



Recognition of Hole Feature and Stock Details of a Prismatic Part from Its STEP AP224 Neutral File

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ABSTRACT: Feature technology transforms the low-level geometrical definition of a 3D model into high level product definition, aiming for full automation of downstream manufacturing activities. Today feature has been implemented in Standard for Exchange of Product (STEP) model specifically in application protocol 224 (STEP AP224), a new ISO standard for neutral file, which utilizes manufacturing feature definition based on prismatic and turning parts. Hence, this research is carried out to study how STEP AP data is stored and how the features can be extracted. The extracted feature details are used in development of CAPP, integration of CAD, CAPP and CAM and finally to automate the generation of NC code for the milling process. This work is limited to recognition of isolated milling features and stock details.

Keywords: Feature technology, NC code generation, neutral file, STEP, computer aided design (CAD), Computer aided manufacturing (CAM).

I. INTRODUCTION

The integration of manufacturing resources are very essential for automation, this can be achieved by integrating seamlessly CAD and CAM through CAPP [1]. Computer Aided Process Planning is a key link for integration. So it is important to have a well designed and performing CAPP system for linking. The present day CAD systems are utilizing Feature Technology as a means to integrate Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM). The feature technology uses two approaches to create high level feature definition in terms of information content are,
-Feature Based Design (FBD)
-Feature Based Recognition (FBR)

The feature based design is used in all most all CAD software's available today, where as the feature based recognition is used in CAM systems.

Most of the neutral files DXF, ACIS, IGES, SET, VDAFS etc., available today employ Boundary Representation (B-Rep) data model for communication or for translation of data among dissimilar CAD systems. This B-Rep data when used in CAM, most of the feature data are not available and are lost, this is mainly because of feature information represented in B-Rep model and makes difficulty in recognizing features [2].

When STEP is compared with above mentioned neutral files, it uses feature technology for data sharing and file transfer. Apart from this, it applies object oriented

techniques to map the relationship among the data structure of file.

This research work aims to recognize basic features of a CAD model and part details using its neutral file STEP AP224 and this feature information can be further utilized for downstream activities of manufacturing. The work will also emphasize on the procedures or methods developed to recognize the features and part details.

II. LITERATURE SURVEY

The research on feature technology has become the major topic in past decade in spite of not knowing the impact of feature technology on the downstream activities of manufacturing. Since earlier work on feature technology by Jung Hyun Han [3], the research has grown substantially. Initially efforts were made to recognize isolated features [4,5] and followed by recognition of interacting feature [6,7]. Despite the fact that, feature recognition system is capable of recognizing interacting features, its implementation on downstream activities of manufacturing has not been carried out. Further research now include manufacturability of the features [8,9]. Feature technology research has matured adequately and has been implemented in STEP AP224 [10]. Feature technology research has shifted to the development of translator to accommodate STEP AP224. Some of the related work in this field is carried out by Bhandarkar [11] Jamaludin Md. Taib *et al* [12].

However, translation of STEP AP224 files has not been addressed adequately. Study on the characteristics of the STEP AP 224 is essential as to develop the methods to extract data of the features. Initial work in this area, is limited to basic features. In this paper the, data related to stock and hole feature is extracted.

III. DATA EXCHANGE WITH STEP

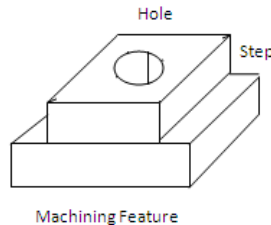
STEP (Standard for Exchanging of Product model data) provides a standardized method to define all data which are necessary for the description of the CAD model throughout its life cycle. STEP is an ASCII file and provides product information along with mechanism and definition for product data exchange and sharing among dissimilar Computer Aided (CAX) systems. STEP was first issued in 1994 and this standard categorizes the CAD data based on Application Protocols (AP). APs are used to define the collection of information required for the base exchange unit. Each AP is known as a formal document that describes the activities of the lifecycle of a CAD model. Currently, there are a few APs that are actively used in research and industries. For examples, AP 201 is explicit drafting, AP 203 for 3D design and the latest one, AP 238 is for STEP-NC manufacturing. This research work is a part of an ongoing research work on development of software for process planing and to integrate the CAD and CAM through this developed CAPP. The best application protocol suits for the CAPP is AP 224, because it provides complete manufacturing information apart from geometrical and topological information to machine single piece mechanical parts.

Step AP 224

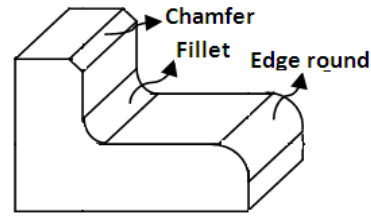
STEP AP224 defines data models which are ranging from low level geometrical definition to high level product definition. The topology and geometrical information of B-Rep model relates to low-level definition and data related to manufacturing and machining feature are related to high level product definition. The STEP AP 224 contains following groups of manufacturing features which are shown in Fig. 1.

- (a) Machining features
- (b) Transition features
- (c) Replicate features
- (d) Compound features

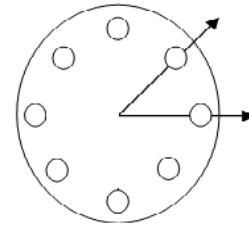
a.



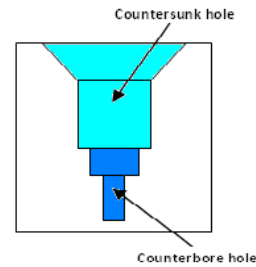
a. Machining feature



b. Transition features



c. Replicate features



d. Compound features

The main focus of this work is on extraction of stock and feature data, which is based on manufacturing features associated with the milling processes (operations).

V. THE EXPRESS LANGUAGE

Express is a data specification language as defined in ISO 10303. It consists of language elements which allow an unambiguous data definition and specification of constraints on the data defined. The EXPRESS language is used to describe the structure of the STEP file through pre-processor. The STEP file as mentioned above is an ASCII file and can be opened using any text editor. EXPRESS maps the relationship between information by binding and inheritance data storage. A portion of STEP file written through EXPRESS language, which defines a cylindrical surface is shown below:

```
#853=CYLINDRICAL_SURFACE
(",#852,0.5625000000000000);
#852=AXIS2_PLACEMENT_3D (",#849,#850,#851);
#849=CARTESIAN_POINT(",(10.062500000000000,-
0.3000000000000000,6.312500000000000));
#850=DIRECTION (",(0.0,1.0,0.0));
#851=DIRECTION (",(0.0,0.0,1.0));
```

In the above example , the line #853 shows the entity CYLINDRICAL_SURFACE(), it has three parameters or arguments, the first one is a label, denoted by ‘ ‘ which is empty, the second parameter is an entity identification number or simply a line number #852 which refers to another entity on the line #852 i.e “AXIS2_PLACEMENT_3D(). The third parameter is numerical value which represents the radius of the cylindrical surface. The entity AXIS2_PLACEMENT_3D() which has four entities, label, Cartesian point, axis direction and reference direction.

The STEP AP 224 physical file

The STEP file begins with a key word ISO-10303-21 and is followed by a header section. Header basically contains the descriptions of the file, such as name of the file, date, time, stamp, author's name, organization name that generates the file and type of file schema or name etc. The header section of a sample STEP file is shown below. It begins with the word HEADER and ends with ENDSEC indicating the end of section.

```
HEADER;
FILE_DESCRIPTION (('AP224 File'),'2;1');
FILE_NAME('ka571_997865309_1.224','1998-09-
30T12:14:10-05:00', ('Mr M Woods'),('LSC: RAMP
Consultancy'),'RPTS MP 6.0','PTC Pro/ENGINEER
Version 18.0','Mr M Woods');
FILE_SCHEMA(('FEATURE_BASED_PROCESS_PL
ANNING')); ENDSEC;
```

The HEADER section is followed by the DATA section. This section is divided into two groups, the geometrical and topological elements. This data is similar to other application protocols that have topology and geometrical B-Rep data, such as AP203 and AP214. The data ends with the highest topology in B-

Rep, which is solid. In STEP, the solid is equivalent to MANIFOLD_SOLID_B-REP() entity. Therefore this entity can be inferred as the end of the topological and geometrical data. The other data that is unique to STEP AP224 is the manufacturing feature definition. A feature in STEP AP224 is defined by sweeping a profile along a given path and its end profile condition. Each feature is described by its geometrical and parametric shape. The geometrical shape will inform the type of profile, path and end profile condition while, the parametric shape will establish the parameter for each profile, path and end profile condition. For instance, a through hole is described by sweeping of a closed circular profile, swept along a linear path and the end condition is ‘through’ while for the parameter shape will ascertain the radii of the circular profile and the distance of the sweeping. These data are stored through binding and inheritance.

Due to the method of storing data, which is by binding and inheritance, frequent line referring is always required. It becomes more cumbersome when the reference line has to be referred to the previous line, causing the reader to read backwards. This becomes worst when a line can be a mixture of numbers and string. This causes every line to be handled as strings. In order to differentiate between numbers and strings, the system has to truncate certain string and convert them to numbers. The truncating and converting process requires the system to fully understand every single line.

Hole Feature and Stock Details Extraction

The CAD model of a part is shown in Fig. 1, consists of a machining feature through hole, which is to be recognized from neutral file STEP AP224 along with part details i.e part width, length, height and its material. A computer program has been developed to recognize the feature and part details in Visual Basic 6.0 based on the algorithm (procedure) developed for extraction of stock and feature details. The details of algorithm and information needed for development of algorithm is as follows.

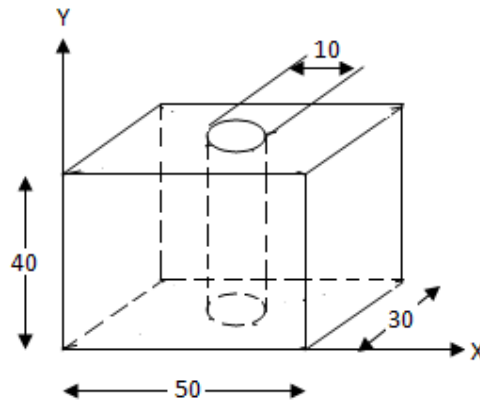


Fig. 1. Part model with Through Hole Feature.

Algorithm for Extraction of Part (Stock) Details:

As mentioned above the part is a prismatic part having geometrical data width, length and height and these data need to be extracted from STEP AP224 neutral file of the part model. Below a section of STEP AP224 file is shown, which defines the stock of the part,

```
#1474=AXIS2_PLACEMENT_3D('orientation',#1471,
#1472,
#1473);#1471=CARTESIAN_POINT(",(0.0000000000
0,-0.000000000000,0.0));
#1472=DIRECTION (",(0.0,0.0,1.0));
#1473=DIRECTION (",(1.0,0.0,0.0));
#1483=(LENGTH_MEASURE_WITH_UNIT()MEAS
URE_REPRESENTATION_ITEM()MEASURE_
WITH_UNIT (LENGTH_
MEASURE(50.000000000000),#15) QUALIFIED
_REPRESENTATION_ITEM ((#1475,#1476,#1477))
REPRESENTATION_ITEM('length'));
#1492=(LENGTH_MEASURE_WITH_UNIT()MEAS
URE_REPRESENTATION_ITEM()MEASURE_WITH_
UNIT(LENGTH_ MEASURE(40.000000000000),#15)
QUALIFIED_REPRESE NTATION_ITEM
((#1484,#1485,#1486)) REPRESENTATION
_ITEM('width');
#1501=(LENGTH_MEASURE_WITH_UNIT()MEAS
URE_
REPRESENTATION_ITEM () MEASURE_WITH_UNIT
(LENGTH_ MEASURE (30.00000000000000),#15)
QUALIFIED
_REPRESENTATION_ITEM((#1493,#1494,#1495))R
EPRESENTATION_ITEM('height'));
#1502=BLOCK_SHAPE_REPRESENTATION ('base
shape', (#1474,#1483,#1492,#1501),#34);
```

The above section of the STEP file contains the geometric details of the stock. For extracting the details, the following algorithm (procedure) is developed.

Read the STEP AP224 file from its ASCII form. Search for the keyword or entity name BLOCK _ SHAPE _ REPRESENTATION () from the file. Extract all the parameters or arguments of the above entity. The first parameter is a string 'base shape', the second parameter is a group of four line identification numbers (id's) and last is again line identification number of STEP file. Store the second parameter i.e group of parameters in an array (first array), each one is a line number in the STEP file in the form of #---. Search for STEP file lines corresponding to the line id's in the array. The first element of the array points to the line having entity AXIS2_PLACEMENT_3D (). The above searched entity has four parameters which describe the orientation of the part model, extract and store these parameters in another (second) array [1-7].

The second element of the array points to the line having entities LENGTH_MEASURE () and REPRESENTATION_ITEM (). Each entity has one parameter, the parameter of LENGTH_MEASURE()

entity indicate distance or dimension i.e LENGTH_MEASURE(dist) and parameter of REPRESENTATION_ITEM() entity is a string and it is the parameter of the stock which is 'length', i.e REPRESENTATION_ITEM ('length'). Extract the parameters 'dist' and 'length' from two entities and they represent the length of the stock in the form similar to length = dist. The third element points to the line having the same entity names as above i.e in step 8, the entity LENGTH_MEASURE() has the same parameter 'dist' where as the second entity REPRESENTATION_ITEM() has the parameter 'width'. Extract the parameters 'dist' and 'width' from two entities and they represent the width of the stock in the form similar to width = dist. The fourth element of the array points to the line having the same entity names as above, the entity LENGTH_MEASURE () has the same parameter 'dist' where as the second entity REPRESENTATION_ITEM () has the parameter 'height'. Extract the parameters 'dist' and 'height' from two entities and they represent the height of the stock in the form similar to height = dist. The data extracted with above procedure represent width, length and height of the stock [8-10]. Search for the elements of another array (second array) pointing to STEP file lines. The first element of the array contains a string 'orientation', which mentions the orientation of the part with respect to current coordinate system. The second element of the array points to a line number of STEP file which has the entity CARTESIAN_POINT() and has two parameter, the first one is blank and the second is group parameter containing three numeric values, representing a Cartesian point X – coordinate , Y – coordinate and Z – coordinate respectively. Extract these coordinates and store in respective variables. This Cartesian point represents the location of the part model in current coordinate system [11-12]. The third element of the array points to a line number of STEP file which has the entity DIRECTION (). The entity has two parameters, the first one is blank and the second is group parameter containing three numeric values, representing X, Y and Z direction respectively. Extract these direction values and store in respective variables. The axis direction of the part is represented by value 1, in the above example it is (0.0,0.0,1.0), means the axis is in the direction of Z axis. The fourth element of the array points to a line number of STEP file which has the entity DIRECTION (), it is considered as reference direction. The entity has two parameters, the first one is blank and the second is group parameter containing three numeric values, representing X, Y and Z direction respectively. Extract these direction values and store in respective variables. The axis direction of the part is represented by value 1. In the above example it is (1.0,0.0,0.0), means the reference axis is in the direction of X axis [13-14]. The output of the above procedure is shown below in Table 1 [15].

Table 1: Part Model Details.

Extracted data from STEP AP224 File for STOCK	
length	50.0
width	40.0
height	30.0
Part position	X =0.0
	Y =0.0
	Z =0.0
Orientation	0.0,0.0,1.0
	1.0,0.0,0.0

Extraction of Hole Feature Details

The presence of any machining feature in the STEP AP224 file is identified by occurrence of three entity, INSTANCED_FEATURE (), MACHINING_FEATURE () and actual feature occurrence like SLOT(), HOLE(), POCKET() etc., and these entities are used in identifying features. The occurrence of these entities in the STEP file is as follows,

```
#...=(CHARACTERIZED_OBJECT()FEATURE_DEFINITION()
INSTANCED_FEATURE()MACHINING_FEATURE
() FEATURE_NAME()SHAPE_ASPECT());
```

In the above STEP line, we can notice the presence of all the three entities, in place of actual feature name, it mentioned as FEATURE_NAME (). In this place any of the feature name, which are defined in the STEP AP224 application protocol may appear. The features can also be identified by using only feature name as key word. In this research work, to identify the machining feature its name is used as key word in the format as defined in the STEP AP224.

The round hole shape is formed by sweeping circular closed profile along a linear path. The circular closed profile and linear path are represented by two entities CIRCULAR_CLOSED_PROFILE () and PATH_FEATURE_COMPONENT () respectively.

To identify and extract the data of machining feature round hole from STEP file, the following procedure is developed

Search the STEP file for the occurrence of the key word ROUND_HOLE (). The sample result of the above step is shown below [1],

```
#1630=(CHARACTERIZED_OBJECT(",")FEATURE_DEFINITION()INSTANCED_FEATURE()MACHINING_FEATURE ()
ROUND_HOLE()SHAPE_ASPECT(",",#43,.T.));
```

Search for the entities CIRCULAR_CLOSED_PROFILE() and PATH_FEATURE_COMPONENT(), which define the shape of the hole feature, using their names as key words in STEP file and the result is shown below [2],

```
#1643=CIRCULAR_CLOSED_PROFILE('circular profile',"
# 1642,.F.);
```

```
#1658=PATH_FEATURE_COMPONENT ('linearpath
','linear ',#1657,.F.);
```

```
#1663=(LENGTH_MEASURE_WITH_UNIT()MEASURE_REPRESENTATION_ITEM()MEASURE_WITH_UNIT
(LENGTH_MEASURE(40.0),#15)REPRESENTATION_ITEM('distance'));
```

Extract the first parameter of the entity PATH_FEATURE_COMPONENT () which is a string 'linear path' and store in a variable [3].

Next extract the parameter value of the entity REPRESENTATION_ITEM () again which is a string 'distance' and store in a variable [4].

Lastly extract the parameter value of the entity LENGTH_MEASURE () which is a numerical value, representing the value of 'distance' parameter of entity REPRESENTATION_ITEM () which is 40 mm. The depth of the hole is equal to the length of the linear path, which is represented by REPRESENTATION_ITEM ('distance'), I.e depth = 40 mm [5].

Next search for location of feature, which can be identified by searching the entity Cartesian point using its name as key word, i.e CARTESIAN_POINT(). The sample result is shown below [6],

```
#1687=CARTESIAN_POINT (",(0.0, 0.0 , 0.0));
#1688=DIRECTION (",(0.0,-1.0,0.0));
#1689=DIRECTION (",(1.0,0.0,0.0));
```

Extract the parameters of above entities i.e CARTESIAN_POINT(),DIRECTION() and DIRECTION() and store in variables.

The other important parameter of the hole, which is necessary for machining, is diameter of the hole [7].

This parameters can identified by searching two entities LENGTH_MEASURE () and REPRESENTATION_ITEM () using their names as key word which contain the parameter value for diameter. The sample result of the search is shown below,

```
1652=(LENGTH_MEASURE_WITH_UNIT()MEASURE_REPRESENTATION_ITEM()MEASURE_WITH_UNIT
(LENGTH_MEASURE(10.000000000000),#15)QUALIFIED_REPRESENTATION_ITEM(
(#1644,#1645,#1646)) REPRESENTATION_ITEM('diameter'));
```

#Extract the parameters of the entity LENGTH_MEASURE () and REPRESENTATION_ITEM () which are respectively 10.0 cm and 'diameter', i.e diameter = 10.0 mm.

The final attribute of hole to be extracted is its bottom condition [8]. The hole can have different bottom conditions. The probable bottom conditions for hole are shown below in Fig. 2.

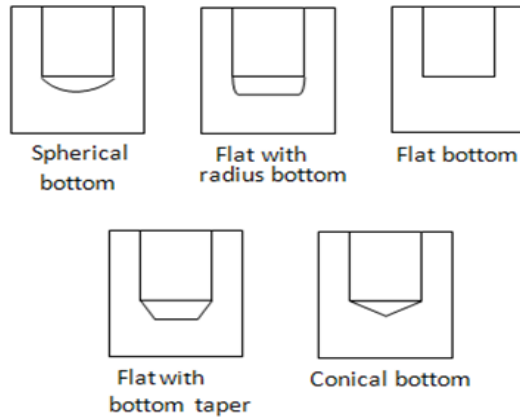


Fig. 2. Hole bottom Conditions.

The hole bottom condition can be extracted by searching the entity HOLE_BOTTOM () by its defined name. The sample result of the search is shown below
 #1675=HOLE_BOTTOM ('bottomcondition','through',#1674 ,.F.);

Extract the parameters of the entity HOLE_BOTTOM (), which has its first parameter 'bottom condition' and second parameter is 'through', which indicates that the hole feature has through hole.

The output of the above algorithm is shown below in Table 2,

Table 2: Hole Feature Details.

MACHINNG_FEATURE()	ROUND_HOLE()	
CIRCULAR_CLOSED_PROFILE()	'circular profile'	
PATH_FEATURE_COMPONENT()	'linear path'	
REPRESENTATION_ITEM()	Distance (Hole Depth)	40
CARTESIAN_POINT() Hole location	X	0.0
	Y	0.0
	Z	0.0
DIRECTION () Orientation of hole feature	Direction	0,-1,0
	Reference Direction	1,0,0
	diameter	10.0
	Bottom condition	through

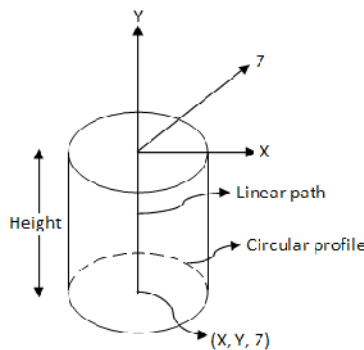


Fig. 3. Hole Feature Details.

Extraction of Stock Material

The material information of the stock should be extracted, which helps in selecting proper cutting tools and machining parameters to machining the part. The entity MATERIAL_DESIGNATION () provides the information about material of the stock in its parameter. Search for the entity MATERIAL_DESIGNATION () from STEP file using its name as key word and the

result is shown #2392=MATERIAL_DESIGNATION ('steel to MoD DEF 13(B) Group 2', (#2391)); Extract the first parameter of the entity, which indicates the material of the stock, in this case it is steel.

Architecture of the current system:

The figure below shows the the architecture of the system. Which is self explanatory and shows how flows seamlessly from CAD to CAM.

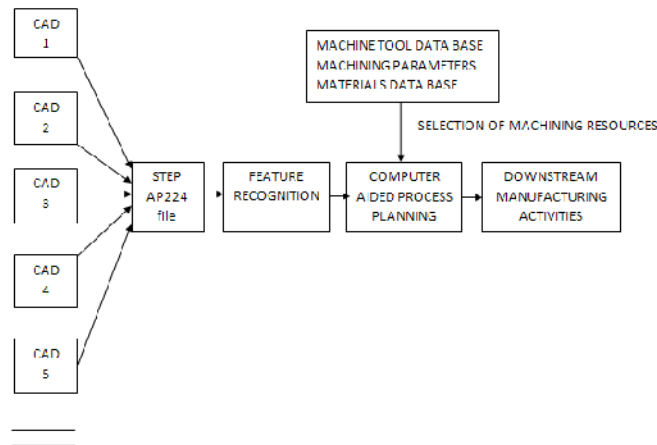


Fig. 4. Architecture of the system.

CONCLUSION

Neutral file concept has helped to transfer CAD model data among dissimilar computer aided systems giving rise to platform independency. The ISO designed and developed neutral file STEP, which is globally accepted by industries is more predominant over all other neutral files because of its strong data model. The computer aided process plans developed based on STEP, especially AP224 neutral file are more flexible and becoming universal. In this research, machining feature circular through hole and part details are recognized from the developed methods (algorithms) using STEP AP224 neutral file and this data will be used for the development of computer aided process planning, which automates the downstream manufacturing activities for realization of part by generating NC code automatically.

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