NATIONAL CONFERENCE ON ICT EMPOWERED TEACHING, LEARNING AND EVALUATION (NCICT-2016)

International Journal of Advanced Scientific Technologies in Engineering and Management Sciences (IJASTEMS-ISSN: 2454-356X) Volume.2,Special Issue.1Dec.2016

CNC R-PARAMETER PROGRAMMING ON MILLING MACHINE

K. Neelakanta, M.usharani, V.Arundhati, B.Rama Sanjeeva Sresta
Department of Mechanical Engineering,
Srinivasa Ramanujan Institute of Technology, Ananthapuramu, (A P), India
E-mail: kallamadineelakanta@gmail.com , usha.malekar@gmail.com , mail4arundhati@gmail.com , Sresta16388@gmail.com

Abstract— This paper develops the new CNC program. CAM Software is used for development of computer numerical control programs based on the applications of CNC machines. Generally the Templates are used to measure the grooves on turning rolls. Using this simple CNC Programming we can generate 'n' number of Templates but all will have the same dimensions. If the groove size changes then the template dimensions should also be changed according to the groove dimensions. Then a new CNC program has to be developed for this, which requires a lot of programmer's effort and time. In this paper a R- Parametric programming for milling a square (or) diamond profile template on CNC milling machine will be undertaken. The advancement in CNC programming is using the R-Parameters which enable to produce many templates with similar shape but different dimensions I.e., repetitive type of job in nature. This reduces the programmer's effort and time spent on programming and works even when the cam system collapse.

Index Terms—computer numerical control, computer aided manufacturing, program, parameters

I.INTRODUCTION

Templates' are conventionally cut by using fitting Techniques. CNC Programs Can be developed using different CAM softwares like Master CAM, Gibbs CAM, Edge CAM, etc. by Using Gibbs CAM for developing NC programs. There are so many Profiles similar in shape and differ in dimensions. For such type of profiles when they are huge in quantity, Programming in NC coordinate System using CAM softwares or by manually is tedious, laborious, time consuming , lengthy process. To overcome this type of situations parametric programs have been developed to write the programs for similar shaped objects. Parameters are the advanced programming techniques developed and incorporated in the machine controller itself by the CNC machine tool manufacturer.

CNC machines are supported by SINUMERIC Controller which accepts the R – parameters as advanced programming techniques, incorporated in the machine controller. Writing programs for similar shaped profiles using advanced programming techniques reduces the Programmers effort and minimizes the time consumption. It can be very much useful even when CAM system collapses.

2. Numerical Control Defined:

2.1 NC PART PROGRAMMING:

The command specifies a position in the Cartesian coordinate system (x,y,z) or motion (work piece travel or cutting tool travel), machining parameters and on/off function. The part program is written manually or by using computer- assisted language such as APT (Automated Programming Tool). Sequence number (n-words),Preparatory word (g-words),Coordinates and interpolation parameters x-,y-,z-,a-b-,c-,I-,j-,k, words, Feed rate (f-words). Cutting speed (s-word), miscellaneous function (m-word), End of block (EOB)

Flow of CNC processing

1. Develop the part drawing
2. Decide which machine will produce the part
3. Choose the tooling required
4. Decide on the machining sequence
5. Do math conclusions for the program coordinates
6. Calculate the speeds and feed required for the tooling and part material
7. Write the NC program
8. Prepare setup sheets and tool lists
9. Send program to machine
10. Verify the program
11. Run the program if no changes are required

Three major phases of CNC program

The following program shows the three major phases of a CNC program %9999

```
N5   G90      G70
N10  MO6      T2
N15  MO3      S1200
N20  G00      X1Y1
N25  Z0.125
N30  G01      Z-0.125  F5
N35  G01      X2 Y2
N40  G00      Z1
N45  X0Y0
N50  MO5
N55  M30
```

Program setup

% Program start
9999 program address number
N5 G90 G70 use absolute units, and inch programming
N10 M06 T2 Stop for tool change, use tool#2
N15 MO3 S1200 Turn the spindle on CW to 1200rpm.

The program setup phase is virtually identical in every program. It always with the program starts flag (%sign). Line two always has a program number (up to four digits, 0000 to 9999).Line three is the first that is actually numbered. It begins with N5 (N for sequence number, 5 for block number 5). We can use and number incrementing upward. We use increments of 5 in example. Incrementing in this way enables you to insert up to 4 new lines between lines when we are editing program. Block 5 tells the controller that all distances (X and Z coordinates) are absolute, that is, measured form the origin point. It also instructs the controller that all coordinates are measured in inch units.

The program set up contains all the instructions that prepare the machine for operation.

The setup phase may also include such commands as coolant on, cutter compensation
cancel, or stop for tool change.

Material Removal

N20 G00 X1 Y1 Rapid to (X1, Y1) from origin point
N25 Z0.125 Rapid down to Z0.125
N30 G01 Z-0. 125 F5 Feed down to z-0.125 at 5 ipm
N35 G01 X2 Y2 Feed diagonally to (X2, Y2)
N40 G00 X1 Rapid up to Z1
N45 X0 Y0 Rapid to X0, Y0

The material removal phase dealer exclusively with the actual cutting feed. It contains all the commands that designate linear or circular feed moves, rapid moves, canned cycles such as grooving of profiling, or any other function required for that particular part.

System shutdown

N50 M05 Turn the spindle off
N55 M30 End of program

The system shutdown phase contains all those G- and M- codes that turn off all the options that were turned on in the setup phase. Functions such as coolant and spindle rotation must be shutoff prior to removal of the part from the machine. The shutdown phase also is virtually identical in every program.

2.3 R- PARAMETER PROGRAMMING:

R- Parameter programming is advantages for different profiles of same shape. This enables us to run the program for N-number of profiles of the same shape but with changes in values.

A parameter comprises the address R and a number with up to 3 digits. In the basic Configuration of SINUMERIK 810-M, 1000 (ie R00 to R999) parameters are available in the controller. A parameter can be assigned instead of a value to all addresses with the exception of N is N5 Z = R5 $^{L}_{F}$

Parameter Definition

The parameter definition is used to assign certain numeric values with signs to the various parameters. The parameters can be defined either in part programs or in subroutines.

R1 = 10 $^{L}_{F}$

The parameter definition, the subroutine call and the switching functions may be written in a single block. The value defined for a parameter is assigned direct to the address.

Example: %5772
N1 ……
N20 R1= 10 R29=-20.5 R5=50 LF
N25 L51 P2 LF
N30 M02 LF
L51
N1 Z= - R5 B= -R1 LF
N2 X= -R29 LF
N50 M17 LF

In the above N20 Parameter definition, N25 Call of subroutine 51, 2 pass

Parameter calculations

Parameter linking:

All four basic arithmetic operations are possible with parameters. The linking sequence is, however, crucial to the result of the calculation.

TABLE.1

Arithmetic operation	**Programmed executed Arithmetic operations**
Definition	R1 = 100
Assignment	R1 = R2
Negation	R1 = - R2
Addition	R1 = R2+R3
Subtraction	R1 = R2 – R3
Multiplication	R1 = R2 * R3
Division	R1 = R2/R3

Value assignment amongst parameters:

If the value of one parameter is to be assigned to another parameter, the following is valid R1 = R3 $^{L}_{F}$

Calculation using numbers and parameters

Addition and subtraction of number and parameters in conjunction with addresses

X=10+R100 The "+ "sign must always be entered.
Example: N10 R1=9.7 R2=-2.1 LF
N15 X=20.3+R1 LF

N20 Y=32.9-R2 LF

N25 Z=19.7-R1 LF

Result: X=30: Y=35: Z=10.

Calculation using numbers and parameters:

It is possible to multiply, divide, add or subtract absolute numbers and R- parameters.

R10 = 15 + R 11

Example:

The parameter R2 shall be divided by 2

R3 = 2 Definition of auxiliary parameter

R1 = R2/R3. The result of the calculation is contained in R1 ; the values of R2 and R3 (auxiliary parameters) are retained.

Parameter string:

R1=R2 +R3 - R4 ∗ R 5/R6……R10

All 4 basic arithmetic operations are permissible in any sequence. It is possible to link up to 10 Parameters together in a parameter string.

The calculations are performed as follows:

Step1 : R1 = R2

Step2 :R1 = R1 + R3

Step3 :R1 = R1 – R4

Step4 : R1 = R1 ∗ R5

Step5 : R1 = R1 / R6

Step1 : R1 = R2

Step2 : R1 = R2 + R3

↓

Step3 : R1 – R4

↓

Step4 : R1 ∗ R5

↓

Step5 : R1 / R6

↓

R1

Instead of a link R parameter(not a result parameter),constants and pointers (pointers to R parameters) are allowed with address P in the parameter string.

Example :

R1 = R2 + 10.5 – P3 ∗ R5 / R6……

The result parameter must be an R parameter P3 :

P address of pointer

3 pointer to R parameter R3 , i.e. the contents of R3 are the address of an R

Parameter whose value is included in the parameter string.

Value range: Minimum value: $1 * 10^{-8}$

Maximum value : 99999999

Display: Floating point (± . 8) to (± 8.0)

Functions supported by SINUMERIC 810 M Milling machine:

These are the special functions represented by symbol @. The following are defined in the program key:

@ 630	:	Calculates sine value.
@ 631	:	Calculates cosine value.
@ 632	:	Calculates tangent value.
@ 634	:	Calculates arc sine value.
@ 613	:	Calculates the square root value.

Format: N_ @ 613 R23 R22 Execution is square root of R22 and its value is stored in R23 .I.e. square root of R22 = R23.

3. Case study – I

Simple CNC Program developed for Square profile template.

%MPF 3065

N005(NC PROGRAM FOR SQUARE PROFILE TEMPLATE DRG 3065)

N010(USE 6MM END MILL)

N015 G90 G17 M07 M03

N020 G0 G90 G54 X-.3451 Y-15.001

N025 T1 D1

N030 G42 G1 Y-.0005 F1

N035 X145.5429

N040 Y20.0005

N045 X125.5424

N050 G2 X121.3029 Y22.3502 I0 J4.9995

N055 G1 X100.7091 Y55.3002

N060 G3 X92.2287 Y60.0005 I-8.4804 J-5.3002

N065 G1 X53.3137

N070 G3 X44.8333 Y55.3002 I0 J-10.0005

N075 G1 X24.2395 Y22.3502

N080 G2 X20. Y20.0005 I-4.2395 J2.6498

N085 G1 X-.0005

N090 Y-10.0005

N095 M5 M8

N100 M2