EXTRACTION OF CYLINDRICAL FEATURES FROM NEUTRAL DATA FORMAT FOR CAD/CAM INTEGRATION

N. Ismail[1], M.R. Osman[1], C.F. Tan[2], S.V. Wong[1] and S. Sulaiman[1]
[1]Department of Mechanical and Manufacturing Engineering, Universiti Putra Malaysia, 43400 UPM, Malaysia
[2] Department of Design and Innovation, Faculty of Mechanical Engineering, Kolej Universiti Teknikal Kebangsaan Malaysia, Ayer Keroh, Melaka, Malaysia
Email: napsiah@eng.upm.edu.my

ABSTRACT

This paper discusses the extraction of cylindrical based features from a neutral data format, namely STEP (Standard for the Exchange of Product Model Data) file produced by any Computer Aided Design (CAD) systems. The cylindrical features can be modelled using feature-based design or Constructive Solid Geometry methods. A rule-based algorithm was developed for the extraction of the cylindrical features. The advantages and disadvantages of this method are also highlighted.

INTRODUCTION

Work on feature-based modeling for CAD/CAM integration has led to the development of two main approaches namely design-by-features and feature recognition. In the design-by-features approach, part models are defined directly by adding, subtracting and manipulating features created as instances of predefined feature types. This approach allows non-geometric information to be stored into the feature model but limits the designer to the use of pre-defined features which thus limits the complexity of the product design that can be represented and making the resulting feature-based model context-dependent. Feature recognition involves computationally recognising features from conventional geometric models or from neutral data format such as IGES, STEP. This approach avoids the limitation of design-by-features by attempting to identify features from already designed component description but requires a complex analysis of the geometric model.

Gao et al (2004)[1] discussed conversion algorithm coaxial hole-series machining feature based on the design feature model for gear box components. The planar-type machining features and non-geometrical attribute features are also studied. The converted machining features model can be transferred to process planning system using STEP file.

Another work by Cicirello and Regli (2001)[2] presented the approach to using 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. The approach is using automatic feature recognition, based on the FBMach system from Allied Signal to generate feature data to be used in indexing algorithms.

Han et al. (2001)[3] 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 that was proposed uses STEP as input and output formats. STEP is the interface for portability between CAD and planning systems, feature recognition for manufacturability and setup minimisation, feature dependency construction, and generation of an optimal feature-based machining sequence.

Bhandarkar and Ragi (2000)[4] 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 STEP file can be exchanged between various companies and can serve as input to further downstream activities such as process planning, scheduling and material requirement planning (MRP).

Henderson and Anderson (1984)[5] used logic rules for cylindrical hole's recognition. An example logic rule used is as follows:

IF *a hole entrance exists*
AND *the face adjacent to the entrance is cylindrical, and the face is convex,*
AND the next adjacent face is a plane,
AND *this plane is adjacent only to the cylinder,*

THEN *the entrance face, cylindrical face and plane comprise a cylindrical hole.*

Work by Abdalla et al. (1994)[6] used logic rules for the recognition of a hole and it can be defined through the following rules:

If
(There is a circular top edge) and
(There is a circular bottom edge) and
(There is a cylindrical face) and
(There is a top face) and
(There is a bottom face) and

Then
(The feature is a hole)

Rule-based and Edge Boundary Classification (EBC) technique for recognition of holes and bosses features were presented by Ismail et al. 1997[7]. The main advantage of this method is that it can be applied for holes features having multi curve edge loop (MEL) as a result of interacting with other features that would not be indentified by other techniques found in literature. As an example, the rule for simple and complex (multi curve edge loop) through holes is as follows:

If a cylindrical face exists
AND tp_1 *for edge loop* SEL_1 *is off object*
AND tp_2 *for edge loop* SEL_2 *is off object*
AND tp_m *for mid-point of virtual line* is *off object*

Then *the feature is through hole*

This paper discusses the extraction of cylindrical features for CAD/CAM integration from STEP files. Rule based algorithm is used for the extraction of feature and its geometrical data. The features considered in the case study are holes that commonly found in tool and die industry and machined components.

CYLINDRICAL FEATURES DEFINITION

Noort et al. (2002)[8] defined form features as regions of the part that have some functional meaning. The form features contain class-specific design information that is captured by means of feature elements and feature constraints. Feature elements are shapes and user-defined variables. Features constraints can be, for example, a geometric distance face-face constraint, a dimension constraint, which specifies a dimension to be within a given range, and on-boundary constraint, which specifies feature face to be on the boundary of the part.

The data structure of cylindrical features consists of circular edges and cylindrical face. The *circular edge* is a set of connected edges that may form the closed boundary of a non-self-intersecting face. Non-linear (curve/circular) edges are formed by cylindrical or conical faces.

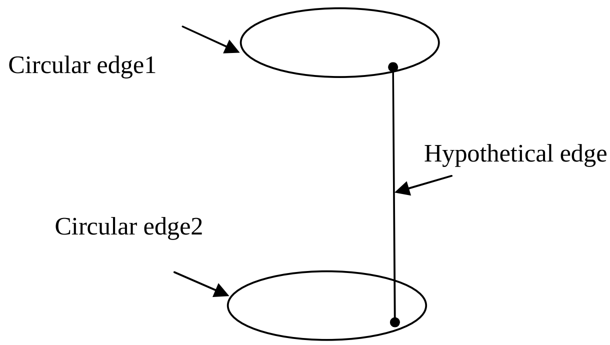


Figure 1: Circular edge of a Cylindrical Face

Hole and bosses features are cylindrical features. Hole can be further divided into simple holes, counter-sunk holes and counter-bored holes. The holes can be created to a specific depth or completely through the body [9].

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 [10].

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 [11].

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 [12].

Table 1: The brief description of some STEP elements

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.
LINE	A line in which the geometry is defined by the associated surface, boundary and vertices.

Figure 2 is part of STEP file for blind hole. Geometrical data of CYLINDRICAL_SURFACE shows that the x-axis and the y-axis of the CYLINDRICAL_SURFACE are the same with first circle and second circle. The radius of CYLINDRICAL_SURFACE, whether the first CIRCLE or second CIRCLE, have the same values. It proves that first circle and second circle.

FEATURE EXTRACTION MODULE

Figure 3 shows system developed in this research. The extraction module recognises cylindrical features from STEP file using Rule-based technique and have the following capabilities:

a) retrieve the associated entity data name of the current solid model being process from the database

b) extract all relevant geometric and topological data and pre-processing the information into a format suitable for use by the rule-based technique

c) process geometric and topological data using interface programming software

d) perform feature recognition by pattern matching and extraction of feature parameters from geometric database

```
#23=CARTESIAN_POINT('',(5.,5.,10.));          } 1st circle with the radius of 2.5 mm
#27=CIRCLE('',#26,2.5);                       } and centre of the circle, X5, Y5, Z10
…………………………………………………….

#68=CARTESIAN_POINT('',(5.,5.,10.,));
#72=PLANE('',#71);
#73=ADVANCED_FACE('',(#33,#67),#72,.T.);

…………………………………………………….
#182=CARTESIAN_POINT('',(5.,5.,4.,));         } 2nd circle with the radius of 2.5 mm and
#186=CIRCLE('',#185,2.5);                     } centre of the circle, X5, Y5, Z4
…………………………………………………….

#193=CARTESIAN_POINT('',(5.,5.,4.,));
#197=PLANE('',#196);
#198=ADVANCED_FACE('',(#192),#197,.F.);
…………………………………………………….

#205=CARTESIAN_POINT('',(5.,5.,10.,));        } cylindrical_surface with the radius
#209=CYLINDRICAL_SURFACE('',#208,2.5);        } of 2 mm
```

Figure 2: Part of STEP file for blind hole

The rule for through hole at XY plane is written as follows:

If

CiX and *CiY* for a circle same with *CiX* and *CiY* for other circle and also same with *CyX* and *CyY* for one of the cylindrical
And
The *CIRADIUS* for same circle same with *CIRADIUS* for other same circle and also same with *CYRADIUS* for the same cylindrical
And
CIPLANE for one of the circle is *FALSE*
And
CIPLANE for other circle also must be *FALSE*

Then

The result is ***Through Hole***

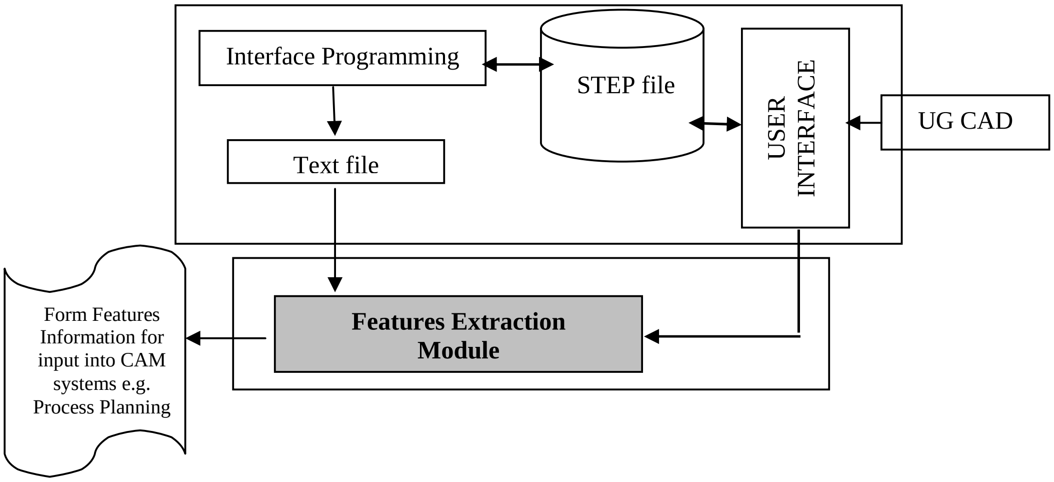
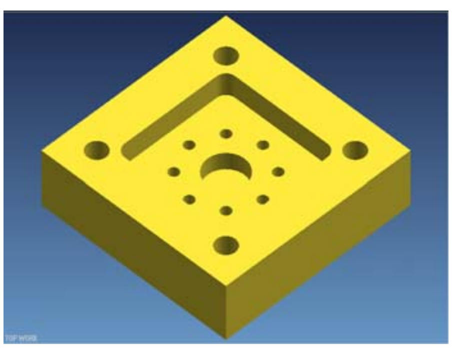


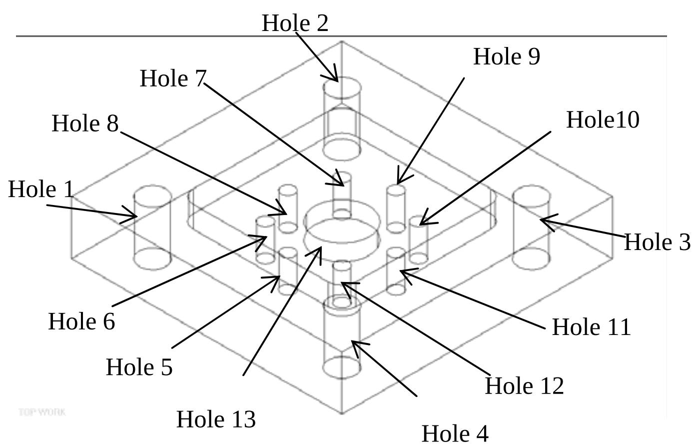
Figure 3: Framework of recognizing features

RESULTS AND DISCUSSION

The case study's test part is as shown in Figure 4. The part consists of four through holes, diameter 40mm, eight through holes, diameter 24mm, and one blind hole. The partial result of features recognised is shown in Table 2. All features labelled in Figure 4b are recognised. The cylindrical features are in xy, xz and yz plane with correct length and radius of each hole.



a) Solid Model



b) Wireframe Model

Figure 4: Test part

Table 2: Partial Recognition Result

Feature no. 1:	Through Hole
The Length (mm):	10
The Radius (mm):	20
Plane:	xy
The Cartesian_Point of Feature no. 1:	
x =15	
y =15	
z =20	
Feature no. 2:	Through Hole
The Length (mm):	10
The Radius (mm):	20
Plane:	xy
The Cartesian_Point of Feature no. 2:	
x =15	
y =85	
z =20	
Feature no. 3:	Through Hole
The Length (mm):	10
The Radius (mm):	20
Plane:	xy
The Cartesian_Point of Feature no. 3:	
x =85	
y =85	
z =20	

Feature no. 4:	Through Hole
The Length (mm):	10
The Radius (mm):	20
Plane:	xy
The Cartesian_Point of Feature no. 4:	
x =85	
y =15	
z =20	
Feature no. 5:	Through Hole
The Length (mm):	5
The Radius (mm):	12
Plane:	xy
The Cartesian_Point of Feature no. 5:	
x =5	
y =30	
z =12	
Feature no. 6:	Through Hole
The Length (mm):	5
The Radius (mm):	12
Plane:	xy
The Cartesian_Point of Feature no. 6:	
x =35.8578	
y =35.8578	
z =12	

The case study shows that the cylindrical features, namely through holes in xy, xz, and yz plane can be recognised easily and the length and radius for each hole can also be retrieved. The system recognised 4 through holes diameter 40mm with 10mm length, 8 through holes diameter 24mm with 5mm length and 1 blind hole diameter 30mm with 3mm length. The database of this system can be upgrade to accommodate other features such as protrusion type features. The extraction of features data is tackled by developing a rule-based technique for each feature.

CONCLUSION

The main advantage of this method is any CAD system can be used to modelled the part. The shortcoming of using neutral data format method is that, a hole that is drilled at an angle to the entrance face (elliptical edges) would not be recognised as the rule for a hole specifies a circular edge at the entrance face and bottom. The work reported that only negative (depression) features are recognised. Extended work is currently being undertaken to include recognition of prismatic part that have slot, pocket, step features.

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