

## **The IGES, DXF and STEP Exchange Formats**

### **IGES**

IGES (Initial Graphics Exchange Specification) was the first specification for CAD data exchange published in 1980 as a NBS (National Bureau of Standards) report in USA. IGES version 1.0 was accepted and released in 1981 as an ANSI standard. All major CAD vendors support IGES and it is currently by far the most widespread standard for CAD data exchange.

IGES was originally developed for the exchange of drafting data like 2D/3D wireframe models, text, dimensioning data, and a limited class of surfaces. Due to criticism and bad experience with the data transfer using IGES, the standard has been gradually extended and developed concerning supported entities, syntax, clarity, and consistency. The current version, IGES 5.2, provides the following capabilities:

- Geometry: 2D/3D wireframes, 2D/3D curves and surfaces, CSG (since version 4.0 in 1988), B-Rep (since version 5.1 in 1991);
- Presentation: Drafting entities for technical drawings;
- Application dependent elements: Piping and electronic schematics, AEC elements;
- Finite Element Modeling: Elements for FEM systems.

IGES specification defines the format of the file, language format, and the product definition data in these formats. The product definition includes geometric, topological, and non-geometric data. The geometry part defines the geometric entities to be used to define the geometry. The topology part defines the entities to describe the relationships between the geometric entities. The geometric shape of a product is described using these two parts (i.e. geometry and topology). The non-geometric part can be divided into annotation, definition, and organization. The annotation category consists of dimensions, drafting notations, text, etc. The definition category allows users to define specific properties of individual or collections of entities. The organization category defines groupings of geometric, annotation, or property elements.

An IGES file consists of six sections: Flag, Start, Global, Directory Entry, Parameter Data, and Terminate. Each entity instance consists of a directory entry and parameter data entry. The directory entry provides an index and includes attributes to describe the data. The parameter data defines the specific entity. Parameter data are defined by fixed length records, according to the corresponding entity. Each entity instance has bi-directional pointers between the directory entry and the parameter data section.

The size of IGES files and consequently the processing time are practical problems. IGES files are composed of fixed format records and each entity has to have records in both the directory entry section and the parameter data section with bi-directional pointers. This causes also errors in pre- and post-processor implementations.

IGES is under control of the NCGA (National Computer Graphics Association) and is part of the U.S. Product Data Association (USPRO) and the IGES/PDES Organization (IGO). The NCGA administers the National IGES User Group (NIUG), which provides access to information on IGES.

## **DXF**

DXF (Data eXchange Format) was originally developed by Autodesk, Inc., the vendor of AutoCAD. It has become a "de-facto" standard among most CAD vendors and is in wide use to exchange 2D/3D wireframe data. All implementations of AutoCAD accept this format and are able to convert it to and from their internal representation. A DXF file is a complete representation of the AutoCAD drawing database thus some features or concepts can't be used by other CAD systems. The DXF version R13 supports wireframe, surface, and solid representations.

A DXF file consists of four sections: Header, Table, Block, and Entity section. The header section contains general information about the drawing. Each parameter has a variable name and an associated value. The table section contains definitions of line types, layers, text styles, views, etc. The block section contains entities for block definitions. These entities define the blocks used in the drawing. The format of the entities in the block section is identical to entities in the entity section. The entity section contains the drawing entities, including any block references. Items in the entity section exist also in the block section and the appearance of entities in the two sections is identical.

Variables, table entries, and entities are described by a group that introduces the item, giving its type and/or name, followed by multiple groups that supply the values associated with the item. In addition, special groups are used for separators such as markers for the beginning and end of sections, tables, and the file itself. Group codes are used to describe the type of the value, and the general use of the group.

## **STEP**

STEP (STandard for the Exchange of Product model data) is a new International Standard (ISO 10303) for representing and exchanging product model information. It includes an object-flavored data specification language, EXPRESS, to describe the representation of the data. STEP defines also implementation methods, for instance, a physical transfer file, and offers different resources, e.g. geometric and topological representation.

The development of STEP started in 1984 as a worldwide collaboration. The goal was to define a standard to cover all aspects of a product (i.e. geometry, topology, tolerances, materials, etc.), during its lifetime. This kind of attempt was not been made before. STEP is a collection of standards to represent and exchange product information. The main parts of STEP are already international standards, while many parts are still under development. The development is performed under the control of the International

Standards organization (ISO), Technical Committee 184 (TC184, Industrial Automation Systems), Subcommittee 4 (SC4, Industrial Data and Global Manufacturing Programming Languages).

The objective of STEP is to offer system-independent mechanism to describe the product information in computer aided systems throughout its lifetime. It separates the representation of product information from the implementation methods. Implementation methods are used for data exchange. The representation offers a definition of product information to many applications. STEP provides also a basis for archiving product information and a methodology for the conformance testing of implementations.

EXPRESS is a formal data specification language used to specify the representation of product information. The use of a formal data specification language facilitates development of implementation. It also enables consistency of representation. STEP specifies the implementation methods used for data exchange that support the representation of product information.

STEP does not only define the geometric shape of a product: it also includes topology, features, tolerance specifications, material properties, etc. necessary to completely define a product for the purposes of design, analysis, manufacture, test, inspection and product support. The use of STEP is still very modest but it is growing all the time. The majority of CAD system vendors has implemented or is implementing STEP pre- and post-processors for their CAD systems. STEP is an evolving standard that will cover the whole product life cycle in terms of data sharing, storage and exchange. It is the most important and largest effort ever established in engineering domain and will replace current CAD exchange standards.