

# GSPCB ENGINEER CONFIRM

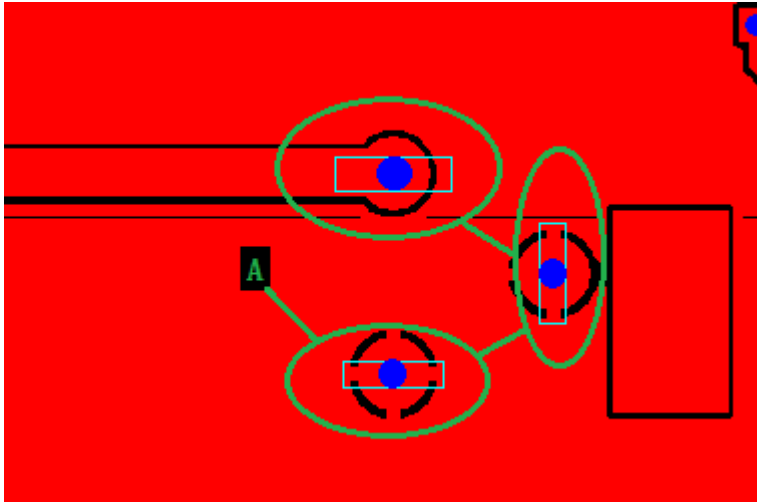
June 23, 2014

## PCB Engineering Question of P/N: blitz\_rev\_d

1. Customer supplied a milling layer, see below A point in figure1 shown, but the slots' size is bigger than the PAD size, and we suggest to ignore this three slots and follow the drill holes to built, please confirm.

Remark: the pro. Gerber file has followed the suggestion to do.

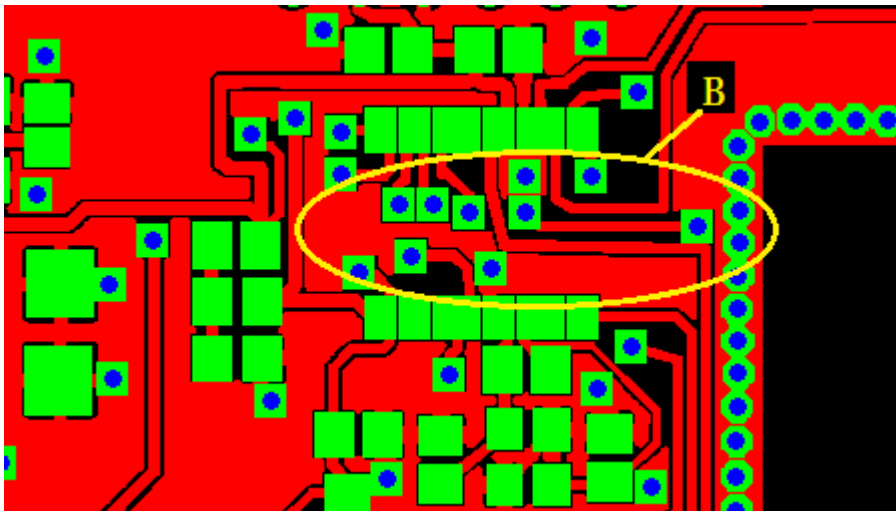
Customer reply:



2. See below B point in figure shown, there are solder mask opening for all via holes, we suggest the solder mask follow MASTER GERBER to built, please confirm.

Remark: the pro. Gerber file has followed the suggestion to do.

Customer reply:



## *GSPCB ENGINEER CONFIRM*

3. See below figure shown, blue is outline and four non-plated holes on broken TAB, and green is paster part for stencil, please confirm if it's ok.

Remark: the pro. Gerber file has followed the suggestion to do.

Customer reply:

