Application of dynamic milling in stainless steel processing

Wenju Shan

1 Zhuhai College of Jilin University, Zhuhai, Guangdong Province, China
E-mail: 36486846@qq.com

Abstract. This paper mainly introduces the method of parameter setting for NC programming of stainless steel parts by dynamic milling. Stainless steel is of high plasticity and toughness, serious hard working, large cutting force, high temperature in cutting area and easy wear of tool. It is difficult to process material. Dynamic motion technology is the newest NC programming technology of Mastercam software. It is an advanced machining idea. The tool path generated by the dynamic motion technology is more smooth, more efficient and more stable in the machining process. Dynamic motion technology is very suitable for cutting hard machining materials.

1. Cutting characteristics of stainless steel
Stainless steel generally refers to steel that resists atmospheric corrosion and resists corrosion in chemical mediator. Stainless steel is divided into martensitic stainless steel, ferritic stainless steel and austenitic stainless steel. Compared with medium carbon steel, stainless steel is hard-to-cut material, and its relative machinability is about 0.3 - 0.5. [1]

2. About dynamic motion technology
Mastercam is a commonly used CAD/CAM integration software. Consecutive years ranked first in the global number of similar software installed. dynamic motion technology is the newest NC programming technology of Mastercam software. It is different from traditional processing, and it is a breakthrough advanced processing idea.

A smooth tool path can improve machining speed more than any machine tool. With less retract and better tool contact control more billets can be safely removed in less time. Dynamic motion technology provides accurate cutting patterns and enables accurate machining of workpieces. Almost all machine tools can experience the acceleration of the speed and efficiency of the dynamic toolpath.

Dynamic milling uses cutting tool edges to cut. Instead of simply calculating the moving path of the tool, a series of algorithms are used to analyze the process of tool cutting and material removal. Adjust the cutting according to the movement of the cutter in the process, with stable material removal rates for rapid, smooth cutting. [2]

Dynamic milling with smooth tool path can uses the entire side edge of the milling cutter to cut, not only shorten the processing time of the workpiece, but also avoid unnecessary sudden swerve in the movement of the spindle of the machine tool. The cutting method can ensure the uniform fracture and stable discharge of the chip. A smooth chip removal process can quickly remove heat from the process and prevent heat from accumulating in the machined area. The surface hardening process of the material is suppressed and the tool wear during processing is reduced, Therefore, reduce the difficulty of processing hard cutting materials.
3. **3D dynamic milling parameter setting**

CNC programming of the workpiece using Mastercam2017, The Part drawing is shown in Figure 1.

![Figure 1 The part drawing](image)

The workpiece is made of stainless steel, which is difficult to process. It has the advantages of simple structure, low surface roughness and low tolerance grade, but its material removal rate is relatively large. Milling efficiency can be improved by using 3D dynamic milling. The following is a brief introduction to the parameter setting process of dynamic milling.

### 3.1 Workpiece blank setting

The blank is the state before the workpiece is machined. It should be set according to the actual processing. The correct blank setting can guarantee the correctness of the actual simulation before the actual processing. After drawing the solid model according to the drawing, ensure the center of the upper surface part of the workpiece is located at the origin of the system by drawing the boundary box. Then enter the milling Modular, Sets the blank size in the machine property group.

### 3.2 Tool parameter setting

Because the stainless steel material belongs to the difficult to processing material, therefore selects the good cutting performance carbide cutting tools to carry on the processing. The process of the workpiece is subjected to rough machining and finishing operation. At the beginning of the rough, you can use a larger diameter tool to improve the material removal rate. A flat end mill with a diameter of 16mm is used here. In finishing, in order to guarantee the processing of R3mm fillet, the ball end mill with 6mm diameter is selected. [3]

After the tool size is set, the machining parameters of the tool must be set. The main machining parameter settings include the choice of cutting speed and feedrate per blade. Then the spindle speed and feed speed are calculated automatically.

Reference NC machining manual, The cutting speed range of carbide milling cutters for milling of stainless steel is 90-100m/min. The range of feedrate per blade is 0.007~0.009*dmm/z. In accordance with the above range, select the cutting parameters and input to the tool parameter settings dialog box, automatically calculate the spindle speed and feed speed. [4]

### 3.3 Dynamic optirough parameter setting

The goal of roughing is to remove as much material as possible from the workpiece in the shortest possible time. Select the dynamic optirough command icon, select the whole entity as the drive surfaces, and draw a rectangle of 61*61mm size as the containment boundary. Select the 16mm flat end mill as the tool used for roughing. Then set the cut parameter. The cut parameter setting is shown in figure 2.
The key of parameter setting is the reasonable selection of step size and the setting of minimum toolpath radius. There are many factors that affect the choice of step size, and it is difficult to choose with a uniform standard. It can only be selected based on some principles or recommendations, and then adjust it according to the specific situation and experience.

The step size includes the settings of cutting depth and the side cutting depth. It determines the processing efficiency. However, excessive step size will lead to an increase in cutting resistance. The fracture of the cutting edge may occur during machining. [5] Generally choose the larger cutting, depth, and then select the corresponding side cutting depth according to different materials. For the flat end milling cutter, the cutting depth generally should not exceed 1.5 times of the diameter of the tool. This workpiece most deep processing position is 20mm, so the cutting depth can be set to 20. Input 20 to the stepdown dialog box in Reference NC machining manual. The side cutting depth of stainless steel materials can be set to 10% - 15% of the tool diameter. Input 10% to the stepover dialog box. In dynamic optirough, you also need to set up an important parameter, "Stepup." This parameter is mainly to set the Z layer of milling. This value determines the surface quality of the workpiece after roughing. The smaller of the value setting, the more uniform the rough machining allowance. Input 0.5 to the stepup dialog box.

Minimum toolpath radius determines the degree of smoothness of the toolpath. The bigger of the radius setting, the smoother of the toolpath, the more stable of the machining process. However, excessive minimum tool radius may result in increased machining allowance at corners. Input 25% to the minimum toolpath radius dialog box. Then set stock to leave, input 0.3 to the leave stock on drive walls dialog box, input 0.5 to the leave stock on the drive floors dialog box.

Then set the linking parameters. In order to reduce the time to raise the tool, set the lifting operation to "Minimum Vertical Retract". In the rough machining operation, the total tolerance value can be set to a larger value to improve the software operation speed. It can be set to 20-30% of the allowance. Set the total tolerance to 0.06 on the Arc Filter/Tolerance tab, turn on the “Line/Arc Filtering Setting” option. Create toolpath after setting is complete. Rough milling theory processing time is about 6 minutes.

3.4 Dynamic finishing parameter setting

The aim of finishing is to guarantee the geometrical dimension and surface precision of the parts. Sometimes, due to the larger diameter of roughing tools or rough machining parameters, the allowance values in some areas are obviously larger than the parameter settings. In order to ensure finish machining smoothly, the machining allowance must be uniform. Remaining operations are usually required. Copy and paste previous rough machining operations. Select the ball end mill with 6mm diameter. Check the "Rest material" option. Set the “Compute remaining stock option from” to “Toolpath group only”. Input 20% to the “stepover” dialog box. Input 10% to the “stepup” dialog box. Limit the processing depth...
from -9 to -14.7, Remaining is applied to the flat region of the workpiece. Remaining theory processing time is about 2 minutes. [6]

In the surface finishing setting, the machining parameters of the curved surface in the flat region and the steep region need to be set separately. The necessary clearance operations are required in the small fillet area. Select “hybrid” command icon, then select all surfaces except the plane region as the machining surface, and select the other surfaces as interference surfaces. Input 0 to the leave stock on drive walls dialog box, input 0 to the leave stock on the drive floors dialog box, input 0.1 to the check walls and floors dialog box. Then input 0.1 to the “3D stepover” dialog box. Other parameters follow default, Generate finishing operations toolpath. Finally, complete the programming of the horizontal and vertical planes with flat end mill. Finish milling theory processing time is about 8 minutes. The machining simulation results are shown in Figure 3.

Figure 3 The machining simulation results

4. Conclusion
Through the examples above, we can get a conclusion, that is, the dynamic milling motion of Mastercam is a safe, efficient programming. The dynamic milling programming is a more reliable method compared to the traditional milling methods, especially suit the stainless steel NC milling.

Acknowledgement
Premier-Discipline Enhancement Scheme Supported by Zhuhai Government Fund. (Fund code: 2015YXXK14)

References
[1] DH. Zhang, NC machining Handbook, Chemical industry press,2013, vol. 2, pp. 364–365.
[2] Q. Guo, Research on the methods and technologies for high performance peripheral machining of complex surface, Dalian University of Technology 2013, pp. 1–5.
[3] JX. Pan, Application of Mastercam software in high speed milling, NC Technology, 2006, vol. 1, pp. 43–46.
[4] Wg. Chen, Hao. Chen, NC machining tool, material structure and selection Handbook, China machine press, 2016, pp. 230–232.
[5] LD. Kang, FZ. Yang, CM. Tong, Research status and analysis of key technologies for high speed machining of aluminum alloy, Modern Manufacturing Engineering, 1 (2014) 83-86.
[6] YB. Wang, A side edge milling method based on Mastercam programming, Die manufacturing technology vol. 7, pp. 74–76, 2015.