



## Application of AutoCAD's 3D Modeling Function in Industrial Modeling Design

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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

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### 1 INTRODUCTION

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,

determine the reasonable structure and shape of parts, and store automatically generated engineering drawing files in the database. The file performs process design and NC machining programming, and directly controls the NC machine tool to manufacture. Bodein et al [6] believe that the purpose of CAD / CAE / CAM system integration is to provide a more efficient design / analysis / manufacturing integration system that covers a certain type of products. The processing power of a single computer is limited. Only through the Internet can resource sharing be achieved, and parallel design and collaborative design have a basis for implementation. Li et al [7] point out that a complex CAD system may itself constitute a computer network. It connects a server with multiple workstations. All public information required for design, such as graphics, data, parts, codes, etc., are stored in a public database on the server, and each workstation can share the data in it through the network. Zhang et al [8] say that workstations can also exchange the intermediate and final processing results required by each other through the network. CAD is the earliest and most widely used in the machinery manufacturing industry. Maddah et al [9] believe that using CAD technology for product design can enable designers to "throw off the drawing board", update traditional design ideas, realize design automation, reduce product costs, and increase the competitiveness of enterprises and their products in the market. In addition, this can also enable enterprises to change from original serial operations to parallel operations, establish a new design and production technology management system, shorten product development cycles, and increase labor productivity. Desgagné-Lebeuf et al [10] point out that the major aviation, aerospace and automotive manufacturing giants not only widely use CAD technology for product design, but also invest a lot of manpower, material resources and funds for the development of CAD / CAM software in order to maintain their technological leadership and the international market advantage.

Based on the establishment of the three-dimensional feature database of plastic parts and the database of plastic properties, this paper develops a feature modeling module. Through the three-dimensional modeling of the plastic part, the shape feature information obtained from the three-dimensional plastic part solid graphics and its attribute feature information are combined to obtain a full information model of the plastic part, thereby providing strong technical support for the design and manufacture of injection molds. And the processed plastic parts are used to form the molded parts.

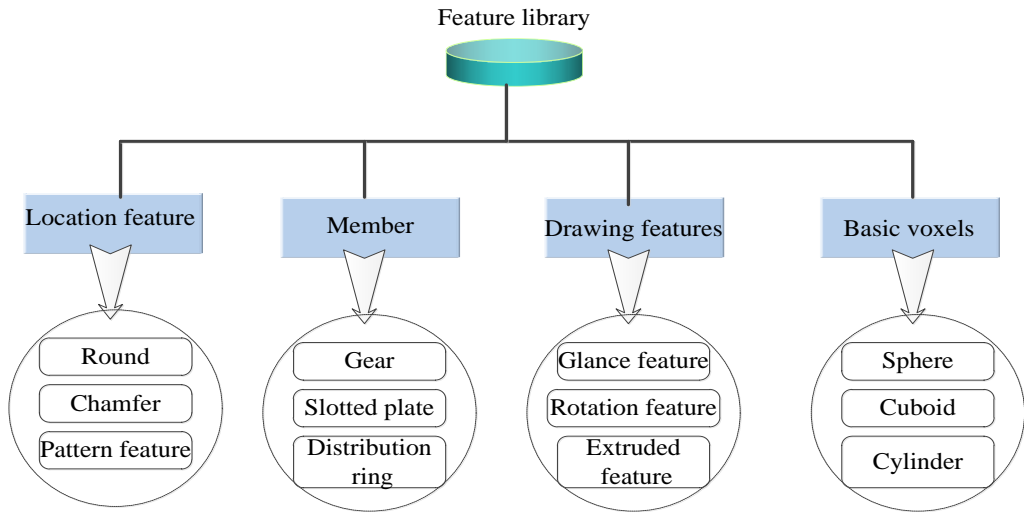
## 2 DESIGN OF PLASTIC FEATURE LIBRARY

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.

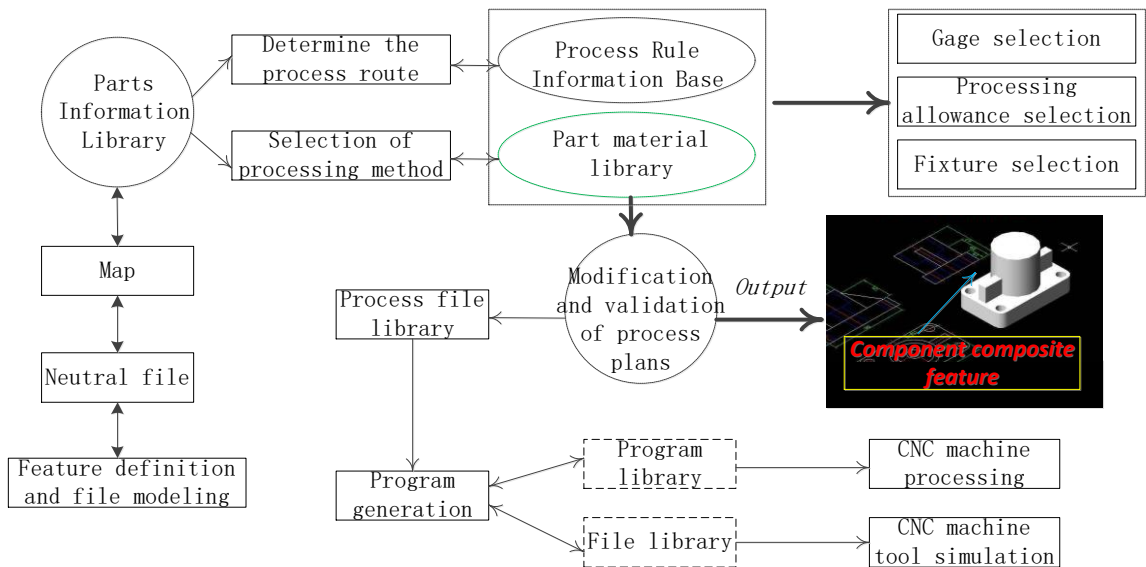
Basic features and drawing features are relatively simple geometric shapes. In practical applications, plastic parts are generally composed of basic voxels and drawing features. However, a careful analysis of various plastic parts shapes reveals that there are some more common and active shape features, 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 2.

### 2.1 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.



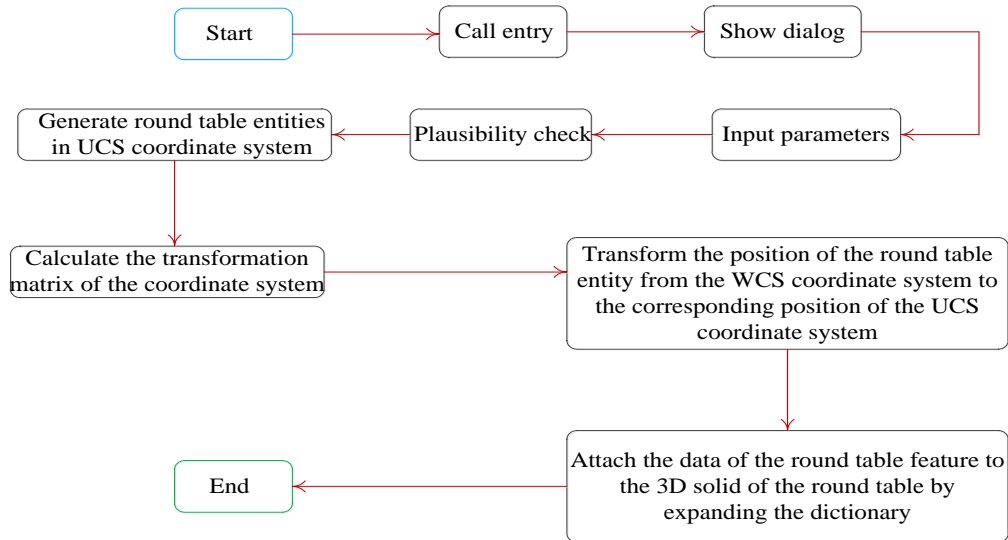
**Figure 1:** The hierarchical structure of the feature library.



**Figure 2:** Information system flow chart.

The shape of the round 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.



**Figure 3:** Flow chart of round table.

## 2.2 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.

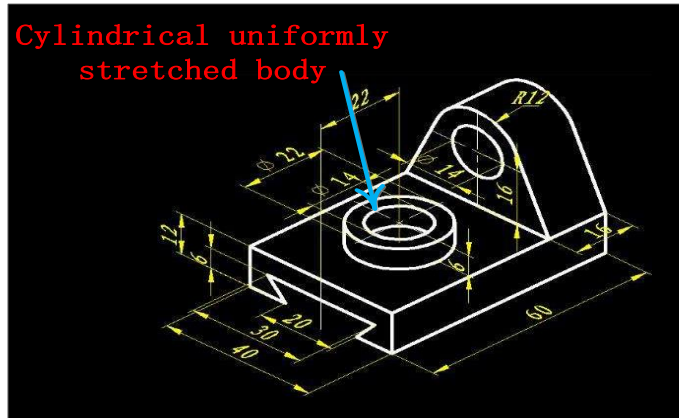
The input of data of uniformly distributed extruded body features on a cylindrical surface is based on the AutoCAD dialog box. Several parameters of the feature shape are determined and entered in the form of edit boxes. However, when determining the cross-sectional shape of the extruded body, you must 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.



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

### 3 RESEARCH ON AUTOCAD TO CREATE ORTHOGRAPHIC DIGITAL MODEL OF OBJECT

#### 3.1 Object Modeling Method

Part design uses feature modeling technology to build a three-dimensional solid model of the part. Features are divided into the following five categories according to the method of generating features:

- (1) Features based on sketches: Basic solid features formed by extruding, rotating, sweeping, and lofting 2D sketches;
- (2) Modified features: Features formed by chamfering, rounding, drafting, shelling, and adding threads according to process requirements;
- (3) Transform feature: A feature formed by transform operations such as translation and mirroring of existing solid features;
- (4) Surface features: For parts with complex geometric shapes, the solid features generated by obtaining the required surface shape according to the surface shape;
- (5) Boolean feature: A feature generated by a Boolean operation between existing entities.

Through various feature creation and editing commands, supplemented by the system's powerful parametric modeling capabilities, a 3D solid model of the part that meets the design requirements can be quickly established. According to the needs of modeling, the key parameters of various features and the constraint relationship between geometric elements can be parameterized, and the mutual relationship between each parameter can be established, and the part model can be quickly modified through parameter driving.

The basic process of solid modeling based on sketch features generally requires that the basic features be established first. For parts with complex structures, it is necessary to follow the solid configuration method (stacking method and cutting method), continue to add forming features, and then add necessary modification features according to the process characteristics of the manufactured part until the modeling is completed.

When designing from the top to the bottom, after entering the assembly design workbench, we first determine the assembly relationship framework between the various parts of the product, and then add and design parts one by one based on this. The design workbench designs each part one by one, and then enters the assembly design workbench to insert the existing parts and perform constraint assembly.

For products designed using the top-down method, the relative installation positions and fits between components are related. If you adjust the overall design parameters or change the shape and position of a component, it may cause the related components to fail. The law is suitable for developing new product designs.

Parametric design is a design method that uses a set of parameters to define the geometric pixel (voxel) size values and agree on the size relationship, which is provided to the designer for geometric modeling. The solution of the parameters is relatively simple, and there is a corresponding relationship between the parameters and the control size of the design object, and the modification of the design result is driven by the size. In the production, the structure and shape of the design object are often shaped. Serialized standard parts belong to this type.

Modification of the size parameters will result in a complete update of the relevant dimensions in other relevant modules. The reason for using this technology is that it completely overcomes the unconstrained state of free modeling, and the geometry is firmly controlled in the form of size. If you want to modify the shape of the part, you only need to edit the value of the size, achieving a change in shape.

Variable design is a design method that uses a constraint-driven approach to change the geometric model composed of geometric constraints and engineering constraints. Graph theory and stable numerical solution techniques are often used to solve geometric constraints. Variable design is to achieve the goal of variable design by actively applying constraints, and to solve the problems of under- and over-constraints.

The variable modeling technology retains the advantages of parameterized technology based on features, full data correlation, and size-driven design modification, but makes a fundamental change in the definition of constraints. The variable technology further distinguishes the size "parameters" defined by the parameterization technology into shape constraints and size constraints, instead of constraining all geometric figures with only the size as in the parameterization technology. Because in the conceptual design stage of the development of a large number of new products, the designer first considers design ideas and concepts and reflects them in certain geometric shapes. The exact dimensions of these geometric shapes and the strict dimensional positioning relationship between the shapes are still difficult to fully determine in the initial stage of the design, so naturally it is hoped that the existence of under-constraints is allowed in the initial stage of the design.

### **3.2 Create a Solid Model of the Object**

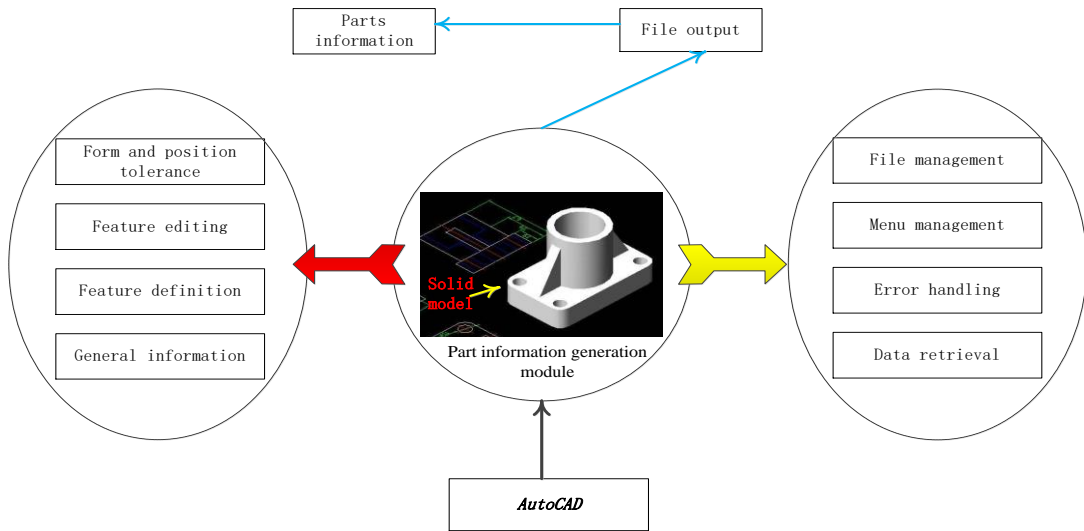
The solid model is the main body of the virtual projection system. To create a virtual projection system, you should first create a solid model. In order to make it easier for learners to understand, here we choose the common parts as the research object, as shown in Figure 5.

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.

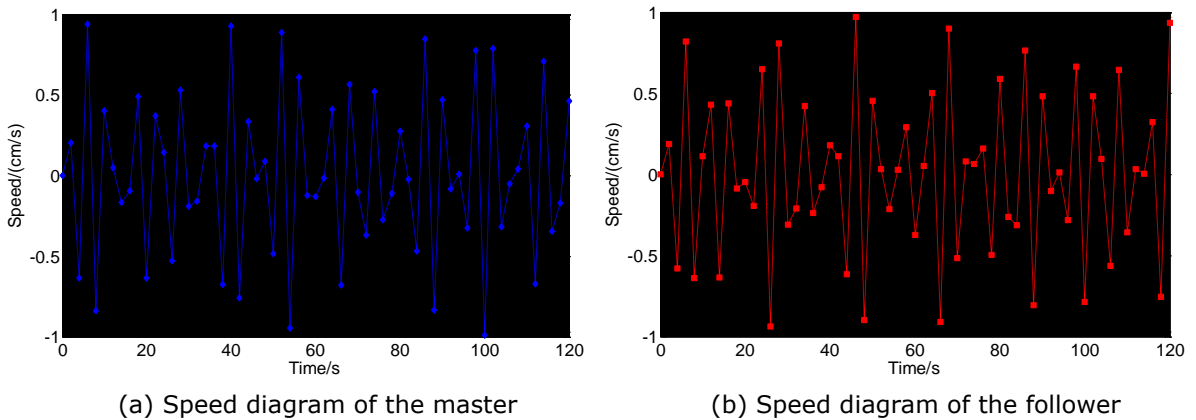
The above Pad (Extrude) and Pocket (Extrude Cutout) features are based on sketch features. When creating these features, you first select the work surface, and then enter the sketcher

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.

In order to facilitate the modification of the model, it can be parameterized and variable. For example, when designing a sketch, you can define parameters and add parameters to some graphic elements that may be modified later, so that you can modify the model by modifying the parameter size. Speed diagrams of the master and follower are shown in Figure 6.

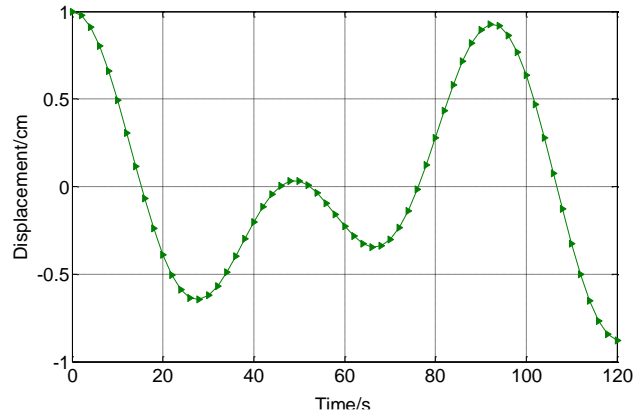


**Figure 6:** Speed diagram of the master and follower.

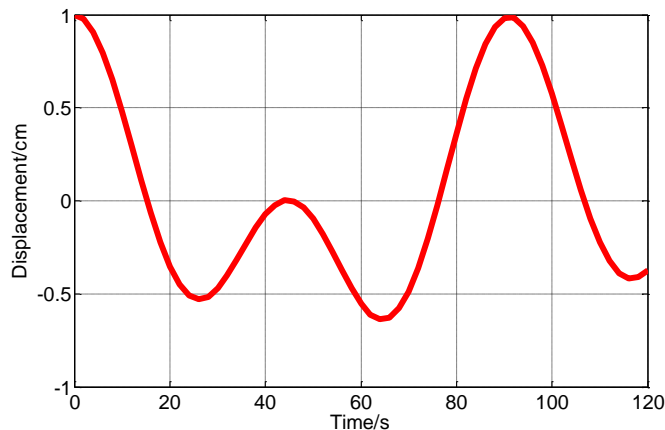
### 3.3 Method for Creating a Digital Image of an Orthographic View of an Object

After obtaining the object and its virtual three-projection plane system, if the projection view can be displayed on the three projection planes, it should be said that a complete digital model of the virtual projection system is created.

In order to better simulate the projection of objects and allow learners to understand the drawing rules in the view, the solid display mode is used to obtain the front view, left view, and top view of the solid model. The view in this display mode not only shows the shadow cast by the entity on the projection plane, but also the visible outline (thick solid line) and invisible outline (dashed line) of the view on the pattern. Finally, the system's built-in image capture command was used to capture the graphics of the three views and save them as corresponding graphic files for mapping projection views on the virtual projection surface. The displacement diagrams of the master and follower are shown in Figure 7.



(a) Displacement diagram of the master.



(b) Displacement diagram of the follower.

**Figure 7:** Displacement diagram of the master and follower.

### 3.4 Orthographic View of the Mapped Object on the Virtual Projection Plane

Based on the above work, in order to be able to simulate the orthographic projection of an object in the constructed virtual projection system, it is sufficient to display the orthographic image of the object on three virtual projection planes. To this end, the graphic files of the three solid views that have been prepared in advance are imported as background maps corresponding to the three virtual projection surfaces, respectively, and the work of mapping the projection views on the three virtual projection surfaces is completed. The result of the virtual digital model of the three-projection plane system is shown in Figure 8.

After the view file is mapped on the virtual projection surface, the positional relationship between the entity and its virtual projection may not meet the visual orthographic projection

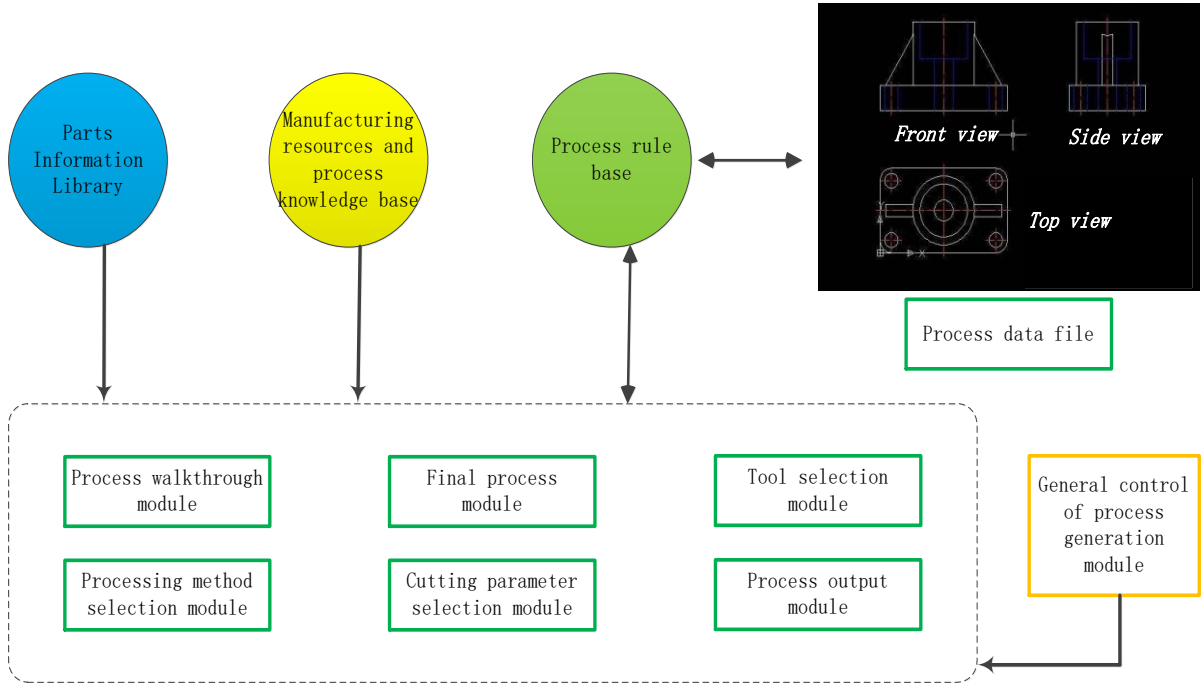


effect. It is necessary to further adjust the three virtual projection surfaces from the top, bottom, front and back, and left and right positions of the solid model, and the size of the virtual projection background map and its relative position with the solid model need to be changed simultaneously so that the relative position between them approximates the orthographic relationship. In general, in order to obtain a relatively satisfactory realistic projection effect, you need to try to recapture the graphic file of the view and redefine its aspect ratio.

It can be seen that compared with the conventional graphic orthographic model, the virtual digital model has the following characteristics:

- (1) The virtual projection of an object is not just an abstract contour figure, but also has a projection and a contour ridge that reflects the projected view;
- (2) Visible edges on objects are shown as thick solid lines, and invisible edges are shown as dashed lines.

The virtual object orthographic digital model can more realistically and intuitively show the process of generating orthographic views of real parts, and has the characteristics of strong reality, intuitive understanding, and vivid image. This method of creating a three-sided orthographic digital model of an object is a novel method that uses digital means to interpret the orthographic view generated in an engineering drawing.



**Figure 8:** Virtual digital model of the three-projection plane system.

**4 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 modeling 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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