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# Advance Parametric 3D Modelling and Drafting of Mechanical Component using AutoCAD

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**Abstract:** *The research of engineering drawing design based on solid model is a hot spot in the research and application of 3D CAD technology. Three-dimensional design is an inevitable trend in the application of CAD technology. For different types of plastic parts, based on a comprehensive analysis and comparison of various schemes of plastic part feature classification, this paper uses 3D feature modeling technology and parameterization technology to establish a 3D feature library of injection molded products. Entity extended database technology is applied to make features carry non-geometric information. This paper proposes a method of creating "orthographic digital model of object" using 3D CAD software. This digital model displays both the physical model and its projection on the virtual projection surface, which can more realistically and intuitively show the process of generating an orthographic view of an object. This is a novel explanation that uses digital means to interpret the orthographic projection view in engineering drawings.*

**Keywords:** AutoCAD; 3D Modeling; Industrial Modeling Design

## I. INTRODUCTION

During the last few decades CAD software are extensively used to increase the productivity of the designer, improve the quality of design and communication through documentation and to create database for manufacturing. This is due to their benefits such as lesser design lead times i.e. lesser time to market, easy modifications, improved quality and productivity, less number of design iterations, etc. A variety of CAD software are available in the market made more and more generalised so as to fit the purpose of wide range of industries. But it is difficult to find CAD software which will meet all the requirements of particular industry. Hence there must be some facility in the CAD software so as to model them to satisfy the specific need of the industry. This can be done by customizing the CAD software according to the application programming interface (API) in which various objects of the CAD software can be used in the program and application which is termed as 'Automation'.

Three-dimensional AutoCAD technology is a hot spot in current CAD research and application, and digital design of engineering drawings based on solid models is an important aspect of this research and application [1]. Three-dimensional design is an inevitable trend in the application of CAD technology [2]. With the widespread application of AutoCAD technology, design departments and manufacturing companies have basically eliminated the drawing board, and realized the digital design of engineering drawings, making the work of drawing more scientific and standardized [3]. With the marketization of 3D CAD technology, people began to use 3D parametric software such as CATIA, UG, Pro / E, and CAXA for more advanced engineering design, and 3D solid models can be used to generate 2D engineering drawings associated with it [4]. It realized the design reuse of 3D solid model information from 2D engineering drawings. CAD system integration is a major trend in the current development of CAD technology. With the integration of CAD technology, there has been a modern integrated manufacturing system composed of CAD / CAE / CAM and other functions. Lee et al [5] point out that in this system, designers can use computers to perform motion analysis, dynamic analysis and stress analysis,

For many decades, one of the main disciplines of the system of training the highest technical qualifications (engineers of mechanics, engineers of builders and other specialists) is descriptive geometry. It was created more than two

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hundred years ago [1, 2] by the French scientist Gaspar Monge, "descriptive geometry was cultivated in the technical school as a science, without which it is inconceivable the formation of an engineer" [1]. As a rule, the "birth" of any product, practically regardless of the field of application, is associated with three stages: design and conceptualization of the idea in the engineer's mind, creation and processing of documentation, production of the product itself on the basis of this documentation. Obviously, these stages are not only closely interrelated, but they do not exhaust all the nuances of preparing and ensuring the production of the finished product. If the product is a construction, in reality a three-dimensional, then the second stage necessarily includes the presentation of information about this design in the form convenient for processing it. Currently, this is a three-dimensional computer model [3, 4, 5], the necessary calculations [6 – 12] and drawing. Drawing, according to Monge, is the language of technology, and the grammar of this language is descriptive geometry [1]. The technology of creating 3D models of three-dimensional objects and the development of design documentation on their basis became available to students and teachers of universities in the early years of the 21st century. Currently, the design process carried out by the vast majority of enterprises is based on the following scheme: the creation of a 3D model of the product, the execution on its basis of calculations (kinematic, dynamic, strength and others), the development of the designed product on their basis and, in a semi-automatic mode, the development drawings and other documentation necessary for the subsequent organization of his (product) production. At first glance, with such an approach, the knowledge of many disciplines studied in higher education institutions is not necessary. The authors may exaggerate, but among such disciplines, mathematics – numbers can be multiplied on a calculator, and, for example, solving an equation in Maple, MatLAB or MathCAD, mechanics – problems in the theory of mechanisms and machines, machine parts, material resistance are easily solved in within the framework of many 3D modeling systems: Autodesk Inventor Professional, Autodesk Simulation, SolidWorks, descriptive geometry – for example, the line of intersection of geometric objects can be constructed without difficulty in any of the three-dimensional modeling systems. However, the essence of higher education, according to the authors, is not the acquisition of a set of skills and the ability to use them mechanically, but the formation of an engineer's special structure of thinking and knowledge base in a whole range of sciences that allowhim to creatively approach the task assigned to him, just copy, but get the best possible solution.

## II. OVERVIEW OF SYSTEM

During the last few decades CAD software are extensively used to increase the productivity of the designer, improve the quality of design and communication through documentation and to create database for manufacturing. This is due to their benefits such as lesser design lead times i.e. lesser time to market, easy modifications, improved quality and productivity, less number of design iterations, etc. A variety of CAD software are available in the market made more and more generalised so as to fit the purpose of wide range of industries. But it is difficult to find CAD software which will meet all the requirements of particular industry. Hence there must be some facility in the CAD software so as to model them to satisfy the specific need of the industry. This can be done by customizing the CAD software according to the application programming interface (API) in which various objects of the CAD software can be used in the program and application which is termed as 'Automation'.

### A. Limitations of CAD geometric modelling

CAD software's have made their place in the industry due to many advantages they have. However, they suffer from following limitations.

- Skilled manpower required
- Time consuming modelling processes
- Higher cost of software as well as higher wages of the operator
- Chances of human errors and drafting errors are not eliminated.

**B. Alternative Methods**

In the present scenario the above problems are tackled by one of the alternative solutions as stated below [1].

- By storing the master file for each type of component and then modifying it as per requirement
- By using the 'Variable table' provided by the CAD software
- By using the part family option if provided by the CAD software

**C. Limitations of existing methods**

All the methods discussed above, suffer few of the drawbacks as discussed below.

- **A graphical master file is required to be stored:** The biggest limitation of all above mentioned methods is that, all of them require at least one model file to be stored. The graphical nature of these model files requires more of the memory space to store. This problem more prominent when number of variations in the part goes on increasing goes on increasing.

- **Shape changes cannot be accommodated:** Though above methods are effective in case of dimensional changes, they are not at all useful to accommodate the shape variation in the components. Some of the features can definitely be suppressed or unsuppressed and few shape changes can be achieved but upto a certain extent. These methods are not available for assembly and drafting. By none of the above methods, repetitive drafting can be eliminated.

- **Skill required cannot be completely eliminated:** All the above-mentioned methods still require good knowledge of the CAD software rather greater knowledge and understanding of the CAD software that required for straight forward modelling. None of these methods can reduce the skill requirement. The wages are directly linked with the skill requirement and hence cannot prove useful for considerable reduction in cost.

- **Error may creep in due to mistakes in inputting data from catalogues:** The changes in the dimensions are to be achieved by changing the values manually wherein thee new values for the required model are generally available from the different catalogues (manufacturing catalogues) available in the design department. Errors may creep is due to mistakes in inputting data from catalogues. There is no any foolproof system to avoid any such mistakes and such errors may not be identified and thus the model created may be a faulty model.

- **Accidental changes in master files are possible:** This is one of the risky limitations of all the three methods. As the master files are in soft form only and are directly available for the user. There are also some changes of accidental changes in the master file. If such changes are saved this may cause major problem, as the data related to master file itself is lost forever. The change may or may not be repairable.

- **Being soft in nature may get corrupt or accidentally be deleted:** The model files stored are also softcopies only so they may be any time get damaged. The files may accidentally delete. Thus, the master file is required to be created again and reducing the reliability of the method.

**D. Part Libraries and e-drawings**

Every design calls for standard part such as bolts, screws, nuts, or linear actuators. Modelling these components repetitively takes a lot of time. There are various part libraries according to the different standards. Once installed, these part libraries can be viewed with the Library Browser so that parts can be inserted into your drawings [2].

On insertion, each part is converted into a block which can be re-inserted many times. Block, a group of entities can be inserted into the same drawing more than once with different attributes locations, scales and rotation angle. Such an instance of a block is usually called an "insert". Inserts have attributes just like entities and layers. An entity that is part

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of an insert can have its own attributes or share the attributes of the insert. The power of inserts is that you can modify the block once and all inserts will be updated accordingly. Blocks can contain useful text, dimensions and reference notes [3].

Many software provide most commonly required models on the internet in the form of e-drawings for almost all CAD software. e-drawings are the premier 2D and 3D design communication tool for internal and external design teams which deliver a rich collaborate tool set and enable to speed up the design process. [4].

The electronic data format of e-drawings files is highly compressed to enable easy transmission via email. edrawings files are created through the use of the e-drawings publisher add-in, which works as a plug-in to many popular CAD products. Once created, an e-drawings file may be sent to anyone via email and support file formats such as CATIA V5 (.CATPart), SOLIDWORKS native files and templates (.sldprt, .slddrw, .sldasm, .prtdot, .drwdot and .asmdot), DXF/DWG files (.dxf, .dwg), ProEngineer files (.prt, .prt, .xpr, .asm, .xas), edrawings native files (.edrw, .eprt, .easm, .edrwx, .eprtx, .easmx). Even the above alternatives have drawbacks of size i.e. require high memory due to graphical nature of the files. The size of a typical e-drawings software is in the range of 10 to 12 GB and that of an e-drawing file is around 15 to 20 MB. It takes much time to download the files from the internet which may be comparatively costlier affair. The market price of an e-drawing software is about 1000$ which is very costly [5].

## III. PROBLEM DEFINITION

As the CAD modelling techniques become more and more advanced, there is a need to complete the product modelling and design changes faster than ever. Updating the assemblies having hundreds of sub-assemblies and parts manually in 3D modelling software is very complicated & time consuming. The CAD models that have large number of parameters and high memory required more graphics and time to process the data. To reduce the development time, minimizing the errors and introduce technologies faster to the market, many companies have been turning more and more to automation process [6]. Automation is a set of technologies that results in operation of systems and machines without significant human intervention by saving the time and also achieving performance superior to manual operation. By using automation it is possible to achieve increase in productivity, quality and robustness along with reduction in time, labour costs and other expenses. The concept of automating the different activities of the design department can better be demonstrated by taking some product [7]. The present study is to automate the different activities of the design department for the basic mechanical components.

## IV. DESIGN OF PLASTIC FEATURE LIBRARY

Based on the analysis of the structure of the plastic part, a systematic plastic feature library was established by referring to the STEP standard and designing the plastic part according to the actual application requirements. The established feature database of plastic parts divides features into four categories: basic voxels, rendering features, components (composite features), and location features.

Basic features and drawing features are relatively simple geometric shapes. In practical applications, plastic parts are generally composed of basic voxels and drawing features. However, a careful analysis of various plastic parts shapes reveals that there are some more common and active shape features, which are composed of basic voxels and drawing features, which are called components (composite features) in this paper. When constructing such features as components, the shape characteristics of plastic parts were analyzed and studied in large quantities, and the most common and universal feature units were extracted from them. The main hierarchical structure of the feature library is shown in Figure 1. The information system process is shown in Figure 1 below.

### A. Construction of Round Table Features

The basic voxel and the three-dimensional shape structure of drawing features are relatively simple, the amount of feature data input is relatively small, and the feature shape can be completed in one step. We take the round-shaped features as an example to explain the construction principles of these relatively simple features. he shape of the round

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table features is relatively easy to implement. First, the rationality check of the input round table feature data must be performed. If the data is not reasonable, we return to the data input interface. Otherwise, in the current WCS coordinate system, a round table entity is generated according to the XY direction radius and height values of the upper and lower end faces of the round table feature. Then, the position transformation matrix between the WCS coordinate system and the UCS coordinate system is established by combining the position coordinate values of the round table feature, and then the position of the round table entity is transformed to the corresponding position in the UCS coordinate system through the matrix. Finally, the data of the features of the round table are added to the 3D solids of the features of the round table by expanding the dictionary. In this way, a feature model carrying non-geometric information of features is formed. Through the feature model, you can query and modify the feature data. The program flowchart is shown in Figure 3.

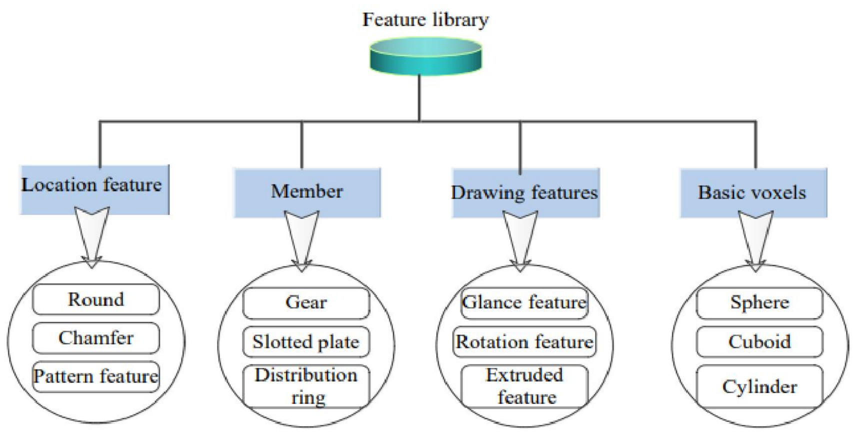


Fig1: The hierarchical structure of the feature library.

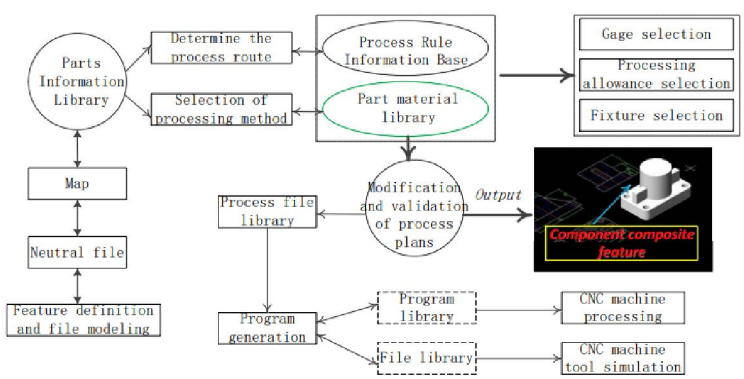


Fig2: Information system flow chart

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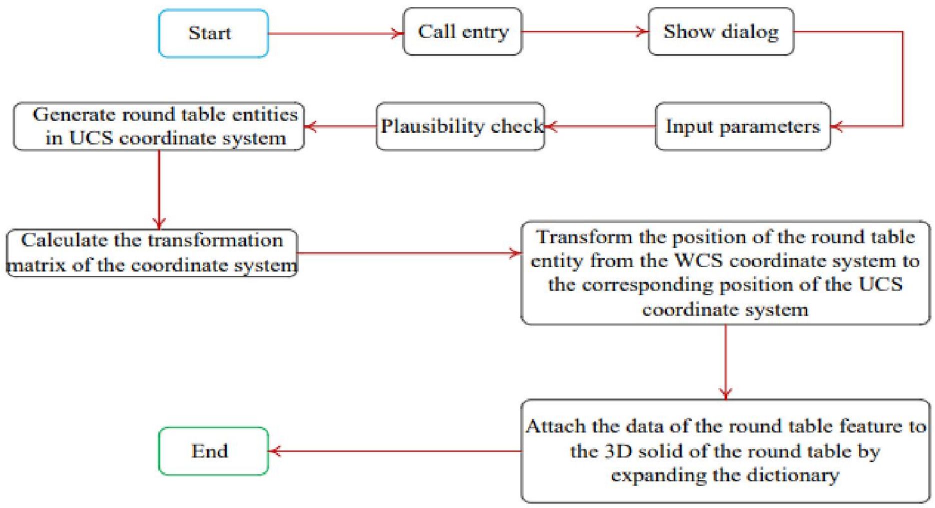


Fig3: Flow chart of round table

**B. Structure of Uniformly Distributed Tensile Body Features (Members) on a Cylindrical Surface**

Components (composite features) are some unit shapes with a certain universal significance and a certain degree of complexity, so the structure of the component shape is relatively complicated, which is composed of basic voxels and drawing features. Now we take the features of uniformly stretched cylindrical surfaces as examples to explain the construction principles of these relatively simple features. The input of data of uniformly distributed extruded body features on a cylindrical surface is based on the AutoCAD dialog box. Several parameters of the feature shape are determined and entered in the form of edit boxes. However, when determining the cross-sectional shape of the extruded body, you must enter the parameters of each endpoint. You must press the input button to save the endpoint parameters. The position of the feature can be selected either on the screen or by entering X, Y, and Z coordinate values. The shape of the uniformly distributed tensile body of the cylindrical surface is relatively complicated, and its modeling process can be divided into six parts. First, the rationality check of the characteristic parameters needs to be performed. Second, you need to generate a cylindrical base feature. Third, you need to generate extruded body assist features. Fourth, according to the angle value between each two extruded bodies, the required extruded bodies are copied at the accurate positions, and then the extruded bodies are combined with the cylindrical base features to generate a three-dimensional uniformly distributed extruded body features on the cylindrical surface shape. Fifth, the position of the feature entity is transformed to the corresponding position in the UCS coordinate system. Sixth, the feature data is attached to the three-dimensional entity of the feature through an extended dictionary. In the modeling of this characteristic entity, it is difficult to stretch the modeling of the body. When extruding a volume, you must first generate closed polylines based on the parameters of each end point of the extruded body's section, set the normal vector of the polyline, and then generate a region object through the pointer of the closed boundary. The object is stretched into a stretched body. According to the requirements, the parameters of the extruded body feature are uniformly distributed on the input cylindrical surface, and then a point is selected as the feature position on the screen, and then the dialog box is closed. The generated feature entity is shown in Figure 4.

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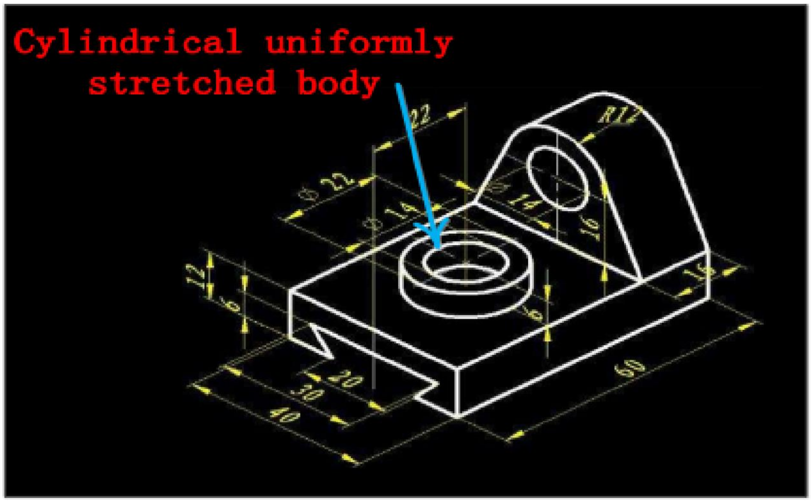

Figure 4: Example of uniformly extruded body features on a cylindrical surface.

**C. Create a Solid Model of the Object**

The solid model is the main body of the virtual projection system. To create a virtual projection system, you should first create a solid model. In order to make it easier for learners to understand, here we choose the common parts as the research object, as shown in Figure 5. The physical analysis of the entity can be regarded as a cutting-type combination obtained by cutting the cuboid body twice. In CATIA V5, this solid model can be constructed from three features, as follows: (1) Create the basic cuboid feature—Pad extruded feature; (2) Create front and rear through-groove features—Pocket extruded cutout features; (3) Create cutout features on the front and right end faces—Pocket extruded cutout features. The above Pad (Extrude) and Pocket (Extrude Cutout) features are based on sketch features. When creating these features, you first select the work surface, and then enter the sketcher workbench to draw the corresponding sketch, then return to the part design workbench to define and create features.

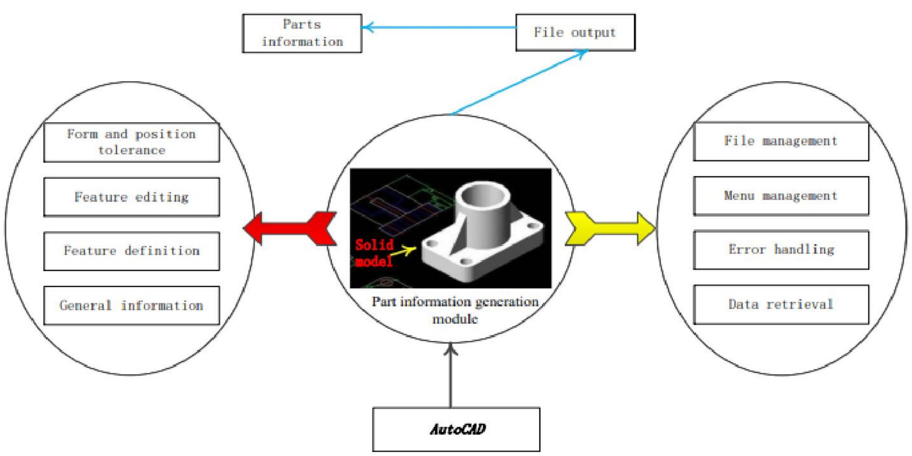

Figure 5: Part solid model and information function generation.

## V. CONCLUSION

Based on the systematic research and analysis of the three-dimensional structure of plastic parts, this paper establishes a feature library of plastic parts, which greatly improves the system's three-dimensional modelling capabilities. This feature not only has a three-dimensional physical structure, but also uses extended dictionary to carry the characteristic information, so that the designer can take out the characteristic information at any time or modify the characteristic information to change the physical structure of the characteristic. This paper proposes a method of creating "orthographic digital model of object" using 3D CAD technology. This digital model is a novel method for digitally interpreting the generation of orthographic projection views in engineering drawings. However, there are still many deep-level problems that need further research and exploration, such as the systematization of generative drafting standard patterns, the systematization of the GB standard parts library, and the methods for standardizing generative drafting methods.

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