
CHAPTER 19

MANUFACTURING INFORMATION DOCUMENTATION AND TRANSFER

Clyde Parrish

PC World, Division of Circuit World Corp., Toronto, Canada

19.1 INTRODUCTION

The manufacturing of printed circuit boards (PCBs) begins with the soft-tooling process. This process is the transformation of customer computer-aided design (CAD) data into the necessary tools required for manufacturing the bare printed circuit board. The typical tools required for manufacturing printed circuit boards include artwork for photoprinting of inner conductive layers, outer conductive layers, and solder mask patterns. Artwork is also created for screen printing patterns for nomenclature and via-plugging layers. Additional tools required include drill and routing numerical controlled (NC) programs, electrical testing netlists and fixtures, and CAD reference soft-tools. During the tooling process, the bill of materials (BOM) and process routing are also defined.

During the tooling process, the customer part numbers are analyzed to determine the compatibility of the design features with the manufacturing process capabilities. Additionally, attempts to optimize the manufacturing of the product at the lowest cost is a primary goal. However, the majority of the costs have been defined before the design is transmitted to the manufacturing site by the PCB designer. An early investment in time by the PCB design team and the manufacturing tooling team can result in the most significant savings in overall product cost.

This chapter describes the PCB tooling process, as defined in Fig. 19.1, including the transfer of information, design reviews, optimization of materials, definition of BOM and routings, tool creation, and additional processes that are required.

19.2 INFORMATION TRANSFER

The tooling process begins with the receipt of information from the customer. This information historically has been transferred via mail or overnight delivery. Increasingly over the past 10 years the communication between design site and manufacturer has been via electronic means to reduce lead times. Unfortunately, although the time required to send information to the manufacturer has decreased to minutes or hours from day(s), a significant issue with the provision of packages to manufacturers is the completeness of the information provided.

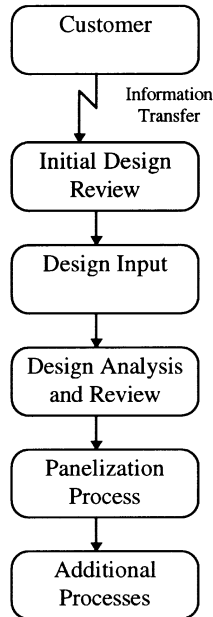


FIGURE 19.1 The soft-tooling process.

The following sections define the basic information required to transmit and methods for communication.

19.2.1 Information Required

Timely tooling of a part number depends on having the correct information. All features required to exist on the PCB must be defined to the manufacturer. The information is defined via design data, drawings, and textual information. The information and common data formats required to permit the tooling of the PCB include the following:

- Part number information

Information: This information defines the part number to be built, including revision number, releases, dates, etc.

Format: This information is typically provided in the part drawing or may be provided as an additional text file.

- Part drawing

Information: This drawing may contain specific design requirements such as material requirements, controlled impedance requirements, solder mask type, nomenclature color, and dimensional tolerances.

Format: Common formats for drawings are HP-GL and PostScript.

- Drill drawing

Information: Although the drill data is provided via data files, this information typically contains only the location of holes and the tool number. The tool number is referenced against the drill drawings to determine the required sizes, plating status, size tolerances, and the total count for verification.

Format: Common formats for drawings are HP-GL and PostScript.

- Subpanel drawing

Information: Many assembly operations require boards to be provided in a subpanel form (many parts on one shippable unit). The drawings will define the orientation and position of each part, the subpanel dimensions, tooling hole information, special markings, and specific manufacturing processes and tolerances.

Format: Common formats for drawings are HP-GL and PostScript.

- Artwork data

Information: This data consists of files for each circuitry, coating (e.g., solder mask), or marking (i.e., nomenclature) layer.

Format: The data required is usually RS-274, commonly called *Gerber data*. Gerber data is provided as a standard output from most PCB CAD systems.

- Aperture list files

Information: The definitions of the shapes used for drawing are required for each layer provided of artwork data. Special shapes like thermal pads should specifically define its method of construction.

Format: This information is usually provided as a text file, although the information may also be defined in the beginning of the individual artwork files, including constructions of complex apertures.

- Drill data

Information: This information may consist of a single or multiple file(s), and defines the location and tool number used for each hole in the PCB. The files required should define all plated, unplated (can be combined with plated if fully defined), buried via, and blind via layers.

Format: Common data files provided are Excellon format.

- Drill tool files

Information: This information describes the size, plating status, layer-from and layer-to (in the case of buried and blind vias), and file names. This information is referenced against the drill drawing.

Format: This information is usually provided as a text file, although the information may also be defined in the beginning of the individual drill files.

- Special requirements information

Information: The drawing or a file should describe any special requirements not defined in other information. It is important for the PCB designer not to assume that requirements are understood, but to refer to specifications or clearly define the requirements.

Format: This information is typically provided in the part drawing or may be provided as an additional text or drawing file.

- Netlist data

Information: Netlist data defines the connectivity of the circuitry.

Format: This information can be provided from the CAD systems in various formats, or it can be extracted from the drill and artwork data. Contact the PCB manufacturer for compatible formats, if the netlist data is to be provided directly. The Institute for Interconnecting and Packaging Electronic Circuits (IPC) has defined a neutral format, IPC-356, which provides all the information necessary for netlist and electrical test fixture creation.

The IPC has defined an alternative neutral format for most of the previously defined data which provides simpler processing at the manufacturer, IPC-350. This format can be generated by most PCB CAD systems and processed by most PCB CAM/tooling systems. The PCB customer should review the compatibility of this format with the PCB manufacturer prior to sending.

19.2.2 Modem Transmission

Currently, the most popular means for transmitting data from design site to manufacturing site is via personal computer modem transmission. Two popular methods for communications are:

1. Dedicated connection to waiting compatible software
2. Bulletin board-type service

Method 1 is the simplest means to initiate communication. Almost any communication package can provide the capability, for example, X-Modem or Windows terminal. Method 2 provides a more robust environment similar in concept to CompuServe. Each user has an individual account logon and password.

19.2.3 Internet Transmission

Several methods are available for connecting to the Internet, depending upon the volume of data, the interactiveness required, and the budget. Typically, a company will utilize a UNIX-

based machine for accessing the Internet. UNIX-based computers are used to provide multi-user/multitasking capabilities (i.e., two or more people may be performing a task at the same time). Personal computers can also provide access depending on the functionality and interactivity desired. The following describes some of the alternatives available:

- *User-Account on Internet Provider's Computer.* This is the least expensive method for gaining access to the Internet. The account is accessed via dial-up modem connection, running a terminal emulation program. Many Internet providers offer this service. Monthly account fees are available for a minimal charge for each account, depending on disk storage requirements. Individual accounts may be maintained for each customer to provide security between customers. Transmission of data from the provider's machine may be provided via popular transfer protocols (e.g., X-Modem), dependent on your Internet provider. The communication speed is limited to modem speed, typically 14,400 baud.
- *Interactive Dial-Up.* This method provides direct communication to customers, through prior contact and the internal initiation of the connection. The communications between the internal computer and the Internet provider computer is typically via Serial Line Internet Protocol (SLIP) or Point-to-Point Protocol (PPP) software. These software packages provide the continuation of network communication from internal systems to the Internet provider. This service can cost less than \$100 per month.
- *Dedicated Dial-Up.* Dedicated dial-up lines are similar to the interactive dial-up, except that a dedicated phone number is provided. The connection to the Internet can be used either for interactive or for continuous-connection dial-up service. This service will cost slightly more than the interactive dial-up service, due to the dedicated line costs at the Internet provider.
- *Leased Lines.* Leased lines provide continuous, high-speed communications between a company and its Internet provider. They are also the most expensive, ranging from several hundred to several thousand dollars per month.

Security of the machines connected to the Internet is of prime importance. Any direct connections (i.e., not the user account just described) to the Internet are usually handled by dedicated machines acting as a firewall to prevent unauthorized entry into a firm's network, as outlined in Fig. 19.2.

There are numerous books available about the Internet and how to connect to it. There are also numerous consulting firms available to assist in the process, which is nontrivial, to prepare a secure connection.

19.3 INITIAL DESIGN REVIEW

The purpose of the initial design review is to determine the potential fit of the product to the manufacturing facility, determine general cost information, and prepare for tooling. Proper upfront analysis of a product prior to manufacturing or tooling, results in a reduction in waste of time and materials.

It is the responsibility of the manufacturing site to determine the fit of a given product to its capabilities. PCB manufacturing sites should monitor and maintain a list of manufacturing capabilities and a technology roadmap of where the facility is developing additional process capabilities. This list of capabilities will define the acceptability of a product for manufacturing, or whether the product is a research and development project.

19.3.1 Design Review

Reviewing the incoming package for design requirements (e.g., line width and spacing) versus the PCB manufacturing capabilities will define the capability for manufacturing, and provide

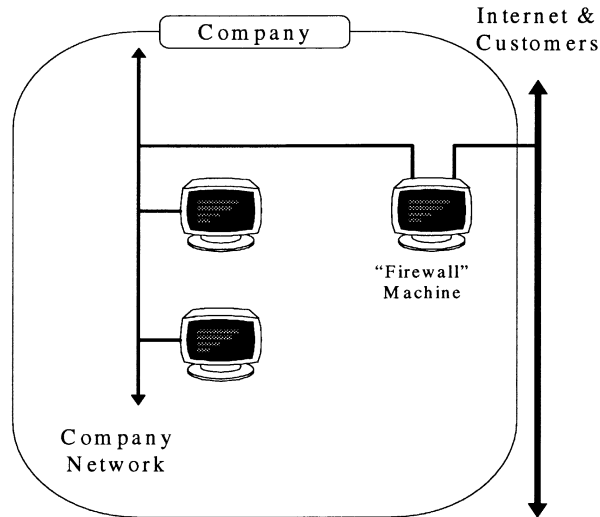


FIGURE 19.2 Typical Internet connection.

a prediction of the resulting yield. Among the design characteristics that should be reviewed against process capabilities are the following:

Item	Issue
Maximum number of layers	Higher-layer-count products require greater control of processes and tooling tolerances.
Board thickness	Some process or handling equipment may have limitations on board thickness (either too thin or too thick).
Minimum feature width	Finer line widths require better control of artwork and process tolerances. In addition, the relationship of line width to copper weight is significant in providing a well-defined line. Fine lines with lower copper weights are easier to manufacture than fine lines with higher copper weights.
Minimum feature spacing	Finer feature spacing requires better control of artwork and process tolerances. In addition, the relationship of feature spacing to copper weight is significant. Fine spaces with lower copper weights are easier to manufacture than fine spaces with higher copper weights.
Minimum finished hole size	Smaller holes require higher manufacturing process capabilities and imply (due to design characteristics) finer tolerances on tooling and internal registration systems.

Maximum aspect ratio	Small holes and thick boards result in difficulty during the plating processes and can result in defective products that may or may not pass electrical testing. Plating high-aspect-ratio holes requires chemistry and process parameter enhancements.
PCB dimensional tolerances	Fine PCB profile or cutout tolerances may result in punching/blanking requirements versus routing, or changes in the routing parameters or programming.
Feature-to-feature tolerances	Location of features on the PCB to other features may require alternative materials or process changes to reduce the tolerances.
Hole size tolerances	Consistency in the plating process versus selected drilling hole size and plating densities have significant impact on the capability to control hole tolerances. Adjustments to the PCB design, drill hole size selection, or process parameters may be required to produce an acceptable product with tighter tolerances.

19.3.2 Material Requirements

Determination of the bill of materials is required during the initial analysis of the design. The determination of the BOM and other material-processing requirements will define the manufacturing facility's capability to produce and its material cost structure. In addition, the definition of the material requirements will be the basis for the generation of the process traveler requirements.

The primary materials requiring definition are those included in the BOM, including laminates, prepregs, copper foil, solder mask, and gold. The materials may be explicitly defined by the PCB customer (e.g., usage of a specific solder mask), or may be implied in the drawings or specifications provided with the tooling package.

Several factors impact the selection of the raw materials, including the following:

- Customer-defined physical constraints, for example, the definition of the physical dimensions between conductive layers
- Customer specification of electrical properties, for example, the definition of the impedance requirements on certain layers
- Manufacturing process capabilities related to lamination thicknesses and tolerances
- Specification of material dielectric requirements, for example, the usage of FR-4 or polyimide
- Specifications of physical operating parameters, for example, the minimum requirements of the glass transition temperature

The determination of the laminates, prepregs and copper foils are based upon the following:

- Standard constructions for the PCB manufacturer of a defined PCB layer count, final thickness, copper weight, and dielectric spacing.
- Custom constructions based on defined physical constraints (e.g., minimum dielectric spacing). These custom constructions are defined through knowledge of the lamination pressing

thickness of materials vs. copper circuitry densities, and the availability of materials from suppliers.

- Custom constructions based on defined electrical property constraints. These custom constructions are typically defined via equations or software models provided with certain product parameters.

The determination of solder mask is based on the customer specifications and drawings. Once the acceptability of the solder masks is defined, the selection of the acceptable masks by the PCB manufacturer is based on either preferred process (due to volume or cost) or the design characteristics' interaction with the solder mask. These design characteristics include the following:

- *Tenting of vias*—A dry film solder mask may be preferred over liquid photoimageable solder mask with a secondary via plugging process.
- *Platable area densities/higher external copper weights*—Thin solder masks may not be able to ensure coverage of high plating.
- *Secondary processes*—Post-solder-mask processes may chemically or mechanically alter the appearance of certain solder masks.

The determination of gold requirements are based on the thickness and area of the gold. These factors can be used to calculate the requirement of gold per PCB product.

19.3.3 Process Requirements

The selection of the proper product routing (or traveler) is critical to the upfront analysis of the product acceptability to manufacturing. Considering a typical multilayer PCB product, the product routings can be broken into two parts: the innerlayer pieces and the outerlayer piece.

The product routings of the innerlayer pieces are fairly standard, and are typically as follows:

Step	Typical process requirements
Layer cleaning	Cleaning of the laminate surface via either mechanical or chemical cleaning processes
Imaging	Coating of the laminate with photoresist material and the exposure of the photoresist with artwork defining the innerlayer pattern
Develop-etch-strip (DES)	Developing of the photoresist, etching of the exposed copper, and stripping of the remaining photoresist
Innerlayer inspection	Inspection of the innerlayer piece to the PCB design intent
Oxide	Coating of the innerlayer piece with an oxide layer prior to lamination

One of the few decision points in innerlayer manufacturing is whether the product will require inspection. This decision can be based on the manufacturing facility's process capabilities and the design of the specific innerlayer piece. For example, if the process capability for innerlayer manufacturing of designs at 0.008-in lines and spaces is 100 percent yield, and the product has been designed at or above 0.008-in lines and spaces, then the product may not

require inspection. Typically, the design package will identify the design technologies (e.g., line width and spacing), however, these should be confirmed during the design analysis and review stage.

The product routings of the outerlayer pieces define the finished product appearance and, as a result, are more complex. Assuming a pattern plating process, the typical outerlayer process routings for a SMOBC/HASL (solder mask over bare copper/hot-air solder level) product is as follows:

Step	Typical process requirements
Lamination	Lamination of innerlayer pieces with prepreg and copper foils to create the outerlayer piece
Drilling	Addition of the holes providing pathways for electrical conductivity between outer layer and innerlayer pieces, and other PCB design purposes
Electroless copper plating	Deposition of copper on the surface and in the holes of the product, providing the conductivity necessary for electroplating
Imaging	Coating of the panel with photoresist material, the exposure of the photoresist with artwork defining the outerlayer pattern, and the developing of the outerlayer pattern
Electroplate copper	The plating of the final circuitry of the product
Tin plating	Sacrificial plating over the final circuitry of the product
Strip-etch-strip (SES)	Stripping of the remaining photoresist, etching of the exposed copper, and stripping of the sacrificial plating
Solder mask	Coating of the bare copper product with either dry film or liquid photoimageable solder mask, exposing of the panel with the PCB customer-supplied artwork pattern, developing of the pattern (exposing the sites requiring solder), and curing of the solder mask
Nomenclature	Screening of the nomenclature onto the panel and curing
HASL	Coating the exposed copper sites with solder, leveling to the customer's requirements of thickness
Depanelization	NC routing of the PCB products from the manufacturing panel
Electrical testing	Electrical testing of the product for conformance to the PCB design
Inspection	Verification of product conformance to specifications prior to shipment to the PCB customer

After the electroplate copper operation, the process may change to meet the surface finish requirements. Some of the alternative steps affecting the outerlayer product routing include the following:

Item	Typical process requirements
Selective gold plating	Before the SES process, the panel is coated with additional resist and the selective sites are exposed. These exposed sites are then plated with nickel and gold. The panel will then proceed through the SES process. The HASL step would be omitted.
NPTH in planes without annular ring	A secondary drilling operation will be required to drill the holes, if the hole is to be located through the copper or the hole/feature size is beyond the process capabilities of tenting. This step may occur at the depanelization step.
Scoring	Additional tooling holes may be required to register the panel to the scoring machine blades. Programming of the scoring machine will be required to set the locations and depth of the cuts. This step may occur prior to NC routing of the product.
Countersink holes	The countersinking process is normally performed prior to depanelization.
Gold finger plating	Plating of nickel and gold over the finger area on connectors. This process typically requires shearing/NC routing separation of the panel to place the fingers at the edge of the remaining pieces, taping the product to expose only the fingers, stripping of the solder, plating nickel, and plating gold, and removal of the tape. These operations occur prior to the depanelization process. Finger plated products usually require chamfering of the fingers.
Via plugging	Plugging of vias may occur prior to solder mask or after the HASL operations.
Chamfer	After depanelization of the products from the manufacturing panel, the PCB products are processed through a machine to add an angle to the edge of the part or fingers.
Organic coating	Products requiring organic coating would omit the HASL step and be coated prior to inspection.

19.3.4 Panelization

The selection of the manufacturing panel is one of the most important steps in achieving product profitability. Several factors affect the selection of panel sizes to produce a specific product. These include:

- Material utilization
- Process-specific constraints
- Process limitations

All of these factors will impact the selection of the manufacturing panel size and the profitability of the product.

19.3.4.1 Material Utilization. The material manufacturing costs, which correspond to 30 to 40 percent of total costs, are directly related to the square inches of material processed. The material costs can include the following: laminate, prepreg, copper foil, solder mask, photoresists, drill bits, chemicals, etc. Generally, these materials are consumed relative to the panel area manufactured.

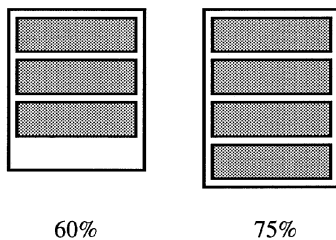


FIGURE 19.3 Selection of the best panel to increase the material utilization of the shippable product.

The appropriate manufacturing panel size (refer to Fig. 19.3) should be selected such that the shippable product consumes the highest percent of the manufactured panel as possible, thus reducing waste material and product costs.

During the design phase of the PCB, the board profile is defined. The impact of designing the profile, which results in poor manufacturing panel utilization, significantly impacts the cost of the product. During the definition of the PCB board profile, the PCB designer should consult with his manufacturing site to provide DFM feedback. Odd-shaped PCB profiles can still result in good material utilization, through nesting of the PCB on the panel.

General limitations in the selection of the appropriate panel sizes include the following: minimum spacing between products for depanelization processes (typically 0.100 in) and minimum panel border-to-product spacing to permit tooling and registration systems (typically 0.600 in).

19.3.4.2 Product Process-Specific Constraints. In the selection of manufacturing panel sizes, there may be processing constraints which restrict the usage of specific panels or limit the method of placing the shippable product on the panel. For example, the manufacturing of products with gold tab plating may require additional spacing between products and restrictions on the rotation and nesting of products. These constraints may force the usage of panel sizes below the optimal material utilization.

19.3.4.3 Process Limitations. Process limitations may require the usage of nonoptimal panel sizes. For example, due to a product's registration requirements, additional tooling may be required, thus reducing the available area for shippable product and reducing the material utilization. Another example is the limitations of some processes to permit larger panel sizes due to the physical processing constraints of the equipment.

19.3.5 Initial Design Analysis

After the initial design review, the basics of the design are known and a decision on whether to manufacture the design has to be made, and if so, the cost must be determined. Once the cost has been determined, the product price can be provided to the PCB customer.

The material and process requirements with the manufacturing panel selection define the basics of the product costs. These factors, combined with the prediction of yield based on the design review, will allow the product to be costed.

19.4 DESIGN INPUT

The input of the design into the CAM system is primarily performed by the loading of Gerber data into the PCB CAM system, after defining the apertures and shapes to be used. Alternatively, most PCB CAM systems accept the IPC-350 format.

The information loaded into the PCB CAM systems includes all artwork layers (e.g., circuitry layers and solder mask layers) and the drill files. Although some PCB CAM systems can accept NC routing files (i.e., board profiling), these files are normally not part of the PCB design system's capabilities to create, and therefore are not provided to the PCB manufacturer.

Prior to the loading of the PCB design files, the aperture codes within the design file need to be related to physical shapes within the PCB CAM system. These shapes are usually round, square, and rectangular, but may also include complex shapes (e.g., thermal relief pads for innerlayer planes). The complex shapes may need to be created on the CAM system from information provided by the PCB designer. The complete definition of these complex shapes by the PCB designer is critical to the success of the resulting design. Incomplete descriptions could result in nonfunctional designs.

During the loading process of the design files, it is important to review any log files and screen messages for missing aperture definitions or damaged design files.

After loading the files, the design should be reviewed for any problems with the interpretation of the data or apertures while aligning the data for further processing.

The PCB CAM operator must ensure that the apertures match those defined by the PCB designer and that the design input process has proceeded without failure, since subtle errors could result in nonfunctional designs that may not be determined within the manufacturing site but only at the PCB customer site.

19.5 DESIGN ANALYSIS AND REVIEW

The design analysis and review step is a refinement of the initial design review performed earlier by focusing on the actual design data. The design analysis and review consists of the following steps:

- Design rule checking
- Manufacturability analysis
- Single image edits
- DFM (design for manufacturability) enhancements

These steps provide the final checking of the PCB design requirements against the capabilities of the PCB manufacturer and preparation of the design for manufacturing.

19.5.1 Design Rule Checking

Design systems lay out circuits to defined rules; however, these systems may fail to adhere to these rules, because of either system failure or manual intervention. The purpose of reviewing the data package, in addition to reviewing the documented design constraints, is to confirm that the product can be produced at the manufacturing site and to the expected manufacturing yields. The analysis of the provided data describing the circuitry of the PCB to the manufacturing facility's production capabilities is critical to the success of the product.

The following information defines typical design rule checking that is performed, the reasons for checking, and the method of checking. Also, see Fig. 19.4.

DRC item	Purpose of check	Method of checking
Drill layer duplicate coordinates	Duplicate drill holes consume manufacturing capacity and may result in broken drill bits.	Most PCB CAM systems have an explicit check for this problem, provided a given radius of tolerance.
Minimum drill hole-to-hole spacing	Drill holes too close may result in broken material between the holes.	Most PCB CAM systems have an explicit feature spacing check.
NPTH-to-board edge minimum spacing	Drill holes too close to the board edge may result in broken material at the board edge.	Merge routing profile onto a copy of the NPTH layer and execute a spacing check of the pad to trace (the route layer would appear as a trace).
Missing drill holes	Missing holes may result in nonfunctional designs or products that cannot be used due to missing mounting or tooling holes.	Pad registration checking against a circuit reference layer, with verification of pads without PTH holes.
Extra drill holes	Extra drilled holes could result in nonfunctional designs from cutting of traces or shorts (if the holes become plated).	Pad registration checking against a circuit reference layer, with verification of PTH holes without pads.
Board edge-to-board edge minimum spacing	Routing cuts leaving minimal material may result in fractured or broken material between routing.	Feature spacing checking of the routing layer as a design layer, without considering electrical connectivity.
Copper-to-board edge minimum spacing	Tolerances of the routing operation may result in copper exposed at the edges of the product.	Feature spacing checking of the routing layer as a design layer against the circuitry layer.
Minimum annular ring	Most customer specifications define the minimum acceptable annular ring on PCB products. Annular ring is dependent on the design and the registration tolerances of the manufacturing processes.	Most PCB CAM systems have an explicit annular ring check.
Minimum pad-to-pad spacing	In addition to feature spacing and potential shorts, pad spacing affects the capability to electrically test products.	Most PCB CAM systems have an explicit feature-to-feature spacing check.
Minimum pad-to-track spacing	Feature spacing below manufacturing capabilities could result in shorts and poor product yields.	Most PCB CAM systems have an explicit feature-to-feature spacing check.
Minimum track-to-track spacing	Feature spacing below manufacturing capabilities could result in shorts and poor product yields.	Most PCB CAM systems have an explicit feature-to-feature spacing check.
Copper-to-NPTH minimum spacing	NPTH locations too close may prevent proper tenting of holes and require secondary drilling operations. NPTH too close may result in damaged features (e.g., cut traces).	Feature minimum spacing against a copy of the design layer merged with the NPTH layers.
Minimum line width	Line widths below manufacturing capabilities could result in poor product yields.	Some PCB CAM systems have an explicit check for this problem, others may require review of the apertures used and highlighting of the apertures for visual inspection.
Track termination without pad	Although this may be design intent, missing pads may be the result of poor design information or loading failures. These problems can result in nonfunctional designs.	Most PCB CAM systems have an explicit check for this problem.
Pad stack alignment	Misaligned pad stacks may result in unpredictable annular ring results, incorrect compensations of registration, and product scrap.	Most PCB CAM systems have an explicit pad registration check.

DRC item	Purpose of check	Method of checking
Minimum solder mask pad clearance	Solder mask pad clearances below manufacturing capabilities could result in solder mask on the pads, and poor product yields.	Check solder mask layer (as a circuit layer) against the circuitry layer using an annular ring check.
Minimum solder mask edge-to-feature spacing	Solder mask edge to feature spacing below manufacturing capabilities could result in exposed features and poor product yields.	Feature minimum spacing against a copy of the design layer merged with the solder mask layer.
Minimum solder mask annular ring for NPTH	NPTH solder mask clearances may need to be larger to prevent ghosting of the solder mask from light diffraction through the product.	Check solder mask layer (as a circuit layer) against the NPTH layer, verify for no matches of the NPTH to solder mask layer.
Solder mask-to-board edge minimum clearance	PCB edge clearances may need to be larger to prevent ghosting of the solder mask from light diffraction through the product.	Feature minimum spacing against a negative copy of the solder mask layer merged with the routing layer.
Solder mask minimum web	Solder mask webs below manufacturing capabilities could result in solder mask breakdown and poor product yields. This problem potentially results in the PCB assembler with solder bridging defects.	Use minimum feature-to-feature spacing on the solder mask layer.
Plane-to-board edge minimum clearance	Spacing below the tolerances of the routing operation may result in copper exposed at the edges of the product.	Feature spacing checking of the routing layer as a design layer against the plane.
Minimum plane layer annular ring	Annular ring below manufacturing registration and tolerance capabilities could result in open connections, and poor product yields.	Most PCB CAM systems have an explicit annular ring check.
Minimum plane layer clearance	Plane clearances below manufacturing registration and tolerance capabilities could result in shorts and poor product yields.	Most PCB CAM systems have an explicit annular ring check.
Plane-to-plane isolation	Plane layer isolation typically results from either incorrect designs or interpretation of aperture lists; the result is a nonfunctional product.	Most PCB CAM systems have an explicit layer-to-layer isolation check.
Nomenclature-to-NPTH minimum spacing	Nomenclature spacing below manufacturing capabilities may result in nomenclature in the hole from misregistration or bleeding of the nomenclature ink.	Feature minimum spacing against a copy of the nomenclature layer merged with the NPTH layers.
Nomenclature-to-solder mask minimum spacing	Nomenclature spacing below manufacturing capabilities may result in nomenclature on product features from misregistration or bleeding of the nomenclature ink.	Feature minimum spacing against a copy of the nomenclature layer merged with each solder mask layer.
Nomenclature-to-feature minimum spacing	Nomenclature spacing below manufacturing capabilities may result in nomenclature on product features from misregistration, bleeding of the nomenclature ink or skipping of the screen over features resulting in illegible markings.	Feature minimum spacing against a copy of the design layer merged with the nomenclature layer.
Nomenclature minimum feature sizes	Nomenclature sizes below manufacturing capabilities could result in illegible nomenclature and poor product yields.	Some PCB CAM systems have an explicit check for minimum line width, others may require review of the apertures used and highlighting of the apertures for visual inspection.

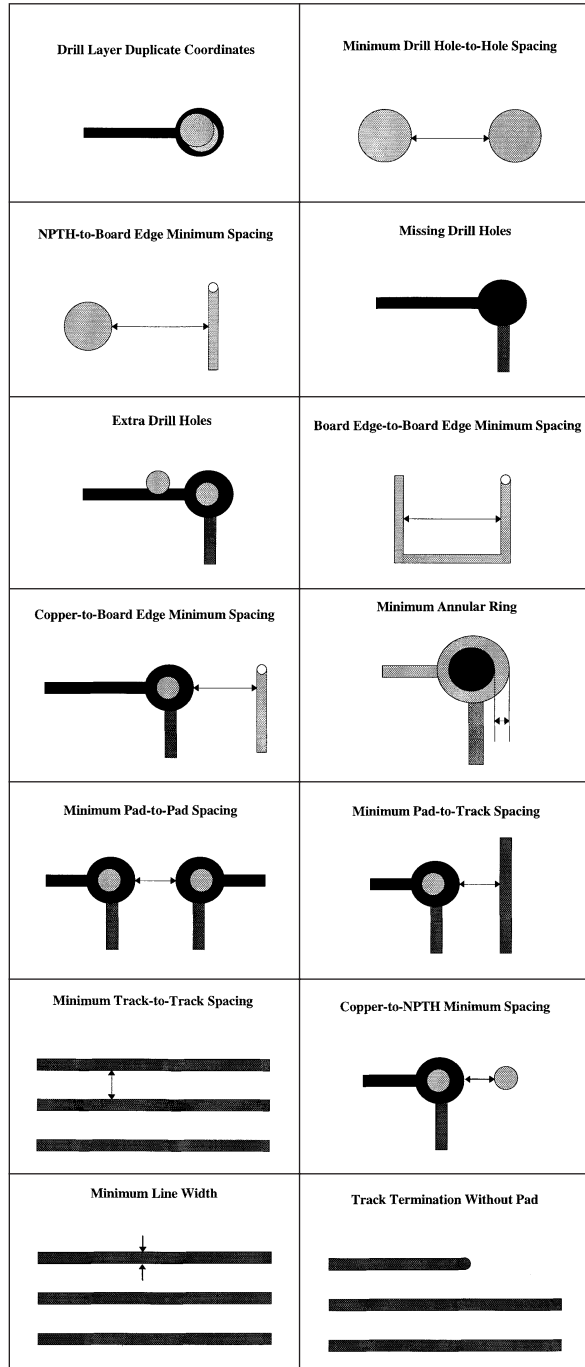


FIGURE 19.4 Examples of design rule check elements.


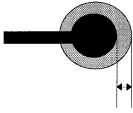
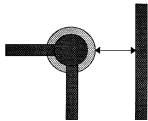
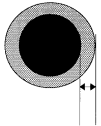
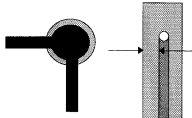
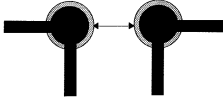
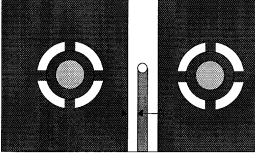
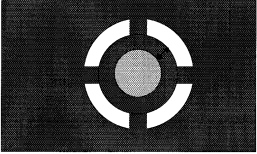
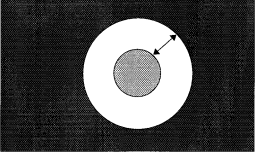
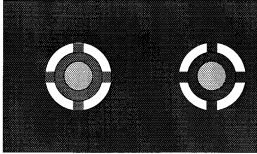
<p style="text-align: center;">Pad Stack Alignment</p> 	<p style="text-align: center;">Minimum Soldermask Pad Clearance</p> 
<p style="text-align: center;">Minimum Soldermask Edge-to-Feature Spacing</p> 	<p style="text-align: center;">Minimum Soldermask Annular Ring For NPTH</p> 
<p style="text-align: center;">Soldermask-to-Board Edge Minimum Clearance</p> 	<p style="text-align: center;">Soldermask Minimum Web</p> 
<p style="text-align: center;">Plane-to-Board Edge Minimum Clearance</p> 	<p style="text-align: center;">Minimum Plane Layer Annular Ring</p> 
<p style="text-align: center;">Minimum Plane Layer Clearance</p> 	<p style="text-align: center;">Plane-to-Plane Isolation</p> 
<p style="text-align: center;">Nomenclature-to-NPTH Minimum Spacing</p> <p style="text-align: center;">U38 → ●</p>	<p style="text-align: center;">Nomenclature-to-Soldermask Minimum Spacing</p> <p style="text-align: center;">U38 → ●</p>
<p style="text-align: center;">Nomenclature-to-Feature Minimum Spacing</p> <p style="text-align: center;">U38 → ●</p>	<p style="text-align: center;">Nomenclature Minimum Feature Sizes</p> <p style="text-align: center;">U38</p>

FIGURE 19.4 (Continued) Examples of design rule check elements.

19.5.2 Manufacturability Review

The design rule checking results are reviewed against the PCB manufacturing capability matrix, with dispositions being made to the acceptability of violations. The PCB customer may be contacted regarding design violations for corrected designs to be retransmitted or to permit design changes to be performed by the PCB manufacturer.

Additionally, the factors reviewed during the initial design review are revisited to confirm the results based upon design actuals. If differences are noted, product yields and/or cost predictions may need to be altered.

19.5.3 Single Image Edits

In general, all designs have information which must be removed before the design can be manufactured. Most of these features are used as references during the PCB layout phase to assist the designer in understanding the available real estate. However, these references must be removed or modified to manufacture the board to design intent. Among these typical items are the following:

Item	Impact if not removed/modified
NPTH pads on outerlayers	Plated holes
Routing crop marks	Copper at board edge
Direct contact plane markings of drilled holes	Open circuits
Clip nomenclature with solder mask data	Nomenclature on pads/in holes

19.5.4 DFM Enhancements

The manufacturability of the PCB can be improved through design enhancements. Many of the enhancements are capable of being performed on CAD systems only (or through specialized post-CAD software); other enhancements can be performed at the manufacturing site.

Ideally, all design enhancements would be performed at the design site to maintain the consistency of products from multiple manufacturers; however, some of the DFM enhancements can impact the CAD system's capability to provide further design changes. Specifically, the optimization of track-to-feature spacing could result in blocked routing channels, preventing an autorouter from performing. The following are typical DFM enhancements (see Fig. 19.5a–e):

Item	Result from enhancement
Copper balancing	The balancing of copper on the surface of the design will improve the distribution of plating (in the pattern-plating process), preventing areas of overplating due to the isolation of features. This improvement is noticed in the ability to control the plated hole diameters to tolerances and the reduction of plating heights which impact solder mask coverage. (See Fig. 19.5a.)
Teardropping	As annular rings decrease due to tighter design specifications, the capability to have defects due to registration/annular ring failures increases. The addition of teardropping increases the pad-to-trace junction size, improving the reliability of

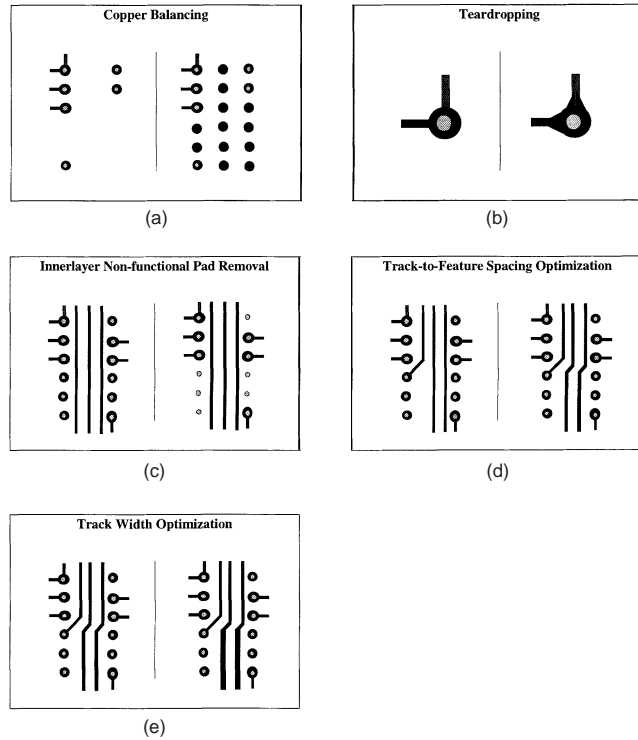


FIGURE 19.5 Typical DFM enhancements.

- Removal of nonfunctional pads on innerlayers

Innerlayer shorts result from several factors, among them the proximity of circuitry to other circuitry. The occurrence of shorts can be correlated to the running length of the circuitry versus the running length at some minimum spacing between circuitry (at the process capability of the operation). Reducing the total distance at minimum spacing will reduce the probability of shorts, resulting in improved yields. (See Fig. 19.5c.)
- Optimization of track-to-feature spacing

As defined in the nonfunctional pads discussion, the occurrence of shorts can be correlated to the running length of the circuitry versus the running length at some minimum spacing between circuitry (at the process capability of the operation). Reducing the total distance at minimum spacing will reduce the probability of shorts, resulting in improved yields. (See Fig. 19.5d.)
- the connection and improving product yields. (See Fig. 19.5b.)

Track width optimization

The occurrence of opens can be correlated to the running length of the circuitry versus the running length at minimum track width (at the process capability of the operation). Reducing the total distance at minimum track width will reduce the probability of opens, resulting in improved yields. (See Fig. 19.5e.)

19.6 PANELIZATION PROCESS

The panelization process results in the placement of the PCB single images in the locations defined in the panelization definition step during the initial design review. In addition to the placement of the PCB single images, the following features may be added to the panel for manufacturing.

Item	Purpose
Customer coupons	Provided to the customers for their inspection of the PCB product. These coupons are typically treated as separate PCB single images.
Internal coupons	During the manufacturing process, destructive testing may be required to confirm product quality. These coupons are typically treated as separate PCB single images.
Tooling holes	Holes used for registration of manufacturing panels to artwork, drilling, or routing operations are added to the panel borders.
Outerlayer thieving patterns	Plating thieving patterns are sometimes needed to balance the platable area across the panel.
Innerlayer venting patterns	The addition of copper to innerlayers outside the PCB single image provides for more consistency in the lamination process.
Textual markings	Operator aides may be provided outside the PCB single image to improve manufacturing quality.

19.7 ADDITIONAL PROCESSES

An important step in all the processes already defined is the necessity for proper information management. Archival of PCB customer-supplied information and the files generated from the tooling process are critical for disaster recovery. Archiving systems exist in the market that also provide for centralized data management and distribution of the information to the various departments within an organization.

In addition to the basic tooling steps already defined, most PCB manufacturing sites require the following information to be created:

- CAD reference files
- Electrical test netlist and fixture creation files

The CAD reference files, if required, are generally created within the PCB manufacturing site. These files prepare *automated optical inspection* (AOI) systems with data that confirms that manufactured pieces match, within tolerances, the PCB design.

Electrical test netlist files may be either created internally or sent to an outside contractor to prepare the files and the resulting fixture. These files use as input either the Gerber or IPC data provided by the PCB customer.