

INTRODUCTION TO NC & CNC

Numerical control (NC) is a method of automatically operating a manufacturing machine based on a code. The numerical data required to produce a part is provided to a machine in the form of a program, called part program.

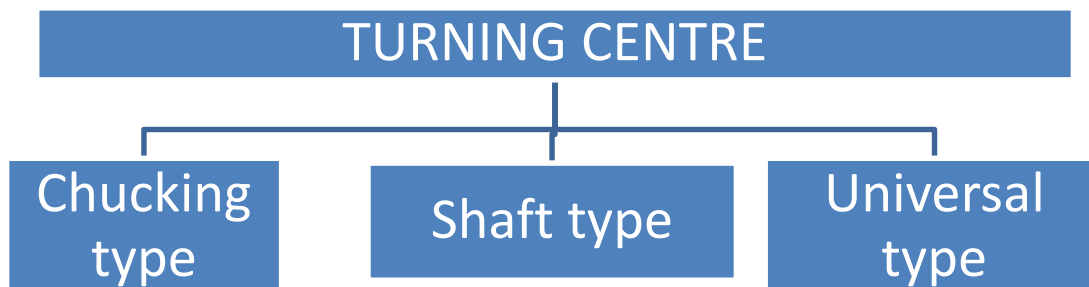
Numerical controllers were incorporated in machine tools. Then realized through computer hardware and software. Technology was renamed as **COMPUTER NUMERICAL CONTROL (CNC)** machines.

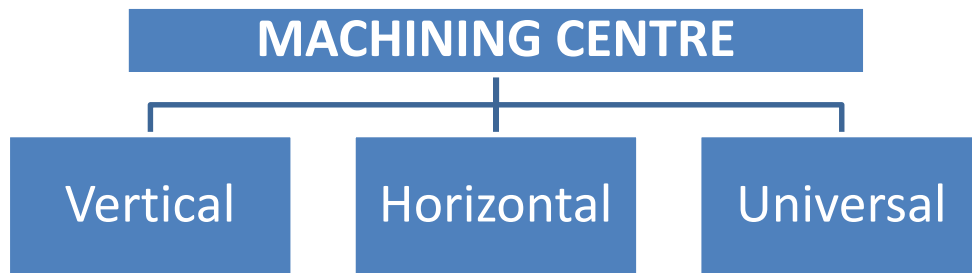
ADVANTAGES OF CNC OVER NC / CONVENTIONAL:

- ❖ Easier programming.
- ❖ Dependence on human skill & effort is reduced.
- ❖ Higher accuracy in mass production is possible.
- ❖ Complex geometry is produced as cheaply as simple one.

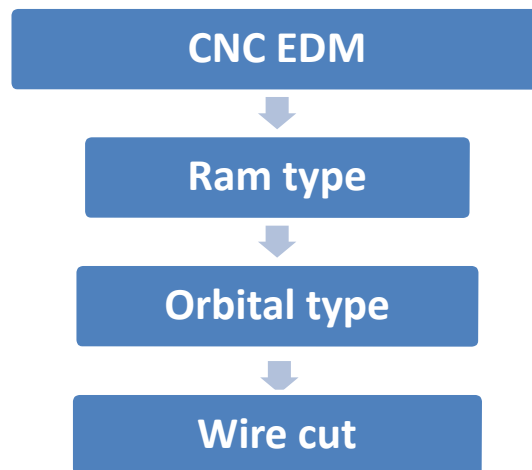
TYPES OF CNC MACHINES

1. Turning Centre
2. Machining Centre
3. Turn Mill (vertical turning centre)
4. CNC Grinding Machines
5. CNC – EDM
6. CNC drilling machine

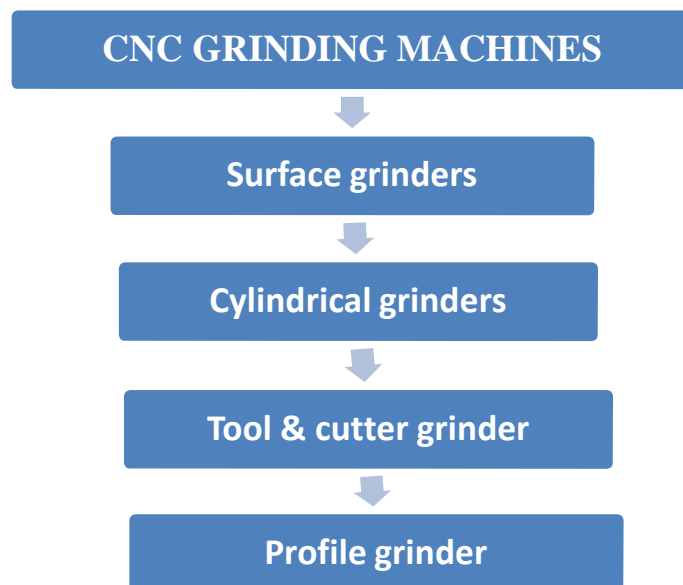




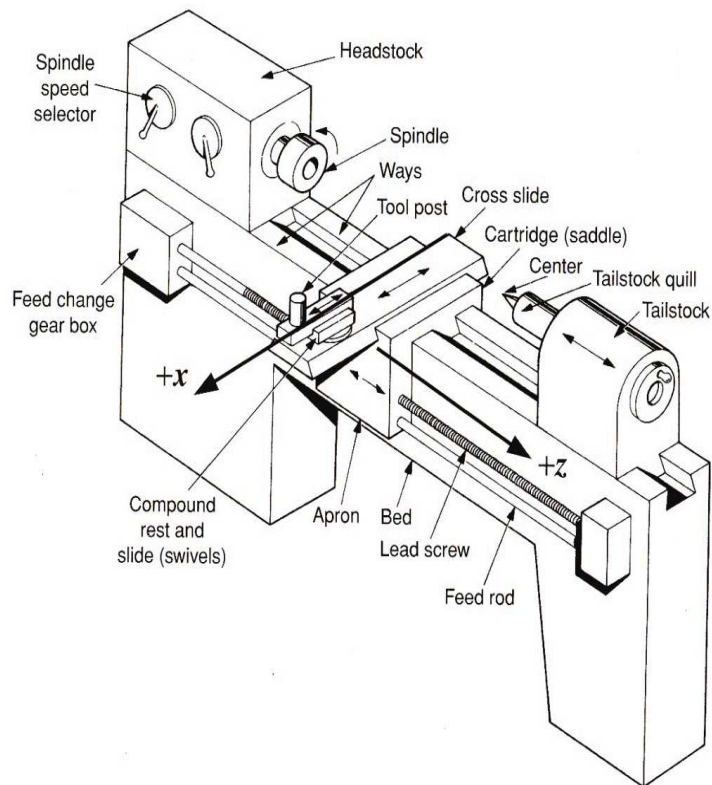
CNC – Electrical Discharge Machine:



CNC – Grinding Machine:



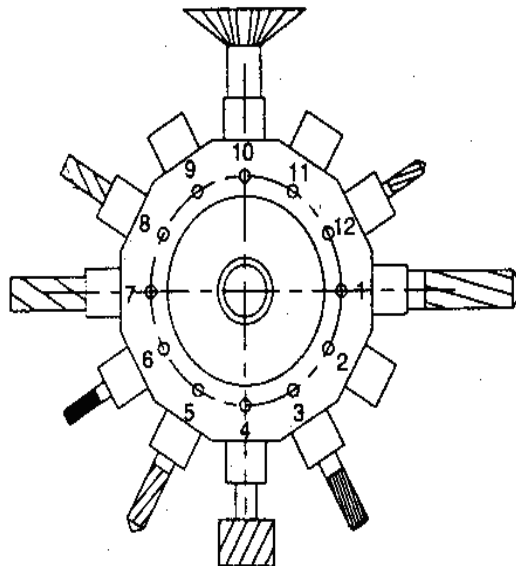
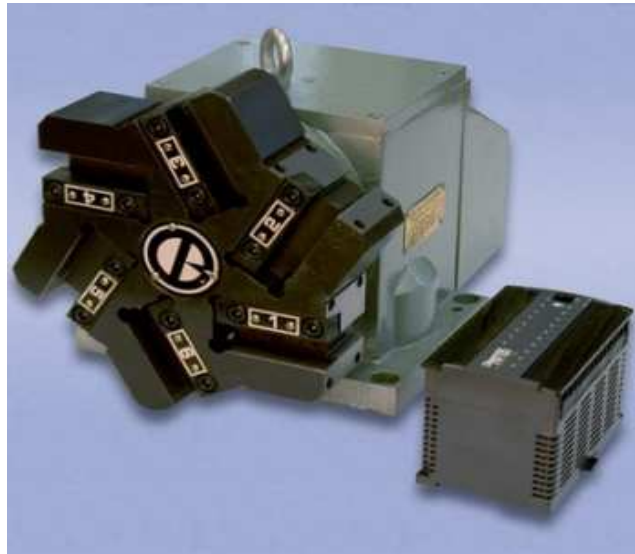
MACHINE AXES OF MOTION



WORK HOLDING DEVICES (CHUCK)



TOOL HOLDING DEVICE (Turret head)



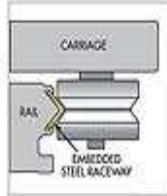
Typical tool turret used in CNC

Slideways And Motion Transmission By Linear Motion

Bearing & Recirculating Ball Screw And Nut Arrangements.

Linear Motion Bearing

PRECISION STEEL RACEWAY
Durable anodized aluminum extrusion with a precision embedded hardened steel raceway.



TESTED TOUGH*

Carriage wheels allow for distribution of force throughout the integral v-rail and tolerates high load capacities up to 1250lbs.

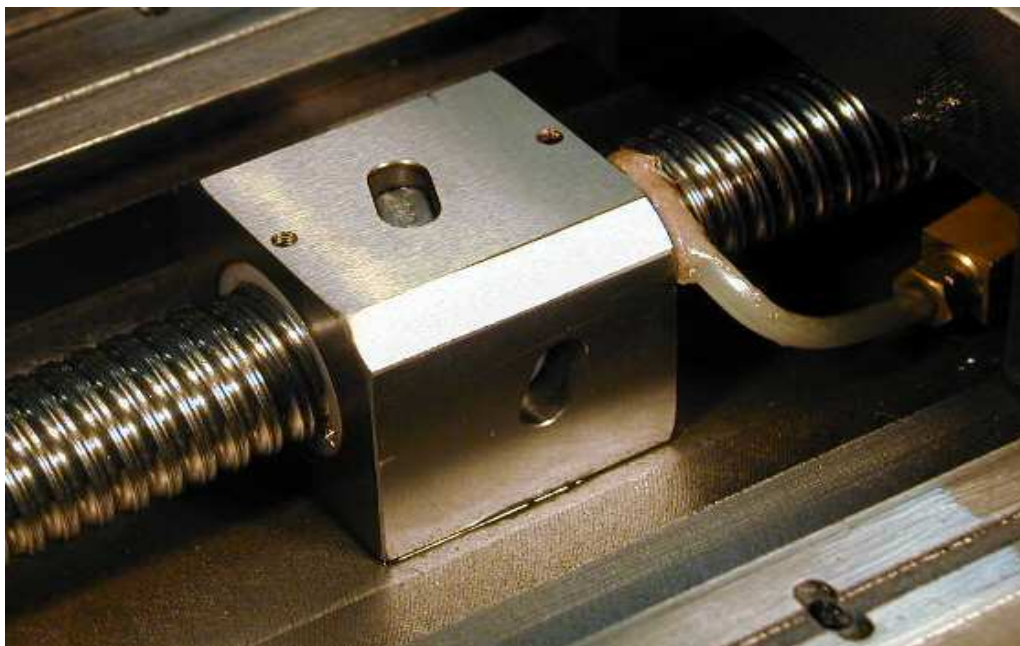
**SIMULTANEOUS
INTEGRAL MILLING
OPERATION**

HIGH ACCURACY

Innovative in-house machining process (SIMO™) provides simultaneous milled surfaces that results in extreme rail accuracy and consistency.



Recirculating Ball Screw And Nut Arrangements.



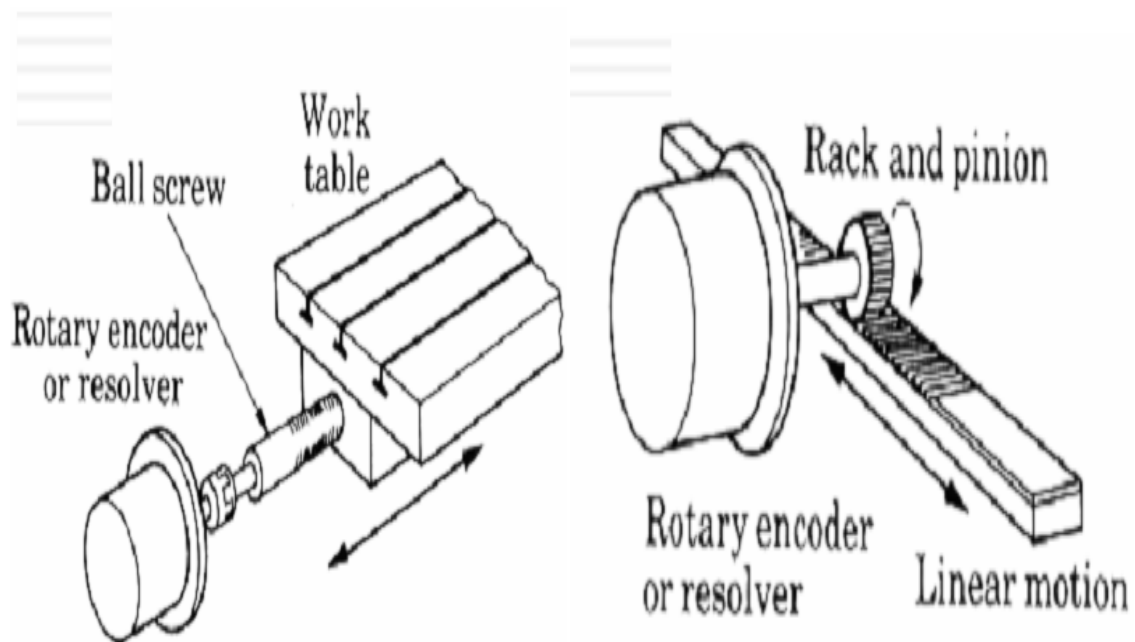
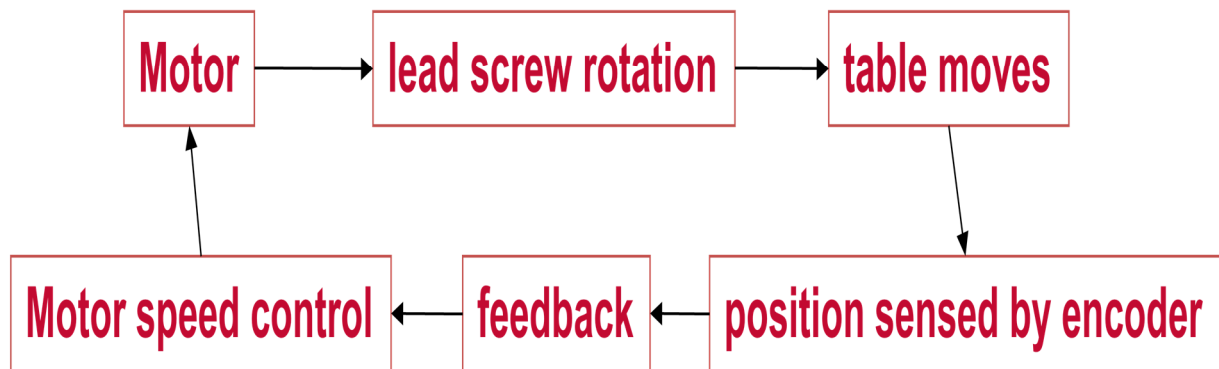
Servo motor:

In order to obtain fast response, a special type of motor called servo motor is used to give power for the slides. Any number of components of mechanical, electrical, hydraulic and pneumatic to control (feedback) the position of machine slide, then that system is known as servo system.

The servo motor can be directly coupled or drive is transmitted through a toothed belt drive.



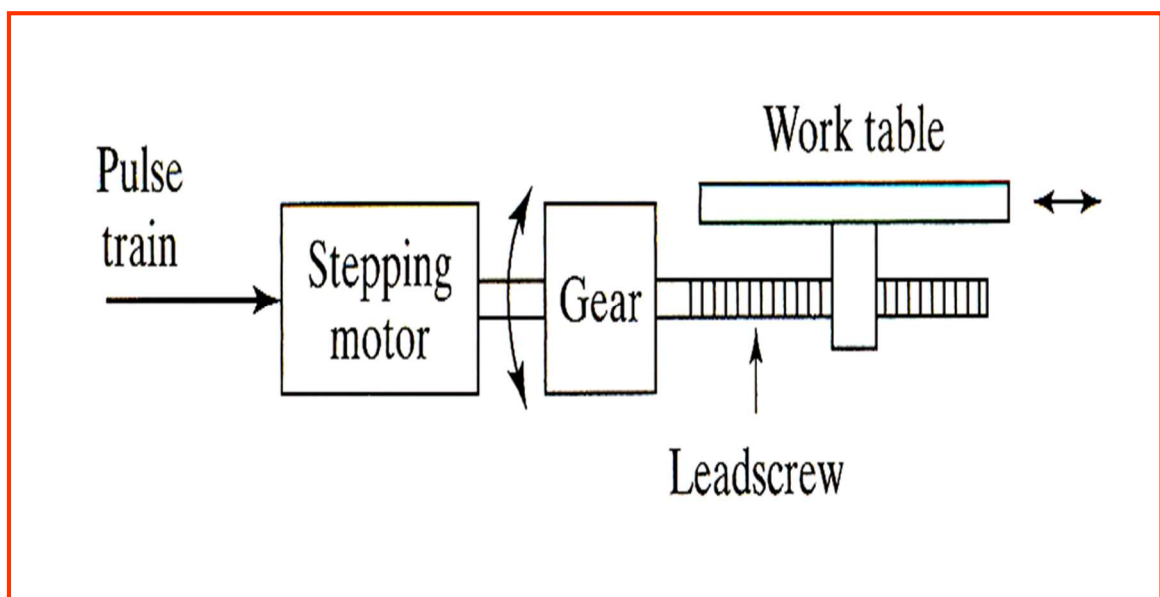
Components of Servo-motor controlled CNC



Loop Systems for Controlling Tool Movement

(Open Loop System)

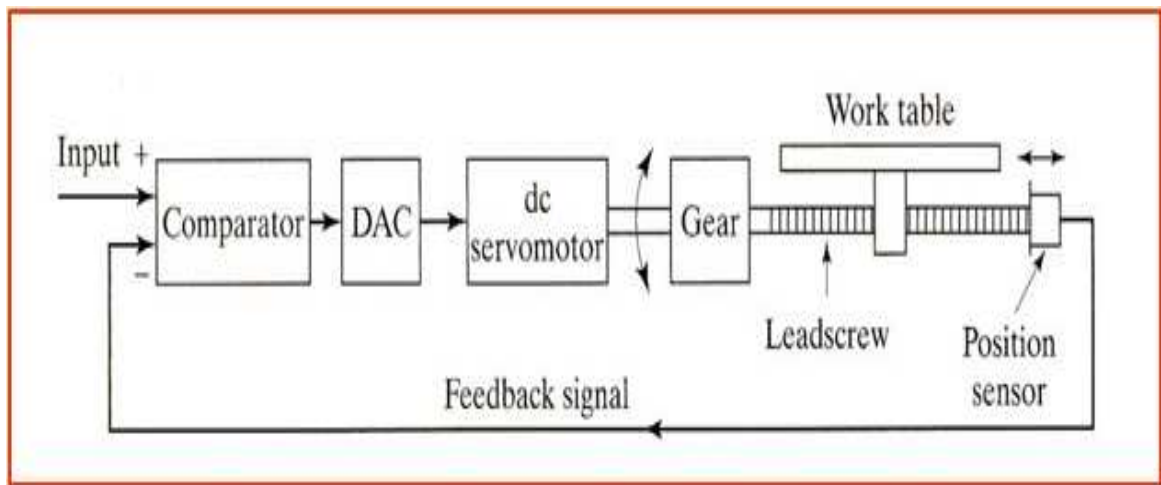
Uses stepping motor to create movement. Motors rotate a fixed amount for each pulse received from the MCU. The motor sends a signal back indicating that the movement is completed. No feedback to check how close the actual machine movement comes to the exact movement programmed.



Loop Systems for Controlling Tool Movement

(Closed Loop System)

AC, DC, and hydraulic servo-motors are used. The speed of these motors are variable and controlled by the amount of current or fluid. The motors are connect to the spindle and the table. A position sensor continuously monitors the movement and sends back a single to Comparator to make adjustments.



Machine Control Panel:

It is the direct interface between the operator and the NC system. During programme execution, the CNC controls the axis and spindle, movements. But at the same time, the machine is to be prepared for some tasks like,

- ✿ Fixing reference point.
- ✿ Loading the system memory with the required part programme.
- ✿ Loading and checking tool offsets etc.

OPERATION MODES:

Manual Mode:

Allows slide movements manually.

Manual Data Input (MDI) mode:

Allows building of new programmes on interactive basis, editing and executing programmes block by block on interactive basis.

Automatic mode:

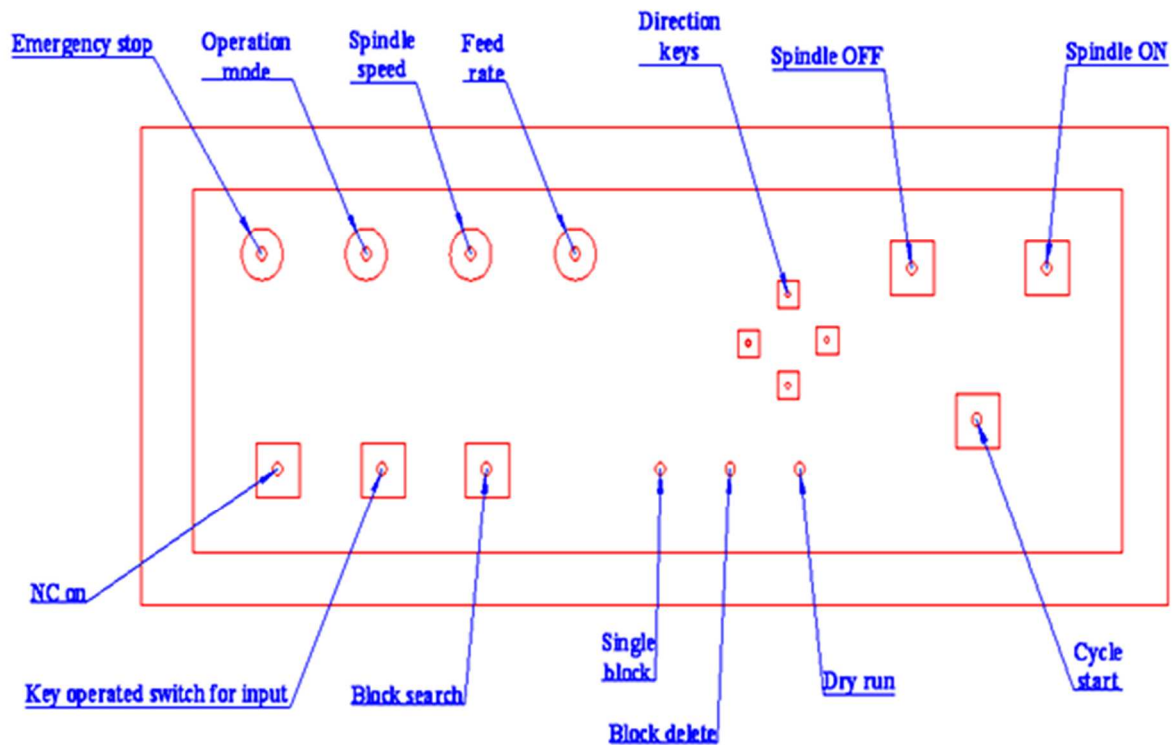
Allows execution of the part programme one block after another automatically.

Reference Mode:

Allows the machine to be referenced to its home position to effect all the compensations.

Input Mode:

Allows loading the part programme, machine setup data, tooling offsets etc., to the memory.



Operator Control Panel:

It provides a two way communication between the user and the CNC system and machine tool. This consists of two parts as shown in figure.

- Video Display Unit (VDU)
- Keyboard

Video Display Unit (VDU):

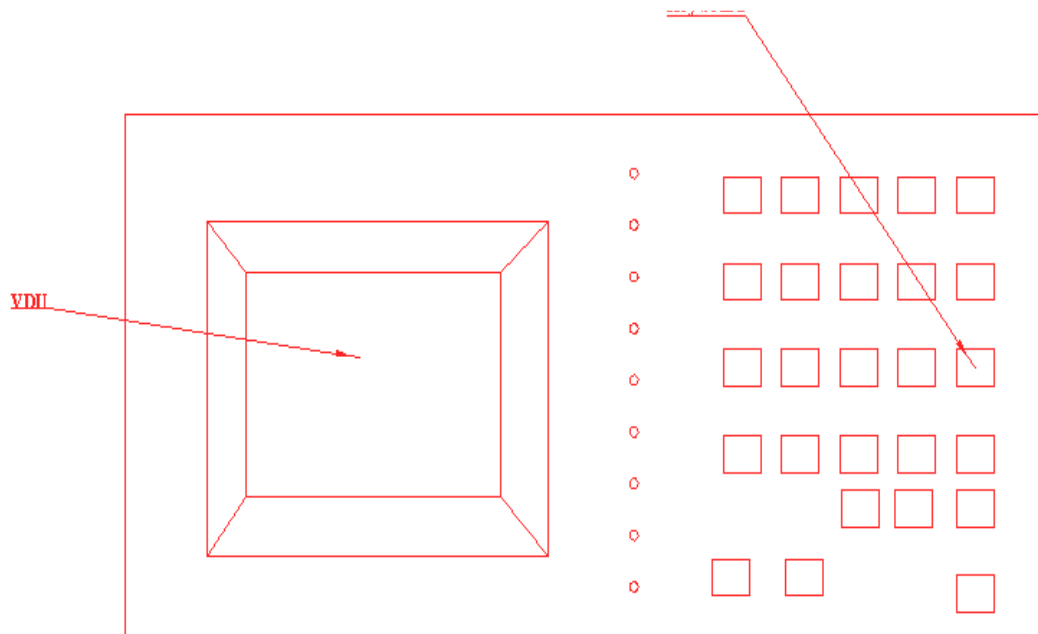
It displays the status of the various parameters of the CNC systems and the machine tool. It also displays current information's such as

- Information about the programme block on current execution
- Actual position value, current feed rate and spindle speed.
- Active G functions and M functions
- Main programme number and subroutine number
- Alarm messages etc.

Keyboard:

It is useful for the following purposes

- Editing of part programs, tool data and machine parameters
- Selection of operating modes such as manual data input, jog etc.
- Selection of feed, speed etc
- Execution of part programmes
- Execution of other tool functions etc.



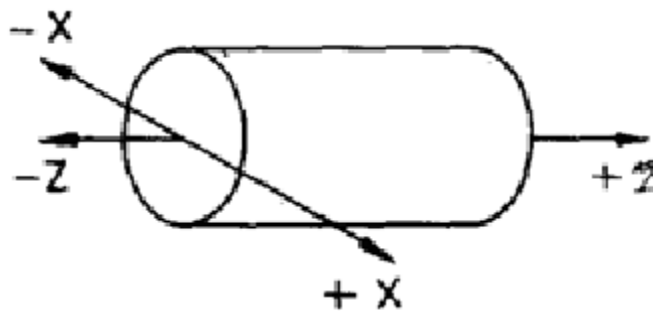
Flow of Computer-Aided CNC Processing

- 1) Develop or obtain the 3D geometric model of the part, using CAD.
- 2) Decide which machining operations and cutter-path directions are required (computer assisted).
- 3) Choose the tooling required (computer assisted).
- 4) Run CAM software to generate the CNC part program.
- 5) Verify and edit program.
- 6) Download the part program to the appropriate machine.
- 7) Verify the program on the actual machine and edit if necessary.
- 8) Run the program and produce the part.

Data required for manual part program:

- 1) Specification of coordinate system.
- 2) Specification of axes.
- 3) Specification of machines.
- 4) Specification of reference points in machines and work pieces.
- 5) Specification of tools.
- 6) Method of holding work pieces.
- 7) Parameters such as speed, feed, depth of cut etc..
- 8) Sequence of operation.

COORDINATE SYSTEM:



CNC machine tool axis system for Lathe

For turning operation there is only two axes X and Z. Z is the axis of rotation of the work piece. X is the radial direction of the cutting tool as shown in the above figure.

ZERO POINTS AND REFERENCE POINTS:

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

There are two methods of specifying the zero points

- Fixed zero
- Floating zero

Fixed zero:

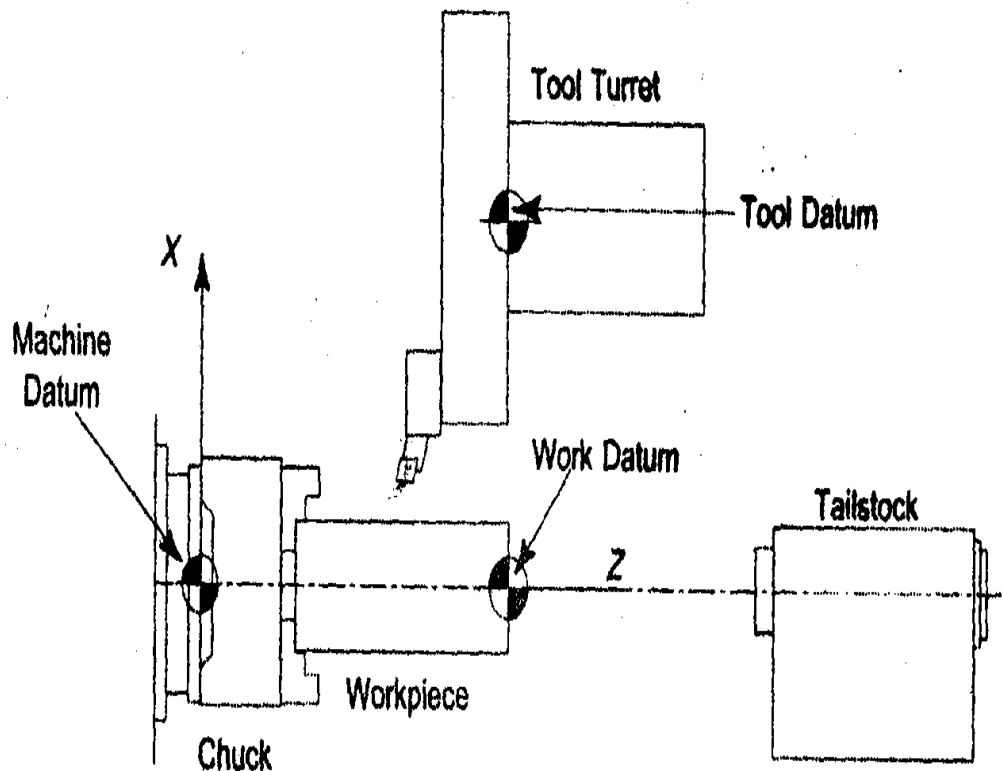
It is the zero point of the NC machine tool fixed by the manufacturer. It is always located at same position the machine tool.

Floating zero:

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

Machine Zero point:

This is specified by the manufacturer of the machine. This is the zero point for the coordinate systems and reference points in the machine, the machine zero point can be the centre 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 calibrating and for controlling the measuring system of the slides and tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches.

Work piece Zero Point:

This point determines the work piece coordinate system in relation to the machine zero point. The work piece zero point is chosen by the programmer and input into the CNC system when setting up the machine. The position of the work piece zero point can be freely chosen by the programmer within the work piece envelope of the machine.

Tool Reference Points:

When machining a work piece, it is essential to control the tool point or the tool cutting edges in precise relationship to the work piece along the machining path. Since tools have different shapes and dimensions, precise tool dimensions have to be established beforehand and input into the control system. The tool dimensions are related to a fixed tool setting point during pre-setting.

BASIC CONCEPT OF PART PROGRAMMING:

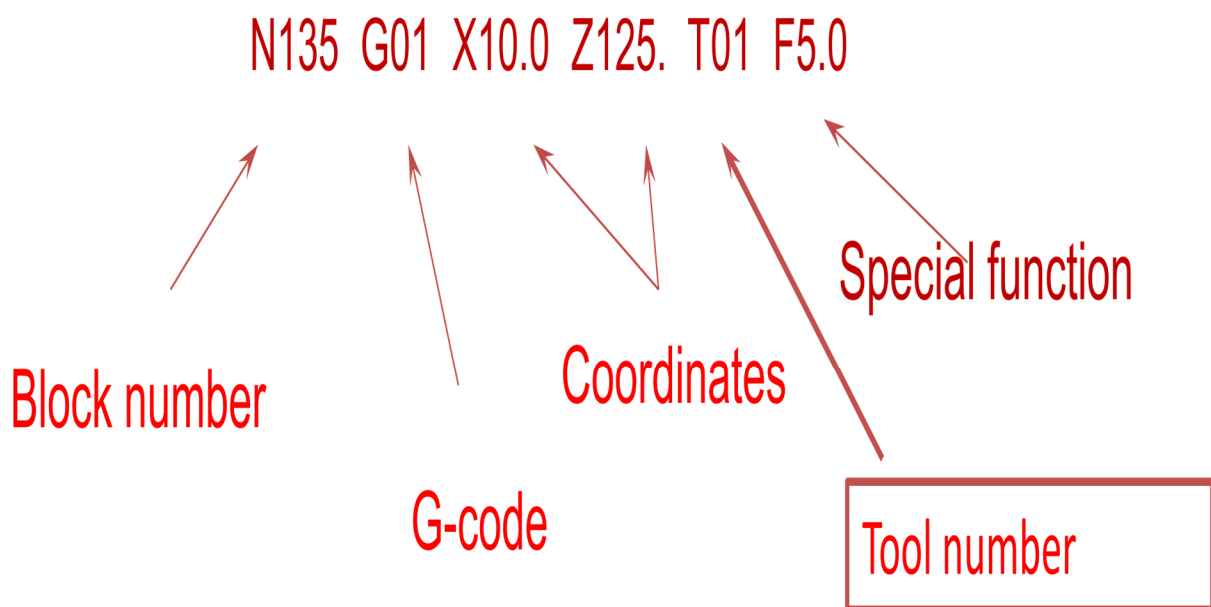
Part programming contains geometric data about the part and motion information to move the cutting tool with respect to the work piece.

Basically, the machine receives instructions as a sequence of blocks containing commands to set machine parameters; speed, feed and other relevant information.

A block is equivalent to a line of codes in a part program.

N135 G01 X10.0 Z125. T01 F5.0

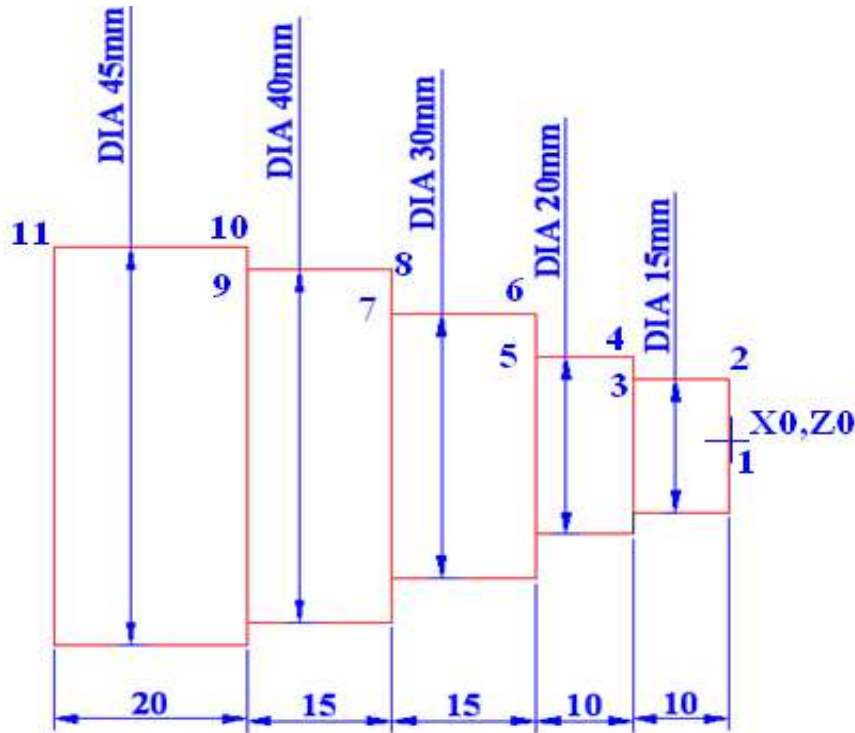
A block is equivalent to a line of codes in a part program.



CNC COORDINATES DIMENSIONING:

Absolute dimensioning:

In absolute dimensioning, the coordinates of a point in a work piece are always defined with respect to the origin.



All dimensions are in 'mm'

Incremental dimensioning:

In Incremental dimensioning the co-ordinates of a point in a work piece are always defined with respect to previous point.

Absolute Method of CNC Programming:

POINTS	X	Z
1	0	0
2	15	0
3	15	-10
4	20	-10
5	20	-20
6	30	-20
7	30	-35
8	40	-35
9	40	-50
10	45	-50
11	45	-70

Incremental method of CNC programming:

POINTS	X	Z
1	0	0
2	15	0
3	0	-10
4	5	0
5	0	-10
6	10	0
7	0	-15
8	10	0
9	0	-15
10	5	0
11	0	-20

FUNCTION OF KEYS

AUTO	MACHINING AUTOMATICALLY
EDIT	TO EDIT THE PROGRAM
MDI	MANUAL DATA INPUT
COOLANT	COOLANT ON/OFF MANUALLY
SPDL CW	SPINDLE ROTATION CLOCKWISE
SPDL CCW	SPINDLE ROTATION ANTI CLOCKWISE
SPDL STOP	SPINDLE ROTATION STOP
REF	HOME POSITION
JOG	MANUAL MOVEMENT (X & Z MOVEMENT)
HANDLE	MANUAL MOVEMENT (X & Z MOVEMENT)
SINGLE BLOCK	TO RUN THE PROGRAM (LINE BY LINE OR BLOCK BY BLOCK)
DRY RUN	MACHINE WARM UP CONDITION
TUR MAN	TO ROTATE THE TURRET HEAD AT HOME POSITION
EMG REL	STOP AT EMERGENCY
EDIT KEY	MACHINE ON /OFF
CHUCK OPEN/CLOSE	TO OPEN & CLOSE THE CHUCK
HOLD	TEMPORARY STOP
START/CYCLE START	TO START THE PROGRAM
STOP/CYCLE STOP	TO STOP THE PROGRAM
LUB MAN	MANUALLY LUBRICATION

REFERENCE POINT SETTING

1. Press the REF KEY and then press the +X and + Z KEYs to move the TURRET HEAD to the home position.
2. Press the JOG KEY and then press the TUR MAN KEY to rotate the TURRET HEAD for required tool in a cutting position.
3. If it is not indexed the TURRET HEAD then go to first point.

TOOL OFFSET SETTING

1. Press the MDI mode and then press PROGRAM mode
2. Type G97 S1000 M4 and press the EOB and INSERT KEYs.
3. Then press the CYCLE START KEY.
4. Press the HANDLE KEY, (to move the tool X and Z axis for pressing the X and Z KEY with rotation of handle)
5. Facing the work piece and then press the RESET KEY.
6. To press the OFFSET KEY in the key board and then press the OFFSET KEY in the screen, again press the GEOMETRY KEY in the screen.
7. To move the cursor by using arrow keys in the required tool number in the Z position.
8. Type Z0. And press the MEASURE KEY in the screen.
9. Press the MDI KEY and then press the program KEY. Type the tool number (for example T505) and press the EOB and INSERT KEYs.
10. Then press the CYCLE START KEY.
11. Press the POSITION KEY in the key board to see the tool position in Z=0.00
12. The same procedure is to be followed for turning operation. Only difference is, to turn the work piece in a particular length and measure the diameter and it is to be entered the value of X50. (For example) in place of X position in the screen.

CNC LATHE

G codes & Its Functions

G Code	Functions
G00	Positioning (Rapid Travels)
G01	Linear Interpolation
G02	Circular Interpolation(CW)
G03	Circular interpolation(CCW)
G04	Dwell Time
G20	Input in “Inch”
G21	Input in “mm”
G28	Return to Reference (home) Position
G40	Tool Nose Radius Compensation Cancel
G41	Tool Nose Radius Compensation Left
G42	Tool Nose Radius Compensation Right
G50	Max. Spindle Speed Setting
G70	Finishing Cycle
G71	Stock Removal in Turning
G72	Stock Removal in Facing
G75	Grooving Cycle
G76	Thread Cycle
G96	Constant Surface Speed Control
G97	Constant Surface Speed Cancel
G80	Canned Cycle Cancel
G90	Absolute Programming
G91	Incremental Programming

Syntax of G CODES

G CODES	SYNTAX
G00	G0 X Z F;
G01	G01 X Z F;
G02	G02 X Z R F;
G03	G03 X Z R F;
G04	G4 X
G28	G28 U0 W0
G50	G50 S M
G70	G70 P Q F
G71	G71 U R G71 P Q U W F
G72	G72 U R G72 P Q U W F
G75	G75 R; G75 X Z. P Q F;
G76	G76 P Q R; G76 X ZP Q F
G96	G96 S M
G97	G97 S M

M codes & Its Functions

M Code	Functions
M00	Program Stop
M01	Optional Stop
M02	Program End
M03	Spindle CW
M04	Spindle CCW
M05	Spindle Halt
M08	Coolant ON
M09	Coolant OFF
M10	Chuck / Collet Close
M11	Chuck / Collet Open
M12	Tail Stock Quill IN
M13	Tail stock quill OUT
M30	Program End & Rewind
M98	Sub Program Call
M99	Nesting
M80	Chuck Outer Clamping
M81	Chuck Inner Clamping

Miscellaneous Function (M Codes)

Miscellaneous Function perform a variety of auxiliary commands, such as stopping the program, starting or stopping the spindle or feed, tool changes, coolant flow etc., which control the machine tool. This is denoted by “M”. These functions actually operate some controls on the machine tool and thus affect the running of the machine. Miscellaneous commands are normally placed at the end of the block.

M00: Program Stop

By inserting M00 in a program, the cutting cycle is stopped after the block containing M00 code. This facility is useful if an inspection check is necessary during an operation. The cycle is then continued by a cycle start.

Example: M00**M01: Optional Stop**

Cycle operation is stopped after a block containing M01 is executed. This code is only effective when the optional stop switch on the machine control panel has been pressed.

Example: M01**M02: Program End**

M02 halts program execution. To execute the program once again, the system must reset.

Example: M02**M03: Spindle Rotation Clockwise**

An M03 instruction starts spindle rotation clockwise. It requires a speed within the range 100 to 3000 RPM

Example: M03 S2200**M04: Spindle Rotation Counter Clockwise**

An M04 instruction starts spindle rotation counter clockwise. It requires a speed within the range 100 to 3000 RPM

Example: M04 S2200**M05: Spindle Stop**

M05 commands instruction stop spindle rotation. It is good programming practice to issue an M05 before a tool change, and at the end of a program.

Example: M05**M06: Tool Change**

The M06 instruction commands to change a different tool.

Example: M06 T01**M08: Coolant On**

M08 turns the coolant on

M09: Coolant Off

M09 turns the coolant off

M10: Chuck Open

M10 commands Chuck open

M11: Chuck Close

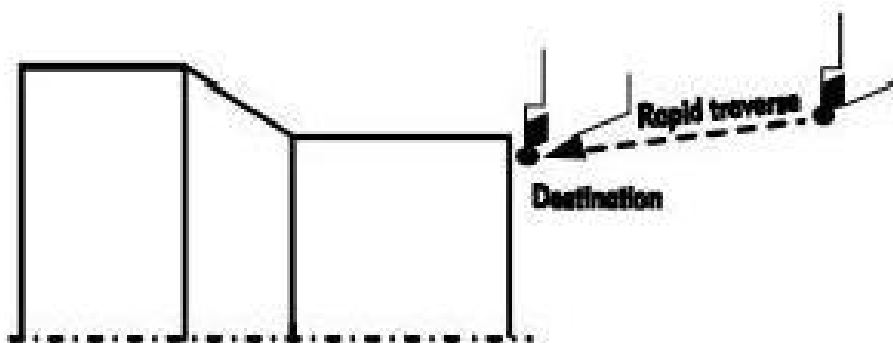
M11 commands Chuck close

M30: Program Stop and Rewind

This command is used to stop the spindle, turns the coolant off, terminates and reset the CNC program.

M98: Sub-Program Call**M99: Sub-Program Exit****Preparatory Functions (G Codes)**

This is denoted by “G”. These are preset function associated with the movement of machine axes and the associated geometry. It prepares the machine control unit for the instruction and data contained in the block. It has two digits.

G00: Positioning (Rapid travels)

The rapid traverse instruction is identified by the program word G00. A rapid traverse instruction traverses the tool to the target point at maximum traverse

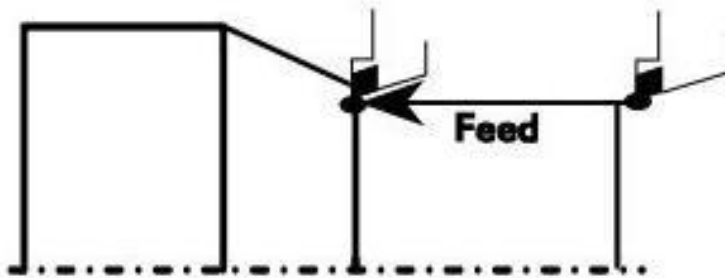
rate. The tool normally takes the shortest path from the starting point to the destination point.

Format : G00 X__ Z__

Example : G00 X20 Z10

Here the tool is moved to X20mm and Z10mm at the maximum traverse rate.

G01: Linear Interpolation



G01 traverse the tool along a linear path to the given target point with the feed rate. The feed rate determines the speed with which the work piece is machined.

Format: G01 X__ Z__ F__

Where

X **Desired coordinate in X axis**

Z **Desired coordinate in Z axis**

F **Feed rate**

Example:

G00 X45 Z10;

G00 X45 Z1;

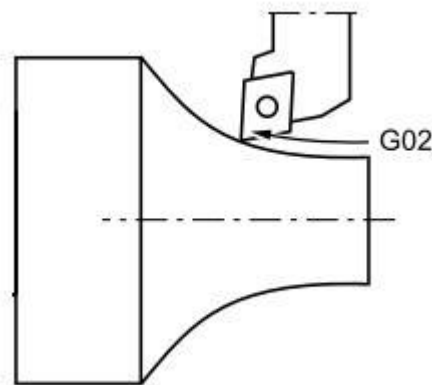
G01 X45 Z-40 F100;

```
G00 X45 Z0;  
G00 X40 Z0;  
G01 X40 Z-40;  
G00 X40 Z0;
```

Circular Interpolation:

G02: Circular Interpolation (clockwise direction)

The tool traverses from a starting point to a given target point along a circular path, and then it is called circular interpolation.



Format: G02 X__ Z__ R__ F__

Where

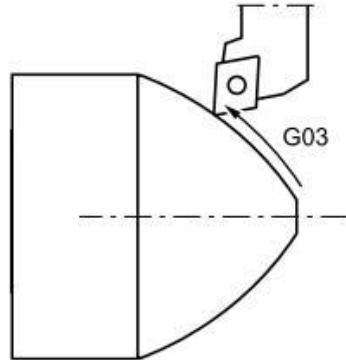
X Desired coordinate in X axis
Z Desired coordinate in Z axis
R Radius of arc
F Feed rate

Example:

```
G00 X45 Z1;  
G02 X30 Z-30 R15 F100;  
G00 X45 Z0;
```

G03: Circular Interpolation (Counter clockwise direction)

The tool traverses from a starting point to a given target point along a circular path, and then it is called circular interpolation.



Format: G03 X__ Z__ R__ F__

Where

X Desired coordinate in X axis

Z Desired coordinate in Z axis

R Radius of arc

F Feed rate

Example:

G00 X0 Z0;

G03 X30 Z-15 R15 F100;

G00 X45 Z-15;

G04: Dwell

The command G04 causes the program to wait for a specified amount of time. The time can be specified in seconds with the “X” prefixes or in milliseconds with the “P” prefix. One of the uses of this code is to get a sharp corner on the profile of the work piece in cutting feed. It is also used at the end of drilling cycle.

During cutter motion, a deceleration at the end of the motion specified by one statement and acceleration at the start of the motion specified by the next statement are usually applied automatically by the controller.

Format: G04 X_ _

Example:

G04X1.5

G04 P1500

G20: Inch mode input

All the input parameters will be taken as imperial values. That is, they will specify inches.

G21: metric mode input

All the input parameters will be taken as metric values. That is, they will specify millimeters.

G28: Go to Reference point (Home Position)

This command specifies automatic return to the reference point for the specified axes. The coordinates defined in this command is an intermediate coordinate and is commanded by absolute or incremental value. The G28 block is used to position the tool at the intermediate point of all specified axes at the rapid traverse speed, and then move to the reference point at the rapid traverse rate. In general this command is used for automatic tool changing. For safety reasons the cutter radius compensation, and tool length compensation should be canceled before executing this command.

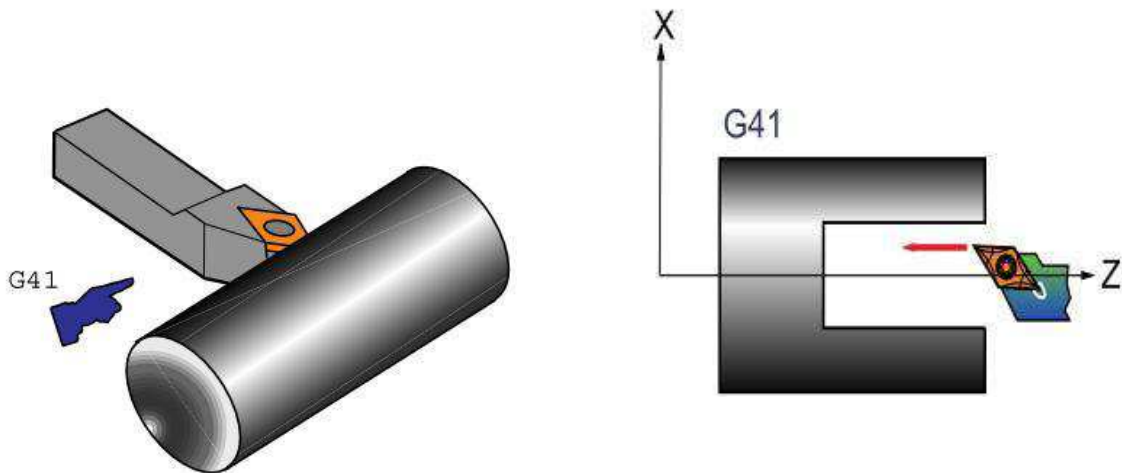
Example: G28 U0 W0 –Makes the cutting tool to move to the reference point automatically.

G40: Tool Nose Radius Compensation Cancel

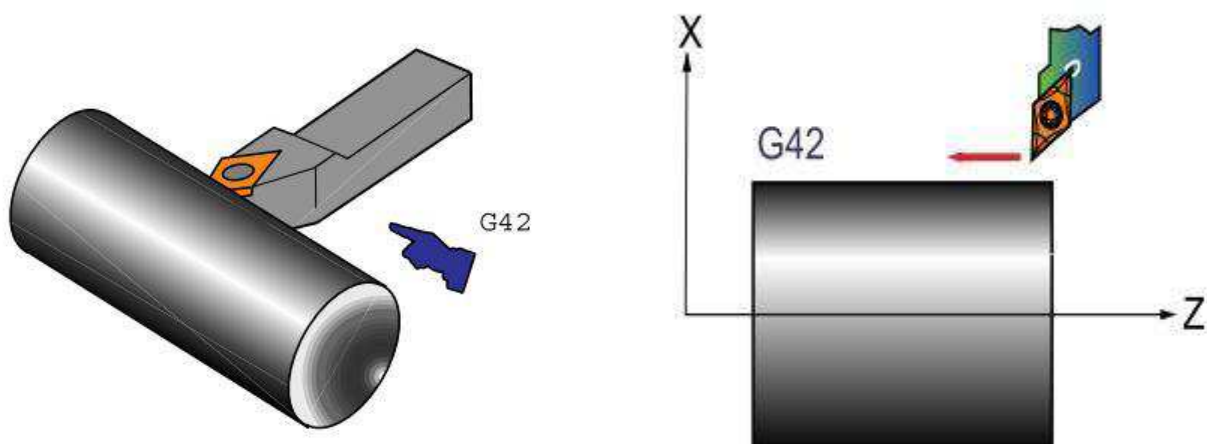
The command G40 deactivates the tool nose radius compensation.

G41: Tool Nose Radius Compensation Left

The command G41 activate tool nose radius compensation left, tool operates in machining operation to the left side of the profile or contour.

**G42: Tool Nose Radius Compensation Right**

The command G42 activate tool nose radius compensation right, tool operates in machining operation to the right side of the profile or contour.



G50: Maximum Spindle Speed Setting

To set the maximum spindle speed at machining operation.

Format: G50 S_ _ M_ _

Example: G50 S3000 M04

G96: Constant Surface Speed

The cutting speed during turning is the peripheral speed of the work. The peripheral speed of a rotating work represents the peripheral path in a given unit time.

The advantage of the Constant Surface Speed can be evident through a parting operation. During parting, the diameter of the work where cutting is taking place is steadily decreasing. The cutting efficiency can only be maintained if the spindle speed is increased at a corresponding rate. So the speed where the cutting is taking place is constant.

Example: G96 S100 M04

G97: Constant Surface Speed cancel

This command cancels constant surface speed.

Example: G97 S1000 M04

G98: Feed per Minute

This command coupled with the F word is used to specify feed rate per minute. This will be specified by mm/ min.

G99 Feed per Revolution

This command coupled with the F word is used to specify feed rate per revolution. This will be specified by mm / rev.

Program build-up for CNC Lathe using FANUC System

CNC Program can be divided into three parts.

1. Start-up Program
2. Profile Program
3. End of the Program

1. Start-up Program

O1000

G21 G98

G28 U0 W0

M06 T1

M03 S1500

G00 X32 Z5

Explanation:

O1000	While writing a program on FANUC controller first line has to be started with letter "O" followed by four digit number which specifies the program number.
G21 G98	G21 -specifies that program is done in metric units G98 gives the unit of feed in mm / min
G28 U0 W0	Makes the tool to go to home position. U & W are secondary movements about x and z axis.
M06 T1	Tool Change with tool position No: 1
M03 S1500	Makes the spindle rotation in clockwise with spindle rotates at 1500 RPM
G00 X32 Z5	G00 gives rapid position of the tool to a point X32 Z5. This is just above the billet. This point is called as the Tool entry Point.

2. Profile Program: Profile program is based on the given part drawing.

3. End of the Program:

G28 U0 W0

M05

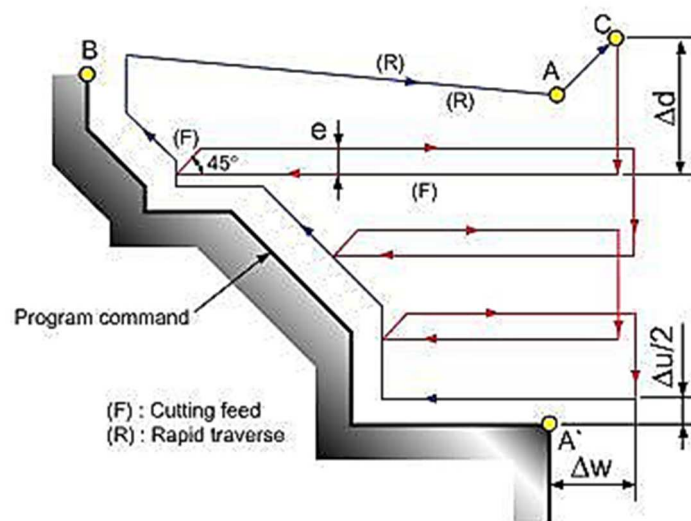
M30

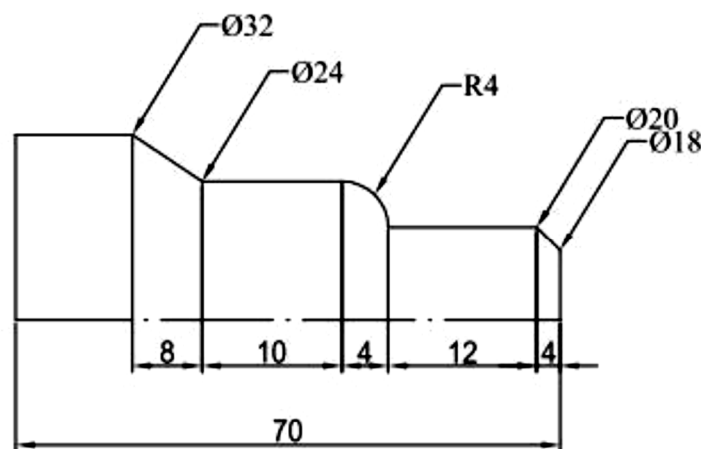
Explanation:

G28 U0 W0	Makes the tool to go to home position. U & W are secondary Movements about X and Z axis.
M05	Stop the Spindle
M30	Programme end and rewind

G71 Stock Removal Turning (or) Canned Cycle (or) Multiple Turning Cycle:

Multiple turning cycles is used when the major direction of cut is along the “Z” axis. This cycle requires two blocks are needed to specify the all parameters.



Format:**G71 U__ R__****G71 P__ Q__ U__ W__ F__****U** - Depth of cut in Z axis**R** - Relief Amount**P** - Starting block of the profile**Q** - Finishing block of the profile**U** - Finishing Allowance in X axis**W** - Finishing Allowance in Z axis**F** - Feed Rate**G71 Stock Removal Turning (or) Canned Cycle (or) Multiple Turning Cycle:****Example:****O1007**

G21 G98 -Initial Settings

G28 U0 W0 -Going to home position

M06 T1 -Tool Change Position No. 01

M03 S1500 -Spindle clockwise with 1500 RPM

G00 X32 Z5-Tool Moving to Tool Entry Point of X32 Z5 at Rapid Traverse

G71 U0.5 R1 -Calling G71 Cycle and defining Cycle Parameters
G71 P1 Q2 U0.1 W0.1 F100 -Defining Cycle Parameters
N1 G01 X18 F100
G01 Z0
G01 X20 Z-4
G01 Z-16
G03 X24 Z-20 R4
G01 Z-30
N2 G01 X32 Z-38
G28 U0 W0
M06 T2
M03 S1800
G00 X32 Z5
G70 P1 Q2 S1800 F80
G28 U0 W0
M05
M30

G70 Finishing cycle:

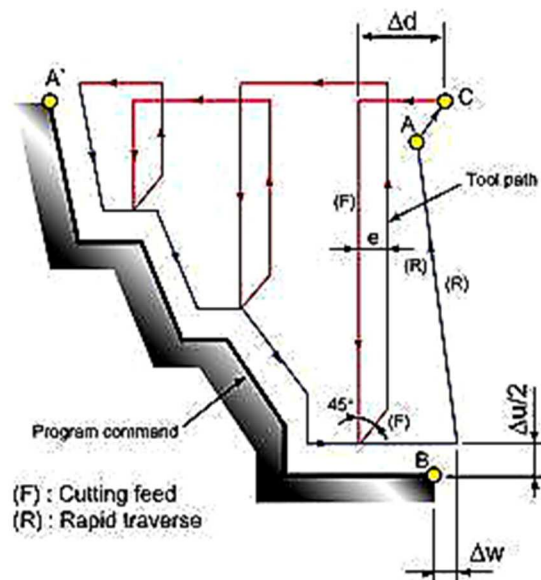
Used with G71 to give the finish cut to remove the finish allowance in X and Y direction given in G71 block.

Format: G70 P__ Q__ F__

P - Starting block number as in G71
Q - Finishing block number as in G71
F - Feed Rate

G72 Multiple Facing Cycle:

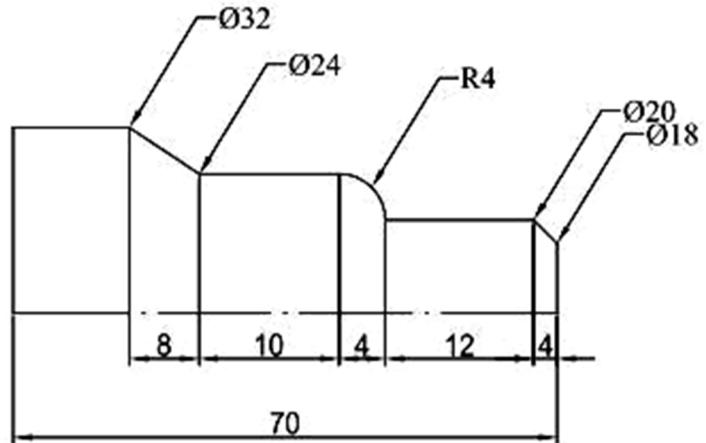
Multiple facing cycles is used when the major direction of cut is along the “X” axis. This cycle requires two blocks are needed to specify the all parameters.

**Format:**

G72 W (Δd) R (e)

G72 P (A') Q (B) U (Δu) W (Δw) F

- W** - Depth of cut in Z axis
- R** - Relief Amount
- P** - Starting block of the profile
- Q** - Finishing Allowance in X axis
- W** - Finishing Allowance in Z axis
- F** - Feed Rate

G72 Multiple Facing Cycle:**Example:**

O1005

G21 G98

-Initial Settings

G28 U0 W0

-Going to home position

M06 T1

-Tool Change Position No. 01

M03 S1500

-Spindle clockwise with 1500 RPM

G00 X33 Z5

-Tool Moving to Tool Entry Point of X33 Z5 at Rapid Traverse

G72 W0.5 R1

-Calling G72 Cycle and defining Cycle Parameters

G72 P1 Q2 U0.1 W0.1 F100

-Defining Cycle Parameters

N1 G01 Z-38 F100

G01 X32

G01 X24 Z-30

G01 Z-20

G02 X20 Z-16 R4

G01 Z-4

N2 G01 X18 Z0

G28 U0 W0

M06 T2

M03 S1800

G00 X33 Z5

G70 P1 Q2 S1800 F80

G28 U0 W0

M05

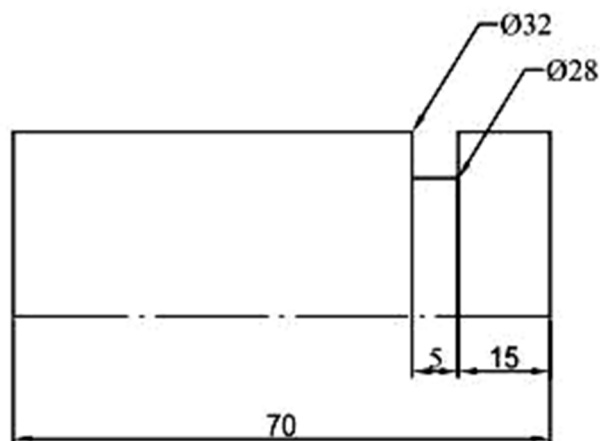
M30

G75 Grooving Cycle:

This cycle is designated for grooving. This cycle also requires two blocks are needed to specify the all parameters.

Format:**G75 R (e)****G75 X Z P Q F**

- R** -Return Amount, mm
- X** -Total Depth along X axis, mm
- Z** -Total Width along Z axis, mm
- P** -Depth of Cut in X axis (in Micron)
- Q** -Stepping distance in Z axis (in Micron)
- F** -Feed Rate, mm

Example:

O1009

G21 G98

G28 U0 W0

M06 T1

M03 S400

G00 X33 Z-18

G75 R1

G75 X28 Z-20 P50 Q1000 F40

G28 U0 W0

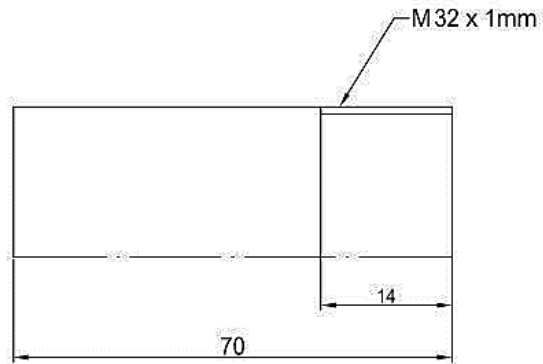
M05

M30

G76 Multiple Threading Cycle:

Thread cutting cycle can be commanded by the G76 command. This cycle also requires two blocks are needed to specify the all parameters.

Format:**G76 P (m) (r)(a) Q (Δ admin) R (d)****G76 X Z P (k) Q (Δ d) F****m** -No. of repeats for finishing operation**r** -Chamfering amount**a** -Tool angle, degree**Q** -Minimum Cutting Depth, (in Micron)**R** -Finishing Allowance, (in mm)**X** -Minor Diameter, mm**Z** -Thread Length, mm**P (k)** -Thread Height, (in Micron)**Q (Δ d)** -Depth of cut for first pass (in Micron)**F** Pitch of the thread, mm

Example:

O1010

G21 G98 -Initial Settings

G28 U0 W0 -Going to home position

M06 T1 -Tool Change Position No. 01

M03 S1500 -Spindle clockwise with 1500 RPM

G00 X32.5 Z5 -Tool Moving to Tool Entry Point X32.5 Z5 at Rapid Traverse

G76 P040060 Q50 R0.01 -Calling G76 Cycle and defining cycle parameters

G76 X30.774 Z-14 P613 Q100 F1

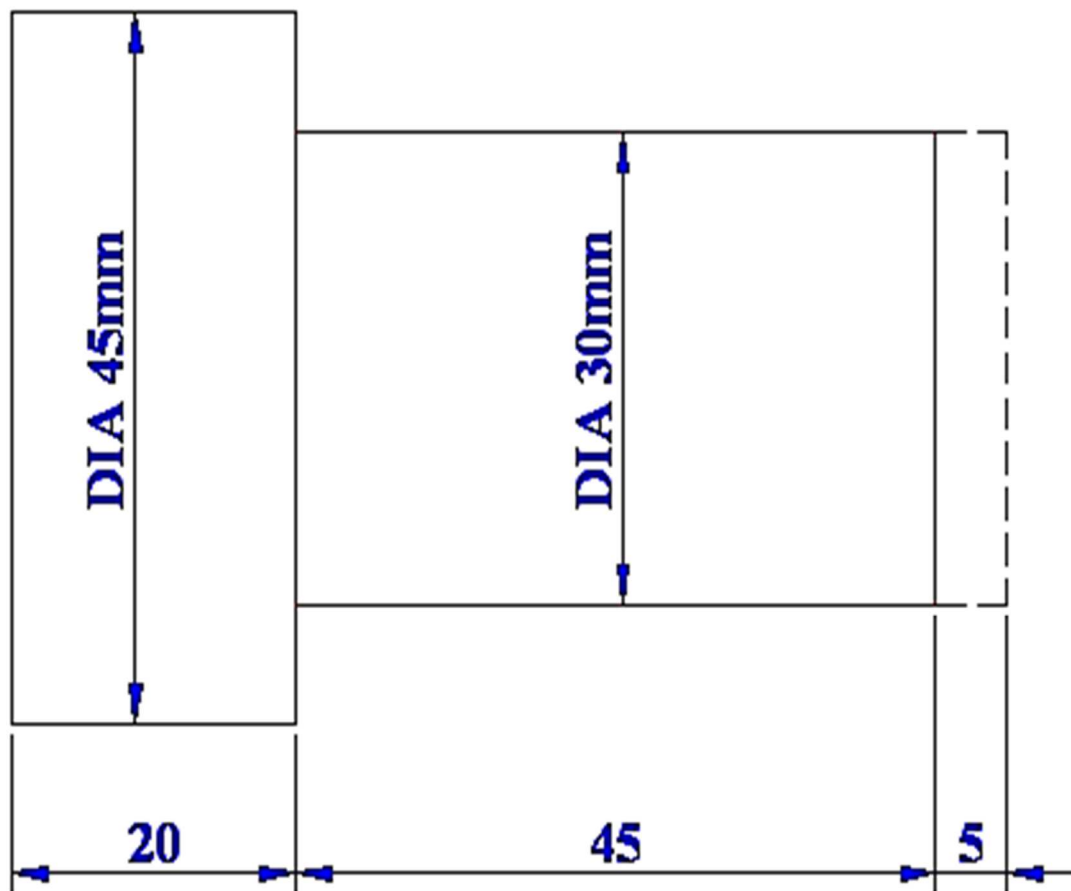
G28 U0 W0

M05

M30

SAMPLE PROGRAM FOR CNC **LATHE**

FACING & PLAIN TURNING OPERATION

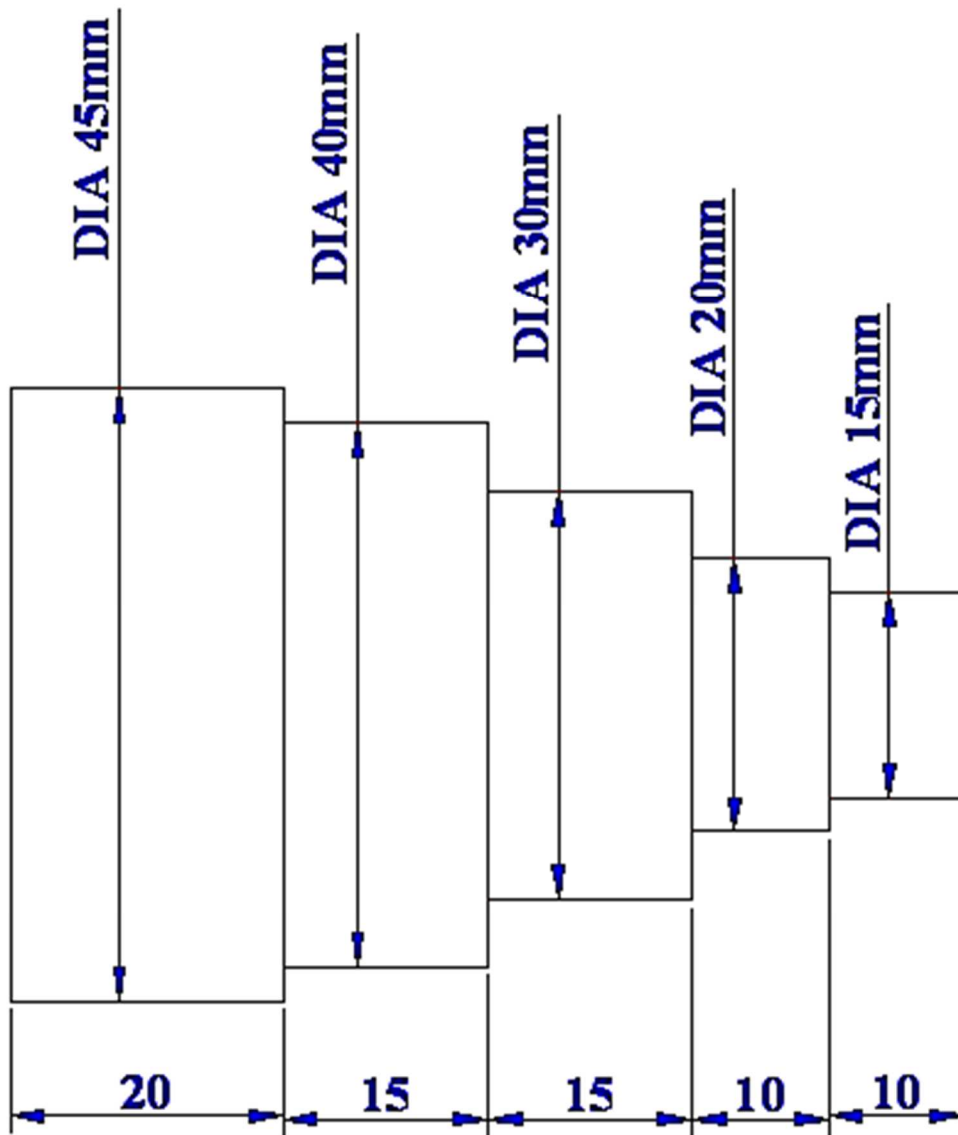


All dimensions are in 'mm'

PROGRAM FOR PLAIN TURNING OPERATION

PROGRAM	EXPLANATION
O 1111;	FANUC control & Program Name
G28 U0 W0;	Home positioning
T505;	Tool Number is 5
G97 M04 S3000 ;	Constant surface speeds cancel.
G00 X60. Z0. M8 ;	Initial position of tool & coolant ON.
G72 W1.0 R0.5 ;	Stock removal in Facing cycle.
G72 P10 Q20 U0 W0 F0.2	
N10 G00 Z-5. ;	Linear interpolation at Z axis.
N20 G01 X-1. ;	Linear interpolation at X axis.
G00 Z20. M5 ;	Tool movement to 20 mm in Z axis
G28 U0 W0 ;	Home positioning
T505;	Tool Number is 5
G50 S3000 M4;	Max. Spindle speed is 3000 rpm & Spindle ON.
G96 S300 M4;	Constant surface speed is 300 mm/min
G00 X60. Z2. ;	Initial position of tool
G71 U1.0 R0.5;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15;	
N10 G00 X30.;	Starting of block.
G01 Z-50.;	Linear interpolation at Z axis.
G01 X45.;	Linear interpolation at X axis.
N20 G01 Z-70.;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
M30 ;	Program end & rewind.

STEP TURNING OPERATION

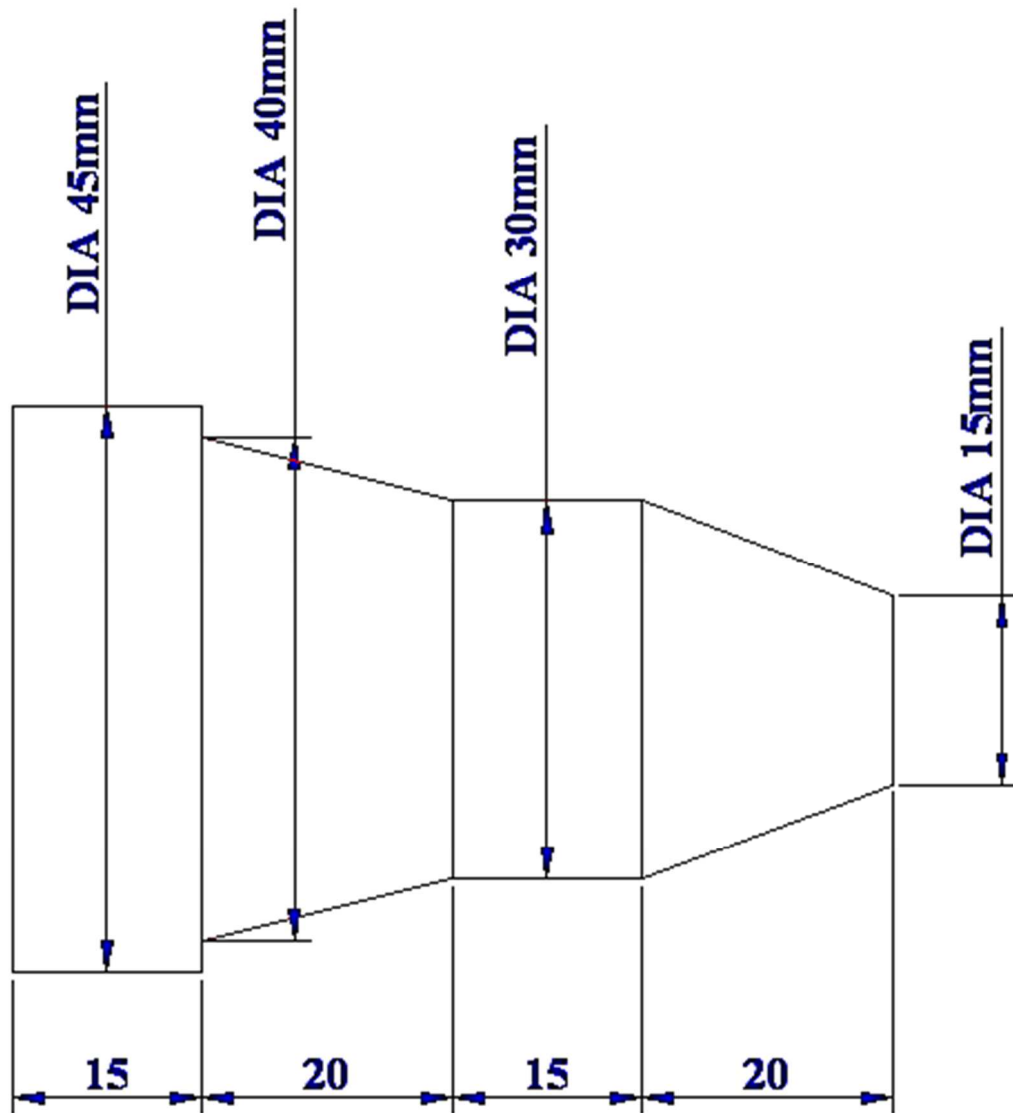


All dimensions are in 'mm'

PROGRAM FOR STEP TURNING OPERATION

PROGRAM	EXPLANATION
O 2233 ;	FANUC control & Program Name
G28 U0 W0 ;	Home positioning
T505 ;	Tool Number is 5
G50 S3000 M4 ;	Max. Spindle speed is 3000 rpm & Spindle ON.
G96 S300 M4 ;	Constant surface speed is 300 mm/min
G00 X60. Z2. M8 ;	Initial position of tool & coolant ON.
G71 U1.0 R0.5 ;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15 ;	
N10 G00 X15. ;	Starting of block.
G01 Z0.;	Linear interpolation at Z axis.
G01 Z-10. ;	Linear interpolation at Z axis.
G01 X20. ;	Linear interpolation at X axis.
G01 Z- 20. ;	Linear interpolation at Z axis.
G01 X30. ;	Linear interpolation at X axis.
G01 Z-35. ;	Linear interpolation at Z axis.
G01 X40. ;	Linear interpolation at X axis.
G01 Z-50.;	Linear interpolation at Z axis.
G01 X45.;	Linear interpolation at X axis.
N20 G01 Z-70. ;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
M30;	Program end & rewind.

TAPER TURNING OPERATION

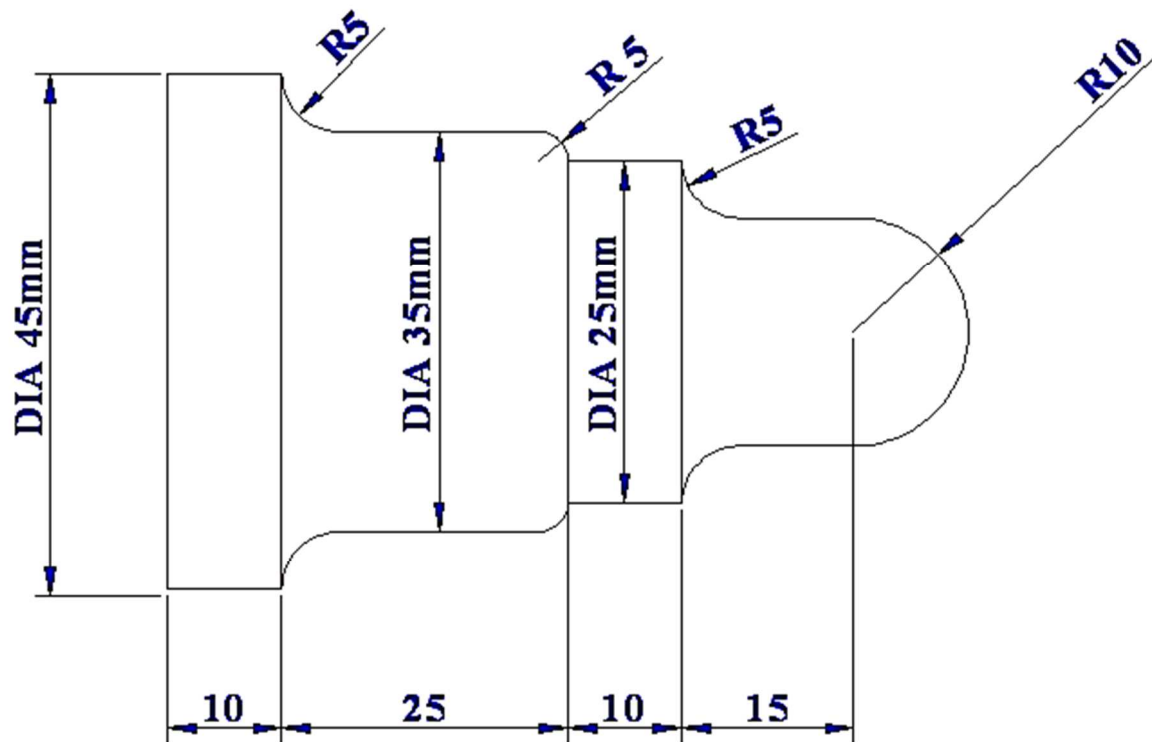


All dimensions are in 'MM'

PROGRAM FOR TAPER TURNING OPERATION

PROGRAM	EXPLANATION
O 3333 ;	FANUC control & Program Name
G28 U0 W0 ;	Home positioning
T505 ;	Tool Number is 5
G50 S3000 M4 ;	Max. Spindle speed is 3000 rpm & Spindle ON.
G96 S300 M4 ;	Constant surface speed is 300 mm/min
G00 X60. Z2. M8 ;	Initial position of tool & coolant ON.
G71 U1.0 R0.5 ;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15 ;	
N10 G00 X15. ;	Starting of block.
G01 Z0.;	Linear interpolation at Z axis.
G01 X30. Z-20. ;	Linear interpolation (Taper line) at X & Z axis.
G01 Z-35. ;	Linear interpolation at Z axis.
G01 X40. Z- 55 ;	Linear interpolation (Taper line) at X & Z axis.
G01 X45. ;	Linear interpolation at X axis.
N20 G01 Z-70. ;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
M30;	Program end & rewind.

CIRCULAR INTERPOLATION OPERATION

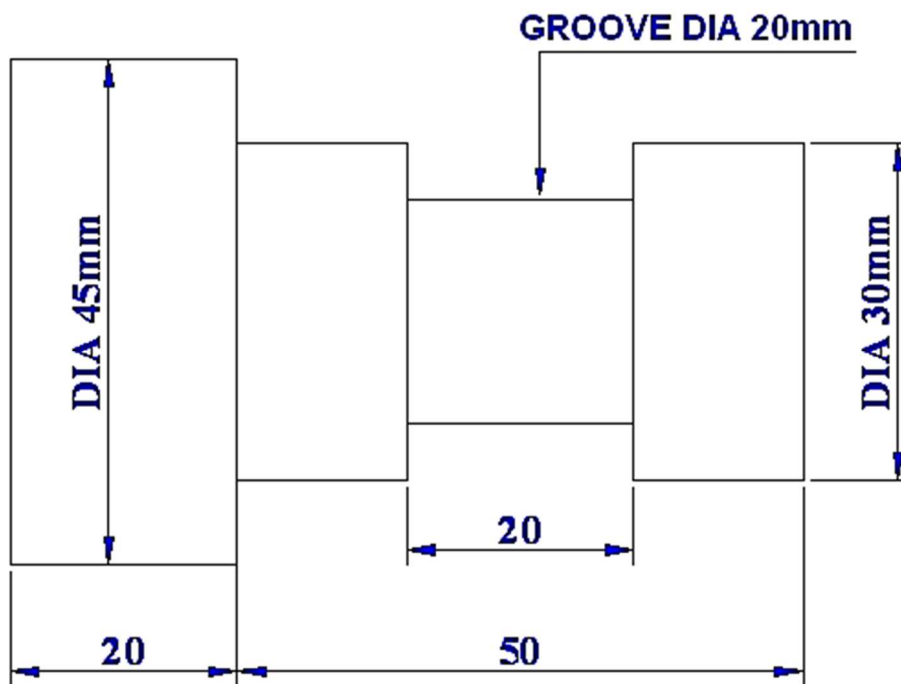


All dimensions are in 'mm'

PROGRAM FOR CIRCULAR INTERPOLATION OPERATION

PROGRAM	EXPLANATION
O 4444 ;	FANUC control & Program Name
G28 U0 W0 ;	Home positioning
T505 ;	Tool Number is 5
G50 S3000 M4 ;	Max. Spindle speed is 3000 rpm & Spindle ON.
G96 S300 M4 ;	Constant surface speed is 300 mm/min
G00 X60. Z2. M8 ;	Initial position of tool & coolant ON.
G71 U1.0 R0.5 ;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15 ;	
N10 G00 X0. ;	Starting of block.
G01 Z0.	Linear Interpolation at Z axis.
G03 X20. Z-10. R10. ;	Circular Interpolation (CCW)
G01 Z-20. ;	Linear Interpolation at Z axis.
G02 X25. Z-25. R5. ;	Circular Interpolation (CW)
G01 Z-35. ;	Linear interpolation at Z axis.
G03 X35. Z-40 R5. ;	Circular Interpolation (CCW)
G01 Z-55. ;	Linear Interpolation at Z axis.
G02 X45. Z-60. R5. ;	Circular Interpolation (CW)
N20 G01 Z-70. ;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
M30;	Program end & rewind.

PLAIN TURNING & GROOVING OPERATION

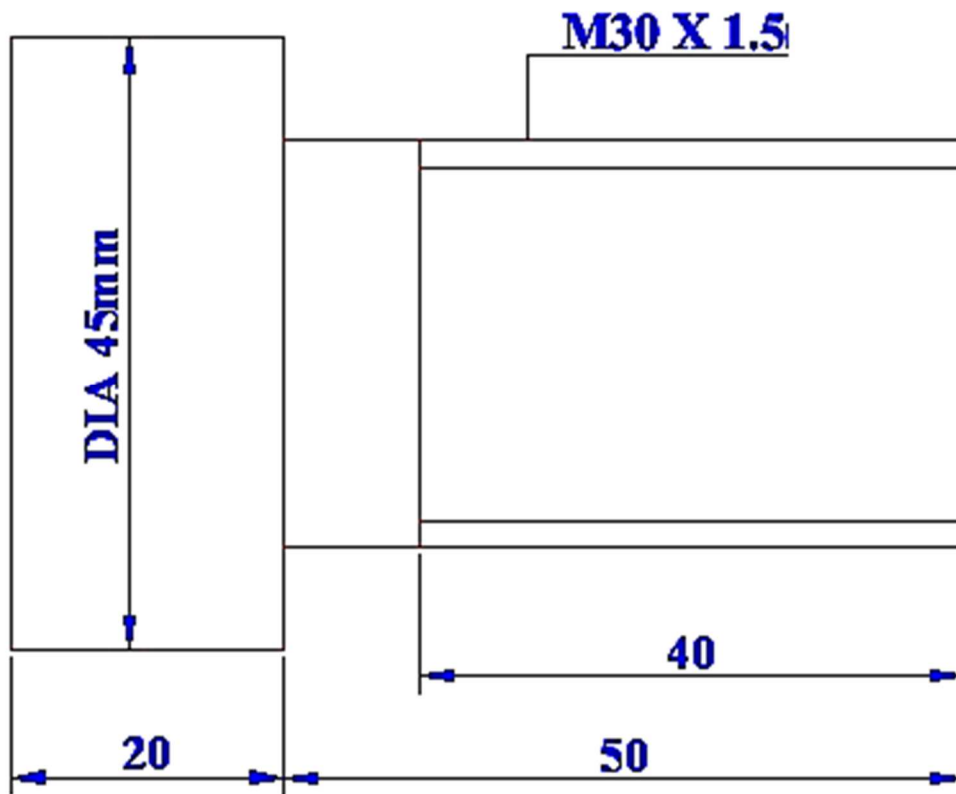


All dimensions are in 'mm'

PROGRAM FOR PLAIN TURNING & GROOVING OPERATION

PROGRAM	EXPLANATION
O 5555;	FANUC control & Program Name
G28 U0 W0;	Home positioning
T505;	Tool Number is 5
G50 S3000 M4;	Max. Spindle speed is 3000 rpm
G96 S300 M4;	Constant surface speed is 300 mm/min
G00 X60. Z2. ;	Initial position of tool
G71 U1.0 R0.5;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15;	
N10 G00 X30.;	Starting of block.
G01 Z0.	Linear Interpolation at Z axis.
G01 Z-50.;	Linear interpolation at Z axis.
G01 X45.;	Linear interpolation at X axis.
N20 G01 Z-70.;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
T707;	Tool Number is 7 (GROOVE TOOL)
G50 S2000 M4;	Starting of block.
G96 S300 M4;	Linear interpolation at Z axis.
G0 X60. Z2.;	Linear interpolation at X axis.
G00 X60. Z-18.;	Ending of block.
G75 R0.5;	Finishing cycle.
G75 X20. Z-35. P1000 Q1000 F0.1;	Tool return @ X axis.
G00 X65. Z5.;	Tool return @ Z axis .
G28 U0 W0	Home positioning
M30 ;	Program end & rewind.

PLAIN TURNING & THREADING OPERATION



All dimensions are in 'mm'

PROGRAM FOR PLAIN TURNING & THREADING OPERATION

PROGRAM	EXPLANATION
O 7777;	FANUC control & Program Name
G28 U0 W0;	Home positioning
T505;	Tool Number is 5
G50 S3000 M4;	Max. Spindle speed is 3000 rpm
G96 S300 M4;	Constant surface speed is 300 mm/min
G00 X60. Z2. ;	Initial position of tool
G71 U1.0 R0.5;	Stock removal in turning cycle.
G71 P10 Q20 U0.5 W.02 F0.15;	
N10 G00 X30.;	Starting of block.
G01 Z0.	Linear Interpolation at Z axis.
G01 Z-50.;	Linear interpolation at Z axis.
G01 X45.;	Linear interpolation at X axis.
N20 G01 Z-70.;	Ending of block.
G70 P10 Q20 F0.1;	Finishing cycle.
G00 X80. M9;	Tool return @ X axis & coolant OFF.
G00 Z40. M5;	Tool return @ Z axis & spindle OFF.
G28 U0 W0;	Home positioning
T202;	Tool Number is 7 (THREADING TOOL)
G97 M3 S500;	Constant surface speeds cancel.
G00 X30. Z5.;	Initial position of tool
G76 P020060 Q50 R50;	Threading cycle operation
G76 X28.05 Z-40 P975 Q200 F1.5;	
G00 X50. Z50.;	Tool return @ Z axis & X axis.
G28 U0 W0;	Home positioning
M30 ;	Program end & rewind.

CNC MILLING

CNC MILLING MACHINE (CNC MACHINING CENTRE)

CNC Machining center is a machine tool capable of performing multiple machining operations on workpiece in one setup under CNC system.

Typical machining operations performed on machining centre include Milling, Drilling Boring, Reaming and Tapping.

CNC machining centres are usually equipped with the following features to reduce Nonproductive time.

- Automatic Tool Changing
- Automatic workpiece positioning
- Automatic pallet changer

CNC machining centres are classified as follows

1. Vertical machining centre
2. Horizontal machining centre
3. Universal machining centre

Vertical Machining centre

A vertical machining centre has its spindle on a vertical axis relative to the work table.

A vertical machining centre is typically used for workpiece that require machining from the top.

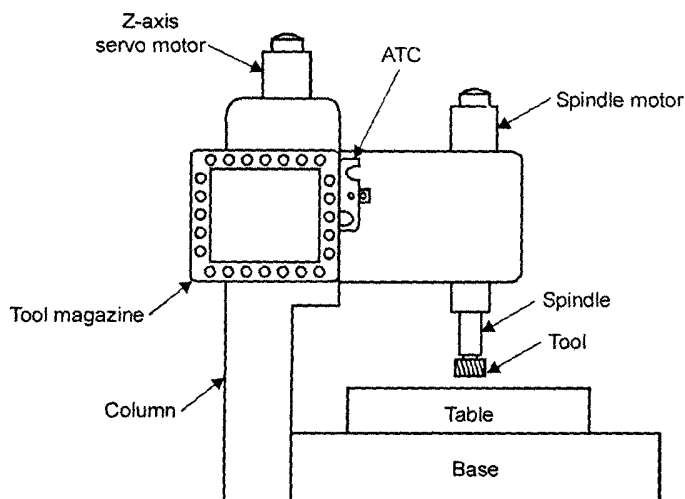


Figure 3: CNC Vertical machining centre

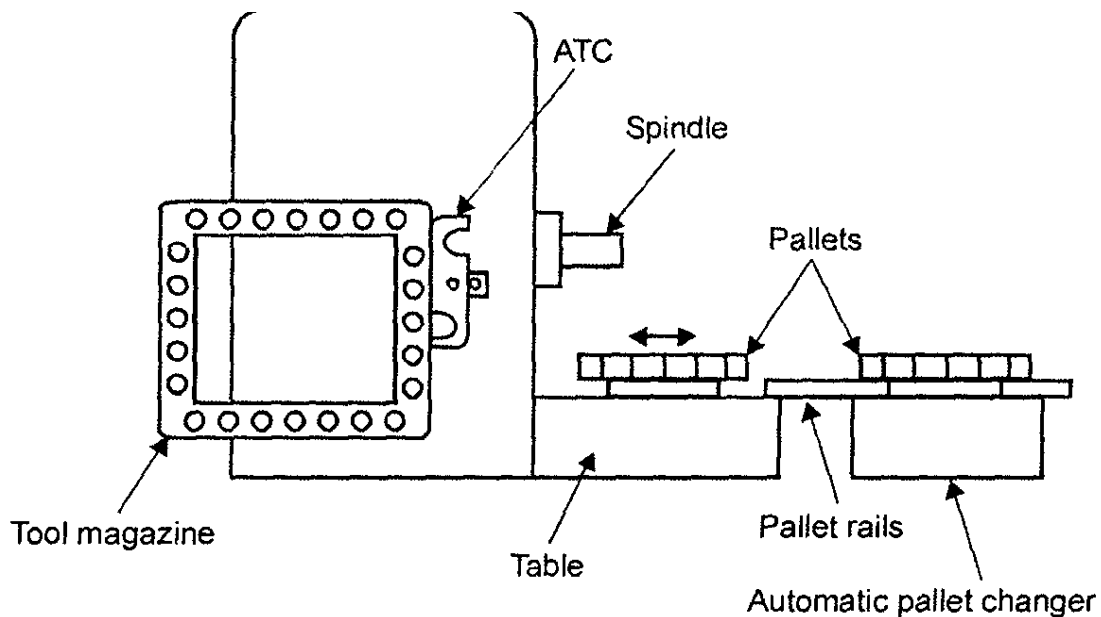
Most of the general machines come with three axes. However, machines with more than three axes also are available.

For example, the spindle can be swiveled in one (or) two axes. These are required for machining sculptured surfaces.

Horizontal Machining centre

A horizontal machining centre has its spindle on a horizontal axis. These machines are used for machining heavier workpiece 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 horizontal machining centre provides a fourth axis. These machines are used for machining the prismatic (box like) components.

Figure 4: CNC Horizontal machining centre

The availability of rotary table makes it possible for machining 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 axis horizontal machining centre, then it will be possible to machine complex.

G – CODES & M – CODES FOR CNC MILLING

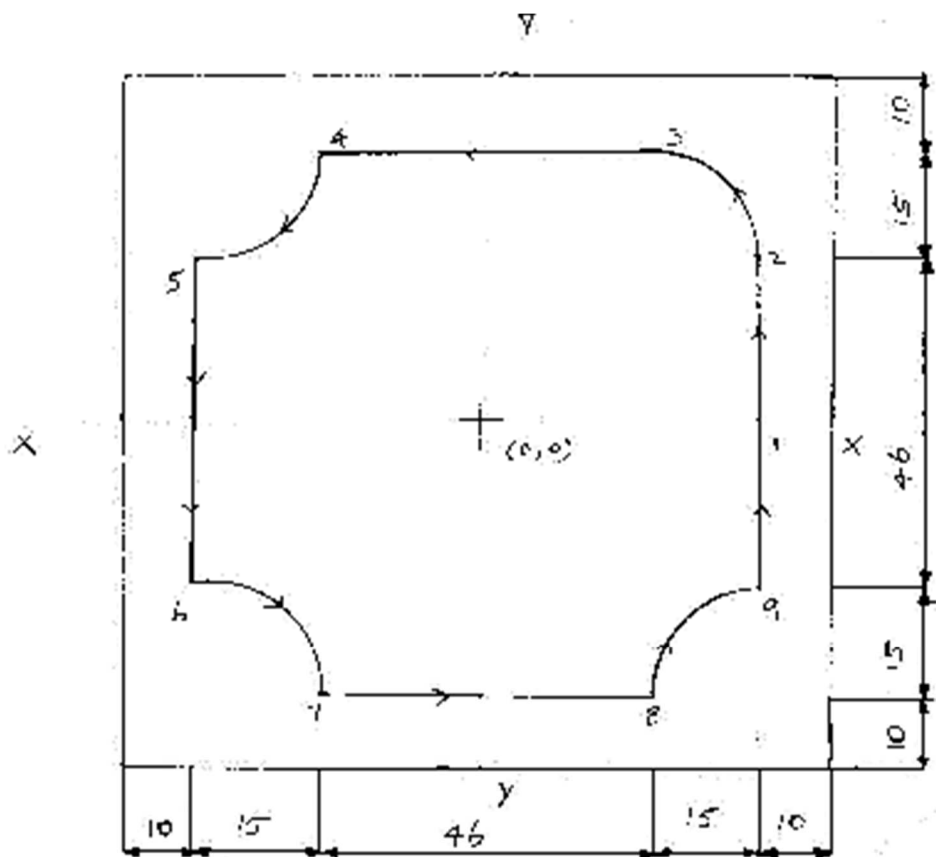
G CODES (PREPARATORY FUNCTION)		
G00	-	Rapid traverse positioning
G01	-	Linear interpolation (cutting feed)
G02	-	Circular interpolation CW direction
G03	-	Circular interpolation CCW direction
G04	-	Dwell
G15	-	Polar co-ordinate command cancel
G16	-	Polar co-ordinate command
G17	-	XY plane section
G18	-	ZX plane section
G19	-	YZ plane section
G20	-	Input in inch
G21	-	Input in mm
G28	-	Automatic return to reference position
G40	-	Cutter compensation cancel
G41	-	Cutter compensation left
G42	-	Cutter compensation right
G43	-	Tool length compensation +
G44	-	Tool length compensation –
G49	-	Tool length compensation cancel
G53	-	Machine co-ordinate system setting cancel
G54	-	Work piece coordinate system 1 selection
G55	-	Work piece coordinate system 2 selection

G56	-	Work piece coordinate system 3 selection
G57	-	Work piece coordinate system 4 selection
G58	-	Work piece coordinate system 5 selection
G59	-	Work piece coordinate system 6 selection
G73	-	Peck drilling cycle
G74	-	Left – hand tapping cycle
G80	-	Canned cycle cancel
G81	-	Drilling cycle
G82	-	Drilling cycle (or) counter boring cycle
G83	-	Peck drilling cycle
G84	-	Tapping cycle
G85	-	Boring cycle
G86	-	Boring cycle
G88	-	Boring cycle
G89	-	Boring cycle
G90	-	Absolute dimensioning programming
G91	-	Incremental dimensioning programming
G94	-	Feed per minute
G95	-	Feed per revolution
G96	-	Constant surface speed control
G97	-	Constant surface speed control cancel
G98	-	Canned cycle return to initial level
G99	-	Canned cycle return to reference level

M CODES (MISCELLANEOUS FUNCTION)		
M00	-	Program stop
M01	-	Optional stop
M02	-	Program end
M03	-	Spindle rotation CW direction
M04	-	Spindle rotation CCW direction
M05	-	Spindle stop
M06	-	Tool change
M07	-	Coolant through tool ON
M08	-	Coolant ON
M09	-	Coolant OFF
M30	-	Program end and rewind
M65	-	X – Axis mirror ON
M66	-	Y – Axis mirror ON
M68	-	All axis mirror OFF
M10	-	Vice open
M11	-	Vice close
M13	-	Coolant ON spindle rotation CW direction
M14	-	Coolant ON spindle rotation CCW direction
M98	-	Sub program call
M99	-	Sub program exit

SAMPLE PROGRAM FOR CNC MILLING

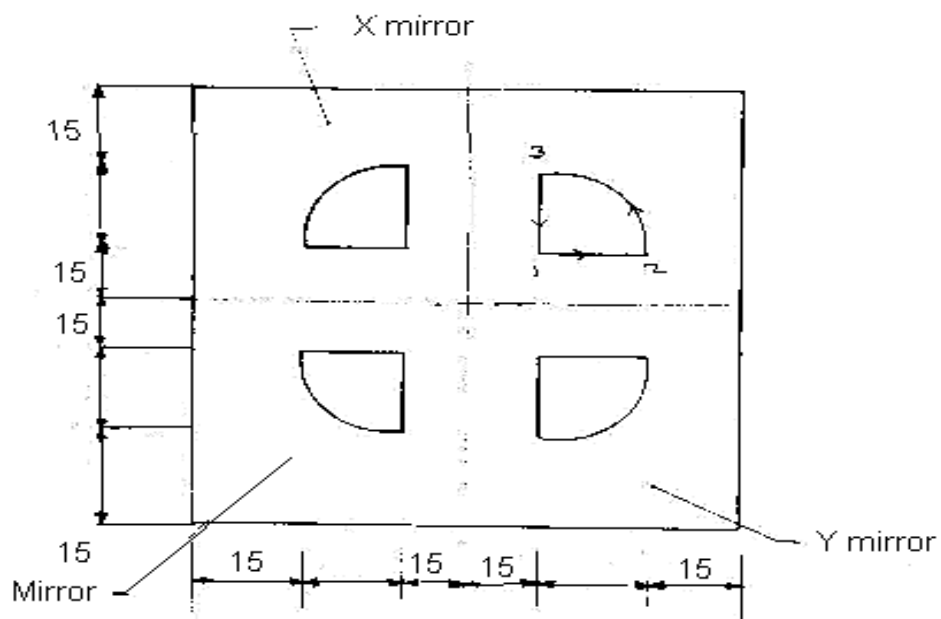
Linear and Circular Interpolation Operation



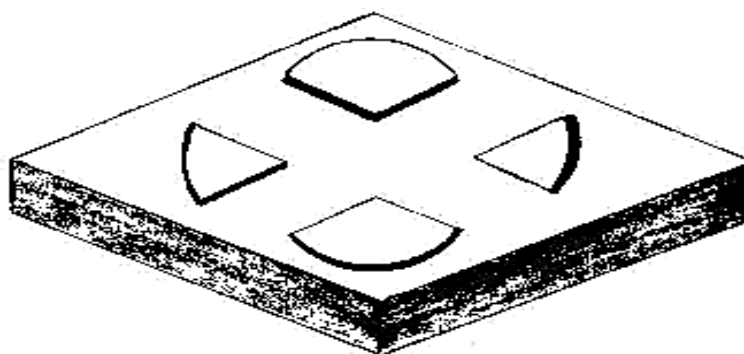
ALL DIMENSIONS ARE IN MM

PROGRAM	EXPLANATION
O 1234	O – Fanuc control name
G17 G21 G40 G80 G54 G94	1234 – program name
M06 T10	M06 – Tool change
M03 S1000	M03 – Spindle rotation (clockwise direction)
M08	G91 – Incremental position
G91 G28 X0 Y0 Z0	G28 – Return to reference point
G90 G00 X0 Y0 Z2	G90 – Absolute position
G00 X38 Y0 Z2	G00 – Rapid position
G01 X38 Y0 Z -5 F200	G01 – Linear interpolation
G01 Y23	G02 – Circular interpolation (clockwise direction)
G03 X23 Y38 R15	G03 – Circular interpolation (anti-clockwise direction)
G01 X -23 Y38	G17 – X-Y plane selection
G02 X -38 Y23 R15	G21 – Data input in mm
G01 X -38 Y -23	G40 – Cutter compensation cancel
G02 X -23 Y -38 R15	G80 – Canned cycle cancel
G01 X23 Y -38	G54 – Co-ordinate setting
G02 X38 Y -23 R15	G94 – Feed in mm/min.
G01 X38 Y0	M08 – Coolant ON
G00 Z2	M09 – Coolant OFF
G91 G28 X0 Y0 Z0	M05 – Spindle speed
M09	M30 – Program end
M05	
M30	

MIRROR IMAGE BY USING SUB PROGRAM



ALL DIMENSIONS ARE IN MM



SUB PROGRAM:	EXPLANATION
O 2456	O – Fanuc control name
G90 G00 X0 Y0 Z2	2456 – program name
X15 Y15	G90 – Absolute position
G01 X15 Y15 Z -10 F200	G00 – Rapid position
X35 Y15	G01 – Linear interpolation
G03 X15 Y35 R20	G03 – Circular interpolation (anti-clockwise direction)
G01 X15 Y15	M99 – Sub program end
G00 Z2	G17 – X-Y plane selection
M99	G21 – Input in mm
MAIN PROGRAM:	G40 – Cutter compensation cancel
O 1234	G80 – Canned cycle cancel
G17 G21 G40 G80 G54 G94	G54 – Co-ordinate setting
G91 G28 X0 Y0 Z0	G94 – Feed in mm/min.
M06 T10	G91 – Incremental position
M03 S1000	G28 – Return to reference point
M08	M06 – Tool change
M98 C2456	M03 – Spindle rotation in clockwise direction
M21	M08 – Coolant ON
M98 C2456	M98 – Sub-program call
M23	M21 – X mirror ON
M22	M22 – Y mirror ON
M98 C2456	M23 – X, Y mirror OFF
M23	M09 – Coolant OFF
M21	M05 – Spindle stop
M22	M30 – Program end
M98 C2456	
M23	
G91 G28 X0 Y0 Z0	
M09	
M05	
M30	