CNC R-PARAMETER PROGRAMMING ON MILLING MACHINE

K. Neelakanta, M.usherani, 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-, J-, j-, k- words, Feed rate (f-words), Cutting speed (s-words), 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

<table>
<thead>
<tr>
<th>N5</th>
<th>G90</th>
<th>G70</th>
</tr>
</thead>
<tbody>
<tr>
<td>N10</td>
<td>MO6</td>
<td>T2</td>
</tr>
<tr>
<td>N15</td>
<td>MO3</td>
<td>S1200</td>
</tr>
<tr>
<td>N20</td>
<td>G00</td>
<td>X1Y1</td>
</tr>
<tr>
<td>N25</td>
<td>Z0.125</td>
<td></td>
</tr>
<tr>
<td>N30</td>
<td>G01</td>
<td>Z-0.125</td>
</tr>
<tr>
<td>N35</td>
<td>G01</td>
<td>X2 Y2</td>
</tr>
<tr>
<td>N40</td>
<td>G00</td>
<td>Z1</td>
</tr>
<tr>
<td>N45</td>
<td>X0Y0</td>
<td></td>
</tr>
<tr>
<td>N50</td>
<td>MO5</td>
<td></td>
</tr>
<tr>
<td>N55</td>
<td>M30</td>
<td></td>
</tr>
</tbody>
</table>
Program setup

Parameter programming is advantageous for different profiles of the 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 \( \frac{1}{2} \).

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.

\[ R1 = 10^2 \frac{1}{2} \]

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

N20 R1= 10 R29= LF

N15 X=20.3+R1 LF

N15 Z= R5 LF

N40 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 deals exclusively with the actual cutting feed. It contains all the commands that designate linear or circular feed moves, rapid moves, canned cycles such as 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
Calculation using numbers and parameters:
It is possible to multiply, divide, add or subtract absolute numbers and R- parameters.

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 * R5 / 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 = R1 - R4
Step4 : R1 = R1 * R5
Step5 : R1 = R1 / R6

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