May 2010

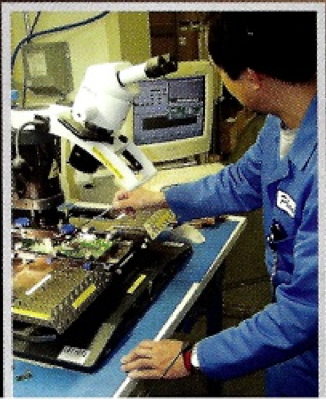
CircuiTree

First in Everything PCB

Fantastic Finishes

ENEPIG: A Final Finish That's Time Has Come

IN THIS ISSUE

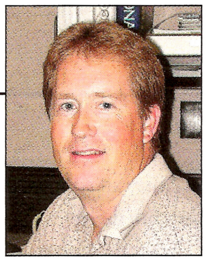


- PCB Prototype Manufacturing
- Does America Need Manufacturing?

A bnp media PUBLICATION

circuitree.com

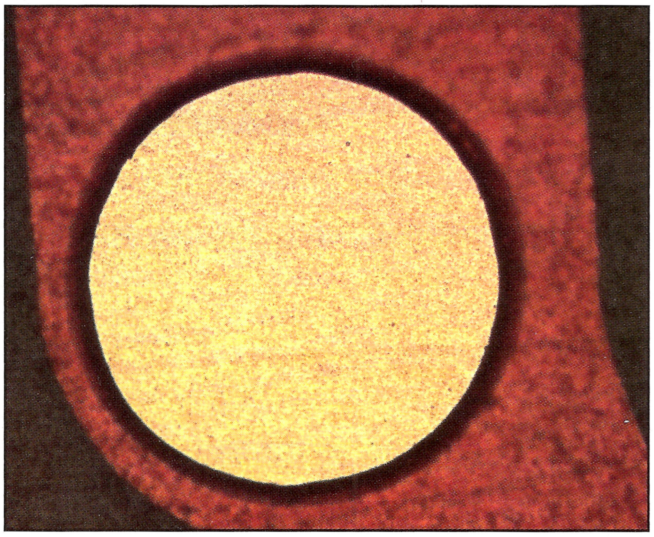
By Mark Finstad



No Mo' Flo'?

Q: ***I recently received some flex circuits and all the pads have a lot of adhesive squeeze out around the cover openings which significantly reduces the area I have to solder. The manufacturer tells me this is normal. I don't have room to increase my pad size, so is there any way to decrease the amount of adhesive squeeze out?***

A: I guess that I would first need to know your definition of "a lot" of squeeze out. Your flex vendor is correct when he told you that adhesive squeeze out is normal. However, the amount of squeeze out can vary "a lot" from one flex circuit to another, and is determine by many different factors. The single biggest factor is the amount of adhesive used to bond the cover to the circuit. Naturally, the thicker the adhesive, the more flow you will get on your solder pads. The manufacturers are always doing a balancing act to try to provide enough adhesive to ensure a good bond and to fill between all conductors, but at the same time keep the squeeze out to a minimum. Most manufactures will use about 1 mil of adhesive per ounce of copper as a general rule of thumb.

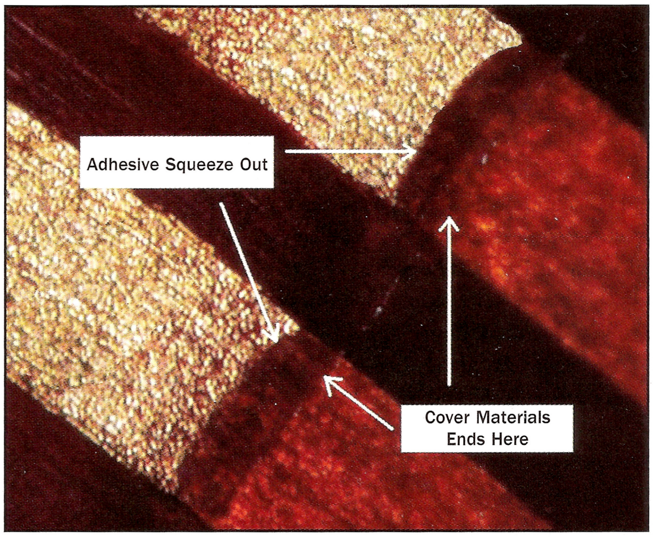


The cover opening on this pad was laser ablated so there is no adhesive squeeze out

However, there could be other factors that drive your manufacturer to a thicker adhesive, which in turn, will result in more squeeze out. I would first ask your flex manufacturer if they could reduce the cover adhesive thickness without adversely affecting the lamination process. If they cannot reduce the adhesive thickness, they may be able to adjust the lamination parameters to reduce the amount of adhesive flow on the solder pads. The type and thickness of the conformal lamination pads can also have a major effect on the adhesive flow. Some lamination materials inhibit adhesive flow more than others, so there may be some adjustments that the manufacturer can make to limit how much adhesive ends up on the pads.

There are cases, especially where you have heavy copper traces, where the manufacturer has to use a lot of adhesive to fully encapsulate and fill between conductors. In cases such as this, there can be a lot of flow, and it can be difficult to reduce it and still get good lamination results. There are some design tricks that you might be able to use that can help. If your flex circuit is two or more layers with plated through holes, size your cover opening to be slightly larger than the solder pad. The pad itself will act as a dam for the adhesive everywhere except the spot where the trace comes in. You may still get some flow in that area, but 270 degrees of the pad should be virtually free of adhesive flow.

Solid planes also tend to aggravate the flow problem because there are no conductors to fill between. Make sure that all of the solderable areas in a plane have thermal pads, and then slightly oversize the cover openings over those pads. There will still be some flow where each "spoke" comes in to the thermal pad, but the rest of the pad should be clean. Finally, you can always have the cover access openings laser skived. Modern lasers can effectively remove all cover and adhesive material over a pad without damaging the pad. The benefits of this method is that you can eliminate the cover drilling step, the cover registration will be nearly perfect (the laser will target etched fiducials), and there will be zero adhesive flow. The downside of laser skiving is time and cost. If you have hundreds or thousands of pads per circuit, laser skiving is not a good option, but if you have a relatively small number of pads, the laser would be your best bet.



This cover opening was punched and then the cover was laminated resulting in the adhesive squeeze out around the perimeter of the opening

The Flexperts are Mark Finstad and Mark Verbrugge