Vol. XIV & Issue No. 06 June - 2021
INDUSTRIAL ENGINEERING JOURNAL

CNC MACHINE - A BOON FOR ENGINEERING INDUSTRIES FOR PERFORMING INNOVATIVE MACHINING OF CYLINDRICAL AND PRISMATIC PARTS

P. K. Basak

Dr. Manish Gangil

Abstract:

Computer Numerical Control (CNC) is the technology for automated control of machine tools. It uses programmable logic, that in, data is the form of letters, numbers, symbols, words or a combination of them for controlling the functions of the machine tools. The very first CNC Machine was developed by Richard Kegg in collaboration with the Massachusetts Institute of Technology, USA in the year 1952 by retrofitting electronic gadgetry to a conventional milling machine. With the development in the field of computers, computer aided design (CAD), computer aided manufacturing (CAM), CNC machines became more & more sophisticated. At present, CNC technology has been extended to almost all types of machine tools, be it metal cutting, metal forming & other types of machines. This is due to the fact that CNC Machines offer a host of advantages like higher productivity, better accuracy, lesser operator skills requirement etc. On a conventional Turning machine, the machining of a profile like ellipsoid on cylindrical jobs requires special form tools and skilled operators. On a two-axis CNC Turning machine, such jobs can be conveniently machined by using an appropriate part program by dividing such profiles into small segments and making part programs for each small segment which add up to the profile. The part programs for the machining of other profiles on cylindrical jobs can be very easily made by using the Canned cycles provided by the CNC controller manufacturer. On a conventional boring machine, the machining of bores requires a size-specific boring tool. Also, the machining of internal and external threads on bores and outside diameters are quite cumbersome and require skilled operators. On a CNC machining centre, bores can very easily be machined using a deep shoulder end mill and the Circular Interpolation feature of the CNC controller. The machining of both internal and external threads of any size can be very conveniently performed by using a thread milling cutter and the Helical Interpolation feature of the CNC controller. In this paper, innovative machining of both cylindrical and prismatic parts shall be presented.

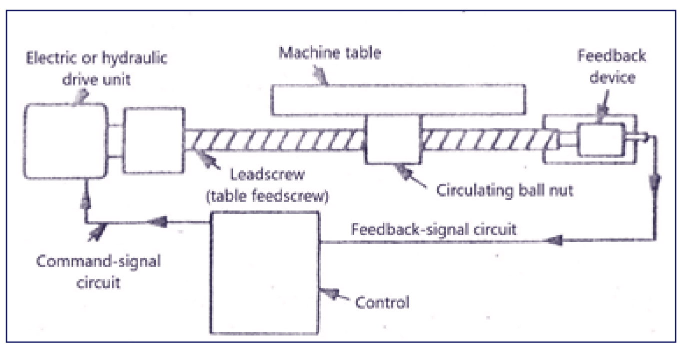
Keywords: CNC Machine Cylindrical parts, prismatic parts, linear interpolation, circular interpolation, helical interpolation, parametric programming.

1. INTRODUCTION

CNC (Computer Numerical Control) is a technology which permits automatic operation of a machine tool or process through a series of coded instructions consisting of numbers, letters and other symbols. CNC is a means to achieve automatic precise control of coordinated multi axis motions of machine tools.

The principle of operation of CNC Machines can be explained with the help of Fig. 1 below. A set of instructions called CNC Part Program which defines relative movement of the machine slides with respect to cutting tool, selection of machine spindle RPM etc. is fed to the CNC Controller. The CNC Controller which consists of electronic hardware & software, reads and interprets the CNC part program and sends signals to servo motors which are connected to machine slides by means of recirculating ball lead screws and nut assembly. The digital signals cause the leadscrew to turn resulting into the linear motion of the machine slides. The pitch of the ball lead screw determines the distance travelled by the table due to each revolution of the servo motor. The use of recirculating ball screw and nut reduces friction, backlash and wear resulting into low torque requirement of servo motor for slide movement. This also leads to better accuracy of movements. The dynamic response of the system is also improved. A feedback device mounted either at the machine slide or at the servomotor measures the displacement or position of the machine slide. The measured and the programmed or target positions are compared, and the servosystem ensures that the difference between these two are nullified and the correct position is achieved. Since positioning is done by electronic means, it is possible to achieve higher accuracy and repeatability.

Figure 1: CNC machine operation



1.1 CNC Programming Fundamentals

CNC Machines are controlled by means of numbers, letters and other symbols. For the machining of a job or a part on a CNC Machine, a set of alphanumeric commands in appropriate format is required to be inputted, which is interpreted and converted into signals that provide control of the machine by it. Such a set of commands is the documentation of the sequence of operations to be performed on the CNC machine and is called a part program. Thus, a part program is a detailed set of alphanumeric commands or instructions incorporating axis movement data, technological data (speed, feed, etc.), auxiliary and tool functions to carry out manufacturing operations. These alphanumeric instructions are converted into control pulses by the CNC controller which result into machine movements and other functions.

A part program is composed is composed of several lines of command. Each line of command is called a Block. One block is separated from another block with end-of-block command, LF. A block is a collection of words. A word is made of address followed by numbers e.g., X-100 is a word. A word may be composed of various codes or letters which are explained below:

1. Sequence Number word is composed of letter N followed by a number, e. g. N 10.
2. Preparatory word is composed of preparatory code (G) followed by a number. G codes prepare the CNC controller unit to perform a certain function. For example, G01 prepares the CNC controller unit for linear interpolation (movement) along a straight line joining two points. G codes are of the following two types:
   a. Modal G code: This type of G code is effective till another G code of the same group is written to supersede it.
   b. Non-modal or One-shot G code: This type of G code is effective in a particular block where it is appearing.
3. Coordinate word: There are six possible motions or coordinates, three linear motions along three linear axes X, Y & Z & three rotational motions in three rotational axes A, B & C. The rotational axes are those rotational motions about X, Y, & Z respectively. In addition to these there may be U, V, & W axes or coordinates which are parallel to X, Y & Z axes respectively. These coordinate words may be described by either X, Y, Z, U, V, W, A, B & C or a combination of these letters.
4. Feedrate (F) word: The feedrate is the advancement or movement of the cutting tool per revolution of spindle or tool or it is the movement of the tool per minute along the direction of the cut. Feedrate is indicated in mm/revolution or mm/minute.

   If Fr is the feedrate per revolution

   Fm is the Feedrate per minute

   N is the revolution per minute of the spindle

   then, Fm = Fr x N
5. Cutting speed (S) word: This word specifies the cutting speed of the process. Sometimes it is specified in revolution per minute & sometimes it is specified in surface speed per minute, which is calculated by the formula,

$$V = \frac{\pi DN}{1000}\ m/min$$

where, V = Surface speed in metres/min.

D = Job diameter

N = Spindle RPM

6. Tool selection (T/D) word: For CNC lathes having Automatic Tool Turret, tool selection is made by T-word. For CNC Machining centres, tool selection is made by D-word.
7. Miscellaneous Function (M) word: The miscellaneous function word is used to specify certain miscellaneous or auxiliary functions of the CNC machines. It is usually the last word in a part program block. At the end of each block a symbol LF is used to separate it from other blocks.

A typical part program block may look like as follows:

N100 G00 x 100 Y 100 Z 100 F 500 S 300 D1 M3 LF

where, N100 is the block (line) number

G00 is the Preparatory function

X100, Y100, Z100 are the axes addresses

F500 is the feedrate

S300 is the RPM

D1 is the Tool Data Number

M3 is the Miscellaneous code

LF is used to specify the end of the block

Some of the G-codes are explained below:

Code	Function
G00	Move in a straight line at rapid speed
G01	Move in a straight line at a feedrate
G02	Clockwise Circular arc at feedrate
G03	Counter-clockwise circular arc at feedrate
G04	Dwell: stop for a specific time
G17	Select X-Y plane
G18	Select X-Z plane
G19	Select X-Z plane
G40	Tool Cutter Compensation off (radius compensation)
G41	Tool Cutter compensation left (radius compensation)
G42	Tool cutter compensation right (radius compensation)
G43	Apply tool length compensation (plus)
G49	Tool length compensation cancel
G53	Cancel work offsets
G54	First work offset (zero offset)
G55	Second work offset (zero offset)
G64	Contouring mode, block transition without speed reduction
G84	Tapping cycle
G90	Absolute Programming

Some of the M-codes are explained below:

Code	Function
M00	Program stop (non-optional)
M01	Program stop (optional, selectable)
M02	End of Program
M03	Spindle on (clockwise)
M04	Spindle on (Counter-clockwise Rotation)
M05	Spindle stop
M06	Tool change
M11	Unbraking and releasing of all coordinates
M17	End of Subroutine
M30	End of Program Rewind & Reset Modes
M44	Gear Range 4 (25, 800 RPM)
M98	Sub program call
M99	End of Subprogram

1.2 Absolute and Incremental Programming

Absolute coordinates are based on the origin (0,0). In absolute programming all coordinate values are calculated from the fixed origin (0,0) of the coordinate system. This type of coordinate system is selected by using the code G90. In incremental programming system, all coordinates describing the shape of the workpiece are calculated from the end point of the previous block. That is, the distances are calculated from the preceding point. This type of coordinate system is selected by G91.

1.3 Circular Interpolation

Circular interpolation means movement of tool along a circular arc to the commanded end position. It may be a complete circle or a portion of it. The motion may be clockwise or counter clockwise (G02/G03). Circular interpolation requires the following information

1. An endpoint
2. A Feedrate
3. A direction of movement
4. Distance from the starting point of the circle to the centre of the circle along the two axes of the plane in which motion is taking place.

Two axes are necessary to complete circular motion. The correct plane (G17-G19) must be used. The distances along the two axes from the start point to the centre of the circle with appropriate sign must be used (I, J, K).

1.4 Helical Interpolation

As a machining process, helical interpolation involves simultaneous circular movement in the X and Y axes combined with an axial feed (Z axis) at a defined pitch. It requires simultaneous motion in all three axes, X, Y and Z. The basic programming procedure for helical interpolation is selecting a circular interpolation plane using a plane selectiog command (G17, G18, G19), selection a command (G02/G03) for clockwise or counter clockwise motion, designating the two axis addresses for circular interpolation and the address of one axis for linear interpolation.

1.5 Parametric Programming

While writing part programs for families of parts, programming with parameters (variables) can save an immense amount of effort and time. In Sinumerik 850M CNC Controller, Parameters (Variables) are designated by the letter R followed by a number of maximum 3 digits. The R parameters can do addition, subtraction, multiplication, division, trigonometric functions. It can also perform other logical operations, comparisons and transfers to zero-offset which greatly simplifies the part program. R-parameters are used for programming a general part program for a family of parts of various sizes. In such programs, the variables are designated by R-parameters.

A glimpse of the various functions performed by the

R-parameters is illustrated below:

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
Sine Value	@ 630
Cosine Value	@ 631

In Sinumerik 850M CNC Control system of Siemens, part-programs are designed by % MPF followed by 4-digit numbers & these are concluded by the code M30. Subprograms, which are basically similar to main programs, are used for repeated machining. These programs are designed by %SPF followed by 3-digit numbers & concluded by the code M17. A subprogram can be called in the main program by the address L followed by its identification number. A subprogram is a small program which can be called in the main part program more than once. The content of the subprogram is not required to be reproduced in the main program but only its name is sufficient to call them for execution in the main program.

2. LITERATURE REVIEW

2.1 Cao Schuling and Guan Jinbiao (2018) [1] presented PowerMLL which is a CAD/CAM package. This package is handy for making CNC Part Programs for complex parts. The input for this CAD/CAM package is 3-dimensional CAD model. This package is similar to other CAD/CAM packages like UG-NX.

2.2 Mohite, Tiwari et al (2017) [2] dealt upon the dressing parameters like dressing depth of cut (um), dressing cross feed rate (mm/min), drag angle of dresser, and the number of passes for their effect on the surface finish of the machined surface.

2.3 Rudrapati et al (2016) [3] presented the effect of machining parameters like spindle speed, feed and depth of cut on the surface finish achieved on an Aluminium alloy workpiece. They also developed a mathematical formula of surface finish.

2.4 Osman et al (2014) [4] dealt upon optimization of material removal rate in case of roughing operations for Rapid Manufacturing process.

2.5 Nakao & Urata [5] presented their study on a CNC free form surface machining system with a rotating cylindrical tool like End Mill. A numerical model for this purpose has been developed by them.

2.6 Sprott et al (2008) [6] presented their paper on cylindrical milling or flank milling of ruled surfaces. They presented a mathematical equation of ruled surfaces.

3. PROGRAMMING EXAMPLES

Engineering industries manufacturing various products are required to carry out machining of various components or parts. The various parts may be cylindrical or prismatic in nature. The cylindrical parts may be shafts of various diameters and lengths requiring machining of various profiles. The prismatic parts may require machining of bores and threads of various sizes.

In this paper, parametric programming technique has been used to carry out the machining of various cylindrical and prismatic parts on various CNC machines. The advantage of parametric programming is that it can be used for the machining of various sizes of jobs, and only the values of the parameters require change. This can be taken care during machining at the shop floor level, and no new part program is required for the machining of each and every job.

Parametric programming examples for the machining of the profiles on cylindrical parts using the machining cycle L95 of Siemens has been presented.

Parametric programming examples for the machining of bores using the circular interpolation feature of the CNC controller and an end milling cutter has been presented.

Parametric programming examples for the machining of threads of various sizes has been presented using the helical interpolation feature of the CNC controller and a thread milling cutter.

Figure 2: CNC Slant Bed Lathe

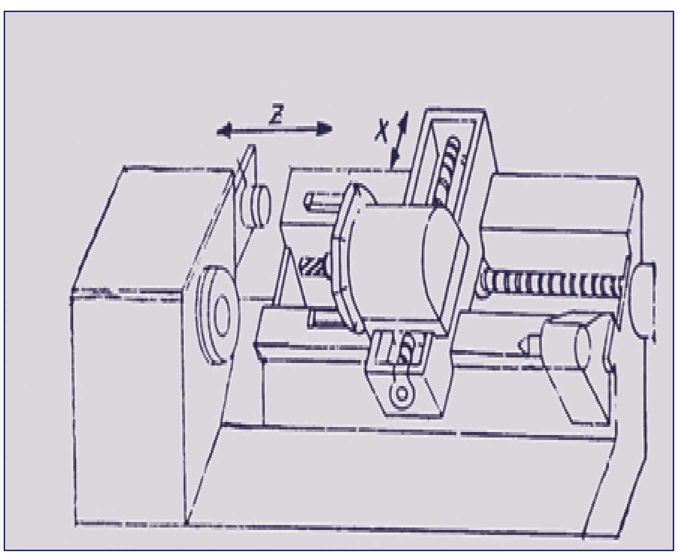


Figure 3: CNC Horizontal Machining Centre

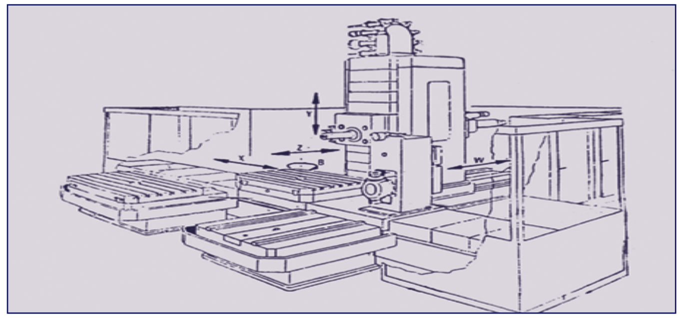
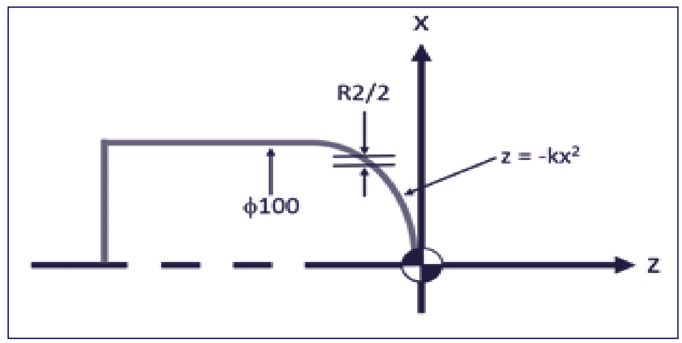


Fig. 2 shows a two-axis CNC Slant Bed Lathe and Fig.3 shows a CNC Horizontal Machining Centre. Cylindrical components are machined on the CNC Slant Bed Lathe, and the Prismatic components are machined on the CNC Horizontal Machining Centre.

The example below is for the machining of a paraboloid surface on a CNC Lathe having two axes of simultaneous movement as shown in Fig. 4.

Figure 4: Paraboloid surface profile



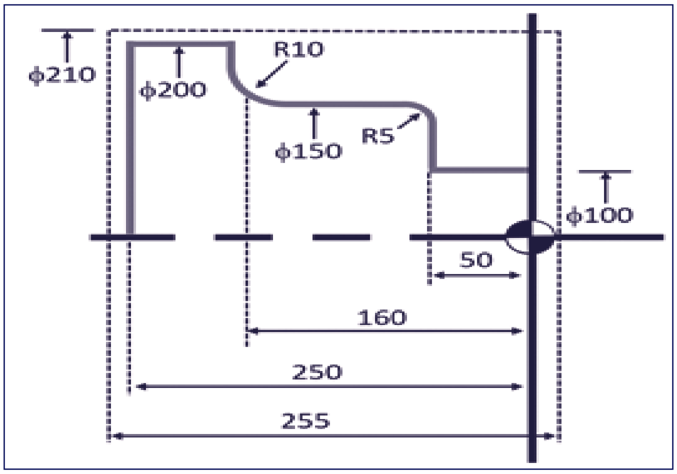
% 1111 LF

N10	(PROGRAM FOR MACHINING OF PARABOLOID SURFACE) F
N20	G54 G90 G95 LF
N30	G00 X 150 Z 10 LF
N40	T0101 (ROUGH TURNING TOOL PCLNR 3232) LF
N50	G96 S150 M03 LF
N60	X 10 210 LF
N70	(R0=K, R1 = STARTING X, R2 =FINAL X, R3 = INCREMENTAL X, R4 = CALCULATED X, R5 = CALCULATED Z) LF
N80	(Z = -K.X.X) LF
N90	R0 = 0.01 R1 = 0 R2 = 100 R3 = 1 LF
N100	L100 LF
N110	G00 X 110 Z 100 LF
N120	T0 M5 LF
N130	M30 LF
L100	LF
N10	(SUBPROGRAM FOR THE MACHINING OF PARABOLOID SURFACE) LF

N20 R4 = R1 LF
N30 R5 = R5.R0 LF
N40 R5 = R5.R0 LF
N50 G1 x = R4 Z = R5 F0.2 LF
N60 R4 = R4 + R3 LF
N70 @ 143 R4 R4 R2 K - 30 LF
N75 (IF R4 IS GREATER THAN R2, GO TO THE NEXT LINE, ELSE GO TO LINE NO 30) LF
N80 M17 LF

The next example is for the machining of a workpiece depicted in Fig. 5

Figure 5: Complex profile with circular interpolation



The CNC Part Program for the machining of this component out of the raw material of size 210mm dia & 255 mm length is depicted below:

% 1112 LF
N10 (PROGRAM FOR CNC LATHE SLANT BED TYPE) LF
N20 G54 G90 G95 LF
N30 T0101 (ROUGH TURNING TOOL PCLNL 3232) LF
N40 G96 S140 M04 LF
N50 G0 x 220 Z10 LF
N60 R20 50 R21 100 R220 R24 1.5 R250.5 R265 R270 R 280.3 R 2911 L95 F0.3 LF
N70 R24 1.5 R 25 0.5 R 270 R28.0.2 R 29 21 L95 LF
N80 M00 LF
N90 M05 LF
N100 M30 LF
L50 LF
N10 (SUBPROGRAM FOR THE JOB SHOWN IN FIG. 5) LF
N20 G01 X 100 Z 6 LF
N30 G01 Z - 50 LF
N40 G01 X 140 LF
N50 G03 X 150 Z - 55 IO K - 5 F0.15 LF
N60 G01 Z - 150 F0.2 LF
N70 G02 X 170 Z - 160 I10 K0 F0.15 LF
N80 G01 X 200 F0.2 LF
N90 G01 Z - 250 F0.2 LF
N100 G00 X 220 Z 20 LF
N110 M05 LF
N120 M30 LF

The description of the various parameters is given below:

R20	Subprogram no. which describes the contour of the job
R21	Starting point of the contour in X
R22	Starting point of the contour in Z
R24	Finishing cut in X
R25	Finishing cut in Z
R26	Depth of cut in X
R27	0 (no tool nose radius compensation)
R28	Feedrate
R29	11 (Cut parallel to Z-axis)
R29	21 (Cut Parallel to Contour along Z-axis)

Machining of Bores

On a conventional Boring machine, bores are machined using the Boring tool of various sizes.

On a CNC Machining Centre or a CNC Horizontal Boring machine, bores can be machined by using any of the following methods:

1. By using Boring tools and using a linear motion along the depth of the bore.
2. By using deep shoulder end mills & the circular interpolation feature of the CNC machine.

A part program for carrying out boring operation using a boring tool may be represented as below:

% MPF 1100

N10 G54 G90 G17 LF
N20 T1=1 M06 LF
N30 M42S300 LF
N40 M3 LF
N50 G0 Z 50 D2 LF
N60 X0 Y0 LF
N70 G1 Z – 100 F 200 LF
N80 M5 LF
N90 G1 & 10 F1000 LF
N100 M30 LF

Part program for machining of bores on a CNC Machining Centre, using a deep shoulder end mill & circular interpolation

The following example part program is written for the Sinumerik 850M CNC Controller.

Let R02 = Bore radius
R03 = Cutter radius
R05 = Offset in Y direction

Part Program for machining a bore of 220 mm dia. using a deep shoulder end mill of 60mm dia. is given below:

```
        % MPF 1300
N2      (MACHINING A BORE OF 220MM
        DIA) LF
N5      (USE MILLING CUTTER DIA 60MM) LF
N10     T1=2 M06 (CHANGE TOOL) LF
N20     G54G90G94G17 LF
N30     M44S800 LF
N40     M03 LF
N50     G00 Z 30 D02 LF
N60     G00 X0 Y0 LF
N70     R02=110 R03=30 R05=50 LF
N80     L500 (CALL SUBPROGRAM NO 500) LF
N90     G00 G90 X0 Y0 LF
N100    Z 30 LF
N110    D0 M05
N120    M30 LF
        % SPF 500
N2      (SUBPROGRAM NO. 500) LF
N10     R06 = R02 + R05 R07 = R06/2 LF
N20     G01 G17 G41 D02 X0 Y=-R5 F500 LF
N30     G91G03 G64 X0 Y=R06 I0 J=R07 F300 LF
N40     G03 X0 Y0 I0 J=-R02 F 250 LF
N50     G03 X0 Y=-R6 I0 J=-R07 F600 LF
N60     G00 Z 100 LF
N70     G90 G40 X0 Y0 LF
N80     M17 LF
```

For the machining of threads on conventional drilling machine, tapping is done using a tapping attachment and the exact size taps.

On a CNC Machining Centre, machining of the threads can be performed by either of the following two methods:

1. Tapping using a tapping Cycle & tap.
2. Thread whirling using a thread milling cutter and the helical interpolation feature of the CNC controller. In thread whirling, threads can be generated by a thread milling cutter of a a smaller sized milling cutter than the threaded hole size. The only requirement for the thread milling cutter is that it should have exact form of the thread & it should be suitable for the pitch of the thread. Usually the thread cutting cycle is represented by L84.

Part Program for the machining of a thread on a CNC machining Centre is as follows:

```
 % MPF84
N10     G90 S48 M03 F460 LF
N20     G00 D01 Z 50 LF
N30     X100 Y150 LF
N40     R02=360 R03=250 R06=4 LF
N45     R07=3 R09=4 (PITCH) R11=3 LF
N50     L84 LF
N60     D0 M05 LF
N70     M30 LF
```

Machining of Thread M130 x 4 P using Thread Whirling Technology & Depth 60 mm

1. First of all, a bore of (130 - 4) = 126 mm. dia. is machined, and a relief groove of 4 mm. width is machined at the end of the hole depth.
2. Next a thread milling cutter of 80 mm. dia. & 4 mm. pitch is used to generate the thread M125 x 4P using helical interpolation.

The CNC part program is given below:

```
        % MPF 3000
N5      (SINUMERIK 850M CNC CONTROLLER)
        LF
N10     (LOAD THREAD WHIRLING HEAD DIA
        80MM) LF
N15     T1=2 M06 (CHANGE TOOL NO 2) LF
N20     G90 G54 G94 G17 LF
N25     M44 S795 LF
N30     M3 LF
N35     G0 Z30 D2 LF
N40     G0 X0 Y0 LF
N45     L200 (SUBPROGRAM FOR THREAD
        WHIRLING) LF
N50     G0 G90 X0 Y0 LF
N55     Z100 LF
N60     D0 M5 LF
N65     M30 LF
        % SPF 200
N5      (SUBPROGRAM FOR THREAD
        WHIRLING) LF
N10     R60=0 R61=1 R62=16 (NO. OF THREAD
        PITCH) LF
N20     G1 G17 G41 D2 X0 Y –30 F1000 LF
N30     Z–62 M11 (POSITION THE TOOL AT THE
                END OF THE HOLE) LF
N40     G91 G3 G64 X 0 Y95 Z 2 I0 J 47.5 F300 LF
N50     G3 X0 Y0 I0 J-65 Z4 F1192 LF
N60     R60 = R60 + R61 LF
N70     @ 126 R62 R60 K – 50 LF
N75     (GO TO BLOCK NO 50 TILL R60 IS
        GREATER THAN R62) LF
N80     G3 X0 Y-95 I0 J–47.5 F600 LF
N90     G0 Z50 LF
N100    G0G90G40 X0 Y0 LF
N110    M17 LF
```

Calculation of the Thread Milling Cutter Feed at the Circular Thread dia.

Let, Vfm = Feed of the Thread milling cutter centre.
Vf = Feed at the circular thread dia.
n = Spindle RPM
Fz = Feed / insert / revolution = 0.25 mm / insert production
Z = No of inserts on the thread milling cutter = 6 in this case of 80 mm. dia.

Vfm is calculated by using the formula

$$Vfm = \frac{(Thread\ dia. - Thread\ milling\ cutter\ dia.) \times n \times Fz \times z}{Thread\ dia.}$$

$$= \frac{(130-80) \times 795 \times 0.25 \times 6}{130} = 458.65\ mm/minute$$

$$Vf = \frac{Vfm \times 130}{(130-80)} = 1192.5\ mm/minute$$

4.0 CONCLUSION

Thus, we observe that with the advent of CNC Machines, machining of complex profiles on cylindrical components, and the machining of bores and threads on prismatic parts can be performed with ease without the help of form tools. This has opened up new possibilities for the benefit of engineering industries.

5.0 ACKNOWLEDGEMENTS

The authors wish to acknowledge with thanks the contributions of various persons directly or indirectly involved in writing this paper. The authors wish to acknowledge with thanks the suggestions given by colleagues, particularly R. Aind & A. K. Pippal who are working in a heavy electrical factory in the field of CNC Part Programming. The authors also wish to thank RKDF University for giving permission for writing this paper.

REFERENCES

1. *Cao Schuling and Guan Jinbiao, Research o CNC Machining Process and Programming Technology of Complex parts based on PowerMLL, Advances in Computer Science Research (ACSR), Vol 83, 8th International Conference on Mechatronics, Computer and Education Informationization (MCEI 2018)*
2. *D.D. Mohite, N. Tiwari et al, Modelling and Optimization of Dressing parameters of CNC Cylindrical Grinding wheel for minimum surface roughness, International Journal of Engineering Research and General Science, Vol 5, Issue 4, July-August 2017.*
3. *R. Rudrapati et al, Turning Parameters on Surface roughness in CNC Turning of Aluminium alloy using hybrid RSM cum TLBO approach, IOP Conference Series: Material Science and Engineering 149 (2016) 012039.*
4. *Mohammed Nafis Osman Zahid, Keith Case et al, Optimization of roughing operations in CNC machining for rapid manufacturing process, Production & Manufacturing Research, Vol. 2, No 1, 519-529, 2014*
5. *Y. Nakao & E. Urata, Study on a CNC Free form Surface Machining System with a Rotating Cylindrical Tool, Kanagawa University, Japan*
6. *K. Sprott, B. Ravari, Cylindrical milling of ruled surfaces, International Journal of advanced manufacturing technology, September 2008 (2008) 38: 649-656*
7. *Chottopadhyay, A.B., Machining & Machine Tools, Wiley, 1 Jan 2017.*
8. *Manocher Djassemi, An efficient CNC Programming approach based on Group Technology, Journal of Manufacturing Systems, Vol 19, Issue 3, 2000, pp213-217.*
9. *Srivasalu, M., Mani Prasad N., Study on Computer Numerical Control Machines, International Journal of Advanced Scientific research, Vol 1, Issue 1, April 2016, pp 13-17.*
10. *Chau-Hwa Chang, Michel A. Melkanoff, NC Machine Programming & Software Design, Prentice Hall International Inc., a division of Simon & Schuster, Englewood, Cliffs, N. J07632.*
11. *Vishal, S., An Introduction to NC/CNC Machines, S.K. Kataria & Sons.*
12. *Chryssolouris, George, Manufacturing Systems, - Theory & Practice, Springer Verlag.*
13. *Daga, S. N., Basak, P.K., Sethi Sumit, Aind, R., Parametric Programming: A Productivity Enhancement Tool for CNC Machines, 12th National Convention of Production Engineers, The Institution of Engineers (India), Jabalpur Local Centre, 18-19 Oct 1997, pp 99-105*
14. *Programming Manual, Sinumerik 850M of Siemens, Germany*
15. *Michael Mattson, Delmar Cengage Learning, 2010*
16. *Mike Lynch, Managing Computer Numerical Control Operations, Society of Manufacturing Engineers, 1995*
17. *Li Y. G., Fang T. L., Cheng S. J., et. al., Research on Feature based Rapid Programming for Aircraft NC Parts, Journal of Applied Mechanics & Materials, 2008, 10-12: 682-687*
18. *Jiao H., Research on NC Turning Technique based on Finishing Machining, Journal of New Technology & New Process, 2013*

AUTHORS

Shri PK Basak, Research Scholar, Sri Satya Sai College of Engineering, RKDF University, Aero City Road, Near RGPV, Abbas Nagar, Gandhi Nagar, Bhopal – 462 033, (MP) Email: pkbasak07@gmail.com

Dr. Manish Gangil, Professor, Sri Satya Sai College of Engineering, RKDF University, Aero City Road, Near RGPV, Abbas Nagar, Gandhi Nagar, Bhopal – 462 033, (MP) Email: rkdfbhojpal@gmail.com