

Research and Implementation of CATIA Tool Integration Technology Based on CAA

Abstract: In order to implement the integration of the tool libraries from CATIA and the tool database, CATIA software was further developed using CAA. CAA macro-based integration project about the two libraries is proposed, and the development process is presented. In CAA environment provided by CATIA, though further developed using CAA, calling for information, converting information and valuating information of CATIA and TOOLMANAGER were researched. Finally, the dynamic calling, association and driving of CATIA-based tool information were implemented successfully.

Streszczenie. W artykule analizuje się możliwości oprogramowania CATIA służącego do projektowania typu CAD/CAE/CAM. W analizie wykorzystano architekturę typu CAA (component application architecture) (Badania możliwości i zastosowania oprogramowania CATIA przy wykorzystaniu architektury CAA)

Keywords: CATIA, TOOLMANAGER, Tool library, CAA, Secondary development

Słowa kluczowe: oprogramowanie CATIA, systemy CAD.

Introduction

CATIA is a CAD/CAE/CAM integration software of French Dassault Systems, which can provides users an integrated digital environment and complexity requirements for users from design to manufacture¹. CATIA provides a 2.5-axis, 3 axis and multi-axis CNC machining modules. Users can compile NC programming using CATIA, both can call a tool from CATIA tool library, and can manually enter the parameters to create the tool². To manage the growing types of knives, many enterprises generally use a professional tool management system, such as TOOLMANAGER software which is a highly integrated tool information management system developed by German company. It can effectively manage the tool component, tool components, machine setting bills, tool map and tool supplier information. The tool library mainly offer help for managing tools for the tool room staffs, and facilitate the staffing process three-dimensional technology and CNC programming.

CATIA has powerful CNC machining modules. The basic operation methods of CNC machining include setting processing elements, setting the tool path, setting tool parameters, setting feed/back knife, cutting simulation and generating NC programs. Selecting tool is very important in the NC content, for each machining operation needs to specify a processing tool. At present, in the enterprises tool management, the users use the tool and maintain it themselves for each business unit, and users cannot exchange of information and share data. For example, during the CNC programming, staff use CATIA tool management functions and redefine their own tool models and data. The NC programming staff cannot get the latest stock information and use of information, resulting in not purchasing the tool to affect the production cycle. NC programmers do not know the information has resulted in inconsistencies in tool selection, and increased manufacturing costs. With the product mix changes and adjustments, the types and specifications of the tool will continue to change purchased by machinery processing enterprises. If the CNC staff cannot grasp the information of existing enterprise tool, use the tools which do not match the existing tools, will lead to the applicability of NC program reduced, seriously affecting the production schedule. Therefore, achieving the integration of CATIA tool and business management tool is very important.

Therefore, we studied the CATIA integration technology of tool library. Basing on an enterprise digital manufacturing, CATIA software is further developed using Component

Application Architecture (CAA), and dynamic integration of CATIA and TOOLMANAGER tool library is achieved, and it has been successfully applied in the enterprisement³. The CNC staff can call tool directly from CATIA tool library, and can grasp the company's existing tools, and improve the efficiency of CNC programming, and ensure the applicability of CNC code. It provides a good basis for successful implementation of digital manufacturing system.

2. A further developing method for CATIA

As powerful engineering software, CATIA has a strong opening performance. Users can follow their own needs, using different ways to the development in various degrees. CATIA secondary development interface communication with external interface is in two ways: in-process applications approach and out-process applications approach. In in-process applications, CATIA software and scripts to run in the same process address space, such as macro (Macro) mode. In the CATIA environment, the menu record Macro, and generate VB Script sequence. When the macro starts running, CATIA is in a non-activated and variables cannot be stored between calling macro. This relatively simple approach can be completed in the CATIA environment. In out-process applications, CATIA and external applications run in different process address space. In the case of CATIA running, external process control CATIA through the interface, it can create and modify CATIA environment and geometry data, size, etc. It can also support Object Linking and Embedding.

Specifically, There are two main ways to CATIA secondary development: using a macro to the second development and using of CATIA component application architecture (CAA) to the further development [4,5].

2.1. Using a macro to the further development

Users use macro recording a series of operation, automatically generated code, and use VBScript as editing tools, it is a customized interactive way. CATIA provides CATIA Automation API for VBScript on the secondary development, Automation API has the ability to communicate with any OLE-compatible platform, Automation interface can call "InputBox" and "MsgBox" function to get user input information and output [6]. For NT users, you can apply more complex Visual Basic to define the input and output interfaces. An icon can be associated with a macro in running, and be into CATIA display frames.

The characteristics of using the macro to the further development of CATIA are convenient, not only to achieve

the desired function, but also to record the macro method and obtain the required procedures. But the disadvantage is that CATIA cannot perform other operations when the macro is running. The function need be achieved was limited.

2.2. Using the component application architecture (CAA) on the further development of CATIA [7,8]

CAA is an acronym Component Application Architecture, is a platform of Dassault Systems, which expanding products and customizing and developing for customers. It uses object-oriented programming language; object-oriented programming (Object-Oriented- Programming, OOP) has become mainstream in the field of software development and design. CAA's development can be seen as a combination and expansion of its component objects. CAA uses COM technology and OLE technology, COM as a software architecture has a better module independence, scalability, making it easier to CAA programming and become standardized, so that the code more concise.

In the support of the CAA structure, Dassault Systems as can be established as the building block, this structure is very suitable for the growth and development of the system. CATIA uses CAA on the further development, its benefits are: reusability, abstraction, encapsulation, etc. However, the complexity of Dassault Systems and depth involved in the content of CAA itself, the second development using CAA must have some complexity and difficulty. Users familiar with the relevant application system of Dassault Systems, they must also have basic knowledge of software development, the basic programming skills of C + + (or Java), COM technology, etc.

3. Integration of CATIA tool library using a macro

3.1. Scheme of integrated tool library

In order to achieving integration, when NC program was written and executed tool selection function, CATIA tool class and the subordinate TOOLMANAGER class can be list. On the other hand, CATIA to be able to dynamically query the tool data in TOOLMANAGER and the selected tool can be returned to the CATIA environment, finally parameterization drive tool graphics. When NC machining tool is selected using CATIA, the view of the tool data is the dynamic tool library, it can realize integration of CATIA and TOOLMANAGER.

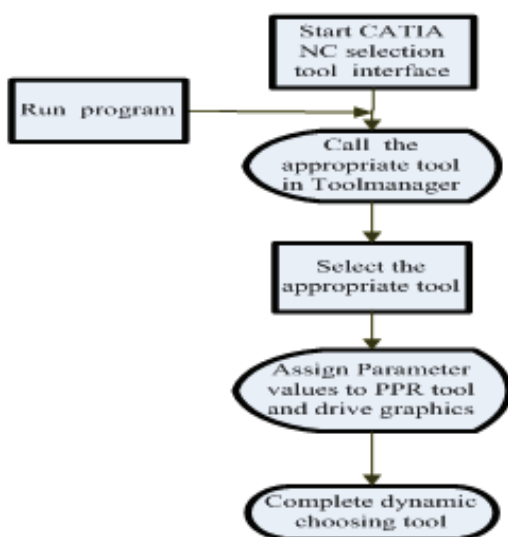


Fig.1. Tool library integrated solution

The integration above mainly is to call tool from TOOLMANAGER tool Library, and then assign the property value of tool to CATIA NC machining tool module model. The use of macros in the implementation for the further development can not call the CATIA function. Though further developed using CAA, calling for information, converting information and valuating information of CATIA and TOOLMANAGER is researched, finally, the dynamic calling, association and driving of CATIA-based tool information is implemented successfully. Fig.1 shows the Tool library integrated solution.

3.2. Development process of the integrated tool library

To ensure the integrity and accuracy of tool information in library TOOLMANAGER tool library, and NC programmers can select the needed tool, before carrying out an integrated development tool, the description field for each type of tool in TOOLMANAGER tool library must be sorted combining with enterprise tool existing type for tools, and make it fit the classes and attributes of CATIA tool. To make CATIA tool class and TOOLMANAGER class correspond, Because CATIA tool that can add new classes, we need to complete further development of CATIA. The development process is as follows:

- (i) The corresponding file in CATIA V5 R16 ----- \ Program Files \ Dassault Systemes\ B16 \ intel_a \ resources \ graphic replaced by *. feat file in FEAT folder.
- (ii) Use WordPad to open MfgUserAttributes.xml in CATIA V5 R16-----\ Program Files \ Dassault Systemes \ B16 \ intel_a \ startup \ Manufacturing \ Samples, and modify the file contents in the second row: Results are as follows: <!DOCTYPE CATSpecs SYSTEM"D:\ProgramFiles\DassaultSystemes\B16\ intel_a\resources\dtd\MfgUserAttributes.dtd">. In this document, we can add tool properties in accordance with the provisions meaning of the field.
- (iii) Start WINDOWS/system32/cmd.exe:Run MfgResourceAttr.exe in the test folder, compile MfgUserAttributes.xml in \Program Files\Dassault Systemes\B16\ intel_a\startup\Manufacturing\ Samples directory, add tool properties to Manufacturing Resources Extensions. Feat and Manufacturing Literals. feat, and complete the tool additional properties. The additional Properties of tool is the basis of building this module.

4. Implementation of integrated

Combining with actual situation of enterprises, this paper use the method of developing macro to make the second development of the CATIA, and achieve integration of CATIA tool and enterprise tool database, the following is system operation interface. The CATIA tool dynamic calling data and driving module has two main functions:

- tool data call function: According to the selected class of tool dynamic call tool data tool.
- tool data-driven features: basing on the selected tool, drive the parameters graphics and tool parameters.

The use as follows: First, start the CATIA V5 R16, into the CNC machining module, select the required CNC machining tool, and then start the tool selection interface, finally implement the integration of two software.

5. Conclusion

We studied a technology of integration research tool library of CATIA in shaanxi scientific technology research project, which is named discrete manufacturing enterprise

decision support system based on heterogeneous information integration(project no:2010K01-068). Tool database creating theory and structure of CATIA were analyzed, CATIA secondary development methods were compared and studied. The methods of CATIA integrated tool database using of macro were proposed. Combining with the actual situation of business, the software module is realizing integrated of CATIA tool library. The successful integration of CATIA tool database and tool library can supply service as follows:

- (i) For NC programming, directly select tool from CATIA tool library, greatly reducing the work of the NC programmer to set parameters of the tool, improving the efficiency of NC programming.
- (ii) The integrating of CATIA tool library, it makes tool selected by NC programming closely link to the production site tool, ensuring the applicability of NC program.
- (iii) Improve automation of the enterprise digital manufacturing system and degree of integration.

This work was supported by Shaanxi Provincial scientific and technological research projects under Grant Nos. DF0102110401.

REFERENCES

- [1] Delacur, Jacques,Cuinier, and Jean-Luc,Presentation of the first PLM integrated Optical Simulation Software for the Design and Engineering of Optical Systems, Proceedings of SPIE - The International Society for Optical Engineering,vol.5249, 2004,pp.42-53.
- [2] Xu Peng-cheng, The Road of Digital Manufacturing System, Aeronautical Manufacturing Technology,2007, pp.68-71.
- [3] Dong Yi-xin,Xi Ping, Secondary development of CATIA interface,Aeronautical Manufacturing Technology,2006, pp.83-86.
- [4] Dong Yi-xin, Xi Ping, The Movement Simulator of Five-axis NC Machine Based on Secondary Development of CATIA, Mechanical Engineer,2005,pp.41-43.
- [5] Chen Ming, Deng Shi-fu, Zhu Rui, Zhou Lai-shui, CAD/CAE Integration Based on CATIA Platform, Journal of Computer-Aided Design & Computer Graphics,vol.18, 2006,pp.1078-1082.
- [6] Deng Dong-mei, Zhou Lai-shui, Chen Gong, An Lu-ling, and Li Wei, Design and Implementation of CATIA-Based Construction Tool for Component Library,Journal of South China University of Technology, vol.35,2007,pp.138-142.
- [7] Wang Su, Chen Nan-fei, Zhu Yu-ming, Zhu Xin-xiong, CATIA-based Modeling Method for Object Made of Functionally Gradient Materials, Chinese Mechanical Engineering,vol.18, 2007,pp.454-456.
- [8] Yang Liu-hui, Zhang He-ming, Research and Application of COM-Based Product Information Integration in CATIA Environment, Computer Engineering and Applications, 2001,pp.132-134.
- [9] Guo Zhe, Zang Shun-lai, Wei Gong-ji, Chen Feng, and Guo Cheng, Secondary development technology of CATIA V5 and its application in design of stamping dies, Die & Mould Industry, vol.32, 2006, pp.1-4.
- [10] Li Wei, Zhang Yong-yan, Development and application of conversion software of NC program based on CATIA, Aeronautical Manufacture Technology,2001, pp.55 - 57.

Authors:*School of Mechano-electronic Engineering, Xidian University, 187#Xi'an,710071, P.R.China E-mail: kongxg@vip.sina.com.*