A Review on Parametric Programming Techniques Utilized for Advanced CNC Machines

Swapnil R. Deogade[1] Dipti N. Kashyap[2]
[1]M.Tech. Scholar [2]Assistant Professor
[1,2]Department Mechanical Engineering
[1,2]YCCE, Nagpur, Maharashtra - India

Abstract— this paper describes how the Parametric Programming is applicable in CNC machining. Parametric programming has the potential to increase efficiency of CNC operations but still it is less frequently utilized features of CNC machines. Feature based parametric programming are beneficial to companies with manufacture of GT (group technology) where the parts with similar design undergoes in a machine cell. The advantages of blending the concept of group technology and parametric programming for CNC machining operations are illustrated with the help of a part family.

Key words: CNC machining, Parametric Programming, Group Technology, Macro.

I. Introduction

For Modern Computer Numerical Control (CNC) machines, parametric programming has the potential to bring higher efficiency to manufacturing industries. There are several applications of parametric programming to CNC operations. These include generating a single CNC program for parts with similar design, inventing macros for machining custom design features, and developing subprograms for a group of parts that are not similar in design but require similar machining operations. In all these applications, parametric programming can significantly reduce the part programming time and effort which in turn leads to shorter throughput and product development times. These applications particularly fit group technology manufacturing in which similar parts are grouped into part families and then processed by a number of machine tools within a cell or by a single multi-tasking machining center. The two common approaches in group technology are grouping based on design similarity, and grouping based on similar machining requirements. Parametric programming can be applied to part families formed by either of the two grouping approaches, as illustrated in this study. This paper evaluates the feasibility of using parametric programming in the implementation of feature-based machining. Custom Macro B was used in this study [1].

II. Parametric programming

Parametric programming is a G/M code programming in which axis position (x, y, z, a, etc.), feed and speed functions can be specified by a parametric expression. Parametric programming can be compared to any computer programming language like BASIC, C Language, and PASCAL. However, this programming language resides right in the CNC control and can be accessed at G code level, meaning you can combine manual programming techniques with parametric programming techniques. Computer-related features like variables, arithmetic, logic statements, and looping are available. Like computer programming languages, parametric programming comes in several versions. The most popular is Custom Macro B (used by Fanuc and Fanuc-compatible controls) [2]. Parametric programming is applied to CNC operations in generating a single CNC program for parts with similar design, inventing macros for machining custom design features, and developing subprograms for a group of parts that are not similar in design but require similar machining operations. Parametric programming can significantly reduce the part programming time and these applications particularly fit group technology manufacturing in which similar parts are grouped into part families and then processed by a number of machine tools within a cell or by a single multi-tasking machining center. For example, several cylindrical parts may have two common parameters such as diameter and overall length. A single parametric subprogram can be called up from the main program to machining such a group of similar parts. Upon loading the main program, the values of two parameters entered; then, these values are transferred to the parametric subprogram. This approach could minimize the number of program changeover and reduces the redundant code in part program and shorten the length of program [3].

III. Difference between CAM generated and macro program

In the conventional CNC part program, G-Codes are used for specific functions. For example, modal G codes such as G1, G2 and G3 are for linear interpolation, clockwise interpolation and counter-clockwise interpolation respectively while the canned cycle like G81 is for drilling a hole. The same codes may be used repeatedly in order to create or to cut a given machining feature. [2] This technique is applied in CAM to generate NC program as shown in Fig. 1 (b). Therefore, the size of NC programs generated by commercial CAM system is usually very large. When using parametric programming, the routines can be written as simple as shown in Fig. 1 (a) and makes it much shorter compared to CAM.

```
WHILE [#118LT [#111-#117-#23]] DO1
#118= [#118+#10]
IF [#118GE [#111-#117-#23]] THEN #118= [#111-
#117-#23]
Y [#102-[#112-#117-#23]]
X [#101-[#118-#23]]
Y [#102+ [#112-#117-#23]]
X [#101+ [#118-#23]]
Y [#102-[#112-#117-#23]]
X#101
END1
```

Fig. 1 (a) Macro program


```
N126 G1 Z69.025
N128 X-5.
N130 Y-120.
N132 X5.
N134 Y-110.
N136 Z79.025
N138 G0 Z80.
N140 X2.5 Y-112.5
N142 Z79.025
N144 G1 Z68.05
N146 X-2.5
N148 Y-117.5
N150 X2.5
N152 Y-112.5
N154 Z78.05
N156 G0 Z80
N158 X5. Y-110.
N160 Z79.025
N162 G1 Z68.05
N164 X-5.
N166 Y-120.
N168 X5.
N170 Y-110.
N172 Z78.05
```

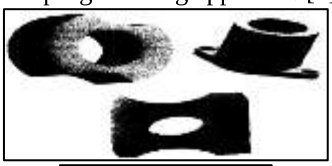
Fig. 1 (b) CAM generated NC program

Macro is very similar to subroutine. The different is that macro enables user to specify arguments and control the variables. With macros, repetitive cycle can be defined. It may be considered as the highest level of NC programming. This technique is more powerful and flexible. In the conventional CNC programming, there is limitation in terms of function of each G-Code. Designed in separated programs, macros can be called by the main program or other macros using macro number. Another advantage of macro is since it is similar to the BASIC language; user can make specific functions that are not provided by the CNC maker by using macro language.

IV. APPLICATION OF PARAMETRIC PROGRAMMING TO CNC MACHINES ARE SHOWN WITH FEW CASES

A. CASE 1

A group of non-similar parts are shown in Fig. 2. Excluding center hole these parts require different machining operations. Figure shows a CNC part program for machining a hole using a conventional G/M code programming approach. This program must be modified for each part. The difference between the two formats is that the parametric program can be called from any program to machine a hole of any size at any location, while the conventional program can only machine a specific hole size. The advantage for CNC machine users is that the operator will only enter the value of hole diameter, depth of hole, feedrate and spindle speed (parameters P 10 to P16). Tool position coordinates in the part program would not be changed as it is the case in conventional G/M programming approach. [1]



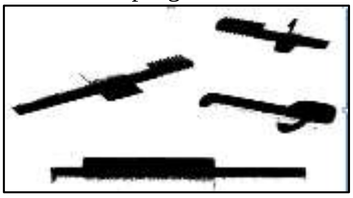
```
O100
G92 X0 Y0 Z1
S1000 M03
G00 X2.5 Y1
M06 T01
G43 H1 Z.1
G01 Z-.5 F10
Y2.125 F15
G02 J-2.125
G01 Y1.0
M30
```

```
O100
P10=.75 (DIAMETER OF END-MILL
        CUTTER)
P11=2.5 (X POSITION OF HOLE CENTER)
P12=1.  (Y POSITION OF HOLE CENTER)
P13=.5  (DEPTH OF HOLE)
P14=1.5       (DIAMETEROF HOLE)
P15=20        (FEEDRATE IN IPM)
P16=1000      (SPINDLESPEED RPM)
(G92 X0 Y0 Z1 (SET INITIAL TOOL
POSITION)
S [P16] M03
G00 X [P11] Y [P12] (RAPID TO HOLE)
(CENTER)
M06 T01
G43 H1 Z0.1 (RAPID TO Z=0.1 ")
G01 Z [-P13] F [P15/2]
Y [P12+[P14/2]-[P10/2]] F [P15]
G02 J [-[P14/2]-[P10/2]]
G01 Y [PI2]
G00 Z.1
M30
```

Fig 2 Parts with similar design feature

Fig. 2 (a) Conventional G/M program for machining a hole

Fig. 2 (b) Parametric program for machining a hole



B. CASE 2

A group of similar parts are shown in Fig. 3. This group represents a family of parts with similar design features such as thread and side steps. By changing the value of major parameters, such as diameter and overall length, a larger number of parts can be included within the same part family. The average number of NC instructions for machining all parts is 82 lines (Fig.3 a), while the number of NC instruction in corresponding parametric program is 38 lines (Fig.3 b). [3]

Fig.3. Parts with similar design

Fig. 3 (a) Partial part program for parts in Fig. 3

```
(Cylindrical Parts)
N002 G50 X14. Z5. S1000
N003 G00 T0505    (Left Shoulder)
N004 G97 S1000 M03
N005 G00 X1.4 Z-.1
N006 G96 S1000
N007 G99
N008 G71 P008 Q030 U0. W0. D2000 F.01 N008 G00
X1.044
N009 G99 G01 X.644 F.003
...............
...............
...............
N128 G00 X1.5 (Cutoff the Part)
N129 Z-2.5
N130 G01 X0 F.005
N131 G00 G40 X10 Z10 TI300
N132 M05
N133 M30
```

```
O2      (Cylindrical parts)
G20
G0 T0101
G97 S1500 M3
G96 S1200
```



```
(Right Shoulder)
(P10 EXTERNAL DIAMETER OF BAR)
(P11 LENGTH OF RIGHT SHOULDER)
(P13 SHOULDER DIAMETER)
     IF P11=0 GOTO N3
     G0 X[P10]+.1 Z.1
     G71 P1 Q2 I0 K0 R0 D.05
N1 G0 X [P131]
     G1 Z0 Z [-P11] X [P10]
N2 (MACHINING THE UNC THREAD, H=.6495P
N3 (P12  # OF PASSES)
     (P13 MAJOR DIAMETER)
     (PI4  # OF THREADS/INCH)
     (PI5 LENGTH OF THREAD)
     IF P12=0 GOTO N4
     G0 T0202
     G0 Z.5
     P6=0
     WHILE P6 LT [P12]
     P6=P6+1
     G0 X[P13-[P6*[[.6495/P 14]/P 12]]]
     G32 Z[-P15] F.005
     G0 X[P13]
     GOZ.1
     WEND
          (LEFT SHOULDER)
          (FIRST CHECK IF IT EXISTS)        N4
     (P16  DISTANCE FROM RIGHT FACE TO
          SHOULDER)
     (P17  OVERALL LENGTH OF PART)
     (P18  DIAMETER OF LEFT SHOULDER)
N3  IF P16=P17 GOTO N10
     G0 T0303
     G0 X[P10] Z[-P16]
     G71 P5 Q6 I0 K0 R0 D.05
N5 GO X[P18]
     G1 Z[-P16] Z[-P17] X[P18]
N6
N10 G0 T0404  (CUTOFF THE PART)
     G0 X [P10] +. 1 Z[-Pl7]
     G1 X0
     G1 X[P10]+1
     M5
     M30
```

Fig. 3 (b) parametric program for parts in Fig. 3

Few of the part families and their parametric programs are shown below [3]

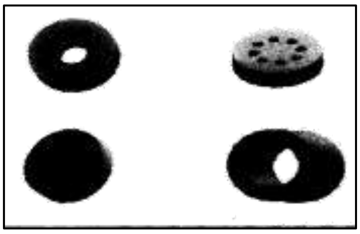


Fig.4. Part family 1

C. *Parametric Part Programs*

```
0002    (PART FAMILY 1)
(P10    # OF HOLES)
(P11    RADIUS OF HOLE CIRCLE)
(P12    EXTERNAL DIAMETER)
(P13    THICHNESS OF PART)
(P14       Z INCREMENT)
(P150 CENTER POCKET RADIUS)
(P155 CUT WIDTH)
G92 X [P11] Y0
S1000 M03 D1 (EXTERNAL CONTOUR)
M06 T01 G43 H1
GO Z0
GO X [-P12] Y0
G1 Z [-P13] F15
G02 X [P12] Y0 R [P12] F80
X [-P12]
GO Z.1
GO X0 Y0 (CENTER POCKET)
F20
G24 Z [-P13] Q [P14] F10
M06 T02 (BOLT HOLE PATTERN)
GO X [P11] YO
G81 Z [-P13] R.1 F20
GO AA0 AB0 R [P11]
AA0 AB [360/P10]
P6=0
WHILE P6 LT [[P10]-1]
P6=P6+l
GO AA [[360/ [P 10]]*P6]
     AB [[360/ [P10]]*[P6+1]] R [P11] F20
WEND
M05
M30
```

```
0003              (PART FAMILY 2)
(P10              # OF SLOTS)
(P11              THICKNESS OF PART)
(P12              LENGTH OF PART)
(P13              WIDTH OF PART)
(P14              # OF PASSES IN Z)
(P15               DEPTH OF SLOTS)
G54
T01 M6
S1000 M3
G0 X0 Y0 Z0
P6=0
WHILE P6 LT [P14]
P6=P6+l
G1 Z [-[P11/P14]*P6] F5
Y [P13] F50
P7=0
WHILE P7 LT [[P10]]
P7=P7+1
G1 X [[P12/ [[P10] +l]]*P7]
G91
Y [-P15]
Y [P15]
G90
WEND
X [P12]
Y0
X0
GO Z.1
WEND
M30
```

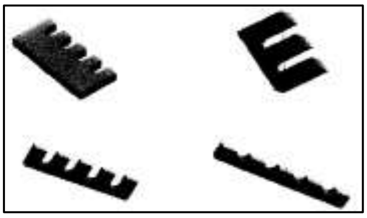


Fig. 5: Part Family 2

V. CONCLUSION

Parametric programming serves as the tool for improving the efficiency of programming CNC machines. This paper showed the characteristics of parametric programming. Two cases of the application of parametric programming are used to show the difference between a conventional CNC program and its equivalent parametric program. It is recommended that GT users adopt parametric programming for large part families or whenever there is a growing trend in the size of the part family to minimize program changeovers and the number of similar NC files. Methodology of parametric programming for error compensation can carried out. The resulting errors are introduced as compensation values to the conventional tool movements along the programmed tool path. This can result in a complex tool path. Parametric programming is applied to handle this complexity for error compensation. Parametric programming is the best-kept secret and less utilized features as well of modern CNC machines. Surveys on successful applications of parametric part programming in group technology facilities would be an appropriate extension to this study.

REFERENCES

[1] Manocher Djassemi, "A Parametric Programming Technique For Efficient CNC Machining Operations" Computers ind. Engg Vol. 35, Nos 1-2, pp. 33-36, 199.

[2] M.A. Razak, M.R. Ibrahim, S. Sulaiman, A. Jusoh, A. Zakaria, "Parametric Programming in Feature-Based Machining" International Journal of Engineering & Technology Sciences (IJETS) 1(4): 218-225, 2013 ISSN 2289-4152.

[3] Manocher Djassemi, "An Efficient CNC Programming Approach Based on Group Technology" Journal of Manufacturing Systems Vol. 19/No. 3, 2000.

[4] Z.-Q. Liu, "Methodology of Parametric Programming for Error Compensation on CNC Centres" Int J Adv Manuf Technol (2001) 17:570–574 © 2001 Springer-Verlag London Limited

