THE GATE COACH

Revised Study Material

For

GATE – PSU

Mechanical Engineering

Production Technology 1

The Gate Coach

28, Jia Sarai, Near IIT

Hauzkhas, New Delhi – 16

(+91) 9818652587,

9873452122

PRODUCTION TECHNOLOGY

VOLUME I

CONTENTS

1 METROLOGY	Introduction	3
	Measurement	4
	Inspection	4
	Terminologies	6
	Standard gauges and limit gauges	13
	Angular Measurement	17
	Surface finish	25
	Interferometry	29
	Comparators	32
	Errors	35

2 METAL CUTTING	Machining & Elements of machining	43
	Orthogonal and oblique cutting	43
	Single point cutting tool	44
	Too signature	46
	Machining mechanics	48
	Merchant circle	52
	Metal cutting theories	57
	Tool failure and tool life	58
	Tool materials	62
	Economics of metal cutting	64

3 MACHINING	3A – Lathe	67
	3B – Shaper and planer	86
	3C – Drilling	89
	3D – Reaming	95
	3E – Milling	97
	3F – Grinding and Finishing	111
	3G – Broaching	121
	SOLVED EXAMPLE ON METAL CUTTING & MACHINING	126

4 GEAR MANUFACTURING	Basic use and applications	135
	Types of gears	135
	Specifications of gears	136
	Gear material	136
	Gear geometry	137
	Manufacture of gears	137
	Finishing of gears	144

5 JIGS AND FIXTURES	Introductions	146
	Consideration in designing	146
	Main principles of design	147
	Main elements	148
	Degree of freedom	148
	Location principles	149
	Locating devices	151
	Types of jigs	153
	Types of fixtures	154

6 UNCONVENTIONAL MACHINING	Electric Discharge Machining	155
	Electro-Chemical Machining	161
	Chemical Machining	168
	Electron Beam Machining	170
	Laser Beam Machining	174
	Ultrasonic Machining	178
	Abrasive jet machining	180
	Water Jet machining	184
	Plasma Arc machining	189

7 ADVANCED MACHINING METHOD	Introduction	192
	Advantages	192
	Elements of CNC	193
	Part Programming Fundamentals	194
	Principles of CNC	194
	Precisions in CNC Machining	198
	G and M codes	200

8 ELEMENTS OF COMPUTATION	The FORTRAN Programming language	203
	Basic elements of FORTRAN	203
	Data types	204
	Undeclared Names	205
	Assignments and Arithmetic operations	206
	Inputs and Outputs	208
	Arrays in FORTRAN	211

CHAPTER 7 ADVANCED MACHINING METHODS

INTRODUCTION

Numerical control of a machine may be defined as the method of automation in which various functions of machine tools are controlled by letters, numbers and symbols. Basically NC machines runs on a program fed to it. The program consists of precise instructions about methodology of manufacture as well as the movements. Since the machine is controlling point for the product manufacture, the machine become versatile and can be used for any part. All the function of an NC machine tool are therefore controlled electronically, hydraucally or pneumatically.

DEFINITION

Computer Numerical Control (CNC) is one in which the functions and motions of a machine tool are controlled by means of a prepared program containing coded alphanumeric data. CNC can control the motions of the workpiece or tool, the input parameters such as feed, depth of cut, speed, and the functions such as turning spindle on/off, turning coolant on/off.

ADVANTAGES OF NC MACHINE OVER CONVENTIONAL MACHINE TOOL:

1. Parts can be produced in less time and therefore are likely to be less expensive.
2. Parts can be produced more accurately, even for smaller batches.
3. The operator involvement is reduced to minimum, hence less scrap is generated.
4. Part program takes care of the geometry generated, so there is no need of expensive jigs and fixtures.
5. Inspection time gets reduced since all the parts in the batch would be identical.
6. The need for certain type of form tools is completely eliminated in NC machines. Because the profile to be generated can be programmed.
7. Lead time needed for the job is to be put on the machine tool can be reduced to a great extent.

LIMITATIONS

1. High costs of NC machine tools.
2. Cost and skill of the operator required for NC machine is higher.
3. Special training is needed for personnel manning the NC machine tools.
4. Higher maintenance costs.
5. Higher running costs due to automatic operation.

APPLICATIONS

The applications of CNC include both for machine tool as well as non-machine tool areas. In the machine tool category, CNC is widely used for lathe, drill press, milling machine, grinding unit, laser, sheet-metal press working machine, tube bending machine etc. Highly automated machine tools such as turning centre and machining centre which change the cutting tools automatically under CNC control have been developed. In the non-machine tool category, CNC applications include welding machines (arc and resistance), coordinate measuring machine, electronic assembly, tape laying and filament winding machines for composites etc.

ELEMENTS OF A CNC

The principle of operation of a NC machine tool has been shown in figure. The basic information that has to be input in to the system consists of the part geometry, cutting process parameters followed by the cutting tool used. This part program is entered in to the controller of tha machine, which in turn runs the machine tool to make the part.

A CNC system consists of three basic components:
1. Part program
2. Machine Control Unit (MCU)
3. Machine tool (lathe, drill press, milling machine etc)

PART PROGRAM

The part program is a detailed set of commands to be followed by the machine tool. Each command specifies a position in the Cartesian coordinate system (x,y,z) or motion (workpiece travel or cutting tool travel), machining parameters and on/off function. Part programmers should be well versed with machine tools, machining processes, effects of process variables, and limitations of CNC controls. The part program is written manually or by using computer assisted language such as APT (Automated Programming Tool).
The program can also be written in higher levellanguages such as UNIAPT, COMPACT II etc. these programs have to be converted in to the earlier mentioned machine tool level program with the help of processors. It is similar to the practice by which computer programs written in HLL such as FORTAN are converted into the relevant computer machine language with the helpof a suitable compiler.
The programs can also be developed directly using the CAD/CAM systems such as Unigraphics, Pro Engineering etc. This would alsorequire a post processor.

MACHINE CONTROL UNIT

The machine control unit (MCU) is a microcomputer that stores the program and executes the commands into actions by the machine tool. The MCU consists of two main units: the data processing unit (DPU) and the control loops unit (CLU). The DPU software includes control

system software, calculation algorithms, translation software that converts the part program into a usable format for the MCU, interpolation algorithm to achieve smooth motion of the cutter, editing of part program (in case of errors and changes). The DPU processes the data from the part program and provides it to the CLU which operates the drives attached to the machine lead screws and receives feedback signals on the actual position and velocity of each one of the axes. A driver (dc motor) and a feedback device are attached to the lead screw.
The CLU consists of the circuits for position and velocity control loops, deceleration and backlash take up, function controls such as spindle on/off.
MCU controls the motion of cutting tool, spindle speeds, feed rate, tool changes, cutting fluid application and several other functions of the machine tool.

MACHINE TOOL

The machine tool could be one of the following: lathe, milling machine, laser, plasma, coordinate measuring machine etc. Figure 3 shows that a right-hand coordinate system is used to describe the motions of a machine tool . There are three linear axes (x,y,z), three rotational axes (i,j,k), and other axes such as tilt (9) are possible.

PART PROGRAMMING FUNDAMENTALS

Steps to be followed while developing CNC part programs:

1. **Process Planning**

Process plan is a detailed plan of the steps involved in manufacturing a given part. It includes

- Machine tool used
- Fixture required
- Sequence of operations
- Cutting tools required
- Process parameters

2. **Axes selection**

All the CNC machine tools rely on the axes system for describing the axes motion. To correctly describe the motion, it is necessary to establish the axes system to be followed with the particular part.

3. **Tool Selection**

The choice of cutting tool is very important function, since for a given operation many tools are feasible, but some of them would be more economical than others Therefore in the economy of manufacture, it is essential to choose the right tool for the job.

4. **Cutting process parameters planning**

For a given tool and the operation selected, the appropriate process parameters are to be selected. There are to be generally taken from the handbooks supplied by the cutting tool manufacturer or based on the shop experiences.

5. **Job and tool set up planning**

This basically is aimed at setting job on the machine tool and adjusting the cutting tool to correct position. This is important since the accuracy of the geometry generated by the CNC machine tool is dependent on the initial position carefully defined.

6. Machining path planning

A careful planning of the tool path ensures that the requisite manufacturing specifications are achieved at the lowest cost.

7. Part Program writing

It deals with the actual writing of the part programs undertaking the format and syntax restrictions in to account.

8. Part program proving

Once the program is made, it should be verified before it can be loaded on the machine tool controller for the manufacture of the component. A faulty program may cause damage to the tool, workpiece and the machine tool itself.

9. NC documentation

This is the most essential aspect of the CNC manufacturing practice. It should include :

- Component drawing
- Process planning sheet
- Tool cards
- Setting cards
- Programming sheets
- Punched paper tape

PRINCIPLES OF CNC

Basic Length Unit (BLU)

Each BLU unit corresponds to the position resolution of the axis of motion. For example,
1 BLU = 0.0001" means that the axis will move 0.0001" for every one electrical pulse received by the motor. The BLU is also referred to as Bit (binary digit).
Pulse = BLU = Bit

Point-to-Point Systems

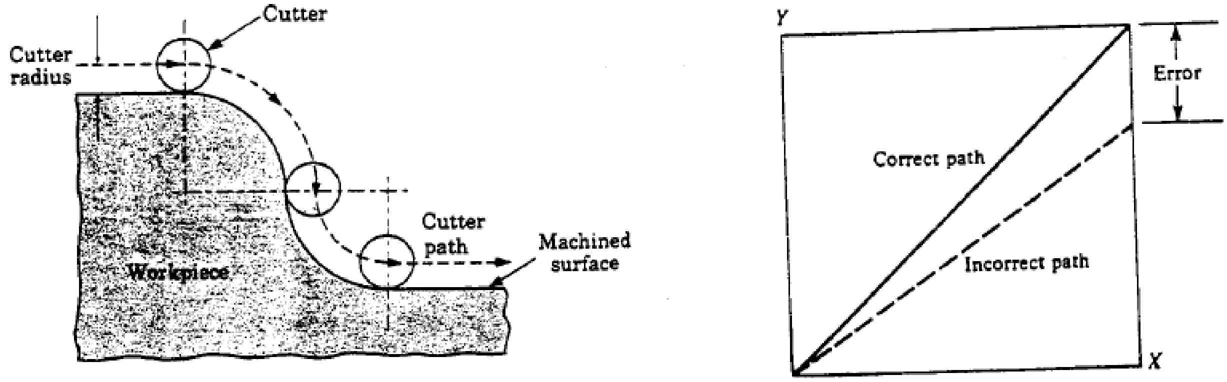
Point-to-point systems are those that move the tool or the workpiece from one point to another and then the tool performs the required task. Upon completion, the tool (or workpiece) moves to the next position and the cycle is repeated. The simplest example for this type of system is a drilling machine where the workpiece moves.

In this system, the feed rate and the path of the cutting tool (or workpiece) have no significance on the machining process. The accuracy of positioning depends on the system's resolution in terms of BLU (basic length unit) which is generally between 0 .001" and 0.0001" .

Continuous Path Systems (Straight cut and contouring systems)

These systems provide continuous path such that the tool can perform while the axes are moving, enabling the system to generate angular surfaces, two-dimensional curves, or three dimensional contours. Example is a milling machine where such tasks are accomplished

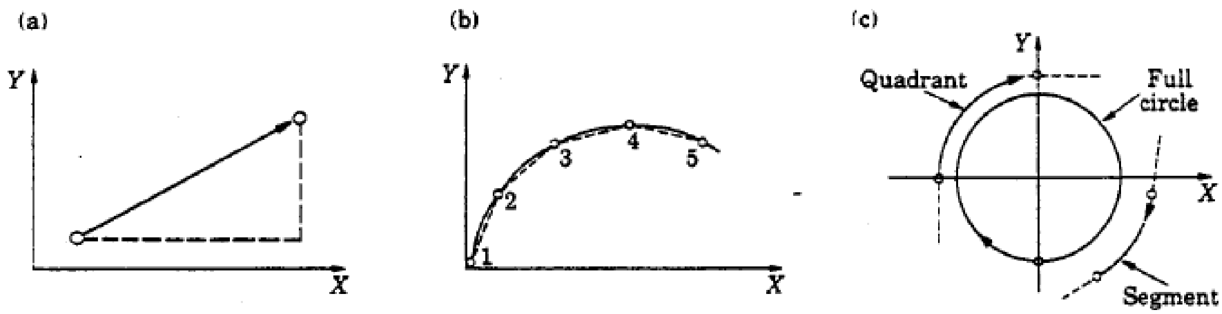
Each axis might move continuously at a different velocity. Velocity error is significant in affecting the positions of the cutter. It is much more important in circular contour cutting where one axis follows sine function while the other follows cosine function.



(a) Continuous path cutting and (b) Position error caused by the velocity error

Interpolator

The input speed of l in/sec in example 2 is converted into the velocity components by an interpolator called the linear interpolator whose function is to provide the velocity signals to x and y directions. Similarly we have circular and parabolic interpolators.



. Types of interpolation (a) linear, (b) continuous path approximated by incremental straight lines, and (c) circular

Incremental and Absolute systems

CNC systems are further divided into incremental and absolute systems .In incremental mode, the distance is measured from one point to the next. For example, if you want to drill five holes at different locations, the x-position commands are x + 500, + 200, + 600, - 300, -700, -300. An absolute system is one in which all the moving commands are referred from a reference point (zero point or origin) . For the above case, the x-position commands are x 500,700, 1300, 1000, 300, 0. (Figure 8) . Both systems are incorporated in most CNC systems. For an inexperienced operator, it is wise to use incremental mode.

The absolute system has two significant advantages over the incremental system

1. Interruptions caused by, for example, tool breakage (or tool change, or checking the parts), would not affect the position at the interruption.

If a tool is to be replaced at some stage, the operator manually moves the table, exchanges the tool, and has to return the table to the beginning of the segment in which the interruption has occurred. In the absolute mode, the tool is automatically returned to the position. In incremental mode, it is almost impossible to bring it precisely to that location unless you repeat the part program

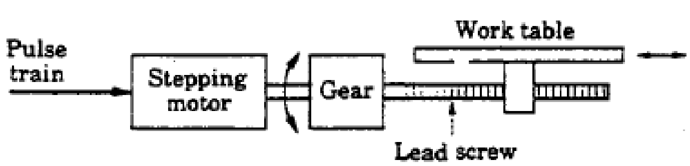
2. Easy change of dimensional data

The incremental mode has two advantages over the absolute mode .

1. Inspection ofthe program is easier because the sum of position commands for each axis must be zero . A nonzero sum indicates an error . Such an inspection is impossible with the absolute system.
2. Mirror image programming (for example, symmetrical geometry of the parts) is simple by changing the signs of the position commands.

OPEN LOOP CONTROL SYSTEMS

The open-loop control means that there is no feedback and uses stepping motors for driving the lead screw. A stepping motor is a device whose output shaft rotates through a fixed angle in response to an input pulse. The accuracy of the system depends on the motor's ability to step through the exact number. The frequency of the stepping motor depends on the load torque. The higher the load torque, lower would be the frequency. Excessive load torque may occur in motors due to the cutting forces in machine tools. Hence this system is more suitable for cases where the tool force does not exist (Example: laser cutting).



Open loop control system

The stepping motor is driven by a series of electrical pulses generated by the MCU. Each pulse causes the motor to rotate a fraction of one revolution. The fraction is expressed in terms of the step angle, α, given by

$$\alpha = 360/N \text{ degrees}$$

Where

N = number of pulses required for one revolution

If the motor receives "n" number of pulses then the total angle,

$$A = n(360/N)$$

In terms of the number of revolutions, it would be (n/N)

If there is a 1 :1 gear ratio between the motor and the lead screw, then the lead screw has (n/N) revolutions . If the pitch of lead screw is p (in/rev), then the distance travelled axially, say x, can be used to achieve a specified x-increment in a point-to-point system.

The pulse frequency, f, in pulses/sec determines the travel speed of the tool or the workpiece.

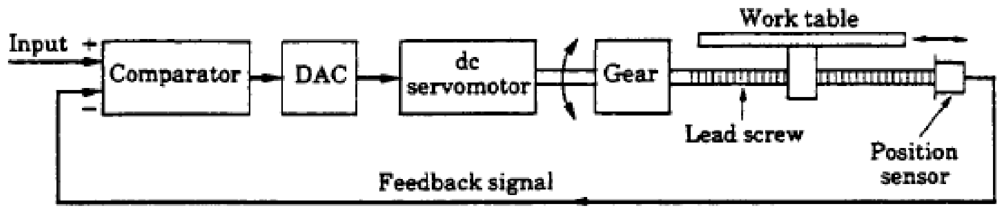
60 f = N (RPM) where N = number of pulses per revolution,

RPM = RPM of the lead screw

The travel speed, V, is then given by V = p (RPM) where p pitch in in/rev

CLOSED-LOOP CONTROL SYSTEMS

Closed -loop NC systems are appropriate when there is a force resisting the movement of the tool/workpiece. Milling and turning are typical examples. In these systems the DC servomotors and feedback devices are used to ensure that the desired position is achieved. The encoder consists of a light source, a photo detector, and a disk containing a series of slots. The encoder is connected to the lead screw. As the screw turns, the slots cause the light to be seen by the photo detector as a series of flashes which are converted into an equivalent series of electrical pulses which are then used to characterize the position and the speed. The equations remain essentially the same as open-loop except that the angle between the slots in the disk is the step angle, a.



Closed loop control system

Both the input to the control loop and the feedback signals are a sequence of pulses, each pulse representing a BLU unit. The two sequences are correlated by a comparator and gives a signal, by means of a digital-to-analog converter, (a signal representing the position error), to operate the drive motor (DC servomotor).

PRECISION IN CNC MACHINING

The combined characteristics of the machine tool and the control determine the precision of positioning. Three critical measures of precision are :

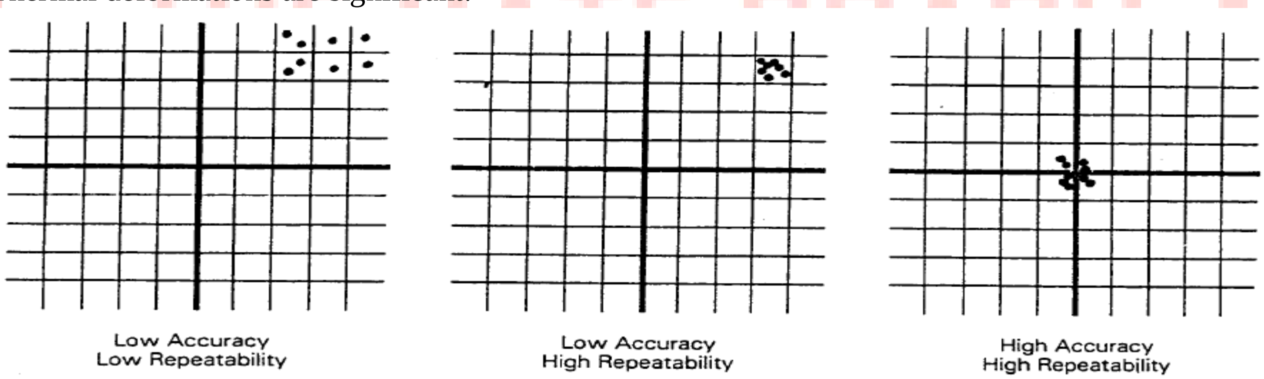
Control resolution (BLU) is the distance separating two adjacent points in the axis movement (The smallest change in the position) . The electromechanical components of the positioning system that affect the resolution are the lead screw pitch, the gear ratio, and the step angle in the stepping motor (open loop) or the angle between the slots in the encoder (closed-loop).
The control resolution for a 1 :1 gear ratio of a stepped motor is,

Resolution = p/N where p = pitch, and $N = 360/\alpha$

Features smaller than the control resolution could not be produced. The programming resolution cannot exceed the control resolution.

Accuracy of a CNC system depends on the resolution, the computer control algorithms, and the machine inaccuracies. The inaccuracy due to the resolution is considered to be (1/2) BLU on the average. The control algorithm inaccuracy is due to the rounding off the errors in the computer which is insignificant. The machine inaccuracy could be due to several reasons (Described below). The designer minimizes this inaccuracy to be under (1/2) BLU and hence
Machining Inaccuracy
Accuracy = (1/2) Resolution + Machining inaccuracy = BLU

Repeatability is a statistical term associated with accuracy. It refers to the capability of a positioning system to return to a programmed point, and is measured in terms of the errors associated with the programmed point. The deviation from the control point (error) usually follows a normal distribution in which case the repeatability may be given as +/- 3a where a is the standard deviation. The repeatability is always better than the accuracy. The mechanical inaccuracy can be considered as the repeatability. Cutting tool deflection, machine tool chatter, mechanical linkage between the lead screw and the tool, and thermal deformations are the chief contributing factors. The lead screw transmits the power to the table or tool holder by means of a nut that engages the lead screw. This will create what is known as "backlash” due to the friction between the screw and the nut. If the nut consists of ball bearings, the friction is reduced. Thermal deformations are significant.



For example, a temperature difference of 1 °C along 1000 mm can cause an error of 0.01 mm.

Part programming for CNC

The transfer of an engineering blueprint of a product to a part program can be performed manually using a calculator or with the assistance of a computer language. A part programmer must have an extensive knowledge of the machining processes and the capabilities of the machine tools. In this section, we describe how the part programmers execute manually the part programs.

First, the machining parameters are determined. Second, the optimal sequence of operations is evaluated. Third, the tool path is calculated. Fourth, a program is written. Each line of the program, referred to as a block, contains the required data for transfer from one point to the next.

A typical line for a program is given below.
N100 G91 X -5.0 Y7 .0 F100 S200 T01 M03 (EOB)
The significance of each term is explained below.

Sequence Number, N
Consisting of typically three digits, its purpose is to identify the specific machining operation through the block number particularly when testing a part program.

Preparatory Function, G
It prepares the MCU circuits to perform a specific operation. G91 implies incremental mode of operation.

Dimension Words
1. Distance dimension words, X,Y,Z
2. Circular dimension words, I,J,K for distances to the arc centre
3. Angular dimensions, A,B.C

While (1) and (3) are expressed either by incremental or absolute mode, (2) is always in given in incremental mode. All angular dimensions are specified in revolutions or degrees.
In the above block, X moves a distance of 5 in. in the negative direction while Y moves a distance of 7 in. in the positive direction. Other axes remain stationary. In some systems, actual distances are used. In others, the dimension words are programmed in BLUs.

Feed rate, F
It is expressed in in/min or mm/min and, is used in contouring or point-to-point or straight-cut systems. For example, a feed rate of F100 implies 100 in/min or 100 mm/min.
Feed rates are independent of spindle speed.
In linear motions, the feed rate of the cutting tool is not corrected for the cutter radius . But in circular motions, the feed rate should be corrected for the tool radius as follows:
For cutting around the outside of a circle, the plus sign in the above equation is used, and the Feed rate is increased. For cutting around the inside of a circle, the minus sign is used, and the feed rate is decreased.

Spindle speed, S
Programmed in rev/min, it is expressed as RPM or by a three-digit code number that is related to the RPM.

Tool word, T
Consisting of a maximum of five digits, each cutting tool has a different code number. The tool is automatically selected by the automatic tool changer when the code number is programmed in a block.
Miscellaneous Function, M
Consisting of two digits, this word relates to the movement of the machine in terms of spindle on/off, coolant on/off etc.

EOB
F = [(part contour radius ± tool radius)/part contour radius] (required feedrate)
The EOB character is used at the end of each block to complete a line.

Table . .1
Preparatory commands (G-code)
G00 Point-to-point positioning
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation
G04 Dwell
G05 Hold
G33 Thread cutting, constant lead
G40 Cancel tool nose radius compensation
G41 Tool nose radius compensation - left
G42 Tool nose radius compensation - right
G43 Cutter length compensation
G44 Cancel cutter length compensation
G70 Dimensions in inches
G71 Metric dimensions
G90 Absolute dimensions
G91 Incremental dimensions
G92 Datum offset
Table . .2 Miscellaneous commands (M-code)
M00 Program stop
M01 Optional stop
M02 End of program
M03 Spindle start clockwise
M04 Spindle start counter-clockwise
M05 Spindle stop
M06 Tool change
M07 Mist coolant on

M08 Flood coolant on
M09 Coolant off
M10 Clamp
Mll Unclamp
M13 Spindle clockwise, coolant on
M14 Spindle counter-clockwise, coolant on
M30 End of tape, rewind

G code Group Function
GOO Positioning (rapid traverse)
GO1 01 Linear interpolation (cutting feed) .
G02 Circular interpolation CW
G03 Circular interpolation CCW
G04 Dwell
G07 00 SIN interpolation (designation of virtual axis)
G09 Exact stop check
G10 Offset amount and work zero point offset amount setting
G17 Designation of X-Y plane
G18 02 Designation of Z-X plane
G19 Designation of Y-Z plane
G20 06 Inch input
G21 Metric input;
G22 04~ Stored stroke,limit,ON
G23 Stored'--s troke"- limit OFF
G27 Return to reference paint
G28 Return to reference-pant:_-_. ..
G29 00 Return from reference point `'
G30 Return to 2nd reference point 4
G31 Skip function
G40 07 Tool diameter compensation cancel
G41 Tool diameter compensation to left
G42 Tool diameter compensation to right
G43 Tool length compensation
G44 08 Tool length compensation
G49 Tool length compensation cancel
G45 Tool offset expansion
G46 00 Tool offset reduction
G47 Tool offset double expansion
G48 Tool offset double reduction
G50 11 Scaling cancel
G51 Scaling
G52 00 .Local coordinate system setting
G53 Machine coordinate system selection

G54 Work coordinate system 1 selection
G55 12 Work coordinate system 2 selection
G56 Work coordinate system 3 selection
G57 Work coordinate system 4 selection
G58 Work coordinate system 5 selection
G59 Work coordinate system .6 selection
G60 00 I One directional positioning
G61 Exact stop check mode
G63 13 I Tapping mode
G64 Continuous cutting mode
G65 00 Macro call
G66 14 Macro modal call A
G67~ C Macro modal call cancel
G73 + Peck dril ling cycle
G74 Reverse tapping cycle
G76 I Fine boring
G80 ` Canned cycle cancel
G81' Dr illing cycle, spot boring
G82 Drilling cycle, counter boring
G83 09 Peck drilling cycle
G84 (Tapping cycle
G85 Boring cycle
G86 I Boring cyc le
G87
r
Back boring cycle -'"
G88 Boring cycle
G89 I Boring cycle
G90 03 L Absolute programming
G91 Incremental programming
G92 00 Programming of absolute zero point
G98 10 I Initial level return (canned cycle)
G99 R point level return (canned cycle)
G501 15 Programmable mirror image cancel
G511 Programmable mirror image