Research on CNC Lathe Programming and Improving Machining Accuracy

Juan Zhang
Engineering Training Center, Shenyang Aerospace University, China.

Abstract. Nowadays, CNC technology is widely used in the manufacturing industry, and the maturity of CNC technology is also improving. By applying CNC technology to the process of CNC lathe processing, the accuracy of parts can be effectively improved and the quality of parts processing can be ensured. The development of the mechanical manufacturing industry and the improvement of manufacturing capabilities create good conditions. This paper mainly studies how the importance of programming in CNC lathes and how to use improved programming procedures to improve the machining accuracy of CNC lathes.

1. Introduction
CNC lathe programming for the use of CNC machine tools is the most important step. Combine the needs of part processing to complete the automatic programming of CNC lathes, it is able to flexibly use familiar program instructions to complete reasonable and efficient processing and support parts with satisfactory accuracy. Compared with other processing methods, the machining accuracy of CNC lathe is higher. To master programming skills, you first need to understand the role of various instructions, and clarify the effects of the combination of instructions, in order to complete the preparation and control of the program according to actual needs. In the operation of CNC lathe, reasonable programming and correct calling of the program are the most important and core content.

2. Programming Focus Of Cnc Lathe

2.1. Establish a coordinate system in the programming of CNC lathe
In the CNC lathe programming work, in order to show the movement of each axis of the lathe better, and to describe its movement in a streamlined language, a standard coordinate system has gradually formed in this industry and must be strengthened. The coordinate system in CNC lathe programming includes: machine coordinate system, working coordinate system, absolute coordinates and relative coordinates. When the machine is working, its arm must move. This requires a standard to measure its moving direction and moving distance. Therefore, a coordinate system can be established on the machine, called the machine coordinate system. Then determine the working coordinate system. The zero point of the working coordinate system of the working parts of the CNC lathe is usually set on the center line of the main shaft and close to the left and right end faces of the working original. The last very important step is to determine that the coordinates of all coordinate points in the coordinate system are calculated according to the coordinates of the same fixed point. When performing the main lathe operation, it is necessary to pay attention to distinguish between absolute coordinates and relative coordinates.
2.2. Reasonable use of code
We need to use different code to program the corresponding parts in combination with the actual situation, so that the efficiency of part processing can be effectively improved, because the requirements for Kyoto are different for different types of parts, in this way, the working efficiency of the part processing can be effectively improved. As for the method, the instructions that need to be used can be considered according to the requirements of the parts. Since the machining requirements of the lathe are various, the programming instructions are also very large, because in the process of programming, it is necessary to be familiar with the characteristics of each instruction, so that complete part processing tasks perfectly.

2.3. Improve accuracy
There must be errors in the design of the parts, the parts are difficult to achieve 100% accuracy, and different parts have different errors, so we must pay attention to this when using CNC lathes to machine parts. In the past, in the case of processing, it was not usually combined with all the parts, and all were processed according to a fixed standard, so that the processed parts would have a large error and the accuracy was seriously insufficient. We can analyze the basic dimensions of the part first, then program it, and compensate for the error with different tools, so that the accuracy of the part will be significantly improved. In addition, tolerance belts and basic dimensions can be adjusted, which also increases part accuracy.

3. Example

3.1. Actual case analysis
According to the part programming process shown in the figure, the blank material is 45 steel.

(1) Parts drawing analysis the part is composed of outer cylindrical surface, outer conical surface, circular arc surface and thread. The shape is more complicated. The Material is 45 steel, and the blank size is selected $\phi 35$mm.

![Fig 1: part drawing](image)

Determine the clamping method of the work-piece, since the work-piece is a solid shaft type part, and the length of the shaft is not too long. Therefore, the right End face and the outer circle of the
work-piece are used 35mm as the positioning Reference. The work-piece is clamped using a common three-jaw chuck, and the right end of the work-piece is taken as the origin of the work-piece coordinate system.

Determine the CNC machining tool, machining process and selection of cutting amount. According to the shape and processing requirements of the part, select the following tool: T01 bit 90° external turning tool, T02 is a 5mm wide cutting knife, T30 is a 60° thread turning tool.

3.2. CNC lathe end face machining programming and existing problems
Turning the end face is the first process of turning, which is especially important. The quality of the end face will seriously affect the smoothness and dimensional accuracy of the end face of the machined part. At present, in the machining and machining of end face machining CNC lathe programs, the machining end faces are usually processed with the following degrees:

N10 T0101 M03 S400; (T01 is a face turning tool)
N20 G00 X52 M08; (Gross outer diameter is $\phi$ 50mm)
N30 G01 X0 F0.15; (F value is selected by material condition)

Obviously, the above program instructions and formats are error-free, but in the actual processing process, the end face of the work-piece obtained by machining is often rough and the processing effect is not satisfactory. Especially close to the center of the work-piece, if the amount of backing knife is too large, it is prone to a sickle accident, and sometimes even cause personal injury to the operator. Therefore, analyzing the causes of this problem, finding solutions and improving procedures is key.

Solution

\[ v = \frac{r \cdot w}{1000} = \frac{2 \pi r \cdot n}{1000} \] . \( v \) is the cutting line speed (m/min), D is the diameter of the machined part, n is the spindle speed. The cutting end face is proportional to the diameter. When the end face is rough, the linear velocity will decrease, and the cutting speed of the tool will be inconsistent with the given spindle speed of the lathe, which will result in a dirty surface of the work-piece, especially near the center position. The end face is processed by the above procedure, and the surface roughness of the work-piece cannot be ignored.

When solving this problem, consider using the constant line speed to control the G96 command. This command is commonly used in surface treatments such as taper, end face, and arc in lathe machining. The linear velocity of the spindle specified by S in the G96 command is in m/min. This command is generally used when the section of the turning disc part or the diameter of the part varies greatly, so that the diameter can be changed, but the linear speed of the main shaft does not change, so that the cutting speed is constant, and the surface roughness of the work-piece remains the same. It is known from: \( n = \frac{1000 \cdot V}{\pi \cdot d} \) that the smaller d is, the larger n is. Therefore, we have to limit the maximum linear velocity to the spindle. Expressed with G50 S. Many beginners are unfamiliar with the G96 instructions, or the understanding is not thorough enough, so that it is rarely considered to solve such problems with this instruction.

Effective use of instruction programming must understand the meaning of the G96 command. The so-called constant line speed control mainly means that the line speed behind S can be kept constant. As the position of the tool changes, the line speed can be used to calculate the spindle speed. G96 is executed. The constant line speed control of the main motor realizes constant, stable and maintenance cutting rate of the work-piece processing process by changing the rotation speed to ensure the machining accuracy. The G96 needs to use G50 to set the maximum spindle speed limit. When the weak tangential speed is constant, it will change with the size of the machined parts. The smaller the diameter, the easier the spindle speed will be too large, resulting in an accident. So the program in question can be changed to the following program:

N10 G50 S2000; (Indicates that the maximum speed limit is 2000r/min)
N20 G96 M03 S200 T0101; (Indicates that the cutting point line speed control is less than 200m/min)
N30  G00  X55  Z0;  
N40  G01  X0  F200; (F value is selected by material condition) 
N50  G97  S400; (The speed is the spindle speed after the constant line speed is canceled, and the system starts the default G97 program) 

Programming with G96 or machining with a lathe with a main motor that can achieve constant line speed control can solve the above problems to a large extent, but if programming is not possible with programming G96, it is not possible to control the motor with constant line speed. The machine tool can then be made as follows:

N10  T0101  M03  S400; (T01 is a face turning tool) 
N20  G0  X52  Z0  M08; (Gross outer diameter is $\phi$ 50mm) N30  G1  X4  F0.18; 
N40  X0  F0.08; 

When the degree of use of the tool is approaching the center of the workpiece, the radial feed rate is lowered to improve the surface finish of the machined part near the center of the work-piece. If the spindle speed is constant, the required precision processing can be completed according to the requirements of the part processing design, and the F value in the program can be adjusted according to the processing experience, so as to achieve the desired processing effect.

4. Conclusion
Reasonable programming and correct calling of programs in CNC machine tools are the most important and core content. Even if the programming procedure is correct, the lack of experience will still result in an unsatisfactory processing result, therefore, CNC lathes not only greatly reduce the burden on the staff, but also improve work efficiency and part accuracy. When programming and machining CNC lathes, we must

First establish a coordinate system, plan each part, and then use different programming instructions according to the size and precision required by different parts to meet the needs of various parts.

References
[1] Z.H.Yu: Thoughts on the Programming and Operation of CNC Lathe [J], (2015)No.11,p: 299. (In Chinese).
[2] X.Y.Yang, C.L.Cao: CNC lathe programming processing scheme determination and detail processing in programming(Jiangxi Chemical Industry) [J], (2014) No.3,p:12-14. (In Chinese).
[3] Q.W.Wang, X.W.Xu: Discussion on the Simple Skills of CNC Lathe Programming [J], (2011)No.8, p: 178. (In Chinese).
[4] G.X.Ren: CNC lathe machining process and programming operation(Beijing: Mechanical Technology Press) [M].
[5] J.C.Mo: Discussion on the Programming Skills of CNC Lathe(Guangdong Education) [J],(2015)No.4,p:116-117. (In Chinese).