SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, New Delhi / Affiliated to Anna University, Chennai / Accredited by NAAC)

Dindigul – Palani Highway, Dindigul – 624 002

DEPARTMENT OF MECHANICAL ENGINEERING

Value Added Courses Summary 2018-2019

Course Name	CNC Programming
Course Duration	30 hours
Year offered	II year Mechanical Students
Course Instructors	Mr.R.Suresh & Mr.V.Ramasamy, Professor/Mech. Engg, SSMIET Dindigul
Course Outcome	1. Students should be able to understand how to properly orient and set up a work piece in the CNC machine.. 2. Students should be able to calculate optimal cutting speeds and feed rates for different materials. 3. Students should be capable of completion of CNC programming projects that simulate real-world machining scenarios.. 4. Students should be able to identify and rectify errors in CNC programs through simulation.
Course Type	Self Framed
Assessment Mode	
Attendance	30 hours
Number of Participants	43
Scheme of Exam	Evaluation test through offline mode

DEPT. OF MECHANICAL ENGG. * SSMIET, DINDIGUL

Course Coordinator

HoD/Mech.Engg

SSM Institute of Engineering and

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (HUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu Post,
Palani Road, Dindigul - 624 002.

SSM

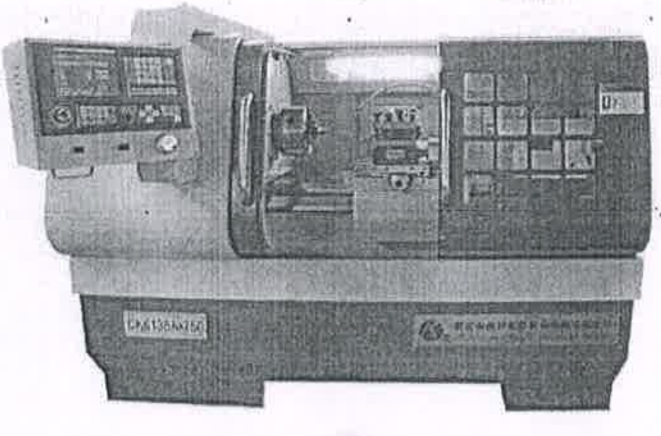
INSTITUTE OF ENGINEERING AND TECHNOLOGY

ETERNAL EXCELLENCE

Value Added Course

on

CNC PROGRAMMING



Prepared By

SURESH.R

RAMASAMY.V

ASSISTANT PROFESSOR

DEPARTMENT OF MECHANICAL ENGINEERING

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
DINDIGUL-PALANI HIGHWAY, DINDIGUL-624002
PHONE: 0451-2448800 – 2448899 (100 LINES) FAX: 0451-2448855
EMAIL: ssmietdgl@gmail.com Website: www.ssmiet.ac.in

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D.,
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
DEPARTMENT OF MECHANICAL ENGINEERING
Value Added Course Program
SECOND YEAR STUDENTS - Batch:2017 to 2021 (15days Attendance)
Course Name - CNC PROGRAMMING

S.No	REG.NO.	NAME OF THE STUDENT	06.08.2018	07.08.2018	09.08.2018	10.08.2018	16.08.2018	20.08.2018	21.08.2018	23.08.2018	24.08.2018	28.08.2018	29.08.2018	10.09.2018	14.09.2018	17.09.2018	19.09.2018
			1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
		A Section															
1	922117114003	AKTHARALI S	/	/	/	/	/	/	/	A	/	/	/	/	/	A	/
2	922117114006	ARULSELVAN K	/	/	/	/	A	/	/	A	A	/	/	/	/	/	/
3	922117114007	ARUN KUMAR E	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
4	922117114011	ARUNKUMAR M	/	A	/	/	A	/	/	/	/	A	/	A	/	/	/
5	922117114012	ASHLEY SACHIN A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
6	922117114028	GAJENDREN R	/	/	/	/	/	A	/	/	A	/	/	/	/	/	/
7	922117114029	GOUTHAM SANKAR K	/	/	/	/	/	/	/	/	/	A	/	/	/	/	/
8	922117114031	GUNAKARAN C	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/
9	922117114032	GUNA SEKAR S	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
10	922117114033	HARI HARAN N	/	/	/	A	/	/	/	/	/	/	/	/	/	A	/
11	922117114041	JAYAPRATHAP N	/	/	/	/	/	A	/	/	A	/	/	/	/	/	/
		B Section															
12	922117114044	JEROME F	/	/	/	/	/	A	/	/	A	/	/	/	/	/	/
13	922117114053	MANICKAVEL V	/	/	/	/	/	/	/	/	/	A	/	/	/	/	/
14	922117114054	MANIKANDAN P	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
15	922117114055	MANIKANDARAJA M	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
16	922117114058	MATHANRAJ G	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
17	922117114059	MATHESHKUMAR B	/	/	/	A	/	/	/	/	/	/	/	/	/	A	/
18	922117114064	MONISHKUMAR M	/	/	/	A	/	/	/	/	/	A	/	/	A	/	/
19	922117114065	MUJIPUR RAHMAN J	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/
20	922117114066	NAGARAJ A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
21	922117114067	NAGA SARAVAN B	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/
22	922117114070	NAVEEN G A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
23	922117114071	NAVEEN BHARATH V	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
24	922117114072	NAVEENKUMAR S	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
25	922117114075	NITHIS C	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
26	922117114076	NITHIS KUMAR K	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/
27	922117114078	PANDIYA RAJ B	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
28	922117114081	PERIYASAMY K	/	/	/	/	/	/	/	/	/	A	/	/	/	/	/
29	922117114083	PREMKUMAR C	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/
30	922117114084	PUGAZHENTHI M	/	/	/	/	/	/	/	/	/	A	/	/	/	/	/
31	LEME1718309	SRI GANESH	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/
32	LEME1718310	SRI SUDHARSANA SAKRAVARTHI	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
		C Section															
33	922117114091	ROBINS CHRISTOPHER L	/	/	/	/	/	/	/	/	A	/	/	/	/	/	/
34	922117114092	ROKESHWARAN R	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/
35	922117114093	SAKTHI SEENIVASAN S	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
36	922117114103	SATHYANARAYANAN V B	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
37	922117114104	SEENU K	/	/	/	/	/	/	/	/	A	/	/	/	/	/	/
38	922117114105	SHANE D	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
39	922117114116	VASANTH S	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/
40	922117114117	VETRIVEL P A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
41	922117114118	VIGNESH B	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
42	922117114119	VIGNESH G	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/
43	922117114126	YOKESWARAN M S	/	A	/	/	/	/	/	/	/	A	/	/	/	A	/

19/9/18

Course coordinator

(R.Ganesh AP/Mech)

(V.Ramasamy AP/Mech)

SSM Institute of Engineering and Technology * DINDIGUL-624 002

24/9/18

HoD/Mech

PRINCIPAL

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS),
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

CNC Programming, Manufacturing and Measuring

INTRODUCTION ABOUT CNC

Introduction

The Term Numerical control is a widely accepted and commonly used term in the machine tool industry. Numerical control (NC) enables an operator to communicate with machine tools through a series of numbers and symbols.

NC which quickly became Computer Numerical Control (CNC) has brought tremendous changes to the metalworking industry. New machine tools in CNC have enabled industry to consistently produce parts to accuracies undreamed of only a few years ago. The same part can be reproduced to the same degree of accuracy any number of times if the CNC program has been properly prepared and the computer properly programmed. The operating commands which control the machine tool are executed automatically with amazing speed, accuracy, efficiency, and repeatability.

The ever-increasing use of CNC in industry has created a need for personnel who are knowledgeable about and capable of preparing the programs which guide the machine tools to produce parts to the required shape and accuracy. With this in mind, the authors have prepared this textbook to take the mystery out of CNC - to put it into a logical sequence and express it in simple language that everyone can understand. The preparation of a program is explained in a 'logical step-by-step procedure, with practical examples to guide the student.

1. Cartesian coordinate system

Almost everything that can be produced on a conventional ma- chine tool can be produced on a computer numerical control machine tool, with its many advantages. The machine tool movements used in producing a product are of two basic types:

1. Point to Point (straight-line movements) and
2. Continuous path (contouring movements).

The Cartesian, or rectangular, coordinate system was devised by the French mathematician and philosopher Rene' Descartes. With this system. any specific point can be described in mathematical terms from any other point along three perpendicular axes. This concept fits machine tools perfectly since their construction is generally based on three axes of motion (X, Y, Z) plus an axis of rotation. On a plain vertical milling machine. the X axis is the horizontal movement (right or left) of the table, the Y axis is the table cross movement (toward or away from the column), and the Z axis is the vertical movement of the knee or the spindle. CNC systems rely heavily on the use of rectangular coordinates because the programmer can locate every point on a job precisely.

SSM Institute of Engineering and Te... DINDIGUL-624 00.

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

When points are located on a work piece two straight intersecting lines, one vertical and one horizontal, are used. These lines must be at right angles to each other and the point where they cross is called the origin, or zero point (Fig. 1)

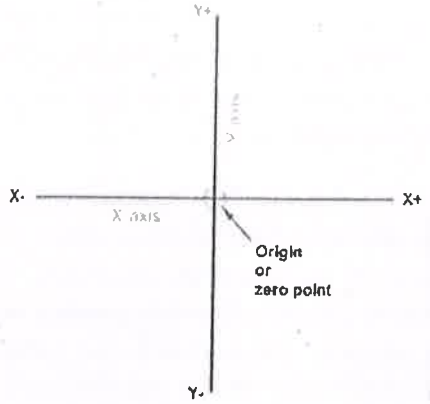


Fig.1. Intersecting lines form right angles and establish the zero point

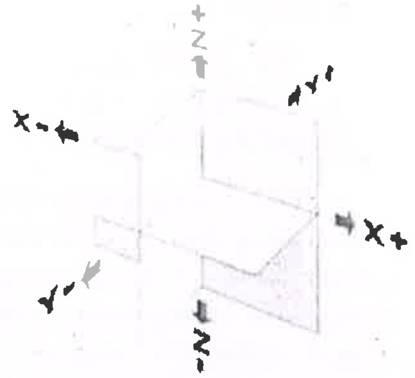


Fig.2. The three dimensional coordinates planes (axes) used in CNC.

The three-dimensional coordinate planes are shown in Fig. 2. The X and Y planes (axes) are horizontal and represent horizontal machine table motions. The Z plane or axis represents the vertical tool motion. The plus (+) and minus (-) signs indicate the direction from the zero point (origin) along the axis of movement. The four quadrants formed when the XY axes cross are numbered in a counter clockwise direction (Fig. 3). All positions located in quad- rant 1 would be positive (X+) and positive (Y+). In the second quadrant, all positions would be negative X (X-) and positive (Y+). In the third quadrant, all locations would be negative X (X-) and negative (Y-). In the fourth quadrant, all locations would be positive X (X+) and negative Y (Y-).

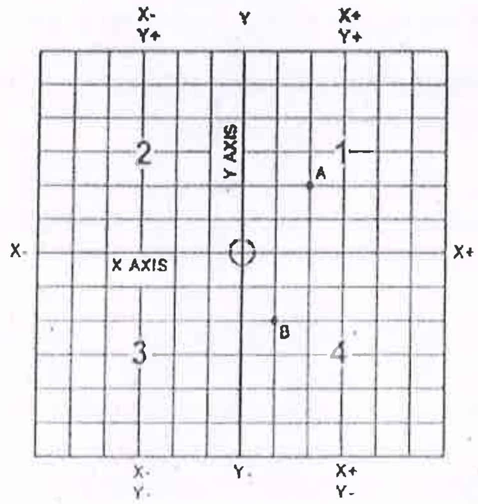


Fig.3. The quadrants formed when the X and Y axes cross are used to accurately locate points from the XY zero or origin or point.

In Fig. 3, point A would be 2 units to the right of the Y axis and 2 units above the X axis. Assume that each unit equals 1.000. The location of point A would be X + 2.000 and Y + 2.000. For point B, the location would be X + 1.000 and Y - 2.000. In CNC programming it is not necessary to indicate plus (+) values since these are assumed. However, the minus (-) values must be indicated. For example, the locations of both A and B would be indicated as follows:

A	X2.000	Y2.000
B	X1.000	Y-2.000

2. Machines Using CNC

Early machine tools were designed so that the operator was standing in front of the machine while operating the controls. This design is no longer necessary, since in CNC the operator no longer controls the machine tool movements. On conventional machine tools, only about 20 percent of the time was spent removing material. With the addition of electronic controls, actual time spent removing metal has increased to 80 percent and even higher. It has also reduced the amount of time required to bring the cutting tool into each machining position.

2.1 Machine Types

2.1.1 Lathe:-

The engine lathe, one of the most productive machine tools, has always been an efficient means of producing round parts (Fig.4). Most lathes are programmed on two axes.

- ❖ The X axis control the cross motion of the cutting tool.
- ❖ Negative X (X-) moves the tool towards the spindle centerline; positive X moves the tool away from the spindle centerline.
- ❖ The Z axis control the carriage travel toward or away from the headstock.

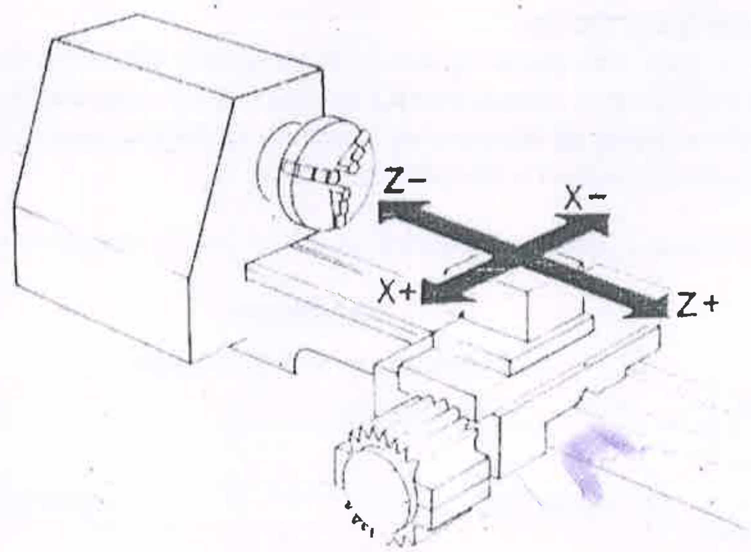


Fig.4. The main axes of a lathe or turning centre

2.1.2 Milling Machine

The milling machine has always been one of the most versatile machine tools used in industry (Fig. 5). Operations such as milling, contouring, gear cutting, drilling, boring, and reaming are only a few of the many operations which can be performed on a milling machine. The milling machine can be programmed on three axes:

- ❖ The X axis controls the table movement left or right.
- ❖ The Y axis controls the table movement toward or away from the column.
- ❖ The Z axis controls the vertical (Up and Down) movement of the knee or spindle.

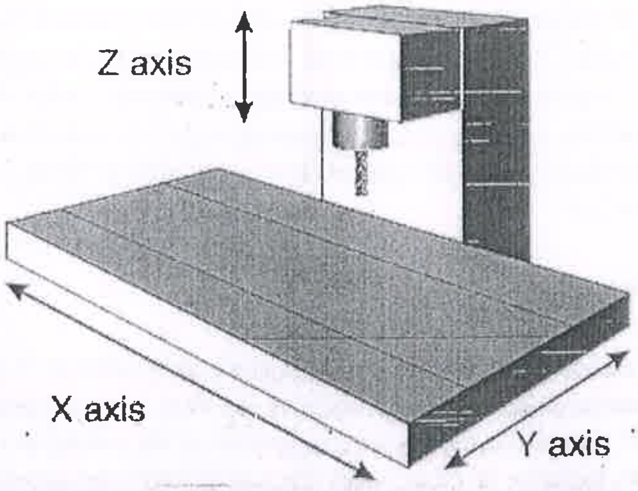
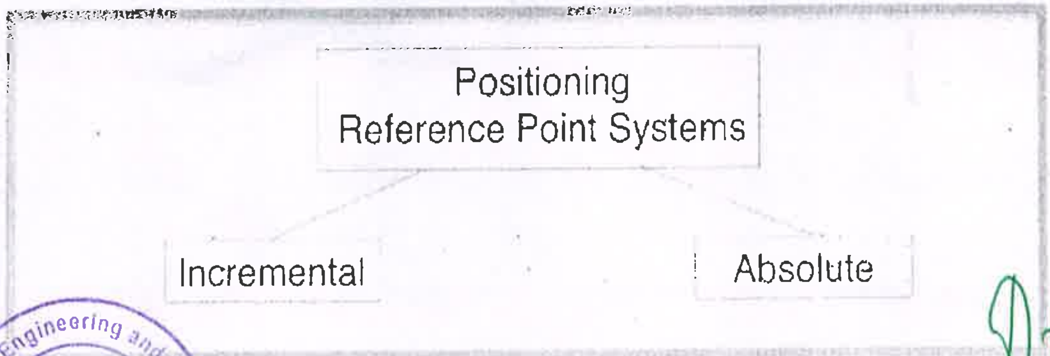


Fig.5. The main axes of a vertical machining centre.

3. Programming Systems

Two types of programming modes, the incremental system and the absolute system, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time. Most controls on machine tools today are capable of handling either incremental or absolute programming.



3.1 Incremental Program:

Incremental program locations are always given as the distance and direction from the immediately preceding point (Fig. 6). Command codes which tell the machine to move the table, spindle, and knee are explained here using a vertical milling machine as an example:

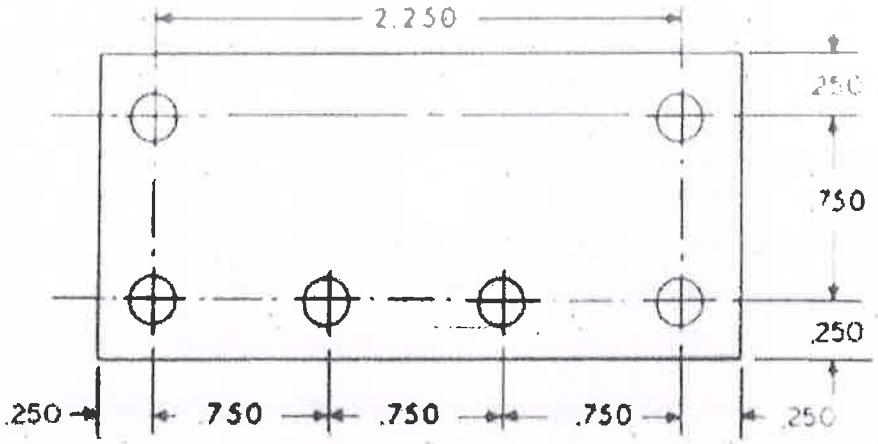


Fig. 6 A work piece dimensioned the incremental system mode.

- A "X plus" (X+) command will cause the cutting tool to be located to the right of the last point.
- A "X minus" (X-) command will cause the cutting tool to be located to the left of the last point.
- A "Y plus" (Y+) command will cause the cutting tool to be located toward the column.
- A "Y minus" (Y-) command will cause the cutting tool to be located away from the column.
- A "Z plus" (Z+) command will cause the cutting tool or spindle to move up or away from the work piece.
- A "Z minus" (Z-) command will cause the movement of the cutting tool down or into the work piece.

In incremental programming, the G91 command indicates to the computer and MCU (Machine Control Unit) that programming is in the incremental mode.

3.2 Absolute program:

Absolute program locations are always given from a single fixed zero or origin point (Fig. 7). The zero or origin point may be a position on the machine table, such as the corner of the worktable or at any specific point on the workpiece. In absolute dimensioning and programming, each point or location on the workpiece is given as a certain distance from the zero or reference point.

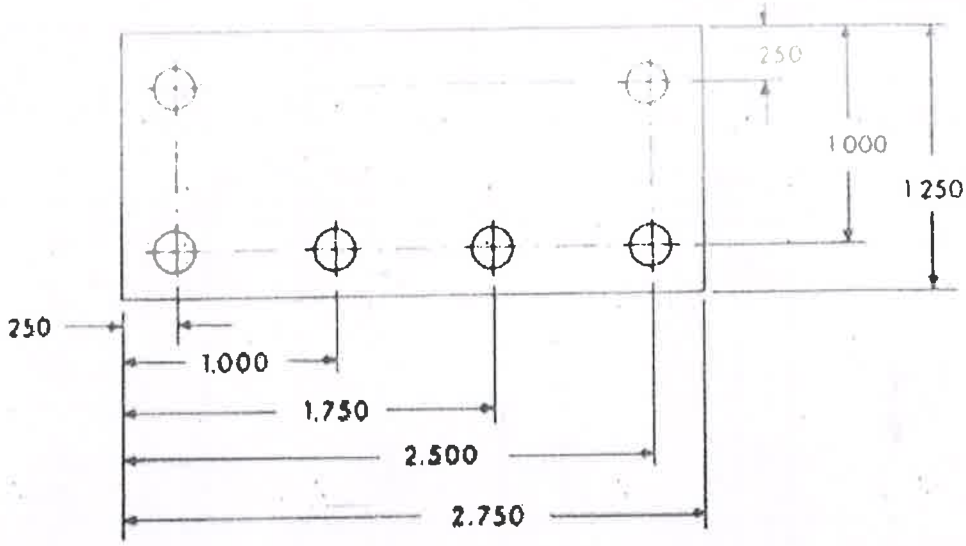


Fig. 7 A workpiece dimensioned in the absolute system mode.

- A "X plus" (X+) command will cause the cutting tool to be located to the right of the zero or origin point.
- A "X minus" (X-) command will cause the cutting tool to be located to the left of the zero or origin point.
- A "Y plus" (Y+) command will cause the cutting tool to be e located toward the column.
- A "Y minus" (Y-) ~~command~~ will cause the cutting tool to be located away ~~from~~ the column.

In absolute programming, the G90 command indicates to the computer and MCU that the programming is in the absolute mode.

3.3 Point-to-Point or Continuous Path:

CNC programming falls into two distinct categories (Fig. 8).

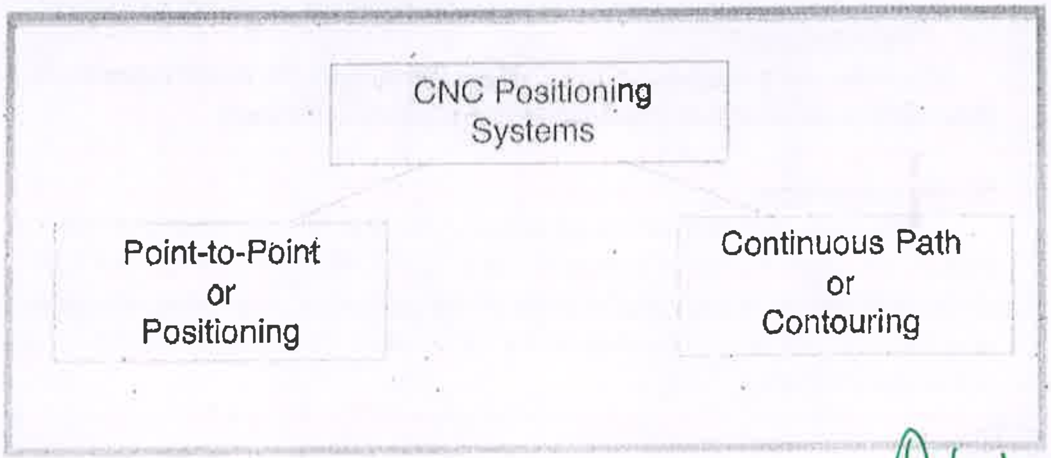


Fig. 8 Types of CNC positioning systems

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL K... M.E., Ph.D., (NUS)
SSM Institute of Engineering and Technology
Kuttathupatti Village, ...Dindigul (Po),
Palani Road, Dindigul - 624 002.

The difference between the two categories was once very distinct. Now, however, most control units are able to handle both point-to-point and continuous path machining. A knowledge of both programming methods is necessary to understand what applications each has in CNC.

3.3.1 Point-to-Point Positioning:

Point-to-point positioning is used when it is necessary to accurately locate the spindle, or the workpiece mounted on the ma- chine table, at one or more specific locations to perform such operations as drilling, reaming, boring, tapping, and punching (Fig. 9). Point-to-point positioning is the process of positioning from one coordinate (XY) position or location to another, performing the machining operation, and continuing this pattern until all the operations have been completed at all programmed locations.

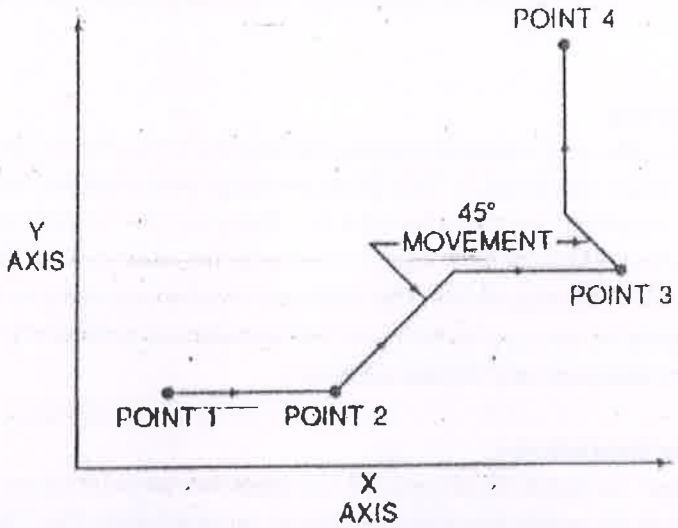


Fig. 9 The path followed by point-to-point positioning to reach various programmed points (machining locations) on the XY axis.

In Fig. 9 point 1 to point 2 is a straight line, and the machine moves only along the X axis; but points 2 and 3 require that motion along both the X and Y axes takes place. As the distance in the X direction is greater than in the Y direction, Y will reach its position first, leaving X to travel in a straight line for the remaining distance. A similar motion takes place between points 3 and 4.

3.3.2 Continuous Path (Contouring):

Contouring, or continuous path machining, involves work such as that produced on a lathe or milling machine, where the cutting tool is in contact with the workpiece as it travels from one programmed point to the next. Continuous path positioning is the ability to control motions on two or more machine axes simultaneously to keep a constant cutter workpiece relationship. The programmed information in the CNC program must accurately position the cutting tool from one point to the next and follow a predefined accurate path at a programmed feed rate in order to produce the form or contour required (Fig. 10).

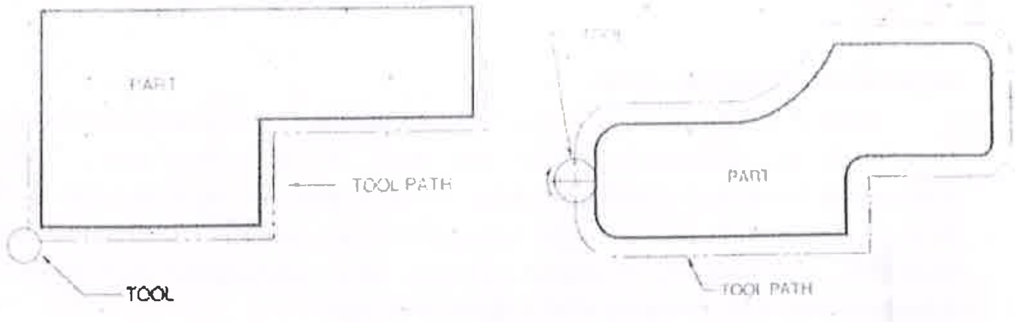


Fig. 10 Types of contour machining (A) Simple contour; (B) complex contour

3.4 Interpolation

The method by which contouring machine tools move from one programmed point to the next is called interpolation. This ability to merge individual axis points into a predefined tool path is built into most of today's MCUs. There are five methods of interpolation: linear, circular, helical, parabolic, and cubic. All contouring controls provide linear interpolation, and most controls are capable of both linear and circular interpolation. Helical, parabolic, and cubic interpolation are used by industries that manufacture parts which have complex shapes, such as aerospace parts and dies for car bodies.

3.4.1 Linear Interpolation:

Linear Interpolation consists of any programmed points linked together by straight lines, whether the points are close together or far apart (Fig. 11). Curves can be produced with linear interpolation by breaking them into short, straight-line segments. This method has limitations, because a very large number of points would have to be programmed to describe the curve in order to produce a contour shape.

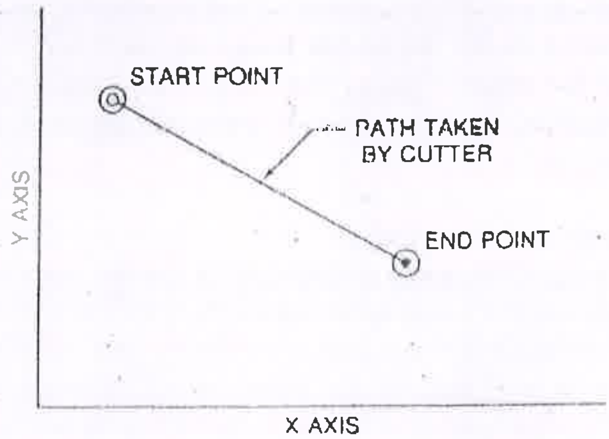


Fig. 11 An example of two-axis linear interpolation.

SSM Institute of Engineering and Technology * DINDIGUL-624 002

Dr.D.SENTHIL ... RAN, M.E., Ph.D., (NUS)
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

A contour programmed in linear interpolation requires the coordinate positions (XY positions in two-axis work) for the start and finish of each line segment. Therefore, the end point of one line or segment becomes the start point for the next segment, and so on, throughout the entire program.

3.4.2 Circular Interpolation:

The development of MCUs capable of circular interpolation has greatly simplified the process of programming arcs and circles. To program an arc (Fig. 12), the MCU requires only the coordinate positions (the XY axes) of the circle centre, the radius of the circle, the start point and end point of the arc being cut, and the direction in which the arc is to be cut (clockwise or counter clockwise) See Fig. 12. The information required may vary with different MCUs.

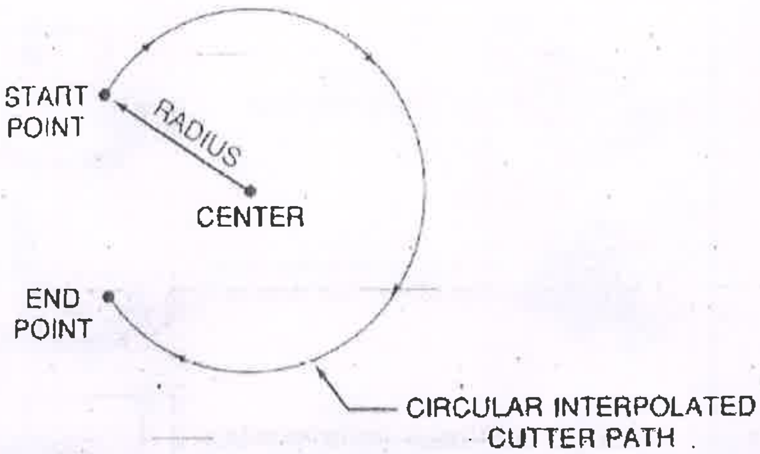


Fig. 12 For two-dimensional circular interpolation the MCU must be supplied with the XY axis, radius, start point, end point, and direction of cut.

3.5 Programming Format

Word address is the most common programming format used for CNC programming systems. This format contains a large number of different codes (preparatory and miscellaneous) that transfers program information from the part print to machine servos, relays, micro-switches, etc., to manufacture a part. These codes, which conform to EIA (Electronic Industries Association) standards, are in a logical sequence called a block of information. Each block should contain enough information to perform one machining operation.

3.5.1 Word Address Format

Every program for any part to be machined, must be put in a format that the machine control unit can understand. The format used on any CNC machine is built in by the machine tool builder and is based on the type of control unit on the machine. A variable-block format which uses words (letters) is most commonly used. Each instruction word consists of an address character, such as X, Y, Z, G, M, or S. Numerical data follows this address character to identify a specific function such as the distance, feed rate, or speed value.

The address code G90 in a program, tells the control that all measurements are in the absolute mode. The code G91, tells the control that measurements are in the incremental mode.

3.5.2 Codes

The most common codes used when programming CNC machines tools are G-codes (preparatory functions), and M codes (miscellaneous functions). Other codes such as F, S, D, and T are used for machine functions such as feed, speed, cutter diameter offset, tool number, etc.

G-codes are sometimes called cycle codes because they refer to some action occurring on the X, Y, and/or Z axis of a machine tool, Fig. 13.

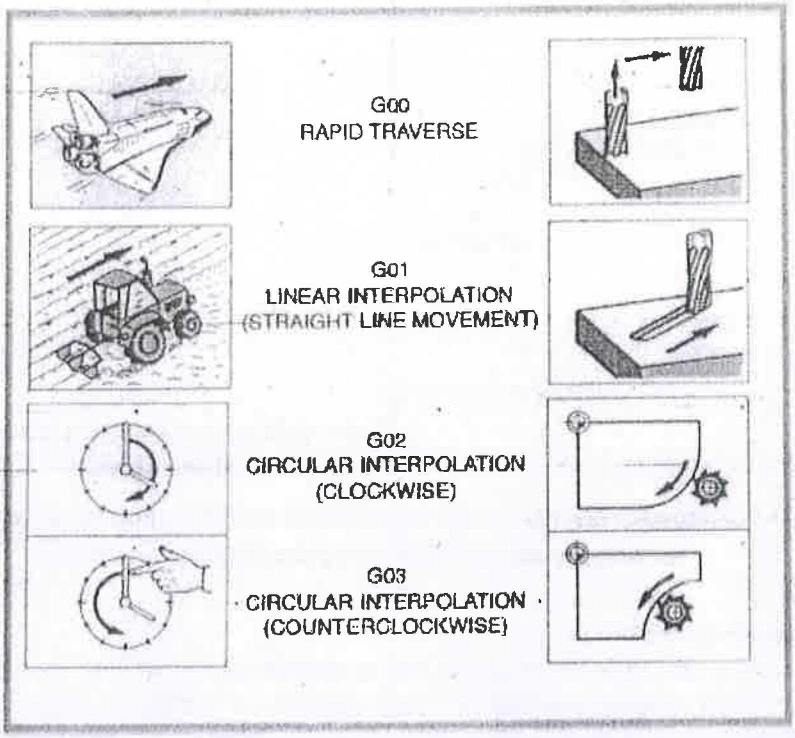


Fig. 13 The functions of a few common G-codes.

The G-codes are grouped into categories such as Group 01, containing codes G00, G01, G02, G03. Which cause some movement of the machine table or head. Group 03 includes either absolute or incremental programming, while Group 09 deals with canned cycles.

A G00 code rapidly positions the cutting tool while it is above the workpiece from one point to another point on a job. During the rapid traverse movement, either the X or Y axis can be moved individually or both axes can be moved at the same time. Although the rate of rapid travel varies from machine to machine, it ranges between 200 and 800 in./min (5 and 20 m/min).

The G01, G02, and G03 codes move the axes at a controlled feed rate.

- G01 used for straight – line movement. (Linear Interpolation)

- G02 (Clockwise) and G03 (Counter Clockwise) are used for arcs and circles. (Circular interpolation)

Group	Code	Function
01	G00	Rapid positioning
01	G01	Linear interpolation
01	G02	Circular interpolation clockwise (CW)
01	G03	Circular interpolation counterclockwise (CCW)
06	G20*	Inch input (in.)
06	G21*	Metric input (mm)
	G24	Radius programming (**)
00	G28	Return to reference point
00	G29	Return from reference point
	G32	Thread cutting (**)
07	G40	Cutter compensation cancel
07	G41	Cutter compensation left
07	G42	Cutter compensation right
08	G43	Tool length compensation positive (+) direction
08	G44	Tool length compensation minus (-) direction
08	G49	Tool length compensation cancel
	G84	Canned turning cycle (**)
03	G90	Absolute programming
03	G91	Incremental programming

(*) - on some machines and controls, these may be G70 (inch) and G71 (metric)

(**) - refers only to CNC lathes and turning centers.

Fig. 14. Some of the most common G codes used in CNC programming.

M or miscellaneous codes are used to either turn ON or OFF different functions which control certain machine tool operations, Fig. 15.

M-codes are not grouped into categories, although several codes may control the same type of operations such as M03, M04, and M05 which control the machine tool spindle.

- M03 turns the spindle rotation in clockwise
- M04 turns the spindle rotation in counter clockwise
- M05 turns the spindle rotation stop.

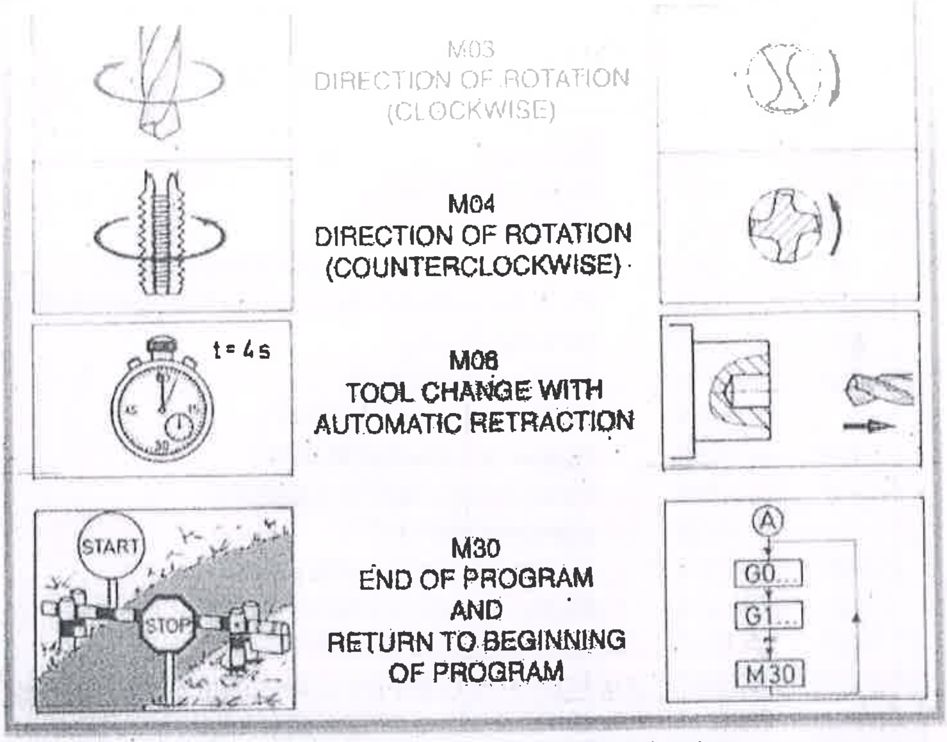


Fig. 15 The functions of a new common M codes.

Code	Function
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping (**)
M11	Chuck - unclamping (**)
M12	Tailstock spindle out (**)
M13	Tailstock spindle in (**)
M17	Toolpost rotation normal (**)
M18	Toolpost rotation reverse (**)
M30	End of tape and rewind
M98	Transfer to subprogram
M99	End of subprogram

(**) - refers only to CNC lathes and turning centers.

Fig. 14. Some of the most common G codes used in CNC programming.

3.6 Block of Information

CNC information is generally programmed in blocks of five words. Each word conforms to the EIA standards and they are written on a horizontal line. If five complete words are not included in each block, the machine control unit (MCU) will not recognize the information, therefore the control unit will not be activated.

Using the example shown in Fig. 17, the five words are as follows:

N001	represents the sequence number of the operation.
G01	represents linear interpolation
X12345	will move the table 1.2345 in. in a positive direction along the X axis.
Y06789	will move the table 0.6789 in. along the Y axis.
M03	Spindle on CW.

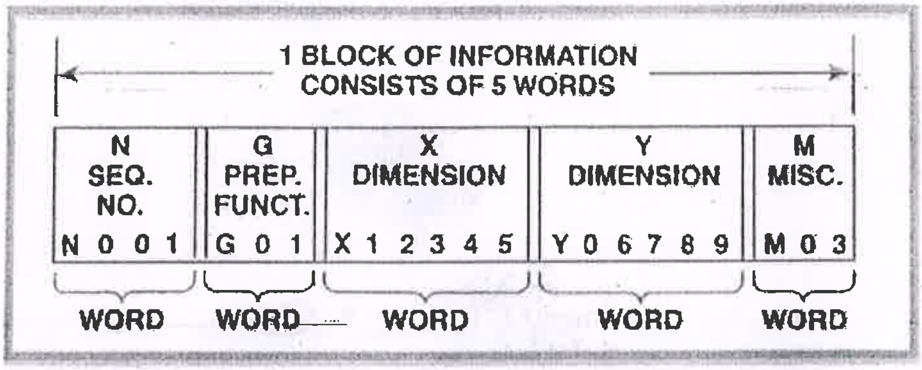


Fig. 17. A complete block of information consists of five words.

4. Programming for Positioning

Before starting to program a job, it is important to become familiar with the part to be produced. From the engineering drawings, the programmer should be capable of planning the machining sequences required to produce the part. Visual concepts must be put into a written manuscript as the first step in developing a part program, Fig. 18. It is the part program that will be sent to the machine control unit by the computer, tape, diskette, or other input media.

The programmer must first establish a reference point for aligning the workpiece and the machine tool for programming purposes. The manuscript must include this along with the types of cutting tools and work holding devices required, and where they are to be located.

4.1 Dimensioning Guidelines

The system of rectangular coordinates is very important to the successful operation of CNC machines. Certain guidelines should be observed when dimensioning parts for CNC machining. The following guidelines will insure that the dimensioning language means exactly the same thing to the design engineer, the technician, the programmer, and the machine operator.

1. Define part surfaces from three perpendicular reference planes.
2. Establish reference planes along part surfaces which are parallel to the machine axes.

3. Dimension from a specific point on the part surface.
4. Dimension the part clearly so that its shape can be understood without making mathematical calculations or guesses.
5. Define the part so that a computer numerical control cutter path can be easily programmed.

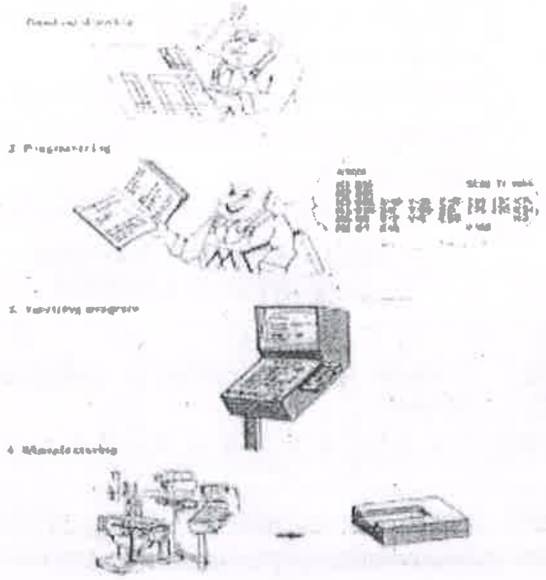
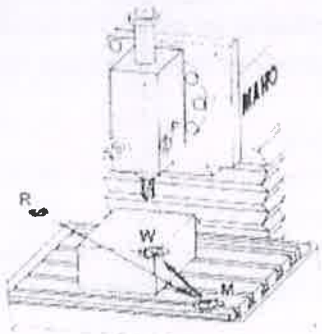


Fig. 18. Steps involved in Part Programming

4.2 Machine Zero Point

The machine zero point can be set by three methods by the operator, manually by a programmed absolute zero shift, or by work coordinates, to suit the holding fixture or the part to be machined.

MANUAL SETTING - The operator can use the MCU controls to locate the spindle over the desired part zero and then set the X and Y coordinate registers on the console to zero.



R = Reference point (maximum travel of machine)
M = Machine zero point (X0,Y0,Z0) of machine coordinate system
W = Part zero point workpiece coordinate system.

Under G54 ... G59 the actual machine coordinates of part zero are stored in the stored zero offsets memory and activated in the part program.

Under G92 the actual machine coordinates are inserted and used on the G92 line of the part program.

ABSOLUTE ZERO SHIFT - The absolute zero shift can change the position of the coordinate system by a command in the CNC program. The programmer first sends the machine spindle to home zero position by a G28 command in the program. Then another command (G92 for absolute zero shift) tells the MCU how far from the home zero location, the coordinate system origin is to be positioned.

The sample commands may be as follows:

N1 G28 X0 Y0 Z0 (sends spindle to home zero position)

N2 G92 X4.000 Y5.000 Z6.000 (the position the machine will reference as part zero)

5. Work Settings and Offsets

All CNC machine tools require some form of work setting, tool setting, and offsets (compensation) to place the cutter and work in the proper relationship. Compensation allows the programmer to make adjustments for unexpected tooling and setup conditions.

5.1 Work Coordinates

In absolute positioning, work coordinates are generally set on one edge or corner of a part and all programming is generally taken from this position. In Fig. 20, the part zero is used for all positioning for hole locations 1, 2, and 3.

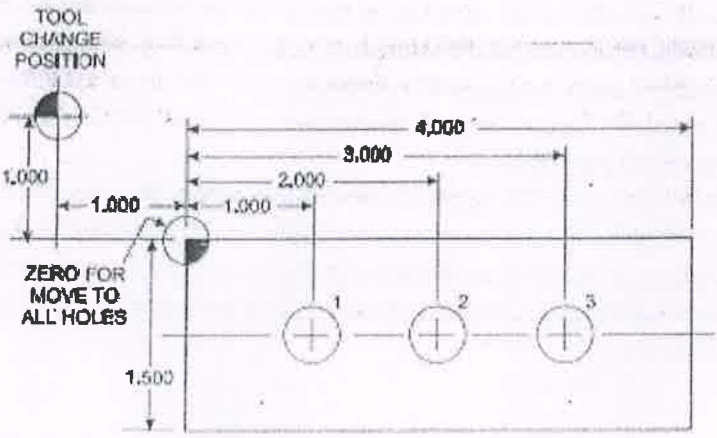


Fig. 20 In absolute programming, all dimensions must be taken from XY zero at the top left-hand corner of the part.

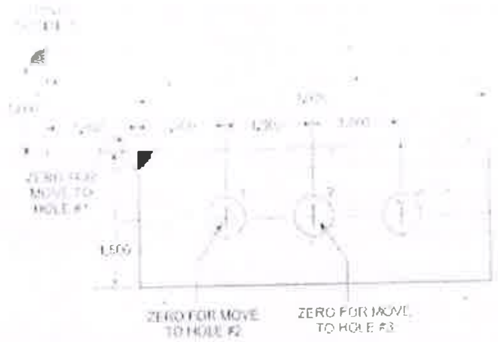


Fig. 20 In incremental programming, all dimensions are taken from the previous point.

In incremental positioning, the work coordinates change because each location is the zero point for the move to the next location, Fig. 21. On some parts, it may be desirable to change from absolute to incremental, or vice versa, at certain points in the job. Inserting the G90 (absolute) or the G91 (incremental) command into the pro- gram at the point where the change is to be made can do this.

5.2 R Plane or Gage Height

~~The~~ The word-address letter R refers to a partial retraction ~~point~~ in the Z axis to which the end of the cutter retracts above the work surface to allow safe table movement in the X Y axes. It is often called the rapid-traverse distance, gage height, retract or work plane. The R distance is a specific height or distance above the work surface and is generally .100 in. above the highest surface of the workpiece, ~~Fig.~~ Fig. 22, which is also known as gage height. Some manufacturers build a gage height distance of .100 in. into the MCU (machine control unit) and whenever the feed motion in the Z axis is called for, .100 in. will automatically be added to the depth programmed.

When setting up cutting tools, the operator generally places a .100 in. thick gage on top of the highest surface of the workpiece. Each tool is lowered until it just touches the gage surface and then its length is recorded on the tool list. Once the gage height has been set, it is not generally necessary to add the .100 in. to any future depth dimensions since most MCUs do this automatically.

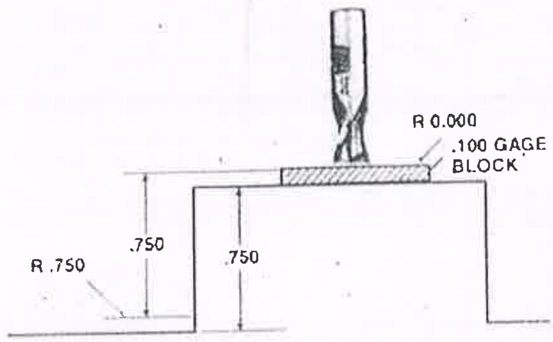


Fig. 22 using a .100 in. Gage block to set the gage height or R0 on work surface.

5.3 Cutter Diameter Compensation

Cutter diameter compensation (CDC) changes a milling cutter's programmed center line path to compensate for small differences in cutter diameter. On most MCUs, it is effective for most cuts made using either linear or circular interpolation in the X-Y axis, but does not affect the programmed Z-axis moves. Usually compensation is in increments of .0001 in. up to +1.0000 in., and usually most controls have as many CDCs available as there are tool pockets in the tool storage matrix.

The advantage of the CDC feature is that it:

1. Allows the use of cutters that have been sharpened to a smaller diameter.
2. Permits the use of a larger or smaller tool already in the machine's storage matrix.
3. Allows backing the tool away when roughing cuts are required due to excessive material present.
4. Permits compensation for unexpected tool or part deflection, if the deflection is constant throughout the programmed path.

The basic reference point of the machine tool is never at the cutting edge of a milling cutter, but at some point on its periphery. If a 1.000 in. diameter end mill is used to machine the edges of a workpiece, the programme would have to keep a .500 in. offset from the work surface in order to cut the edges accurately, Fig. 23. The .500 offset represents the distance from the centerline of the cutter or machine spindle to the edge of the part. Whenever a part is being machined, the programmer must calculate an offset path, which is usually half the cutter diameter.

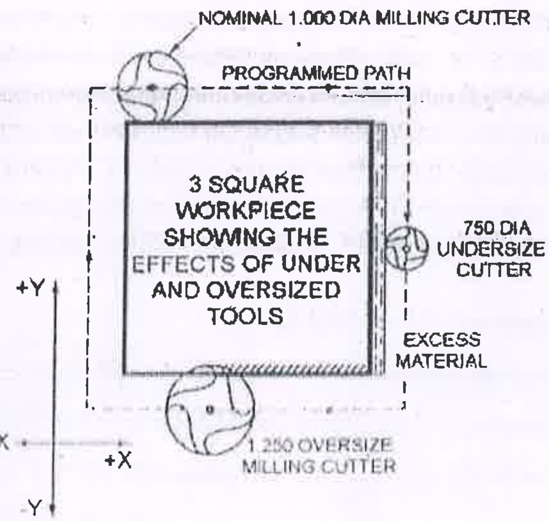


Fig. 23. Cutter-diameter compensation must be used when machining with various size cutters.

Modern MCUs, which have part surface programming, automatically calculate center line offsets once the diameter of the cutter for each operation is programmed. Many MCUs have operator-entry capabilities which can compensate for differences in cutter diameters; therefore an oversize cutter, or one that has been sharpened, can be used as long as the compensation value for oversize or undersize cutters is entered.

6. CNC Bench-Top Milling and Turning Centers

Bench-top teaching machines are well suited for teaching purposes because neither the student nor the teachers are intimated by the size or complexity of the machines. They are easy to program and perform machining operations similar to industrial machines with smaller work piece and lighter cuts. Bench-top machines are relatively inexpensive and ideal for teaching basic CNC programming.

Vertical machining centers and turning centers are the most common CNC machines used in industry. For teaching purposes, two types of CNC Bench-Top machines, the lathe and the mill, will be used because they use the same basic programming features and the Fanuc compatible controls as industrial machines. Most of the G and M codes are the same for CNC Bench-top teaching machines and industrial machines. Since programming codes do vary slightly with manufacturers, it is always wise to consult the programming manual for each specific machine to avoid crashes or scrap work.

The 3-axes bench-top CNC vertical machining center (mill) with the Fanuc compatible controller, is ideal for teaching the basics of CNC mill programming. It includes all important G and M codes, milling cycles, subroutines, etc. and can be programmed in inch or metric dimensions in both incremental and absolute programming. Some models are equipped with a graphics display that allows the operator to test-run the program on the computer screen without cutting a part. This is a safe way to check the accuracy of a program, to prevent crashes and scrap work, without actually running the machine.

The CNC Bench-Top turning center (lathe), is excellent for teaching the basics of CNC lathe programming. It uses the same standard G and M codes as the larger machines, can be programmed in inch or metric dimensions in both absolute and incremental programming. Many teaching machines also are equipped with canned cycle processing and canned thread-cutting cycles. Some models are equipped with a graphic display that allows a student to simulate (test run) the cutting action of the CNC program on the computer screen without actually cutting a part on the machine. This allows the student to check the program for accuracy and make corrections which avoid machine crashes, damage, and scrap parts.

6.1 CNC Programming Hints – MILLING

- Reference point (maximum travel of machine)
- Machine X Y zero point (could be tool change point)
- Part X Y zero point (programming start point)
- Indicates the tool change position. A G92 code will reset the axis register position coordinates to this position.

For a program to run on a machine, it must contain the following codes:

M03 To start the spindle/cutter revolving.

Sxxx The spindle speed code to set the r/min.

Fxx The feed rate code to move the cutting tool or work piece to the desired position.

ANGLES: The X Y coordinates of the start point and end point of the angular surface plus a feed rate (F) are required.

6.2 CNC Programming Hints – TURNING

Indicates the X Z 0 (zero) location which is the starting point for programming.

Indicates the tool-change position.
A G92 code will reset the axis register position coordinates to this position.

For a program to run on a machine, it must contain the following codes:

M03 To start the spindle/cutter revolving.

Sxxx The spindle speed code to set the r/min.

Fxx The feed rate code to move the cutting tool or work piece to the desired position.

TAPERS/BEVELS/ANGLES

The X Z coordinates of the small diameter, the large diameter, and a feed rate must be programmed.

The Z moves the cutting tool longitudinally away from the end of the work piece.

The Z moves the cutting tool along the length of the work piece towards the chuck (head stock).

X moves the cutting tool away from the work diameter.

X moves the cutting tool into the work diameter.

6.3 Fanuc Compatible Programming

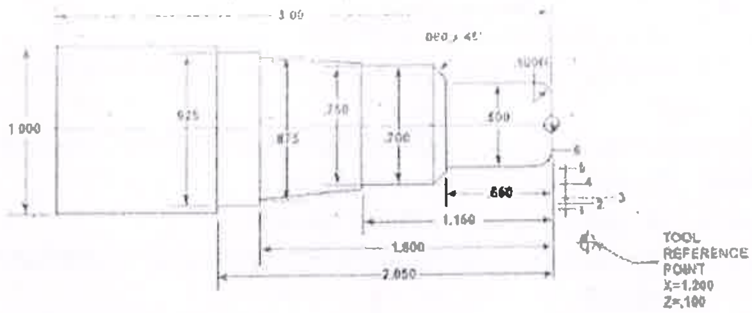
The programming for the Fanuc compatible control is the one most commonly used in industry. Although many controls are similar to the Fanuc control, there are some differences. A few of the main differences are:

1. The G28 code is used to set the programmed offset of the reference point.
2. Codes are modal and do not have to be repeated in every sequence line.
3. All dimensions are entered as decimals.

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

Using the part illustrated in Fig. the programming for a Fanuc compatible control would be as follows:



Turning Programming:

Programming Sequence:

- N05 G20 G90 G40
  - G20 inch data input.
  - G90 absolute positioning mode
  - G40 cancels tool radius compensation.
- N10 G95 G96 S2000 M03
  - G95 feed rate per revolution.
  - G96 constant feed rate.
  - S2000 spindle speed set at 2000 r/min.
  - M03 spindle ON clockwise.
- N15 T0202
  - tool number and offsets.
- N20 G00 X1.200 Z.100
  - G00 rapid traverse mode.
  - X&Z tool reference or change point.
  - X1.200 tool point .100 away from the outside diameter.
  - Z.100 tool point .100 to the right of end of work.

Rough Turning Cycle:

- N25 G73 U.05 R.05
  - G73 rough turning cycle.
  - U.05 .050 allowance on diameter for finish cut.
  - R.05 tool nose radius.
- N30 G73 P35 Q95 U.025 W.005 F.008
  - P35 start block of rough contour cycle.
  - Q95 end block of rough contour cycle.
  - W.005 shoulder allowance for finish cut.
  - F.008 feed rate at .008 per revolution.
- N35 G00 X.300 Z.050
  - G00 rapid traverse mode.
  - X.300 tool point at .300 diameter for start of .100 radius.
  - Z.050 tool point .050 away from end of the part.

SSM Institute of Engineering and Technology DINDIGUL-624 002

Dr.D.SENTHIL KUMARAN. M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology

N40 G01 Z0

G01 linear interpolation (feed).

Z0 tool point touching end of the work.

N45 G03 X.500 Z-.100 R.100

G03 circular interpolation (counter clockwise).

X.500 largest diameter of radius.

Z-100 end of radius on .500 diameter.

R.100 size of the radius.

N50 G01 Z-.650

G01 linear interpolation.

Z-.650 machines .500 diameter to .650 length.

N55 X.580

X.580 tool moves out to the small diameter of .060 x 45° bevel.

N60 X.700 Z-.710

X.700 large diameter of bevel.

Z-.710 end distance of bevel.

N65 Z-1.150

Z-1.150 the .700 diameter cut to 1.150 length.

N70 X.750

X.750 cutting tool feeds out to .750 (small end of taper).

N75 X.875 Z-1.800 (cutting taper)

X.875 large end of taper.

Z-1.800 length that taper is cut.

N80 X.925

X.925 tool feeds out (faces) to .925 diameter.

N85 Z-2.050

Z-2.050 the .925 diameter is cut to 2.050 length.

N90 X1.050

X1.050 the tool is fed out to .050 past the diameter of the part.

N95 G00 X1.200 Z.100 (tool back to tool reference point)

G00 rapid traverse mode.

X1.200 & Z.100 (reference point positions).

Finish Turning:

N100 G72 P35 Q95 F.005

G72 finish turn cycle.

F.005 feed rate .005 per revolution.

N105 G00 X2.000 Z.500

G00 rapid traverse mode.

X2.000 & Z.500 machine home position.

N110 M30

M30 end of program

CNC LATHE AND MILLING MACHINES

AIM:

To study about the functional parts and working principle of the CNC Lathe and Milling machine.

INTRODUCTION

Automation is a technology concerned with the application of mechanical, electronic and computer – based systems to operate and control production.

COMPUTER NUMERICAL CONTROL (CNC)

CNC stands for computer numerical control. It is a machine controlled by a computer. Its external appearance is similar to that of a NC machine. Tape or Computer Keyboard or Tutor Keyboard is used as input media for CNC machines. For NC machines tape is to be fit repeat to produce repeated jobs. But for CNC machine tape is fit once and the program is stored in the memory and can be run repeat to produce repeated jobs.

CNC SYSTEM COMPONENTS

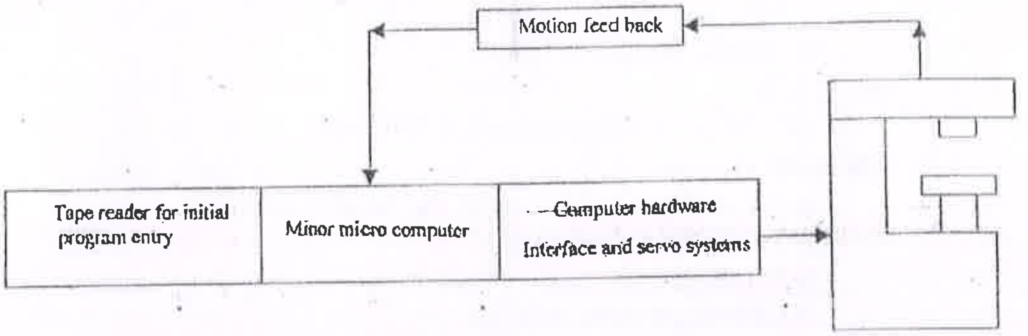


Fig .components of CNC system

WORKING PRINCIPLE OF CNC MACHINES

It contains two distinct controls, one is CNC controller which doses the function of program decoding interpolation, diagnostics machine actuation etc. Another is programmable logic controller (PLC), which dose spindle on off, coolant on off, turret operation etc.

The slides are moved by their own feed drive (AC or DC) servomotors or through balls screw and nut drive. The feed drive controllers the feed drive motors. Suitable transducers fitted either to the table or motor measure the slide position. Also the position is monitored and checked through the feed back transducers to ensure the accuracy of positioning. Spindle is provided with stepped motors of AC or DC. A suitable control is used to vary is speed of the spindle motor. A suitable fed back device attached to the shaft monitors the speed.

Features of CNC machines

- Part program input may be through keyboard.
- Part program is entered in to the computer and stored in the memory. Then it is used again and again.
- The part program once entered can be edited for any errors or design changes.

- Graphical display of the cutter path and shape of the finished job before actually running the program is possible (simulation).
- Tool wear compensation is possible.
- Able to get machine utilization information's like number of components produced, time per component, time for setting the job etc.,
- Sub-program facility for repetitive machining sequences also possible.

Types of CNC machine tools

- CNC Lathe
- CNC Milling Machines.
- CNC Turning Centers.
- CNC Vertical Machining centers.
- CNC Horizontal Machining centers.
- CNC Electrical Discharge Machine.
- CNC wire cut EDM.
- CNC Grinding Machine.

CNC TURNING CENTERS / CNC LATHES

The CNC turning center s is a machine tool capable of performing various turning and related operations, on work piece in one set up under CNC system. These are generally provided with two – axis control, z-axis parallel to the spindle and X-axis perpendicular to spindle axis.

Turning centers are provided with a slant bed to allow for better view of the machining plane as well as for easy placement of the various devices involved in the machine zone. It also provided with a index able tool turret which can hold 8, 12, or 16 tools of various types.

The controllable components of a simple CNC lathe are

- Spindle rotation
- Feed drive
- Work piece clamping device
- Automatic tool changer (ATC)

The other types of popular CNC machines and are generally classified as:

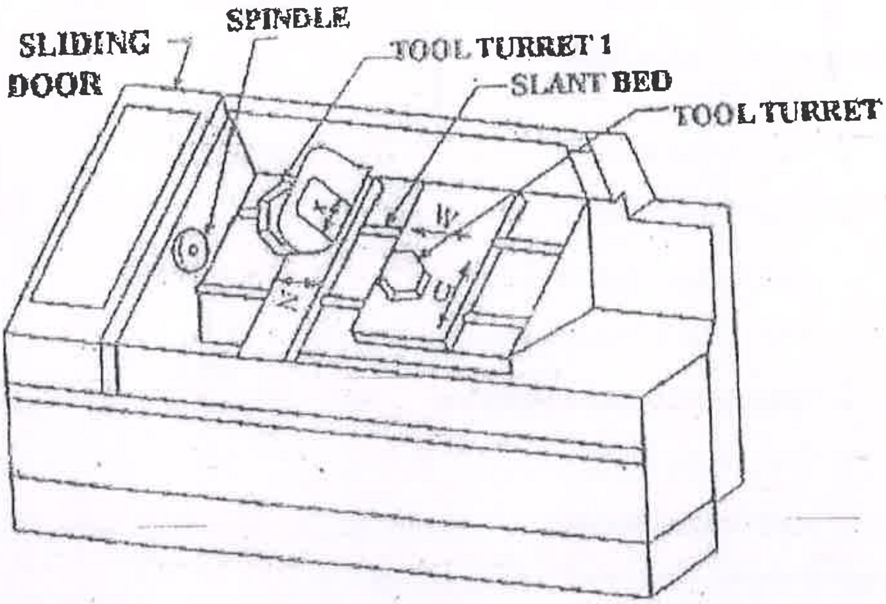
1. Horizontal machines
2. Vertical machines

SSM Institute of Engineering and Technology DINDIGUL-624 002

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

Classification of the Horizontal machines

1. Chucking machines
2. Shaft machines
3. Universal machines



CNC MACHINING CENTERS

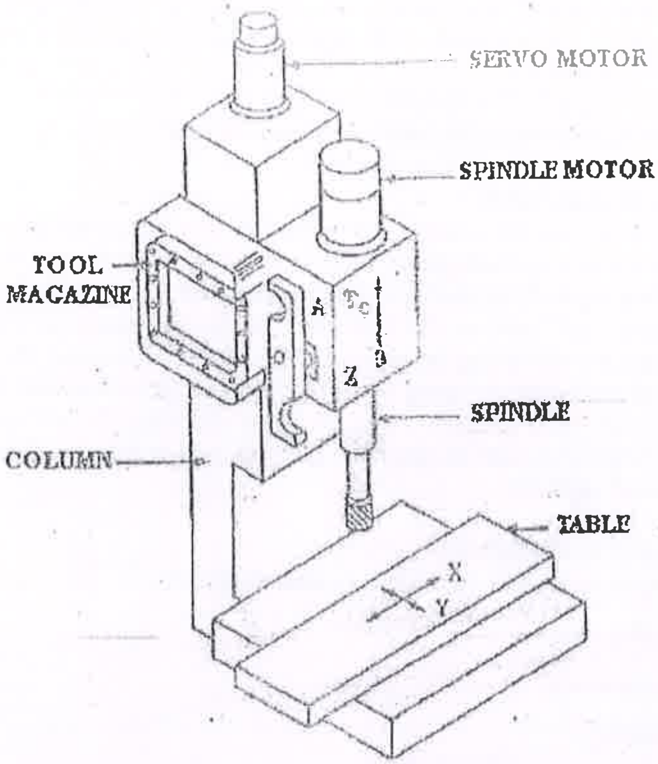
The CNC machining center is a machine tool capable of performing multiple machining operations on work piece in one set up under CNC system. Typical machining operations performed on machining center include milling, boring, reaming, and tapping.

Features of machining centre

CNC machining centers are usually equipped with the following features to reduce non-productive time.

Automatic tool changing

A variety of machining operations require a variety of cutting tool. The tools are stored in a tool magazine that is integrated with the machine tool. When a cutter need to be changed, the tool drum rotates to the proper position. An automatic tool changer (ATC) replaces the tool in the spindle with required tool in the tool magazine. The ATC operates under part program control. The capacity of tool magazine ranges from 16 to 80 cutting tools.



Automatic work piece positioning

Many horizontal and vertical machining centers have the capability to orient the work piece relative to the spindle. This is achieved by means of a rotary table on which the work piece is fixed. The table can be oriented at any angle about a vertical axis to permit the cutting tool for machining almost the entire surface of the work piece in single setup.

Automatic pallet changer

Machining centers are generally equipped with two or more separate pallets. A pallet may be considered as a small table having standard dimensions. While machining is performed on the work piece with one pallet in position at the machine, the other pallets are in a safe location away from the spindle. In this safe location the operator can unload the finished part from the prior cycle and then fix raw work piece for the next cycle, thus not disturbing the machining operation. An automatic pallet changer is used to move these pallets.

Machining centers are classified as follows:

- Horizontal machining centers
- Vertical machining centers
- Universal machining centers

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

Vertical milling machines

Vertical milling machining centers are also a bed type machine with 1. Single spindle and auto tool changers 2. Multi Spindle with turret head (turret machining centers). The structural configuration is as follows:

X-axis traverse provided by table or column.

Y-axis traverse provided by saddle or column or ram.

Z-axis traverse provided by the head stoke.

Horizontal machining centers

A horizontal machining center has its spindle on a horizontal axis. These machines are used for machining heavier work pieces with large metal removal rates. So it requires large and heavier tools. As a result, these machines are provided with heavier tool magazines.

The rotary table used in the horizontal machining center provides both axis. These machines are used for machining the prismatic (box like) components. The availability or rotary table makes it possible for machining of all four faces of the component in a single setup. The rotary table can also have more than one axis rotation capability. If such rotary table is interfaced with a conventional three z axis horizontal machining center, then it will be possible to machine complex sculptured surfaces.

CNC PART PROGRAMMING

The basics steps involved in an NC / CNC programming are

- Process planning
- Part programming
- Program entry
- Program verification or simulation
- Production or machining

Data required for programming

To To write a part programming for CNC machines the following data are required

- Machine tool specification
- Cutting tool specification
- Work measurements
- Speed, feed details
- Sequence of operation

CNC COORDINATE SYSTEM

Machining of a work piece by a CNC program requires a coordinate system to be applied to the machine tool. As all machine tools have more than one slide, it is important that each slide is identified individually. These are three planes in which movement can take place

- Longitudinal

- Vertical
- Transverse

ZERO POINT AND REFERENCE POINT

It is the origin point of the co-ordinate system of the NC machine tool. With respect to this origin point programmer decide the tool position and movements.

Fixed zero (M)

It is the zero point of the CNC machine tool fixed by the manufacturer. It is always located the same position of the machine tool. For drilling, boring, and milling machines it is at the lower left corner of the machine table and for turning center it is on the spindle flange.

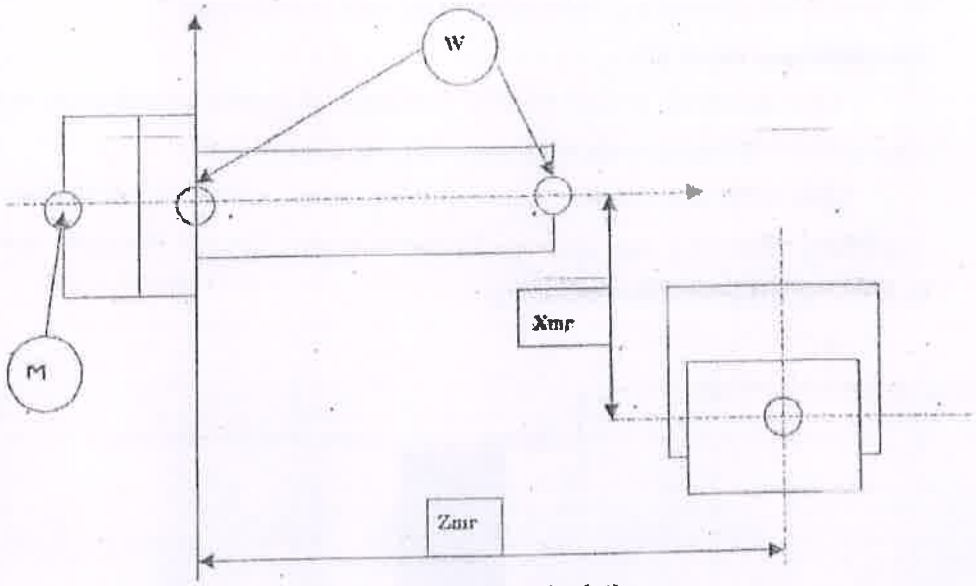


Fig. Co ordinate point for lathe

Floating zero (W)

It is the zero point of the NC machine tool set by the operator at the any position on the machine table / work piece.

Machine Zero Point (M)

This point was specified by the manufacturer of the machine. This is zero point for the co -ordinate systems and reference points in the machine. The machine zero point can be the center of the table or a point along the edge of the traverse range. The position of the machine zero point generally varies from manufacturer to manufacturer. It is also called as home position.

Reference Point (R)

This point serves for controlling the measuring system of the slides and tool traverse. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches.

Work Piece Zero Point (W)

This point determines the work piece co- ordinate system in relation to the CNC system when setting up the machine.

For turned parts, the work piece zero point should be placed along the spindle axis in line into the right hand or left hand end face of the finished contour. For milled parts its generally advisable to use and extreme corner points of the work piece zero points.

Tool Reference Points (T)

When machining a work piece, it is essential to control the tool point or tool cutting edges precise relationship to the work piece along the machine path.

Since tools have different shapes and dimensions, precise tool dimensions have to be establishing before hand and input into the control system. The tool dimension is related to the fixed tool setting point during pre setting.

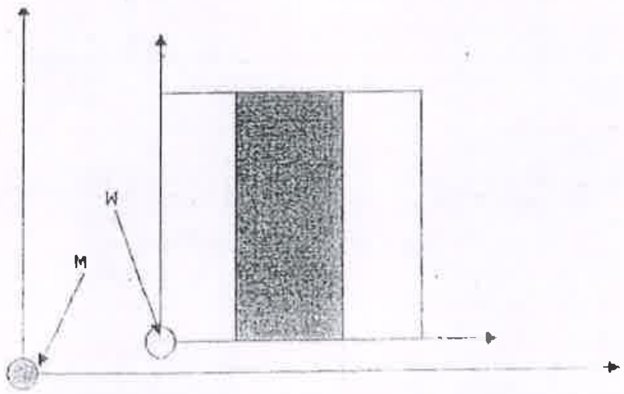


Fig. Co ordinate point for Milling

RESULT:

Thus we have studied about the CNC Lathe and Milling Machines.

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

AIM:

To study of international standards G - codes and M- codes

M codes for CNC Milling and Lathe machine operation:

MILLING			LATHE	
S.No	CODES	DESCRIPTION	CODES	DESCRIPTION
1.	M00	Programming stop	M00	Programming stop
2.	M02	Program end	M01	Optional stop
3.	M03	Spindle clockwise	M02	Program end
4.	M04	Spindle counter clockwise	M03	Spindle clockwise
5.	M05	Spindle stop	M04	Spindle counter clockwise
6.	M06	Tool change	M05	Spindle stop
7.	M08	Coolant on	M06	Tool change
8.	M09	Coolant off	M08	Coolant on
9.	M10	Work Clamp Open	M09	Coolant off
10.	M11	Work Clamp Close	M10	Vice open
11.	M13	Coolant ON CW	M11	Vise close
12.	M14	Coolant ON CCW	M13	Programmed end rewind
13.	M19	Spindle Angular Position	M98	Sub program call
14.	M20	ATC Moves towards spindle	M99	Sub program exist
15.	M21	ATC Move away from spindle		
16.	M22	ATC Arm Down		
17.	M23	ATC Arm Up		
18.	M24	Tool Release from ATC		
19.	M25	Tool clamps by ATC		
20.	M30	Program Reset and rewind		
21.	M32	ATC Rotates CW		
22.	M33	ATC Rotates CCW		
23.	M70	X Mirror ON		
24.	M71	Y Mirror ON		
25.	M80	X Mirror OFF		
26.	M81	Y Mirror OFF		
27.	M98	Sub program call		
28.	M99	Sub program exist		

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN. M.E., Ph.D., (NUS)
SSM Institute of ...

G Codes for CNC lathe operation programming:

SL.NO	CODES	DESCRIPTION	SYNTAX
1.	G00	Fast Traverse	G00 X Z
2.	G01	Linear Interpolation	G01 X F
3.	G02	Clockwise Circular Interpolation	G02 X Z R F or G03 U W R F
4.	G03	Counter Clockwise Circular Interpolation	G03 X Z R F or G02 U W R F
5.	G04	Dwell	G04 X or G04 U or G04 P
6.	G20	Inch Data Input	G20
7.	G21	Metric Data Input	G21
8.	G28	Reference Point Return	G28 U0 W0
9.	G40	Cutter Compensation Cancel	G40
10.	G41	Cutter Compensation Left	G41
11.	G42	Cutter Compensation Right	G42
12.	G70	Finishing Cycle	G70 P Q F
13.	G71	Multiple Rough Turning Cycle	G71 U R G71 P Q U W F S T
14.	G72	Multiple Facing Cycle	G72 U R G72 P Q U W F S T
15.	G73	Pattern Repeating Cycle	G73 U W R G73 P Q U W F
16.	G74	Peck Drilling Cycle	G74 R G74 Z Q F
17.	G75	Grooving Cycle	G75 R G75 X Z P Q R F
18.	G76	Thread Cutting Cycle	G76 Pnpa Q R G76 X Z R P Q F n-No. of passes, p- pull out angle, a- angle of thread
19.	G90	Cutting Cycle A (Box Turning)	G90 X Z F –Turning G90 X Z R F Taper Turning
20.	G92	Single Thread Cutting Cycle	G92 X Z F (F- pitch)
21.	G94	Cutting Cycle B (Box Facing)	G94 X Z F – Facing G94 X Z R F – Taper Facing
22.	G98	Feed Per Minute (Mm/Minute)	G98
23.	G99	Feed Per Revolution (Mm/Rev)	G99

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

G-CODES SYNTAX EXPLANATION FOR LATHE

Operation	Syntax	Description
Fast Traverse	G00 X Z	X- diameter axis, Z- length axis
Linear Interpolation	G01 X F	X- diameter axis, Z- length axis
Clockwise Circular Interpolation	G02 X Z R F	X- diameter axis, Z- length axis, R- radius, F- feed
Counter Clockwise Circular Interpolation	G03 X Z R F	X- diameter axis, Z- length axis, R- radius, F- feed
Finishing Cycle	G71 P Q F	P- starting block No., Q- end block No, F- feed
Multiple Rough Turning Cycle	G71 U R G71 P Q U W F S T	U- depth of cut, R- Relief amount, P- starting block no., Q- end block no., U- finishing allowance in X axis, W- finishing allowance in Z axis, S- speed, T- tool no.
Multiple Facing Cycle	G72 U R G72 P Q U W F S T	U- depth of cut, R- Relief amount, P- starting block no., Q- end block no., U- finishing allowance in X axis, W- finishing allowance in Z axis, S- speed, T- tool no.
Pattern Repeating Cycle	G73 U W R G73 P Q U W F	U,W- relief amount in X & Z axis, R- number of cut, P,Q- start and end number of program, U,W- finishing allowance in X & Z direction, F- feed.
Peck Drilling Cycle	G74 R G74 Z Q F	R- Return amount, Z- total depth, Q- depth of cut, F- feed.
Grooving Cycle	G75 R G75 X Z P Q R F	R- Return amount, X- total depth, Z- total width, P- peck increment, Q- stepping distance, F- feed.
Thread Cutting Cycle	G76 Pnpa Q R G76 X Z R P Q F	n-No. of passes, p- pull out angle, a- angle of thread, Q- min cutting depth, R- finishing allowance, X- minor dia, Z- end position of thread, P- height of thread, Q depth of first cut, F- pitch.
Cutting Cycle A (Box Turning)	G90 X Z F (plain) G90 X Z R F –Taper	X- diameter axis, Z- length axis. F- feed X- diameter axis, Z- length axis, F- feed R- incremental of cut.
Single Thread Cutting Cycle	G92 X Z F	X- depth of cut. Z- length of thread, F- pitch.

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E.,Ph.D.,(NUS)
Principal
SSM Institute of Engineering and Technology
[illegible]
[illegible], Dindigul - 624 002

G CODES FOR CNC MILLING PROGRAMMING

Sl.No	CODES	DESCRIPTION	SYNTAX
1.	G00	Fast Traverse	G00 X Y Z
2.	G01	Linear Interpolation	G01 X Y Z F
3.	G02	Clockwise Circular Interpolation	G02 X Y R F
4.	G03	Counter Clockwise Circular Interpolation	G03 X Y R F
5.	G04	Dwell	G04 X
6.	G20	Inch Data Input	G20
7.	G21	Metric Data Input	G21
8.	G28	Reference Point Return	G28 X0 Y0 Z0
9.	G40	Cancel Tool Radius Compensation	G40
10.	G41	Left Hand Tool Radius Compensation	G41
11.	G42	Right Hand Tool Radius Compensation	G42
12.	G73	High Speed Peck Drilling Cycle	G73 X Y Z Q R K F
13.	G74	Left Hand Tapping Cycle	G74 X Y Z Q R P K F
14.	G76	Fine Boring Cycle	G76 X Y Z P Q R K F
15.	G80	Cancel Canned Cycle	G80
16.	G81	Continuous Drilling Cycle	G81 X Y Z K R F
17.	G82	Continuous Drilling Cycle With Dwell	G82 X Y Z P K R F
18.	G83	Peck Drilling Cycle	G83 X Y Z Q R K F
19.	G84	Right Hand Tapping Cycle	G84 X Y Z K R P F
20.	G85	Boring Cycle With Feed Retraction	G85 X Y Z P K R F
21.	G86	Boring Cycle With Rapid Retraction	G86 X Y Z K R F
22.	G87	Back Boring Cycle	G87 X Y Z P K F
23.	G90	Absolute Coordinate System	G90
24.	G91	Incremental Coordinate System	G91
25.	G94	Feed Per Minute (Mm/Minute)	G94
26.	G95	Feed Per Revolution (Mm/Rev)	G95
27.	G170 G171	Circular Pocking	G170 R P Q X Y Z I J K G171 P S R F B J
28.	G172 G173	Rectangular Pocking	G172 I J K P Q R X Y Z K F G173 I K P T S R F B J Z

G- CODES SYNTAX EXPLANATION FOR MILLING

OPERATION	SYNTAX	DESCRIPTION
Fast Traverse	G00 X Y Z	X- length, Y- width, Z- depth
Linear Interpolation	G01 X Y Z F	X- length, Y- width, Z- depth, F- feed
Clockwise Circular Interpolation	G02 X Y R F	X- length, Y- width, R- radius, F- feed
Counter Clockwise Circular Interpolation	G03 X Y R F	X- length, Y- width, R- radius, F- feed
High Speed Peck Drilling Cycle	G73 X Y Z Q R K F	X, Y- drill position, Z- depth of hole, P- Dwell time, Q- depth of cut, R- incremental distance in Z direction, F- feed.
Left Hand Tapping Cycle	G74 X Y Z Q R P K F	X,Y- drill position, Z- depth of hole, R- incremental distance in Z direction, F- feed
Fine Boring Cycle	G76 X Y Z P Q R K F	X,Y- drill position, Z- depth of hole, Q- shift value, R- incremental distance in Z direction, F- feed
Continuous Drilling Cycle	G81 X Y Z K R F	
Peck Drilling Cycle	G83 X Y Z Q R K F	
Right Hand Tapping Cycle	G84 X Y Z K R P F	X,Y- tap position, Z- depth of hole, R- incremental distance in Z direction, F- feed
Boring Cycle With Feed Retraction	G85 X Y Z P K R F	X,Y- hole position, Z- depth of hole, R- incremental distance in Z direction, F- feed
Back Boring Cycle	G87 X Y Z P K F	X, Y- the next position to bore, Z- depth of hole, R- Z coordinates of the R point, K- number of repeats.
Circular Pocking	G170 R P Q X Y Z I J K	R- position of tool to start cycle, P- (0-roughing, 1- finishing), Q- peck increment each cut, X,Y,Z- coordinates of centers pocket, I- finishing allowance for side, J- pocket finishing allowance for pocket base, K- radius of circular pocket.
	G171 P S R F B J	P- % of cutter movement(50 or 75), S- roughing spindle speed, R- roughing feed in Z direction, F- roughing feed in X,Y direction, B- finishing spindle speed, J- finishing feed.
Rectangular Pocking	G172 I J K P Q R X Y Z K F	I-Length of pocket in X direction, J- length of pocket in Y direction, K- corner radius(always zero), P- (0-roughing, 1- finishing),Q- depth of cut for each pass, R- absolute depth from the Z surface, X,Y- pocket corner, Y,Z- absolute base of pocket
	G173 I K P T S R F B J Z	I-pocket side finish allowance, K- pocket base finish allowance, P- cutter width percentage(50 or 75), T- tool number, S- spindle speed in rpm, R- roughing feed in Z, F- roughing feed along X,Y, B- finishing spindle speed in rpm, J- finishing spindle feed, Z- safety Z position.

RESULT:

Thus the international standards G - codes and M- codes are studied

EX.NO: -   CIRCULAR INTERPOLATION   DATE:-

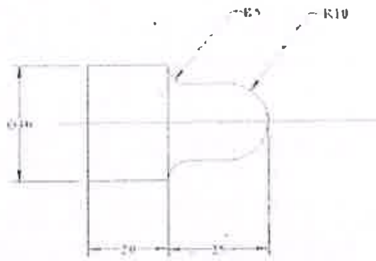
AIM:-

To develop a part program for Circular Interpolation and simulate in the software

SOFTWARE REQUIRED:

CNC Lathe Fanuc controlling simulation software.

PROGRAM:



CIRCULAR INTERPOLATION PROGRAM	**DESCRIPTIONS**
O2094	Program Number
[BILLET X30 Z60	Material size
G21 G98	Metric unit and feed in mm/rev
G28 U0 W0	Tool ref.point
M06 T0101	Turret select tool No 1
M03 S1500	Spindle speed 1500 rpm
G00 X30 Z1	Program ref point
G90 X28 Z-20 F40	Step turning dia 30 to28 for length20
X26	Step turning dia 28to26 for length20
X24	Step turning dia 26 to24 for length20
X22	Step turning dia 24 to22 for length20
X20	Step turning dia 22 to20 for length20
G01 X15 Z0	Tool moves to first circular cut ref point
G03 X20 Z-10 R10 F40	Ist Counter clock wise circular cutting R10
G01 X21 Z-10	Tool moves rapidly upward
G00 X21 Z0	Tool goes to ref point
G01 X10 Z0	Depth of cut
G03 X20 Z-10 R10 F40	IInd CCW circular cutting R10
G01 X21 Z-10	Tool moves rapidly upward
G00 X21 Z0	Tool goes to ref point
G01 X5 Z0	Depth of cut
G03 X20 Z-10 R10 F40	IIIrd CCW circular cutting R10
G01 X21 Z-10	Tool moves rapidly upward
G00 X21 Z0	Tool goes to ref point
G01 X0 Z0	Depth of cut
G03 X20 Z10 R10 F40	IVth CCW circular cutting R10
G01 X31 Z-10	Tool moves rapidly upward
G01 X31 Z-20	Tool moves toIInd circular cut ref point
G01 X25 Z-20	Depth of cut
G02 X30 Z-25 R10 F40	Ist CW circular cutting R10
G01 X31 Z-25	Tool moves rapidly upward
G00 X31 Z-20	Tool goes to ref point
G01 X20 Z-20	Depth of cut
G02 X30 Z-25 R10 F40	IInd CW circular cutting R10
G01 X31 Z-25	Tool moves rapidly upward
G00 X31 Z0	Tool goes to ref point
G28 U0 W0	Tool home
M05	Spindle off
M30	Program end

RESULT: Thus the part program for Circular Interpolation is developed and simulated in the software.

EX NO: - MULTIPLE TURNING DATE:

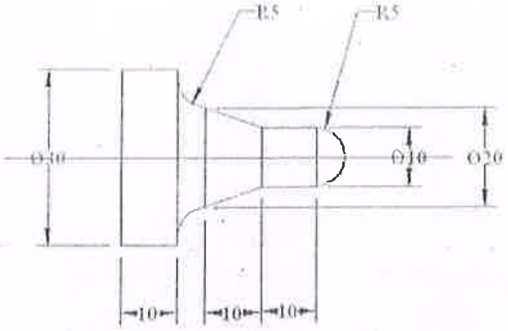
AIM:-

To develop a part program for Multiple Turning and simulated in the software.

SOFTWARE REQUIRED:

- CNC Lathe Fanuc controlling simulation software.

PROGRAM:



MULTIPLE TURNING	DESCRIPTIONS
O2095	Program Number
[BILLET X30 Z40	Material size
G21 G98	Metric unit and feed in mm/rev
G28 U0 W0	Tool ref.point
M06 T0101	Turret tool position1(Rough turning tool)
M03 S1800	Spindle speed 1500 rpm
G00 X31 Z0	Program ref point
G71 U2.0 R1.5	Multiple turning doc 2 & tool retraction1.5
G71 P10 Q50 U1.0 W1.0 F40	Multiple turning from point1 to point 5
N10 G01 X0 Z0	Tool moves to Start profile point
N20 G03 X10 Z-5 R5	Tool moves to 2nd profile point
N30 G01 X10 Z-15	Tool moves to 3rd profile point
N40 G01 X20 Z-25	Tool moves to 4th profile point
N50 G02 X30 Z-30 R5	Tool moves to 5th profile point
G28 U0 W0	Tool home
M05	Spindle off
M06 T0202	Turret tool position2(finishing tool)
M03 S1500	Spindle speed 1500 rpm
G00 X31 Z0	Program ref point
G70 P10 Q50 F40	Multiple turning finishing cycle from 1to5
G28 U0 W0	Tool home point
M05	Spindle off
M30	Program end

RESULT:

Thus the part program for Multiple Turning is developed and simulated in the software.

EX NO: - MIRRORING DATE:-

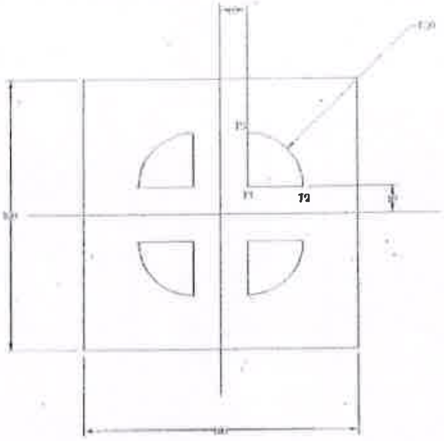
AIM:-

To develop a part program for Mirroring with subroutines and simulate in the software.

SOFTWARE REQUIRED:-

CNC Fanuc milling controlling simulation software.

PART DRAWING



PROGRAM:

MIRROR PROGRAMMING	DESCRIPTIONS
O6656	Program Number
[BILLET X100 Y100 Z10	Material size
G21 G94	Metric unit and feed in mm/rev
G91 G28	Tool ref.point
M06 T0101	ATC select tool No 1
M03 S1500	Spindle speed clock wise 1500 rpm
G90 G00 X15. Y15. Z5	Program start point
M98 P6666	Sub program calling for Ist operation
M70	X axis mirroring ON
M98 P6666	Sub program calling
M80	X axis mirroring OFF
M71	Y axis mirroring ON
M98 P6666	Sub program calling
M81	Y axis mirroring OFF
M70	X axis mirroring ON
M71	Y axis mirroring ON
M98 P6666	Sub program calling
M80	X axis mirroring OFF
M81	Y axis mirroring OFF
G91 G28 X0. Y0. Z0.	Tool goes to home point
M05	Spindle off
M30	Program end
O6666	Sub program Number
G90 G00 X15. Y15. Z5.	Tool goes to start point 1
G01 Z-1. F80.	Depth of cut given to 1mm
G01 X35. Y15.	Tool moves to start point 2
G03 X15. Y35. R20. F60.	Tool moves to start point 3
G01 X15. Y15. F80.	Tool comes to start point 1
G00 Z5.	Tool moves upward
G00 X0. Y0.	Tool goes to origin
M99	Sub program exit

RESULT: Thus the part program for Mirroring is developed and simulate in the software

EX NO: RECTANGULAR AND CIRCULAR POCKETING DATE:-

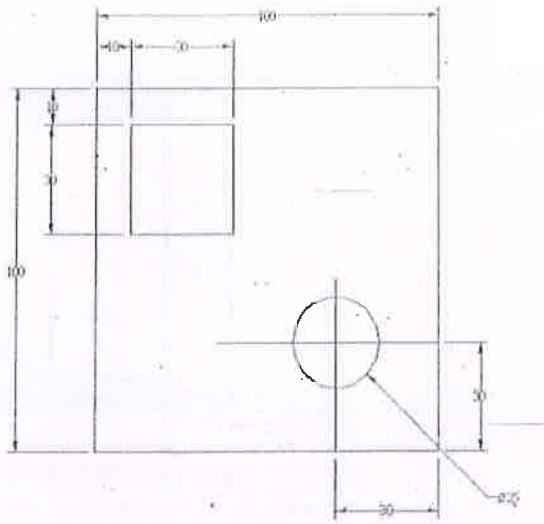
AIM:-

To develop a part program for Rectangular and Circular Pocketing and simulate in the software

SOFTWARE REQUIRED:

CNC Fanuc milling controlling simulation software.

PART DRAWING



PROGRAM:

POCKET PROGRAMMING	DESCRIPTIONS
O4633	Program Number
[BILLET X100 Y100 Z10	Material size
G21 G94	Metric unit and feed in mm/rev
G91 G28 X0. Y0. Z0.	Tool ref.point
M06 T01 01	ATC select tool No 1
M03 S1500	Spindle speed clock wise 1500 rpm
G90 G00 X0. Y0. Z5.	Program start point
G172 P0 Q1 R0. X10. Y60. Z-3. I30. J30. K0.	Rectangular pocketing rough cycle (G172 and G173)
G173 P75 S1500 R40 F80 B1500 J0.1 K0.1 T1 Z5	
G172 P1 Q1 R0. X10. Y60. Z-3. I30. J30. K0.	Rectangular pocketing finishing cycle (G172 and G173)
G173 P75 S1500 R40 F80 B1500 J0.1 K0.1 T1 Z5	
G170 P0 Q1 R0. X70. Y30. Z-5. I0.1 J0.1 K12.5	Circular pocketing rough cycle (G170 and G171)
G171 P75 S1200 R40 F80 B2000 J40	
G170 P1 Q1 R0. X70. Y30. Z-5. I0.1 J0.1 K12.5	Circular pocketing finishing cycle (G170 and G171)
G171 P75 S1200 R40 F80 B2000 J40	
G90 G00 Z5.	Tool moves upward
G91 G28 X0. Y0. Z0.	Tool goes to home point
M05	Spindle off
M30	Program end

RESULT:

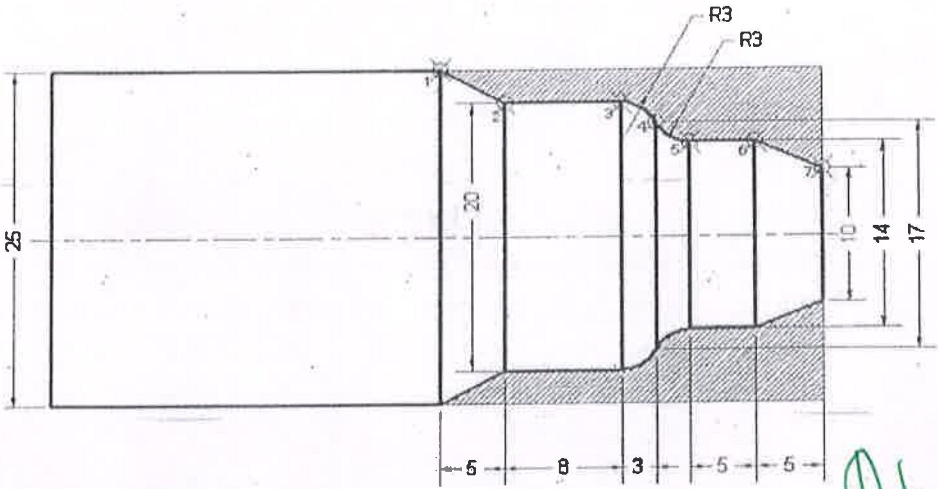
Thus the part program for Rectangular and Circular Pocketing is developed and simulate in the software.

EXERCISES:

Write down the CNC program for the following Diagram:

Write a CNC Part Program to manufacture the following object using standard G and M codes. Also check the tool path simulation using appropriate CAM software.

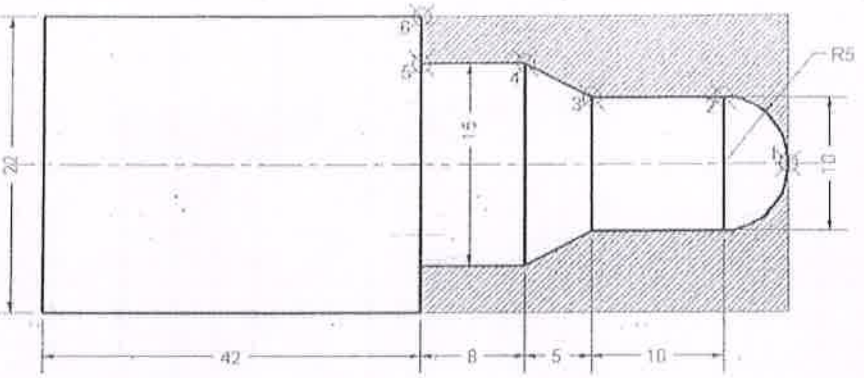
Multiple Facing Cycle: *Billet size: - ϕ 25 x 70 mm*



Dr.D.SENTHIL [illegible] M.E.,Ph.D.,(NUS)
SSM Institute of [illegible] and Technology
Kuttathupatti [illegible] (Po),
Palani Road, Dindigul - 624 002.

Write a CNC Part Program to manufacture the following object using standard G and M codes. Also check the tool path simulation using appropriate CAM software.

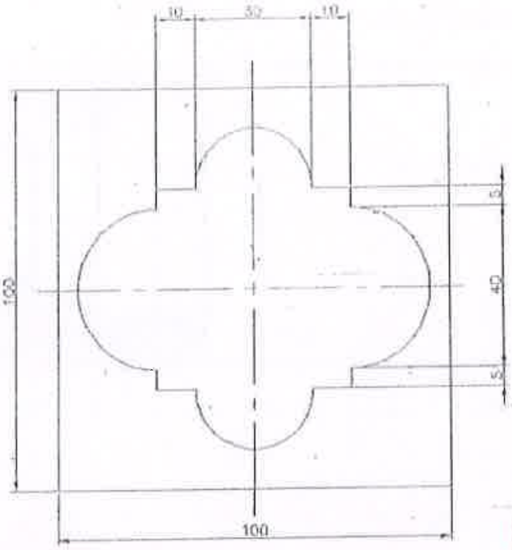
Multiple Turning Cycle: Billet size: - ϕ 22 x 65 mm



Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

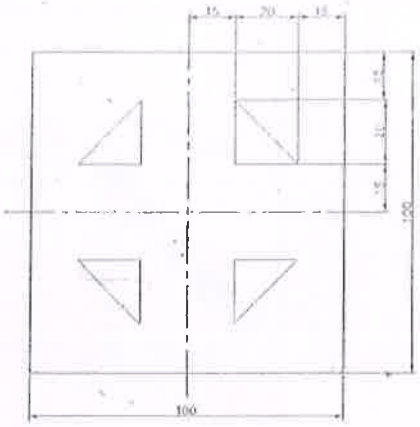
Write a CNC Part Program to manufacture the following object using standard G and M codes. Also check the tool path simulation using appropriate CAM software.

Simple Contour Milling: Billet size: - 100 x 100 x 10 mm; Depth of cut – 1 mm



Write a CNC Part Program to manufacture the following object using standard G and M codes. Also check the tool path simulation using appropriate CAM software.

Mirroring Cycle: Billet size:-100 x 100 x 10 mm, Depth of cut – 1 mm

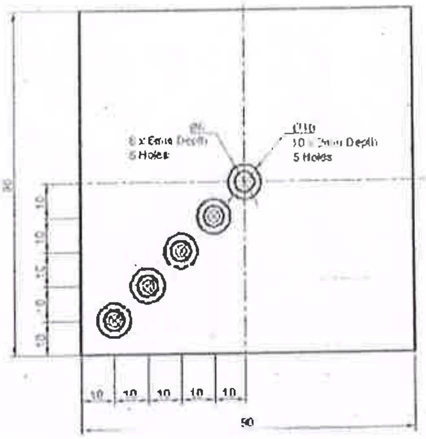


Dr.D.SENTHIL KUMARAN, M.E., Ph.D.,
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

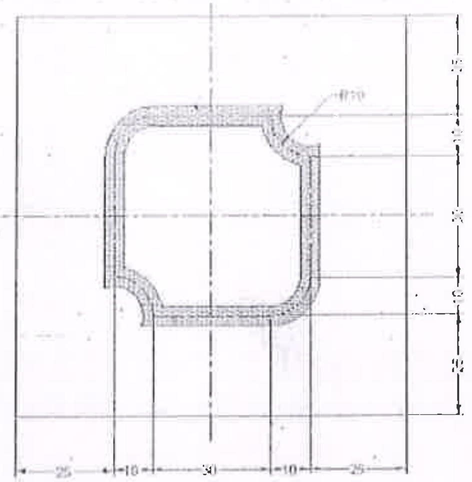
Write a CNC Part Program to [illegible] the following object using standard G and M codes.
Also check the tool path simulation using appropriate CAM software.

Drilling & Boring Using Incremental Co-ordinates: : Billet size : 100 x 100 x 10 mm
Drilling Tool : ϕ 6 Boring Tool : ϕ 10



Write a CNC Part Program to manufacture the following objects using standard G and M codes. Also check the tool path simulation using appropriate CAM software.

Simple Contour Milling: Billet size - 100 x 100 x 10 mm; Depth of cut = 1 mm



SSM Institute of Engineering and Technology
* DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul- Palani Highway, Dindigul – 624 002.

Department of Mechanical Engineering

Value Added Course (2018-2019) Odd Semester

Course Name : **CNC Programming**

Course Coordinators: Mr.Suresh.R& Mr.Ramasamy.V

MARKS STATEMENT FOR VALUE ADDED COURSE

S.No	Reg.No	Name of the Student	Marks Scored
1.	922117114003	AKTHARALI S	75
2.	922117114006	K.ARUL SELVAN	85
3.	922117114007	E.ARUNKUMAR	85
4.	922117114011	ARUN KUMAR M	90
5.	922117114012	ASHLEY SACHIN	75
6.	922117114028	GAJENDREN R	85
7.	922117114029	GOETHEM K	90
8.	922117114031	GUNAKARAN C	75
9.	922117114032	GUNA SEKAR. S	85
10.	922117114033	HARI HARAN. N	85
11.	922117114041	JAYA PRATHAP	85
12.	922117114044	JEROME F	90
13.	922117114053	MANICKAVEL V	75
14.	922117114054	MANIKANDAN P	85
15.	922117114055	MANIKANDARAJA M	85
16.	922117114058	MATHANRAJ G	90
17.	922117114059	MATHESHKUMAR B	90
18.	922117114064	MONISHKUMAR M	85
19.	922117114065	MUJIPUR RAHMAN J	85
20.	922117114066	NAGARAJ A	85
21.	922117114067	NAGA SARAVAN B	90
22.	922117114070	NAVEEN G A	90
23.	922117114071	NAVEEN BHARATH V	85
24.	922117114072	NAVEENKUMAR S (12-07-2000)	90
25.	922117114075	NITHIS C	95
26.	922117114076	NITHIS KUMAR K	90
27.	922117114078	PANDIYA RAJ B	85

SSM Institute of Engineering and Technology, Dindigul - 624 002.

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

28.	922117114081	PERIYASAMY K	90
29.	922117114083	PREMKUMAR C	85
30.	922117114084	PUGAZHENTHI M	95
31.	922117114309	SRI GANESH	85
32.	922117114310	SRISUDHARSANA SAKRAVARTHI M	90
33.	922117114091	ROBINS CHRISTOPHER L	80
34.	922117114092	ROKESHWARAN R	85
35.	922117114093	SAKTHI SEENIVASAN S	85
36.	922117114103	SATHYANARAYANAN V B	85
37.	922117114104	SEENU K	70
38.	922117114105	SHANE D	85
39.	922117114116	VASHANTH S	90
40.	922117114117	VETRIVEL P.A	95
41.	922117114118	VIGNESH B	85
42.	922117114119	VIGNESH G	95
43.	922117114126	YOKESWARAN M S	90

Faculty Incharge

HoD/Mech.Engg

Principal

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	Srigaresh
Register Number	922117114309
Year &Section	II &C

1. What does CNC stand for?
   a. Computer Networking and Control
   b. Computer Numerical Control
   c. Centralized Numerical Code
   d. Control Number Computing
2. Which programming language is commonly used in CNC programming?
   a. Java
   b. C++
   c. G-code
   d. Python
3. What does G-code represent in CNC programming?
   a. Graphic Code
   b. Geometry Code
   c. General Code
   d. Gears Code
4. In CNC programming, what does the term "RPM" stand for?
   a. Rotations Per Minute
   b. Revolutions Per Millisecond
   c. Radii Per Minute
   d. Rapid Programming Mode
5. What is the purpose of the M00 code in CNC programming?
   a. Stop
   b. Pause
   c. End of Program
   d. Spindle Speed Control
6. Which code is used for tool change in CNC programming?
   a. T-code
   b. C-code
   c. M-code
   d. G-code
7. What does the term "Dwell" refer to in CNC programming?
   a. Tool movement
   b. Rapid traverse
   c. Pause in the program
   d. Cutter compensation

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

Dr.D.SENTHIL KUMARAN, M.E., Ph.D.,
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

8. What is the function of the G90 code in CNC programming?
   a. Absolute positioning
   b. Incremental positioning
   c. Circular interpolation
   d. Tool compensation
9. Which G-code is used for linear interpolation in CNC programming?
   a. G01
   b. G02
   c. G03
   d. G04
10. What is the purpose of the G28 code in CNC programming?
   a. Tool change
   b. Return to home position
   c. Rapid traverse to a point
   d. Linear interpolation
11. What does the term "DRO" stand for in CNC machining?
   a. Digital Readout
   b. Drill Rotation Offset
   c. Data Recovery Operation
   d. Dynamic Rapid Override
12. What does the F-code represent in CNC programming?
   a. Feed rate
   b. Tool radius compensation
   c. Spindle speed
   d. Tool length offset
13. What does the G41 code indicate in CNC programming?
   a. Cutter compensation left
   b. Cutter compensation right
   c. Circular interpolation clockwise
   d. Circular interpolation counterclockwise
14. Which code is used for rapid traverse in CNC programming?
   a. G00
   b. G01
   c. G02
   d. G03
15. What does the term "CNC Interpolation" refer to?
   a. Tool wear compensation
   b. Smoothing of toolpath
   c. Automatic tool change
   d. Toolpath optimization
16. What is the purpose of the G54-G59 codes in CNC programming?
   a. Tool offset selection
   b. Spindle speed control
   c. Coolant flow control
   d. Rapid traverse positioning
17. Which code is used for canceling tool length offset in CNC programming?
   a. G40
   b. G41
   c. G42
   d. G43
18. What does the term "CNC Macro" refer to?
   a. Small-sized CNC machines

b. Customizable and reusable code segments
c. High-speed machining
d. CNC simulation software

19. What is the purpose of the G50 code in CNC programming?
   a. Set work coordinate offsets
   b. Cancel tool radius compensation
   c. Set maximum spindle speed
   d. Enable cutter compensation

20. Which code is used for canceling the tool length and radius offsets in CNC programming?
   a. G49
   b. G50
   c. G51
   d. G52

85/100

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu

Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855

E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING

(06.08.2018-19.09.2018)

Name of the Student	VIGNESH.G
Register Number	9221171144119
Year &Section	II & 'C'

1. What does CNC stand for?
    a. Computer Networking and Control
    b. Computer Numerical Control
    c. Centralized Numerical Code
    d. Control Number Computing
2. Which programming language is commonly used in CNC programming?
    a. Java
    b. C++
    c. G-code
    d. Python
3. What does G-code represent in CNC programming?
    a. Graphic Code
    b. Geometry Code
    c. General Code
    d. Gears Code
4. In CNC programming, what does the term "RPM" stand for?
    a. Rotations Per Minute
    b. Revolutions Per Millisecond
    c. Radii Per Minute
    d. Rapid Programming Mode
5. What is the purpose of the M00 code in CNC programming?
    a. Stop
    b. Pause
    c. End of Program
    d. Spindle Speed Control
6. Which code is used for tool change in CNC programming?
    a. T-code
    b. C-code
    c. M-code
    d. G-code
7. What does the term "Dwell" refer to in CNC programming?
    a. Tool movement
    b. Rapid traverse
    c. Pause in the program
    d. Cutter compensation

SSM Institute of Engineering and Technology · DINDIGUL-624 002

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul 624 002.

8. What is the function of the G90 code in CNC programming?
   a. Absolute positioning
   b. Incremental positioning
   c. Circular interpolation
   d. Tool compensation
9. Which G-code is used for linear interpolation in CNC programming?
   a. G01
   b. G02
   c. G03
   d. G04
10. What is the purpose of the G28 code in CNC programming?
   a. Tool change
   b. Return to home position
   c. Rapid traverse to a point
   d. Linear interpolation
11. What does the term "DRO" stand for in CNC machining?
   a. Digital Readout
   b. Drill Rotation Offset
   c. Data Recovery Operation
   d. Dynamic Rapid Override
12. What does the F-code represent in CNC programming?
   a. Feed rate
   b. Tool radius compensation
   c. Spindle speed
   d. Tool length offset
13. What does the G41 code indicate in CNC programming?
   a. Cutter compensation left
   b. Cutter compensation right
   c. Circular interpolation clockwise
   d. Circular interpolation counterclockwise
14. Which code is used for rapid traverse in CNC programming?
   a. G00
   b. G01
   c. G02
   d. G03
15. What does the term "CNC Interpolation" refer to?
   a. Tool wear compensation
   b. Smoothing of toolpath
   c. Automatic tool change
   d. Toolpath optimization
16. What is the purpose of the G54-G59 codes in CNC programming?
   a. Tool offset selection
   b. Spindle speed control
   c. Coolant flow control
   d. Rapid traverse positioning
17. Which code is used for canceling tool length offset in CNC programming?
   a. G40
   b. G41
   c. G42
   d. G43
18. What does the term "CNC Macro" refer to?
   a. Small-sized CNC machines

b. Customizable and reusable code segments
c. High-speed machining
d. CNC simulation software

19. What is the purpose of the G50 code in CNC programming?
   a. Set work coordinate offsets
   b. Cancel tool radius compensation
   c. Set maximum spindle speed
   d. Enable cutter compensation

20. Which code is used for canceling the tool length and radius offsets in CNC programming?
   a. G49
   b. G50
   c. G51
   d. G52

9/10

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul 624 002.

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	AktharAli. S
Register Number	922117114003
Year &Section	II 'A'

1. What does CNC stand for?
   a. Computer Networking and Control
   b. Computer Numerical Control
   c. Centralized Numerical Code
   d. Control Number Computing
2. Which programming language is commonly used in CNC programming?
   a. Java
   b. C++
   c. G-code
   d. Python
3. What does G-code represent in CNC programming?
   a. Graphic Code
   b. Geometry Code
   c. General Code
   d. Gears Code
4. In CNC programming, what does the term "RPM" stand for?
   a. Rotations Per Minute
   b. Revolutions Per Millisecond
   c. Radii Per Minute
   d. Rapid Programming Mode
5. What is the purpose of the M00 code in CNC programming?
   a. Stop
   b. Pause
   c. End of Program
   d. Spindle Speed Control
6. Which code is used for tool change in CNC programming?
   a. T-code
   b. C-code
   c. M-code
   d. G-code
7. What does the term "Dwell" refer to in CNC programming?
   a. Tool movement
   b. Rapid traverse
   c. Pause in the program
   d. Cutter compensation

8. What is the function of the G90 code in CNC programming?
    a. Absolute positioning
    b. Incremental positioning
    c. Circular interpolation
    d. Tool compensation
9. Which G-code is used for linear interpolation in CNC programming?
    a. G01
    b. G02
    c. G03
    d. G04
10. What is the purpose of the G28 code in CNC programming?
    a. Tool change
    b. Return to home position
    c. Rapid traverse to a point
    d. Linear interpolation
11. What does the term "DRO" stand for in CNC machining?
    a. Digital Readout
    b. Drill Rotation Offset
    c. Data Recovery Operation
    d. Dynamic Rapid Override
12. What does the F-code represent in CNC programming?
    a. Feed rate
    b. Tool radius compensation
    c. Spindle speed
    d. Tool length offset
13. What does the G41 code indicate in CNC programming?
    a. Cutter compensation left
    b. Cutter compensation right
    c. Circular interpolation clockwise
    d. Circular interpolation counterclockwise
14. Which code is used for rapid traverse in CNC programming?
    a. G00
    b. G01
    c. G02
    d. G03
15. What does the term "CNC Interpolation" refer to?
    a. Tool wear compensation
    b. Smoothing of toolpath
    c. Automatic tool change
    d. Toolpath optimization
16. What is the purpose of the G54-G59 codes in CNC programming?
    a. Tool offset selection
    b. Spindle speed control
    c. Coolant flow control
    d. Rapid traverse positioning
17. Which code is used for canceling tool length offset in CNC programming?
    a. G40
    b. G41
    c. G42
    d. G43
18. What does the term "CNC Macro" refer to?
    a. Small-sized CNC machines

SSM Institute of Engineering and Technology

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

b. Customizable and reusable code segments
c. High-speed machining
d. CNC simulation software

19. What is the purpose of the G50 code in CNC programming?
    a. Set work coordinate offsets
    b. Cancel tool radius compensation
    c. Set maximum spindle speed
    d. Enable cutter compensation

20. Which code is used for canceling the tool length and radius offsets in CNC programming?
    a. G49
    b. G50
    c. G51
    d. G52

75/100

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	Nithis C
Register Number	922117114075
Year &Section	II & B

1. What does CNC stand for?
   a. Computer Networking and Control
   b. Computer Numerical Control
   c. Centralized Numerical Code
   d. Control Number Computing
2. Which programming language is commonly used in CNC programming?
   a. Java
   b. C++
   c. G-code
   d. Python
3. What does G-code represent in CNC programming?
   a. Graphic Code
   b. Geometry Code
   c. General Code
   d. Gears Code
4. In CNC programming, what does the term "RPM" stand for?
   a. Rotations Per Minute
   b. Revolutions Per Millisecond
   c. Radii Per Minute
   d. Rapid Programming Mode
5. What is the purpose of the M00 code in CNC programming?
   a. Stop
   b. Pause
   c. End of Program
   d. Spindle Speed Control
6. Which code is used for tool change in CNC programming?
   a. T-code
   b. C-code
   c. M-code
   d. G-code
7. What does the term "Dwell" refer to in CNC programming?
   a. Tool movement
   b. Rapid traverse
   c. Pause in the program

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul 624 002.

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

d. Cutter compensation

8. What is the function of the G90 code in CNC programming?
   a. Absolute positioning
   b. Incremental positioning
   c. Circular interpolation
   d. Tool compensation
9. Which G-code is used for linear interpolation in CNC programming?
   a. G01
   b. G02
   c. G03
   d. G04
10. What is the purpose of the G28 code in CNC programming?
   a. Tool change
   b. Return to home position
   c. Rapid traverse to a point
   d. Linear interpolation
11. What does the term "DRO" stand for in CNC machining?
   a. Digital Readout
   b. Drill Rotation Offset
   c. Data Recovery Operation
   d. Dynamic Rapid Override
12. What does the F-code represent in CNC programming?
   a. Feed rate
   b. Tool radius compensation
   c. Spindle speed
   d. Tool length offset
13. What does the G41 code indicate in CNC programming?
   a. Cutter compensation left
   b. Cutter compensation right
   c. Circular interpolation clockwise
   d. Circular interpolation counterclockwise
14. Which code is used for rapid traverse in CNC programming?
   a. G00
   b. G01
   c. G02
   d. G03
15. What does the term "CNC Interpolation" refer to?
   a. Tool wear compensation
   b. Smoothing of toolpath
   c. Automatic tool change
   d. Toolpath optimization
16. What is the purpose of the G54-G59 codes in CNC programming?
   a. Tool offset selection
   b. Spindle speed control
   c. Coolant flow control
   d. Rapid traverse positioning
17. Which code is used for canceling tool length offset in CNC programming?
   a. G40
   b. G41
   c. G42
   d. G43
18. What does the term "CNC Macro" refer to?

a. Small-sized CNC machines
b. Customizable and reusable code segments
c. High-speed machining
d. CNC simulation software

19. What is the purpose of the G50 code in CNC programming?
a. Set work coordinate offsets
b. Cancel tool radius compensation
c. Set maximum spindle speed
d. Enable cutter compensation

20. Which code is used for canceling the tool length and radius offsets in CNC programming?
a. G49
b. G50
c. G51
d. G52

95/100

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po).
Palani Road, Dindigul 624 002.

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	B.Pandiyaraj
Register Number	922117114078
Year &Section	II & B

FEEDBACK FORM

1. Course objective and scope in the industry (Please put ✓ mark)	❑ Excellent ☑ Good ❑ Average ❑ Poor
2. Knowledge and exposure of the trainer in the domain (Please put ✓ mark)	❑ Excellent ☑ Good ❑ Average ❑ Poor
3. Content coverage (Please put ✓ mark)	❑ Excellent ☑ Good ❑ Average ❑ Poor
4. Usefulness (Please put ✓ mark)	❑ Excellent ☑ Good ❑ Average ❑ Poor
5. Explanation and Clarity (Please put ✓ mark)	❑ Excellent ☑ Good ❑ Average ❑ Poor

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	Vetrivel.P.A.
Register Number	922717114117
Year &Section	II &C.

FEEDBACK FORM

1. Course objective and scope in the industry (Please put ✔ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor
2. Knowledge and exposure of the trainer in the domain (Please put ✔ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor
3. Content coverage (Please put ✔ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor
4. Usefulness (Please put ✔ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor
5. Explanation and Clarity (Please put ✔ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

SSM Institute of Engineering and Technology DINDIGUL-624 002.

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E-mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	JEROME F
Register Number	922117114044
Year &Section	II – B

FEEDBACK FORM

1. Course objective and scope in the industry (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor,
2. Knowledge and exposure of the trainer in the domain (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
3. Content coverage (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
4. Usefulness (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
5. Explanation and Clarity (Please put ✓ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY
Dindigul-Palani Highway, Dindigul – 624 002, Tamilnadu
Tel. No:0451-2448800-899 (100 lines) Fax : 0451-2448855
E mail : ssmietdgl@gmail.com

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL COURSE ON CNC PROGRAMMING
(06.08.2018-19.09.2018)

Name of the Student	Naveen G A
Register Number	922117114070
Year &Section	II - B

FEEDBACK FORM

1. Course objective and scope in the industry (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
2. Knowledge and exposure of the trainer in the domain (Please put ✓ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor
3. Content coverage (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
4. Usefulness (Please put ✓ mark)	☑ Excellent ☐ Good ☐ Average ☐ Poor
5. Explanation and Clarity (Please put ✓ mark)	☐ Excellent ☑ Good ☐ Average ☐ Poor

Naveen G A

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

SSM Institute of Engineering and Technology * DINDIGUL-624 002 *

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul-Palani Highway, Dindigul-624 002.

Department of Mechanical Engineering

Certificate of Completion

This is to certify that Mr. GAJENDREN R (922117114028) of has successfully completed the value added course on *"CNC Programming"* organized by the **Department of Mechanical Engineering, SSM Institute of Engineering and Technology**, Dindigul from 06.08.2018 to 19.09.2018.

Co-ordinator

Head/Mech.Engg

Principal, SSMIET

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

SSM Institute of Engineering and Technology * Dindigul-624

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul-Palani Highway, Dindigul-624 002.

Department of Mechanical Engineering

Certificate of Completion

This is to certify that Mr. NAGA SARAVAN .B (922117114067) of has successfully completed the value added course on *"CNC Programming"* organized by the **Department of Mechanical Engineering, SSM Institute of Engineering and Technology**, Dindigul from 06.08.2018 to 19.09.2018.

Co-ordinator	**Head/Mech.Engg**	**Principal, SSMIET**

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002.

SSM Institute of Engineering and Technology · DINDIGUL 624 002

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul-Palani Highway, Dindigul-624 002.

Department of Mechanical Engineering

Certificate of Completion

This is to certify that Mr.PANDIYA RAJ B (922117114078) of has successfully completed the value added course on *"CNC Programming"* organized by the **Department of Mechanical Engineering, SSM Institute of Engineering and Technology**, Dindigul from 06.08.2018 to 19.09.2018.

Co-ordinator | **Head/Mech.Engg** | **Principal, SSMIET**

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po).
Palani Road, Dindigul - 624 002.

SSM Institute of Engineering and Technology - DINDIGUL-624 002.

SSM INSTITUTE OF ENGINEERING AND TECHNOLOGY

Dindigul-Palani Highway, Dindigul-624 002.

Department of Mechanical Engineering

Certificate of Completion

This is to certify that Mr. PRAVEEN KUMAR M (922117114306) _of has successfully completed the value added course on *"CNC Programming"* organized by the **Department of Mechanical Engineering, SSM Institute of Engineering and Technology**, Dindigul from 06.08.2018 to 19.09.2018.

Co-ordinator

Head/Mech.Engg

Principal, SSMIET

Dr.D.SENTHIL KUMARAN, M.E., Ph.D., (NUS)
Principal
SSM Institute of Engineering and Technology
Kuttathupatti Village, Sindalagundu (Po),
Palani Road, Dindigul - 624 002

SSM Institute of Engineering and Technology • DINDIGUL-624 002