to **-0.07**

- LEAST Density Level, Side Lead (Concave), Pitch > 0.50 and <=0.65 mm had incorrect value for SIDE
 - SIDE value was -0.70 mm and changed to -0.07
- MOST Density Level, Outward L Lead, Pitch > 0.40 and <=0.50 mm had incorrect value for HEEL
 - HEEL value was 20.00 mm and changed to 0.20 mm
- MOST Density Level, Rectangular End Cap, Size <= 0.50 mm, TOE
 - TOE value was 0.60 mm and changed to 0.06 mm

Version 2017.09

NEW / ENHANCED:

- V2017 FPX File Format:
 - Going forward, all saved FPX files will be upgraded to 2017 format
 - Your V2012 FPX file will automatically be upgraded when you save your FPX file to V2017 format
 - V2017 FPX files are not backward compatible with older versions of Library Expert
 - Highly recommended that all team members upgrade to V2017.09 at the same time
- Library Editor:
 - Updated FPX file format to accommodate unique/complex packages
 - This makes for easier viewing and creation
 - See [this video](#) for more details
- Parts on Demand:
 - Added 455 new Unique packages to POD
- Calculator:
 - Added “Bead” to the Molded Body component family
 - Added the supporting 3D STEP model

FIXED:

- Calculator:
 - SOD component family – fixed auto-generated Assembly Polarity Marker
 - Fixed an issue with Chip part courtyards when using 3 place dimensions
- Library Editor:
 - Fixed an issue with deleting columns at the end of a FPX file and then selecting Datasheet
 - Fixed an issue with Sorting Column data when importing new FPX files into an existing FPX file
- Altium:
 - Fixed an issue with FP Designer parts with SMD pads on Top & Bottom sides
- PADS VX:

- Fixed an issue for 3D STEP with LGA component family
- Fixed the sizing of the “Type” to map to the Legend Height
- Xpedition:
 - Irregular Rotated Paste Mask on a thermal tab now rotate properly

Version 2017.08

NEW / ENHANCED:

- Help & User Guide:
 - Created a new video for the Free BOM Builder Service - [Click Here!](#)
- Calculator:
 - Added alphanumeric pin assignment text in the component dimension panel for BGA, LGA, CGA and PGA; New UI Text – Increments: alpha on D axis; numeric on E axis
- Library Editor:
 - New “Tools > Check Manufacturer Names” feature: Checks names to standardize and ensure the spelling matches POD
 - Added new feature for importing Excel spreadsheet data
 - Pasting from Excel can now be by column and can be in any order (row 1 column headers must match the fpx header)
 - Pasted columns can be by manufacturer (required), part number, case code, logical description, data sheet
 - When changing the manufacturer name an option was added to apply the change to all similar manufacturer names

FIXED:

- Calculator:
 - 2-pin packages – fixed the auto-generated Polarity Dot to adhere to Legend to Pad Preferences
 - Updated the Mounting Hole calculator
- Library Editor:
 - Tools > Regenerate Library > Names – SOD was regenerating name changes prefix to CAP
 - Copy / Paste cells did not update text case
 - Saving an Excel spreadsheet generated FPX to file alternating blank rows are added

Version 2017.07

NEW / ENHANCED:

- Calculator:
 - Chip – added a new component family for Antenna – ANTC
 - When selecting the Red indicator light in the viewer a new message appears and the OK button updates the Data
 - Added alpha-numeric pin text in the thumbnail pictures for BGA, LGA, CGA and PGA
- 3D STEP:
 - Low Quality version is now available

FIXED:

- Calculator:
 - TO-92 – Added the missing A1 dimension to the thumbnail picture
 - Side Concave Package (2, 4 pins) – Synced pin names C & A with Preferences 1 & 2
 - DPAK – Fixed an issue when adding polarity marking and mirror footprint
- FP Designer:
 - Fixed an issue when adding multiple corner radius pad stacks
 - 3D STEP has been adjusted so that the 3D model will offset to a checkdown-list of Extents depending on whether you included them or not. The first that it finds, it will use. In order, It will center to the Component Outline Extents, then the Assembly Extents, then the Courtyard Extents, then the Extents of anything available. This should solve any offsetting problems.
- 3D STEP:
 - Updated PTH Radial Disk
 - Connected Model for Resistor, Varistor, and Thermistor families
 - Connected Model for Thermistor family
 - SOT23, SOT143, SOT223
 - Fixed a problem with the backside negative Z-axis gull-wing lead
 - SOT223
 - Put in a new body. The old one had an issue with PADS 3D.
- CADSTAR:
 - Fixed an issue when dealing with multiple alphanumeric pin numbers (i.e. 2A, 2B, etc.)
- Allegro/OrCAD PCB:
 - When the Anti-pad Size is smaller than the Pad Size, the translator will force it to be larger by 0.005 mm. This is done to bypass the needless warnings from Allegro. It shouldn't have any effect on the manufacturing.

- Fixed an issue that was generating duplicate pad-stacks for symmetrical pads at different orientations as found on a Chip Resistor, etc.
- Added an extra line when setting the unit's precision

Version 2017.06

NEW / ENHANCED:

- Help:
 - Added 70 new educational videos that cover every feature in Library Expert
 - Introduced BOM Builder program. Exchange your component list spreadsheet for POD parts.
- Calculator:
 - Added 3 new component families
 - Radial Molded, Fuse
 - SOD, Non-polarized
 - MELF, Non-polarized Diode

FIXED:

- Calculator:
 - Fixed the Courtyard Excess. When swelling solder mask, the Courtyard Excess was determined by the Mask and not the Pad.
 - All Header component families, the upper left "Pin Pattern" option was disabled
 - QFN component family - Fixed an issue with Rectangular Lead shape, changing pad shape did not work correctly
 - Fixed a Pad-to-Pad rule violation that trimmed the pad but not the solder mask
- FP Designer:
 - The Finish tab was not counting the Thermal Pad in the total pin quantity
- Preferences:
 - Added a note to the Preferences > Rules > Minimum Spacing Pad-to-Pad "for SMD use only (not PTH)"
- 3D STEP:
 - Fixed an issue with offset body outline in FP Designer
 - Removed the Polarity Marking for 2-Pin DFN Non-polarized Diodes
 - Updated SOT23, removed the Polarity Dot because no mfr. package has one
 - Updated PTH Radial Disk with Offset Leads - fixed a bug with the new Radial Disk Pin Lead that was not offsetting the leads properly per their Z (in 3D Mechanical Coordinates) location
- Allegro/OrCAD PCB V17.2:
 - Solved an issue when dealing with irregular pad stacks with corners
 - Fixed an issue where Chamfered Pads in FP Designer were not rotating correctly

- P-CAD:
 - The error about Capture files has been fixed

Version 2017.04

NEW / ENHANCED:

- Preferences:
 - Added Oblong pad shape to these component families:
 - SOD
 - SON w/rectangle leads with and without Thermal Pad
 - QFN w/rectangle leads with and without Thermal Pad
- FP Designer:
 - Removed a restriction of 50% pad width for Chamfered pad shape, now unlimited
- Help > Topics:
 - Added 70 new educational training videos to the program Help > Topics

FIXED:

- Calculator:
 - In the calculator dimension panel the Drafting > Courtyard > on/off switch was disabled
- Calculator to FP Designer:
 - Side Concave Package (2, 4-pins) swapped D & E dimensions going to FP Designer
- Preferences:
 - Fixed an issue with SOT Pin Pitch < 0.40 mm
- Print Feature:
 - Fixed an issue where the print feature was disabled
- Allegro/OrCAD PCB:
 - Fixed an issue with Drafting Line Arcs not coming out as intended
 - Fixed an issue with 16.6 and earlier that was causing the suppression of the inner layers of a through-hole pad stack. V17.2 scripts are unaffected
 - Fixed a rotation issue with irregular polys when using PADS to CAD
- PADS Layout:
 - Parts from FP Designer with pads that had corner radius on 2 sides crashed Library Expert

Version 2017.03

NEW / ENHANCED:

- Library Editor:
 - When updating a FPX Row, added option to throw a warning message if the new footprint name is different
- General:
 - Help file included with installer and no longer needs to be downloaded

FIXED:

- Calculator:
 - MELF 3D STEP model color assignment image (preferences) was clipped
 - Fixed an issue where the Legend and Assembly Ref Des Label was stuck on 270 degrees
 - Now the Labels are rotated correctly reflecting the rotation and length of the part
 - Label orientation value not displaying correctly in the query properties dialog form
 - When moving a rotated Calculator part to FP Designer, the Legend in FP Designer did not rotate
- FP Designer:
 - Pad Stacks with Offset Pads was not working as intended
 - Removed code for offsetting slotted holes but kept offset pads
 - If you have a FPX files with offset slotted holes, the pad stacks need to be recreated
 - The center of the slotted hole is the new pad stack origin
 - Fixed an issue with vias being added to a thermal pad and having the code recognize them as vias
 - Fix issue with thermal vias getting deleted after selecting the OK button
 - Added “via” as a new pad stack type
 - Updated via pad stack to remove thermal pattern for direct connection to plane
- Preferences:
 - When changing units to Mils, the Library Editor “Tools > Regenerate > Names” was not working correctly
 - When changing units to Mils, selecting FP Designer defaulting to Millimeters instead of Mils
 - Changing the Legend Label line width changed the Assembly Label line width (not the Legend)
- PADS to CAD:
 - Fixed an issue when importing older PADS V2004 & V2007 format p/d files
- Altium:
 - Updated progress bar in the CAD tool output menu

- When creating 3D STEP from FP Designer now sets to center origin
- Xpedition:
 - Translation of square thermal pads during PADS to CAD
 - Pad Stack Naming when using multiple complex pad stacks of the same name
 - Addressed issues dealing with Rotation of pad stacks

Version 2017.02

NEW / ENHANCED:

- Library Expert Lite
 - Removed pin quantity limits
- Calculator:
 - Axial Lead Pitch is now auto-calculated
 - Added Thermistor component family to Radial Disk and Radial Disk w/Offset Leads
 - Updated Footprint Name for TO92. Added a dash – after TO92- to conform to the standard.
- FP Designer:
 - Add new “Via” tab in Pad Stack Designer to insert a Via matrix to Thermal Pads
 - Now supports Contour Courtyards
 - Added “Via” as a terminal type
- Calculator and FP Designer Drafting Icon - changed text "Polarity Symbol" to "Drafting Symbols"
 - Added Legend Bottom
 - Added Assembly Drawing Bottom
- Preferences:
 - Courtyard feature to define separate excess distances for package body and pad
 - When changing Pad Shape, a notification appears to ask if you want to Change All
 - Added controls for Contour Courtyard minimum Gap of the contour bend
 - Solder Joint Goals now default to IPC-7351C but the user can switch back to IPC-7351B
 - When working in Mil Units, the user can now select fractions of a mil option

FIXED:

- FP Designer:
 - Fixed an issue when a part has multiple pins with the same pin name
 - No more hole offsets in pad stacks (because some pad tools can only handle pad offsets). This will effect some FP Designer parts if they were made with hole offsets.
 - Through-hole parts with pin rotations that weren't 0, 90, 180 or 270 had hole-slot rotations that didn't match the pad rotation
 - Automatic courtyard perimeter wasn't conforming to pad stacks with pad offsets or were associated with pins with orientations that weren't at 0, 90, 180 or 270.
- KiCAD Interface:
 - It was disabled in Library Expert Lite