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Neutral Representation of Tolerance Information for Process Planning Using STEP AP 224

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The focus of CIM is on information as it is the crucial element for integrating all the manufacturing activities. CAPP, as one of the key elements in CIM, needs to extract the manufacturing information such as machining features and precision specifications like surface roughness and tolerances from a geometric model in order to link CAD and CAM. However, these data are not real attributes of the geometric model in most of the current CAD systems. Therefore, human interpretation is inevitable for further processing of CAD model for downstream application like process planning or inspection. This paper proposes a scheme to represent the manufacturing information in a neutral format using STEP technology in order to enable downstream users such as process planner and inspection planner to make correct decisions on process selection, processing conditions, etc. It is shown that by using STEP AP224 manufacturing information encompassing machining features, surface roughness, dimensional and geometric tolerances can be completely represented together with part geometry, which certainly contributes to successful implementation of CIM.

Key Words : Manufacturing Information, Machining Feature, Tolerance, Surface Roughness, STEP

1. Introduction

CIM (Computer Integrated Manufacturing) is a conceptual basis for integrating the applications and information flow of product design, process planning, production planning, and manufacturing processes. The focus of CIM is on information as it is the crucial element linking all facets of the manufacturing enterprise. While the geometry information is created from the design activity, the manufacturing information is concerned with the process planning, production planning and plant operations. Given a part geometry, CAPP (Computer Aided Process Planning), the bridge between CAD (Computer Aided Design) and CAM (Computer Aided Manufacturing), generates a sequenced set of instructions to manufacture the specified part. To do so, CAPP has to extract manufacturing information such as machining features and precision specifications including surface roughness, dimensional and geometric tolerances in order to select the necessary processes and determine the operation conditions. Despite a lot of efforts in the past to interlink design and process planning, sharing of manufacturing information still remains as a bottleneck (Roh 1993; Maropoulos 1995). One of the reasons is that the tolerance and surface finish data are not embedded in the geometric model. At a glance, CAD models seem to incorporate these data as seen in the drawings. However, as a matter of fact, these data are not real attributes of CAD models but simply represented as texts on the drawing like technical notes. This results from the shortcoming that most of the current CAD systems do not have the appropriate data structure to accommodate them. Therefore, when a CAD

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model is to be transferred to downstream users like process planner or inspection planner, every user repeatedly needs to regenerate the necessary manufacturing information through human intervention. To avoid this inefficiency, a data structure should be developed, in which manufacturing information as well as geometry data can be stored. At the same time, a neutral format for the representation would be desirable for facilitating the interface between disparate computer systems. STEP, which is a standard for the exchange of product model data defined as the international standard ISO 10303(Owen 1993; ISO 1994a; 1994b), includes not only geometrical but also technical and managerial information, and thus gives a clue to the solution.

The goal of this research is to propose a method to represent manufacturing information in a neutral format in order to enable downstream users to make correct decisions on process selection, processing conditions, etc. , which forms the prerequisite for automating post-CAD activities like process planning, assembly planning or inspection planning, as shown in Fig. 1. In order to implement the proposed scheme for representing manufacturing information, three problems should be solved, namely, recognition of machining features, incorporation of technical information such as surface roughness and tolerance, and realization of neutral interface. As the feature recognition kernel, *IF^2, Integrated Incremental Feature Finder*, is used. The kernel was developed at USC (Han 1996; Han and Requicha 1998a) and has been extended at NIST(National Institute of Standards and Technology) and Sung Kyun Kwan University to include the setup cost minimization. For preparing technical information, a tolerance processor is developed. To ensure the neutrality of the outcome, the STEP application protocol AP224, which stands for mechanical part definition for process planning using machining features(ISO 1994b), is applied.

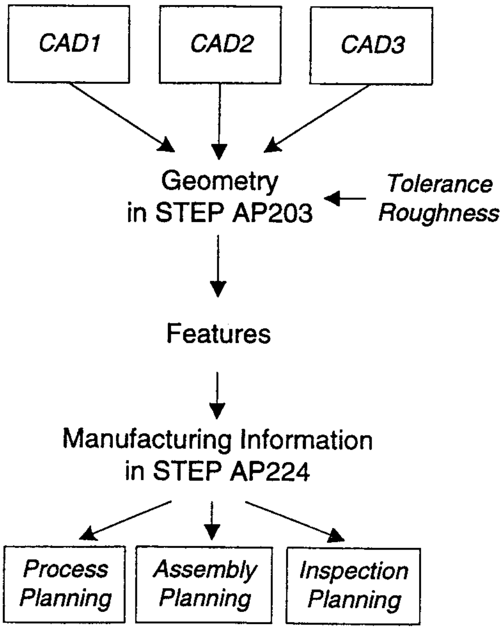


Fig. 1 Use of manufacturing information represented in a neutral format

Before going into details of the proposed framework, previous works on feature recognition, handling of technical data, and STEP application are briefly described.

2. Previous Works

Feature recognition has been the subject of research since the seminal work of Kyprianou (1980). Among a number of methods, four distinct approaches are currently attracting attention; graph pattern matching, convex hull decomposition, cell-based decomposition, and hint-based reasoning. Consult the work by Han(1998b) for a critical survey of these approaches. Despite two decades research, the impact of features technology has been insignificant, and the results have rarely been transferred into industry. One of the reasons is that feature recognition approaches proposed so far have not been in accordance with the requirements of CAPP. Either the coverage of feature recognition is limited to some ideal geometric shapes, or they do not sufficiently take into account the manufacturability issue like manufacturing cost minimization(Park et. al. 1995; Suh 1994; Han 1997; Park et. al. 1999). Meanwhile, some feature recognition approaches focused on manufacturability are known. Gupta(1994) used branch-and-bound algorithm to generate an optimal feature model. Similarly, Sormaz(1994) used

A* algorithm for optimal process planning. However, feature precedence relations are not precisely defined in their systems. *IF^2, Integrated Incremental Feature Finder*, utilized in this research, takes into account the manufacturing setup cost minimization with the aid of search algorithm (Han 1996; Han 1997; Han and Requicha 1998a).

Technical information such as dimensions, surface condition and tolerance of geometric characteristics dictates the machining requirements and crucially affects the product cost. Therefore, these specifications have been principally examined from the viewpoints of functionality and cost (Park et. al. 1996a; Park et. al. 1996b; Choi et. al. 1998; Choi et. al. 1999). Few works have been done on incorporating the technical information into geometric model, and commercial CAD systems have disregarded this issue. Bley et al. (1999) and Wittmann (1999) suggested a concept of tolerance information system which provides designers with an integrated environment to make use of tolerance related information such as cost, machining time, and feasibility. Even though they take into consideration both the functional and the manufacturing viewpoints of tolerance, their approach may be regarded as a kind of technical information management dedicated to a CAD system in use. Ha et al. (1999) proposed a tolerance representation scheme to integrate geometry and tolerance information. Through a user interface, tolerance types and values can be assigned to the selected entities. The outcome is an integrated geometry and tolerance model in an ad hoc format. Moreover, the system is bound to a specific geometric kernel, ACIS, requiring the geometric model to be exclusively in ACIS format. Thus, the suggested system is very restricted in its portability because of its geometry input method only through ACIS and the output format for tolerance model that is neither standard nor neutral.

Data exchange not only between CAD packages but also between CAD, CAPP, and CAM systems can be effectively done through a neutral standard format. Among many data exchange formats developed, DXF (Drawing Transfer File), IGES (Initial Graphics Exchange Standard) and STEP (STandard for the Exchange of Product model data) are most widely accepted. In contrast to DXF and IGES, STEP is aimed to define a standard file that includes all information necessary to describe a product from design to production. It supports multiple application domains, for instance, mechanical engineering, electronics, architecture (Owen 1993). STEP AP224, which is a mechanical part definition for process planning using machining features, contains all of the information needed to manufacture the required part, including materials, part geometry, dimensions and tolerances, applicable notes and specifications, and administrative information. Current scope of the STEP224 is restricted to single mechanical part manufactured by milling or turning process (ISO 1994b; SCRA 1995b). Although a lot of works applying STEP AP203, configuration-controlled design, have been reported in the literature (Kim 1997; Ahn and Yoo 1998), researches dealing with STEP AP224 are rarely found. SCRA (South Carolina Research Authority) (1995a; 1995c) team has conducted a couple of researches investigating the CAD-independent applicability of STEP standards in interfacing design and process planning (SCRA 1995a; SCRA 1995c). It has been attempted to implement the information flow from design to manufacturing in a distributed and networked environment by describing manufacturing information such as material, process property, specifications, surface property, and administrative data in a note block of STEP file. However, the relationship between these data and geometric entities is not represented, and therefore intervention of a process planner is still required to retrieve the manufacturing information.

3. Preparation of Manufacturing Information

In order to preserve the independency, STEP AP203 is taken as the input file format, which can be generated from most of commercial CAD systems. Among the various methods for geometry representation provided by STEP AP203, only

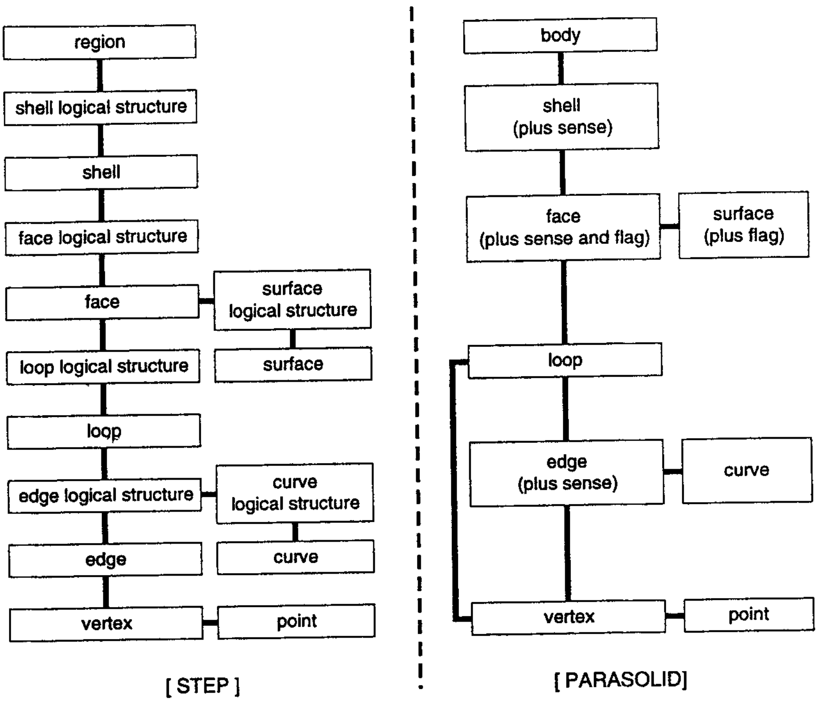
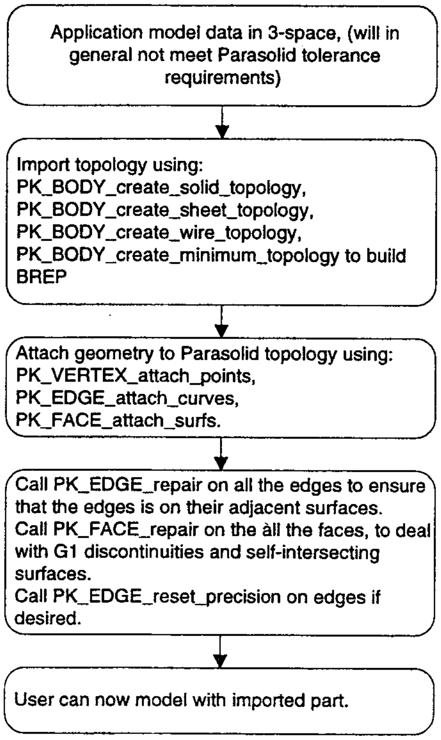


Fig. 2 Entity structures of STEP AP203 and Parasolid

the Brep (Boundary Representation) is taken into account in this study because most mechanical parts can be modeled based on BRep. For interpreting STEP files, the EXPRESS information model for AP203 is compiled to produce C^{++} classes using ROSE library (STEP-Developer 1996). EXPRESS is a data definition language widely adopted in the STEP society (Schenk and Wilson 1994). By linking these classes created by ROSE together, the developed application program can interpret and convert STEP data into Brep data structure. The BRep entities are then translated into a Parasolid model by using Parasolid API (Application Protocol Interface) functions, where Parasolid is a commercial geometric modeling kernel (UniGraphics 1998). Since the entity structure of STEP AP203 is not identical to that of Parasolid as shown in Fig. 2, STEP's BRep model should be processed to match with that of Parasolid. Figure 3 describes the detailed procedure of STEP to Parasolid translation.

Given the geometric model in Parasolid, tolerance information is to be attached to the Parasolid model. Technical information can be clas-



Application model data in 3-space, (will in general not meet Parasolid tolerance requirements)

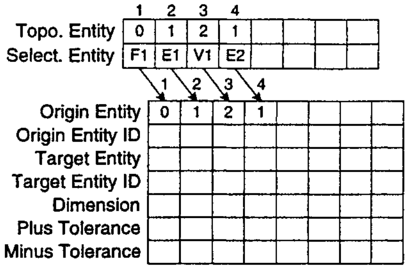
Import topology using:
PK_BODY_create_solid_topology,
PK_BODY_create_sheet_topology,
PK_BODY_create_wire_topology,
PK_BODY_create_minimum_topology to build BREP

Attach geometry to Parasolid topology using:
PK_VERTEX_attach_points,
PK_EDGE_attach_curves,
PK_FACE_attach_surfs.

Call PK_EDGE_repair on all the edges to ensure that the edges is on their adjacent surfaces.
Call PK_FACE_repair on the àll the faces, to deal with G1 discontinuities and self-intersecting surfaces.
Call PK_EDGE_reset_precision on edges if desired.

User can now model with imported part.

Fig. 3 Procedures of STEP AP 203 to Parasolid translation sified into two groups depending on whether it is self-referenced or needs a cross-reference. To the former belong surface roughness, straightness, flatness, cylindricity, and so on. Dimensional tolerance, parallelism, concentricity, perpendicularity, angularity, etc. are typical examples of the latter. The self-referenced tolerance can be treated as an attribute of an entity. For instance, the surface roughness is stored simply as a surface attribute of a Parasolid model. In contrast, the cross-referenced tolerance implies a characteristic between two entities. To this belong linear dimensional tolerance, perpendicularity, parallelism, angularity, concentricity, and so on. For example, an entity couple for linear dimensional tolerance can be face to face, face to edge, face to vertex, edge to vertex, or vertex to vertex. Since Parasolid does not provide a data structure to incorporate tolerance information, it is necessary to implement an appropriate one in order to store tolerance values within the Parasolid model. Figure 4 shows a specially designed 2D array data structur-

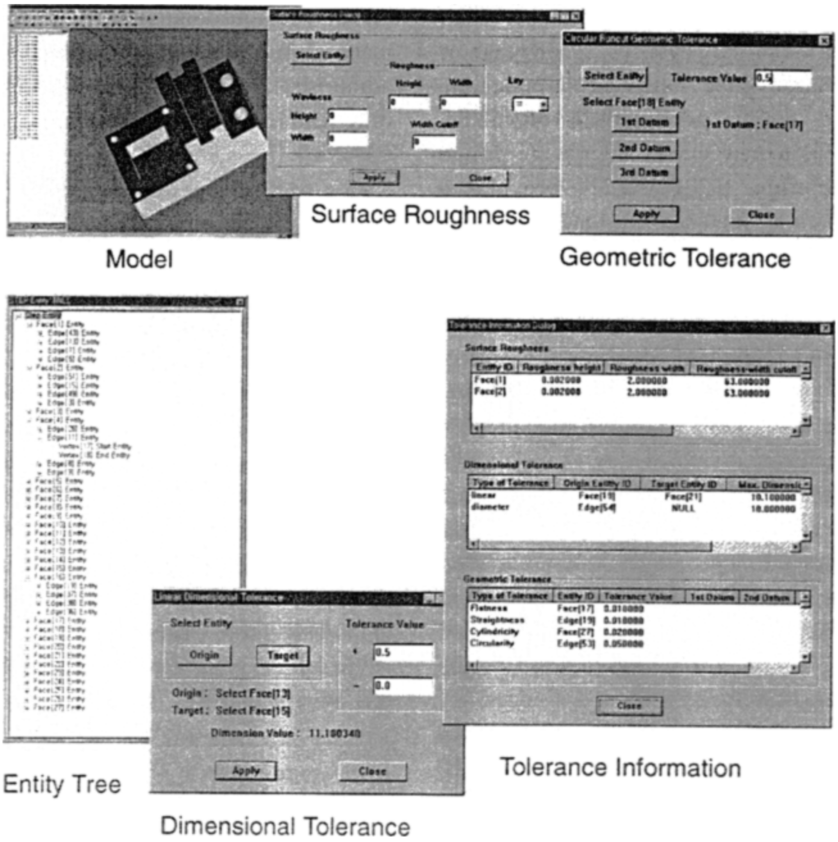


	1	2	3	4				
Topo. Entity	0	1	2	1				
Select. Entity	F1	E1	V1	E2				

	1	2	3	4				
Origin Entity	0	1	2	1				
Origin Entity ID								
Target Entity								
Target Entity ID								
Dimension								
Plus Tolerance								
Minus Tolerance								

```
typedef struct
{
    Origin Entity;
    Origin Entity ID;
    Target Entity;
    Target Entity ID;
    Dimension;
    Plus Tolerance;
    Minus Tolerance;
};
```

Fig. 4 Data structure for linear dimensional tolerance



Tolerance Value 0.5

Select Face[18] Entity

1st Datum : Face[17]

\+ 0.5

\- 0.0

Origin : Select Face[13]

Target : Select Face[15]

Dimension Value : 11.180340

Surface Roughness

Entity ID	Roughness height	Roughness width	Roughness-width cutoff
Face[1]	0.002000	2.000000	63.000000
Face[2]	0.002000	2.000000	63.000000

Dimensional Tolerance

Type of Tolerance	Origin Entity ID	Target Entity ID	Max. Dimensio
linear	Face[19]	Face[21]	10.100000
diameter	Edge[54]	NULL	10.000000

Geometric Tolerance

Type of Tolerance	Entity ID	Tolerance Value	1st Datum	2nd Datum
Flatness	Face[17]	0.010000		
Straightness	Edge[19]	0.010000		
Cylindricity	Face[27]	0.020000		
Circularity	Edge[53]	0.050000		

Fig. 5 Snapshots showing technical information processing

e to store the datum and the target entity for linear dimensional tolerance. Tolerance value is assigned interactively via a graphical user interface as depicted in Fig. 5. The datum and the target entity are selected from the visualized model, and the tolerance types and allowance values are assigned. Some information relevant to manufacturing, for example special comments, can also be added in the form of text attributes. From the Parasolid geometric model with tolerance information added, *IF^2* generates an optimal feature model that minimizes the overall setup cost.

4. Tolerance Information Representation Using STEP AP 224

Machining features with relevant technical information are crucial for process planning. Automatic input of this information to a process planning system has been a troublesome problem. For a long time there has been no better alternative for that than part description language or part classification code based on group technology. The recent development of STEP standard opens a new era of interfacing design and manufacturing. STEP AP224 provides a good foundation for sharing manufacturing information between design and manufacturing engineering. Part 47, shape tolerance resource model, specifies the resource rules to represent dimensions and tolerances of product geometry (ISO 1999a). It belongs to *Integrated Generic Resources* on the conceptual layer that provide the generic integrated information model, while Part 519, *Application Interpreted Construct*: geometric tolerances, groups the information resource entities for geometric tolerances that make it easier to express identical semantics in application protocols (ISO 1999b). AP 224, also called Part 224, is an *Application Protocol* on the external layer which analyzes and utilizes the entities defined in the integrated resources from the viewpoint of a specific application domain. In addition, application protocols can also contain the conformance requirements and the characteristics of implementation methods. Therefore, it is considered sufficient to consult the specific application protocol when developing an application program. Although STEP AP224 includes all the necessary manufacturing information like material, specifications and special notes, or other administrative information as shown in Fig. 6, only feature related information is considered in this work.

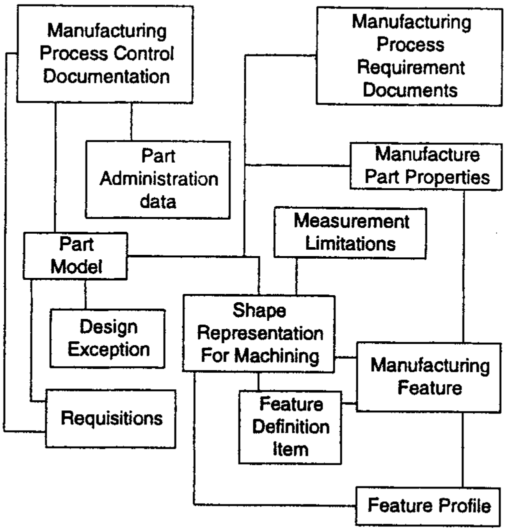


Fig. 6 STEP AP224 schema

In a reverse way of interpreting STEP AP203 file as explained in the previous section, machining features and tolerance information as well as geometry can be converted into STEP AP224 format. Beginning with the lowest elements like vertex to higher ones like loop or face, the Parasolid entities are transformed into STEP 224 entities using ROSE functions. Since the schema of AP 224 does not include the entities for storing tolerance data as shown in Fig. 7, tolerances cannot be assigned in a unique manner. A simple method to represent the tolerances in STEP is to store them as plain text, which is the way taken in the RAMP project at NIST (SCRA 1995a; SCRA 1995c). If the surface quality data are expressed in this way, it looks as follows:

```
# 808 = PROPERTY_DEFINITION('surface_roughness', 'a=63.000000 b=0.010000 c=0.005000 d=0.002000 e=2.000000 f=\X\A1\X\BF', $);
```

The drawback of this approach is that the format

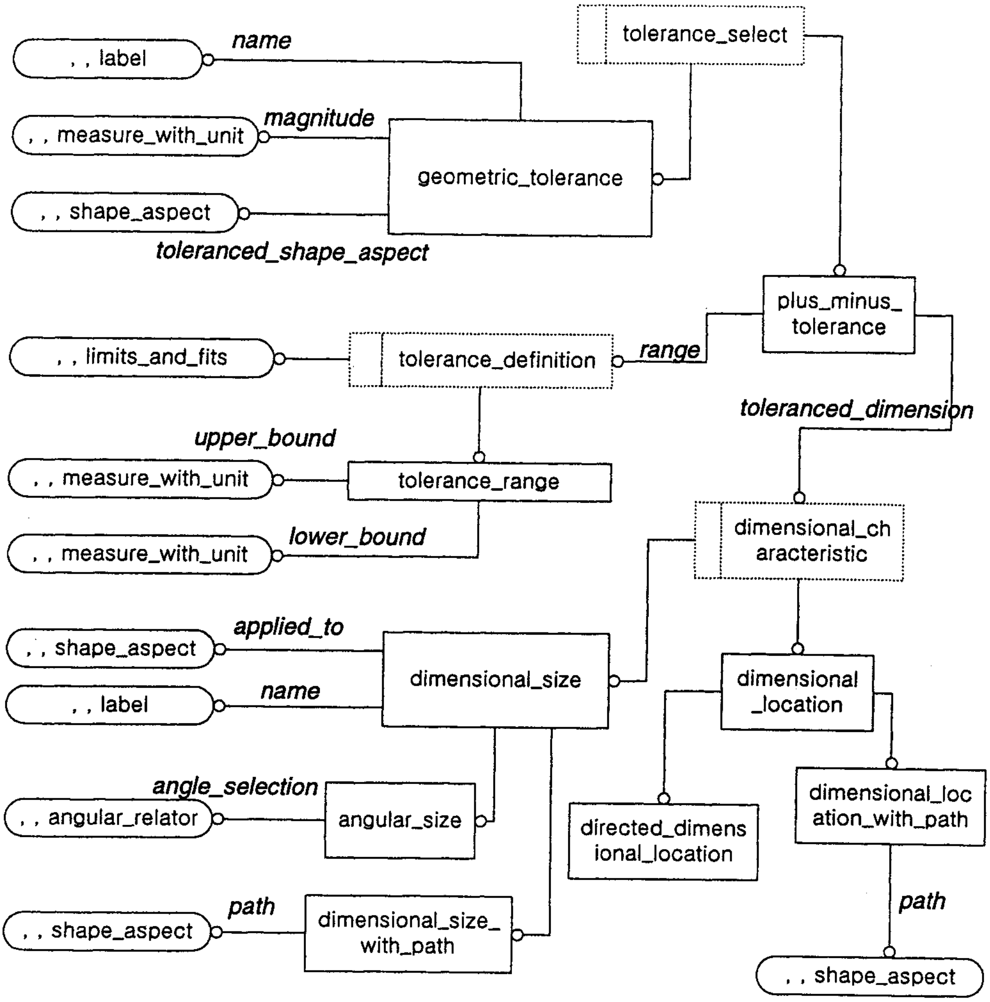


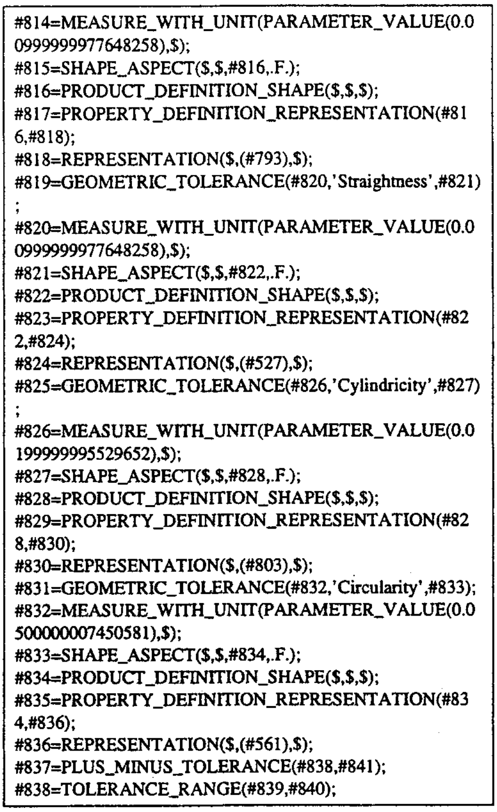
Fig. 7 Schema of AP 224 for dimensional and geometric tolerances

and meaning of every position in the text must be specified when a STEP file is delivered. In the example shown above, one has to know the alphabets **a** to **f** mean roughness width, roughness height, roughness width cutoff, waviness width, waviness height, and lay respectively. Another method would be to extend the AP 224 schema to include tolerance values. In this case, one needs to make the extended schema known to the users. Representation of surface roughness using the extended schema looks as follows:

```
#807=SURFACE_ROUGHNESS($, $, #813, #808,
#809, #810, #811, #821, \X\A1\X\BF’,”);
#808=MEASURE_WITH_UNIT(PARAMETER_VALUE
(0.009999999776482582),$);
#809=MEASURE_WITH_UNIT(PARAMETER_VALUE
(63.),$);
#810=MEASURE_WITH_UNIT(PARAMETER_VALUE
(0.00499999988824129),$);
#811=MEASURE_WITH_UNIT(PARAMETER_VALUE
(2.),$);
#812=MEASURE_WITH_UNIT(PARAMETER_VALUE
(0.0020000000949949),$);
#813=SHAPE_ASPECT($,$,#814,.F.);
```

In this research, both methods have been implemented.

Once a physical STEP file of AP224 format is generated, downstream users can proceed with



```
#814=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.0
0999999977648258),$);
#815=SHAPE_ASPECT($,$,#816,.F.);
#816=PRODUCT_DEFINITION_SHAPE($,$,$);
#817=PROPERTY_DEFINITION_REPRESENTATION(#81
6,#818);
#818=REPRESENTATION($,(#793),$);
#819=GEOMETRIC_TOLERANCE(#820,'Straightness',#821)
;
#820=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.0
0999999977648258),$);
#821=SHAPE_ASPECT($,$,#822,.F.);
#822=PRODUCT_DEFINITION_SHAPE($,$,$);
#823=PROPERTY_DEFINITION_REPRESENTATION(#82
2,#824);
#824=REPRESENTATION($,(#527),$);
#825=GEOMETRIC_TOLERANCE(#826,'Cylindricity',#827)
;
#826=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.0
199999995529652),$);
#827=SHAPE_ASPECT($,$,#828,.F.);
#828=PRODUCT_DEFINITION_SHAPE($,$,$);
#829=PROPERTY_DEFINITION_REPRESENTATION(#82
8,#830);
#830=REPRESENTATION($,(#803),$);
#831=GEOMETRIC_TOLERANCE(#832,'Circularity',#833);
#832=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.0
500000007450581),$);
#833=SHAPE_ASPECT($,$,#834,.F.);
#834=PRODUCT_DEFINITION_SHAPE($,$,$);
#835=PROPERTY_DEFINITION_REPRESENTATION(#83
4,#836);
#836=REPRESENTATION($,(#561),$);
#837=PLUS_MINUS_TOLERANCE(#838,#841);
#838=TOLERANCE_RANGE(#839,#840);
```

Fig. 8 Excerpt from a STEP 224 file representing dimensional and geometric tolerances

their process planning or inspection planning work, with provision of a STEP 224 interpreter, which is to be mentioned in the example of following section. Figure 8 shows the dimensional and geometric tolerance description part of an example STEP 224 file.

5. Example

An example describing the workflow to prepare manufacturing information from a geometric model and represent it in a neutral data format is shown in Fig. 9. At first a STEP 203 file generated from an arbitrary CAD system, in this work from UniGraphics modeler, is imported and each line of the STEP file is interpreted according to the protocol of AP203. The interpreted geometry is transformed into Parasolid entities, which can be visualized in a solid representation to confirm the correctness of STEP 203 import. Using Parasolid model, required tolerance values like dimensional or geometric tolerance are assigned. From the geometric model with this technical information machining features are extracted to give the feature list together with technical information, which are then translated into a STEP AP224 format. STEP 224 file is CAD-independent and contains the relevant data to manufacture the modeled part including surface roughness, dimensional and geometric tolerances.

6. Discussion and Conclusion

Three issues necessary to interface design and manufacturing engineering have been addressed: recognition of machining features, handling of technical information, and implementation of neutral interface. Emphases have been put on the representation of tolerance information by using the neutral product data format STEP. Unlike the works done by SCRA, where manufacturing data were described as text in a note block of STEP file, a proper data structure to store various types of tolerance and surface finish data has been proposed and implemented. This kind of implementation can help CAPP system extract manufacturing information contained in STEP AP224 file more easily, regardless of the CAD systems in use. The author believes that this work contributes towards removing the main barriers, namely, difficulty in recognizing machining features and manufacturing informations, to computer-automated downstream systems like process planning or inspection planning. Despite the achievements in handling manufacturing information, recognition of relevant features of complex shapes still remains as a bottleneck. Without a mature feature recognizing system, a seamless interface between design and manufacturing can not be accomplished.

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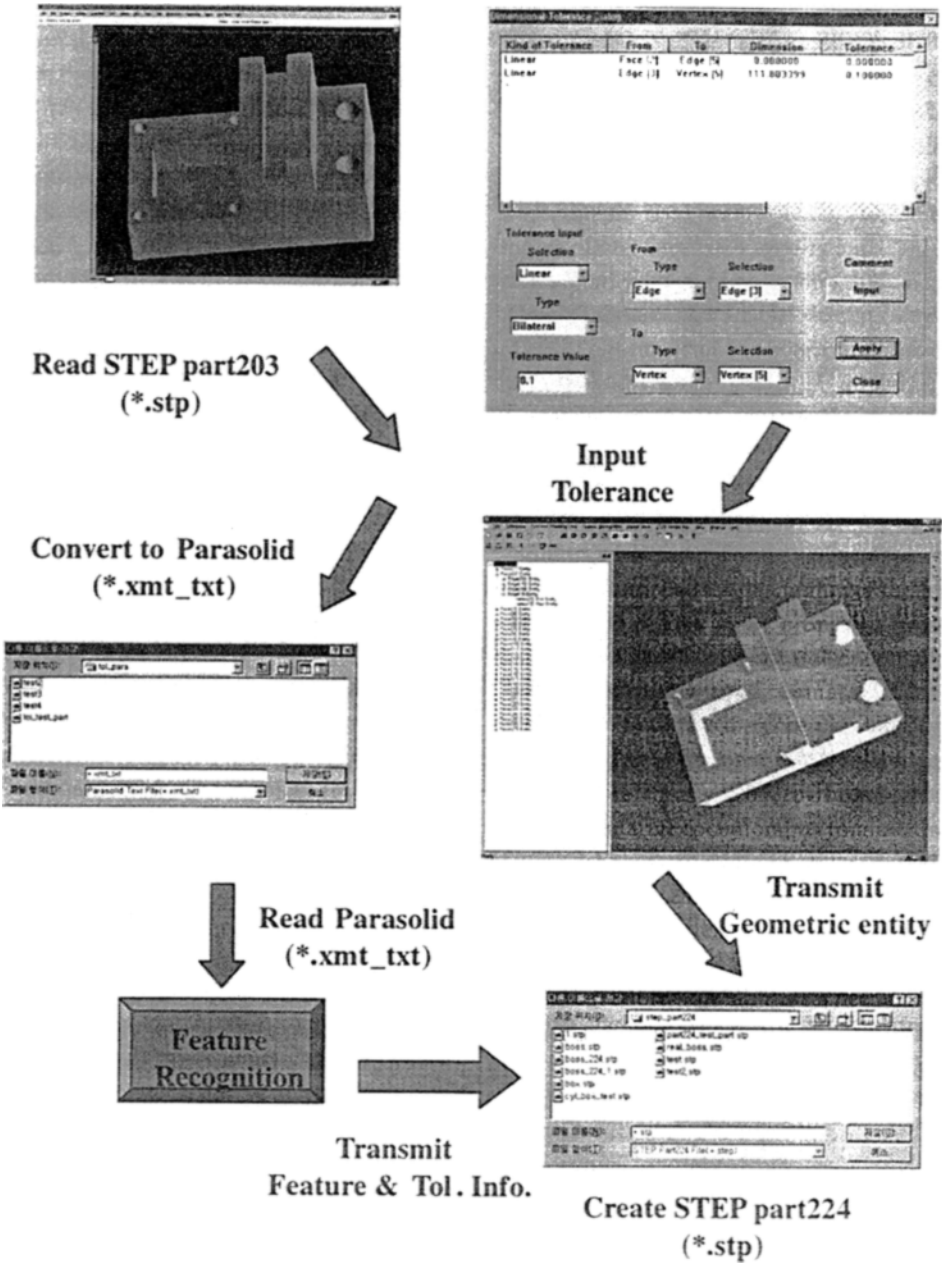

Fig. 9 Workflow to prepare manufacturing information from a geometric model

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