



## PCB DESIGN ANALYSIS

Company Name: \_\_\_\_\_ Contact \_\_\_\_\_  
 Address: \_\_\_\_\_ Phone : \_\_\_\_\_  
 \_\_\_\_\_ Email: \_\_\_\_\_  
 \_\_\_\_\_

We aim to supply you back your Design Analysis within 5 days of receiving the complete data package, although we can offer a quicker service if required. We will not be held responsible for any claims or damages arising from the use of the data or information that we supply to you.. Please compress the whole data package (Zip, Tar) and email it aslong with this form to [jd.info@jdphoto.co.uk](mailto:jd.info@jdphoto.co.uk). The form MUST be signed to acknowledge these conditions.

Signed: ..... Name:..... Date: .....

**Please tick all / any boxes for which you require analysis. Where required, please supply your values for each test.  
 If no values are supplied, we will use a pre-defined values set.**

### PROJECT

Project Name: \_\_\_\_\_ Approx PCB Size: \_\_\_\_\_ x \_\_\_\_\_

### Data Format supplied :

- Gerber 274X data
- Gerber 274D data
- ODB++ or ODB Xml
- BARCO DPF
- Other... \_\_\_\_\_

### I will also supply

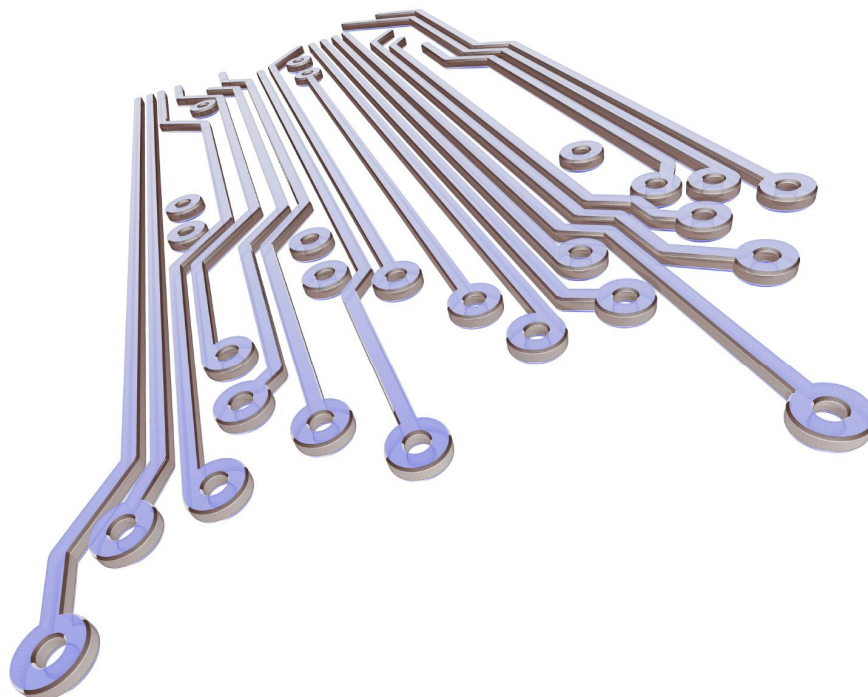
- IPC-D-356 Netlist
- NC Drill
- NC Mill Rout Data
- Pads ASCII Netlist

### NETLIST COMPARISON

- .. **Shorts**  
Multiple external nets touch one internal net.
- .. **Opens**  
Multiple internal nets touch one external net.
- .. **No Copper**  
No pad exists at the external net location.
- .. **No External Net Point**  
No external net point exists at an internal net point.
- .. **Pad Size Differences**  
The probe point pad size in the external net differs from the internal net pad size.

## DESIGN RULE CHECKS

- .. **Pad / Pad** *value:*  
*The minimum spacing allowed between pads.*
- .. **Pad / Track** *value:*  
*The minimum spacing allowed between pads and Tracks.*
- .. **Track / Track** *value:*  
*The minimum spacing allowed between Tracks.*
- .. **Border** *value:*  
*The minimum spacing allowed between any item and the border*
- .. **Min Pad** *value:*  
*The minimum pad size allowed.*
- .. **Min Track** *value:*  
*The minimum Track size allowed.*
- .. **Stubs**  
*A Track stub is any Track that touches a pad or Track on one end, but does not on the opposite end.*
- .. **Drill / Drill** *value:*  
*The minimum spacing required between adjacent drill locations.*
- .. **Drill Annulus** *value:*  
*The minimum annular ring required between conductive layers and drill layer.*
- .. **Drill / Copper** *value:*  
*The minimum space required between copper entities and drills. Separate tolerances are available for plated through-holes (PTH) and non-plated through holes (NPTH).*
- .. **Pad / Top Mask & Pad / Bot Mask** *value:*  
*The minimum annular ring required between top / bottom side pads and the top / bottom solder mask.*
- .. **Drill / Top Mask & Drill / Bot Mask** *value:*  
*The minimum annular ring required between the drill layer and the top / bottom solder mask.*
- .. **Missing Top Mask & Missing Bot Mask**  
*Checks for pads on the top / bottom side conductive layer without openings on the top / bottom solder mask.*
- .. **Pad / Top Silk & Pad / Bot Silk** *value:*  
*The minimum clearance required between top / bottom side pads and the top / bottom silkscreen layer OR the corresponding mask layer.*
- .. **Missing Drill**  
*Checks for pads that do not have a corresponding drill hit.*
- .. **Coincident Drill**  
*Checks for drill hits that are in the same location but are different sizes.*
- .. **Redundant Drill**  
*Checks for drill hits of the same size that are in the same location.*
- .. **Mill Path**  
*Checks compensated mill paths for errors introduced by the compensation: arcs that implode, paths that cross over themselves, and any break tabs that become invalid because the compensated path is too short for the tab to fit.*



---

## **DESIGN FOR FABRICATION:**

- .. **Acid Traps** *value:*  
*An acid trap is an area where etching solution accumulates but does not flow out during manufacturing. This causes over-etching, which hurts yield. This analysis detects potential acid traps in your design.*
- .. **Copper Slivers** *value:*  
*Copper Slivers are areas of copper that are so narrow that they will likely flake off. This command detects those potential slivers on the selected conductive layers in your design. Any areas of copper that are less than the Size you specify will be considered copper slivers.*
- .. **Resist Slivers** *value:*  
*Similar to acid traps, resist slivers are small areas of resistive material that have a surface area too small to adhere to the board, and can therefore flake or peel off. Resist areas less than the value you specify are considered resist slivers.*
- .. **Top Mask Slivers & Bot Mask Slivers** *value:*  
*Solder mask slivers are areas where the resist is so narrow that they will likely flake off, float, and redeposit themselves in an area that might be soldered later. Any resist areas that are less than the Size you specify will be considered possible mask slivers.*
- .. **Isolated Thermals** *value:*  
*Over-etching of surrounding items can result in a thermal being isolated from the rest of a negative plane. This analysis function allows you to oversize the data on the negative plane layers by a specified amount, and determine its effects on the connection of the thermal to the negative plane. You specify the amount of Over Etch that you wish to apply to the data on the specified plane layer.*
- .. **Starved Thermals** *value:*  
*The starved thermal analysis verifies whether each thermal connection to the negative plane is valid, or has been constricted by adjacent data that is too close (or overlapping). You specify the Percentage of the thermal's tie width that can be unblocked by objects. The Clear value is how many thermal ties must be clear in order for a thermal to be considered not starved. If you specify 0, all ties must be clear. If you select 1, one tie must be clear.*
- .. **Top Solder Bridges & Bot Solder Bridges** *value:*  
*Openings for pads on a mask layer may be oversized too much, and expose an adjacent Track or another conductive object. The copper for that pad may accidentally get too close during fabrication, and create an unwanted connection, or bridge, to the adjacent object. The Bridge Distance is the distance between the pad and objects in the same mask opening, where the solder can possibly create a bridge. If the adjacent object closer to the pad than this distance, it will be identified as a possible bridge.*
- .. **Mask Misalignment Bridging** *value:*  
*The Position Tolerance tests for shifting of the placement of mask openings. It over sizes mask openings by the amount you specify, to check if other conductive objects are then exposed by the mask opening. If an object is exposed, and is also within the specified Bridge Distance to the pad, then an error is logged.*
- .. **Pin Holes** *value:*  
*A pin hole is a void in an area of solid copper that can cause acid to pool and over-etch the surrounding copper, or can cause resist flakes, potentially causing conductivity problems. You specify the maximum Size of the pin holes to be detected*
- .. **SMD Pitch** *value:*  
*Checks the centre to centre spacing of adjacent SMD pads, to determine if they are too close.*
- .. **SMD Spacing** *value:*  
*Checks the edge-to-edge spacing of adjacent SMD pads, to determine if they are too close.*
- .. **Layer Registration** *value:*  
*Checks your drill layer against the top mask and any other conductive layers you select. If two items are within the tolerance distance you specify, they are considered to be at the same location.*

---

## **EMBEDDED PASSIVES ANALYSIS**

- .. **Alignment** *value:*  
*Checks that the resistor shape is centred (horizontally and vertically) between the termination bars. An error is reported if the resistor shape is out of alignment by more than the specified tolerance.*
- .. **Bar Overlap** *value:*  
*Checks that the resistor shape overlaps the two termination bars by the specified amount.*
- .. **Bar Extension** *value:*  
*For additive passives, this checks whether the termination bars extend beyond the resistor shape by the specified amount. For subtractive passives, this checks whether the resistor shape extends beyond the termination bar by the specified amount.*
- .. **Bar Width** *value:*  
*Checks if the termination bars are at least as wide as the specified amount.*
- .. **Minimum Size** *value:*  
*Checks that the active resistor area (not including the overlap) is at least the specified X:Y size.*
- .. **Maximum Size** *value:*  
*Checks that the active resistor area (not including the overlap) is no larger than the specified X:Y size.*
- .. **To Copper** *value:*  
*Checks the distance of the resistor shape against any adjacent copper. If they are closer than the specified distance, an error is reported.*
- .. **To Drill** *value:*  
*Checks the distance of the resistor shape against any drills on the same net. If they are closer than the specified distance, an error is reported.*

---

## **FURTHER ANALYSIS & OPTIMISATIONS**

- .. **Solder Mask Optimisation**  
Optimisation of solder mask layers using a comprehensive set of rules that are designed to give you maximum possible clearance while maintaining optimal coverage. Support for fine-pitch pads, with block openings for those areas to prevent web (sliver) problems between pads.
  - .. **Paste Mask Optimisation**  
Optimisation of paste mask using a comprehensive set of rules that are designed to give you optimal reduction across a wide variety of pad shapes and sizes. Special support for fine-pitch pads allows for independent X and Y reductions to eliminate paste build-up between pads.
  - .. **Find Duplicates**  
The Find Duplicates command finds all occurrences of duplicate data.
- 

## **DESIGN INFORMATION**

- .. **Min Air Gap**  
*Reports the shortest distance between any two items in the design.*
- .. **Conductive Layer Count**  
*Reports the total number of conductive layers.*
- .. **Board Size**  
*Reports the extents of the Border layer, if one exists. Otherwise the extents of the conductive layers is reported (drawing and other graphical layers are ignored).*
- .. **Hole Count**  
*Reports the tools used on each NC layer, and the number of holes (hits) created by each.*
- .. **Hole Size Count**  
*Reports the number of tools used on each NC layer (does not report tool numbers or hits).*
- .. **Buried Vias**  
*Reports the upper- and bottom-most (start and end) layers for buried vias.*
- .. **Blind Vias**  
*Reports the upper- and bottom-most (start and end) layers for blind vias.*
- .. **Min Pad / Drill**  
*Reports the minimum annular ring spacing from copper pads to plated through-hole drill hits.*
- .. **Min Drill / Drill**  
*Reports the minimum spacing between drill hits.*
- .. **Min NPTH / Copper**  
*Reports the minimum spacing between non-plated through-hole drill hits and copper on external and internal layers.*
- .. **Min Pad / Pad**  
*Reports the minimum spacing from one pad to another for each conductive layer.*
- .. **Min Pad / Track**  
*Reports the minimum spacing from a pad to a Track for each conductive layer.*
- .. **Min Track / Track**  
*Reports the minimum spacing from one Track to another for each conductive layer.*
- .. **Min Pad / Mask**  
*Reports the minimum annular ring spacing from mask openings to their corresponding copper pad, on the top and bottom layers.*
- .. **Min Thermal / Drill**  
*Reports the minimum annular ring from thermal relief to plated through-hole drill hits.*
- .. **Min PTH Pad Size**  
*Reports the size of the smallest pad that has been pierced by a plated through-hole drill hit, for each conductive layer.*
- .. **Min Track Width**  
*Reports the smallest Track width in the design.*
- .. **Min SMT Pitch**  
*Reports the minimum centre to centre spacing of SMD pads. A pad must be identified as SMT in the Aperture List in order to be checked.*
- .. **Legend on PTH Count & Legend on NPTH Count**  
*Reports the number of occurrences of the silkscreen legend touching pads on the top and bottom layers.*
- .. **Missing Hole Count**  
*Reports the number of instances where a through-hole pad on a conductive layer appears to be missing a drill hit.*
- .. **Mill Path Length**  
*Reports the total linear distance each tool travels, per NC layer.*
- .. **Max PTH Registration**  
*This shows the greatest amount of misalignment of data on drill and conductive layers. It reports the maximum spacing of plated through-hole drills (on an NC layer) from their corresponding pads (on a conductive layer).*
- .. **Exposed Copper Count**  
*Reports the number of occurrences of Tracks exposed by adjacent mask openings.*
- .. **Copper Sliver Count** *value:*  
*Reports the number of occurrences of copper slivers. Any areas of copper that are less than the Size you specify will be considered possible copper slivers.*
- .. **Mask Sliver Count** *value:*  
*Reports the number of occurrences of mask slivers. Any resist areas that are less than the Size you specify will be considered possible mask slivers.*