Model Exchange Formats

Every CAD/CAM system uses a proprietary format to store the solid models internally, which may be quite different from other systems. The internal format primarily depends on the geometric kernel, which in turn may be proprietary or public (such as ACIS, PARASOLIDS and CASCADE). To facilitate exchange of solid models between two or more systems, the models are exported and imported in widely agreed upon standard formats. Several formats are available today. One of the earliest de-facto formats was DXF used in AutoCAD program (Autodesk, Inc.). A few other formats have been promoted by industry consortiums or countries (for example, VDAF in Germany). Currently, the most widely used exchange formats include IGES, STL and STEP, described here.

IGES: It stands for Initial Graphics Exchange Specification. It was primarily meant for surfaces, though the latest versions of IGES fully support solid models using Boundary Representation. It is the most widely used format for models with intricate contoured surfaces. |An IGES format file essentially contains an analytic representation of surface patches belong to the part. The accurate definition of surface is necessary for numerically-controlled (NC) manufacture of tooling (patterns, core boxes and moulds). However a few major CAD/CAM system developers have extended IGES in different ways, leading to minor losses of data and errors during exchange between systems. However, even a minor loss makes the file useless for NC manufacturing. Patching up or fixing the errors in an IGES file imported from a different system requires considerable effort, leading to loss of productivity.

STL: It stands for Stereolithography Tessellated Language, which originated from rapid prototyping systems. It stores the Boundary Representation of a solid model in terms of triangular facets. Thus curved surfaces have to be decomposed into a number of facets by the CAD system before exporting. Higher faceting gives a more accurate surface, but increases the memory and computation requirement. The degree of faceting can be controlled during file export by specifying the extent of deviation from the original surface. The STL file essentially contains the (x, y, z) coordinates of vertices belong to each triangular facet. It is a well defined and easy to handle format, and has therefore quickly gained wide support in CAD/CAM community. It is the preferred format for visualization and analysis programs, since these do not require accurate surface data. It is however, not suitable for NC manufacture, particularly with coarse faceting.

STEP: The Standard for Exchange of Product data is proposed and managed by International Standards Organization. The STEP handles not only geometric data related to a product, but also non-geometric information such as bill of materials, physical location of database, conformance testing procedures, display attributes, application protocols and data required by specific applications. A protocol for casting application (ISO 10303-223: Exchange of design and manufacturing product information for cast parts) has been proposed by ISO and is in the process of ratification. The protocol includes design data for cast parts (geometry, materials, tolerances, required physical and mechanical properties, and required tests); process plans for parts made by sand, die and investment casting processes; characterisation of products used to make cast parts (moulds, dies, equipment, materials and consumable items); and specifications for patterns and die assemblies. The proposed standard also includes input to and output from casting process simulation software; data exchange between customer and foundry, within the foundry, and between the foundry and supplier; use of data for foundry automation and shop floor control; and use of data for archival of design and manufacturing data. The STEP is expected to standardize product data management and eliminate the need for developing translators and formatters for different applications.

Most solid modelling systems maintain two or more representation schemes, and support several model formats for importing and exporting. In general, the CSG/features representation is useful for modifying specific features and retracing the steps during modelling, whereas the Boundary Representation is useful for visualization, geometric property computation and analysis. However, the specific format for storing CSG or Boundary information varies between different modelling systems. It is therefore not surprising for translation errors to crop up when a solid model is exchanged between two or more different systems, especially for parts having intricate curved surfaces with fillets.