The PCB Library Expert Solution

PCB Libraries, Inc.

What's YOUR Problem?

- Changing customer or manufacturing requirements

- Companies with multiple CAD tools and have different formats with various levels of quality

- Companies outdated/obsolete libraries
  - Migrate to an IPC compliant library
  - Transition from Inch to Metric
  - Apply consistency to libraries
    - 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

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: <u>footprints</u> *and* <u>3D models</u>
    - 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

PCB Library Expert
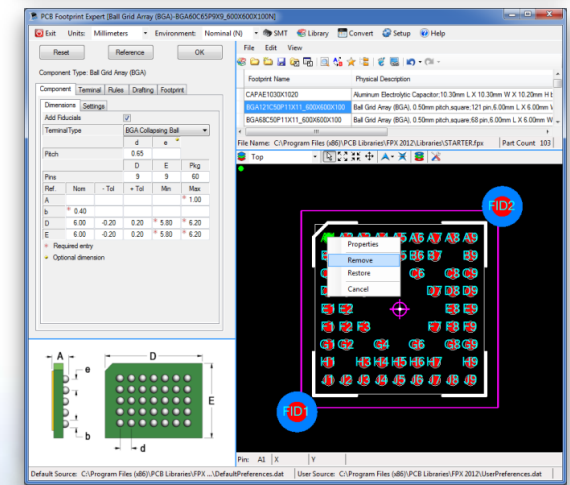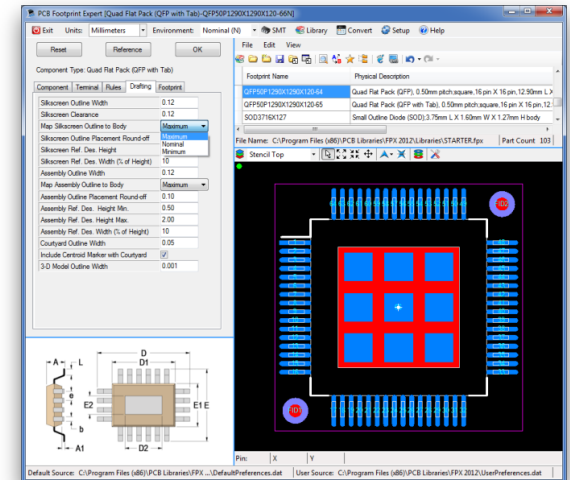
PCB Library Expert - Levels

Level I – FREE

- Browse and quality control component data

- Build one part at a time (does not safe preferences or component data)

Level II – Component Data

- Search, modify, and save to "FPX data files" with the Librarian feature

Level III – CAD Ouput

- Output component FPX data to any major CAD format

PCB Library Expert - Usage

FREE!

Level 2
- ✓ *Level 1, plus*
- ✓ Edit / Modify
- ✓ Save to "FPX" file (library documentation)
- ✓ Print datasheet for record-keeping

Level 1
- ✓ Verify
- ✓ Browse / Search
- ✓ Quality Control

Level 3
- ✓ *Levels 1 & 2, plus*
- ✓ Create directly to CAD format
- ✓ Output FPX data to CAD format
- ✓ Output 3D models (3D STEP)

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
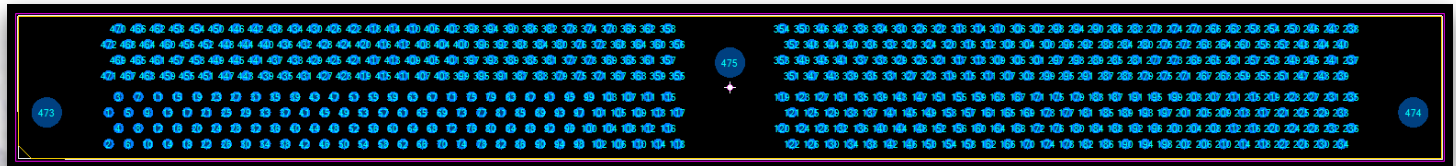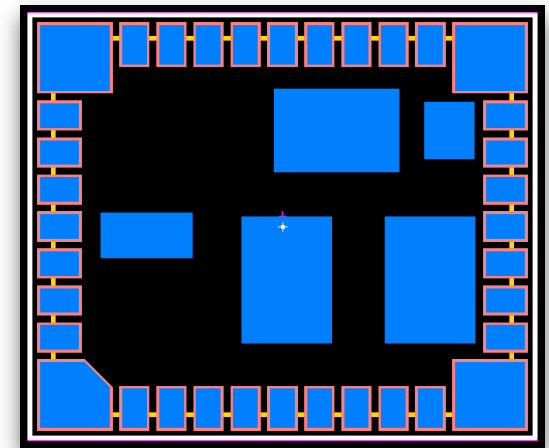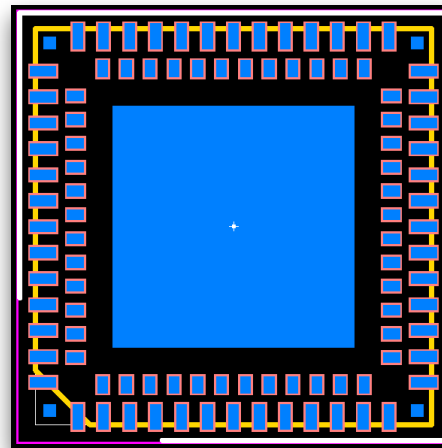
PCB Library Expert Preference Rules

- 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
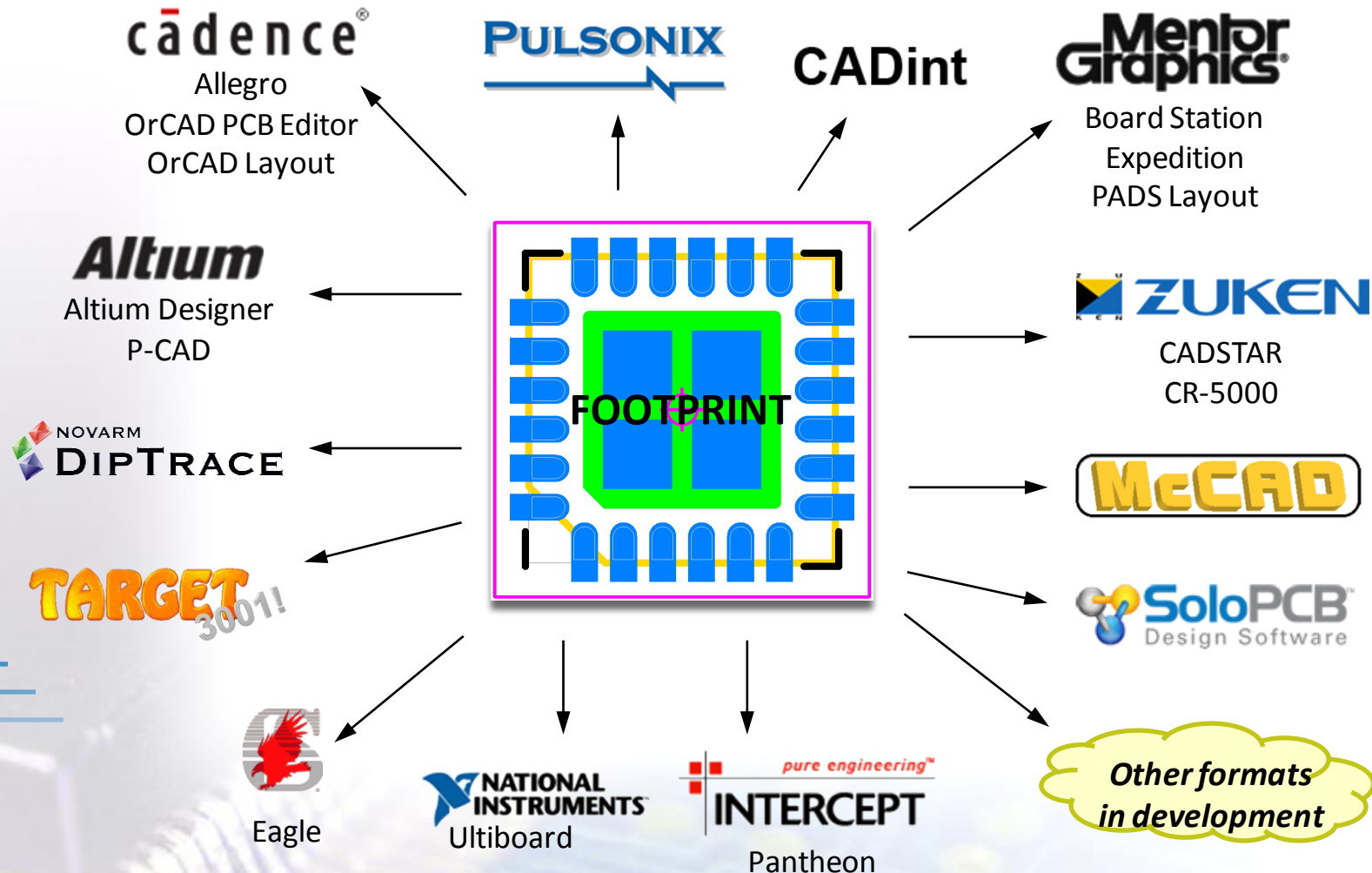
PCB Library Expert Calculator Features

- 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

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

PCB Library Expert CAD Tool Interfaces

cadence®
Allegro
OrCAD PCB Editor
OrCAD Layout

PULSONIX

CADint

Mentor Graphics®
Board Station
Expedition
PADS Layout

Altium
Altium Designer
P-CAD

NOVARM DipTrace

ZUKEN
CADSTAR
CR-5000

FOOTPRINT

McCAD

TARGET 3001!

Eagle

NATIONAL INSTRUMENTS™
Ultiboard

INTERCEPT *pure engineering™*
Pantheon

SoloPCB Design Software

Other formats in development

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

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

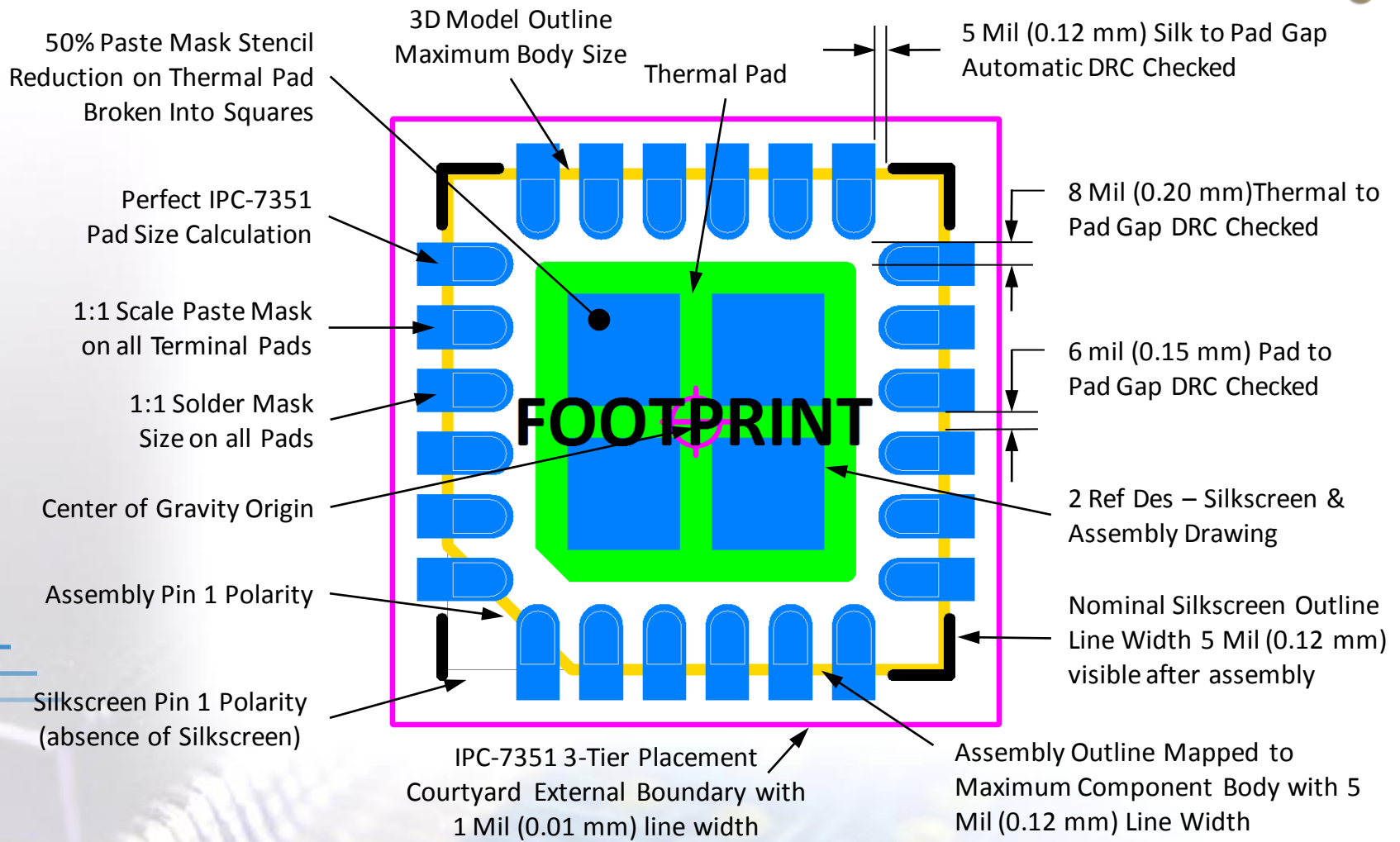*Entered by User*

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 Line Width

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

Footprint Library Elements (QFN)

50% Paste Mask Stencil Reduction on Thermal Pad Broken Into Squares

3D Model Outline Maximum Body Size

Thermal Pad

5 Mil (0.12 mm) Silk to Pad Gap Automatic DRC Checked

Perfect IPC-7351 Pad Size Calculation

8 Mil (0.20 mm) Thermal to Pad Gap DRC Checked

1:1 Scale Paste Mask on all Terminal Pads

6 mil (0.15 mm) Pad to Pad Gap DRC Checked

1:1 Solder Mask Size on all Pads

FOOTPRINT

2 Ref Des – Silkscreen & Assembly Drawing

Center of Gravity Origin

Assembly Pin 1 Polarity

Nominal Silkscreen Outline Line Width 5 Mil (0.12 mm) visible after assembly

Silkscreen Pin 1 Polarity (absence of Silkscreen)

IPC-7351 3-Tier Placement Courtyard External Boundary with 1 Mil (0.01 mm) line width

Assembly Outline Mapped to Maximum Component Body with 5 Mil (0.12 mm) Line Width

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.

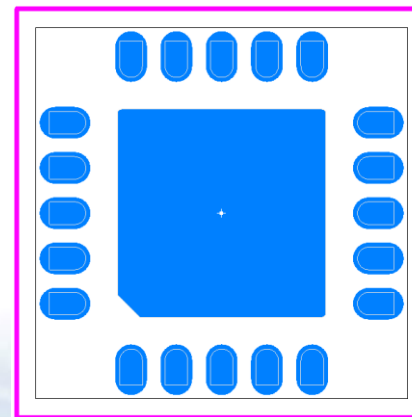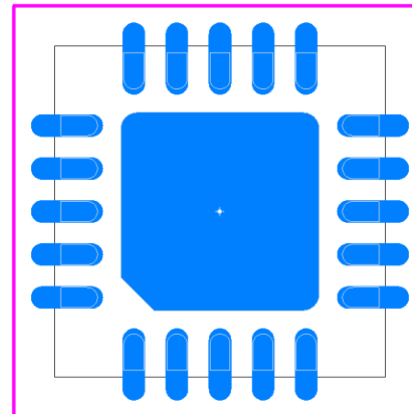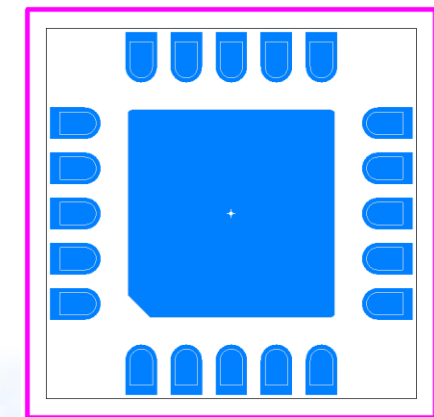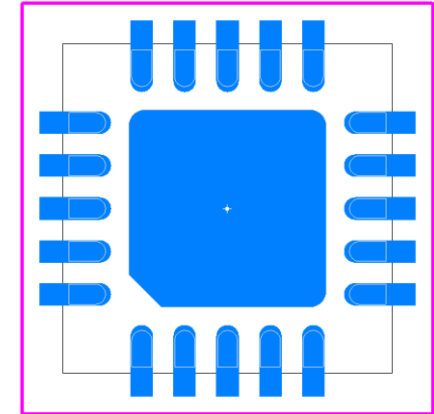- **User:** Definable preference rules created by the customer.

Density Level A   **Density Level B**   **Density Level C**   **Manufacturer / User**
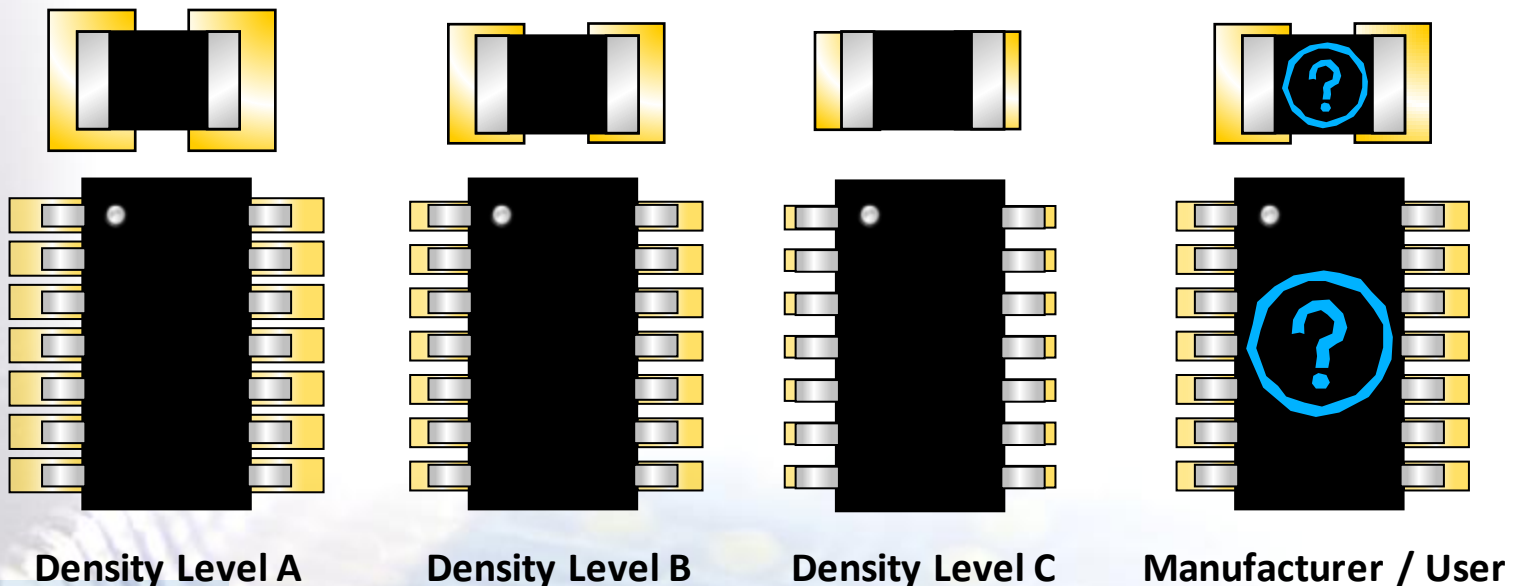
Component Pad Shapes

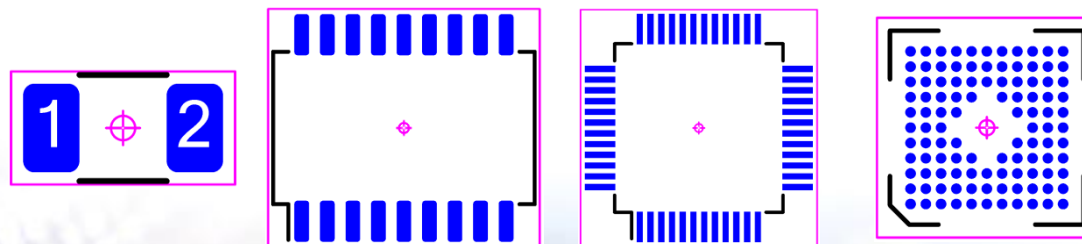**Rectangle or Rounded Rectangle**
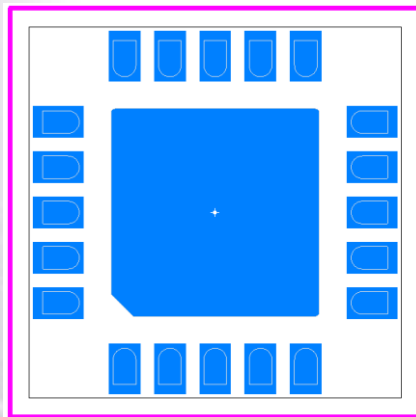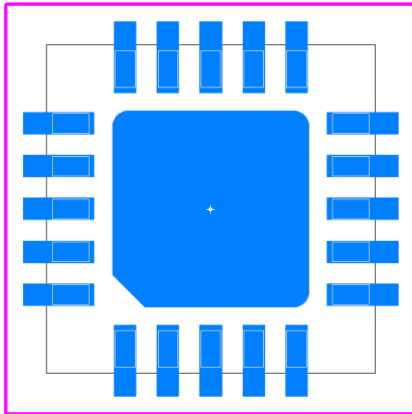
Oblong

D-shape

Zero Component Orientation

Zero Orientation with Pin 1 in Upper Left Corner Introduced in 2007 in the IPC-7351A Publication
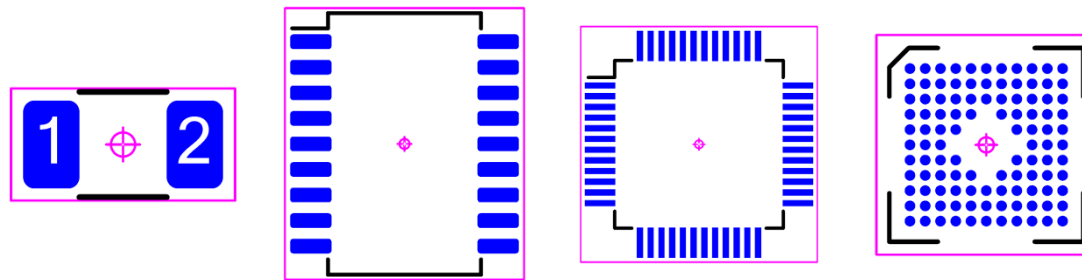
Zero Orientation with Pin 1 in Lower Left Corner Introduced in 2009 in the IEC 61188-7 publication

SMT Component Family Categories

BGA (Ball Grid Array)

AEC (Aluminum Electrolytic Capacitor)

CQFP (Ceramic Quad Flat Pack)

CFP (Ceramic Flat Pack)

CGA (Column Grid Array)

CHP (Chip Rectangular End Cap)

CHPA2 (Chip Array 2-Side Flat or Concave)

CHPAX (Chip Array 2-side Convex)

CHPA4 (Chip Array 4-side Flat or Concave)

1DIOSC (Diode, Side Concave)

DFN 2-lead

TO (DPAK)

XTAL (2-pin Crystal)

LCC (Leadless Chip Carrier)

LGA (Land Grid Array)

MELF (Metal Electrode Lead Face)

MLD (Molded Body Diodes & Capacitors)

OSCC (Oscillator, Corner Concave)

OSCS (Oscillator, Side Concave)

OSCJ (Oscillator, J-Lead)

OSCL (Oscillator, L-Bend)

PLCC (Plastic Leaded Chip Carrier)

QFN (Quad Flat No-Lead)

QFP (Quad Flat Package)

SOIC/SOP (Small Outline Package)

SOD (Small Outline Diode)

SODFL (Small Outline Diode)

SOFL (Small Outline Flat Lead)

SOJ (Small Outline J-lead)

PSON (Pull-back Small Outline)

SOT (Small Outline Transistor)

SOTFL (Small Outline Transistor)

SOT23 (Small Outline Transistor)
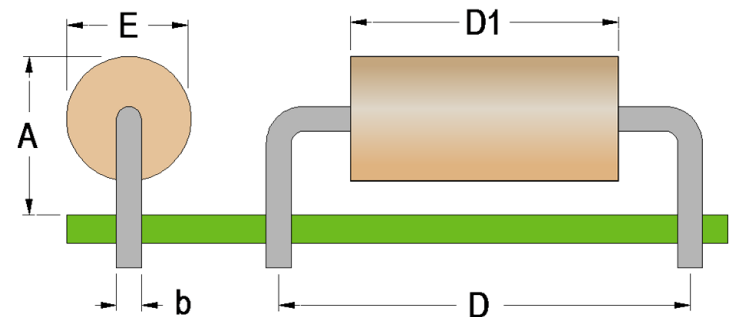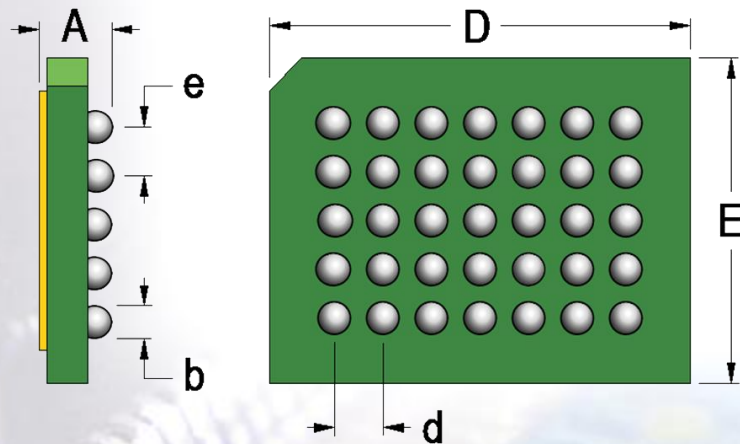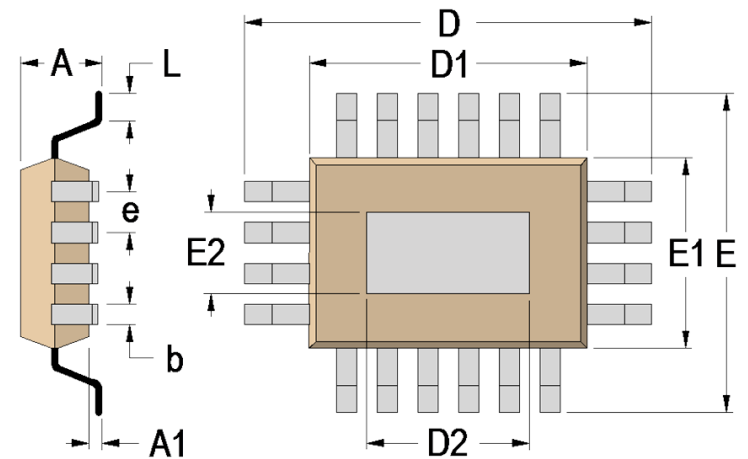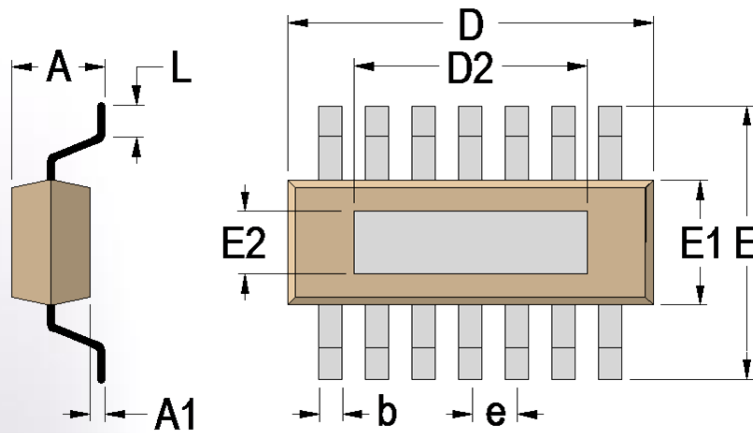
SOT143 (Small Outline Transistor)

SOT223 (Small Outline Transistor)

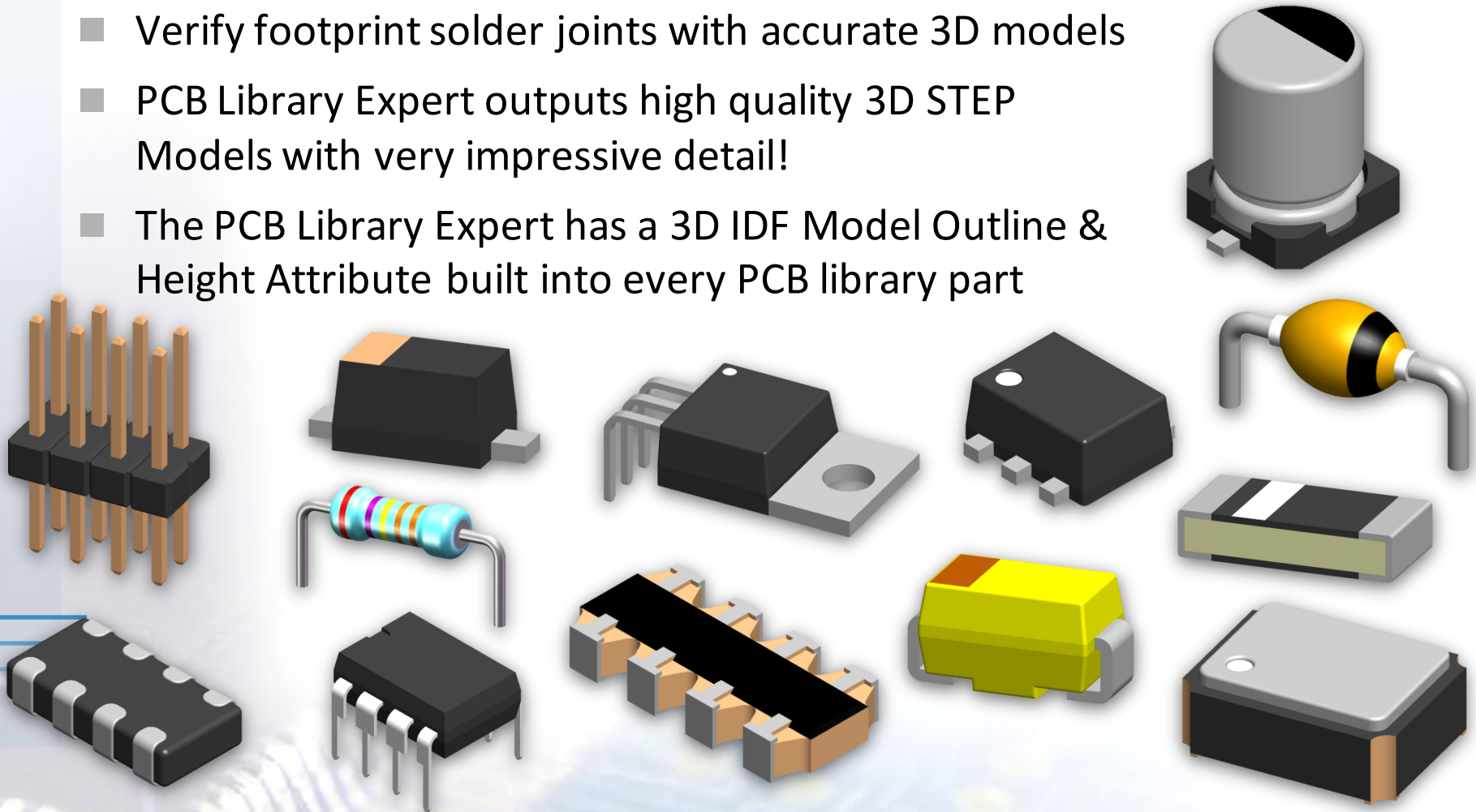DFN 3-lead (Dual Flat No-lead)

DFN 4-lead (Dual Flat No-lead)

….and many more to come

JEDEC Component Dimensions

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 Distributors

With distributors world-wide, we speak your language!

- Belgium
- China
- Denmark
- Germany
- India
- Italy
- the Netherlands

- Poland
- South Africa
- South Korea
- Sweden
- United Kingdom
- United States

view growing list of distributors: www.PCBLibraries.com/Distributors

PCB Libraries, Inc.   -   1153 S. Lee Street, #220   -   Des Plaines, IL 60016

Tel: 847.557.2300   -   Fax: 847.557.2319   -   www.PCBLibraries.com