



## Case One - Telecommunications Leader

### A major, well-known telecommunications company verifies the importance of Netlist Compare

A2Ts output their PCB design database into Gerber Files, as commonly done. This particular design specifically called for square pads on the plane layers. However, the output showed round clearances on the plane layers. The CAD system's DRC approved this discrepancy with no problem.

### The Gerbers were sent to fabrication...

- PCB CAD design rule checking validates design with no clearance violations including plane layer connectivity
- Gerber files created with square pads on plane layers with round clearance pads

The fabricated bare boards **FAILED** due to large number of shorts on the plane layers.

### With Translation Comes Risk

Verifying the CAD netlist matches the final netlist extracted from the Gerber will ensure original design intent is maintained. Netlist Comparison is key in this process. Many common errors uncovered through Netlist Compare include:

- Accidental inclusion/exclusion of traces, vias, etc on Signal layers
- Text or reference designators placed on signal layer instead of a silkscreen
- Placement of drafting items on ALL layers instead of specific layers

### The Result

After the bad boards were produced, A2Ts requested CAM350 software be brought in to trouble shoot the problem. Netlist Compare immediately detected the plane layer mismatch. Precautions were taken, discrepancies resolved, and successful boards were produced.

### In Conclusion

A2Ts now uses CAM350 as the final validation before committing any designs to Fabrication.

### Netlist Compare

Netlist Compare literally compares two netlists. It accurately compare your original CAD netlist with the extracted one in Gerber or with your fab house's artwork.