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# GEOMETRIC DATA EXTRACTION METHODOLOGY FROM STEP FILE FOR CYLINDRICAL PARTS

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## ABSTRACT

ISO STEP (Standard for the Exchange of Product model data) files are widely used in the manufacturing industry to exchange 3D design data. Extracting features from these STEP files can be a powerful tool for various applications, such as computer-aided design (CAD) and computer-aided manufacturing (CAM). The extraction of features from ISO STEP files can facilitate various tasks including automatic recognition of standard components, identification of geometric relationships, and generation of interactive visualizations. This enables users to streamline their design processes, improve collaboration, and enhance overall productivity. Hence, data extraction from STEP-file is becoming essential for automated manufacturing of parts particularly cylindrical ones. The paper presents methodology to extract all the related entities, vertices and cartesian points of the cylindrical parts. An integrated computer package is developed to analyze and extract the geometrical data that define parts' feature for cylindrical parts. A methodology to extract all the geometrical details of the parts stored as STEP file AP203 is proposed. To check the suitability of the proposed methodology and validate the computer package, several cases covering all the features that may exist in any cylindrical parts were considered. The studied cases were created using Inventor package version 2023. A software package written by Microsoft Visual Studio 2022 - C# is designed, tested, and applied to the considered cases. The system proved capable of getting all the geometrical details for internal and external surfaces of straight cylinders, tapered cylinders, conical surfaces, recesses, fillets, internal drill, Chamfering and undercut. Several case studies were considered. Some are given in this paper. The obtained results verified the success of the developed software.

**KEYWORDS**: STEP (10303-21), Feature recognition, Microsoft C# -2022 programming, CAD files

# إستخلاص البيانات الهندسية للأجزاء الأسطوانية من ملف STEP

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**الملخص**

تستخدم ملفات ISO STEP على نطاق واسع في عمليات التصنيع لتبادل بيانات التصميم ثلاثي الأبعاد. يعد استخراج البيانات من ملفات STEP أداة قوية لتطبيقات مختلفة، مثل التصميم المدعوم بالحاسوب (CAD) والتصنيع المدعوم بالحاسوب(CAM) . استخلاص بيانات التصميم من ملفات STEP يمكن أن يسهل أداء مهام مختلفة مثل التعرف التلقائي على المكونات القياسية، وتحديد العلاقات الهندسية، وإنشاء تصورات تفاعلية. مما يتيح للمستخدمين تبسيط عمليات التصميم الخاصة بهم، وتحسين التعاون، وزيادة الإنتاجية. ومن هنا يصبح استخراج البيانات من ملف STEP أمرًا أساسيًا لتصنيع الأجزاء تلقائيًا خاصة الأسطوانية. تقدم هذه الورقة البحثية منهجية لاستخراج جميع الكيانات ذات الصلة، والنقاط الرئيسية، والنقاط الكارتيسية للأجزاء الأسطوانية. ولتحقيق هذا الهدف فقد تم تطوير حزمة حاسوب متكاملة لتحليل واستخراج البيانات الهندسية التي تحدد ميزات الأجزاء للأجزاء الأسطوانية.

تم اقتراح منهجية وتصميم برنامج حاسب لاستخراج جميع التفاصيل الهندسية للأجزاء المخزنة كملف (STEP AP203) .وللتحقق من ملاءمة الطريقة المقترحة والتحقق من صحة الحزمة الحاسوبية، تم دراسة العديد من الحالات التي تم إنشائها باستخدام حزمة Inventor الإصدار 2023. تم تصميم وكتابة الحزمة الحاسوبية المستخدمة في البحث بواسطة Microsoft Visual Studio 2022 - C#، وتم اختبارها وتطبيقها على العديد من الحالات.

أثبت النظام المقترح قدرته على الحصول على جميع التفاصيل الهندسية للأسطح الداخلية والخارجية للأسطوانات المستقيمة، والأسطوانات المدببة، والأسطح المخروطية، والتجاويف، والتقوسات، والحفر الداخلي، والتشطيب الزاوي، والتقليم السفلي. تمت مراجعة العديد من دراسات الحالة. تم تقديم بعضها في هذه الورقة. أكدت نتائج الدراسة نجاح البرنامج المقترح.

**الكلمات المفتاحية:** ملفات (STEP 10303-21)، استخلاص البيانات الهندسية، ملفات التصميم بمساعدة الحاسب (CAD)، مايكروسوفت C# إصدار2022.

# 1. INTRODUCTION

Extracting features from these STEP files can be a powerful tool for various applications, such as computer-aided design (CAD) and computer-aided manufacturing (CAM). The process of feature extraction from ISO STEP files involves identifying and extracting specific geometric information from the 3D models represented in these files. This method gives engineers and designers the capability to analyze and manipulate the data in a more intuitive and efficient manner. Exploring improvements in recognizing features in computer-integrated manufacturing, a thorough look at important research; show how things are changing.

In the 21st. century global market, innovation will be the key to future success. Traditionally, the efficient design and manufacture of a component has required design engineers to have an effective knowledge of the manufacturing process to be used. This has meant that the design, whilst satisfying the function and operating standards of the service requirements has also been constrained by the level and spread of process knowledge/industry contacts which the individual designer/design group had acquired. Once a design has been accepted, the ability of the manufacturer to influence the previously made design decisions to allow ease of production is always limited to small modifications. At this stage prospects of total process route changes are normally considered impractical [1].

Nowadays all aspects of life involve information that is stored digitally as data. In the industry or privately, data is stored as files on hard- drives either locally, on servers, or dispersed in cloud systems. Standard file formats simplify the re-use of data by providing portability that enables data file exchange and database sharing. ISO STEP (Standard for the Exchange of Product model data) files are widely used in the manufacturing industry to exchange 3D design data [2]. The goal of STEP is to offer to the public a consistent suite of data definitions for all major engineering domains. Being consistent means that interoperability is not only possible within the same domain, but also between overlapping domains [3].

Sharing product design information with other downstream applications, such as process planning, is a major barrier to developing an integrated manufacturing system. Part of this shortcoming is due to the difference in product data descriptions, since a design is geometry based, whereas process planning is manufacturing feature based. The implementation of automatic feature recognition (AFR) techniques is considered an indispensable concept for transferring product data between computer-aided design (CAD) and automatic computer-aided process planning (ACAPP). This is accomplished using one of the international Product Data Exchange standards, such as DXF, IGES, or STEP files. Despite different AFR techniques and systems have been developed to serve this aim, each of them has limitations. The most important limitation is that each system is restricted to a specific set of pre-defined manufacturing features. This means that even when the system tries to cover as many as possible of the existing features that are predefined, it is always possible to create a new feature based on the specific requirements and designer creativity. Consequently, the new feature is not included in the compiled database and, hence, will not be recognized [4].

Automated tools which can understand and interface with CAD models are of significant research interest due to the potential for improving efficiency in manufacturing processes. At present, most research into the use of artificial intelligence to interpret three-dimensional data takes input in the form of multiple two-dimensional images of the object or in the form of three-dimensional grids of voxels. The transformation of the input data necessary for these approaches inevitably leads to some loss of information and limitations of resolution. Existing research into the direct analysis of model files in STEP format tends to follow a rule-based approach to analyze models of a certain type, resulting in algorithms without the benefits of flexibility and complex understanding which artificial intelligence can provide [5].

Direct information transfer using feature technology from CAD to CAPP will reduce many problems. But most of the existing CAD system does not provide part feature information. In other words, CAPP systems do not understand the three-dimensional geometry of the designed parts from CAD systems in term of their engineering meaning related to manufacturing or/and assembly. To solve the CAD and CAPP interface problem, feature recognition is one the most efficient approaches. ISO 10303 is an ISO standard for the computer interpretable representation and exchange of product manufacturing information [6]. To enhance the interoperability and integration of CAD/CAM system, a generic feature recognition system will be needed incompliant with (STEP) neutral file for seamless manufacturing. A methodology to recognize manufacturing features of prismatic parts from ISO 10303 STEP file is essential for manufacturing processes [7].

Intelligent manufacturing systems of the future produce small lot sizes with high flexibility, which require quick and automated decisions. The key to the system integration is the product data exchange among various systems, which use a simplified and generalized methodology of extracting manufacturing feature information from STEP files. The extracted data can be used to generate controller dependent NC codes [8].

A CAD/CAM system has become a dominant system in modern production techniques. In the attempts to facilitate the transfer of design data and geometry between the huge varieties of the international working CAD/CAM systems, the STEP file format is introduced (ISO STEP AP-203). Extraction of the entities is the first step in automated programming either for process planning, product classification or in programming for Numerical Control Machine Tools [9].

The efficient execution of process planning activities requires knowledge from several distinct domains. A good concept provides the manufacturing personnel with a method for comparing the newly designed part with a pool of validated models to identify the most similar one. Each previous CAD part model is linked with the necessary manufacturing information, so that the initial values for the manufacturing process are available without a time-consuming testing phase. The method is divided into three main steps: the global similarity comparison, the segmentation of the part and the local similarity comparison. In every cluster, the segmented geometries are again compared for similarity. The combination of the first and the second ranking results in a global similarity hierarchy for the newly designed part [10].

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, the most widely used is the STEP file format [11]. The emerging paradigm of Agile Manufacturing has imposed additional requirements of neutral format, so that form-feature information can be readily shared among multiple partners of a virtual enterprise. Recently, the STEP format has emerged as the means for neutral form exchange of product related data. The STEP efforts have broken down the domain of manufacturing related activities in the form of application protocols [12].

Automated feature recognition (AFR) is considered a critical node for integration of CAD/CAPP/CAM. The CAPP module requires the implementation of a feature recognition procedure, so that decisions relating to steps of process planning activities can be made automatic. STEP file information used to generate manufacturing parameters becomes an important task [13]. Existing AFR methods consider feature geometries, and the recognized machining features have limited process attributes for process planning. With the model-based definition becoming more advanced in the industry, AFR methods need to be improved and upgraded. The feature recognition results can be directly applied for downstream machining process planning [14].

Machining feature recognition is a key task in the intelligent analysis of 3D CAD models as it represents a bridge between a part design and the manufacturing processes required for manufacture and can, therefore, increase automation in the manufacturing process. As 3D model files do not naturally conform to the fixed size necessary as the input to most varieties of neural network, most existing solutions for machining feature recognition rely on either transforming CAD models into a fixed shape representation, accepting some loss of information in the process, or employ rigid rules-based feature extraction techniques prior to applying any learning-based algorithm, resulting in solutions which may display high performance for specific applications but which lack in the flexibility provided by a purely learning-based approach [15].

The current study aims to design, build, and evaluate a software system capable of reading and analysing STEP file format to extract all features of cylindrical parts in oreder to build a system capable for CAPP for turning machines.

## 2. Methodology

The arrangement by which the STEP file was designed is unique. It consists of two main sections. The first section contains general remarks about the creation of the file and other information which is not related to the technical data. The second section contains all the necessary geometrical and technical details. The detailed structure of the technical section is illustrated in **Fig.1**.

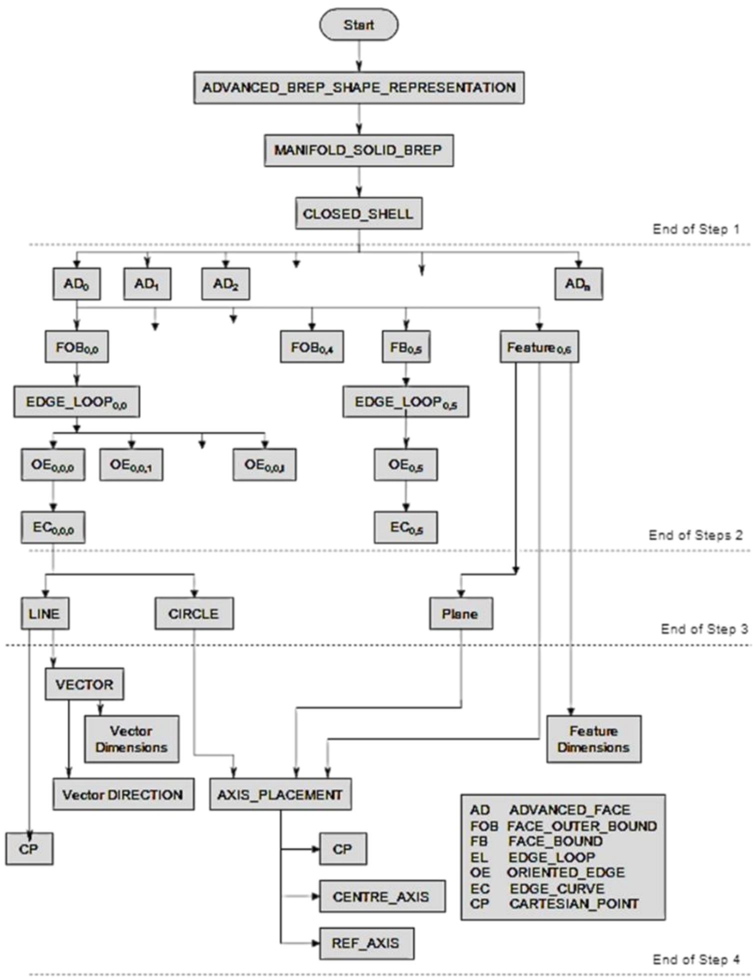


**Fig 1.** Structure of the STEP File.

The proposed methodology for the extraction of the technical data is suggested to be as follows:

- After reading the text file, digitizing it to words and saving the words in an array, the first step is followed by searching for the block containing the "ADVANCED_BREP_SHAPE_REPRESENTATION" which leads to "MANIFOLD_SOLID_BREP" and later to "SOLID_SHELL."
- The number of features is found from the "ADVANCED_FACE" together with both the "FACE_OUTER_BOUND" and the "FACE_BOUND," **Fig. 2** and **Fig. 3**.

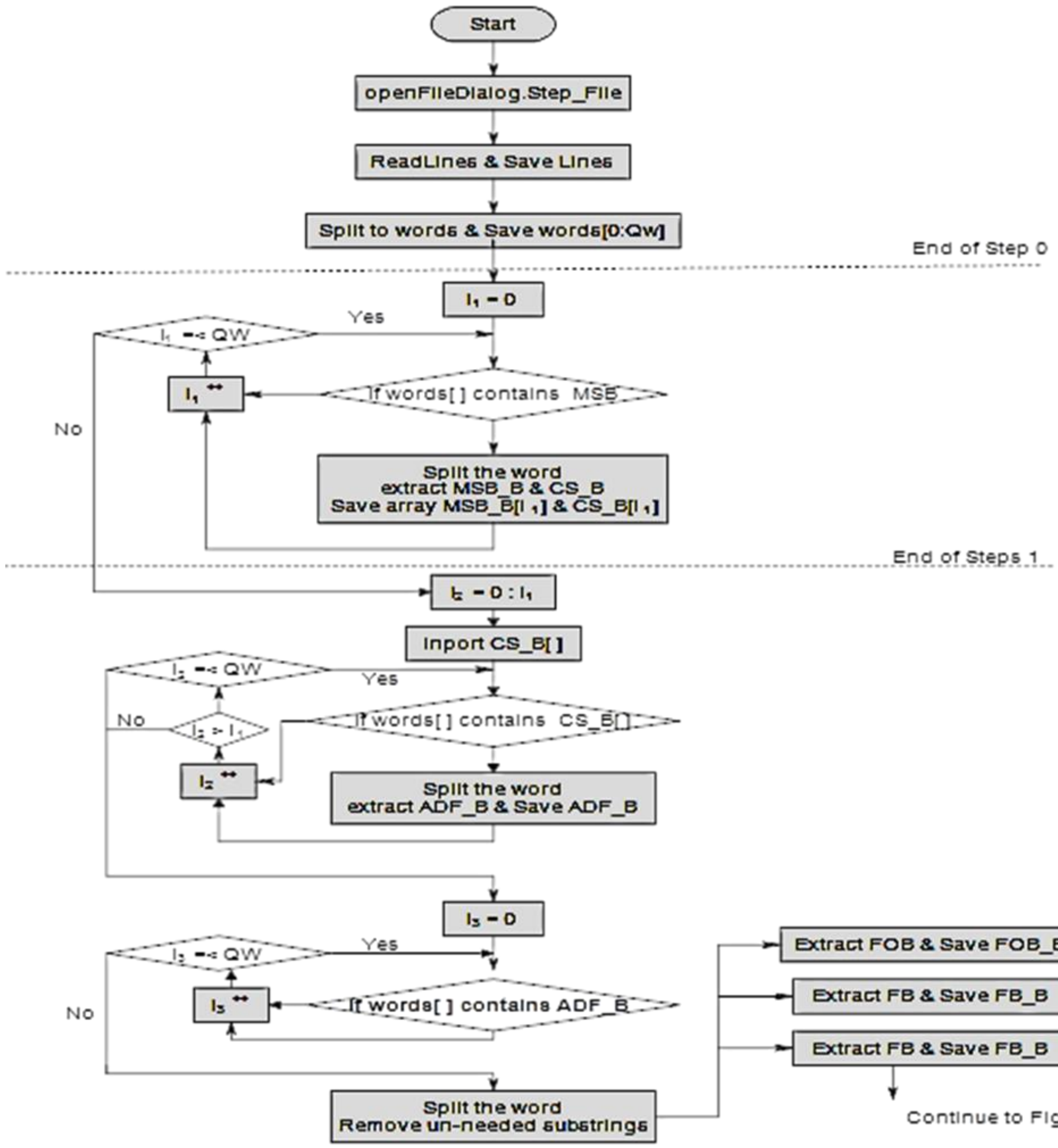


**Fig 2.** Software algorithm1.

The detailed entities of the edge loops, edge curves down to the geometrical elements of the types of features with their identification geometrical elements of circles, planes, lines, vertex points and other Cartesian points. A Computer package written in Sharp C "C#" – Version 2022 is designed giving the part shape and the full detail of the extracted data.

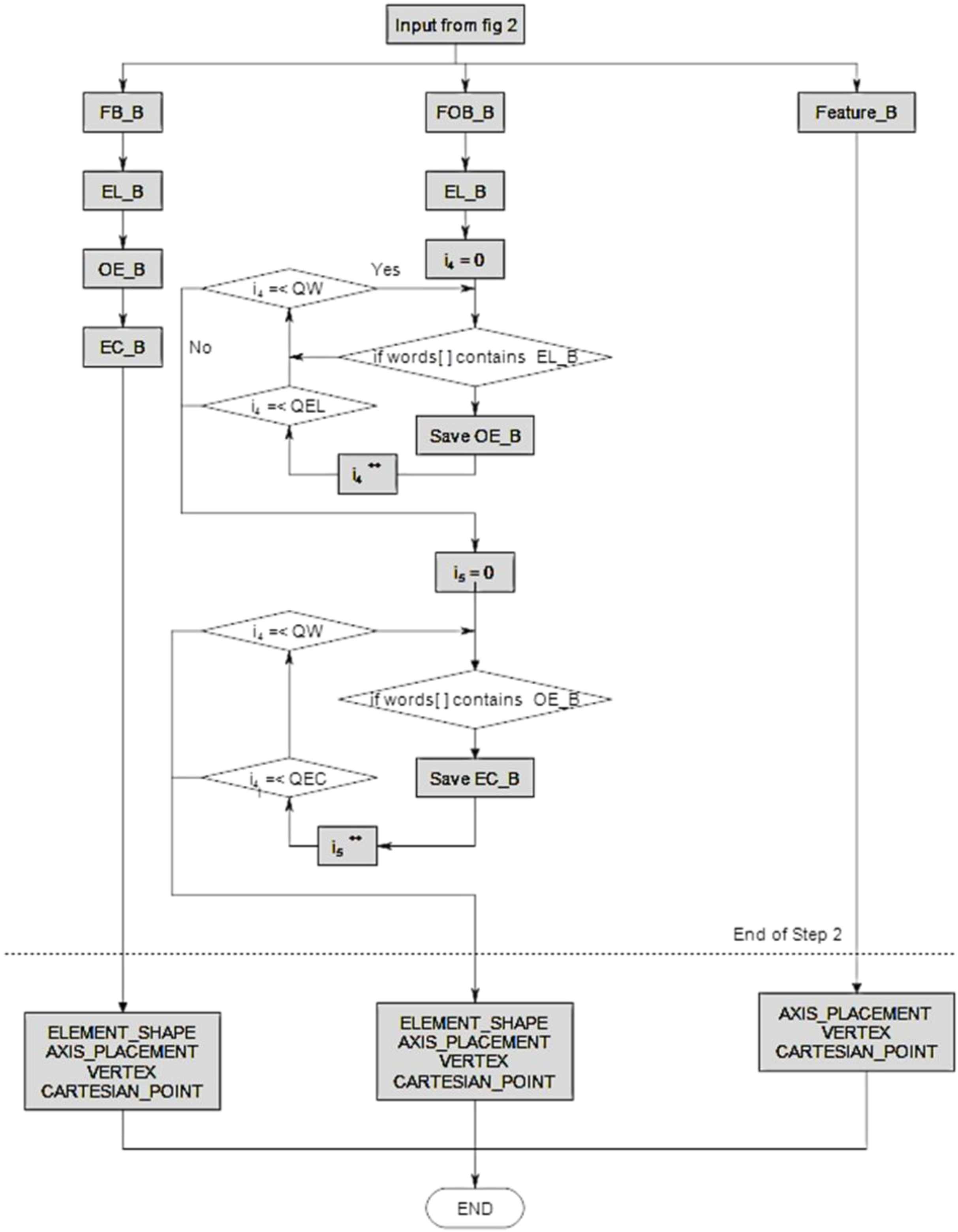


**Fig 3.** Software algorithm2.

## 3. RESULTS AND DISCUSSION

To check the suitability of the proposed methodology and validate the computer package several cases were considered. **Fig 4** shows one such cases which was created using Inventor package version 2023. The part was meant to have almost all the cylindrical features.

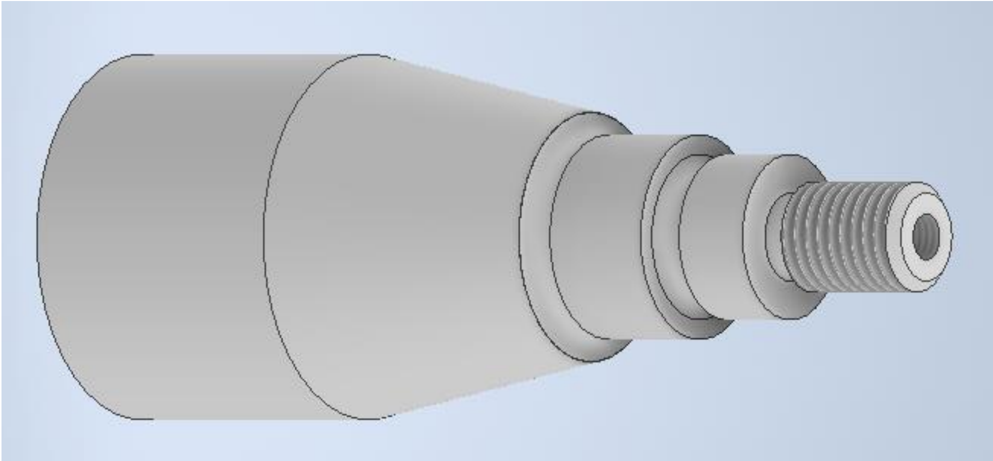

**Fig 4:** Case study (Straight Cylindrical + External Taper + Undercut + Threaded part)

**Fig. 5** shows a screen shot of the designed interface with all the extracted data.

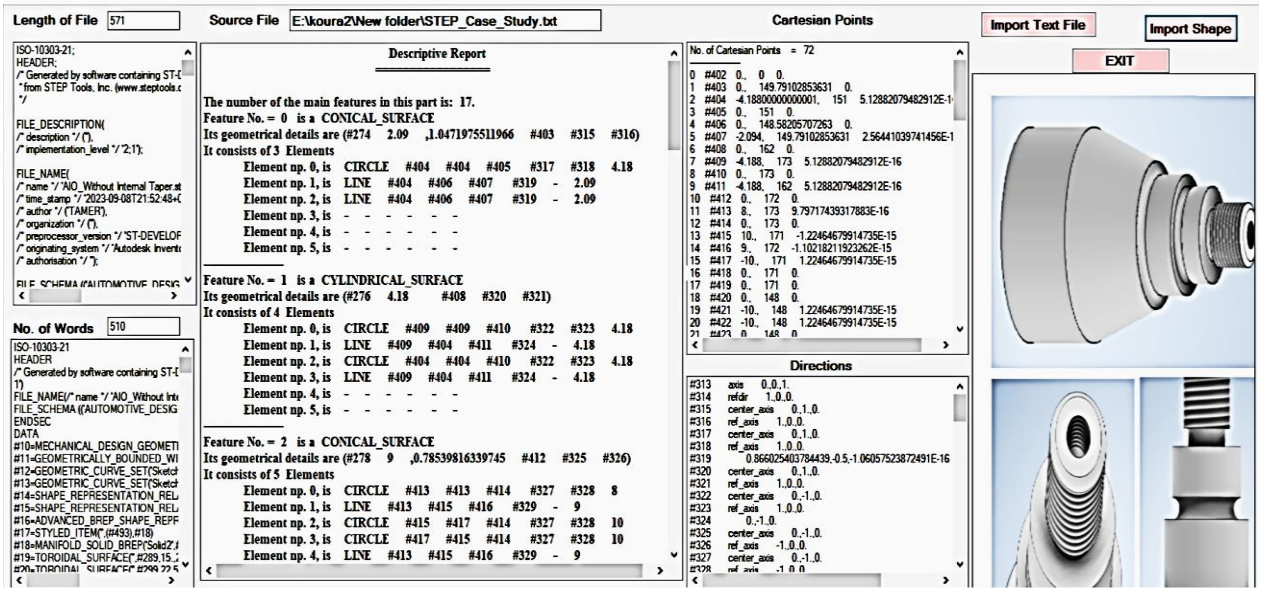


**Fig 5.** Screenshot of the Interface.

The interface contains six main groups. The first lists the STEP file as imported from the STEP-CAD File. The second group lists the extracted words of the file as obtained from the output of step "0" in the algorithm shown in **Fig. 2**. The third group gives a full descriptive report of all the features with the related geometrical data and their element components. In the figure, the first three features are shown with the rest of the features can be seen through scrolling up and down. The boundary elements for each feature (circles and lines) are shown together with their geometrical dimensions. The directions of the "ref_axis" and "center_axis," the vertices and the Cartesian points are shown in groups four and five. The six groups show an image of the part.

## Conclusions

In this paper, a methodology to extract all the geometrical details of the parts stored as STEP file AP203 is proposed. A software package written by Microsoft Visual Studio V2022 - C# is designed, tested, and applied to several cases. The system proved capable of getting all the geometrical details for internal and external surfaces of straight cylinders, tapered cylinders, conical surfaces, recesses, fillets, internal drill, Chamfering and undercut.

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