A Conversion Platform for Multi-Software NC Programming

Pengyan Yin, Yidu Zhang, Qiong Wu, Shuguang Chen
School of Mechanical Engineering and Automation Beihang University Beijing, China

*Corresponding author e-mail: wuqiong@buaa.edu.cn, yinpy@buaa.edu.cn,
yidzhang@buaa.edu.cn, sgchen@buaa.edu.cn

Abstract. In this paper, a conversion platform based on UG CAM and HyperMILL is proposed combining the advantages of both. Complicated features can be programmed with 5-axis machining in HyperMILL, then transformed into tool path files that can be recognized and processed by UG CAM through the platform. Relevant tool information can also be obtained and output during the conversion process for unified management. A case study is presented to validate the proposed methodology and platform.

1. Introduction
In nowadays, computer-aided manufacturing (CAM) plays a significant role in shortening product development cycle, improving the quality of product and enhancing industry competitiveness. As a high-end application of manufacturing technology, CAM has a rapid application trend in aerospace manufacturing industry [1, 2]. With the improvement of industrialization level and the development of intelligence, high-speed and high-precision processing has become the trend of modern manufacturing industry, 5-axis linkage NC machining technology has become the main technology gradually.

The advantages of 5-axis CAM software processing technology are mainly reflected in the following aspects [3]:
- Shorten the processing time: 5-axis machining can reduce the processing process and the number of clamping times, thus greatly shortening the overall processing time and delivery cycle;
- Improve the quality of product: using a shorter tool can reduce the deviation and obtain higher machining precision and more stable surface quality;
- Reduce the cost: 5-axis machining can extend the service life of the tool and reduce the use of other processing equipment.

At present, one of the most widely used CAM software in aerospace manufacturing enterprises is UG, which provides an integrated function from designing to processing and facilitates the file management of factories [4]. However, there are still some problems in the 5-axis machining function of UG. Taking the processing of casing parts as an example, the 5-axis programming process is complex for complicated features such as convex platform and mounting edge, which has low programming efficiency, insufficient optimization of path planning, and programmers are required to have extensive experience. Besides, it is easy to encounter such situations as tool bumping during processing. Therefore, it is necessary to use an excellent 5-axis CAM processing software cooperatively to improve these problems.

Hyper MILL is a new generation of CAM system oriented towards the whole model object and the process feature. It is the first software in the world to put forward intelligent tool shaft automatic
avoidance technology using computer, as well as its independent 5-axis linkage, dynamic change of tool shaft inclination and other functions [5]. All the processes can be completed with once clamping in HyperMILL, and it is recognized as one of the best 5-axis processing software [6]. However, it has poor editability of the tool path and the complex post-processing. Table 1 shows the performance comparison between UG and HyperMILL.

**Table 1** Performance comparison

| Performance          | Software   |
|----------------------|------------|
|                      | HyperMILL  | UG CAM    |
| Ease of Operation    | √          |           |
| Tool Path Editability| √          |           |
| Post-processing      | √          |           |
| 2-axis               | √          |           |
| 3-axis               | √          |           |
| 5-axis               | √          |           |

The symbol means better perform.

In contrast, UG and HyperMILL have complementary characteristics in some extent. On the other hand, considering that the enterprises have experience in using UG for many years with precious accumulation of technical documents, combining the two is the most effective way. That is, HyperMILL is used for 5-axis programming of complicated machining features, while UG is used for general feature programming, as well as unified post-processing and data management as usual. In this paper, the establishment of the platform for conversion of documents and information between UG and HyperMILL is studied.

### 2. Proposed Methodology

The process of implementing the combined use of UG and HyperMILL is shown in Fig. 1. The main research contents can be summarized by sorting out the process:
Cross-software’s Link Issues: It is convenient for users to call HyperMILL directly by linking to UG CAM, thus reduce the time of human participation and operation steps of interaction between UG and HyperMILL.

Model Creation: The model is usually created in UG and it will be a waste of time if the model is recreated in HyperMILL. Therefore, it is necessary to solve the problem for HyperMILL to call the model established by UG.

```
31: oM[1]( z[/15.5025] )
31: oM[2]( x[/-0.132941] y[/-3.361233] z[/15.5025] u[/0.0000
31: arg(T: firstPosition:=0)
31: proc(firstPosition5X)
10: nctx(APP)
10: nctx(0)
10: tr(FX)
14: aliasUse(FX)
31: oL[3]( z[/0.0025] )
12: end ctx link
31: oL[4]( z[/-2.9975] )
12: G1 pth
10: tr(FX)
14: aliasUse(FX)
31: oL[5]( x[/0.123682] y[/3.361627] )
31: oL[6]( x[/0.384966] y[/3.342042] )
31: oL[7]( x[/0.641273] y[/3.302594] )
31: oL[8]( x[/0.89778] y[/3.243519] )
31: oL[9]( x[/1.140963] y[/3.165167] )
31: oL[10]( x[/1.381416] y[/3.068003] )
31: oL[11]( x[/1.61365] y[/2.952606] )
```

(a)

```
$\$ISO
UNITS/MM
TOOL PATH/D10.0000R5.0000, TOOL
TLDATA/ballMill,10.0000,5.0000,75.0000,0.0000
MSYS/150.00000000000000,150.000000000000000120.000000
$ centerline data
PAINT/PATH
PAINT/SPEED,10
RAPID
GOTO/0.0000,0.0000,15.5025,0.0000,0.0000,1.0000
RAPID
GOTO/-0.1329,-0.1329,15.5025
FEDRAT/MMPM,50.0000
GOTO/-0.1329,-0.1329,0.0025
GOTO/-0.1329,-0.1329,2.9975
FEDRAT/MMPM,200.0000
GOTO/0.1264,-3.3616,-2.9975
GOTO/0.3850,-3.3420,-2.9975
GOTO/0.6413,-3.3026,-2.9975
GOTO/0.8938,-3.2435,-2.9975
GOTO/1.1410,-3.1652,-2.9975
GOTO/1.3814,-3.0680,-2.9975
GOTO/1.6137,-2.9526,-2.9975
```

(b)

Fig. 2. Part of the content from the tool path file. (a) Obtained from HyperMILL. (b) Obtained from UG CAM
Tool Path Conversion: The tool path editing function of HypeiMILL is not convenient enough compared with UG CAM, and Hyper MILL does not provide a fully open interface to export tool path files to other software. So it is necessary to load the tool path obtained by HyperMILL into UG CAM for unified post-processing. The tool path file obtained by Hyper MILL is in pof format, while UG’s is in cls format. To ensure that the UG can read the tool path created by Hyper MILL, converting the pof file into cls format without losing information is required.

Information Extraction of Cutting Tool: For the convenience of tool management and data inheritance, the tools created in Hyper MILL should also be stored in UG. So the platform needs to get the tool information from the tool path file when the conversion finished.

The above task requirements can be achieved through the VS2010 platform using the VB.NET language.

2.1. Cross-software’s Link Issues

Data interface is an important way to realize mutual calls between different software and data exchange.

As an integrated NC programming CAM software, Hyper MILL provides direct data interface to some other CAD platforms, such as SolidWorks and Inventor, but it does not have such an integrated interface for UG. The better way is to locate the starter program of Hyper MILL, and make it start with the help of ‘Shell’ function when a button is pressed on the platform.

![Fig. 3. The principle of conversion from HyperMILL into UG](image)

```vbnet
If AllLine(i).Contains("nextToolType") Then
    Dim ToolType1 As Integer = InStr(1, AllLine(i), ",")
    Dim ToolType2 As Integer = InStr(3, AllLine(i), ",")
    ToolType = Microsoft.VisualBasic.Mid(AllLine(i), ToolType1 + 1, ToolType2 - ToolType1 - 1)
End If
If AllLine(i).Contains("cornerRadius") Then
    Dim Radius1 As Integer = InStr(1, AllLine(i), ",")
    Dim Radius2 As Integer = InStr(3, AllLine(i), ",")
    Radius = Microsoft.VisualBasic.Mid(AllLine(i), Radius1 + 1, Radius2 - Radius1 - 1)
    Radius = Format(Val(Radius), "0.00")
End If
If AllLine(i).Contains("vtaperAngle") Then
    Dim vtaper1 As Integer = InStr(1, AllLine(i), ",")
    Dim vtaper2 As Integer = InStr(3, AllLine(i), ",")
    vtaperAngle = Microsoft.VisualBasic.Mid(AllLine(i), vtaper1 + 1, vtaper2 - vtaper1 - 1)
    vtaperAngle = Format(Val(vtaperAngle), "0.00")
End If
```

![Fig. 4. Part of the code about information extraction](image)
2.2. Model Creation
Hyper MILL provides a direct invocation interface to the geometric model of UG, that is, the model established by UG can be opened directly in its CAD platform, but the interface is not free. After testing, it is found that when the model established by UG is saved in step format, Hyper-CAD can read it without information missing as well. Then the model can be loaded automatically when HyperMILL is opened.

2.3. Tool Path Conversion
The tool path files obtained from UG and Hyper MILL are as shown in Fig. 2 below. These files contain almost all the information during the programming process, including software version, process name, processing mode, time, coordinates, tool information, and tool path location and so on. By comparing the two files, the following rules can be summed up:

The tool path file of Hyper MILL contains complete information, which can be processed and extracted to obtain the required position information and tool information.

The tool position information in Hyper Mill’s tool path file is six valid digits, while the four valid digits are retained behind the decimal point in the tool path file of UG CAM.

The coordinate and tool information expressions of Hyper MILL tool path file and UG CAM tool path file are different and need to be converted.

The principle of conversion can be seen in Fig. 3. After reading into the pof file, the content is segmented with line breaks as a symbol, traversing and querying the key words of the file which are needed, extracting the corresponding data and making required adjustments, then writing into the cls file in right position. Through the code, the platform can realize the conversion of the tool path file from HyperMILL into UG CAM.

![Fig. 5. The interface of the developed platform.](image-url)
2.4. Information Extraction of Cutting Tool

In UG CAM, taking ball milling tool as an example, six items of data need to be obtained, including diameter, lower radius, taper angle, length, cutting length and flutes. By extracting these data, the parameters used to create the tool path in HyperMILL can be displayed on the platform, and the tool can be created with the same parameters in UG CAM. Part of the code is shown in Fig. 4.

3. Case Study

The proposed methodology was tested with a case study. The platform developed for the case can be seen in Fig. 5 and the 3-D solid model was created in UG CAD shown in Fig. 6. The part contains three types of machining features. Feature 1 and feature 2 are common plane features among them, which can be directly programmed in UG CAM. While feature 3 is a hook face feature, programming in HyperMILL will be more effective. The following will focus on the verification of programming and conversion process based on feature 3.

Fig. 6 The 3-D solid model for the case

Fig. 7 The tool path track generated in HyperMILL

Fig. 8 The converted tool path track in UG CAM
Fig. 9. The platform interface after the conversion.

Clicked the "OPEN" button to activate HyperMILL when it came to the programming of feature 3. The tool path then could be generated by steps including creating operations, tools, geometry, and setting cutting parameters, then saved the file in pof format. The tool path track generated in HyperMILL is shown in Fig. 7.

On the platform, the file address to read and export could be selected by users. After the conversion, the file get in last step can be generated into a tool path file in cls format, which can be opened and simulated in UG CAM as well. At the same time, the corresponding tool parameters will also be displayed in the "Tool Parameters" column on the platform. The tool path track and cutting tool parameters shown in UG can be seen in Fig. 8 and Fig. 9 respectively.

4. Conclusion
The methodology and platform presented in the paper was successfully validated with a case study by combining UG CAM with HyperMILL. Firstly, HyperMILL can be opened directly using the interface functions through the platform. By extracting the position information and cutting tool information in the tool path file generated by HyperMILL, the information can be arranged in the same format as the files generated by UG CAM during the conversion. Besides, the related cutting tool parameters can be output for later management through the platform. In the process of programming in UG, HyperMILL can be used to deal with the complicated processing features. This method can effectively solve the deficiency of UG programming in 5-axis function and bring more benefits to enterprises as well.

5. Acknowledgment
Thanks are due to Zihan Gao for assistance with the programming and to Xuezhi Liao for valuable discussion.
6. References

[1] H Liu, “On CAD/CAM Technology in Mechanical Design and Processing,” Modern Industrial Economy and Informationization, vol. 141, Sept. 2017, pp. 107–108, doi:10.16525/j.cnki.14-1362/n.2017.09.48.

[2] H Zhao, “Applications of CAD / CAM Technology in Machine Design and Processing,” Agricultural Science and Technology and Equipment, vol. 210, Dec. 2011, pp. 14-16, doi:10.16313/j.cnki.nykyzb.2011.12.004.

[3] S.G Han, J. Zhao, X.X Chen, A.H Li, and Y.W Dong, “Research of Numerical Programming Technology and Its Typical Applications,” Modular Machine Tool and Automatic Manufacturing Technique, vol.3, Mar. 2012, pp. 100-103, doi:10.3969/j.issn.1001-2265.2012.03.026.

[4] D.W Liu and C.Q Xue, “The Research of Secondary Development Technology Based on UG,” Machine building and Automation, vol. 34, Jan. 2005, pp. 80–83, doi:10.3969/j.issn.1671-5276.2005.01.023.

[5] X.J Liu, “Application of HyperMILL in Five-axis NC Machining Center,” National Brazing Equipment Conference, Sept. 2013, pp. 216-220.

[6] P.W Qin, “Creation of Automatic Programming 5 Axes Tool Path of hyperMILL,” Aviation Manufacturing Technology, vol. 3, Feb. 2010, pp. 98-99.