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Using semantic Geometric Dimensioning and Tolerancing (GD&T) information from STEP AP242 neutral exchange files for robotic applications

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Abstract

This article proposes the use of Geometric Dimensioning and Tolerancing (GD&T) information from STEP AP242 for robotic manufacturing applications. This information can be directly added to the relevant features of the 3D model as per Model Based Definition methodology during the design phase of the product life cycle. STEP AP242 neutral exchange files enable the availability of product definition at the downstream operations, thus completing the Digital Thread as part of Industry 4.0 practices. This article discusses two methods; using custom Unicode strings and the standard entities; for including GD&T using the STEP AP242 Edition 2 neutral file exchange format. A method to form a single Unicode string to add all the GD&T information to the STEP files is described in this paper. The GD&T information in the Unicode string is fully semantic and can easily be parsed to extract the relevant PMI for tolerance analysis. A novel process of extracting and interpreting the relevant PMI for robotic manufacturing applications is described in detail. This article discusses various applications of this information for robotic manufacturing through two use-cases using the second edition of the STEP AP242 standard.

Keywords Robot · Assembly · Welding · STEP AP242 · ISO 10303 · Model based definition · Product manufacturing information · PMI · Geometric dimensioning and tolerancing · GD&T

Abbreviations

AM	Application Module
AP	Application Protocol
API	Application Programming Interface
ASME	American Society of Mechanical Engineers
ATI	Advanced Technologies for Industry
AWS	American Welding Society
CAD	Computer Aided Design
DT	Digital Thread
FCF	Feature Control Frame
GD&T	Geometric Dimensioning and Tolerancing
ISO	International Standards Organisation
JSDAI	Java Standard Data Access Interface
MBD	Model Based Definition
MBE	Model Based Enterprise
NC	Numerical Control
OEM	Original Equipment Manufacturer
PMI	Product and Manufacturing Information
QIF	Quality Information Framework
SME	Small and Medium Enterprise
STEP	STandard for Exchange of Product Data

1 Introduction

Robotics is one of the 16 Advanced Technologies for Industry (ATI) identified by the European Commission. The successful implementation of Industry 4.0 depends on the close integration of design and manufacturing by establishing a Digital Thread (DT) of product and process data which facilitates the automation of industrial operations. The use of industrial robots is increasing in automotive and electronics industries [1,2] much faster than in other industries. The main reason for this trend is that the product variation is low in these industries, and the production volumes are high. This reduces the cost of automation and increases productivity, thus providing more significant rewards for using robotics.

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The adoption of robotic automation in the case of Small and Medium Enterprises (SMEs) is less than half compared to that of large enterprises [3]. Generally, SMEs handle a variety of products in small volumes, which necessitates changing the robot program every time the product changes. This increases the cost of automation and reduces the production time, hence less use of robots in SMEs. The time and cost of robot programming can be reduced by using the product information from the design phase for automated programming. The establishment of a DT enables product data sharing to the downstream manufacturing operations. Some robot programming research use CAD software to define the motions relative to the product design such as Autopass [4], HighLap [5,6], Archimedes 2 [7], or more recently the work of Perzylo et al. [8], and Pane et al. [9]. The CAD data is used to associate end-effector motions or control strategies based on the relative geometric dimensions in the design. Mohammed et al. [10] used the assembly constraints from STEP AP242 files for robotic assembly operations to exemplify a direct connection between robot programming and the neutral exchange format.

Another aspect that limits the use of industrial robots is uncertainty. The uncertainties can be grouped as product, manipulator, and environment or work-cell uncertainties. The effects of these uncertainties can be mitigated by calibration of robots, using sensors, active or passive compliance, and careful placement of parts and robots in the work-cell. These approaches require process-specific tuning and calibration, often stemming from the experiential knowledge of the process planner or automation provider, or are arrived at by rigorous experimentation with real parts. This represents a disconnect between the design process and the downstream manufacturing process.

Traditionally the product uncertainties are handled by specifying the tolerance limits. The designers design the products to ideal size and shape to perform their function. However, it is impossible to manufacture the parts to exact nominal dimensions due to the limitations in manufacturing processes, as noted by Srinivasan [11]. Hence the designers specify some deviation from the nominal dimensions within which the product can perform its intended functions. These deviations from the nominal dimensions are known as part tolerances. As the part tolerances become tighter, the manufacturing cost increases, and if the tolerances become loose, the part may not function as expected.

While the tolerances on dimensions allow for variation in size, the Geometric Dimensioning and Tolerancing (GD&T) captures part variations in the size, form, and shape. GD&T is an important aspect of the Product and Manufacturing Information (PMI) which captures non-geometric product data. The variations in the individual parts affect the overall product assembly and the assembly process itself. These variations should be considered in robotic assembly for the planning, programming, and successful completion of the assembly tasks.

Proctor et al. [12] suggested the use of GD&T information for robotic manufacturing. They suggested that the accuracy of manipulators can be increased by using part tolerances for robotic path planning. It was also suggested that the GD&T information could be used for robot selection and part placement in work cells. Proctor et al. [13] used semantic GD&T and Quality Information Framework (QIF) to handle the uncertainty in robotic manufacturing and assembly operations.

This paper follows up on the idea by Proctor et al. [13] and discusses the use of GD&T information directly from the neutral CAD files as input to define robotic tasks. The STEP AP242 Ed2 file format is used for this purpose. The contributions of this paper are:

1. A proposal for the use of GD&T from design phase for robotic automation of assembly and welding operations,
2. A description of two methods to semantically add all the GD&T information to the STEP AP242 files.
3. A description of the process of extracting the relevant PMI for robotic applications.
4. A formation of a single Unicode string to capture the dimension, datums, Feature Control Frames, and other related PMI.
5. A presentation of use-cases where this GD&T information can be used for robotic applications.

This paper is structured as follows. Section 2 presents the basics of robot programming and the relevance of GD&T for robotic assembly and welding. Model-Based Definition (MBD) is presented in Sect. 3. Section 4 describes the STEP AP242 standard briefly. Section 5 explains the concepts of geometric dimensioning and tolerancing and its effects on assembly. Section 6 describes the methods of including the GD&T information in STEP AP242 files using custom Unicode string and using the entities defined in the STEP AP242 standard. Section 7 presents various use-cases for the applications of tolerancing information from STEP AP242 for robotic manufacturing and assembly processes. The concluding discussion is in Sect. 8.

2 Robot programming

The product data from annotated 3D CAD models can be used for robot programming and control for various manufacturing tasks like assembly and welding. The three traditional methods of robot programming are (1) teach-pendant programming, (2) offline programming, and (3) programming by demonstration [14]. As robots have high repeatability, both teach-pendant programming, where the technician jogs the robot to the desired positions, and programming by demonstration, where the technician physically guides the robot through the process, utilize the repeatability of modern industrial robots [15]. Most industrial robots are capable of repeating a motion with high accuracy. This is because the errors caused by varying end-effector payload, inverse kinematics deviations, or link and gear flex are sufficiently deterministic [16–18]. For offline programming, the precise calibration of both the robot and its environment is required to keep a direct correspondence between the programmed scenario and the physical setup [19]. In both cases, the repeatability and calibration errors can be handled using sensor-feedback control strategies.

Procter et al. [13] describe a method to combine and evaluate the uncertainty of the robot kinematics and the GD&T information to make better choices regarding which robot hardware to use and to improve or verify motion plans. Pane et al. [9] describe extracting sensor-feedback control strategies from CAD constraints and suggest inferring control gains based on GD&T information. These are examples of how a direct connection between the design process and programming of the automation process can be of value. The GD&T information can also potentially serve a purpose during the deployment of an automation process by evaluating the source of failures during production. For production lines with various part designs, providing information about which features the robot is expected to have interacted with during a failure, and their GD&T information can allow for faster failure-recovery and evaluation of the source of the error.

In many SMEs, a different company often provides robotic automation solutions that may use different CAD vendors and software solutions than the SME itself. This means that the GD&T information should be provided in a neutral exchange format to facilitate its usage in robotic automation solutions.

2.1 Relevance of GD&T in robotic assembly

The two essential parameters of any insertion operation are clearance and insertion length. Clearance is also represented as the clearance ratio, and it is defined as the ratio of the clearance to hole diameter [20]:

$$r_c = (d_h - d_p)/d_h \tag{1}$$

where r_c is the clearance ratio, d_h is the hole diameter, and d_p is the peg diameter.

Part clearances, clearance ratio, and insertion lengths can be used to estimate the contact states and force needed to complete the assembly. The clearance ratio also determines the maximum permissible tilt during the assembly process. Simunovic [21] used the part dimensions and insertion lengths to estimate the type of contact and insertion forces during assembly. Whitney [20] used the clearance ratio and insertion lengths to estimate the contact states, forces and jamming during peg-in-hole assembly. Clearance also determines the maximum tilt and the conditions of jamming in a chamferless assembly as described by Haskiya et al. [22]. The applicability of contact and force models also depends on the part clearances [23].

Part clearances also determine the suitability of the robotic manipulator and the success of the assembly operation. It also determines the control methodology needed for completion of the assembly [24,25]. ElMarghy et al. [26] discussed the effects of part dimensions and tolerances along with manipulator accuracy and repeatability on the success of assembly operation. As the uncertainty-to-clearance ratio increases, the completion of assembly becomes difficult with position control alone [27,28]. Hence depending on the clearance, an appropriate control strategy can be selected [29]. As the clearance ratio decreases, the success of assembly decreased [28,30].

The extraction of part dimensions and tolerances from the STEP files is used to calculate the effective clearance, which can be used to estimate the contact states and possibility of jamming; the assembly forces needed; decide the suitability of a manipulator for the assembly task; and to decide the control strategy.

Once the toleranced features and the tolerance values are extracted, these values are used to calculate the variation of these features from the nominal values. These values are used for defining the constraints for the robotic assembly tasks. Automated tolerance analysis programs can calculate the variation of toleranced features and their propagation in the entire assembly. Effective (worst possible) clearances between mating parts can be evaluated using these tolerance values. Considering the size tolerances, the effective clearance can be calculated as [26]

$$c_e = ((d_h + t_h) - (d_p + t_p))/2 \tag{2}$$

where c_e is the effective clearance, t_h is the tolerance on hole diameter, and t_p is the tolerance on peg diameter. Using Monte-Carlo simulations and other automated computer programs, the tolerance information can be analyzed to describe the clearances that can be expected.

Many designs have multiple features that can come into contact during the assembly operation. Providing the complete design with PMI makes it possible to evaluate the assembly process to determine which features are relevant to the process. For robot programs, the change in relevant features can require a change in control strategy required depending on the tolerance values.

2.2 Relevance of GD&T in robotic welding

PMI in general will be very useful in the automation of welding process. The PMI along with product geometry can be used for automatic programming of welding robots. This will decrease the dependence on costly and time consuming manual programming and increases its adoption in SMEs where changing product design requires frequent changes in robot programs. Mohammed et al. [31] discussed the importance of PMI in robotic welding and described the reuse of welding information for STEP AP242 files. In addition to welding annotations, GD&T information will also be useful in automated robotic welding.

The quality and the success of the weld depends on the root gap and the fit–up of the components being welded. These two parameters depend on the seam geometry, edge preparation and material thickness. The geometric variations of the parts being welded also effect the seam position, orientation and geometry, thus affecting the weld path, positioning and orientations of weld gun. The effects of these variations are limited by specifying various tolerances. The tolerances that impact the weld quality can be divided into two groups.

- Groove tolerances: The tolerances specified for edge preparation are considered as groove tolerances. These are applied to root gap, bevel angles and root face. *AWS D1.1 Structural Welding Code – Steel* gives the tolerances for welding of steel structures. *ASME B16.25–2017: Buttwelding ends* standard specifies the edge preparation and tolerances for buttwelding of piping components. These tolerances can be easily included in the Unicode welding annotations described by Mohammed et al. [31].
- General tolerances: The tolerances specified on the welding components that affect the seam geometry, position and fit–up are considered in this group. The standard *ISO 13920:1996 – Welding – General tolerances for welded constructions – Dimensions for lengths and angles – Shape and position* specifies the size and positional tolerances for welded structures.

In manual welding, the welder senses the variations and errors and makes adjustments to successfully weld the parts. The product variations should be taken into account in robotic welding processes to avoid product quality issues and rejections. Generally in robotic welding, sensors are used to scan and extract the seam and groove geometry [32]. The sensing operation is planned based on the human input. This can be improved using the information from the CAD files. This paper demonstrates how the groove tolerances can be included in the welding annotations.

3 Model based definition (MBD)

The designers define a product and create 3D product models using commercial CAD software. Then the 2D manufacturing drawings are prepared from the 3D CAD models by adding the necessary PMI. Therefore, the 2D manufacturing drawing becomes the master and source of product information, and these drawings are the input to the downstream manufacturing operations. Consumption of information from 2D drawings at downstream operations requires manual intervention to identify and recreate the necessary data for that specific operation. Recreating data at each downstream operation is time-consuming and may introduce human errors. This involvement may lead to product quality issues and delays as a result of miscommunication between the design and the manufacturing teams. These problems can be avoided by reusing the product data from the design phase in the downstream manufacturing operations. The manual recreation of information from 2D drawings is eliminated by adopting a Model Based Definition (MBD) methodology.

In MBD, the PMI is semantically added to the relevant features of the 3D CAD model during the design phase. Then the 3D model carries the critical dimensions, GD&T, surface finish, and other needed information along with the product geometry, making it the master of product information [33,34]. Figure 1 shows a 3D CAD model with all the PMI attached to its features. These annotated 3D models are the basis of connected DT in a Model Based Enterprise (MBE). The product information will be available to the downstream operations for direct use like NC machining and automated inspection [35]. Many contemporary applications of MBD are mentioned by Goher et al. [36].

The MBD created by the designers is shared with the downstream operators and subcontractors. Designers use commercial CAD software to prepare the 3D models and MBDs. Sharing the MBDs in native CAD formats is not feasible and is not a good practice as this requires the same commercial CAD software. Not all the stakeholders have access to the CAD software, nor are they knowledgeable about using that software. Most subcontractors are SMEs who serve various original equipment manufacturers (OEMs). They cannot purchase the licenses of all the CAD software used by different OEMs. Hence a neutral exchange format is needed, facilitating the sharing of product designs to all stakeholders avoiding the dependency on commercial CAD software in the downstream operations. STEP AP242 is such a neutral exchange file format that can carry both the product geometry and PMI. The following section describes this exchange file format.

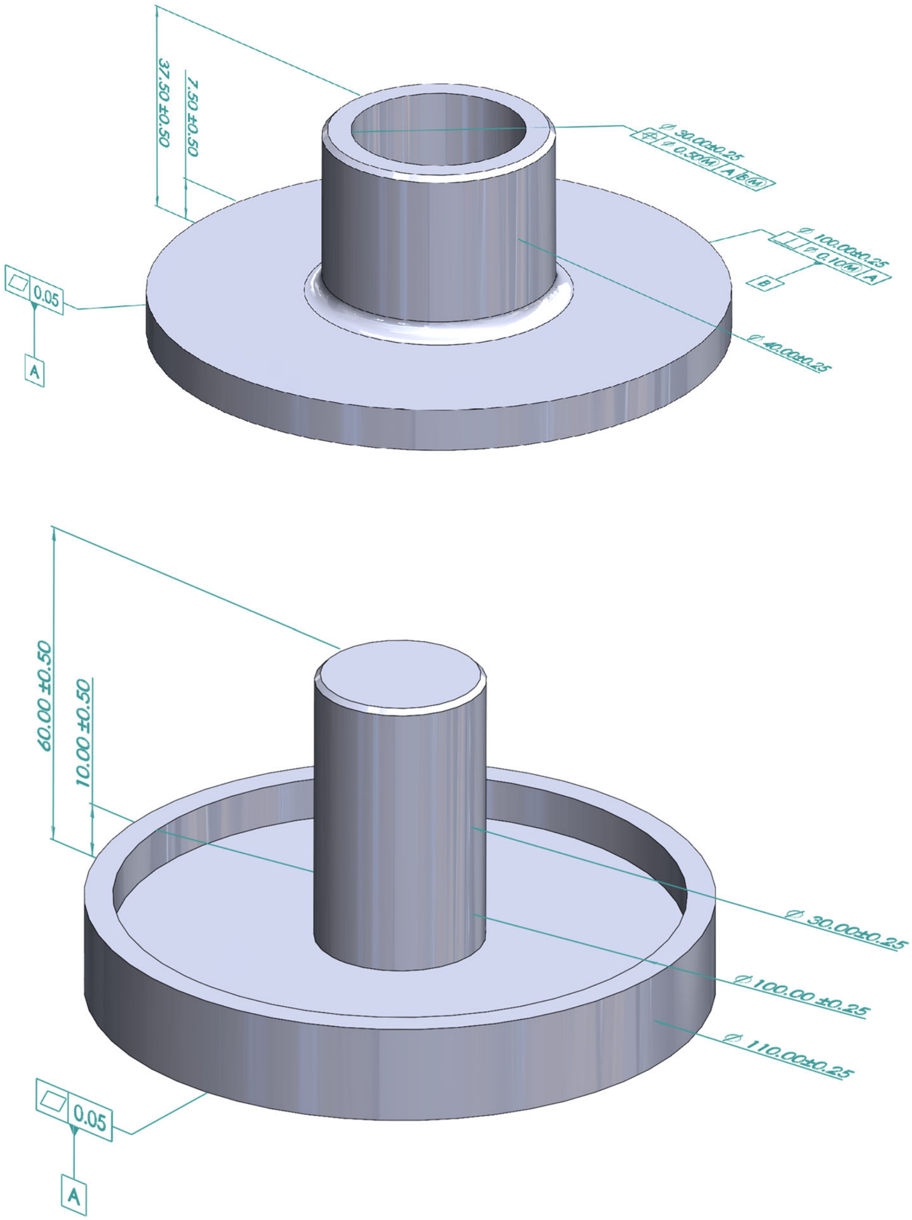

Fig. 1 Parts with GD&T annotations

4 STEP AP242

STandard for the Exchange of Product Model Data (STEP) is one of the industry's most widely used neutral file formats. STEP AP242 is part of a family of ISO (International Standards Organization) standards *10303: Automation systems and integration – Product data representation and exchange*. The *AP242: Managed model–based 3D engineering* is one of the latest application protocols that deals with 3D semantic PMI [37]. The latest (second) edition of this Application Protocol (AP) was released in 2020 [38]. STEP AP242 replaced AP203 (Configuration controlled 3D designs of mechanical parts and assemblies) and AP214 (Core data for automotive mechanical design processes). The AP203 and AP214 were developed for aerospace and defense and automotive industries respectively [39,40]. Using STEP AP242, MBD can be created as per *ASME Y14.41–2019: Digital Product Definition Data Practices* [41]. The use of STEP AP242 in the industry increases with increasing digitization. Many researchers are working on the capabilities of STEP AP242 and its applications in industry like automated tolerance analysis [42], CAD/CAE integration, and smart manufacturing [43–45].

STEP AP242 allows the addition of PMI both semantically and non-semantically. The non-semantic addition of PMI is for presentation purposes and not relevant for robot programming, and only semantic elements are considered for PMI addition and extraction for constraint definition. The following information is available in STEP AP242 files.

Product Geometry The exact product geometry is translated into STEP, and it can be readily extracted and used to identify part features.

Product Structure and Assembly Information STEP AP242 files also carry the product structure in terms of the constituent parts of the assembly and their relative positions. Using the Mating Capability Domain Model of STEP AP242 Ed.2, mating and joint type can also be included in the assembly files.

Critical Dimensions and GD&T The dimensions critical for quality and the GD&T can be semantically added to the part features in a STEP AP242 file. These can be used to extract the part features' sizes and locations and estimate possible deviation from the nominal design values.

Annotations Other PMI like surface finish, thread specifications, and welding symbols can be included in the STEP files in the form of annotations. Mohammed et al. [31] presented two different ways in which the welding annotations can be added to the STEP AP242 files.

Properties Product properties like material, weight, and envelope dimensions can be extracted from the STEP AP242 files. These can be used to estimate the automation of handling and packaging.

Notes Other relevant information can also be included in the MBD in the form of text annotations. These may include the special tools, fixtures, and essential process-related information which can be used to define additional constraints on the manipulator.

4.1 Structure of STEP files

A STEP file is a text-based file that encapsulates all the product details in entities. These files are called Part-21 files because this text encoding is done as per Part-21 of ISO 10303 standard [46]. These files are also known as physical files and have the extension '.STP' or '.STEP'. These files are human-readable and facilitate the writing, reading, and exchange of product data. The significant entities used for representing product geometric information are

Geometric Entities The sub-classes of 'geometric_representation_item' are considered as geometric entities.

Topological Entities The sub-classes of 'topological_representation_item' are considered as topological entities. There are some entities like 'vertex_point' and 'edge_curve' which are sub-classes of both 'geometric_representation_item' and 'topological_representation_item'. Such type of entities are also considered as topological entities.

Presentation Entities The entities from 'Presentation appearance' schema, along with the entities from other schemas of 'Visual appearance (Part: 46)' are used to define the graphical appearance of the model. These entities are considered as 'presentation entities'.

Representation Entities Entities from the schemas of 'Fundamentals of product description and support (Part: 41)' are considered as representation entities. These entities are used to identify the product and classify and establish relationships among products. Various sub-classes of 'representation' and 'representation_relationship' entities, which are used to establish relationships between different entities, are also included in this category.

Figure 2 shows the different types of entities and their relationships for a rectangular block. The geometric entities represent the basic geometric elements of product design like a Cartesian point, a line, or a plane. These entities are combined or referred to in topological entities to form the features of the product geometry like a vertex, an edge or a face of the product. The higher entities bind all the topological entities and represent the entire product, for example, the 'Mechanical_Design_Geometric_Presentation_Representation' entity.

Where possible, the annotations are attached to the topological entities to refer to the part features that are affected by these annotations. This is also suggested by the CAx–IF recommendation [47], where it recommends using the 'advanced_face' entity for part features while attaching the GD&T annotations. This helps identify the part features quickly and enhances the reuse of product data in the downstream operations.

5 Geometric Dimensioning and Tolerancing (GD&T)

The practice of GD&T is mainly based on the set of standards from ISO and *ASME Y14.5: Dimensioning and Tolerancing standard* [48]. The standard ISO 1101 [49] deals specifically with form, orientation, location, and run-out tolerances. As the part features deviate in form, orientation, location, and run-out from the ideal values, they affect the assembly process also.

Each of these tolerances defines a tolerance zone that limits the deviation of the part feature from its ideal shape, size, and position. The shape of the tolerance zone depends on the characteristics of tolerance, and the size depends on the tolerance value. Table 1 gives the shape of tolerance zones defined by the type of the tolerance.

Figure 3 shows a part with flatness tolerance and the resultant tolerance zone. The flatness tolerance on the top surface of the part means that all the points on that surface lie within the tolerance zone formed by two parallel planes, separated by a distance equal to the tolerance value. The effects of the tolerances vary depending on how they are attached to the feature.

Fig. 2 Structure of STEP file: geometric, topological and representation entities

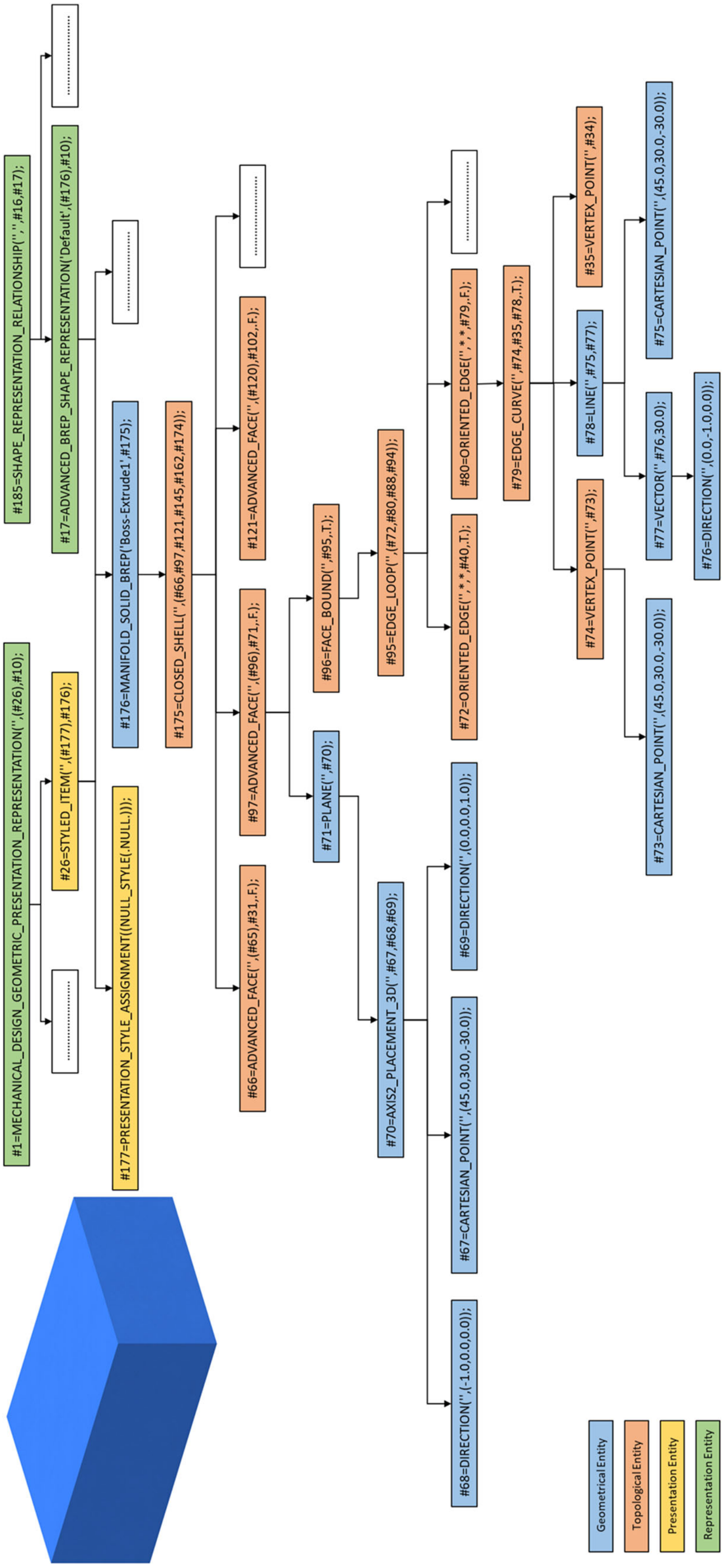

Table 1 Geometric tolerances and their tolerance zones

Type	Characteristic	Symbol	Modifier/element applied to	Tolerance zone
FORM	Straightness	⏤	None	Two parallel lines
	Flatness	⏥	None	Two parallel planes
	Circularity	○	None	Two concentric circles
	Cylindricity	⌭	None	Two concentric cylinders
PROFILE	Profile of a Line	⌒	None	Two parallel lines in the cross-section
	Profile of a Surface	⌓	None	3D Volume along the feature
ORIENTATION	Angularity	∠	None	Two parallel planes
			Diameter	Cylindrical
	Perpendicularity	⊥	None	Two parallel planes
			Diameter	Cylindrical
	Parallelism	//	None	Two parallel planes
			Diameter	Cylindrical
RUNOUT	Circular Runout	↗	Applied to a surface of revolution	Two concentric circles
			Applied to a surface perpendicular to the axis	Two circle of equal diameter separated along the axis by a distance equal to the tolerance value
	Total Runout	⌰	Applied to cylindrical surfaces	Two coaxial cylinders
			Applied to a perpendicular to the axis	Two parallel planes separated by distance equal to the tolerance value
LOCATION	Position	⌖	None	Two parallel planes
			Diameter	Cylindrical
	Concentricity\Coaxiality	◎	None	Cylindrical
	Symmetry	⌯	None	Two parallel planes

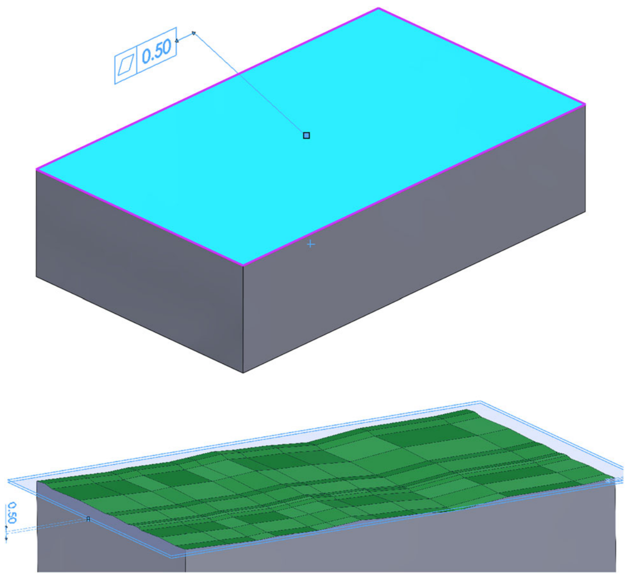

Fig. 3 Part with flatness tolerance and the resulting tolerance zone

1. If the Feature Control Frame (FCF) is attached directly to the feature, it applies to the feature surface.
2. If the FCF is attached to the dimension, it affects the feature's mid-plane, axis, or centerline.
3. For some characteristics like the profile of a line, if the FCF is attached to the surface or the edge of a surface in the perspective view, it is applied to the entire surface. But if it is attached to a line on a surface, it affects all the parallel line elements on that surface.

6 Including GD&T information in the STEP files

Most of the commercial CAD software support semantic annotation of GD&T in their native formats, even though there are some limitations [50]. Generally, the annotations do not retain their semantic property when saved as STEP AP242 files using most commercial CAD software. Except for two of the CAD software evaluated by Lipman and Filliben, all others need third-party converter tools to generate the STEP AP242 MBD files from the native CAD formats [51,52]. Below are some of the combinations in which the dimensions, FCFs, tolerances and datums can be represented on a 3D model:

1. Individual Dimension which can be basic, reference, or with tolerances. The tolerances can be unidirectional, bidirectional equilateral or bidirectional unequal.
2. Independent datum call outs
3. Independent FCFs
4. Dimensions with datum call outs
5. Dimensions with FCFs
6. Dimensions with both FCFs and datum call outs
7. FCFs with datum call outs

An example is shown in the annotations on the two components of a peg-hole assembly in Fig. 1.

This section describes two methods by which GD&T annotations can be semantically and directly added to the STEP AP242 files as per CAx recommendations [47,53]. JSDAI API [54], a Java based API for the Standard Data Access Interface [55], is used to edit the STEP AP242 files.

6.1 Using unicode text

The GD&T information and feature control frames can be included in the STEP files using Unicode strings based on the CAx recommendations of *PMI Unicode String Specification Examples and Mapping Strategies* [53]. All the GD&T symbols defined in ISO and ASME standards are represented using Unicode characters. Table 2 shows the Unicode and custom codes of GD&T symbols.

6.1.1 Forming the unicode string

A similar methodology is used as described by Mohammed et al. [31] in forming the Unicode string for GD&T annotation. Below are the key rules followed for this purpose.

1. The Unicode string starts with a six-letter combination ‘GDTXXX’. Here the first three letters ‘GDT’ stand for ‘Geometric Dimensioning and Tolerancing’.
2. The next three letters are used to indicate the dimensions, FCF, and datum callouts, respectively. The possible combinations of dimensions, FCFs and datum callouts are described in Sec. 6. Table 3 shows the letter codes used for this purpose. The letter ‘N’ is added in the corresponding location when any of these items are not present in the annotation.
3. Special symbols like all-round can be added after this six-letter combination using a region separator ‘\w’ (see Point- 8 below).
4. The Unicode characters are placed between ‘`\X2\`’ and ‘`\X0\`’.
5. For abbreviations, custom codes and identifiers, a 4–letter combination like ‘TXXX’ is used.
6. The first letter ‘T’ in the 4–letter combination denotes ‘Tolerancing’.
7. To make all the values semantic, each and every value is separated as opposed to the CAx recommendation [53]. ‘\u’ is used to separate different values.
8. ‘\w’ is used to separate different types and regions.

The flag notes are not included in the Unicode string, as they can be attached to the part features as separate text annotations.

Figure 4 shows a feature control frame for a position tolerance attached to a dimension with equal bilateral tolerances. The following Unicode string captures the information in this annotation formed using the methodology described in this section.

```
’’GDTEFN\w\w\u\X2\2300\X0\\u30.00\uTPOM
\u0.25\w\u\X2\2316\X0\
\u\X2\2300\X0\\u0.50\u\X2\24C2\X0\\uA
\uB\u\X2\24C2\X0\’’.
```

This Unicode string is added to the STEP AP242 files using the entity ’TEXT_LITERAL’. ’SHAPE_REPRESENTATION’ entity is used to associate this annotation with the corresponding part feature semantically.

6.2 Using the entities from STEP AP242 Ed2

All the GD&T information can be included in the STEP AP242 files using the entities defined in the standard. The methodology followed is given in CAx recommended practices for ”Representation and Presentation of Product Manufacturing Information (PMI) (AP242)” [47]. From this recommendation, only ’representation’ aspects are relevant as the information is semantically represented and reusable for robotic applications. This CAx recommendation uses the entities defined in the first edition of STEP AP242.

The authors of this article have analyzed the second edition of STEP AP242 and performed a comparative study against the first edition for the present work. The second edition has a set of changes in type and entity definitions from the first edition. However, these changes do not affect the methodology described in the CAx recommendation. Hence, the authors suggest using the same methodology described by CAx using the second edition of STEP AP242. The entities from the following Application Modules (AM) can capture the GD&T information of the parts: Dimension tolerance (Part: 1050), Geometric tolerance (Part: 1051), and Default tolerance (Part: 1052).

Table 2 Geometric dimensioning and tolerancing symbols, their unicode from CAx recommendation [53] and the custom codes

GD&T symbol	Description	Unicode	Custom code
∠	Angularity	2220	–
︵	Arc Length	FE35	–
↔	Between	2194	–
Ω	Capital Omega	03A9	–
℄	Center Line	2104	–
↗	Circular Runout	2197	
○	Circularity	25CB	–
◎	Concentricity	25CE	–
⌲	Conical Taper	2332	–
Continuous Feature	Continuous Feature	–	TCOF
Controlled Radius	Controlled Radius	–	TCTR
⌴	Counterbore	2334	–
⌵	Countersink	2335	–
⌭	Cylindricity	232D	–
⌀	Diameter	2300	–
Depth	Depth	–	TDPT
Envelope Requirement	Envelope Requirement	–	TEPR
⏥	Flatness	23E5	–
Ⓕ	Free State	24BB	–
Ⓘ	Independency	24BE	–
Ⓛ	Least Material Condition	24C1	–
Ⓜ	Max Material Condition	24C2	–
⫽	Parallelism	2AFD	–
⏊	Perpendicularity	23CA	–
Plus/Minus	Plus/Minus	–	TPOM
⌖	Position	2316	–
⌒	Profile of a Line	2312	–
⌓	Profile of a Surface	2313	–
Ⓟ	Projected Tolerance Zone	24C5	–
Radius	Radius	–	TRDS
Ⓡ	Reciprocity	24C7	–
Ⓢ	Regardless of Feature Size	24C8	–
⌳	Slope	2333	–
⍵	Small Omega	2375	–
Spherical Diameter	Spherical Diameter	–	TSPD
Spherical Radius	Spherical Radius	–	TSPR
Spot Face	Spot Face	–	TSTF
□	Square	25A1	–
Statistical Tolerance	Statistical Tolerance	–	TSTT
⏤	Straightness	23E4	–
⌯	Symmetry	232F	–
Ⓣ	Tangent Plane	24C9	–
⌰	Total Runout	2330	–
▷	Translation Modifier	25B7	–
Ⓤ	Unilateral/Unequally Disposed	24CA	–

Table 3 Keywords and their abbreviations [53]

Keyword	Abbreviation (identifier)
Basic Dimension	B
Reference Dimension	R
Dimension with Unidirectional Tolerances	S
Dimension with Equal Bidirectional Tolerances	E
Dimension with Unequal Bidirectional Tolerances	U
Feature Control Frame (ASME)/Tolerance Frame (ISO)	F
Datum Feature Symbol	D
Datum Target Symbol	T
AllAround Symbol	TAAS
All Over Symbol	TAOS
Derived Feature	TDFT

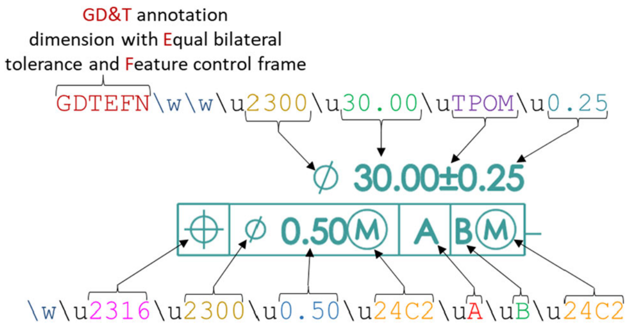

Fig. 4 GD&T annotation with the corresponding unicode string

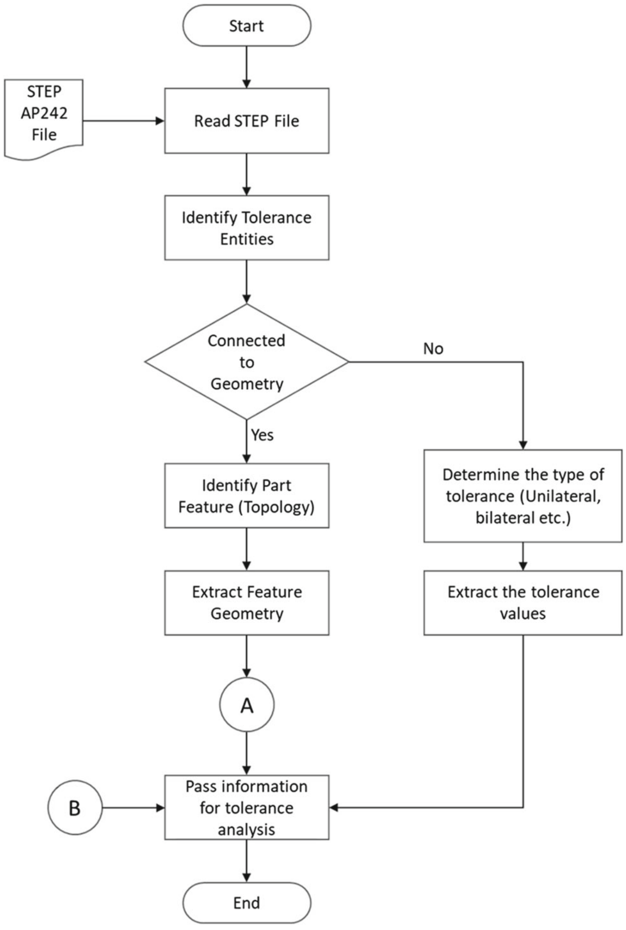

Fig. 5 Extraction of GD&T information from STEP AP242 entities

The entities from the first three AMs are mainly mapped to the entities of Integrated Generic Resource: Shape variation tolerances (Part: 47). The Part: 47 has three schemas that cover the information requirements for GD&T defined in these AMs. The schemas of Part: 47 are ‘Shape aspect definition, Shape dimension and Shape tolerance’. Using the entities from these AMs and following the methodology of CAx recommendation, all the GD&T information can be added semantically to the part features.

Another useful AM is ‘Extended geometric tolerance’ (Part:1666). This AM specifies the entities for defining the tolerance zone boundaries and can be used for automated tolerance analysis and quality analysis.

6.3 Extracting the GD&T information

Extraction of GD&T information from the Unicode string is straightforward. From the ‘SHAPE_REPRESENTATION’ entity, the part feature and the corresponding Unicode tolerance annotation can be identified. The feature geometry is extracted using the standard geometric and topological entities of STEP AP242 standard, and the Unicode string is extracted from the entity ‘TEXT_LITERAL’. A parser is developed to parse the Unicode annotation and extract the tolerance information. This information is used in forming the tolerance zones and tolerance analysis.

The overall process for extracting the tolerance information while using the standard entities from STEP AP242 requires more steps than the Unicode string. A flowchart of the process is shown in Fig. 5. After reading the STEP file, the program searches and identifies the tolerance entities. These are the entities from the schemas of Shape variation tolerances (Part: 47) of the standard. These entities are checked to determine whether these tolerances are connected to the geometry of part features. If the tolerance entities refer to ‘SHAPE_ASPECT’ or ‘COMPOSITE_GROUP_SHAPE_ASPECT,’ then these tolerances are attached to the part features. This corresponds to the path ‘yes’ after the ‘Connected to Geometry’ check in Fig. 5. In this case, the corresponding part feature and its geometry are extracted from the topological and geometric entities.

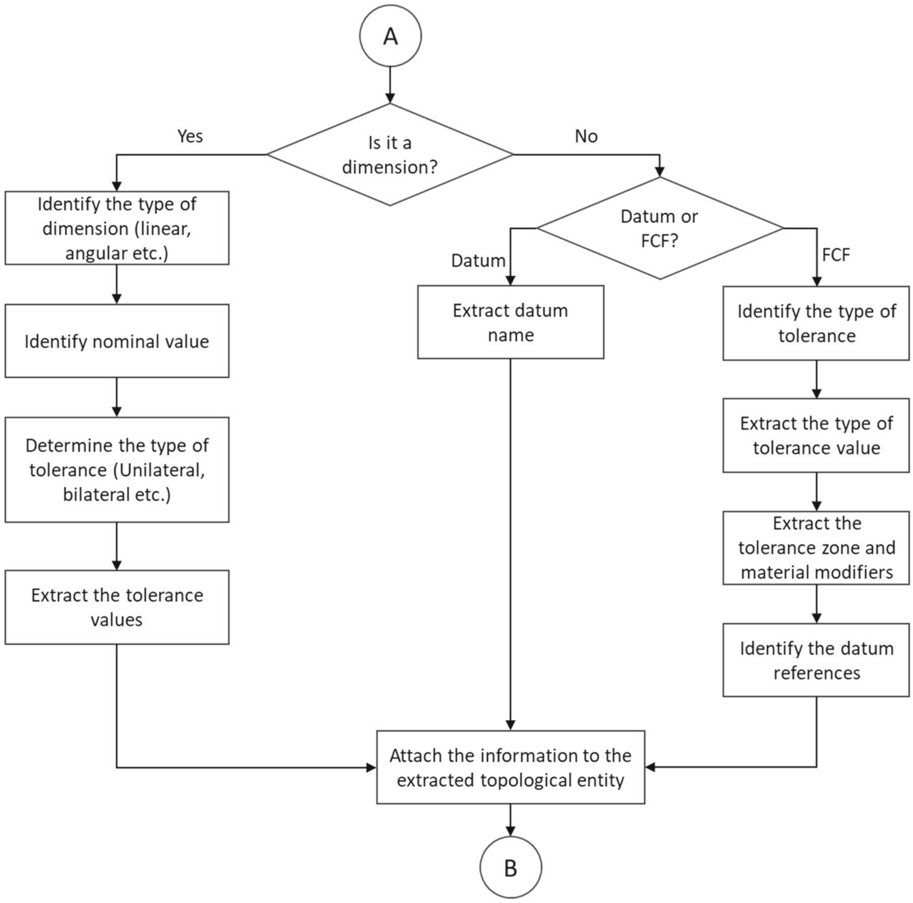

Fig. 6 Identification and extraction of dimension and geometric tolerances and datum features

The default tolerances are not attached to any particular geometric feature, hence do not refer to 'SHAPE_ASPECT' or 'COMPOSITE_GROUP_SHAPE_ASPECT' entities. This corresponds to the path 'no' after the 'Connected to Geometry' check in Fig. 5. These default tolerances apply to all the dimensions for which specific tolerances are not given. Traditionally these tolerances are shown in the 'title block' of the manufacturing drawing. The type and values of default tolerances are extracted from the entities of Part: 1052 (Default tolerances). The values of these default tolerances are then passed on to the tolerance analysis programs.

After extracting the topological and geometric details of the entities, the attached dimension and tolerance information is extracted. The tolerance information is inferred from various entities from the schemas of Part: 47 (Shape variation tolerances). Figure 6 shows the important steps in this process. First, it is determined whether the annotation is a dimension or has a datum or an FCF. If a dimension is attached to the feature, then the type of the dimension, i.e., dimension of size or location or distance or angle, is determined. Then the nominal value is identified. The type and value of tolerance affecting this dimension are determined. The dimensions and tolerance values from the standard entities can be readily extracted from the entities corresponding to Part: 1050 (Dimensions tolerances).

If a datum callout is attached, the name is identified and attached to the part feature. The datum target values are not considered in this paper. In the case of an FCF, all the values related to the geometric tolerances along with the modifiers are extracted as shown in Fig. 6. These values can be extracted from the entity mappings defined in Part: 1051 (Geometric tolerances). Once all the information is extracted, it is linked to the corresponding part feature. The tolerance information and the corresponding feature geometry are further used to define the tolerance zones and perform tolerance analysis, described in the next section.

7 Use-cases

7.1 Assembly use-case

This process is demonstrated using a motor assembly design provided by Mjøs Metallvare shown in Fig. 7. The tolerances applied to these three major motor sub-assemblies are also shown. The tolerance values shown are created for demonstration purposes and do not correspond with the true tolerance values in the product. Figure 8 shows the portion of

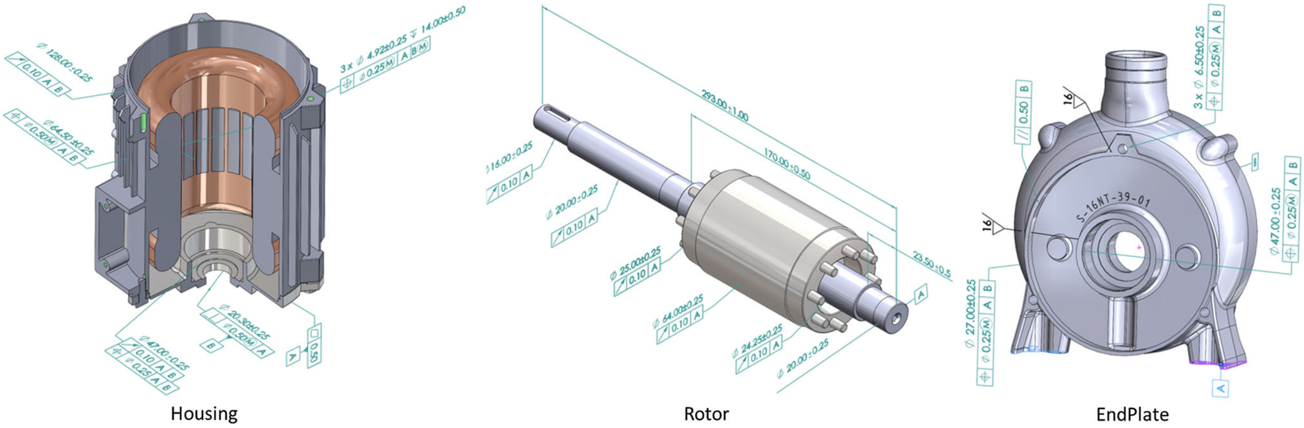

Fig. 7 Motor sub-assemblies with GD&T annotations

Fig. 8 Portion of STEP file showing the entities for semantic tolerance and datum annotations

```
#4402=DATUM('Datum_A',$,#23,.F.,'A');
#4403=DATUM_FEATURE('Datum_A',$,#4406,.T.);
#4404=SHAPE_ASPECT_RELATIONSHIP('Datum_A',$,#4402,#4403);
#4405=GEOMETRIC_ITEM_SPECIFIC_USAGE('Datum_A-to-BottomFace',$,#4406,#16,#1591);
#4406=SHAPE_ASPECT('Flatness_Datum_A',$,#23,.T.);
#4407=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.5),#9);
#4408=FLATNESS_TOLERANCE('Flatness_BottomFace',$,#4407,#4406);
```

a STEP file with standards entities adding a flatness tolerance and a datum callout attached to the FCF of the flatness tolerance. The clearance ratios between the Rotor-Housing and between Rotor-EndPlate are calculated using these tolerance values. The overall process for calculating the effective clearance from the mating features and their tolerances is shown in Fig. 9. The lowest clearance ratio value is considered for programming of the assembly processes where mating/insertion co-occurs at two features. In the present use-case, the insertion/mating happens at two features simultaneously during the assembly of Rotor with Housing. The different stages of this assembly are shown in Fig. 10. The start and end of these two stages of mating can be estimated from the insertion lengths, and the insertion lengths are calculated from the mating constraints and the tolerance values. In this case, the overall insertion length is from the bottom of the bearing seat to its corresponding mating surface on the bearing, approximately 162 mm. The Rotor reaches the Housing in the initial approach and achieves rough alignment. During this stage, the manipulator has much freedom in angle and position. Precise control is unnecessary, and the manipulator path can be planned based on position control.

In the first stage, the first possible contact situation occurs between the magnet region of the Rotor with the windings in the Housing. At the beginning of this stage, the clearance between the components is about 3 mm, and it reduces to 0.25 mm as the insertion proceeds. Precise position control is needed from this stage.

The second stage starts when the bearing starts mating with the bearing seat in the bottom plate of the Housing sub-assembly. This stage starts after insertion of about 142 mm and ends after another 19 mm insertion. The clearance between the housing seat and the bearing outer diameter is around 0.03 mm, requiring sensor-feedback control.

The assembly task is completed when the bearing is seated in the Housing. Successful assembly is achieved when a sufficient force is detected and the Rotor is placed within the expected region identified by tolerance analysis.

Similarly, the assembly of EndPlate with the rest of the motor can be divided into first stage when it starts mating with the shaft and the second stage when the top bearing starts mating with the bearing housing on the EndPlate. In this step, the first stage has very high clearance, and the clearances in the second stage are the same as that of the second stage of the earlier step.

7.2 Welding use-case

The same process is applied to the T–joint of two pipes shown in Fig. 11. In case of structural welds involving beams, the weld seams are along the edges of mating surfaces. These can be directly identified and extracted from the CAD models. Extraction and identification of welds involving cylindrical parts like pipe welds are a bit complicated. Generally welders cut the profiles in the pipes before welding using wrap-around template curves. When these weld joints are properly modelled in the CAD files, then the seam curves will be readily available in the STEP files. The 'B_SPLINE_CURVE_WITH_KNOTS' entity from the STEP AP242 file shown in Fig. 12 gives the weld seam of the T-joint shown in Fig. 11. From this, the nominal geometry of the seam curve was derived. The information from

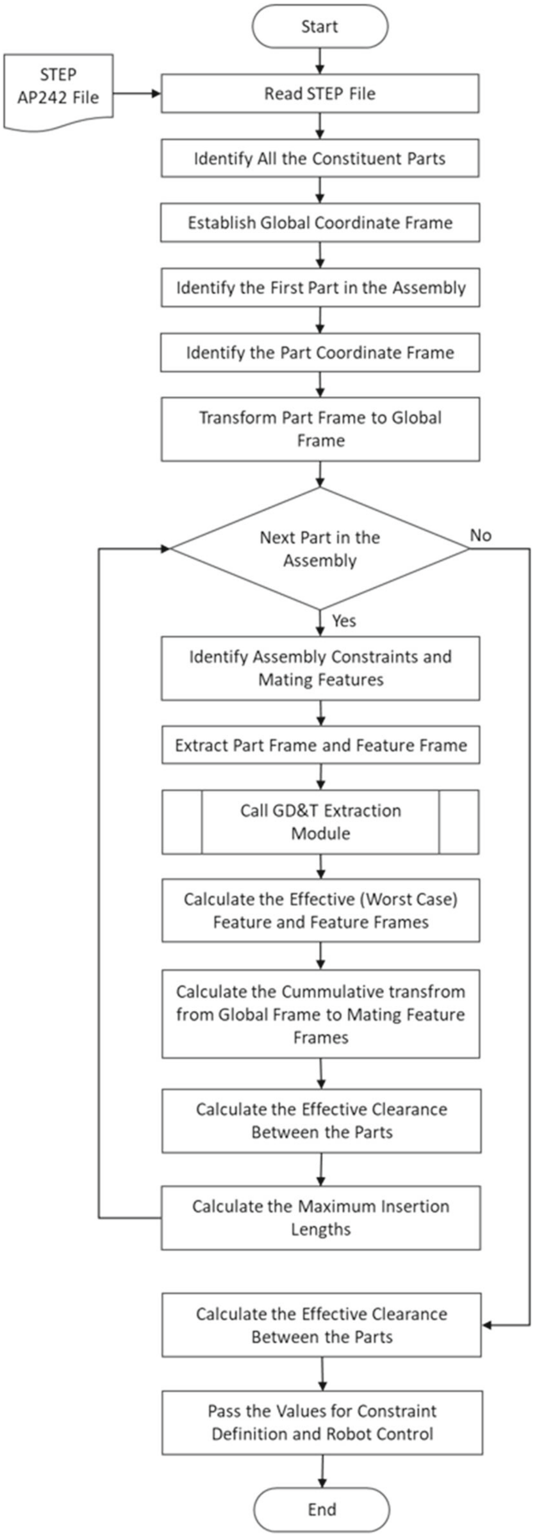

Fig. 9 Process for extracting and using assembly and tolerance information

GD&T annotations is used to calculate the deviations from the nominal geometry. The identified weld seams will be in the local coordinate frame of the mating features. These have to be represented with respect to the coordinate frame of the welding cell in which the weld torch is described.

The Unicode string for welding annotation is included as 'TEXT_LITERAL' entity. The groove geometry was extracted from the weld annotations. The weld groove is bevel with a bevel angle of $37.5° \pm 2.5°$ and root face of 1.6 ± 0.4 mm. The Unicode weld string is modified by adding the tolerance value after the nominal value.

- The tolerance value is separated from the nominal value by '\v' as shown in Fig. 12.
- If the tolerance is unilateral the tolerance value has '+' or '-' sign, which ever is applicable. If the tolerance is bilateral the value does not carry any sign.

The groove geometry with the seam can be used as an input for defining the sensing window for planning the scanning using a visual sensor system.

8 Conclusions

This paper described how GD&T information could be included in STEP AP242 files as a neutral exchange file format between in-company processes or automation providers for SMEs. Two methods were described: one used a Unicode string and the other used standard entities from the second edition of the STEP AP242 standard. A process to extract the included GD&T information from STEP files is presented in this paper. The relevance of this GD&T information for robot programming was discussed, and a motor assembly was used as a use-case, demonstrating the use of this information. This GD&T information is used to estimate the worst-case boundaries of the mating features. This information is also used to calculate the clearance ratio and insertion length. These two are essential parameters in an assembly operation that decide the mating forces and can be used to decide the control method for robotic assembly.

The tolerance information can be combined with statistical quality control and machine learning methods to automatically estimate the clearances for the same or similar products in an assembly line. This will increase the effectiveness and applicability of the robotic assembly for batch production.

The Unicode string approach to annotate GD&T is simple and allows for fast parsing and backward compatibility

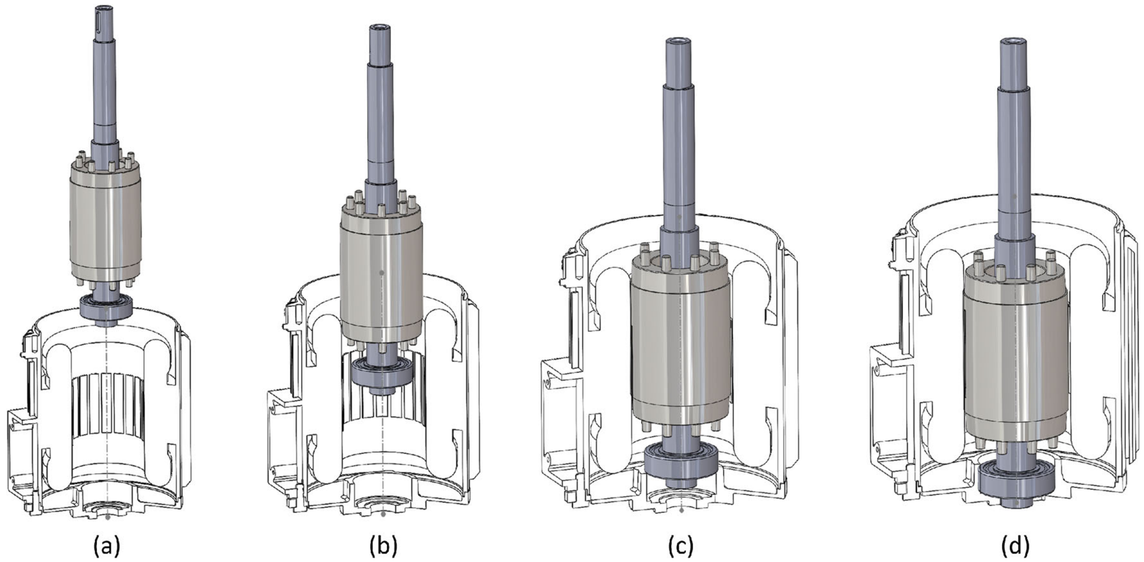

Fig. 10 Rotor—Housing assembly stages. **a** Initial approach. **b** First stage. **c** Second stage. **d** Final stage

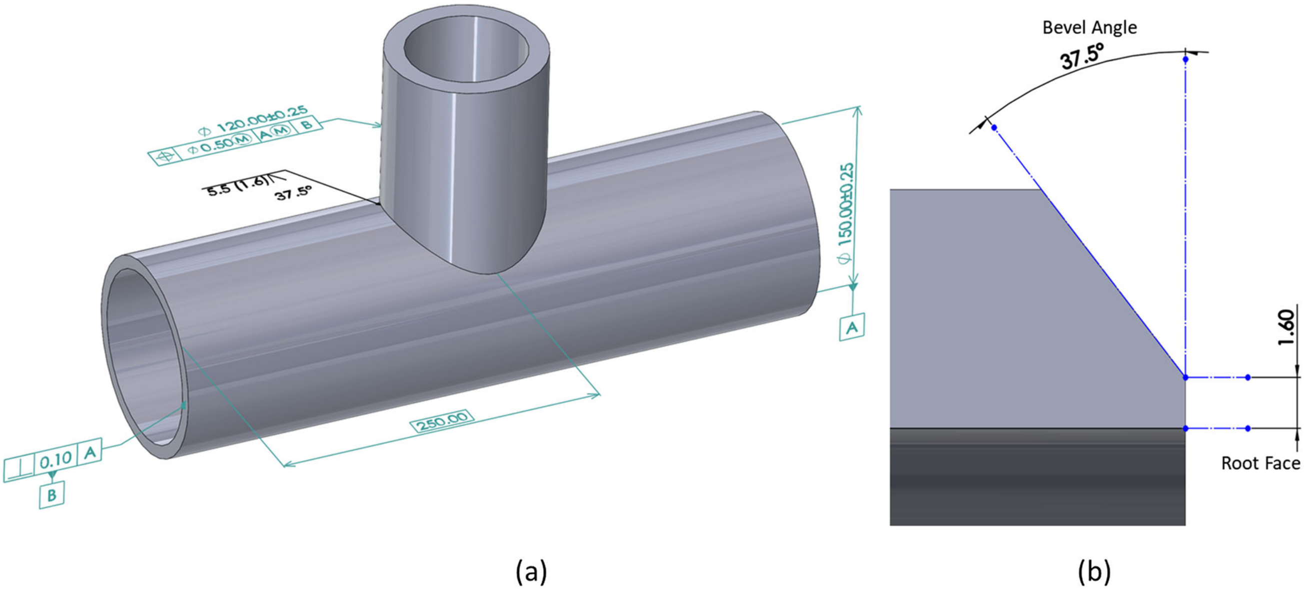

Fig. 11 **a** T-joint of pipes with GD&T and weld annotations. **b** Groove geometry

Fig. 12 Portion of STEP file showing the entities for weld seam and weld annotations

```
#361=B_SPLINE_CURVE_WITH_KNOTS('',3,(#269,#270,#271,#272,#273,#274,#275,#276,
#277,#278,#279,#280,#281,#282,#283,#284,#285,#286,#287,#288,#289,#290,#291,
#292,#293,#294,#295,#296,#297,#298,#299,#300,#301,#302,#303,#304,#305,#306,
#307,#308,#309,#310,#311,#312,#313,#314,#315,#316,#317,#318,#319,#320,#321,
#322,#323,#324,#325,#326,#327,#328,#329,#330,#331,#332,#333,#334,#335,#336,
#337,#338,#339,#340,#341,#342,#343,#344, #345,#346,#347,#348,#349,#350,#351,
#352,#353,#354,#355,#356,#357,#358,#359,#360),.UNSPECIFIED.,.T.,.F.,(4,2,2,2,2,
2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,2,
2,4),(0., 0.03125,0.046875,0.0625,0.09375,0.125,0.15625,0.1875,0.203125,0.21875,
0.25,0.265625,0.28125,0.3125,0.328125,0.34375,0.375,0.40625,0.4375,0.453125,
0.46875,0.484375,0.5,0.515625,0.53125,0.5625,0.578125,0.59375,0.625,0.640625,
0.65625,0.6875,0.703125,0.71875,0.75,0.765625,0.78125,0.8125,0.84375,0.875,0.90625,
0.9375,0.953125,0.96875,0.984375,1.),.UNSPECIFIED.);
#362=EDGE_CURVE('',#252,#252,#361,.T.);
#439=TEXT_LITERAL('Weld Annotation','WLDNNN\\w\\w\\u\\u\\u\\u\\u\\u\\u\\u\\w5.5
\\u1.6\\v0.4\\uWBAS\\u\\u\\u37.5\\v2.5\\u\\x\\u\\u\\w',$,$,$,$);
```

with previous STEP application protocols. Using the STEP AP242 second edition entities for annotation allows for more granular control and a direct linking between zero, one, or multiple STEP entities. However, it is not available in all CAD software systems yet. In either approach, the availability of GD&T information for use in downstream processes is an essential step towards the generalization of robot programming methods that can help close the gap between design and production.

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Declarations

Conflict of interest The authors declare that there is no conflict of interest involved in this research work.

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