Research on Numerical Control Programming Error Analysis of Complex Free-form Surface Machining Accuracy

Fang He¹, Jihui Luo², Meizi Tian¹, Renqi He¹*

¹School of Robot Engineering, Yangtze Normal University, Chongqing, 408100, PR China
²College of Materials Science and Engineering, Yangtze Normal University, Chongqing, 408100, PR China
*Corresponding author’s e-mail: 903380717@qq.com

Abstract: In order to improve the machining precision of NC machining of complex free-form surface, the factors affecting the precision of NC machining of free-form surface are analyzed. The main links of NC programming are studied. The method of reducing NC programming error and improving machining accuracy is proposed.

1. Introduction
Surface processing is mainly accomplished by CNC milling machine or multi-axis machine tool with machining center. For surface processing, especially complex surface (such as blades, dies), how to reduce machining errors and ensure machining accuracy has always been a research topic in the field of CNC machining. There are many errors in the whole process chain of surface machining. In order to improve the accuracy of surface machining, it is necessary to carefully analyze the error sources in the whole process chain of surface NC machining, and then take corresponding measures to avoid or compensate the errors according to the characteristics of the error sources [1].

In the whole process chain of surface NC machining, every error source in the whole process chain from part drawing to finished product will affect the processing accuracy of the surface. The main error sources in the whole process chain can be summarized as follows: spline approximation error of surface, roundness error of numerical calculation, interpolation principle error, control system algorithm error, feedback error of detection, geometric error of machine tool, tool factor error, workpiece factor error, stress deformation error, thermal deformation error and so on[2]. Error sources can generally be divided into three categories: the first is programming errors \( \delta_p \), which are generated in the programming process chain. The second category is process system error \( \delta_s \), which refers to the whole process system (including CNC machine tools, tools, fixtures and workpieces) involved in the cutting process. It mainly includes geometric error \( \delta_h \) of machine tools, control system error \( \delta_c \), feed system error \( \delta_f \), workpiece error \( \delta_w \), tool error \( \delta_t \), etc. The third type is the personnel error \( \delta_r \), which is caused by the operator's technical level. The total error of NC machining of curved surface should be the synthesis of various errors, that is:

\[
\delta = f (\delta_h, \delta_p, \delta_c, \delta_f, \delta_w, \delta_t, \delta_r, ...) 
\]

The total error of surface NC machining should be controlled within the requirement of surface machining accuracy. In this paper, the main error sources such as \( \delta_h, \delta_c, \delta_f \) or \( \delta_t \) are studied and discussed in many literatures, but \( \delta_p \) or \( \delta_r \) are not studied and discussed in depth. In this paper, the main error elements of programming error \( \delta_p \) are analyzed and discussed.
2. Connotation of programming error source

CNC machine tools can process a variety of shapes, sizes and precision parts, because there are programmers to develop different processing procedures. Surface Machining Programming Error $\delta_p$ Although the proportion of its total processing error $\delta$ is small, the general $\delta_p=(0.1~0.2)\delta$. However, the level of programming technology will determine the quality of processing procedures and programming time, thus affecting the entire workpiece processing production efficiency and processing quality. Therefore, reasonable determination and control of programming errors is an important issue in the programming of the program [3].

The automatic programming method commonly used in surface machining programming, the error element in the whole process chain of programming can be divided into four groups. One group is forced error $\delta_B$, which is the error caused by using approximate method to force surface modeling in front treatment. The two groups are numerical calculation errors, which are errors caused by numerical calculation in the programming process, including interpolation error $\delta_I$, line spacing error $\delta_H$ and rounding error $\delta_Y$. Three groups are to choose unreasonable error $\delta_X$, which is the error caused by the unreasonable selection in the planning tool path. The four groups are the test error, which is the error caused by the program test of tool interference test and post processing to generate the final processing program, which mainly includes the interference error $\delta_G$ and the post-processing error $\delta_Z$.

The sub-error source of the whole process chain compiled by the surface machining program can also be expressed as

$$\delta_p = f(\delta_B, \delta_I, \delta_Y, \delta_H, \delta_X, \delta_G, \delta_Z)$$

The machining error of $\delta_B$ in the approximation of part contour curve by near calculation method is called approximation error. The machining error of $\delta_I$ is called interpolation error when the interpolation segment approximates the contour curve of the part. When the $\delta_Y$ is based and the node is processed, the machining error produced when the fractional pulse circle is rounded into an integer pulse is called the rounding error. $\delta_H$ for Tool path planning, the machining error caused by cutting line spacing is called line spacing error. $\delta_X$ for automatic programming when the establishment of knife position files, due to the selection of tools, cutting parameters, advance and retreat knife mode, walking knife route and other unreasonable caused by the processing error, known as the choice error. $\delta_G$ is a machining error caused by the interference of tool CC point and theoretical surface, which is called interference error. The error $\delta_Z$ caused by the program inspection for the post-processing of the tool bit file is called the post-processing error.

3. Numerical computational error source

3.1 approximate error

If the surface is the original contour shape expressed as a series of phenotypic points, when the spline surface is approximated or fitted, the difference between the surface shape expressed and the original surface shape of the part is the approximation error or the fitting error $\delta_B$. This error only appears in the surface machining (such as reverse engineering) which is represented by a series of phenotypic points, which mainly depends on the reference profile error, the measurement error, and the finite error of the type value point. Therefore, on the premise of certain reference plane, it is an effective method to reduce $\delta_B$ by improving the measurement precision and increasing the measuring point as much as possible, then we use reasonable surface modeling method (such as NURBS method) to interpolate and fit it, and then perform a better fairing process (such as the energy-light method). In order to reduce the approximation error and improve the approximation precision. For surface modeling with drawings, the average size and average tolerance should be taken into account when considering the influence of tolerance requirements on modeling accuracy.

3.2 Interpolation error

When machining parts with NC machine tool, according to the interpolation different function of NC system, the contour of part surface can be approximated by linear-arc-parabola. When the surface
contour curve is approximated by a straight line or an arc, the maximum difference between the approximation curve and the actual surface contour curve is called interpolation error $\delta_I$. If the contour curve of the surface is made up of the same interpolation function as that of the NC system, such as regular surface machining such as sphere, cone or rotating paraboloid, it can be programmed by hand, the circular interpolation of cross section is completed in G17 mode (in XOY plane) with the circular interpolation function of G02 or G03 of numerical control system, and the step motion of the other direction (XOZ plane) is completed by programming the macro instruction variable at the same time, then there is no interpolation error (except for the small error generated by the interpolation method relative to the machining accuracy), that is $\delta_I=0$. In practical production, $\delta_I\neq0$ is usually required to process complex curved surfaces with automatic programming.

According to the principle of NC machining, in the multi-axis machining surface, the tool motion path produced by system interpolation is mostly the tool motion path of a curve (parameter line or section line) on the surface, which is composed of several small straight lines (or circular arc segments). As shown in figure 1, CC$_1$, CC$_2$ represents the start and end point of the tool contact of the interpolation line segment on the surface, CL$_1$, CL$_2$ is the starting point and end point of the corresponding tool point.

$\delta t$ denotes the linear forcing error, and $\delta_I$ is the normal vector rotation error and $\delta_I$ denotes the amount of deviation from the theoretical trajectory (theoretical surface) of the actual cutting path (actual machined surface), that is, the interpolation error (or chord height error), sometimes known as cutting tolerance, it must be less than the specified maximum allowable cutting tolerance, general taking $\delta_I<0.05\sim0.1\delta$. It can be obtained from the graph:

$$\delta_I = \delta t + \delta n = K_f L^2 (1 + R K_f) / 8$$

In the formula, $R$ and $K_f$ are the effective cutting radius of tool and the normal curvature of the machining surface along the feeding direction in the interpolation line section. $L$ is the chord length, which is approximated as step of the interpolation.

It can be known from the geometric relations of the formula that for a certain curved surface region, the larger $L$ leads to the $\delta_I$ and the error of the processing is larger. Conversely, the smaller error of the machining leads the the $\delta_I$ and the interpolation step size is small, and the programming and the processing efficiency is lower. Therefore, to ensure the processing accuracy, the following ways are often used for compensation:

1. Reduce interpolation step method. Increase the number of interpolation nodes and release the interpolation step size $L$ to reduce $\delta_I$.

2. Normal vector rotation error compensation method. For convex curved surface, the cutter cores CC$_1$ and CC$_2$ are moved outwards in the direction of normal direction of the machined surface.

$$\delta_e = R K_f^2 L^2 / 8$$

The value is such that the distance from the center of the tool to the interpolation line is close to the tool radius.

3. Tolerance symmetry method. In order to make the tool motion trajectory closer to the surface contour, the $\delta_I$ is symmetrically offset compensated by the theoretical contour of the relative surface in the close plane according to the principle of least squares. For the negative error caused by machining the convex surface, the tool core is offset by half a $\delta_I$ correction amount along the normal direction of the surface, so that the cutting tolerance is close to the theoretical contour, the positive error caused by the concave surface is Then, a half $\delta_I$ correction amount is moved inward along the surface normal.
3.3 Line spacing error

When performing tool path planning, the machining error caused by the cutting line spacing is called the line spacing error $\delta H$ (or residual height).

The cutting pitch (or cutting line spacing) is also called the cutting line width and should be set according to the tool condition and surface machining accuracy requirements [5]. As shown in Fig. 2, the cutting distance of the finished surface is closely related to the effective cutting radius $R$ of the tool, the effective radius of curvature $\rho$ of the surface to be machined, and the maximum allowable machining error $\tau$ of the residual height error.

The residual height $\delta H$ formed between the intersection line of the cutting surface between the two cutting lines and the theoretical curved surface should satisfy $\delta H < \tau$. The figure shows:

$$l \leq 2 \frac{2R\tau}{\sin \beta - Rk_b} \tag{2}$$

Where $\beta$ is the heel angle of the end mill, $k_b$ is the normal curvature of the machined surface along the cutting direction.

For surface finishing, controlling the residual height in actual programming is accomplished by changing the form of the tool rail and adjusting the line spacing. Generally, the residual height is controlled within the surface roughness $Ra$ required for the machined surface. The programmer should adopt different tool path forms and flexibly adjust the line spacing according to the different conditions of different processing areas. In order to achieve a certain processing accuracy and at the same time ensure the processing efficiency, the control of the residual height is mainly to make the residual height on all the surfaces relatively uniform, that is, to make the in-plane line spacing substantially consistent in different areas of the surface.

In general, when the radius of curvature is large, there is no sharp angle, and the machining accuracy is not high, the fixed step size and the cutting line width are used for programming; when the radius of curvature is small, there is a sharp angle, and the machining precision is high, the fixed string height error is adopted. And the way the residual height is programmed. When the regular surface machining is manually programmed, the machining accuracy is ensured by limiting the increment of the cutting travel to the step parameter; when the automatic surface machining is used for automatic programming, the spacing parameter is limited by the dialog box to ensure the machining accuracy.

3.4 Rounding error

The rounding error $\delta Y$ is the value of less than one pulse equivalent in the calculated value when data processing (such as calculating the base point of the curved surface curve, the coordinates of the node or the center of the circle), and is generated by rounding into an integer pulse value by using the rounding method. The value of $\delta Y$ does not exceed half of the pulse equivalent. Therefore, when using programming, use the reference coincidence and absolute coordinate programming as much as possible to prevent the accumulation of rounding errors. For example, when the pulse equivalent is 1 $\mu$m, the third digit after the decimal point must be retained in the numerical processing calculation of the base point (or node) so that $\delta Y < 0.5$ $\mu$m.
4. Selective error source

The irrationality error $\delta_X$ is selected as the machining error caused by the unreasonable selection of the tool, cutting parameters, advance and retreat mode, and path of the tool when designing the tool position file [6].

4.1 Tool error

According to the characteristics and machining requirements of machined surfaces, the selection of cutting tools should focus on the shape of the tool, the material of the tool, the size of the tool, the angle of the edge, and so on, in order to ensure the precision of the machining of the curved surface. The tool error in programming is mainly caused by the tool size. Tool selection should take into account:

- The tool radius should be less than the minimum curvature radius of the machined surface recess, so $R < 1/K_{\text{max}}$, where $K_{\text{max}}$ is the maximum normal curvature of the machined surface recess.
- Working efficiency. The larger the radius of the tool, the greater the cutting width at the same residual height, resulting in higher machining efficiency.
- Method vector rotation error. From formula $\delta_n = R K_f L^2 / 8$, you can see that the normal phase vector rotation error is proportional to the tool radius. For convex surfaces, the bigger the tool radius is, the better it is theoretically, but the proper tool radius must be chosen felicitously, especially when the normal phase vector rotation error is not compensated, the normal phase vector rotation error should be checked, if the error exceeds, the tool radius should be reduced as appropriate to reduce the normal phase vector rotation error.
- Attention should be paid to the error between the tool error compensation calculated for the tool position and the actual tool radius in multi-axis machining.
- Value range. Take the tool radius as far as possible in accordance with the specifications or standard series.
- The size of the tool shall match the size of the machined surface. Shouldn’t be here that use a spherical knife with a large radius while having a small machining surface, otherwise the tool can easily interfere (or collide) with the machined surface [7].

4.2 Cutting parameter error

In programming, the main cutting parameters are main spindle speed, feed speed, cutting depth and so on, in which feed speed is the main factor affecting interpolation accuracy:

$$L = \frac{T F}{60}$$  \hspace{1cm} (3)

In this formula, $L$ is the interpolation step, the unit is $\mu$m, $T$ is the interpolation period, the unit is the ms, $F$ is the feed speed and the unit is the $\mu$m/ms. Obviously, the faster the feed speed, the greater the step size and the lower the interpolation accuracy; the slower the feed speed, the smaller the step size and the higher the interpolation accuracy. Therefore, the selection of feed speed should be based on the requirement of machining precision, and on the premise of satisfying the production efficiency, select a smaller feed speed as far as possible, reduce the interpolation error, and improve the machining precision.

4.3 Start point, forward and retreat tool mode error

The starting point is the position of the tool when it starts to feed the tool at the feed speed before the surface finishing. To avoid high-speed rotating tool with G00 mode (moving track is often broken line) speed degree and the interference collision of work piece (or other tools) ensures the normal cutting speed of cutting tool, and the starting point (lower cutting point) is usually located above the machined surface or at a relatively empty position outside the blank profile of the surface. Avoid bumping surfaces and cutting tools.

The cutting tool mode (or route) refers to the movement and route of the tool approaching or leaving the machined surface of the work piece. The commonly used way is along the given vector direction and along the coordinates. Z axis direction, along surface or wheel the tangent vector direction of the profile, according to the spiral direction and other ways. In surface finishing, in general, convex surfaces usually choose to advance and retract in the direction of the tangent vector of
the surface or in the direction of the arc of the contour; concave surfaces usually choose to advance and retract in the direction of the arc of the contour or in the direction of the spiral. In this way, the tool can be prevented from leaving marks on the work piece advance and retreat, and the surface of the curved surface can be protected to ensure the processing precision.

4.4 Tool path error

Table 1. The characteristics and application of the commonly used cutting method in surface finishing

| Feeding way          | Characteristic                                                                 | Application                                                                 |
|----------------------|-------------------------------------------------------------------------------|-----------------------------------------------------------------------------|
| Surface finish parallel | Generate a set of cutter paths that are parallel to each other at a particular Angle | Finish the surface with small slope and gentle transition                    |
| Par.steep            | Generate tool path for removing residual material on curved slopes            | A surface, such as a steep incline, that is substantially parallel to a given direction |
| Radial finishing     | Generate radial tool path                                                     | A surface, such as a sphere, having radiative properties                     |
| Projection finishing | Project an existing tool path or geometry onto a surface to generate a new tool path | Finish the projection surface                                                |
| Streamline finishing | Generate tool path along curved surface streamline direction                 | To finish a single or contiguous surface, such as a corrugated surface       |
| Contour finishing    | Generate tool path along contour of surface                                   | The surface of straight wall or side wall with small slope                    |
| Contour and shallow  | Generate a tool path to remove part of the surface debris. The shallow area is determined by the slope | A surface with a flat finish to complement a steep surface                   |
| Pencil               | Generate a tool path to remove the intersecting parts of the surface         | Semi - fine or supplementary processing, remove the corner of the surface residual |
| Leftover             | Generate a tool path to remove the remaining material from the larger tool    | Semi - finishing, removing the residual material left by roughing broadsword |
| Scallop              | Generate a set of cutter paths around the workpiece surface with equal steps in the 3d direction | Semi - finished or finished surfaces with more variation in slope            |

Table 1 shows the commonly used cutting methods in the form of tool motion trajectory for surface finishing. Each of these methods has its own advantages and disadvantages and its application. In surface finish machining of the CAM programming was, therefore, must fully understand the CAM software provides the characteristics and applications of different kinds of feeding, according to the
different characteristics of the processed surface or area, processing requirements, and the characteristics of the cutting tools, machining path to be shortest, smooth cutting, machining accuracy and higher efficiency as the principle, especially should satisfy uniform surface cutting width and the height of the residual, selecting rational or a combination of feeding way, feeding way make the processed surface in theory and actual cutting all can get good precision [8].

5. Interference error
The interference error $\delta_G$ is the machining error caused by the interference between the CC point of the tool and the theoretical surface. With ‘CAM’ The tool path is displayed in the software, so as to judge whether the interference phenomena such as over-cutting, undercutting or collision occur in the process of surface machining. Simultaneous judgment the choice of the tool, the way to enter and retreat, whether the route is reasonable. Through tool path inspection, overcutting, undercutting, collision, etc., The commonly used verification methods are display method, section method, Numerical method, Simulation method and envelope surface method. In production, according to different characteristics of machined surfaces, machining requirements and cutting tool characteristics, reasonable selection is often made. Verification method or validation method the combination of, Observe areas where interference is possible, adjust parameter settings when problems are found, and re-check reasonable. Here, the leakage of observation is checked, the rationality of parameters is adjusted, and some errors are brought to the local area of the surface [9].

6. Post processing error
Post processing error $\delta_z$ caused by post-processing of the tool position file for program inspection. Post-processing is a tool position. Source the process of converting files into NC programs that can be executed by a specified CNC machine. Among them, coordinate transformation and machining mode and selected numerical control machine tool the type is closely related, In terms of error, Control the following aspect: 1 Corner direction problem. If the corner component of the two knife points is $\alpha_1, \alpha_2$. If $\alpha_1 - \alpha_2 > 180^\circ$, then $\alpha_2 = \alpha_2 - 360^\circ$. If $\alpha_1 - \alpha_2 < -180^\circ$, then $\alpha_2 = \alpha_2 + 360^\circ$. 2 Tool length compensation problem [10]. By improving the measurement accuracy, the length error is minimized and the machining error is reduced as much as possible. 3 Processing feed rate. If the two-tool position only has the tool axis vector change, the feed rate must be calculated from the corresponding limit value to prevent the error of the tool mark.

7. Conclusion
1) There are many error elements in the programming error, which should be based on the character of each element. Sex, take the corresponding avoidance or compensate measures as far as possible elimination or decrease error to ensure surface machining accuracy.
2) According to the theory of error composition, the following results can be obtained:

$$\delta_p = \sum \delta_e - \sum \delta_i \leq 0.1 \times 0.2 \delta$$

$\delta_e$ is the external tolerance error, the basis of trimming the amount of wear for the fitter; $\delta_i$ is the content difference error.
3) Only by summing up practical experience and improving programming technology, can we effectively avoid unreasonable error selection, post-processing error, reduce interference error, compensate numerical calculation error as much as possible, reduce the error caused by programming, and ensure that the error caused by programming can be effectively avoided. Surface machining precision to improve.

Acknowledgements
The project funded by Chongqing education committee science and technology research projects (KJ1401208, KJ15012025) and Fu Ling science and technology committee key research project (FLKJ2013XJYB004, FLKJ2013XJYB005).
Refferences

[1] LiuQitong.etc,“Research on Hybrid Tool Paths Generation Algorithm for NC Machining of Free-form Surfaces” [J]. Machine Tool & Hydraulics. Vo.46, No.19, pp.8-14, 2018.

[2] HeRenqi.etc,“The research on Surface manufacturing error and control methods” [J]. International Journal of Advancements in Computing Technology. Vo.4, No.19, pp.40-48, 2012.

[3] WangAiling.etc, “Modern NC Programming Technology and Its Application” [M]. Beijing: National Defense Industry Press,2010.

[4] ZhouXiaohong.etc,“Error Source and Countermeasure in NC Machining” [J]. Manufacturing automation. Vo.8, No.24, pp.20-23, 2002.

[5] YangChangqi.etc,“High Precision Calculation of Line Spacing in Multi-axis Machining of Free-form Surface” [J]. journal of chongqing university. Vo.5, No.27, pp.1-4, 2004.

[6] CheShufa.etc,“Error Analysis of NC Machining of Three-coordinate Surface with Spherical Cutter” [J]. Machine Tool & Hydraulics. Vo.3, No.5, pp.294-295, 2003.

[7] HeRenqi.etc,“Tool Path Designing for NC Machining of Complex Profile” [J]. International Journal of Digital Content Technology and its Applications. Vo.7, No.6, pp.668-675, 2013.

[8] WangWeibing.etc, “Practical Course of MasterCAM NC Programming” [M]. Beijing: Tsinghua University press,2011.

[9] FanLiuqun.etc,“Research on the Method of Improving the Motion Accuracy of NC Machine Tool” [J]. Journal of Tongji University. Vo.4, No.28, pp.443-448, 2000.

[10]LiHong.etc,“Research on NC Machining Position Error Model and Its Measurement and Compensation Method” [J]. Journal of Hunan Light Industry College. Vo.4, No.1, pp.37-43, 2004.