Optimization of machining parameters based on VERICUT three-axis milling

Xiaolei Li*, Wei Li and Xuerui Chen
School of Mechanics and Transportation, Southwest Forestry University Kunming, China

*E-mail: 2419476704@qq.com

Abstract: For the processing of complex parts, it is necessary to determine the optimal processing path and processing technology, in order to ensure its processing accuracy and processing efficiency. Because the complexity of parts, the application of equipment, the method of setting up the process have a direct impact on the processing effect, how to obtain the best processing effect and quality has become the problem that must be solved in the process of intelligent manufacturing. This paper introduces the method of obtaining the best machining parameters of complex parts by using VERICUT software. The experimental results show that the machining parameters obtained by this method can improve the efficiency and quality of machining complex parts by using CAM software. Through this way to complete the processing of complex parts for the complex parts processing technical personnel to provide a certain technical reference and reference role.

1. Introduction
In the process of NC programming, even if the parameters and specifications of each element of the process system are clear, use general CAM system to program different parts, there will also be a lot of repetitive setting of processing parameters[1]. When the part has complex surface, manual programming calculation is cumbersome, heavy workload, and even impossible to achieve, at this time, we should use computer automatic programming to improve efficiency. Computer automatic programming is often based on the set speed processing, for the same NC program, the feed rate is fixed, it cannot be adjusted according to the geometric characteristics of the workpiece, fixed feed rate affects machining efficiency. Therefore, in order to meet the requirements of machining technology, it is necessary to seek the optimal cutting parameters for the optimization of NC program.

2. Main contents and technical route
At present, CNC milling optimization mainly adopts online optimization and offline optimization. Online optimization is mostly based on adaptive control technology, which combines sensors and algorithms, continuously monitors machining parameters and optimizes cutting parameters with algorithms in real time during machining [1]. Off-line optimization can be summarized as tool path optimization, empty tool stroke optimization, and cutting parameter optimization by using software or algorithm to optimize tool path or related machining parameters. Wang Jiahai of Tongji University used VERICUT to optimize milling parameters, which was realized by optimizing feed speeds under different milling forces [2]. This optimization significantly reduced the time of idle cutting. According to the geometric characteristics of complex surfaces, Zhao Junhua of Southwest University optimized the NUMERICAL control program by using zone milling to improve the processing quality and...
efficiency[3]. Firstly, the 3d modeling of parts was completed, and the model was imported into CAM software for process design and simulation processing, and the post-processing program of corresponding processing equipment was obtained. Secondly, the simulation of the processing equipment was established based on VERICUT machine, on the model into the rear of the parts processed handler, virtual machining of parts, because in the CAM software set in the feed speed is fixed, cannot satisfy the needs of different process conditions on the speed in the machining, so on the premise of meet its processing technology, using VERICUT software to optimize the cutting parameters for different process conditions, to improve the machining efficiency. The virtual machining method can be used to simulate the machining process and cutting parameters, and then the final machining program can be applied to the machining of real equipment, so as to ensure the machining precision and quality of the machining parts, shorten the machining time and improve the machining efficiency.

3. MASTERCAM tool path simulation and post location management

3.1 Process Analysis

As shown in Figure 1, the workpiece is processed. The bottom of the workpiece is a square outline, while the top is a sphere. The outer contour of the sphere is uniformly distributed with 4 circular arc grooves. Blank is 60 mm × 60 mm × 40 mm aluminum alloy square, clamping vise the blank, the workpiece circular arc and outer contour processing on the surface, only one time clamping can complete all parts of the processing. CNC milling is one of the most common metal processing methods, which is widely used for rough and finish machining [4]. The processing technology of the workpiece is: (1) shape milling, (2) groove digging (- general groove digging), (3) surface rough machining -- parallel milling, (4) finishing outside contour, (5) surface finishing -- parallel milling

3.2 Tool path simulation

Figure 1. Three views of the artifact.

Figure 2. Tool path.
After determining the processing technology, the tool path should be compiled. Excellent tool path can guarantee the surface quality and processing efficiency of the parts [5]. Will a 60 mm × 60 mm × 40 mm blank model is imported into the MASTERCAM software, according to the above process analysis planning good cutting tool path and the processing parameters and the cutting tool, set up to simulate the processing to prepare. The processing plan is shown in Figure 2.

3.3 Verification of solid cutting
In order to verify the correctness of tool path, the MASTERCAM simulation processing function is used to detect whether the tool path is correct, whether there is interference and collision, etc. [6]. The six operations in Figure 2 are verified by solid cutting in MASTERCAM, and the workpiece shown in Figure 3 is obtained, which meets the expected requirements.

![Figure 3. Tool path.](image)

3.4 Post-processing
Post-processing is to convert the tool track generated by the system into G code instructions recognized by the machine tool in combination with the specific machine tool, which can be directly input into the CNC machine tool for processing. Convert the tool position file of the MASTERCAM above into a NUMERICAL control program that can be recognized by the CY-VMC850 vertical machining center.

4. Virtual simulation system construction and simulation processing
VERICUT simulation software was used to verify the numerical control program, and the machining results were judged in advance, so that the simulation system could give full play to its role of machining error correction, so as to ensure the machining accuracy and reliability of the workpiece. The virtual simulation process is as follows [7]:

4.1 Configuration of project tree
VERICUT software builds virtual NUMERICAL control system by configuring project tree, which includes control system, NUMERICAL control machine tool, accessory components, coordinate system, G code offset, machining tool, numerical control program, etc. UG software for 3 d modeling of machining center and export machine tool components with STL model, under the design BASE node of the tree of VERICUT topological structure according to the movement of the machine to add the corresponding machine tool components and import the corresponding STL model, after waiting for import fixture model set machine after stroke limit and limit parameters such as virtual machine after completed. The FANUC01-D NC system of the machining center is called from the VERICUT software library to configure the design tree of the above components.

4.2 Work piece installation and tool setting
Workpiece alignment tool includes the definition of program zero and the determination and setting of the tool program. The definition of program zero is used to determine the location of the tool, and the determination and setting of the tool program is realized by G code bias. Add a 60° in VERICUT 40 mm blank, adjust the position relationship between the blank and jig, and move them to the appropriate location, position as shown in figure 4. Since the NUMERICAL control program USES G54 to establish the workpiece coordinate system, the zero point of the program USES "work bias" and registers select "54".

4.3 Add tools and simulate machining
Add the cutter to the VERICUT tool library, whose cutter Numbers are # 2 and # 5 respectively, in which # 2 is the end mill with a diameter of 16mm and # 5 is the ball end mill with a diameter of 8mm. The clamping point, automatic aligning point and tool length compensation parameters of the two knives are set respectively. The numerical control program was imported into the numerical control program node of the design tree and the simulation was carried out. The processed workpiece was shown in Figure 5. The processed workpiece was consistent with the expectation.

5. Optimization of cutting parameters

5.1 Optimization principle
Formula (1) is the relationship between milling speed $V$ and spindle speed $n$:

$$V = \frac{\pi nd}{1000} \quad (1)$$

Where: $d$ is the diameter of the tool. As shown in Equation (2), is the relationship between the feed rate $f_Z$ and the feed speed $V_f$:

$$V_f = nf_Z \quad (2)$$
Where: $Z$ is the number of cutting tool teeth. It can be seen from equations (1) and (2) that, when the cutter is determined, the milling speed $V$ is mainly determined by the spindle speed $n$, while the feed speed $V_f$ is mainly determined by the spindle speed $n$ and the feed quantity $f_Z$ per tooth.

5.2 Optimization method
VERICUT provides NC optimization module. The optimization method is to optimize the spindle speed and feed speed in the processing process without changing the tool motion trajectory and the tool axis vector in the source program. Therefore, the optimized program is completely consistent with the tool path of the source program without causing quadratic errors and errors. In the optimization process, when the margin is small, the speed is improved. When the margin is large, reduce the speed, and then modify the program, and the new speed code will be inserted into the source program, forming a new NC program [8].

VERICUT cutting speed optimization method: Constant volume removal rate Cutting optimization: this method changes the feed speed according to the size of the material removal volume, this method is mainly used in the rough machining stage where the machining allowance changes greatly. (2) Optimization of constant cutting thickness mode: the method is mainly used for semi-finishing and finishing, which can improve the processing efficiency and surface quality. Combine the two methods, in the semi-finishing and finishing, VERICUT combined with the above two optimization methods, automatic comparison of the two methods of feed speed, select a smaller feed speed inserted into the program.

5.3 Optimization Steps
2 # cutters are mainly used for contour milling in rough machining and are therefore optimized with constant volume removal. Open the interface of tool manager and select # 2 tool to add optimization. #2 tool is mainly used for rough machining. Select the optimization method for volume removal, and the specific optimization parameters are shown in Figure 6. The optimized NC program is interrupted by software, with a new feed speed inserted at each step.

![Figure 6. Setting of cutting parameters.](image)

The same operation is carried out for #5 cutter, which mainly focuses on surface finishing. Constant cutting thickness optimization method is adopted to improve the surface machining quality of parts. The optimized NUMERICAL control program was imported into the software and simulated. The simulation results were consistent with the expected results, and the display time of the processing report was relatively reduced. The exported processing time report was shown in Figure 7, with the processing time reduced by 37%.
The comparison of NUMERICAL control programs before and after optimization is shown in FIG. 8. It can be seen from FIG. 7 and 8 that the original NUMERICAL control program was interrupted and new feed speeds were inserted in some lines, which greatly improved the processing efficiency of the optimized NUMERICAL control program.

Figure 8. Comparison of NUMERICAL control programs before and after optimization.

6. Processing verification
After optimization of NC program into the CY-VMC850 CNC machining center and the clamping a 60 mm 60 mm 40 mm blank, knife after the execution of NC program, after the machining process and workpiece as shown in figure 9.

7. Conclusion
The NC program obtained by optimizing the cutting parameters runs smoothly in the machining center spindle during the cutting without obvious impact and interference. The surface quality of the workpiece
after the machining is good. The measurement verifies that all the points meet the expected requirements, and the surface quality is greatly improved compared with that before the optimization. The correctness of simulated machining eliminates interference risk and provides safety guarantee for actual machining. The results show the feasibility of optimizing NC program by optimizing cutting parameters. It provides a feasible method for optimizing NC program.

Reference

[1] Keyi Zhu. Development of automatic Programming System for NC Machining of Die Profile [J]. Manufacturing Technology and Machine Tools, 04(2019).
[2] Hengli Liu, Yong WANG, Jingchuan DONG. Adaptive on-line optimization of numerical control milling parameters based on dynamic characteristics [J]. Manufacturing automation, 41(2019).
[3] Jiahai Wang. Numerical control milling parameter optimization based on VERICUT software, (2019).
[4] Yonghong Dai, xiong Han, hongying Zhu, xinghua Luo. Research on surface Quality of CNC Milling based on Equipment factor [J]. Manufacturing Technology and Machine Tools, 07(2019).
[5] Junhua Zhao, Li Li, Lingling Li, Conbo Li. The Method of milling and Machining Optimization for high efficiency and Energy Saving Complex Surface based on Zoning and number control [J]. China mechanical engineering, 30(2019).
[6] Guang Yang. Numerical Control Machining and Simulation of five-axis Integral Impeller [D]. Xi'an: Xi'an University of Science and Technology, (2013).
[7] Yang Yang, Liang Zhang, Caijun Liu, Lile Yang, Liangzhao Xiong. Research on the Whole body Number Control and Addition of complex ring frame work Pieces for a certain type of cabin [J]. Manufacturing Technology and Machine Tools, 10(2019).
[8] Deying Kong. Numerical Control Machining Simulation and Optimization based on UG/CAM and VERICUT [J]. Machine tools and hydraulics, 23 (2011).