



Milling Features Recognition Methodology of Rectangular Fillet Blind Pocket for Universal Data Structure

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ABSTRACT

Various studies have focused on developing an interface system that could integrate the machining process with CNC (Computer Numerical Control) machine tools by producing tool paths using STEP (Standard for the Exchange of Product model data) file as an input function. Studies to recognize features in STEP file by encoding the data of geometric entities of a CAD (Computer-Aided Design) design has been done by many researchers. However, they only focus on Cutter Location (CL) which eventually describes the machining profile in the x and y planes without considering the depth of the machining feature in the z-axis. Identifying and recognizing machining features in the STEP file is important since STEP is a platform for converting industry-standard CAD systems and software data models. The aim of this paper is to identify profiles of geometric machining features for rectangular fillet blind pockets using the Geometric Data Extraction (GDE) approach. Three cases have been carried out namely single rectangular fillet blind pocket, double rectangular fillet blind pocket and triple rectangular fillet blind pocket. The results were verified by manual graph plotting and automatic graph plotting using CPM 3D Plotter.

1. Introduction

STEP is a high-level programming language that allows solid models to be created using CAD software. The model data structure can provide a two-way data flow that can transfer data between CAD, CAM (Computer-Aided Manufacturing), and CNC platforms without redundancy or risk of information loss [1]. Object-oriented modelling should be utilized to define the exchanged data as there should be no data leakage on the Computer Assisted Technology (CAx) data chain [2]. In this study, the CNC milling machine is the most suitable for the tasks. Production processes known as CNC machining are controlled and operated by computers comprise cutting speed, tool and workpiece interface temperature, feed rate and, cutting depth in order to form workpieces made of materials

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such as metal, wood, plastic, foam, etc. [3-4] Although there are numerous types and procedures used in CNC machining, their fundamental principles typically remain the same. The following steps make up the fundamental CNC machining process: designing a CAD model, converting a CAD file to a CNC programme, setting up the machine, and carrying out machining operations on the machine [5].

Many types of research had been done regarding automatic feature recognition of STEP files such as linear features [6], circular features [7] and the latest studies involving full profile features (a combination of linear features and circular features) [8]. However, they only focus on Cutter Location (CL) which eventually describes the machining profile in the x and y plane without considering the depth of the machining feature in the z-axis. Therefore, a practical system of identifying and recognizing machining features such as pocket and boss (blind pocket) in the STEP file and translating it into G-code had not really taken off. The loss of geometric data while generating G-codes from a CAD file as users edit and fill in the specifics of the lost data had frequently been encountered in this research area [9].

The 3D solid model is created by the user or programmer using CAD modelling software as the product model and is stored in a STEP file format [8, 10-11]. The STEP-data structure consists of a text format comprising general information which are the cartesian points, lines, vectors etc. However, it is difficult to identify and recognize the geometric machining features. In this study, identifying geometric machining features in a STEP file was conducted for a rectangular fillet blind pocket using the Geometric Data Extraction (GDE) approach. Identifying and recognizing the machining features of the models in the STEP file is important since the file could describe product data through the product life cycle. Three cases have been carried out with different numbers of rectangular fillet blind pockets which are a single rectangular fillet blind pocket, double rectangular fillet blind pocket and triple rectangular fillet blind pocket. The results were verified by a manual graph plotting and automatic graph plotting using CPM 3D Plotter.

2. STEP Data Structure

An initial concentration on design and manufacturing applications that is standard, ISO 10303 informally known as STEP (Standard for the Exchange of Product model data) addressed this issue. STEP is the standardized data format for machining part technical specifications and exchanging CAD data. This new open-source format is also a CAD-independent format that can only be used on 3D drawings [10-12]. Any CAD software system such as SolidWorks 2021, Autodesk Inventor, CATIA, etc can be used to extract all relevant data from this STEP file. STEP-data encoding and decoding in pre-processors are widely used in CAD/CAM software packages.

STEP can be generated into G-code for CNC machining, a low-level programming language instructing the machine to move the tool path in the desired direction. There was no unified standard that CNC machine manufacturers used to encode the CAD design before developing the STEP file. On CNC machines, post-processors must decode the CAD model and encode it into G-code. CAD models and G-code become incompatible when machines use different standards. Engineers will need more time to make these machines function correctly. Production may be delayed because of this incompatibility. Figure 1 shows the structure of the STEP file. All of the descriptions of STEP File structure elements are listed in Table 1.

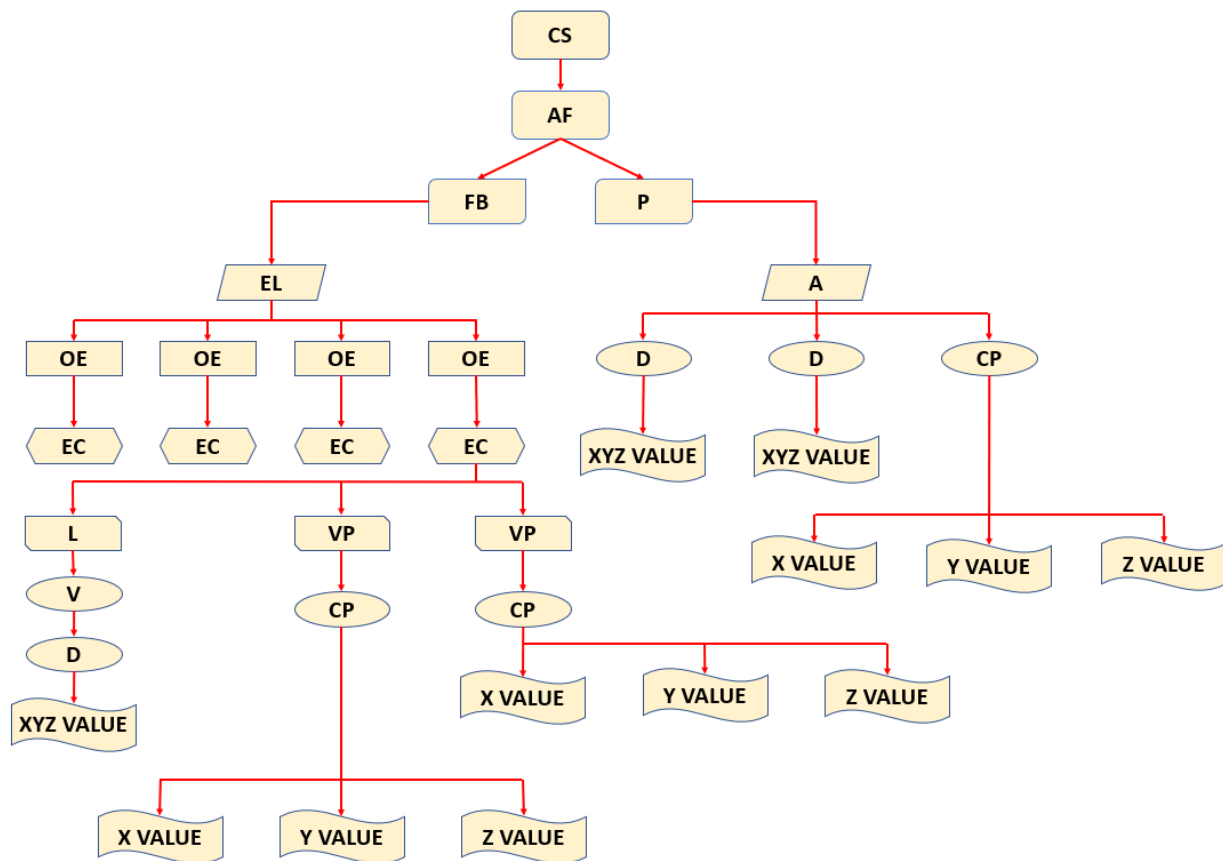


Fig. 1. STEP File Structure

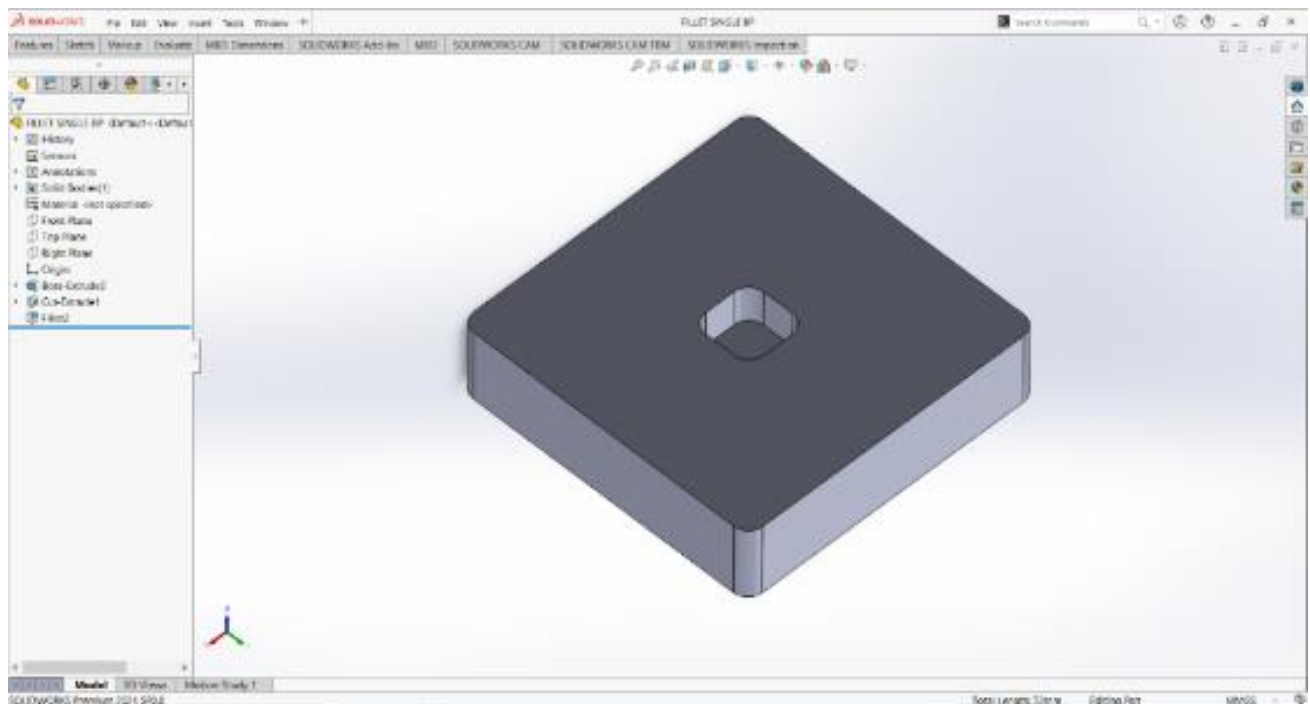
Table 1
 Description of STEP File Structure Elements

STEP Element	Description
CLOSED_SHELL	A group of one or more faces leap a region in three-dimensional spaces and separates the space into two districts, one bounded and the other unbounded.
ADVANCED_FACE	A face related to a set of surfaces.
FACE_BOUND	A loop for bounding a face.
EDGE_LOOP	A path in which the start and end vertices are identical.
ORIENTED_EDGE	An edge is constructed from the original edge and contains the direction (orientation) information. It will be equivalent to the original edge if the orientation information is not comprehended.
EDGE_CURVE	Contain geometry fully defined.
VERTEX_POINT	Define the geometry of a vertex.
CARTESIAN_POINT	Address a point in Cartesian space. The hierarchy of the different elements of the STEP file starts from the Closed Shell to move toward the Cartesian Point.
LINE	A line segment lies with the Cartesian Point plotted on a line.
VECTOR	A quantity that contains directions.
DIRECTION	The path that will move the cutter.
PLANE	The local surface of the x-axis and y-axis, with the z-axis direction as the normal surface.
AXIS2_PLACEMENT_3D	Contain a point through which the normal surface passes and two direction entities define the local surface of the x-axis and z-axis for the plane.

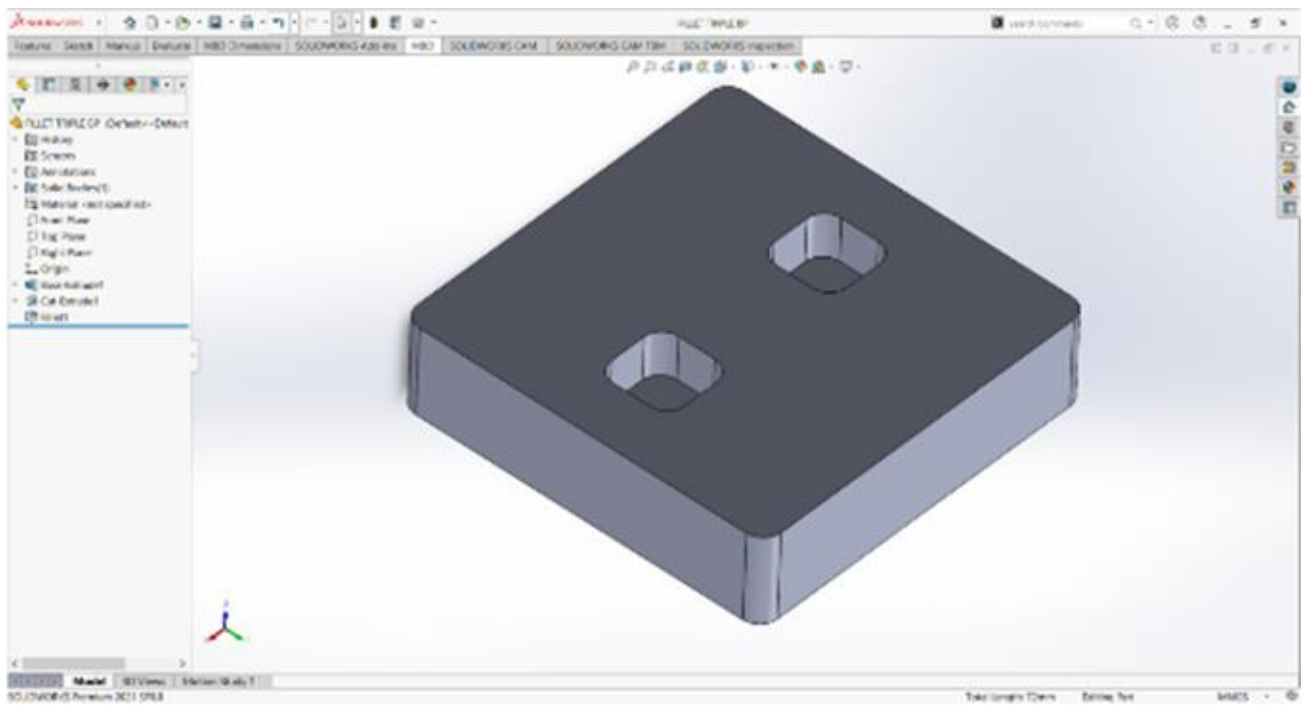
3. Proposed Methodology

As for this research, STEP AP 242: Managed Model Based 3D Engineering file format will be used to analyse and identify profile geometric comprises machining features for the rectangular fillet blind pocket. STEP AP 242 file will exploit the cartesian point coordinates to ease the recognition process. Then, the data structure of a STEP file as shown in Figure 1 is analysed, and the GDE approach is performed to classify existing geometric data. The GDE algorithm is developed based on a rule-based approach that simplifies and facilitates data representation and removal.

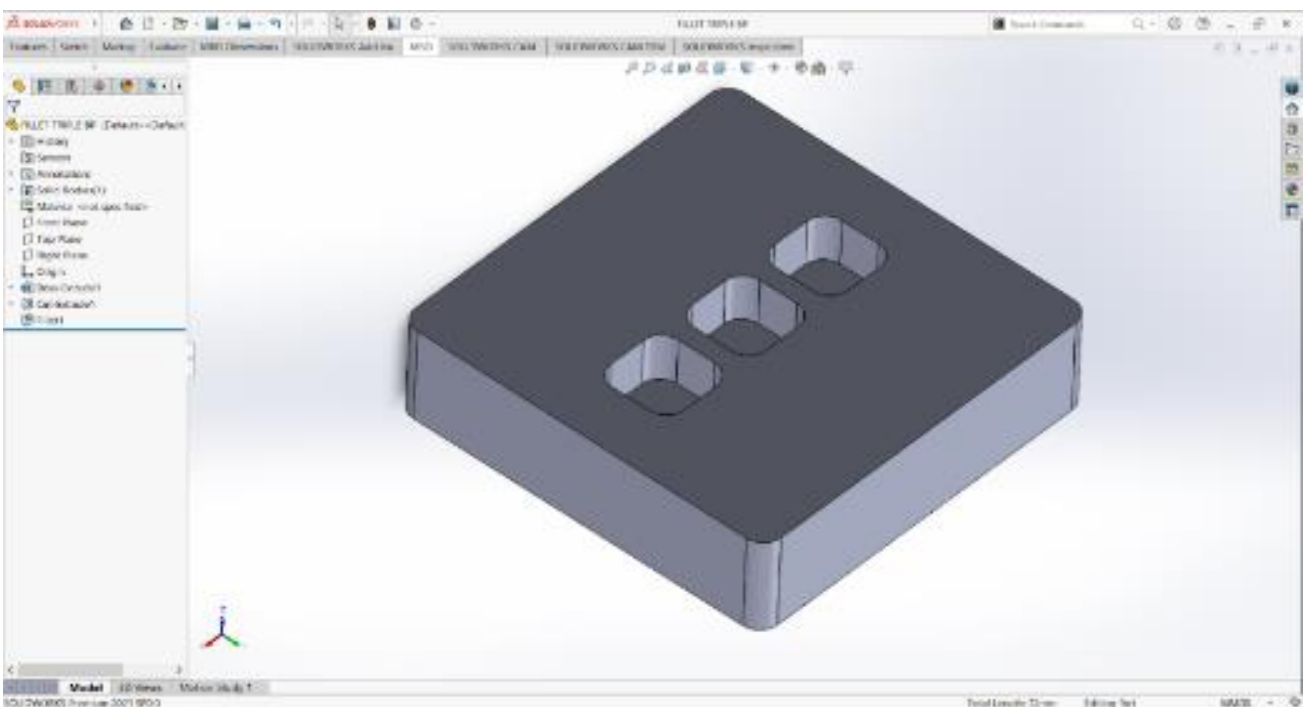
Three case studies were conducted in identifying the geometric machining features in the STEP file which are single rectangular fillet blind pocket, double rectangular fillet blind pocket and triple rectangular fillet blind pocket. All the cases were designed in CAD software (SolidWork2021) with the same size of block which is 100 mm (length) x 100 mm (width) x 24 mm (thickness) and also the same size of each fillet blind pocket that is 20 mm (length) x 20 mm (width) x 12 mm (depth), respectively. These 3D CAD models are converted into STEP AP242- Managed Model Based 3D Engineering files. Figure 2 depicts the 3D CAD models using SolidWork2021 for (a) single rectangular fillet blind pocket, (b) double rectangular fillet blind pocket, and (c) triple rectangular fillet blind pocket.



(a) Single Rectangular Fillet Blind Pocket



(b) Double Rectangular Fillet Blind Pocket



(c) Triple Rectangular Fillet Blind Pocket

Fig. 2. 3D CAD Models using SolidWorks 2021 (a) Single Rectangular Fillet Blind Pocket (b) Double Rectangular Fillet Blind Pocket (c) Triple Rectangular Fillet Blind Pocket

4. Case Studies Analysis

The geometric database is created by analysing the geometric information available in the STEP AP 242 file using the boundary representation (B-rep) technique [13]. There are two types of information in the B-Rep model: geometric data to describe coordinates, positions in 3D space, and curves, and topological data to connect multiple geometric entities such as solids, surfaces, loops and

edges [14-16]. There is a hierarchy in choosing the factors that define the geometry as all of the data from the STEP file is studied as a file, and the faces, edge loop, edges, vertices, and Cartesian points are one by one being traced separately. A specific order is followed while extracting data from STEP files [17]. As for this study, CARTESIAN_POINT with x, y, and z coordinates values will be extracted from STEP files in defining the CAD model structure for each case.

In this study, the pocket feature can be identified from the face loop by extracting the dimensions of the constant value coordinate and the changing value of the coordinate in a face. The face bound flag denotes which face has an inner loop, and if it is a blind pocket, only one face will be having a face bound. In the case of a through-the-pocket, both planar faces will show an inner loop, and the faces can be identified by checking parallel and perpendicular faces for the inner loop. The changing coordinate value along the z-direction is taken as height or depth, and the varying value of the other two coordinates inside a planar outer bound gives the length and breadth of the pocket in the inner loop [18]. The depth of the blind pocket can be identified when the z-value of CARTESIAN_POINT is (-12) as the depth is 12mm and the thickness of the block is 24 meaning the z-value of CARTESIAN_POINT is (-24mm). Table 2 listed a comparison of all the information in the STEP AP242 file for all cases.

As can be seen from Table 2, the reference number line (final line) was increasing when the number of pockets increased constantly with 282 total increments same goes for the total of ADVANCE_FACE with a total of 5 increments. Then, the difference number line for CLOSED_SHELL also has a correlation (increments of 282) with the number of pockets applied. This is due to the more features added to the model, the B-rep model will encode more geometric information such as more information about the geometric data and topological data. The details of geometric data and topological data are documented in Sharizam *et al.*, [7].

Table 2
 Comparison of STEP AP242 File for all cases

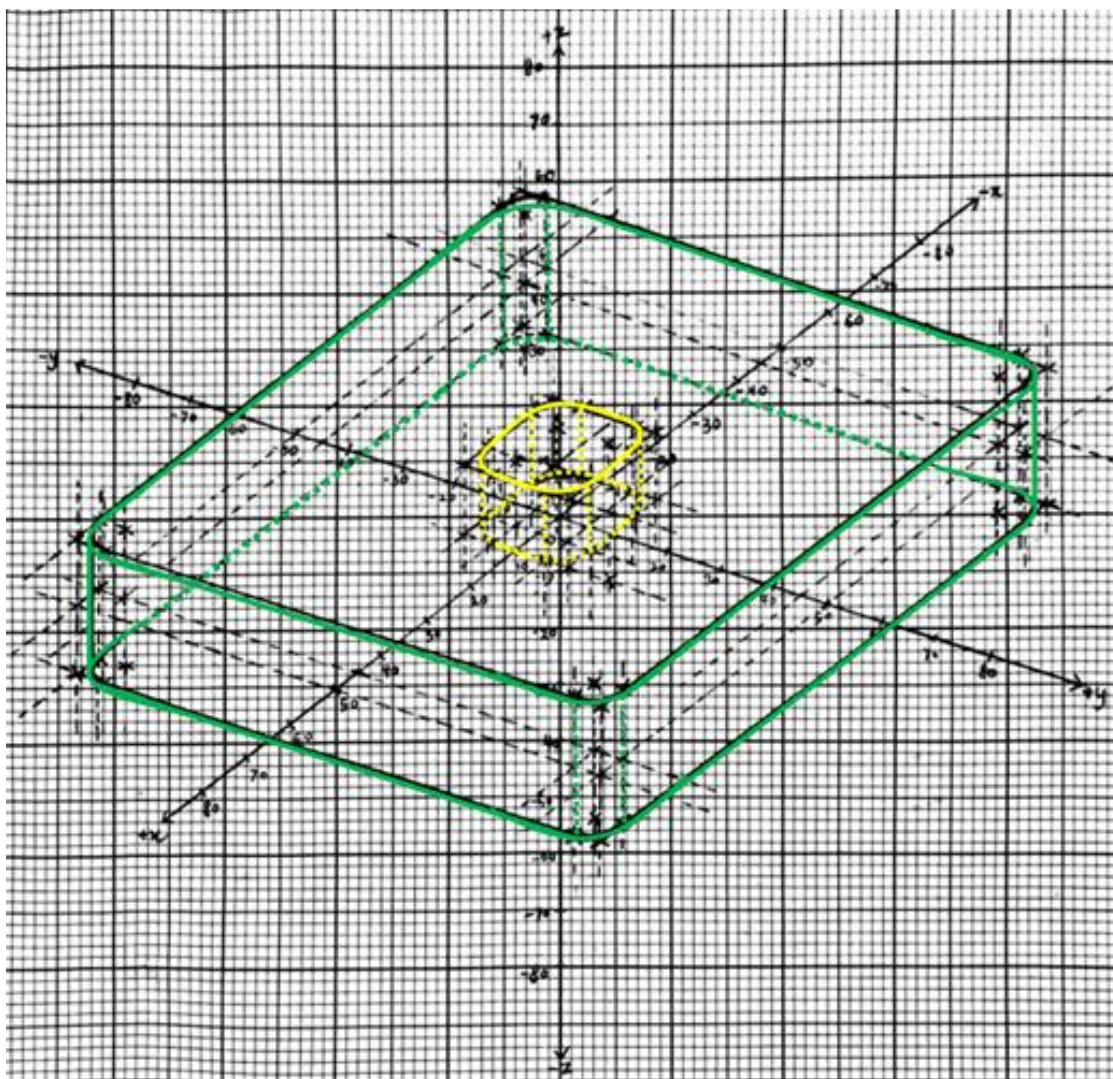
CASES	Reference Number Line (Final Line)	Changes of Header	Total of ADVANCE_FACE	CLOSED_SHELL
Single Rectangular Fillet Blind Pocket	#634	NO	19	#597=CLOSED_SHELL(#66,#106,#148,#188,#245,#269,#309,#349,#373,#398,#415,#439,#461,#479,#533,#550,#567,#584,#596)
Double Rectangular Fillet Blind Pocket	#916	NO	24	#879=CLOSED_SHELL(#68,#110,#141,#172,#203,#378,#409,#459,#490,#507,#532,#557,#574,#598,#610,#627,#652,#677,#701,#733,#767,#779,#803,#825,#837,#854,#866,#878)
Triple Rectangular Fillet Blind Pocket	#1198	NO	29	#1161=CLOSED_SHELL(#66,#108,#148,#179,#221,#263,#296,#338,#369,#428,#459,#499,#530,#570,#594,#625,#658,#700,#722,#753,#784,#806,#824,#848,#864,#888,#921,#952,#974,#1050,#1068,#1080,#1098,#1120,#1132,#1144,#1160)

5. Result and Discussion

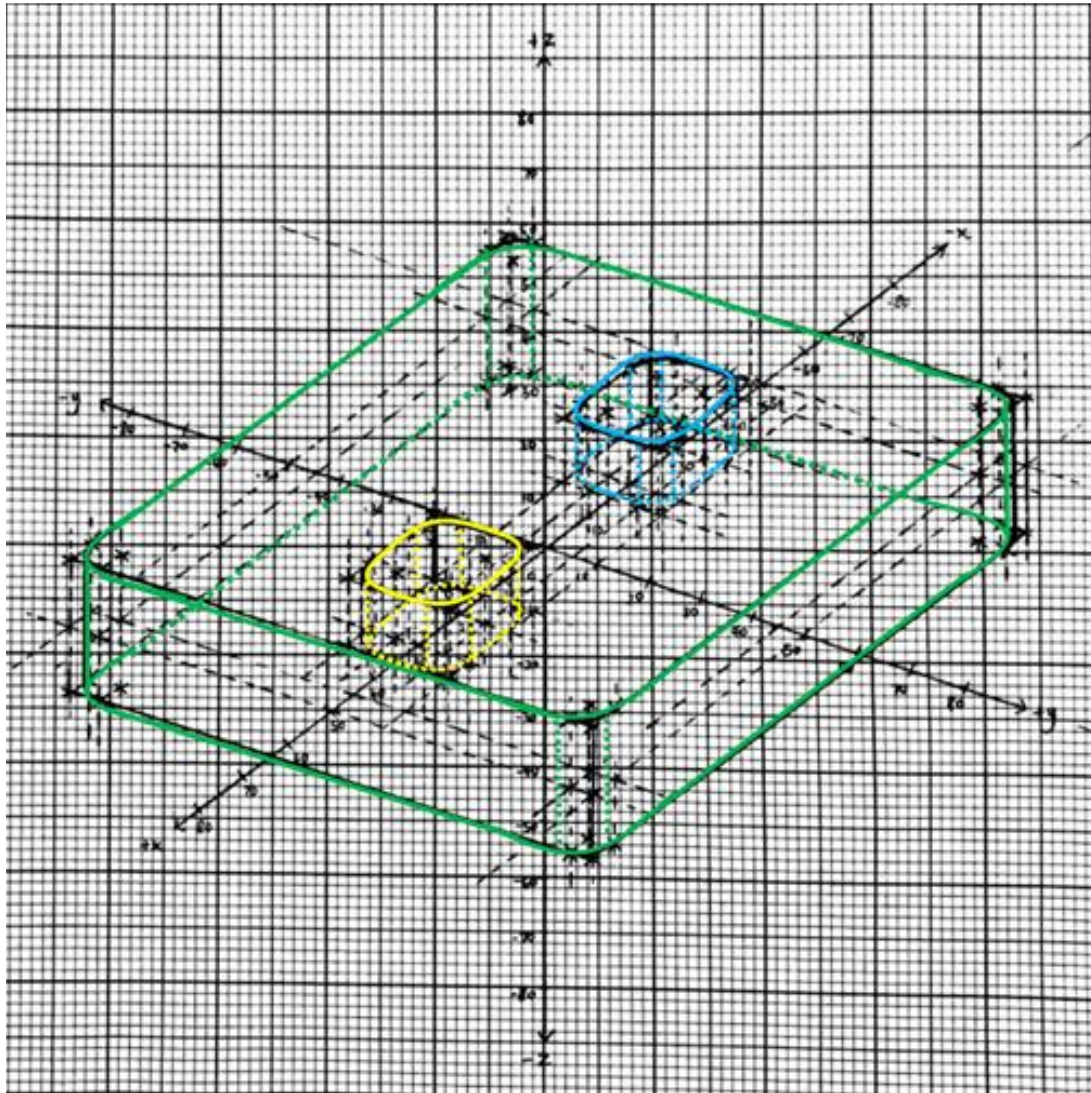
5.1 Validation Process

As to validate the identifying data process from the STEP AP 242 file for all cases, manual draft plotting using grid paper (Figure 3) and automatic graph plotting through CPM 3D PLOTTER (Figure 4) were conducted based on the CARTESIAN_POINT analysed through STEP AP242 file for each case.

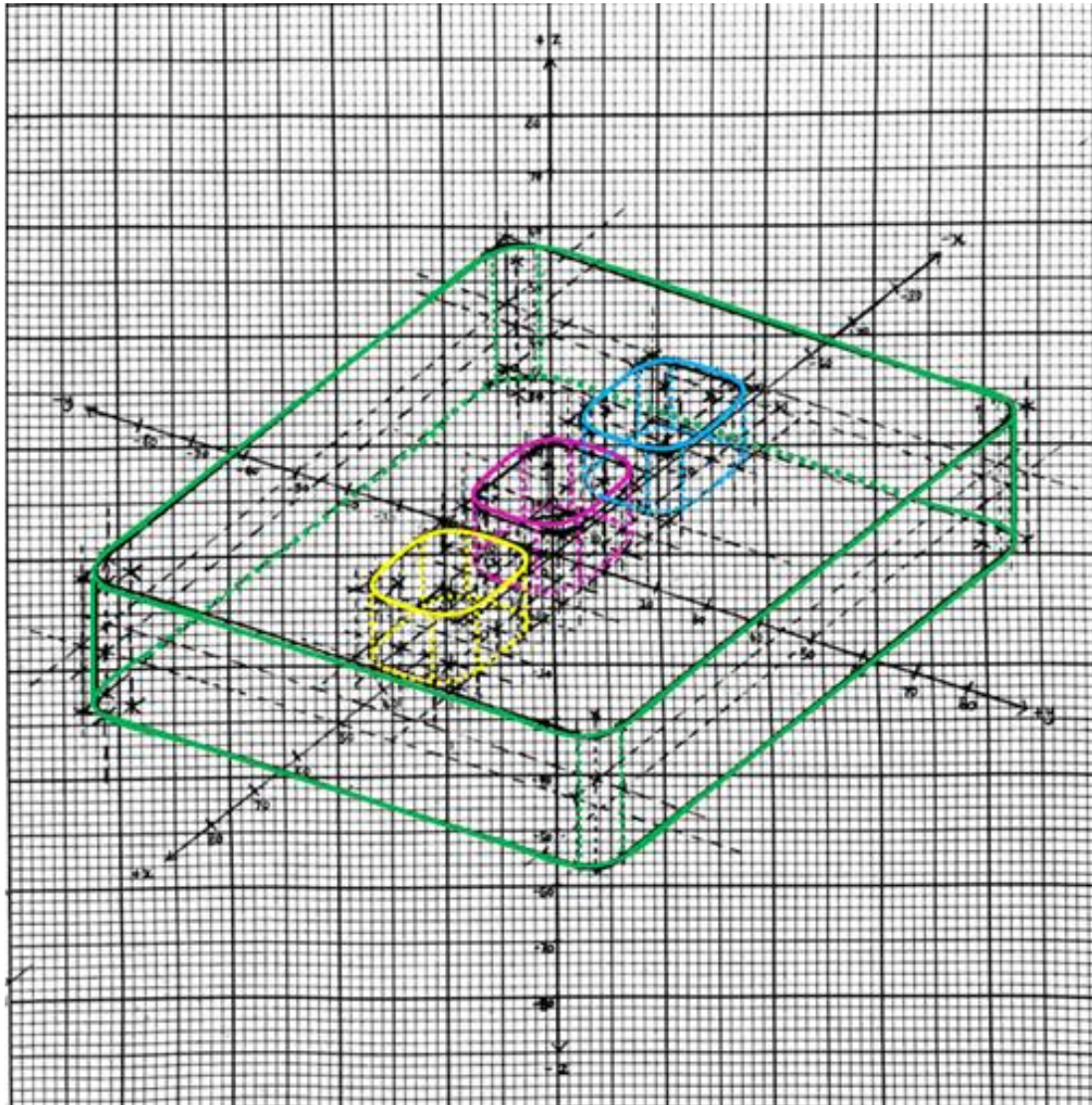
Figure 3 shows the manual draft plotting using grid paper of the 3D model after recognizing the features from the STEP AP242 file for all cases. The green line represents the edge of the block. The yellow, purple and line dotted represent the edges of the pocket. Meanwhile, the 3D model automatic graph plotted was constructed using CPM 3D PLOTTER as shown in Figure 4. These results have good agreement with the data extracted from the STEP AP 242 file. The hierarchy of geometric data must be understood in order to comprehend a STEP file, and each keyword denotes a different geometric entity.



(a) Single Rectangular Fillet Blind Pocket

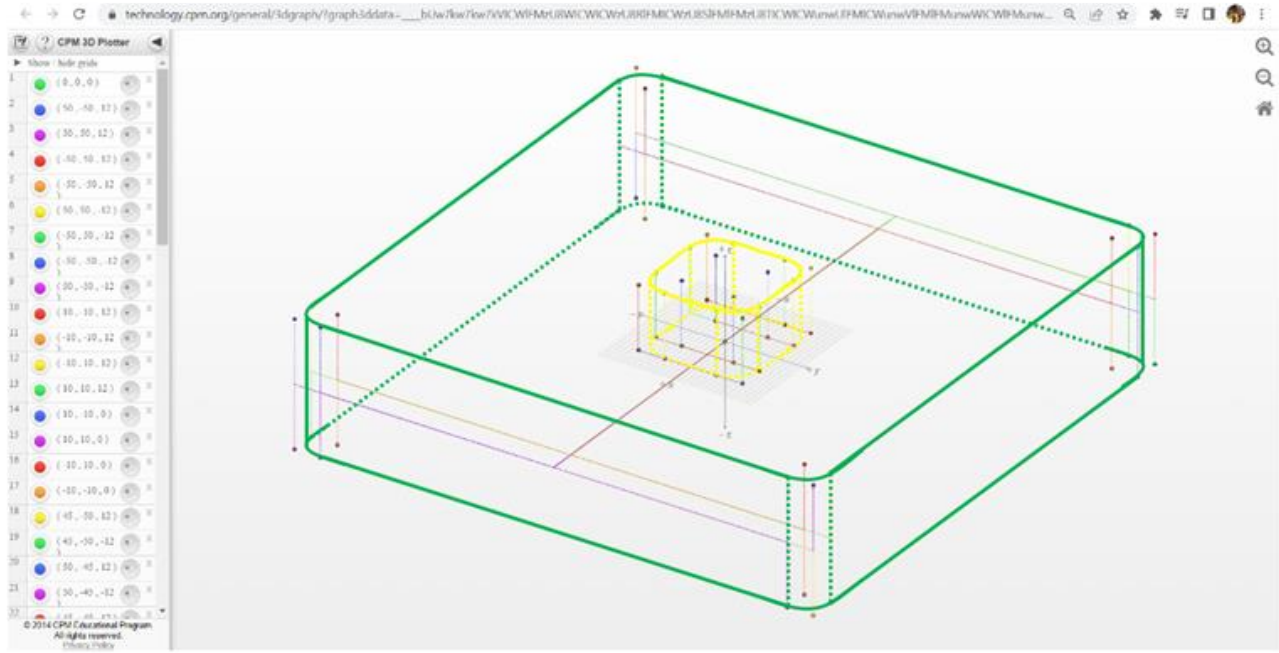


(b) Double Rectangular Fillet Blind Pocket

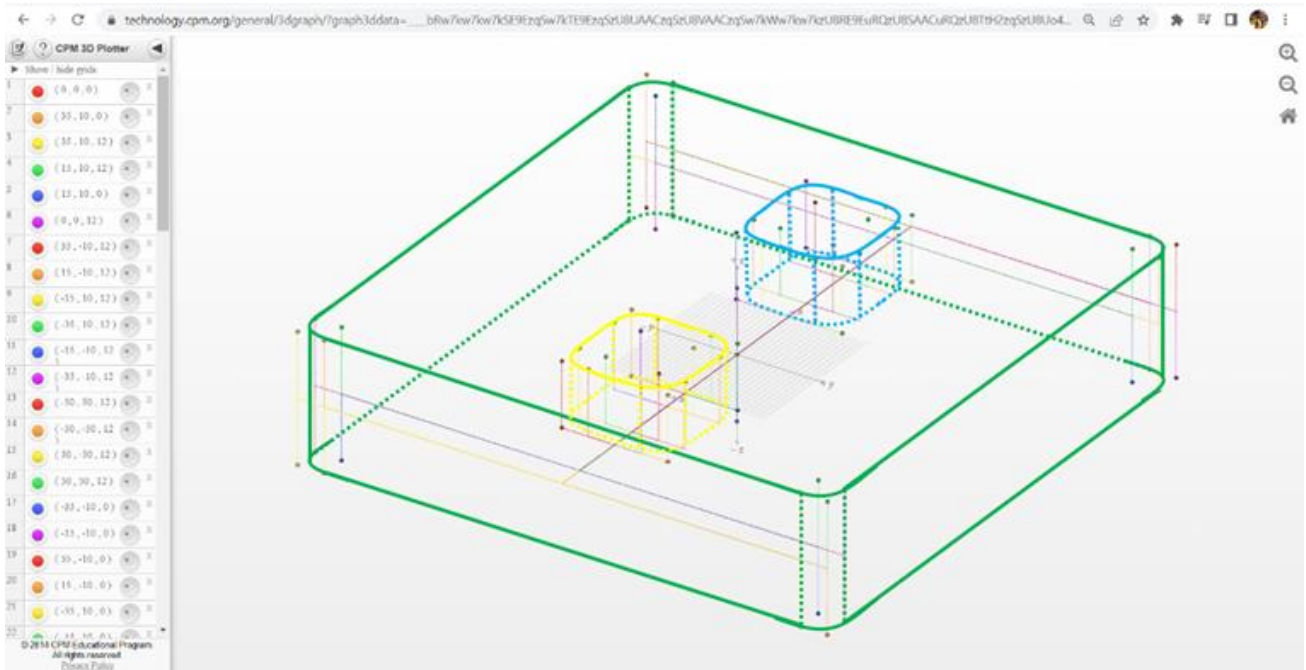


(c) Triple Rectangular Fillet Blind Pocket

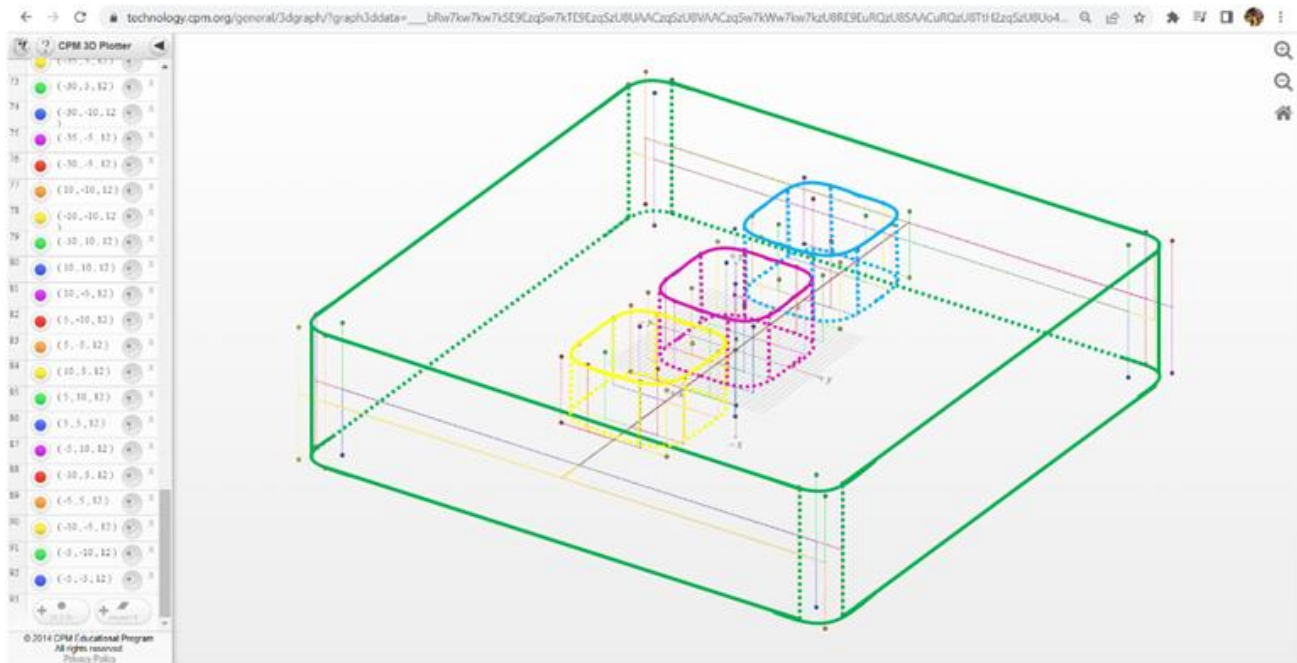
Fig. 3. 3D CAD Models using manual draft plotting (a) Single Rectangular Fillet Blind Pocket (b) Double Rectangular Fillet Blind Pocket (c) Triple Rectangular Fillet Blind Pocket



(a) Single Rectangular Fillet Blind Pocket



(b) Double Rectangular Fillet Blind Pocket



(c) Triple Rectangular Fillet Blind Pocket

Fig. 4. 3D CAD Models automatic graph plotting using CPM 3D Plotter (a) Single Rectangular Fillet Blind Pocket (b) Double Rectangular Fillet Blind Pocket (c) Triple Rectangular Fillet Blind Pocket

6. Conclusion

Feature recognition plays a key element in achieving the integration of design and manufacturing. The methodology suggested in this study is a practical approach to recognising the machining features of blind pockets with different numbers of rectangular cuts from the STEP file. Although many reviewers have dealt with feature recognition methodologies, it is though needing development over the preceding methodologies appearing in the literature. The STEP file has the proficiency to produce a good and generic representation of the simple and compound merchandise data in that the feature, geometry, topology & manufacture data are associated. The geometric string hierarchy of the STEP file for these cases will be developed for future work.

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