

Customization of Catia V5 for Creating Different Types of Holes on Disc Wheel

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Abstract- This paper includes the customization of Catia V5 by using macros (Program code written in certain programming language) for creating different types of holes on disc wheel. i.e. how the three dimensional solid modeling software (like Catia V5) can be used to automate the repetitive tasks. The input data from user is used to create holes on disc surface in part modeling environment of Catia V5. Generated part model can be used to draft different views of the disc wheel which can directly be used as production drawing

Key words- customization of Catia V5, macros, automobile disc wheel.

I. INTRODUCTION

Automobile wheel include two main parts welded to each other, which are the rim and the disc. Tyre is fitted on the rim and wheel disc serves as connection between rim and wheel hub. While modeling the disc of wheel we have to create different holes on disc surface. Three types of holes are required on simple disc surface - Centre bore hole, Stud hole, Vent hole.

Types of stud hole:

- 1) Simple stud hole
- 2) Conical stud hole (uni or bilateral)
- 3) Spherical stud hole (uni or bilateral)

Macro means program written for specific task. For developing advanced macros for special needs Catia V5 is an open system. Macros may be useful for creating, analyzing, measuring, modifying, translating, optimizing surfaces, solids, wireframes and more. Macros are useful for part operation, assembly operation and all multidisciplinary applications. Languages used for developing code are VBScripts and VBA. For creating macros for holes we need inputs, outputs and necessary supporting data. First we create basic architecture and process flow of the program code then we develop the code and verify it for different combinations.

II. RELATED WORD

Eric Bush [5] explains that about 90% of the generative shape and part design workbenches can be duplicated in macros. His work explains about recording of macros for automate the repetitive task. A macro can do any task repeatedly without taking a break or pausing to make complex decisions.

Maria Gloria, Del Rio-Cidoncha [3] and co-authors describes how automation in solid modeling can be approached by using macros for some tasks using Catia V5. Modeling of solids in Catia V5 can be done by three tools-Power Copy, User Feature and Macros. Catia V5 offers the possibility of automating tasks (Customization of software for particular task).

By using these three tools we can customize/automate Catia V5. Since macro has advantage over other two, macros are used mostly. Creation of user interface forms and modules increases flexibility in Visual Basic Application.

III. DISC AND HOLES

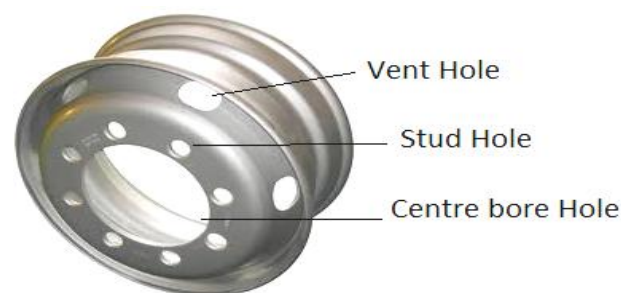


Fig. 1. Disc Wheel

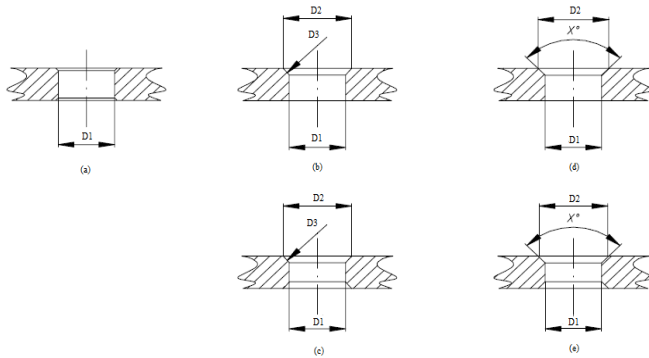


Fig. 2. Types of Stud Hole: (a) Simple, (b) Conical uni, (c) Conical bilateral, (d) Spherical uni, (e) Spherical bilateral.

Disc includes three types of hole already named. Stud hole include three types shown in Fig. 2. For individual type separate code is written and then assembled together. Stud hole contains uni or bi-lateral countersink. Vent holes are provided for ventilation and mass reduction. The Holes created are as per EUWA standards [6]. While modeling, creation of vent hole on curved surface is somewhat complicated task. Because for vent hole creation we require inclined plane and proper references.

IV. CATIA V5

CATIA (Computer Aided Three Dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by French company Dassault Systems and it is marketed world-wide by IBM. Catia is the world's leading CAD/CAM/CAE software. For developing advanced macros for special needs Catia V5 is open system. A macro is a series of functions, written in a scripting language, that you group in a single command to perform the requested task automatically. These macros may be useful for creating, analyzing, measuring, modifying, translating, optimizing surfaces, solids, wireframes and more. Macros are used to save time, reduce the possibility of human error by automating repetitive processes, standardization, improving efficiency, expanding Catia's capabilities, and for streamlining tasks. For creating basic structure and basic flow of program we require inputs, outputs, and supporting data from the user.

CATScript, VBScript and VBA (Visual Basic Application) are the languages used for coding. Each has its own advantages and disadvantages. For present application we select VBA. Access to the Catia object model is provided using scripts in different ways depending on the operating system and on the application that can share their own objects with Catia. The CATScript language is a Catia specific language which was designed to allow compatibility between the Unix Basic Script engine and the Windows VBScript engine. As Catia doesn't anymore hosts the basic script engine on Unix since V5R7, it has only been kept for compatibility purpose. It is actually processed by a VBScript engine after removal of the typing information.

V. CATIA V5 MACROS

A macro is a series of functions, written in a scripting language, that you group in a single command to perform the requested task automatically. In simple it is a piece of

code written in certain programming language which groups a set of operation that defines a certain task. For each task separate code is written and assembled together by using forms in VBA. Some of the macros we developed:

- Centre bore hole macro
- Simple hole macro
- Conical one side macro
- Conical two side macro
- Spherical one side macro
- Spherical two side macro
- Form showing macro for centre bore and stud holes
- Circular vent hole macro
- Form showing macro for vent hole.

CATIA Customization/Automation Objects

Catia provides customization capability. In Catia the part objects, which are used for developing part model i.e. three dimensional object are structured under a automation tree [2] as shown in the Fig. 2. As and when needed the part object can be extracted with the macro which is programming code for customization or automation of Catia V5. This part document automation object tree gives the general idea about customization structure.

Part Document Automation Objects

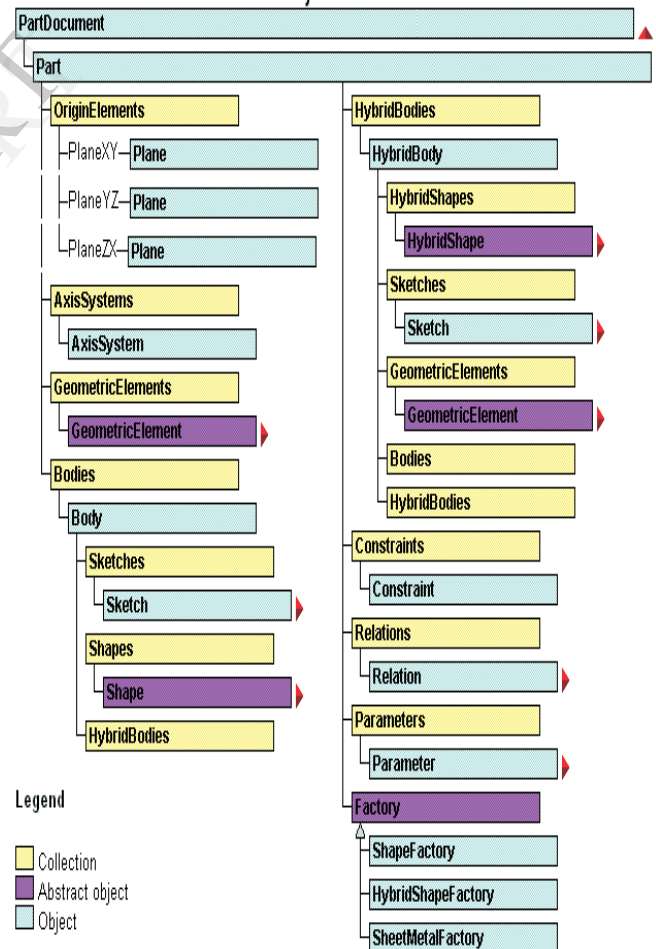


Fig. 3. Part Modeling Object Tree

The Part Document object contains, or includes, the part tree structure starting with the Part object located at the top of the part specification tree. These Part Document objects are: OriginElement, AxisSystems, GeometricElements, Bodies of Part objects are: Sketches, Shapes, HybridBodies, HybridShapes, Constraints, Relations, Parameters, Factory.

VI. METHODOLOGY

Generation of three dimensional part model

For creating holes on disc surface user has to import disc geometry. User has to select few inputs like type of hole, type of stud hole, required number of holes, spacing between holes, thickness of disc, required dimensions of holes. With this, all parameters are send to a subroutine to create holes on disc surface. In normal procedure for hole creation, user has to create planes at different stages. Every time he has to start from basic, drawing a sketch is repetitive process. By using macros we can complete modeling of disc wheel in short time.

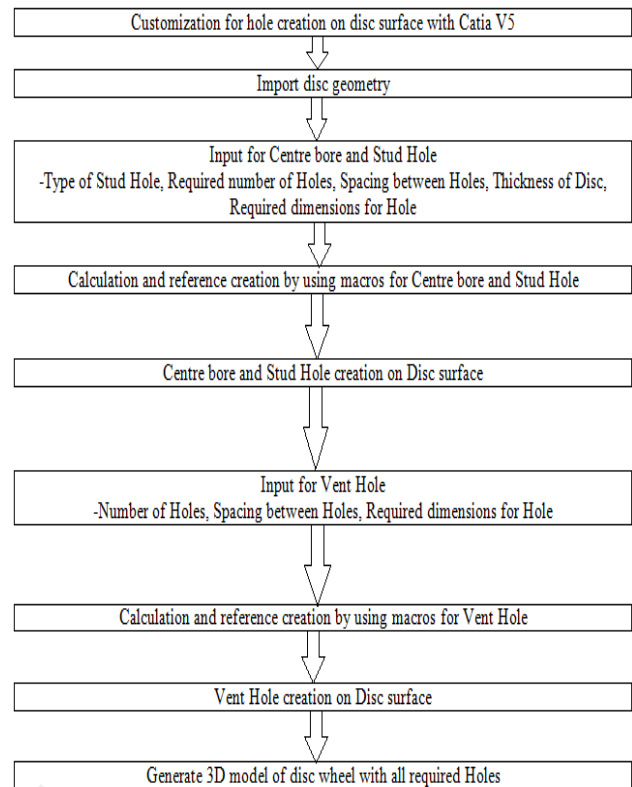


Fig. 4. Customization Information Flow

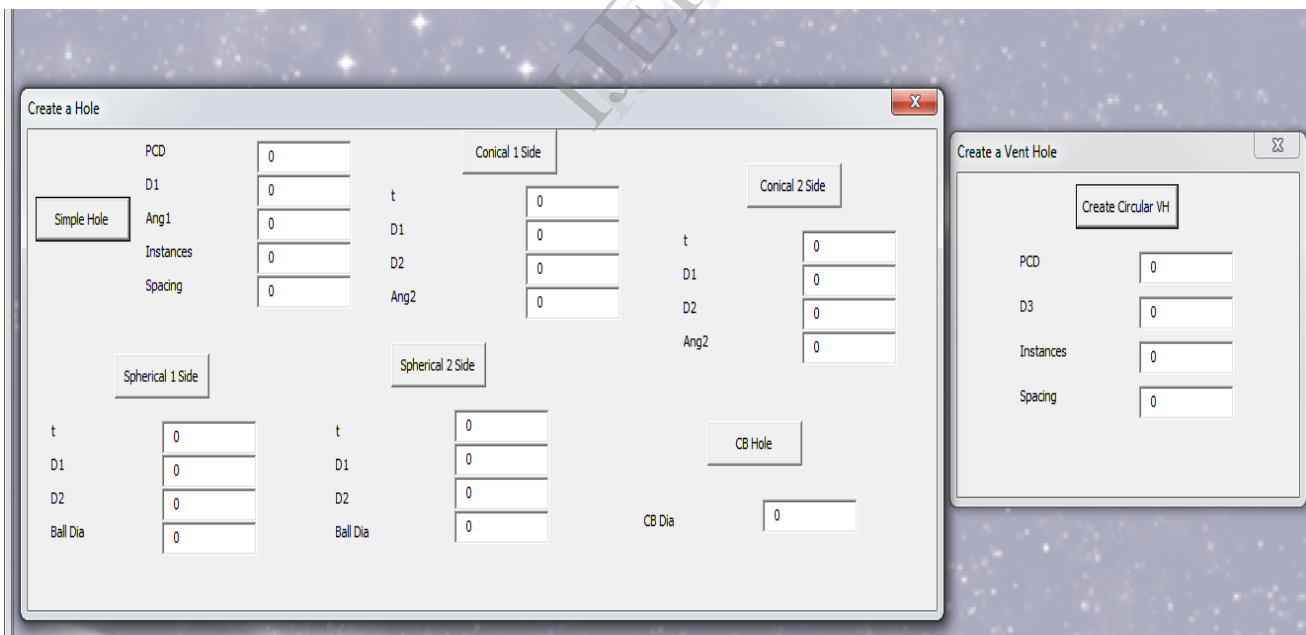


Fig. 5. Input Parameter Form

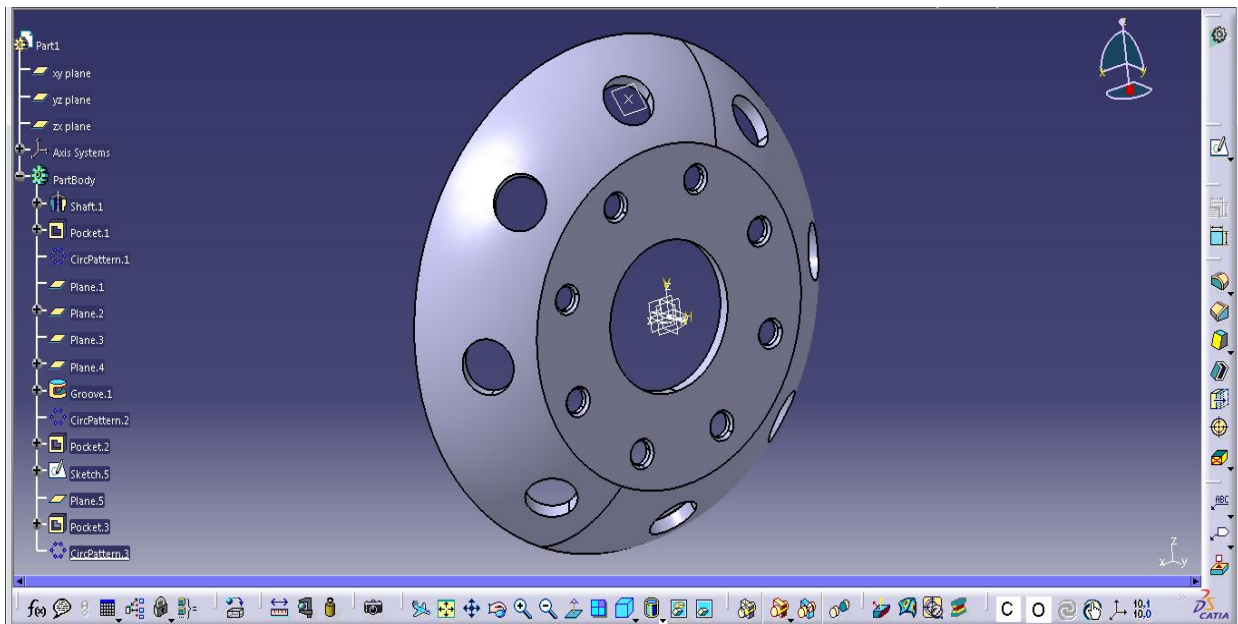


Fig. 6. Holes Created on Sample Disc

VII. CONCLUSION

Holes are created using macros. With this time required for creating holes on Disc surface and generating part model (Three dimensional model) of Disc Wheel is reduced to few minutes. This part model can be used to create different views of the Disc and then Disc Wheel which can directly be used as production drawing. Customization/Automation reduces the possibility of human error along with standard procedure is set for modeling. Thus, this customization/automation reduces modeling time which in turn reduces lead time.

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