A Convenient Part Library Based On SolidWorks Platform

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Abstract—3D part library is an ideal approach to reuse the existing design and thus facilitates the modeling process, which will enhance the efficiency. In this paper, we implemented the thought on the SolidWorks platform. The system supports the functions of type and parameter selection, 3D template driving and part assembly. Finally, BOM is exported in Excel format. Experiment shows that our method can satisfy the requirement of die and mold designers.

Keywords—Intelligent, SolidWorks, automatic assembly, part library.

I. INTRODUCTION

IN the traditional mold design process, the designers are doing complex and tedious work. The efficiency and quality cannot completely satisfy the demand of the rapid development and improvement in industry. Designers usually have to do the same or similar work repeatedly, which brings much inconvenience and mistakes. If most of the parts used in the mold are standardized and a perfect part database is built, it could provide the designers an intelligent design system of standard parts for modeling, which can save the time and costs for the mold maker [1]. On the other hand, Solidworks is becoming more and more widely used in industrial production in the world. As one of the most mainstream Computer Aided Design software in the market, Solidworks can be expanded by the third party very well. It is well known that Solidworks itself cannot satisfy all the need of intelligent in mold design. Therefore, the research and development of intelligent design system for die and mold based on Solidworks is necessary.

It is a good conception to build a part library based on SolidWorks. The research holds some theoretical and practical meanings. Firstly, it combines the traditional industry technology with the advanced computer skills so that it can enhance the design level and improve the ability to develop new products. Secondly, the research emphasizes on the development based on Solidworks and it is contributed to the expansion of mold design on various platforms and serve different purpose of the business. Thirdly, it aims at the utility and legerity and optimizes the process of mold design and reaches the intelligence, which brings handy greatly. Although there are many standards to build and share part libraries, e.g. ISO13584 (an international standard for the representation and exchange of CAD part libraries), many companies still want to establish their own dedicated part library system which are in line with national and enterprise's standards and integrated

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seamlessly with their design environment for improving product design efficiency. Yang et al. [2] developed a CAD part library in agreement with ISO 13584 on WinSTEP. Gu et al. [3] developed a typical part library based on UG. Wu et al. [4] implemented a mechanical standard part library on SolidEdge. Yan et al. [5] develop three-dimensional standard part library of riveting joint on Pro/Engineer. Kim et al. [6] constructed a 3D CAD library for press die design system on Catia.

II. ANALYSIS AND DESIGN

A system in accordance with the designers' quests should take the running environment, the operating system, the database system, the object and also the function demand into consideration. For the first and most important issue, the system should have good stability so that it can run in very different condition and can react immediately to the possible fault. It must keep running well and guarantee that the data of guest are protected from damage and lost. In addition while mistake appears, the system must report the actual problem and make the proper treatment. Next, a friendly interface should be established and easy to be accepted by the users. On the other hand, many data are come into being with the system running. Too much data may lead to the reduction of the speed of computer and system and the trouble while looking for necessary information. So a good data management should be accompanied with the system. Then the system should be able to expand quickly to reply to the change in mold species and the increasing number of standard parts as well as the standard. Last but also essential, the system should be accurate and fast. Even a little deviation in design may cause a large waste of costs and time as well. As a result, abundant amount of tests should be made in order to ensure the accuracy.

According to the analysis above, the system should have several functions. Firstly, a template library should be established. In mold designing, the utility of standard parts could reduce the cycle of design process and enhance the efficiency. Meanwhile, a good template library will permit the user call and modify and parameterize the part quickly. The establishment of standard parts library is characterized as common and easy to parameterize and be modeled. Secondly, the standard part will be additionally processed for the reason that many parts in mold are similar and the only difference between them is that some of the parts are cut or some assemble holes is punched. The additional process is good for declining the amount of template so that the parts can be called easily. After getting the proper template, user should import the parameterized part into the assembly and put it onto the appropriate location, preparing to be assembled. However, the

fact cannot always be perfect. Not only the location but also the direction sometimes does not fit the final gesture in the mold. The user is able to change the direction and position after importing. Then, to become a complete product, all the parts should be assembled together. In Solidworks and some other CAD software, assemble often comes with many steps such as selecting the element, choosing the right way of mate and so on. Fortunately, the secondary development provides a good platform to solve this problem. And the pattern to add mates in molding can be concluded to a common form so that we can simplify the move. Finally, the bill of material is an essential part in production designing and manufacturing, which can directly show the material, the amount, the producer and many other information of parts in a mold and other goods, etc. When finishing the design, the system can automatically read the specific property and write it to the database like Excel used in the research. These are the main function of the intelligent design system for molding based on Solidworks.

III. System Architecture and Functions

Our system is divided into three subsystems. The first one is the type-parameter selection subsystem, the second is the 3D modeling subsystems and the last one is the plug-in subsystem which is embedded into SolidWorks.

The type-parameter selection subsystem (T-PSS) can either run independently without CAD software or be used as a server and embedded in clients. It provides the information about the type and related modeling parameters. Its main target is to help users select the type of the standard part and choose the characteristic parameters. It is comprised by the part library and the some useful method to make choice. In the light of the function, the interface has three main classes. In the first step, users can use individual ways to choose the templates according to the picture or the name or some superior ways. In the second order, users can select the exact model depend on its character, material. And in the next stage, users can parameterize the chosen part. 2D and 3D model are shown to tell the dimension.

3D modeling subsystem is responsible for driving geometrical templates according to the parameters received from the former subsystem.

The third one aims at bridging the gap between the first two. It is invoked by users' operation on menus which are embedded into Solidworks platform, and it connects with the CAD system through dynamic link. This plug-in system is based on the Solidworks API SDK to make the menu in Solidworks in order to make the system like one part of the CAD software and let users operate more easily. There are also some options related to the function of the system. The CAD system, the tool to make mold has to make true the function of the system. Secondary development based on Visual Basic is the main method to customize the interface. The task CAD system should fulfill is to transform the format of the part, cut the part, rename the part and copy the part file to the appointed folder, open the part file and add the component to the assembly, move and rotate the component in the assemble environment, add mate to the assembly and create a BOM table. The system

handles the main function in designing with Solidworks including parametric molding, man-machine interaction system and so on and finally builds an intelligent design system for molding. Although the interface system has been loaded on some other CAD software also by us two years ago and some of them has put into use, it still remained empty in Solidworks.

The research has separated the program into small and absolute unit and studied and researched one by one. At last, they were combined together and become an entirety that can run fluently. The secondary development of Solidworks is based on a relatively easy, widely used and powerful programming language, Visual Basic. Solidworks has provided a wonderful exploit tool and interface to link the software and human together. Any original move in Solidworks can be replaced by a function. In the importing module, the key point is to transform the format and the function is like load, saveas, opendoc, filecopy, and filename and so on. All the function has its own definition and parameter so that it has its own usage. In parameterizing module, the main idea is that the shape of the template does not change while the size changes. We can have the part by only making over the characteristic dimension of the template in the library. A small system was built to simulate the large one and some standard parts are used to imitate the true ones. In the program, dimensions are taken out from the model and could be replaced by assigning. One of the functions in Solidworks can help the intelligent cutting. It is the compress of the cutting model. If the cutting has to be shown, the cutting is removed from compress. It can merge the template that the difference only exists in small place and can reduce the amount of the template. The rotation and translation of the component uses the transform matrix. However, there are some other ways of rotation that can reach the same result. The mate of the vision coordinate system is applied in the research. In adding mates, user must select the element as well as the type. The type in mold design is somewhat single while it can be more intelligent. The system can detect the selected object type and mate type according to the type of the part and then choose the way to coordinate.

The human-computer interaction can make the system more intelligent and Solidworks is available for it. Through the handler can monitor user's operation and make reactions. Traversal is the main idea of creating BOM table. Reading all the property in assembly tree and filter out needed ones and then writing into the database is the way. Arrays are used to record the number and property. At last, the system is designed into the menu of Solidworks so that users can open it quickly. VB addin provides such function can users can customize their own menus.

The system basically meets its demand of intelligent and can bring much convenience to the designers. However, it also remains some problems to improve in the future. For example, the stability and the ability to deal with the unexpected fault is not good and more tests should be made. In addition, more function can be added into the system such as generating the dimension of template according the mating elements and so on.

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IV. IMPLEMENTATION AND EXAMPLES

Based on the above description, we have developed a part library based on SolidWorks for both die and mold part. Firstly, menus are created by SolidWorks APIs, as shown in Fig. 1.

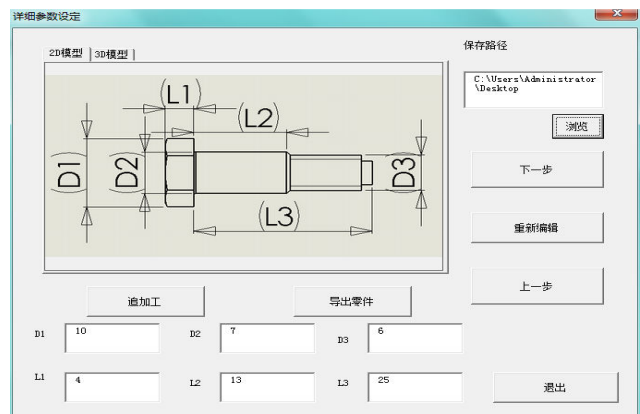Fig. 1 Menus for plug-in module of system

The T-PSS can be processed into three steps: the first step is to select the class to which the type belongs, and the second step is to select the actual type form the class while the last one is to decide the parameters of this type, the whole steps are shown in Fig. 2.
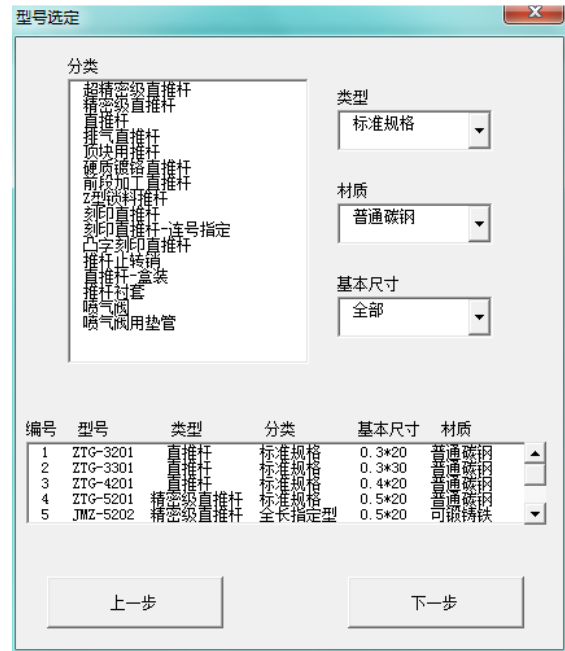
(a) Class Choice
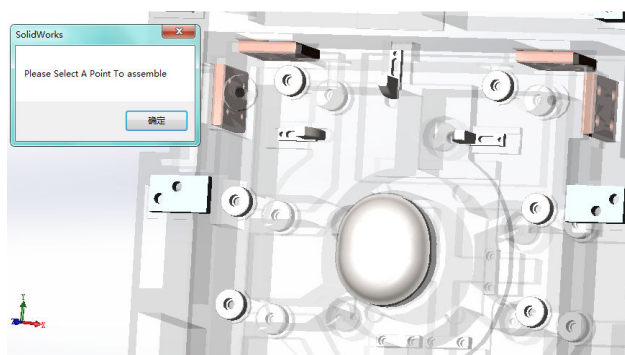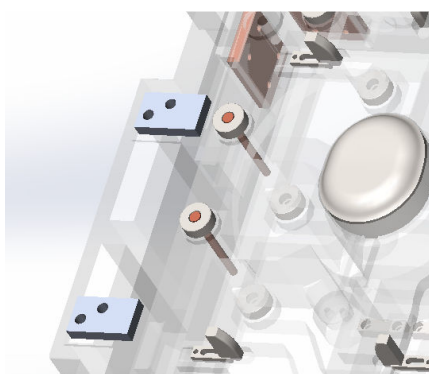

(b) Type Choice

(c) Parameter Choice

Fig. 2 Type-parameter Selection Subsystem

After all the type and parameters are decided, the driving subsystem will send the parameters to the template for this type, and then the finial model is created and imported into SolidWorks. The user has the initiative to select a position to insert the model and the related constraints will be automatically added to form a rational assembly as shown in Fig. 3. The proposed system also support several file formats to be chosen such as .stp, .iges, .x_t, etc.

The chosen example is an ultraprecise straight push rod. The system read the user's selected types of parts and related parameters to automatically convert the driven template into STEP parts, and then the file is processed by translating, loading and storing to obtain the required Solidworks components and is imported to the assembly document. The inserting position is decided by the user through the pickup by mouse and system will finish the assembly stage.

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(a) Select the Position to Insert

(b) Automatic Assembly

Fig. 3 Assemble the model into SolidWorks

Repeat the above process until the design has completed. The system supports the function to output the BOM in Excel format. The system will automatically traverse the assembly tree to get the part attribute needed and write them into the Excel file, several customized items such as type, order, material and quantity, etc can be listed in the table shown in Fig. 4.

Fig. 4 BOM Example

V. Conclusion and Further Work

In this paper, we propose the method to implement a part library on SolidWorks platform, by which it is very easy to generate general part according to practice and assemble it into the model. With the tool, the users can easily put the desired parts into their designing models, which will greatly quicken the process and improve the efficiency. In the further, we will further investigate and support more convenience and usability of the library to the user. Firstly, the class the quantity will be greatly increased; secondly, the user interface will be friendlier, and finally, more CAD platforms will be considered to be transplanted on for those designer whose work on Pro/E, UG, CATIA, etc.

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