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Procedia CIRP 18 (2014) 238 – 243

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International Conference on Manufacture of Lightweight Components – ManuLight2014

A Feature Recognition System using Geometric Reasoning

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Abstract

In this paper, an automatic feature recognition system is proposed using object oriented approach. The part data is extracted from STEP file, which is the standard format for the most known CAD systems. The object oriented structure found quite useful for the generation of part geometric database. Both simple and interacting machining features considered in the study. Different algorithms developed for the extraction of feature faces along with their geometric properties and dimensions. The logical rules then formulated based on geometric reasoning to recognize the selected machining features. A case study presented to validate the proposed methodology.

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Peer-review under responsibility of the International Scientific Committee of the "International Conference on Manufacture of Lightweight Components – ManuLight 2014"

Keywords: Feature Extraction; STEP; Machining Features; Geometric Reasoning; Object Oriented

1. Introduction

The successful link between CAD and CAM will be defined by the efficient methods that can process a decomposed object from the CAD database and reconstruct the entities for downstream applications [1]. Features form the basis for linking CAD with downstream manufacturing applications [2].

Although various types of features have been investigated, [3] the most common type of feature is the form feature, which contains both shape information and parametric information. Examples of form features common in many shape models are round holes, slots, bosses, and pockets.

Many alternative techniques have been devised for creating feature-based models. These can be divided into two main categories based on whether features are extracted from geometry (FfG), or on whether geometry is created from the features (GfF). Ismail et. al., and Tan et.al., [4, 5] proposed a rule based approach for the recognition of hole feature from STEP file. Mangesh et al., [6] developed the system essential for connecting the design stage with process planning stage. Zhang et al., [7] proposed a STEP-based product model data exchange framework for virtual enterprises that demonstrated the related data translation between an IGES file and a STEP AP203 file for data exchange of heterogeneous CAD systems. Nasr [8] proposed a methodology for extracting manufacturing features from CAD system using Initial Graphics Exchange Specification (IGES) format as input and translates the information in the file to manufacturing information. Lu [9] proposed an algorithm for the identification of design and machining features from a data exchanged part model. Alan and Shou [10] developed a volume decomposition approach for building an interface that bridges the gap between a 3D CAD model and an automated process planning system.

In this paper, both simple and complex features were catered for automatic recognition process using C++ programming language. The STEP AP 203 file was used as for the generation of part geometrical database. The significance of using STEP was its capability of

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Peer-review under responsibility of the International Scientific Committee of the "International Conference on Manufacture of Lightweight Components – ManuLight 2014"
doi:10.1016/j.procir.2014.06.138

describing product data throughout the life cycle of product.

2. Proposed Methodology

The methodology adopted in the paper intends to bridge the information gap between Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) using object oriented technique. The 3-D solid model created using CATIA V5. The geometrical and topological information of the model represented in the form of STEP AP 203 file. The STEP data then converted into B-rep data structure using C++ programming language to generate part geometric database.

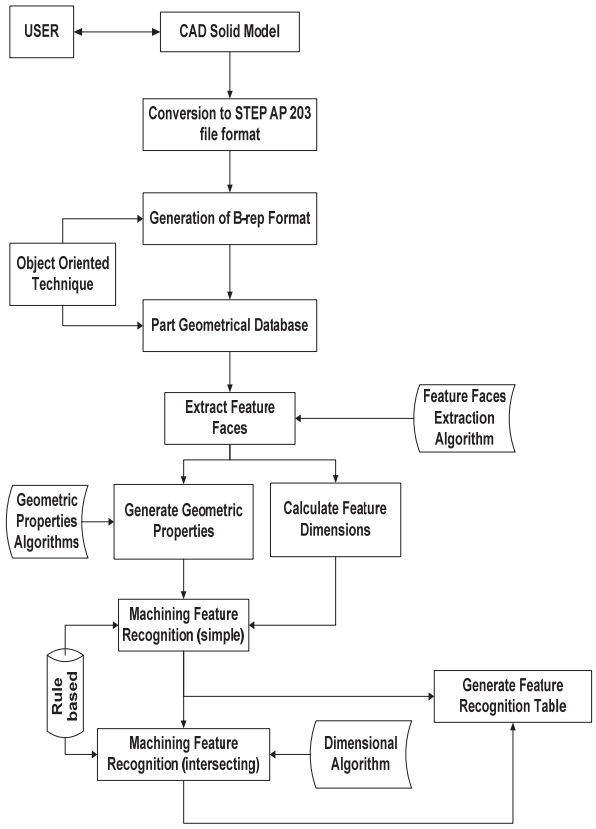

Fig. 1. Proposed Methodology

The developed algorithms (coded using C++) used to recognize and group feature faces and find out their geometric properties along with their dimensions. A separate dimensional algorithm developed to obtain the dimensions of intersecting features. The rule based designed to recognize simple or intersecting features based on geometric reasoning. The methodology presented in the form of flow chart in Fig.1.

2.1 Part Geometric Database

The part geometric database is created by analyzing the geometric information available in the STEP AP 203 file using B-rep Technique. The B-Rep consists of a topological description of the solid model. In this scheme, boundaries of component are represented in the form of shell. The shell actually a union of faces that have specific boundaries called loops. The loops are formed by the lines called edge curves. Each edge or curve is represented by its vertices. By using this information, every face in B-rep solid is defined by its faces, loops, edges and vertices along with the surface type and the normal vector direction. A fraction of part database is shown in Fig. 2.

```
Closed Shell:
---------BEGIN OF FACE 1 -----------
PartBody
Face Outer Bound :
Edge Loop:
Curve: 1
Vertex Point ( 1 ): Cartesian Point: (100,0,0)
Vertex Point ( 2 ): Cartesian Point: (100,100,0)
Line Edge Curve:
Curve: 2
Vertex Point ( 3 ): Cartesian Point: (0,100,0)
Vertex Point ( 2 ): Cartesian Point: (100,100,0)
Line Edge Curve:
Curve: 3
Vertex Point ( 4 ): Cartesian Point: (0,0,0)
Vertex Point ( 3 ): Cartesian Point: (0,100,0)
Line Edge Curve:
Curve: 4
Vertex Point ( 4 ): Cartesian Point: (0,0,0)
Vertex Point ( 1 ): Cartesian Point: (100,0,0)
Line Edge Curve:

PlaneAXIS2 Placement 3D:
Cartesian Point: (0,0,0)
Axis2P3D Direction , Point: (0,-0,1)
Axis2P3D XDirection , Point: (0,1,0)
---------End OF FACE 1 -----------

---------BEGIN OF FACE 2 -----------
PartBody
Face Outer Bound :
Edge Loop:
Curve: 5
Vertex Point ( 2 ): Cartesian Point: (100,100,0)
Vertex Point ( 5 ): Cartesian Point: (100,100,100)
Line Edge Curve:
```

Fig. 2. Part Geometric Database

2.2 Feature Library

In this paper, the machining features are classified as simple and intersecting features. The simple features recognized as single entity whereas the intersecting features are hard to recognize as single entity. The feature interaction defined [3] as "an intersection of feature boundaries with those of other features such that either the shape of the semantics of a feature is altered

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