Analysis on the Design of the Huazhong Numerical Control Lathe Trapezoidal Threads Parametric based on the Macro Program

Tang Bo
Engineering Training Center
Shenyang Aerospace University
Shenyang, Liaoning, China

Abstract — The trapezoidal thread processing skills belong to the content that the senior technicians should grasp. Compared with the common thread of NC machining, the NC machining process of the trapezoidal thread is more complex, and the difficulty of programming is greater. The paper summarizes the NC machining and teaching experience for many years, based on the macro program for parametric programming function instruction of trapezoidal thread processing, and it is applied in practice, the good results have been achieved. The trapezoidal thread processing process has been solved by the stratified cutting tool force allocation problem of the cutting edge, and the breaking edge phenomenon can be effectively avoided; Meanwhile, the parametric program editor made of trapezoidal thread the numerical control turning macro program can be applied to all kinds of inside and outside of the trapezoidal thread processing, which can reduces the trapezoidal thread of the NC programming and machining difficulty.

Keywords - Macro program; HuazhongNC lathe; Trapezoidal thread; Parameterized

I. INTRODUCTION

The trapezoidal thread is a common, ordinary lathe, in the production of trapezoidal thread processing is one of the most basic elements, but in the use of numerical control lathe for trapezoidal thread processing, often because of reasons of processing technology, cutting tool, but few, even it is put forward on the NC lathe is not trapezoidal thread processing, obviously this idea is wrong. Process analysis is reasonable, in fact, as long as the rational use of cutting tool and use processing instructions properly, can process a qualified trapezoidal thread in CNC lathe. The CNC programming as one of the key technology of NC machining, the program compiled largely determines the quality and efficiency of product processing accuracy and production efficiency.

II. THE THEORETICAL BASIS

A. The threading tool material selection

Commonly used threading tool materials have high speed steel and carbide two categories: HSS threading tool, sharpened up easily, and has good toughness, point is not easy to split, threads of the turning surface roughness is small, but poor thermal stability of high speed steel, suitable for low speed cutting thread; Carbide threading tool of high hardness, good heat resistance, but poor toughness, apply to the high speed turning thread.

B. Thread Angle on turning Angle

Threading, due to the influence of the helix, cutting plane and the position change, tool rake Angle and back Angle at work, before the grinding Angle (static) before and after the grinding Angle (static) after different numerical. The degree of change depends on the size of the work piece thread Angle, triangle thread the thread Angle is compared commonly small, are small impact. But in the trapezoid and thread pitch larger influence is big. Therefore, in the threading tool grinding, the effect must be considered.

1) The changes of the Angle of the tool on both sides.
   The work on either side of the turning tool after angle generally is $3^\circ \sim 5^\circ$, When there is no thread Angle (such as transverse feed tank turning), after work on either side of the turning tool Angle and Angle of the same after grinding ($\alpha_{OL}$) should be equal to the Angle and screw thread Angle after work ($\psi$). In order to ensure enough strength of the cutting tools, turning on the right side of the grinding Angle ($\alpha_{OR}$) shall be equal to the job after minus the thread Angle. That is:
   $$\alpha_{OL} = (3^\circ - 5^\circ) + \psi$$
   $$\alpha_{OR} = (3^\circ - 5^\circ) - \psi$$
   The situation is the opposite when turning left thread.

2) The change of tool rake Angle. Due to the position changed, work on either side of the tool rake Angle or Angle before ($\gamma_0$) grinding are not equal.
   If grinding rake Angle are on either side of the tool is $0^\circ$, Thread, right at the turning on the right side of the work for negative rake Angle, cutting is not smooth, chip removal is difficult. In order to improve the situation, will be turning tool cutting edges on both sides of the plane perpendicular to the spiral line clamp, makes the left side of the blade Angle are before work is $0^\circ$. Or the rake Angle for grinding is $6^\circ \sim 8^\circ$, the smooth cutting tool and chip removal.

3) The name of the trapezoidal thread parts, and a formula to calculate the following table.
III. THE MACRO PROGRAM AND NC PROGRAMMING BY HAND

As software development, the computer automatically programming software is widely used at home and abroad, there are few programmers who choose CNC programming method for production and processing by hand. Many programmers have ignored the role of macro program in numerical control programming, the macro program can be made manually programmed is more intelligent, and can improve the capability of NC programming of numerical control programming.

Macro programming because it is in the general manual programming instruction on the basis of development, but the general manual programming instructions in the application of NC machining range of reduced, resulting in enterprise production practice, very few people will use the macro program. Practice shows that the macro program has an irreplaceable role in the numerical control processing, many programmers think learning will have a difficult macro program, and there is no real realize the effect of macro programming, therefore the macro program is not in our country were popular in the field of NC machining. Macroinstruction program of less number of lines, convenient and modify, just change some parameters of the assignment will be able to become a new processing procedure, greatly reducing the new program's establishment and inspection time, during the processing machine tool reads a macro instruction, reading speed, and much higher than the read speed of computer programming software, without waiting for the phenomenon in the machine, machining efficiency and greatly improve the surface quality of parts.

NC programming macro definition:

In the numerical control manual programming, use some numerical variable assignment method to express, and read the numerical control system of numerical value is variable, and the variable assigned values are regularly changes, when programming can also be added a mathematical calculation and logic operation, using this method of manual program called macro program.

The advantage of using NC macro program programming:

a. Although a lot of numerical control processing enterprises at home and abroad are using computer automatic programming software for NC machining program, regardless of the program size using the programming software programming. In this case the programmer programming thinking will be greatly restricted, if meet with complex parts will not start, even cannot be solved by programming software. If in the heart of the NC machining program by hand has the advantage of parts with manual programming, programmers programming skills will be gradually improved. On the basis of general macro instructions into the manual programming, the NC machining program is that use the variable makes the regular shape of part programming easier. When writing program more readable, easy to program changes and read. NC machine tools is also very like to read such programs while reading program, because the macro program can make the CNC machine to read faster, machine processing and faster response speed.

b. Programmers often encountered in NC machining of curved surface machining, surface in the use of computer programming software programming will be found that the number of rows in a lot of NC machining program, even the capacity of the NC machine tool can not meet such procedures. At this time can only be used online transmission method for NC machining of the program, and in the use of computer numerical control processing program, as long as there is a parameter change in the programming software, the NC machining tool motion trajectory is calculated, a new this work can only be done on the computer. In calculation of cutter path at the time of calculation speed is related to the computer's hardware configuration, program calculation be transmitted to the CNC machine again. This method of programming will waste more processing time. If you will regularly when NC machining parts surface shape the macro program to programming can make the process more concise, processing performance will be better.

c. A lot of mechanical parts processing quantity, the shape of the parts don't need too much mathematical calculation, parts of the size tolerance and behavior very close tolerance requirement. Due to mechanical parts machining quantity, efficiency is the numerical control processing enterprises key consideration. Using the macro program can play a biggest machine tool processing efficiency, save processing time. In processing machine parts if there are any shapes similar shapes, programmers just change one or several variables for new parts processing, the program modified don't need to program for processing, because it is the excellent program after many processing and tested.

d. The macro program in processing machinery parts, because of the mechanical parts many shapes are simple plane, inclined plane, cylindrical and convex cylindrical cavity, such as when programming does not need to do too much math. In processing machinery parts, often encounter the shape of the general manual programming instructions will not be able to process, such as ellipse, hyperbola, parabola, involutes shape, etc. Such shapes available macro program compiled with do not need mathematical calculation values, as long as the trigonometric function relation or curve equation can be expressed in macro instruction, so the macro program application in the fields of mechanical processing has a very wide range of prospects.

e. Some of NC machining parts shape the use of computer programming software program won't work, such as elliptical shape threads, thread pitch can change, and processing thread on the sphere, these complex shape only using the macro program is the best solution.
IV. EXPLORING THE TRAPEZOIDAL THREAD PROCESSING TECHNOLOGY

A. The trapezoidal thread processing technological process

Trapezoidal thread processing technology is mainly with the aid of computer software to realize the control of working process, the process mainly includes the following aspects: first, adjust the trapezoidal thread processing precision of the NC machine tools used, and then the corresponding NC machine tools to add the selected materials, in the numerical control machine tool before formal job, the corresponding machine operators must test the accuracy of machine tools, to ensure the accuracy of the machine tool processing conforms to the requirements of the, which meet the needs of processing. Second, numerical control machine operator need to use the standard in the process of machining machine tool cutting method and the clamping way, we commonly used in the manufacturing process of the clamping way mainly through the use of three catch plate fixed at one end of the work piece clamping, on the other side of the work piece with top to resist, at the same time, in order to avoid in the process of machine tool work piece axis vibration caused by the machine tools machining precision no problem occurred, one end of the NC machine tool clamping usually needs to be set into terraced. In addition, in order to avoid the deformation of trapezoidal thread appears problem, machine tool operation personnel should pay attention to in the process of machine tool processing leading to resist components cannot be too loose or too tight, when a certain amount of space needed.

B. The trapezoidal thread cutting tool selection and the choice of cutting method

There are two main types of materials used in the trapezoidal thread turning tool, namely, cemented carbide and high speed steel. The rationality of the threading tool material machining efficiency of machine tool and its own life has important influence. Easy to HSS tool grinding blade, sharp edge, good toughness, and the point is not easy to break, so it is mainly suitable for cutting plastic material, but because of its poor resistance to high temperature, at the time of high speed rotating vulnerable to wear and tear. Cutting brittle materials, such as casting copper, cast iron carbide cutting tools are suitable. Carbide cutting tools with high speed cutting low wear rate, high efficiency, and high temperature resistance is strong, but caused by the cutter blade easily happen through or fracture problem.

The trapezoidal thread turning processing method mainly includes two kinds, namely, high speed and low speed. The low speed turning method is mainly suitable for single piece production or those who are higher in the production of trapezoidal thread processing precision requirement. The traditional turning processing methods mainly include grooving, turning around, straight entered, oblique entered and stratified cutting method, etc. Straight entered, for example, is mainly along the transverse cutter feed, but due to the cutting feed mainly adopts three blades, so the tool wear is very serious, and it's easy to have a firm knife and so on.

C. The trapezoidal thread turning processing process

Trapezoidal thread turning processing process mainly includes two kinds of machining process, namely rough turning machining process and finish turning process. In the machining process of the specific parts often need to first after rough lanes to realize the preliminary processing of the components (mainly slot), in order to ensure the applicability of the rough machining components, ensure it won't appear in finish turning processing deformation problem, need in the process of machining in radial processing room reserved 0.1 0.2 mm, also need to make sure that the top of trapezoidal thread diameter and the gap between the reserve of bottom diameter 0.2 0.4 mm, this is also the effective measures to ensure the device processing accuracy. In the process of machining fine turning, turning usually requires two fine processing, and will be introduced to the G92 system dynamic error detector in the process of the turning the two fine processing, machine tool turning measurements usually 0.2-0.3 mm, this time will need to measure and adjust the fine turning inner diameter size, this step is mainly implemented by finishing tool, generally need to be changed back turning to 0.25 0.55 mm. Due to coarse turning processing and fine processing is adopted in the process of straight into the cutting method, the blade of the finishing tool when processing needs to have certain requirements, namely to make sure that the blade width equal to the width and thread slot. Then need to Seiko program compiled, but need to prepare for the radial dimensions when compiling thread bottom diameter, engineering transmission rate needs to be set to 120-150 r/min, finishing processing technology.

V. THE DIFFICULTY OF THE NUMERICAL CONTROL TURNING TRAPEZOIDAL THREADS PROCESSING

A. The CNC turnings can't directly use ordinary lathe of trapezoidal thread processing method

The trapezoidal thread processing methods such as ordinary lathe use about cutting method, straight slot and ladder slot cannot be directly used in the numerical control turning. Because NC turning is cancelled on ordinary lathe mechanical transmission chain, through mounted on the spindle at the ends of the synchronous belt is connected to the spindle pulse encoder, thus constituted the main shaft and the big slide between the drive screw transmission chains. Spindle when threading, pulse encoder signal output at the same time two road, all the way by the programmer in the spindle speed and pitch value given in the process, determine the speed of servo motor, guarantee the main shaft and two kinds of servo motor speed form a strict ratio; Another way is to control the positioning of the Z axis to ensure that the threading tool in the process of multiple circular cutting, tool tip is always in the spiral groove teeth. If suffering pierced a knife or cutting tool in machining process damaged need to replace the threading tool and make the numerical control lathe when they stop rotating spindle pulse encoder to stop working, and the two way to stop the output signal, at this
time to install the threading tool is difficult to accurately in a threading tool before the turning out of the spiral groove, so as to increase the difficulty of the knife, appear even tooth phenomenon.

B. trapezoidal thread cutter easy to occur in the numerical control turning processing knife phenomenon

Due to no small skateboarding devices on NC lathe, unable to realize trapezoidal thread cutting tool in cutting process of moving, the 3 cutting edge at the same time cause the lathe tool for cutting, resulting in a large cutting force, a knife phenomenon. On the plain lathe processing trapezoidal thread, the whole process in the control of the operator, if found a knife or other abnormal phenomena, can timely interrupt processing and take corresponding measures to prevent damage of the cutting tool. CNC machined trapezoidal thread, the machining process is controlled by the NC program in advance, when appear abnormal phenomenon such as operator is likely to be caused by not timely interrupt processing cutting tool damage.

The Trapezoidal Thread Processing

(1) The straight entered

Threading tool X to intermittent feed to the depths of the teeth Using this method processing trapezoidal thread, thread turning tool cutting, the three sides were involved in the first several times because of cutting quantity is not big, no problem, as the cutting depth increases gradually, which causes the difficulty in processing chip, cutting force and cutting heat increases, the blade wear serious, also can produce "Mr. Knife" and "blasting knife" phenomenon. The method of NC lathe can be used instruction G92, but obviously, this method is not suitable for turning trapezoidal thread, commonly used for pitch or lead less than 3 mm triangular thread processing.

(2) The oblique entered

Threading tool along the direction of the tooth type Angle of oblique intermittent is feed to the depths of the teeth. Using this method processing trapezoidal thread, thread turning tool is always only one on the left side of the cutting edges for cutting, so that the clearance is smooth, the stress of the blade and heating straighter entered improved, not easy to cause "Mr. Knife" phenomenon in turning. This method can be used on the CNC lathe G76 instruction to implement.

(3) The art of using saber

Threading tool along the direction of the tooth type Angle staggered intervals feed to the tooth depth. The class method can overcome the straight entered machining defects, the method is tool side force, working steady, not easy to produce "Mr. Knife" and "blasting knife" wait for a phenomenon, it will be the trapezoidal thread cutting feed process law, so that we can use the macro program variable control technology, realization of trapezoidal thread of parametric programming and machining. The method can be used on the CNC lathe G32 instruction to implement the transferring.

The Parametric Design

(1) The processing method of choice

Based on trapezoidal knowledge processing (as shown in figure 2) artifacts, on the CNC lathe processing trapezoidal thread, adopt a clip on the three jaw chuck clamping. In order to facilitate the knife and program, the program from the center of the origin set on the right side of the workpiece end face, and because of high speed machining is trapezoidal thread, so the selection of carbide cutting tools.

When the high-speed cutting trapezoidal thread, in order to prevent "collapse" blade, requirements in the trapezoidal thread machining, cutting force can't be too big. Cutting tool cannot simultaneously on three sides, therefore cannot use the thread cutting instructions G92 for straight entered or straight into the cutting method, also cannot use G76 thread cutting composite cycle instruction to processing. According to the long-term practical experience, the use of thread cutting instructions G92 and use subroutine calls for stratified cutting method. As shown in figure 2, the effect is good, and safe, reliable, and easy.

(2) The choice of cutting tool geometry

Under conditions of high speed cutting of trapezoidal thread, first calculate the spiral Angle, in order to correctly determine the geometric Angle of grinding tool. Make the tool sharper and is advantageous to the chip breaker, by using only one tool "button" phenomenon will not occur.
(3) The Angle of the trapezoidal thread turning processing

In order to the smooth completion of trapezoidal thread in CNC lathe processing, and ensure the stability of processing quality, also need to trapezoidal thread turning tool geometric angles are the correct grinding.

a. The trapezoidal thread turning tool on both sides of the blade sharpening of anterior horn

Trapezoidal thread pitch is relatively large, because the cutting plane is changed; base level also produces change subsequently. Back before the blade on one side of the spiral Angle is negative, this is very averse to processing. In longitudinal trapezoidal thread grinding of tool rake Angle, can be on the front knife grinding out a thread Angle at the same time, make the negative rake Angle is changed to zero, thus greatly improves the cutting effect.

b. The tool knife Angle correction

CNC machined trapezoidal thread, the work piece tooth type Angle is half Angle on tooth type parameters guided, nothing to do with vertical lathe tool rake Angle and blade Angle. But in order to guarantee the tool has enough strength and reduce the surface roughness of the work piece when machining, trapezoidal thread turning tool knife sharp corners slightly less than the best tooth type Angle.

Coarse turning is finished, if the speed directly transferred to the low speed fine turning call the original program, will mess tooth, collapse edge or crash, so we are going to solve before the low speed turning tool teeth problems. When considering the low speed turning tool feed speed is slow, we can use the naked eye to observe threading tool with the turning thread profile chamfer whether alignment, specific operation method is as follows:

a. To change the work piece coordinate system, make the tool cutting thread does not contact surface, after rough turning parked the roughing tool in position X200 z50’s, at this point in the entry mode input G50 X192 after implementation, namely change the coordinate system, the equivalent of the coordinate system origin along the X axis moving direction of the 4 mm, which is slightly larger than a tooth high distance. At this time will lathe spindle speed down, such as transferred to 25 r/ min, it is to run the program, roughing tool surface, the turning is less than the position of near surface.

b. The tool on the trapezoidal thread slot back to out of the turning, because the tool feed speed is slow, at this point we can see the turning tool with the original turning trapezoidal thread slot is not coincidence, tool offset a short distance, as shown in figure 5, the purpose is to make tool on the turning of the trapezoidal thread slot again. The principle of operation and on the NC lathe turning multiple threads is the same, is by changing the threading tool before turning the starting point of the axial position to achieve a goal, which is to modify the above program G00 X40 Z - 20 of 20. We can judge by naked eye needs to adjust about distance, if changes to Z - 20-21, after running the program, found that turning tool and the turning out of the trapezoidal thread of trough is still not fully to is. It is to modify the Z value, run the program again, until the turning tool with trapezoidal thread slot for completely.

c. To restore the original work piece coordinate system, start to finish. In order to facilitate understanding and less error-prone, still moving tool to X200 z50’s position, under the entry, perform G50 X208, repair the original work piece coordinate system, to run the program, can finish turning trapezoidal thread at low speed. Fine turning is by changing the threading tool before turning the starting point of the axial location method to repair the light of the trapezoidal thread two side, at the same time by measuring, control cutting the number of threads to suit the requirements of dimensional accuracy. Through experiment, in the high-speed and low-speed cutting revolutions are fixed, the tool need to offset displacement is fixed, with this data, after turning collapse edge, or need to change after wear knife can need not repeat steps, directly in the low speed fine turning will be the starting point of threading migration corresponding position. In this paper, the machining examples of the numerical control machine tool plant in FANUC NC lathe, and when the lathe spindle speed from 560 r/min speed to 25 r/min when trapezoidal screw lathe tool need to shift to the left of 1.8 mm on the Z axis.

VI. THE MEASUREMENT OF THE TRAPEZOIDAL THREAD

The Common thread tooth type is a lot of more phyletic, the diameter size is measured with vernier caliper and micrometer only, while the tooth type test requires special detection tools. Measuring triangle thread by thread micrometer is most easy and convenient; for the batch production of the thread, you must choose special testing tools, such as thread gauge and ring gauge can greatly improve the efficiency of detection. Should choose three stitches trapezoidal thread measurement method more

Figure 2 Processing the Finished Product

DOI 10.2013/IJSSST.a.17.07.10 10.5 ISSN: 1473-804x online, 1473-8031 print
accurate. Here introduces three stitches measuring method of measuring method of trapezoidal thread. Three stitches measurement method is to measure the external thread pitch diameter of a more sophisticated way. Some precision demand is higher, suitable for measuring the thread Angle is less than 4° artifacts. Measurement when placed three diameter equal to the amount of needle in the thread that corresponds to the spiral groove, micrometer measure both sides of the distance between the needle vertex M, select 5 points record number, take the average.

VII. SUMMARY

From the numerical control thread processing principle, the parameterized macro program is suitable for all kinds of internal and external screw thread processing, especially suitable for larger thread pitch and worm processing. When the pitch or modulus increase, the tooth depth is increasing, more can show around mobile in reducing cutting force in the process of cutting edge. Due to allow the user to use macro program variable arithmetic and logical operations and conditional transfer function, so as to make the program with similar processing operation is more convenient and easier. The operators can be the same processing operation such as fixed processing cycle as generic programming, and use a simple instruction when processing out user macro program, greatly reduce the workload of programming.

ACKNOWLEDGMENTS

This paper was supported by 2014 Shenyang Aerospace University non doctoral Youth Growth Fund (Tech) Project: "Research On The Application Of Parametric Machining Of Trapezoidal Threads: The Huazhong CNC Lathe Based On The Macro Program (201419Y)"

REFERENCES