An investigation into laying out a Customer PCB for double sided attach of SFF Plugable modules.

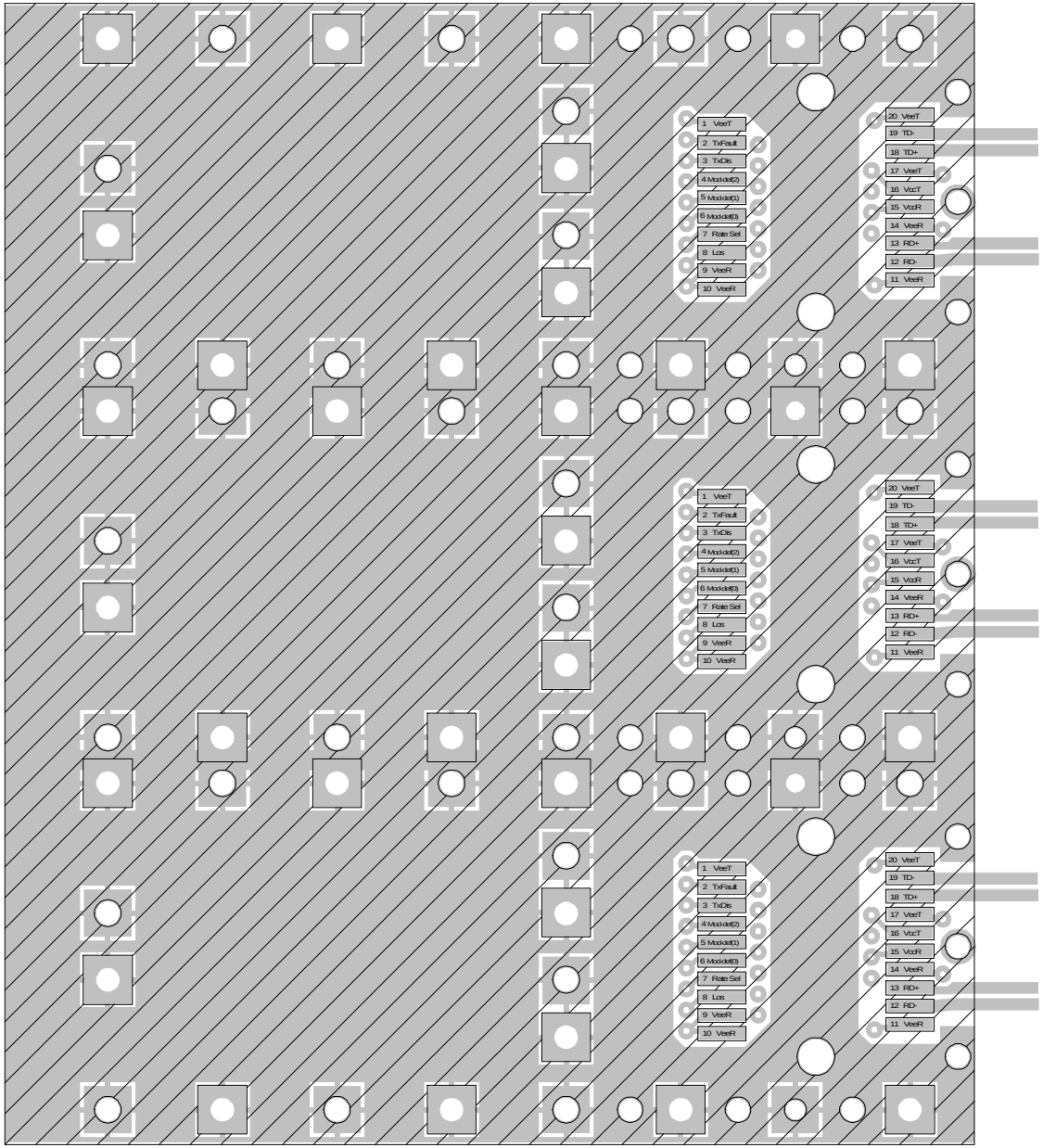
2/15/00 P.J.Edwards, Tyco Electronics.

The essence of the problem was to demonstrate how difficult (or easy) it is to lay out a pcb to accept double sided module attach, whilst keeping to industry standard guidelines for pcb layout. This demonstration shows how one could attach six module cages, three on the top side of the pcb and three on the bottom side, whilst maintaining 16.25mm (640mil) module to module spacing, per the recommended pcb layout drawings for a single module.

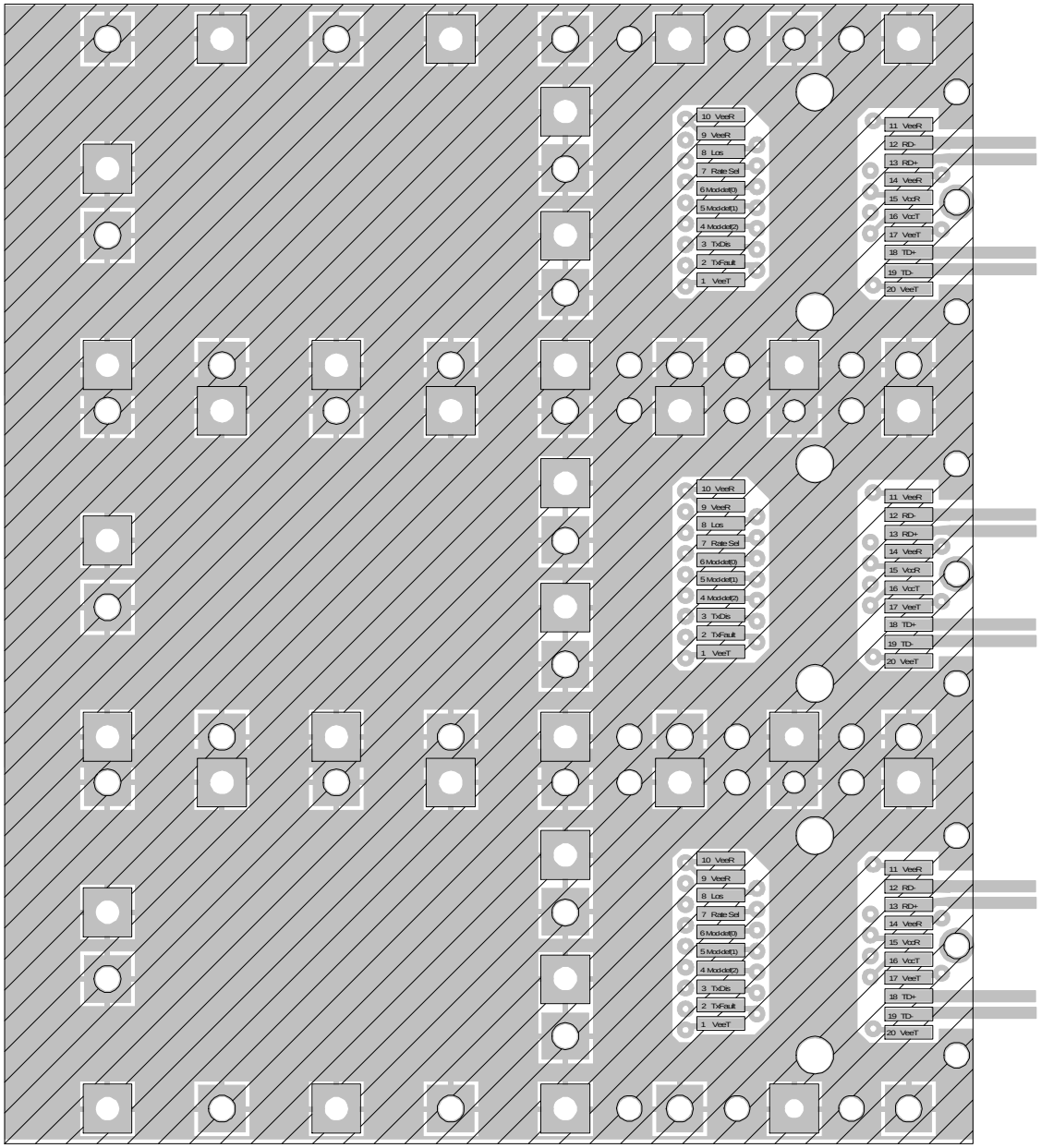
Some points to note are:

- The minimum thickness of this Customer pcb must be 3mm (120mil) to allow clearance for the release mechanisms of modules on opposing sides of the pcb.
- It has been assumed that four conductor layers would need to be shown. These would be: Top metal (for Chassis ground, surface mount pads for the module connector, and high speed signal traces); Inner layer 1 (for slow speed signals pertinent to the top modules); Inner layer 2 (for slow speed signals pertinent to the bottom modules) and Bottom metal (see top metal explanation).
- No attempt was made to route digital power and ground as these would probably be presented to the modules via localized power planes.
- 8mil lines and spaces are used as a minimum.
- 12 mil vias with 28mil diameter pads are used.
- 2mil soldermask keepout from copper is used.
- High speed lines are 20mils wide, with an 8mil gap between them and a 12mil distance to a digital ground layer on the next layer. This gives 50 Ohm traces
- The cross-hatching on the outer metal layer views shows the soldermask locations.

Top Metal and Soldermask.



Bottom Metal and Soldermask (viewed from topside).



Inner slow speed signal trace layers (viewed from top).