ODB++: THE MOST EFFECTIVE COMMUNICATIONS FORMAT FOR TRANSFERRING PCB DESIGN DATA TO MANUFACTURING

MAX CLARK, MENTOR GRAPHICS

WHITEPAPER

P C B   M A N U F A C T U R I N G

Competing in global markets requires OEMs and suppliers to continuously seek ways to more effectively communicate manufacturing requirements when bringing products to market. The collection of manufacturing requirements can be referred to as the product model. For the suppliers, the product model defines 'what' needs to be manufactured, but does not define 'how' the manufacturing process will be accomplished. When complete, the product model contains the manufacturing content necessary to convey the fabrication, assembly and test requirements to suppliers. The method used to communicate the product model is a key factor in determining process efficiencies, the level of product quality and ultimately the rate a product moves to volume production, which determines the actual time-to-market. When an effective product model is used to streamline the manufacturing process, many segments of the design and manufacturing operations can contribute to the successful launch of a new product.

But today there are fundamental inefficiencies used in communicating product models into manufacturing that lead to unnecessary delays and additional risks that could largely be avoided if improved communication practices were implemented. One fabrication house estimates that upwards of 25% of all data packages received today have some form of a data integrity issue.

Adopting a common and comprehensive product model between design and manufacturing will allow both the OEMs and suppliers to more effectively:

- Meet product requirements while reducing costs
- Minimize production delays
- Optimize manufacturing productivity through improved knowledge exchange including proper IP management

There have been industry driven initiatives to improve upon the data exchange formats available. The most widely adopted improvement was based upon enhancing a format originating in the 1960s with the creation of Gerber RS274X. But before you can begin to consider the data exchange format of choice, you should determine what the data requirements to support the manufacturing processes are.

COMMUNICATING TODAY'S PRODUCT MODEL

The primary method of communicating a product model into manufacturing today is based on a series of independent files, in several different formats, each defining a segment of the manufacturing requirements. The most common bare PCB formats are Gerber to represent physical board layers, Excellon to represent the mechanical interconnect information and an IPC 356 file to represent the electrical intent. For each product model, there will most certainly be multiple Gerber files, possibly multiple Excellon files and hopefully a single IPC 356 file. To effectively convey the assembly requirements, a non-standard column based list is typically provided containing the OEM's internal part numbers, x and y locations, rotations, side, etc. A bill of material (BOM), at times along with an Approved Vendor List (AVL), is also provided to connect the OEM's internal component part number to a component supplier and supplier part number. There may be additional files in yet different formats containing, fabrication and assembly documentation, loaded board test access locations and other product related content.

The challenge in using this method of introducing a product model into manufacturing is that the reconstruction of the design requires reverse engineering efforts to recreate the product model's manufacturing intent. Most CAM software solutions, in order to ensure manufacturing guidelines are met, require some form of board construction information or buildup. In the case of Gerber data, the layer representations have to be organized into design buildup positions and purposes, the Excellon drill information has to define the hole finish types and depth through the buildup, component locations need to be aligned to outer copper layers, etc. Each of these steps is error prone and quite often incomplete or inconsistent manufacturing data results. For example, drill count information does not match the drill drawing, electrical content is inconsistent with the software extracted netlist, component rotations are not consistent with component land patterns, test point locations do not align with copper outer layer pads, etc.

BARE PCB CONSTRUCTION

Many of the product model files just mentioned were created to convey mechanical instructions to production equipment. This is especially true in the case of bare PCB manufacturing. All forms of Gerber are used to convey how the multiple phototools used in the imaging process should be created by mechanical or laser based photoplotters. The Excellon format is used to instruct mechanical drill and routing equipment how to respond when creating holes or removing boards from a fabrication panel.

FILENAME:	DESCRIPTION:	TYPE OF FILE:
readme.txt	This readme file.	ASCII Text
ncdrill1.tap	Plated/NONPlated thru drill file NCtape	
nc_tools.txt	Drill Tool file	ASCII Text
nc_param.txt	Drill Parameter file	ASCII Text
Art_param.txt	Gerber parameters	ASCII Text
Art_aper.txt	Gerber Apertures	ASCII Text
tpst.art	Top Pastemask	Gerber File
tsmk.art	Top Soldermask	Gerber File
tslk.art	Top Silkscreen	Gerber File
lyr1.art	Layer 1 - TOP Signal Traces	Gerber File
lyr2.art	Layer 2 - INT1 Signal Traces	Gerber File
lyr3.art	Layer 3 - GND1 Plane	Gerber File
lyr4.art	Layer 4 - INT2 Signal Traces	Gerber File
lyr5.art	Layer 5 - INT3 Signal Traces	Gerber File
lyr6.art	Layer 6 - PWR1 Plane	Gerber File
lyr7.art	Layer 7 - INT4 Signal Traces	Gerber File
lyr8.art	Layer 8 - INT5 Signal Taces	Gerber File
lyr9.art	Layer 9 - PWR2 Plane	Gerber File
lyr10.art	Layer 10 -INT6 Signal Traces	Gerber File
lyr11.art	Layer 11 -INT7 Signal Traces	Gerber File
lyr12.art	Layer 12 - GND2 Plane	Gerber File
lyr13.art	Layer 13 -INT8 Signal Traces	Gerber File
lyr14.art	Layer 14 -BOT Signal Traces	Gerber File
bpst.art	Bottom Pastemask	Gerber File
bsmk.art	Bottom Soldermask	Gerber File
bslk.art	Bottom Silkscreen	Gerber File
bprobe.art	bottom probe	Gerber File
fab.art	Fabrication Drawing	Gerber File
tasy.art	Top Assembly	Gerber File
basy.art	Bottom Assembly	Gerber File
tasy.pdf	Top Assembly	PDF file
basy.pdf	Bottom Assembly	PDF file
fab.pdf	Fabrication Drawing	PDF file
place.rpt	Placed component	TXT file
netlist.rpt	Netlist	TXT file

Figure 1: Typical listing of all files that need to be commu–nicated to the PCB manufacturer when using traditional Gerber files.

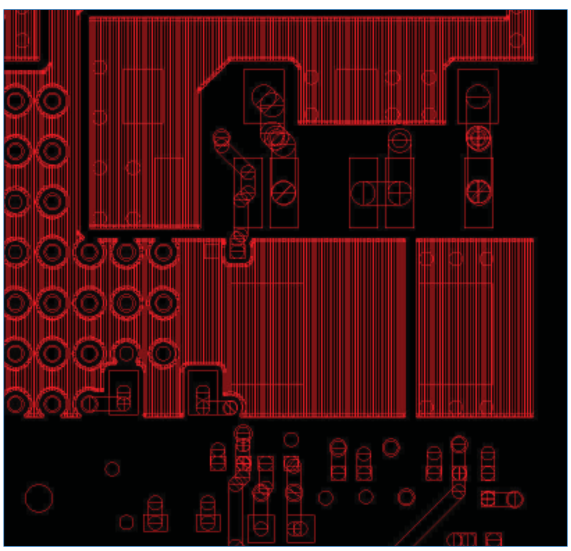


Figure 2: This image of filled-in surface area is based on Gerber RS274D data files.

No matter which format is used, one common characteristic is that the representation of the product model has to be based on the simplest and most primitive elements defined by the format, regardless of how that might compromise the clarity of the content. While this might have not been an issue when used for the intended purpose, the advent of more sophisticated CAM systems found these primitive data formats challenging.

The representation of board design layers that should have been a simple collection of objects, ended up as a collection of overly complex objects that enlarged the amount of data. Also, basic design intent, such as finish types for drilled holes, was also not an integral part of any format. This information ended up embedded in drill tables and drawings for manufacturing consumption. Later, CAM systems found it necessary to reverse the effect of the over simplification through the use of software. These actions, at the time necessary and with good intention, add unnecessary risk and require additional engineering resources.

PCB ASSEMBLY PROCESS

The basic product model information required to complete the assembly process is the x/y locations, rotations, and the possible component part numbers for each of the component placements. The variety of manufacturing equipment providers and machine capabilities are so variable that no single format was deemed sufficient to cover all possibilities. As a result no "quasi-standard" format was selected to represent even the basis of a product model. In addition, the solder paste land patterns that are used to create the solder stencil, which then is used in the application of the solder paste, must be present in the product model.

Since no standard was established, today's solutions are typically based on multiple non-standard formatted files that are either in text or Microsoft Excel formats. Much like the CAM systems used in bare PCB manufacturing, the assembly CAM systems are then required to implement solutions to read these non-standard formats in an attempt to reconstruct the assembly representation of the product model.

COMPONENT ORIENTATION

These are processing challenges that increase risk and therefore require additional resources during this reverse engineering process. Component rotation (orientation) is based on the EDA system's representation, which is not necessarily consistent with the component orientation as present in the assembly CAM system. Therefore, through the use of a CAM system, resources are assigned to complete the product model as a final assembly by once again processing the bare PCB's Gerber-based outer layer information. Typically, this would include the outer copper and silkscreen layer information. Using the outer layer copper, silkscreen, and component location/rotation information, the CAM system attempts to recreate the assembly representation.

Assigning component rotations begins with aligning the placement coordinates with the outer layer land patterns and then managing the component rotation through the use of polarity indicators. Polarity indicators are usually found on the silkscreen and are used to define the orientation of the component as found in the placement file. There can be a number of errors starting with simply misinterpreting the intended rotation assignment or an inconsistency in the component rotations as defined in the original data source.

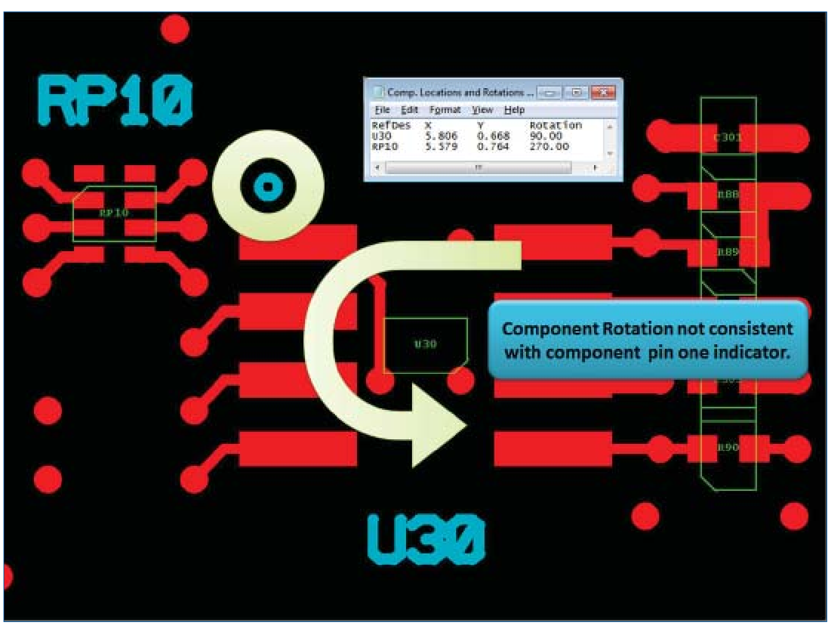


Figure 3: Image of a board outer layer, associated silkscreen and a component centroid indicator, along with a text file indicating the component rotation. Should something convey the message "what is 0, 90, 270, 360, etc. degrees?"

In addition to processing the outer copper layers, as described earlier, if the design solder paste definition was included using the traditional Gerber method, the assembly CAM system could also process this content. The solder paste would then be aligned using the same methods as the outer copper and silkscreen layers, along with the components.

Next, using reverse engineering methods, the solder paste openings are reconnected to copper landings and, ideally, back to a component to recreate a land pattern representation. Finally, using the land pattern representations, the assembly manufacturer's optimal solder paste stencil can be designed.

This reverse engineering process is again time-consuming while adding no actual value to the final assembly. There are also risks that an error within the land pattern assignment could lead to an incorrect solder stencil design resulting in quality related issues identified at the very end of the manufacturing process.

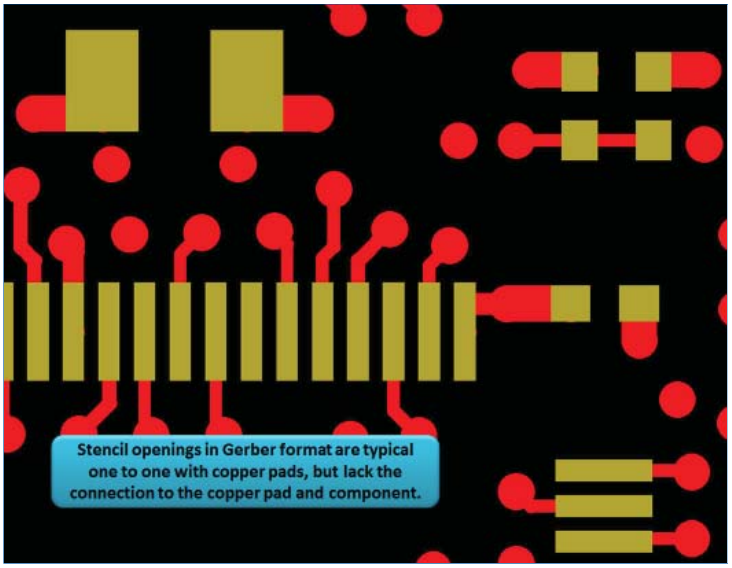


Figure 4: Here, the solder paste opening can be seen with a component land pattern extracted for the base Gerber data.

FINAL TEST

Access to critical pad locations on an assembled finished product is an essential part of final test. The test equipment requires the placement of test pins such that proper contact with a copper pad is possible. Inadequate test coverage or insufficient test access will compromise quality testing during final production.

In many cases designers go to great lengths to ensure their board designs have allowances for proper testing. However, there are no standard methods today for conveying this vital information. In some cases the test locations can be determined by a unique pad size on the outer layer. In other cases reference designators with a unique prefix are provided. Even a text file with x/y and side locations have also been created and then sent to manufacturing. In all the cases above, CAM systems will attempt to take the outer layer information and reverse engineer in the initial test location as planned by the designer. In the event the testing is found to be inadequate, the CAM system will attempt to improve coverage working with the product model's outer layers.

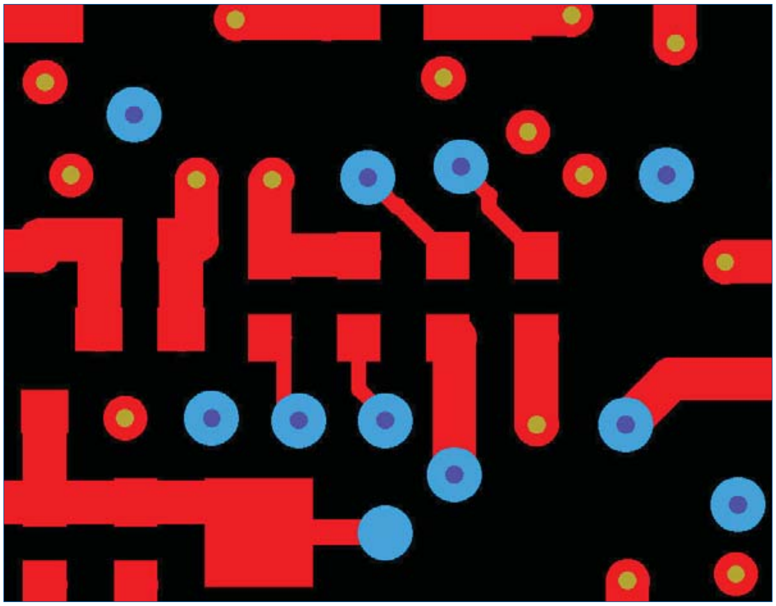


Figure 5: This image shows the test locations by determined pad sizes.

The task of creating an acceptable test strategy through this type of process is user intensive. Many process steps introduce unnecessary risks along the way. All of which can lead to test procedures not as complete as possible or test implementation challenges that go undetected until well within production.

EFFECTIVE PRODUCT MODEL COMMUNICATION

Addressing the need for effective design to manufacturing communication begins with the product model that contains the basic buildup information and other related manufacturing content is then associated with it. The buildup representation acts as the central repository for what was once multiple files, eliminating the potential errors of assigning layer types (signal, power/ground, solder mask, etc.) and layer positions as the basis of the product model. Mechanical and assembly data elements are represented right along with the board copper layer data elements. The buildup information can be used to maintain the interdependencies between other data elements, empowering other CAM systems to harness those relationships in order to improve production efficiencies.

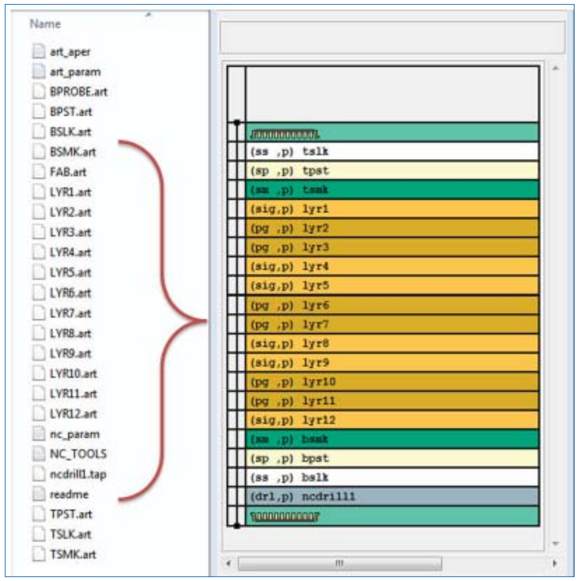


Figure 6: The filenames can be represented in matrix form, where the matrix clearly illustrates layer types, polarity, drill depth, and other necessary fabrication information.

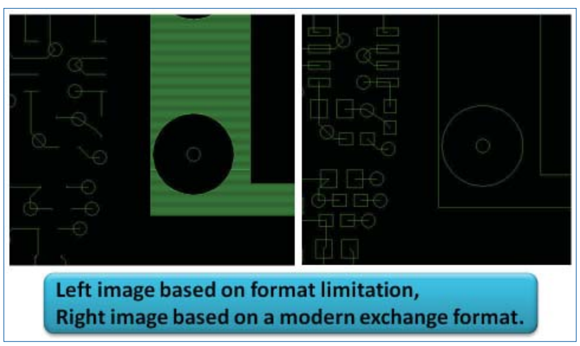
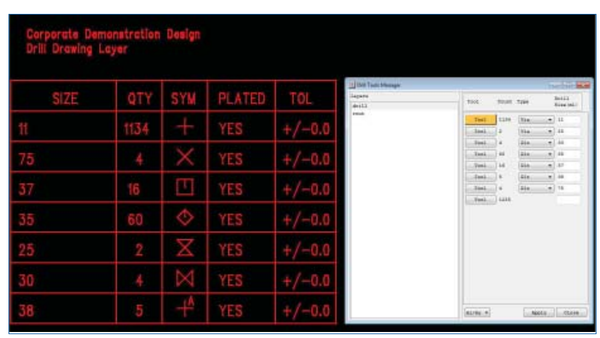


Figure 7: On the left is an image of drawn SMD pads and filled in areas, with the same representation in ODB++ on the right.

The product model then represents the graphical content in a comprehensive fashion that is completely independent of historical restrictions. For example, surface pad locations that were once drawn with multiple data elements would be represented as a single element with a land pattern definition. Solid areas previously filled in with multiple drawn features would be represented by complex, yet simpler to manage, surface polygons. Representing the data in this fashion would improve on the communication of complex manufacturing requirements while enabling manufacturing facing CAM systems to efficiently manage production requirements.

With the basic product model organized and represented in an intelligent manner, the product model should begin to expand to also eliminate manual assignments of important manufacturing requirements. As an example, the final finish and tolerances of each drill hole requires further definition. Therefore the final finish method of plated, non-plated, via, tolerances, etc. and tolerance information assigned to each drill size should be an essential part of the product model. Having this information as part of the product model reduces the risk of manufacturing errors by minimizing the occurrences of incorrectly sized and improperly processed holes. Both of these errors will typically result in costly scrap and a delay in product delivery.



SIZE	QTY	SYM	PLATED	TOL
11	1134	+	YES	+/-0.0
75	4	X	YES	+/-0.0
37	16		YES	+/-0.0
35	60		YES	+/-0.0
25	2		YES	+/-0.0
30	4		YES	+/-0.0
38	5		YES	+/-0.0

Figure 8: Here is the traditional drill table on the left, and the same information with the drill tools manager.

Thus far the entire product model definition has included the requirements in order to construct a bare

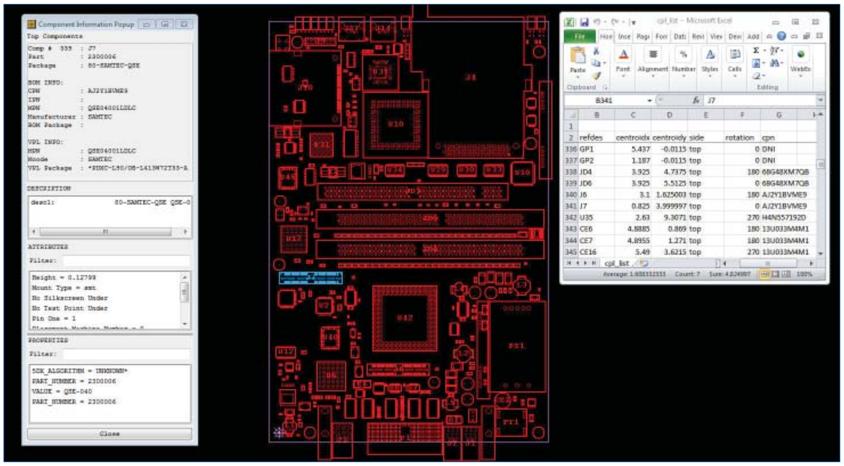


Figure 9: Here we compare the component placement list with the ODB++ component layer display.

PCB prior to assembly. A comprehensive product model should contain the final component placements, rotations and component geometries. In addition, within the product model the possible component manufacturers and part numbers required for component ordering and assembly should be assigned. This information enables the procurement of required components and the creation of the manufacturing process definition (MPD), which later drives the entire product assembly process. Having the component package types, lead forms, physical sizes and weights enables the automated process of creating pick and place programs for an assembly line and facilitates the creation of manual assembly procedures.

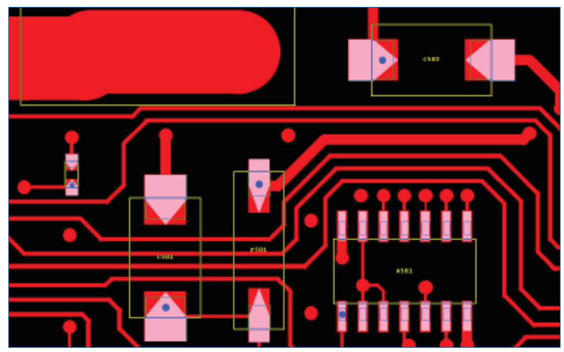


Figure 10: This complex solder stencil design includes a VPL package with lead contact area. Solder design should include the use of home plates and possible a dog bone.

Having a correct representation of the product model in order to create assembly machine programs is important, but of nearly equal importance is an optimal solder stencil. Solder stencils are a critical part of the manufacturing definition because they control the amount and shape of the solder paste applied to each surface mount location. A well-defined product model would maintain the connections between the copper landings to each lead of the component package. In addition the type of lead, gull wing, J- lead, ball, etc., would also be assigned to each of the component leads. Later, the combination of component package, lead form and copper landing together would be used to define the shape and size of solder stencil opening to create in order to apply the appropriate amount of solder paste.

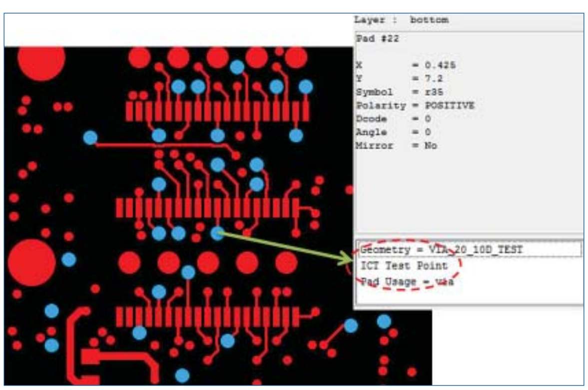


Figure 11: This figure shows the test location by ODB++ feature attribute and the test point potentials.

One of the very final production process steps required is test. The product model should carry the designer's intended test locations. The test locations are defined by a relationship assigned to select copper pads. The lack of a solder mask opening for the defined pad would indicate a test pin with the ability to penetrate the solder mask is required or the location is not suitable. With a complete product model, the CAM system would be capable of automatically increasing test coverage while considering the designer's intent, thereby ensuring the highest quality test possible.

HOW EFFECTIVE IS YOUR PRODUCT-MODEL EXCHANGE RELEASE TO YOUR SUPPLY CHAIN?

In an ever increasing competitive landscape, the design/supply chain, as defined by the PCB manufacturing process, should constantly look to reap the benefits of their existing investments. This is not just true in the sense of technology, but also design/manufacturing expertise, lessons learned from past experiences, valued partner relationships and implementing already available best practices. When the supply chain has optimized the data exchange of design information into the various manufacturing areas, each area will be able to optimize their individual manufacturing processes by obtaining the maximum benefits on their individual investments, thereby further streamlining efficiencies into the process and in turn delivering a quality product at an optimal price.

Existing today is an alternative to the lower-level task driven formats such as Gerber and Excellon. The ODB++ format has the ability to transfer all, or user-selected portions, of the PCB product-level intelligence required to support the PCB manufacturing processes (fabrication, assembly, test). Therefore, the ODB++ format enables design-to-manufacturing integration within the fabrication, assembly and test processes, while still protecting the design's intellectual property (IP). In doing so, the ODB++ format eliminates the daunting and risky tasks required to reconstruct the product-model from multiple legacy formats and the processing of additional information received in non-standard formats. These same product-model reconstruction tasks are often repeated multiple times within the PCB manufacturing supply chain.

The infrastructure currently within the ODB++ format is based on expert knowledge gained over years of industry use and in many ways captures the best practice requirements in defining the multitude of PCB manufacturing processes. The ODB++ format harnesses all the data requirements for the design-to-manufacturing exchange of a product model that meets, and in many ways exceeds, the need for an effective and efficient data-exchange process between OEMs and their suppliers. Using the ODB++ format, within the PCB manufacturing supply chain, enables all parties to invest more in improving key business and manufacturing processes that are critical to their success.

Corporate Headquarters
Mentor Graphics Corporation
8005 SW Boeckman Road
Wilsonville, OR 97070-7777
Phone: 503.685.7000
Fax: 503.685.1204

Sales and Product Information
Phone: 800.547.3000
sales_info@mentor.com

Silicon Valley
Mentor Graphics Corporation
46871 Bayside Parkway
Fremont, CA 94538 USA
Phone: 510.354.7400
Fax: 510.354.7467

North American Support Center
Phone: 800.547.4303

Europe
Mentor Graphics Deutschland GmbH
Arnulfstrasse 201
80634 Munich
Germany
Phone: +49.89.57096.0
Fax: +49.89.57096.400

Pacific Rim
Mentor Graphics (Taiwan)
Room 1001, 10F
International Trade Building
No. 333, Section 1, Keelung Road
Taipei, Taiwan, ROC
Phone: 886.2.87252000
Fax: 886.2.27576027

Japan
Mentor Graphics Japan Co., Ltd.
Gotenyama Garden
7-35, Kita-Shinagawa 4-chome
Shinagawa-Ku, Tokyo 140-0001
Japan
Phone: +81.3.5488.3033
Fax: +81.3.5488.3004

MF 2-12 MISC1490-w