The NC Machining Post-Processing Technology Based on UG

Sharifa Magambo¹, Liu Ying ²

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 fund 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.Ousterhout. 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 agility 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 strok 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 ± 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.
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.

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.

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 "M03", "M05" 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

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 Version6.0.0”, then changing 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

<table>
<thead>
<tr>
<th>The General Processor Program</th>
<th>The Special Processor Program</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>N0010 G40 G17 G90 G70 N0020 G91 G28 Z0.0 N0030 T01 M06 N0040 G06 G90 X-1.2504 Y-3.686 Z200 M03 N0050 G43 Z-3.937 H00 N0740 M02</td>
<td>G40 G17 G49 G80G90; T02M06; tToolName=MILL_12_R0.8DE SCRIPTON=Milling ZD=12.001=50.001=75.001; M09; M05; G91G28Z0.0; M02; M30;</td>
<td>Change tool The information of tool Return zero</td>
</tr>
</tbody>
</table>

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


