
# The State of CAD Data Exchange October 2004

## "Are we nearly there yet?"

"Are we nearly there yet?" the dreaded question we hear wailing from the back of the car. "Yes, nearly there" is our cheery response. We didn't mean it. We might be miles away, hours more of tedious journey. The PLM industry forges forward, the CAD users seeking interoperability cry out "Are we nearly there yet?", and the vendor side of the industry in a single chorus replies "Yes, nearly there".

The vast majority of day to day CAD translation is now satisfied by good explicit translation between CAD systems. That is to say, solids are translated to solids, surfaces to surfaces and wire frame, points, layers, colours and so on, to their equivalents. The translation products have matured and the user can expect good results. For example one Theorem customer, a UK automotive OEM, recently batch translated more than 30,000 models in less than 12 hours with only 16 failures, A success rate of 99.95%. with this kind of success rate, why are we not able to say, "Yes, we're nearly there"?

The answer is that things are changing. Opportunities for improvement arise even with the CAD systems themselves. For example, a CATIA V5 assembly can comprise CATIA V5 parts and referenced geometry from a number of other CAD systems including native CATIA V4 parts. The opportunity exists therefore for a single translator to translate the hybrid assembly, and this capability has already been incorporated into Theorem's recent Version 8 release.

The need for fast, accurate explicit translation still exists and just as new requirements have appeared new solutions are required. 3D models have always had the ability to hold more than just geometry and the ASME 14.41 standard now defines how 3D annotation, including dimensions and tolerancing data should be held. This means that the user requirement for manufacturing detail to be passed with a translated model now has a technical solution and the translation vendors have a set of common standards against which they can provide workable solutions rather than just technical solutions. These pictures show a real production example of CATIA V5 model data, with 3D annotations and the resultant translation into VisMockup for enterprise wide use of the information.

There have also been changes in the use of some pre-existing data formats. Open associations have been formed to encourage some formats to become standards. The JT format was originally introduced as a "light weight" format for visualisation of CAD models in other business related applications. JT enables drawings, 3D models and assemblies to be viewed and used throughout an enterprise. It has been shown to be quite adequate for some types of CAD data exchange and since the formation of the JT Open association it is likely that it will continue to develop in importance and functionality. One major automotive OEM has terabytes of data in JT format and it may well decide that this will be the standard for exchange of information between itself and its world-wide supply chain. An action like this does not diminish the number of translations required, it will simply replace or even add to CAD to CAD translation with CAD to JT translation. Perhaps JT to CAD translation will be the next natural development. JT may therefore have the effect of increasing the number of translations that take place.

While one "open standard" develops, another is announced, and 3D XML is championed by some as the next "open standard". There is no doubt that the 3D XML protagonists have a huge following and we can expect 3D XML to become widely used, but is it going to replace all the collective CAD data held in different formats? Probably not. The most likely outcome is that both "standards" will be widely used and that they will need to co-exist. Does this mean in the future a translator to move data between JT and 3D XML?

Any consideration of standards in the CAD data translation environment must also include some thoughts about STEP. Once considered by some to be nothing more than a "super IGES" STEP has slowly but surely become a real workable standard in some major OEM's and their supply chains. Perhaps one of the best-kept data exchange secrets is that STEP specifications have now extended to include Feature and History structure. There are formal specifications for a modular extension to AP203 and it will be finalised as AP203 Edition II. It has been tested in productive use by a group of major industrial organisations, and at least one vendor of translators already has a STEP product with Feature and History capability.

So, even in this fairly simple overview of the developing requirements of the end user, and the evolution of the new standards from the industry we see a continued state of flux and a continued need for translators and their further development.

A clear indication that CAD data translation is now an accepted part of the complex mechanical design world is the emerging trend towards building translators into other environments and other applications. The stability and dependability of explicit translators now means that there is considerable benefit to be gained by initiating not just batch translations from other applications, but also from managing the pre and post processed data in product life-cycle management (PLM) systems. The result is a requirement for an automated job manager to schedule the translations, initiate the translations, perhaps share resources across a network and deliver a variety of different file types to a PLM system.

Another major change is the way in which translators are now being developed. Whereas once it was necessary to reverse engineer a translator, that is to actually de-code the source and target CAD file formats, this is no longer necessary. The PLM vendors now provide API's (application programming interfaces) that enable authors of translators to create new translators very quickly and to have a high degree of success even with early releases. This move away from dependence on reverse engineered code also reduces the risk of translators becoming redundant through a change in the data format of a CAD system.

So, the question "are we nearly there yet?" doesn't help. It is more useful to ask the question "Where are we going?", or perhaps "What do I want to do with the translated geometry?" A user who asks these questions and carefully considers the answers, has the highest likelihood of getting the right solution for his own data translation requirements.

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