



Introduction

Net name changes to the Schematic Netlist can cause DRCs and un-routes for nets connected to planes. When a plane's net name is changed via the schematic netlist and imported into the board, all items associated to the net name do NOT get updated. The Cline (etch) connecting the pin to the via, the plane (shape) and the via itself may not get renamed to the new net name causing DRC errors. This occurs when:

1. A net name was never specified on the schematic and the packager created a net name. When an ECO is done on the schematic, the packager may assign a new undefined signal name (un1...).
2. The power/ground net names are changed in the front end tool.
3. The power/ground net names contained special characters on the schematic and the packager removed those characters. +5_vcc became +5vcc. Packager XL may not be set up with a directive to also remove the underscore.

The reason items do not get updated with the new net name is because the connectivity model cannot allow netrev to change the net name connected to a shape. If some pins on the original net moved to a new net, while other pins remained on the old net, netrev cannot determine which net would be correct. Therefore it does not change the original net name. If the entire net name is renamed, and the original net name does not exist in the PST files, then netrev will change the shape name to a 'dummy net'. This causes all original etch and vias to inherit the 'dummy net' name. A DRC will be generated from each pin on the net which has a new name to the etch and vias originally associated with them. It is not possible to reconnect the pins with new net names to the etch and vias tied to the shape. Options to resolve this are:

1. Change all shapes where net name changes will occur with the new netlist to "Dummy Net" then import the new netlist.
 - a. For each shape, Shape - Select Shape or Void/Cavity (Allegro) or Shape – Select Shape or Isolation/Cavity (OrCAD), then LMB (left mouse button) the shape.
 - b. Options Menu change Assign net name from old net name to Dummy Net. RMB - Done.
 - c. Import the new netlist using File > Import > Logic (Allegro) or Import > Netlist (OrCAD).
 - d. Repeat the steps a-b but change the net name to the new one.
2. Rename the nets in Allegro to match the new netlist before importing the netlist (Allegro PCB Designer only).
 - a. Logic > Net Logic (You will need to set a user preference to allow this "logic_edit_enabled")
 - b. Select the net to be renamed under the "Options" tab
 - c. Select the "Rename" button
 - d. Enter the new net name, select OK
 - e. Import the new netlist using File > Import > Logic (Allegro) or Import > Netlist (OrCAD).

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.