

# The NC Machining Post-Processing Technology Based on UG

Sharifa Magambo<sup>1</sup>, Liu Ying<sup>2</sup>

Tianjin University of Technology and Education

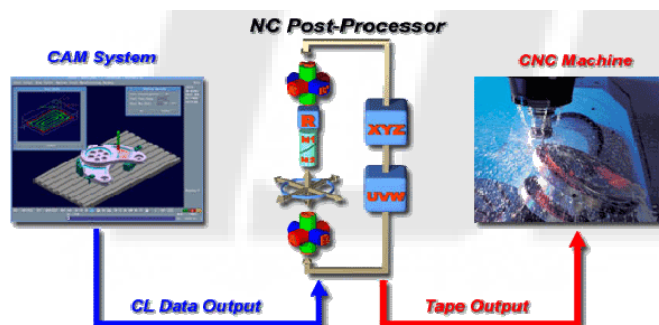
Department of Mechanical Manufacturing and Automation, Tianjin, China

**Abstract:** Post-processing technology is the key to CNC automatic programming technology and an important module of the CAD / CAM system. Post-processing technology converts the NC program that is produced by the CNC automatic programming tool into the file that can be identified by CNC system. Moreover, the generated tool path files must match with the CNC system. Post-processing techniques and UG software are briefly introduced. Using them, a post-processor for FANUC CNC systems is created. The program that generated by the special processor is contrasted with the general processor. The results illustrate that using the dedicated post-processor in engineering applications can improve programming efficiency and processing reliability.

**Keywords:** UG CAM; UG/Post; Post Builder; FANUC NC System

## 1. Introduction

The main task of NX post-processing is to convert the machining tool path source file generated by UG CAM into the NC code files that NC program can be identified by CNC system and machine tool, so this process is an important link of the UG and NC machining. Generation of tool path files must be processed into the NC program format that can be accepted by NC system, then can effectively drive machine processing. This is because different machine use different NC system, which lead to the use of the NC program format is not the same. The conversion process is called for "post-processing"[1]. There are two kinds of post-processing methods: Graphics Post processor Module that referred to as GPM and NX/Post post-processing editor that was provided by UG NX6 [2, 3]. The Post Builder module that's provided by UG/Post can be used to generate special post processor, which can solve the disadvantages of the general post-processor and improve the reliability of the program. Because it has advantages as follows: the specification of program format; errors can be found easily; given the corresponding information of the cutting tool, it is convenience for the user to know the tool.



**Figure 1:**How post-processing links CAM and CNC machine.

## 2. UG/Post

UG/Post mainly consists of processing output manager, events generator, event handling files and definition file, etc[4,5]. The comprehensive application of TCL language

realizes the development of UG/Post, which is an explanation type computer language which is developed by John K.O usterhout. There are two approaches: manual programming and Post Builder.

Post Builder is the tool provided by UG/Post. It's very convenient to create and modify post-processing. By means of graphic interface interaction of post-processing constructor, the user can agilely define and establish NC program output format and content, as well as each event processing mode.

## 3. Created the post-processor for FANUC NC system

Although users can use some of the machine post-processor provided by UG software to generate NC files, which can be used just for a small amount of modification. However, it is not convenient and easy to appear all sorts of mistakes, so it can't meet the demand. Then, it is necessary to create the post-processor for the requirements through the Post Builder in according to the actual situation of the machine tool.[7]

The example that shows how to create the post-processor for FANUC NC system is in follows. Here, XHK714 three axis vertical machining center with FANUC NC system is chosen as machine tool. The tool's stroke is X 660mm, Y 460mm and Z 620mm. The Table can load 400kg. The Spindle maximum speed is 8000rpm. The tool's repeat positioning accuracy is  $\pm 0.005$  mm.[8,9]

### 3.1 Creating the post-processor by the Post Builder

Entering the Post Builder module and creating the post-processor of FANUC system. Click [start]-[all program]-[UGS NX6.0] and [machining tools]-[post-processing constructor] in Windows XP operating system. Open the Post Builder post-processing constructor. As shown in Fig.2.



Figure 2: Version 6.0.0-License Control

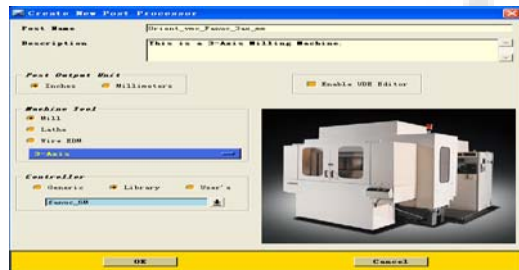


Figure 3: Create New Post Processor

Selecting the [File]-[New] in Fig.2 dialog, then pop-up a dialog box of "Create New Post Processor". According to the actual situation of the machine tool, setting the machine name, description, units of measurement and machine type in the dialog, as shown in Fig.3.

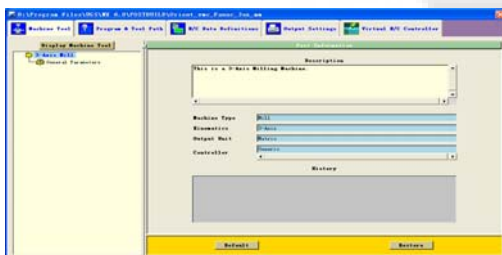


Figure 4: Edit interface.



Figure 5: Setting the machine parameter.

Completing the choice of machine tool and NC system. Determine the related events variable of post-processing in Fig.3 dialog, then click "OK". Pop-up a dialog box of edit, as shown in Fig.3. Consist of five options composition: Machine Tool; Program & Tool Path; NC Data Definition; Output Setting; Generate Virtual N/C Controller.

### 3.2 Machine Tool—Setting the machine parameters

Click on the "Display Machine Tool", then pop-up a dialog box of "three axis Machine diagram". Determine the workpiece coordinate system, click "closed" that on the top right corner. Setting the parameters of the machine tool stroke limit and fast moving speed, as shown in Fig.5.



Figure 6: Setting NC program starting formats.

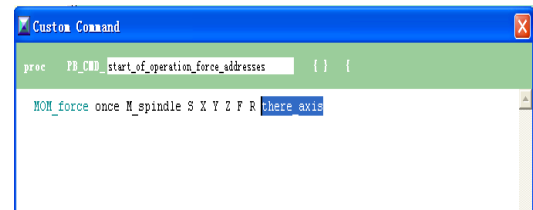


Figure 7: Custom Command dialog box.

## 4. Program & Tool Path

Make the starting format of the NC program: Choose "Program and Tool Path", click "Program Start Sequence" and Delete "MOM\_set\_seq\_on". Click "G40 G17 G90 G71", then pop-up a dialog box of "Block: absolute\_mode" and move "G71" to the recycle bin. Add three address block: G\_adjust/G49-Cancel Tool Len Adjust, G\_motion/G80-Cycle Off and G/G\_MCS Fixture Offset to program line. Then set "G49 and G80" as "forced Output"; set "G" as "options". as shown in figure 6.

Make the starting format of the operation: click the "Operation Start Sequence", then click **PB\_CMD\_start\_of\_operat...**, Set the corresponding machine tool of the post-processor as 3 axis in the "Custom Command" dialog, as shown in Fig.7. Select MOM\_set\_seq\_on, Operator Message and MOM\_set\_seq\_of and click the button of "Add Word", then drag them to the below of "PB\_CMD\_start\_of\_operation\_force\_address program". Input "\$mom\_Path\_name" in the text box of "Operator Message", as shown in Fig.8.

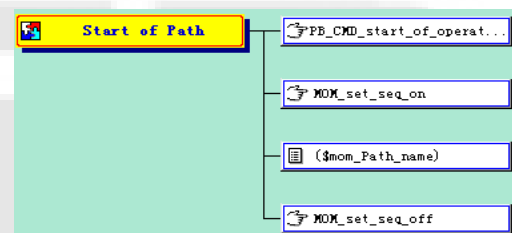


Figure 8: Set up Start of Path.

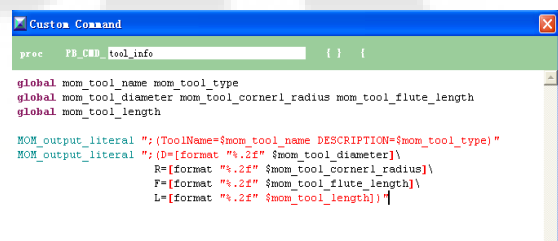


Figure 9: Custom Command box.

Setting up the automatic tool change and given the basic information of tools: Select the "Custom Command" in the

dialog box of “Program & Tool Path”. Then drag to the between T M06 and T program lines of the Auto Tool Change, Change the text box information of PB\_CMD into the tool\_info, and input corresponding content in the text box, as shown in Fig.9.



Figure 10: Select Tool Path/Motion .

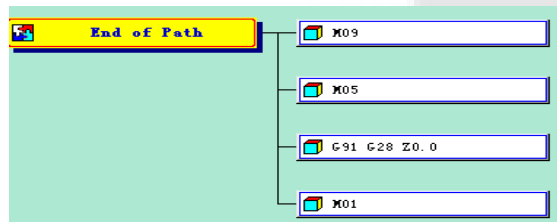


Figure 11: Set up End of Path.

Setting up the movement instruction of NC program for comply with the requirements of machine tool: switch the dialog box to the Tool Path/Motion, appear three marked on the right : Linear Move, Circular Move and Rapid Move, as shown in Fig.9. Setting the end format of operation: selecting the “Operation End Sequence”. Add the address block of "More/M\_coolant/M09", "M05" and "M01" in the pop-up dialog, click the "OK". Continue add the "G91", "G28" and "Z0.0", as shown in Fig.11. Setting up the end format of NC program: click the “Program End Sequence”, delete two program lines of "PB\_CMD\_nurbs\_end\_of\_program" and "MOM\_set\_seq\_off". NC Data Definition used define the NC data format (Eg: add "; " in a single program), as shown in Fig.12.

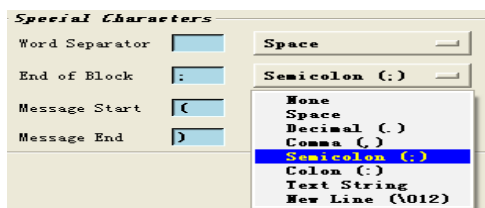


Figure 12: NC Data Definition

```
*****
WIRE_KIN_4_AXIS,$UGII_CAM_POST_DIR$weds.tcl,$UGII_CAM_POST_DIR$weds.def
MILL_3_AXIS,$UGII_CAM_POST_DIR$mill3ax.tcl,$UGII_CAM_POST_DIR$mill3ax.def
MILL_3_AXIS_TURBO,$UGII_CAM_POST_DIR$mill3ax.turbo.tcl,$UGII_CAM_POST_DIR$mill3ax.turbo.def
MILL_4_AXIS,$UGII_CAM_POST_DIR$mill4ax.tcl,$UGII_CAM_POST_DIR$mill4ax.def
MILL_5_AXIS,$UGII_CAM_POST_DIR$mill5ax.tcl,$UGII_CAM_POST_DIR$mill5ax.def
LATHE_2_AXIS,$UGII_CAM_POST_DIR$lathe2.tcl,$UGII_CAM_POST_DIR$lathe2.def
LATHE_2_AXIS_TURRET_REF,$UGII_CAM_POST_DIR$lathe2.turret_ref.tcl,$UGII_CAM_POST_DIR$lathe2.turret_ref.def
MILLTURN,$UGII_CAM_POST_DIR$millturn3axis.mill.tcl,$UGII_CAM_POST_DIR$millturn3axis.mill.def
MILLTURN_MULTI_SPINDLE,$UGII_CAM_POST_DIR$millturn4axis.mill.tcl,$UGII_CAM_POST_DIR$millturn4axis.mill.def
Orient_vac_Fanuc_3ax_mn,$UGII_CAM_POST_DIR$Orient_vac_Fanuc_3ax_mn.tcl,$UGII_CAM_POST_DIR$Orient_vac_Fanuc_3ax_mn.def
TOOL_LIST(text),$UGII_CAM_POST_DIR$post_tool_list_text.tcl,$UGII_CAM_POST_DIR$post_tool_list_text.def
TOOL_LIST(html),$UGII_CAM_POST_DIR$post_tool_list_html.tcl,$UGII_CAM_POST_DIR$post_tool_list_html.def
OPERATION_LIST(text),$UGII_CAM_POST_DIR$post_operation_list_text.tcl,$UGII_CAM_POST_DIR$post_operation_list_text.def
OPERATION_LIST(html),$UGII_CAM_POST_DIR$post_operation_list_html.tcl,$UGII_CAM_POST_DIR$post_operation_list_html.def
*****
```

Figure 13: Post-pro files added to template\_post.dat

## 5. Adding the files of the post-processing

Click “save”. Saving the file of post-processing that was settled to the directory that was defined by user, selecting the [Utilities]/[End Template Posts Data File] in the dialog of “UG/Post Builder Version 6.0.0”, then choosing the line of TOOL LIST(text)...and click [NEW], as shown in Fig.13.

## 5.1 The program that generated by the special processor is contrasted with the general processor

The General Processor Program	The Special Processor Program	Note
% N0010 G40 G17 G90 G70 N0020 G91 G28 Z0.0 N 0030 T01 M06 N0040 G0 G90 X-1.2504 Y.9468 S200 M03 N0050 G43 Z.3937 H00 N0740 M02 %	% G40 G17 G49 G80 G90; T02M06; ;(ToolName=MILL_12_R0.8DE SCRIPTON=Milling ;(D=12.00F=50.00L=75.00); M09; M05;  G91G28Z0.0; M02; M30; %	Change tool The information of tool        Return zero

At the above tablet, it is obviously that the program formats of starting and ending isn't specification and isn't easy to find errors for the general processor. However, for the program of special processor, because the specification of program format uses the semicolon to end the each program segment, it's very convenient for users to check program. Moreover, in the specification of program format, the corresponding informations of the tool are given and it can automatically return zero after the end of program.

## 6. Conclusions

Post-processing technique is the main modules of CAD/CAM software, and the important content of NC machining programming technology. NC machining programming directly affects the processing quality and production efficiency. So to produce the efficient and accurate NC program is an important link in the NC processing. Because the different NC system is used in different machine, it leads the different usage of the NC program format. User needs to build the special post-processor to meet all requirements of the machine tool, that generate the NC code files (G code) that can be identified by machine tool. Finally, the high quality and reliable NC program is obtained, which can improve the efficiency of the nc processing and shorten the time of processing.

## References

- [1] JiSeon Hwangt,Tien-Chien Challg. Five-axis machining of compound surfaces using flat and filleted-endmills.Computer-Aided Design,1998,30(8):(641-647).
- [2] Yi Deng,HaoGe Peng,Qi Xie.The research present situation and the development tendency of CAM post processing technology [J]. Journal of hunan college engineering .2003.12(4):(46-48).“In Chinese”
- [3] Lin Tang. Nc post-processing techniques[J].<New Technology & New Process>Digital technology and mechanical processing technology and equipment .2008(7):(24-25).“In Chinese”
- [4] J.Xia, Q.J.Ge.An exact representation of effective cutting shapes of 5-axis CNC machining using rational Bezier and B-spline toll motions. International Conference on Robotics & Automation Seoul Korea.2001 (21-26).
- [5] Shitian Zhao,Dianzhu Sun. Processing center special post-processor research and application based on the UG/Post [J].Journal of Shandong university of science

- and technology .2005(5):(56-60). "In Chinese" youth. UG-based CNC programming and machining process simulation [J]. Inner Mongolia University of Technology, 2007, (4): 1.
- [6] Ma Zhicheng. CNC technology [M]. Beijing: Beijing Institute of Technology Press, 2012, (1): 3-6.
- [7] Rong-shean Lee etc. Tool Path Generation and Error Control Method for Multi-Axis NC Machining of Spatial Cam [J]. Int J Mach. Tools Manufacture, 1998, 38 (4): 277 ~ 290.
- [8] Wei Yanbo. Surfaces Based on UG bowl CNC machining technology research [D]. Shandong: Shandong University, 2010, (9): 2-5.
- [9] Shen Ligu. Foreign CNC technology research trends and development trends [J]. Machinery Industry Automation, 1996, (2): 19 ~ 23.
- [10] While the birch. Extra ultra high-speed CNC control system development [J]. Manufacturing Technology & Machine, 2000, (5): 12 ~ 14.
- [11] Zhang. CNC technology trends \_ intelligent CNC system [J]. Manufacturing Technology & Machine, 2000, (4): 7-9.
- [12] Zhou Yanyou the world of CNC system development and China CNC transformation [J]. Chinese Machine Tool Industry specific information, NO: 990308.61,2000.
- [13] Guo Qian. CNC automatic programming technology status and development direction [J]. Value engineering, 2011.
- [14] Wang Yu, Ma Haojun, Gao Chonghui et al. A computer aided tool selection system for 3D die/mould-cavity NC machining using both a heuristic and analytical approach[J].International Journal of Computer Integrated Manufacturing,2005,18(8):686~701.