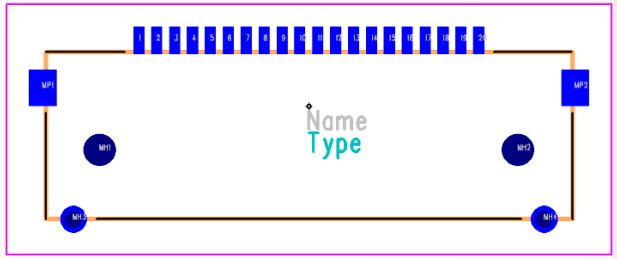
PCB Libraries Footprint Expert - Comparison with other Online Sources

SAMTEC - FCS8-20-01-L-S-A-TR

	Source 1	Source 2
		
AUTO-GENERATED library parts use predefined or user-modified rules – share across your organization; **RULES-based, auto-generated part library**	**MANUALLY CREATED** by many different people, using **countless rules** – results in a board with inconsistently built parts, **far more prone to introduction of costly errors**; massive variations in parts inconducive to standardized rule-based library
Built to exact datasheet specification, either Metric or Mils	**Units vary based on source**, and Metric often converted from Mils with rounded-off measurements; **converting pin pitch often results in leads not fully centered on pads**
User-defined consistency of line widths for all parts	**Many different line widths** used across different footprints due to **lack of standard rules**
Accurate contour courtyards are auto generated	**Sloppy or non-existing contour courtyards**
Relevant data: Footprint Name, Physical and Logical Descriptions, datasheet link, and more	**Nothing, or inconsistent random data**
Auto-generated keep outs on all non-plated holes for copper poly shapes, traces and vias	**Missing keep-out will cause DRC errors** at the manufacturer if copper, traces, and vias are placed within 0.25 mm of the non-plated hole
Pin 1 polarity markers for silkscreen and assembly	**Pin 1 markings missing or vary** in shape and in size
Footprint Origin at center of gravity	**Footprint Origins randomly placed**
Uniform, silkscreen outlines are whole objects mapped to the maximum package dimensions	**Silkscreen is fragmented** and mapped to the nominal package dimensions
Clear, consistent, intelligently placed markings to **minimizes use of board space**	Inconsistent and extra markings and excessive courtyards **requires more placement space**
Consistent silkscreen trimming around all pads	**Inconsistent and random silkscreen trimming** around pads
Solder Mask swell auto-applied and configurable	**Inconsistent solder mask swell** from one footprint to another
Through-hole **pad stack hole sizes automatically defined** by maximum terminal lead diameter, and **pad size auto-calculated** by 1.5 x hole diameter	**Lack of published, consistent rules** to define through-hole pad stacks make it impossible for proper quality control of resulting footprint, **results in capriciously calculated footprints**
Non-plated holes have smaller pad sizes to **prevent copper slivers after drilling process**	Non-plated holes have a pad size equal to the hole size create **risk of copper slivers** after the drilling process; the **plane antipad is randomly defined with no obvious rules**

2024-07-26

The Footprint Expert automates footprints to perfection. ***Professional, detailed, and accurate parts – all built consistently with identical rules, regardless of how many people are involved in your library development. Contact us for more samples, details, demo, or evaluation license!***

847.557.2300 - Sales@PCBLibraries.com

PCB Libraries