A Paradigm shift in the machining of bores & threads using circular and helical interpolation on CNC Machines

P. K. Basak[1, a)] and Manish Gangil[2, b)]

[1]*Research Scholar, RKDF University, Bhopal*
[2]*Professor, Sri Satya Sai College of Engineering, RKDF University, Bhopal*

Abstract. *The machining of bores and Threads constitute a significant portion of machining activity in an engineering industry. The machining of bores of different sizes requires different boring tools of different sizes. Boring tools are size-specific. These tools are quite expensive and an inventory of various sizes of boring tools is to be maintained for carrying out machining of different sizes of bores. Similarly, for the machining of threaded holes, different sized taps are required for the machining of different sized tapped holes. The taps are also size – specific. For example, for the machining of a tapped hole of size M 36 x 4, a tap of the same size in required. With the advent of CNC Technology, the feature like circular interpolation & helical interpolation have proved out to be very helpful for the machining of bores & threaded holes respectively. The machining of bores & threaded holes has become quite easy and convenient. On a CNC machine a bore can be generated by a deep shoulder end mill by using circular interpolation feature of the CNC controller. In circular interpolation, a free size milling cutter which is smaller than the bore size is used to generate the bore. Threads can also be machined on a CNC machine by a tread milling cutter using the helical interpolation feature of the CNC controller. Here also, a free size thread milling cutter of a particular pitch, which is smaller than the thread size is utilized to generate the threaded hole. In both the cases of bore and threaded hole machining a free size cutter of smaller size can generate a wide variety of sizes of bores & threaded holes. This has led to a drastic reduction of inventory of cutting tools, thus saving a lot of costs of tooling This has led to a paradigm shift in machining of bores & threaded holes. This has been made possible by the CNC Technology.*

Keywords. CNC Machine, Circular Interpolation, Helical Interpolation, Parametric programming

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 servomotors 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 servomotor. 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.

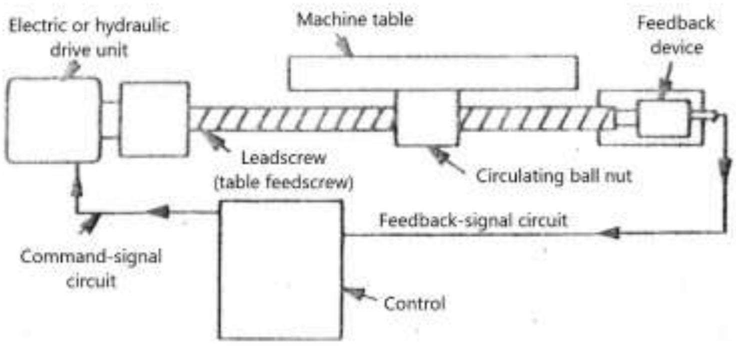

Figure 1

CNC 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 in 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 are 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

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.

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).

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 selection 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.

Parametric Programming

While writing part programs for families of parts, programming with parameters (variables) can save an immense amount of efforts 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
Sin Value	@ 630
Cosine Value	@ 631

In Sinumerik 850M CNC Control system of Siemens, park-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.

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 and mill & circular interpolation

The following example of the 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 200 mm dia. using a deep shoulder end mill of 60mm dia. is given below:

```
      % MPF 1300
N2    (MACHINING A BOARD OF 200MM 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=100 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 from 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 M125 x 4 P using Thread Whirling Technology

1. First of all, a bore of (125 - 4) = 121 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 Y92.5 Z 2 IO J 46.25 F300 LF
N50   G3 X 040 IO J-62.5 Z 4 F 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 X 0 Y-92.5 I0 J - 46.25 F66 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

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

$$= \frac{(125-80\,) \times 795 \times 0.25 \times 6}{125} = 429.3\ mm/minute$$

$$Vf = \frac{Vfm\ \times\ 125}{(125-80)} = 1192.5\ mm/minute$$

Conclusion

Thus, we observe that with the advent of CNC Machines, machining of bores and threads are not dependent anymore on the specific sizes of boring tools and threading tools. Bores and threads can be machined with free size cutters. This has provided a lot of flexibility and freedom to the machining of bores and threads. Rather, this has provided a paradigm shift in their machining methodology.

Acknowledgements

The authors wish to acknowledge with thanks the contributions of various persons directly or indirectly involved in writing this paper. The author number 1 has drawn heavily from his experience of working in an engineering manufacturing company as a CNC Part Programmer, Process Planner in writing this paper. The authors also wish to thank RKDF university for giving permission for presenting this paper.

References

1. Chottopadhyay, A.B., *Machining & Machine Tools*, Wiley, 1 Jan 2017.
2. Manocher Djassemi, *An efficient CNC Programming approach based on Group Technology*, Journal of Manufacturing Systems, Vol 19, Issue 3, 2000, pp213-217.
3. 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.
4. Chau-Hwa Chang, Michel A. Melkanoff, *NC Machine Programming & Software Design*, Prentice Hall International Inc., a division of Simon & Schuster, Englewood, Cliffs, N. J07632.
5. Vishal, S., *An Introduction to NC/CNC Machines*, S.K. Kataria & Sons.
6. Chryssolouris, George, *Manufacturing Systems, - Theory & Practice*, Springer Verlag.
7. 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
8. Programming Manual, Sinumerik 850M of Siemens, Germany