A New Methodology for Recognition of Milling Features from STEP File

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Abstract

In recent years, various researchers have come up with different ways and means to integrate CAD and CAM. Automatic feature recognition (AFR) from a CAD solid model for down stream applications like process planning and NC program, greatly contribute to the level of integration. When generating G&M codes from CAD DXF file, it leads to the loss of geometric information and the user is to edit and fills the details of the lost data. STEP is an international standard for geometric and non geometric data transfer between CAD, CAE and CAM and it replaces the IGES and DXF. For that reason this paper proposes an automatic feature recognition methodology to develop a feature recognition system using STEP file.

The proposed methodology is developed for 3D prismatic parts that are modeled any CAD software having STEP output file format. A JAVA program is used to implement the geometric data extraction algorithm, which has been developed for extracting the geometric information from the STEP file. A feature recognition algorithm is used to recognize the different features of the part such as slot, pocket etc based on geometric reasoning approach by taking B-rep data base as input. The authors present an example to demonstrate the application of the proposed methodology.

Keywords

STEP, Geometric Data Extraction, Concavity, Feature Recognition.

Introduction

Feature recognition is the ability to automatically or interactively identify and group topological entities, such as faces in boundary representation (B-rep) solid models into functionally significant features such as holes, slots, pockets, fillets, ribs etc. In order to integrate CAD and CAM an interface for neutral (mostly geometric) data exchange is needed between CAD and CAM. Many data exchange formats have been developed in the past, their primary purpose is to exchange geometric data. Most widely used data exchange formats like IGES, DXF and PDES will transfer only the geometric information and they may not support the data throughout the life cycle. Most promising solution to these problems is development of a new standard that will support the data throughout product life cycle. STEP is such a standard which gives the explicit and complete representation of the product data throughout the life cycle, independent of any particular system.

The integration of Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) has received significant attention in the recent years according to the development of faster computing power tools. However, the actual integration between CAD and CAM, for the downstream applications such as process planning, can be achieved only when the manufacturing information can be obtained directly from 3D solid model and hence automate the process planning functions (Chang, H-C., Lu, W. F., and Liu, F. X.,2002). This automatic extraction of manufacturing information from CAD systems play an important role to facilitate the concurrent engineering concept in order to achieve the link between the design and manufacturing activities. This successful link can be considered as fundamental step to automate the product development from the design stage all the way to manufacturing and shipping stages. Hence, the total life cycle of the product can be reduces dramatically (Bhandarkar, M. P., Downie, B., Hardwick, M., and Nagi, R., 2000).

The proposed methodology is developed for 3D prismatic parts that are modeled any CAD system having STEP output file extension. The system takes a part 21 file format as input and translates the information in the file into manufacturing information using JAVA program. The boundary (B-rep) geometrical information of the part design is analyzed by a feature recognition program that is created specifically to extract the features from the geometrical information based on a geometric reasoning approach by using object oriented design software which is included in JAVA language.

Related Work

There have been considerable researches on the feature recognition systems. Automated feature recognition has been an active research area in solid modeling for many years and is considered to be a critical component for integration of CAD and computer-aided manufacturing. Mike Pratt (Mike Pratt and William C.Regli, 2000) gave an overview on the three major algorithmic approaches for feature recognition and mentioned several drawbacks of them also proposed several open research areas. JungHyun Han (JungHyun Han, 1996) made a survey on feature recognition and merits of several algorithms of feature recognition: graph pattern matching, cell based decomposition, convex hull decomposition and Hint based reasoning. In graph-based approach, boundary representation of the part is converted into a graph which involves a set of nodes and their attributes. Joshi and Chang (Joshi, S. and Chang, T.C., 1988) developed a graph named the Attribute Adjacency Graph (AAG) to represent features in which each face of the part is represented as a node, and each edge or face adjacency is represented as an arc. Sashikumar Venkataraman (Sashikumar Venkataraman., 2001) presented a graph based frame work for feature recognition. The feature recognition step involved finding similar sub graphs in the part graph. The novelty of this framework lied in the usage of a rich set of attributes to recognize a wide range of features efficiently. W.F. Lu (W.F. Lu., 2003) gave an approach to recognize features from a data exchanged part model. A litany of algorithms for the identification of design and machining features are proposed. A.F.M. Anwarul Haque (A.F.M.

Anwarul Haque.,2001) explained manufacturing feature recognition of a rotational component using DXF file. In this work geometric information of a rotational part is translated into manufacturing information through a Data Interchange Format (DXF). Emad S. Abouel Nasr (Emad S. Abouel Nasr., 2006) discussed a methodology for extracting manufacturing features from CAD system. The system takes a neutral file in Initial Graphics Exchange Specification (IGES) format as input and translates the information in the file to manufacturing information. The boundary (B-rep) geometrical information of the part design is then analyzed by a feature recognition program that is created specifically to extract the features from the geometrical information based on a geometric reasoning approach.

Structure of STEP File

The STEP file structure is language based and is described by an unambiguous context free grammar to facilitate parsing by software. The grammar is expressed in Wirth Syntax notation (S. Ma, Y. Mareachal and J.L. Coulomb, 2001). The information contained with the file is in free format and thus not column dependent. The STEP file is begun by the keyword ISO-10303-21and is terminated by keyword END-ISO- 10303-21, and in similar fashion sections are delimited by keywords. The contents of the sections are limited to the entity instances, i.e., the description of the object of interest. Briefly the data format is as follows. Each entity instance has an identifier of the form #N. where N is a unique integer. Each individual entity has a name. The data for an entity instance follows the type name and is enclosed in parentheses. A datum can be either "primitive" like integer, real or string, etc., or it may be a reference to another entity instance within the file. Such a reference has the form #N where N is the entity number of the reference instance. Entities may be referenced before they are defined within the file.

A STEP file consists of three types of data (David Loffredo, 2000) namely: Descriptive, Geometrical and Topological, and is divided into two major sections: Header section and Data section. The information about the STEP translator version and the type of CAD software used to build the model is included in the Header section. The Data section consists of geometrical entity definitions and topological elements like faces, loops and bounds. Reference between elements is provided by instance ids or pointers (which may be nested). These instance ids or pointers by themselves have no semantic meaning except to identify an instance in a STEP file. The sequence of instances in a STEP file is not specified by the standard. The STEP standard consists of many parts. The entire model is represented by a variety of geometrical entities and topological elements arranged in, the data section. A brief description of some of the important STEP data elements are given below.

Closed-Shell: A collection of one or more faces which bounds a region in three dimensional space and divides the space into two regions, one finite and the other infinite.

Face-Surface: A type of face in which the geometry is defined by the associated surface, boundary and vertices

Face-Bound*:* A loop used for bounding a face.

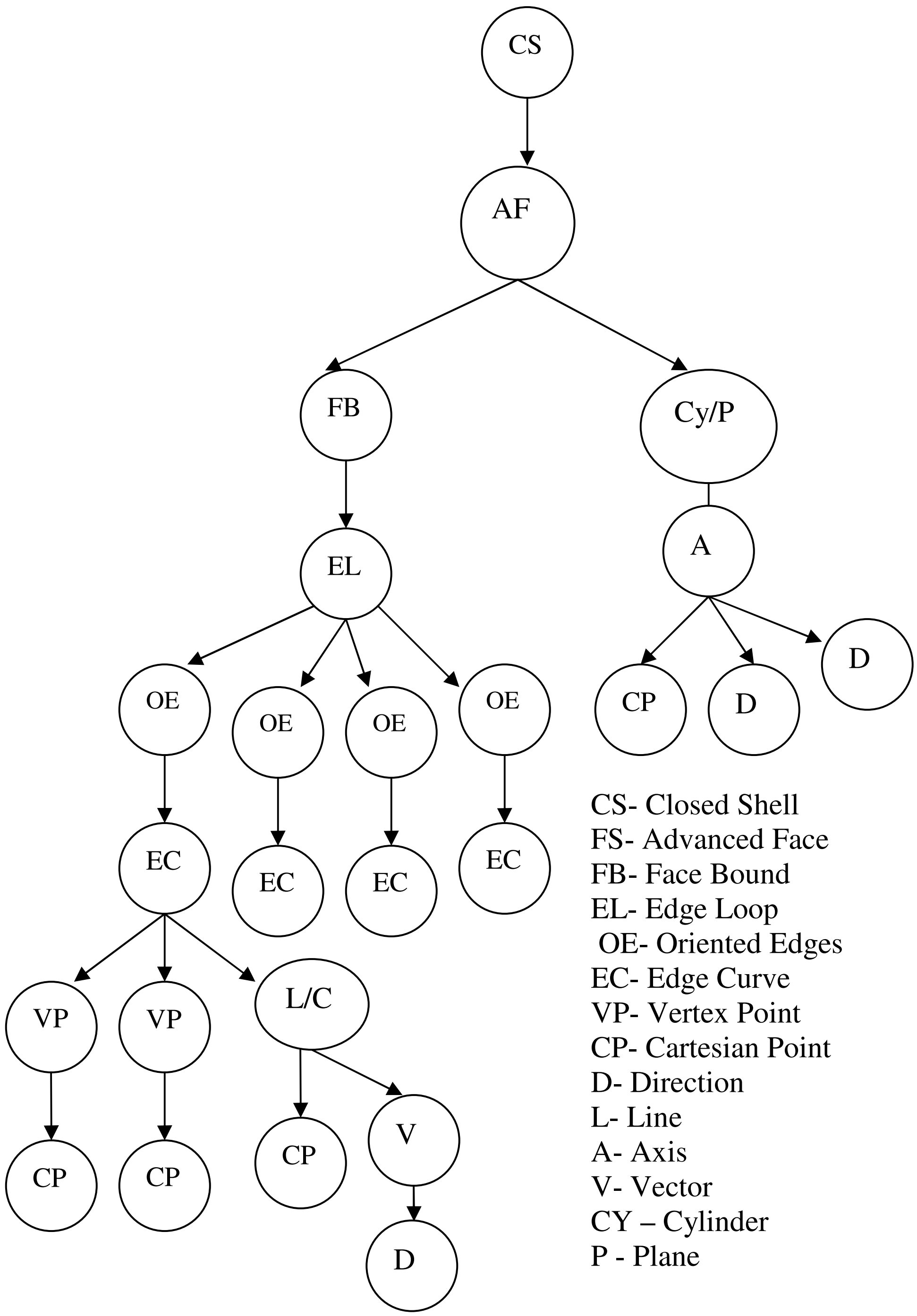
Edge-Loop*:* A path in which the start and end vertices are the same.

Oriented-Edge: An edge constructed from another (original) edge and containing the direction (orientation) information. The ORIENTED-EDGE will be equivalent to the original edge if the orientation information is not included.

Edge-Curve: A type of edge which has its geometry fully defined.

Vertex-Point: A point defining the geometry of a vertex.

Cartesian-Point: Address of a point in Cartesian space.



CS- Closed Shell
FS- Advanced Face
FB- Face Bound
EL- Edge Loop
OE- Oriented Edges
EC- Edge Curve
VP- Vertex Point
CP- Cartesian Point
D- Direction
L- Line
A- Axis
V- Vector
CY – Cylinder
P - Plane

Figure 1: Structure of STEP file

Geometric Data Extraction Algorithm

After a thorough study of the STEP file an attempt has been made to develop an algorithm to extract the geometric information (B-Rep database) from the STEP file. The algorithm will determine the type and orientation of each face using the B-Rep data base including all topological and geometrical information for the object and the B-rep data base is modified accordingly. Algorithm for extracting the geometric information from STEP file is shown in Figure 2.

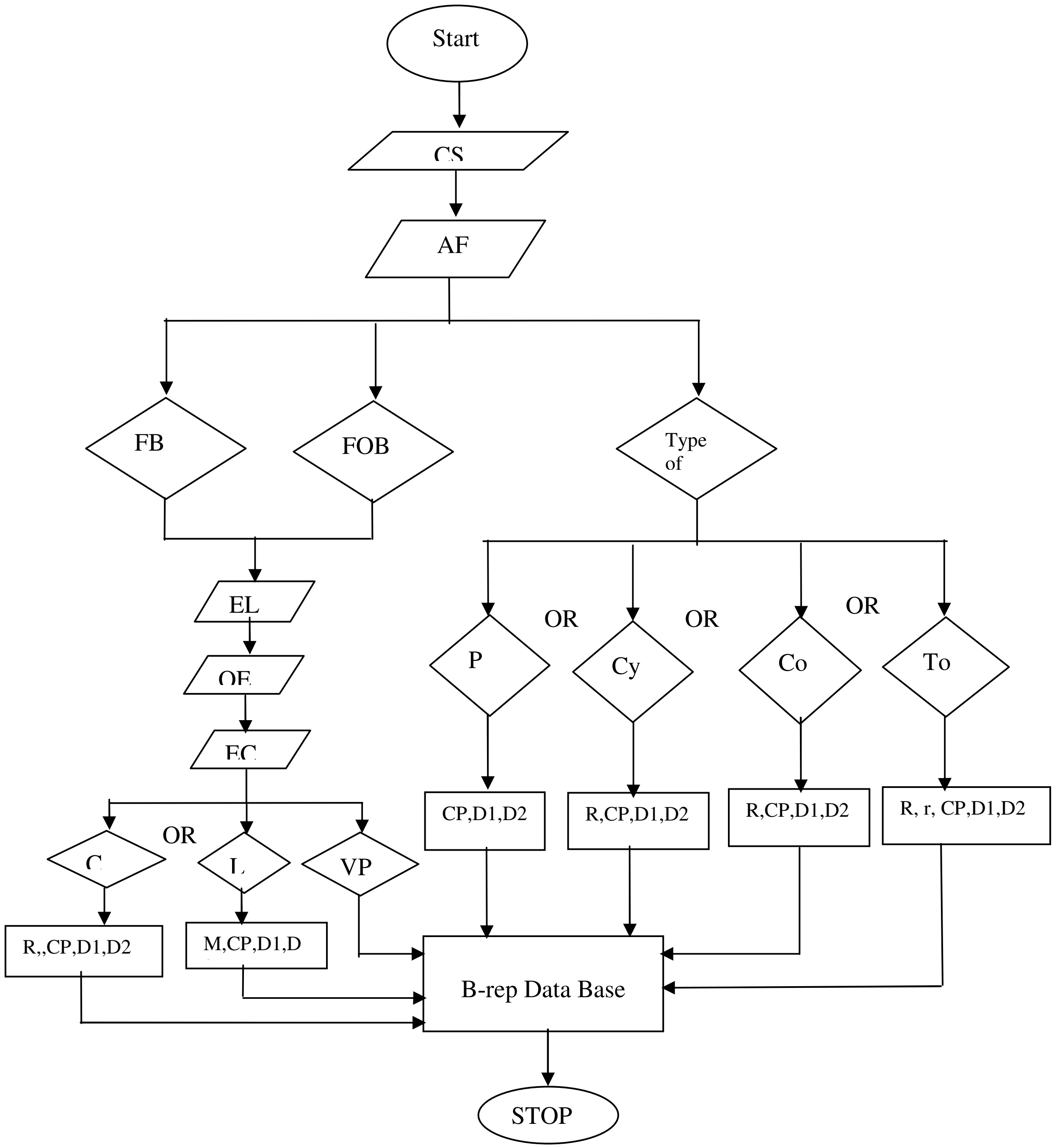


Figure 2: Algorithm for extracting the geometric information

A generalized JAVA program has been written to extract the geometric information from STEP file of any part which has been modeled on any platform.

Methodology Used to Identify the Features

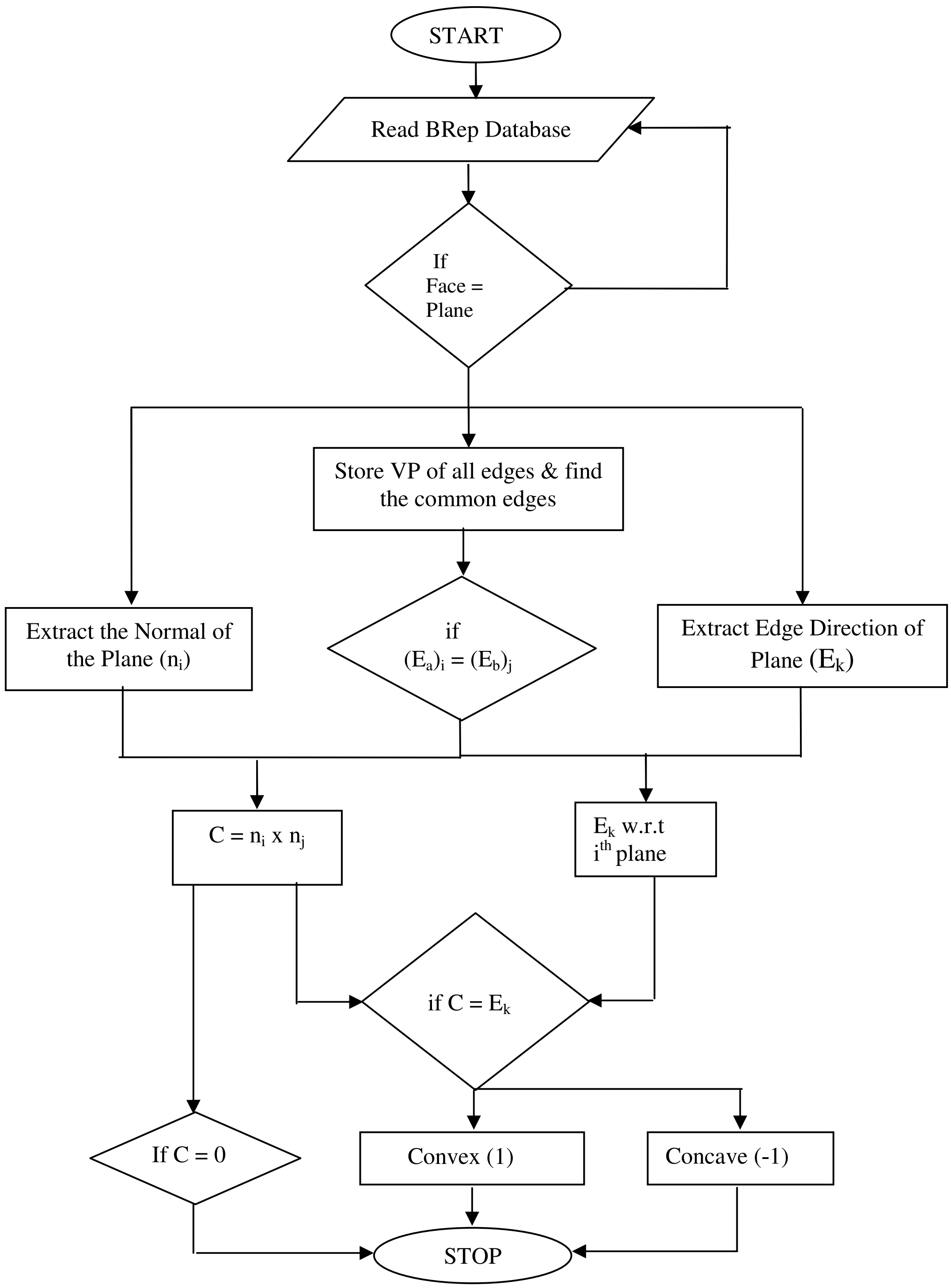


Figure 3: Algorithm for find out the convex/concave faces

The methodology used in this paper for the feature extraction is primarily based on the principle of *concave decomposition of edges* (Emad S. Abouel Nasr and Ali K. Kamrani, 2006). The root faces of a form-feature are extracted by decomposing the object at concave edges is the basic principle of the proposed heuristics. The heuristic similarly implies that a face whose all neighboring faces are at convex angle does not form part of a feature

The boundary faces of the feature are also recognized as well as root faces. Once the boundary and root faces associated with each feature on the object are identified, it is not too difficult to determine titles and characteristics of the features. The input to the system is a B-Rep data base. Algorithm for find out the convex/concave faces from the B-rep data base obtained from geometric data extraction algorithm is shown in figure 3. JAVA program has been written to implement this algorithm.

This algorithm will determine the relational topology which is basically converted with adjacency relationships between faces and edges. The attributes used for face and edge are shown in Table 1 and Table 2 respectively. These attributes and relationship between all faces and edges of the given object are converted in to relation matrix. This is well explained in the next section with example.

Type of Face	**Attributes**
Plane (between same faces)	0
Convex	1
Concave	-1

Table 1: Face attributes

Table 2: Edge attributes

Methodology Via Example

The example part with different feature which has been modeled in CATIA with the face numbers are shown in figure.4 and a part of STEP file is also shown in Table 3.

Type of Edge	**Attributes**
Non-adjacent	2
Convex	1
Concave	-1

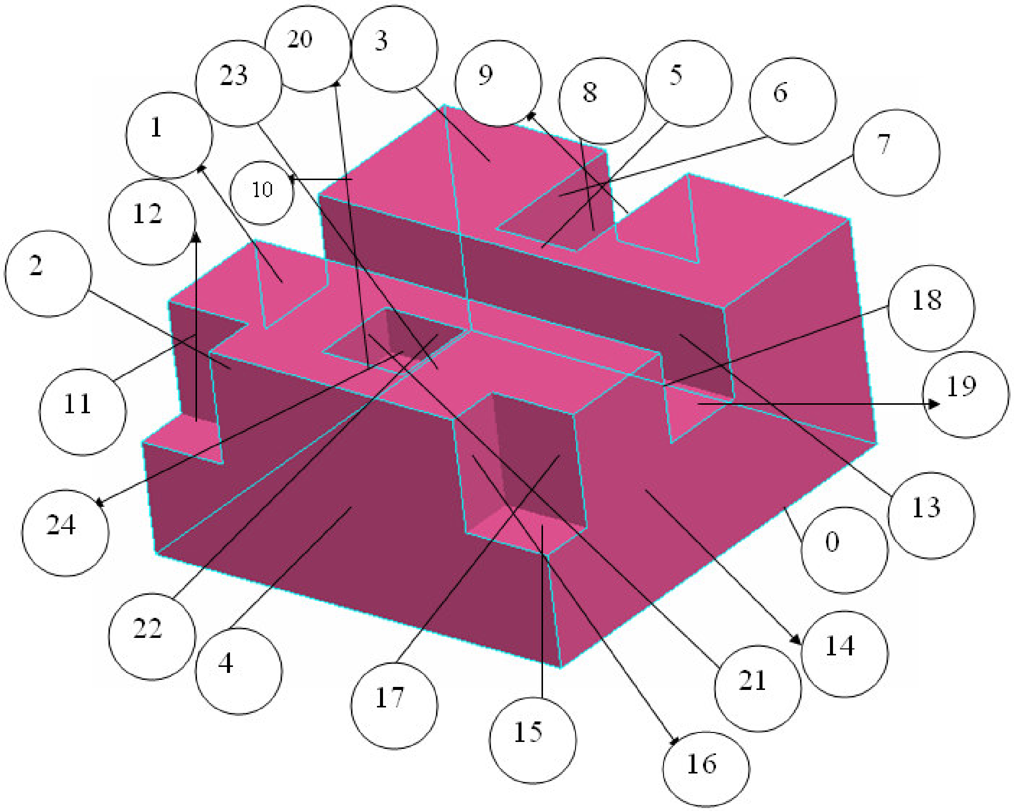


Figure 4: Example part with different features

```
ISO-10303-21;
HEADER;
FILE_DESCRIPTION(('CATIA V5 STEP Exchange'),'2;1');
FILE_NAME('C:\\Documents and Settings\\devs\\Desktop\\milling2.stp','2008-04-
11T20:00:34+00:00',('none'),('none'),'CATIA Version 5 Release 11 (IN-9)','CATIA V5 STEP AP203',' none');
FILE_SCHEMA(('CONFIG_CONTROL_DESIGN'));
ENDSEC;
DATA;
#5=PRODUCT('Part2','','',(#2)) ;
#1=APPLICATION_CONTEXT('configuration controlled 3D design of mechanical parts and assemblies') ;
#14=PRODUCT_DEFINITION(' ',' ',#6,#3) ;
#16=SECURITY_CLASSIFICATION(' ',' ',#15) ;
#15=SECURITY_CLASSIFICATION_LEVEL('unclassified') ;
#47=CARTESIAN_POINT(' ',(0.,0.,0.)) ;
.
.
#44=(NAMED_UNIT(*)SI_UNIT($,.STERADIAN.)SOLID_ANGLE_UNIT()) ;
#46=(GEOMETRIC_REPRESENTATION_CONTEXT(3)GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT
((#45))GLOBAL_UNIT_ASSIGNED_CONTEXT((#41,#42,#44))REPRESENTATION_CONTEXT(' ',' ')) ;
ENDSEC;
END-ISO-10303-21;
```

Table 3: STEP out put file for the above example

The algorithm given in the section 4 will determine the type and orientation of each face using the B-Rep data base including all topological and geometrical information for the object and the B-rep data base is modified accordingly. B-rep Data base extracted for the above part from its STEP file (for the first three faces only) is shown in Table 4.

```
0||1||0||200||0||0||0||0||LINE||200||0||-
1||0||null||null||null||PLANE||null||0||0||0||0||0||1||1||0||0||
0||2||0||0||0||200||0||0||LINE||200||1||0||0||null||null||null||PLANE||n
ull||0||0||0||0||0||1||1||0||0||
0||3||200||0||0||200||200||0||LINE||200||0||1||0||null||null||null||PLAN
E||null||0||0||0||0||0||1||1||0||0||
0||4||200||200||0||0||200||0||LINE||200||-
1||0||0||null||null||null||PLANE||null||0||0||0||0||0||1||1||0||0||
1||1||0||160||100||0||160||50||LINE||50||0||0||-
1||null||null||null||PLANE||null||0||200||0||-1||0||0||0||-1||0||
1||2||0||160||100||0||40||100||LINE||1200||0||-
1||0||null||null||null||PLANE||null||0||200||0||-1||0||0||0||-1||0||
1||3||0||40||100||0||40||50||LINE||50||0||0||-
1||null||null||null||PLANE||null||0||200||0||-1||0||0||0||-1||0||
1||4||0||0||50||0||40||50||LINE||40||0||1||0||null||null||null||PLANE||n
ull||0||200||0||-1||0||0||0||-1||0||
1||5||0||0||0||0||0||50||LINE||50||0||0||1||null||null||null||PLANE||nul
l||0||200||0||-1||0||0||0||-1||0||
1||6||0||200||0||0||0||0||LINE||200||0||-
1||0||null||null||null||PLANE||null||0||200||0||-1||0||0||0||-1||0||
1||7||0||200||0||0||200||50||LINE||50||0||0||1||null||null||null||PLANE|
|null||0||200||0||-1||0||0||0||-1||0||
1||8||0||200||50||0||160||50||LINE||40||0||-
1||0||null||null||null||PLANE||null||0||200||0||-1||0||0||0||-1||0||
2||1||0||160||100||23||160||100||LINE||23||1||0||0||null||null||null||PL
ANE||null||0||160||0||0||1||0||-1||0||0||
2||2||0||160||100||0||160||50||LINE||50||0||0||-
1||null||null||null||PLANE||null||0||160||0||0||1||0||-1||0||0||
2||3||23||160||50||0||160||50||LINE||23||-
1||0||0||null||null||null||PLANE||null||0||160||0||0||1||0||-1||0||0||
2||4||23||160||100||23||160||50||LINE||50||0||0||-
1||null||null||null||PLANE||null||0||160||0||0||1||0||-1||0||0||
3||1||130||1200||100||130||80||100||LINE||40||0||-
1||0||null||null||null||PLANE||null||0||0||100||0||0||1||1||0||0||
3||2||200||80||100||130||80||100||LINE||70||-
1||0||0||null||null||null||PLANE||null||0||0||100||0||0||1||1||0||0||
3||3||200||0||100||200||80||100||LINE||80||0||1||0||null||null||null||PL
ANE||null||0||0||100||0||0||1||1||0||0||
3||4||1200||0||100||200||0||100||LINE||80||1||0||0||null||null||null||PL
ANE||null||0||0||100||0||0||1||1||0||0||
3||5||1200||0||100||1200||200||100||LINE||200||0||1||0||null||null||null
||PLANE||null||0||0||100||0||0||1||1||0||0||
3||6||200||200||100||1200||200||100||LINE||80||-
1||0||0||null||null||null||PLANE||null||0||0||100||0||0||1||1||0||0||
3||7||200||1200||100||200||200||100||LINE||80||0||1||0||null||null||null
||PLANE||null||0||0||100||0||0||1||1||0||0||
```


Table 4: B-rep Database for the example part

Each row of B-rep data base contains the face number, edge number, vertex points of the corresponding edges, type of edge and its details and type of face (Plane/Cylindrical surface/Toroidal surface/Conical surface) and its details respectively. The algorithm given in the section 5 will determine relational topology which is basically concerned with the adjacency relationships between the faces and edges are then determined. The output obtained from the algorithm given in section.5 is shown in figure 5. By using the attributes obtained from the convex/concavity algorithm for face and edges and the relations between all faces and edges of the object are converted into a relation matrix. The relation matrix derived for the example part is given in Table 5.

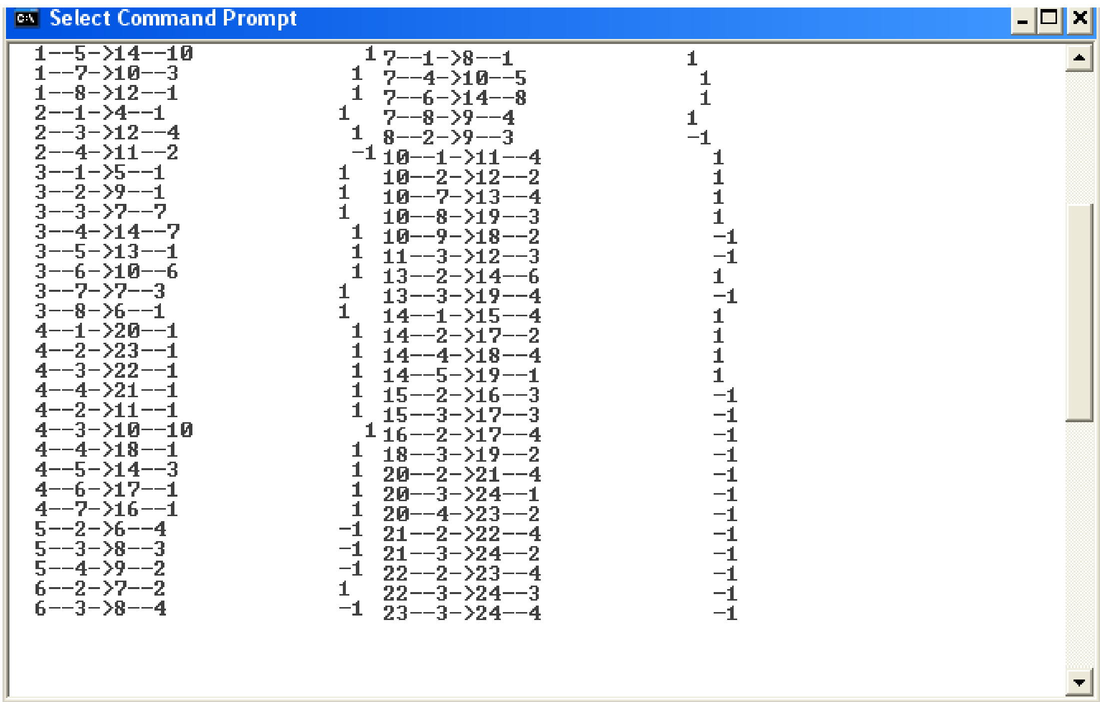


Figure 5: Out put from the algorithm given in section.5

Feature Extraction from the Relation Matrix

The feature extraction session is performed by scanning this matrix starting from the first row. The first negative entry starts the feature extraction cycle. A negative entry in a off-diagonal cell (i,j), the faces Fi and Fj are defined as the root faces of the current (candidate) feature. Any of column for face having an off-diagonal (-1) entry in the i_{th} row is also flagged as the root faces, while columns for face having an off-diagonal (1) entry in the i_{th} row is also flagged as the boundary faces. The cycle is finished when no new root face is found.

Different features identified from relation matrix are shown in Table 6. The three faces 2, 11 &12 having the concavity relation between each other as shown in table 6.1 and will together form a Blind Step. Similarly 15, 16 & 17 will also form a Blind Step as shown in table 6.2. Face 13 having the concavity relation with face 19 and non-adjacency relation with 18 and the face 18 is having the concavity relation with face 19 as shown in Table 6.3, will together form a Through Slot. Similarly the other features Blind Slot (5, 6, 8 & 9) and Rectangular Pocket (20, 21, 22, 23 & 24) are shown in table 6.4 and 6.5.

	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24
0	0	2	2	2	1	2	2	1	2	2	1	2	2	2	1	2	2	2	2	2	2	2	2	2	2
1	2	0	1	2	1	2	2	2	2	2	1	1	2	2	1	2	1	1	1	2	1	1	1	1	2
2	2	1	0	2	1	2	2	2	2	2	2	-1	-1	2	2	2	2	2	2	2	2	2	2	2	2
3	2	2	2	0	2	1	1	1	2	1	1	2	2	1	1	2	2	2	2	2	2	2	2	2	2
4	1	1	1	2	0	2	2	2	2	2	1	2	1	2	1	1	1	2	2	2	2	2	2	2	2
5	2	2	2	1	2	0	-1	2	-1	-1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
6	2	2	2	1	2	-1	0	2	-1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
7	1	2	2	1	2	2	2	0	1	1	1	2	2	2	1	2	2	2	2	2	2	2	2	2	2
8	2	2	2	2	2	-1	-1	1	0	-1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
9	2	2	2	1	2	-1	2	1	-1	0	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
10	1	1	2	1	1	2	2	1	2	2	0	1	1	1	2	2	2	2	1	1	2	2	2	2	2
11	2	1	-1	2	2	2	2	2	2	2	1	0	-1	2	2	2	2	2	2	2	2	2	2	2	2

12	2	2	-1	2	1	2	2	2	2	2	1	-1	0	2	2	2	2	2	2	2	2	2	2	2	2
13	2	2	2	1	2	2	2	2	2	2	1	2	2	0	1	2	2	2	2	-1	2	2	2	2	2
14	1	1	2	1	1	2	2	1	2	2	2	2	2	1	0	1	2	1	1	1	2	2	2	2	2
15	2	2	2	2	1	2	2	2	2	2	2	2	2	2	1	0	-1	-1	2	2	2	2	2	2	2
16	2	1	2	2	1	2	2	2	2	2	2	2	2	2	2	-1	0	-1	2	2	2	2	2	2	2
17	2	1	2	2	2	2	2	2	2	2	2	2	2	2	1	-1	-1	0	2	2	2	2	2	2	2
18	2	1	2	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	0	-1	2	2	2	2	2
19	2	2	2	2	2	2	2	2	2	2	1	2	2	-1	1	2	2	2	-1	0	2	2	2	2	2
20	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	0	-1	2	-1	-1
21	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	-1	0	-1	2	-1
22	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	-1	0	-1	-1
23	2	1	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	-1	2	-1	0	-1
24	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	-1	-1	-1	-1	0

Table 5: Relation matrix

6.1 Blind Step-1

	2	11	12
2	0	-1	-1
11	-1	0	-1
12	-1	-1	0

6.3. Through slot

	13	18	19
13	0	2	-1
18	2	0	-1
19	-1	-1	0

6.2 Blind step-2

	15	16	17
15	0	-1	-1
16	-1	0	-1
17	-1	-1	0

6.4 Blind slot

	5	6	8	9
5	0	-1	-1	-1
6	-1	0	-1	2
8	-1	-1	0	-1
9	-1	2	-1	0

		20	21	22	23	24
6.5 Rectangular pocket	20	0	-1	2	-1	-1
	21	-1	0	-1	2	-1
	22	2	-1	0	-1	-1
	23	-1	2	-1	0	-1
	24	-1	-1	-1	-1	0

Table 6: Features identified from Relational Matrix

Conclusions

Feature recognition plays a key factor toward achieving the integration of design and manufacturing. Although many researchers have dealt with feature recognition problems, it is still need improvement over the previous methodologies found in the literature. The methodology discussed in this paper has several advantages over other methods suggested in the literature. First, The STEP file has the ability to provide a good a generic representation of the simple and compound product data in which the feature, geometry, topology, and manufacturing data are associated. Second, the proposed methodology is flexible to the variations of the STEP file format from different vendors that offer different CAD systems. The algorithm for geometric data extraction from STEP file gives the full details of the geometry including the normal and edge direction of the plane, which reduces the complexity while finding the concavity relation between different faces.

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