01 Jan 1995

Automatic PCB Inspection Systems

M. Moganti

Fikret Erçal
Missouri University of Science and Technology, ercal@mst.edu

Recommended Citation

M. Moganti and F. Erçal, "Automatic PCB Inspection Systems," *IEEE Potentials*, Institute of Electrical and Electronics Engineers (IEEE), Jan 1995.
The definitive version is available at https://doi.org/10.1109/45.464686

Automatic PCB inspection systems

Computer vision makes inspecting circuit boards for defects easier

There are more than 50 process steps required to fabricate a printed circuit board (PCB). To ensure quality, human operators simply inspect the work visually against prescribed standards. The decisions made by this labor intensive, and therefore costly, procedure often also involve subjective judgements.

Automatic inspection systems, however, remove the subjective aspects and provide fast, quantitative dimensional assessments. This means better quality at a lower cost. Due to the following criteria, automated visual inspection has become a part of the manufacturing environment:

- They relieve human inspectors of the tedious jobs involved.
- Manual inspection is slow, costly, leads to excessive scrap rates, and does not assure high quality.
- Multi-layer boards are not suitable for human eyes to inspect.
- Quality standards are so high that sampling inspection is not applicable.
- Production rates are so high that manual inspection is not feasible.
- Tolerances are so tight that manual visual inspection is inadequate.
- Configuration management and defect tracing require computer help.

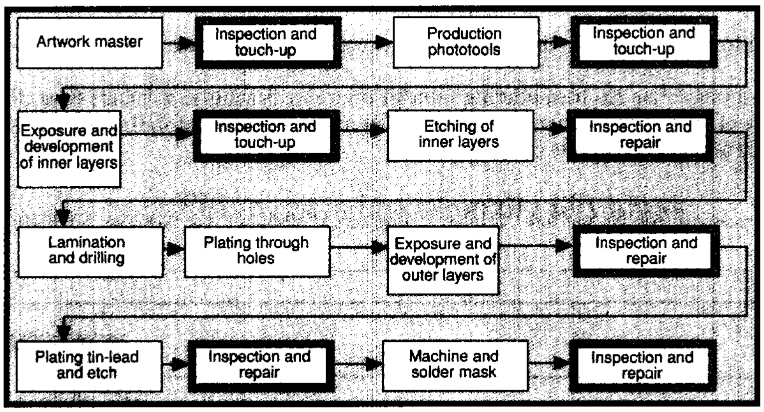


Fig. 1 Stages in multilayer PCB fabrication

Types of inspection

PCB flaw detection procedures can be broadly divided into two classes: electrical/contact methods and non-electrical/non-contact methods. Even though many design parameters can be successfully checked by electrical test, it has limitations that could allow defective products to pass. Potential defects such as line width or spacing reductions are not detected, nor are cosmetic defects or those caused by process problems. Defects such as copper on an inner layer, which may cause failure of the final board, are also missed.

Further, electrical testing is very setup-intensive—as boards come to be designed on grids of less than 0.1 inches, the fixtures necessary for testing become extremely complicated and expensive. Electrical testing, therefore, augments visual inspection but cannot replace it. The double lined boxes in Fig. 1 designate stages at which electrical testing can, in principle, be applied. Some of the non-contact automatic inspection methods that are currently available in industries are:

Automatic visual/optical inspection. Automatic optical inspection systems detect the same type of surface related defects as manual inspection. Optical testers can find bare board defects other than shorts, and opens, such as line width errors, pad mousebites, and trace misplacements. Automatic optical inspection (AOI) has the following characteristics that the contact testing (electronic testing) does not have:

- It recognizes potential defects such as out-of-specs, line widths, line spacing, voids, pin holes, etc. These are not always traceable by contact testing methods. Narrow lines burn up over long periods or under fairly large currents. Also, in high frequency circuits, these defects may cause leakage, parasite capacitance, impedance or mutual inductance.
- An AOI system is not confined by a design grid during inspection—unlike most electrical testing equipment.
- AOI can inspect artwork and provides strict product control from the onset of production.

X-ray imaging. X-ray imaging is used for rapid and precise measurements of multilayer PCBs. Based on the measurements of individual test pads, the system supplies specific information on layer registration, distortion and the torsion of the layers. X-rays also reveal minute defects, such as hairline cracks, which escape other methods of inspection. SMT (surface mount technology) defects such as heel cracking, voids, component misalignment, bridging, insufficient solder, excess solder, solder threads and balls, poor wetting, and bent leads can be detected using X-rays.

Scanned-beam laminography. Laminography provides cross-sectional X-ray imaging which separates the top and bottom sides, or any other layer of the PCB, into cleanly separated images. The basic principle of laminography is to move the X-ray source and the X-ray image detector around on opposite sides of the object. As long as the X-ray beam always passes through the same points in the object and the same points in the detector simultaneously, a cross-sectional image is formed in real-time. By changing the size of the X-ray scanning circle, the fields of view and magnification of the image can be varied on the fly. This enables inspection of fine-pitch components at high magnification, and of other components at normal magnification, to optimize throughput.

Madhav Moganti & Fikret Ercal

0278-6648/95/$4.00

Ultrasonic imaging. Ultrasonic imaging technology best detects solder-joint defects such as internal voids, cracks, and disbands. Defects or the juxtaposition of dissimilar materials make their presence known by reflecting (echoing) the high-frequency pulses. In practice, this system is limited to simple solderjoint geometries.

Thermal imaging. Thermal imaging systems indicate hot spots on operating PCBs indicating shorts and overstressed components. Compared to optical and X-ray systems, thermal inspection systems are less automated.

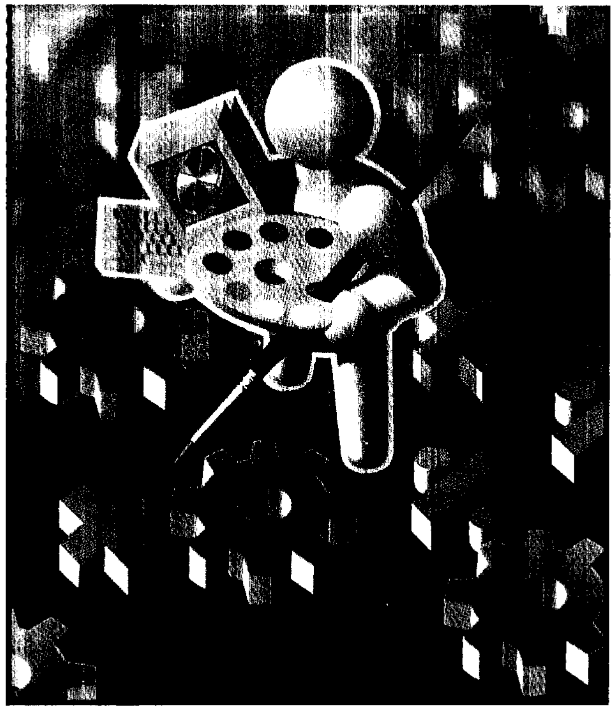
Stages in multi-layer PCB fabrication

In the multi-layer PCB fabrication, several process steps, where quality control is essential to the final product, are inspected and defects corrected. Figure 1 depicts these fabrication stages along with the different points where inspection could be applied. Except for the final inspection task, which detects cosmetic defects, all the others are crucial in the fabrication process.

1) Artwork masters: These are silver halide 1:1 scale transparencies of the conductor pattern on film. Artwork masters are produced in most cases by computer driven photoplotters. Defects on the master will directly affect all ensuing production batches of the given PCB. Therefore, a great deal of work is usually put into the incoming inspection and touch-up of artwork masters. The most common defects on artwork masters are caused by scratches and dust particles. These show up on the finished product as opens, shorts, and pinholes or copper splashes.

2) Phototools: These are silver halide or diazo transparencies obtained by contact printing from the artwork master. They are used for the actual exposure of the boards. The inspection needs and defects of artwork masters applies also to phototools.

3) Inner and outer layers after exposure and development: These are sheets of copper-clad laminate, overcoated with photoresist, with the conductor pattern exposed on it. Many defects found after etching are the defects that are already present in the photoresist pattern. Inspection at this stage is beneficial because it allows touch-ups, or stripping and repeating the photoresist process. Defects caused by dust and other foreign particles during exposure can be added to the list of defects mentioned for artwork and phototools.



© T. Lochray Inc., The Image Bank

4) Inner layers after etch and strip: This is the point at which most of the visual inspection is invested in the multilayer manufacturing cycle. This is the last inspection stage before lamination. After this point, a defective inner layer in the multilayer board is not repairable. Defects previously mentioned can appear at this stage. Also, defects such as over- and under-etch, which lead to narrow conductors or spacings and excess copper, can appear.

5) Outer layers after etch: As in the case of inspection of the inner layers after etch, this is the last point at which repairs can be made on the conductor pattern. The AOI problem is different at this stage because of the appearance of holes and the necessity of inspecting their annular rings for width and breakout.

6) Inspection after machining and solder masking: This is mainly a cosmetic final inspection. Defects in the conductor pattern can hardly be detected at this stage because the solder mask obscures the conductors.

Defects

A variety of defects can afflict the copper pattern of PCBs; however, they all do not mean immediate rejection of the board. The types of faults range from hair-line (e.g., size equal to 100 microns) breaks and bridges as small as one mil (0.001 inches) between conductor paths, to unacceptable enlargements and reductions in line widths, to poorly formed plated through holes.

Figure 2 shows an artificial defect free PCB image pattern. This figure depicts through hole PCB patterns, printed wiring board patterns, and surface mount PCB patterns in the same image. Because most of the defects are common to all three varieties of boards, the three different patterns are shown in one example image.

Figure 3 shows the same image pattern as in Fig. 2 with a variety of defects shown in it. Though each defect shown in the figure is a representative example for that particular defect, the shape and size of the defect varies from one occurrence to another. Smaller and smaller lines, and spaces make these defects more serious, more likely and harder to detect. Studies show that open/partial open, short, pinhole, breakout, overetch, underetch are the most frequent defects that occur. These defects are caused due to one or more of the following errors:

- thermal expansion of the artwork during printing, or by defective etching,
- dirt on board, air bubbles from electrolysis,
- incorrect electrolysis timing,
- mechanical misregistrations, and
- distortions due to warping, etc.

Components involved

A typical inspection process involves observing the same type of object repeatedly to detect anomalies. The process involves digitization of the object to be inspected for visual data. The analysis involves the processing of the image to enhance relevant features and the detection of defects. Industrial PCB visual inspection ideally requires a cost-effective off-the-shelf system. This means it should be designed to take into account operation speed, reliability, ease of use, and modular flexibility. This way the system can be adapted to different inspection tasks. The main hardware components of the inspection system are the material and component handling system, illumination system, image acquisition system, and the processor.

Material and component handling system. This system comprises the mechanism which presents the part or assembly, denoted material, in different orientations to the components of the automated visual inspection system.

Illumination system. Suitable lighting and viewing conditions facilitate inspection, avoiding the need for complex image processing algorithms. The main parameters that characterize the suitability of an illumination system are: a) *intensity*, b) *uniformity*, c) *directionality*, and d) *spectral profile.*

The relative importance of these parameters and the degree to which each one must be controlled are largely governed by the surface characteristics of a given PCB and the constraints imposed by the camera. Among the illumination techniques, the most commonly used are: standard light sources, indirect and back lighting, fluorescent lighting, reflected lighting, diffuse illumination, fiber-optic, quartz-halogen light sources, and such.

Image acquisition system. Images are usually acquired by use of a camera or a digitizer that acts as a sensor. There are several types of cameras available; the appropriate type is dictated by use. Examples of different types are the television camera (a charged coupled device (CCD) camera) and the laser scanner camera. AOI System Corp. developed the AOI-20 system that utilizes as many as 20 CCD cameras. The MOP 5002 system operates with one or two high precision lenses cameras which scan the PCB image through linear CCD sensors. These high precision lenses guarantee the maximum possible resolution and a wide depth of field.

Processor. The processor system usually consists of a high speed computer system. Usually, most of the commercially available systems have special processors designed solely for inspection purposes. A commercially available inspection system, AOI-20, uses a high speed parallel processing system.

Algorithms

A large number of PCB inspection algorithms have been proposed in the literature to date. Figure 4 shows the classification of these algorithms. In general, they fall into one of three categories: reference comparison (or referential approaches), non-referential approaches, and hybrid approaches.

The *referential* methods execute a real point to point (or feature to feature) comparison whereby the reference data from the surface image of a "good" sample is stored in an image database. These methods detect errors such as missing tracks, missing termination, opens, shorts, and so forth.

Image subtraction is the simplest and most direct approach to the PCB inspection problem. The board is scanned and its image is compared against the image of an ideal part. Figure 5 shows this comparison process a logical XOR operation on the subimage patterns of the PCB. The advantage of this method is that it is trivial to implement in specialized hardware. Therefore high pixel rates can be obtained. Another advantage is that it allows for verification of the overall defects in the geometry of the board. This technique suffers from many practical problems, including registration, color variation, reflectivity variation, and lighting sensitivity.

Feature matching is an improved form of image subtraction. In this method, the extracted features from the object and those defined by the model are compared. The advantages are that it greatly compresses the data for storage, reduces the sensitivity of the input data and enhances the robustness of the system. The disadvantages are that it requires a large data storage for the ideal PCB patterns, and precise registration is necessary for comparison. Also, the method is sensitive to illumination and digitization conditions, and lacks flexibility.

Model-based methods perform inspection by matching the pattern under inspection with a set of predefined models. These involve graph matching methods, syntactic methods, and tree matching methods. These matching methods are based on the structural, topological, and geometric properties of the image. The idea is based on the topological/structural comparison. The standard model obtained from the conductor and insulator image patterns of the reference PCB are compared with those of inspection boards.

Non-referential methods do not need a reference pattern. They work on the idea that a pattern is defective if it does not conform with the design specification standards. These methods are also called design-rule verification methods or generic property verification methods. They basically use the design-specification knowledge in verifying the board to be inspected. Applying the design-rule verification process directly to the image patterns is a time consuming process, hence the response time of the system decreases. Usually these methods process/transform the image into a form which reduces the verification time.

Ejiri of Hitachi Limited developed the classic *expansion-contraction* technique that assumes defects exist in a high first-order spatial-frequency domain (viz., patterns that are small relative to the acceptable patterns). Expansion-contraction methods employ morphological operations such as erosion, dilation, expansion, contraction, and thinning in the pre-processing stage.

Design specification information is embedded so that transformations generate images that can be easily interpret-

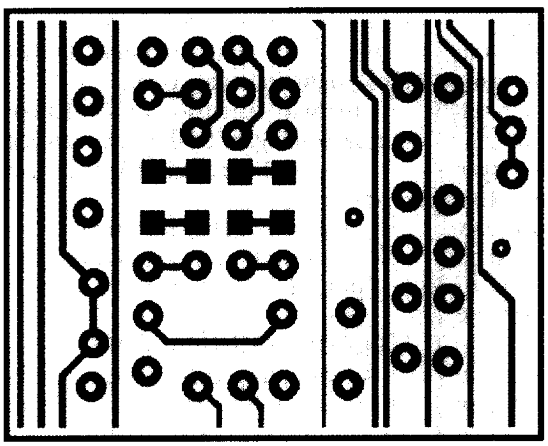


Fig. 2 Good PCB patterns

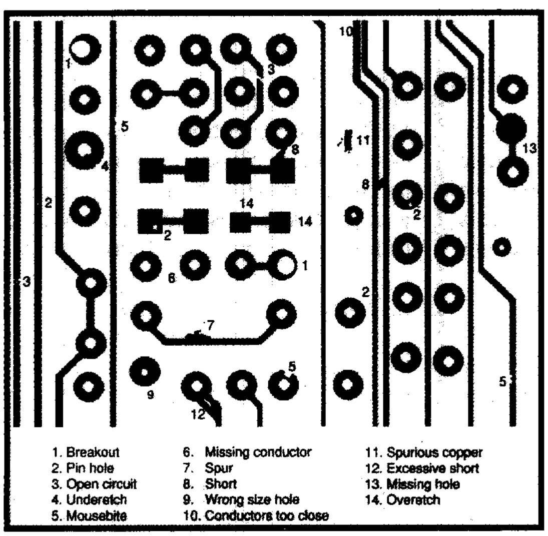


Fig. 3 Defective PCB patterns

ed for defects. Encoding techniques also transform the image patterns. The verification phase interprets these transformed patterns by extracting the topological features and imposing localized constraints such as minimum or maximum widths to detect anomalies.

The advantage of non-referential methods is that speed is maximized and computer storage requirement is minimized. The disadvantage is that they identify only some kinds of defects, such as verifying widths and spacing violations. They may miss flaws that do not violate the rules, such as shorts that are identical to conductors. Also, another drawback of this inspection is that it requires the standardization of the conductor trace types.

The *hybrid* flaw-detection techniques use both referential and design-rule techniques. Thus, they exploit the strengths while overcoming the weaknesses of each of the methods. These methods cover a large variety of defects compared to either referential or non-reference methods alone.

For example, most of the design-rule verification methods are limited to verifying minimum conductor trace and land widths, spacing violations, defective annular ring widths, angular errors and spurious copper. Printed circuit board errors which do not violate the design rules are detected by reference comparison methods. These methods can detect missing features or extraneous features such as isolated blobs. The design rule process detects all defects within small and medium sized features; the comparison methods are equally sensitive right up to the largest features.

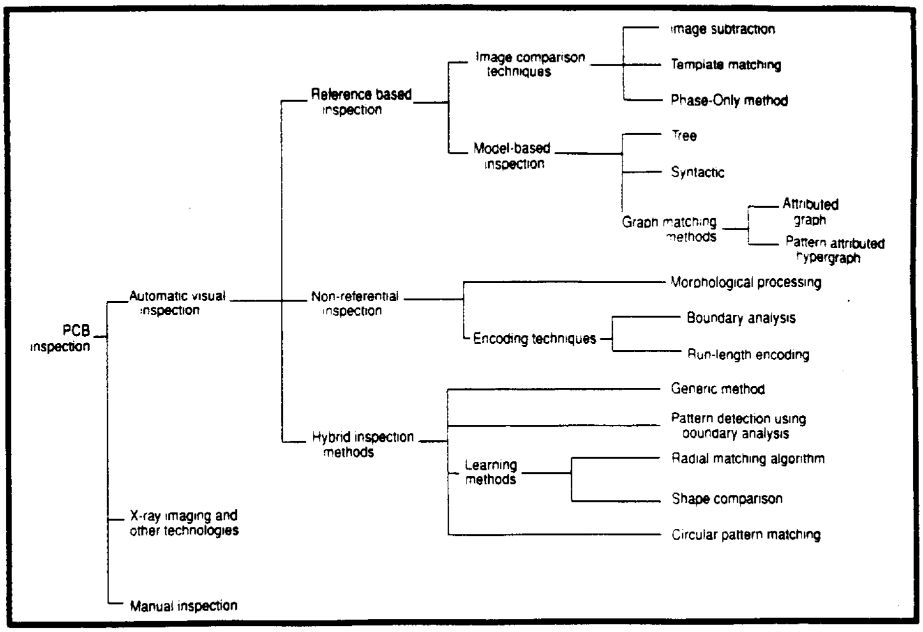


Fig. 4 Classification of inspection algorithms

Commercial systems

Many factors must be considered in designing a commercial inspection system: hardware, software, system throughput, versatility, and reliability. Versatility refers to the number of different inspections the system can perform. Some commercial systems run the gamut from inspecting holes and measuring dimensions to inspecting complete bare boards. Some can make exact measurement of board features or perform inspection in-line with the production process. For manufacturing, the most complete (and most expensive) systems can execute all these functions. The following lists capabilities and features a typical commercial optical PCB inspection system is expected to have:

• System capability: 1) Minimum flaw that can be repeatedly detected at the stated escape rate—*2.0 mil.*; 2) Scan rate—*4.0 square feet / minute*; 3) Panel through-put—*inspect both sides* of 18 x 24 *inch panel (85% active) including setup, loading, scanning, and unloading at a rate of 40 panels/hour*; 4) Typical pixel size—*1.0 mil.*; 5) False alarm rate (fail good product)—*less than 2.0 per square foot*; 6) Escape rate (pass bad product)—*less than 1.0 per 100 square feet (depends on defect criteria)*; 7) Gagging capability (where specified)—*measure feature size to ±1.0 mil.*

• Typical dimensions of panels to be inspected: 1) Panel dimension—20" x 26"; 2) Scan area—18" x 24"; 3) Nominal conductor width—*5 mil.;* 4) Nominal conductor spacing— *5 mil.;* 5) Pad size—*round or rectangular pads of dimension between 3 and 10 mil.;* 6) Conductor via hole diameter size—*0.5 mil or larger.*

• Types of panel to be inspected: 1) Conductor layout—*all possible line orientations and power/ ground layers;* 2) Photoprinted boards —*all commercial photoresist types;* 3) Inner-layer metalization —*drilled and undrilled PCBs in copper technology;* 4) Artwork—*most forms including silver-halide and diazo on both mylar and glass substrate;* 5) Finished boards—*without solder and prior to solder mask;* 6) Substrates—*FR$_4$, polymide and other common substrate material.*

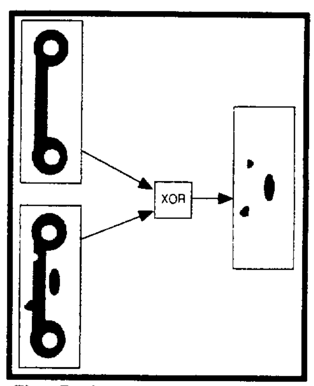


Fig. 5 Image subtraction

• Types of defects to be inspected:

1) Voids—*any void in a conductor that exposes bare substrate material and exceeds 5% of the design width*; 2) Shorts—*Any short with a width in access of 0.001 mil at any point*; 3) Opens—*Any conductor open exceeding 0.01 mils in width*; 4) Spacing—*Any metalization that reduces the space between conductor by more than 5% of design spacing*; 5) Extraneous metal—*Any isolated spot whose area exceeds 2 square mils.;* 6) Artwork—*Any defect violating the above rules for voids, spacing, or extraneous metal; as well as any pinhole in excess of 0.5 mil.*

Table 1 lists the main products available on the market. Most systems use the hybrid inspection techniques—design rule checking and comparison methods—jointly. These systems can test more items with greater accuracy than before. To improve image quality, makers use different kinds of illumination from reflected light, transmitted light and fluorescent light from multiple light sources. Processing speeds continue to accelerate.

Some makers now adopt multi-processing systems. AOI System Corporation developed the AOI-20 product, which employs as many as 20 CCD cameras and performs parallel pro-

Best Copy Available

System	Inspection Methods	Image System	Resolution	Scan Rate	Features/Benefits
AOI System AOI-20	Design Rule Checking (8 kinds of detection sensors) and Comparison method	20 CCD Cameras Reflection/Transmission lighting	1 mil	6.00 sq. ft/min	Continuous operation is possible through the use of conveyor system
Mania MOP-5002	Simultaneous use of Design Rule Checking and Image Comparison	Two CCD Cameras Halogen Lamp Lighting	1 mil	6.00 sq. ft/min	Menu driven user friendly software for easy and fast setup. Fast unit under test change over using patented vacuum adaptor system
Dai-Nippon Screen OPI-5220	Design Rule Checking and Comparison method	LED light, CCD line sensor Reflection/Transmission lighting	1 mil	19.20 sq. ft/min	Complete Comparison Inspection Inspection function of product with special shape
Shin-Nippon Steel PT-2130	Design Rule Checking and Comparison method	Halogen Lamp, Multi-directional illumination Speedy CCD Camera	1 mil	33.33 sq. ft/min	Continuous Variable Resolution (0.2 to 1 mil) Fastest Speed
Orbotech PC-1411	Design Rule Checking and Comparison method (Golden Board or CAD download)	Reflective and Diffusive Omni lighting	Fixed Resoution 0.5 mil	18" x 24" panels 45 sides/hour	Low cost startup, On-line verification
Orbotech PC-1450	Design Rule Checking and Comparison method (Golden Board or CAD download)	Reflective and Diffusive Omni lighting	Variable Resoution 0.25-0.9 mil	18" x 24" panels 38-160 sides/hour	3-10 mil line width technology
Orbotech PC-1490	Design Rule Checking and Comparison method (Golden Board or CAD download)	Reflective and Diffusive Omni lighting	Variable Resoution 0.20-0.5 mil	18" x 24" panels 45-130 sides/hour	3-6 mil line width technology for high volume PCB shops
Orbotech V-309i/x	Design Rule Checking and Comparison method (Golden Board or CAD download)	Flourescent technology (Blue Laser)	Variable Resoution 0.4-1.0 mil	18" x 24" panels 77-180 sides/hour	3-10 mil line width technology for high volume PCB shops
Orbotech Vision Blaser	Design Rule Checking and Comparison method (Golden Board or CAD download)	Flourescent technology (Blue Laser)	Variable Resoution 0.25-0.5 mil	12" x 12" panels 80-180 sides/hour	2-4 mil line width technology for high volume PCB shops

Table 1 Commercially available bare PCB inspection systems

cessing. Even a slow system with 1 *mil* resolution attains a processing speed of 6.00 ft^2/min; fast systems reach 33.33 ft^2/min. Converting this value to a pre-pixel speed reveals an astonishing speed of 10 *ns/pixel*.

With progress in diminishing pattern thickness, developers have improved resolution. The 1 *mil* resolution of the early days now has reached 0.2 *mil*. In addition to handling testing processes, these systems now display defective locations, components, incorporate correction machines and accommodates computer-integrated manufacturing (CIM).

Summary

Utilizing advances, machine vision may answer the manufacturing industry's need to improve product quality and increase productivity. The major limitation of existing inspection systems is that all the algorithms need a special hardware platform to achieve the desired real-time speeds. This makes the systems extremely expensive. Any improvements in speeding up the computation process algorithmically could reduce the cost of these systems drastically. However, they remain a better option than increasingly error prone, and slow manual human inspection.

Also, forefront in the challenges confronting the automated visual inspection research is the development of generic inspection equipment. Hardware and software that is capable of handling a wide variety of inspection tasks. Systems in the future will be easier to operate.

Read more about it

• Madhav Moganti, Fikret Ercal, Cihan H. Dagli, and Shou Tsunekawa, "Automatic PCB Inspection Algorithms: A Survey," to appear in *Computer Vision and Image Understanding Journal* (also as *Csc-93-27 Technical Report*, Dept. of Comp. Sci., Univ. of Missouri-Rolla.)

• Eduardo Bayro-Corrochano, "Review of Automated Visual Inspection 1983 to 1993 - Part I & II," *SPIE-Intelligent Robots and Computer Vision XII*, Vol. 2055, pp. 128-172, 1993.

• Hisashi Tsunekawa, "Latest Image Evaluation Systems Aid Efforts for Product Quality," *Journal of Electronic Engineering*, No. 306, Vol. 29, pp. 72-77, June 1992.

About the authors

Madhav Moganti is a doctoral student in the Department of Computer Science at the University of Missouri-Rolla.

Dr. Fikret Ercal is currently an associate professor at the University of Missouri-Rolla, Computer Science Dept.