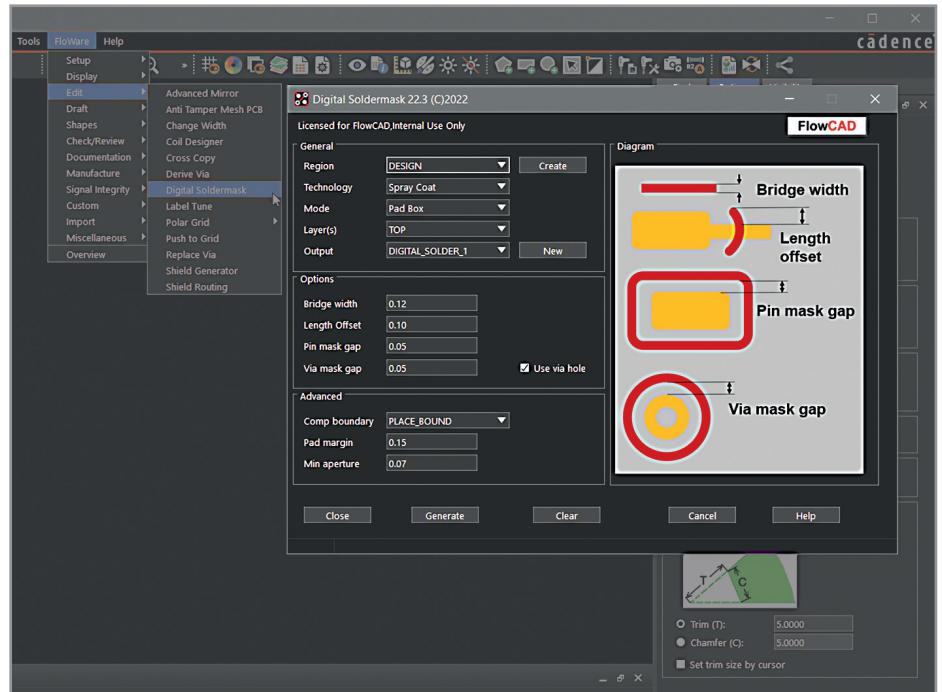


Digital Solder Mask



Minimum land widths, minimum distances and offsets can easily be generated

Digital Solder Mask in OrCAD and Allegro

A new generation of inkjet printers enables the selective application of solder resists during circuit board production. This results in new areas of application and a changed process in the design flow. In addition to material savings, this process also opens up new, creative and technically different solutions. These are of particular interest in the area of high voltage and power electronics, for example for e-mobility.

The solder resist (also called solder mask or resist) is now mostly applied in a pouring or spraying process or over the entire surface by screen printing. To do this, the outer layers of the printed circuit board are completely coated with photosensitive paint and pre-dried. In subsequent process steps, the resist is exposed and developed. In order to keep the relevant areas free of solder resist during the manufacturing process, a special mask is output by the EDA software.

In further steps during assembly production, soldering paste is applied to the exposed areas, components are placed and then the solder melts in the reflow oven and the components are soldered. The solder resist prevents the solder from getting to varnished areas when soldering. Solder resist was origi-

nally developed for wave soldering systems to avoid solder bridges and reduce solder consumption. Solder resist has other functions that play a role in a printed circuit board: It protects the copper conductor tracks from corrosion and protects them from mechanical damage. Furthermore, solder mask improves electrical properties such as dielectric strength and serves as a resist for the final surface (e.g. ENIG) that is applied at the end of the process. In addition, however, it also influences the impedance of conductor tracks.

Inkjet Technology

Similar to an inkjet printer, an inkjet print head can be used to spray paint instead of ink and thus apply it specifically to the circuit board. Research in this area has increased since around 2017. The research results are now ready for the market and are being used in the first commercial projects.

Material Savings

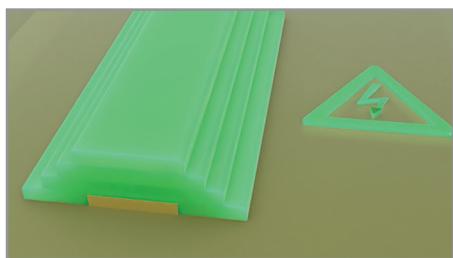
The topic of sustainability is also becoming increasingly important in electronics. With the pouring and spraying process, the entire surface of the printed circuit board is covered with the colored paint. The soldering areas

are then freed from solder resist again. In the screen printing process, however, there is also the possibility of leaving free the areas on which solder paste is then applied. With this non-full-surface application of the lacquer, the registration, i.e. the precise superimposition of the copper conductor pattern and the lacquer, is significantly worse than in the photo-technical process, so that soldering areas have to be saved to a much larger extent.

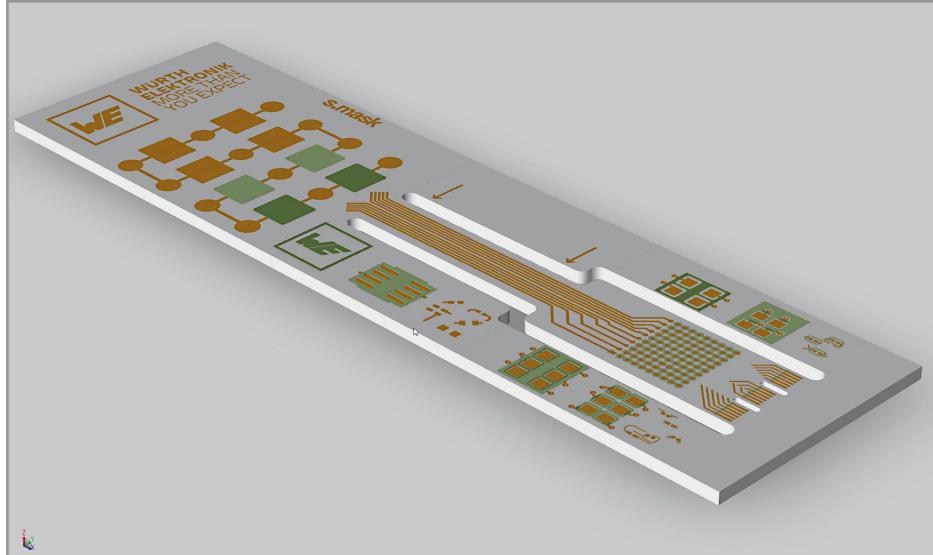
Since most paints are green, printed circuit boards are generally known as green-colored circuit carriers, although the base material (FR-4) is actually yellowish. The appearance of the stop function can be reconsidered by the targeted application of solder mask. To prevent the solder from running, a small dam is sufficient as a boundary to the connection PAD. The rest of the line does not have to be covered with a reflow soldering process. This is suitable, for example, for embedding technology (components within the printed circuit board) in the soldering process. Since the assembled inner layers are pressed further in further process steps after the components have been soldered, a surface with good adhesion is required, which should be free of solder resist. If the entire inner layer were coated with solder mask, there could be areas where the layers are subject to delamination.

High Voltage

The epoxy resin solder mask also has insulating properties. Since it has only a small thickness during the screen printing process, it does help with dielectric strength, but has not yet been considered an insulator. However, if you apply several layers of paint with an inkjet printer, the thickness becomes so great that the insulating effect is significantly increased. The overlapping of the conductor track edges, which is always very small due to the process technology, can be selectively and partially overprinted and thus reinforced. This opens up new possibilities for insulation in the area of high voltage and power electronics. This is of particular interest in the area of e-mobility, since high voltages are often laid on printed circuit boards.



Protection against high voltage

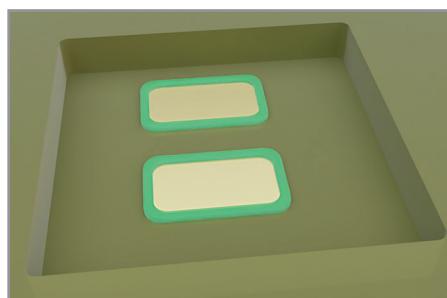


PCB to demonstrate the technology by Würth Elektronik

Cavities and Rigid-Flexible Assembly

If components are to be assembled on different levels, then there is not only one top layer on which components are assembled, but it can happen in areas of the circuit board that the assembly level is different.

In such a case, the application of solder mask using the screen printing process is usually impossible. With an inkjet printer that can print in depth, areas in cavities (indentations in the FR-4 material) can be easily coated with varnish. The same also applies to flexible areas that are brought out of the printed circuit board as inner layers and on which components are to be attached.



Solder mask on inner layer in cavity

Different Serial Number or Barcode on Each Board

As with digitization in the print area on paper, individual data can even be printed with solder resist for each panel or each individual circuit board. This data no longer comes from the PCB software such as OrCAD and Allegro, but is supplemented in the

CAM at the circuit board manufacturer in defined areas.

EDA Integration

An integration in the EDA tool is very important, because only the PCB designer knows which areas in the PCB can be freed and where the varnish is required to stop the solder or to protect the circuit board from environmental influences or high voltages. Furthermore, the manual effort or the additional documentation for the designer to transmit the required information to the CAM at the circuit board manufacturer without errors is reduced to almost zero, which saves costs and communication.

The subsequent cumbersome and error-prone reworking of the data sets in the CAM is therefore no longer necessary. A whole range of new functions for creating digital masks can be used in OrCAD and Allegro PCB Tools to develop specifications for digital printing. The necessary functions in the software were defined by FlowCAD together with Würth Elektronik. It is possible to define all currently known applications in the EDA tool and to output the corresponding production documents. The masks for the digital solder resist can be automatically generated with the corresponding rules of the circuit board manufacturer for minimum land widths, minimum distances and offsets. If generation is not possible, there are corresponding error messages. Digital solder resist masks using inkjet printing enable new technical solutions and are attractively priced.

More Information:

T +49 89 45637-770

www.FlowCAD.com