

Three popular PCB Design Software

Which software is suitable for beginners? What software can greatly help PCB designers? There are many kinds of PCB design software in the market, including paid software and free software. China PCB manufacturer introduce three kinds of software with high popularity rate at present: PADS, Altium Designer (AD), and Cadence allegro. What are their advantages and disadvantages? Let's take a look below.

1. PADS

More people use this software, which is user-friendly, simple, and easy to use. Relatively speaking, it is more convenient and easy to operate than Protel. This software is more suitable for low-end design, which can be called the king in this level. Its characteristic is it's easy to interact with schematic, unlike Allegro, which sometimes fails to connect the schematic and PCB. In addition, it also supports real-time modification of the network, which is suitable for little changes or simple board wiring.

2. Altium Designer (AD)

For most PCB engineers, the first design software they used is AD. As the basic and entry-level hardware design software, AD is suitable for drawing up the simple single- and double-sided boards and four- and six-layer boards. Through the perfect combination of various technologies including schematic design, circuit simulation, PCB drawing, and signal integrity analysis, engineers can easily conduct the PCB design. If engineers can be proficient in this software, the quality and efficiency of the circuit design will be greatly improved.

3. Cadence allegro

The biggest advantage of this software is that it is very convenient to repair the wiring, and tends to be intelligent. It can display DRC in real time. In addition, line sticking, wiring and winding are also very convenient. Nowadays, the advantages of teamwork are more and more obvious, and Allegro has more advantages in multi-person cooperation than other PCB software.

PCB learning is a long process, and each software also has its own characteristics and advantages. PCB engineers should choose the right software according to their own design needs, so that PCB design will be more efficient and convenient.