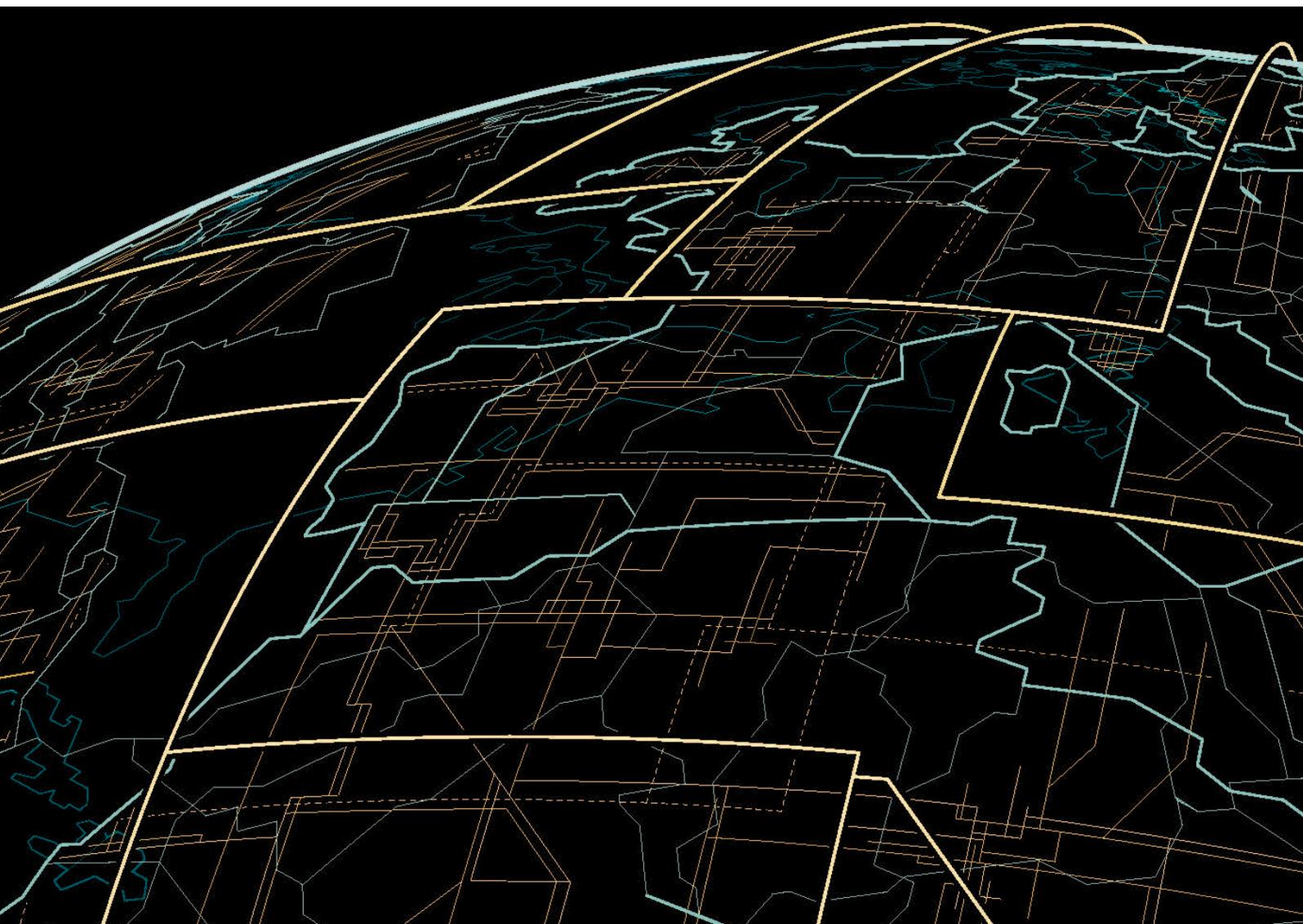


APPLICATION NOTE AN003

Protection for OEM data communications, signal and telephone systems

ESP AN003 for ESP PCB/06D, ESP PCB/15D,
ESP PCB/30D, ESP PCB/50D, ESP PCB/TN,
ESP PCB/06E, ESP PCB/15E, ESP PCB/30E,
ESP PCB/50E



Protection for OEM data communications, signal and telephone systems

Circuit board layout and design

All Furse PCB protection products are capable of handling 10 kA of surge current.

The ESP PCB/** Series are electrically the same as the (wire-in) Furse ESP D, SL, E and TN Series protectors (e.g. ESP 06D, ESP SL06, ESP 15E, ESP TN).

All these units offer a high level of protection, however with the PCB mount versions the track layout or choice of connectors may restrict the unit's performance as the line and earth tracks need to be able to handle 10 kA.

If the track fails before 10 kA, the surge protection offered will be limited to what the track can handle before breakdown.

Some guidelines for suitable track layouts are outlined opposite.

The track width connected to the output pins does not affect the surge current capabilities, however care must be taken to ensure the transient is not picked up on the output tracks.

The dirty ('line') tracks should be routed parallel and as close together as possible. This should also be implemented on the clean tracks, however clean (outgoing) tracks should never be routed close and parallel to line (incoming) tracks or dirty barrier earth connections as the transient can be re-introduced after the protector due to electromagnetic coupling (Figures C and D opposite).

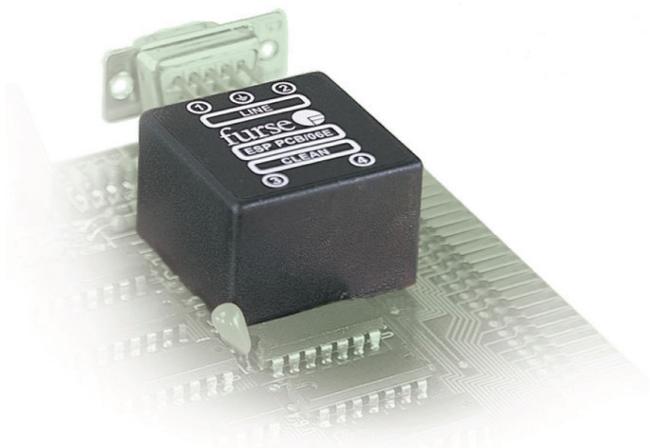
If it is unavoidable the clean tracks can cross the line tracks at 90°.

The use of an earth layer or plane is highly recommended as this reduces the electromagnetic field produced by a transient discharging to earth considerably, and hence the chance of the transient being picked up on the clean tracks.

If multiple lightning barriers are used on a PCB, dirty line and clean lines should be kept at least 20 mm apart (Figure B).

This separation distance must still be implemented on multi-layer PCBs, as the interference will easily pass through the board. If the PCB layout requirements cannot be met, the wire-in Furse D, SL, E and TN series equivalents can be used.

For further queries regarding suitable track layouts, please contact Furse.



Good layout and design**Do**

- Keep the input 'line' tracks as close together as possible
- Keep the output 'clean' tracks as close together as possible
- Use the largest track width the board can accommodate for the 'line' inputs
- Balance out the large track width with good track separation to ensure adequate creepage and clearance
- Use both top and bottom copper layers on the PCB (if possible)
- Use the largest pad size possible, as small solder joints can break down with a transient overvoltage

- Position the unit as close to the line input of the PCB as possible, minimising the track distance
- Use a high PCB copper plating level, as thicker plating will considerably increase the current handling
- Connect the earth to the main star point of the earthing system, routing away from all other connections
- Use an earth layer/plane (if possible), as this will greatly minimise inductance and electromagnetic coupling

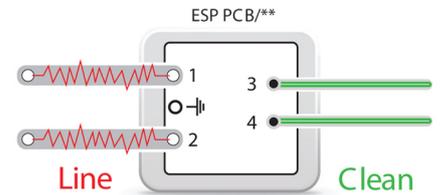


Figure A: Maximum line to clean separation. Large input tracks and pads (using top and bottom copper layers). Earth pin is bonded to an earth layer / plane.

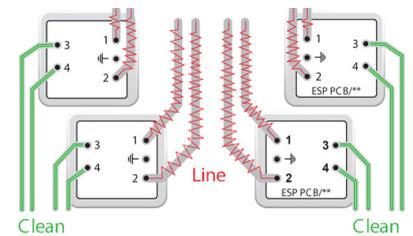


Figure B: All dirty ('line') incoming tracks are separated from the clean output tracks, individual line and clean tracks are routed close together. Earth pins are bonded to an earth layer/plane.

Bad layout and design**Don't**

- Route the line inputs (including earth) close to the clean outputs
- Create large loops with the line or clean tracks as this will increase electromagnetic coupling

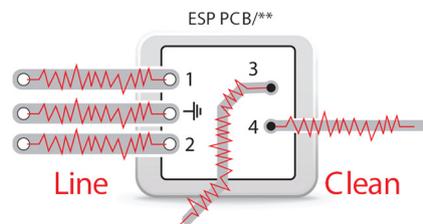


Figure C: Output tracks will create a large induction loop and are too close to input tracks. Input pads could be made larger. Transient will be re-introduced after protector.

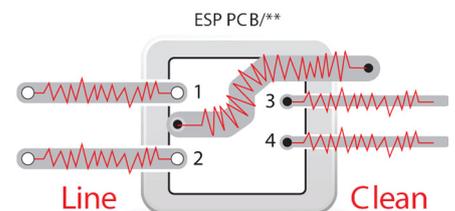


Figure D: Earth track too close and parallel to output tracks. Input pads could be made larger. Transient will be re-introduced after protector.



—

ABB Ltd.

Tower Court
Foleshill Enterprise Park
Courtaulds Way
Coventry CV6 5NX
National Tel: 0333 999 9900
National Fax: 0333 999 9901
E-Mail: LV.Enquiries@gb.abb.com

www.abb.com/lowvoltage

