# *CH- 03 CNC MILLING MACHINE*

## Contents:-


1.Introduction.

2. Types of machining centre.

2.1Horizontal Machining centre.

2.2) Vertical Machining centre.

3.Axis Identification in Milling Machine.

4.Automatic Tool Changer.

5.Tool Magazine:-.

6.Positions on CNC Machine:

7.Part program for Drilling Operation

8.Part program for Slot milling:

9.Part program for straight surface

10.Part Program on Canned cycle for drilling

11.Part program by use of Subroutine

12.Diff. btw. Subroutine and Canned cycle:-.

13.Cutter Radius Compensation:

14.CAPP


By- SUMIT GANGULY

## 1.INTRODUCTION :-

Milling is the process of cutting and drilling material (like wood or metal). A milling machine, regardless of whether it's operated manually or through CNC, uses a rotating cylindrical tool called a milling cutter. It is held in a spindle and can vary in form and size. The capability of CNC machines have also been enhanced by addition of automatic tool changer and tool magazine, thus making them versatile. The CNC milling machine with such capabilities is called as machining centre.

Fig-01 ( Milling machine)

Fig-02 (CNC MILLING MACHINE)

## 2.Types of Machining Center:-

1) Horizontal Machining centre.

2) Vertical Machining centre.

3) Universal machining centre.

### 2.1) Horizontal Machining centre:-

Horizontal machining occurs on a horizontal machining center (HMC), which employs a spindle that is parallel to the ground floor. With a horizontally oriented spindle, tools stick out of the side of the tool holder and cut across the side of a workpiece. They are suitable for large work piece requiring Machining on all the surface. This machining centre are very heavy in construction.

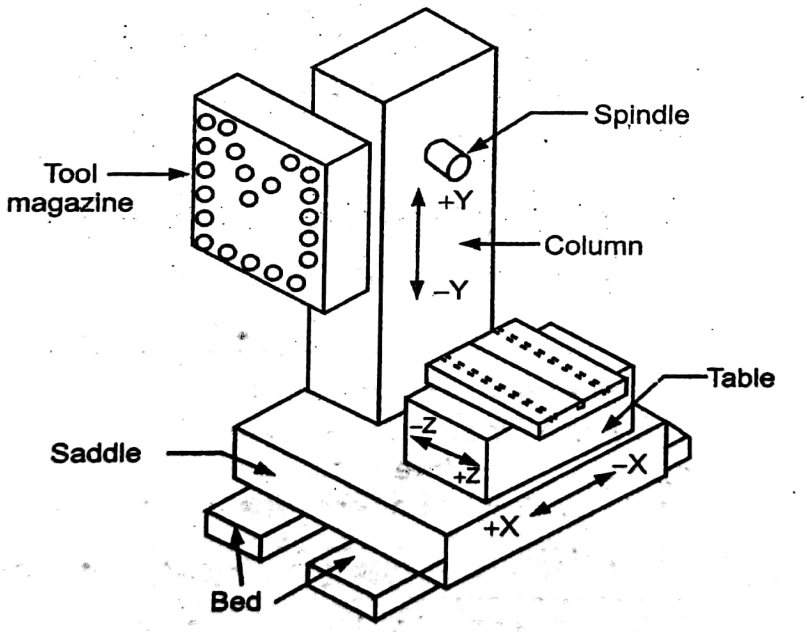



Fig -04

**Parts of Horizontal Machining centre:**

**1. Bed:**- it is heavy cast iron structure which carries the complete machine.

**2. Saddle:-** it is mounted over the guide ways on the bed and carries the column over it. The saddle provide the X axis movement of the machine.

**3. Column:-** it is mounted over the saddle and provides Y axis linear movement to the spindle 4.
**4.Table:-** it is made up of cast iron and mounted over the guide ways machined on the saddle.The table provide Z axis movement.

**5. Tool changer:-** it is an automatic device used to change the tool from the Machine spindle.

**6. Spindle:-** it is mounted on the head stock and provides Z axis rotational movement.

**7. Servo system:-** This system consists of servo motors and feedback system.it enables accurate and rapid movement along X, Y and Z axis.

## 2.2) Vertical Machining centre:-

Vertical machining occurs on a vertical machining center (VMC), which employs a spindle with a vertical orientation. With a vertically oriented spindle, tools stick straight down from the tool holder, and often cut across the top of a workpiece.

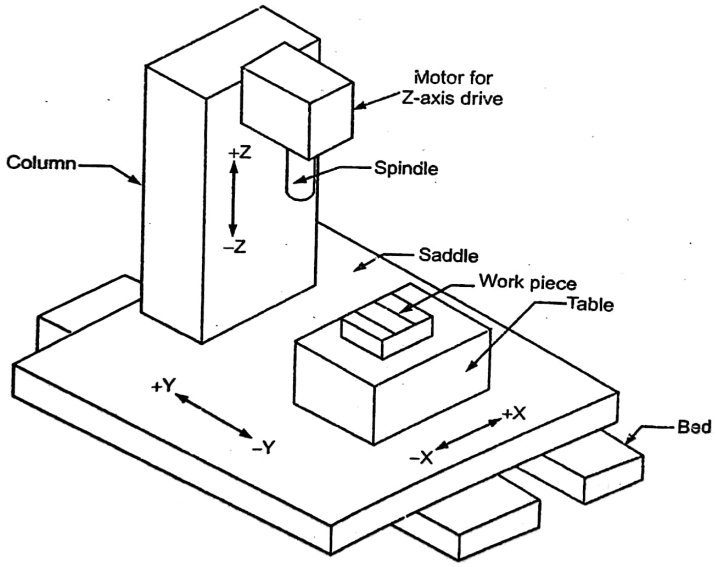



**Fig-04**

**Parts of vertical machining centre:-**

**1. Bed:-** It is a heavy cast iron structure which supports the machine.

**2. Saddle:-** It is mounted over the guide ways on the bed and carries the column over it. The saddle generally provides the X axis movement of the Machining centre.

**3. Column:-** It is mounted over the saddle and provides X axis or Y axis linear movement.

**4. Table:-** It is made of cast iron and mounted over the guide ways machined on the saddle. The table provides X axis movement on the machine center.

**5. Tool changer:-** It is an automatic device used to change the tool from the Machine spindle.

6. Spindle:- It is mounted on the machining head which is mounted on the column. It provides Z axis movement.

**7. Servo system:-** This system consists of servo motors and feedback system. It enabled accurate and rapid movement along X, Y and Z axis.

## 3.Axis Identification in Milling Machine:-

**The axes on the Machining centre are divided into two types:**

**1. Linear axes:-** X,Y and Z are identified as linear axes.

**2. Rotary axes:-** A,B and C axes are identified as rotary axes.

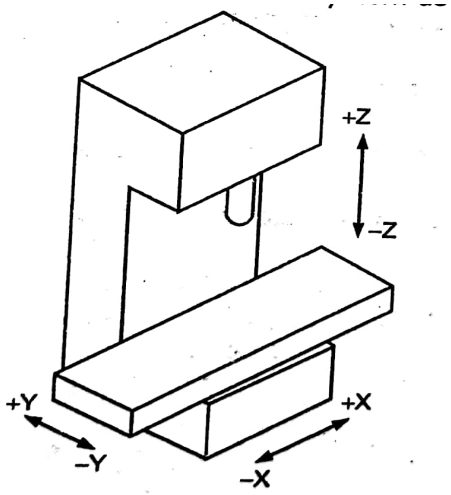



Fig-05 ( Linear axis).

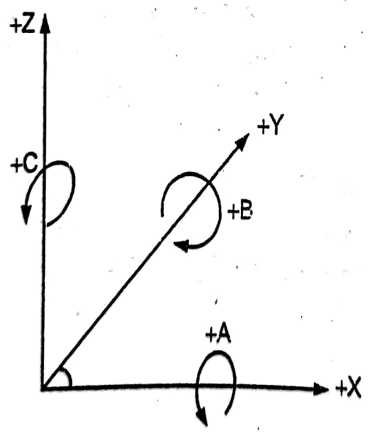



Fig-06 ( rotary axis)

Z-Axis Which axis is which depends on the orientation of the spindle. The axis of motion that is parallel to the spindle axis is always the Z-axis.

X-Axis The mill axis that moves right and left (as the operator is facing the front of the mill) is the X-axis for both vertical and horizontal spindle mills.

Y-Axis The Y-axis on mills is either the in-and-out motion--toward and away from the operator (vertical spindle) or up-and-down motion (horizontal spindle).

A-Axis it is the axis of rotary motion of a tool along X axis.

B-Axis It is the axis of rotary motion of a tool along Y axis.

C-Axis It is the axis of rotary motion of a tool along Z axis

## 4.Automatic Tool Changer:-

An Automatic tool changer or ATC is used in computerized numerical control (CNC) machine tools to improve the production and tool carrying capacity of the machine. ATC changes the tool very quickly, reducing the non-productive time. Generally, it is used to improve the capacity of the machine to work with a number of tools. It is also used to change worn out or broken tools.

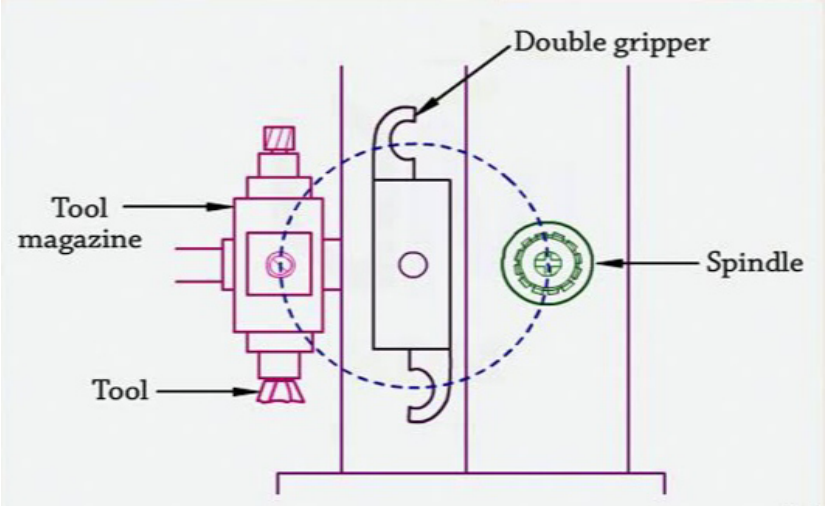



Fig-07

## 5.Tool Magazine:-

The tool magazine is used to store the tool required during the machining Operation. Store capacity of tool magazine starts with about 12 tools and can go upto 200 or even more, while 30 to 60 being most common capacity. The tools are identified by tool number using the same number as in the part program.

### 5.1Types of tool magazine:

**5.1.1) Drum type:** In drum type the tools are arranged around a drum which is generally circular in shape. The drum rotates for the purpose of tool change to bring the required tool to the tool change arm.

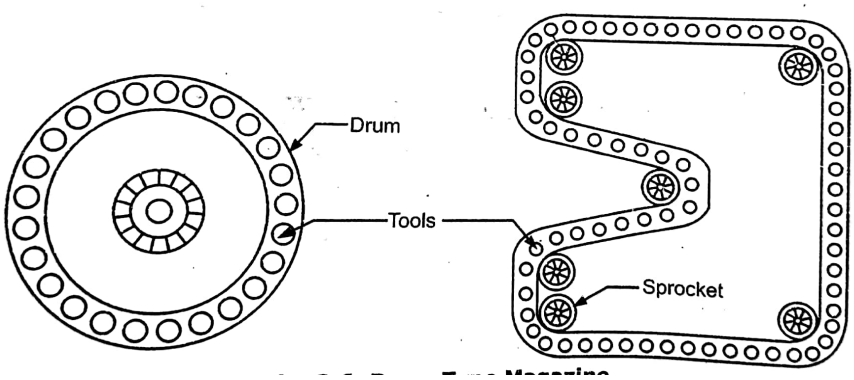



Fig- 08( Drum type). Fig-09 ( chain type)

**5.2.2) Chain type:-** The chain type magazine is used for storing large number of tools with required flexibility. The tool are held in a pocket. The sproket are attached to the chain which moves on the sproket.

## 6.Positions on CNC Machine:

**6.1. Machine zero position:-** Each CNC machine has a built-in location that is called machine zero. This point typically is located at the farthest positive direction along the X, Y, and Z axes, and it cannot be changed by anyone after it leaves the original manufacturer.

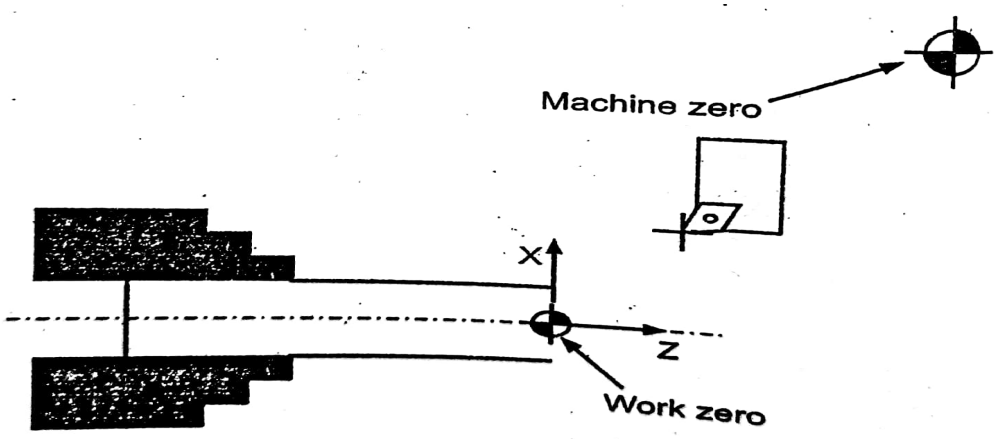



Fig -10 ( Machine and workpiece zero position for cnc lathe)

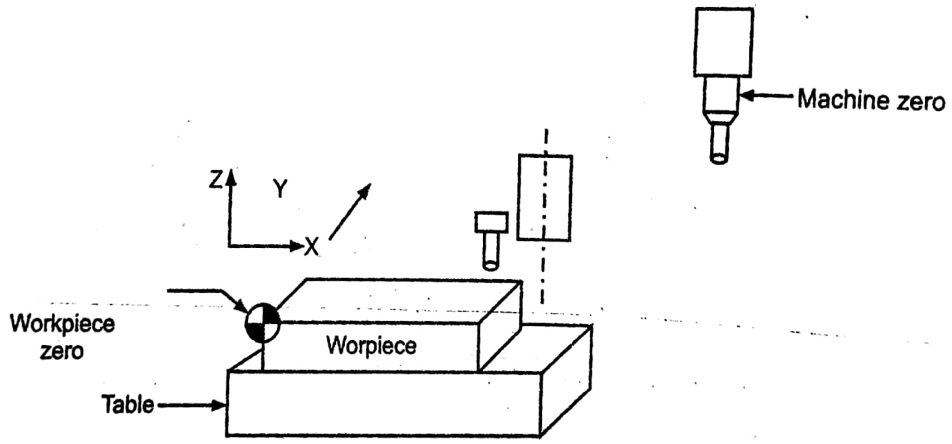
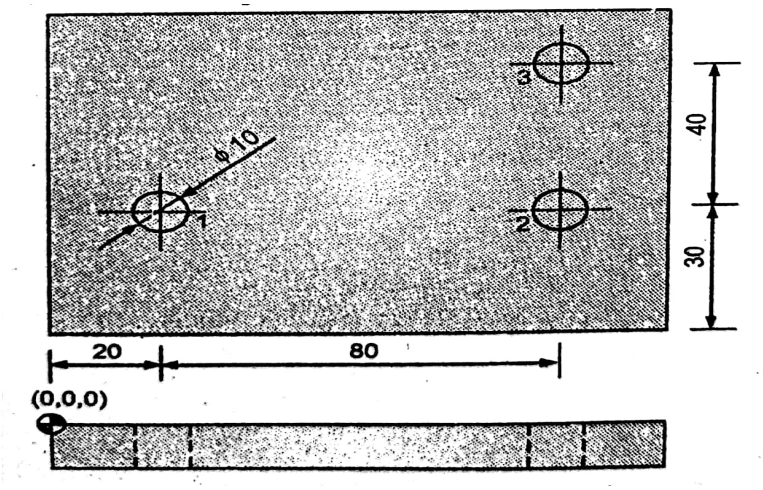



**Fig-11 ( Machine and workpiece zero position for cnc milling)**

**6.2. Workpiece zero position:-** It is the position of cutting tool on the workpiece. It can be selected anywhere on the workpiece. Generally in the south west position of the workpiece for Milling and it is in the front face and at the centre of the cylindrical workpiece for the lathe.

## 7.Part program for Drilling Operation:-

**Ques 1)** Write a program to drill the holes for the job shown in the figure. The depth of workpiece is 5 mm.



**Program no- 0101**

N10 G90 G21 G94 EOB

N20 M03 S1100 M08 EOB

N30 GOO Z4 EOB

N40 G00 X20 Y30 EOB

N50 G01 Z-5 F90 EOB

N60 GOO Z4 EOB

N70 G00 X100 Y30 EOB

N80 G01 Z-5 F90 EOB

N90 G00 Z4 EOB

N100 G00 X100 Y70 EOB

N110 G01 Z-5 F90 EOB

N120 GOO Z4 EOB

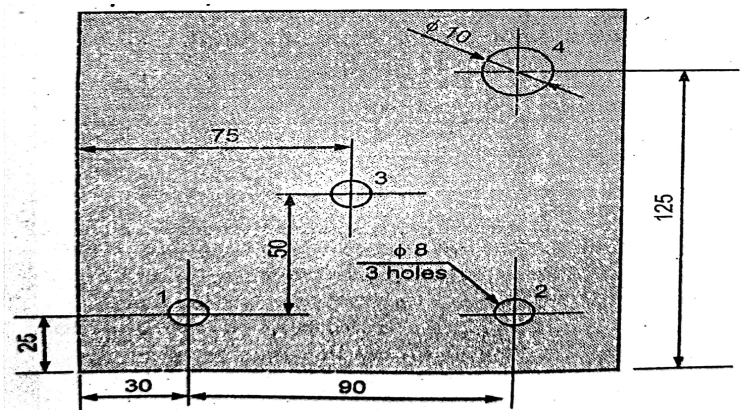
N130 G28 EOB

N140 M05 EOB

N150 M09 EOB

N160 M30 EOB

**Ques 2)** write a program to drill the four holes in the workplace as shown in the figure. The depth of the plate is 6 mm.



## Program no:- 0102

N10 G90 G21 G94 EOB

N20 M06 T01 EOB

N30 M03 S1100 M08 EOB

N40 GOO Z5 EOB

N50 G00 X30 Y25 EOB

N60 G01 Z-6 F50 EOB

N70 G00 Z5 EOB

N80 G00 X120 Y25 EOB

N90 G01 Z-6 F50 EOB

N100 GOO Z5 EOB

N110 G00 X75 Y75 EOB

N120 G01 Z-5 F50 EOB

N130 GOO Z5 EOB

N140 G28 EOB

N150 M05 M09 EOB

N160 M06 T03 EOB

N170 M03 S1100 M08 EOB

N180 GOO Z5 EOB

N190 G00 X120 Y125 EOB

N200 G01 Z-5 F50 EOB

N210 GOO Z5 EOB

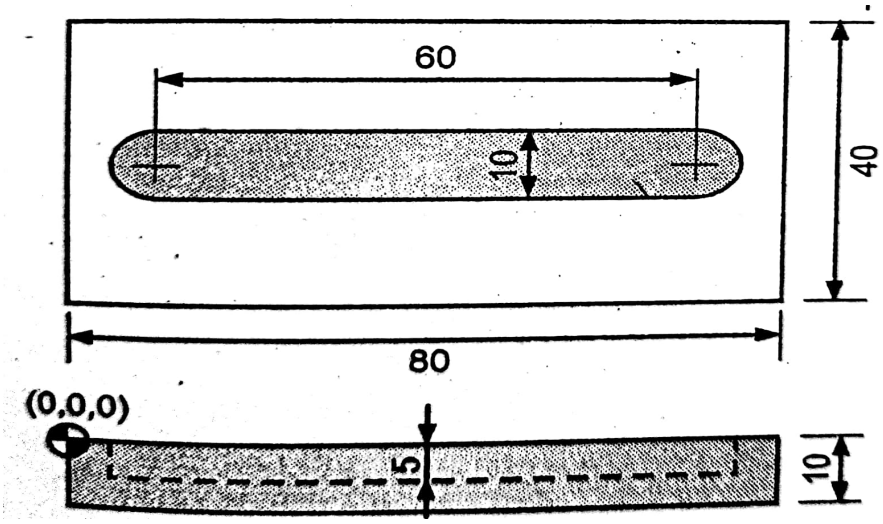
N220 G28 EOB

N230 M05 M09 EOB

N240 M30 EOB

## 8.Part program for Slot milling:

**Ques 3)** write a program to machine the 10 mm slot in the workpiece as shownin the figure. Use 10 mm diameter end mill. The depth of the slot is 5 mm. use a depth of cut of 2.5 mm.




**Program no:- 0103**

N10 G90 G21 G94 EOB

N20 M06 T01 EOB

N30 M03 S900 M08 EOB

N40 G00 Z5 EOB

N50 G00 X10 Y20 EOB

N60 G01 Z-2.5 F50 EOB

N70 X70 Y20 EOB

N80 Z-5 EOB

N90 X10 EOB

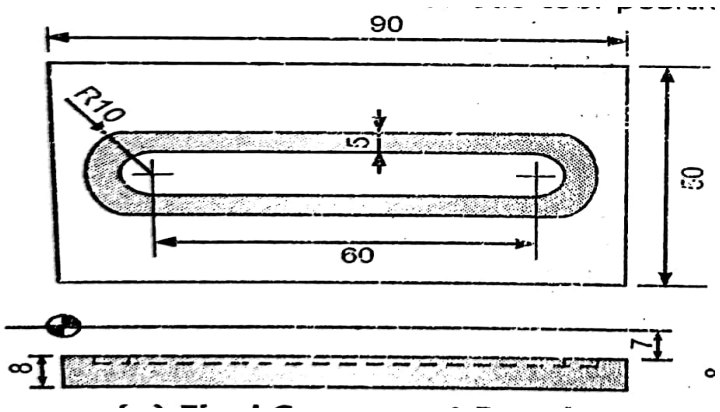
N100 GOO Z5 EOB

N110 G28 EOB

N120 M05 M09 EOB

N130 M30 EOB

**Ques 4)** write a program to machine the 5mm slot inside the workpiece as shown in the figure .use a 5 mm diameter end mill and the depth of the slot is 2 mm.




## Program no:-0104

N10 G90 G21 G94 EOB

N20 M06 T03 EOB

N30 M03 S800 M08 EOB

N40 GOO ZO EOB

N50 G00 X15 Y32.5 EOB

N60 G01 Z-9 F50 EOB

N70 X75 EOB

N80 G02 X82.5 Y25 R5 F50 EOB

N90 G02 X75 Y17.5 R5 F50 EOB

N100 G01 X15 F50 EOB

N110 G02 X7.5 Y25 R5 F50 EOB

N120 X15 Y32.5 R5 F50 EOB

N130 GOO ZO EOB

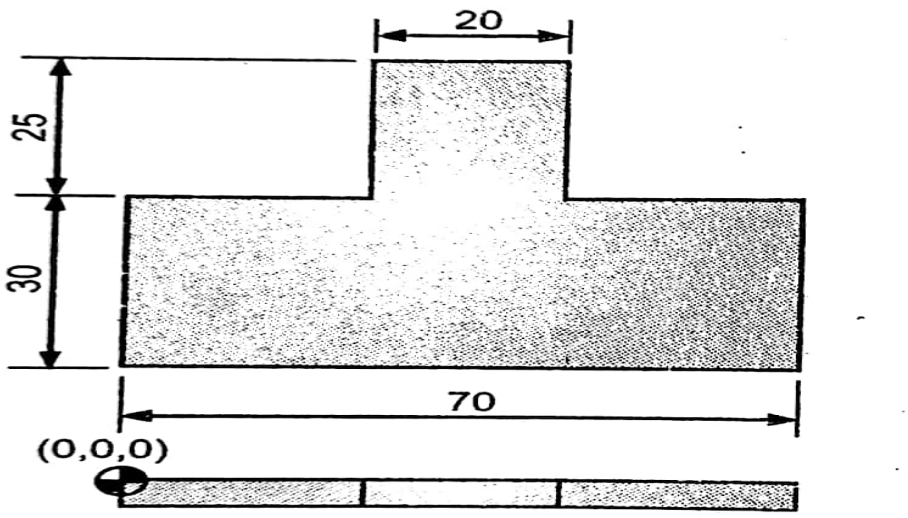
N140 G28 EOB

N150 M05 M09 EOB

N160 M30 EOB

## 9.Part program for Machining straight surface:-

Ques 5) write a part program to machine around the outside of the part as shown in the figure. Use a 6 mm diameter end mill. The part depth is 3 mm. Use feedrate 120 mm/ minute.




## Program no:- 0105

N10 G90 G21 G94 G40 EOB

N20 M06 TO1 EOB

N30 M03 S700 M08 EOB

N40 GO0 Z5 EOB

N50 G00 X-10 Y-3 EOB

N60 G01 Z-3 F120 EOB

N70 X-3 EOB

N80 X73 EOB

N90 Y33 EOB

N100 X48 EOB

N110 Y58 EOB

N120 X22 EOB

N130 Y33 EOB

N140 X-3 EOB

N150 Y-10 EOB

N160 GOO Z5 EOB

N170 G28 EOB

N180 M05 M09 EOB

N190 M30 EOB

## 10.Part Program on Canned cycle for drilling:

When a same operation has to be repeated for number of times then the program becomes lengthy, time consuming, and uses more computer space. All these factors can be eliminated by using Canned cycle. Canned cycle may be defined as a set of instructions stored in the memory of the system to perform a fixed sequence of operation.

### List of canned cycle:

G80 Cancel Canned cycle

G81 Standard Drilling

G82 Spot Drilling

G83 Deep Hole Peck Drilling

G84 Right Hand Tapping

G85 Reaming/Boring

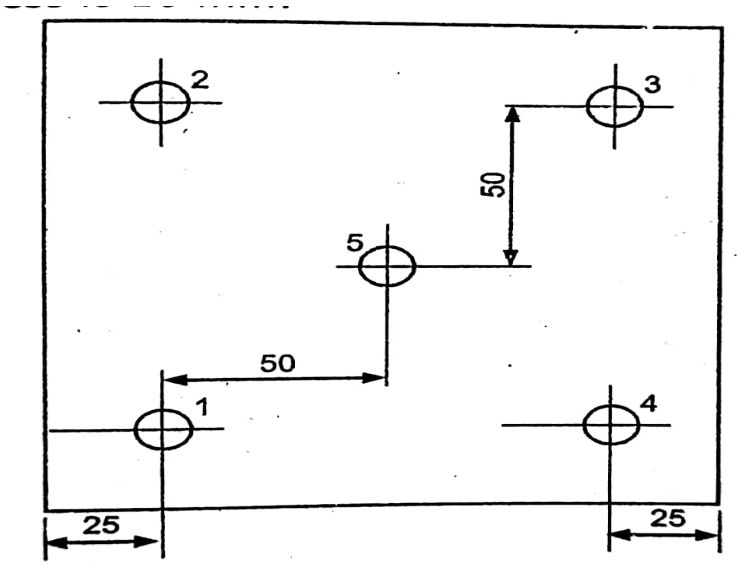
G86 Boring

G87 Back Boring Cycle

G88 Manual Dwell Boring Cycle

G89 Dwell Boring Cycle

**Ques 6)** write a program to drill 5 holes of 8 mm diameter the plate thickness is 10 mm.




**Program no:- 0106**

N10 G90 G21 G94 G40 EOB

N20 M03 S1100 M08 EOB

N30 G00 Z5 EOB

N40 G00 X25 Y25 EOB

N50 G81 X25 Y25 Z-10 R5 F90 EOB

N60 Y125 EOB

N70 X100 EOB

N80 Y25 EOB

N90 X75 Y75 EOB

N100 G80 X150 Y150 Z5 EOB

N110 G28 EOB

N120 M05 M09 EOB

N130 M30 EOB

## 11.Part program by use of Subroutine:-

When an identical machining Operation is to be performed repeatedly then the general method of forming a program will be time consuming and will use more computer memory space. In such case subroutine is a powerful time saving technique. A subroutine is an independent program similar to any usual program. It can be called anywhere in the main program and for any number of times.

**The block instructions of subroutine is:**

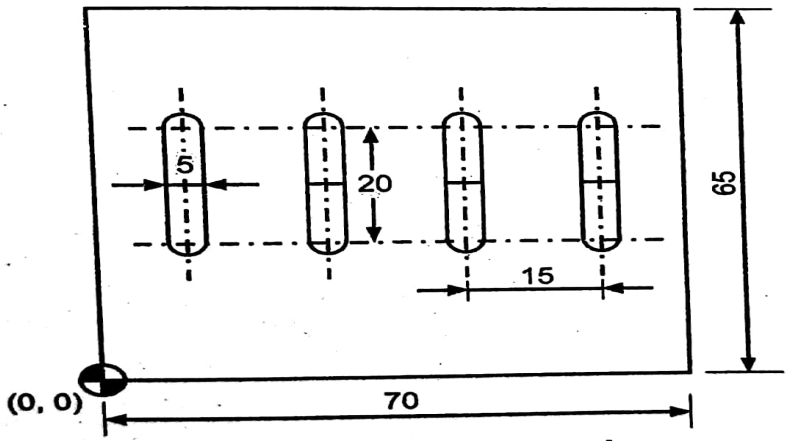
N30 M98 P50 L1 EOB

M98- Instructs the controller to jump to a subroutine.

P50-program number to be used.

L1-Number of time to execute subroutine.

**Ques 7)** Write a program to slot a mill shown in the fig. A) speed= 450 r.p.m. B) feed= 35 mm/min. C)Depth of cut=3mm D) Tool diameter =5 mm.



## Subroutine Program Number 54

N10 G91 EOB

N20 G01 Z-8 F35 EOB

N30 Y20 EOB

N40 GOO Z8 EOB

N50 G90 EOB

N60 M99 EOB

## Main program:

## Program no:-0107

N100 G90 G21 G94 EOB

N110 M03 S450 M08 EOB

N120 GOO ZO5 EOB

N130 X12.5 Y22.5 EOB

N140 M98 P54 L1 EOB

N150 G00 X27.5 EOB

N160 M98 P54 L1 EOB

N170 GOO Y42.5 EOB

N180 M98 P54 L1 EOB

N190 G00 X57.5 EOB

N200 M98 P54 L1 EOB

N210 G28 EOB

N220 M05 M09 EOB

N230 M30 EOB

## 12.Diff. btw. Subroutine and Canned cycle:-

| Subroutine cycle | Canned cycle |
|---|---|
| 1. It is separate which is called in the main program. | 1. It is not a program but part of the main program. |
| 2. It is called and ended by miscellaneous function. | 2. It is called and ended by preparatory function. |
| 3. It is used when multiple passes are required at different locations. | 3. It is used when multiple passes are required at the same location. |
| 4. The cutter part for every point is to be given by the programmer. | 4. The cutter path for every pass is generated by the controller. |

## 13.Cutter Radius Compensation:

It is also called as cutter offset. A workpy is machined by the periphery of the cutting tool and not by the centre of the cutter. When cutter compensation is used the cutter diameter can be ignored and the tool path can be developed for the centre line of the tool rather than the point on the periphery.Compensation Is done by offsetting the tool path by the distance equal to the radius of the cutter.

### What G codes are used for tool radius compensation?

G40: Tool radius compensation cancellation

G41: Tool radius compensation turning the left mode ON

G42: Tool radius compensation turning the right mode ON

## 14.Computer Assisted Part Programming (CAPP)

Manual part programming is suitable for simple jobs but for complicated jobs part programming can be time consuming, lengthy and subject to errors. Therefore computers are used for such jobs. Computer Assisted Part Programming plays a major role in such situations. In CAPP the computer is used to perform calculation which other wise would have done by the

programmer. In CAPP the machine instructions are written in English like statement that are translated by the computer into the low level machine code that can be interpreted and executed by the machine tool controller. The CAPP requires both human and computer involvement for doing any job. And they are termed as:

## 14.1) Part programmer job:

**a) Define the part geometry:-** Any work piece geometry is generally composed of basic geometric elements. It is part programmers task to identify and list the geometric elements of which the part is composed of.

**b) Specifying tool path and operation sequence:-** After the part geometry has been defined, the part programmer must specify the tool path that the cutter will follow.

## 14.2) The computers job:

**a) Input translation:-** part programmer enters the program written in APT. The input translation unit convert it into computer usable form.

**b) Arithmetic calculation:-** Computer solves the mathematics required to generate the required surface.

**c) Cutter offset computation:-** The computer automatically offset the tool path according to the periphery of the cutter.

**d) Post processor:-** It is a seperate computer program written to prepare punched tape.

## Important links:-

Horizontal machining centre:- https://youtu.be/KKB6HT9Cqqc

Vertical machining centre:- https://youtu.be/oxxvlx1FObQ

Automatic tool changer:- https://youtu.be/tvhEea2adz0