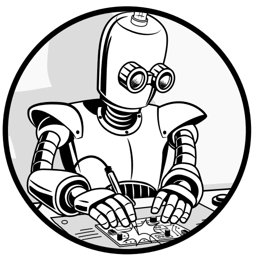
5

THE LAYERS OF A PCB



PCBs are more than just copper; they consist of multiple layers that each serve a different function. This chapter introduces several layer types, from stacked conductive copper sheets that create electrical connections to solder mask, silkscreen, mechanical features, and conceptual layers that organize information in the design software. You'll then use this knowledge to build on the exercise from Chapter 4, transforming your basic pad-only footprint into a comprehensive component definition that incorporates all the necessary layers.

In your PCB design software, the default perspective is looking straight down through the board from the top layer. You can view a single layer at a time, every layer, or a customized subset of layers. We'll discuss the different kinds of layers in a PCB design with that perspective in mind.

Signal Layers

Signal layers are sheets made of conductive material, typically copper, that allow you to solder components to the board and route connections between them. PCBs are sometimes called *printed wiring boards (PWBs)*, which gives you a better sense of signal layers' primary use: replacing the physical wires between components with pathways known as *traces*.

Signal layers are drawn with *positive polarity*, meaning anything shown on that layer in the design tool will be copper on the finished board (see Figure 5-1). Most PCBs are manufactured using a subtractive process for signal layers: The layer starts completely covered with copper, and then copper is removed from all areas except those specified in the design. Within the last decade, PCBs have increasingly been built with additive technology like 3D printing, which adds material only where it's needed and thus reduces waste.

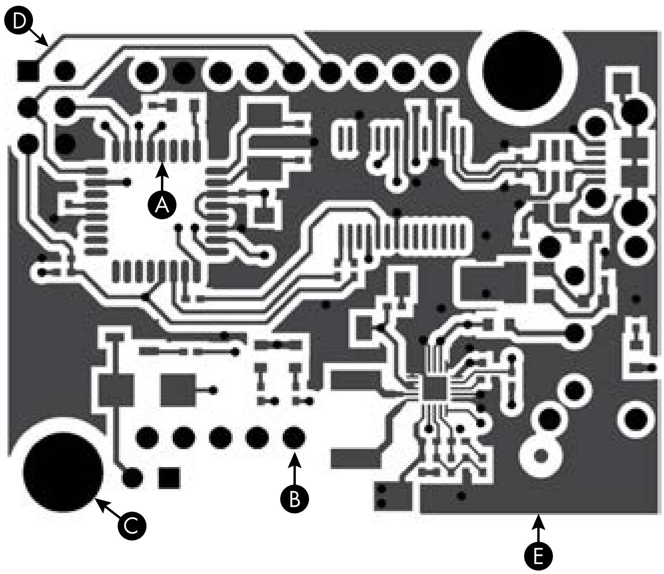


Figure 5-1: A signal layer, with copper areas shown in black

The signal layer shown in Figure 5-1 includes several typical features (which we'll revisit in Chapter 8):

- Pads for surface-mount (A) and through-hole (B) components. The pads of a component footprint are the exposed, conductive, solderable areas that will be used to attach pins to the PCB.
- Mounting holes (C). Most or all of the copper defining a hole will be removed during drilling, unless the hole is also plated (used to electrically tie the board to the chassis). We'll discuss this in more detail in Chapter 8.
- Traces connecting components together (D).
- *Polygons* (E), which are larger areas of copper that help distribute power and dissipate heat.

Signal layers normally come in pairs. Simple PCB designs use two or four signal layers. Extremely complex boards might use 20 or more. We'll discuss how to determine the appropriate number in Chapter 6. A PCB is often described by how many signal layers it contains (for example, a board with four signal layers is called a *four-layer board*), but this doesn't account for the many other layers the software uses to define it.

Solder Mask

Solder mask is a material that is commonly used to cover the outer signal layers of a PCB. It has several important properties. First, it gives the PCB its visible color. The most common color is green, but many others are available as well. Some companies use different solder mask colors to distinguish between different versions of a PCB: prototype versus production, normal versus extended temperature range, or assembled with leaded versus lead-free solder. Others use a color that matches the company's branding. Using a PCB manufacturer's default color will generally result in the lowest cost and fastest fabrication time, especially for small-quantity orders.

Second, solder mask is nonconductive, so covering two adjacent traces with solder mask won't short them together electrically. It should not be relied on as an insulator, however. Solder mask can wear off through friction, heat, or chemical or mechanical means. If you rest a PCB against a metal surface, you shouldn't assume the solder mask will prevent shorts on the board.

Third, solder mask protects the conductive copper on the PCB's outer layers. Over time, those conductive surfaces can oxidize, reducing their performance. Covering them with solder mask significantly slows this process.

Finally, the main function of solder mask is to prevent the flow of solder. Although solder mask doesn't act as a thermal insulator, placing it between solderable areas that shouldn't be electrically connected stops solder from creating a bridge that shorts those areas together. For example, imagine trying to solder the small-outline integrated circuit (SOIC) and resistor to the copper shown in Figure 5-2 without using solder mask.

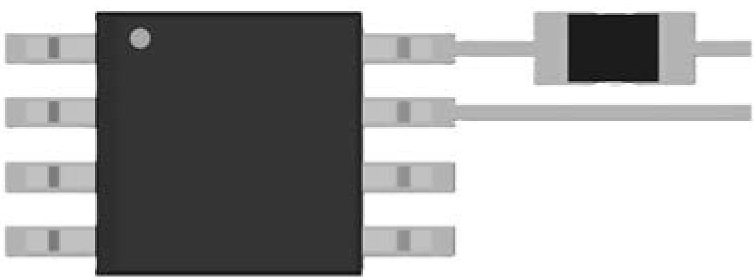


Figure 5-2: Soldering challenges without solder mask

Because the pins of the SOIC are so close together, it would be difficult to solder a single pin without getting solder on the adjacent ones. Solder would also flow away from the pins on the traces, risking shorts in other areas and requiring more solder to ensure solid connections. Adding solder mask, as shown in Figure 5-3, greatly simplifies the task by keeping solder where it should be.

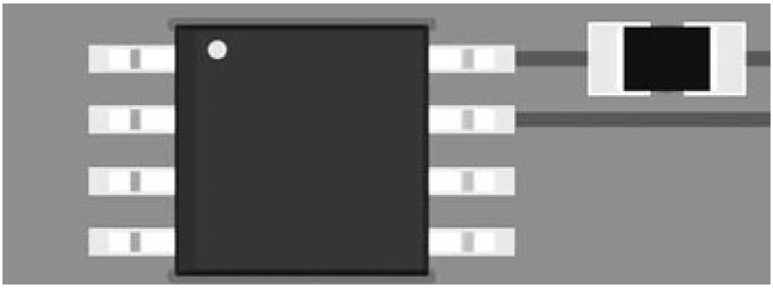


Figure 5-3: A circuit with solder mask

Openings in the solder mask define exposed conductors for components and test points. Since solder mask covers almost all of the PCB, this layer is drawn with *negative polarity*—that is, it shows where solder mask will *not* be placed. Most design tools automatically generate the solder mask layer based on rules set for an individual pad or the entire PCB project. The main rule sets the clearance between a copper feature, such as a pad, and the solder mask. Figure 5-4 shows the solder mask layer corresponding to the signal layer shown in Figure 5-1.

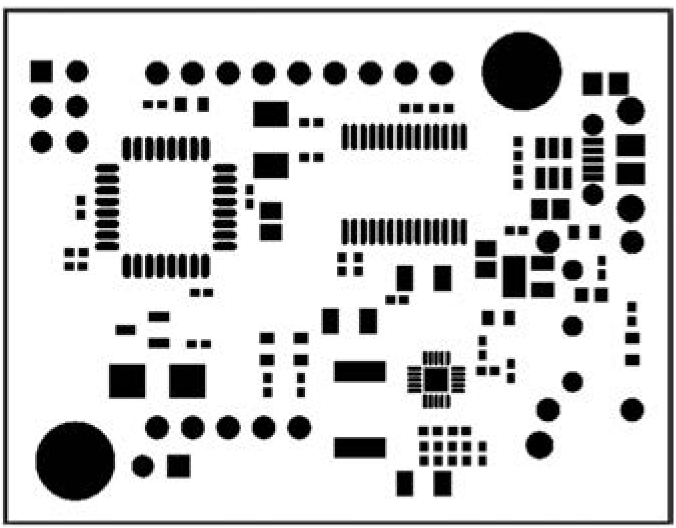


Figure 5-4: The top-layer solder mask for the circuit from Figure 5-1. The black areas won't have solder mask.

Solder mask material is often applied over the entire PCB in liquid form and then cured with ultraviolet (UV) light. A stencil based on the solder mask layer blocks the light from areas that shouldn't have solder mask so that they remain liquid. Once the UV cure is complete, the remaining solder mask is cleaned from the PCB.

When planning the solder mask layer, you must decide how to lay it out relative to the component footprint pads. You have two options, which are shown below the rectangular pad and connecting trace in Figure 5-5.

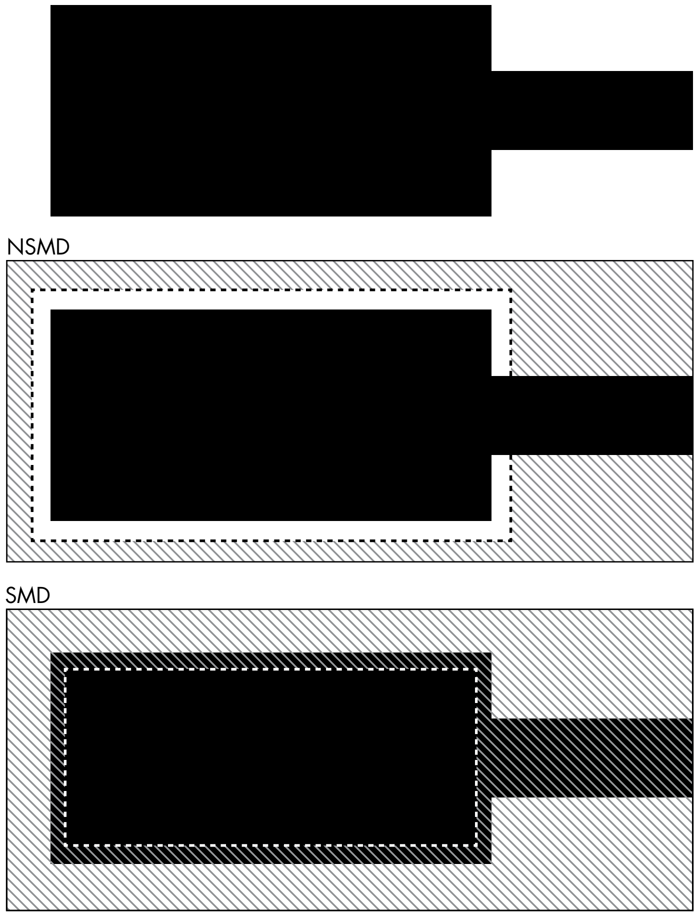


Figure 5-5: Defining component footprint pads using solder mask

In the middle image, the dashed black line defines the opening in the solder mask. The solderable area will be the entire copper surface of the rectangular pad plus a small portion of the trace. Since the solderable area is defined by the copper, not the solder mask, this approach is known as a *Non-Solder Mask Defined (NSMD)* pad. Solder pools on the exposed copper due to its high thermal conductivity, while avoiding the base laminate and areas covered in solder mask.

The solder mask opening at the dashed white line in the bottom image of Figure 5-5 indicates a *Solder Mask Defined (SMD)* pad, where the solder mask defines the amount of exposed copper. In this case, the mask overlaps the sides of the pad and covers the entire trace.

SMD pads are often used for very small components like 0201 and 01005 passives, as well as for BGAs, where you need tighter control over the solderable area. The overlap of solder mask on the copper can help hold those small pads in place. It also ensures that only the pad is exposed, not the connecting trace as well. NSMD pads, on the other hand, tend to make for stronger connections since solder covers the full surface of the pad and wraps around its edges. With this approach, you can make the pad a little smaller, since you don't have to allow for the overlapping solder mask, leaving you more room on the PCB.

Since the benefits of NSMD pads are more widely applicable, use them as a general rule and use SMD pads only if the component datasheet calls for it.

Silkscreen

Silkscreen, also called *legend* or *overlay*, marks the outer surface layers of a board with useful information. The silkscreen layer goes on top of the solder mask. Common silkscreen markings include reference designators from the schematic; polarity marks and pin 1 identification marks to ensure components are oriented correctly; and part numbers, bar codes, or corporate logos. Figure 5-6 shows the silkscreen corresponding to the signal layer shown in Figure 5-1.

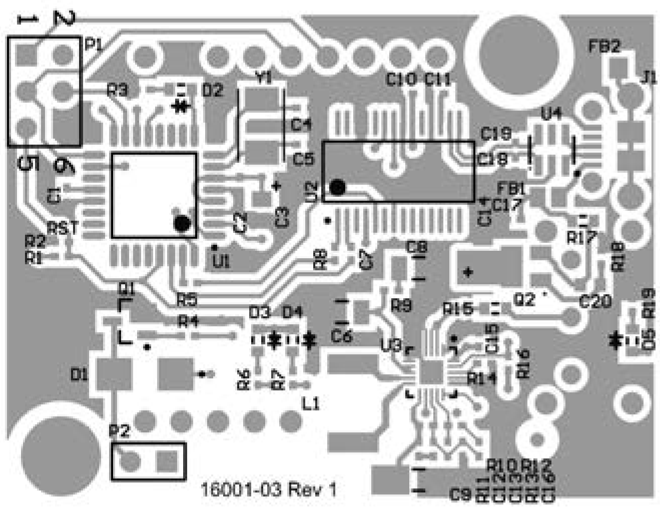


Figure 5-6: Silkscreen (black) for the top layer (gray)

Silkscreen can be used on both the top and bottom of the PCB and comes in a variety of colors. For the best legibility, choose a color that contrasts well with the color of the solder mask (white silkscreen is popular for this reason). You can also improve legibility by balancing the line width and letter height of any silkscreened text (see Figure 5-7).

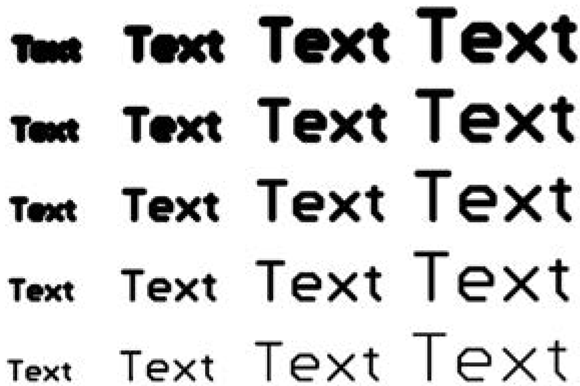


Figure 5-7: Silkscreen legibility at various text heights (40, 60, 80, and 100 mils) and line widths (8, 12, 16, 20, and 24 mils)

Choosing a line width that's too heavy can make the letters bleed together and the text hard to read, while too thin isn't as visually appealing.

Paste Mask

PCBs with surface-mount components are often assembled with *solder paste*, a toothpaste-like combination of powdered solder and flux paste, rather than spools of solid solder. Solder paste is applied to the PCB, the components are put into position, and the assembly is heated in a reflow oven. The components are initially held in place by the tacky paste and then by surface tension as it melts. As the temperature falls, the paste solidifies, leaving the components soldered to the PCB. We'll discuss how to use solder paste in Chapter 11.

Solder paste can be applied by hand or selectively placed by machine, but it's commonly applied through a stencil that has holes where the paste goes. The *paste mask layer* uses negative polarity to define the stencil holes. Figure 5-8 shows the paste mask corresponding to the signal layer from Figure 5-1.

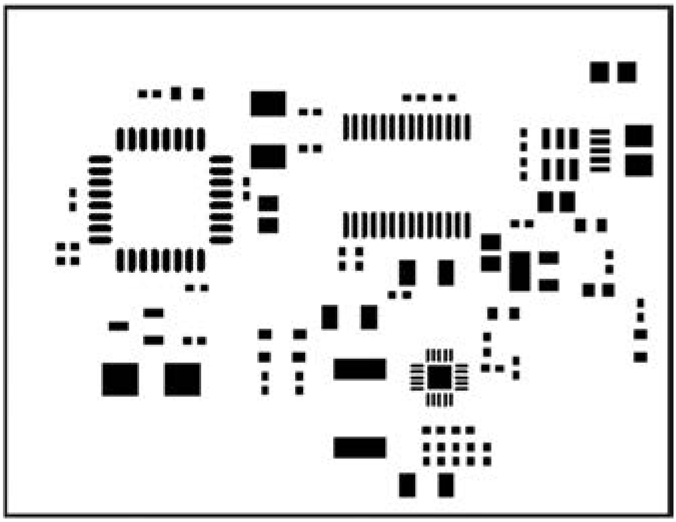


Figure 5-8: The top-layer paste mask for the circuit from Figure 5-1

The black areas define the holes in the solder paste stencil. Placing the stencil on top of the PCB and applying paste through the holes leaves solder paste only in the areas where it's needed.

The paste mask layer is an example of a design layer that isn't used directly in PCB fabrication. It's still part of the design, though, as it defines where surface-mount components will be soldered on the PCB.

Mechanical Layers

Most PCB design tools allow you to add design specifics or documentation using *mechanical layers*. For example, you could have layers dedicated to design dimensions; notes on the PCB fabrication or assembly; 3D models of components; or mechanical engineering input, such as drawings of the desired board shape (known as the *board outline*), connector locations, or mounting holes.

Mechanical layers often serve as a reference rather than part of the finished PCB. One exception is the board outline, which could be copied from another mechanical layer or created from scratch on a dedicated layer. Footprint libraries often inadvertently include information on the outline layer, but nothing should exist here other than the board shape. If you have a 3D model of a component on this layer, for example, the PCB manufacturer might misinterpret that information and try to cut a hole in the shape of the component.

Other Layer Types

Two final layers to discuss are the multilayer and keepout layers. Consider the pad of a through-hole component. If it's on a two-layer board, the pad needs to go on both the top and bottom layers. Rather than having to draw it on both layers, you can draw it once on the *multilayer layer* to apply it to all copper layers. If you switch to a four-layer board later, the design will automatically update to include the pad on all four.

The *keepout layer* also applies to all copper layers at once, but it defines areas where nothing can be placed. Say your design includes an inductor with high switching currents and you want to avoid coupling noise or inducing currents in nearby circuits. You could use the keepout layer of the inductor's footprint to define an area underneath the inductor that prohibits circuits or traces on any other layer of the board. If you need to keep an area clear only on a specific layer or layers, you instead use a *keepout designation* on the relevant layer(s). For example, if you need to keep the area in front of a connector clear for the mating connector to be attached, you would define a keepout only on the connector's layer.

Now that you've seen the various physical and conceptual layers that make up a PCB, let's revisit the component footprint from Chapter 4 to see how it might encompass them.

Features of a Complete Component Footprint

The quantity, size, shape, and relative positions of the pads are critical to ensuring a component will fit on the PCB, which is why they were the focus of the footprint design exercise in Chapter 4. There's more to a component footprint than just the pads, however. When you define pads on a signal layer, they'll generate information on the corresponding solder mask and paste mask layers, for example.

You can manage the settings for these layers globally, or you can customize them for individual components through their pad definitions. The main property to define for both the solder mask and paste mask is the *expansion*, or the distance around the pad on the relevant layer. For example, if you set the pad size to a 20 × 30-mil rectangle with a 2-mil solder mask expansion, the opening in the solder mask would be 24 × 34 mils. That would keep solder mask 2 mils away from the pad at all points. For SMD pads, use negative expansion values to make the solder mask opening smaller than the pad itself. Setting a paste mask expansion of –2 mils for the same 20 × 30 pad would make the paste mask opening 16 × 26 mils. For guidance on setting expansion values, consult the component datasheet.

Pin 1 Markings

Marking pin 1 or other important pins in a component footprint can help ensure components are placed properly during assembly. There are several ways to do this. The first is by using a unique pad shape, as you did for the custom footprint design in Chapter 4. Surface-mount components often use a rectangular pad for pin 1 and rounded pads for the other pins, while through-hole components typically use a square pad for pin 1 and circular pads for the rest (see Figure 5-9).

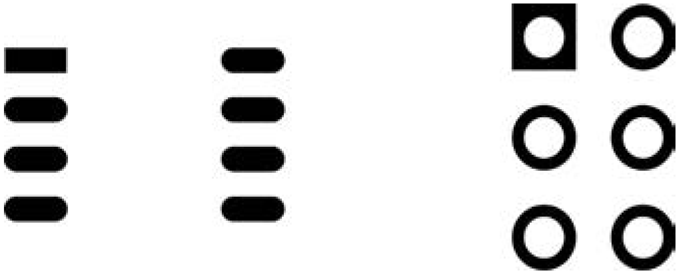


Figure 5-9: Unique pin 1 pads for a surface-mount (left) and a through-hole (right) component

Another option is to use silkscreen dots or other markings to identify pin 1 and a silkscreen outline to indicate the size of a component body (see Figure 5-10). These features are manually added to the footprint's silkscreen layer, at the discretion of the designer, and reduce the likelihood of manufacturing questions or assembly errors.

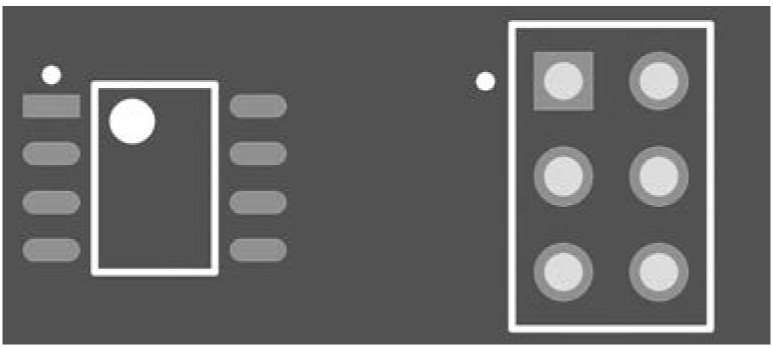


Figure 5-10: Marking part footprints with silkscreen

While silkscreen is useful for showing the 2D outline of the physical component, modern PCB design tools also allow you to add 3D models.

3D Models

Using 3D models of components helps ensure sufficient clearance between components and provides a more holistic view of the entire product, including the assembled PCB and external elements like the housing or mating connectors. Component 3D models are normally added to the PCB footprint. They can range from simple boxes indicating the maximum dimensions to more elaborate representations of the physical part, as seen in Figure 5-11.

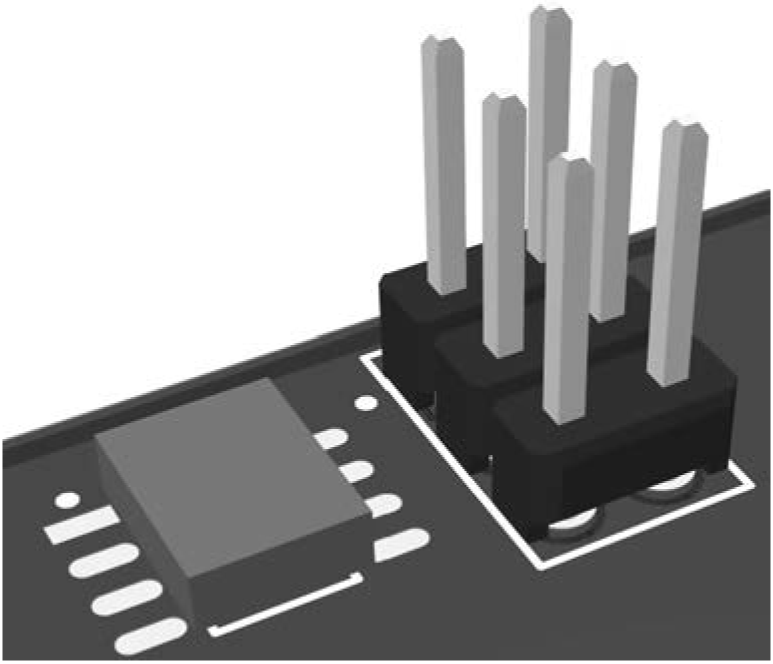


Figure 5-11: Modeling PCB components in 3D

You can build models for parts from scratch in the design tool, obtain them from the manufacturer, or find them on third-party websites such as 3D ContentCentral.

Keepout Designations

PCB footprints might include a keepout designation. A single-layer keepout might reserve space for a mating connector, as discussed previously, or prevent a component's conductive surface from shorting nearby signals. A keepout layer (which, as already noted, applies to all copper layers) is typically used to limit electromagnetic interference; for example, it might define a clearance area between a conductive surface and a Bluetooth antenna.

Other Features

Some footprints include additional information relevant to the part, perhaps on a mechanical or documentation layer. For example, an edge-mount connector may include a reference line on a mechanical layer indicating the edge of the board. Embedding that information in the footprint ensures that the part will be properly aligned. Not every footprint needs all of these features; consult the datasheet to determine which attributes to include.

Exercise 5: Complete Your Custom Footprint

This exercise expands on the custom footprint you started in Chapter 4, adding other layers of information. Let's assume the datasheet defines the following properties for the footprint:

- Pads are 1 mm wide and 0.25 mm tall.
- Pitch is 0.5 mm.
- Spacing between each column of pins is 3 mm.
- Solder mask expansion is 0.05 mm.
- Paste mask expansion is 0 mm.
- Component body is 2.5 × 2 mm and 1 mm tall.

You already used the first three properties to create the pads. Now you can update those pads to include the solder mask and paste mask expansion.

Define the Solder and Paste Mask

Enter the solder and paste mask values in the pad configuration dialog as shown in Figure 5-12.

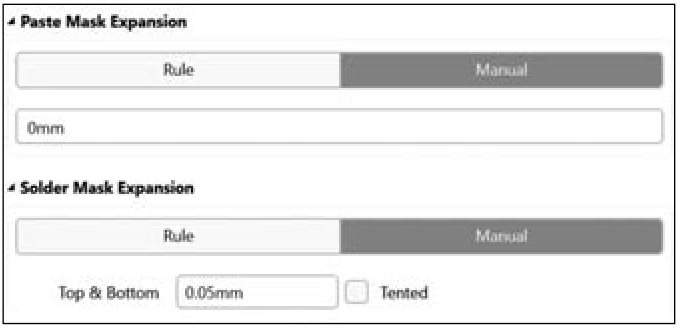


Figure 5-12: Pad configuration

The manual settings option allows you to set the expansion just for this component. Using the rule-based settings option instead would apply the project-wide expansion settings, which are defined as a design rule (covered in Chapter 9).

Add Silkscreen

You're using a unique pad shape to designate pin 1, but you could also use a silkscreened circle, arrow, or text to mark it or to convey other information. Generally, you don't need to add reference designators to the footprint explicitly; the design tool will add them automatically to the silkscreen layer when you place the footprint on the PCB. Make sure your silkscreen doesn't cover any copper.

Create a 3D Body

Including 3D models of your components on the footprint isn't required, but as noted earlier, they can help you better visualize your board and highlight any component clearance issues. Figure 5-13 shows two options: a basic box that represents the maximum component size and a more realistic model.

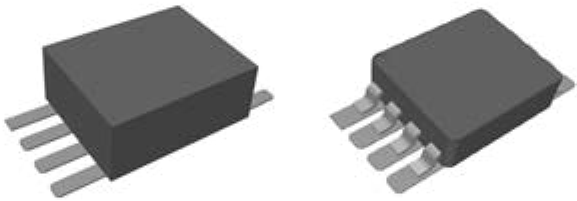


Figure 5-13: A rough 3D body created with simple extrusion (left) and a more detailed model (right)

Not all design tools have 3D capabilities. If yours does, consider adding 3D bodies to your footprints to make your finished design look more professional.

Define Keepouts

If your datasheet or your intended application calls for it, add any necessary keepout definitions to your footprint.

Summary

This chapter introduced several layer types used in PCB design, including signal layers, solder and paste mask layers, and silkscreen. In the exercise, you added the layers necessary to complete the custom footprint you started in Chapter 4. In the next chapter, you'll learn how to determine the optimal number and arrangement of layers to reduce manufacturing costs and headaches.