



INTEGRATING 3D WITH PRINTED CIRCUIT BOARD DESIGN

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INTRODUCTION Packaging and printed circuit board design have always been closely related disciplines. As little as 10 years ago, printed circuit boards (PCB) were designed by a member of a mechanical engineering department. When integrated schematic/PCB CAE tools became available, electrical engineers accepted the tasks related to layout. Today, almost all printed circuit board design falls upon electrical engineering or a closely integrated board design department.

In general, electrical engineers are not equipped with the skills or knowledge to design complex mechanical packages. Close collaboration between mechanical engineers and electrical engineers must be established early in a project. Despite this collaboration, tightly packaged designs often require re-spins because mechanical constraints are not understood, overlooked, or simply neglected.

This article examines the prevailing method of data exchange between PCB and Mechanical Engineering. We will also examine leading data formats and an emerging technique for direct 3D model transfer from PCB into packaging software, without the aid of data translation.

DATA FORMATS

DXF - Drawing Interchange Format or Drawing Exchange Format - is a file format introduced by Autodesk in 1982 to facilitate AutoCAD drawings to be opened by other programs. DXF was introduced as a method to transfer vector data using standard ASCII text files. As Autodesk introduced new versions of their AutoCAD DWG file format, the DXF version changed as well, causing compatibility issues with other programs.

DXF has become less useful because it can no longer support complex objects. Certain object types, including solids and regions, can not be supported at a sufficient level to allow users to export as DXF.

DWG - Developed in the late 1970's by AutoCAD. Autodesk owns and develops the DWG file format. The DWG format changes every few years as Autodesk adds new features to AutoCAD. The DWG format, along with DXF, have become the *de facto* standard file format for CAD drawing exchange. It is estimated that there are over one billion DWG files in existence. Autodesk licenses a developer toolkit called "RealDWG" so other software can implement their DWG technology. Almost all commercial application software developers and Autodesk's competitors now choose to support DWG format.

IGES - The Initial Graphics Exchange Specification (pronounced eye-jess) is a data format that allows the digital exchange of information among graphic (vector) Computer-aided design systems.

The IGES standard is developed and maintained by the IGES/PDES Organization (IPO); a program area of the U.S. Product Data Association. In addition to the general specification, IGES also includes application protocols in which the standard is interpreted to meet discipline-specific requirements. Using IGES, a CAD user can exchange product data models in the form of circuit diagrams, wireframe, freeform surface or solid modeling representations. Applications supported by IGES include engineering drawings, models for analysis, and other manufacturing functions.

IGES was started in 1979 by a group of CAD users and vendors, including Boeing, General Electric, Xerox, Computervision and Applicon with the support of the National Bureau of Standards and the U.S. Department of Defense (DoD). The name was chosen to avoid any suggestion of a database standard that would compete with the proprietary databases used by the different CAD vendors.

Since 1988, the DoD has required that all digital Product Manufacturing Information (PMI) for weapons systems contracts be delivered in electronic form, specifically in IGES format. As a consequence, any CAx software vendor who wants to market their product to DoD subcontractors must support the import and export of IGES format files.

IGES has generated warehouses full of magnetic tapes and CD-ROMs of digital PMI for the automotive, aerospace, and shipbuilding industries, as well as for weapons systems from Trident missile guidance systems to entire aircraft carriers. IGES provides a way to access this data decades from now, years after the vendor of the original design system has gone out of business. Today, plug-in viewers for Web browsers allow IGES files created 20 years ago to be viewed from anywhere in the world.

After the initial release of STEP in 1994, interest in further development of IGES declined, and Version 5.3 (1996) was the last published standard.

VRML Virtual Reality Modeling Language, pronounced *vermal* is a standard file format for representing 3-dimensional interactive vector graphics, designed particularly for the World Wide Web.

VRML is a text file format where vertices and edges for a 3D polygon can be specified along with the surface color, image-mapped textures, shininess, transparency, and so on. URLs can be associated with graphical components so that a web browser might fetch a web-page or a new VRML file from the Internet when the user clicks on the specific graphical component. Animations, sounds, lighting, and other aspects of the virtual world can interact with the user or may be triggered by external events such as timers. A special Script Node allows the addition of program code (e.g., written in Java or JavaScript (ECMAScript)) to a VRML file.

VRML files are commonly called "worlds" and have the .wrl extension. Although VRML uses a text format, they may often be compressed so that they transfer over the internet more quickly. Many 3D modeling programs can save objects and scenes in VRML format.

While VRML still enjoys widespread use, particularly in education and research where an open specification is most valued, it has now been superseded by X3D. The MPEG-4 Interactive Profile (ISO/IEC 14496) was based on VRML (now on X3D), and X3D is largely backward-compatible with it. VRML is also still used as a file format for interchange of 3D models, particularly from CAD systems.

STEP - Standard for the Exchange of Product Model Data - also known as ISO 10303.

STEP started in 1984 as a successor of IGES. The objective of STEP is to provide a mechanism that is capable of describing product data throughout the life cycle of a product, independent from any particular system. STEP can be used to exchange data between CAD, CAM, PDM/EDM and other systems. STEP is addressing product data from various industries such as mechanical, automotive, aerospace, building construction, ship, oil & gas, process plants and others.

STEP consists of several hundred parts and every year new parts are added or new revisions of older parts are released. Each part has its own scope and introduction

APs the top parts. They cover a particular application and industry domain and hence are most relevant. Every AP defines one or several Conformance Classes, suitable for a particular kind of product or data exchange. To provide a better understanding, information requirements and usage scenarios an informative application activity model (AAM) is added to every AP.

STEP primarily defines data models using the EXPRESS modeling language. Application data according to a given data model can be exchanged either by a STEP-File, STEP-XML or via shared database access using SDAI. Many critics point out that the XML standards for e-commerce are being developed much more quickly than STEP.

Parametric Solid modeling CAD

Solid modelers have become commonplace in engineering departments in the last ten years due to faster PCs and competitive software pricing. Solid modeling software creates 3D representations of components for design and analysis. Interfaces can include programmable macros, keyboard shortcuts and dynamic model manipulation. Designs are usually done within context of the whole product using component assembly modeling methods.

Solid models may be built using a group of sketcher-based features; e.g. 2-D sketches that are swept along a path to become 3-D. These may be cuts, or extrusions.

Another modeling technique is 'surfacing' (Freeform surface modeling). Surfaces are defined, trimmed and merged, and filled to make solid. The surfaces are usually defined with datum curves in space. Surfacing is more difficult but better applicable to some manufacturing techniques like injection molding.

Parametric modeling uses parameters to define a model (dimensions, for example). The parameter may be modified later, and the model will update to reflect the modification. Typically, there is a relationship between parts, assemblies, and drawings. A part consists of multiple features, and an assembly consists of multiple parts. Drawings can be made from either parts or assemblies.

For example, a shaft created by extruding a circle, 100 mm. A hub is assembled to the end of the shaft. Later, the shaft is modified to be 200 mm long. When the model is updated the shaft will be 200 mm long, the hub will relocate to the end of the shaft to which it was assembled and the engineering drawings and mass properties will reflect all changes automatically.

3D parametric solid modeling is used in many CAD products including **Autodesk Inventor**, SolidWorks, Pro/ENGINEER, and Solid Edge.

Evolution

Printed circuit board design has migrated from mechanical engineering to electrical engineering. Commercial products such as camcorders, cell phones, and other small products are driving complex packaging. PCB design and electrical design software operate seamlessly, however the need for a seamless mechanical interface is required.

A majority of PCB designs start with a parametric design created in 3D. A file is exported or saved as DXF, then imported into the PCB CAD system. Some experimentation is required since DXF is not native to either software application. Incompatibility issues with the DXF format may require changes to scaling, unit size, or origin settings to achieve satisfactory results. In any case, 3D information can not be exchanged using DXF. Height restrictions or constraints must be conveyed using 2D elements in the drawing or some other means of communication must be used.

Using the DXF file, a board designer can now place components. Other factors such as fabrication, manufacturing, and/or electrical constraints will determine component placement. The design is saved as DXF then transferred back to the mechanical engineer. Some mechanical designers use look up tables or libraries to “recreate” a 3D file. Some designers choose not to convert back to 3D, but will instead overlay the flat DXF file over the original parametric model. This is the same technique as using a light table, except the monitor is the electronic light table. A visual check light table check is often employed instead of relying on software for verification.

This import/ export process may be repeated several times until both packing and electrical constraints are satisfactorily met. The process is labor intensive, requires manual intervention and is inherently error prone.

The flow for data transfer is shown in **figure 1**.

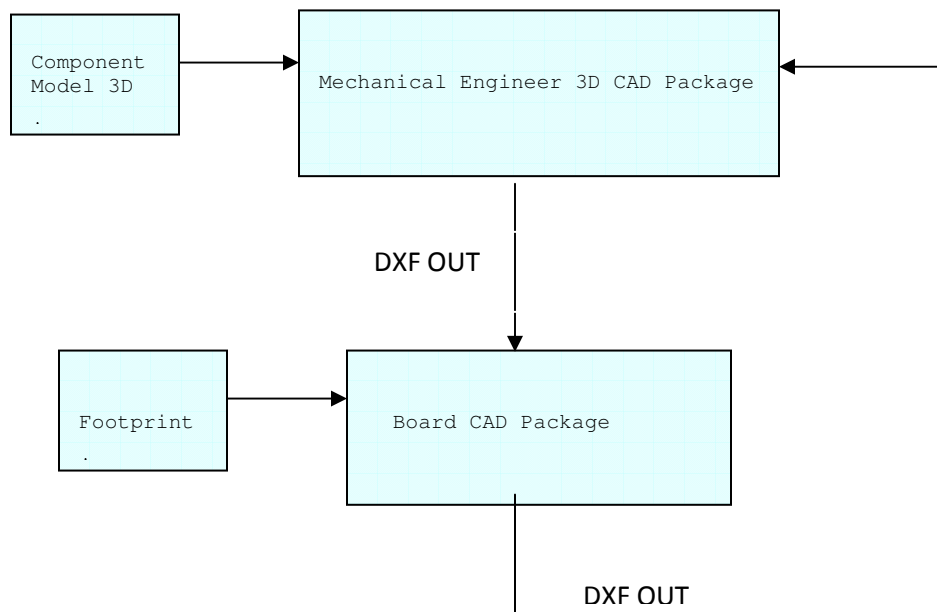


Figure 1

The inherent problems are

- Multiple translations to/ from DXF are required

- DXF does not support complex features

Height information is not available

Look up tables and height cross referencing is required.

Correlation of 3D information is subject to errors

STEP and PCB

An alternative method is to integrate the 3D model with the PCB component. (see Fig 3) A component is generally created with electrical, fabrication, simulation, and manufacturing elements. PCB components should also include the 3D model. STEP models are a good choice since they enjoy industry support from popular software applications such as AutoDesk Inventor, Solid Works, and Pro-Engineer. Connector manufactures such as AMP and Samtec are also supplying STEP models.

The manufacture's datasheet or STEP model only includes information relative to the part. Features such as keepouts, pad sizes, and copper must be included in the 3D model. Leaded components such as, capacitors, resistors, and Power Devices, require additional space for lead bending or mounting. The manufacture's datasheet does not consider assembly, fabrication or manufacturing tolerances, therefore ownership of the 3D models should be shared between mechanical engineering and PCB design.

As PCB and electrical software have become integrated, PCB and packaging software should be just as seamless. Component models should include clearances, keepouts and pads. When SMD footprints are modeled, origins should always be in the middle of the component. There are no clear guidelines for thru-hole components, however origins of each PCB component should match the origins of the STEP model. If the origins do not match, the resulting data exchange will be skewed.

Conclusion:

Despite the availability of integrated tools many designers and engineers are exchanging DXF or DWG files. Anytime a file is manually adjusted there is room for errors. Employing these outdated techniques will continue to lead to errors. Light table verification should be replaced with software error checking.

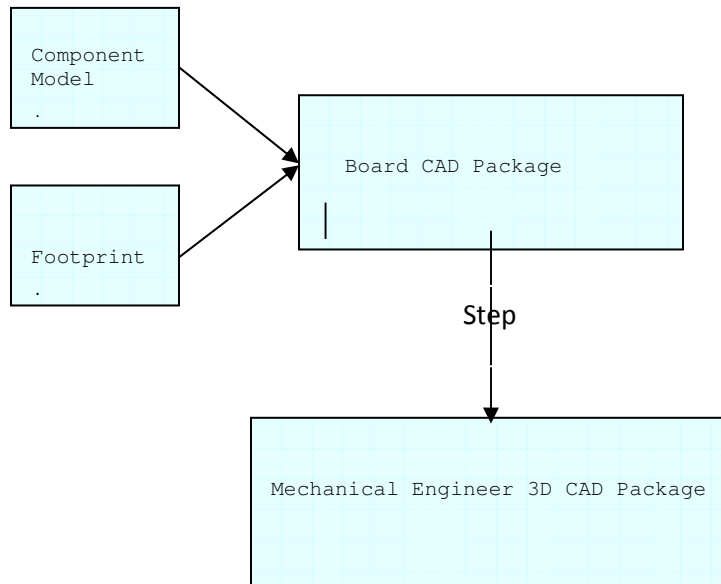


Figure 2

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