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An approach for interlinking design and process planning

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Abstract

Interlinking design and process planning plays a key role in realizing Computer Integrated Manufacturing (CIM). Given a part geometry from a CAD system, CAPP generates a sequenced set of instructions to manufacture the specified part. In order to do that, CAPP has to recognize manufacturing features of the part and the relevant information about precision requirements such as surface roughness as well as dimensional and geometric tolerances. Since geometric models from most of the current CAD systems do not incorporate this manufacturing information, human intervention at the first stage of CAPP is inevitable. This has been a major hindrance to information flow between design and process planning. This paper proposes an approach for interlinking CAD and CAPP, and describes the relevant efforts towards it: recognition of machining features, handling of manufacturing information, and implementation of a neutral interface using ISO 10303-224.

Keywords: Design; Process planning; Manufacturing information; Machining feature; Tolerance; Surface roughness; STEP

1. Introduction

Computer Integrated Manufacturing (CIM) 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 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, Computer Aided Process Planning (CAPP), the bridge between Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM), generates a sequenced set of instructions to manufacture the specified part. To do that, CAPP has to extract manufacturing information such as machining features and precision specifications including surface roughness, and dimensional and geometric tolerances in order to select the necessary processes and determine the operation conditions. Despite a lot of effort done in the past to interlink design and process planning, sharing of manufacturing information still remains a bottleneck [1–3]. One of the reasons is that 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 text on the drawing the same as technical notes. This results from most of the current CAD systems not having the appropriate data structure to accommodate them. Therefore, when a CAD model is to be transferred to downstream users such as the process planner or the inspection planner, every user repeatedly needs to regenerate the necessary manufacturing information through human intervention. To avoid this inefficiency, an integrated product model should be achieved, in which manufacturing information and geometry data can be stored together. At the same time, a neutral format for the representation would be desirable for facilitating an interface between disparate computer systems. STandard for the Exchange of Product (STEP) model data, which is defined as the international standard ISO 10303 [4–6], includes not only geometry but also technical and managerial information, and thus gives a clue to the solution.

This paper proposes an approach to interlink design and process planning by representing manufacturing information together with part geometry in an integrated product model based on the STEP neutral format. Fig. 1 shows the problems to be considered for interlinking design and process planning, namely the recognition of machining features, the incorporation of manufacturing information such as surface

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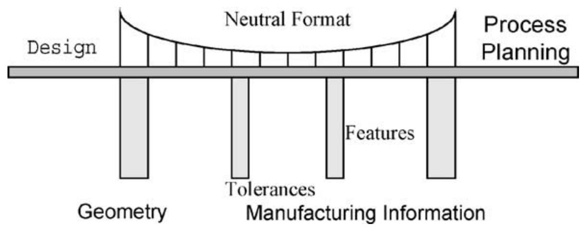

Fig. 1. Elements needed to bridge design and process planning.

roughness and tolerances, and the implementation of a neutral interface.

2. Previous work

2.1. Feature recognition

Feature recognition has been the subject of research since the seminal work of Kyprianou [7]. 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 [8] 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 manufacturability issue such as manufacturing cost minimization [9]. At the same time, some feature recognition approaches focused on manufacturability are known. Gupta [10] used a branch-and-bound algorithm to generate an optimal feature model. Similarly, Sormaz [11] used A^* algorithm for optimal process planning. However, feature precedence relations are not precisely defined in their systems. *Integrated Incremental Feature Finder* (IF^2) , utilized in this research, takes into account the manufacturing set-up cost minimization with the aid of a search algorithm [9,12–14].

2.2. Tolerance information processing

Technological information such as dimensioning, 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. Few works have been done on incorporating the technological information into the geometric model, and commercial CAD systems have disregarded this issue. Bley et al. [15] and Wittmann [16] suggested a concept of a 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 a kind of technical information management dedicated to a CAD system in use. Ha et al. [17] 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 the tolerance model that is neither standard nor neutral.

2.3. Neutral interface for manufacturing information exchange

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, Drawing Transfer File (DXF), Initial Graphics Exchange Standard (IGES) and STEP model data are the 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 [4]. STEP AP224, 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. The current scope of the STEP224 is restricted to a single mechanical part manufactured by a milling or turning process [6,18]. Although a lot of work applying STEP AP203, configuration-controlled design, has been reported in the literature [19,20], research dealing with STEP AP224 is rarely found yet.

The South Carolina Research Authority (SCRA) team has conducted a couple of researches investigating the CAD-independent applicability of STEP standards in interfacing design and process planning [21,22]. An attempt has been made to implement 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 a STEP file. However, the relationship between these data and geometric entities is not represented, and therefore the intervention of a process planner is still required to retrieve the manufacturing information. In addition, tolerances are stored as plain text, which results in the format and meaning of every position in the text having to be specified when a STEP file is delivered.

3. Proposed framework

Fig. 2 shows the data flow diagram of the proposed framework. It comprises a STEP AP203 interpreter, a Parasolid translator and manipulator, a tolerance processor, a feature recognizer, and a STEP AP224 generator. STEP AP203 interpreter extracts geometric entities from AP203 files, which are then converted into a Parasolid model by the Parasolid translator. Parasolid is a commercial geometric modeling kernel [23], which is used as the fundamental geometry kernel in this whole framework. The use of Parasolid is merely trivial: it could be any other one such as ACIS. The generated model is to be investigated against its correctness using the Parasolid manipulator. The tolerance processor assigns the relevant technological information such as surface roughness, and dimensional and geometric tolerances. The outcome is a geometry model with tolerance assignments, from which machining features are extracted by feature recognizer. Together with the geometric model given are the machining features and tolerance data stored in a STEP AP224 file. Once a physical STEP file of AP224 format is generated, any downstream activities including process planning or inspection planning can be automated regardless of the computing environments in use. Due to the neutrality of the input and output format, the proposed system can provide a general interface between arbitrary CAD and CAPP systems.

4. Preparation of manufacturing information

In order to preserve independency, STEP AP203 is taken as the input file format, which can be generated from most commercial CAD systems. Among the various methods for geometry representation provided by STEP AP203, only Boundary Representation (BRep) 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 the ROSE library [24]. EXPRESS is a data definition language widely adopted in the STEP society [25]. 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 Application Protocol Interface (API) functions. Since the entity structure of STEP AP203 is not identical to that of Parasolid, STEP's BRep model should be processed to match with that Parasolid. Fig. 3 describes the detailed procedure of STEP to Parasolid translation.

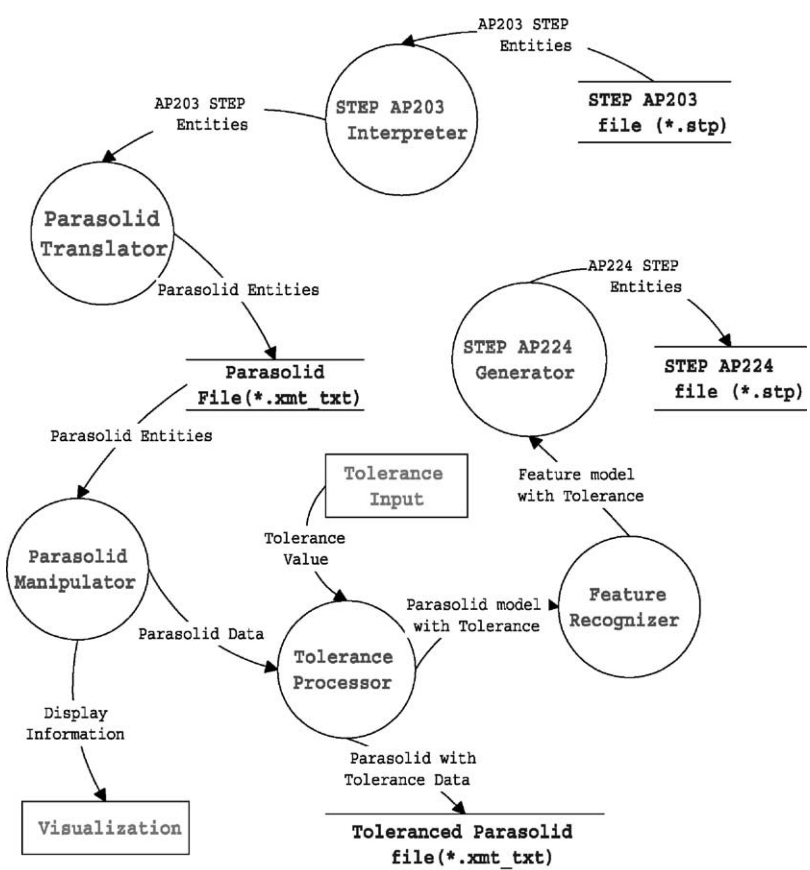

Fig. 2. Data flow diagram for the proposed framework.

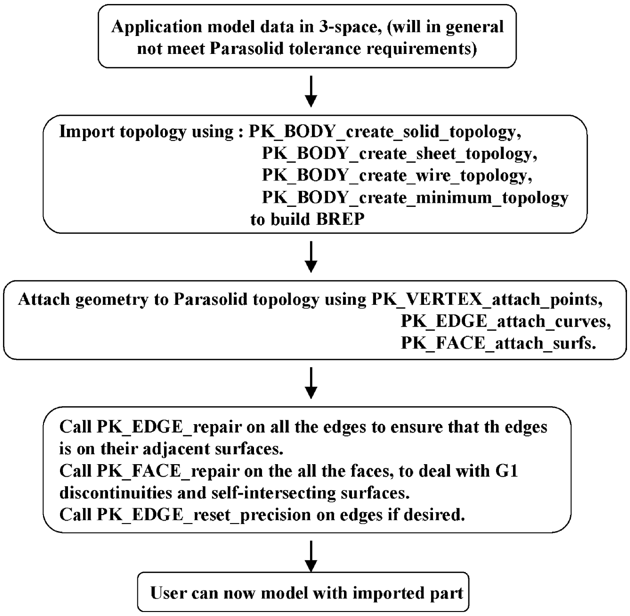

Fig. 3. Procedure to convert STEP AP203 to Parasolid.

Given the geometric model in Parasolid, tolerance information is to be attached to the Parasolid model. Technological information can be classified 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 entity. For instance, the surface roughness is stored simply as a surface attribute of the Parasolid model. In contrast, the cross-referenced tolerance implies a characteristic between two entities. 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, the same as most geometric modelers, it is necessary to implement an appropriate one in order to store tolerance values within the Parasolid model.

Fig. 4 shows an especially designed 2D array data structure to store the datum and the target entity for linear dimensional tolerance. A 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, machining features are to be extracted. As the feature recognition kernel, IF^2 is used, which was developed at the University of Southern California [12] and has been extended at the National Institute of Standards and Technology and Sung Kyun Kwan University to include set-up cost minimization [9,25]. IF^2 generates an optimal feature model that minimizes the overall set-up cost, as shown in Fig. 6.

5. Neutral representation of manufacturing information using STEP

Machining features with relevant technological 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 this than part description language or part classification code based on group technology. The recent development of the 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, the shape tolerance resource model, specifies the resource rules to represent dimensions and tolerances of product geometry [26]. It belongs to *Integrated Generic Resources* on the conceptual layer that provide the generic integrated information model, while 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, the application protocols also contain the conformance requirements and the characteristics of implementation methods. Therefore, it is considered sufficient to consult the specific appli-

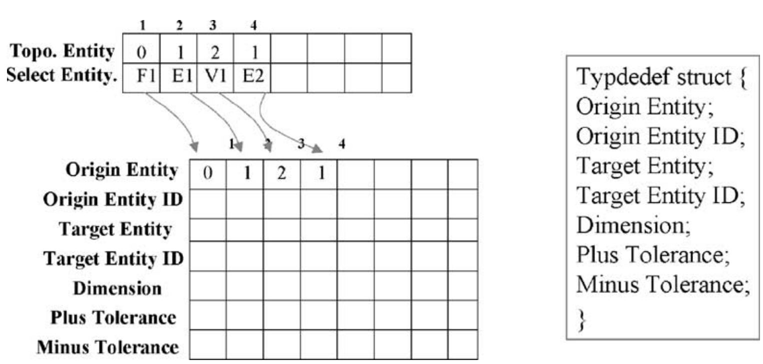

Fig. 4. Data structure for linear dimensional tolerance.

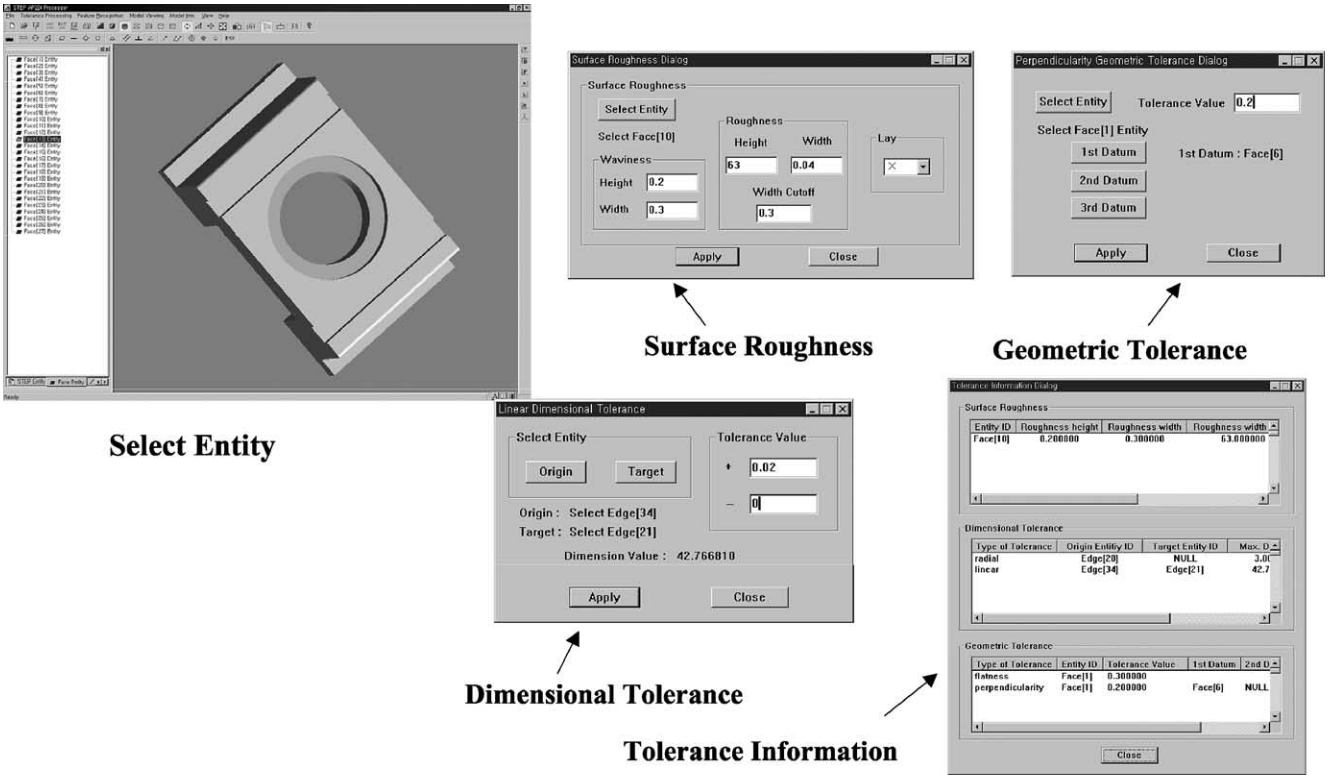

Fig. 5. Snapshot of tolerance processing.

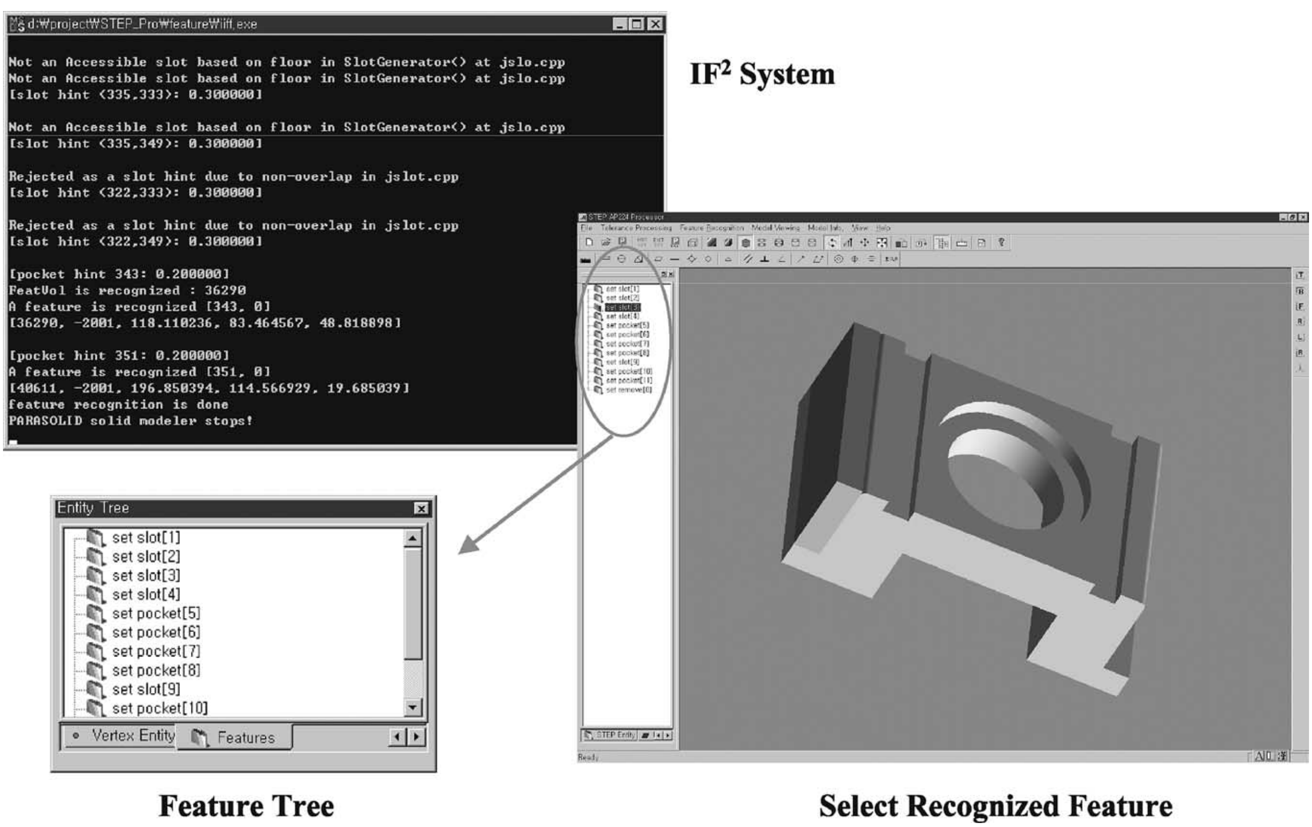

Fig. 6. Machining features recognized by IF^2.

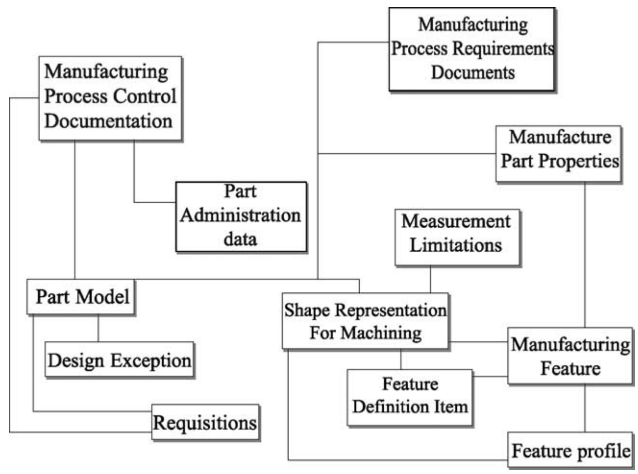

Fig. 7. Basic structure of STEP AP224 schema.

cation protocol when developing an application program. Although STEP AP224 includes all the necessary manufacturing information such as material, specifications and special notes, or other administrative information as shown in Fig. 7, only feature related information is considered in this work.

In a similar but reverse way of interpreting the 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 such as vertex to higher ones such as loop or face, the Parasolid entities are transformed into STEP 224 entities using ROSE functions. Once a physical STEP file of AP224 format is generated, the process planning system can proceed with its process planning task, if provided with a STEP 224 interpreter. Fig. 8 shows the dimensional and geometric tolerance description part of an illustrative STEP 224 file.

6. Implementation

The framework presented in this paper has been completely implemented. The whole system is developed using Visual C++ 6.0, and operated in Windows NT 4.0 on PC. For handling STEP files ST-Developer 7.0 is used, and for geometry processing solid modeling kernel Parasolid v.11.0. The graphical user interface including model visualization is coded mainly using OpenGL.

Fig. 9 depicts the workflow to prepare manufacturing information from a geometric model in order to interlink design and process planning. First a STEP 203 file generated from an arbitrary CAD system, in the case of our work from a 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 check the correctness of the STEP 203 import. Within the

```
#814=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.00999999
977648258),$);
#815=SHAPE_ASPECT($,$,#816,.F.);
#816=PRODUCT_DEFINITION_SHAPE($,$,$);
#817=PROPERTY_DEFINITION_REPRESENTATION(#816,#818);
#818=REPRESENTATION($,(#793),$);
#819=GEOMETRIC_TOLERANCE(#820,'Straightness',#821);
#820=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.00999999
977648258),$);
#821=SHAPE_ASPECT($,$,#822,.F.);
#822=PRODUCT_DEFINITION_SHAPE($,$,$);
#823=PROPERTY_DEFINITION_REPRESENTATION(#822,#824);
#824=REPRESENTATION($,(#527),$);
#825=GEOMETRIC_TOLERANCE(#826,'Cylindricity',#827);
#826=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.01999999
95529652),$);
#827=SHAPE_ASPECT($,$,#828,.F.);
#828=PRODUCT_DEFINITION_SHAPE($,$,$);
#829=PROPERTY_DEFINITION_REPRESENTATION(#828,#830);
#830=REPRESENTATION($,(#803),$);
#831=GEOMETRIC_TOLERANCE(#832,'Circularity',#833);
#832=MEASURE_WITH_UNIT(PARAMETER_VALUE(0.05000000
07450581),$);
#833=SHAPE_ASPECT($,$,#834,.F.);
#834=PRODUCT_DEFINITION_SHAPE($,$,$);
#835=PROPERTY_DEFINITION_REPRESENTATION(#834,#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.

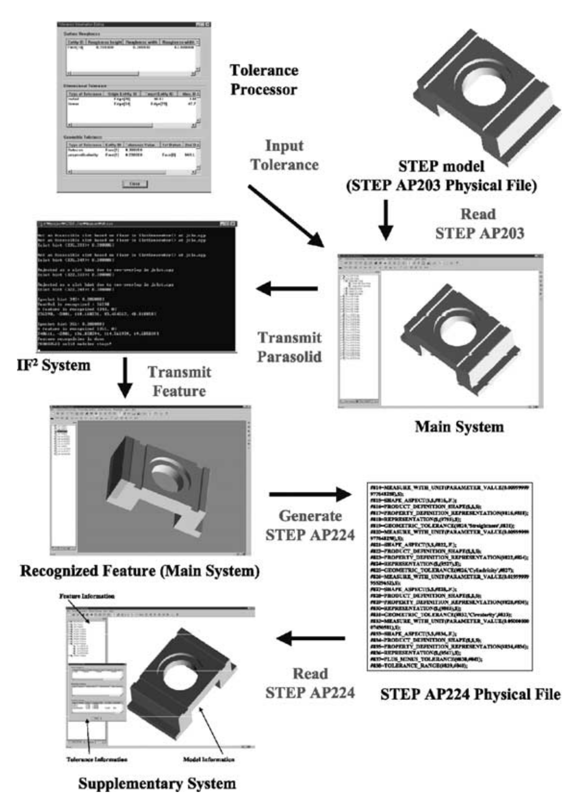

Fig. 9. Workflow in the proposed framework to prepare neutral manufacturing information from a geometry model.

Parasolid model, required tolerance values such as surface roughness, and dimensional or geometric tolerances are assigned. From the geometric model with this technological information, machining features are extracted to give the feature list, by applying IF^2, which recognizes machining features using hint-based reasoning and performs cost optimization with the aid of a search algorithm. This manufacturing information, incorporating technological data as well as machining features, is then translated into a STEP AP224 format. A physical STEP 224 file is CAD-independent and contains the relevant data to manufacture the modeled part including surface roughness, dimensional and geometric tolerances. The supplementary system shown in Fig. 9 is a STEP AP224 interpreter that aims to check the correctness of the generated AP224 file. It can also be used as a pre-processor for a process planning system to import a STEP AP224 file.

7. Discussion and conclusions

Three issues necessary to interlink design and manufacturing engineering have been addressed: recognition of machining features, handling of technical information, and implementation of a neutral interface. Emphasis has been put on the representation of tolerance information by using the neutral product data format STEP. A proper data structure to store various types of tolerance and surface finish data has been proposed and implemented. This can help a CAPP system extract manufacturing information contained in STEP AP224 file more easily, regardless of the CAD systems in use. The authors believe that this framework contributes towards removing the main barrier to computer-automated downstream systems such as process planning or inspection planning, i.e. difficulty in recognizing machining features and tolerance information. Despite the achievements in handling manufacturing information, recognition of relevant features of complex shapes still remains a bottleneck. Without a mature feature recognizing system, a seamless interface between design and manufacturing could not be automated. Therefore, more efforts is needed in the future in extending the capability of feature recognition.

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