

Feature recognition for CAD/CAM integration using the STEP file: A review

Avtar Singh

Assistant Professor, Yadavindra College of Engineering, Talwandi Sabo (Punjab) India - 151302

Abstract

Feature recognition is the technique used to extract the various features of CAD part model. This technique is efficiently converting the low level information of part model into the high level of information. The high level information contains the topological information which is required for feature recognition. There are several approaches currently in use for feature recognition. STEP file in some approaches the topological information contained by the STEP file is used as input for feature recognition technology. STEP file is Standard for the Exchange of Product Data, an ISO standard exchange format. This format can be used as standard exchange format among the various CAD part modeling tools. In this paper the techniques are elaborated which used the STEP file as input for feature recognition.

Keywords: Feature Recognition, CAD part model, Topological information, STEP file.

1. Introduction

Traditionally the various activities of process planning were performed by the process planner. The outcome of a good process plan has been depending on the skill and experience of the process planner. This activity also took too much time resulting in the increase in lead time of product to market. To reduce the manual efforts the feature recognition techniques have been developed for CAD/CAM integration. CAD/CAM integration is a challenging task and requires the consistent efforts to achieve the goal. This can be achieved by the extracting the information from CAD part model. The CAD part model contains the low level information of the geometric entities which has to be translated into the high level information for feature recognition. Feature recognition is the technology which reconstructs the decomposed CAD model for downstream application. It is a simple and efficient method which provides the efficient results for recognition of features of CAD part model.

There are various approaches available for the feature recognition namely graph based, cell decomposition, convex hull decomposition rule based feature recognition approach. Each technique requires the low level information of the CAD model. STEP file is a standard exchange of product format which can be used as the input for feature recognition. The STEP file is in the format of ISO – 10303 which is an international acceptable standard for exchanging data between different CAD/CAM. It represents a viable alternative to the standards and proprietary data formats, and is a proven way to ensure fast, reliable data exchange between partners. In this paper an attempt has been made to review the various techniques in which STEP file is used as the input.

2. Feature

Features are a product's generic shapes or characteristics that have some engineering significance about the product. For example, in the machining domain, where a product is made by material removal; the feature set includes holes, slots, and pockets [1]. In other words this can be defined as a region of a part with some interesting geometric or topological patterns. This geometric or topological information is the adjacency of different surfaces of product which is used for feature recognition. Various researchers classify features for feature recognition which is discussed in the next section of this paper.

2.1 Classification of Features

Features are domain specific as shown in [Fig 1] so they can be classified accordingly. Broadly, features can be classified as

- i) Isolated features those that do not share the volume with the other feature
- ii) Compound feature is the combination of two single features. It handles the situation in which a large single feature contains certain smaller attached single features.

- iii) Interacting and intersecting features are also the form of compound feature. Feature interactions are intersection of feature boundaries with those of others feature such that either the shape or the semantic of a feature are altered from the standard or generic definition [2].

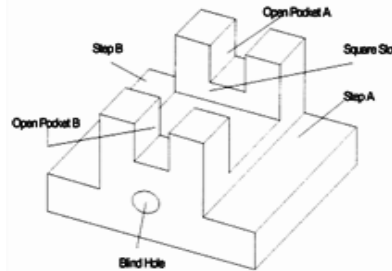


Figure 1 Types of features [1]

3. Recognition of feature using STEP file

Standard Exchange of Product data (STEP) file is the unique format used for the exchange of low level information among the various modeling tool. This format contains the boundary representation (B-Rep) information of the CAD part model. B-Rep describes the geometry of object in terms of boundaries, namely vertices, edges, and surfaces. In order to represent a solid object by its surfaces the orientation of each surface must be defined as the interior or exterior of the object. The B-Rep information is the easiest way to describe a solid model. There are several approaches developed for feature recognition of CAD part model which are discussed below.

Borker and Puri [2] proposed a system of feature recognition for 3D components for automated process planning activities in a computer integrated manufacturing environment. The features were identified using the STEP information of the CAD 3D model for downstream application. This paper also proposed the efforts for the integration of two activities such as feature recognition and process planning.

Bhandarkar and Nagi [3] developed a feature extraction system that can store the feature data in a computer interpretable format and which can be transmitted between various locations. The extracted form features data are saved in the STEP part 21 file formats which can be read by the CAD system. The output file generated by the feature extraction system contains all the geometrical and topological information to completely define the solid and the feature information.

Ismail et al. [4] described the system for the extraction and recognition of feature from step file. The feature recognition system is developed using the expert system shell to recognize through and blind hole.

Dereli and Filiz [5] use the STEP for interfacing of design to process planning. The methodology used in the development of the interface, use of both automatic feature recognition and feature based design technologies. Using the ability of the STEP, a generic scheme is developed in which the feature is treated as a combination of faces to which the geometric and technical information is associated. The limitation of this methodology the initial blank is cubical billet.

Holland, et al. [6] describe the development of a CAD linked standalone computer based metalforming process module which can be used for the purpose of process planning in the field of design. The input for the feature recognition system is an ISO 10303(STEP) format text file. The computer program has been written to dissect the STEP output from a CAD drawing into manufacturing processes element suitable for the metal forming. This module recognizes the limited numbers of features from the STEP input file.

Madan et al. [7] proposed the methodology for the determination of the parting line and parting direction. In this research STEP file of the part is used to extract the die casting feature which consist of protrusion and depression regions of the part. The features are classified single, double, or multiple withdrawal directions. Geometric

modeling is used for the feature recognition which includes the nested and interacting feature. Parting line is determined based on selected candidate parting direction considering process constraint and priorities.

Venu et al. [8] developed an approach to recognize the prismatic parts feature STEP AP 203 file format with JSDAI interface. This technique was called Object Slicing Approach. The various feature of the part were stored in the manufacturing feature pattern library on the imaginary planes. The features present in the solid model were compared with available library of the features for recognition using iteration process. Further this work may be found suitable for the recognition of curved, intersecting and other complex features.

Yifei et al. [9] reported the work related to the integration of CAD and CAPP. The STEP file of AP214 was used for exchange of data CAD model data. Rule based approach was used to develop an algorithm for basic features.

Sreeramalu et al. [10] developed an integration of CAD with CAPP using STEP file. The STEP file contained the geometrical and topological information of the part related to the manufacturing data. 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 implementing the turning feature recognition algorithm. A turning feature recognition algorithm developed for the recognition of turning features. This turning feature recognition algorithm has the ability to recognize the different features like Cylindrical, Toroidal, Conical, Threading, radial and axial holes etc. with attributes of the features.

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