

The PCB Footprint Expert Solution

PCB Libraries, Inc.



Footprint Expert FPX File

- Footprint Expert FPX file contains:
 - Necessary for Footprint & 3D STEP creation
 - Component Dimensions
 - Mfr. Recommended Pattern
 - Footprint Name
 - Physical Description of Package
- FPX File also contains Manufacturer Attributes
 - Library Management Data
 - Mfr. Case Code
 - Mfr. Name
 - Mfr. Part Number
 - Mfr. Logical Description
 - Mfr. Datasheet link

FPX Data Files: Component Data

- Component Family Category
- Component Dimensions
- Footprint Name
- Physical Description

*Created by
PCB Footprint Expert*

- Manufacturer Package Case Code
- Component Manufacturer
- Logical Part Number
- Logical Description
- Datasheet Web-link
- Component Reseller Part Number
- Component Reseller Link to Purchase Part

Entered by User

Footprint Expert FPX File

- The user can add custom columns to the FPX file
 - Corporate Part Number
 - Vendor Data
 - Values
- The FPX file row content never changes
 - The manufacturer never changes the part number, logical description or associated case code
- The FPX file constantly grows with new data
- The user can rebuild an entire CAD library from the FPX file based on new library construction rules

PCB Footprint Expert Library Data Manager

- Parts on Demand (POD) Subscription available to access millions of dynamic and modifiable component packages
- Build your FPX file once and output many different libraries with different rules and CAD formats
- Batch Create an entire FPX file of thousands of parts in seconds
- Part Library Manager with many advanced editing features
 - Undo/Redo, Find/Replace, Copy/Paste, Add/Delete Rows & Columns
- Sort data by column attributes
- Quickly move data from one FPX file to another
- Link to web datasheets or network PDF datasheets
- Search by component family categories
- Web-link checker verifies all your datasheet links in the background
- Quickly locate duplicate entries in any column

PCB Footprint Expert Preference Options

- The Footprint Expert applies your preference options with the component dimensions to auto-generate the perfect footprint
- Define your personal preference options and/or default options
 - Minimum pad to pad, pad to thermal, gang mask, thermal pad stencil
 - Select your Pad Shape – Oblong, Rectangle, D-Shape, Rounded Rectangle
 - Drafting rules for silkscreen, assembly, 3D model, courtyard and ref des
 - Component family rules for both surface mount and through-hole
 - Component terminal rules for 21 different lead forms
- Create multiple option files for different manufacturing applications
 - Rigid Board, Flexible circuit, Wave solder or any manufacturer specs
- Create multiple option files
- Share your Option files with every FPE User for consistent quality

Some PCB Footprint Expert User Preferences

Drafting Options

- Silkscreen Outline Line Width
- Silkscreen Outline Polarity Marker
- Map Silkscreen to Nom or Max Body
- Silkscreen to Land (Pad) Clearance
- Silkscreen Place Round-off
- Silkscreen Ref Des Height
- Assembly Outline Line Width
- Assembly Outline Polarity Marker
- Map Assembly to Nom or Max Body
- Assembly Outline Place Round-off
- Assembly Ref Des Min/Max Heights
- Courtyard Line Width
- 3D Model Colors

Design Rule Options

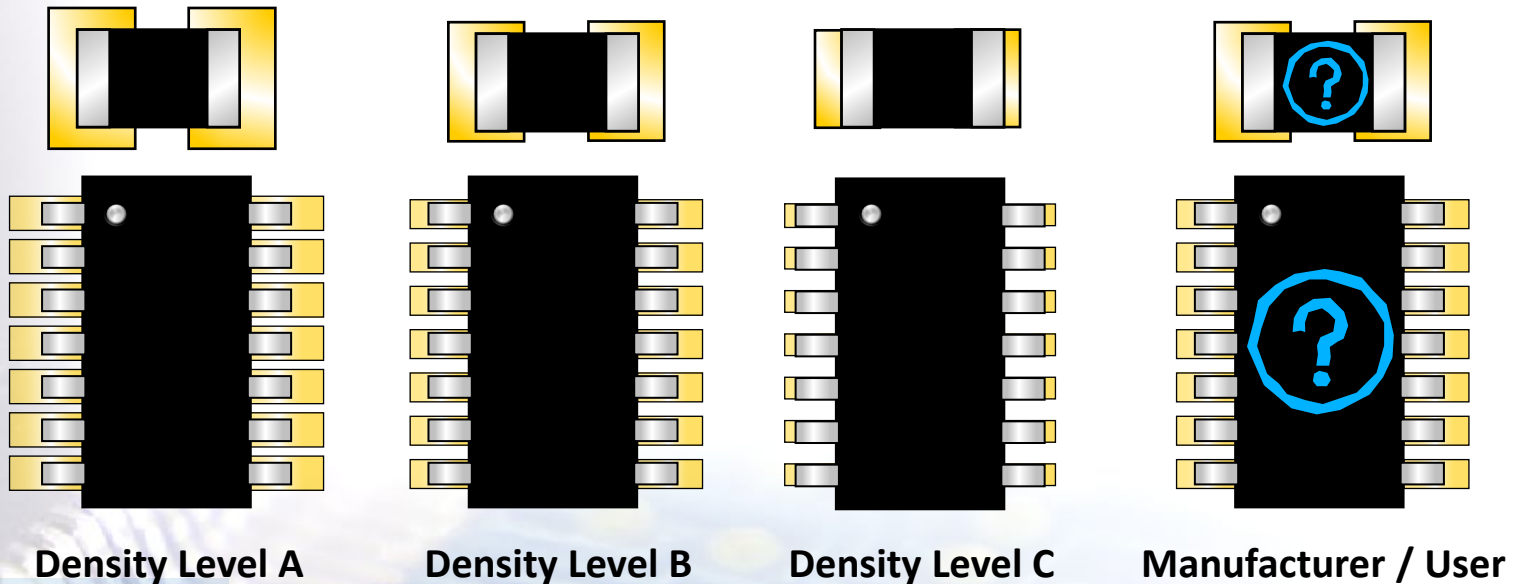
- Metric, Mils, Micrometers, Inch
- 3-Tier Environment or User
- Pad Shape – Rectangle, Oblong, D-shape
- Land to Land Clearance Min.
- Land to Thermal Pad Clearance
- Gang Mask Contour or Block
- Minimum Pad Trim Height
- Rounded Rectangle % of Width
- Rounded Rectangle Max Radius
- Rounded Rectangle Round-off
- Solder/Paste Mask Over/Under
- Thermal Paste Mask Reduction
- Local Fiducial Sizes & Min Pitch

PCB Footprint Expert Calculator Features

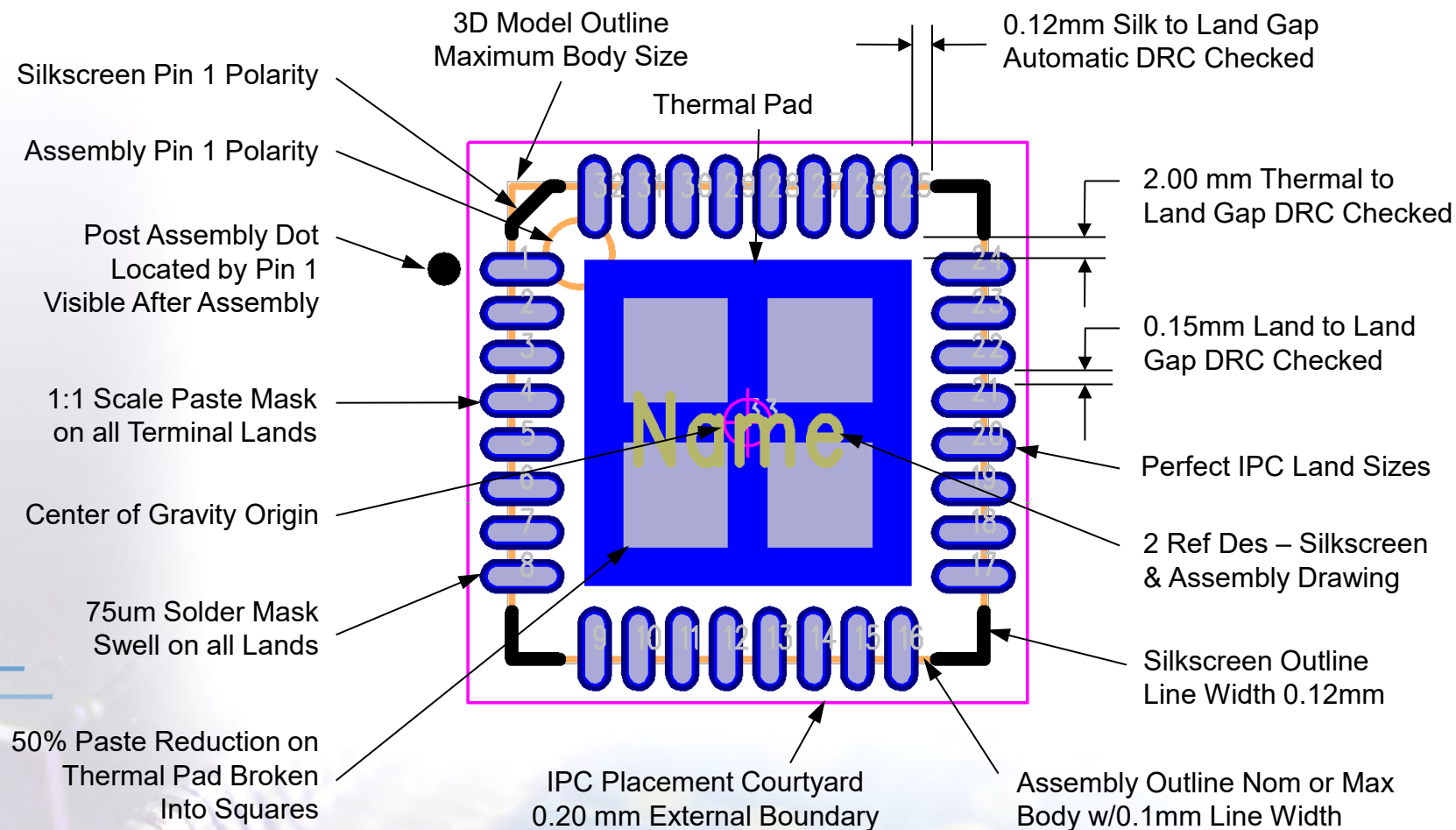
- On-line DRC checking with trim pad feature to adhere to your rules
- Change Units, Environments, Rules and Drafting objects on the fly
- Displays the component superimposed on the footprint
- Easily turn layers, elements on/off and change color display
- Use recommended component mfr. footprint dimensions
- Access all solder joint goal data for Toe, Heel and Side fillets
- Rotate and Mirror component and footprint as needed
- JEDEC dimension letters make it easy to transpose dimensional data
- Ability to trim pads under component package
- Local Fiducials on/off switch for BGA and QFP component families
- Enter Min/Max or Nom + Tolerance dimensions
- Pin renumber or rename to any alphanumeric character

Five Tolerance Settings

- **IPC Density Level A:** Maximum (Most) – For low-density product applications.
- **IPC Density Level B:** Median (Nominal) – Moderate level of component density.
- **IPC Density Level C:** Minimum (Least) – High component density typical of portable and hand-held product applications.
- **Manufacturer Recommended:** Component manufacturer recommended pattern.
- **User:** Definable preference rules created by the customer.

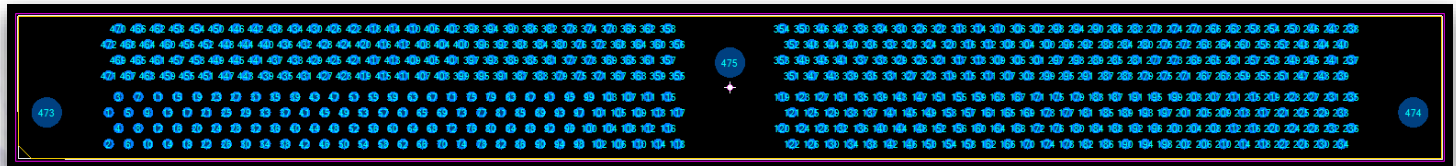
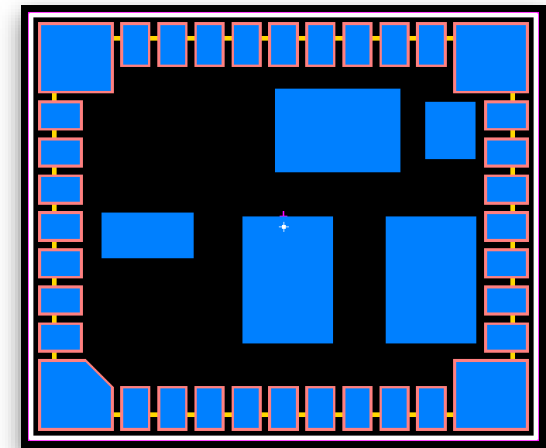
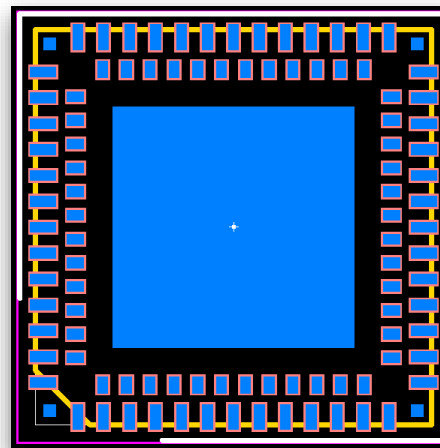


Footprint Library Elements (QFN)

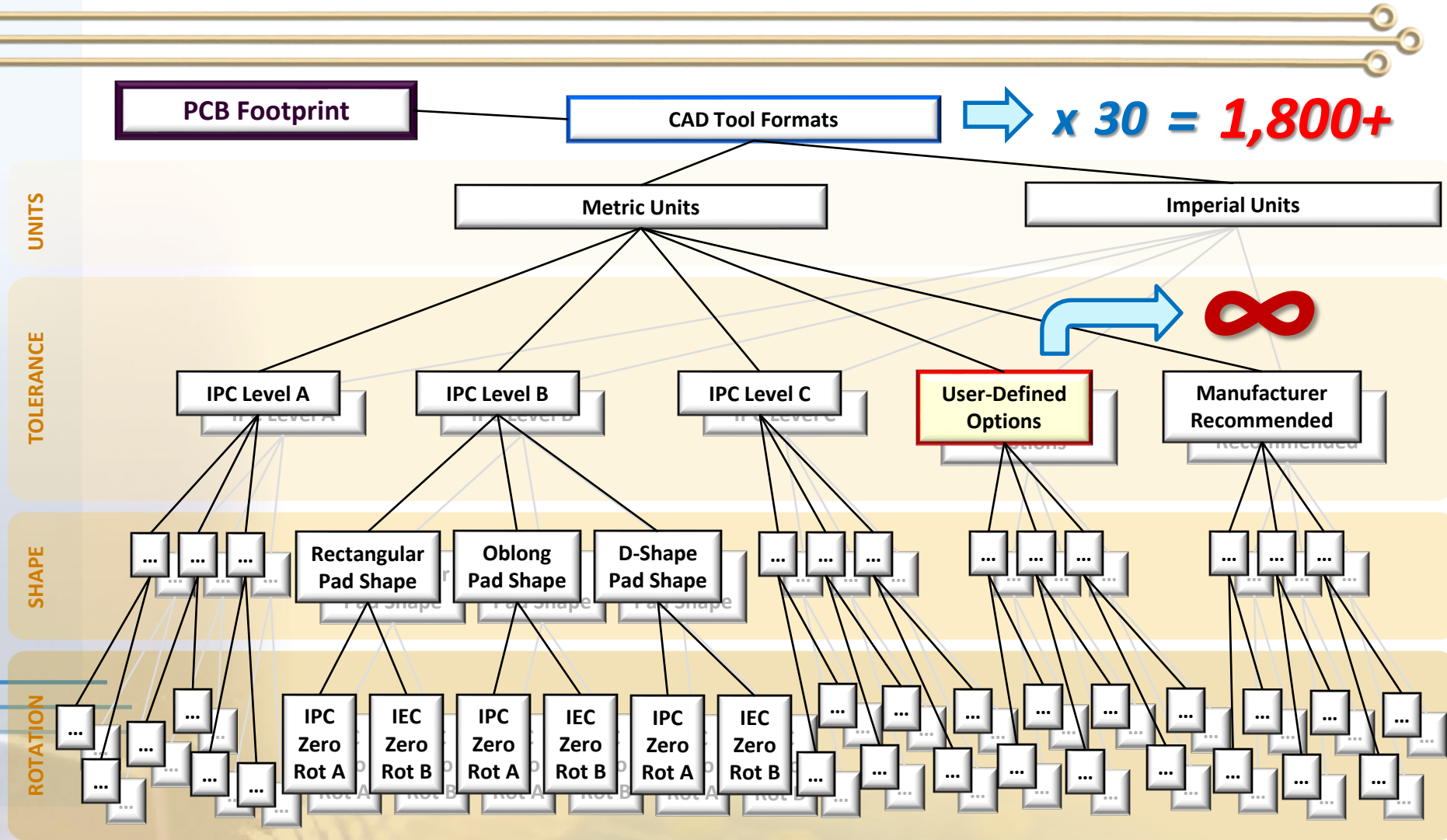


The “Footprint Designer” Module

- Traditional footprint software only calculates standard parts, constraining usage to only 50% of the components in the industry.
- PCB Footprint Expert also creates footprints for components with the following characteristics:
 - Asymmetrical
 - Various sizes of pads
 - Different pad shapes
 - Slotted holes
 - Pads on different grids
 - Import X/Y coordinates
 - Save data to FPX library



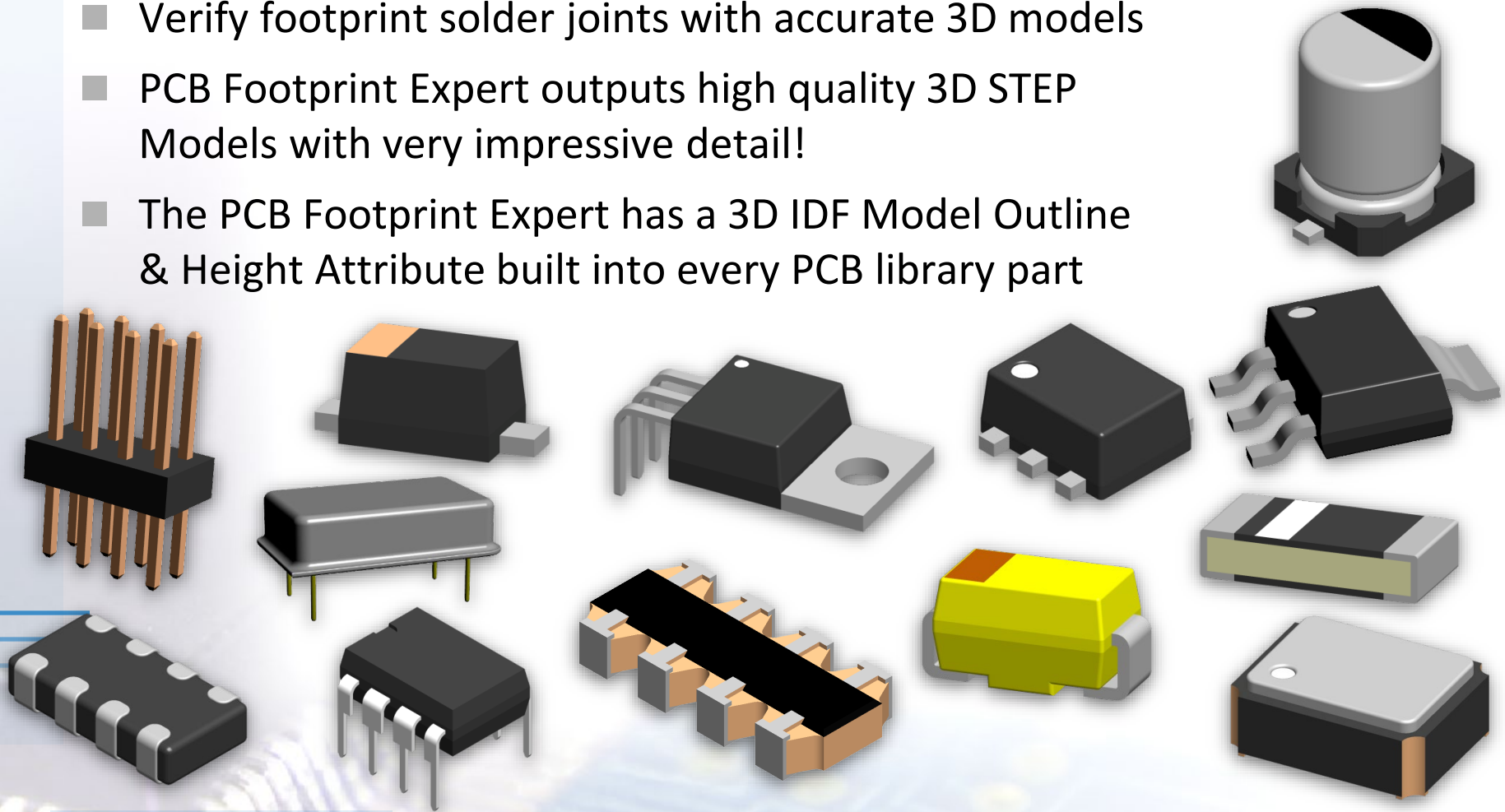
1,800+ Footprint Variations



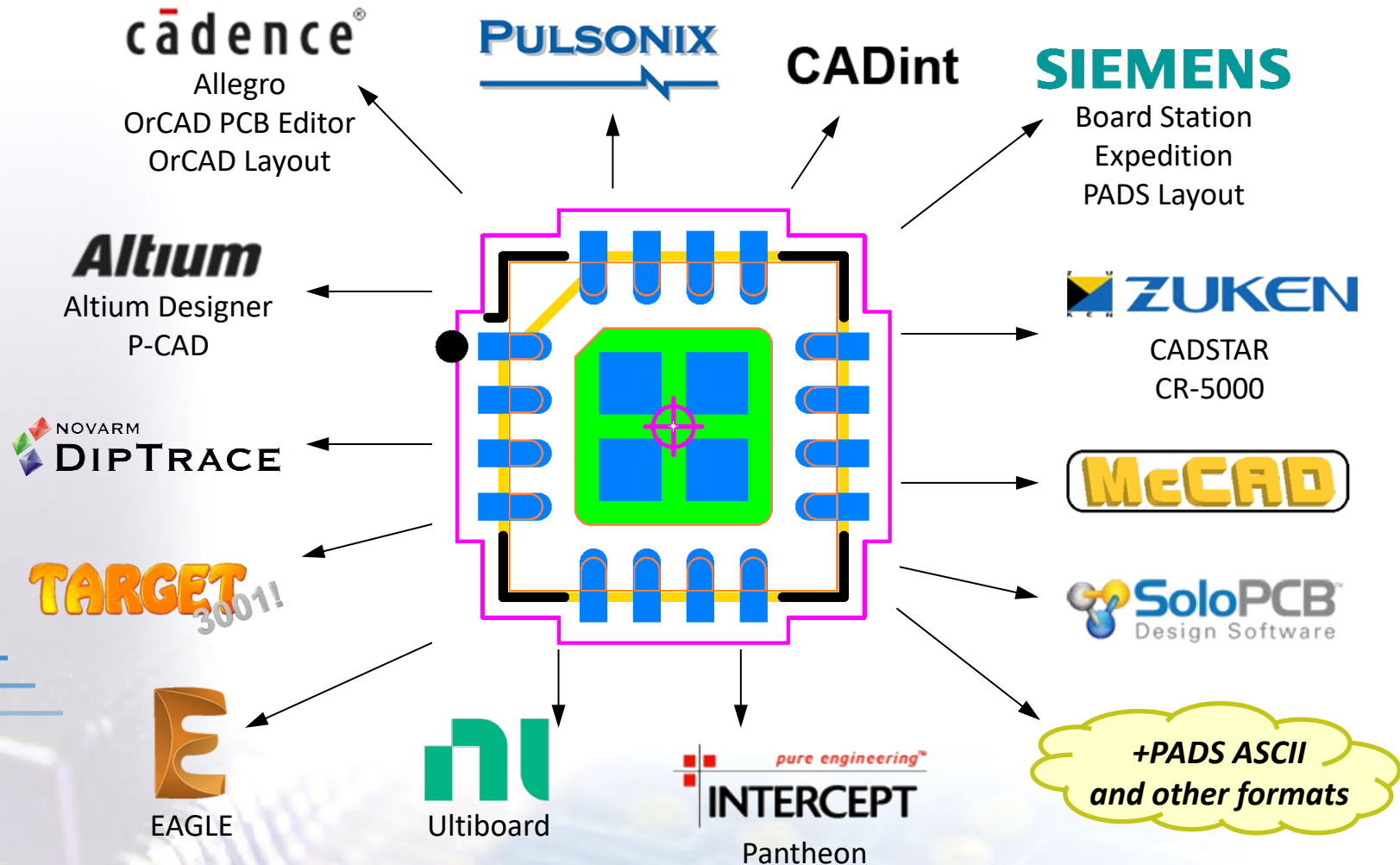
30 CAD tools X 2 units = 60 x 5 tiers = 300 X 3 pad shapes = 900 X 2 Rotations = 1,800+ variations

3D STEP Model

- Verify footprint solder joints with accurate 3D models
- PCB Footprint Expert outputs high quality 3D STEP Models with very impressive detail!
- The PCB Footprint Expert has a 3D IDF Model Outline & Height Attribute built into every PCB library part



PCB Footprint Expert CAD Tool Interfaces



Who the PCB Footprint Expert Benefits

- Large companies who use multiple CAD tools and want the same library quality in every CAD format
- **Anyone** who needs to replace or upgrade their entire library
 - Migrate to an IPC compliant library, or transition Inch to Metric
 - Apply consistency to libraries: footprints *and* 3D models
 - Touched by many different people with various skills
 - Built using many different rules
 - Created over many years
 - CAD library is poor quality and needs overhaul
 - Reduce a long term project to several days
- **Anyone** who needs flexibility to easily reconstruct an entire PCB library with totally different rules based on future needs