Scanning for Electronic Data

By Ken Rockette

New electronic data for manufacturing can be created out of old artwork.

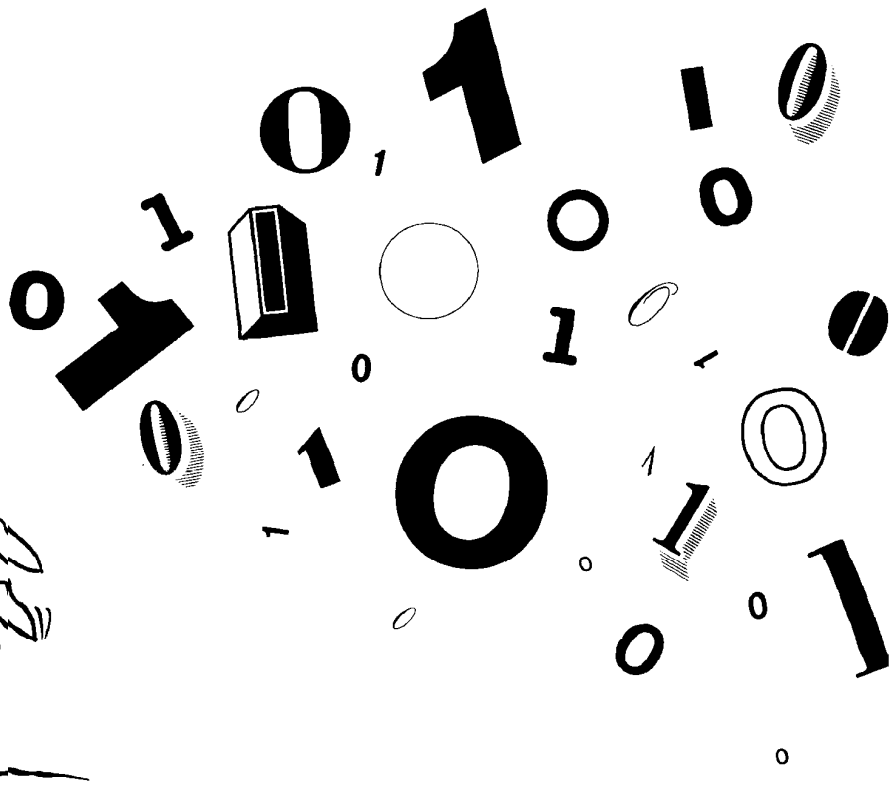
One by-product of the thrust toward computer-integrated manufacturing (CIM) in the industrial workplace is electronic data. As companies slowly but steadily retool, the demand for electronic data becomes a problem. New production equipment, now used to produce old products, needs electronic documentation that doesn't exist. It must be generated. Scanning has become an important tool in creating the required data.

A good example of this situation is the procurement of spare parts used on systems for which manufacturing information doesn't exist or is incomplete. Infinite Graphics Inc. (IGI, Minneapolis, MN) has recently handled a similar job supplied by a major aerospace customer. The only documentation available on PC fabrication consisted of very old and well-used aperture cards.

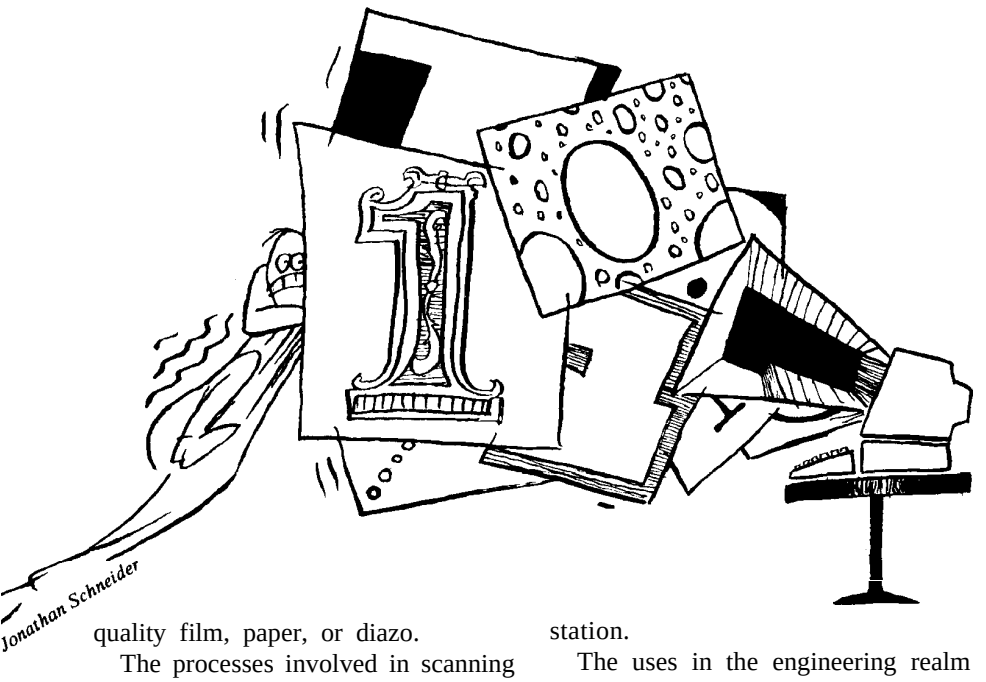
In normal circumstances, a complete redesign would have been required. A combination of scanning and reverse engineering resulted in the delivery of 600 PCB layers with complete manufacturing data within four weeks—at less than one-third the cost of a redesign. In many situation scanning can reduce costs and in prove turnaround time.

WHAT IS SCANNING?

Scanning is the use of optoelectric technology to create a digital database out of existing PCB artwork. The image can be produced on high-



Jonathan Schneider



quality film, paper, or diazo.

The processes involved in scanning are numerous. The procedure begins with a laser scan of the original artwork to create a raster file. The raster file is a graphics database that breaks the images into pixels, a binary database in which a 1 indicates a black dot, and a O indicates a clear dot.

Next, a raster-to-vector conversion is initiated. A vector database contains the locations for the end points of lines and origins of pads using an X, Y coordinate mapping system and origins of pads.

At IGI, the next step is to convert the vectorized data into an IGI internal format. The data are edited and manipulated in this state. The data is further translated into Gerber file data format, a drill tape format, or some other manufacturing format. Usually, the photoplotting process requires Gerber data, so a file is created. The Gerber file can now be used to photoplot new artwork or make modifications on a CAD design station.

The uses in the engineering realm vary, but the most prominent application for this type of high-precision graphic service is phototooling. IGI uses an Optrotech scanning system for initial input. A variety of workstations and in-house custom software is used to edit and otherwise prepare the data.

WHO USES SCANNING?

Basically, two groups in the electronics industry use scanning services: PCB design houses and PCB fabricators. The main reason a design house requires scanning is to obtain the electronic data. In these situations, the data have either been lost or have become obsolete due to subsequent modifications to the design. In many instances, even though the initial design was created on a computer, the database may not have been updated to reflect all of the modifications.

Even in today's CAD/CAM-oriented production environment, customers still supply PCB shops with film. The film then needs to be translated into Gerber data to be used efficiently in board manufacture.

A FIVE-TIER SYSTEM

Based on the complexity of a particular scanning job, IGI has developed a unique five-tier approach to processing these jobs. This method has helped to alleviate some of the common problems associated with achieving accurate or complete scanning. Most problems occur when customers' needs and expectations aren't fully understood. Once these have been established, job processing will be smoother.

Level 2. Scan, align to specifications, scale to correct size, and check for completion. When two-sided or multiple layers are being scanned, layer-to-layer registration is important. For example, the data generated from the existing artwork film may fluctuate in size over time due to temperate changes. As a result, the layer-to-layer registration would be inaccurate.

Using in-house design capabilities, IGI goes into the scanned database itself and modifies the data to acceptable registration alignments. Finally, we check for job completeness, ensuring that the Gerber file database is square.

Level II. This process includes the same steps as Level I as well as database "clean-ups." Clean-up refers to the process of correcting any discrepancies in line widths, removing characters translated from foreign particles picked up during the scan, and checking the integrity of the line-to-pad connections. These changes are made through our in-house CAD system to achieve a higher-quality board representation.

Level III. This is comprised of the same steps occurring in Levels I and H, in addition to snapping the pads to grids or drill data locations. Vector-

filled pads are replaced with a single pad flash. Because drawn pads will not be snapped to the drill layer, this procedure is necessary to help ensure layer-to-layer registration during the manufacturing process. Snap-to-grid can achieve this by aligning all layers involved to a common X, Y coordinate or group point, with the first or master layer being the reference location.

Snap-to-drill data is another form of layer registration and is the more desirable and accurate of the two methods. Snap-to-drill data involves using the drill data as a reference point to align the layers of the board.

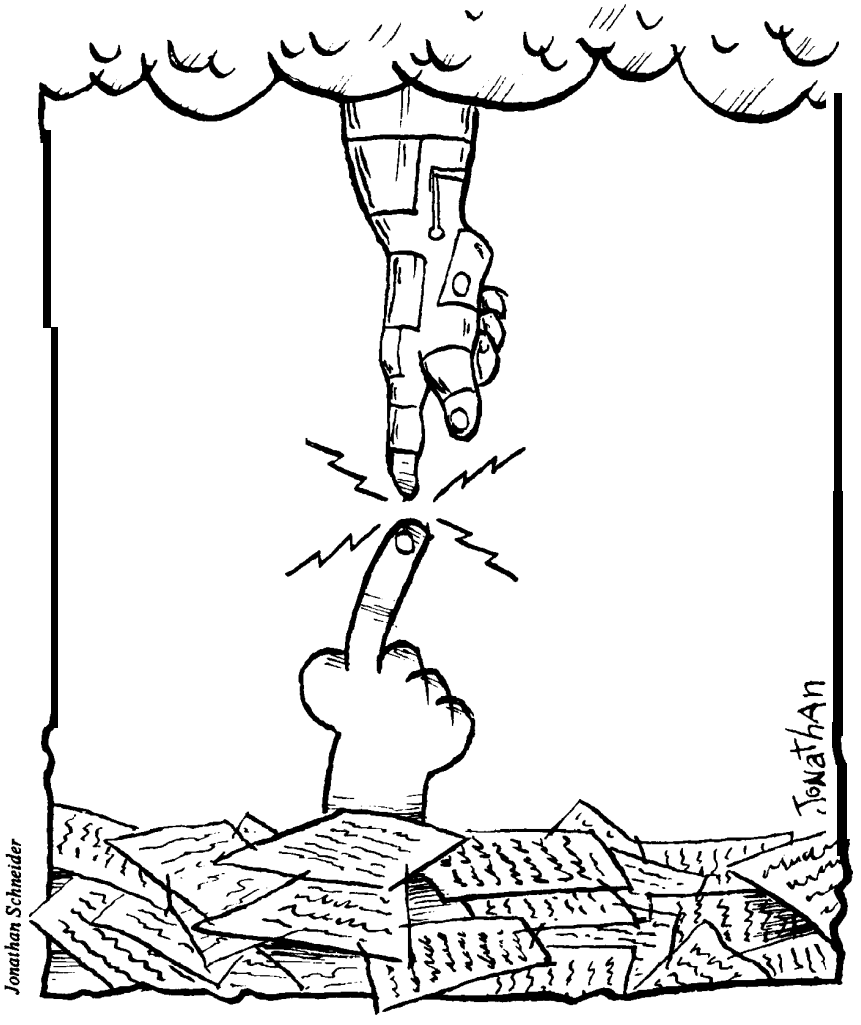
The format used to obtain information for the drill data can be a drill tape, floppy disk, or magnetic tape. A drill tape is simply a roll of paper tape (usually supplied by the customer) that contains X, Y coordinates of drill hole locations coded as a series of punched holes. If a drill tape is not available, one can be created from the artwork. IGI's method of replacing vectors with single-pad flashes has the following advantages:

- It will reduce the size of the database.
- It makes the database easier to manipulate.
- It will provide cleaner, sharper images.
- It enables smoother execution of the snap-to-pad process.

Level IV. This includes the steps in Levels I through III as well as design rule checking (DRC). Clean-up is used in Level 111. However, there is another process that locates the minimum spacing violations between pad-to-pad, line-to-line, and pad-to-line on the board. DRC can ensure that the manufactured board will meet the applicable specifications.

Level V This step consists of full reverse engineering, which is the re-creation of the PCB. This process is used when the customer wants a high-quality database that meets original product specifications. Input ranges from a customer's sample of an existing board to engineering drawings or aperture cards.

Various custom software programs are employed with the digitized data, and, depending on the circumstances, so are high levels of manual PCB design effort.



FUTURE OF SCANNING

Clearly, the number of applications for this process is enormous. IGI estimates that there are millions of part numbers that could require this documentation procedure.

With time, the scanning process will become increasingly automated through the development of more efficient software. However, the procedure will begin at a scanner where the initial digitizing will occur. ■

ACKNOWLEDGMENT

I would like to recognize the following people from IGI for their assistance in putting this article together: Robert Larsen, supervisor of custom design services; John Thompson, supervisor of panelization and scanning; and Fred Snyder, director of marketing.

Ken Rockette is marketing manager with Infinite Graphics Inc., Minneapolis, MN.