Mechatronics and Automation Technology
J.-Y. Xu (Ed.)

doi:10.3233/ATDE231095

Development of a Drilling Cycle Program with Decreasing Depth of Pecking Drilling

Xinze LIN[a], Ping LIU[b,1] and Min WU[b]

[a] *NINGBO HANVOS KENT SCHOOL, The Barstow School Ningbo Campus, Ningbo, Zhejiang 315202, China*
[b] *Intelligent Equipment Research Institute, Ningbo Polytechnic, Ningbo, Zhejiang 315800, China*

Abstract. Deep hole machining accounts for a large proportion in machining. Due to the special depth of the hole being processed, the conventional deep hole drilling fixed cycle can lead to overheating of the tool after several times of drilling. Using the macro functions of the FANUC system, a new G code is developed to gradually reduce the depth of the drilling hole when processing deep holes. After being verified by the GLU28x40 CNC, the newly defined G code is input and programmed according to the prescribed syntax, Equivalent to a fixed loop implanted in the control system. When machining deep holes, it can effectively alleviate the problem of excessive tool temperature during the pecking drilling process and improve processing accuracy.

Keywords. Macro program; Deep hole drilling; FANUC system.

1. Introduction

Deep hole machining, as a special metal cutting process, is an important branch of mechanical processing, originating in the manufacturing of gun barrels and barrels in the military industry. Therefore, its application to civil products (such as engines, machine tool manufacturing, automobiles, textile machinery, petrochemical industry, and instrument manufacturing) is relatively late. With the rapid development of the mechanical processing industry, the proportion of deep hole machining is increasing [1], In ordinary machining, hole processing accounts for 20% to 30%, while in hole processing, deep hole processing accounts for 40% [2,3]. At the same time, higher requirements are put forward for the processing accuracy of deep holes. It can be seen that deep hole processing technology occupies a very important position in the mechanical manufacturing industry.

In the FANUC Series 0i Mate-MC system, there are two drilling cycles: G73 high-speed chip removal drilling cycle and G83 chip removal drilling cycle. G73 is applied to the processing of shallow holes, with less tool retraction action per processing; G83 is used for deep hole processing. In order to effectively remove chips during each processing, the cutter is retreated to the lower cutting point of the drilling cycle (hereinafter referred to as the R point). However, if the hole is too deep, after

[1] Corresponding Author, Ping LIU, Intelligent Equipment Research Institute, Ningbo Polytechnic, Ningbo, Zhejiang 315800, China; Emial: 52599570@qq.com.

several times of drilling, and after returning to the R point, there is a problem of insufficient cooling, which can affect the machining accuracy of the hole. When drilling deep holes, the depth of the drilling is gradually reduced, that is, the decreasing depth pecking cycle can effectively solve the cooling problem in drilling. However, the FANUC system does not provide a fixed cycle for this function.

Using the macro functions of the FANUC system, redefine a new G code, and develop a drilling cycle with a gradually decreasing pecking depth on the FANUC system through a user-defined G code method. This solves the problem of tool overheating in deep hole processing, effectively improves cooling, and improves the processing accuracy of deep holes. The new G code is used in the same way as the system's inherent fixed cycle code, equivalent to a fixed cycle embedded in the control system, This G code can be used even if the programmer does not understand the basic principles of macro programs.

2. Method of User-defined G Code

In the FANUC system[4], a maximum of 10 custom G code macro programs are allowed to be defined. The G code is similar to the inherent G code of the system and is used in the same way as fixed loop code, that is, defined within a program segment. When defining the G code, it is necessary to specify the meaning of the letter address used. The method of calling a macro program using G code can be used even if the programmer has little programming experience and does not have the knowledge to understand the macro program. Without knowing the serial number of the macro program being called, it can be used like standard G code (such as G83).

In the FANUC system, it is specified that G code can only be used to call macro programs using 9010-9019, and the G code used to call macro programs can be 1-9999. However, only unused G code can be used during the definition, otherwise the original G code will be redefined. At the same time, the G code program number of calling macro programs (9010-9019) is specified by specific parameters (6050-6059), and the relationship is shown in table 1. For example, 6050 is 183, G183 will call O9010. If the program corresponding to the defined G code does not exist, the system will recall the alarm message "NUMBER NOT FOUND".

Table 1. Correspondence between parameter number and program number when the G code calls macro program

Parameter No.	program No.
6050	O9010
6051	O9011
6052	O9012
……	……
6058	O9018
6059	O9019

3. Drilling Cycle Development with Decreasing Drilling Depth

3.1. Definition of the G183 Code Macro Program

Using the macro function of the FANUC system, define a new G code (G183), which has the drilling cycle function of gradually reducing the drilling depth during deep hole machining. Store 183 in the parameter 6050 in table 1, with the corresponding program number O9010, and the resulting tool point is shown in figure 1.

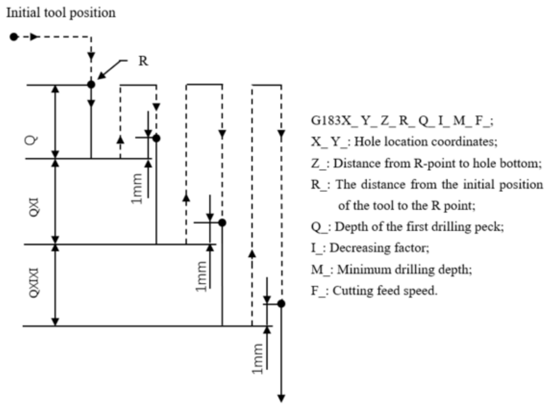

Figure 1. Milling machine descending drilling fixed cycle

Code usage considerations:

1.Firstly, the tool quickly moves to the XY coordinate point (the axial position of the current hole), and then rapidly feeds along the Z-axis direction to the point R to start drilling. The tool trajectory is the same as the G83 cycle standard, but the pecking depth gradually decreases with the influence of a decreasing factor (i.e., Q-$Q \times I$ -$Q \times I \times I$...);

2. All machining and tool retraction operations are performed along the axial direction of the hole (here, the Z-axis direction), which is shown separately in this figure for clarity;

3. When the calculated drilling depth $(Q \times I^n)$ is less than the specified minimum drilling depth M, the subsequent drilling depth is assigned a value of M;

4. All tool retraction operations are fast feed;

5. When drilling each time, the tool quickly feeds from point R to a distance of 1mm from the previous drilling depth, and then increases the depth of the hole with the new drilling depth to start the feeding action;

6. When machining to the bottom of the hole, stop. Finally, retract the tool to the point R or the initial Z coordinate position. To meet the requirements of deep hole machining accuracy, the final pecking drill should have a machining action from the point R to the bottom of the hole.

3.2. Algorithm of the G183 Code Macro Program

The According to the definition requirements of G183 code, the algorithm of its macro program O9010 is as follows:

1. Storing the original position of the Z coordinate; Quickly move to the XY coordinate position (hole center); Quickly move to the cutting point R of the drilling cycle; Calculate<required machining depth>(the starting point of drilling is point R);

2. If<machining depth required>≤ Q, skip to step 7;

3. Drill from the R point to the hole depth Q at the set feed speed; Storing the current Z coordinate position; Set<Required Processing Depth>=<Required Processing Depth>- Q; Quickly retract the cutter to the R point.

4. If<machining depth required>≤ M (minimum drilling depth) or a new drilling depth value (QXI), skip to step 6;

5. Calculate the new drilling depth. If this value is<M, assign the hole depth to M; Quickly move to a depth of 1mm in the Z direction from the last drilling depth; Feeding action, increasing the hole depth with a new pecking depth; Storing the current Z coordinate position; Quickly retract to point R; Set<machining depth required>=<machining depth required>-<current drilling pecking>;

6. Tap the required depth of drilling, quickly move to a position 1mm from the last machining depth in the Z direction, and then drill to the bottom of the hole with a feed action;

7. Single continuous processing until the final depth;

8. If G99 is currently active, quickly retract the cutter to the R point; If G98 retracts the tool to the original Z coordinate position at the current effective block speed

9. Return to the calling program

The flowchart of the algorithm is shown in figure 2.

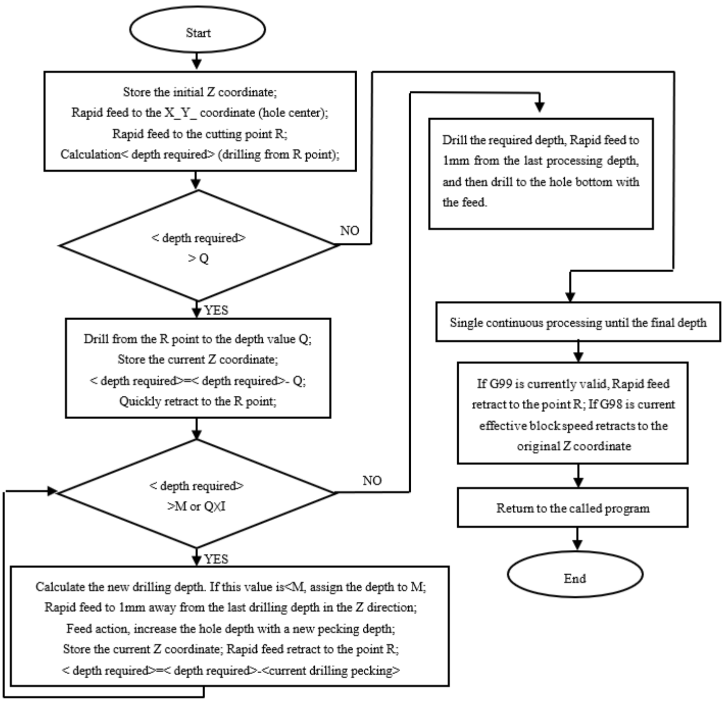

Figure 2. Flow chart of CNC decreasing drilling fixed cycle algorithm

3.3. Compilation of the G183 Code Macro Program

In this example, all coordinate values are in the absolute coordinate G90 mode, and the macro program uses parameters to specify type I. The meanings of the variables to be used are as follows:

X (#24) = X coordinate of the hole center
Y (#25) = Y coordinate of the hole center
Z (#26) = Z coordinate of the hole center
R (#18) = Z coordinate of the point R
Q (#17) = the first drilling depth
I (#4)= Decreasing value (each drilling depth is equal to the previous drilling depth multiplied by the decreasing value)
M (#13) = the minimum drilling depth
F (#9) = Feed
#101= Z coordinate of the stored tool initial position
#102= Z coordinate storage of the stored tool initial position<machining depth required>(remaining depth to be drilled)
#103= The Z coordinate after each machining is stored, and this value is updated after each machining
#104= Current drilling depth (multiplied by the value of the last drilling depth by # 4)

The specific procedure of macro O9010 is as follows:

```
O9010 (REGRESSIVE PECKING ON MILL M)
IF [#24 EQ#0] THEN #3000=1(SPECTIFY HOLE X- COORDINATE);
IF [#25 EQ #0]THEN #3000=2(SPECTIFY HOLE Y-COORDINATE);
IF [#26 EQ #0] THEN # 3000=3(SPECTIFY HOLE DEPTH)
IF [#18 EQ #0] THEN #3000=4(SPECTIEYR- POINT);
IF [#17EQ#0THEN #3000=5(SPECTIFY FIRST PECKLENGTH);
IF [#13 EQ#0] THEN #3000=6(SPECTIFY MINMUM PECKLENGTH);
IF #4EQ #07 THEN #3000=7(SPECTIFY REGRESSION FACTOR);
IF #9EQ#0] THEN #9=#4109;
IF #9 EQ 0] THEN #3000=8(SPECTIFY FEEDRATE)
(ALARM IF THERE IS AN INPUT ERROR)
#101=#5043;( STORE INITIAL Z COONRDINATE)
G00X#24￥#25; (RAPID FEED TO HOLE CERTER)
Z#18; (RAPID FEED TO POINT R)
#102=#18-#26; (CALCULATE THE REQUIRED MACHINING DEPTH)
IF[#102 LE 0] THEN #3000=9(IMPROPER R OR Z SPECIFIED)
#17=ABS[#17]; (BOTH POSITIVE AND NEGATIVE VALUES OF THE FIRST DRILLING DEPTH MUST BE PROCESSED ALONG THE NEGATIVE DIRECTION OF THE Z AXIS)
#13=ABS[#13]; (BOTH POSITIVE AND NEGATIVE VALUES OF MINIMUM DRILL DEPTH)
IF [#102 LE #17]GOTO10; (IF THE PRCESSING DEPTH IS LESS THAN OR EQUAL TO THE FIRST DRILLING DEPTH,SKIP TO N10 DRILLING INSTEAD OF DRILLING)
G01 Z[#18-#17]F#9;  (THE PECKING DISTANCE IS CALCULATED FROM THE R POINT AND THE FIRST DRILLING IS COMPLETED)
#103=#5043;( STORE THE CURRENT Z COORDINATES)
```

```
#102=#102-#17; (UPDATE REUIRED MACHINING DEPTH)
G00Z#18; (RAPID FEED RETRACTS TO THE POINT R)
#104=#17; (INITIALIZE THE CURRENT DRILLING DEPTH)
WHILE [[#102GT #13] AND [#102 GT [#104 *#4]]]DO1;( ENTER THE
DRILLING CYCLE.IF THE MACHINING DEPTH IS REUIRED TO BE LESS
THAN OR EQUAL TO THE MINIMUM DRILLING DEPTH OR THE NEW
DRILLING DEPTH, EXIT THE CYCLE)
#104=#104*#4;
IF [#104 LT #13]THEN #104=13; (CALCULATE THE NEW DRILLING
DEPTH)
G00Z [#103+1];(RAPID FEED TO THE POSITION 1mm FROM THE LAST
MACHINING DEPTH IN THE Z DIRECTION)
G01Z[#103-#104]( FEED,INCREASE THE HOLE DEPTH WITH A NEW
PECKING DEPTH)
#103=#5043; (STORE THE CURRENT Z COORDINATE)
G00 Z#18;(RAPID FEED RETRACTION TO THE POINT R)
#102=#102-#104;( UPDATE REUIRED MACHINING DEPTH)
END 1;(JUMP TO THE WHILE SECTION AND VERIFY THE CYCLE
CONDITIONS AGAIN)
G00 Z [#103 +1];
G01Z#26; (REMAINING DEPTH OF DRILLING)
GOTO 20; (JUMP TO N20 AND FINISH THE FINAL RETRACTION)
N10G01Z#26E#9;(SINGLE FEED ACTION REACHES THE FINAL DEPTH)
N20G00Z#18;(RETRACT TO THE POINT R)
If [# 4010 EQ 99] GOTO 30;(IF G99 IS VALID,JUMP TO THE END OF THE
PROGRAM)
G00 Z#101 (IF G98 IS VALID, RETRACT THE TOOL TO THE ORIGINAL Z
COORDINATE)
N30 M99;(RETURN TO THE CALLED PROGRAM)
```

3.4. Examples of the G183 Command Application

The G183 applies the mm mode and can only be called in absolute coordinate mode. To illustrate the purpose of the G183 command, the hole shown in figure 3 is processed. The hole is very deep (the ratio of hole depth to diameter is large), so the G183 code is required for processing.

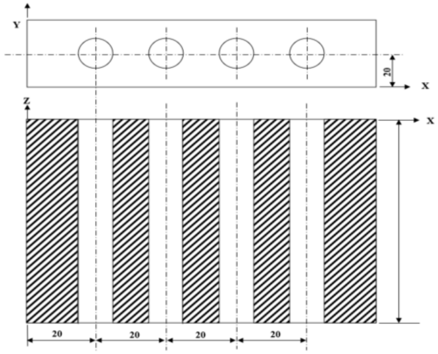

Figure 3. The plate with deep holes

G21 G94 G54;	G183 X20 Y10 Z-53 R2 Q10 10.8 M5 F20;
G91 G28 Z0;	G183 X40Y10Z-53 R2 Q10 I0.8 M5;
G28 X0 Y0;	G183 X60 Y10 Z-53 R2 Q10 I0.8 M5;
M06 T01;	G98;
G90 G00 X0 Y0;	G183 X20 Y10 Z-53 R2 Q10 I0.8 M5;
G43 H01 Z1000;	M09;
M03 S1000;	M05;
M08;	G91 G28 Z0;
G99;	M30;

In this example, in order to increase universality, the first three holes are processed using the G99 mode, and the fourth hole is processed using the G98 mode. Therefore, after each processing of the first three holes, the tool should stay at the point R (2mm from the upper surface of the workpiece), and at the same time, after completing the fourth hole, it is necessary to retract the tool to the initial Z coordinate position (100mm from the upper surface of the workpiece). Since F is a modal code, it is not necessary to set the feed speed for the last three calls, because if a macro is called, If there is no special feed speed setting, the macro will automatically use the previous feed speed.

Macro calls (G65 or other methods) should be made in different program segments, as shown in the above example, G98 and G99 cannot appear in the same program segment as G183. If a G code is specified on the left side of a macro call instruction in the same program segment, it will be ignored by the control system, and the G code on the right side of a macro call will cause an alarm, because this G code will be considered a parameter of a macro, and the G code is not allowed as a parameter of a macro. In this case, the rules of M98 are different, It does not ignore the G code or alarm, but does not wait for the G code execution to complete. Therefore, if it is a motion code (such as G01), the subroutine will start executing when the tool is still moving. Therefore, in order to avoid any conflict, the subroutine and macro must be called in separate program segments.

In the FANUC system, there are some fixed loops and G codes with special functions, which are optional functions that require additional payment, but are expensive. Users can develop and design their own required macro instructions through the macro functions provided by the system to expand the processing functions of the machine tool and meet special processing requirements.

4. Conclusion

The drilling function of the FANUC system for gradually reducing the depth of deep hole drilling has been extended through custom G code (G183). When processing deep holes on a CNC milling machine equipped with the FANUC system, only the syntax requirements need to be set. Even if the programmer does not understand the basic principles of macro programs, G183 code can be used (the use method is similar to other fixed cycle functions of the system). After processing deep holes on the GLU28x40 CNC, it can effectively alleviate the problem of excessive tool temperature during the drilling process and improve processing accuracy. This method can also be used to expand other processing codes of CNC machine tools and expand the processing range of the machine tool.

Acknowledgements

This work was supported by 2022 Visiting Engineer of Colleges and Universities in Zhejiang Province(Project No. FG2022037).

References

[1] Li X B, Zheng J M,Yu B, Du Y Q, Zhou Y N. Analytical Model of Hole Diameter and Self-Guiding Machining Mechanism of BTA Deep Hole Drilling[J]. Materials, 2022, 15(15).
[2] Guba Nikolai, Schumski Lukas, Paulsen Tebbe, Karpuschewski Bernhard. Vibration-assisted deep hole drilling of the aluminum material AlMgSi0.5[J]. CIRP Journal of Manufacturing Science and Technology, 2022, 36.
[3] Wang M Y . Exploration into the Macro Programming of CNC Lathes Ellipses[J]. Applied Mechanics and Materials, 2015, 3759(727-728).
[4] BEIJING-FANUC.BEIJING-FANUC 0i-MA System Operation ManualB-63514C/01. 2003, 06.