

## Experiment No.-1

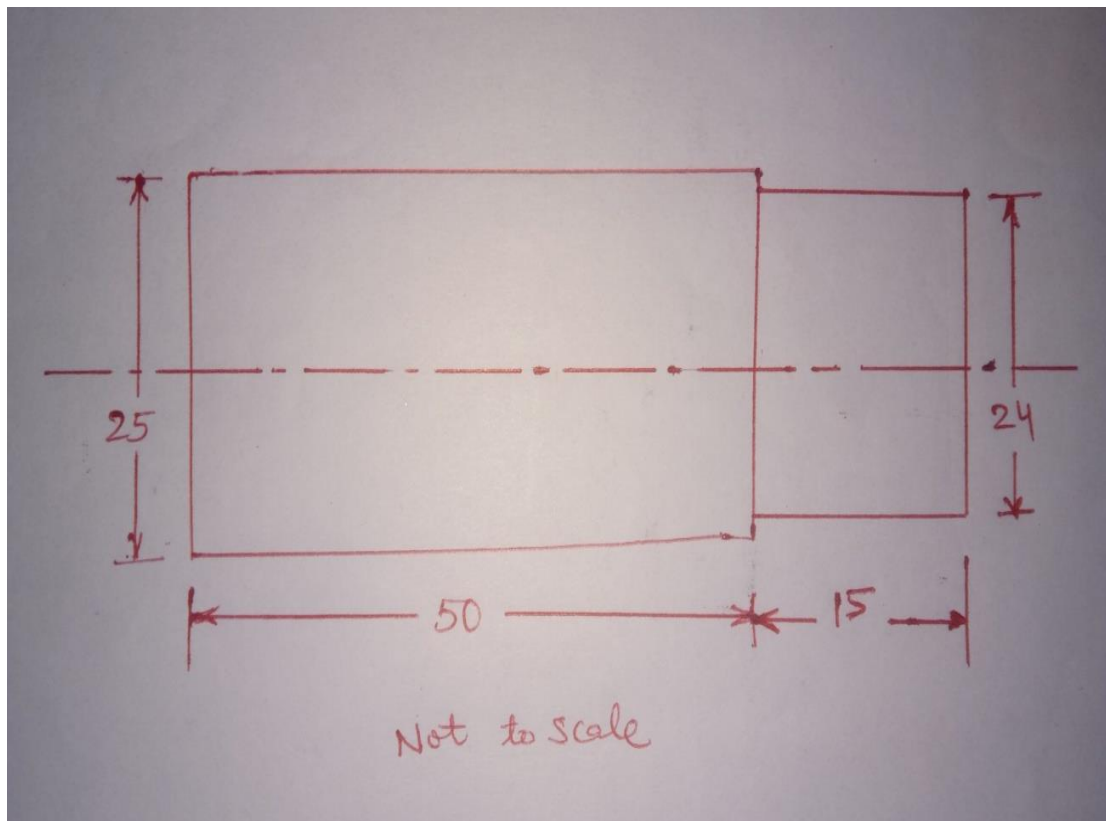
### Object:

To prepare part program for plain turning operation.

### Procedure:-

- Make part drawing (Taking suitable dimension).
- Prepare part program using G and M codes in cut viewer turn software.
- Run the program in simulation mode on PC.
- Observe the satisfactory machining of job on PC as per drawing/Sketch

### Part Program:-



Program statement	Description
(TOOL/STANDARD, 40, 40, 0, 10, 3)	Turning tool description (Non-executable)
(STOCK/ 50, 25, 0, 0)	Stock description (Non-executable)
N10 M03 S2000	Spindle start clockwise with spindle 2000 RPM
N20 G00 X25 Z2	Rapid tool positioning (dia 25, length Z=2) up to reference point
N30 G01 X24	Linear interpolation up to X=24, with 1mm cut
N40 Z-15 F80	Up to length -15mm with feed rate 80mm/min
N50 G00 X25 Z2	Rapid tool positioning up to reference point
N60 M30	Program End and Rewind

**Result:** - Part program of the given dimension has been prepared and also run on the software successfully.

## Experiment No.-2

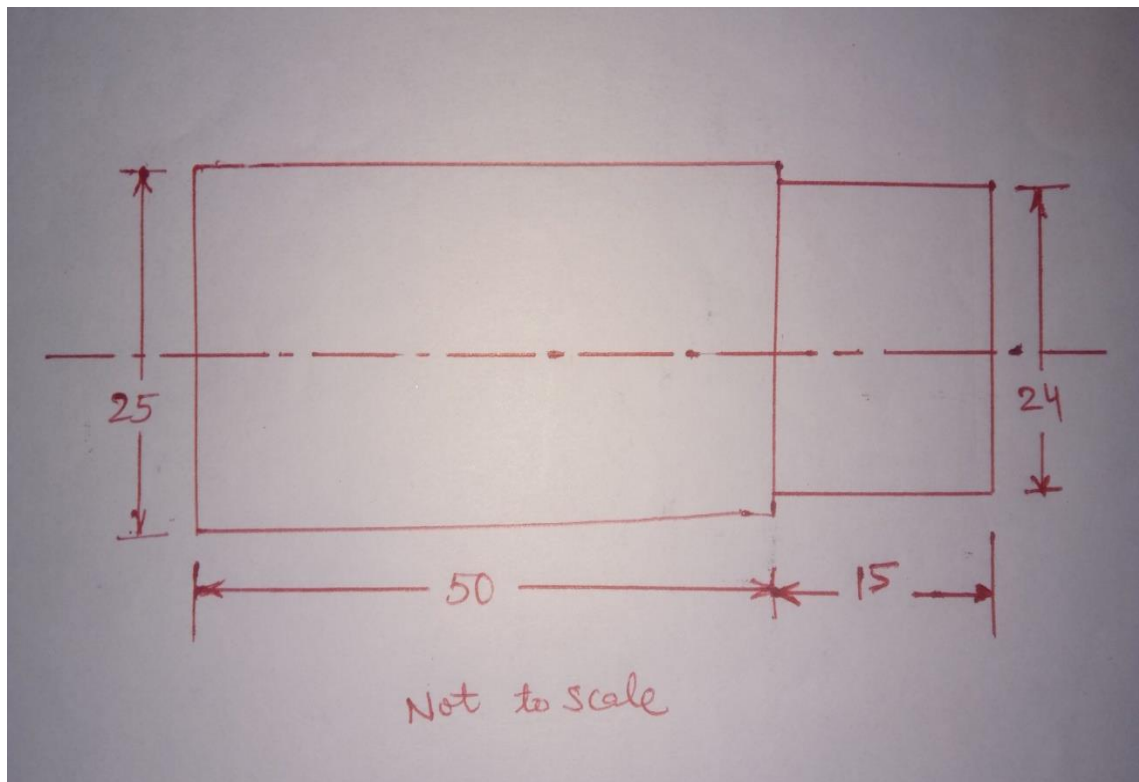
### Object:

To prepare part program for turning operation.

### Procedure:-

- Make part drawing (Taking suitable dimension).
- Prepare part program using G and M codes in cut viewer turn software.
- Run the program in simulation mode on PC.
- Observe the satisfactory machining of job on PC as per drawing/Sketch

### Part Program:-



Program statement	Description
(TOOL/STANDARD, 40, 40, 0, 10, 3)	Turning tool description (Non-executable)
(STOCK/ 50, 25, 0, 0)	Stock description (Non-executable)
N10 M03 S2000	Spindle start clockwise with spindle 2000 RPM
N20 G00 X25 Z2	Rapid tool positioning (dia 25, length Z=2) up to reference point
N30 G01 X24	Linear interpolation up to X=24, with 1mm cut
N40 Z-15 F80	Up to length -15mm with feed rate 80mm/min
N50 G00 X25 Z2	Rapid tool positioning up to reference point
N60 M30	Program End and Rewind

**Result:** - Part program of the given dimension has been prepared and also run on the software successfully.

### Experiment No. – 3

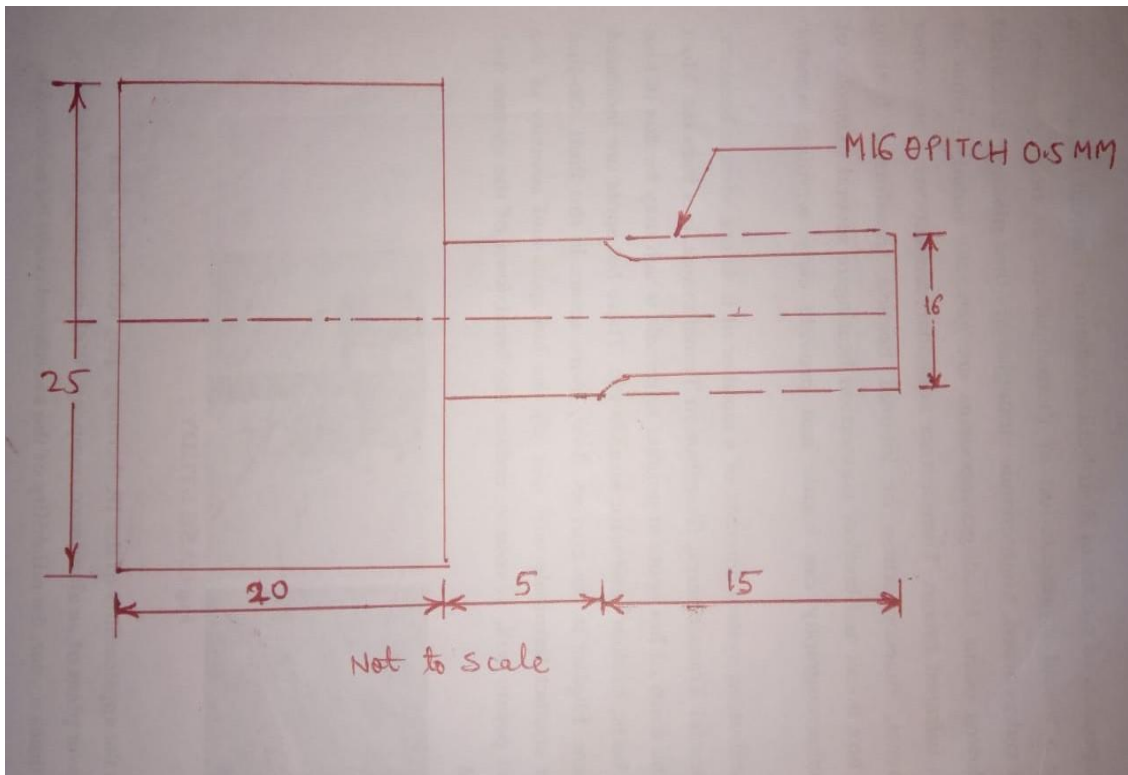
#### Object:

To prepare part program of given drawing for threading operation.

#### Procedure:-

- Make part drawing (Taking suitable dimension).
- Prepare part program using G and M code and cut viewer software.
- Keyed in the program on PC.
- Run the program on PC.
- Observe the satisfactory machining of job on PC as per drawing/Sketch

#### Part Program:-



**Program statement****Description**

(TOOL/STANDARD, 40, 40, 0, 10, 3)

Turning tool description (Non-executable)

(STOCK/ 40, 25, 0, 0)

Stock description (Non-executable)

N10 M03 S2000

Spindle start clockwise with spindle 2000 RPM

N20 T1

Tool No.1 Present for cutting operation

N30 G00 X25 Z2

Rapid tool positioning (X=25, Z=2) up to the reference point

N40 G90 X24 Z-20 F80

Diameter cutting cycle with 1mm cut up to -20 length with feed F=80mm/min

N50 G90 X23 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N60 G90 X22 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N70 G90 X21 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N80 G90 X20 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N90 G90 X19 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N100 G90 X18 Z-20

Diameter cutting cycle with 1mm cut up to -20 length

N110 G90 X17 Z-20

Diameter cutting cycle with 1mm  
cut up to -20 length

N120 G90 X16.8 Z-20

Diameter cutting cycle with 1mm  
cut up to -20 length

N130 G00 X30 Z40

Rapid positioning up to the  
reference point (i.e. X=30, Z=40)  
for Tool change

(Tool/Thread, 40, 20, 5, 90)

T3

Tool No.3 present for cutting  
operation

N140 G00 X17 Z0

Rapid positioning up to the  
reference point (i.e. X=17, Z=0)

N150 G01 X16.8

Linear interpolation up to given  
point (0.1mm cut)

N160 G32 Z-10 F1

Thread cutting cycle with 0.1mm  
cut up to -10mm length with feed  
rate 1mm

N170 G01 X18

Calling of Tool back with safe  
amount (i.e. X=18) for next cutting  
operation

N180 G01 Z2

Calling of Tool back with safe  
amount (i.e. Z=2) for next cutting  
operation

N190 G01 X16.7

Linear interpolation up to given  
point (0.1mm cut)

N200 G32 Z-10 F1

Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm

N210 G01 X18

Calling of Tool back with safe amount (i.e. X=18) for next cutting operation

N220 G01 Z2

Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation

N230 G01 X16.6

Linear interpolation up to given point (0.1mm cut)

N240 G32 Z-10 F1

Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm

N250 G01 X18

Calling of Tool back with safe amount (i.e. X=18) for next cutting operation

N260 G01 Z2

Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation

N270 G01 X16.5

Linear interpolation up to given point (0.1mm cut)

N280 G32 Z-10 F1

Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm

N290 G01 X18

Calling of Tool back with safe amount (i.e. X=18) for next cutting operation

N300 G01 Z2

Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation



N310 G01 X16.4	Linear interpolation up to given point (0.1mm cut)
N320 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N330 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N340 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation
N350 G01 X16.3	Linear interpolation up to given point (0.1mm cut)
N360 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N370 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N380 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation
N390 G01 X16.2	Linear interpolation up to given point (0.1mm cut)
N400 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N410 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N420 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation

N430 G01 X16.1	Linear interpolation up to given point (0.1mm cut)
N440 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N450 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N460 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation
N470 G01 X16.0	Linear interpolation up to given point (0.1mm cut)
N480 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N490 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N500 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation
N510 G01 X15.9	Linear interpolation up to given point (0.1mm cut)
N520 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N530 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N540 G01 Z2	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation

N550 G01 X15.8	Linear interpolation up to given point (0.1mm cut)
N560 G32 Z-10 F1	Thread cutting cycle with 0.1mm cut up to -10mm length with feed rate 1mm
N570 G01 X18	Calling of Tool back with safe amount (i.e. X=18) for next cutting operation
N580 G01 Z30	Calling of Tool back with safe amount (i.e. Z=2) for next cutting operation
N590 M30	Program End and Rewind

**Result:-** Part program of the given dimension has been prepared and also run on the software successfully.

## Experiment No. - 4

**Object:-** To study G and M codes commonly used in the CNC lathe Trainer and CNC Mill Trainer.

### **THEORY:-**

In order to facilitate various functions of the CNC machines there has been developed a specific code structure by various CNC software developing companies (eg. FanucFota, Siemens etc.). These codes are accepted globally and can be divided into two broad categories:

- (i) G-CODES
- (ii) M-CODES

### **LIST OF COMMONLY USED G-CODES**

#### **CNC Milling G Code List**

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G17	X Y plane selection

G18	Z X plane selection
G19	Y Z plane selection
G28	Return to reference position
G30	2nd, 3rd and 4th reference position return
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation + direction
G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G68	Coordinate rotation

G69	Coordinate rotation cancel
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
G80	Canned cycle cancel
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
G98	Return to initial point in canned cycle

G99	Return to R point in canned cycle
-----	-----------------------------------

### CNC Lathe G Code List

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G09	Exact stop
G10	Programmable data input
G20	Input in inch
G21	Input in mm
G22	Stored stroke check function on
G23	Stored stroke check function off
G27	Reference position return check

G28	Return to reference position
G32	Thread cutting
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G70	Finish machining cycle
G71	Turning cycle
G72	Facing cycle
G73	Pattern repeating cycle
G74	Peck drilling cycle
G75	Grooving cycle
G76	Threading cycle
G92	Coordinate system setting or max. spindle speed setting
G94	Feed Per Minute
G95	Feed Per Revolution
G96	Constant surface speed control
G97	Constant surface speed control cancel



## **DESCRIPTION OF G-CODES**

### **G00 (Rapid Traverse)**

G00 is used for positioning the tool rapidly at the desired point. In rapid traverse, the machine moves at a prespecified, rapid rate. It is used when cutting tool is not cutting the material, but is being positioned with respect to the work piece. G00 helps in saving machining, time as rapid traverse rate is much higher compared to feed rate. For the machine manufacturer sets rapid traverse, feed rate, user has no control over it. Hence it can be different for different machine tools. For example,

G00 X 0.0 Z 2.0

implies rapid traverse of the tool to X = 0 and Z = 2 coordinates.

### **G01 (Linear Interpolation)**

G01 is used to move the cutting tool with the feed rate. Hence, it is used when material has to be actually removed from the work piece. Feed rate has to be specified by the programmer in the part program. For example, the command

G01 X - 1.0 Z - 35.0 F 30

moves the tool to X = - 1 and Z = - 35 at feed rate 30

### **G02 (Circular Interpolation, Clockwise)**

G02 is used when machining has to be done in a circular path in clockwise direction. While using this command, other information that has to be specified is: Plane of machining (i.e. XY/YZ/ZX), position of end point of arc, radius of the arc. For example,

G02 X 10.0 Z - 10.0 R10.0 F 20

causes tool to move clockwise along arc of radius 10 from current position to X = 10, Z = - 10 at feed rate of 20.

### **G03 (Circular Interpolation, Counterclockwise)**

G03 is used when machining has to be done in a circular path in counter clockwise direction.

#### **G04 (Dwell)**

G04 causes program to wait for a specified time. It can be used to rotate the tool at a fixed position for a specified period of time.

The format of the command is

G04 X(t)

where t is the dwell time.

For example, if a drill or reamer should dwell in a hole for 5 seconds; the command can be written as

G04 X5.0

#### **G17, G18, G19**

G 17, G 18 and G 19 are used to select XY, XZ and YZ planes respectively in milling operation. For example, G 17 G03 X200.0 Y25.0 R 25.0 indicates movement of tool along circular arc in anticlockwise direction in XY plane.

#### **G20 and G21**

G20 and G21 is used to enter input data in inches and millimeter respectively.

#### **G28**

G28 is used to move the tool to a reference point via an intermediate point. The movement is accomplished in rapid traverse mode.

#### **G32, G78**

It is used for cutting straight or tapered threads, but since it does not allow automatic return to the start point, G78 i.e. multiple threading cycle is used. G78 is

canned cycle unlike G32. The format of this cycle comprises of two blocks. For example,

G78            P021056        Q180        RO.18

G78            X20            Z-5        RO        P1000        Q200        F2.0

In the first block for P021056, 02 refers to number of finishing cuts, 10 is the chamfer value and 56 is flank angle of thread in degrees. Q is the minimum cutting depth in microns and R is finishing offset (allowance) in millimeter. In the second block, X and Z are the coordinates. R is incremental taper value with sign (R = 0 for cylindrical threads), P is the thread depth in microns (always positive). Q is cutting depth of first cut in microns (radius value, without sign), F is thread pitch in millimeter.

#### **G40, G41, G42 (Cutter radius compensation)**

An amount equal to its radius uses G41 and G42 to offset the cutter from its predesigned path. For example, consider end milling of the rectangular plate as shown in figure 3.1. Usually, the edges of the plate are defined as the trajectory of the cutter, while in fact, the axis of the tool should get offset from the edges by an amount equal to radius of the

cutter and move along dotted path shown in the figure. In order to offset the tool from I workpiece, by an amount equal to its radius, workpiece shape is programmed with cutter compensation made by the programmer. During machining, if offset value equal to cutter radius is set in the CNC controller, the tool path is offset automatically. The offset amount is set in the offset memory. G41 is used when cutter moves on the left side of the workpiece and G42 is used when cutter moves on the right side of the workpiece. G40 and G41 are used in conjunction with G00 or G01. The radius compensation is cancelled with G40. It is incorporated in the same block as G00 or G01 or in the proceeding block.

#### **G43, G44, G49**

In NC and CNC machines, the tool magazine carries a number of tools. Tools of different lengths are present in the magazine. Hence, it becomes necessary that their length or height should be compensated. G43 and G44 are used for giving tool length compensation while G49 is used to cancel tool length compensation. For example,

G43 TO2 HO2

G44 TO3 HO3

The CNC software, has a table which has tool information such as tool number, code number and offset value. Refer the following table.

<b>Tool Number</b>	<b>Code number</b>	<b>Offset value (assumes)</b>
T001	H001	200
T002	H002	150
T003	H003	-100
T004	H004	-300

For positive offset value, G43 is used while for negative offset value, G44 is used. G49 is used at the end of the program to cancel tool-length offset value.

### **G90 and G91**

In part programming, options for absolute and incremental dimensioning systems are available. In absolute programming, the origin is fixed, while in incremental programming, for every next point, previous point, becomes the origin. G90 is used for absolute dimension programming; G91 is used for incremental dimension programming.

If G90 is used, command for defining point B will be:

G90 X 45 Y 50

If G91 is used, command for defining point B will be:

G91 X 35 Y 40

## **G92**

It is used to define coordinate system setting. It is used to specify the location of system origin, relative to starting point of the cutting tool in milling, drilling and some lathe machines.

## **G94, G95, G98, G99**

In drilling or milling operation, G94 is used to define feed rate in mm per minute while G95 is used to define feed rate in mm per, revolution. In turning operation, G98 is used for feed rate in mm per minute while G99 is used for feed rate in mm per revolution.

## **LIST OF COMMONLY USED M-CODES**

### **CNC Milling M Code List**

<b>M code</b>	<b>Description</b>
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M06	Too change
M07	Coolant ON – Mist coolant/Coolant thru spindle
M08	Coolant ON – Flood coolant

M09	Coolant OFF
M19	Spindle orientation
M28	Return to origin
M29	Rigid tap
M30	End of program (Reset)
M41	Low gear select
M42	High gear select
M94	Cancel mirrorimage
M95	Mirrorimage of X axis
M96	Mirrorimage of Y axis
M98	Subprogram call
M99	End of subprogram

### CNC Lathe M Code List

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW

M04	Spindle start reverse CCW
M05	Spindle stop
M08	Coolant on
M09	Coolant off
M29	Rigid tap mode
M30	End of program reset
M40	Spindle gear at middle
M41	Low Gear Select
M42	High Gear Select
M68	Hydraulic chuck close
M69	Hydraulic chuck open
M78	Tailstock advancing
M79	Tailstock reversing
M94	Mirrorimage cancel
M95	Mirrorimage of X axis
M98	Subprogram call
M99	End of subprogram

## **DESCRIPTION OF M COMMANDS**

### **M00 (Program Stop)**

This command is used to stop the execution of part program. Main spindle, feed and coolant will be switched off. Also, the chip protection door can be opened without triggering an alarm. M00 can be used if an inspection check is required during an operation.

### **M01 (Optional Program Stop)**

M01 also works like M00, but only if "Programmed Stop Yes" is switched on by soft key in the menu PROGRAM CONTROL.

### **M02 (End of Program)**

It works like M30. It halts program execution, spindle is turned off and tool moves to extreme right on the Z-axis.

### **M03 (Spindle Start, Clockwise Direction)**

It is used to switch on the spindle in clockwise direction. Note that spindle speed has to be programmed, chip protection door should be closed and workpiece properly damped for executing M03.

### **M04 (Spindle Start, Counterclockwise Direction)**

It is used to start the spindle motion in counterclockwise direction. Conditions described under M03 apply here also.

### **M05 (Spindle Stop)**

It is used to stop the spindle motion. In turning operation, M05 can be used to stop spindle for inspection of workpiece or change of cutting tool. It can be used at the end of program, however at the program end, the main spindle is also automatically switched off.



### **M00 (Tool Change)**

If the NC machine has tool magazine or tool drum with multiple tool capacity. M06 is used to change the tool. It is used with T word, which specifies the tool number.

### **M07, M08 and M09**

M07 is used to switch the coolant on in flood mode while M08 is used to switch coolant on in mist mode. M09 is used to switch coolant off.

### **M10, M11**

M10 is used to clamping the fixture while M11 is used to unclamp it.

### **M13, M14, M17**

M13 turns the coolant on and starts the spindle motion in clockwise direction. M14 also switches the coolant on but starts spindle motion in counterclockwise direction. M17 switches off both coolant and spindle.

### **M30**

With M30, all machine drives are switched off and control returns to start of program.

### **M37, M38, M39**

M37 sets mirroring about current X position of tool while M38 sets mirroring about Y position of tool. M39 disables mirroring.

### **M60**

In case NC/CNC machine makes use of multiple pallets for loading-unloading of workpiece from the machine, M60 can be used for changing pallets.

### **M98, M99**

M98 is used to call a subprogram or function. In this command, the subprogram number and the number of repetitions have to be specified. M99 is used to jump back to the start of the program or the specified block number.