DEVELOPMENT OF FEATURE RECOGNITION SYSTEM FOR CAD/CAM INTEGRATION

TAN CHEE FAI
PROF. MADYA LT. KOLONEL IR. MOHD HAZANI HJ. SHAFIE
SIVAKUMAR A/L DHAR MALINGAM

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Development of Feature Recognition System for CAD/CAM Integration

C. F. Tan[1], N. Ismail[4], M.H. Shafie[2], D. Sivakumar[3] and A. Abdullah[5]

[1]Department of Design & Innovation
[2]Department of Automotive
[3]Department of Structure & Materials
Faculty of Mechanical Engineering
Kolej Universiti Teknikal Kebangsaan Malaysia
Locked Bag 1200, Ayer Keroh,
75450 Melaka, Malaysia.
cheefai@kutkm.edu.my

[4]Department of Mechanical and Manufacturing Engineering
Faculty of Engineering
Universiti Putra Malaysia
43400 UPM Serdang, Selangor, Malaysia

[5]Office of the Science Advisor,
Prime Minister's Dept.,
Level 2, Perdana Putra Buidling (West Wing),
62502 Putra Jaya, Malaysia.

ABSTRACT

This paper describes a methodology to integrate design and manufacturing using Feature Recognition System (FRS) for achieving concurrent engineering goals. The Feature Recognition System retrieves the geometrical data from Standard for the Exchange of Product Model Data (STEP) file of the CAD/CAM system. A filtering program is developed to extract the geometrical data for feature recognition process. The rule-based technique is applied to recognize features. The test part, namely, ANC-101 designed by Advanced Numerical Control (ANC) program of CAM-I is used to test the FRS capability.

1. INTRODUCTION

The integration of CAD/CAM (Computer-Aided Design/Computer-Aided Manufacturing) is the important aspect for computer-aided engineering. The CAD/CAM integration leads to Computer Integrated Manufacturing (CIM) where this is a primary system in realizing the development of agile manufacturing, concurrent engineering, information technology and production processes [1]. Advancement in design efficiency can be achieved using feature-based design approach. Feature-based product modeling is a concept and technique that has gained a large number of commercial advocates in CAD/CAM development since the late 1980s. A complete product description consists of various feature types, e.g. form features, tolerance features, assembly features, functional features and material features. A complete product definition can be created using the feature-based approach to provide an intuitive design environment for engineers [2].

The work on feature modelling has developed two main approaches, namely, design by features and feature recognition. In the design by features approach, the designer uses the combination of features to create the parts directly. The feature recognition approach is developed to extract the manufacturing information that is recognized from the computer-aided design (CAD) database into computer-aided manufacturing (CAM) database. The features can be used to subtract higher level manufacturing data from lower level or geometrical computer-aided data.

The feature recognition system can be developed to identify the manufacturing data either direct from CAD system or from neutral file format such as STEP (Standard for the Exchange of Product Model Data). Cicirello and Regli [3] presented the approach to use machining features as an index-retrieval mechanism for solid models. One of the technical approaches for this research is to perform machining features extraction to map the solid model to a set of STEP machining features.

Han et al. [4] proposed the work to integrate feature recognition and process planning in the machining domain. The purpose of the work is to achieve the goal of CAD/CAM integration. The system used STEP to interface for portability between CAD and manufacturing systems. Bhandakar and Ragi [5] developed feature extraction system takes STEP file as input and to define the geometry and topology of a part. In addition, the system generates STEP file, as output with form feature information is AP224 format for form feature process planning.

The aim of this paper is to describe the development of feature recognition system to recognize prismatic part for CAD/CAM integration from STEP files. Rule based algorithm is used for extraction of feature and its geometrical data. The rule-based technique is applied to recognize features. The ANC-101 test part is used to test the FRS capability.

2. DEFINITION OF FEATURES

A feature can be defined as characteristic of the part, which carries significance or higher semantic meaning to particular application. These applications could be manufacturing, engineering, design, assembly, etc [5].

In brief, a feature is a parametric shape associated with attributes such as intrinsic geometric parameters (length, width, and depth), position and orientation, geometric tolerances, material properties, and reference to other features. Feature also provides access to related production process and resource models. Thus, features have a higher level semantically than the primitive elements used in ordinary CAD systems [6].

3. STANDARD FOR THE EXCHANGE OF PRODUCT MODEL DATA (STEP)

The purpose of STEP is to build a common standard that ensures the product data can be communicated electronically across different platforms, e.g. CAD, CAM and CAE. The STEP standard differs from IGES by incorporating a formal object-oriented model for data exchange [7].

STEP enables all individuals contributing to the design, manufacturing, marketing and supply of a product and its components to contribute to, to access, and to share information. STEP aims at eliminating the concept of "islands of automation". STEP also attempts to unite manufacturing efforts among corporate partners, distant subsidiaries and suppliers across diverse computer environments. STEP addresses the issues of diversified engineering applications and covers security aspects, which become relevant now that several companies would be sharing the same product information [8].

The STEP neutral file is a text file that contains geometrical data of a component including boundary representation data such as shells, faces, vertices, surface geometric data such as planes, cylinders, cones, curve geometric such as lines, circles, ellipses, b-spline curves. The brief description of some STEP elements is provided as shown in Table 1 [9].

Table 1. The brief description of some STEP elements

STEP Element	Description
CARTESIAN_POINT	Address of a point in cartesian space.
ADVANCE_FACE	The face that associated with a type of surface
CYLINDRICAL_SURFACE	A face of cylinder in which the geometry is defined by the associated surface, boundary and vertices.
CIRCLE	A circle in which the geometry is defined by the associated surface, boundary and vertices.
PLANE	A plane in which the geometry is defined by the associated surface, boundary and vertices.

4. SYSTEM IMPLEMENTATION

Fig. 1 shows the architecture of Feature Recognition System. The feature recognition algorithm is developed based on the geometrical data from STEP file of CAD system. STEP or ISO 10303 is an international standard that provides a computer-interpretable and exchangeable product data model. The purpose of STEP is to build a common standard that ensures the product data can be communicated electronically across different platforms, e.g. CAD, CAM and CAE.

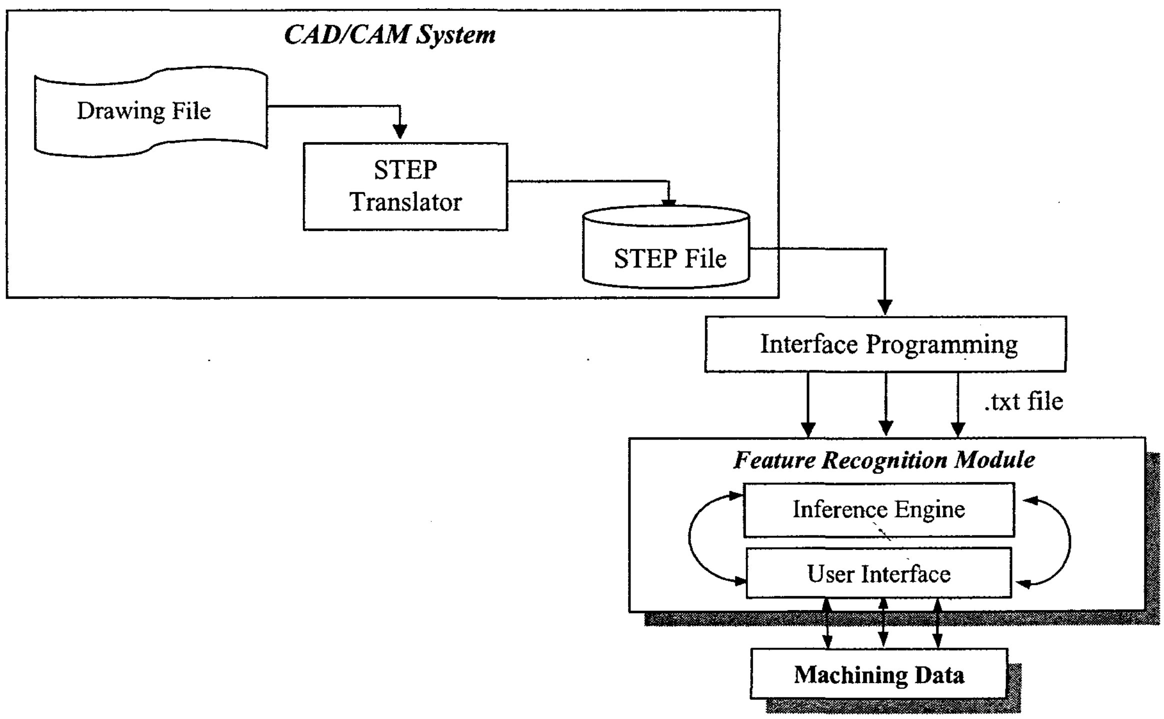

Fig. 1. The architecture of recognizing features for machined Parts

The expert system shell is used in this work to develop the hole recognition system. The set of rules for recognition system is generated by using rule-based technique.

The filtering program is developed using Microsoft Visual C++ 6.0. The filtering program will input and process the STEP file; then the program will retrieve the geometrical data that is needed and output as a text file. The text file will be input for Feature Recognition System. The part of the output file for a model is display in Table 2.0 that consists of circular data (coordinate and radius) and ellipse data (coordinate, major length, minor length and radius).

Table 2.0. Part of the output file for a model

CIRCLE-1
CiX1=4.
CiY1=9.
CiZ1=10.
CiRADIUS1=1.
CIRCLE-2
CiX2=4.
CiY2=7.
CiZ2=10.
CiRADIUS2=1.
ELLIPSE-1
ElX1=15.93261531631
ElY1=5.
ElZ1=10.
ElMAJOR1=1.10337791896249
ElMINOR1=1.
CIRCLE-3
CiX3=5.
CiY3=14.
CiZ3=10.
CiRADIUS3=1.5

5. RESULT AND DISCUSSION

Fig. 2 shows the test part called ANC-101 designed by Advanced Numerical Control (ANC) program of CAM-I [10] to test the feature recognition capability. The ANC-101 part is purely a theoretical part without any real function but provides a range of features that can be used to test feature recognition systems. The partial result of feature recognized is shown in Table 3. The system managed to recognize boss, blind holes and through holes in the test part.

Table 3. Partial recognition result

Through_hole, 6, 40, (11,11,40);	Blind_hole, 2, 10, (64,86,100);
Through_hole, 6, 40, (76,11,40);	Blind_hole, 2, 10, (67,97,100);
Through_hole, 6, 40, (76,134,40);	Blind_hole, 2, 10, (64,109,100);
Through_hole, 6, 40, (11,134,40);	Blind_hole, 2, 10, (55,118,100);
Blind_hole, 8, 25, (23,86,100);	Blind_hole, 2, 10, (44,121,100);
Boss, 8,30, (44,25,89);	Blind_hole, 2, 10, (32,118,100);
Blind_hole, 8, 25, (23,86,100);	Blind_hole, 2, 10, (23,109,100);
Blind_hole, 5, 15,(44,97,100);	Blind_hole, 2, 10, (20,97,100);
Through_hole, 11, 95, (44,97,95);	Blind_hole, 2, 10, (64,86,100);

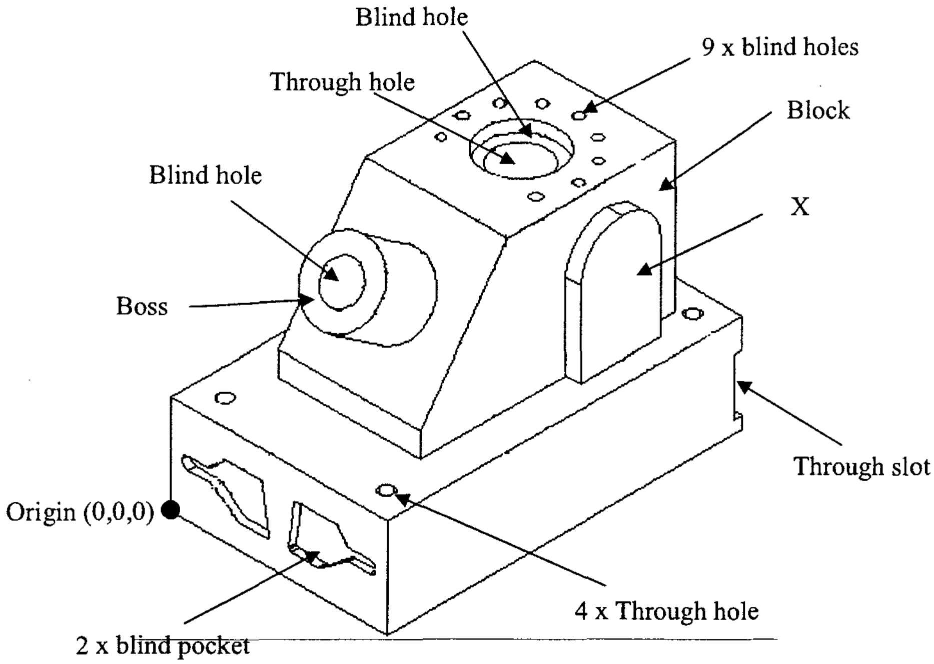

Fig. 2. ANC-101 test part

The case study shows that cylindrical features, namely, through hole, blind hole and boss can be recognized by the developed system. The system shows errors when recognizing the blind pocket, through slot, blind pocket, x and block. The rule of this system can be upgrade to accommodate more features, such as polyhedral features, conical features and interacting features.

6. CONCLUSION

Feature Recognition System has successfully been developed to recognize the cylindrical features. The manufacturing data such as length of hole, radius of the hole and coordinate of the hole can be retrieved to be used in downstream manufacturing activities such as process planning. Further improvement to the system is in progress include recognition of polyhedral features such as pocket, slot and step features.

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