EAGLE CIRCUITS

APPLICATION NOTES

Filled Conductive Vias

Designers, fabricators, assemblers and material suppliers are always working on new methods of increasing the density of PCBs, while reducing cost, especially with technology driving electronic devices to become smaller and faster. More efficient use of "real estate" needs to be achieved since plated through holes and lands take up a large fraction of the available surface area in many of today's designs. For example, many PCBs with BGA substrates have a "dog bone" design with vias connected to pads for accepting solder balls (see Figure1).

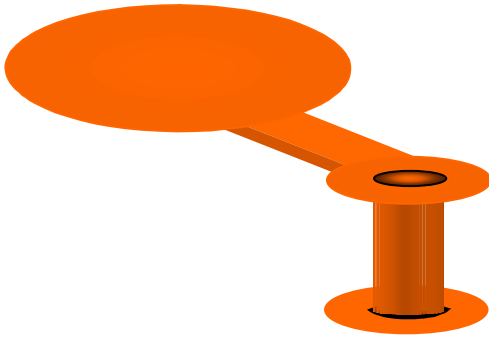

Figure 1. Typical "dog bone" pattern for BGA

Via-in-pad and buried via constructions have become a reliable and cost effective way of overcoming this problem. Filled vias with thermally cured epoxies or liquid photoimageable soldermasks are prone to shrinkage and out-gassing. This can occur at curing or during assembly and can lead to mechanical failures of the vias.

Likewise, simply having a via-in-pad design does not provide a solution because a "capillary effect" draws solder away from the joint and into the via. This leads to poor solder joints and this affects long term reliability. However, via-in-pads filled with conductive paste are proving to be a widely accepted, relaible, and cost effective solution.

Plating the conductive paste, leaves a coplanar surface. This surface is solderable and has high electrical and thermal conductivity. This gives it a distinct advantage over other filling materials.

Eagle Circuits currently uses Dupont's CB100 Silver conductive paste. It consists of a one part silver/copper/epoxy. The typical characteristics of the conductive plug are shown in Table 1. (Pinted Circuit Fabrication - November 1997).

Eagle Circuits uses the following process to fabricate boards with filled conductive vias. Multilayer panels are initially laminated and then drilled. The panels are processed through a permanganate desmear line and then through an electroless copper line. The panels are then electroplated with copper so that 0.4 - 0.5mils of copper is plated on the surface and inside the holes. These panels are then laminated with dry film resist, exposing the vias to be filled. The conductive silver paste is then screened into the vias. The dry film resist is removed and the surface is scrubbed to achieve a coplanar finish. Dry film resist is applied once again and the finished circuit image is exposed. This exposed image is then electroplated with copper to enhance the electrical contact of the plug to the circuitry and to achieve better solderability. The panels can now be fabricated using the standard manufacturing process. An example of the via-in-pad filled with conductive silver epoxy is shown in Figure 2.

No additional drill files or Gerber data need to be generated in order to manufacture PCBs with via-in-pad filled with conductive epoxy. We recommend placing a note in the manufacturing

specification, identifying the pads that need to be filled.

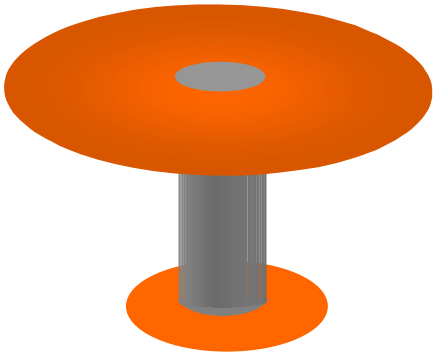

Figure 2. Typical via-in-pad filled with conductive epoxy

For PCBs requiring buried vias, we would require an additional drill file for the vias to be filled. Again, we recommend placing a note in the manufacturing specification requesting buried and filled vias

The conductive paste has become widely used and accepted in industry for applications such as via-in-pad and buried vias. Further information can be obtained from sales@eagle-circuits.com

Table 1. Typical characteristics of the filled conductive via plug

Test	Typical Result
Volume Resistivity	0.00016cm
Plug Resistivity (0.013" of Ø, 0.025" thick substrate)	<10mΩ per plug
Thermal Conductivity	>5.2 Watt/m°K
CTE – linear	<35 PPM/°C
Thermal Cycle(-65°C to +125°C)	No degradation after 1000 cycles
Thermal Sress10s in Sn/Pb to at 228°C)	No degradation after 5 dips
Paste Adhesion 0.055" of Ä, 0.055" thick substrate)	7-10 lbs. on BT resin and FR4