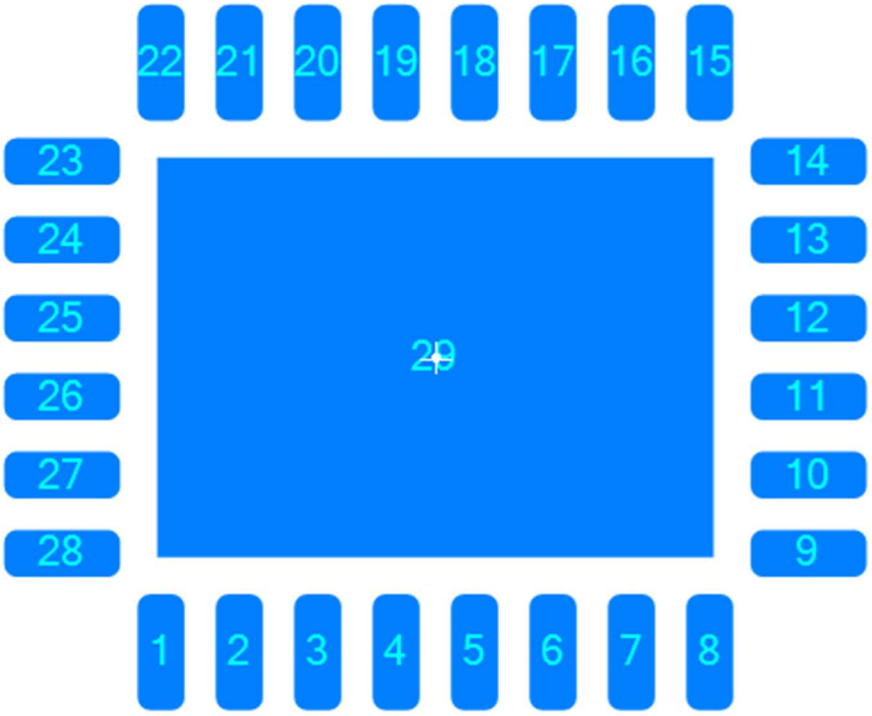
Manufacturer Recommended Patterns

The misconception that simply using the manufacturer recommended pattern will keep you safe is enough to get you in serious trouble, and over time, create mayhem in your CAD library. You need much more than the recommended pattern for a reliable PCB design!

Most component manufacturers provide recommended solder patterns in their datasheets, but there are two major issues you need to consider:

1. What rules were used to calculate the mfr. recommended solder pattern, and are they consistent across your library of parts for different manufacturers?
2. Since manufacturers provide no drafting outline data, how are you consistently creating these and applying them throughout your library over time?
3. Did the component manufacturer create reference PC boards and run them through stress tests to check their solder joint goals against the IPC J-STD-001 standard?
4. Component manufacturer recommended patterns are typically designed using nominal package dimensions; does the recommended pattern also accommodate the minimum and maximum material condition of the package tolerances specified in their datasheet?

Here is a Manufacturer Recommended Pattern with rounded rectangle pad shape. All component manufacturers (except Texas Instruments) recommended rectangle pad shape. However, paste mask stencil apertures are laser cut with rounded corners.



© 2025 PCB Libraries, Inc.

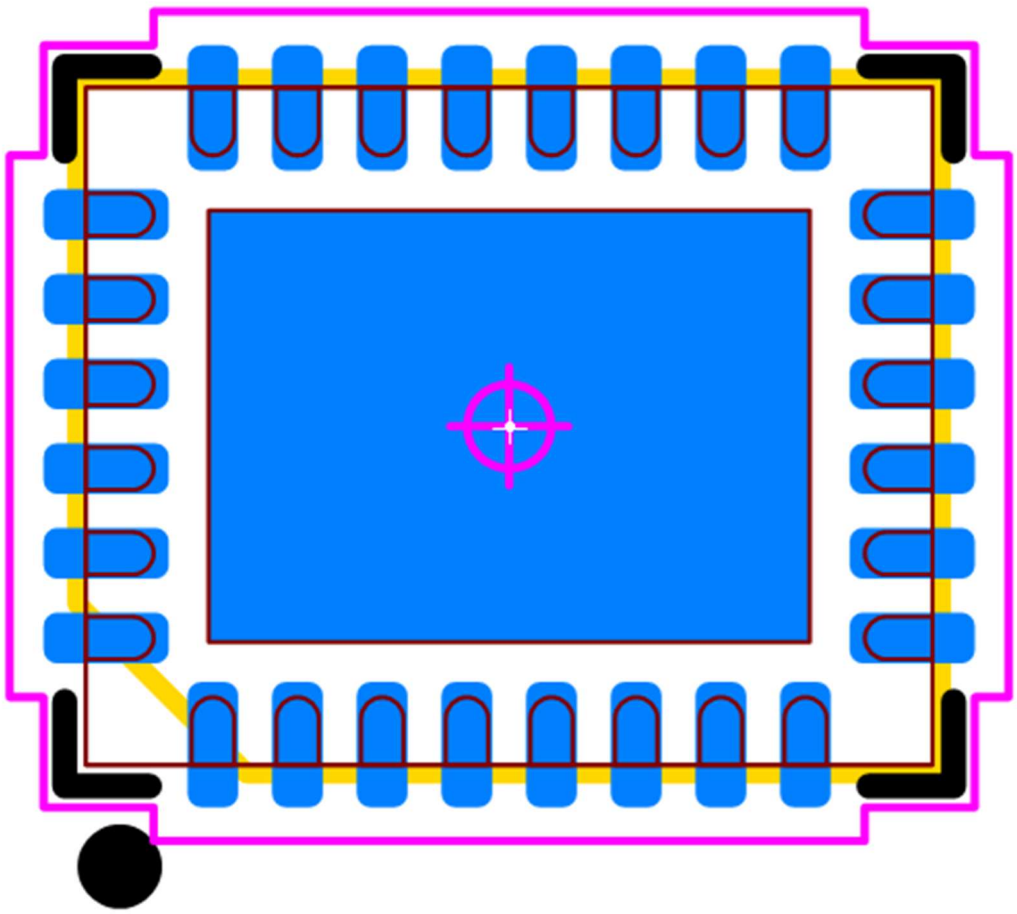
This is obviously missing the six critical drafting outlines. Traditionally, these have been manually created with a plethora of inconsistent rules that get adjusted over time:

1. Silkscreen
2. Assembly
3. Courtyard
4. Origin
5. Component
6. Terminals

It takes more time to manually add all the necessary Drafting Outlines than it does to create the Manufacturer Recommended Pattern, and these are just as important as the pattern itself!

We offer technology that automatically generates these from package dimensions! All drafting outline widths and spacings are defined in your personal master Options file. This guarantees consistent quality throughout your entire CAD library. You can also turn off any drafting outline that you do not want in your CAD library, but easily "bring it back" if you decide later that you need it!

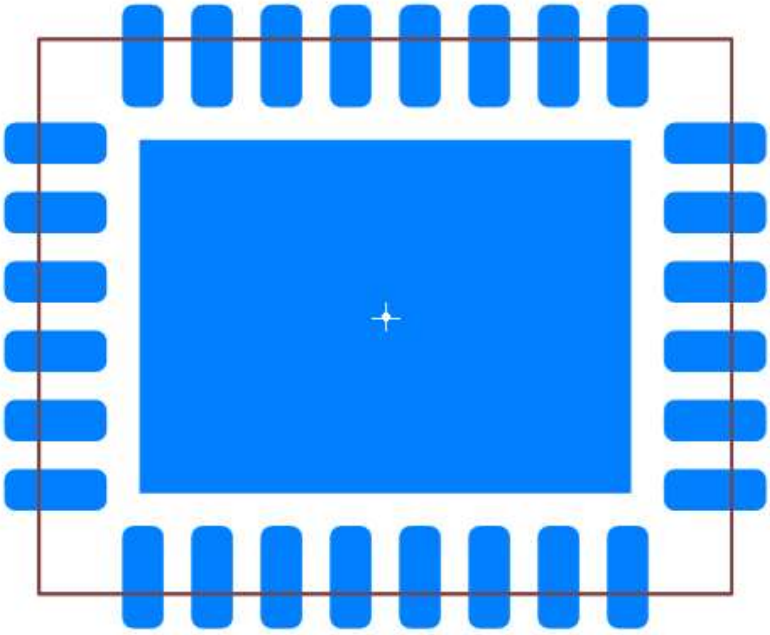
This is your end goal, read on for details how to reach it:



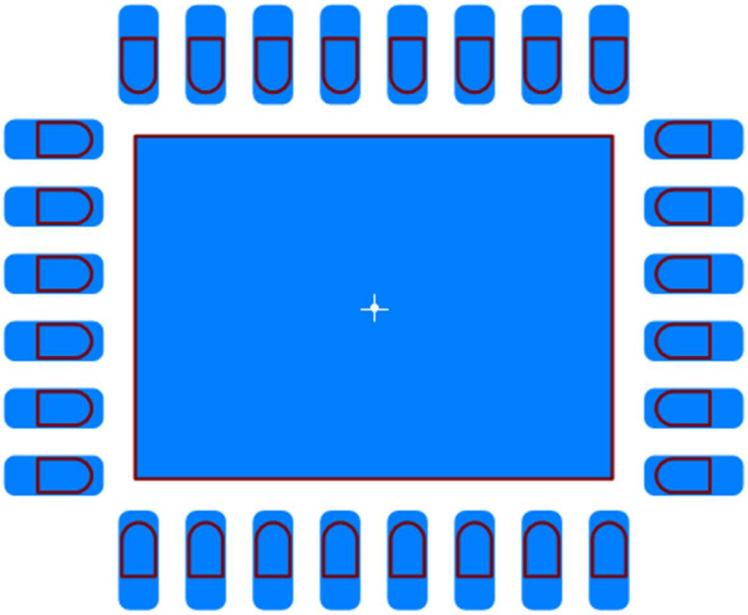
© 2025 PCB Libraries, Inc.

The drafting outlines are important in the following ways:

1. The Component Outline is crucial for accurate assembly placement and helps ensure your components fit properly on the board. It helps with verification during visual inspections, and facilitates repair, rework, and future updates to the design.

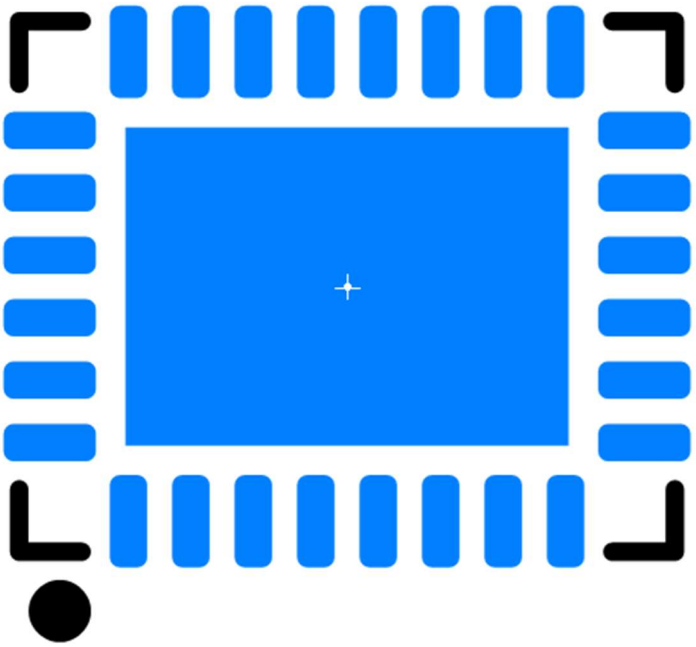


2. The Terminal Outlines are likewise critically important. They represent where on the metal terminal leads the physical package makes contact to the PCB. These are essential for reliable soldering and proper electrical connectivity. Each terminal lead outline must be entirely located on a pad. If the pad is not 100% on a metalized terminal outline, then the manufacturer's recommended pattern is not good, and you will get a bad solder connection.

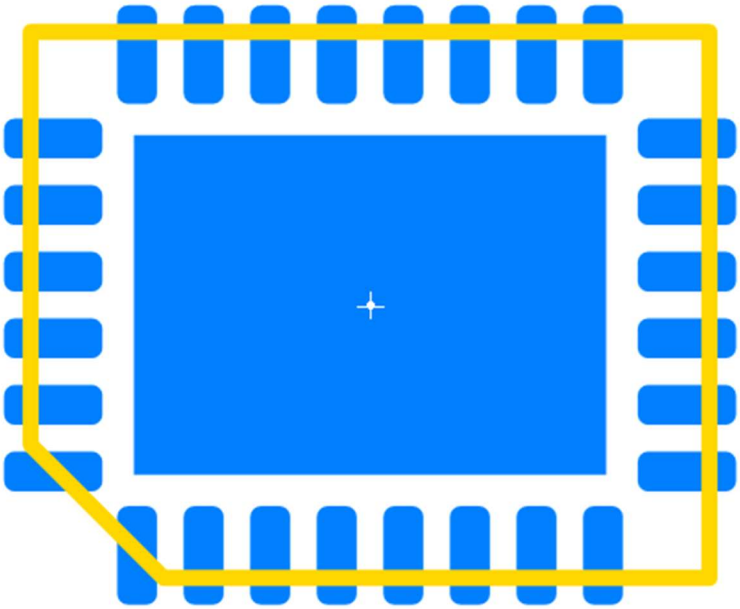


© 2025 PCB Libraries, Inc.

3. The Silkscreen Outline enables proper component orientation and should be created so that it's visible after assembly. Silkscreen outlines should not be placed under the package body – they are useless there and could potentially lift low profile or micro-miniature components off the PCB surface resulting in poor solder joints. The silkscreen data includes a post assembly inspection dot located by Pin 1, which helps with final assembly inspection.

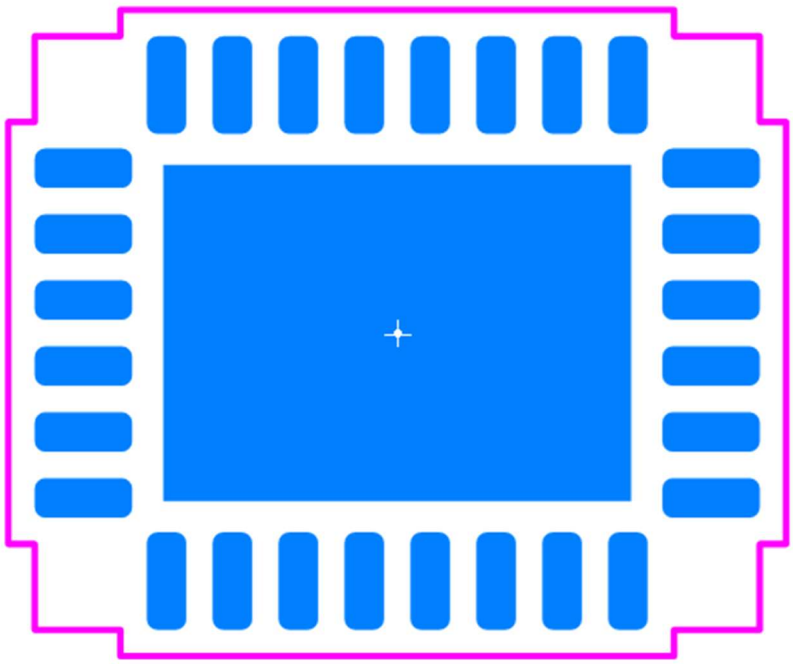


4. The Assembly Outline is a closed polygon. A chamfered corner is located by Pin 1 if the component is polarized or cannot be inverted during assembly. The assembly outline is important because it provides clear reference points for automated pick-and-place machines. A well-defined Assembly Outline also facilitates design verification and quality control.

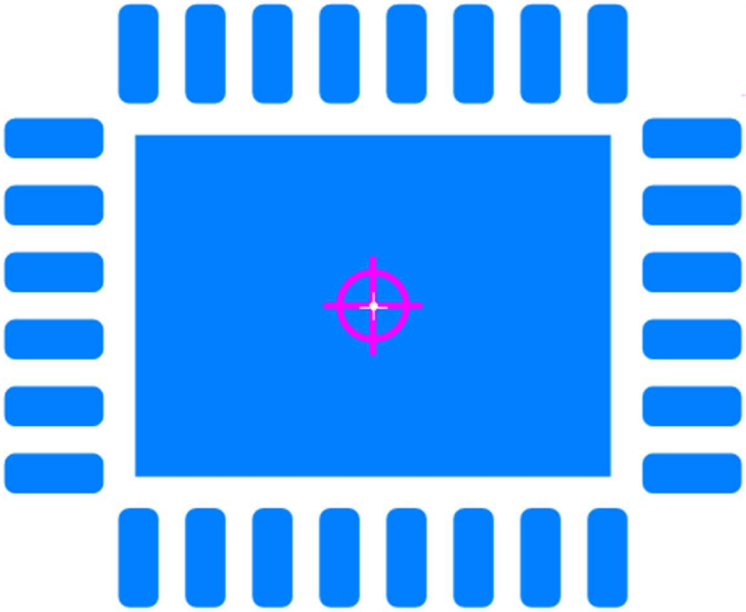


© 2025 PCB Libraries, Inc.

5. The Courtyard Outline is extremely important in defining the required minimum clearance area required for a component, ensuring there is no interference with neighboring components during assembly or operation. The ideal shape is contoured around the package body and pads, which is the default setting in the Footprint Expert. The courtyard excess value for Package Body and Pads can be set separately.

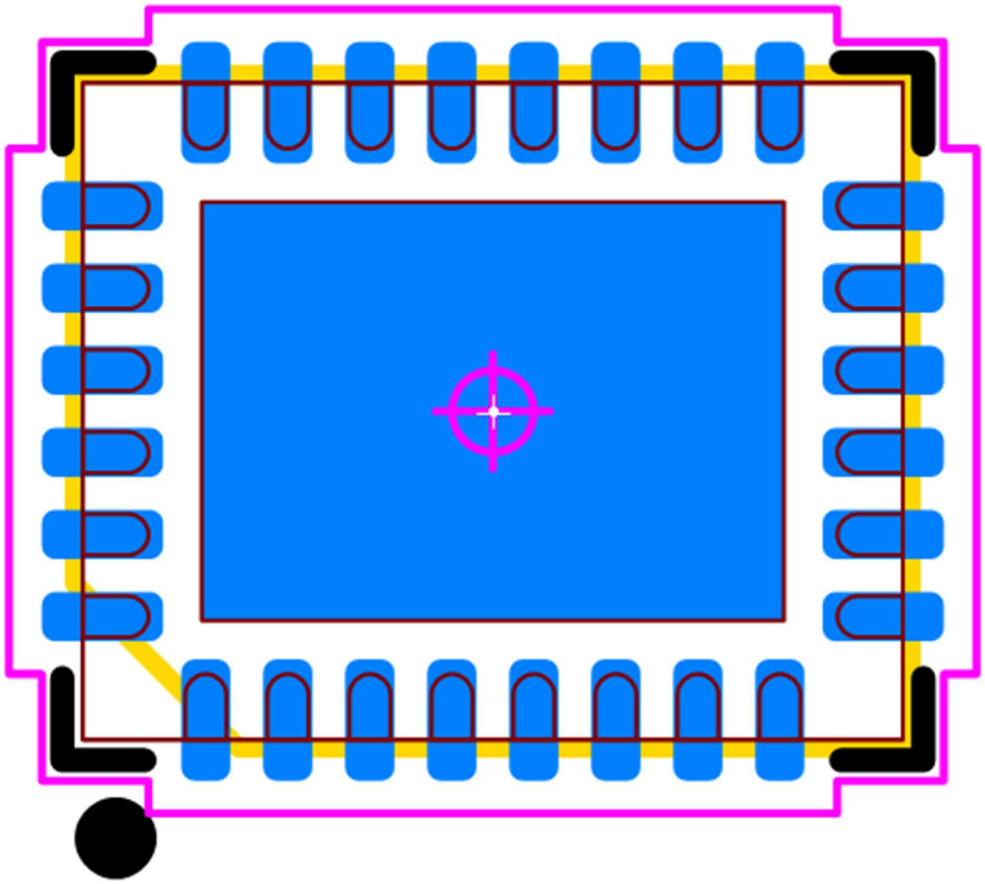


6. The Origin Outlines are placed on an independent layer. The size & outline width are user definable. Accuracy is essential for accurate alignment because it serves as a reference for positioning the component on the PCB. A precise origin simplifies machine programming and enhances design portability across tools.



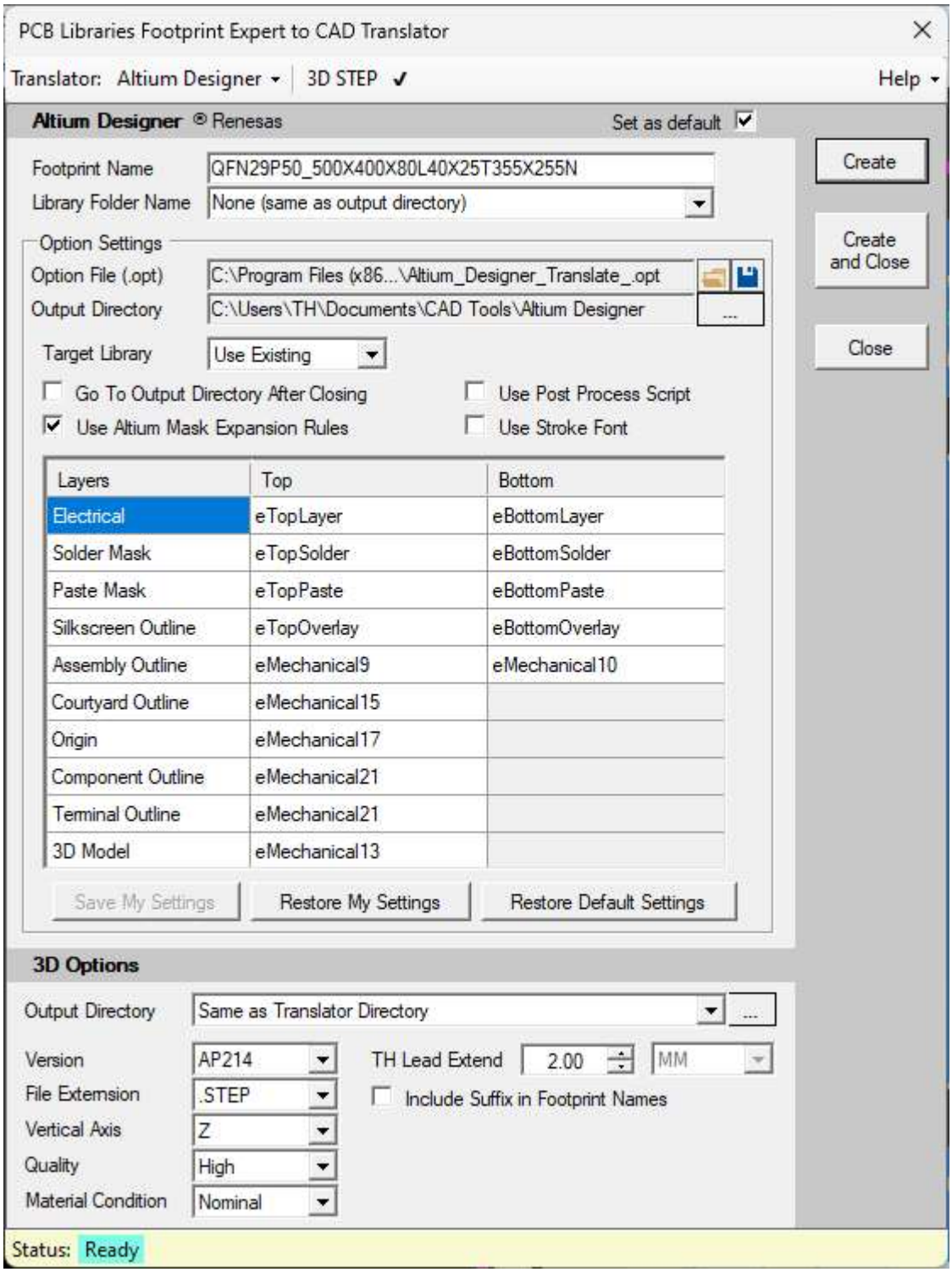
© 2025 PCB Libraries, Inc.

The Footprint Expert autogenerates all these drafting outlines from the same component dimensions used to calculate and generate the component footprint. It comes with pre-defined rules commonly used, which can be easily customized to your specific needs. The Footprint Expert allows you to modify this set of rules, then batch a new library in seconds with the modified rules!



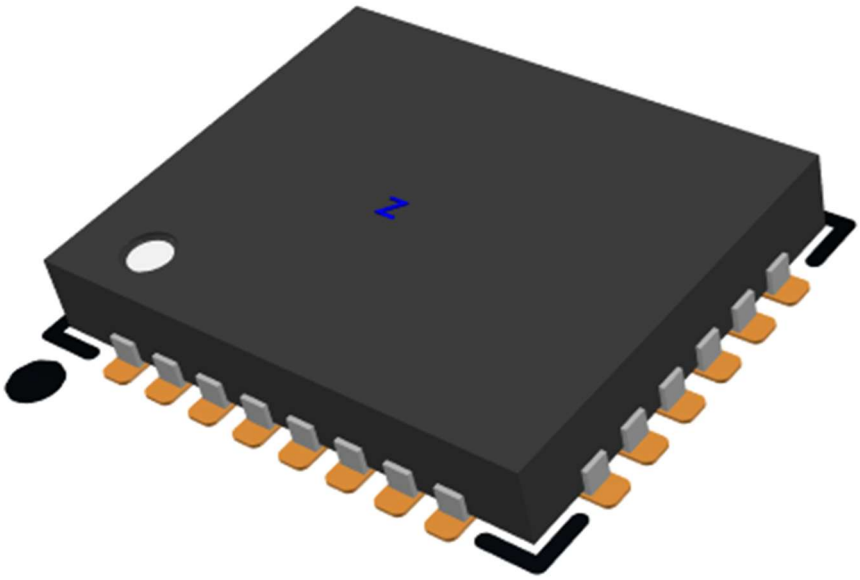
© 2025 PCB Libraries, Inc.

The CAD tool interface in Footprint Expert allows you to choose the layer assignment for each drafting outline. Here is a sample layer assignment for a CAD tool.



© 2025 PCB Libraries, Inc.

Using the package dimensions, Footprint Expert also creates a high-quality 3D STEP model.



Welcome to the future of PCB design with the PCB Footprint Expert software program - your ultimate solution for creating precise, error-free PCB footprint solder patterns in just minutes!

Create IPC or FED compliant footprints using package dimensions and your personal Options for pad stack rules, drafting outlines and solder joint goals.

- IEC_61188-7 - Zero Component Orientation
- IPC J-STD-001 - Solder joint goals for Toe, Heel and Side
- IPC-7352 - Mathematical model for pad stack calculation
- FED Vol 18 - Proportional Land Dimensioning Concept

Get the ultimate solution for creating precise, effortless, error-free PCB footprint solder patterns! Register on the PCB Libraries website and request your free evaluation license and web-based demo today!

www.PCBLibraries.com

© 2025 PCB Libraries, Inc.