

# Auditing a Printed Circuit Board Fabrication Facility

**Greg Caswell**

## Introduction

DfR is often requested to audit the PCB fabrication process of a customer's supplier. Understanding the process variations and nuances is vital to performing a viable analysis. This white paper delineates DfR's approach to this task. The audit encompasses all levels of PCB fabrication, from generic multilayer rigid boards to complex rigid-flex configurations.

Audits can be performed to assess the viability of a new supplier, to assure their compliance with our customer's requirements, or to identify an issue that has evolved into a failure analysis situation.

## Revenue/Stability

DfR typically requests information regarding the financial stability of the supplier to assure their growth status and profitability. This is important feedback to our customer as it provides insight into the viability of the supplier.

## Personnel Present at Audit

The audit report identifies the individuals that were involved in the audit.

## Materials and Process Audit

This section of the white paper provides insight into DfR's assessment of the process operations observed during a typical audit of a PCB fabrication facility.

## Planning Engineering

The first step in most PCB fabrication facility processes involves evaluating the Planning Engineering function whereby an engineer verifies the customer data, the QA codes, specifications, and anything beyond the drawings and special instructions.

The Planning Engineer normally builds the in-house traveler, identifies the materials to be used for fabrication, the stackup to meet electrical requirements, any impedance controls, and creates the lay-up sheet identifying the copper weights per layer and the dielectrics. In addition he performs a net-list verification, reviews the customer SOW, ECNs, IPC-6012 requirements, panel size, and special codes. If required, the Planning Engineering function also incorporates the customer HATS coupons into the lay-up. Finally, Planning Engineering's work triggers the CAM/CAD activity and Document Control.

Verification of all of these activities is the first stage a detailed audit.

## **CAM/CAD**

The CAM activity usually sets up a customer part number and drawing for the project, creates a job number, links in the input data from the customer, sets up the layers using internal naming structures, and performs any edits needed to match internal processes with the customer supplied data. CAM also checks the customer supplied data against the drill files and sets the lines and spaces necessary for each layer. Finally the CAM operation sets up the panelization for the project including the inclusion of specified coupons.

All of these actions are verified in an audit.

## **Documentation and Configuration Control**

Documentation packages are then prepared for each production run and are maintained for many years, with 5 years being typical, unless the customer requires a different timeline. All Standard Operating Procedures should also be readily available for review.

## **Training**

The auditor should also verify the internal training program for operators to assure that they are all qualified to manufacture the customer's products.

## **Process/Materials/Equipment Changes since DATE**

The audit should also determine what recent changes, if any, in processes and materials have been implemented as these changes can have a profound impact on the fabrication process.

## **Fabrication Flow**

The following paragraphs delineate a typical PCB fabrication flow and DfR's assessment approach.

### **Incoming Receiving**

The audit should verify that the materials received are verified against the list of materials called out for the project. Thickness of the cores, copper weight, and layer identification are ascertained at this operation. The auditor should verify that the materials selected meet the procurement requirements by picking a random lot for assessment

Prepreg received should be stored in a temperature and humidity controlled area. Typically, prepreg should be moved to the fabrication process on a FIFO basis to control storage times. How prepreg is handled and transferred is a critical operation and needs to be audited.

### **Inner Layer Imaging, Development, Etching, Resist Strip, Handling**

The audit then examines how the cores are cleaned and how the wet lab monitors the samples.

All processes audited should have process controls in place, checks for verification, planned maintenance, and calibration controls that are all defined by panel size.

The imaging process used to produce the artwork and the process used to perform the exposure operation are next audited to assure that the set up and the artwork match the panel structure.

### **Inner Layer AOI and Visual Inspection**

Layer inspection is audited next, including the method used to monitor the materials being transferred to this operation. Good PCB fabricators will do this via an airlock to minimize any oxidation.

The audit must examine the AOI tool used and how it is linked to the CAD data. Quality Control tally sheets are typically used to record defects identified in the inspection of line widths and spacings as identified on the traveler. All data should be recorded including information on the type of defect and its location.

### **Inner layer preparation for lamination, Lay-up, Lamination, Break Down**

The bond film operation is next in the process and is typically fully automated. The materials being readied for lamination are baked if necessary and the information is logged. Sets of lay-up materials are generated and are presented to the lamination presses as completed stackups.

The process engineer sets the recipes for the presses through linking the parameters to the specific job number. He selects the correct recipe and logs it for verification purposes. A control monitor watches the temperature, pressure and vacuum levels to assure compliance.

The laminated boards then go through a cleaning operation to remove excess material from the edges of the laminate and the stackup is then verified again for thickness using a laser thickness measurement system. All data should be recorded.

Each panel is typically then serialized and coupons are cut out for microsectioning to confirm that inner copper layers are in the correct sequence and that the correct layers of pre-preg have been utilized.

The process engineer then gives the approval for the stackups to proceed to the drilling operation.

### **Drilling**

Normally, prior to the drilling operation the stackups are X-rayed to set fiducials to estimate the center point for setting the tooling holes. Drilling is normally accomplished with automated drilling systems. The number of drill hits should be monitored as a function of a drill list that defines replacement requirements based on size and speed. Drill bits are new and need to be replaced per the direction on the traveler.

### **Hole Preparation and Hole Metallization**

Hole preparation can take many forms to accomplish deburring. This could involve a brushing technique as well as a plasma cleaning operation. The audit should verify that the holes are completely clear.

## **Copper Plating if strike**

DfR would typically audit the electroless processing tanks in the production area.

At the control console the operator should select from a list of materials and cycle times for the specific project and enter that data into a log to confirm his selection.

The process engineer should be the only person who can change the recipe and operators should only be able to select it from the menu on the control computer. The only action required by the operator at this point is to load and unload the panels as all else is typically automated.

## **Copper Plating, Copper Analysis, Copper Properties Testing**

PCB fabricators typically have several pattern plate processes that can be utilized; Nickel only, fused Sn/Pb and HASL, ENIG, ENIPIG, Ni with hard gold, and Ni with soft gold.

All process parameters for each of these variations should be monitored. DfR recommends using the TrueChem software for this purpose.

## **Resist Strip, outer layer Etch, Removal of Etch plating**

The plasma etching process for etchback and desmear is the next process operation. The operator should log the film type, the exposure level, the age of the bath and the chart for day/day settings.

Etching is normally based on the starting copper weight versus the speed of the process to maintain control. It is also at this point in the process that controlled impedances in the board are tested and verified.

100% of the pieces should be inspected at this point, looking for surface issues, burrs and holes.

## **Solder Mask, Development, and Cure**

Resist is typically applied in a clean room environment following the requirements of the traveler including hold time and laminate.

The material is then photo-processed where the operator selects the resist height and logs the data into the tally sheet and shop order. DfR recommends the Laser Direct Imaging process for this function.

Typical soldermask processing uses a Liquid Photo Imageable (LPI) material, with a 100% inspection of the solder mask images after processing. Dry or Wet mask processes, if used, should also be audited.

## **Marking/Silk Screening**

Marking and Silk Screening are another typically automated process with a first piece inspection performed to assure correct marking including date code and serial number. Following both the mask and silkscreen operations the process involves a cure of both materials. Controls for these processes is part of the audit.

## Solder Coating

A vertical dip coating process for applying a HASL finish to the circuit boards is a fairly generic process for solder coating. The boards are fluxed, dipped in a solder bath for a controlled period of time using set programs, are air knifed on the way out and placed in a rack to cool for a short time before being washed. Post process cleaning is then typically performed.

Coupons are removed after this operation including all process and HATS coupons which are then sent to a microsection lab for analysis.

## Electrical Testing

Flying probe and fixture probe testing are basic elements of a PCB testing capability. Measurements can be made at different voltage levels using a sampling plan.

## Final Inspection and End Item Data Package

IPC 600 certified inspectors should perform the final inspections of the circuit boards using stereozoom microscopes. On-site training should be provided to these inspectors. If defects are noted, they should be marked for touch up whereby the boards are then sent to a final inspection. Coupons, cross sections, and full documentation should be prepared for delivery with the completed product.

## Cross Section Laboratory

Cross sections for analysis of the processes should be prepared using multiple grind steps with increasingly finer grit sand paper followed by multiple polish steps.

A full lab report should be delivered with the customer's deliverables.

## Shipping

A *Requirements Document* should be used to summarize yield figures, start dates, date codes, along with special notes per customer requirements.

## Documentation Review

DfR also audits documentation typical of those used to monitor and control the process.

## Summary

The audit process described in this white paper addresses a typical printed circuit board fabrication process. Not all facilities have all of the operations described, but rather have different approaches. In any case, the flow in the audit would encompass the unique processes to assure compliance.