Research on Optimization Design of xcv-855 Machining Center Post-processing Based on Mastercam9.1

Xuanlin Ye

College of mechanical and electrical engineering, Yunnan Open University, Kunming 650500, China

Abstract. According to the structural characteristics of the four-axis machine tool of xcv-855 machining center and the requirements of NC program, MP language is used to optimize it on the basis of the post-processing file attached by Mastercam 9.1, so that it can automatically add M10 / M11 in the four-axis fixed axis machining, M29 in the rigid tapping, and the output angle is 0 ° ~ 360 °. Taking the actual processing of the filter head array drum as an example, the correctness of the developed post-processing file is verified. The practical results show that there is no alarm in the process of processing, and the processed parts can meet the specified accuracy requirements, so as to verify the correctness of the post-processing file.

Keywords: Mastercam9.1; Post processing; Optimal design; Xcv-855 machining center.

1. Introduction

With the rapid development of CAD / CAM technology, Mastercam9.1 occupies a large market in the field of manufacturing. However, due to the diversity of CNC machine structure and CNC system in the market, different machine tools have different requirements for the format of NC code[1-2]. The NC program generated by the post-processing file attached to Mastercam9.1 software can not fully meet the actual production requirements, and needs a lot of repair the improvement work seriously affects the processing efficiency and does not give full play to the advantages of computer.

2. Characteristics of the Structure and Program Format of the Four Axis Machine Tool and Problems Raised

2.1. Structural features of xcv-855 machining center

Xcv-855 machining center is a four-axis CNC machine tool, equipped with FANUC CNC system. It adopts the form of three-axis linear machine tool and the fourth axis CNC rotary table.

2.2. Program and program format features

(1) The program format of xcv-855 machine tool includes program name, program start, tool exchange, machining process, cutting cycle and program end. The program name of FANUC CNC system starts with% and the program name starts with O followed by 4-digit integer, such as o1234. (2) Four axis function: under the mode of automatic and MDI, the fourth axis rotation is controlled by program. M10 and M11 command control the clamping and loosening of the rotary console, and the rotary table can rotate only when it is loosened. That is to say, M11 appears before angle A in NC program, M10 appears after angle A, and the workpiece can be processed only when it is clamped. (3) Only one coordinate system is allowed in an NC program. (4) The machine tool needs rigid tapping when tapping. M29 Command + spindle speed at tapping is required before G84 code.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd
2.3. Questions raised

According to the above structural features and program requirements of xcv-855 machine tool, the postprocessing file attached by Mastercam9.1 software cannot meet the NC code format requirements of xcv-855 machine tool.

1) G55, G56, G57, G58, G59, G54.1p1 will appear in the NC program processed by the PST file attached to mastercam9.1 software G54.1p18 and other coordinate systems, and every time a rotation fixed surface is added, a coordinate system will be added to the NC program after being set, as shown in Figure 1. (2) The NC program processed by the processing file attached to mastercam9.1 software does not have M11 and M10 instructions. (3) The NC program processed by the configuration processing file attached with mastercam9.1 software does not have the spindle speed when M29 + tapping. (4) The output angle of the NC program processed by the configuration processing file attached with Mastercam9.1 software is positive and negative continuous, and the angle increases linearly. If the processed part rotates for 8 turns, the output angle is close to 3000 °. After the part is processed, it idles for 8 turns when it returns to zero, which wastes time and seriously affects the production efficiency.

Therefore, it is necessary to develop a special post-processing program for this machine tool, so as to meet the needs of teaching and actual production and processing, and give full play to the best effect of the cam module of Mastercam9.1 software, so as to improve the production efficiency. After a lot of explorations and attempts, the author makes some modifications and optimizations to the post-processing file attached to Mastercam9.1 software, so that the NC program generated after Mastercam9.1 software can run directly on xcv-855 machining center without any alarm. The following is the author's modification and optimization of the self attached processing file of Mastercam9.1 software combined with the existing xcv-855 machining center, and the accuracy of the post-processing file of Mastercam9.1 software is verified with the programming and processing of filter tip arrangement drum parts shown in Figure 2 as an example.

```
N104T1M6
N106G0G90 G54 X-110.Y0.A0.S2200M3
N106G4G1250.M0
N116G9G81Z30.R40.5F25.

N332 G55 X-126.Y0.Z50.A-10.
N334G0G8138.R40.5F25.

N792 G55 X-126.Y0.Z50.A10.
N794G0G8328.340R40.5Q2.5F35.

N5384 A-390.
N5386X-121.6B3Y0.

N6596X0G0Z60
N6596 A-410.

N3020M5
```

Figure 1. Post NC program diagram of self attached post-processing file.

Figure 2. Parts drawing of filter nozzle arrangement drum.
3. Problem Solving

3.1. Analysis and solution to the problems of multi coordinate system

In the four-axis machining of xcv-855 machining center, when the machining content is the same and evenly distributed, but after rotating the fixed face tool path, the NC program will increase with the increase of the number of rotating fixed face coordinate system. Machining the tool path of 18 holes in the circle as shown in Fig. 2. With the increase of the number of rotation and face determination, the coordinate system increases from G54 to G54.1p12, which is easy to cause accidents in actual machining. Solution to the above problem[3]: open the post-processing file attached to Mastercam9.1, find the source code of the block name PWCS in the file, and make modifications, as shown in Table 1.

| Source code | Modified code |
|-------------|---------------|
| if workofs <> prv_workofs | if workofs <> prv_workofs |
| (force_wcs & toolchng), | (force_wcs & toolchng), |
| [ if workofs < 6, | [ if workofs < 6, |
| [ g_wcs = workofs + 54 | g_wcs = 54 |
| *g_wcs] | *g_wcs] |
| else, | else, |
| [ p_wcs = workofs - five | [ p_wcs = workofs - five |
| "G54.1", *p_wcs ] ] | "G54"] ] |

The above G54 is used by the author. If G54 is occupied, just replace G54 with G55 or other coordinates.

3.2. Analysis and solution of M11 and M10 problems

Because the NC program processed by the attached processing file of mastercam9.1 software does not have M11 and M10 instructions, the machine tool will give an alarm and stop running when the program is uploaded to xcv-855 machining center for processing. Solutions to the above problems: open the post-processing file attached to Mastercam9.1 software, find the source code of program block names psof and ptlchg in the file, and make modifications, as shown in table 2 and table 3.

| Source code | Modified code |
|-------------|---------------|
| pindex | pindex |
| if mi1 > one, absinc = zero | if mi1 > one, absinc = zero |
| pcan1, pbld, n, *sgcode, *sgabsinc, pwcs, pxout, pfyout, pfcout, *speed, *spindle, pgear, strcantext, e | pcan1, pbld, n, *sgcode, *sgabsinc, pwcs, pxout, pfyout, pfcout, *speed, *spindle, pgear, strcantext, e |
| pbld, n, "G43", *tlnsgno, pfzout, scoolant, next_tool, e | pbld, n, "G43", *tlnsgno, pfzout, scoolant, next_tool, e |
| absinc = sav_absinc | absinc = sav_absinc |

| Source code | Modified code |
|-------------|---------------|
| absinc = zero | absinc = zero |
| pbld, n, sgabsinc, pwcs, pxout, pfyout, pfzout, pfcout, e | pbld, n, sgabsinc, pwcs, pxout, pfyout, pfzout, pfcout, e |
| pe_inc_calc | pe_inc_calc |

Table 2. Automatically add source code modification in M10 / M11 block name psof.

Table 3. Automatically add M10 / M11 block name ptlchg to modify the source code.
3.3. Analysis and solution of rigid tapping

Since the NC program processed by the attached processing file of mastercam9.1 software does not have M29 and corresponding spindle speed before g84 instruction, the starting position of the processed thread has changed and cannot be fixed, which cannot meet the actual production requirements. The solution to the above problem: open the post-processing file attached to Mastercam9.1 software, find the source code of the program block name ptap, and modify it, as shown in Table 4.

Table 4. Automatically add the M29 block name ptap to modify the source code.

| source code                        | Modified code                        |
|------------------------------------|--------------------------------------|
| pdrlcommonb                        | pdrlcommonb                          |
| pcanl, pbld, n, *sgdrlref,         | pcanl, pbld, n, "M29", *speed, e     |
| *sgdrl, pxout, pyout, pfzout,      | pxout, pyout, pfzout, pbld, n, *sgdrlref, *sgdrl |
| prcout,                            | prcout, *feed, strcntext, e          |
| prdrlout, *feed, strcntext, e      | prdrlout, *feed, strcntext, e        |
| pcom_movea                         | pcom_movea                           |

3.4. Analysis and solution of angle output problem

The output angle of the NC program processed by the configuration processing file attached with Mastercam9.1 software is positive and negative continuous, and the angle increases linearly. If the processed parts are rotated for 8 turns, the output angle is close to 3000°. After the parts are processed, when they return to zero, they idle for 8 turns, which wastes time and seriously affects the production efficiency. If we can make the output angle between 0° ~ 360°, we can solve the problem of turning many circles when returning to zero. Solutions to the above problems: open the post-processing file attached to Mastercam9.1 software, find the source code of the program block name pfcout and pcout, and make modifications, as shown in table 5 and table 6.

Table 5. Modification of source code in 0° ~ 360° block name pfcout.

| source code                        | Modified code                        |
|------------------------------------|--------------------------------------|
| if absinc = zero, cabs, !cinc      | while cabs<0,                          |
| else, cinc, !cabs                  | [cabs=cabs+360]                       |
|                                   | while cabs>=360,                      |
|                                   | [cabs=cabs-360]                      |
|                                   | if absinc = zero, *cabs, !cinc       |
|                                   | else, *cinc, !cabs                    |

Table 6. The output angle in linear movement is modified in the 0° ~ 360° block name pcout.

| source code                        | Modified code                        |
|------------------------------------|--------------------------------------|
| if absinc = zero, cabs, !cinc      | while cabs<0,                          |
| else, cinc, !cabs                  | [cabs=cabs+360]                       |
|                                   | while cabs>=360,                      |
|                                   | [cabs=cabs-360]                      |
|                                   | if absinc = zero, *cabs, !cinc       |
|                                   | else, *cinc, !cabs                    |

To sum up, after modifying and optimizing the post-processing file attached to mastercam9.1, the post-set NC program does not need to be manually modified, and can be directly operated on xcv-855 machine without alarm, thus improving the production efficiency. Save the post NC file attached to mastercam9.1 modified and optimized by the above method as "xcv-855-4axis". Take the automatic programming and processing of filter tip arranging drum as an example to verify the accuracy of the post file "xcv-855-4axis".

4. An Example of Processing the Arrangement Drum of Filter Tip

Mastercam9.1 software has the function of modeling. It uses the cam module to complete the setting of machining type, machining shape, machining tool selection, machining allowance, cutting parameters
and other machining parameters, and programs the tool path. Because the features are evenly distributed on the circumference, only one tool path needs to be made, and the other tool paths can be programmed by the tool path conversion function of mastercam9.1 software. Methods: right click "tool path", "path conversion" and "rotation" in the operation manager, and set parameters in the rotation interface. The number of rotations is n-1, and the rotation angle is 360 / n, where n is the number of characteristic circumference distribution, and then select the right view of "rotation to the viewing angle" to complete the work of centering the remaining 17 holes[4-5]. The simulation processing results are shown in Figure 5. After the solid cutting simulation is completed, save the document and perform post-processing. Select "xcv-855-4axis. PST" as the document, the system automatically generates the NCI file and NC processing program. The NC program after "xcv-855-4axis. PST" file is shown in Figure 3 and Figure 4. The NC program of post-processing is uploaded to xcv-855 machining center for actual cutting verification, which fully meets the requirements of xcv-855 machining center. The actual machining results are shown in Figure 6, and the workpiece is measured. The practical results show that it can meet the accuracy requirements of parts processing, so as to verify the correctness of four-axis post-processing.

5. Conclusion

After repeated modification and debugging, the coordinate system of NC program generated by the post-processing file of Mastercam 9.1 software developed by the author is G54, which can automatically add M10 / M11 in the four-axis fixed axis machining, M29 in the rigid tapping, and the output angle is 0° ~ 360°. Taking the actual processing of the filter head array drum as an example, the correctness of the developed post-processing file is verified. The practical results show that there is no alarm in the process of processing, and the processed parts can meet the specified accuracy requirements, so as to verify the correctness of the post-processing file. It can save the cost of purchasing post-processing module for enterprises and is suitable for further promotion and application.

![Figure 3. Optimized coordinate system and NC program of four-axis unclamping and clamping.](image-url)
Figure 4. Optimized NC program of rigid tapping.

Figure 5. Simulation processing diagram of arrangement drum of filter nozzle.

Figure 6. Actual finished processing drawing.

Acknowledgements
The corresponding author of this paper is Xuan Lin Ye. This paper is supported by basic application research of Yunnan Science and Technology Department, Project number: 2013FD062, The People in charge of the project: Xuan Lin Ye.

References
[1] X.L.Ye, “CAD / CAM Technology Mastercam x5 Example Course,” in Jiangsu Phoenix Education Press. China, 2017, pp. 5-15.
[2] L.C. Lian, X.L.Zhang, “Study on Corresponding Relationship of MasterCAM Post-processor Drilling Cycle Parameters Page,” Machine Tool & Hydraulics, Vol. 42(2015) No. 10, p56-58+135.
[3] X.L.Ye, “Post Processing Optimization and Implementation of VM-32SA Vertical Machining Center Based on MasterCAM9.1,” in Machine Tool & Hydraulics, Vol. 45(2018) No.02, p13-16+5.
[4] X.L.Yan, W.L.Wang, “Solid modeling and NC machining of conical sine groove cam based on Mastercam,” in Manufacturing Technology & Machine Tool, Vol. 58(2009) No.08, p102-105.
[5] Z.W.Lu, Y.Cao, “Modelling and NC Simulating of the Cylindrical Cam Based on Mastercan X6,” in Modular Machine Tool & Automatic Manufacturing Technique, Vol. 54(2013) No.02, p111-113+116.