

The Design Process, Part 5

Building the Library: Drawing things in the library - Footprints

Sharon Hurlley-Durand
27 December 2021

The next step in building the part library is creating the Decal symbols for the parts you will be entering into your design. The Decal is also commonly referred to as the Footprint. The most important part of the footprint is the metal pads that the component will be soldered to. Their size and location must be appropriate to the leads of the component. Also for through hole components, the hole size must be large enough to allow the lead to easily pass through, but not so large that the component can fall out during the assembly process. Typically, I use a hole size that is 10 mils (0.254mm) larger than the widest part of the lead diameter (most leads are round but some are rectangular, these require a diagonal measurement).

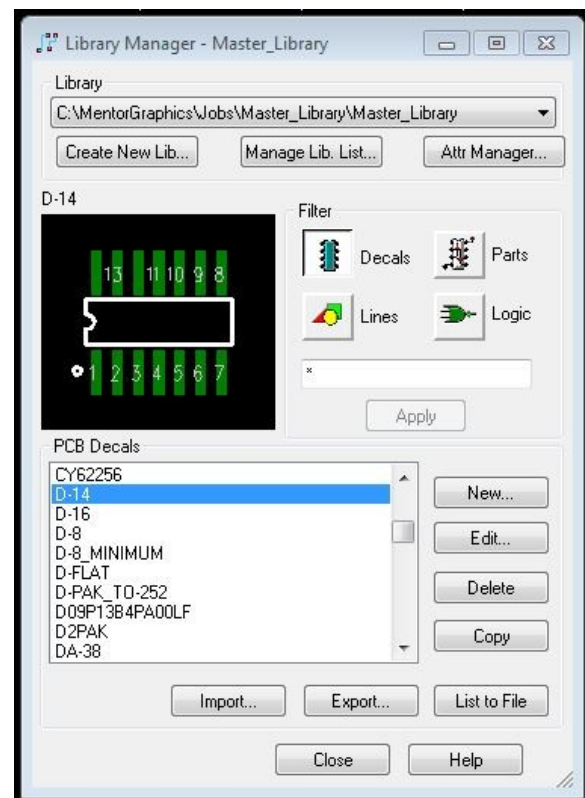
Most surface mount components have pads that are square to rectangular in shape, but there are a lot of odd shaped ones as well. For these types of components, the size and shape of the pad depends a lot on the kind of soldering process. This is where you have to be in communication with your customer and the fabrication house that they are planning to use for their recommendations. As most of our designs are prototypes and will be hand soldered by us in our facility, I have developed a system which works well for hand soldering, typically 20 mils (0.508mm) longer in the direction the soldering iron tip will be applied. I follow the recommended dimensions for the remaining sides of the surface mount pad. This allows adequate room to solder the lead to the pad without disturbing the alignment of the component on its pads. As these components are quite small and light weight, it is easy to dislodge them. And for multi-leaded components, it can be challenging to get them back in place.

To the right we see the Decal section of the Library, where the footprints are drawn. The decal consists of several different layers.

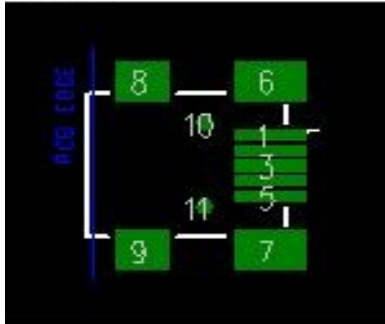
The pads for a through hole component typically have a Bottom Soldermask, a Bottom Pad, a generic Internal Pad, a Top Pad, and a Top Soldermask layer.

The pads for a surface mount component typically have a Top Pad (here shown in green), a Top Soldermask, and a Top Solderpaste.

Some surface mount components also have through hole pads which act as alignment peg locators, or attachments for secure mounting. This is found on a lot of connectors where something may be repeatedly unplugged and replugged. Also, heavy components such as



transformers often require secure attachments. If a part requires a heat sink attached to it, more holes may also be needed.



To the left is an example of a surface mount connector that has both surface mount and through hole pins (the electrical and non-electrical connections to the board). Here Pins 1 through 5 are the surface mount electrical connections to the circuit. Pins 6 through 9 are the surface mount mounting pads which add strength for a secure attachment. And Pins 10 and 11 are non-plated through holes that serve as alignment holes for pegs on the bottom of the component.

The dark blue line and text (which is hard to see in this drawing) reads "PCB EDGE" and it shows where to place the footprint with reference to the board edge so that it is in the correct position for the plug to attach to the connector. Plugs often have bulky housings that require specific clearance distances to possible obstacles. This blue line and text are placed on an internal layer that is not used in the fabrication process and is there only for the layout designer to use. The pin numbers 1 through 11 are also not visible in the final design. They are there for reference during the construction of the Decal.

In addition to the pad layers, decals normally have a Top Silkscreen layer (shown in white in these examples). This usually shows the Outline of the component along with some indicator for Pin 1 to insure that the component is placed in the correct direction. In the example of the D-14 part shown previously, a small circle indicates the location of Pin 1. And in the connector there is a small white number 1 to indicate the location of Pin 1. The small notch in the rectangle also indicates which end of the part has Pin 1, but often the circle or some other mark reinforces the location as often when the part is in place it may cover up the notch. The circle then tells the engineer the location when clarification is needed in the De-bugging process.

There are also Top and Bottom Assembly Drawing layers which duplicate the decal drawings and are used by the assemblers for verification of parts locations and orientations. Other text and drawings may be included on these layers to clarify unusual circumstances. A Bill of Materials generated by the Schematic half of the software will accompany these Assembly Drawings. It lists all of the components associated by their Reference Designators.

One last layer contains all of the information for the Fabrication Drawing and is used as a blueprint for the fabrication of the board. It shows the dimensions of the board outline and the location and sizes of all of the holes along with any necessary information needed to construct the board.