

Designing a PCB Using CAD

Sharon Hurlley-Durand

6 November 2021

There are several CAD programs available to use to design Printed Circuit Boards. Some are very simplistic and others are more complicated and refined. I have been using PADS 9.0 (the 2009 revision) by Mentor Graphics Corporation for the past several years. We originally chose PADS because it was very popular with the fabrication houses (the production facilities that turn your design into a physical product), and because of that, in theory, there should be fewer problems with the manufacturing process. This has turned out to be true and we have since had good results.

As with all software programs there are pluses and minuses, things that should have been done better or completely left out. There are a few things that I dislike about this program, but the parts that work well do get the job done. We stopped upgrading the revisions of the software in 2009 as the parts they were refining were aspects that we never used and we didn't need to have unnecessary business expenses.

One of the aspects that troubles me is that the programmers who wrote this software obviously do not know what reference designators do. And this begs the question of what else do they not understand about the design process. In the past, I tried to explain this reference designator problem to customer support, but the issue was never fixed.

When you construct a library part, a reference designator is assigned to it. If you tell the software that the component is a connector, it assigns the designator "J" or sometimes "P" to the component. This indicates the function of the component. As in the case of a connector, it functions as a device that connects the particular part of the circuit to which it is wired to another off board circuit. This may be as simple as connecting to a power cord to a wall outlet, or to a connector on another board that has a different circuit altogether. Inside of a complex electronic device, such as a computer, there may be several PCB's that serve different functions in a more intricate design.

There are more than one standard for assigning reference designators and they have been modified with usage. I have always used the American National Standards Institute (ANSI) version. But, if you are working for a company or a client, they may have their own usage preference.

A number is added to the reference designator when the component is added to the design to indicate how many of these types of component exist in the design. This serves as an address that aids in knowing where things are located and differentiates one part from another in the circuit. A complex design may have several hundred components of a particular type.

The problem with this software is that it often chooses one of the possible package types instead of the component type, especially when using integrated circuits (otherwise referred to as IC). Integrated circuits are complex components that have many different functions within their package and come in many different types of packages from through hole (original

technology requiring holes in the board for connection) to surface mount (SMT or SMD) to ball grid array (BGA which is a more modern form of surface mount).

The correct reference designator for an IC is "U", but the software usually defaults to "BGA" (package type). It can be manually reset to the correct designator, but if the software is updated to the next revision, all of the settings are reset to the manufacturer's original. It's irritating. It would be like automatically referring to all cars as "LADA".

But, with all of the issues with different design software packages, it is by far much preferred to the old method of doing layout by hand. The process is much faster and the result is more or less permanent and much more accurate than laying down strips of tape and decals on a transparent surface that is later photographed to be used in creating the stencils used by the fabrication houses.

