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TECHNICAL PAPER

Towards the development of a smart manufacturing system for the automated remodeling and manufacturing of standard parts

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Abstract

The standard components of a scalable product incur minor design changes to meet functional requirements, such as a change in load or capacity. Usually, this leads to a complete renewal of the Computer-Aided Design (CAD) and Manufacturing sequence of such parts, which is tedious and time-consuming. This paper proposes developing an integrated smart system for automated remodeling and manufacturing of such reconfigurable standard components. It includes three main steps: automatic remodeling of CAD model, automatic feature extraction from XML, and Computer Numerical Control (CNC) part program generation with minimal human intervention. In case the user adds new features to the standard design of the component, the proposed system can extend its ability by enabling automatic recognition of general machining features using the STEP AP-203 file. The whole development process of standard products is serialized by parameters and computations to achieve an automated design procedure that is integrated into Autodesk Inventor software. It is then combined with the developed algorithm of automatic feature recognition and automated toolpath generation using Vb.net. The modular design of the system allows the reconfiguration of the individual modules to include extra capabilities without affecting the whole system. Finally, a case study was presented for the design and manufacturing of a typical component of the conveyor system to demonstrates the efficacy of the developed system.

Keywords Manufacturing automation · CAD/CAM · STEP · Product modeling · Automatic design

Abbreviations

CAD	Computer aided design
CAM	Computer aided manufacturing
CNC	Computer numerical control
XML	Extensible markup language
STEP	STandard for the exchange of product model data
AFR	Automatic feature recognition
STL	Stereolithography
CAPP	Computer aided process planning
OEM	Original equipment manufacturer
UI	User interface
p_j	Primary dimensions
s_l	Secondary dimensions
β_k	Basic dimensions
α_i	Geometry parameters
M_{para}	Database of manuals
RegEx	Regular expression
Ts	Type of surface
$C_{t,i}$	Center point of top
$C_{b,i}$	Center point of bottom
L_s	Length of the surface
A_i	Advanced face
$V_{i,j}$	Vectices of advaced face
$\hat{\alpha}$	Distance vector
h	Depth of pocket
AF	Advanced face
FB	Face bound
FOB	Face outer bound
$\mathrm{L_S}$	Length of the shaft
$\mathrm{L_O}$	Length of the outer bracket
$\mathrm{L_M}$	Length of the middle bracket
$\mathrm{L_R}$	Length of the roller
$\mathrm{L_n}$	Length of the notch
D	Roller diameter
$\mathrm{d_s}$	Shaft diameter
$\mathrm{w_b}$	Bracket width

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1 Introduction

Today, standard parts are widely used in material handling, die molds, and automobile industries as a component of large assemblies. These parts follow a specific set of protocols (such as company-specific standards, ISO, functional, and safety standard) that improves interoperability, compatibility, repeatability, and quality of the assembled products [1]. Generally, standard parts are outsourced to small/medium scale industries to make the assembly cost-effective. The design and manufacturing of such components require less workforce and resources as compared to custom-made parts. Hence, it is beneficial for any design and manufacturing firm to follow some design protocols while manufacturing such standard parts.

The standard parts incur frequent dimensional changes due to changes in functional requirements (like load, size, and cost) that allow the reuse of existing designs. However, minor design changes often lead to a complete renewal of the CAD model and manufacturing sequence based on application requirements. Typically, in a manufacturing firm, 15% of the manufacturing cycle time is spent in technical decision making, 40% in data table lookout and calculations, and 45% in text and documentation [2]. Thus, data lookup and documentation constitute a significant portion of the total lead time of manufacturing such parts.

To manufacture a product that involves machining in its production route, first, the design engineer calculates the basic dimensions of product geometry. Other necessary dimensions are found from design handbooks, international standards, and industry-specific manuals with additional geometry calculations based on these values. The next step is to communicate these product specifications with the manufacturing engineer, who will then recognize machining features and develop CNC codes for machining the standard part. This traditional approach to design and manufacturing is cumbersome, error-prone, and time-consuming. Thus, to remain competitive in today's volatile markets, more efficient methods to perform these tasks must be deduced. The integration of CAD and CAM could be a solution. This work proposes an interactive approach for the same.

2 Related literature

The integration of CAD and CAM for interactive automated product development pertains to three main research portfolios: automatic design, feature recognition, and part program generation. However, there is a lack of work focused on combining all three research directions.

Potočnik et al. [3] developed a system for the automated design of compound washer dies. They made use of knowledge-based generative models for the proposed system development. Virtual modeling of a shoe for the footwear industry using image processing and geometrical rules was presented by Raffaeli and Germani [4]. They achieved good results, but the method was computationally intensive. Li and Liang [5] proposed an intelligent design method for stamping dies based on parameterization and sketch drive. Wei et al. [6] exemplified a methodology to design the layout of standard parts for the automated design of multi-plate die based on distributed assembly modeling strategy. Carfagni et al. [7] proposed an automated design procedure for vane-motor based on constructive parameters and the mathematical model obtained from the design of experiments approach. Colombo et al. [8] designed a 3D interactive environment to design medical devices based on the patient's data. Bai et al. [9] presented a semantic-based approach for partial retrieval of 3D CAD models for design reuse. However, the CAD model reconstruction was not discussed in this work. Gong et al. [10] devised a decision-making system to develop a method of design for remanufacturing. Their approach was non-empirical that caught the designer's subjective preferences. It is evident from the literature that no system exists, which tries to reduce the remodeling time of standard parts during CAD modeling.

After CAD modeling next step in automatic product development is to identify the machining features. Literature reports various automatic feature extraction/recognition approaches from different part models, such as STL, IGES, and STEP. The feature information is used for downside applications like CNC code generation [11], process planning [12], defeaturing [13], etc. The feature recognition techniques viz., rule-based [11], hint based [14], syntactic pattern recognition [15], attributed adjacency graph-based [16, 17], Artificial neural network-based [18] are most widely used. For the complete automation of manufacturing, a feature recognition technique should be generic and easy to implement on industrial parts. Generally, the research work in this area is specific to the types of machining features or their applications. Liu et al. [19] proposed an algorithm for the extraction of the features of the prismatic part from IGES. Arivazhagan et al. [15] developed a module for recognition of tapered and curved features from the STEP file. Sunil et al. [20] reported the automatic recognition of machining features using artificial neural networks from the SAT file for sheet metal parts. Leo Kumar et al. [21] presented a feature extraction technique from XML file for micromachine components. Rameshbabu and Shunmugam [17] presented a hybrid approach to recognizing prismatic manufacturing features by volume subtraction and face adjacency graph from STEP format. It is seen that if the types of features are known a priori, as in the case of standard parts, the rule-based techniques could be very useful. The rule-based technique on XML and STEP files is easy to implement, fast, and accurate.

Automatic toolpath generation is one of the applications of feature recognition. Sivakumar and Dhanalakshmi [11] developed a Numerical Control (NC) code generation system for cylindrical parts using the STEP file as input. Suh et al. [22] presented a method to generate CNC part programs from the STEP AP-203 file. Han et al. [23] performed feature recognition from STEP, followed by tool path planning and finally cost analysis to prove the requirement of process planning along with feature recognition. Leo Kumar et al. [21] presented a toolpath planning approach for micro-machined components. Patil and Pande [24] reported an intelligent feature-based process planning (IFPP) system for prismatic parts. They used a Feature-Based Modeler (FBM) to make the CAD model using a design by feature approach, followed by Computer Aided Process Planning (CAPP). Khan et al. [12] developed a process planning approach for prismatic die parts using the STEP AP-203 file. Carlos et al. [25] described a method for an internet-based CAD/CAPP/CAM system using STEP-NC. As an end application of an integrated CAD/CAM system, Letaief et al. [26] proposed an approach to use company database and geometrical and topological parameters of the CAD model to estimate manufacturing cost. However, the designing of a new CAD model and machining planning were not devised in this work.

The link between the CAD modeling, feature recognition, and CNC toolpath generation is usually established manually. Literature suggests that scant work is done to integrate them. Most of the approaches either perform automatic remodeling/intelligent design, automatic feature recognition (AFR), or automatic CNC toolpath generation. Thus, there is a need to automate and combine the design process and generation of part programs for manufacturing standard parts to reduce the product development cycle time. This paper is an attempt in this direction. It presents a new interactive way of automatic remodeling and manufacturing of standard parts with minimal human intervention.

The organization of the rest of the paper is as follows. Section 3 explains the different modules of the system. Section 4 presents a case study to show the capability of the developed system, and Sect. 5 presents the conclusion.

3 Modular components of the smart system

Figure 1 shows the flowchart of the system. The method proposed in this paper consists of three main modules—automatic remodeling, feature extraction/recognition, and part program generation. The first module generates the CAD model of the standard part. The system then exports

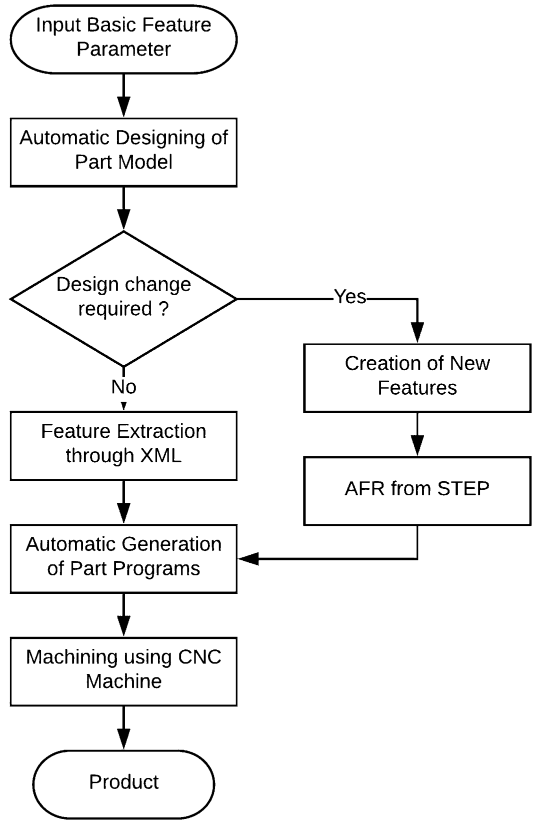


Fig. 1 The proposed methodology for the development of the smart system

the CAD model as XML and extracts the features. If there is any change in conventional design, such as the addition of slots, steps, and tapers, the CAD model is updated manually and exported as a STEP AP-203 file. The system then reads the STEP file to recognize the machining features. The third module generates the CNC part programs using the information provided from the previous module about the machining features.

As noted earlier, the original equipment manufacturers (OEMs) and several small/medium scale industries use CAD and CAM software in tandem to develop a product that is error-prone and inefficient. The proposed system automates the process of CAD modeling and integrates the modules mentioned above to increase productivity.

These modules are configured to create an interactive system [27]. A user interface (UI) is designed for remodeling, AFR, and generate CNC part program as shown in Fig. 13. The remodeled CAD can be visualized. Besides, a decision support system prompts for adding any new features to the product. The user can also edit the CNC part program (if needed) before saving or transferring it to the CNC machine. Thus, interactive design of the system gives the user control over the course of action while automating the whole process to manufacture a product efficiently. The following sections describe the modules of the developed system in detail.

3.1 Automatic remodeling of standard parts

The designing of a product requires finding out the *basic parameters* of the product by experienced design engineers. Then, for each product component, the *primary dimensions* are taken from the databook/manuals corresponding to the basic parameters. The remaining *secondary dimensions* are calculated using these extracted values based on the geometry and topology of the part. For example, in the case of the ball bearing design, *primary dimensions* are the diameter of the hole, the diameter of the balls, the thickness of the bearing, and the width of the bearing. These dimensions can be extracted directly from the standard manuals corresponding to the *basic parameters* viz. load, shaft diameter, and type of fit. And the secondary dimensions, such as the outer diameter of the bearing and the diameter of the labyrinth seal, are calculated using primary dimensions.

The CAD model of the product is designed using the primary $\left(p_j\right)$ and secondary dimensions (s_l). The change in the basic parameters $\left(\beta_k\right)$ changes the primary and secondary dimensions, which often leads to complete redesigning of the product. The proposed method tries to eliminate this shortcoming of the conventional designing process. It involves the extraction of parameters, establishing constraints, scaling, and using empirical relations to carry out this task.

Figure 2 shows the modular diagram of the proposed Automatic remodeling system. A VB.net script was written to link the various modules of the designed system. The user only needs to give new basic parameters as the input. The developed program would then read the values of the modified parameters and update the CAD model automatically using Inventor COM API for remodeling of parts.

The steps to design the system are as follows:

1. Prepare the database of technical handbooks, standard manuals, and company standards stored as individual arrays (M_{para}).
2. Initialize all the geometry parameters $\left(\alpha_i \forall i = (1, n)\right)$ of the CAD model in the database.
3. Extract the values of primary dimensions $(p_j \subset \alpha_i)$ from the database (M_{para}) based on the basic parameters $(\beta_k \subset \alpha_i)$.
4. Calculate the secondary dimensions $(s_l \subset \alpha_i)$ from p_i.
5. Design the CAD model of the standard parts using Autodesk Inventor's part modeling module.
6. Inter-Link the dimensions of the CAD model with the α_i in the Autodesk Inventor.
7. Create the assembly of the complete model by including suitable dimensional constraints and spatial constraints for keeping the relationship between all the modeled standard parts intact.

After designing the system, the modules are interlinked using VB.net script to automate the process. This approach could bring a considerable reduction in the lead time for redesigning the CAD model. Section 4 explains the proposed system with the help of a case study.

3.2 Automatic feature extraction/recognition system

After the remodeling of the CAD model, the next step is to identify the machining features. For standard parts, the system exports the CAD model as XML. Whereas automatic feature recognition (AFR) was used for additional features that are out of the scope of the standard part feature library.

3.2.1 Automatic feature extraction from XML file

The remodeling changes the dimensions of the standard parts. The developed program automatically reads and extracts the parameter names and their corresponding dimensions from the XML file. The *xmlread* function of VB.net was used to read the XML file. Figure 3 shows a sample XML file. A feature library is created in the system to define various features of the part. This feature library is used to interpret the variables extracted by reading the

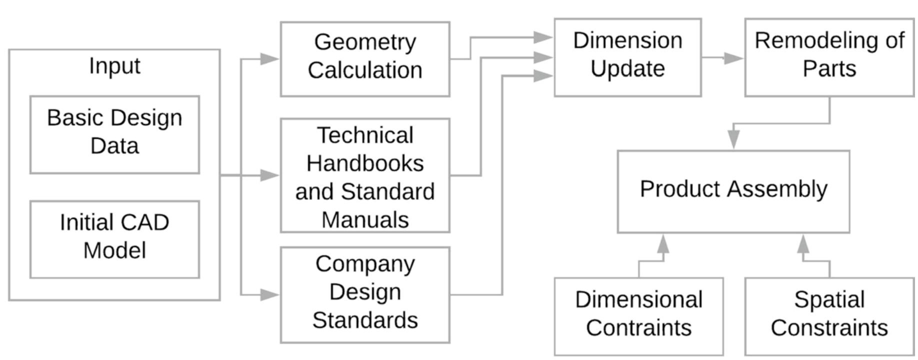


Fig. 2 Modular diagram of the automated remodeling system

```
<?xml version="1.0" encoding="utf-8"?>
<ParamWithValueList xmlns:xsd="http://
  <version>20080502</version>
  <parameterTypes>
    <ParamType>
      <typeName>Inventor</typeName>
      <typeCode>0</typeCode>
    </ParamType>
    <ParamType>
      <typeName>String</typeName>
      <typeCode>1</typeCode>
    </ParamType>
    <ParamType>
      <typeName>Boolean</typeName>
      <typeCode>2</typeCode>
    </ParamType>
  </parameterTypes>
  <parameters>
    <ParamWithValue>
      <name>d0</name>
      <typeCode>mm</typeCode>
      <value>shaft_dia</value>
      <comment />
      <isKey>false</isKey>
    </ParamWithValue>
```

Fig. 3 Sample XML file

XML file, and a suitable algorithm is used to generate part programs for each extracted machining feature.

3.2.2 Automatic feature recognition (AFR) from a STEP file

The STEP (AP-203) file has a hierarchical structure. It starts with the *Shell* that is formed by a combination of *advanced faces*. Each advanced face has *bounds* and *surface types,* which could be *planar, cylindrical, conical,* or *toroidal*. Bounds are of two types, face bound and face outer bound, which specify whether the type of edge loops of the entity are internal or external, respectively. Next, the edge loop is defined by oriented edges, which have edge curves. The edge curves are finally defined as lines or circles along with their vertices.

The hierarchy of the STEP file must be traversed to take out useful information regarding geometrical entities such as the type, coordinate values, and directions. The proposed method uses Regular Expression (RegEx) to perform this operation. A RegEx is a way of describing a search pattern using a unique combination of strings. The RegEx engine matches the defined string pattern and extracts the data. The data is stored in a data grid view, a temporary tabulation of data in VB.net. The arrangement of strings helps in identifying the pattern for feature recognition without reading the STEP file recursively. As opposed to the methods that either use line-by-line text reading or some available tools for reading STEP files, feature recognition performed on the temporary tabulation speeds up the navigation process through the hierarchy of STEP files. The regex search pattern for a few strings in an ISO 10303-21 (STEP) file is given below in Table 1, and Fig. 4 shows the extracted data stored in the data grid view of VB.net.

Figure 5 shows the flowchart for recognizing the turning features from the stored tabulated data grid. The advanced faces are arranged in the table as per their distance from the origin. The system reads the advanced faces and extracts the following information by horizontal lookup.

- Type of surface T_s, i.e., cylindrical, conical
- Center point of top ($C_{t,i}$) and bottom ($C_{b,i}$) face
- Radius of the top ($R_{t,\,i}$) and bottom ($R_{b,i}$) face
- Length of the surface (L_s).

The algorithm checks two advanced faces at a time. A turning feature is present if the two advanced faces are connected at the same point, i.e., C_b of AF_i is equal to the C_t of the AF_{i+1}. If the radius of the second face is greater than the bottom radius of the first face, then the feature is classified as a *step*. Otherwise, it is a *groove*. The profile of the feature depends on the type of surface. Figure 6 shows a typical axisymmetric part with various turning features.

Machining features for vertical machining centers differ from CNC turning features regarding topology and the process of recognition. In this work, basic non-intersecting prismatic machining features are recognized, such as pockets and holes. The developed system uses the pattern matching technique for the recognition of machining features on prismatic parts.

In the STEP file, holes are specified by *cylindrical* advanced_face. It must have bottom face_outer_bound and top face_bound to identify it as a hole. The face_bound center is the center of the hole, and the distance between the face bound and face outer bound gives the depth of the hole.

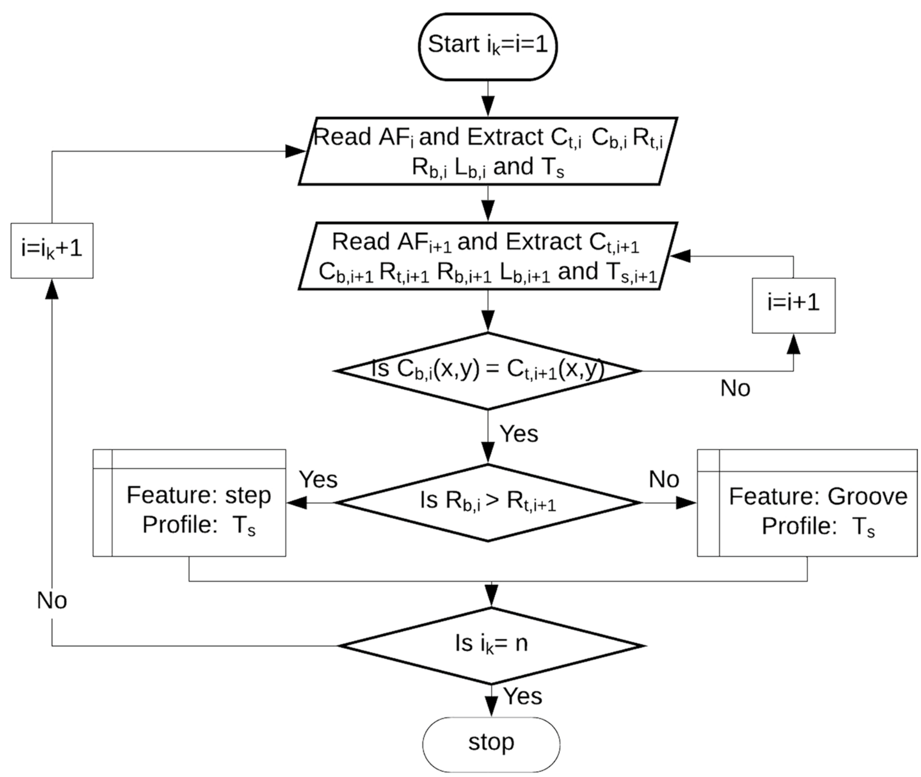
Table 1 RegEx for some geometrical strings of STEP

Advanced face	(?< =\#)(.+)(?=\=ADVANCED_FACE\(\''\,\(\#)
Face outer bound	(\#[0–9]+ =ADVANCED_FACE\(\'[A-Z]*'\,)(.+)(\,\#)+(?< =\(#)([0–9]+)
Face bound	(\#[0–9]+ =ADVANCED_FACE\(\'[A-Z]*'\,)(.+)(\,\#)+(?< =\,#)([0–9]+)
Cartesian point (x)	(?< =\#[0–9]*\=CARTESIAN_POINT \(\'[A-Za-z]*\'\,[\s]*\()(-?[0–9]+\.[0–9]*[E]*[-]*[0–9]*)

	A2D	OE	AF	FOB	FB	FOB	FBel	OEe	EC	CIR(	A2D	EL	A2D	A2D	A2D	CIRf	CIRI	ORI(	surfa	point	point	curv	verte	vcartesic
	316	166	224	81	74	94	95	138	138	114	238	94	0.	0.	0.	240	30.	316	222	118	118	114	318	118
	317	167	225	82	75	96	97	139	139	115	239	95	41...	11...	100.	241	30.	317	223	119	119	115	320	119
	318	168	226	83		98		140	140	116	240	96	44...	11...	0.	243	25.	319	63	120	120	116	323	120
	319	169	227	84		99		141	141	117	241	97	41...	11...	0.	244	25.	321	64	121	121	117	325	121
	320	170	228	85		100		140	142		242	98	44...	11...	100.			322	65	122	123	15	329	122
	321	171	229	86		101		142	143		243	99	41...	11...	100.			324	66	122	124	16	330	123
	322	172	230	87		102		143	144		244	99	31...	10...	100.			326	67	125	124	17	332	124
	323	173	231	88		103		144	145		245	99	34...	10...	90.			327	68	123	125	18	334	125
	324	174	232	89		104		145	146		246	99	31...	10...	90.			328	69	126	122	19	338	126
	325	175	233	90		105		146	147		247	100	34...	10...	100.			337	70	126	127	20	340	127
	326	176	234	91		106		147	148		248	100	31...	10...	100.			343	71	124	127	21	344	128
	327	177	235	92	76	107	108	148	149		249	100	31...	10...	90.			349	72	128	126	22	346	129
	328	178	236	93	79	111	112	143	150		250	100	150.	150.	100.			352	73	128	129	23	353	130
	329	179						149	151		251	101	150.	150.	100.			361		127	129	24	354	131
	330	180						150	152		252	101	150.	50.	100.			367		123	128	25	356	132
	331	181						151	153		253	101	150.	125.	100.			373		129	125	26	358	133
	332	182						147	154		254	101	150.	150.	0.			376		130	131	27	362	134
	333	183						152	155		255	102	150.	150.	100.			377		131	132	28	364	135

Fig. 4 Data grid to store geometrical strings temporarily

Fig. 5 Flowchart to recognize the turning feature



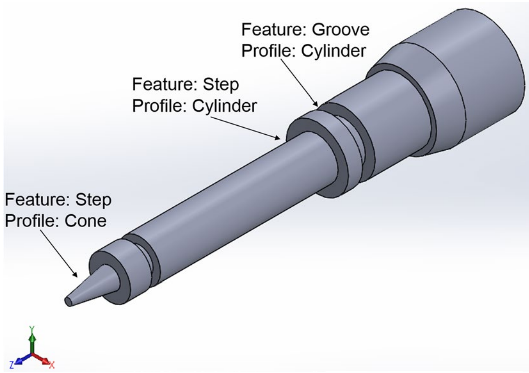


Fig. 6 Axisymmetric part with turning features

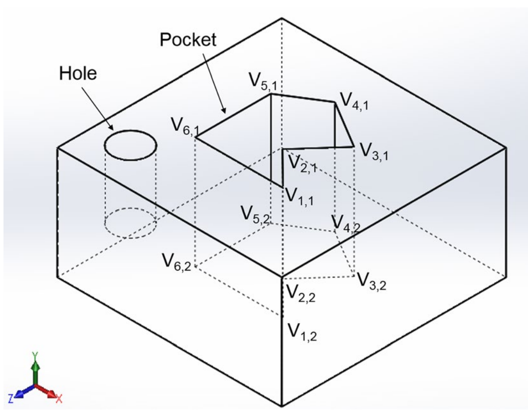


Fig. 7 Prismatic part with milling features

For recognizing a pocket, a *Planar* advanced_face with only one face_bound is identified. Let one such advanced_face A_1 have vertices $V_{1,1}$–$V_{6,1}$ (Fig. 7). If another advanced face A_2 is present in the STEP file with the same number of vertices ($V_{1,2}$–$V_{6,2}$). And the direction vectors of all the vertices from A_2 to A_1 are the same. Then the feature is identified as a pocket. The direction vector $\hat{d}$ of the pocket can be found using Eq. 1.

$$\hat{d} = \frac{\overrightarrow{V_{i,1}} - \overrightarrow{V_{i,2}}}{\left|\overrightarrow{V_{i,1}} - \overrightarrow{V_{i,2}}\right|} = \frac{\overrightarrow{V_{j,1}} - \overrightarrow{V_{j,2}}}{\left|\overrightarrow{V_{j,1}} - \overrightarrow{V_{j,2}}\right|} \quad \forall i,j \in [1,n] \tag{1}$$

where n is the total number of vertices in the selected advanced face. Equation 2 gives the depth of the pocket 'h.'

$$h = \left|\overrightarrow{V_{i,1}} - \overrightarrow{V_{i,2}}\right| \quad \forall i \in [1,n] \tag{2}$$

The proposed method of feature recognition might give better accuracy than the ANN-based feature recognition techniques. In addition, it does not require time for training the network because it uses a rule-based approach. However, for every new feature, new rules need to be written. Therefore, it is most suitable for industries where the types of features remain fixed.

3.3 Automated part program generation system

After extraction of geometry and position of features, the next step is to use this information to generate CNC codes. The output of the feature extraction module is the type, profile, size, and position of the machining feature. The CNC turning machine is used to manufacture the axisymmetric parts, and 3-axis vertical milling machine is used for prismatic parts.

The machining features are projected on the X–Z plane for CNC turning features, where Z is the spindle axis. The width of the workpiece is divided into parallel horizontal segments of the user-defined step size. The intersection of the projected features and the parallel segments forms feed paths for step turning. A plunge toolpath is used for grooves, which is generated by segmenting the projected contour using vertical lines. Ramp-on and ramp-off moves are added at the start and end of the feed paths. The toolpath generated by the developed system was verified using CNC simulation software (Discriminator 2.1.0.32). Figure 8 shows the toolpath to machine the part shown in Fig. 6.

The developed system generates layer by layer roughing toolpath for milling features. The pockets are machined using contour parallel toolpath topology. The contour of the pocket is recursively offset inside to create the toolpath. The offset is done until the width of the contour becomes less than the radius of the tool. Next, the holes are machined by taking the drilling tool above the center point of the hole. Which is then interpolates into the workpiece to the required depth of the hole. Figure 9 shows the simulation of the machining operations (pocketing and drilling) for the part shown in Fig. 7 using simulation software Discriminator 2.1.0.32.

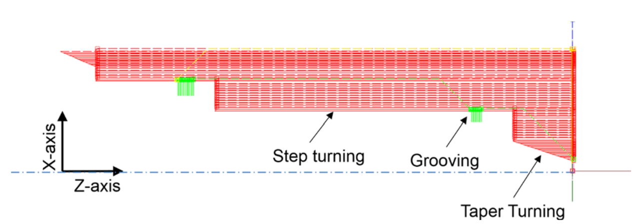


Fig. 8 Simulated toolpath for CNC turning

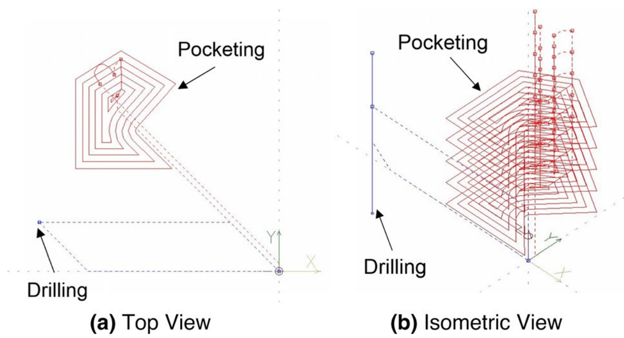


Fig. 9 Simulated toolpath for milling features

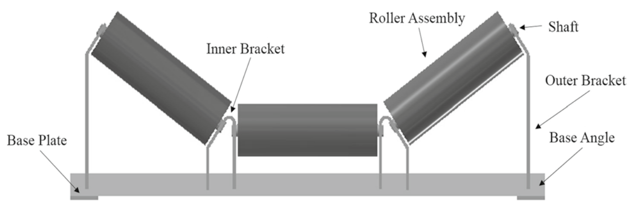


Fig. 10 A CAD model of an idler assembly

4 Demonstration of the proposed system: case study

The proposed smart system is developed in a VB.net programming environment and implemented on a desktop with Windows 7 OS, Intel Core i3, and 4 GB of RAM. The system used the commercial software Autodesk Inventor© for CAD modeling and visualization. The spreadsheet was meant for database mining, and calculation was prepared using Microsoft Excel®. A VB script links the spreadsheet with the CAD package to automatically update the CAD model and generate the CNC part program.

This section presents a case study of an idler assembly, an essential part of the conveyor system. Figure 10 shows the complete assembly of an idler consisting of various components such as shafts, middle bracket, and outer bracket that can be partly manufactured using CNC machines.

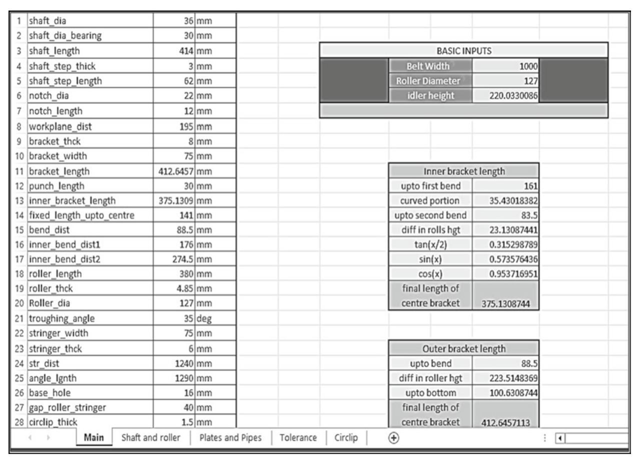
Dimensions of an idler depend on two main basic specifications, viz. belt width and roller diameter. Based on these, primary dimensions are extracted from the standard manual of belt conveyors (IS 11592:2000 Selection and Design of Belt Conveyors—Code of Practice) [28] viz. length of the shaft, the diameter of the shaft, dimensions of the notch. The secondary dimensions are then computed using primary dimensions and geometry calculations. Typical primary dimensions and their dependency on basic dimensions for idlers are shown in Table 1. And

Table 2 Dependency of basic on primary

Primary dimensions	Basic dimensions
Shaft diameter	Roller diameter
Bearing diameter	Belt width
Roller thickness	Idler height
Bracket thickness	
Stringer thickness	
Bracket width	
Troughing angle	

Table 3 Dependency of basic and primary on secondary

Secondary dimensions	Primary and basic dimension
Shaft length	Belt width
Inner bracket length	Belt width, idler height, shaft diameter, troughing angle, roller diameter
Outer bracket length	Belt width, idler height, shaft diameter, troughing angle, roller diameter
Roller length	Belt width, shaft diameter
Notch width	Shaft diameter
Notch length	Shaft diameter, troughing angle



1	shaft_dia	36	mm
2	shaft_dia_bearing	30	mm
3	shaft_length	414	mm
4	shaft_step_thick	3	mm
5	shaft_step_length	62	mm
6	notch_dia	22	mm
7	notch_length	12	mm
8	workplane_dist	195	mm
9	bracket_thck	8	mm
10	bracket_width	75	mm
11	bracket_length	412.6457	mm
12	punch_length	30	mm
13	inner_bracket_length	375.1309	mm
14	fixed_length_upto_centre	141	mm
15	bend_dist	88.5	mm
16	inner_bend_dist1	176	mm
17	inner_bend_dist2	274.5	mm
18	roller_length	380	mm
19	roller_thck	4.85	mm
20	Roller_dia	127	mm
21	troughing_angle	35	deg
22	stringer_width	75	mm
23	stringer_thck	6	mm
24	str_dist	1240	mm
25	angle_lgnth	1290	mm
26	base_hole	16	mm
27	gap_roller_stringer	40	mm
28	circlip_thick	1.5	mm

BASIC INPUTS	
Belt Width	1000
Roller Diameter	127
idler height	220.0330086

Inner bracket length	
upto first bend	161
curved portion	35.43018382
upto second bend	83.5
diff in rolls hgt	23.13087441
tan(x/2)	0.315298789
sin(x)	0.573576436
cos(x)	0.953716951
final length of centre bracket	375.1308744

Outer bracket length	
upto bend	88.5
diff in roller hgt	223.5148369
upto bottom	100.6308744
final length of centre bracket	412.6457113

Main | Shaft and roller | Plates and Pipes | Tolerance | Circlip

Fig. 11 Spreadsheet for database mining and calculations

the dependency of secondary dimensions on basic and primary dimensions is shown in Tables 2 and 3.

Figure 11 shows the spreadsheet prepared to evaluate the primary and secondary dimensions. The cells of primary dimensions use lookup functions to extract values from the database, and the cells with secondary dimensions use geometry equations. For example, Eq. 3–5 are used to calculate the length of the shaft, middle bracket, and outer bracket, respectively (Fig. 12a–c).

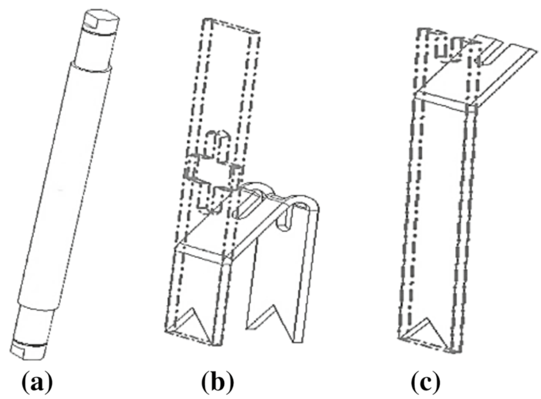


Fig. 12 CAD model of Idler parts **a** Shaft, **b** Middle Bracket, and **c** Outer Bracket

$$L_s = L_s + 2 \times C + w_b \tag{3}$$

$$\begin{aligned} L_m = {} & h_i + D + C + w \times \cos 45 + \frac{w_b}{2} \\ & + \left(2 \times C + \frac{h_b}{2}\right) \times \pi \times \frac{((180 - \emptyset))}{180} \\ & + C + \frac{d_s}{2} + \left(\frac{D}{2} + C + \frac{d_s}{2}\right) \\ & \times \left(2 \times C + \frac{t_b}{2} + \tan \frac{\emptyset}{2}\right) \end{aligned} \tag{4}$$

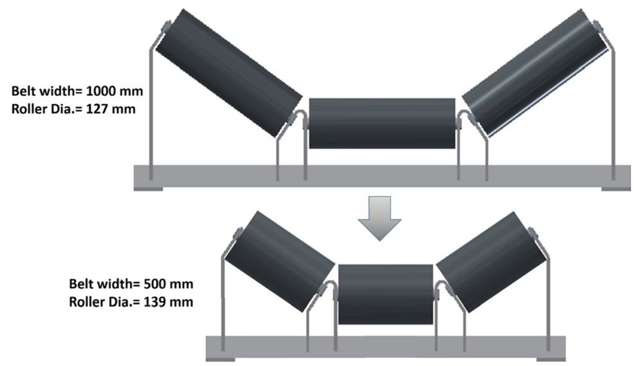


Fig. 14 Automatic remodeling of the CAD model with the change in basic dimensions

$$\begin{aligned} L_o = {} & d_s + D + 4 \times C + \left(L_s - L_n \times 2 + t_b\right) \\ & \times \sin \emptyset - C \times \cos \emptyset + W_g + \frac{h_b}{2} \end{aligned} \tag{5}$$

where L_S, L_O, L_M, L_R, and L_n, are the lengths of the shaft, outer bracket, middle bracket, roller, and the notch, respectively. h_i, is the idler height; D, roller diameter; w, stringer width; w_b, bracket width; W_g, the gap between roller and stringer; h_b, bracket height; d_s, shaft diameter; t_b, bracket thickness; and C is clearance (generally taken as 4 or 5 mm).

Figure 13 shows the user interface developed for the system. A VB script is linked with each button to perform a specific task.

The script reads the modified parameters from the excel file and updates the CAD model using the API functionality

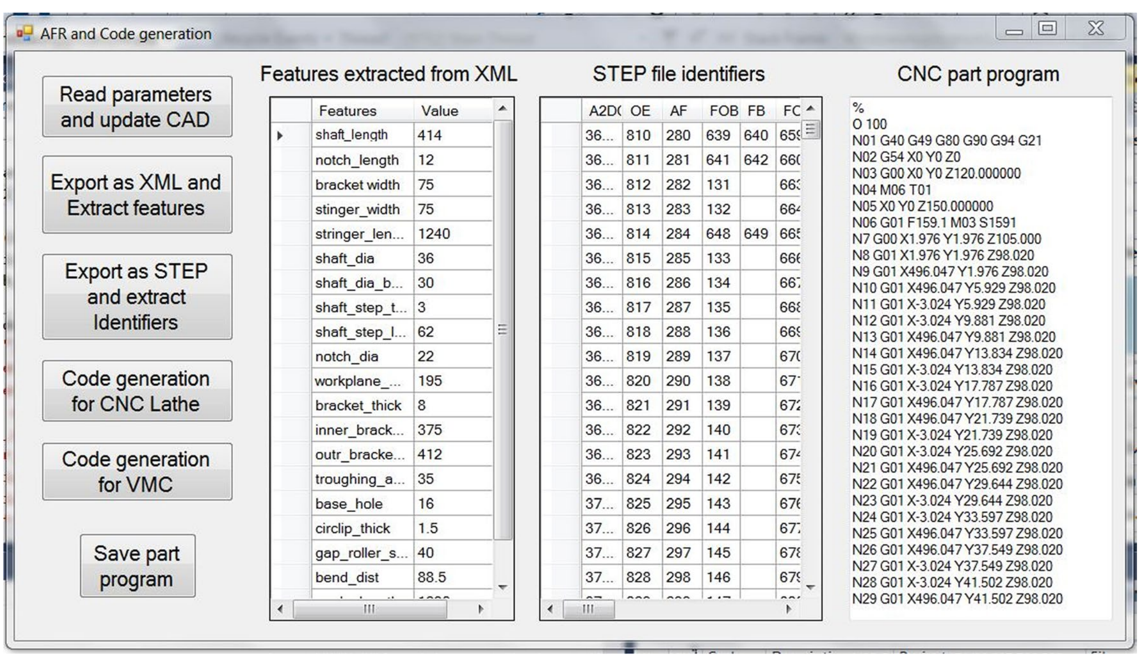


Fig. 13 The user interface of the system

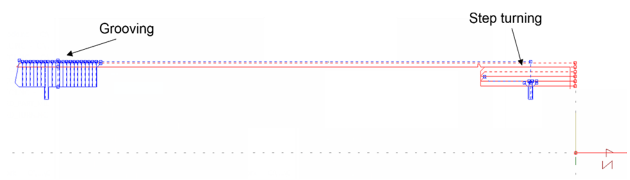


Fig. 15 Simulation of tool paths for the shaft

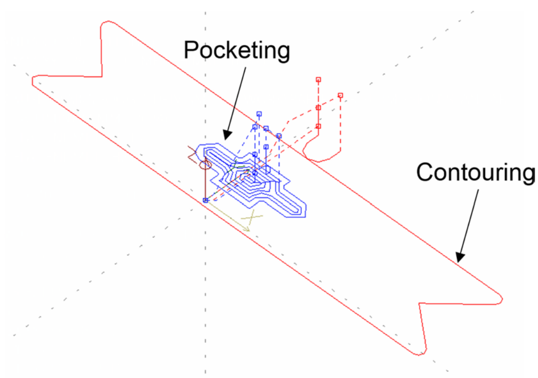


Fig. 16 Simulated toolpath for the middle bracket

of Autodesk inventor. Figure 14 shows the change in the design of the idler assembly on changing the belt width from 1000 to 500 mm and roller diameter from 127 to 139 mm.

The system then exports the CAD model as an XML file and extracts machining features. The main features extracted for the manufacturing of the shaft are *steps* and *grooves*. Whereas, for the middle and outer brackets, it is the external dimensions of the bracket, notch location, and dimensions of the notch. The system enables the addition of new prismatic machining features that are in the scope of the AFR module (such as holes, pockets, and slots) in the standard CAD model. By clicking the *Export as STEP and extract identifiers* button, the system exports the modified CAD model as the STEP AP-203 file and recognize machining features. The machining feature parameters thus obtained are taken as input to generate CNC part programs for turning and milling machines.

A simulation software (Discriminator, Version Discrim_V2_ 1_0_40) was used to check the accuracy of the generated tool paths. Figures 15 and 16 show the tool paths generated for the manufacturing of shaft and middle bracket for 1000 mm belt width and a roller diameter of 127 mm, respectively.

The developed system works well for all the belt widths that are there in the database. The database for the manufacturing of parts can always be extended to produce more variants.

5 Conclusion

This paper proposed a smart system for integrated CAD/CAM. It uses feature-based CAD to enable automatic remodeling and a new efficient Regex-based pattern matching technique for feature extraction and recognition. The recognized features are finally used to generate contour parallel CNC toolpath for milling automatically. An interactive system was established to seamlessly integrate these modules and reduce the manufacturing lead time of standard parts. It is an attractive option for small/medium industries that engage in the manufacture of standard parts, as it improves the efficacy and productivity during the part design and manufacturing stage.

In the future, various other modules can also be added to the system, such as cost estimation, computer-aided design analysis, computer-aided process planning, etc., to utilize its modular design effectively. Although robust, the AFR in the current system only works for prismatic parts. This work can be extended further to include more complicated features using ANN-based feature recognition technique.

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