ROHINI

COLLEGE OF ENGINEERING & TECHNOLOGY

Unit-V

CNC machine Tool & Part Programming

Introduction:—

* Numerical control has been developed out of need for hipher productivity, lower cost & more precise manufacturing.

* In NC System, operation instructions are inputted to the machine as a code of numbers, letters, & special characters. These coded instructions are then automatically carried out in the machine tool in a predetermined sequence with pre set speed, feed, etc., without human interven-tion.

NC & its Components:—

* NC stands for numerical control. It is a technique of automatically operating the machine tool based on a code of letters, numbers and special characters.

* The complete set of coded instructions responsible for executing an operation is called a

"Part Program".

* This Part Program is translated into electric Signals to drive Various motors to operate the m/c to Carry out the required operations.

* NC System Contains the following Components.

1. Program of Instructions
2. Tape Punch
3. Tape Reader
4. Machine Controller
5. NC machine.

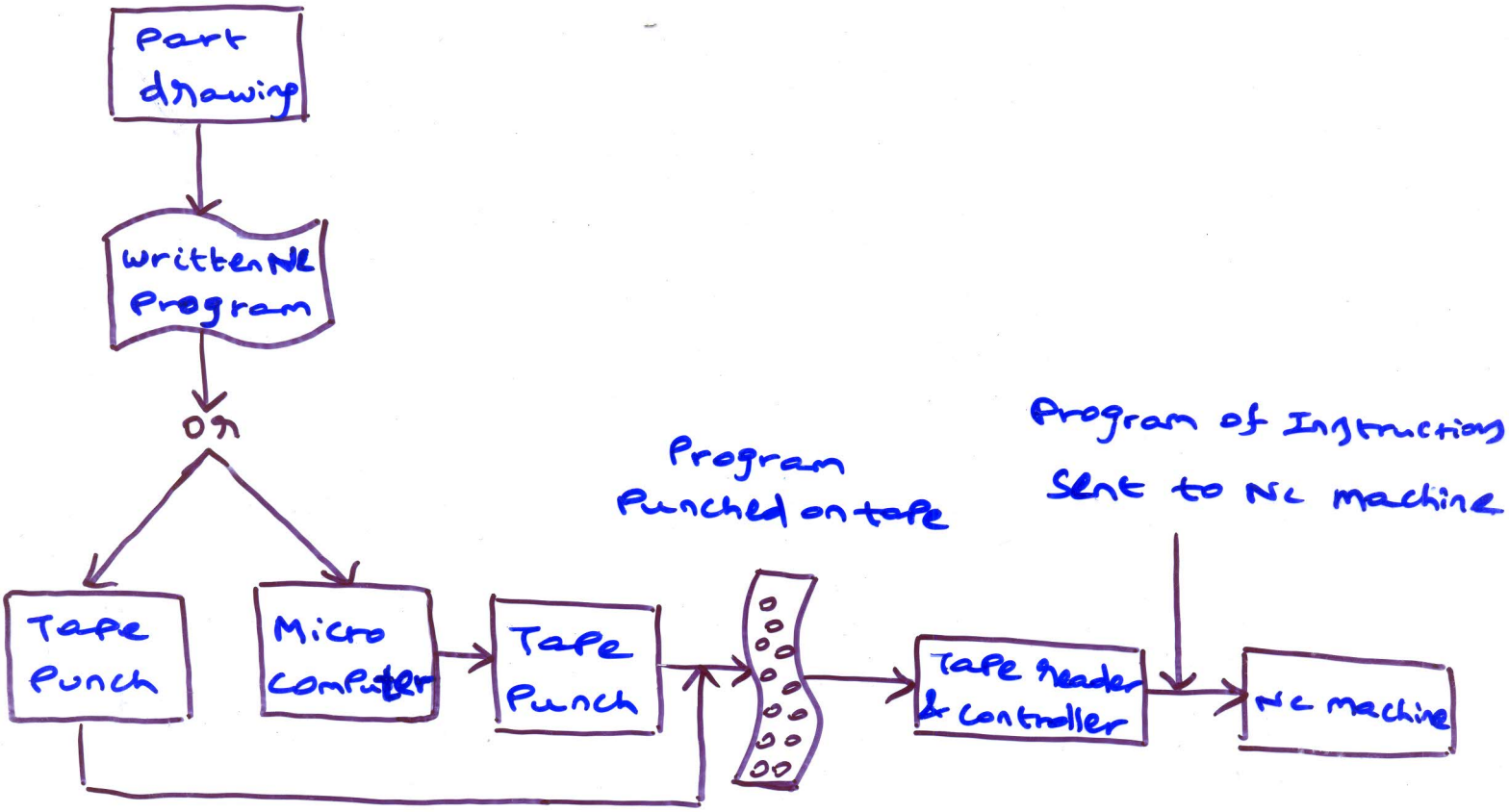
(1) Program of Instructions:-

* The Program of instructions, often called Part Program is the detailed Set of instructions for the NC machine to produce a component. The Part program is a mixture of Alphabetic Codes, numeric data & Special Characters.

* This Part Program is Punched in a input media (usually paper tape) in a Specified format

* This input is read by a tape Reader which Converts the instructions on a Paper tape into the electric Signals & transfers it to the machine Controller to operate the machine Slides & to generate Specific

Surfaces on the job.



(2) Tape Punch:—

* Usually it is a paper tape of 1" width. Paper mylar, Aluminium mylar (or) plastics are also used as a tape materials.

* Punching machine (flexo writers) of various types are used to key in program of instructions to tapes.

* Presently tapes are prepared by micro-computers by keying in the information from the manuscript.

① once the entire program has been entered, it is checked & corrected if needed.

② Then the computer activates the tape punching unit to produce the tape.

(3) the computer can also generate print-out through its printer. (4)

(3) Tape Reader:—

A tape reader reads the hole pattern on the tape & converts the pattern to a corresponding electric signal.

(4) Machine Controller:—

* Controller receives the electric signals from the tape reader & causes NC machine to respond.

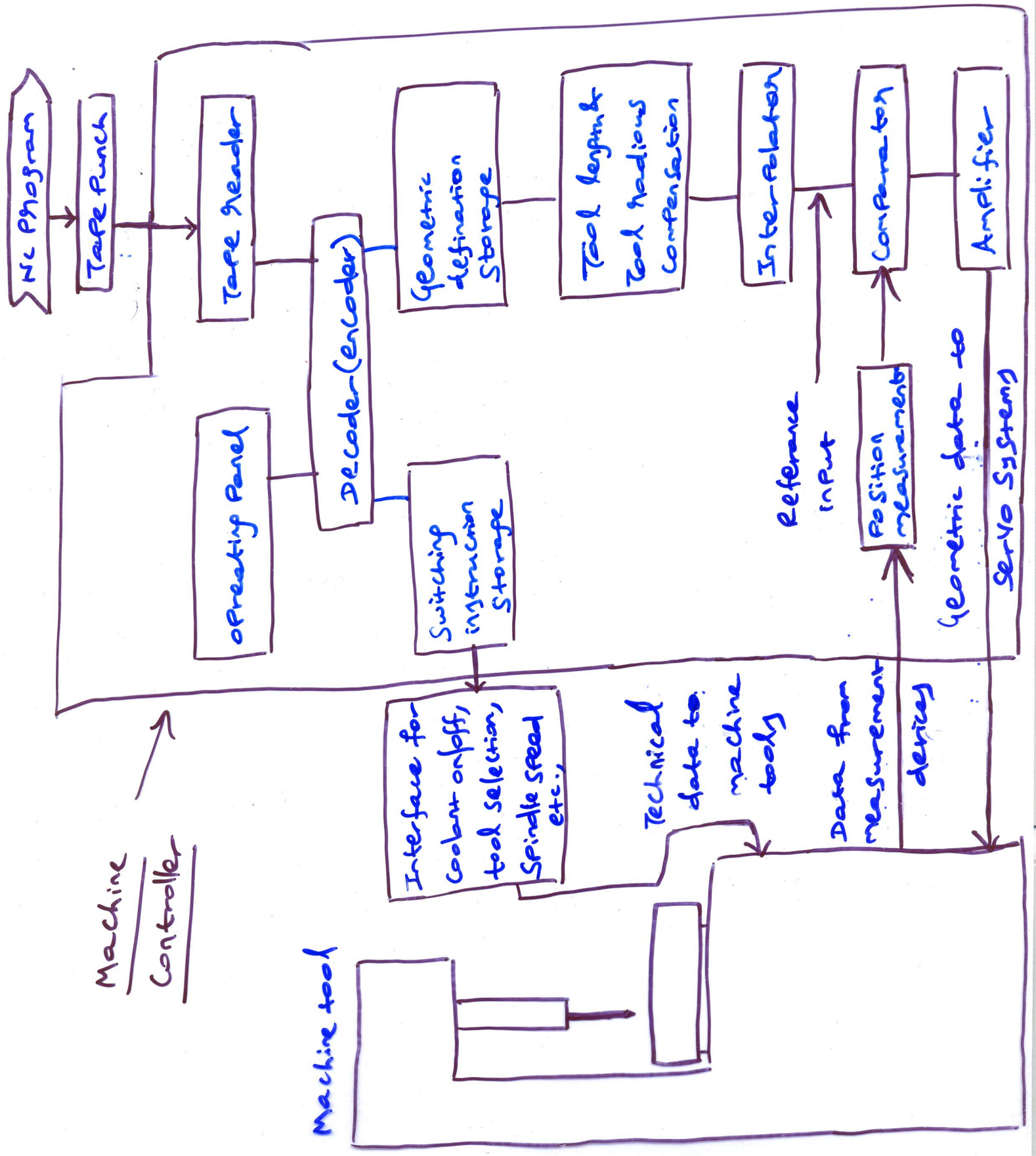
* It containg a decoder/encoder, an interpolater and facilities to execute auxillary functions which are machine dependent.

* the decoder/encoder receives data & stores them in two separate memory locations. one for the part geometry data & the other for process data. Process data includes switching functions for adjusting feed rates, spindle speeds, tool changes, cutting fluid applications etc., Geometric data consists information about tool motion, tool length, tool radious, tool compens-ation etc.,

* The function of interpolater is to break down the curves (or) simple line into small individual increments

for each controlled motion of the machine tool.

* Controller also interfaces drive motors, transducers and other control functions of the machine tool.



NC machine:—

* NC machine responds to the electric signals from the controller. Accordingly the machine executes various slide motions & spindle rotations to manufacture a part.

* Transducers are fitted to feed back

(1) the R.P.M of the spindle (2) the amount of cut on the job.

* NC machine tools range from simple spindle drilling machine to complex machines having multiple motions, tool changers, high capacity tool magazines and multi axis control.

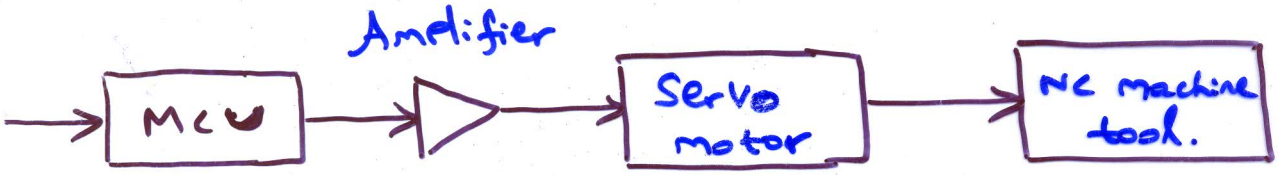
Position & Motion Control in NC machine:—

In NC machine, there is a servo control unit which is a group of electrical, mechanical, hydraulic & pneumatic devices used to control the slide position of NC machine tool. The servo control unit is classified into two types.

a) Open loop system

b) Closed loop system.

Open loop System:-



* It involves feeding of tape, interpretation of information by a tape reader, storing the data in the buffer storage.

* After storing, it is converted into electrical signal and the electrical signal is sent to the machine control unit [MCU]

* the control unit is connected to servo control unit which controls the slide movement.

* In open loop system, there is no feedback devices to ensure whether the obtained slide movement is same as desired (or) not and if not, what error is present.

Closed loop System:-

* It is similar to open loop system, but it carries an additional feed back device

* A feed back device is nothing but an transducer

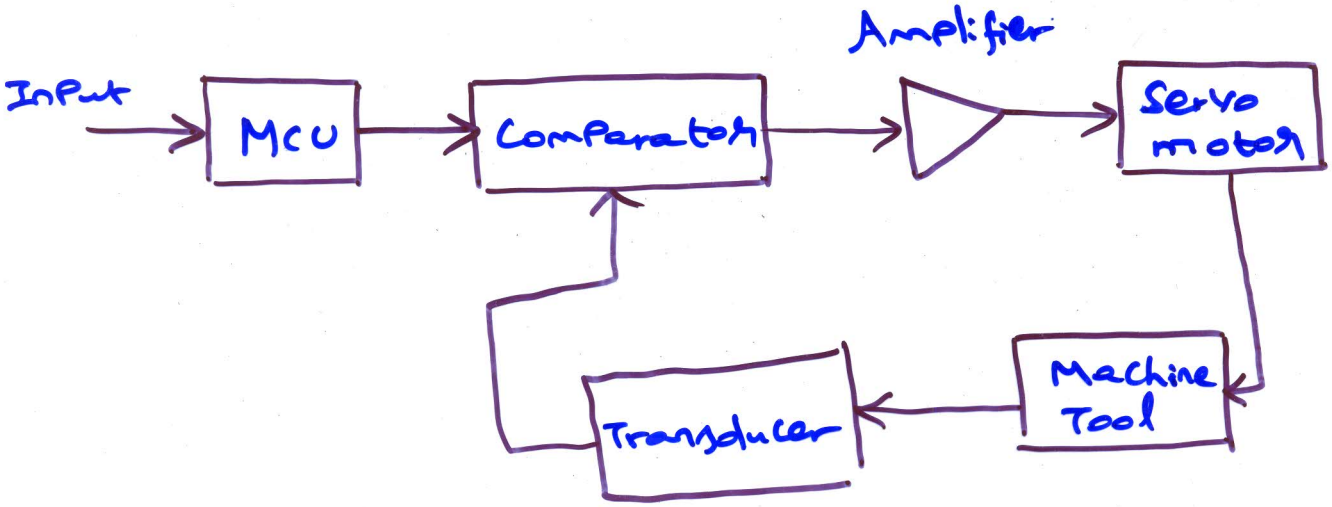
accompanied by a comparator.

* As this is similar to open loop system, the motion is same upto the servo control.

* the transducer fed back the obtained slide displacement to the comparator.

* the comparator compares the obtained slide displacement with the applied slide motion and error if any, is fed back to the servo control unit through an amplifier

* the servo control unit sends correct commands to servo motor & the cycle continues.



* the transducer used in the closed loop system is classified into two types.

a) Analog transducer

b) Digital transducer

a) Analog transducer:—

* It produces a variable electrical voltage which varies with the rotational speed of the shaft.

* This voltage can be easily measured and converted into the linear distances moved by the slides.

b) Digital transducer:—

* It converts the rotary motion of machine screw into the countable electric pulses.

* The number of electric pulses indicates the linear distance moved by the table corresponding to the lead screw rotation.

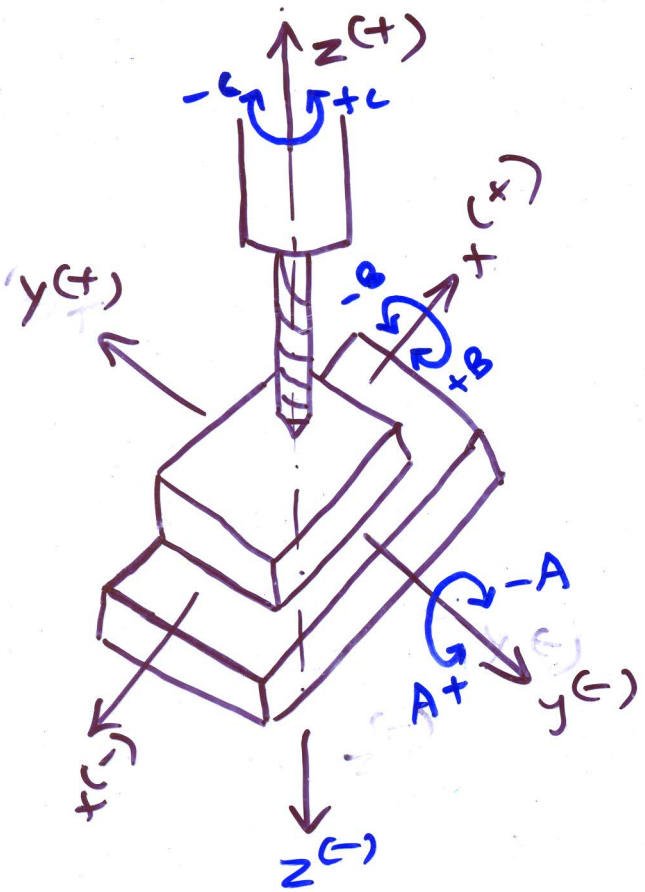
NC machine - Axis of motion:—

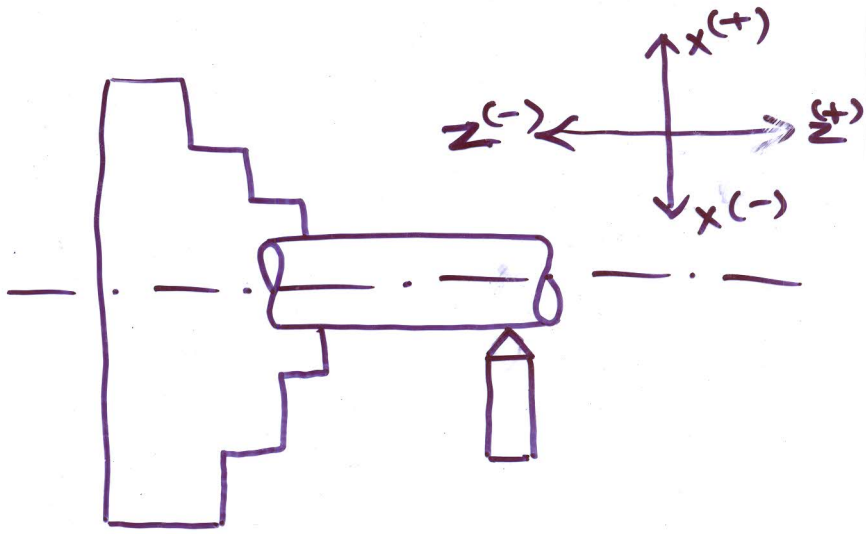
* The location of NC tool at any point in the NC machine is mentioned by Cartesian Co-ordinate System. It consists of x, y, z axes which are mutually perpendicular to each other.

* The NC machine axis of movement follows the right hand rule

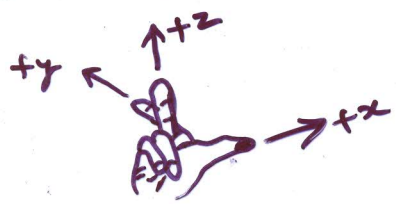
* The axes [x, y, z] of motion in NC machine are specified as follows. (10)

1. Z-axis is always spindle axis
2. X-axis is always horizontal axis & parallel to the surface of the work
3. Y-axis is perpendicular to both X-axis & Z-axis.

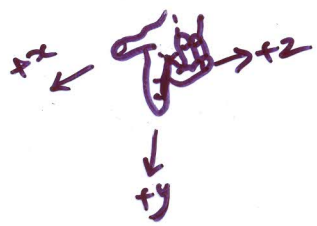




CNC milling



CNC turning



* Rotary motion

1. Rotation about an axis parallel to X-axis is "A."

(2) Rotation about an axis Parallel to Y axis is "B"

(3) Rotation about an axis Parallel to Z axis is "C"

* U, V, W axes are Parallel to X, Y & Z axes respectively.

(11)

Classification of NC System:-

(1) According to the motion Control of tools in NC System.

(1) Point-to-Point NC System

(2) Straight Cut NC System

(3) Contouring NC System

(1) Point to Point (PTP) System:-

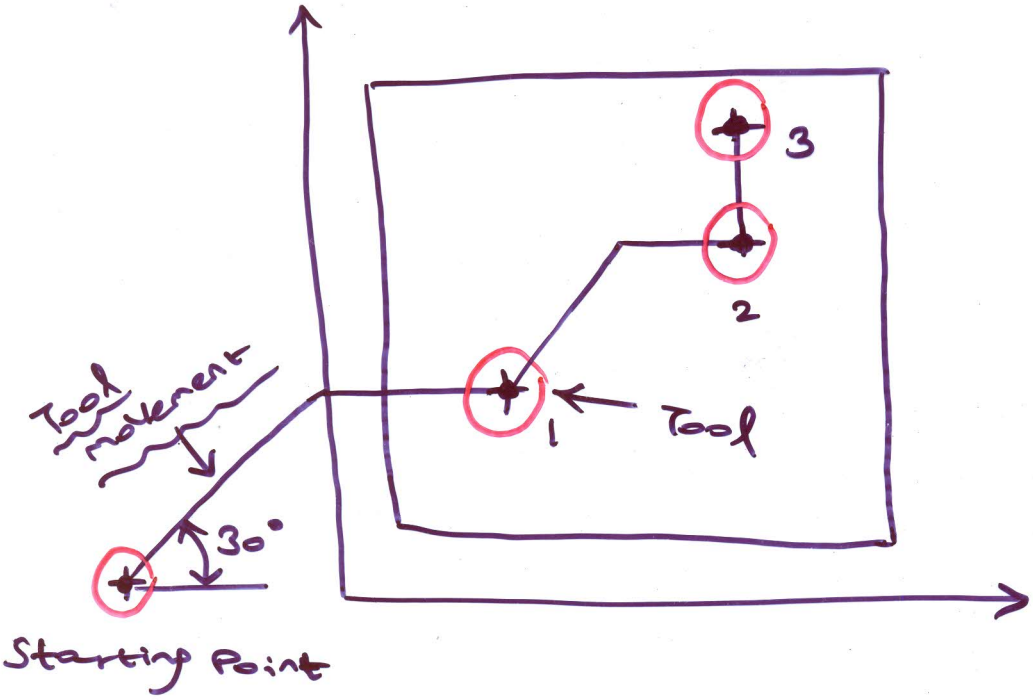
* It refers to the motion of tool at a faster rate to a Point followed by a manufacturing operation at that Point. The drilling operation is one of the operation inwhich PTP System is adopted

* As Shown in figure, the tool is moved from a starting Point to Point 1 & drilling ofn

is done at Point 1. Then the tool is moved to Point 2 & Point 3 followed by the drilling operation at the corresponding Points.

example

NC drilling machine



(2) Straight line (or) Straight Cut System:-

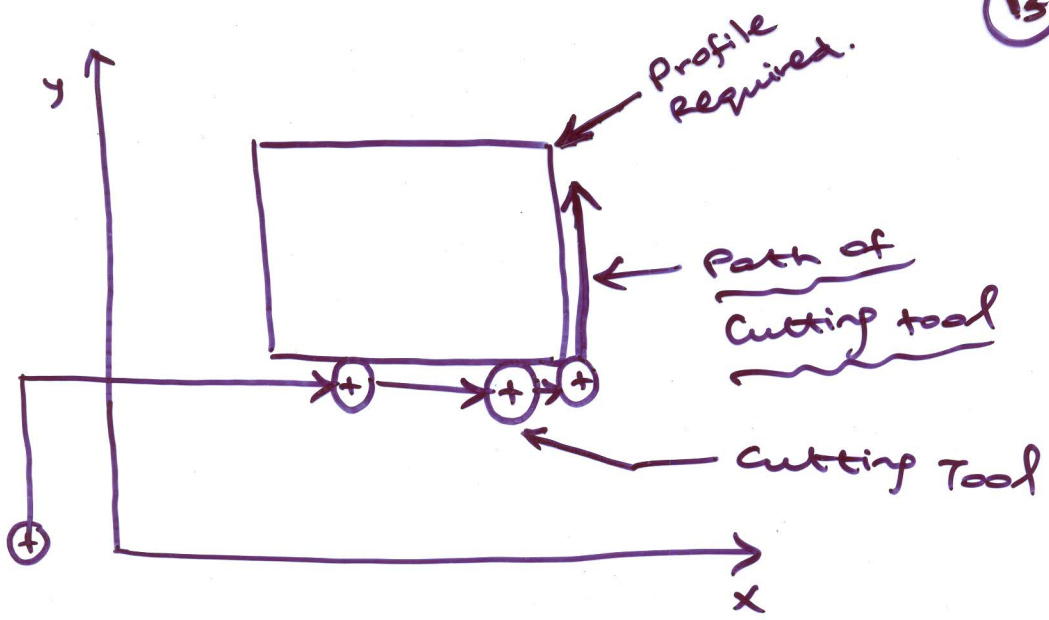
* In Straight Cut NC, the tool moves Parallel to one of the major axis at a desired Rate Suitable for machining.

* It is not possible to combine the axis of motion. Hence the tool motion is only along X-axis, Y-axis, Z-axis. Due to this, Angular Cuts can not be Produced in this type of machine.

* example :-

Milling workpieces of Rectangular Profile

* Any NC machine tool capable of Straight Cut movement can perform Point-to-Point operation also.



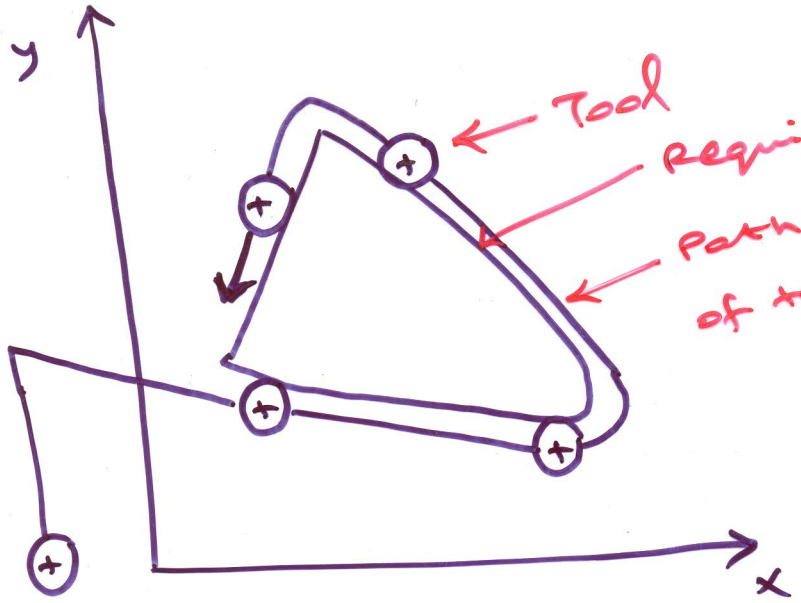
(3) Contouring (or) Continuous NC System:-

* In this type of system, there is a relative motion between the tool & the workpiece.

* All the axis of motion might move simultaneously. Due to this, different curves & Profiles can be cut.

* Actually it is a combination of PTP & Straight Cut NC System.

* An NC machine tool capable of doing Contouring type NC operation is capable of doing PTP & Straight Cut NC operation also.



example

NC milling machine.

Contour control in NC

(2) According to the tool positioning (or) modes of programming

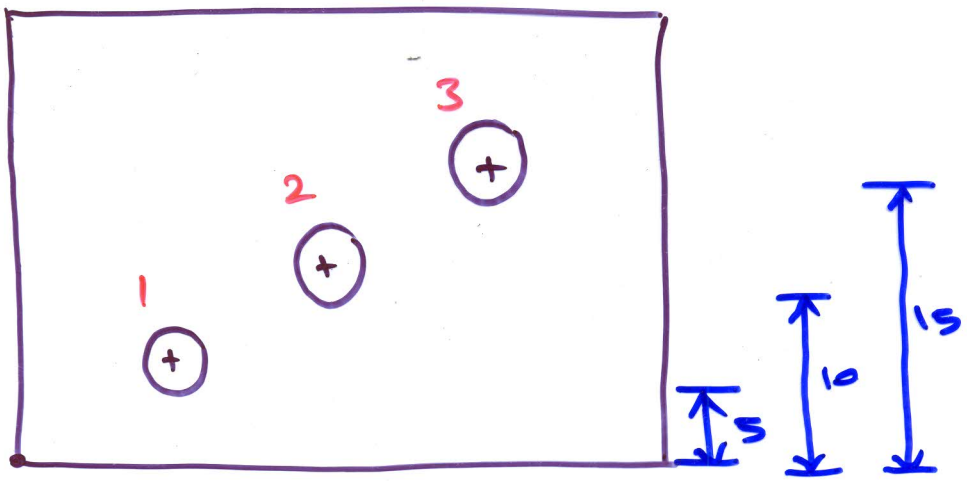
a) Absolute programming NC system

b) Incremental programming NC system.

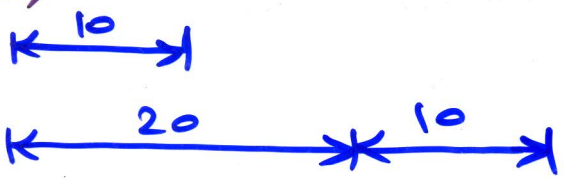
a) Absolute System:-

* In this system, all the position of the tool are measured from the same zero point.

* the following figure shows the positions of the tool measured from the same zero point "A".



A(0,0)



Point 1 = 10, 5

Point 2 = 20, 10

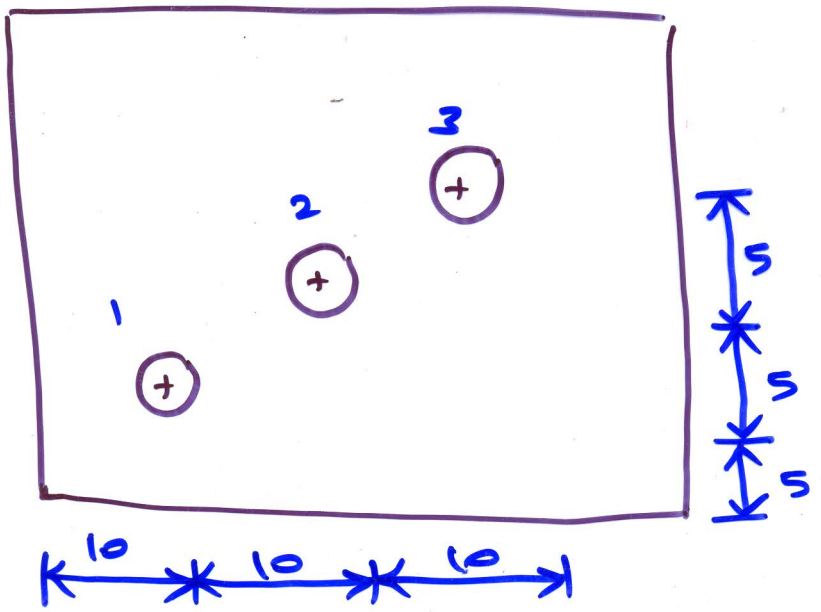
Point 3 = 30, 15

b) Incremental System:-

* An incremental system is one inwhich the reference point to the next instruction is the end point of the preceding operation.

* the main disadvantage of this system is that if an error occurs into the dimension of any location, all the location marked after that will carry the same error.

* the following figure shows the various position of tool as per the incremental system.



Point 1 = 10, 5 ; Point 2 = 10, 5 ; Point 3 = 10, 5

(3) According to the Servo Control System, the NC System is classified into 2 types

a) Open loop control system
b) Closed loop control system

} It was previously discussed.

Zero Point:—

There are two types of zero point in NC machine.

(1) Machine Zero Point
(2) Workpiece Zero Point → fixed zero point
→ floating zero point.

(1) Machine Zero Point:-

* It is called as Reference Point. It is set by the manufacturer when the machine is manufactured. It is far away from the Spindle axis.

* Usually after all the operation is completed on the workpiece, the tool is brought back to the Reference Point.

* Tool Changing also is done after the tool is sent to the Reference Point.

(2) Workpiece Zero Point:-

* It is the origin Point from where all the position of the tool & work table is defined. It is of two types

(a) Fixed Zero Point

(b) Floating Zero Point.

(a) fixed zero point:-

Here, the origin is always predefined and fixed. Usually it is at lower left corner of the work table.

(b) Floating zero point:-

In modern NC machines, floating zero

concept is provided, which allows the operator to define this origin.

* In this type of machine, the operator can set the origin wherever he want it to be set on the workpiece.

* The setting of zero point is done manually by the operator, by bringing X, Y & Z axes to the point at which the origin is to be defined and by pressing the zero button at that point.

NC Part Programming:—

* It is a set of instructions which instruct the machine tool about processing steps to be performed to manufacture a component.

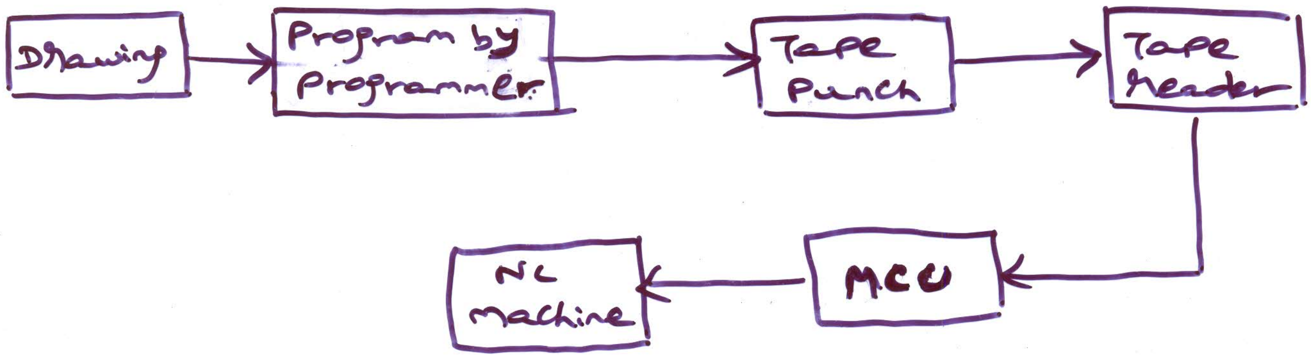
* There are three types of programming techniques

1. Manual Part programming
2. Computer-Assisted Part Programming
3. Manual data input.

Manual Part Programming:—

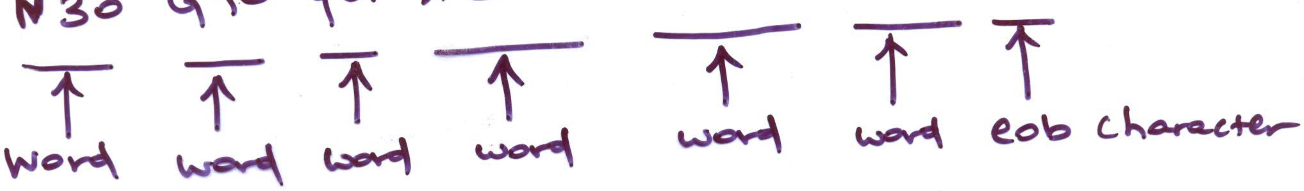
* In manual Part Programming, the NC Part program required for a particular part is written

by a programmer. This part program is typed with a flexo writer where typing causes the typed paper and punched tape to be prepared simultaneously.



* In a part program, each line of instruction is called as a block which is composed of one word (or) an arrangement of words. Blocks may vary in length [(ie) they contain variable number of words]. The block ends with end of block (eob) character. Each word is represented by an adress followed by a number.

(ex):- N30 G90 G01 X30.0 Z-20.0 F150 ;



① Sequence number (N code) :-

This code (or) address is used to indicate the block number. The NC part program contains number of blocks. Each block is identified by the block number. N30 ⟹ Block number: 30.

② Preparatory function (G-codes):—

* This codes inform the controller what type of action is to be carried out.

* In general, G-codes is typed at the beginning of block after N-code so that it can set the control for a particular mode when acting on the other words in the same block (or) all other subsequent block.

* G-codes may be modal (or) non-modal.

* For modal type, G-codes specification will remain in effect for all subsequent block unless replaced by another G-code.

* X, Y, Z codes are used to indicate howmuch distance the tool (or) work table along a particular direction.

③ Feed rate (F-code):—

This code is used to indicate the feed rate by which the tool (or) work table should move along a particular direction. It is in terms of inch/minute (or) mm/minute.

④ Spindle speed (S-code):—

This code is used for the rotation of the

Spindle at a particular rpm.

⑤ Tool number (T-code):—

It is used to indicate which tool is to be used for a particular operation.

⑥ Miscellaneous codes (M-codes):—

* this codes are used to control the parts of the NC machine

Ex:-

N20	M03 S1500;	(Spindle motor on at a speed of 1500 R.P.M)
N30	M09;	(Coolant motor off)
N40	M06 T02;	(Tool should be changed by tool number: 2)

Computer Aided Part programming:—

* In Computer aided Part Programming, much of the computational work needed in manual programming is performed by the computer processor

* In this programming type, the programmer prepares the set of instructions in the high level Computer language which is converted to machine tool level program with the help of processor.

Manual Data input:-

* It is a procedure inwhich the part programmer directly keys in the program into the MCU of the machine tool.

* Most of the modern CNC machine is having this facility.

* This facility helps the Programmer to change any existing program before the machining operations.

Advantages of NC System:-

* High Productivity due to less setup & lead time
* Less Scrap

  As human errors are eliminated, Accurate components are produced. Hence Scrap is reduced.

* High quality due to high accuracy.

* Flexibility in design

  In NC System, Complicated Profile can be easily produced at a faster rate.

* Reduction in inventory
* Safety to the operator

Disadvantages of NC System:-

* High initial cost
* High maintanence cost
* Skilled operator

for part Programming, well trained and highly skilled operator is required.

* The program can not be stored & edited. So for every program, tape punch should be used.

CNC System:-

* CNC stands for "Computer numerical control". CNC machine is having computer attached with machine control unit [MCU].
* Here, the part programming can be fed into the computer memory by pendrive (or) tape punch.
* The part programmes which was fed into the CNC machine can be stored & edited.
* The part programmes is converted into electrical signals by the microprocessor in the computer.

The features of the CNC machine:-

① Cathode ray tube which is capable of simulating cutting parameters and show the positions of machine

table & the cutting tool before the part is actually loaded on the machine tool. Actual cutting position may also be shown when the part is being machined. The entire program also can be listed on the screen.

(2) Provisions of absolute & incremental programming which are incorporated by G90 (absolute) and G91 [incremental] codes.

(3) Provision of inch (or) metric data input through G70 (inch) and G71 (metric) address.

(4) Availability of manual data input (MDI) to incorporate changes in part programming as & when necessary. Lot of programmes can be stored and can be used as & when necessary.

(5) Point-to-Point System, Straight cut System & Contouring path System are available in one CNC machine.

(6) Cutter diameter & length compensations are incorporated.

(7) Provision of high volume program & data storage area for future storage & use are incorporated with hard disks.

(8) Use of canned (or) fixed cycle programming to reduce complexity in programming is allowed.

(9) Incorporation of provision of subroutine/sub programming and macros.

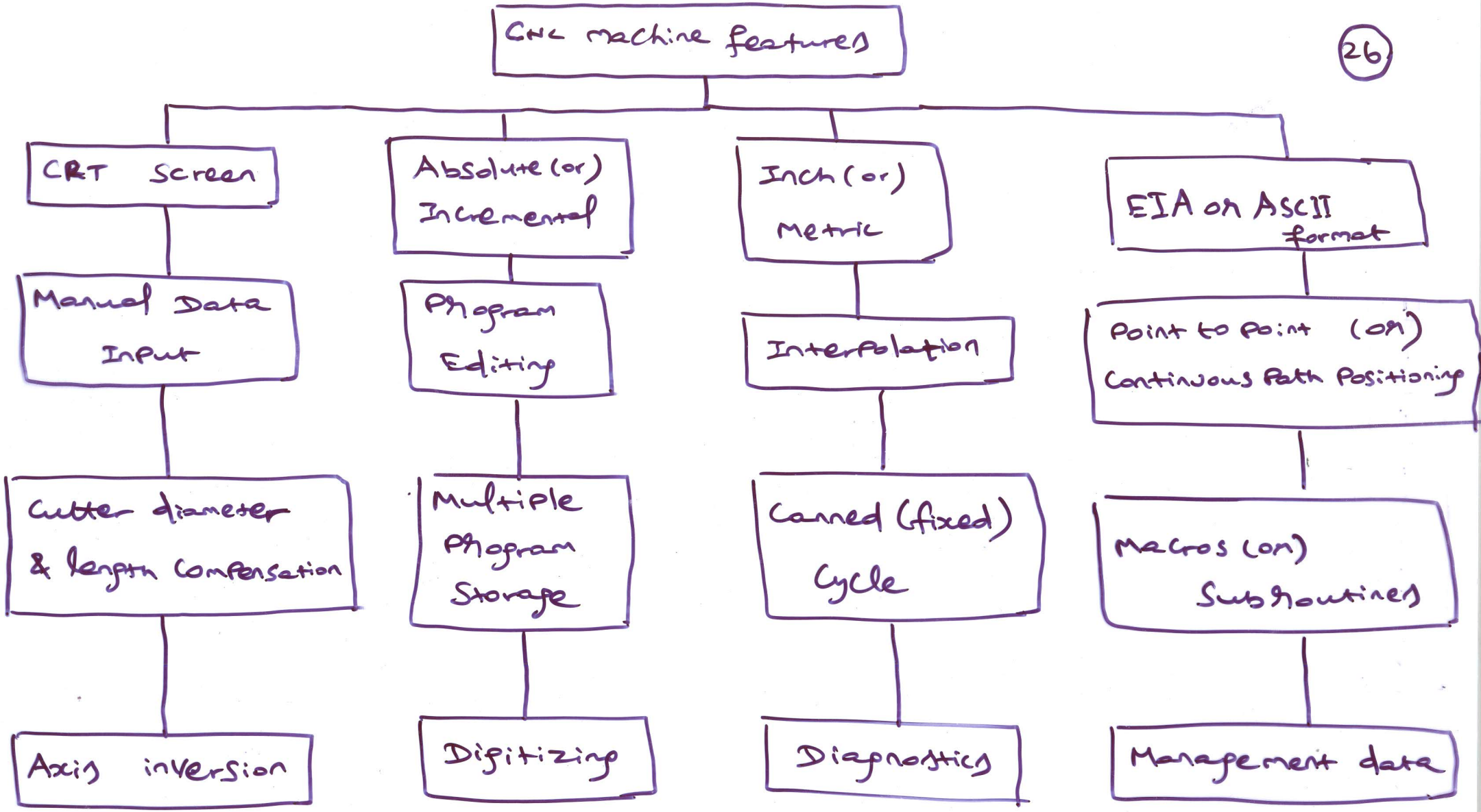
(10) Capability to create axes inversion (mirror image) to produce right (or) left hand part from the same program.

(11) Digitizing to make part programming directly from the existing part.

(12) A program can be loaded in MCU & hence, dependency on tape reader is eliminated.

(13) Adaptable to both EIA or ASCII tape formats

(14) Advanced interpolation methods like helical and cubic make it more versatile. In previous available NC systems only linear, circular and parabolic interpolators are available.


CNC machine features
CRT Screen
Absolute (or) Incremental
Inch (or) Metric
EIA or ASCII format
Manual Data Input
Program Editing
Interpolation
Point to Point (or) Continuous Path Positioning
Cutter diameter & length Compensation
Multiple Program Storage
Canned (fixed) Cycle
Macros (or) Subroutines
Axis inversion
Digitizing
Diagnostics
Management data

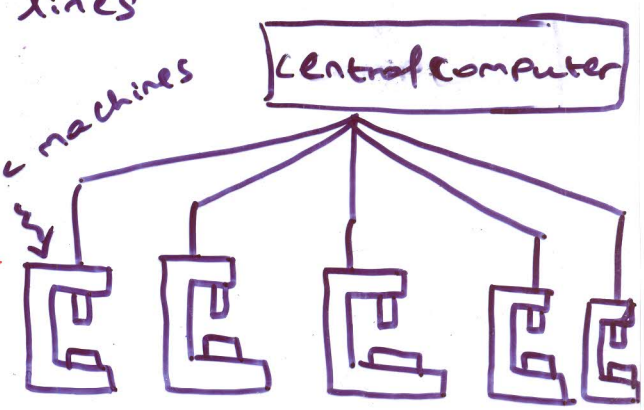
Direct Numerical Control :—

* It is a manufacturing System in which a number of machines are controlled by a Central computer through a direct connection of tele communication lines.

* Instead of using a tape reader as in NC machines, the Part Program is transmitted to the machine directly from the computer memory. One Computer can control more than 200 Separate machines.

* The Direct Numeric Control System consists of four components

1. Central computer
2. Bulk memory which stores NC Part Program
3. Tele communication lines
4. Machine tools



Advantages of CNC machines :—

Advantages of CNC machines are Similar to NC machine. Some additional advantages due to additional feature in CNC machine are as follows.

① Program Storage

As computer is available, multiple programs can be stored in the machine.

② Reliability of System :-

As the data is directly entered with the help of computer, no need to use punched tape. It improves the reliability of the system.

③ Online part programing :-

The part program can be done online with editing if required.

④ Metric (or) inches :-

The program can be written in metric (or) inches through G70 (inch) & G71 (metric) address.

⑤ Interpolation :-

In NC System, there is interpolation for straight and circular path, but in CNC, helical, parabolic and cubic interpolation also can be made.

⑥ Tool compensation :-

For the purpose of tool offset & tool wear, cutter diameter & its length compensation are available in CNC machine

⑦ Axis inversion:—

Capability to create axes inversion (mirror image) to produce right (or) left hand part from the same program.

⑧ canned (fixed) cycle:—

Use of canned (fixed) cycle Programming to reduce complexity in programing is available.

Disadvantages of CNC machine:—

① High initial cost

② High maintanence cost.

③ Costly control system.

④ Need of skilled opreator for part Phogramming.

⑤ whole machine will stop functioning if there is any problem in Computer

⑥ Costly Software

Advantages of DNC System:—

1. Control of more than one NC machine.
2. Elimination of punched tape & tape reader.
3. Convenient Storage of NC part programs in computer files.
4. Greater computational capability & flexibility.
5. The data for tools & cutters can be centrally maintained & updated.
6. The data related to manufacturing can be effectively collected & hence inventory can be better controlled.

Disadvantages of DNC System:—

1. If the central goes down, all machines becomes inactive.
2. Initial cost is too high.

Comparison between NC, CNC & DNC System:- (31)

Sl. no	Parameters	NC	CNC	DNC
1.	Flexibility	Less	High	High
2.	Tape editing	Not possible	Possible	Possible
3.	Productivity	Less	High	Highest
4.	Number of Programs stored	only one at a time	Multiple Programs	Multiple Programs
5.	Number of operations done at a time	one	one	Multiple
6.	Initial cost	Low	High	Highest

Various NC machines:-

1. Single spindle drilling machine
2. Lathe
3. Milling machine
4. Turning centers
5. Machining Centers
   a. Horizontal machining center
   b. Vertical machining center

(1) Single spindle drilling machine:-

* It is the most simple NC machine.
* It is programmed on three axes.

X-axis → Table movement to the right (or) left of the column

Y-axis → Table movement towards (or) away from the column

Z-axis → Up & down movement of the spindle.

(2) Lathe:-

* Machine tool to manufacture & machine round parts
* It is programmed on two axes.

X-axis → Cross motion of the cutting tool.

Z-axis → Carriage travel towards (or) away from the Head Stock

(3) Milling machine:-

* Accepted → one of the most versatile machine tools used in the manufacturing industry.

* It is Programmed on three axes.

X-axis → Table movement left (or) right of the Column

Y axis → Table movement towards (or) away from the Column.

Z axis → up & down movement of the Spindle

(4) Turning Centers:—

* Turning centers are more accurate and productive than engine lathe & has a higher Spindle rate.

* It is having Turret tool head which can accondate 12 Tools and as such can Produce wide range of components without tool change.

* It is Programmed for two axes.

X-axis → Cross wise movement of the Tool

Z-axis → Movement of the Tool towards (or) away from the Head Stock.

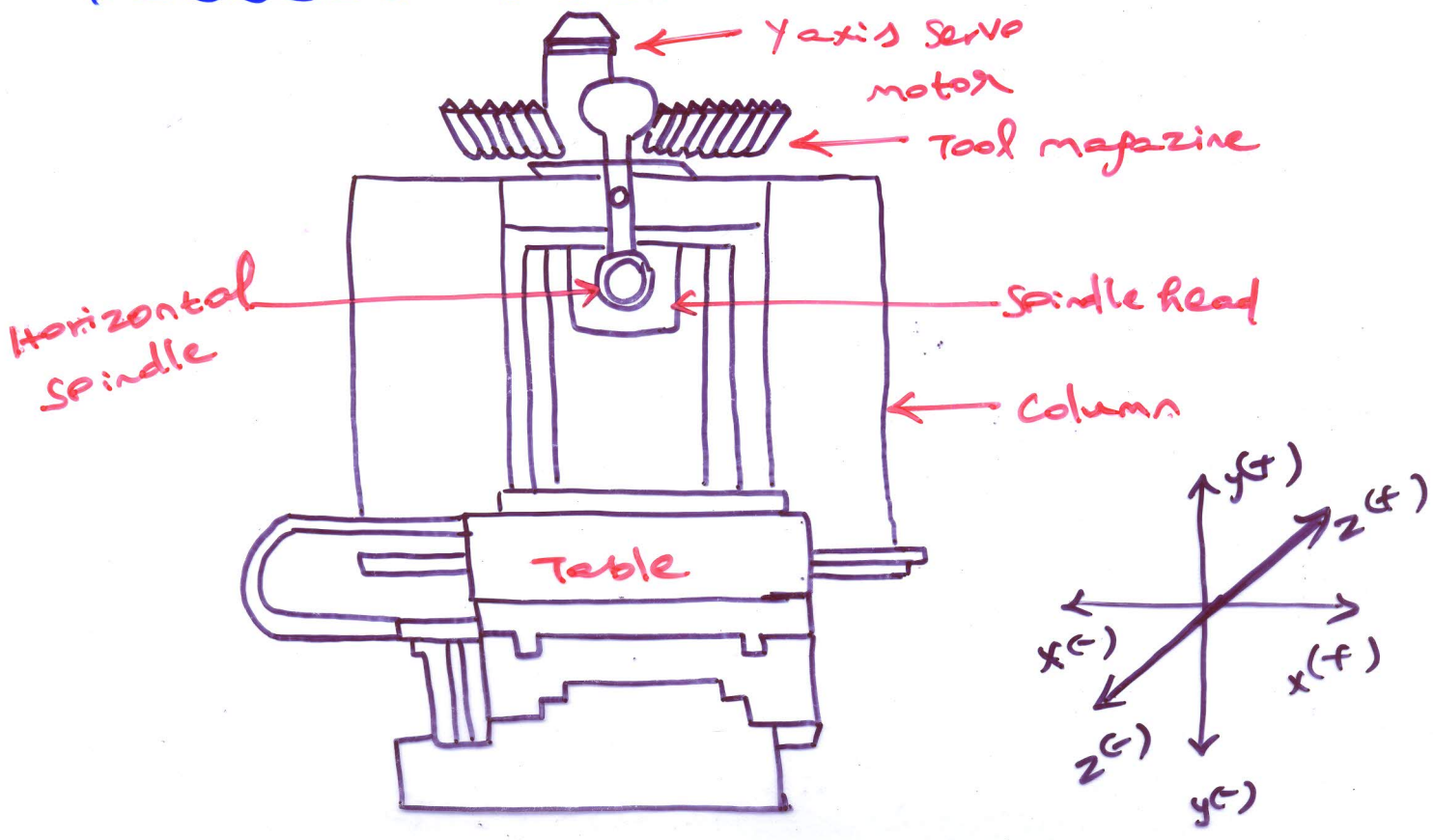
(5) Nc machining centers:—

Machining centers are of two types

1. Horizontal machining center (HMC)
2. Vertical machining center (VMC)

Machining Center are capable of a variety of machining operations. For this reason, a variety of tools are required in a machining center. thus a machining center must have Automatic tool changer (ATC), Automatic pallet changer apart from other parts.

(1) Horizontal machining centers (HMC):—



* Horizontal machining center (HMC) has horizontal spindle head. Here the following movements are there

① X axis for table movement

② Y axis is for vertical movement of the spindle

③ Z axis is for horizontal movement of the spindle perpendicular to x & y axis.

* In this machining center, tool magazine which holds lot of tools is there and Automatic tool changer is there. After completing one operation, Automatic tool changer remove the old tool from the spindle head & fix the new tool into the spindle head.

* Horizontal machining center may be fixed column type (or) travelling column type.

* Fixed column type carries a removable table called as pallet

* After machining, a pallet with a component is removed & another pallet which was already loaded with the workpiece is then clamped

on the pallet receiver & machined.

* In travelling column type machining center, there are two tables. After machining, components are on one table and column travels to the second table which carries a fresh workpiece for machining.

* Components which is to be machined is unloaded from the first table & a fresh workpiece is loaded on it.

* Machine comes under this category

  ① CNC Horizontal milling machine

  ② CNC Grinder. etc.,

Vertical machining center:- [VMC]

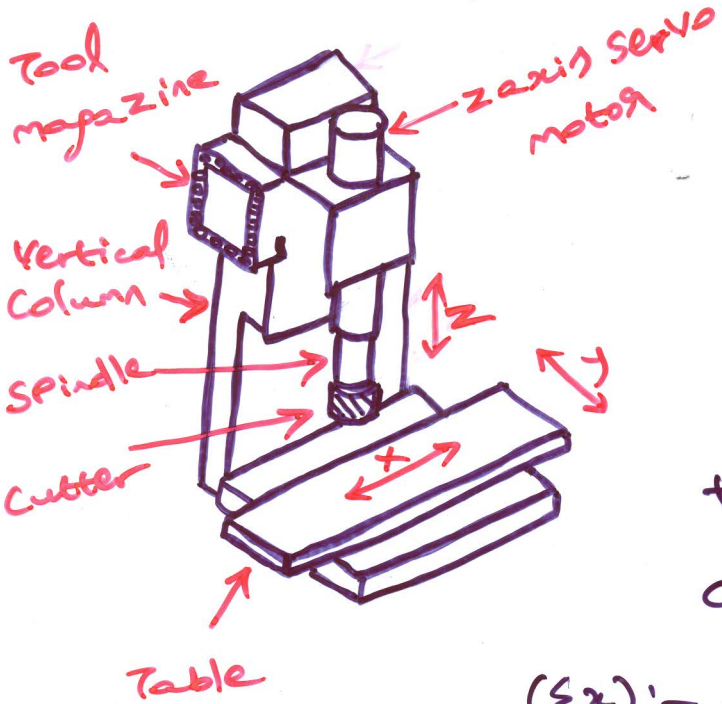
* these machines are generally single spindle machine.

* Vertical machining centers has

  ① x-axis control for the table movement left (or) right

  ② y-axis control for the table movement towards (or) away from the column

  ③ z-axis control for the vertical movement of the spindle



* It carries a Vertical spindle head which can slide along a vertical guide ways provided on the column.

* It has tool magzine, tool transfer arm, Automatic Pallet changer etc.,

(Ex):- CNC Vertical milling machine, CNC drilling machine etc.,

Advantages of machining center:-

1. Machining centers have high metal removal rate capabilities
2. Machining centers are highly versatile and increases productivity.
3. It is more flexible and economical than conventional machines.
4. It is mainly used for mass production.

Automatic Tool Changer (ATC):-

In order to reduce the time for changing over of tools, machining centers are provided with automatic tool changing facility. It is done by

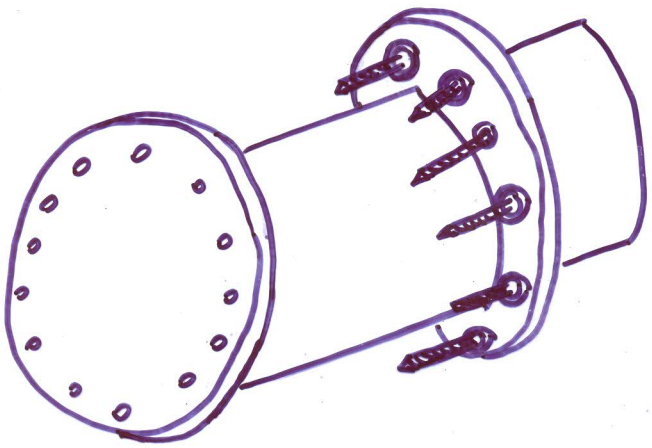
Automatic tool changer (ATC) unit of the machining center. The ATC consists of the following two units.

① Tool magazine ② Tool transfer arm

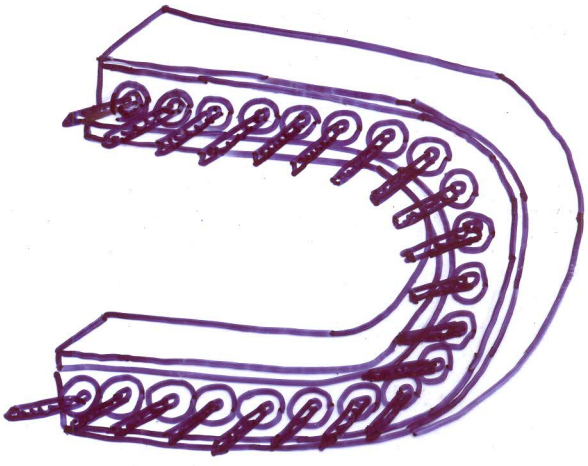
Tool magazine:-

It is a tool storing unit, attached to the machine center. It contains number of pockets (from 10 to 400) to hold number of tools. Each tool is inserted into the pockets and numbered. To select the required tool, in the part program these numbers are denoted.

The tool pockets are arranged in drum (or) chain. Based on that, it is classified as "drum type tool magazine" and "chain type tool magazine".

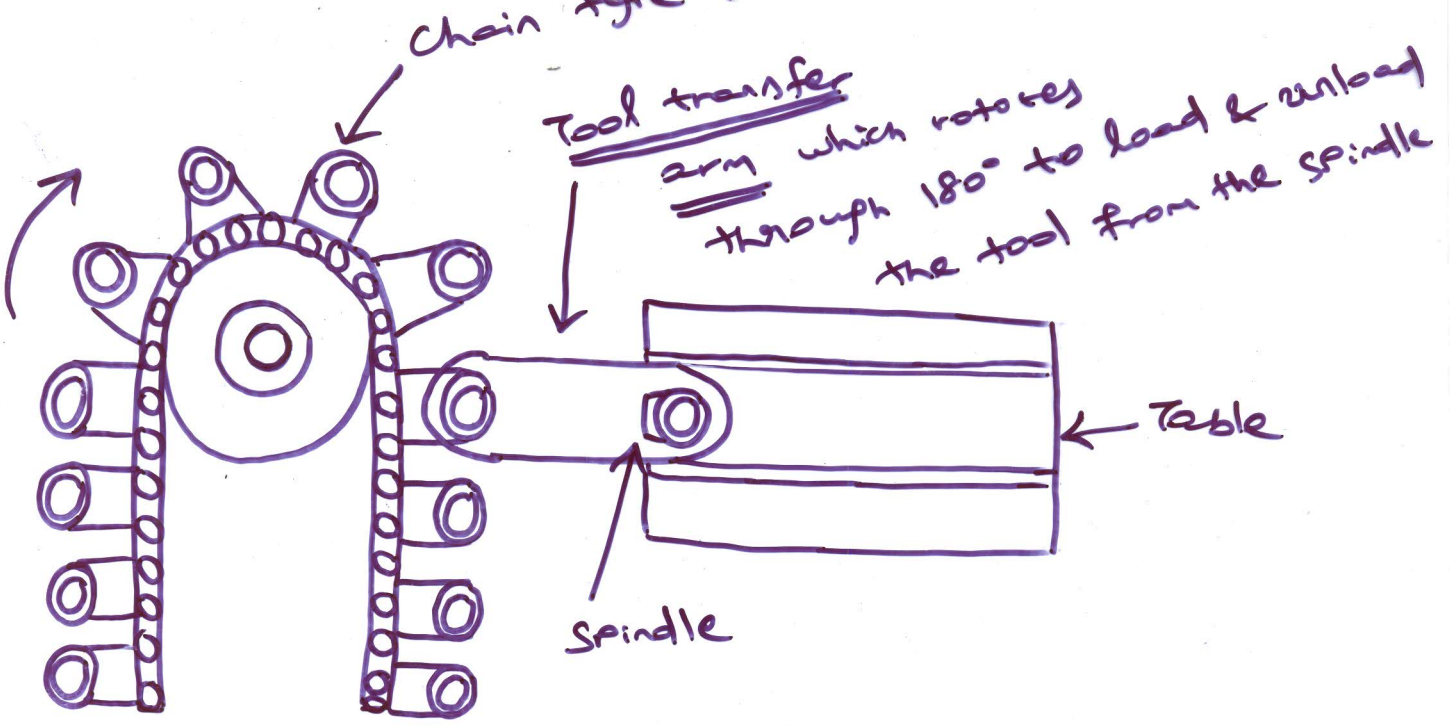


a) Drum type tool magazine



b) Chain type tool magazine

Tool transfer Arm:-



* Tool transfer arm is used to pick the tool from the tool magazine & the spindle and fix the new tool to the spindle & fix the old tool to the tool magazine by rotating through 180°.

* As per the Part Program instruction, after one operation is over, the magazine indexes to the next tool position. Then one end of the tool transfer arm picks the required tool for next operation from the tool magazine and the another end of the arm picks the old tool from the spindle. Then it will rotate through 180° & fix the new tool to the spindle & old tool to the tool magazine.

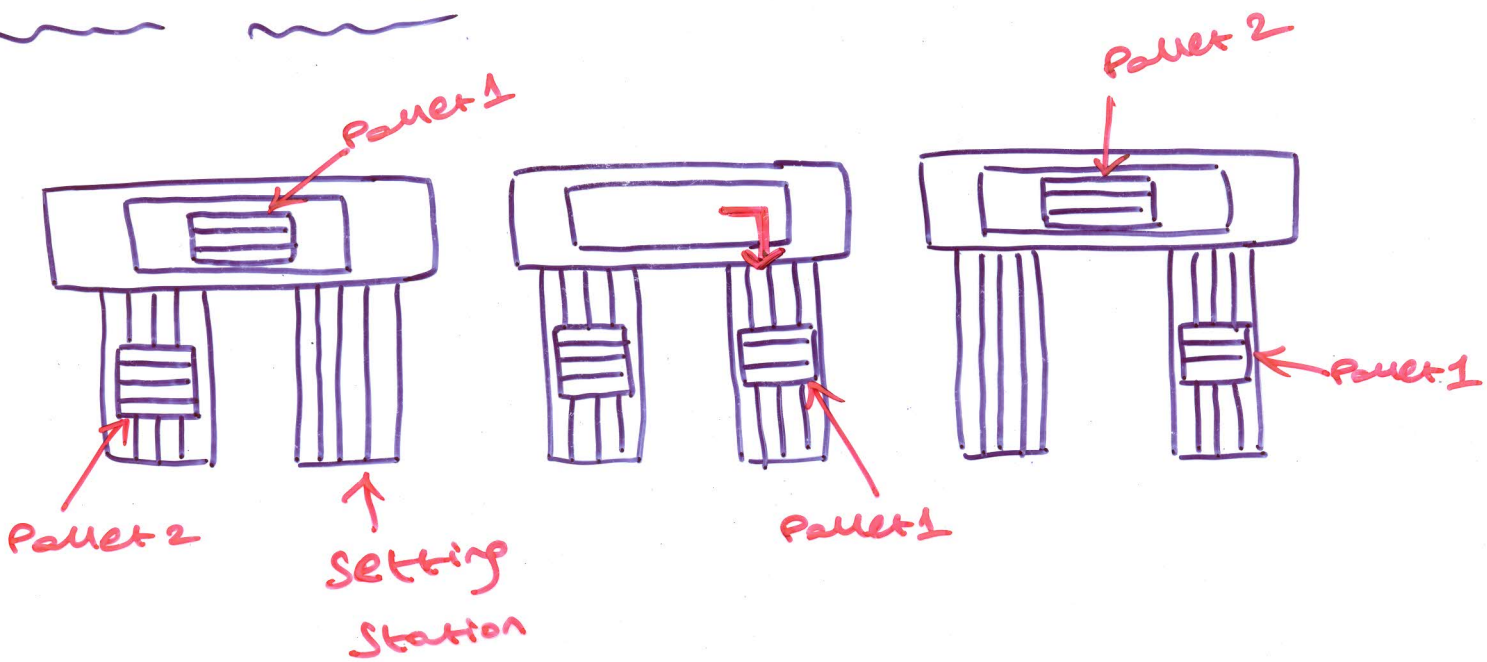
Automatic Pallet Changer:—

* Machine downtime because of loading-unloading, clamping-releasing of the workpiece can be minimized with the help of automatic workpiece loader/unloader System.

* In this System, the workpieces are mounted on the pallet and the pallets are moved around the machine in a logical manner. This System is called as "Pallet Changing System".

* According to the logical movement of the Pallet, the System can be linear (or) Rotary.

(i) Linear motion:—



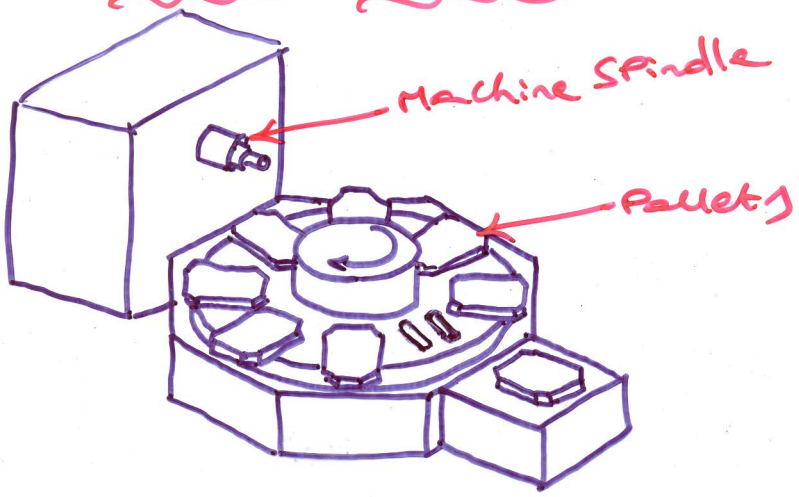
(a) (b) (c)

* In the fig (a), the workpiece on the left side track is waiting for completion of machining operation of earlier workpiece.

* In the fig (b), after completion of earlier workpiece it moves onto the unloading table and the next component is ready to move onto the machining table.

* In the fig (c), the next component moves onto the machining table and this process continuous.

Rotary Pallet Changer System:-



* In this pallet changer system, the number of pallets are clamped on the rotating table.

* For the machining purpose of the workpieces, the table is moved in rotary motion with the help of indexing mechanism.

Canned cycle (or) Multiple cycle (or) fixed cycle:-

A canned cycle is a combination of tool movements that performs any one particular machining function such as drilling, boring, turning etc., this cycle automatically generates multiple movements from a single block.

Ex:- multiple turning cleyle & finishing cycle

G71 U___ R___;

G71 P___ Q___ U___ W___ F___;

G70 P___ Q___ F___;

where,

U → Depth of cut for each pass along x-axis

R → Retraction amount in x axis.

P → Starting block; Q- Ending block;

U → Stock in x axis

W → Stock in y axis

F → feed rate

G71 → Multiple Rough turning clcle

G70 → Multiple finish turning clcle

Subroutine (or) Sub Programe:—

when a component has repetitive pattern machining at different places, in stead of writting the same program each & every time, the Program for the repetitive operation is written & stored in a separate file & is called whereever needed. this program is called as "Sub-programme".

Syntax for Sub Progran

M98 001 1234;

M99;

where,

M98 – Sub Program call;

001 – no of repeats;

1234 – Sub Program number;

M99 – Sub program End.

Interpolation:—

It is the method of specifying the path to be generated for maching. There are the following types of Interpolation.

(1) Linear Interpolation (2) Circular Interpolation

(3) Helical Interpolation (4) Cubic Interpolation.

(5) Parabolic Interpolation.

(1) Linear Interpolation :-

* It is used when a straight line path is to be generated on the workpiece of NC machine.

* The linear path can be Horizontal, Vertical (or) Inclined.

* The G-code — G01 is used for linear interpolation & four parameters [x, y, z & f] are required to specify G01 code.

Syntax for G01 :-

G01 X_Y_Z_F_;

where

x = value for x co-ordinate

y = value for y co-ordinate

z = value for z co-ordinate

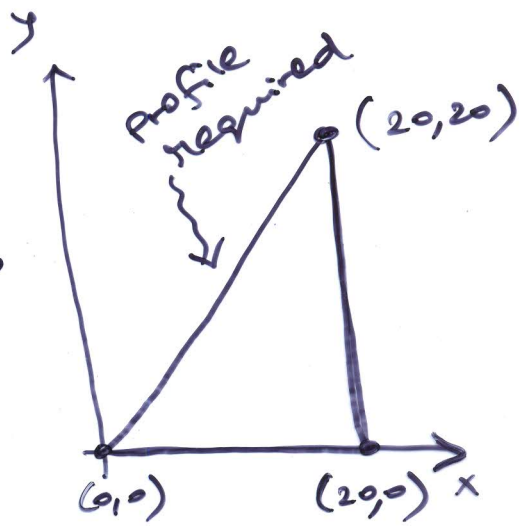
F = feed rate

example :-

G01 X20.0 Y0.0 Z-1.0 F1.5;

X20.0 Y20.0 Z-1.0;

X0.0 Y0.0 Z-1.0;



2. Circular Interpolation:-

* The movement of the tool along a circular path is called as "Circular Interpolation".

* Five Parameters are required for specifying the G-code used for Circular Interpolation [G03 (or) G02]

* G03 code is used for making the Circular path in Counter Clockwise direction and G02 code is used for making the Circular interpolation in a clockwise direction

* Syntax for G03/G02 code:-

G02/G03 X_ Y_ Z_ R_ F_ ;

where

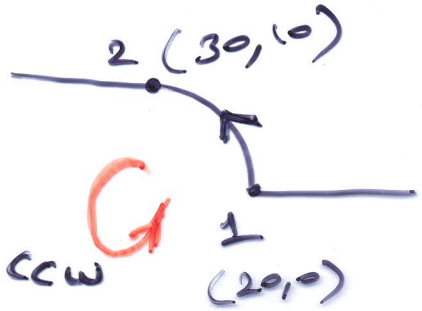
X, Y, Z → Co-ordinate values;

R → Radious of the Arc;

F → Feed rate in mm/rev (or) mm/minute

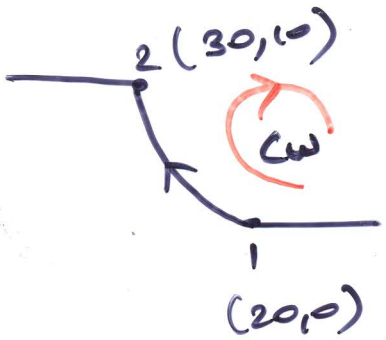
Example:-

(1)



G03 X30.0Y10.0Z-1.0R10.0F1.5;

(2)



G02 X30.0 Y10.0 Z-1 R10.0 F1.5;

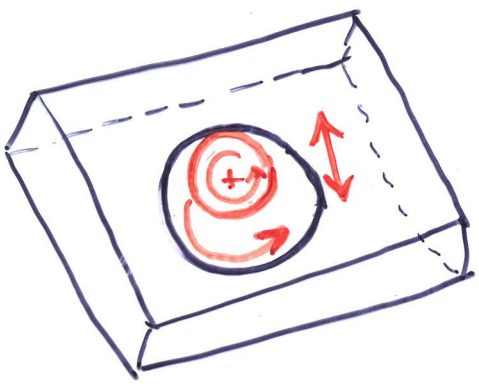
(3) Helical Interpolation:-

* The movement of the cutting along a Helical path is called as "Helical Interpolation".

* To make this interpolation happens, the cutting tool is moved simultaneously along 3 axis X, Y & Z.

* It is mainly used for making Aero Parts.

* By means of Helical interpolation, any bigger dia. hole can be made with smaller diameter tool with less torque & less power.



Producing a bigger hole with a smaller dia tool

Slide movement in CNC machines:-

Precise positioning and repeatability of machine tool slides are the major functional requirements of CNC machines. A plain slide way will give stick-slip. To prevent this, there are different slide way systems are used. These have low wear, negligible stick slip, good damping capability, low co-eff of friction properties.

Requirement of a good slide way system:-

1. Low co-eff of friction
2. Low rate of wear
3. Must have good damping capacity
4. Must give a smooth drive.
5. There must be no stick-slip.

In CNC machines, the following slideways type may be used.

a) Plastic inserted slide ways

b) Recirculating ball screw & nut type slide ways.

a) Plastic inserted slide ways

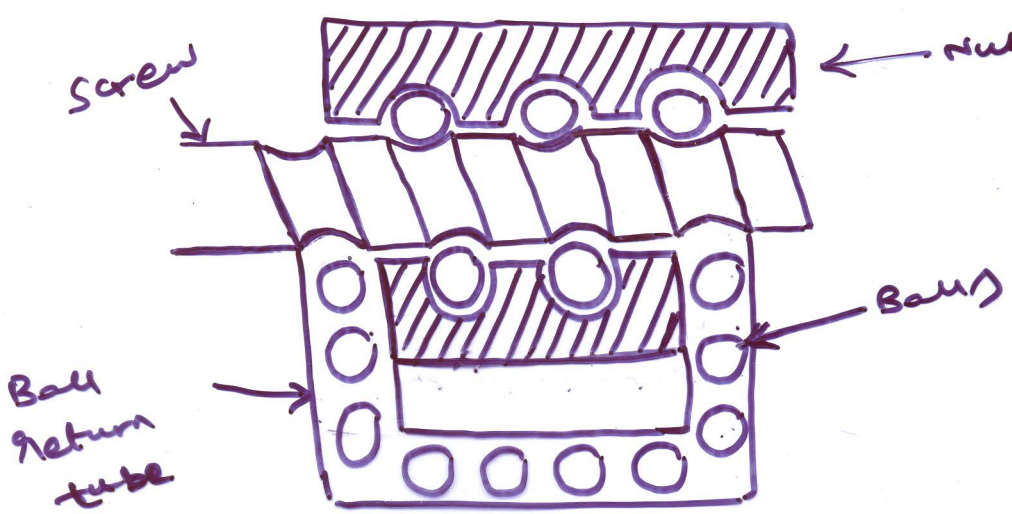
In this slide ways, plastic inserts are bonded to the underside of the sliding members. the inserts can be thermo plastic (or) thermo setting types. For these coated slide ways, the friction is less.

Advantages:-

1. Has less co-eff of friction.
2. Increase in strength
3. Good wearing quality
4. Has self lubricating properties.
5. Worn out inserts may be replaced

b) Recirculating Ball Screws & nut type Slide ways

* Recirculating Ball Screw & nut arrangement are used to transmit motion to the Slides.



* It consists of screw thread which acts as ball race to hold steel balls. The balls are surrounded by a nut.

* The balls rolling in the grooves exit from the trailing end of the nut and are picked up by return tube inserted from outside and are recirculated into the leading end of the nut.

* The rotation of the ball between the screw & the nut moves the slide attached to the nut.

* Here the sliding contact between the screw & nut is changed into the point contact due to the steel ball. So, the friction & hence wear is reduced.

Advantages:—

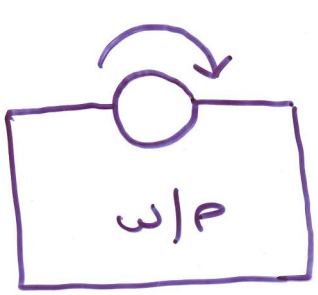
1. Very low coeff of friction
2. Higher transmission efficiency
3. No stick-slip.
4. Backlash between the parts are eliminated by preloading the assembly.
5. Reduced friction & hence wear.

Tool offset (or) compensation :-

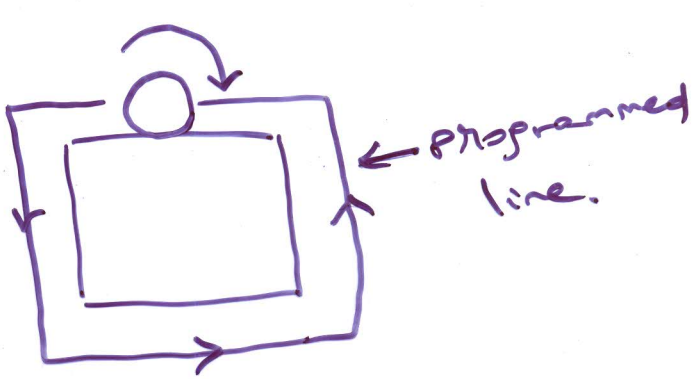
* the word offset refers to the allowance made by the CNC machine for the radious & length of the tool to cut the job. Programming on a CNC machine is always done according to the center point of the cutter. If the offset is not set, the tool will follow a wrong path.

* Since the diameter & length of a tool may vary, an offset value needs to be set so that the tool can be moved to the correct position for the cutting required.

* For example, if you were using a 10mm cutter the workpiece would be reduced by 5mm on each edge (or) 10mm overall. To overcome this the, the radious of cutter is given as a compensation.



no offset



offset is given

- Cutter is moved on the programmed line

Tool length offset (or) Tool length compensation :—

Normally, Part programmes are prepared for a particular tool length. The variation if any in a tool is given in the form of tool length offset and it is the difference between the programmed length and actual length of the tool. The same will be taken into account by the machine during operation.

example:—

G43 H1 Z10;

* H1 refers to tool length compensation of tool one, which has a value of −3.000 in the offset table.
* A minus value increases the tool length whereas the positive value decreases the tool length.

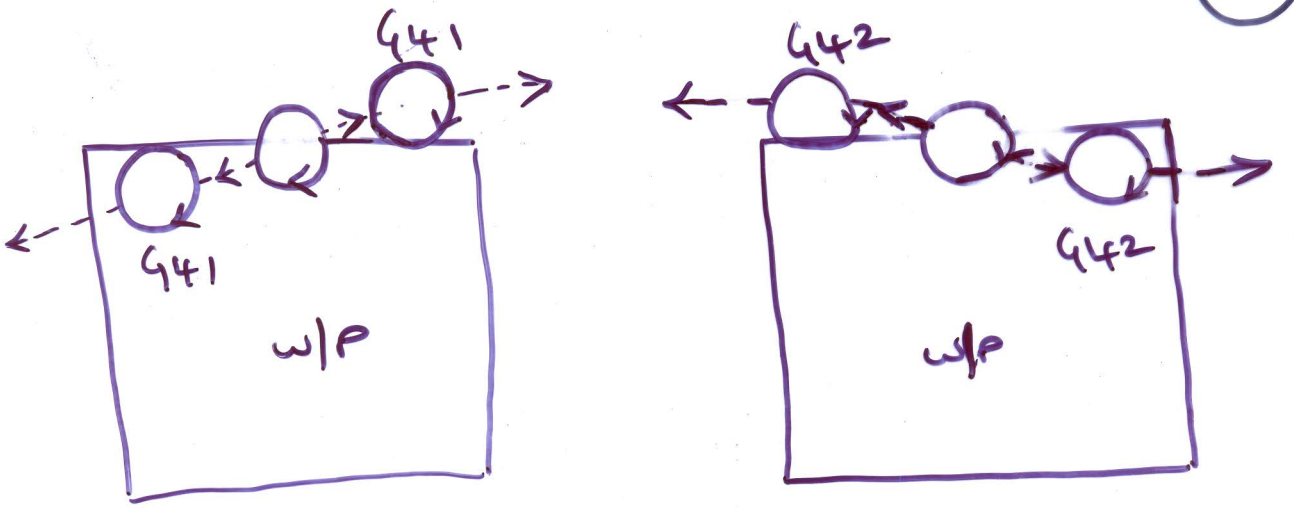
Tool radious (or) diameter compensation:—

Here the tool radious (or) diameter is given as a compensation value

G41 — Cutter radious compensation to the left

G42 — Cutter radious compensation to the right

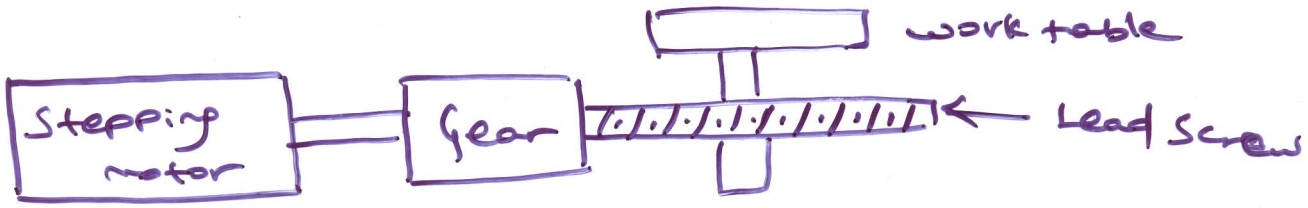
G40 – cancels the cutter radius compensation.



Codes for left cutter compensation

Codes for right cutter compensation

Formulaes used for calculating the slide movement :–



open loop System

Each pulse makes the stepping motor to rotate at an angle (or) a fraction of revolution.

① Step angle, $\alpha = \frac{360}{N}$; N → No of pulses required for 1 revolution of stepping motor.

Let $n \rightarrow$ Total no of pulses received by Stepping motor

(2) ∴ Total angle made by the Stepping motor $\Big\} A \Rightarrow n \times \left(\frac{360}{N}\right)$

(3) no of Revolution $= \frac{n}{N}$ [for 1:1 gear ratio between lead screw & motor shaft]

(4) If the pitch of the lead screw is P (in/rev), then the distance made by the worktable (x) axially

$$x = P\left(\frac{n}{N}\right)$$

(5) Pulse frequency (f) in pulses/sec is given by

$$f = \frac{N \times RPM}{60}$$

(6) Speed of the worktable, $\boxed{v = P \times RPM}$ in/min

Problem:—

(1) A Stepping motor has $N = 150$, $P = 0.2$"/rev; If $n = 2250$ Pulses, what is the distance travelled in x direction? what should be the Pulse frequency for a travel speed of 16 in/min?

Soln:-

① Distance travelled by the worktable axially $\Big\}$ $x = P\left(\frac{n}{N}\right)$

$$= 0.2\left(\frac{2250}{150}\right)$$

$$\boxed{x = 3''}$$

② Pulse frequency, $f = \frac{N\,(RPM)}{60}$

$$= \frac{150 \times \left[\frac{v}{P}\right]}{60} = \frac{150 \times \left[\frac{16}{0.2}\right]}{60}$$

$$= \frac{150 \times 80}{60} = 200\ Hz$$

---

② A stepping motor of 200 steps/revolution, is mounted on the lead screw of a drilling machine. If the pitch is 0.1"/rev & $f = 2000$ Hz, what is the speed of the table?

Soln:-

$N = 200$; $P = 0.1''$/rev; $f = 2000$ Hz; RPM = ?

$$f = \frac{N\,(RPM)}{60}$$

$$\therefore RPM = \frac{60 \times f}{N} = \frac{60 \times 2000}{200} = 600\ RPM.$$

(3) Consider a CNC worktable driven by a closed loop control system consists of a servo motor, lead screw & optical encoder. The lead screw has a pitch $P = 0.2''$ and is coupled to the motor shaft with a screw to motor gear ratio of 1:4. The encoder generates 150 pulses/revolution of the lead screw. If the no of pulses & pulse rate received by the control system are 2250 & 200 Hz respectively, Calculate

a) Table speed b) motor speed in R.P.M

c) Distance travelled by the table

Soln:—

$N = 150$ pulses/rev; pitch of lead screw $\Big\} = P = 0.2''/\text{rev}$

$n = 2250$; $f = 200$ Hz;

(i) RPM for table & motor = ?

(ii) Distance by table axially, x = ?

$$f = \frac{N\,(\text{RPM})}{60}$$

$$\therefore \text{RPM of the table} = \frac{60 \times f}{N} = \frac{60 \times 200}{150} = 80 \text{ rpm.}$$

∴ RPM of the motor shaft } $= 4 * 80$

$= 320$ RPM.

Distance by the table axially } $x = \left(\frac{n}{N}\right) \times p$

$= \left(\frac{2250}{150}\right) \times 0.2 = 3''$

$\boxed{x = 3''}$

Important G-codes

(1) G00 – Rapid traverse :–

Syntax :– G00 X Y Z (Milling)

G00 X Z (Turning)

(2) G01 – Linear Interpolation :–

Syntax :– G01 X Y Z F (Milling)

G01 X Z F (Turning)

F → Feed rate

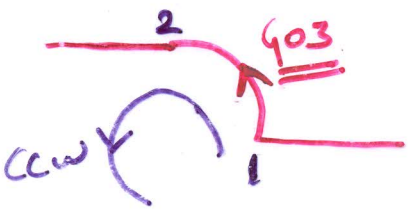
(3) G28 – Go to Reference Point

Syntax :– G28 U0.0 U → Go to reference (X)

G28 W0.0 W → Go to reference (Z)

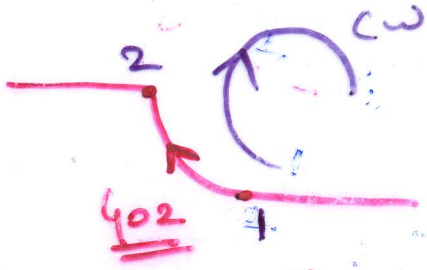
(4) G03/G02 – Spindle on CCW & CW

Syntax :– G03/G02 – X Y Z R F



R → Radius

F → Feed rate



G90 — Absolute Dimensioning

G91 — Incremental Dimensioning

G20 — Imperial Units

G21 — Metric Units

G98 — Feed / minute

G99 — Feed / revolution.

M codes:—

M04 / M03 — Spindle on CCW & CW respectively

M05 — Spindle Stop.

M06 — Tool Change

M07 — Coolant on

M09 — Coolant off

M30 — Program Stop and return

M98 — Sub Program Call

M99 — End of Subprogram.

Multiple/canned cycles:-

① Turning cycle:-

Block ① G71 U_ R_ ;

Block ② G71 P_ Q_ U_ W_ F_ ;

Block ③ G70 P_ Q_ F_ ;

where;

G71 – Rough turning cycle

G70 – Finish turning cycle

Block ①

U → Depth of cut for each Pass.

R → Retraction amount at the end of each Pass.

Block ②

P → Block number for start of opn

Q → Block number for End of opn

U → Stock in x axis

W → Stock in z axis

F → Feed rate

* Feed rate for finishing cycle should be less.

Grooving cycle:-

```
G75  R0.5
G75  X40 Z-10.0 P500 Q1500 F0.05
```

where

R → Retraction amount

P → Depth of cut - Peck amount (Microns)

Q → Movement along z direction (microns)

F → Feed in mm/rev

Drilling cycle:-

```
G74  R0.5
G74  Z-50.0 X0.0 Q500 F0.2
```

where

R → Retraction

Q → Depth of cut - Peck amount [Microns]

F → Feed in mm/rev

Single threading cycle :—

Thread = M24 × 1.2

G92 X23.8 Z-50 F1.2

G92 X23.6

G92 X23.2

G92 X23.0

G92 X22.8

G92 X22.6

G92 X22.4

where

P = Pitch value.

Depth of thread = P × 0.64 P

Minor dia = Major diameter − (2 × Thread depth)

Multiple facing cycle :—

Block : 1 G72 W0.25 R1.0 ;

Block : 2 G72 P10 Q11 U0.0 W 0.5 F0.5 ;

Block : 3 G70 P10 Q11 F0.25 ;

N10 G00 Z0.0 ;

N11 G01 X0.0 ;

Block : 1

W → Depth of cut along z axis

R → Retraction amount

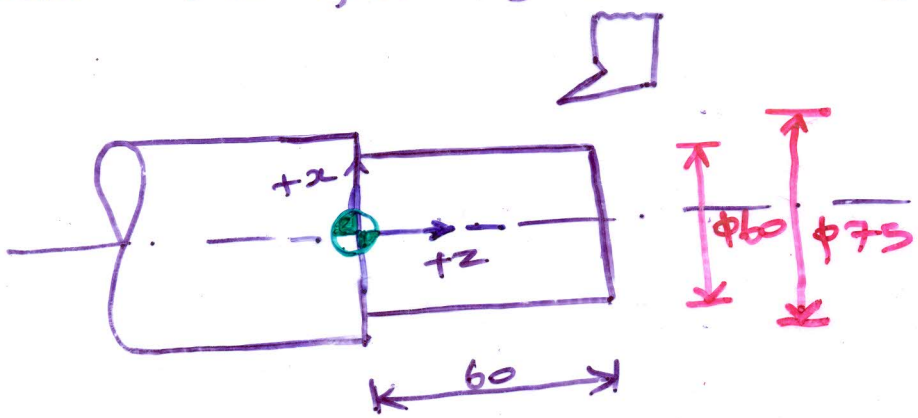
Block : 2

U, W → Stock along x & z axis respectively

G70 — finishing cycle

CNC Part Programming

(1) Write a Manual Part Program to turn the component shown on a CNC Lathe from 75mm bar stock. The following Data may be assumed.



(a) there will be two rough turnings and one finish turning. The first cut is with a depth of 3mm for a length of 58 mm. The second with a depth of 3mm for a length of 59 mm and the third with a depth of 1.5mm for a full length of 60mm.

(b) the shoulder of the workpiece is also machined during each cut.

(c) The spindle speed is 400rpm and the feed rate is 0.5mm/rev.

Make a free hand sketch showing relevant Points of tool position for each of the three turning operations and then write the manual part program. State also what each line of the program does.

```
O1234                          [Program number]
[Billet X75 Z100
G28 U0.0 ;                     [Go to reference Point]
G28 W0.0 ;
G21 G99 G90;                   [Metric Unit & feed /rev & Absoute Programming]
M06 T0101 ;                    [Tool change : 01 & offset no: 1]
M04 S400 ;                     [Spindle on CCW; SPeed = 400rpm]
M07 ;                          [coolant on]
G00 Z62.0 X76.0;               [Rapid traverse]
G71 U0.25 R1.0;
G71 P100 Q101 U0.0 W0.0 F0.5;
N100 G01 X69.0;
N101 Z2.0;
```

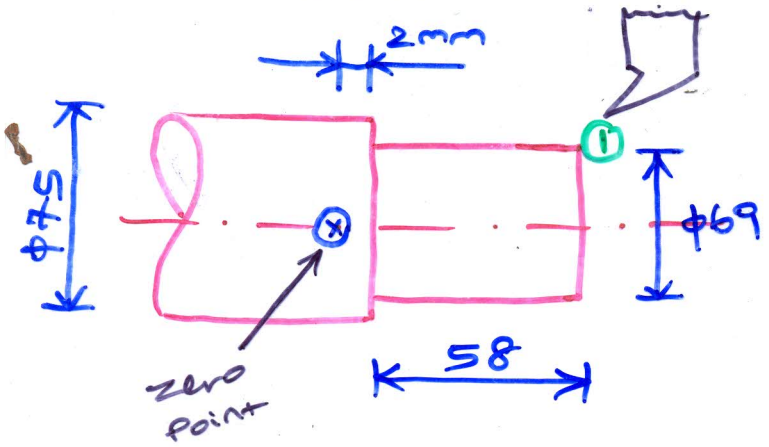
Roughturning : 1
Multiple Rough turning cycle to a depth 3mm & length 58mm

```
G71 U0.25 R1.0;
G71 P102 Q103 U3.0 W1.0 F0.5;
G70 P102 Q103 F0.25;
N102 G01 X60.0;
N103 Z0.0;
```

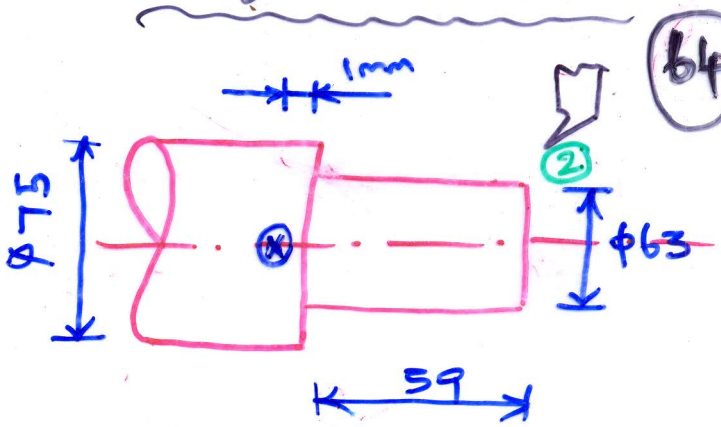
Rough turning : 2
depth :- 3mm
Length :- 59mm
G70 → Finish turning for depth = 1.5 length = 60mm

```
M05;                           [Spindle off]
M09;                           [Coolant off]
G28 U0.0;
G28 W0.0;
M30                            [Program Stop & return]
```

Rough turning:-1

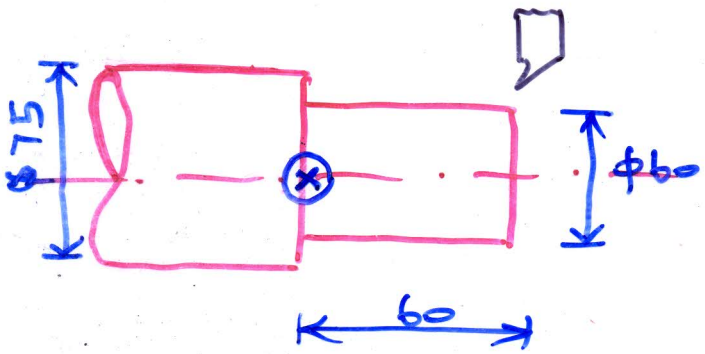


Rough turning:-2

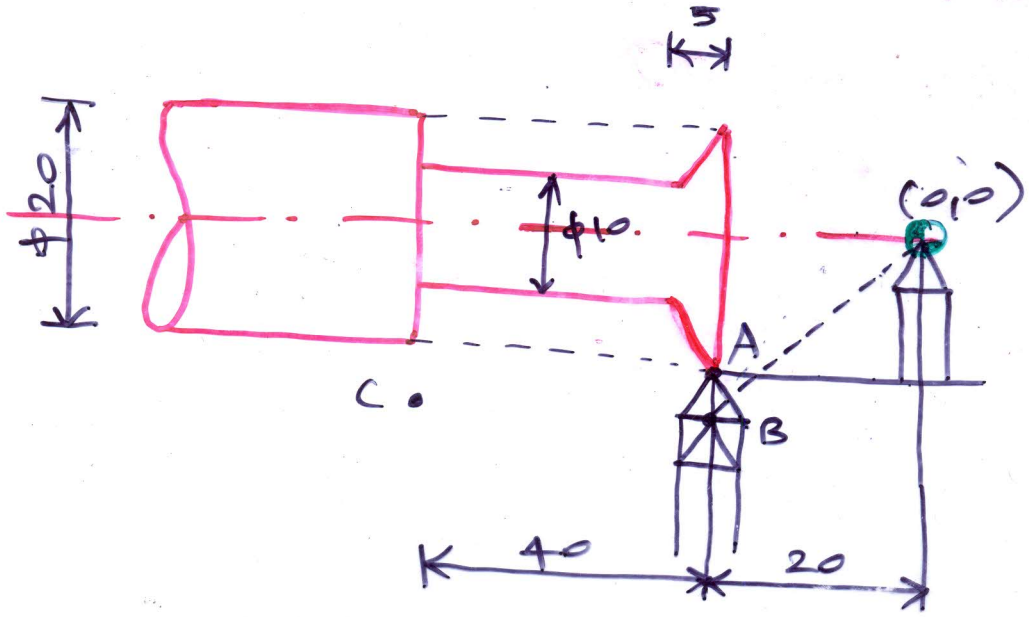


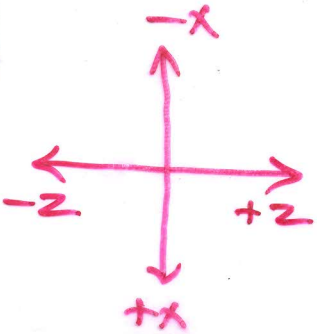
64

Finish Turning



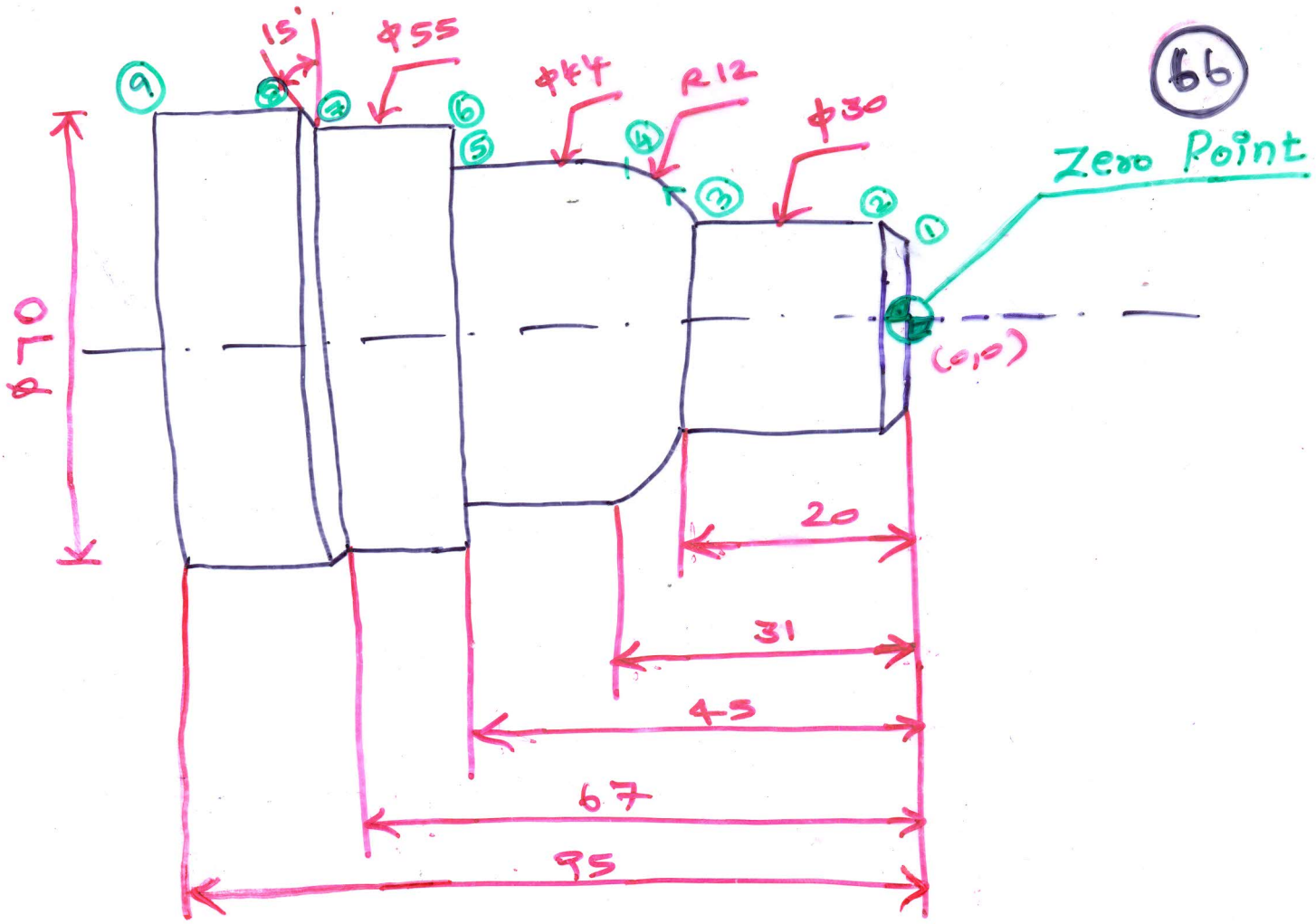
(2) Write a Part Program for the Part Shown below.



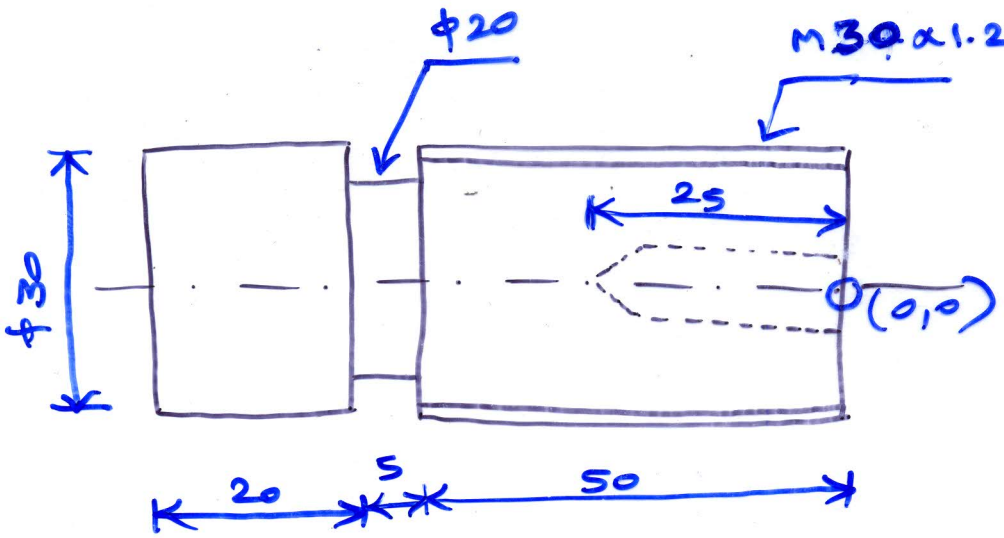


```
O1234                          ( Program number)
[ BILLET X20 Z80
G28 U0.0 ;                     ( Go to reference point)
G28 W0.0 ;
G21 G99 G90;                   [Metric Units & Feed /rev
                                ( Tool change) & Absolute Prog
                                 ramming]
M06 T0101;
M04 S1000;                     ( Spindle on CCW )
M07 ;                          ( Coolant on )
G00 X0.0 Z0.0;                 (Rapid traverse)
    X30.0 Z-20.0;              ( Point B)
    X30.0 Z-60.0;              ( Point C) Rough
G71 U0.25 R1.0;                 ( Multiple turning cycle)

G71 P100 Q200 U0.5 W0.5 F0.5;

G70 P100 Q200 F0.25;           (Multiple finish turning cycle)
N100 G01 X10.0;                (Linear Interpolation)
     Z-25.0;

N200 G01 X20.0 Z-20.0;

M05;                           (Spindle stop)
M09;                           (Coolant off)
G28 U0.0;                      (Go to Reference Point)
G28 W0.0;
M30;                           ( Program Stop & return)
```

(3) A 110mm long cylindrical rod of ϕ75mm is to be turned into a component as shown in fig using a CNC Lathe. Write a CNC program for manufacturing this component.

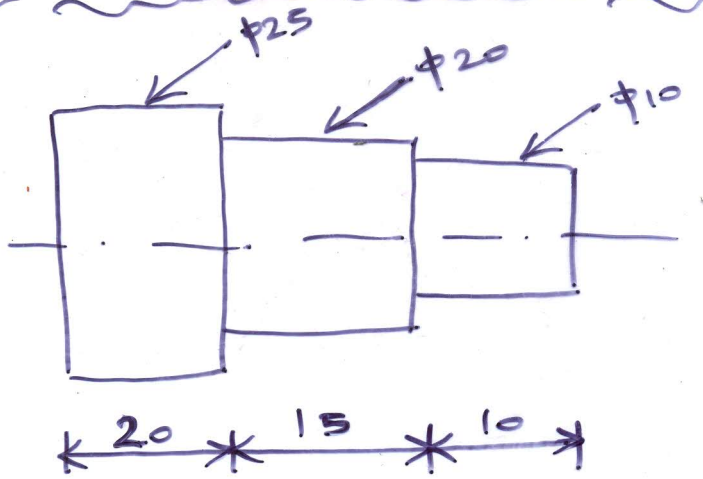


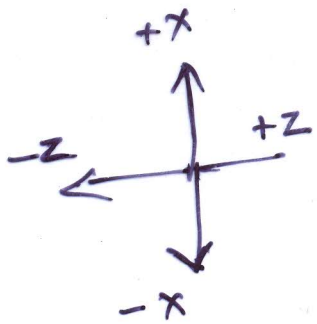

```
O1234 (BILLET X75 Z110)
G28 U0.0;
G28 W0.0;
G21 G99 G90;
M06 T0101;
M04 S1000;
M07;
G00 X76.0 Z2.0;
G71 U0.5 R1.0;
G71 P100 Q200 U0.5 W0.5 F0.5;
G70 P100 Q200 F0.25;
N100 G01 X26.0;
     G01 X30.0 Z-2.0;
     G01 Z-20.0;
     G03 X44.0 Z-31.0 R12.0;
     G01 Z-45.0;
     G01 X55.0;
     G01 Z-67.0;
     G01 X70.0 Z-69.0;
N200 G01 Z-95.0;
M05;
M09;
G28 U0.0;
G28 W0.0;
M30;
```




```
O1234
[BILLET X35 Z85
G21 G99 G90;
G28 U0.0;
G28 W0.0;
M06 T01; M03 S1500; M07;
G00 X35.0 Z2.0;
G71 U0.25 R1.0;
G71 P10 Q11 U0.25 W0.25 F0.5;
G70 P10 Q11 F0.25;
N10 G01 X30.0;
N11 G01 Z-75.0;
G28 U0.0;
G28 W0.0;
M06 T02;
G00 X31.0 Z2.0;
G00 Z-52.0;

G75 R0.5;
G75 X20.0 Z-55.0 P500 Q1500 F0.05;
G28 U0.0;
G28 W0.0;
M06 T03;
G00 X31.0 Z2.0;

G92 X29.5 Z-50.0 F1.2;
G92 X29.0;
G92 X28.5;
G92 X28.44;
G28 U0.0;
G28 W0.0;
M06 T03;
G00 X0.0 Z2.0;
G74 R0.5;
G74 Z-25.0 Q500 F0.2;
G28 U0.0; G28 W0.0;
M05; M09; M30;
```

Multiple turning & multiple facing cycle





O1234

[BILLET X30 Z60

G21 G99 G90;

G28 U0.0;

G28 W0.0;

M06 T0101;

M03 S1000;

M07;

G00 X30.0 Z2.0;

G71 U0.25 R1.0;

G71 P100 Q200 U0.25 W0.0 F0.5;

G70 P100 Q200 F0.25;

N100 G01 X10.0;
Z-10.0;
X20.0;
Z-25.0;
X25.0;
Z-45.0;
N200

G72 W0.25 R0.5;

G72 P101 Q111 U0.0 W0.25 F0.5;

G70 P10 Q11 F0.25;

N101 G01 Z-1.0;

N111 G01 X0.0;

G28 U0.0;

G28 W0.0;

M05;

M09;

M30;

NOTE: G21 P10 in mm

G99; Feed rate / rev

G71, Multiple Turn cycle.

CNC MILLING

G - codes

G94 - Feed / min

G95 - Feed / rev

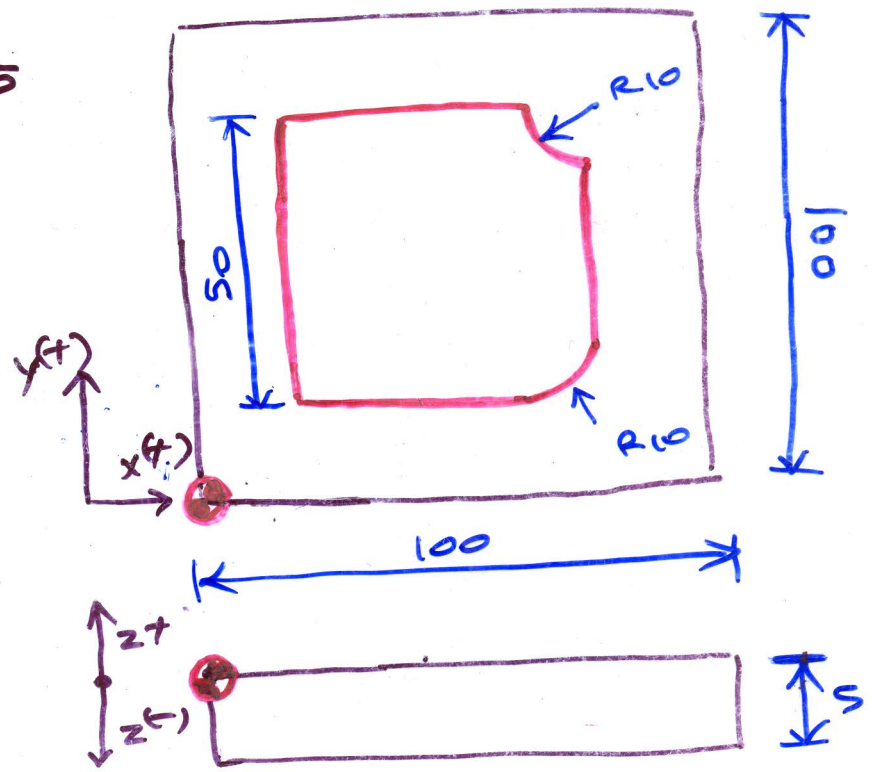
M - codes

M98 - Sub Program call

M99 - End of Sub Program

---

① Write a CNC Program for the following Profile



[BILLET X100 Y100 Z5

[Tool Def T1 D5

G21 G94 G90 G54;

G28 W0.0;

G28 U0.0 V0.0;

M06 T1;

M03 S1000;

G00 X25.0 Y25.0 Z5.0;

G01 Z-1 F60.0;

G01 X25.0 Y75.0;

X65.0 Y75.0;

```
G03 X75.0 Y65.0 R10.0 F90.0;
G01 X75.0 Y35.0 F60.0;
G02 X65.0 Y25.0 R10 F50.0;
G01 X25.0 Y25.0 F60.0;
G01 Z5.0;
G28 W0.0;
G28 U0.0 V0.0;
M05;
M30;
```

70

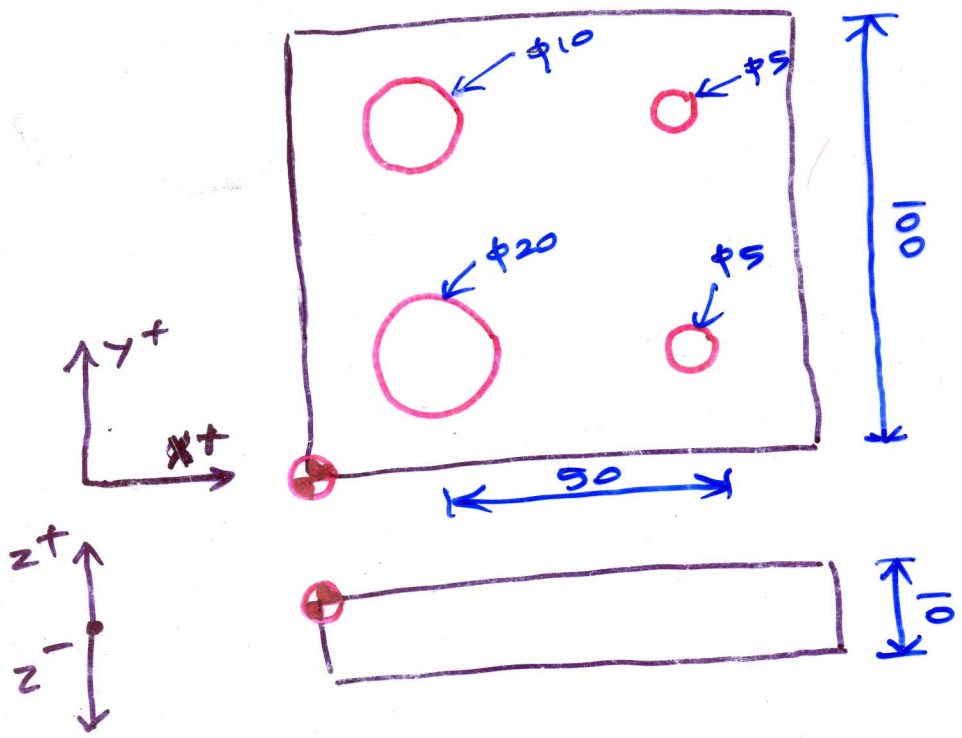
G83 – Peck Drilling Cycle

Syntax:– G83 X Y Z Q R F K

- X & Y : Value of x & y
- Z : Drill hole bottom
- R : Retract Position in Z
- Q : Depth to increase on each Peck
- F : Cutting feed rate
- K : Number of the Peats.

(2) Write a CNC Program for the following Profile

71



[BILLET X100 Y100 Z10

[Tool Def T1 D5

[Tool Def T2 D10

[Tool Def T3 D20

G21 G94 G90 G54;

G28 W0.0;

G28 U0.0 V0.0;

M06 T3;

M03 S800;

M07;

G00 X25 Y25 Z2;

G83 X25 Y25 Z-5 Q1 R0.5 F40 K1;

G54 - offset number

G94 - Feed/min

G90 - Absolute programming

G21 - Metric Units

```
G28 W0.0;
G28 U0.0 V0.0;

M05;
G55;
M06 T02;

M03 S800;

    G00 X25 Y75 Z2;

G83 X25 Y75 Z-5 Q1 R0.5 F40 K1;

G28 W0.0;
G28 U0.0 V0.0;

M05;
G56;
M06 T1;
M03 S800;

    G00 X75 Y75 Z2;

G83 X75 Y75 Z-5 Q1 R0.5 F40 K1;

G00 X75 Y25 Z2;

G83 X75 Y25 Z-5 Q1 R0.5 F40 K1;

G28 W0.0;

G28 U0.0 V0.0;

M05;

M09;

M30;
```

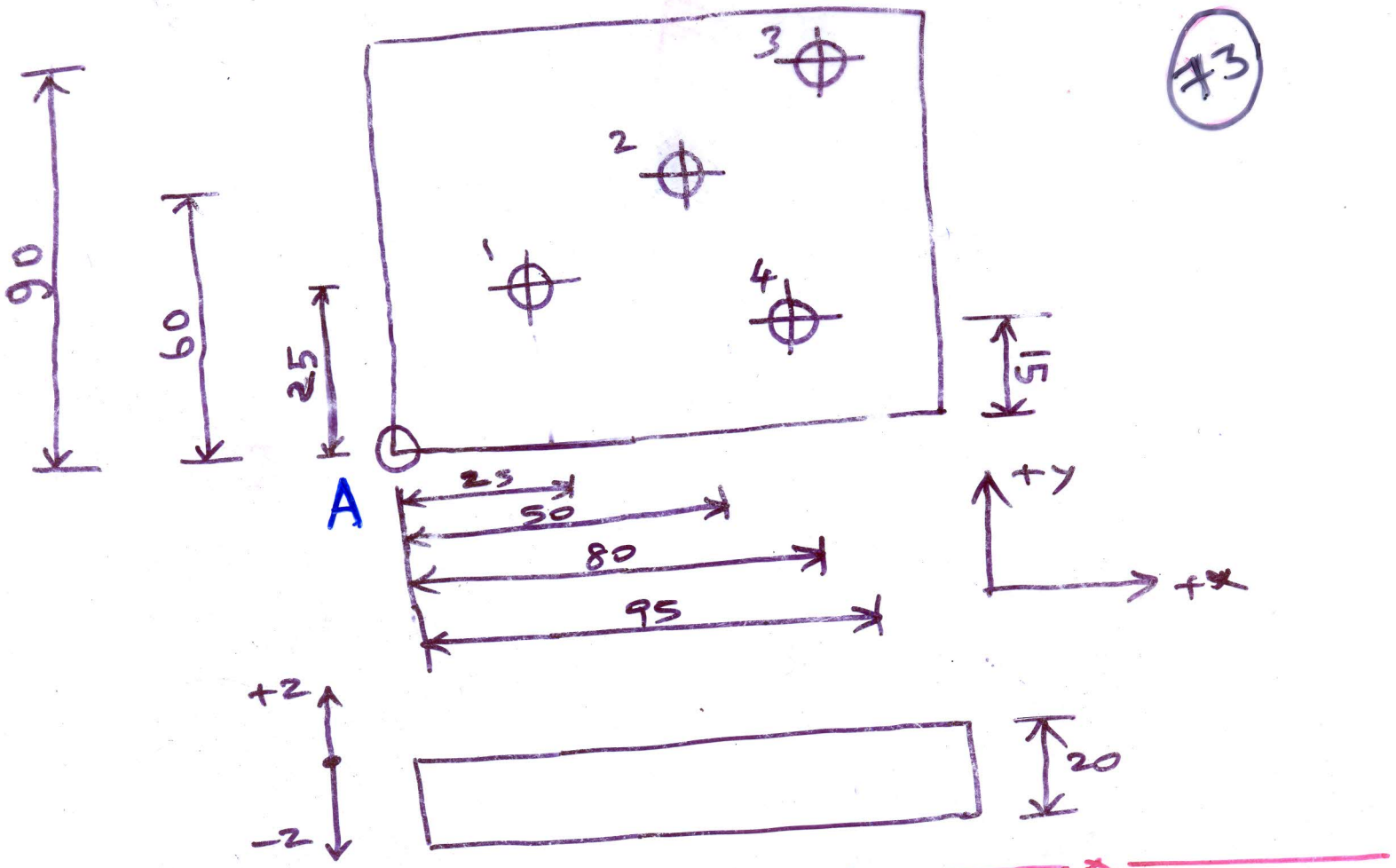
Syntax for Sub Program } M98 P0011234

001 = Number of repeating

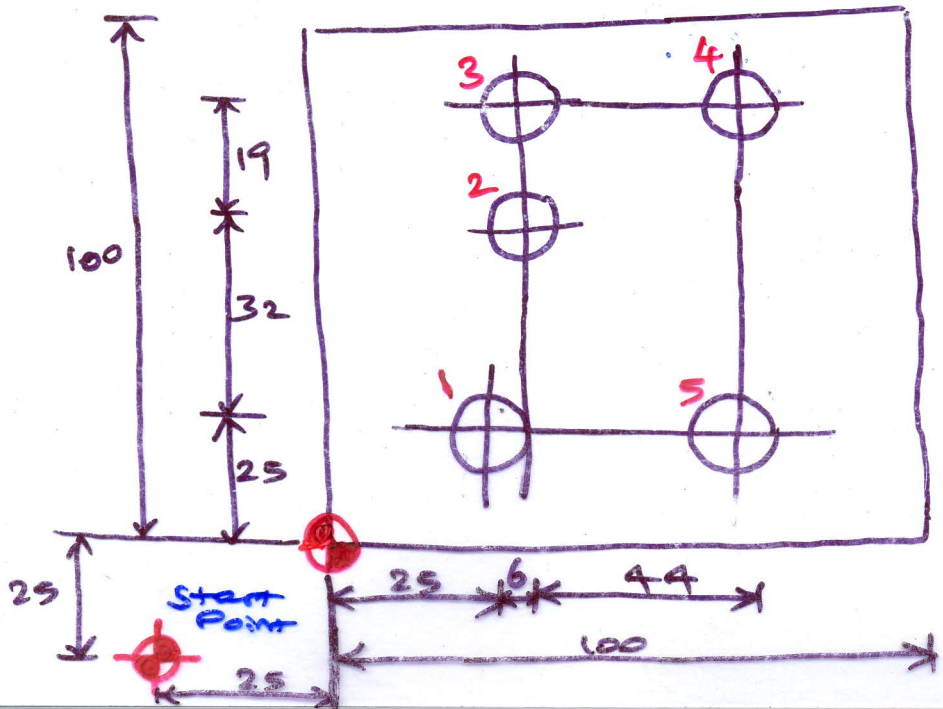
1234 = 4 digit number

(3) Write a Part Program for drilling holes for the Part shown in figure. The plate thickness is 20 mm.

NOV/DEC-2010

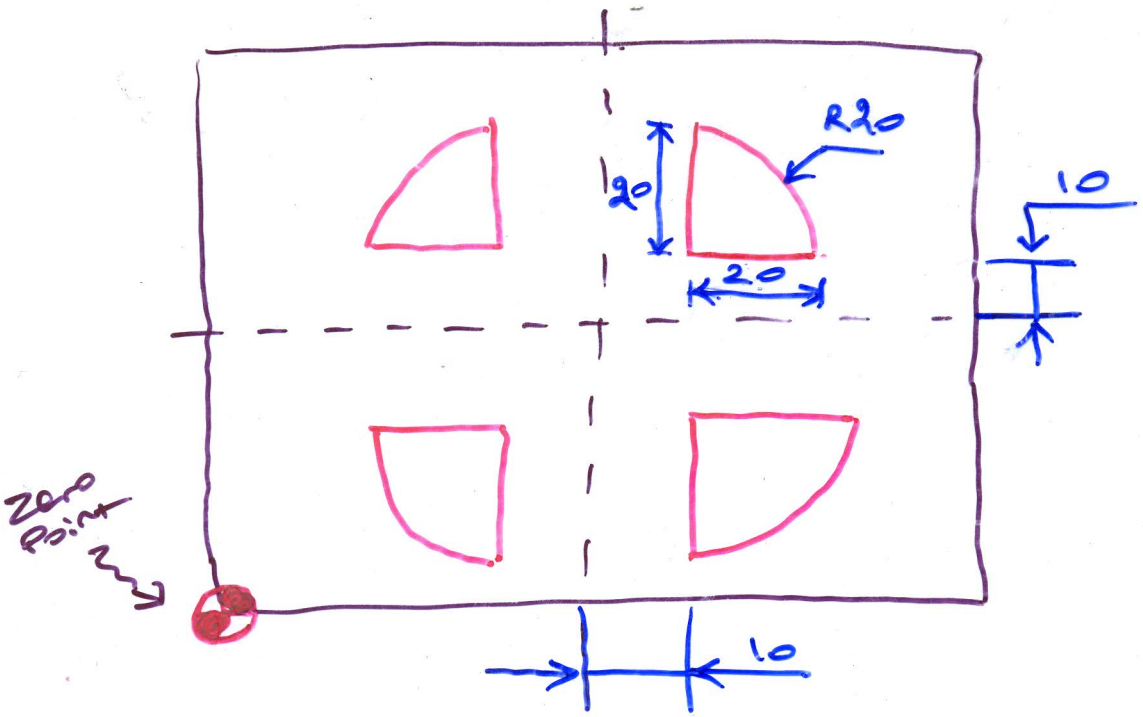


(4) write a Program to drill five hole in the five location.



Write the CNC milling program for the component shown using Mirroring command.

74




```
[ BILLET X100 Y100 Z10
[ Tool DEF T1 D5
[ Edge move X-50 Y-50
G21 G94 G54 G90;
G28 W0.0;
G28 U0.0 V0.0;
M06 T1;
M03 S1000;
M07;
    G00 X0 Y0 Z5;
M98 P0011234;
M70;
M98 P0011234;
M80;
```

Sub program 1234

```
G91;
G00 X10 Y10;
G01 Z-6 F50;
G01 X20 Y0;
G03 X-20 Y20 R20 F30;
G01 X0 Y-20;
G01 Z6;
G00 X-10 Y-10;
G90;
M99;
```

Mcodes for mirroring:-

M70 – X axis Mirror on

M80 – X axis Mirror off

M71 – Y axis mirror on

M81 – Y axis Mirror off

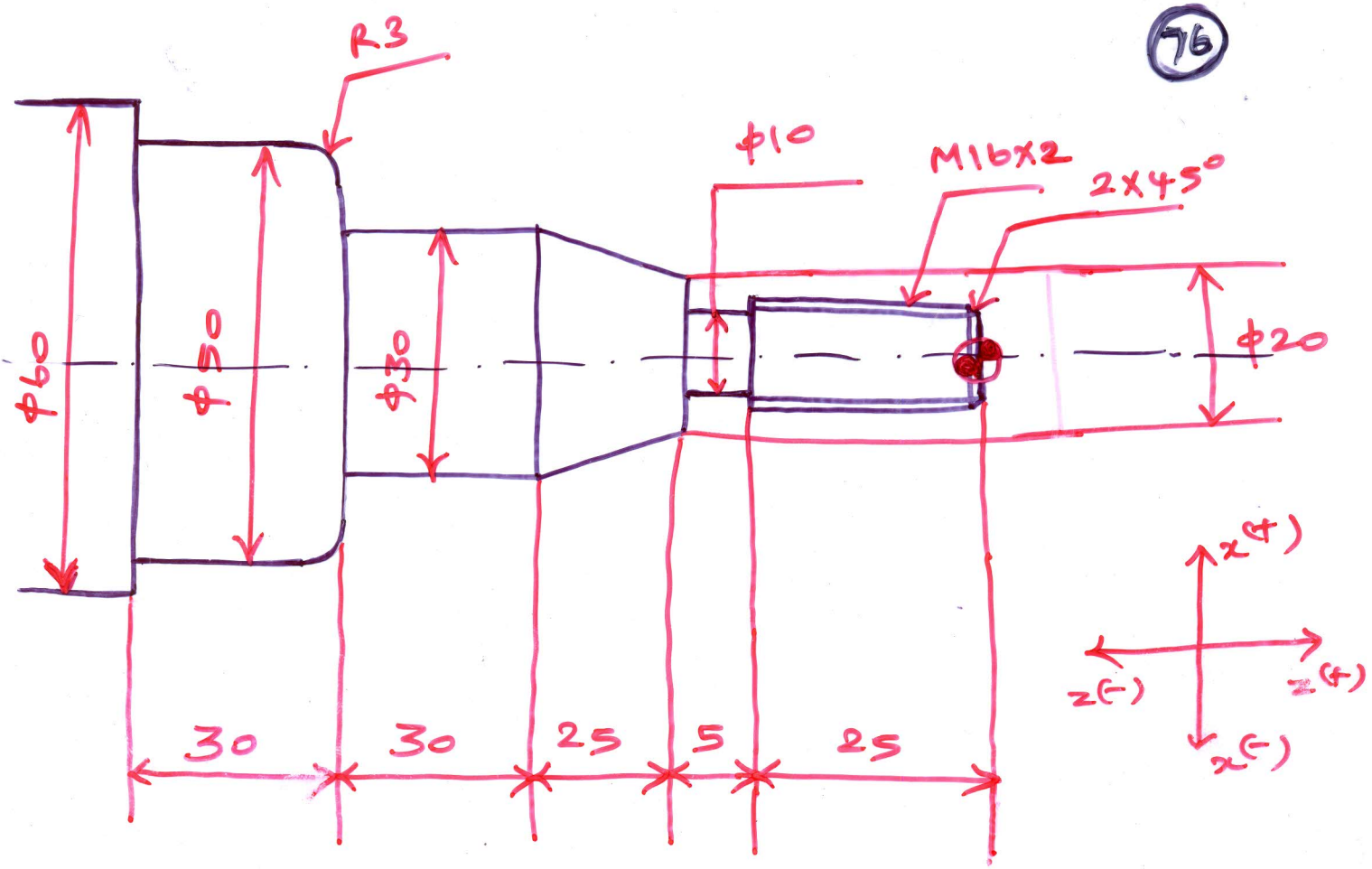
Continuation of the Program

```
M71 ;
M98 P0011234;
M81 ;
M70 ;
M71 ;
M98 P0011234;
M80 ;
M81 ;
G28 W0.0 ;
G28 U0.0 V0.0 ;
M05 ;
M09 ;
M30 ;
```

Write the CNC Program for the Profile shown below

(NOV/Dec'2016)

76



O1234

[BILLET X63 Z130

G28 U0.0;

G28 W0.0;

G21 G99 G90;

M06T0101 [Tool Change to T01 - Single Point Tool]

M04 S1000;

M07;

G00 X63.0 Z2.0;

```
G71 U0.25 R1.0;
G71 P100 Q200 U0.25 W0.0 F0.5;
G70 P100 Q200 F0.25;

N100 G01 X16.0;
        Z-30.0;
        X20.0;
        X30.0 Z-55.0;
        Z-85.0;
        X44.0;
     G02 X50.0 Z-88.0 R3.0;
N200 G01 Z-115.0;
  M05;
  G28 U0.0;
  G28 W0.0;
  M06 T0202;
  G00 X17.0 Z-27.0;
  M04 S500;
  G75 R0.5;
  G75 X10.0 Z-30 P500 Q1000 F0.05;

  M05;
  G28 U0.0;
  G28 W0.0;
```

→ Tool Change : T02 - Grooving Tool
width = 2mm
offset no: 02

[G75 - Grooving cycle]

```
M06 T0303;   [Tool change to T03-threading tool]
G00 X17.0 Z2.0;
M04 S300;
G92 X15.6 Z-25.0 F0.2;
G92 X15.2 Z-25.0;
G92 X14.8 Z-25.0;
G92 X14.4 Z-25.0;
G92 X14.0 Z-25.0;
G92 X13.702 Z-25.0;
M05;
G28 U0.0;
G28 W0.0;
M06 T0101;
G00 X17.0 Z2.0;
M04 S1000;
G01 X12.0 Z0.0 F0.5;
    X16.0 Z-2.0;
M05;
M09;
G28 U0.0;
G28 W0.0;
M30;
```

[Chamfering operation]

G92 = Thread cutting cycle

Pitch value, P = 2

Depth of thread = P × 0.649 = 1.298

Minor dia = Major dia − [2 × depth of thread]

= 16 − 1.298 = 13.702

Depth of cut required = $\frac{16 - 13.702}{2}$ = 1.149

Depth of cut for each pass = 0.2 mm

No of passes = 6

APT Programming:—

* APT means automatically programmed Tools.

* To program in APT, the workpart Geometry must first be defined. Then the tool is directed to various point location and along the surfaces of the workpart to carryout the machining operations.

* There are four types of statements in the APT language

1. Geometry Statements
2. Motion Statements
3. Post Processor Statements
4. Auxillary Statements

① Geometry Statements:—

It defines the geometric elements such as points, lines, circles etc., of the workpart.

Syntax of Geometry Statements

Symbol = keyword to define the geometry | Co-ordinates to define the geometry.

example:— P1 = POINT / 20,30,0

It defines the co-ordinate (20, 30, 0) of a point

and assign the same to "P1".

(2) Motion Statement:—

It describe the path of the cutting tool.

Syntax:—

Motion Command keyword / Co-ordinate Position

Example:—

(1) GoTo/P1 (2) FROM/P0

(3) Post Processor Statement:—

It is used to control the operation of the spindle, feed & other features of the machine.

Syntax:—

Post Processor keyword / related data

example:—

(1) SPINDLE/450 [Spindle Speed = 450 rpm]

(2) FEDRAT/1.5 [feed rate = 1.5 mm/rev]

(4) Auxillary Statement:—

It is used to specify other details like cutter size, tolerance and so on.

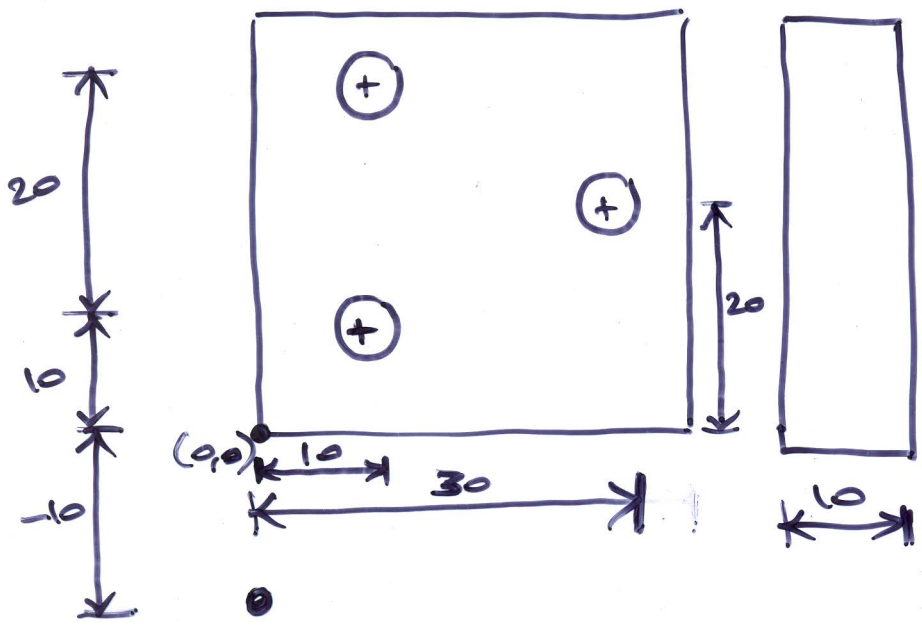
Syntax:-

Auxillary key word | related data.

example:-

CUTTER | 50 [cutter diameter = 50 mm]

(1) Write the APT program for following profile



(Geometry definations)

P0 = POINT | 0, -10, 10 [Initial cutter position]

P1 = POINT | 10, 30, 10

P2 = POINT | 10, 10, 10

P3 = POINT | 30, 20, 10

(PreProcessor/Auxiliary definations)

SPINDLE/550 [Takes the spindle speed 550 r.p.m]

FEDRAT/1.5 [feedrate = 1.5 mm/rev]

COOLT/ON [Coolant motor on]

CUTTER/18 [18mm dia drill bit]

[MOTION Statements]

FROM/P0 [Tools starts from P0]

GOTO/P1 [Tool go to P1]

GO DLTA/0,0,-10 [Drilling takes place at P1 to a depth of -10mm]

GO DLTA/0,0,+10

GO TO/P2

GO DLTA/0,0,-10 [Drilling of hole at P2]

GO DLTA/0,0,+10 [Tools travels back]

GO TO/P3

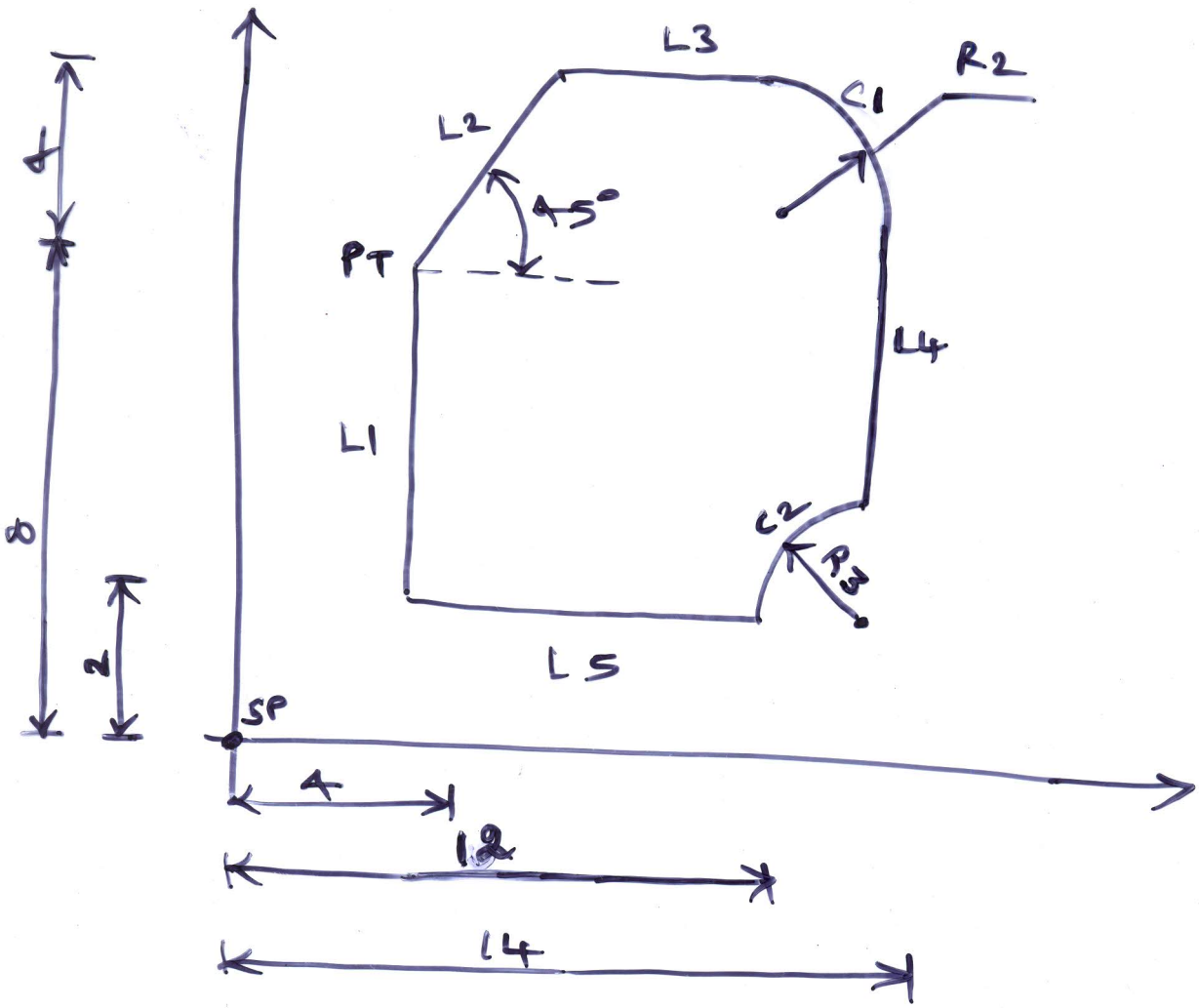
GO DLTA/0,0,-10

GO DLTA/0,0,+10

GOTO/P0

FINI [Finishes line of the Program]

(2) write the APT Program for the following profile



MACHINE/MILL

SP = POINT/0,0,0

L1 = LINE/4,2,0, 4,8,0

PT = POINT/4,8,0

L2 = LINE/PT, ATANGL, 45

L3 = LINE/8,12,0, 12,12,0

L4 = LINE/14,5,0, 14,10,0

L5 = LINE/4,2,0, 12,2,0

C1 = CIRCLE/12,10,0, 2

```
C2 =CIRCLE/14, 20, 3
INTOL/0
OUTTOL/0.005
CUTTER/15
SPINDLE/2000, CCW
COOLANT/ON
FEDRAT/20.0
FROM/SP
GO/TO, L1 [CUTTER to move from current location SP until it is just tangent to L1]
TLLFT, GOLFT/L1, PAST, L2
GORGT/L2, PAST, L3
GORGT/L3, TANTO, C1
GOFWD/C1, TANTO, L4
GOFWD/L4, PAST, C2
GORGT/C2, PAST, L5
GORGT/L5, PAST, L1
GOTO/SP
COOLNT/OFF
SPINDL/OFF
FINI.
```