

The PCB Library Expert Solution

PCB Libraries, Inc.



Updated 2015.05.24

Library Expert FPX File

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

FPX Data Files: Component Data

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

*Created by
PCB Library 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

Library 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 Library Expert Library Data Manager

- Program comes with thousands of predefined 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 Library Expert Preference Rules

- The FPE program applies your preference rules with the component dimensions to auto-generate the perfect footprint
- Define your personal preference rules and/or default rules
 - 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 rule files for different manufacturing applications
 - Rigid Board, Flexible circuit, Wave solder or any manufacturer specs
- Create multiple rule files
- Share your Rules file with every FPE User for consistent quality

Some PCB Library 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

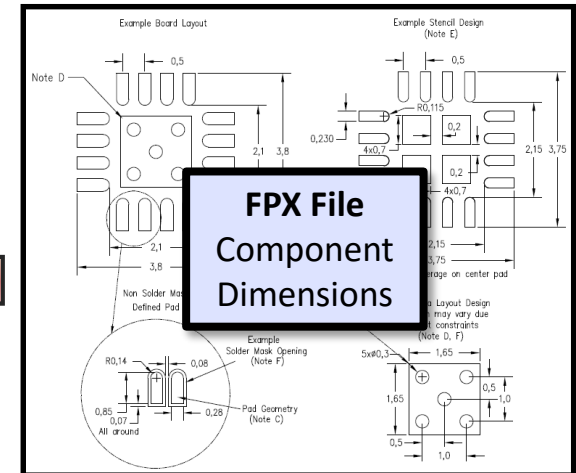
- Metric, Mils, Micrometers, Inch
- 3-Tier Environment or User
- Pad Shape – Rectangle, Oblong, D-shape
- Land to Land Clearance Min.
- Land to Thermal Pad Clearance
- Gang Mask Contour or Block
- Minimum Pad Trim Height
- Rounded Rectangle % of Width
- Rounded Rectangle Max Radius
- Rounded Rectangle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

PCB Library Expert

**Multiple User
and Company Defined
Rules & Preferences**

DAT Preference Files

- Imperial or Metric
- Pad Shapes
- Component Rotations
- Solder Joint Goals
- Rules & Tolerances
- Drafting Line Widths
- Polarity Marking

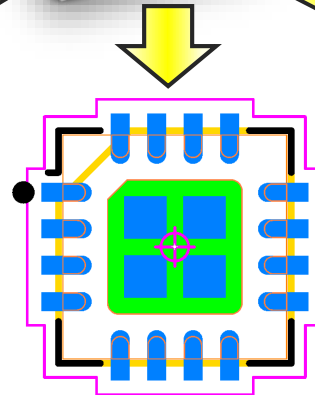


Quad Flat No-Lead (QFN with Tab) - QFN50P500X400X80-29
COMPONENT DIMENSIONS (Units - Millimeter)

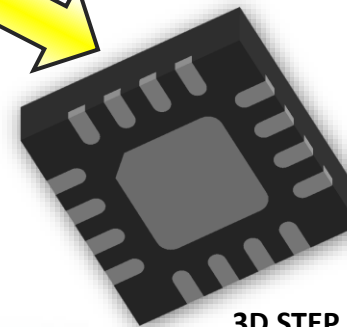
Lead Shape	D5Shape		
Pins Package	28		
Pins Side D	8		
Pins Side E	8		
Pins e	0.50		
	Min.	Nom.	Max.
A			0.80
B	0.20	0.25	0.30
C		0.35	
D	4.90	5.00	5.10
D2			3.55
E	3.90	4.00	4.10
E2			2.55
L	0.30	0.40	0.50
r	0.05		

* Optional entry

**Print Component and
Footprint Datasheet**



**Footprints for
21 CAD Formats**



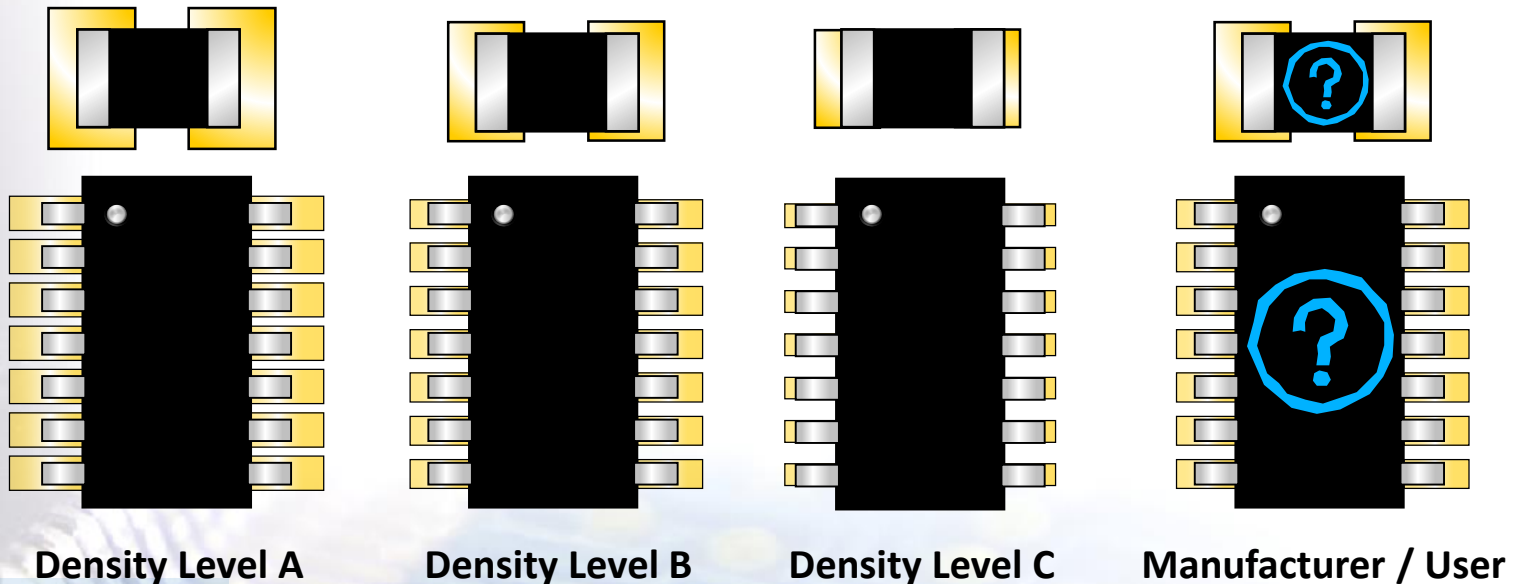
3D STEP Model

PCB Library Expert Calculator Features

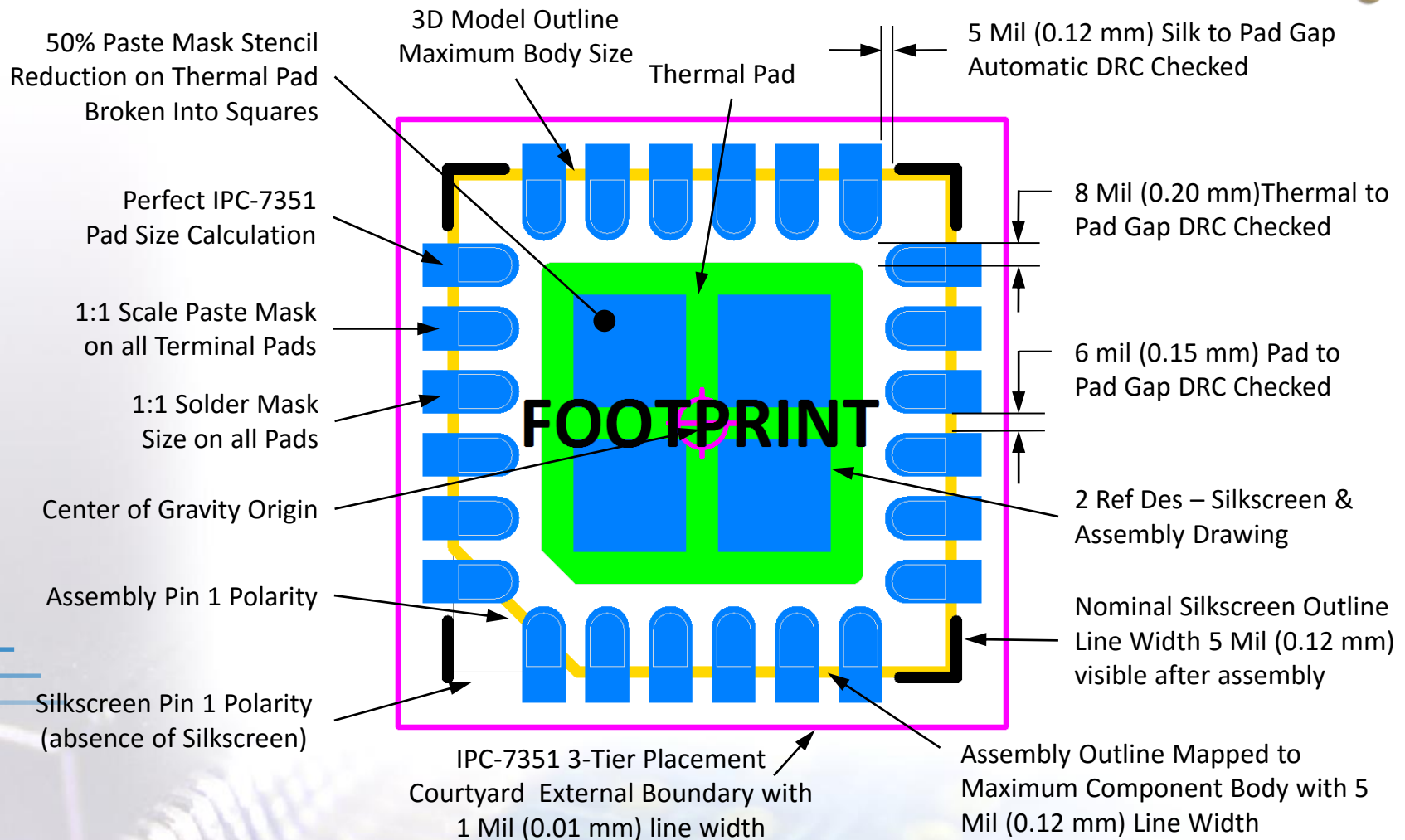
- On-line DRC checking with trim pad feature to adhere to your rules
- Change Units, Environments, Rules and Drafting objects on the fly
- Displays the component superimposed on the footprint
- Easily turn layers, elements on/off and change color display
- Use recommended component mfr. footprint dimensions
- 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.

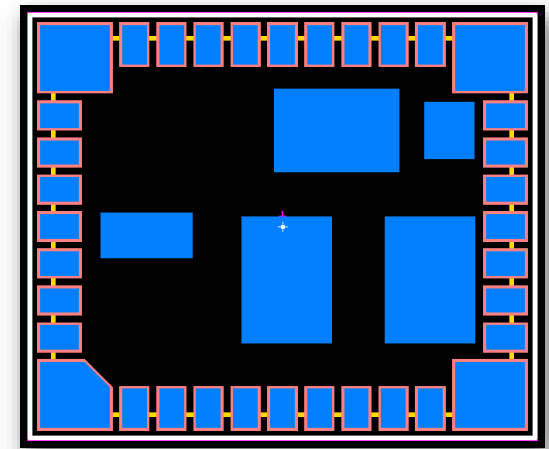
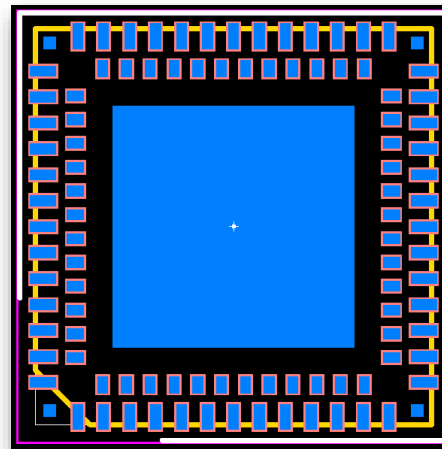


Footprint Library Elements (QFN)



The “Footprint Designer” Module

- Traditional footprint software only calculates standard parts, constraining usage to only 50% of the components in the industry.
- PCB Library 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

PCB Footprint

CAD Tool Formats

⇒ x 30 = 1,800+

UNITS

Metric Units

Imperial Units

TOLERANCE

IPC Level A

IPC Level B

IPC Level C

User-Defined
Options

Manufacturer
Recommended

SHAPE

Rectangular
Pad Shape

Oblong
Pad Shape

D-Shape
Pad Shape

ROTATION

IPC
Zero
Rot A

IEC
Zero
Rot B

IPC
Zero
Rot A

IEC
Zero
Rot B

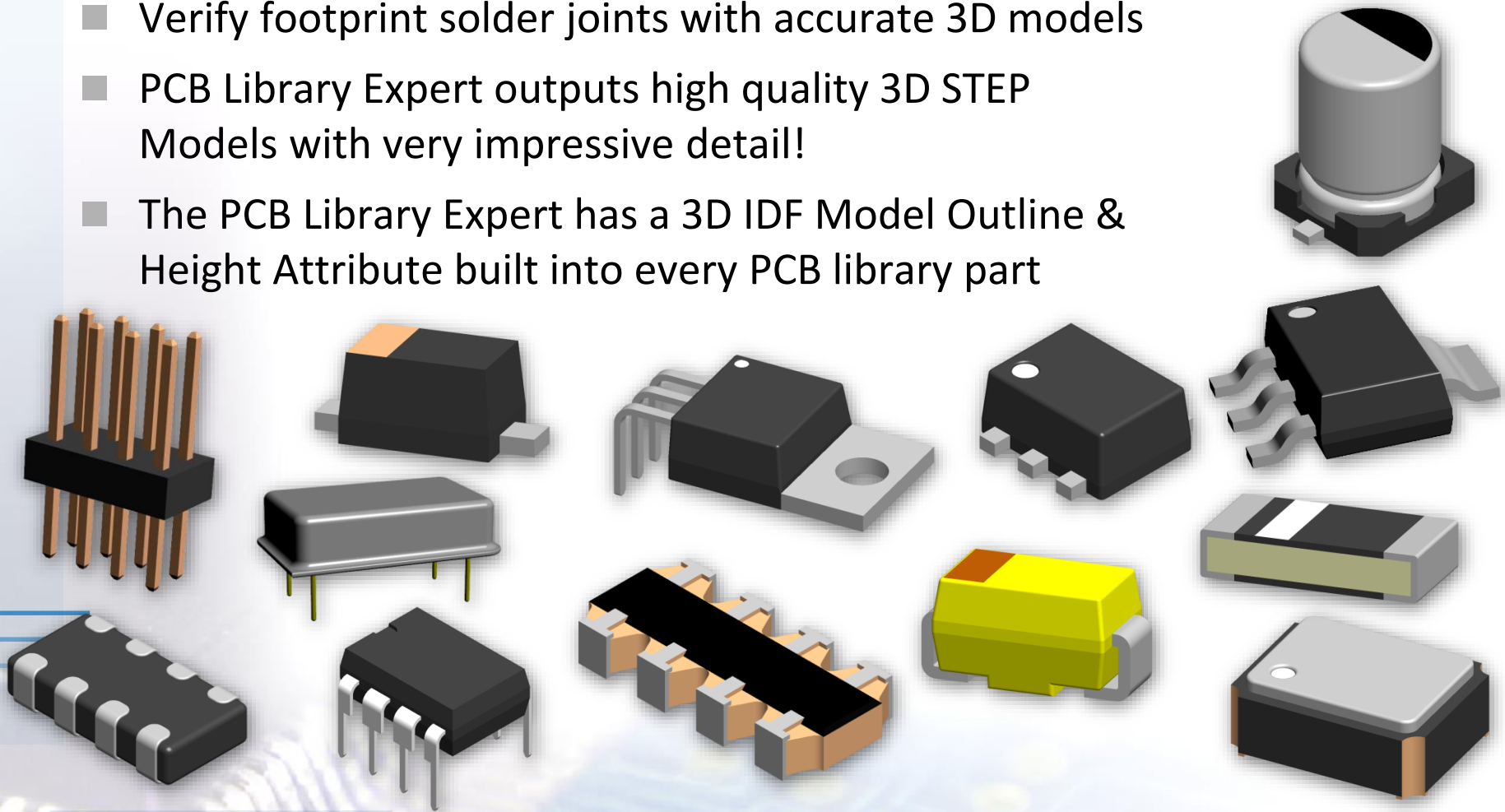
IPC
Zero
Rot A

IEC
Zero
Rot B

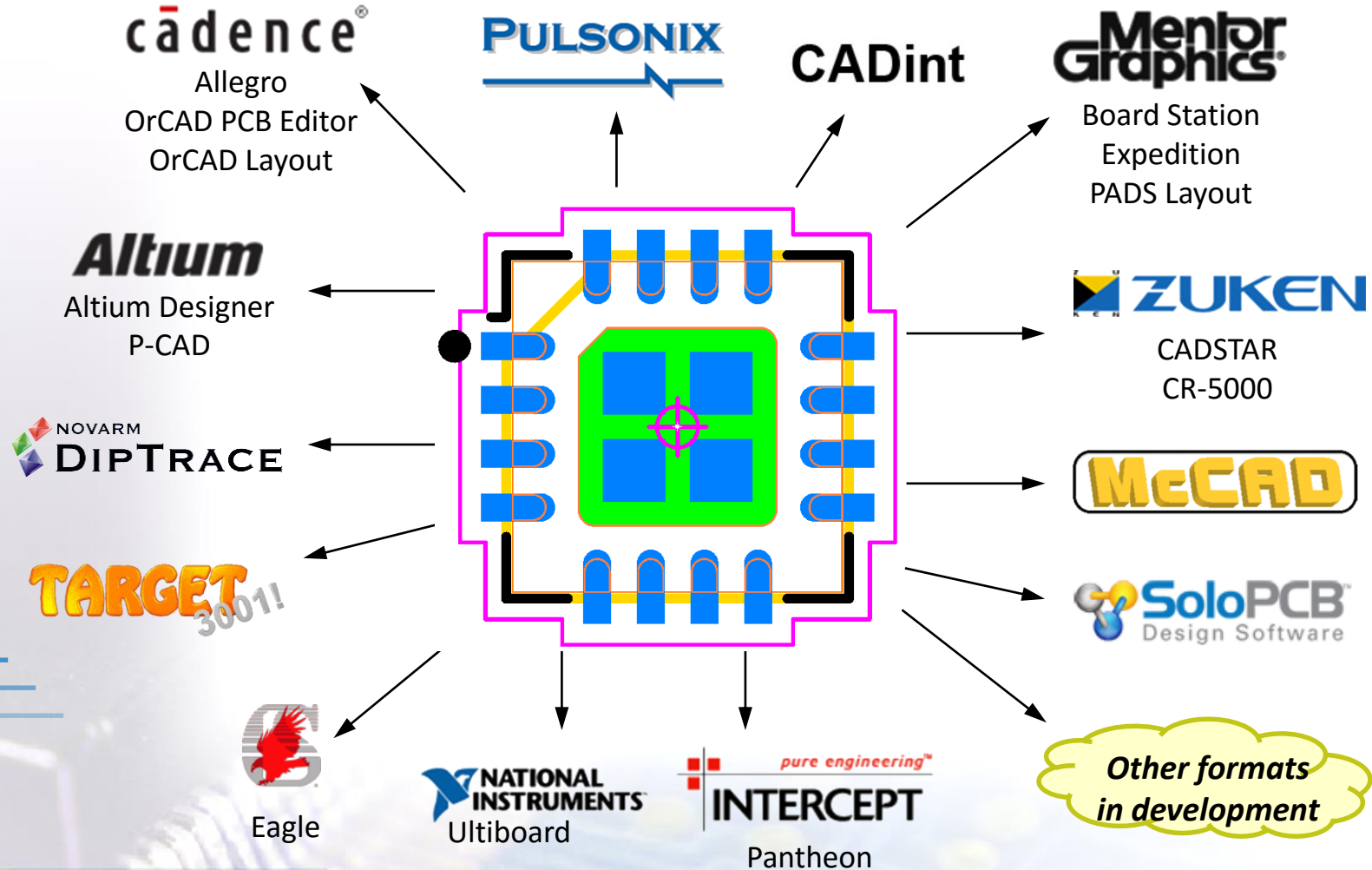
30 CAD tools X 2 units = 60 x 5 tiers = 300 X 3 pad shapes = 900 X 2 Rotations = 1,800+ variations

3D STEP Model

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



PCB Library Expert CAD Tool Interfaces



Who the PCB Library Expert Benefits

- Large companies who use multiple CAD tools and want the same library quality in every CAD format
- Companies who need 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
- Companies who need flexibility to easily reconstruct an entire PCB library with totally different rules based on future needs