Journal of Engineering Mechanics and Machinery (2023)
Clausius Scientific Press, Canada
DOI: 10.23977/jemm.2023.080103
ISSN 2371-9133 Vol. 8 Num. 1

Processing Method of Rectangular Unequal Distance Thread

Liao Hengqiang

Guangdong Huali Technician College, Guangzhou, 511300, China

Keywords: Unequal Distance; Rectangular Thread; CNC Machining; Turning

Abstract: With the improvement of the social demand for our daily life, the products needed in our life are getting higher and higher, and the requirements for the mechanical structure and functions need to be constantly improved, which leads to an increase in the demand for unequal distance threads. This will further improve the processing technology and processing efficiency of unequal distance threads. In the following, we will talk about the processing method of rectangular unequal distance threads, focusing on the basic idea of NC processing of unequal distance threads Programming skills, rough and finish machining methods in CNC machining, tool application.

1. Introduction

Unequal pitch screw threads are widely used in daily life, such as sanitary products, feeding unequal pitch screw mechanisms, machining advance and retreat mechanisms, unequal pitch propeller devices on ships, unequal pitch screw lifting devices, etc. Unequal pitch thread can be divided into two cases according to the pitch: one is unequal pitch of equal width groove thread; the other is teeth with equal width and unequal spacing. Unequally spaced threads can be divided into three types according to the tooth type: triangular unequally spaced threads, trapezoidal unequally spaced threads, and rectangular unequally spaced threads.

In order to improve the processing efficiency of machining unequal pitch threads and save costs, CNC machine tools are now used to process unequal pitch threads, especially unequal pitch threads with equal width slots. CNC machine tools have a higher production efficiency in machining unequal pitch threads. However, the accuracy will be improved if the unequal distance thread is processed by numerical control turning method. The following is an example of machining unequal pitch threads with rectangular teeth with equal width slots.

2. Material selection and tool angle requirements for machining rectangular unequal distance thread turning tools

The shape of turning tools for machining rectangular threads is basically similar to that of cutting tools, which can be divided into rough machining turning tools and finish machining turning tools. In order to prevent vibration or tool breakage caused by excessive cutting force, the edges on both sides of rough machining turning tools do not participate in processing, and the finish machining tools are divided into finish machining left tooth tools and right tooth tools. When selecting forming tools, the influence of thread rising angle and the technical requirements of thread groove width must be considered. Here, the width of the tool head is required to be 3mm.

2.1. Geometric angle parameters for machining rectangular unequal pitch thread turning tool

(1) The length of the cutter head is 2-3 mm longer than the height of the tooth profile.

(2) Roughing turning tool with cutter head width is generally 2mm smaller than the minimum groove width of rectangular unequal pitch thread; finishing turning tools are generally divided into two types: right and left tools for machining tooth profiles. [1]

(3) Rough machining of longitudinal front angle is 6 °for steel parts and 12 °for aluminum alloys.

(4) Finish the left tooth surface for 2 °and the right tooth surface for 12 °at the longitudinal anterior angle.

(5) Rear angle of rectangular unequal pitch thread turning tool (turning right thread) [3].

$\tan\psi = Ph/\pi d2 = 11/(3.14 \times 33) = 0.106$

$\psi = 6°6'$

$\alpha left = 5° + 6° = 11°$

$\alpha right = 5° - 6° = -1°$

among ψ--- thread rise angle

To make the car sharp, α Take about 1 °~3 ° on the right.

(6) The secondary deflection angle of rectangular unequal pitch thread turning tool is generally 2 °. [2]

2.2. Processing technology for machining unequal pitch thread

2.2.1. Determine the tool setting point position.

Generally, the right end face is taken as the tool setting point position, but there is a turning tool for machining the right tooth of rectangular unequal distance thread, so it is necessary to set the tool in the Z-axis direction of the tool retracting groove to the right tool.

1) Workpiece programming origin

On the workpiece coordinate system, determine the calculation and programming origin of the workpiece contour coordinate value, which is called the workpiece programming origin. It belongs to a floating coordinate system, which is used as the origin to establish a rectangular coordinate system for numerical conversion.

On the CNC lathe, the workpiece programming origin is generally set at the intersection of the axis line of the part and the end faces on both sides of the part, as shown in the figure 1: O1 and O2.

Figure 1: O1 and O2

Principles for determining the workpiece programming origin:

(1)The position of the workpiece programming origin should be known on the given drawing.

(2)In the coordinate system established at this point, the relationship between geometric elements shall be concise and clear, so as to facilitate the determination of coordinate values.

(3) It is convenient to set the program origin.

2) Program origin

The program origin refers to the starting point of the tool (tool tip) when the machining program is executed, which is also called the program origin. The position of the program origin is relative to the programming origin position of the workpiece.

Generally, after a part is processed, the tool returns to the program origin position, waits for the command to execute the processing of the next part, and sets the position of the program origin.

2.2.2. The tool material shall be determined

In order to improve production efficiency, cemented carbide or coated tools shall be used for rough machining, and high-speed steel tools shall be used for finish machining to meet the processing quality requirements.

2.2.3. Determine the cutting parameters

In order to improve the production efficiency during rough machining, the spindle speed is 800r/mim, the X-axis feed is 0.2mm, and in order to improve the machining accuracy, the spindle speed is 30r/mim.

2.2.4. Determine the processing plan

The general principle for formulating the processing plan is rough first and then fine, near first and then far, inside first and then outside, with the least program segments, the shortest tool path, and special treatment under special circumstances.

(1) Coarse before refined

In the turning process, the rough machining process shall be arranged first, and the machining allowance of the blank shall be removed in a short time to improve the production efficiency. At the same time, the allowance uniformity of the finish machining shall be met as far as possible. To ensure the finishing quality of the parts, the finishing allowance left after rough machining is more uniform to meet the requirements of finishing.

(2) Minimum program segments

In the processing of CNC lathe, on the premise of ensuring the processing efficiency, the processing of parts can be realized with the minimum number of program segments, so as to simplify the program, reduce the programming workload, reduce the program error rate, and facilitate the inspection and modification of the program.

2.2.5. In order to meet the requirements of machining accuracy

The finishing allowance is 0.2 mm in the X axis direction and 0.2 mm in the Z axis direction on both sides of the tooth.

2.2.6. Use cutting fluid

Rough processing uses cutting fluid to reduce the processing temperature, which can improve the processing efficiency, but also reduce the processing deformation, and the use of cutting fluid can improve the surface roughness of the workpiece.

3. The cutting command for NC lathe to process variable pitch thread is G34

G34 X (U) - Z (W) - F (I) - J-K-R -

F. The I code word is used to specify the thread pitch of the thread, and can process metric or inch variable pitch straight threads, tapered threads, and end face threads.

F: Specify lead.

1: Specifies the number of teeth per inch.

R: Incremental value or decrement value of spindle pitch per revolution, R=F2-F1, R with direction; When F1 > F2, R is negative, which means the pitch decreases; When F1 ＜ F2, the pitch increases when R is positive.

J: Thread tailing is the amount of movement in the direction of the minor axis, unit:; Mm/inch, with positive and negative directions; If the minor axis is the X axis, this value is specified for the radius.

K: Thread tailing is the length in the direction of the major axis. If the major axis is the X axis, the value is specified as the radius; No direction.

The tool path is a straight line from the starting point of X and Z axes to the end point specified by the program section. The coordinate axis with large displacement from the starting point to the end point (X axis is based on the radius value) is called the major axis, and the other coordinate axis is called the minor axis. During the movement, the major axis moves for one lead every revolution of the main axis, and a certain pitch every revolution of the main axis is to continuously increase or decrease the specified value, forming a spiral groove with variable pitch on the working surface, so as to realize the processing of variable pitch threads. During cutting, the tool retraction can be set.

Attention:

1) J and K are modal codes. When J and K are omitted from the next program in case of continuous thread cutting, tail off shall be performed according to the previous J and K values.

2) When J or J, K are omitted, there is no tool retraction; When winning K, press K=J to exit.

3) J=0 or J=0, K=0, no tail off.

4) When J ≠ 0, K=0, press J=K to exit the tail.

5) When J=0, K ≠ 0, there is no tail off.

6) The current program is thread cutting, and the next program segment is also thread cutting. When the next segment starts to cut, the one turn signal of the spindle position encoder is not detected, and thread processing is started directly. This function can realize continuous thread processing.

The GSK980TDC numerical control system is taken as an example to introduce the application of this function.

GSK980TDC has a variety of thread cutting functions that can be used to process threads without undercut. However, because there is a deceleration process in the X axis and Z axis at the beginning and end of thread cutting, the pitch error is large at this time, so it is necessary to reserve the thread lead in length and tool retraction distance at the actual thread start and end.

Under the condition that the thread pitch is determined, the movement speed of X axis and Z axis during thread cutting is determined by the spindle speed, which has nothing to do with the multiplying rate of cutting feed speed. The multiplying rate of the spindle during thread cutting is related to the spindle speed. When the spindle speed changes, errors will occur in the thread pitch due to the acceleration and deceleration of X axis and Z axis. Therefore, the spindle speed is not adjusted during thread cutting, let alone stopping the spindle. Stopping the spindle will cause damage to tools and workpieces.

(1) Groove equal width tooth pitch

The tooth form is pitch form (the first pitch is 4, and the distance between the tool fast feed and the end face of the workpiece is 5mm). The intersection point between the right end face of the workpiece and the axis is the origin of the workpiece coordinate system.

0000l // Main program name

N10 G00 X120 Z80 // Program tool change point position

N20 M03 M08 G97 S800 // Spindle forward turning on cutting fluid speed 800r / min Closing constant linear speed

N30 T010l // Call 1 # cutter compensation

N40 G00 X60 Z8 /// Quickly move to the first tool point
N50 M98 P25 0002 // Call subprogram O0002 for 25 times
N60 G00 X120 // Fast moving X direction away from the workpiece
N70 Z80 // Fast moving z-direction away from the workpiece
N80 T0100 // Cancel the compensation of No.1 knife
N90 M30 // End the program and return to the beginning of the program
O0002 // subprogram name
N10 GOO U-10.2 // Indication of fast approaching workpiece by phase feed along x-axis direction
N20 G34 Z-70 F4 J1 K2 R1 /// Unequally spaced processing thread end point Z-axis end point coordinate 70mm, the lead of the first turn of thread is 4mm, and the increase of each turn is 2mm
N30 G0 U10//Fast and fast tool retraction along the x direction is 10mm.
N40 GOO Z8//Quickly move to the starting point of the next cutting
N50 M99 // End of subprogram

(2)Teeth with equal width and unequal spacing

The use of macro variables simplifies programming when using G34 cycle cutting multiple times. Value: δ 1=4mm, δ 2=4mm, total cutting depth 4mm, total cutting cycle 15 times; The lower feed is 0.5mm, and the decrease is 0.2mm each time. The minimum feed is 0.2mm.

(3) Program [2]

00003 // Main program name
N10 G50 X100.Z50 // Set workpiece coordinate system
N20 M03 G97 M08 S50 /// Spindle rotates forward, turn on cutting fluid at constant speed of 50r/min
N30 T0101 // Call 1 # tool compensation, tool width 5mm
N40 G00 X60 Z8 /// Quickly move to the first tool point
N50 M98 P50 0004 // Call subprogram O0004 for 50 times
N60 G00 X100 ///The tool moves quickly away from the workpiece
N70 Z50 // Fast moving z-direction away from the workpiece
N80 T0100 // Cancel the compensation of No.1 knife
N90 T0202 // Call 2 # cutter compensation, the cutter width is 3mm
N100 G00 X60 Z6.3 // Quickly point the second tool starting point
N110 M98 P50 0005 // Call subprogram O0005,50 times
N120G00 X60 Z5.6 /// Quickly point the third tool point
N130 M98 P50 0006 // Call subprogram 00,006,50 times
N140 G00 X60 Z5 // fast moving x direction away from the workpiece
N150 M98 P50 0007 // Call subprogram O0007,50 times
N160 G00 X100 ///The tool moves rapidly in the x direction away from the workpiece
N170 Z50 // Fast moving z-direction away from the workpiece
N180 T0200 // Cancel the compensation of No. 2 knife
N190 M30 // End the program and return to the beginning of the program
00004 // Main program
G00 X55 Z4 M03 S800
G65 H1 P # 102 Q0.5 First feed: assigned value # 102=0.5
G65 H1 P # 102 Q0 cycle count: assigned value # 103=0
N10 G65 H02 P # 104 Q # 103 R1 Cycle counting start: # 104=# 103+1
G65 H01 P#103 Q#104 #103=#104
G65 H81 P30 Q # 104 R15 Total cutting cycles: # 104=15, transfer to N30
Sequence segment
G00 U-5 feed to 50mm

G65 H01 P # 100 Q # 102 Cutting feed rate: # 100=# 102
G00U # 100 feed
G34 W-78 F3.8 J5 K2 R0.2 variable pitch thread cutting
G00 U5 tool retraction
Z4 Z-axis returns to the starting point
G00 U-5
G65 H3 P # 101 Q # 100 R0.2 Declining connection of cutting feed again: # 101=# 100-2
G65 H1 P # 102 Q # 101 Reassign # 102=# 101
G65 H86 P20 Q # 102 R0.2 Feed rate judgment: transfer to N20 program section when # 102 ≤ 0.2mm
G65 H80 P10 unconditionally transferred to N10 block
N20 G65 H01 P=# 102 Q0.2 Minimum feed: @ 102=0.2
G65 H80 P10 unconditionally transferred to N10 block
N30 M30

4. Conclusion

Due to the rapid development of modern machinery, more and more workpieces need to use variable pitch thread, and the method of machining variable pitch thread is also constantly improving. The efficiency of machining variable pitch thread has been improved from high-speed steel tool to coated tool. Different variable pitch threads have different machining methods, so it must be summarized in actual operation to select the best machining method and programming method.

References

[1] Wang Yumei. Analysis and application of thread processing methods for CNC machining centers [J]. Coal Mining Machinery, 2018, 39 (12): 2.
[2] Zhou Y. CNC Cutting Process of Thread. Machinist, cold working, 2006, 58-62.