Development of Three-Dimensional CAD System for Buoy Hatch Library

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1. INTRODUCTION

It is common to know that buoy is a small-scale water equipment used for data investigation, environment monitoring, weather fore-casting and scientific experiments, etc. The general buoy's physical design considerations are to have storage to store batteries and sensor systems, and appropriate spaces for workers to work inside. Consequently, hatch is a requisite part of buoy which used to act as a waterproof door.1

The mechanical design of the buoy must design to be watertight to protect the system inside from destruction. Hence, the hatch must be well designed. Considering the small size and variety shape of the buoy, a parametric system for buoy design need to be construct to ease the design tasks. Fig.1 is the three-dimensional model of a hatch, which designed using the CATIA environment. By utilizing parametric design, the hatch design can be change to any desired form easily by only changing several main parameters.

The major concern of this research is to simplify the hatch design steps to ease the designer in their design jobs. Utilizing the hatch library, the designer able to avoid the repeated hatch design steps which due to the standard hatch parts already provided by the library.

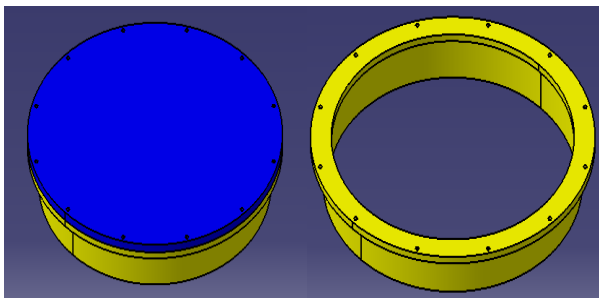

Fig.1 Simplified assembly drawing(left) and main wall body(right) of small scale round shape hatch

This research used CATIA as the main design tool. It is an intelligent software system and widely used in Aerospace, motor vehicles and navigation design area. 2 It support powerful parametric design function, designers can build their own parametric feature library to simplify the design process and fasten the total buoy development time. This paper will take small buoy hatchway design as example and to analyze it in details to accomplish the development of hatch three-dimensional CAD system.

This research study is important for number of reasons. First of all, if take comparison to the traditional design process, there is no necessary for designers to do repetitive work when there are only one or several small parameters need to be changed. Secondly, the study here can solve the problems of serious structure changes leading by small changes of parameters. Thirdly, we will find a common method to build the standard parts library for the parts with same external structure but have different parameters. At the same time, this research will make the library more organize and easier to be modify.

2. MAIN DATA FLOW OF DEVELOPMENT OF THREE-DIMENSIONAL SYSTEM

There are two common methods to build a three-dimensional standard parts library under CATIA platform. One is using commands of formula, design table and catalog to create their own three-dimen-sional Standard Parts Library and another one is to use secondary development interface in programming language development environment. This research will focus on the first method.

The development process was mainly made up by two steps. The first step is to build a parametric model, and the second is to build an integrated catalog as a library. The flow chart below demonstrate the main hatch library development process.

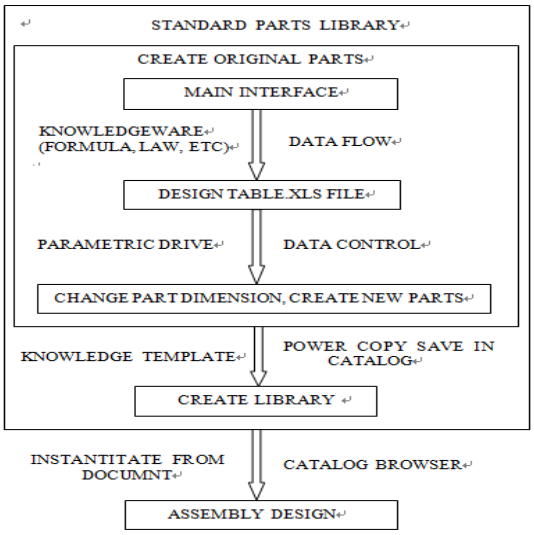

Fig.2 Flow chart of Standard parts library development process

3.PARAMETRIC DESIGN OF ORIGINAL STANDARD PART

Beforehand, a three-dimensional model part need to be create and it will be used as a standard part in the standard part library. Another parts with the same structure but different parameters can be generated only need to change several main parameters. For the main wall of buoy hatch design, command formula can be use to define the wall thickness, the height, the wall diameter, diameter of holes, and width of rim respectively. The parameters and equations will be listed in the structure tree.

After creating the standard part with these parameters, the following step is to create the design table with these data. By using the command design table, we select the option "Create a design table with current parameter values"and filter the main variables defined in parametric design. Save it and the design table will be create successfully.

Double click design table icon in structure tree under relations, the interface with variable data will display as Fig.3. Click icon edit table, the data file will be open and until now related data with all kinds of situation can be added in this file(here we select excel file as the default data file).The final design table will display

automatically as Fig.4 .If we select different data combination, different parts will be auto-created.

4.CREATE PART LIBRARY AND INVOKE PARTS

In order to put the standard part into a library, the commands power copy and save in the catalog are used. During creating the power copy, we should take notice on the referring base when inserting this part in the next invoking. After creation of a catalog, command part family should be added in this catalog and it will include all kinds of parametric parts created in section 3.

Catalog browser is the command PartFamily.1 of invoking parts from standard parts library. When we insert the standard part in assembly module, the referring base should be considered.

The whole development process of standard parts library will complete here.

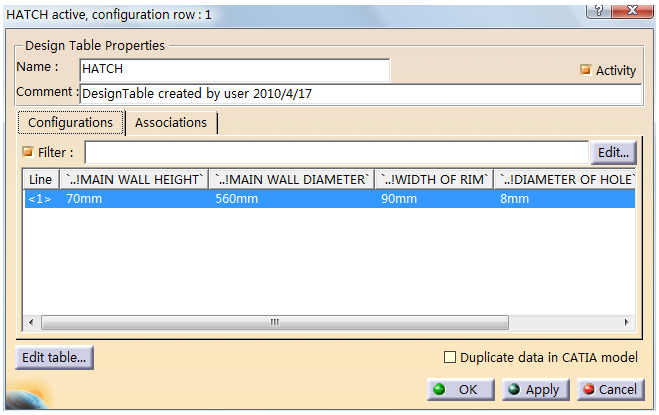

Fig.3 Design table properties box with part parametric variables

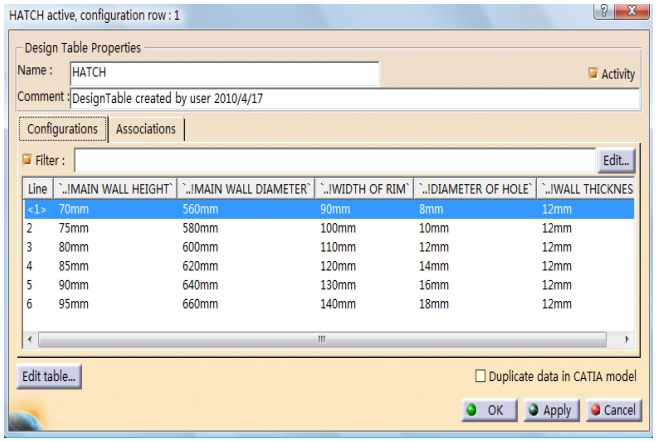

Fig.4 Final Design table properties box with parameters of all kinds of parts

4. CONCLUSION

With the combination of parametric design and building of catalog, standard parts library for small-scale buoy hatch can be construct. This method lead to various convenience to the designer, and should be consider as a priority design step in the buoy design process.

Firstly, the standard part library saves time and avoids repetitive work. 3 Designers can avoid building another part from the first step to the end if this part has a similar outside structure with the existing parts. If they are not exactly the same but only have several similar properties, it also works to invoke these standard parts only by changing some relations.

Secondly, designers can collect and add any kinds of new parts into the parts library in this method. And this makes the parts more organized, systematic and easier to be found.

Thirdly, it is kind of cumulative drawing experience. As we put the parts with the same properties in the same item of the library, it will be more convenient to analyze the features of these parts.

Last but not least, with this method, the production process will be more automatic and it's also helpful for the optimization of production process.

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POSTSCRIPT

For the product design, we invoke the standard part (hatch wall and cover) separately from parts library and assemble them with certain relations. However, if we consider using publication command in assembly module, It is possible to get the parameters data as a common data in the whole product. Then it will create a reference for the next design of hatch cover. If hatch cover uses parametric design with the publication data, the whole product will be related by the data we defined in the parametric design section. Therefore the whole product will be main parameter driven by the data in design table. This method can run successfully in the single product design which means design two part separately, and assemble them one by one. Therefore, it still needs further research on this assumption if it is possible or not to combine the above method with standard parts library. This is one of the next research directions.

On the other hand, there is another method to build the three-dimensional parts library. It is the second development of CATIA. Besides, it will also be another further research directions.

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