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Application of Step AP224 in Computer Aided Manufacturing

G. Ashok
Department of Mechanical Engineering,
BKIT, Bhalki, (Karnataka), INDIA

(Corresponding author: G. Ashok)

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ABSTRACT: The major manufacturing software provider has taken an important step that will enable CNC machine tool users to benefit from emerging data-communications standards designed to promote a truly paperless manufacturing environment. The paperless factory doesn't only mean that product designers, manufacturing engineers and CNC programmers share product manufacturing information strictly by computer rather than paper drawings. "Paperless" implies that data formats requiring human interpretation will be obsolete. That means moving away from the electronic equivalent of an engineering drawing. The key is STEP AP224 interface.

I. INTRODUCTION

Manufacturing personnel have been interpreting information conveyed by engineering drawings for centuries. Making manufacturing far less reliant on the human interpretation of this information (whether delivered on paper or by computer) is the real promise of the paperless factory. It's not only the sheets of pressed wood fiber that must become unnecessary. What must also be eliminated are the numerous steps where in a person must intellectually analyze and assign meaning to information before further computer processing can proceed. In a paperless factory where "paperless" applies in its fullest sense, part production would be seamless and highly automated. Efforts to get manufacturing computer systems to work together seamlessly and achieve higher levels of automation have been going on for years.

In one encouraging development, a manufacturing software provider has completed an interface that allows its CAM offerings to read product data in a standardized file format that includes both machinable features and critical specifications necessary for machining. The format for the product data is defined by STEP Application Protocol 224 (AP224). STEP (Standard for the Exchange of Product model data) is the international standardization effort to alleviate obstacles to the exchange of product data among different and otherwise incompatible hardware and software. STEP is the much-expanded outgrowth of the effort that created IGES, the Initial Graphics Exchange Specification, in the 1980s.

STEP includes the development of a "manufacturing suite" of standardized data codes and file formats (protocols) for the data requirements that arise among the major phases (applications) of designing and manufacturing products. One of those application protocols is AP224.

AP224 specifically supports the transfer of manufacturing product information between design engineering and manufacturing engineering groups. It focuses on mechanical parts and covers all the information necessary to machine these parts. AP224 captures part features, geometric dimensions and tolerances, material, surface finish and other part data that are needed in machining.

Like all STEP application protocols, AP224 encodes the information so that it can be interpreted directly by a computer system. The encoding is neutral—that is, not dependent on any particular computer system.

II. ABOUT STEP AP224

AP224 is one of a set of application protocols created under STEP for the manufacturing community. This "manufacturing suite" represents a number of interrelated standards that address the specific needs that manufacturing companies have for product model data. Because discrete, machined parts are critical to many products, including military aircraft and fighting vehicles, the product model data that covers machining have received a lot of attention. AP224's official title is "Mechanical Product Definition for Process Planning Using Machining Features.

AP224 has two very important aspects that all CNC machine tool users should understand it deals with machining features, and the data codes are for computer interpretation, not just display on a computer, much of the activity associated with NC programming involves recognizing and evaluating features for machining. Without any additional notation or indications, part geometry simply describes a shape, using its surfaces or its volumes as a solid entity. It is necessary to interpret this shape to identify the various portions of this geometry as representations of particular part features such as pockets, bosses, holes, fillets and so on. Likewise, these features must be analyzed according to their tolerances because the tightest and/or most difficult of these will usually determine which machining processes are the most appropriate and which sequence is the most advantageous. Finally, the machining steps and tool paths for producing these features in a blank work piece have to be organized by the NC programmer and processed by the CAM software.

Along with other manufacturing data, AP224 captures information about the machining features embodied in the geometric model of a part. The standard covers 16 different categories of features.

- Boss
- Pocket
- Hole
- Slot
- Protrusion
- Rounded_end
- Outer_round
- Step
- Planar_face
- Revolved_feature
- Spherical_cap
- General_outside_profile
- Thread
- Marking
- Knurl
- General_volume_removal

The standard also describes three different types of feature transitions namely, fillet, edge_ round and chamfer. Replicate features – Three different ways to replicate features**,** there is also a mechanism for describing a pattern of features. Each feature is 'explicitly' described using parameters. Union of one or more feature yields a compound feature.

Open_profile

- Square U profile
- Rounded U profile
- Linear profile
- Vee profile
- Tee profile
- Partial circular profile
- General open profile

Closed_profile

- Circular closed profile
- Rectangular profile
- Ngon profile
- General closed profile

Profiles

Paths

Open_profile

- Linear path
- Complete circular path
- Partial circular path
- General path

Tolerances are very important for manufacturing. AP224 caters for dimensional tolerances, size tolerances, location tolerances and geometric tolerances. The model description also contains various other miscellaneous manufacturing information like material, alternate material, material properties, notes and specifications.

Dimensional Tolerances

Representation of geometric dimensions and of tolerances limiting geometric dimensions. Tolerances for geometric dimensions describe the allowable deviation range and that are characterized by:

Plus-minus-tolerances

Dimension ranges

Dimension limits

Size Tolerances

-Radial

-Diameter

-Curve dimension

-Angular size

Location Tolerances

-Angular

-Location

-Distance along curve

Tolerance values

-Plus minus value

-Tolerance limit

-Tolerance range

-Limits and fits

Geometric Tolerances

-Geometric tolerances support other ISO standards

- ISO 1101 "Tolerancing of form, orientation, location and run-out"
- ISO 5459 "Datums and datum systems for geometric tolerances"

-Geometric tolerances support US national standards

- ANSI Y14 "American Standard, Dimensioning and Tolerance"

-representation of geometric tolerances with a datum reference **Example:** parallelism or perpendicularity

-representation of geometric tolerances without a datum reference **Example:** straightness or flatness

-allows the definition of up to three datum references that are either single datum, compound datum, or datum targets

- additionally the specification of tolerance zones

-definition of the presentation of tolerance information is not dealt with in AP224

-AP224 Part Administration Data

- Approval
  - Manufacture authorization for a part
  - Release authorization of a project order for a part
- Customer_order
  - Request from customer to purchase a desired part contains:
  - delivery date
  - order identification number
  - quantity of parts being manufactured
- Organization
  - responsible for manufacturing
  - purchaser of part
  - approval authority
- Person
  - responsible for manufacturing
  - purchaser of part
  - approval authority

III. BENEFITS OF STEP AP224

1. The use of STEP AP224 driven manufacturing technology will show a distinct advantage in the time required to prepare a part for manufacture over the conventional product data and process planning technology.
2. All tolerance information, material information, machining feature information and other special requirements are completely defined as STEP data.
3. Accurate and complete capture of design for a mechanical piece part in an intelligent digital format
4. Highly economical for small part lots
5. Effectively exchange the product data information between disparate computer systems.
6. Use of product data to enable computer automated downstream systems
7. Electronically archive product data so that it is available when needed in the future

IV. CONCLUSION

The product data representation and exchange standard STEP provides a powerful support for CAD/CAPP/CAM integration. The integrated resources of STEP give a general description of product data, which can be the basis of the global product data model for integration. The implementation methods of STEP ap224 have offered the mechanisms for data exchange and sharing for integration. For specific applications, application protocols are demanded for standard and consistent implementations. The formal description of STEP ap224 guarantees the possibility of software tools to support the implementation methods, so as to the system integration.

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