



RE: ODB++

November 8, 2012

To start with I am a member of the ODB++ Solutions Alliance. I am also a big fan of the ODB++ format. However, ODB++ is a powerful data format but is still experiencing growing pains.

ODB++ was first released as a CAM tool in 1995 by a company called Valor. The XML version that the database operates under was released in 2000. Valor was purchased by Mentor Graphics in 2010. The ODB++ database has gone through several changes and is currently at version 7. There are plans for a version 8 release.

There are two concerns with the database format.

- 1) The database has been and is still a proprietary format. This has been one of the main reasons why acceptance has taken almost 15 years. Valor was a competitor in the CAM industry and had the appearance of pushing a proprietary format upon their competitors from the CAD end of the supply chain. The concerns over the issue of a proprietary format were greatly magnified when Mentor Graphics, a provider of CAD software, purchased Valor. Mentor Graphic has made the database format available to any that apply for it. To date there have been no documented instances of access being restricted by Mentor Graphics.
- 2) When the Gerber data format (RS-274-D) switched over to the extended format (RS-274-X) there were many documented instances of programming errors on both the CAD and CAM sides of the supply chain. The program errors typically resulted in custom features for solder pads being improperly converted. Even though everyone did their jobs correctly the end result were boards that could not be assembled. Over time, both the CAD and CAM programmers were able to import and export reliable versions of the RS-274-X format. Data errors with Gerber are a very rare occurrence. This is mainly due to the fact that the format has not changed for a long time. For all intended purposes, Gerber may be considered a dead language that is still used by the industry at large.

ODB++ may be considered a living language and is prone to all of the growing pains that occurred with Gerber. BBG and our partners have experienced these growing pains first hand as the ODB++ format has progressed. Unfortunately this is a case where change is bad. When a major revision of the database format is released all of the CAD/CAM programmers have to decipher the ODB++ programming specification and write code for it. The problem here is that programmers are people and people make mistakes. A misinterpretation of the specification typically results in a non-conformance on the finished product. Data issues of this nature are typically thrown back at the bare board manufacturer. Hence our reluctance to welcome ODB++ with open arms.

Understanding that the ODB++ database format is going to have another version released in the near future should be noted by all in the supply chain. Precautions should be taken by all parties involved to ensure that the non-conformances if they arise are minimized.

- 1) The return of check plots. In the old days board fabricators used to provide photo plotted images for designers to review. Where plotted film is not logistically feasible due to source location and manufacturing lead times, exporting Gerber files or even PDF views of the working data for review is not a problem. Designers may review areas of concern or custom features readily enough through these data formats.
- 2) Build a small qualification batch prior to building the production order. The pain is less if data errors are found on a few boards as opposed to many.
- 3) When supplying ODB++ as a format, consider sending a backup format to compare against. Gerber as a backup is a wise choice. CAM systems can perform a layer to layer image compare. Load in the ODB++ data and Gerber and have the fabricator compare the two prior to proceeding. Any differences would be reported. Providing a net list of the design database in the IPC-356 format is also a good choice to compare against.

Regards,  
David Duross  
Engineering Director