Introduction of Computer Numerical Control (CNC):

The idea of numerical control started when the automation of machine tools originally incorporated specific concepts of programmable logic. In the beginning, the first NC machines were built back in the 1940s. Slightly more advanced machines came along in the 1950s. These manufacturing machines were constructed based on existing tools that were modified with motors designed to move the controls of the machine. These controls followed specific points that were fed into the machine on punched tape.
The numerical control technology moved into the 1960s and 1970s, a very familiar form of a CNC machine that most would recognize today started taking shape. Digital technology then entered the fray, and automation in production processes became more efficient than ever. In fact, many individuals can purchase – and even design – their own homemade CNC machines. Because of how advanced computers are nowadays, it's more common than ever to find CNC machines in all industries.

What is CNC?

The term "CNC" is a generic term which can be used to describe many types of device, this would include milling machines, Lathe machines, Drilling machines, plotters, CNC Routers, 3D printers, and others. CNC stands for Computer Numerically Controlled and basically means that the physical movements of the machine are controlled by instructions, such as co-ordinate positions that are generated using a computer.
It is typically used to refer to a device which uses a rotating cutting tool/Work peace which moves in 2/ 3 or more axes (X, Y and Z) to cut-out or carve parts in different types of materials. It is incredibly versatile and allow to cut a variety of different types of product and materials. The exact abilities of a machine will vary with size, rigidity and power.

Computer skills for CNC work- One requirement common to all aspects of CNC work is how to use a computer to perform basic tasks. That may need to use a combination of software to design and cut the parts. This will require an understanding of starting and stopping software programs, saving, copying, moving and deleting files, finding files stored on the computer and installing programs and updates.

Conventional Machines- For machining, Conventional machines are controlled by machinists, machinists can freely control the machine axis movements, can choose spindle speed of their liking, can change tool at their will (although all these steps are component material dependent, as for a harder material it has to choose a low-speed etc. but everything is controlled by the machinist.).

CNC Machines:

The CNC machining process are same as conventional machining process, But not everything is in the control of CNC machinist. On a CNC machine every step tool-change/spindle-speed/axis-movement is controlled through CNC machine control. A CNC machinist can only Start/Stop/Control-tool-cutting etc. Every component operation (tool-change/axis-movement/spindle-speed/gear-selection) is stored in the cnc-machine-memory as a list of *Instructions* this is called a Part-Program or just a Program.

CONTROLS PROGRAMMING LANGUAGE:

More commonly used languages are,

(i) APT (Automatically programmed tools): It is a universal part programme language which can specify any component and its machining operations to be carried out on NC machines having upto 5 controlled axes. APT is a powerful and complex language which requires the use of a large computer and is not convenient for simple applications. It uses the nomenclature of FORTRAN (Formulae translation) which in turn is a Universal Computer language developed by IBM.

(ii) ADAPT: was the first of several alternative languages developed from APT. It is simplified version of APT suitable for simple machines.

(iii) EXAPT: (Extended Subjects of APT) is an important language developed by Germany which is made up of 3-parts.
EXAPT 1 for point to point application.
EXAPT 2 for turning work, power and energy
EXAPT 3 for milling type works.

(iv) NELAPT: is a programming language based on APT and developed by the National Engineering Laboratory (NEL)

Examples are:

FEDRAT → Feed rate

GOFWD → Go forward

GOLFT → Go left

TANTO → Tangential to

How the CNC works ?

As we might already have guessed, everything that an operator would be required to do with conventional machine tools is programmable with CNC machines. Once the machine is setup and running, a CNC machine is quite simple to keep running. In fact CNC operators tend to get quite bored during lengthy production runs because there is so little to do. With some CNC machines, even the workpiece loading process has been automated. (We don’t mean to over-simplify here. CNC operators are commonly required to do other things related to the CNC operation like measuring workpieces and making adjustments to keep the CNC machine running good workpieces.)

The Advantages of NC / CNC Machine Too:l

The advantages of a NC/CNC machines are quite obvious. After the initial setup, the additional setup time is eliminated. Only one work-holding fixture is necessary because the transfer to other machines has been eliminated. Setup time is reduced for the same reason. Less time is needed for the change of feeds & speeds, tool selection and insertion, and indexing of the work piece. Machining time can also be predicted because the human element has largely been eliminated during the machining process. A more accurate part is produced because of the elimination of transfer error. Can be manufacturing the parts which are very complex in construction & having very close tolerance.

The Disadvantages of NC / CNC Machine Tool:

As with every system, the CNC systems too have certain disadvantages which are, (a) Higher Investment Cost- It has more sophisticated and complex technology and this technology cost more to buy its non CNC counterpart. So to be use these machines more aggressively than ordinary machines and high utilization is essential to get returns of investment. (b) Higher maintenance cost- maintenance problem becomes more acute and need CNC engineers with certain skill. (c) Costlier CNC Personnel- CNC machine operations require a higher skill level than conventional operations, so skilled personnel with particular field is required. (d) Planned support facility- The planned supports are essential i.e. Part programming, tape preparation, tool pre-setting, fundamental services etc.

The Application of NC / CNC Machine Tool:

The following areas of machining, where the CNC users can expect benefit and improvement by applying the NC/CNC machine tools.

01. To minimize the Lead time in manufacturing.
02. To manufacture such parts which are very complex in construction and it will not possible to manufacture them accurately on conventional machines due to human and machine error involved.
03. To give flexibilities to machine tools to adapt such parts of manufacture, which are frequently subjected to change the design.
04. Repetitive and precision quality parts which are to produce in batch quantity. The batch may be small or medium in quantity.
05. To cut down the investment cost on tooling and fixture inventory as required, when parts are made on a conventional machine.
06. If it is an expensive parts, where mistakes in processing would be costly.
07. The manufactured parts are require 100% inspection.
08. Excessive metal needs to be removed.
09. The manufactured parts having very close tolerance.
10. Large number of operations must be performed on the part in its manufacturing.

The environmental control for NC / CNC Machine Tool:

Protection of machine is very important for efficient working and long life of the machine. Various types of collapsible guards, wipers and telescopic covers are used to protect the machine guide ways, drive screws etc. As well as the following environmental control is necessary to protect the machine life.

01. Dust free floor space and environment.
02. Working temperature should be within control limits.
03. Well air circulation.
04. Space should not be congested, it should be quite open.
05. Electrical power should be regulated.
06. There should be proper disposal point for scrap removal.
07. There should not be presence of any vibrating and noisy source near to the machine.
08. There must be proper lighting and air ventilation system.
09. The machine should be operated by trained person.
10. There must be provision of sufficient supply of coolant & should protect from the moisture.

Importance for the employees to understand CNC programming:

In the past it did not seem necessary for managers, supervisors or engineers to understand the concepts of CNC programming. BUT NOW increasing competition, shrinking margins and high employee turnover demands that people communicate more effectively between disciplines. 7 Easy Steps to CNC Programming. A Beginner's Guide makes it possible for everyone, regardless of their primary job assignment, to understand CNC programming concepts.

- Engineers can more easily design for manufacture.
- Supervisors can be of greater support to operators. Managers/Supervisors can more reliably measure the quality of CNC programs.
- Programmers using Computer Aided Manufacturing (CAM) systems can more reliably desk check programs before sending them to the shop.
- Managers/Supervisors can feel more confident when discussing CNC related issues.
- Managers/Supervisors can garner more respect from CNC operators and programmers.
- Well trained operators increase efficiency and profitability.
- Well trained operators can more easily spot programming errors.
- Well trained operators are less likely to scrap parts.
- Well trained operators are less likely to have crashes and suffer personal injury.

Part Program for CNC Machine Tool:

Part Programme is an important component of the CNC system. It is the procedure by which the sequence of processing steps and other related data is planned and documented on the CNC machine. It is a set of instructions which instruct the machine tool about the processing steps to be performed for the manufacture of a component. The shape of the manufactured components will depend on how correctly the program has been prepared.

Description of the Part Program:

Every Part-program consists of multiple instruction (which tell cnc machine what to do).

In the Part-program, we will see the multiple lines of instruction.

Every line of part-program is called a **cnc program block**.

Every cnc program block might consist of multiple instruction (called G-Codes/M-Codes, Co-ordinate positions, etc)

So these are the instructions which tell cnc machine what to do next, when to change tool, which axis to move, how much spindle speed is required with which gear.

These programs are made/written by **CNC-Programmers**, in the past most of program were written by hand (even today), but nowadays multiple software are available which ease this whole process.

So when the Part-program is complete those are brought to cnc machine, there are multiple mediums and ways to transfer the program to a cnc machine, floppy disks/flash card/networks are some of many ways used these days in workshops.

Now part program is stored inside the cnc machine memory called **CNC Programs Directory**. So now cnc machinist *select* the required part program for execution.

Then that part program is run by cnc machinist by pressing **Cycle-Start**, and now the control is transferred to the **cnc machine control** which now reads the part-program and instructs different parts of cnc machine to do the required operation. This way whole the part-program instructions are executed block by block.

If during this whole process anything goes wrong then cnc machinist can stop the cutting tool motion (Feed-Hold), or he can completely stop the machine (Cycle-Stop). Even cnc machinist can press Emergency-Stop-Button if anything dangerous happen or going to happen.

If everything goes well during part-program execution then machine automatically stops at the end of part program.

Main Program: CNC main program is a series of instructions, assigned to different tools, tool offsets and operations. Within it the various set up conditions for the machining task to be given. It is the original program, which will select for performing automatic operation. In main program may include two or more repetitive instructions by sub-program execution command. During main program execution, sub-program are executed, when execution of the sub-program is finished, the sequence returns to the main program.

Sub Routing Program or Sub-Program: If some repetitive features such as grooves, holes, contour profiles etc. are presenting a component and to make the part programs are proportionally longer. Simpler ways of achieving such repetitions are provided by the facility of repetitive programming techniques are called Sub Program or a branch or an extension of the main program. In fact it is a independent program with all the features of a usual part program are stored in the memory under separate program number and may nested with another sub-program, whenever a particular feature is required within the program, the associated subroutine is called for execution with repeated any number of times with in the main program. Use of sub programming techniques significantly reduces the length of resulting part program, shortens the time and reduces the computer memory space.

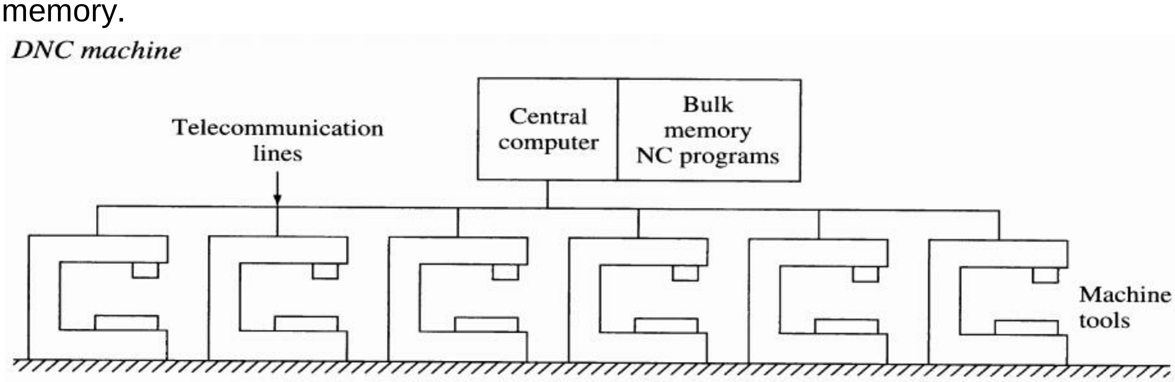
Manual Programming: Manual programming or On machine programming (with out a computer) has been the most common method of preparing a part program. In manual programming all mathematical calculations are done by hand and all instructions are entered manually by control key board entry directly at the machine. It requires the total involvement of the CNC programmer and offers virtually unlimited freedom in the development of the program structure. The latest CNC controls make manual programming much easier than before by using fixed or repetitive machining cycles, variable type programming, graphic tool motion simulation, standard mathematical input and other time saving features. There are some disadvantages associated with manual programming. The most common is the length of time required to develop a full functioning program. The manual calculations, verifications and other related activities are very time consuming. The complex jobs are very difficult to calculate all the data and may come a large percentage of errors and a lack of tool path verification.

Computerized Programming:

Difficult or complex jobs will benefit from a computerized programming system. Technologies such as Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) have been a strong part of this process. Program can generate with in a shortest time by using an inexpensive desktop or a laptop computer. The benefits offered by this technology are too significant just design, drafting, Tool path generation, Tool path verification and programming. Programmed data can be transferred to the CNC machine via a cable or other data transferable method.

Direct numerical control (DNC) / Distributed numerical control:

DNC is a common manufacturing term for networking CNC machine tools. On some CNC machine controllers, the available memory is too small to contain the machining program (for example machining complex surfaces), so in this case the program is stored in a separate computer and sent directly to the machine. If the computer is connected to a number of machines it can distribute programs to different machines as required. Usually, the manufacturer of the control provides suitable DNC software.
Means DNC is-
(1) "A system connecting a set of numerically controlled machines to a common memory for part program or machine program storage with provision for on-demand distribution of data to machines."
(2) Several NC machines are directly controlled by a computer, eliminating substantial hardware from the individual controller of each machine tool. The part-program is downloaded to the machines directly (thus omitting the tape reader) from the computer memory.



CNC Program Blocks:
Every cnc program is a sequence of many cnc program blocks which are written together to form of a complete tool-path for one or many tools.
This tool-path tells cnc machines how a cnc machinist wants his component to be machined.
Every single cnc program block adds / alters / modifies some useful information to a cnc program.

<u>Definition of a Sequence Block or Block Format</u>:

The basic unit of a part program input to the control is called Block. Each block is a individual line contains adequate information for the machine to perform the functions or a movement. Block on turn are made up of words and each words consists of a number of characters. Just like the word is used a single instruction and the block is a multiple instruction to the CNC system and it is terminated by the End of Block character.

It may consists are….

01. Optional block skip (/)
02. Sequence Number (N)
03. Preparatory Functions (G)
04. Dimensional Information's (X ,Y , Z, u, v, w, A, B, C, R, I, J, K etc.)
05. Decimal Point (.)
06. Feed rate (F)
07. Spindle Speed (S)
08. Tool Number (T)
09. Tool offset function (D)
10. Miscellaneous function (M)
11. End of Block (EOB)

01. **Sequence Number (N - word)**→ The first word in every block is the sequence number, it is used to identify the block, written as N1, N2, N3 etc.

02. **Preparatory Function (G - word)**→ The preparatory word prepare the control unit to execute the instructions, that enables the controller to interpret the data which follows and precedes the coordinate words. i.e. G01 is used to prepare the controller for linear interpolation.

03. **Coordinates or Dimensional Information's (X-,Y-,Z-, etc. words**)→ These words give final coordinate positions of the motion with Preparatory command. Like G00/G01 X50 Y45 Z200 . G02/G03 X60 Y55 R10.

04. **Decimal Point (.)**→ In some systems, decimal point is not coded but the control system automatically provides a decimal point at a pre-set position. But in such cases leading zeros may have to be given to the program, e.g. X= 7.09 will be given as X= 7090

05. **Feed Function (F - word)**→ The feed function is used to specify the feed rate in the machining operation. The feed rate is expressed either in millimeter or inch. per minute (mm or inch./min) and millimeter or inch. per revolution (mm or inch./rev). The appropriate preparatory command should be specified.

06. **Spindle Speed Function (S - word)**→ The speed function is used to specify the speed rate of main spindle in the machining operation. The spindle speed is specified either in revolution per minute (rev/min) and meters or feet per minute (m or ft./min). The appropriate preparatory command should be specified.

07. **Tool Selection Function (T - word)**→ The T- word in the part program specifies which tool is to be used in the operation and the tool offsets. It is also needed for machines with programmable tool turret or automatic tool changer (ATC), because there is tool pocket with distinct tool number.i.e.T01, T02, T23 etc.

08. **Tool offset function (D - word)**→ The D- word is denote for tool diameter offset number. It is counted for cutter radius compensation.

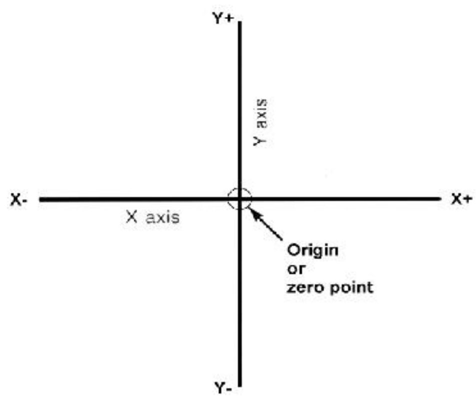
09. **Miscellaneous function (M - word)**→ The miscellaneous function word is used to specify certain machine functions, which do not relate to the dimensional movements of the machine. It may be Spindle ON/OFF, Coolant ON/OFF, etc.

10. **End of Block (EOB)**→ The EOB symbol identifies the end of instruction block or line.

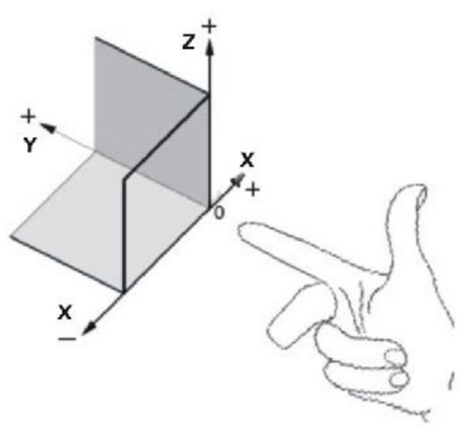
Axis Identification in NC/CNC machines:

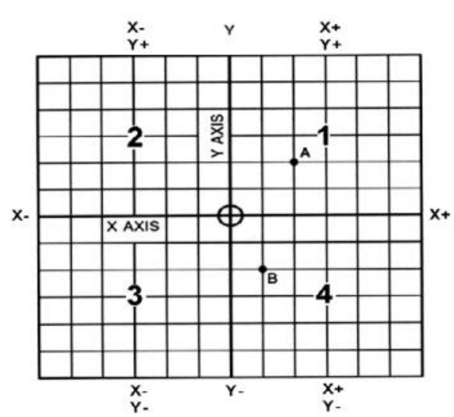
Most of the machine have two or more slide ways, disposed at right angles to each other, along which the slides are displaced and each slide is fitted with a control system. For the purpose of giving commands to the control system the axis have to be identified. The basis of axis identification is the 3-dimensional Cartesian coordinate system and the three axis of movement are identified as X, Y and Z axis. Rotary movements about X, Y and Z axis are designated as A , B and C respectively.

X - axis: The X-axis of motion is always horizontal and is always parallel to the work holding surface. In turning center diameter axis or cross slide motion is X- axis.



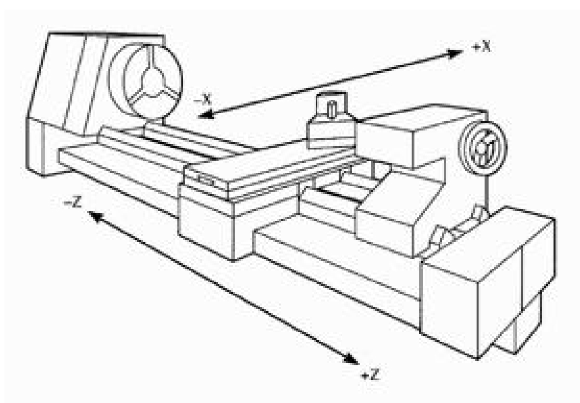
Y - axis: The Y-axis of motion is always at the right angles to both the X-axis and Z- axis. On vertical machining centers, the Y-axis is perpendicular to the column and on horizontal machining centre, the Y-axis is perpendicular to the earth.

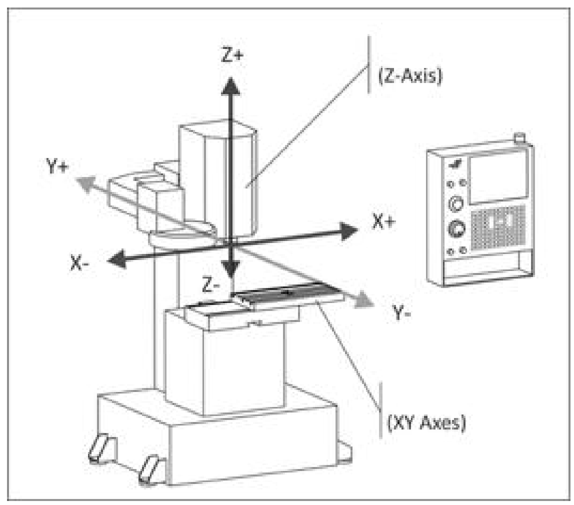




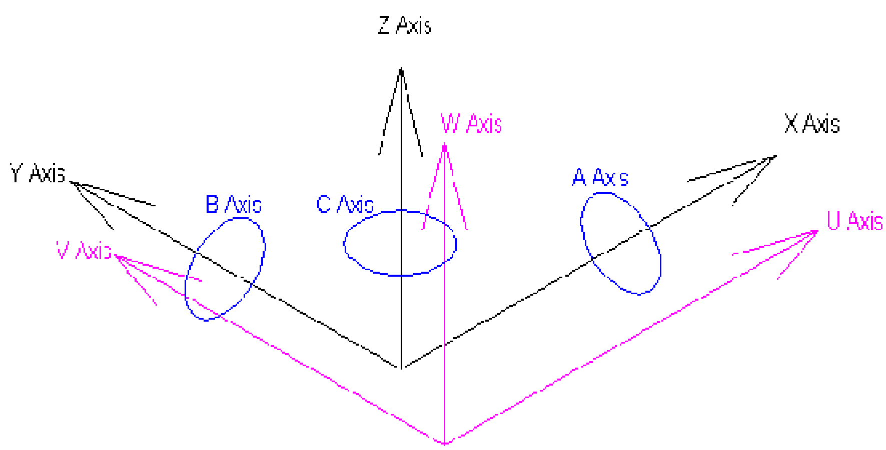
<u>Z - axis</u>: The Z-axis of motion is always the axis of the main spindle of the machine. On vertical machining centers, the Z-axis is vertical and on horizontal machining centre and turning centers, the Z-axis is horizontal. Positive Z movement (+ Z) is in the direction that increases the perpendicular distance between the work piece and the cutting tool.

<u>Important Note</u>→ For axis motion of the machine tool, may moves the work piece or may moves the cutting tool, it is depends upon the machine tool feature. The programmer may confuse with the direction of motion. So in all cases the part program axis motion should be written assuming that the cutting tool moves. Also it is the cutter path which is defined in the program, In case of milling program the axis motion and cutter position should be programmed with respect to centre of the cutter.

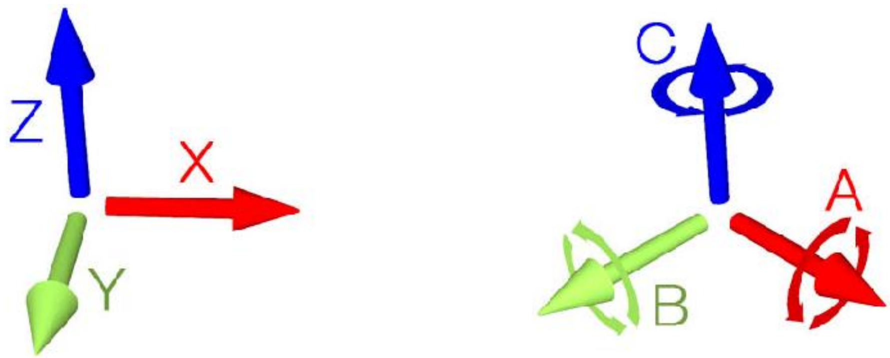




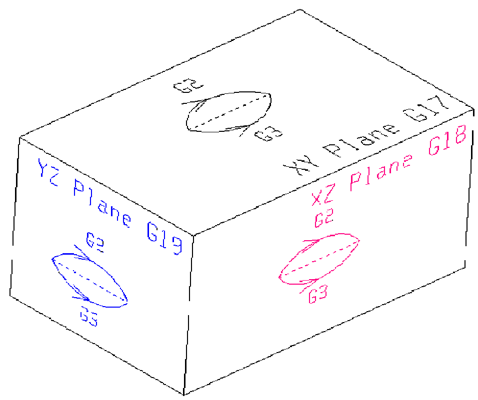
CNC Lathe Machine Axis XZ CNC Milling Machine Axis XYZ



<u>Rotary axis</u>: The rotary motion about the X, Y and Z axis are identified by A, B, C respectively. Rotary motion with parallel to X axis is 'A' axis, parallel to Y axis is 'B' axis and parallel to Z axis is 'C' axis.



<u>Planes in NC/CNC machine</u>: A plane is a surface in which a straight line joining any two of its points will completely lie on the surface. Machining in planes, means the path of cutting tool is a combination of straight lines and arcs and the tool motion in one or two axes always takes place in a plane, designated by two axes i.e. XY, YZ and XZ.



Machine Zero → The machine Zero point is at the origin of the coordinate measuring system of the machine. The machine zero point is fixed and can not be sifted. The machine zero point is also called Home Position of the machine.

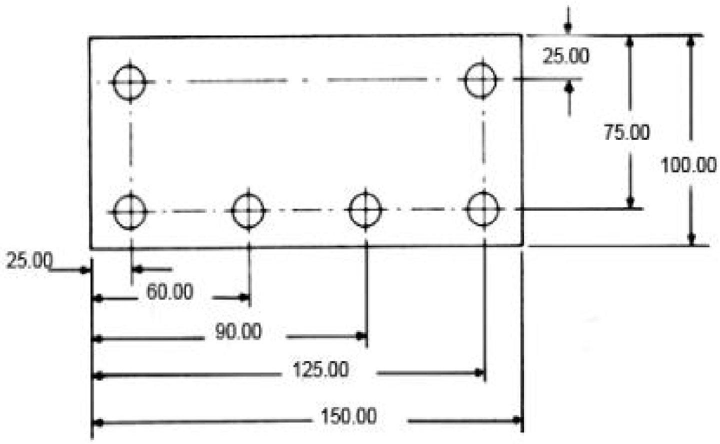
Work Zero → Work piece zero or datum may be defined as a point, line or surface on the component drawing to which all the dimensions referenced. For writing the part program, the programmer should know the relationship between the work piece zero coordinate and machine zero coordinate.

<u>Coordinate system or Dimensioning system</u>:

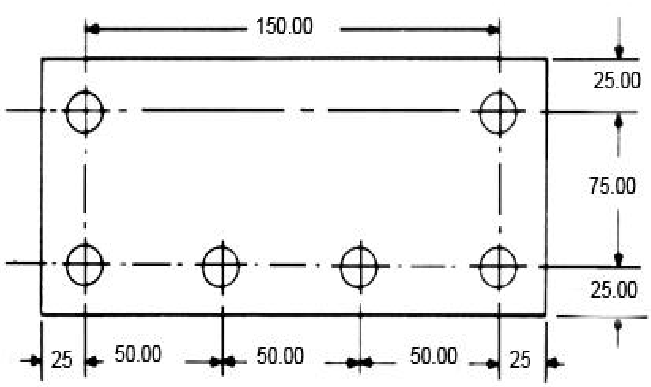
Address in a CNC program that relate to the tool position at a given moment are called the Coordinate Word. It is always take a dimensional value. Dimensional values have a specified point of reference. There are two type of coordinate system or dimensioning system are used to define and control the position of the tool in relation to the work piece. Each system has its own applications and the both system may be used independently or may mixed with in a part program according to the machining requirements of the components. (i) Absolute coordinate system and (ii) Incremental coordinate system.

<u>Absolute Coordinate system</u> → In the absolute programming mode, all dimensions are measured from the point of origin. The origin is the program reference point also known as program zero or datum. The actual motion of the machine is the difference between the current absolute position of the tool and the previous absolute position. The algebraic sign (+) plus or (-) minus refer to the quadrant of rectangular coordinates, not the direction of motion.

The main advantage of absolute programming is the ease of modification by the programmer or by the CNC operator. A change of one dimension does not affect any other dimensions in the program.



Incremental Coordinate system → In the incremental programming mode, also called a relative mode, all program dimensions are measured as departure distances into a specified direction (equivalent to the 'distance-to-go' on the control system). Means the coordinate position of a point are measured with reference to the previous point i.e. the point at which the cutting tool is positioned and is taken as zero or datum point for calculating coordinates of the next point to which movement is to be made. The actual motion of the machine is current position of the tool to the next position. The algebraic sign (+) plus or (-) minus is specify the direction of the tool motion not refer to the quadrant of rectangular coordinates. The main advantage of incremental programming is their portability between individual sections of a program. It is mostly used when developing subprograms or repeating an equal distance. It is not ease of modification by the programmer or by the CNC operator. A change of one dimension will effect on other dimensions in the program.



Preparatory Codes or Geom. setting & movement Commands:

The program address 'G' identifies a preparatory command or Geometrical setting and movement command, often called the G code. This address has one and only objective-that is to preset or to prepare the control system to a certain desired condition, or to a certain mode or a state of operation.

G-code is a language in which people tell computerized machine Tools what to make and how to make it. The "what" and "how" are mostly defined by instructions on where to move to, how fast to move, and through what path to move. The most common situation is that a cutting tool is moved according to these instructions, cutting away excess material to leave only the finished workpiece. Non-cutting tools, such as cold-forming tools, burnishing tools, or measuring probes, are also sometimes involved.The letter **"G"** Generally it is a code telling the machine tool what type of action to perform, such as:

1. Rapid move (transport the tool through space to the place where it is needed for cutting; do this as quickly as possible)
2. Controlled feed move in a straight line or arc line.
3. Series of controlled feed moves that would result in a hole being bored, a workpiece cut (routed) to a specific dimension, or a profile (contour) shape added to the edge of a workpiece.
4. Controlled Speed.
5. Set tool information such as work offset, Tool Dia. and Length offset.
6. Switch coordinate systems
7. Input system mm/inch.
8. Type of Compensation
9. Information of Work offset, etc.

G- Function

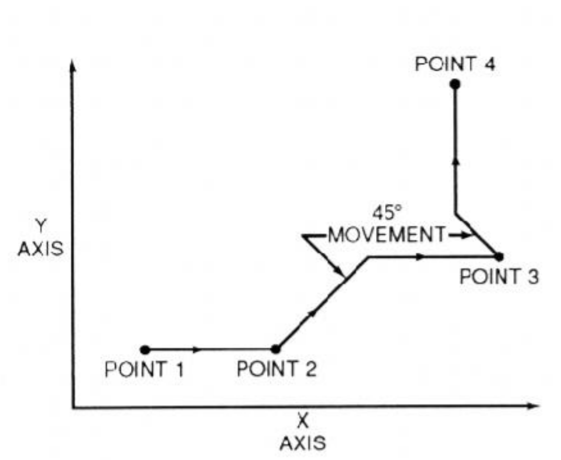
G-codes	**Description**	**Group**
G00	Rapid Traverse with m/c feed	01
G01	Linear Interpolation with given feed	
G02	Circular Interpolation Clock Wise	
G03	Circular Interpolation Counter Clock Wise	
G04	Dwell (waiting time)	00
G17	Plane Selection XY	02
G18	Plane Selection XZ	
G19	Plane Selection YZ	

G20	Input in Inch	06
G21	Input in Millimeter	
G28	Machine Zero return/Return to reference point	00
G40	Tool Nose Radius Compensation Cancel	07
G41	Tool Nose Radius Compensation in Left	
G42	Tool Nose Radius Compensation in Right	
G43	Tool Length Compensation Positive	08
G44	Tool Length Compensation Negative	
G49	Tool Length Compensation Cancel	
G50	Coordinate system setting/max. Speed Setting	00
G54	Work Coordinate system Selection → P1	12
G55	Work Coordinate system Selection → P2	
G56	Work Coordinate system Selection → P3	
G57	Work Coordinate system Selection → P4	
G58	Work Coordinate system Selection → P5	
G59	Work Coordinate system Selection → P6	
G80	Canned cycle cancel	
G82	Spot Drilling cycle	
G83	Peck Drilling cycle	
G84	Tapping Cycle	
G85	Boring Cycle	
G90	Absolute Dimensioning	03
G91	Incremental Dimensioning	
G94	Feed Rate mm. or inch. per minute setting	05
G95	Feed Rate mm. or inch. per revolution setting	
G96	Constant Cutting Speed Setting	17
G97	Constant RPM Setting/ CCS Cancel	

Modal Code → Many program words are modal. The word modal is based on the word 'mode' and means that the specific command remains in this mode after it has been used in the program once. It can only be canceled by another modal command of the same group. In same group may tell one is cancel command for other. For example:- G90- Absolute Dimensioning/ G91- Incremental Dimensioning, G00- Rapid Traverse/ G01- Linear Interpolation, G20- Input in Inch/ G21- Input in Metric, G94- Feed per min/ G95- Feed per revolution, G96- Constant cutting speed/ G97- Constant RPM, etc.

Non-Modal Code → Whose functionality starts and ends in the same block. Means the 'G' code is effective only in the block in which it is specified, is called Non-Modal Code or One Shot Code. For example:- G04- Dwell, G28- Machine zero return, Certain machining instructions, such as tool change, pallet change, indexing table, etc.

Interpolation Functions G00 (Positioning) → The G00 command moves a tool to the position in the work piece system specified with an absolute or incremental command at a rapid traverse rate. In the absolute command coordinate value of the end point and in the incremental command the distance the tool moves is to be programmed. i.e. G00 XP YP ZP ; It moves each axis at its max speed until its vector is achieved. Shorter vector usually finishes first. the programmer needs to consider depending on what obstacles are nearby, to avoid a crash. G00 XP YP ; (Tool moves to XY position with Rapid motion or with machine feed rate.)

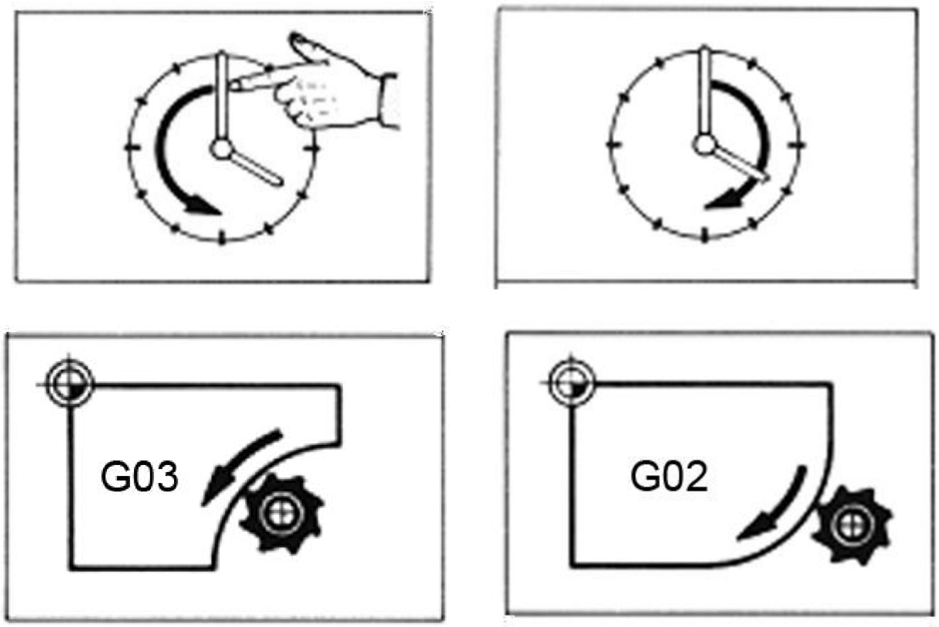


G01 (Linear interpolation) → The tool moves along a straight line to the position in the work piece system, specified with an absolute or incremental command within the shortest possible time at the feed rate specified in F. In the absolute command coordinate value of the end point and in the incremental command the distance the tool moves is to be programmed. The feed rate specified in F is effective until a new value is specified. It need not be specified for each block. If the F code is not commanded, the feed rate is regarded as zero. i.e. G01 XP YP ZP F- ; The G01 command can not functioned without specified Feed. G01 XP YP F_ ;(Tool moves along a line to the XY position with the feed specified in F).

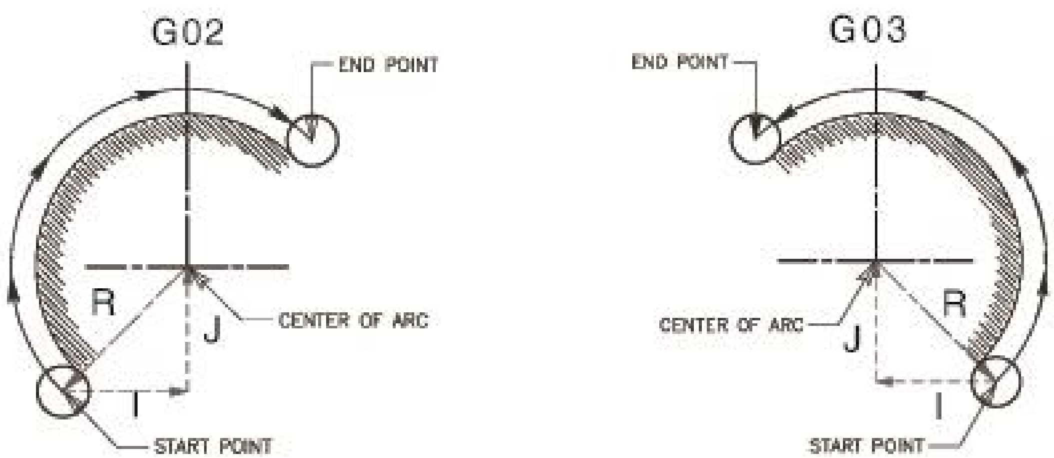
G02/ G03 (Circular interpolation Positioning CW/ CCW) →

The tool moves along an arc line to the position in the work piece system, specified with an absolute or incremental command in the positive or negative direction with the specified arc center by addresses I, J and K or radius R at the feed rate specified in F.

In the absolute command coordinate value of the end point and in the incremental command the distance the tool moves is to be programmed.



<u>I, J and K (Arc Vector Address word)</u> → Some older control systems do not accept the direct radius designation specified by the R address. Instead, the arc vectors I, J and K must be used. The arc vector is specified by addresses I, J and K for the XP, YP and ZP axes respectively. The numerical value following I, J and K, however, is a vector component in which the arc center is seen from the start point and it is always specified as an incremental value. The addresses I, J and K must be signed positive (+) or negative (-) according to the direction.



Note:- I, J and K is the distance between start point and the center point of the arc, i.e. Distance between XP to XP^1 is indicating by 'I', Distance between YP to YP^1 is indicating by 'J', and Distance between ZP to ZP^1 is indicating by 'K'.
'I', 'J', 'K' is always specified as an incremental value.
If the arc is covered the path 180^o or less than 180^o can be specified by using radius (R) of the circle instead of I, J or K. The I, J and K are considered for the arc path more than 180^o.

Dwell Function (G04): The G04 command, by specifying a dwell, the execution of the next block is delayed by the specified time. Which is denoted by letter X or P or F. Delay time 10 seconds denoted like G04 X10.0 or P10000 or F10.0. This function is used in case it is desired that the cutting tool should not immediately return after touching the programmed position, but should wait at the programmed position for some period. For example; In case of machining a Groove, Blind hole drill etc. the tool should delay at the required depth, to ensure the complete removal of the material.

G04 x….(sec), or G04 U…(sec) or G04 P…(micro sec).

Feed Functions (G94 / G95) → While the spindle function control the spindle speed and the spindle rotation direction, Feed rate controls how fast the tool will move to remove excessive material. Usually two types of feed rate are used in CNC programming i. e. Feed rate per minute (G94) and Feed rate per revolution (G95). The value of feed rate is the distance a cutting tool will travel in one minute or in one revolution. Example:- In metric system, the feed rate amount of 200 mm/min will appear in the program as G94 F200.0 and the feed rate amount of 0.2 mm/rev. will appear G95 F0.2.

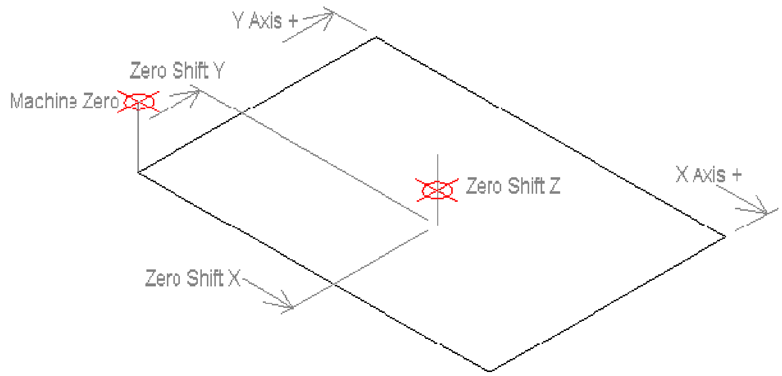
Spindle Speed Functions (G96 / G97) →The spindle function control is the spindle speed and the spindle rotation direction. The spindle speed G96 (Surface Speed) & G97 Revolution per minute)—Programmed spindle speed should be based on the machined material and the cutting tool diameter or the part diameter (Lathes).
In CNC milling, where is cutter diameter is fixed and designate the spindle speed directly in revolution per minute (RPM) no peripheral speed is used. There is no need to use a preparatory command to indicate the RPM setting, it is the control default.
In few CNC Lathe may be equipped with the option of a duel spindle speed selection-direct RPM G97 and a peripheral speed G96. The general rule is that the larger diameter, the slower the spindle speed and incorrect spindle speed will have a negative effect on both the tool and the part. The turning tool has no diameter and the part diameter is used for spindle speed calculation. As the part is being machined, the diameter changes continuously during turning or facing operation and very difficult to calculate the RPM on varying diameter. The solution is to use the surface speed directly with the command G96 and in this mode the actual spindle revolution will increase &

decrease automatically, depending on the diameter being cut. This feature not only saves the programming time, it also allows the tool to remove constant amount of material at all time, thus saving the cutting tool from excessive wear and creating better surface finish. The operations as drilling, reaming, tapping, etc. are common for centre line operation and programmed at the zero diameter (X0). These operations are always programmed in the direct RPM mode, Using G97 command, spindle speed is controlled directly, it does not change.
Example :- G97 S1200 (Constant Spindle Speed 1200 RPM)

Maximum Spindle Speed Setting (G50): When the CNC lathe operates in the Constant Surface Speed mode, the spindle speed is directly related to the current part diameter. If the work diameter is smaller, the spindle speed will be greater and it will be gradually increasing when machining. So the natural question is- what will be happen if tool point is reaches the spindle center line or the tool diameter is zero. The result will be normally the highest spindle rotation available in the machine. This situation is not acceptable; the part may extend from the chuck or fixture. There is simple solution to this problem, using a programming feature maximum speed limit setting (G50) command to preset the highest limit of RPM.
G96 S150 (Constant Cutting Speed setting 150 m/min.)
G50 S1200 (Highest Speed Limit Set 1200 RPM)
Note:- Fraction or decimal values are not allowed and the RPM must always be within the range of any machine specifications.

Planes Selection in NC/CNC machine: The Preparatory commands for plane selection G17 (XY plane), G18 (YZ plane) and G19 (XZ plane) - programming one of them will activate the selected plane only. The proper selection of a machining plane will enable programming various contouring operations using circular and helical interpolation, cutter radius compensation and fixed cycles.
Assigning program zero (G50/G92) :
Keep in mind that the CNC control must be told the location of the program zero point by one means or another. How this is done varies dramatically from one CNC machine and control to another. One (older) method is to assign program zero in the program. With this method, the programmer tells the control how far it is from the program zero point to the starting position of the machine. This is commonly done with a G92 (or G50) command at least at the beginning of the program and possibly at the beginning of each tool.
Another, newer and better way to assign program zero is through some form of offset. Commonly machining center control manufacturers call offsets used to assign program zero fixture offsets. Turning center manufacturers commonly call offsets used to assign program zero for each tool geometry offsets.

Settable Zero off-set (G54 - G59): The zero off-sets are the distance X0, Y0 and Z0 from machine zero to work zero in X, Y and Z directions. The settable zero off-set specifies the position of the work piece zero on the machine i.e. off-set of work piece zero with reference to the machine zero. These off-set values are determined after clamping the work piece on the machine and then the values entered by choice in the off-set table numbered G54 to G59. The value is enabled by the program with work off-set command (G54 – G59).



For measuring zero off-set: (1) locate the edge finder in spindle head. (2) Manually move the table with Jog or Handle feed. (3) Position the spindle and load the edge finder on the surface X or Y direction of the work piece. (4) Release the load on edge finder until the flanges match. (5) Note the X or Y direction reading on the screen which is sowing the distance from the machine zero. (6) The radius of the edge finder should be added / subtracted and must be identified correctly. (7) Noted perfectly and entered into the work off-set table. To measure the Z direction values, mount the tool into the spindle and move down the tool till its end touches surface of Programme zero (may place a feeler or height master, the length of which known) and note the Z reading on the screen with considering feeler height.

<< WORK PROBE		WORK ZERO OFFSET			WORK PROBE >>
G CODE	X AXIS	Y AXIS	Z AXIS		
G52	0.	0.	0.		
G54	-12.5680	-8.4890	-23.1480		
G55	0.	0.	0.		
G56	0.	0.	0.		
G57	0.	0.	0.		
G58	0.	0.	0.		
G59	0.	0.	0.		
G154 P1	0.	0.	0.		
G154 P2	0.	0.	0.		
G154 P3	0.	0.	0.		

ENTER A VALUE

G54 X-806.250
Y-147.100
Z-530.570

G55 X-556.250
Y-197.100
Z-530.570

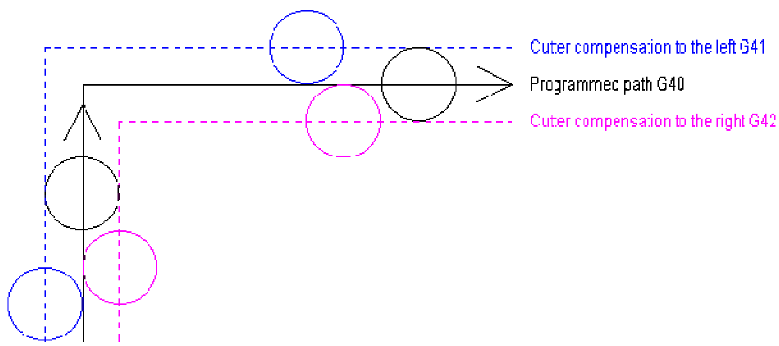
G56 X-306.250
Y-247.100
Z-530.570

Extended Offsets

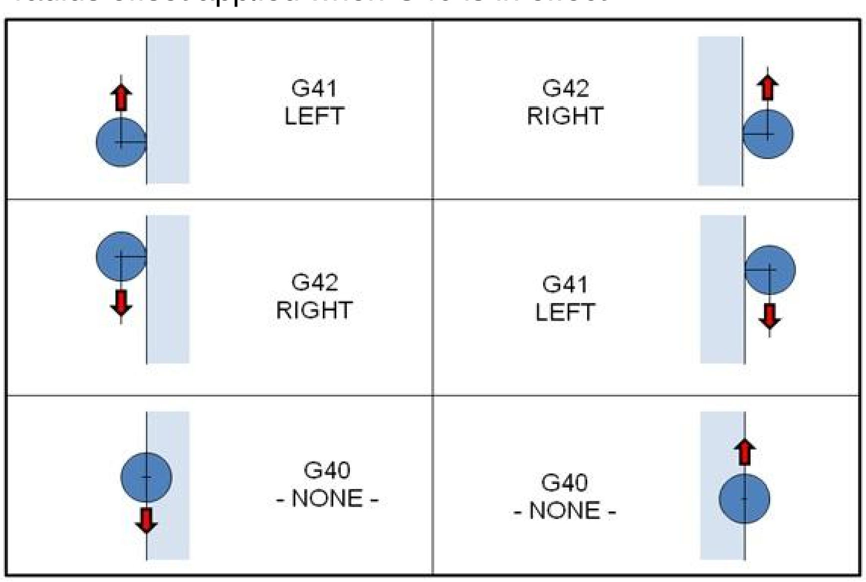
G54.1 P1, G54.1 P2, G54.1 P3, G54.1 P4 etc.

Tool length offset:

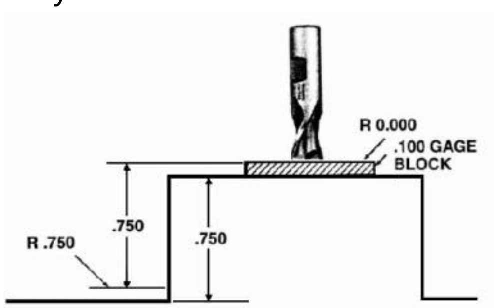
Set the tool length offset for each tool by loading first tool in spindle. Manually move the Z axis down until the tool's tip is near the Z0 position, the programmer had previously established. Get a piece of 0.01 mm shim stock and hold it between the part and the tip of the tool. Carefully and lower the Z axis in 0.001 mm increments until the shim stock can be pulled with a slight drag. Go to your tool length offset page and enter the machine's absolute Z value plus -0.01mm in the tools registry. Repeat procedure to additional tools and -0.01 is added for the shim stock's thickness.

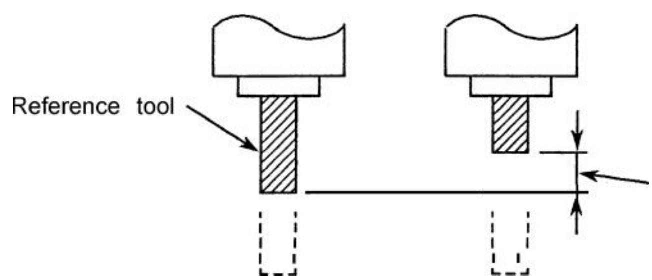


Tool Nose Radius Compensation Functions (G41/ G42): The programmed cutter path is developed with reference to the center of the tool rather than the point on periphery, where the actual cutting takes place. It will result in a work piece is larger or smaller. The diameter of the cutter is entered into the control system; the control system will then generate a new cutter path considering direction of motion around the part contour with cutter radius compensation. It is necessary to indicate whether compensation to be left (G41) or right (G42) of the tool depending on direction of its movement when machining. G41 and G42 is cancelled by the G40 command. There is no cutter radius offset applied when G40 is in effect.



Tool length compensation Functions (G43 / G44): Tool length compensation uses three G codes -- G43, G44, and G49 -- plus the H codes. The H code tells the control which length offset value to use, when length compensation is active (as selected by G43 or G44). Generally, the H code is the same as the tool number. I.e. if you are using tool 4, you would include H4 in the G codes, telling the control to look up the tool length on line 4 of the offset library. Example- T04 M06, G43 / G44 H04. G43 tells the control to begin applying length compensation, by adding the current length offset (selected by the H code) to all Z axis positions. G44 is a rarely-used alternative to G43. It tells the control to begin applying tool length compensation, by subtracting the current length offset from all Z axis positions. In this scheme, larger length offset numbers identify shorter tools (as if they were measured from the table up rather than from the spindle down). G44 is not compatible with the tool measuring methods built into Control's offset library.





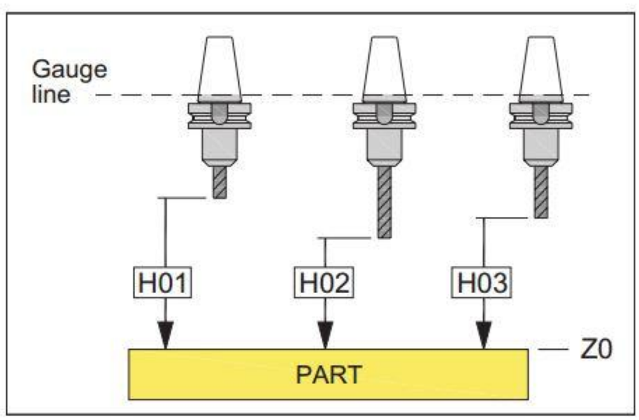


Figure 19-5

Touch-off method of the tool length offset setting

G49 tells the control to stop applying length compensation.

G49 H0

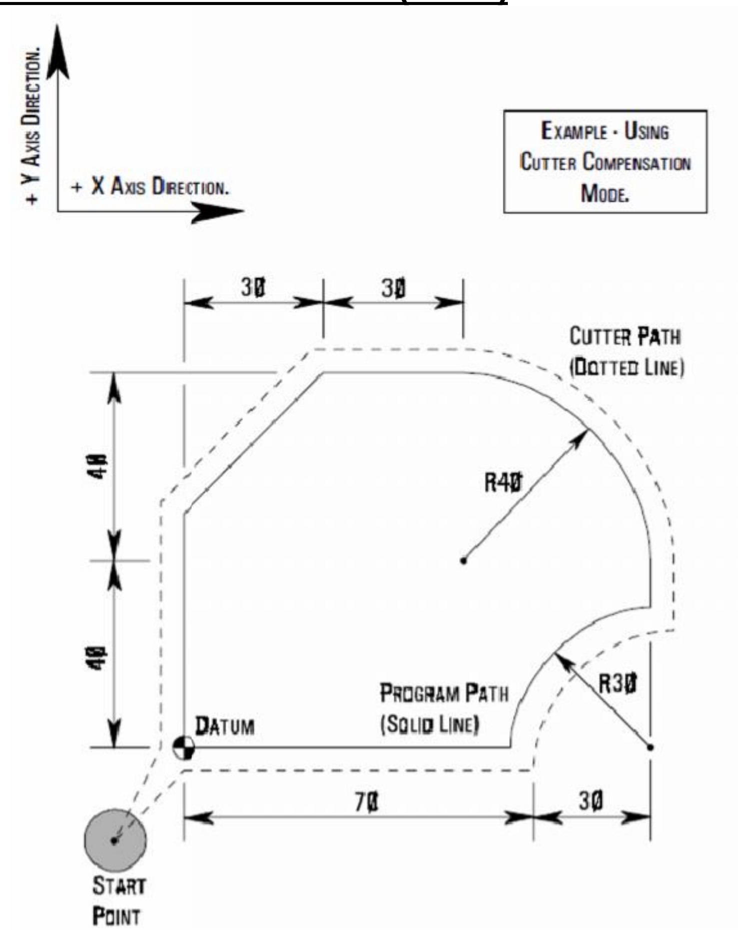
H0 is a special H code. It specifies a length offset value of zero, meaning that no offset will be applied even if G43 / G44 is active.

```
N1 M6 T4     ; load tool 4 (Z at home)
N3 G43 H4   ; turn on length compensation for tool 4
N2 G0 Z2.0   ; rapid down to clearance level
```

at the end of a cut :

N48 G1 X2.0 Z-0.5 ; make final cut at depth
N49 G49 H0 ; turn off tool length compensation
N50 G00 Z200 ; move Z to 200 position.

TOOL NOSE RADIOUS COMPENSATION (TNRC) :



Advantages of Cutter Compensation (TNRC):

1. The mathematical computations for determining a tool path are greatly simplified.
2. Because the geometry and not the tool center are programmed, the same program can be used for a variety of different cutter diameters.
3. When using cutter compensation you are then able to control and adjust for part dimensions using your cutter diameter/radius offsets register.
4. The same program path can be used for the roughing passes as well as finishing cuts by using different cutter offset numbers.

Disadvantages with Cutter Compensation (TNRC):

1. A cutter compensation command (G41, G42 or G40) must be on the same block with an X and/or Y linear command when moving onto or off of the part using cutter comp.
2. You cannot turn on or off cutter compensation with a Z axis move.
3. You can use cutter comp. in the G18 (X, Z) or G19 (Y, Z) planes using G141.
4. You cannot turn ON or OFF cutter compensation in a G02 or G03 circular move, it must be in a linear G00 or G01 straight line move.

When activating TNRC, Care must be taken to:

1. Select a clearance point, without cutter compensation, to a start point in the X and Y axis at least half the cutter diameter off the part before you start initiating cutter compensation.
2. Bring the Z axis down without cutter compensation in effect.
3. Make an X and/or Y axis move with a G41 or G42 call-out on the same line, with a diameter offset Dnn command, which has the cutter diameter value in the offset display register being used.

When deactivating TNRC, Care must be taken:

1. Select a clearance point in X and Y axis, at least half the cutter dia. of the part.
2. DO NOT cancel cutter compensation on any line that is still cutting the part.
3. Cancel of cutter compensation (G40) may be a one or two axis move, but you may need values entered for both X and Y axis.

M CODES – (Misc./ Machine Function Commands)

The program address 'M' identifies a miscellaneous function, sometimes called machine function. Not all miscellaneous functions are related to the operation of a CNC machine- quite a few are related to the processing of the program itself. So the use of miscellaneous functions falls into main groups, based on a particular application;

(**i**) Control of the machine functions—Spindle rotation, Automatic tool change , gear change , pallet change, Coolant operation, Tail stock motion, quill motion etc.

(**ii**) Control of the program execution—Sub-program calls, the change of a job setup, Optional stop, Program end etc.

M00	Optional Program Stop (Non-Conditional)
M01	Optional Program Stop (Conditional)
M02	Program End and all machine functions Stop
M03	Main Spindle ON Forward (Clock Wise)
M04	Main Spindle ON Reverse (Counter Clock Wise)

M05	Main Spindle OFF
M06	Tool Change
M08	Coolant ON
M09	Coolant OFF
M30	Program End with Rewind and all machine functions Stop
M98	Sub-Program call in Main Program
M99	Sub-Program End and return to main program

Note:

M00 : For this command, main spindle stop, cutting oil, motor stop, tape reading stop are carried out.

M01 : While this function is the same as M00, it is effective when The optional stop switch of console is ON. This command shall be over ridded if the optional stop switch is OFF.

M02: Indicates the end of main program and main spindle stop, cutting oil motor stop, tape reading stop.

M07 : is used to turn on mist coolant.(If available)

M08: is used to turn on flood coolant.

M09: turns off the coolant.

M30 : This is the same as M02 and it returns to the starting position of the program when the memory and the tape are running.

M60 : is commonly used to make pallet changes.

**It is favorable for M code to program in a command paragraph independently.

<u>Difference between Optional Program Stop M00 and M01</u>

The main rule of using M00 (Program Stop) is the need of a manual intervention for every part machined. Manual tool change in a program qualifies for M00, because every part needs it. But a dimensional check may not qualify, it is infrequent and M01 (Optional Program Stop) is better choice. Because M01 is functioned via the control panels optional function switch or ON/ OFF button and the setting of the switch will determine whether the program will temporary stop or continues to be proceed. Although the difference between the two functions is slight, the actual difference in cycle time can be significant for large number of parts. To avoid confusion, always inform to the operator why the M00/ M01 function has been used and what the purpose of it is. May like this way-- N39 M00 (REMOVE CHIPS), N53 M01 (PART CHEACKING) etc.

<u>Program End with M30</u> → If the end of program is terminated by the M30 function, the rewind will be performed, means if the last block in the program contains M30 as preferred END and sequence block is allowed to start the block. If the M02 function is used, the rewind will not be performed.

The Milling Process

Milling is the most versatile of machining processes. Metal removal is accomplished through the relative motions of a rotating, multi-edge cutter and multi-axis movement of the workpiece. Milling is a form of interrupted cutting where repeated cycles of entry and exit motions of the cutting tool accomplish the actual metal removal and discontinuous chip generation. Milling has more variations in machine types, tooling, and work piece movement than any other machining method.

All milling machines, from compact tabletop models to the standard vertical knee mill and the massive CNC machining centers, operate on the same principles and operating parameters. The most important of these operating parameters are:

- cutting speed, which is the speed at which the tool engages the work
- feed rate, which is the distance the tool edge travels in one cutter rev..
- the axial depth of cut, which is the distance the tool is set below an un-machined surface.
- the radial depth of cut, which is the amount of work surface engaged by the tool. The capabilities of the milling machine are measured by motor horsepower, maximum spindle speeds and spindle taper size.

Milling Machines & Machining Centers -

Manual mills require that the operator/machinist set all the required parameters, change tools, and manually direct all table movement. However, with CNC capability, work is performed much faster, with exceptional repeatability. In addition, CNC computer programs can be verified and completed graphically before actual metal cutting begins.

A machining center is a machine for both milling and hole making on a variety of non-round or prismatic shapes. The primary types of machining centers are either vertical or horizontal. The vertical type is often preferred when work is done on a single face. With the use of rotary tables, more than one side of a work piece, or several work pieces, can be machined without operator intervention. Vertical machining centers using a rotary table have four axes of motion. Three are lineal motions of the table while the fourth is the table’s rotary axis. Horizontal centers with their horizontal spindles are better suited to larger, boxy work pieces. With a horizontal spindle, a wider variety of work piece shapes are easier to mount and chips fall out of the way better. Like vertical machining centers, horizontal machining centers have multiple-axis table movements. Typically, the horizontal center's table rotates to present all four sides of a work piece to the tooling.

Means **machining center** is a combination of many machine features put into a single machine. This machining center is a more versatile machine tool which can perform various operations such as:

1. It controls movements along 3, 4, 5 axes.
2. It is possible to move column, spindle head table traverse and also carry out table rotation, table tilting, tool changing, Job changing, Dimension Gauging or job probing, coolant supply, by programming codes.
3. The NC system may also move the axis in either point to point system, straight cut system or continuous path control system.
4. The basic control feature include simultaneous movements along all axes, programmable feed rates, spindle speeds and activation of fixed cycles such as, Drilling, Boring, Tapping, Mirroring, etc.
5. Additional features that increase the control capability or, machine operation include.

(a) Sequence number display, (b) Auxiliary function display i.e. spindle clamp or declamp, pallet clamp or declamp etc.(c) Manual data in-put, (d) Tool gauging and selection.(e) Tool length and tool radius compensation. (f) Linear and circular interpolation etc.

(g) Activation of different modes of programming / operation i.e. Block by Block, simulation, manual data input, Dry run full sequence run etc.

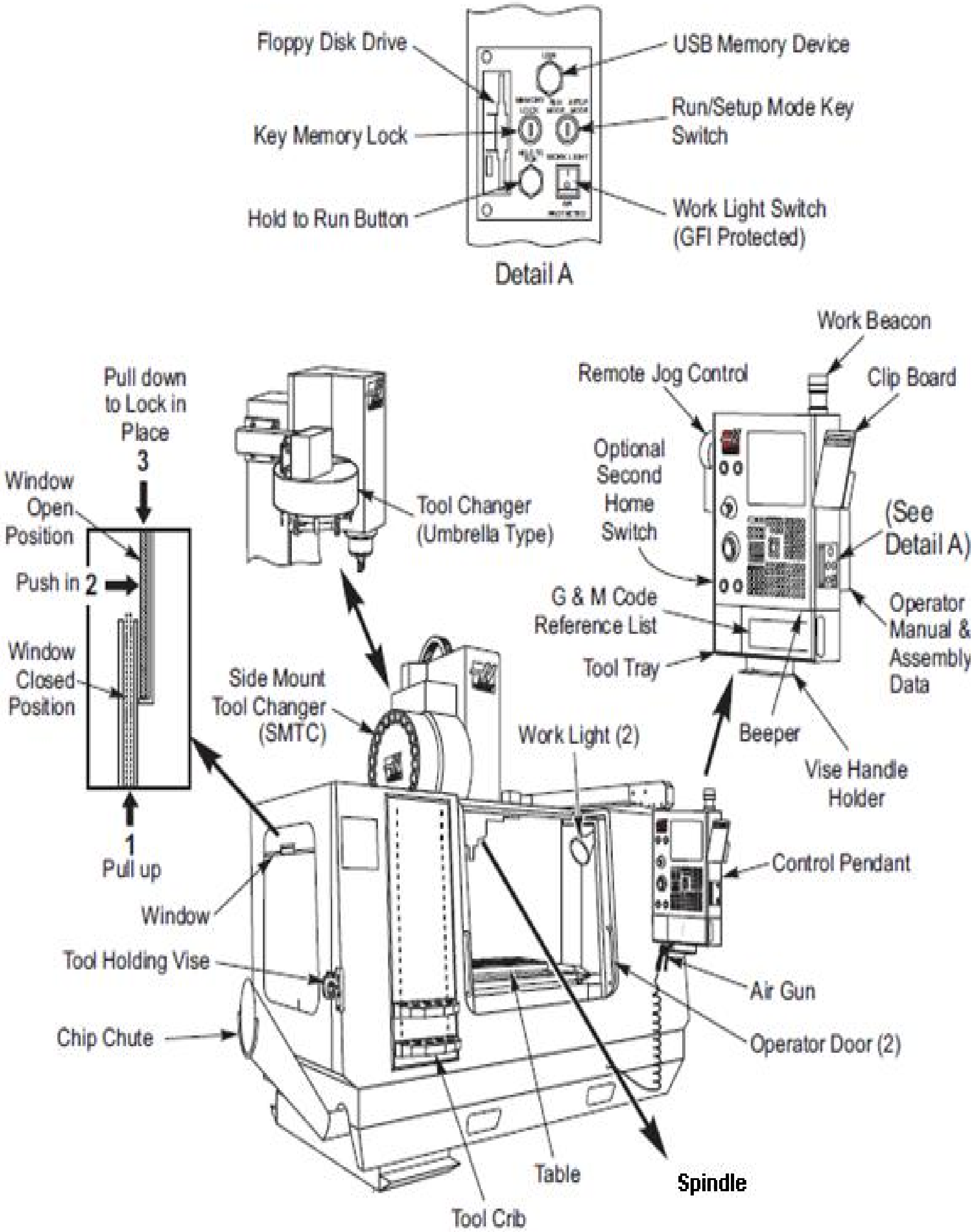
Toolchangers & Cutting Tools

The unique feature of the machining center is the tool changer. The tool-changer system moves tools from storage to spindle and back again in rapid sequence. While most machining centers will store and handle 20-40 individual tools, some will have inventories of over 99.

In general, a milling cutter is a rotary tool with one or more cutting edges, each of which removes a small amount of material as it contacts the workpiece. The variety of cutter types is almost limitless. One of the more basic is the face mill cutter used for milling flat surfaces. Used at high speeds, face mill cutters range from a few centimeters to over a half-meter in diameter. Some face mills will simultaneously mill a shoulder that is square to the surface.

Work that requires edge preparation, shoulders, and grooves, is accomplished with other milling cutters. An end mill cutter is a tool with cutting edges on its end as well as on its periphery. End mills are used for short, shallow slots and some edge finishing. Circular grooving or slotting cutters are more adapted to the making of longer and deeper slots. This is because end mills are susceptible to deflection during heavier cuts. Chamfers and contour milling are performed with specially shaped end mills.

CNC Milling Machine



Operation Control Panel

Operational Panel is depending on the type of the CNC machine and the manufactures specifications. It is common that many machines have used some special feature. The following features cover the most typical and common on the modern operation control panel.

Feature	Description
ON / OFF Switch	Power & control Switch for the main power and the control unit.
Cycle Start	Start Program execution and MDI command execution.
Emergency Stop	Stops all machine activity and turns off power to the control unit.
Feed hold	Temporarily stops motion of all axes.
Single Block	Allows automatic program run one block at a time & press cycle start switch for every block.
Dry Run	Enables program testing at fast feed rate.
Optional Stop	Temporarily stops the program execution (M01 is required in the program.
Block Skip	Ignore blocks preceded with a forward slash (/) in the program.
Spindle Speed Override	Overrides the programmed spindle speed. Usually within 50-150 % range.
Feed rate Override	Overrides the programmed Feed rate. Usually within 0-200% range.
Coolant Switch	Coolant control ON / OFF / AUTO.
Spindle Rotation	Indicates direction of spindle rotation CW / CCW.
Spindle Orientation	Manual orientation of the spindle.
Tool Change	Switch allowing a manual Tool change.
Reference Position	Switches and lights relating to setup of the machine from reference position.
Hand Wheel	Manual Pulse Generator (MPG), used for Axis select and handle Incremental movement.
EDIT mode	Use for to make a new Program and allows to changes to be made to a program stored in the CNC memory.
MEM/AUTO mode	Allows program execution from the memory of the CNC unit and allows automatic operations.
Simulation/Graphics	Allows to graphic representation of the Tool path for program checking.
MDI / MDA	Manual data input / manual data aided, allows manual operations during setup.
JOG mode	Allows axis movement with Jog feed rate for setup.
RAPID mode	Allows axis movement with Rapid feed rate for setup.
Alarm Lights	Red light indicating an error.
Alarm Massage	The Alarm massage display on the screen for measure.

Typical Specification of a CNC Machining Centre

Sl. No.	Description	Vertical Machining Centre	Horizontal Machining Centre
01	Number of Axes	3 Axes (XYZ)	4 Axes (XYZB)
02	Table Dimensions	780 x 400 mm 31 x 16 inches	500 x 500 mm 20 x 20 inches
03	ATC & No. of Tool	32	36
04	Maximum Travel – X axis	575 mm 22.5 inches	725 mm 28.5 inches
05	Maximum Travel – Y axis	380 mm 15 inches	560 mm 22 inches
06	Maximum Travel – Z axis	470 mm 18.5 inches	560 mm 22 inches
07	Table Indexing angle	-------	0.001 degree
08	No. of Slots & Size(width x Pitch)	5nos. & 18x80mm	5nos. & 18x80mm
09	Max. allowable wt. on table	650 Kgs	650 Kgs
10	Spindle Speed	60-8000 rpm	40-4000 rpm
11	Spindle Output	AC 7.5/5.5 KW AC 10/7 HP	AC 11 / 8 KW AC 15 / 11 HP
12	Spindle nose to table distance – Z axis	150 – 625 mm 6 – 24.6 inches	150 – 710 mm 6 – 28 inches
13	Spindle Taper	BT 40	BT 40
14	Feed rate Range	2 – 10000 mm/min 0.001-393 in/min	2–10000 mm/min 0.001-393 in/min
15	Rapid Traverse Rate – XY axis	30000mm/min 1181 inch./min.	30000mm/min 1181 inch./min.
16	Rapid Traverse Rate -Z axis	24000mm/min 945 in/min	24000mm/min 945 in/min
17	Positioning accuracy (X, Y, Z)	+/- 0.005 mm	+/- 0.003 mm
18	Repeatability accuracy (X,Y, Z)	+/- 0.002 mm	+/- 0.002 mm
19	Tool Selection	Random Memory	Random Memory
20	Maximum Tool Diameter	80 mm/3.15 in.	80 mm/3.15 in.
21	Maximum Tool Length	300 mm/11.8 in.	350 mm/13.75 in.

Machine Setup & Basic CNC Milling Operations

Safety Precautions

The safety precautions related to the use of CNC units is essential, it is to be observed by the users & ensure the safe operation of machines equipped with a CNC unit. The operator must become fully familiar with the CNC machine before attempting to operate the machine or to create a program & control the operation of the machine and carefully read the warning and caution.

(a) Warning is applied, when there is a danger of the user being injured or when there is a damage of both, the user being injured and the equipment being damaged, if the approved procedure is not observed.

(b) Caution is applied when there a danger of the equipment being damaged, if the approved procedure is not observed.

1. Immediately after switching on the power, do not touch any of the key on the panel unit.
2. Check the specification of the machine if in doubt.
3. Never attempt to machine a work piece without first checking of the operation.
4. Before operating the machine, thoroughly check the entered data.
5. Ensure that the specified Speed rate & Feed rate is appropriate for the intended operation.
6. When using a tool compensation function, thoroughly check the direction and amount of compensation.
7. Carefully read the Alarm massage display on the screen and take measure. If not the machine may behave unexpectedly.

Personal safety

1. Do not operate with the door open condition.
2. Do not operate with out proper training.
3. Always wear the safety goggles.
4. The machine is automatically controlled and may start at any time.
5. Improperly or inadequately clamped parts may be ejected with deadly force.
6. Do not exceed rated RPM.
7. Higher RPM reduces chuck/ vice clamping force.
8. Hydraulic pressure must be set correctly.
9. Chucks/ vices must be greased weekly and regularly serviced.

Introduction to Cutting Parameter:

The factors that affect the cutting process during the operation in milling is called cutting parameters for milling. The basic Cutting Parameters in milling operation are as follows. (1) Cutting Speed , (2) Feed , (3) Depth of Cut .

Factors are influenced on Cutting Speed

There are two main factors are influenced on cutting speed i.e. 1. Constant factors-- Work Material which to be machined and Cutting Tool material by which operation has to be performed. 2. Variable factors - Machine rigidity, Proper Coolant supply. Operators efficiencies etc.

Charts with suggested speeds and feeds for common materials are available from major tooling companies. These charts are helpful for machining as well as in programming.

Factors are influenced on surface finish

Precision parts require a certain degree of surface finish quality. Technical drawing indicates the required finish for various features of the part. The most important factors influencing the quality of surface finish are spindle speed, feed rate, cutting tools nose radius and the amount of material removed. Generally, a larger cutter radius and slower feed rate contributes towards finer surface finishes.

Cutting Speed:

The speed of milling cutter is peripherical linear speed resulting from rotation. It is expressed in meters per minute.

The cutting speed can be derived from the formula:

$$V_c = \pi dn / 1000 \text{ meter per min.}$$

Where V_c = Cutting speed in m per min.

d = Diameter of the cutter in mm.

n = Cutter speed in RPM

Factors Govern the Cutting Speed: For a given material there will be an optimum cutting speed for a certain set of machining conditions, and from this speed the spindle speed (RPM) can be calculated.

Factors Affecting the Calculation Of Cutting Speed :

1.The material being machined (steel, brass, tool steel, plastic, wood) (see Table below)The material the cutter is made from (Carbon steel, high speed steel (HSS), carbide, ceramics)

2.The economical life of the cutter (the cost to regrind or purchase new, compared to the quantity of parts produced)

3.Cutting speeds are calculated on the assumption that optimum cutting conditions exist, these include:

4. Metal removal rate (finishing cuts that remove a small amount of material may be run at increased speeds)
5. Full and constant flow of cutting fluid (adequate cooling and chip flushing)
6. Rigidity of the machine and tooling setup (reduction in vibration or chatter)
7. Continuity of cut (as compared to an interrupted cut, such as machining square section material in a lathe)
8. Condition of material (mill scale, hard spots due to white cast iron forming in castings).

The cutting speed is given as a set of constants that are available from the material manufacturer or supplier, the most common materials are available in reference books, or charts but will always be subject to adjustment depending on the cutting conditions. The following table gives the cutting speeds for a selection of common materials under one set of conditions. The conditions are a tool life of 1 hour, dry cutting (no coolant) and at medium feeds so they may appear to be incorrect depending on circumstances. These cutting speeds may change if, for instance, adequate coolant is available or an improved grade of HSS is used (such as one that includes cobalt).

<u>Cutting Speeds for Various Materials Using a Plain HSS Cutter:</u>

Material type	Meters per min (MPM)	Feet per min (FPM)
Mild steel	26-30	50-60
Steel (tough)	30-38	100-125
Cast iron (medium)	18-24	60-80
Alloy steels (1320–9262)	20-37	65-120
Carbon steels (C1008-C1095)	21-40	70-130
Free cutting steels	35-69	115-225
Stainless steels (300 & 400 series)	23-40	75-130
Bronzes	24-45	80-150
Leaded steel (Lead Aloy 12L14)	91	300
Aluminum	75-105	25-350
Brass	90-210	300-700

Feed:

The feed in a milling machine is defined as the rate with which the work piece advances under the cutter. The feed is expressed in a milling machine by the following three different methods.

(1) Feed Per Tooth, (2) Feed Per Cutter Revolution, (3) Feed Per Minute

Factors Affecting the Feed Rate:

Increased feed reduces cutting time but it greatly reduces the tool life. The feed depends upon the factors such as size, shape, strength and method of holding the component, the tool shape and its setting, the rigidity of machine, depth of cut, etc. Coarser feeds are used for roughing and finer feeds for finishing cuts.

The feed rate used on milling depends upon several factors, such as

Depth of cut, Tool geometry, Sharpness of cutter, Material of work-piece,

Strength and uniformity of work piece, Types of finish and accuracy required, Power and rigidity of machine, etc.

Depth of Cut:

The depth of cut in milling is the thickness of the material removed in one pass of the work under the cutter. It is the perpendicular distance measured between the original and final surface of the work piece and is expressed in mm.

Offsets Setting

X axis

Clamp the Edge Finder/ Webler / End Mill Cutter in collet adaptor

Hold the collet adaptor in spindle → Start the spindle with 600 rpm → Touch the webler to the reference face of x-axis → Stop the spindle → Press the offset key on keypad → In offset select work zero offset 'X' → Press the key part zero set on keypad.

Y axis

Touch the webler to the reference face of Y-axis → Stop the spindle → Press the offset key on keypad → In offset select work zero offset 'Y' → Press the key part zero set on keypad

Z axis

Clamp the required cutting tool in collet adaptor → Hold the collet adaptor in spindle → Check the zero error of tool presetter first by using dowel pin → Put the tool presetter on top of the work piece → Touch the cutting tool to the top of the tool presetter and make the tool presetter value zero. → In offset select tool geometry 'Z' → Press the key tool offset measure on keypad → Add the height of the tool presetter (-50) & press enter → Move the cutter in upward direction.

CNC Programming

What's the Programmer has to do ?

Before programming a programmer should take care of the following to make the part program true and effective,

01. Study the relevant component drawing thoroughly.
02. Identify the type of material to be machined.
03. Determine the specifications and functions of machine to be used.
04. Decide the dimension and mode of dimensioning (Metric / Inch).
05. Decide the coordinate (Dimensioning) system (Abs. / Inc.).
06. Identify the plane of Cutting.
07. Check the Tooling required.
08. Determine the cutting parameters with the Job & Tool combination.
09. Decide the feed rate.
10. Establish the sequence of machining operation.
11. Identify whether use of special features like Sub-Routing program, Mirroring, Scaling, etc.
12. Decide the mode of storing the part program once it is completed.

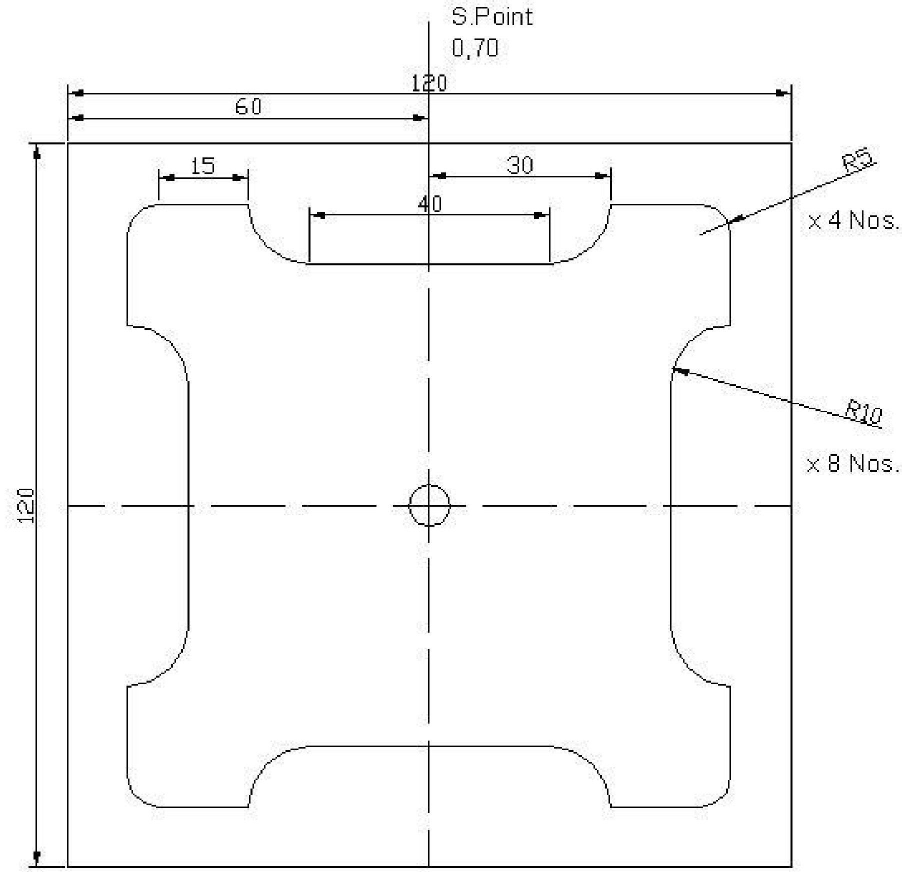
The Sequential steps for to make a Part Programme.

When a programmer making a part program, he should maintain the following sequential steps for to ensure that the tool does not foul with the job or others at automatic machining and to reach a perfect goal.

01. Program Head (Program name or number & May keep some Information's for future use).
02. Select Tool change position.
03. Tool change (Manual or Automatic Tool Changer).
04. Select operational position.
05. Cutting parameter setting (Cutting speed, Feed etc.).
06. Operation.
07. On completion of total operation of that tool, then return to the tool Change position.
08. If there is any other operation, then again proceed from No. 3 to No.7,If not then program End.

PROFILE MILLING

Exercise: Manufacture the following job within 0.01 mm accuracy and 20 mm deep.



Main Programme

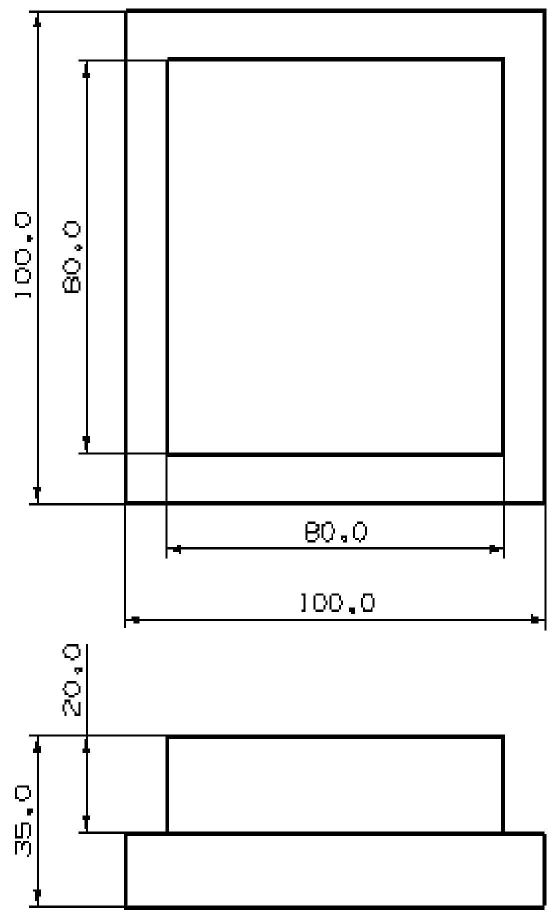
```
O04959;
G00 G54 G90 Z150;
G01 X0 Y70;
T02 D02 M06;
G43 H02 Z50  ;
G00 X0 Y70;
S1000 M03
G00 Z0. F500;
M98 P5060 L20 F200;
G00 G90 Z150;‘
 M05;
 M30;
```

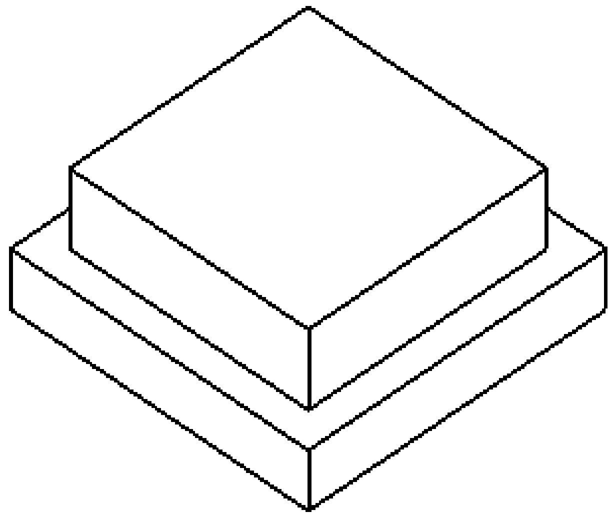
Sub Programme

```
O5060;
G00 G90 X0 Y70 ;
G01 G91 Z-1 F200;
G41 G90 G01 X0 Y40;
G01 X20 Y40;
G03 X30 Y50 R10;
G01 X45 Y50;
G02 X50 Y45 R5;
G01 X50 Y30;
G03 X40 Y20 R10;
G01 X40 Y-20;
G03 X50 Y-30 R10;
G01 X50 Y-45;
G02 X45 Y-50 R5 ;
G01 X30 Y-50 ;
G03 X20 Y-40 R10;
G01 X-20 Y-40;
G03 X-30 Y-50 R10;
G01 X-45 Y-50;
G02 X-50 Y-45 R5;
G01 X-50 Y-30;
G03 X-40 Y-20 R10;
G01 X-40 Y20;
G03 X-50 Y30 R10;
G01 X-50 Y45;
G02 X-45 Y50 R5;
G01 X-30 Y50;
G03 X-20 Y40 R10;
G01 X0 Y40
G40 G01 X0 Y70
M99;
```

STEP MILLING

Exercise:





Aim: Step Milling Operation as per Drawing.
Raw material size: 105 x 105 x 40 mm,
Finished size: 100 x 100 x 35 mm

Main Programme

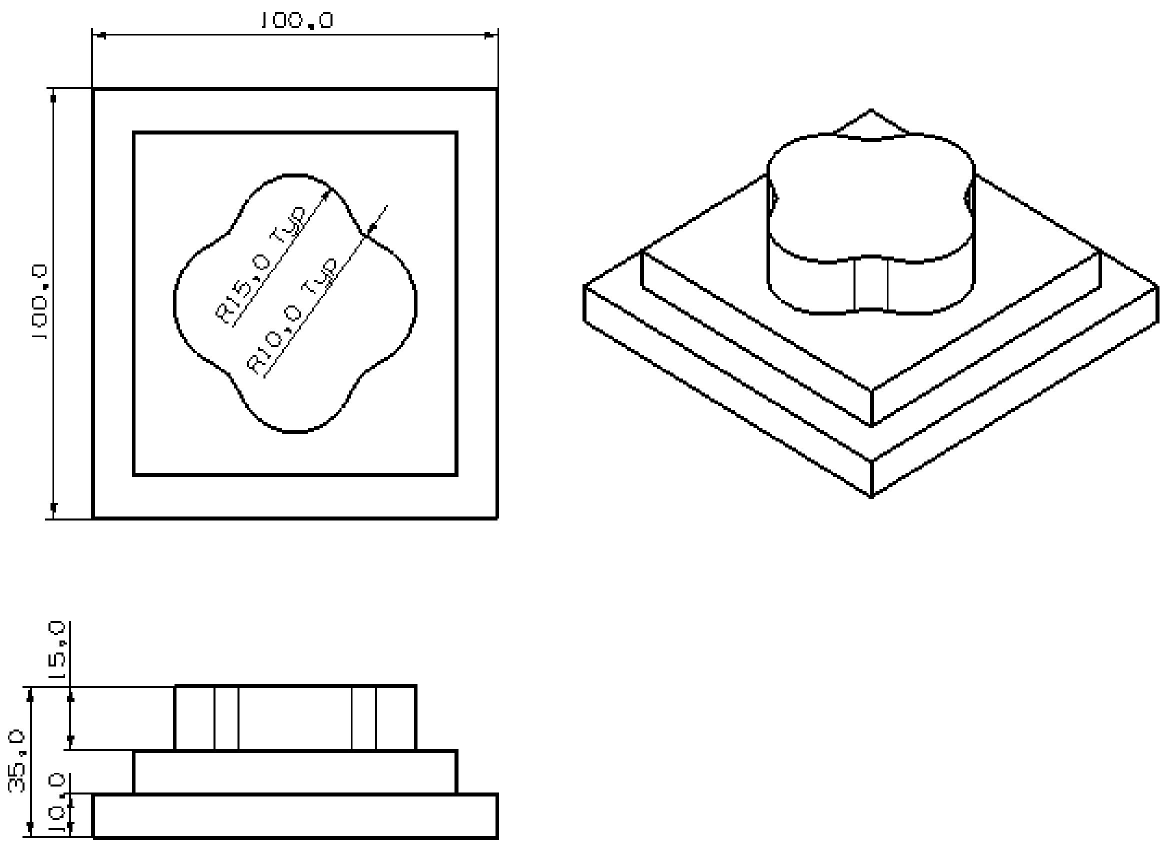
```
O09132;
T1 M06;
S500 M03;
G00 G54 G90 X0 Y0 F500;
G00 G43 H01 Z50.;
G01 Z0. F500;
M98 P12346 L10 F250;'
G00 G90 Z50;
M05;
M30;
```

Sub Programme

```
(ASSUME CUTTER DIA 20MM)
O12346;
G00 G54 G90 X0Y0 ;
G01 G91 Z-2 F250;
G01 G90 X100;
G01 Y100;
G01 X0;
G01 Y0;
M99;
```

PROFILE MILLING

Exercise:



Aim: Profile Milling Operation as per Drawing.

Cutting Tool: End Mill Ø20mm.

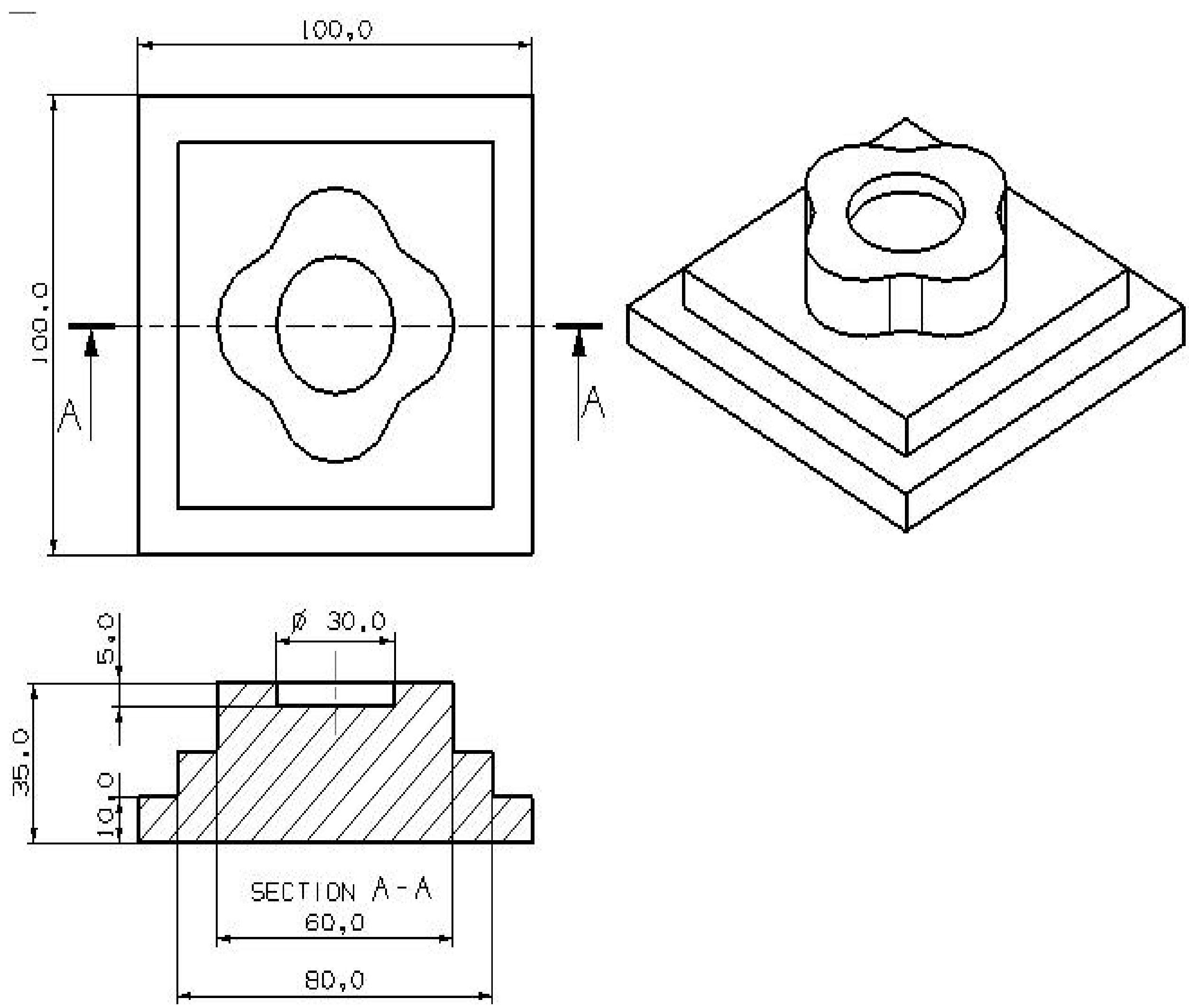
Main Programme

```
O08889;
T1 D01 M06;
S860 M03;
G54 G90 G40;
G00 X0 Y50 F500.;
G00 G43 H01 Z50.;
G00 Z2;
G01 Z0. F500.;
M98 P9132 L16 F340.;
G00 G90 Z50.;
M05;
M30;
```

Sub Programme

```
O09132;
G00 G90 X0 Y50 ;
G01 G91 Z-1 F340;
G90 G41 G01X20 Y50;
G02 X29.9 Y64.1 R15;
G03 X35.9 Y70.1 R10;
G02 X69.1 Y90.1 R15;
G03 X70.1 Y64.1 R10;
G02 X70.1 Y35.9 R15;
G03 X69.1 Y29.9 R10;
G02 X35.9 Y29.9 R15;
G03 X29.9 Y35.9 R10;
G02 X20 Y50 R15;
G40 G01 X0 Y50;
M99;
```

INSIDE MILLING OPERATIONS - POCKET MILLING OPERATION



Main Programme

```
O09132;
O12346;
S540 M03;
G00 G54 G90 X50 Y50 F500;
G00 G43 H01 Z50;
G01 Z1. F500;
M98 P12346 L110 F214;
G00 G90 Z50;
M30;
```

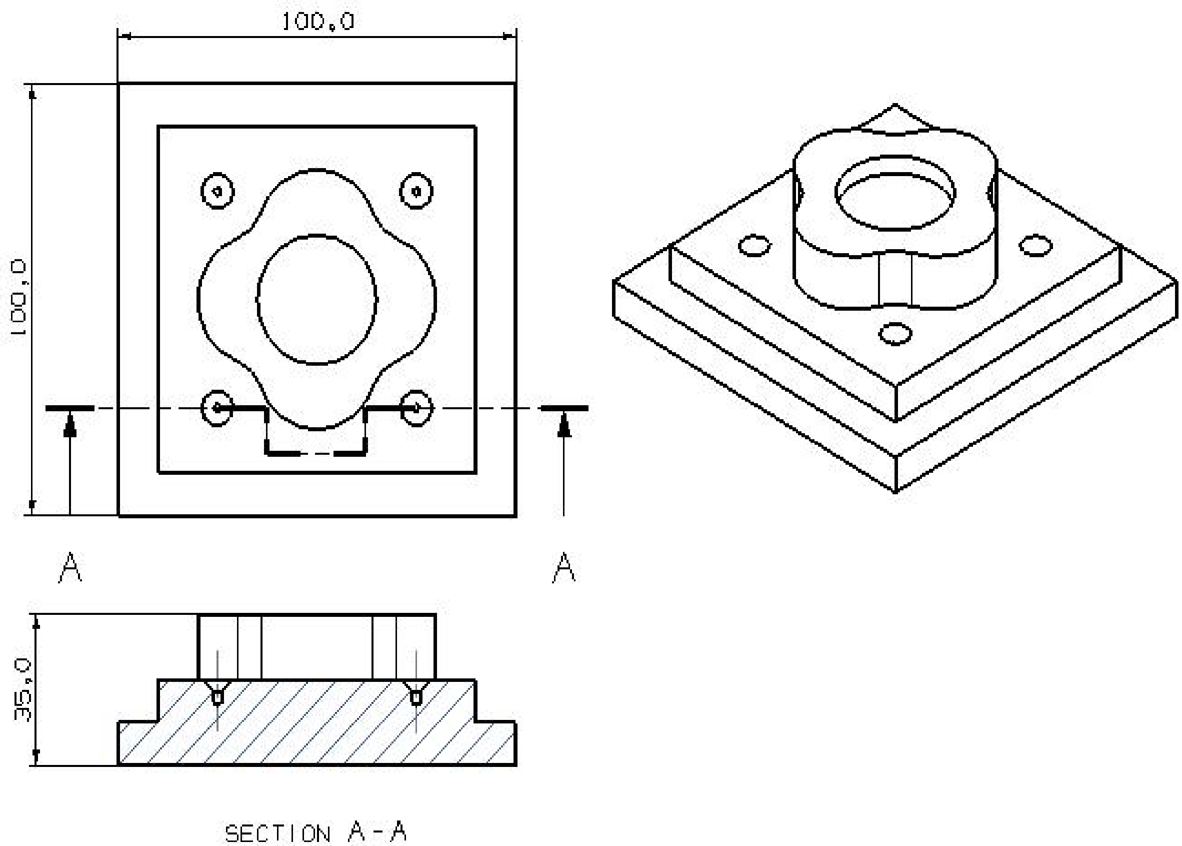
Sub Programme

```
(ASSUME CUTTER DIA 20mm) M06;

G00 G54 G90 X50 Y50;
G01 G91 Z-0.2 F100;
G01 G90 X55 F150;
G02 X45 Y50 R15;
G02 X55 Y50 R15;
G01 X50 Y50;
M99
```

SPOTTING (CENTRE DRILLING) & DRILLING OPERATION

Exercise:

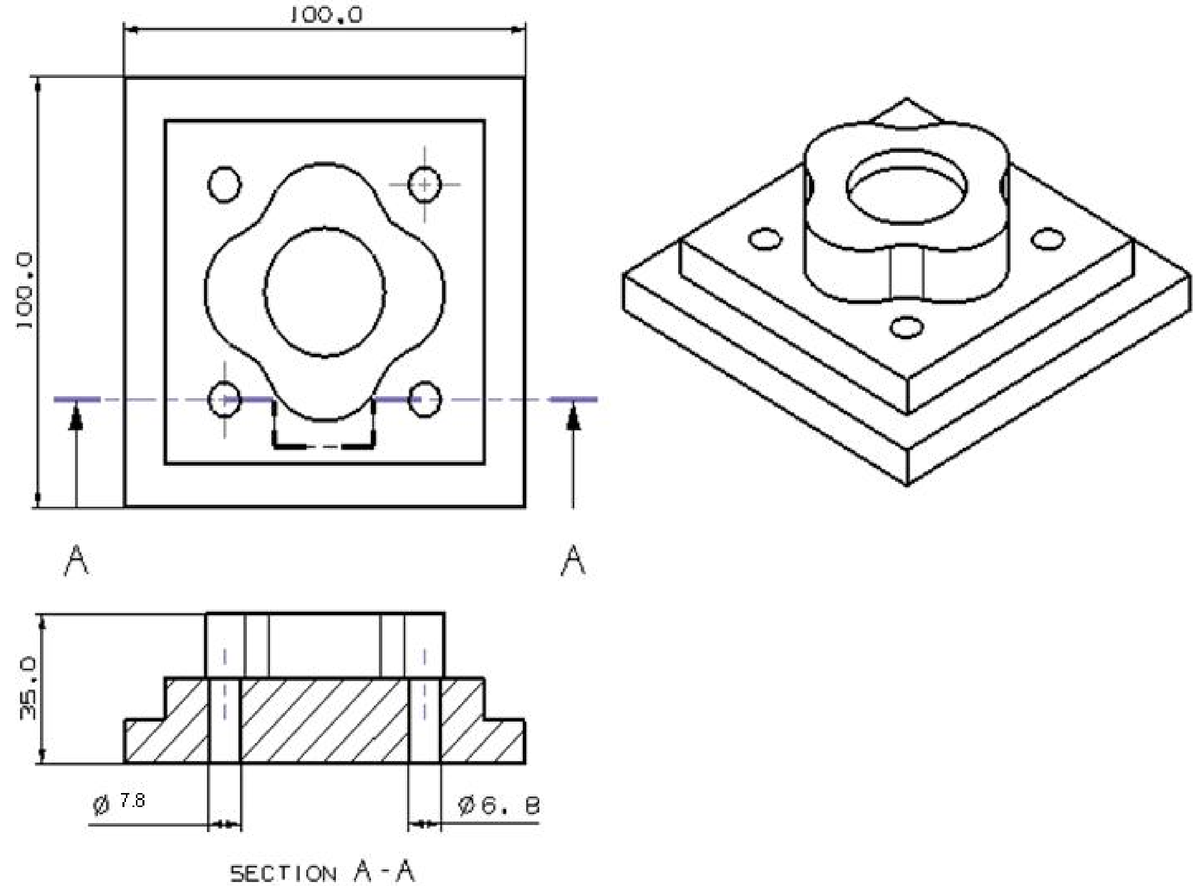


Main Programme

```
O09132;
TI M06;
S2500 M03;
G00 G54 G90 X25 Y25;
G00 G43 H01 Z50.;
G01 Z1 F500;
G83 G98 Z-5.5 L0 Q0.2 F600;
            X75 Y25;
            X25 Y75;
            X75 Y75;
G00 G80 Z100;
M05;
M30;
```

DRILLING OPERATION

Exercise:

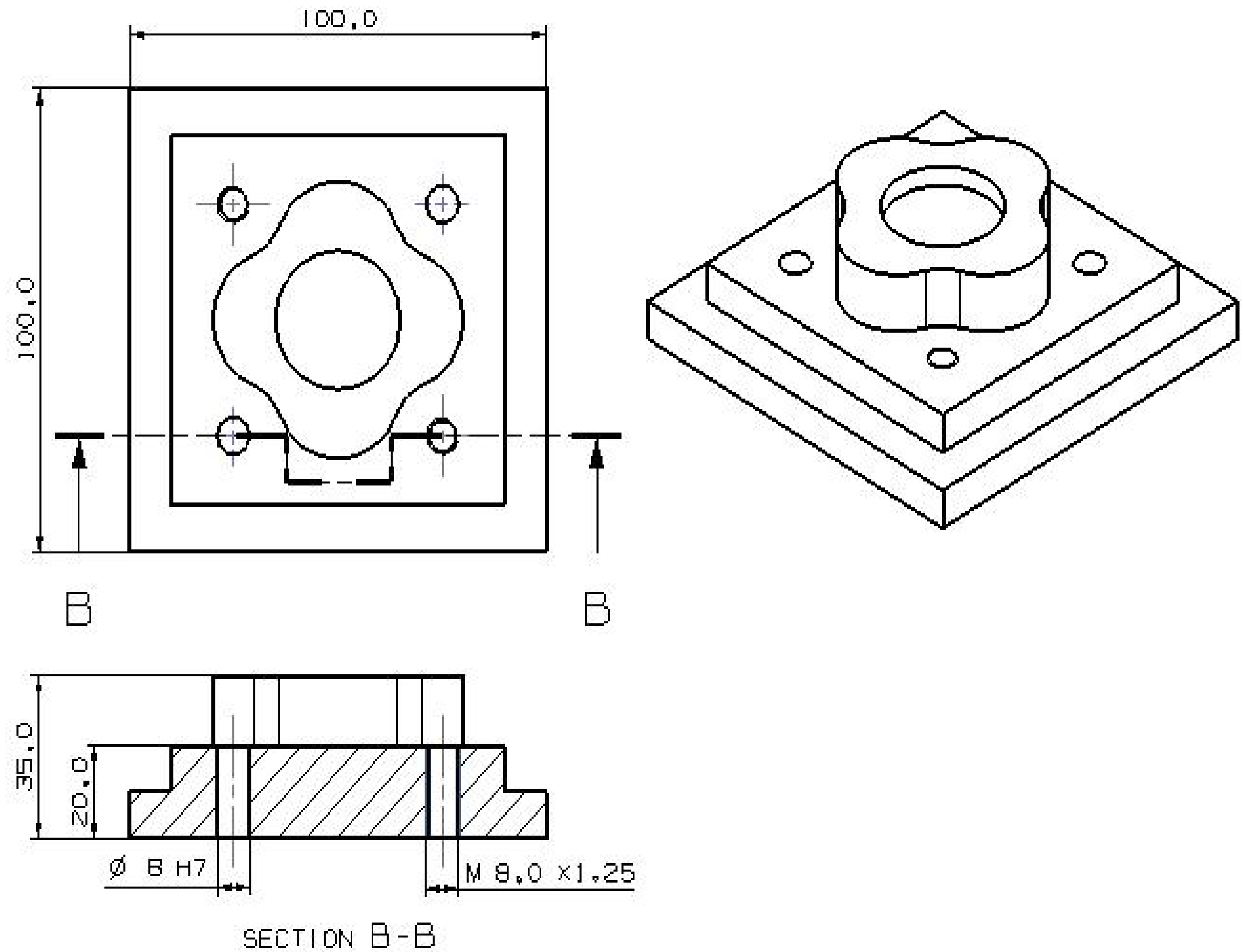


Main Programme

```
O09132 (DIA 7.8);
TI M06;
S102 M03;
G00 G54 G90 X25 Y25;
G00 G43 H01 Z50.;
M08 / M07;
G01 Z1 F500;
G83 G98 Z-40 L0 Q0.2 F500;
          X75 Y75;
G00 G80 Z100;
M09;
M30;
```

```
O09133(DIA 6.8);
TI M06;
S1200 M03;
G00 G54 G90 X75 Y25;
G00 G43 H01 Z50.;
M08 / M07;
G01 Z1 F500;
G83 G98 Z-40 L0 Q0.2 F500;;
          X25 Y75;
G00 G80 Z100;
M09;
M30;
```

TAPPING OPERATION



Aim: Tapping Operation as per Drawing.

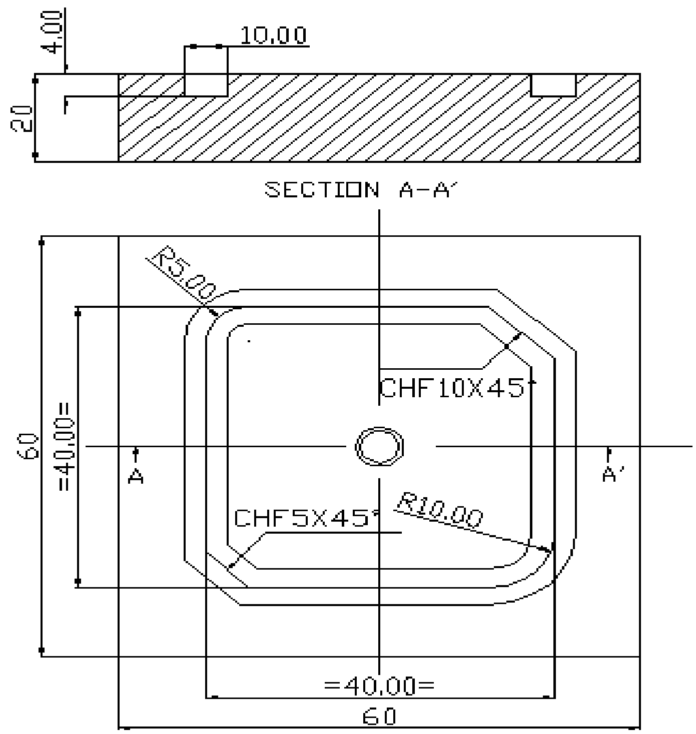
Main Program (Tap M8.0X1.25)

```
O09132;
TI M06;
S20 M03;
G00 G54 G90 X25 Y75 F500;
G00 G43 H01 Z50;
M08 / M07
G01 Z1 F500;
G84 Z-35 R1 F25;
      X75 Y25;
M05;
M09;
M30;
```

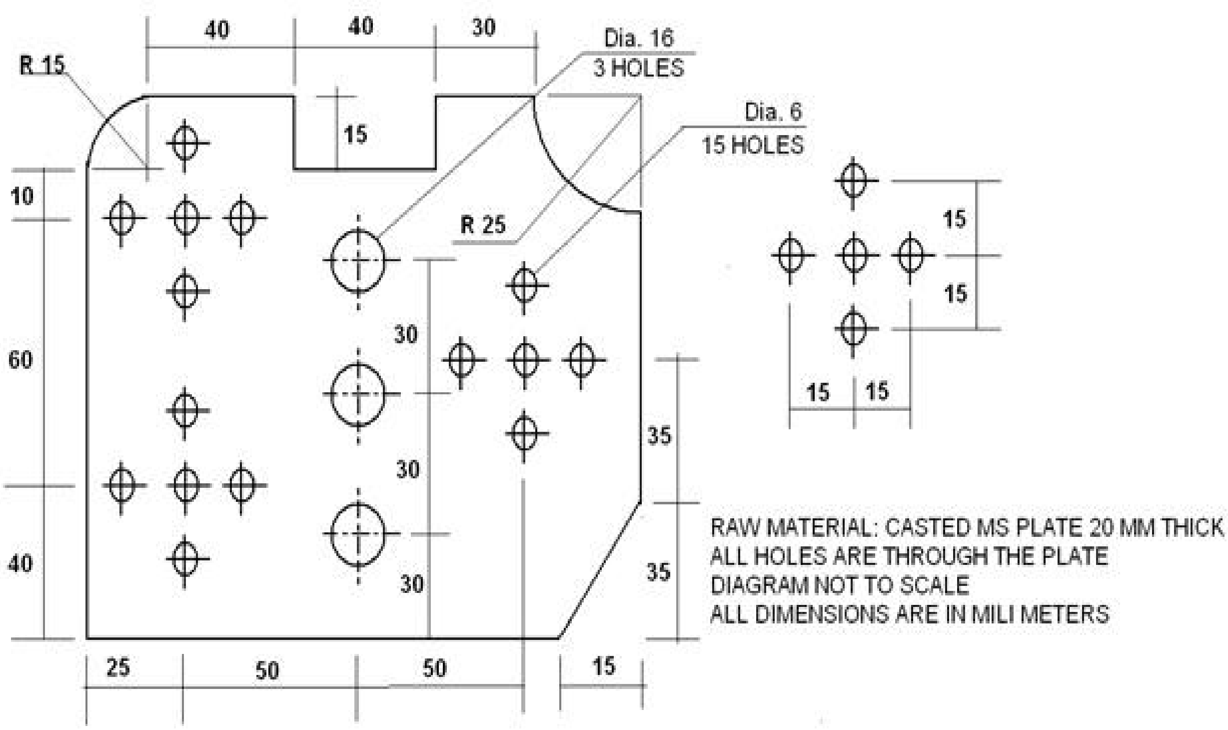
Assignments:

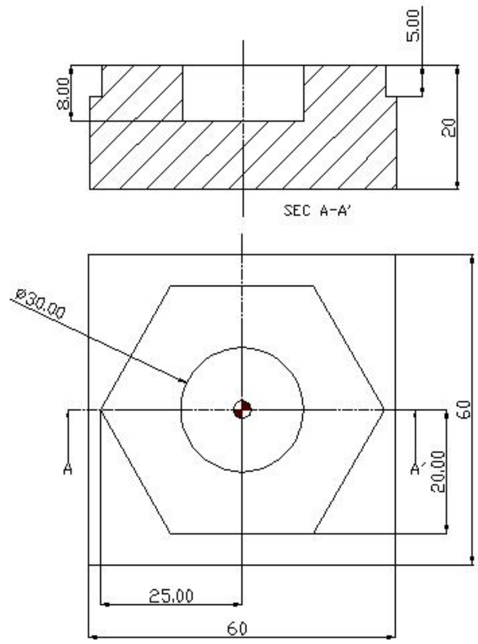
Read the drawing carefully & prepare the program.

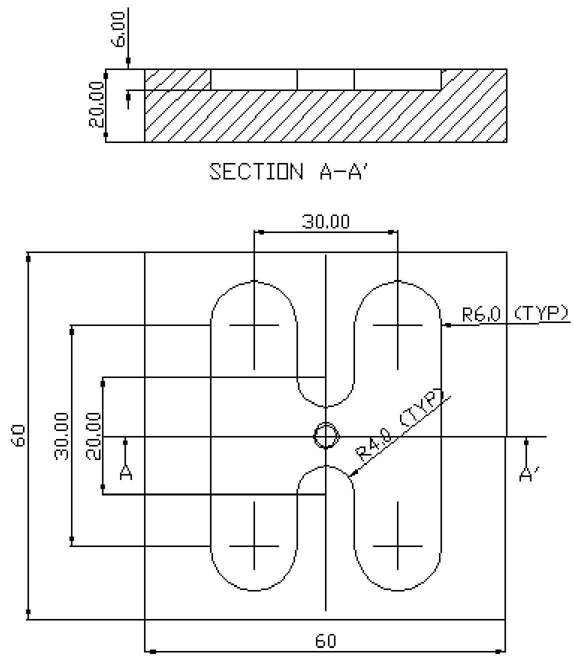
1.

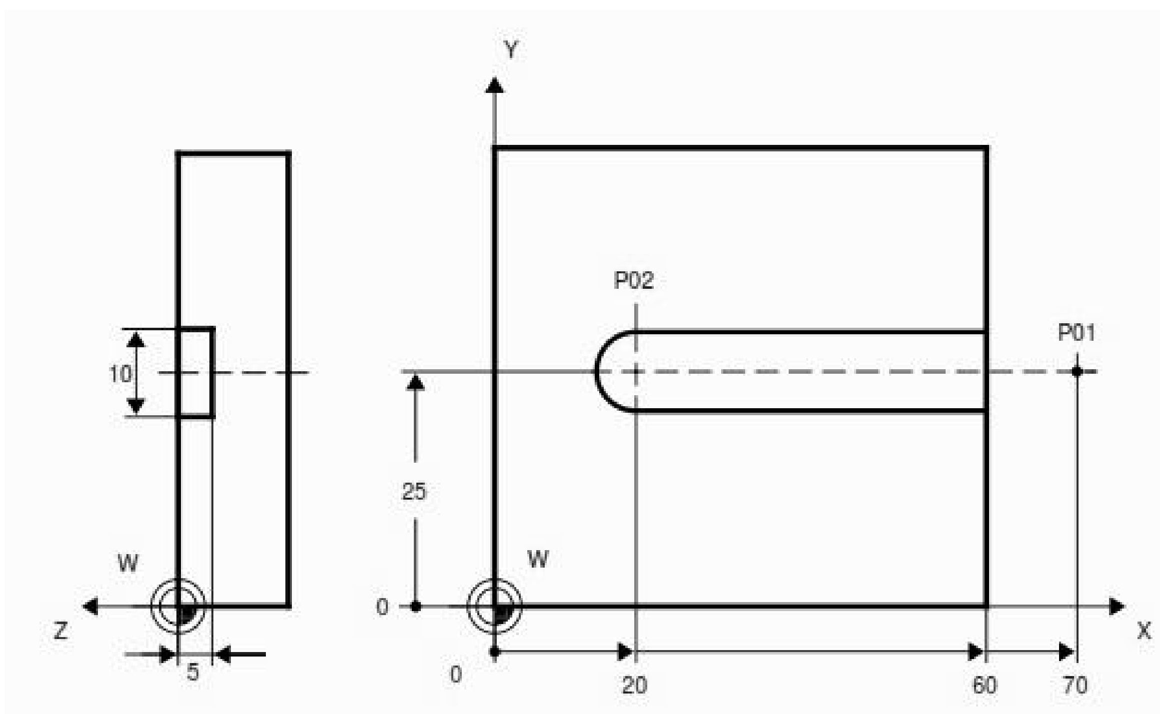


2.

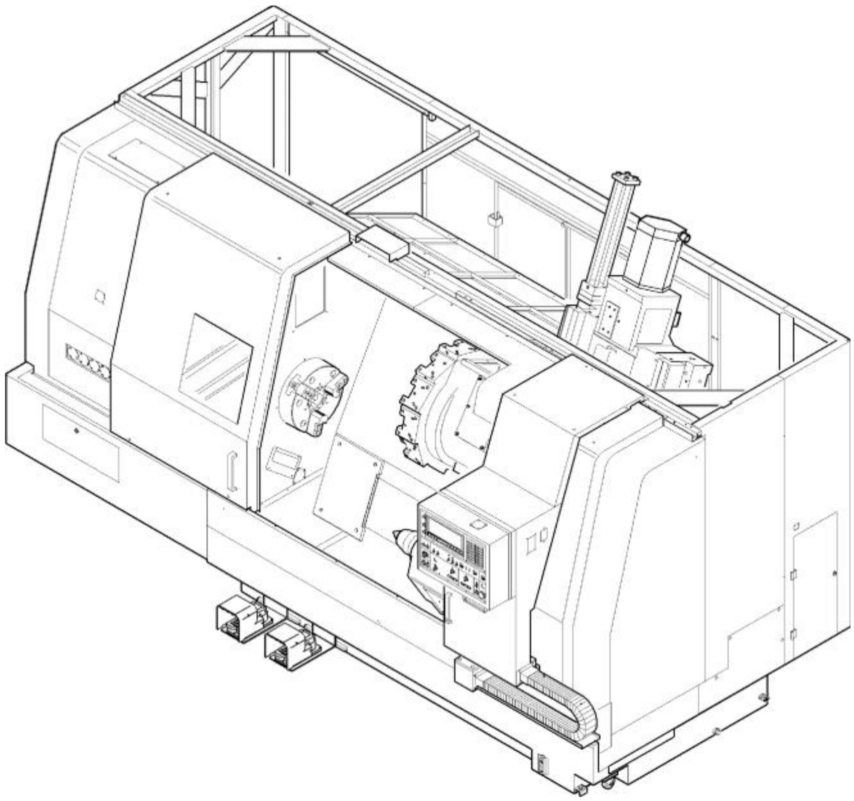



5.00
8.00
20
SEC A-A'
Ø30.00
60
A
A'
20.00
25.00
60


6.00
20.00
SECTION A-A'
30.00
R6.0 (TYP)
R4.0 (TYP)
60
30.00
20.00
A
A'
60


Y
P02
P01
10
25
W
Z
5
0
X
20
60
70

CNC TURNING



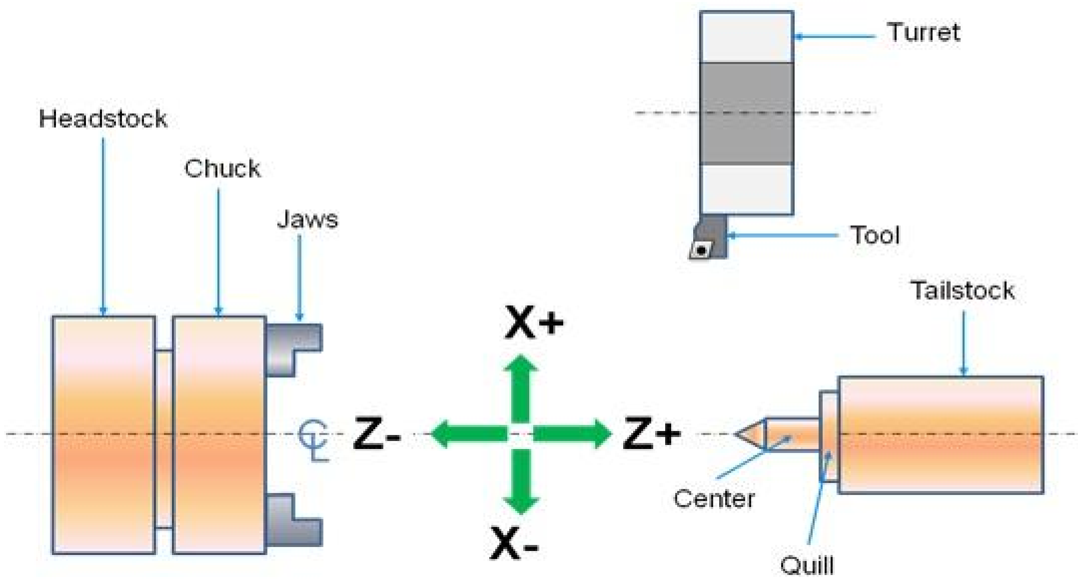
What is CNC Turning?

CNC turning, in the operation of a machine shop, is a method of machining a part in which a pointed cutting tool is fed parallel onto the surface of a material being rotated. This rotation is performed by a lathe, considered to be the oldest machining tool dating far back to ancient Egypt. The lathe secures and spins the part being machined, allowing for a simple single-point cutting tool to remove and shape the material, creating the desired part. This is accomplished along a dual axis of movement that may be a straight line, or along a prescribed set of curves or angles. Turning allows for the creation of varying complex shapes including plain, tapered, contoured, filleted, threaded, and radius profiles. Turning in its purest form involves the spinning of a lathe and the steady hand of the operator applying the cutting tool to the material being machined. Advances in technology have led to the creation of CNC, or Computer Numerical Control, lathes and turning processes. Beyond programming commands into the CNC lathe, the operator is taken out of the equation. An automated system now holds the cutting tool firmly in the lathe and follows a pre-programmed design, allowing for a precise turn, exact tolerances, and an abundance of shapes; straight, conical, curved, or grooved.

The Coordinate System in the Lathe Machine

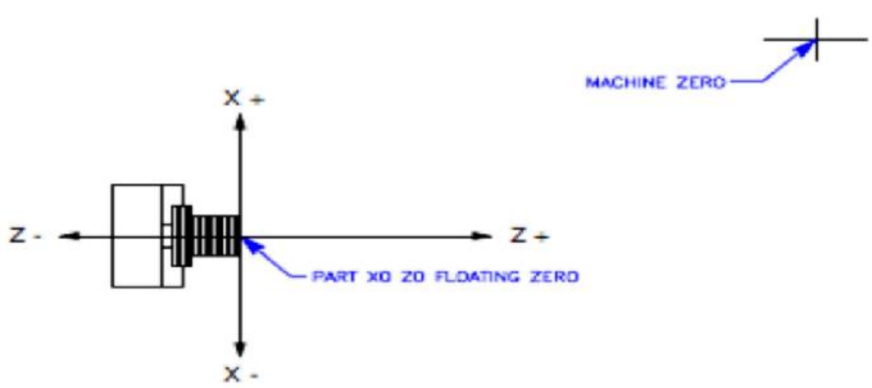
All CNC machines move tools to specific locations described by coordinate systems. With lathes the coordinate system can be simply described as two number lines that intersect.

With lathes the vertical number line is called the **X-axis**. The horizontal number line is called the **Z-axis**. When programming lathes **X0 is always the centerline** of the part you are working on and always X- value is Diameter Value. Therefore an X move from centerline or X0 to X10.0 mm. will only be moving the machine along the X axis 5.0 mm. in the positive direction.

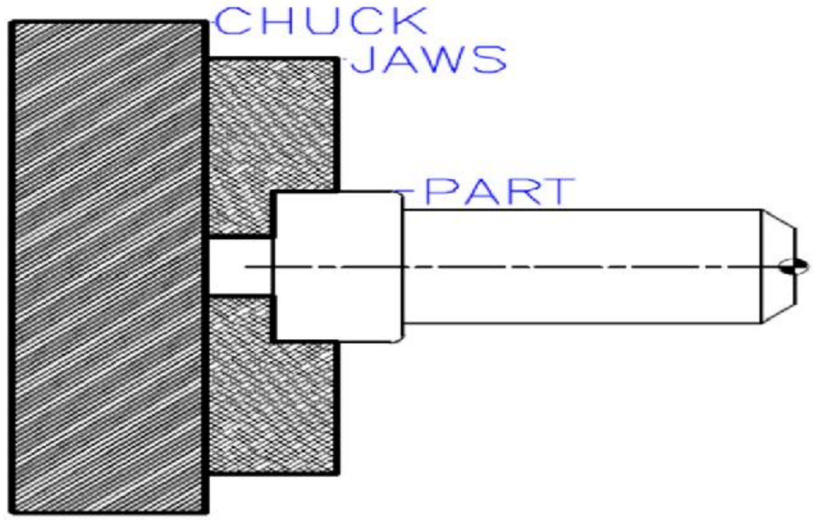


This is crucial to the operation and function of a CNC machine as **all of our programs**, **locations** of **fixtures** and **tooling** are **based** off of **machine home**. At home position the **machine coordinates** are **X0, Z0**. It would not be easy or convenient to write a program using machine coordinates. Instead programs are written with values that would correspond to dimensions found on prints. To do this a secondary floating zero point is established using offsets.

This floating zero is referred to as the **PART ZERO or PART ORIGIN**



When we setup a machine, we need to tell the machine the distance different tools at home position are from the part origin. Each tool is manually touched off the face and diameter and thru key strokes the distances from machine zero to the part zero are saved in X and Z register of the **Tool Offsets Page**.



<u>Absolute and Incremental Positioning</u>

Programmers normally use the front end of our finish machined part as (Z-Zero) and the centerline of part as (X-Zero).
There are two methods used by the programmer to "Steer" our machine.
The first is "ABSOLUTE POSITIONING". Absolute means that X and Z code values are based on the ZERO POINT on the part. If a diameter of 10.0 mm is needed, it is input as X10.0.
The programmer has another tool available to him called "INCREMENTAL POSITIONING". This is movement based on where the machine is currently sitting. It is also called point to point programming. If a change of 2.0 mm. smaller diameter is required of the machine from where it is currently sitting U-2.0 is put in the code.

The letters X&Z represent ABSOLUTE POSTIONING
The letters U&W represent INCREMENTAL POSTIONING
If you are familiar with the mill programming language, absolute and incremental are handled differently for Mills and Lathes. A mill uses G codes (G90 and G91) to go back and forth between the two. Where as a lathe uses the different letters to differentiate them.

G01 X2.000 W-.25 (Move X in Absolute, Z in Incremental)
or
G01 U.5000 Z-.5000 (Move X in Incremental, Z in Absolute)

Preparatory Functions

G-CODE LIST

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04	00	Dwell
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)		Polygonal turning cancel

G51.2 (G251)	G51.2 (G251)	G51.2 (G251)	20	Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON or balance
G69	G69	G69		Mirror image for double turrets OFF or balance cut mode cancel
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
	G90	G90	03	Absolute programming
	G91	G91		Incremental programming
	G98	G98	11	Return to initial level
	G99	G99		Return to R point level

M-CODE LIST

M-CODE	DESCRIPTION	REMARK	M-CODE	DESCRIPTION	REMARK
M00	PROGRAM STOP		M39	STEADY REST 1 UNCLAMP	OPTION
M01	OPTIONAL STOP		M40	GEAR CHANGE NETURAL	
M02	PROGRAM END		M41	GEAR CHANGE LOW	
M03	MAIN-SPINDLE ON FORWARD		M42	GEAR CHANGE MIDDLE	
M04	MAIN-SPINDLE ON REVERSE		M43	GEAR CHANGE HIGH	
M05	MAIN-SPINDLE STOP		M98	SUB-PGM Call IN MAIN PGM.	OPTION
M07	HIGH PRESSURE COOLANT ON	OPTION	M99	END OF SUB-PROGRAM	OPTION
M08	COOLANT ON		M50	BAR FEEDER COMMAND 1	OPTION
M09	COOLANT OFF		M51	BAR FEEDER COMMAND 2	OPTION
M30	PROGRAM END WITH REWIND	OPTION	M52	SPLASH GUARD DOOR OPEN	OPTION

Note:

M00 : For this command, main spindle stop, cutting oil, motor stop, tape reading stop are carried out.

M01 : While this function is the same as M00, it is effective when The optional stop switch of console is ON. This command shall be over rided if the optional stop switch is OFF.

M02 : Indicates the end of main program and main spindle stop, cutting oil motor stop, tape reading stop.

M03 : Main Spindle ON Anti-Clockwise (Reverse).

M04 : Main Spindle ON Clockwise (Forrowed).

M05 : Main Spindle OFF.

M07 : is used to turn on mist coolant.(If available)

M08 : is used to turn on flood coolant.

M09 : turns off the coolant.

M30 : This is the same as M02 and it returns to the starting position of the program when the memory and the tape are running.

**It is favorable for M code to program in a command paragraph independently.

Function	Addres	Meaning of address
Program number	O	Letter for Program number
Block sequence number	N	Sequence Number
Preparatory function	G	Specifies a motion mode (Linear, arc, etc)
Dimension word	X, Z U,W I, K R	Command of moving position(absolute type) of each axis Instruction of moving distance and direction(incremental type) Ingredient of each axis distance of circular center and Value of chamfering Radius of circle, corner R, edge R
Feed function	F,	Designation of feed rate and thread lead
Auxiliary function	M	Command of ON/OFF for operating parts of machine
Spindle speed function	S	Designation of speed of main spindle or rotation time of main spindle
Function (Tool)	T	Designation of tool number and tool compensation number
Dwell	P, U, X	Designation of dwell time
Designation of program	P	Designation of calling number of auxiliary program
Designation of sequence No	P,	Calling of compound repeat cycle, end number
Number of repetitions	L	Repeat time of auxiliary program
Parameters	P.Q,R,I, Etc.	Parameter at fixed cycle

Machine Defaults

The control automatically recognizes these G codes when your CNC lathe is powered up:

G00 Rapid Traverse
G18 X, Z Circular Plane Selection
G20/G21 Input in Inch / mm
G40 Cutter Compensation Cancel
G54 Work Coordinate Zero
G80 Canned Cycle Cancel
G97 Constant Surface Speed Cancel
G99 Feed Per Revolution

G18: G18 is the default command for designating which plane (X,Z plane) a radius is thrown. If an arc is attempted in a G17 (XY plane) or a G19 (YZ plane) the CNC lathe will alarm out.

G20: G20(inch) and G21(mm) make sure setting #9 is set to inch or Metric Normally most shops are either inch or metric. If the wrong setting is active it will be clearly seen in graphics or the machine will give an over travel alarm.

G40: G40 cancels cutter compensation codes G41 or G42. Cutter compensation should be cancelled after it is used.

G54: G54 is however the default value and if not changed will always be active.

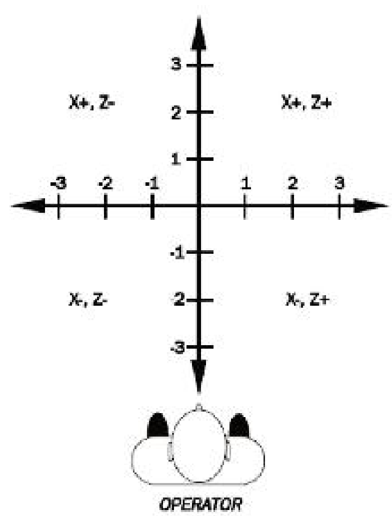
G80: Cancels canned cycles.

G97: Constant spindle speed(RPM). A G97 should be programmed at the beginning and end of every tool in a program with a spindle speed.

G99: Feed per revolution: This is the default. Normally lathes are always in G99 mode, they are never changed to G98 (Feed per minute). The only time G98 would be used if live tooling is being used.

<u>Meaning of Address 'T'</u>

T function is used for designation of tool numbers and tool compensation.
T function is a tool selection code made of **4** digits.
T 02 02 1st 02 for Designation of tool number
2nd 02 for Designation of tool compensation number
Example) If it is designated as (T 02 02)
The first 02 calls the tool number and Second 02 calls the tool compensation value of number, and the tool is compensation as much as memorized value in the storage.
The cancel of tool compensation is commanded as T 02 00
If you want to call the next tool and compensation, you should cancel the tool compensation. For convenient operation, it is recommended to used the same number of tool and compensation. It is not allowed to use the same tool compensation number for 2 different tools. Minimum compensation value : + 0.001mm
Maximum compensation value : + 999.999mm
Tool compensation of X spindle is designated as diameter value.



BASIC MOTION COMMANDS

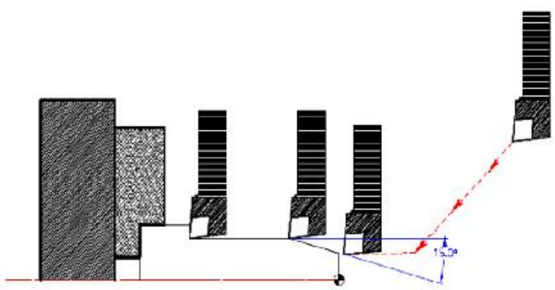
G00 - Rapid Traverse

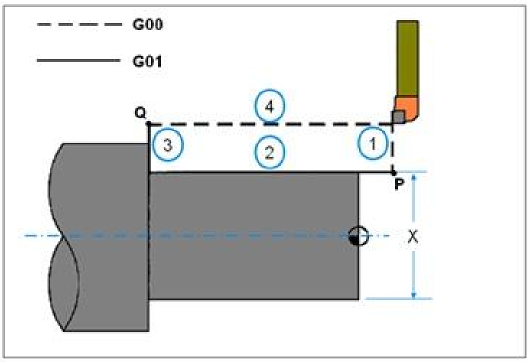
When the tool is moving to a position preparatory to executing a cutting motion or when it is moving to the tool change position, the motion is essentially a waste of time and is executed as fast as possible. The motion is called Rapid traverse. The time taken to execute a rapid motion is also called the Air cut time or non cutting movement.

Format: G00 X_ Z_ (X, Z are the destination coordinates)

G01 - Linear interpolation

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the feed rate.





Format: G01 X__ Z__ F__(X, Z are the destination coordinates F is feed rate,) G01 X20.0 Z-10.0 F0.2 ; X40.0 Z-15.0

G01 need not be repeated in the second line because it is a 'modal command'.

G02 / G03 - Circular Interpolation

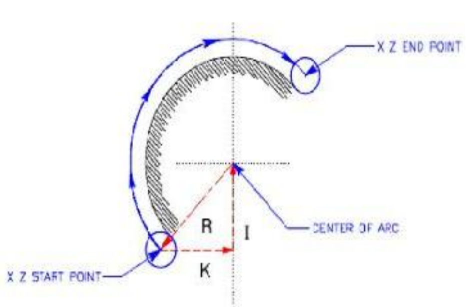
G02 moves along a Clockwise (CW) arc,

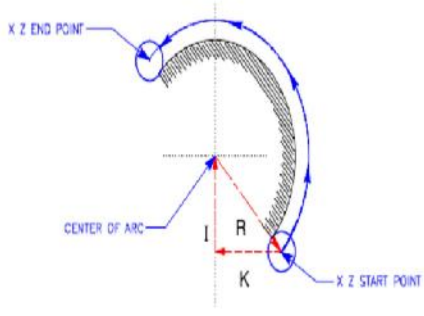
G03 moves along a Counterclockwise (CCW) arc.

G02/G03 X__ Z__ R__ F__

X, Z are the destination coordinates

R is the arc radius, F is the feed rate.





G02- CW Circular Interpolation Motion G03- CCW Circular Interpolation Motion

Circular Interpolation Format

G02 and G03 (Circular Interpolation Clockwise/Counterclockwise) using “I” and “K”. in place of “R” (radius)

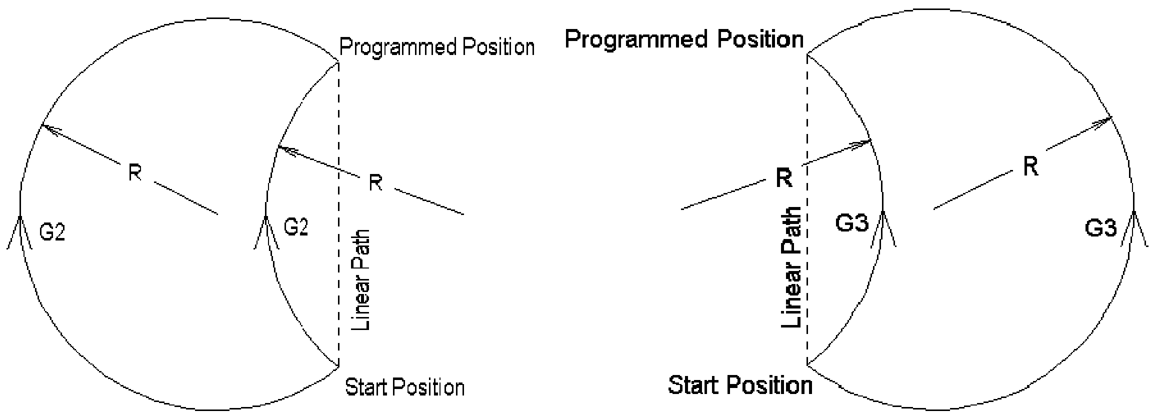
G02 / G03 X__ Y__ I__ K__F__;

It is sometimes easier to program the “R” (radius) word in place of the “I” and “K” words. The “I” and “K” words are used to define the incremental distance and direction from START POINT to the ARC CENTER, which can also be done by using the “R” word in place of “I” and “K”.

‘I’ and **‘K’** are the relative distance from the arc start point to the arc center.

I = X coord. of start point - X coord. of center

K = Z coord. of start point - Z coord. of center



NOTE:

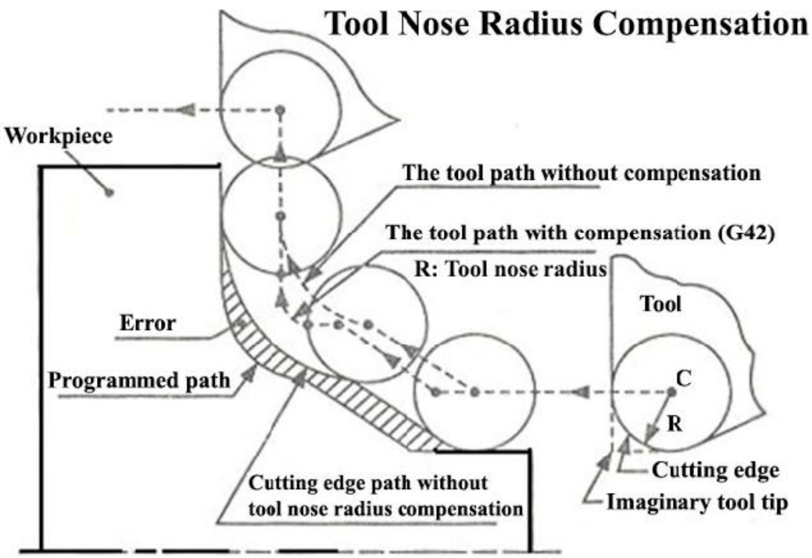
Any radius being cut using the “R” word needs a positive (+) value for a radius of 180 degrees or less. For a radius larger than 180 degrees, it needs to have a minus (-) sign with a radius value.

G04 - Dwell

A dwell command results in a temporary stoppage of all axis motions for a specified duration. The spindle motion is not affected. It is typically used when the tool has reached the final position in an operation and needs to stay there for a few spindle rotations to obtain good dimensional accuracy or surface finish. For example, in a grooving operation when the tool reaches the bottom of the groove it needs to stay there for at least one full revolution. Without a dwell it would retract back instantaneously and result in a non-circular cross section at the groove bottom.

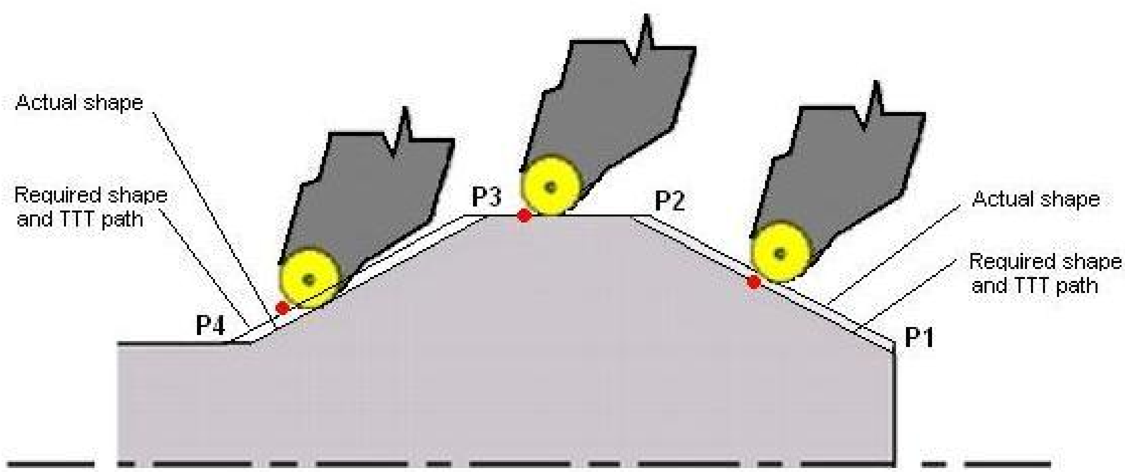
Format: G04 X_ (X is the dwell time in seconds.)

Example: G04 X1.0 (This results in a dwell of 1 second.)



TOOL NOSE RADIUS COMPENSATION (TNRC)

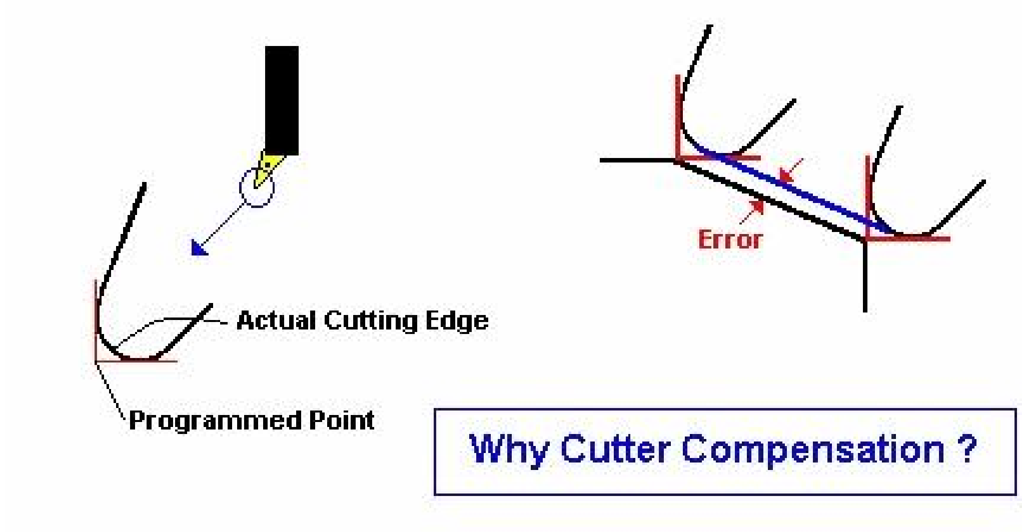
Tool nose radius compensation, or TNRC, is required for generating accurate profiles. When you command the tool to move to a position, you are actually commanding the **Theoretical Tool Tip (TTT)** to move to the position. When doing an operation like contour turning, you just program the contour according to the coordinates in the part drawing. This causes the TTT point moves along the commanded path.



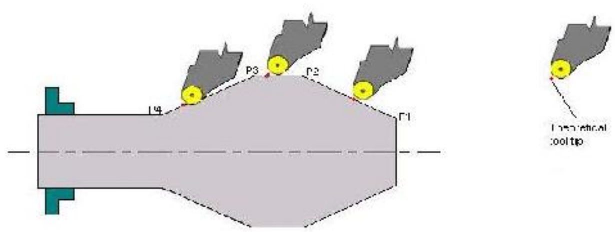
TTT moving along Contour

This is the point on the tool that is used as the reference point for determining tool offsets.

Necessity of TNRC.

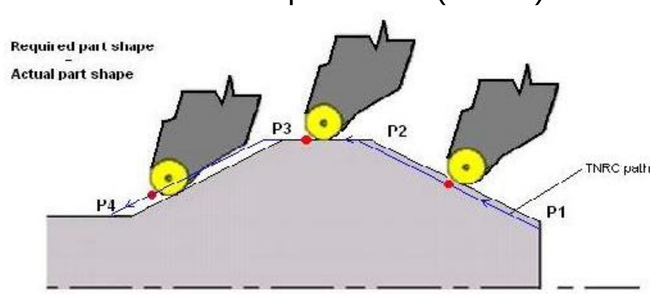


As the tool moves along the programmed contour, the point on the tool nose radius that is actually doing the cutting keeps changing. We actually need the nose radius to be tangential to the part contour at the point where it is cutting, but moving the Theoretical Tool Tip (**TTT**) along the contour does not ensure this. As a result, the tool leaves unmachined material in some areas (P1 to P2 in picture) and digs into the material in some areas (P3 to P4 in picture).



Tool path without TNRC

To get an accurate contour during machining, an alternate tool path is generated such that the nose radius is tangential to the contour. This is the path with Tool Nose Radius Compensation (TNRC).

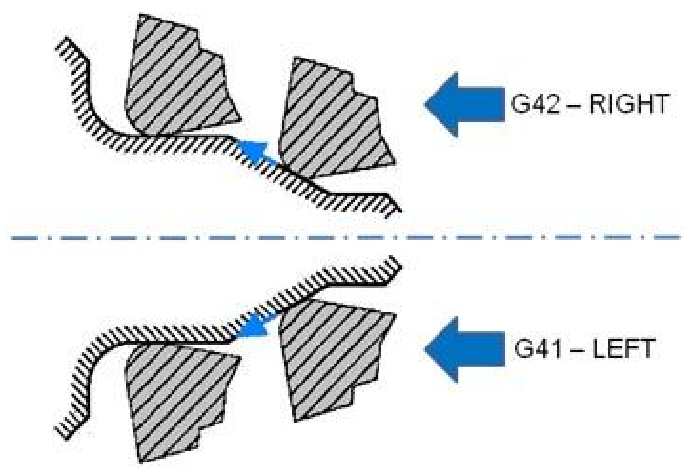


Compensated tool path

Compensation commands.

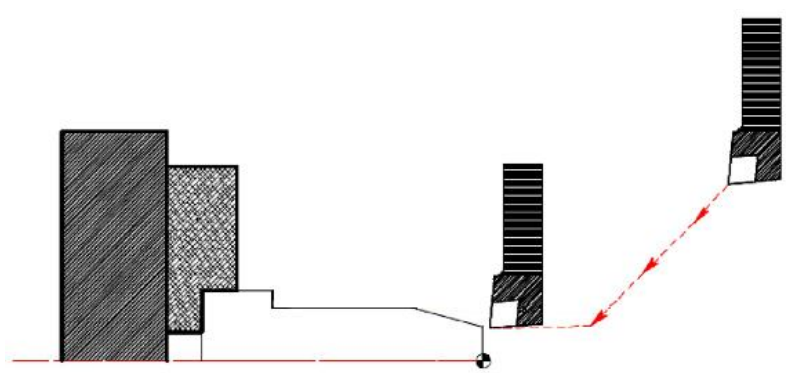
The compensated tool path must be either to the left or the right of the tool path programmed with the coordinates from the part drawing. The direction of compensation depends on the direction of motion and whether the tool is cutting on the inside or outside of the part. In the program you can specify whether the compensation must be to the left or right, and the controller determines the compensated tool path. The tool nose radius too must be specified in separate area of the memory. The commands are:

- G41 Tool nose radius compensation Left
- G42 Tool nose radius compensation Right
- G40 Tool nose radius compensation Cancel



TNRC Left and Right

<u>HOMING IN CNC LATHE</u>



To bring the turret or tool position to machine reference point is known as HOMING. Where X=0 and Z=0. There are 4 procedures for homing.

(A) In ZRO RETURN mode there are two procedures, without help of program i.e. manually.

(B) In MDI mode two procedures are there i.e. with the help of program.

First Procedure for Homing:

Disturb the both axis (x first then z) in –ve direction, then keep the mode selector at ZRN mode, keep the rapid at zero, then press the cycle start button. Then give rapid maximum.

Second Procedure for Homing:

Disturb the both axis (x first then z) in –ve direction, then keep the mode selector at ZRN mode, keep the rapid at zero, then press X+ & Z+ on the JOG table. Then give rapid maximum.

Third Procedure for Homing:

Disturb the both axis (x first then z) in –ve direction, go to MDI mode. Then go to PRG window and write- T0000;

G00 X0 Z0;

Keep the rapid zero, press position switch then press CYCLE start switch, and give the rapid maximum.

Fourth Procedure for Homing:

Disturb the both axis (x first then z) in –ve direction, go to MDI mode. Then go to PRG window and write- T0000;

G28 U0 W0;

Keep the rapid zero, press position switch then press CYCLE start switch, and give the rapid maximum.

Test Program in Dry Run mode.

In this mode the program runs with high feed i.e. 20 m/min. At this time there must be safety distance from the top surface of job. This option will be available from soft menu of the control system.

Single Block Run:

Once Simulation in Graphic mode is satisfactory. Block by Block mode to be tested for the program on Job. This Block by Block mode is used only for checking program error (approximately). Tool movement at a safety distance is carried out in this mode of running programme.

Full sequence Run with Safety distance:

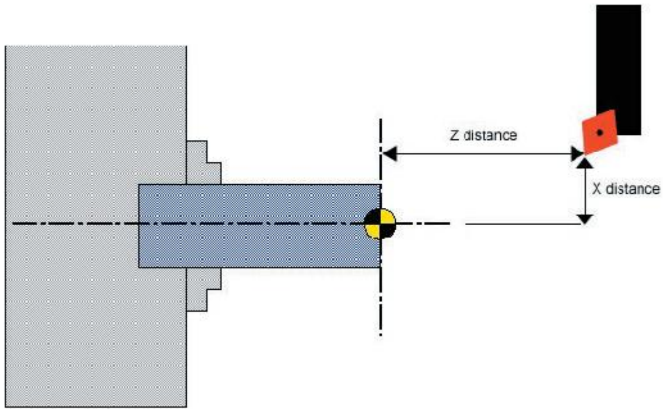
Once the programme testing in a single Block mode is satisfactory then run the program in full sequence mode with full speed & feed. This is actual machining mode in which we will get the finished job. But for safety purpose we can run the program with very less depth and check it and then we can run to complete the job.

Full sequence with full depth:

Once the previous mode is tested the program to be edited with required depth as per the drawing will be run for final operation.

OFFSET SETTING

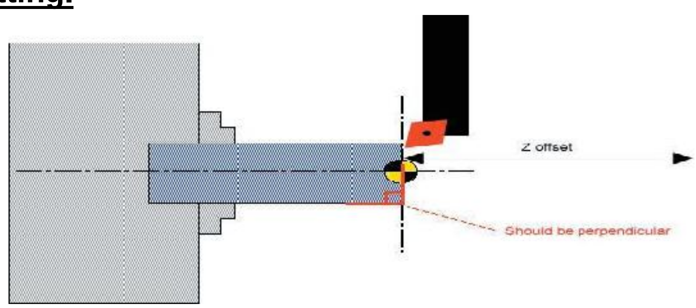
The procedure by which the job reference point or job data is given to the machine is known as offsetting **OR** Offsets are the distances the cutting tool needs to travel, from its 'Home' position to the Workpiece datum in X & Z.



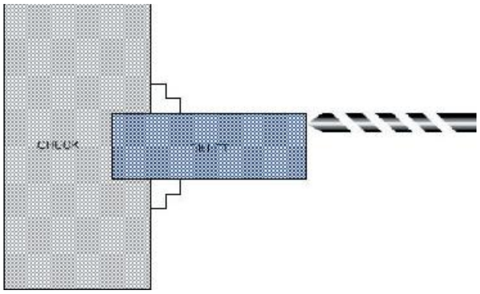
There are two types of offsetting is done in CNC TURNING.

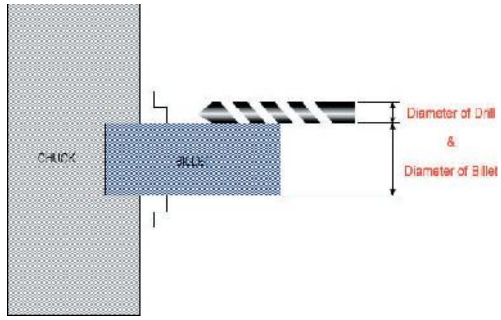
1. Z offsetting or length offset.
2. X offsetting or dia offset.

Z Offsetting.



Manually bring the tool near to the job, rotate the spindle, touch the face of job by providing small step by help of MPG and clean the face of job by bring the tool along x axis. Then go to OFFSET window and put Z0.0, then press the soft key MEASURE. For example: if you are taking tool no 5, then put Z0.0 at serial easy to use. For drill bit touch the tip of the drill on the face of the job.

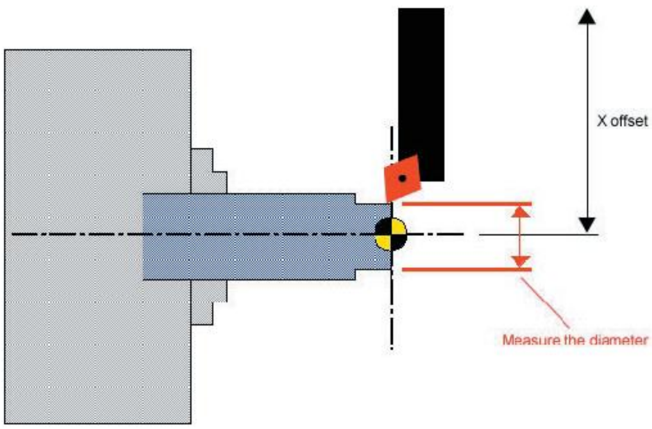




(FOR DRILL)

X Offsetting.

After completing Z offset again bring the tool over the job slightly clean the outer dia of job and stop the spindle. Measure the OD of job by measuring instrument, then go the OFFSET window put the exact value at the same serial no ,then press MEASURE.



For drill bit touch the drill bit to the billet outer diameter, and then enter the sum of the diameter of the billet + the diameter of the drill.

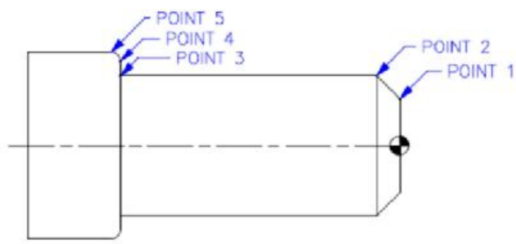
Tool geometry offset Table

Geome - try offset number	OFGX (X–axis geometry offset amount)	OFGZ (Z–axis geometry offset amount)	OFGR (Tool nose radius ge-ometry	OFT (Imaginary tool nose direction)	OFGY (Y–axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	:	:	:	:	:
G05	:	:	:	:	:
:	:	:	:	:	:

Tool wear offset Table

Wear offset number	OFGX (X–axis wear offset amount)	OFGZ (Z–axis wear offset amount)	OFGR (Tool nose radius wear	OFT (Imaginary tool nose direction	OFGY (Y–axis wear offset amount)
W01	0.040	0.020	0	1	0.010
W02	0.060	0.030	0	2	0.020
W03	0	0	0.20	6	0
W04	:	:	:	:	:
W05	:	:	:	:	:
:	:	:	:	:	:

NOTE** For facing, turning, threading, knurling and grooving the offsetting procedure is same. But in drilling the spindle rotation is not required and in boring for X offset you have to clean the inside surface of job and take the internal diameter (ID) measurement for offsetting.



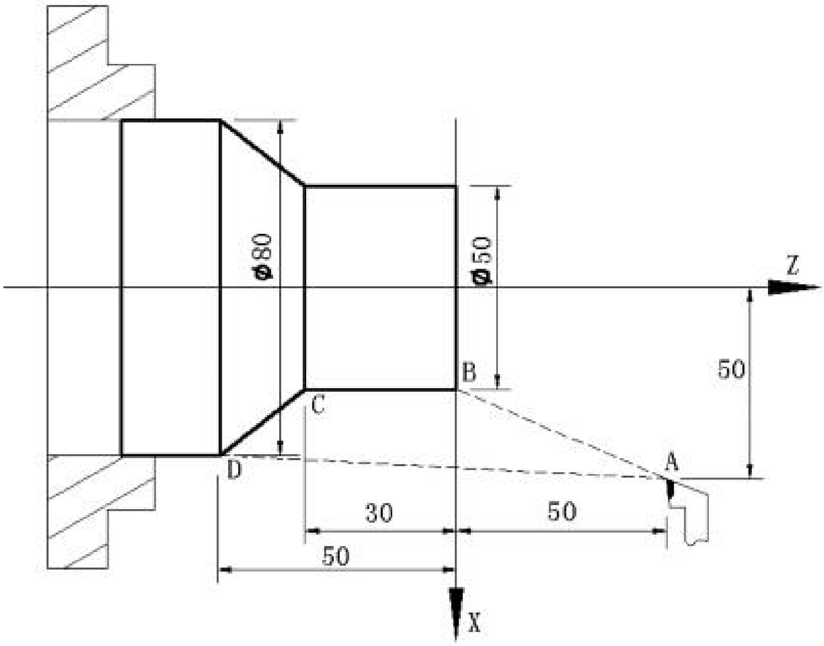
Give the X and Z coordinates for the part below. Note the X values are diameters on CNC Lathes not radii.

	X	Z
Point 1		
Point 2		
Point 3		
Point 4		
Point 5		

The Sequential steps for to make a Part Programme.

01. Program Head (Program name or number & May keep some other Information's for future use).
02. Select Tool change position.
03. Tool change (Manual or Automatic Tool Changer).
04. Select operational position.
05. Cutting parameter setting (Cutting speed, Feed etc.).
06. Operation.
07. On completion of total operation of that tool, then return to tool change position.
08. If there is any other operation, then again proceed from No. 3 to No.7, If not then program End.

EXAMPLE FOR LINE PROGRAMMING



O0001;

```
G28 U0 W0 T0;
T0101; (Tu. cum Facing Tool)
G00 X100 Z50;
G96 S150;
G50 S1500;
G97 F.2;
G00 X85 Z5;
M03 / M04;
(Facing Operation)
G00 Z0;
G01 X0;
G00 Z2;
G00 X85;\
(Turing Operation)
G00 X81;
G01 Z-60;
G01 X85;
G00 Z2;
G00 X76.5;
G01 Z-47;
G01 X82;
G00 Z2;
G00 X72;
G01 Z-44;
G01 X80;
G00 Z2;
G00 X67.5;
G01 Z-41;
G01 X75;
G00 Z2;
G00 X63;
G01 Z-38;
G01 X70;
G00 Z2;
G00 X58.5;
G01 Z-35;
G01 X65;
G00 Z2;
G00 X54;
G01 Z-32;
G01 X60;
G00 Z2;
G00 X51;
G01 Z-30;
G01 X55;
G00 Z2; (Finish Cut)
G00 X0;
G01 Z0;
G01 X50;
G01 Z-30;
G01 X80 Z-50;
G01 Z-60;
G01 X85;
G00 X150 Z150;
M30;
```

CANNED CYCLES

Canned cycle or fixed cycles may be defined as a set of instructions, inbuilt or stored in the system memory, to perform a fixed sequence of operations. The canned cycles can be brought into action with a single command and such reduce the program length, time and effort. It is used for repetitive and commonly used machining operations.
Commonly available fixed cycles are- The operations Drilling, Tapping, Boring, Pocketing , etc. are in milling. And The operations Turning, Facing, Drilling, Grooving, Threading, OD/ID Rough Out, Contour Finishing, etc. are in lathe.

Machine Cycles for Turning and Grooving

The following is a list of the canned cycles that can be used for turning and grooving for the
Fanuc lathe controls.

G90 O.D./I.D. Turning Cycle Modal
G92 Thread Cutting Cycle Modal
G94 End Face Cutting Cycle Modal
G70 Finishing Cycle
G71 O.D./I.D. Stock Removal cycle
G72 End Face Stock Removal Cycle
G73 Irregular Path Stock Removal Cycle (Contour Parallel Turning)
G74 End Face Grooving or Turning with Chip Break Cycle
G75 O.D./I.D. Grooving or Turning with Chip Break cycle
G76 Thread Cutting Cycle, Multiple Pass

A machine cycle is used to simplify programming of a part. Machine cycles are defined for the most common, repetitive operations such as turning, facing, threading and grooving. There are both modal and non-modal machine cycles. Modal cycles, such as turning cycle G90, remain in effect after they are defined. After any subsequent X or Z-axis positioning, the canned cycle is executed again. Modal canned cycles remain in effect until canceled by a G80, G00, end of program, or RESET. Non-modal machine cycles are effective only for the block that contains them, but will perform a series of machining moves to perform that command.

Fanuc G90 Turning Cycle

Fanuc G90 turning cycle is used for simple turning however multiple passes are possible by specifying the X-axis location of additional passes. Called with many names like – Fanuc G90 Outer Diameter/Internal Diameter Cutting Cycle or G90 Straight cutting cycle

G90- Programming Format

G90 X—Z--; OR G90 U—W--;

X – Diameter to be cut. Z – End point in Z-axis.

U – X-axis incremental distance. W – Z-axis incremental distance.

Fanuc G94 Facing Cycle

Fanuc G94 G code is used for rough facing.

It is used for simple facing (one-pass facing) however multiple passes are possible by specifying the Z-axis location of additional passes.

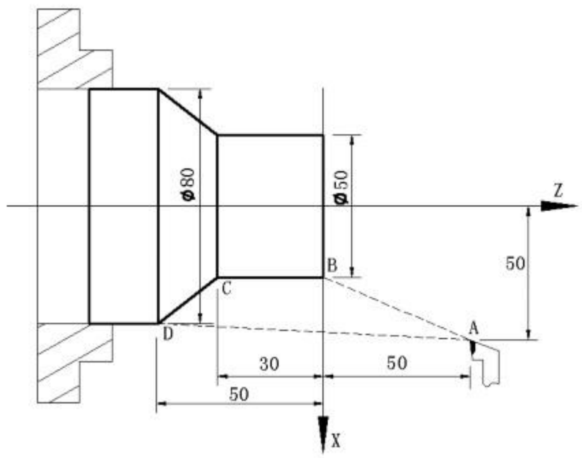
G94- Programming Format

Fanuc G94 facing cycle is very simple to program and use. G94 G code parameters are explained below,

G94 X… Z… OR G94 U……W…..

X: End point in X-axis. Z: End point in Z-axis.

U: X-axis incremental distance. W: Z-axis incremental distance.



O0001;	G94 Z2;	G00 X0;
G28 U0 W0 T0;	Z1;	G01 Z0;
T0101; (Turning cum Facing Tool)	Z0;	X50;
G00 X100 Z50;	G90 X81;	Z-30;
G96 S150;	X76.5;	X80 Z-50;
G50 S1200;	X72;	Z-60;
G97 F.2;	X67.5;	G00 X150 Z150;
G00 X85 Z5;	X63;	M30;
M03 / M04;	X58.5;	
	X54;	
	X51;	

Fanuc G92 Threading Cycle Format

Fanuc G92 threading cycle gives the cnc machinist the flexibility to control every thread pass depth, so the cnc program blocks will be the calculated diameters for each thread pass and cycle cancelled by G00 code.

G92 X__Z__F__;

Where, X = Current diameter of the thread pass

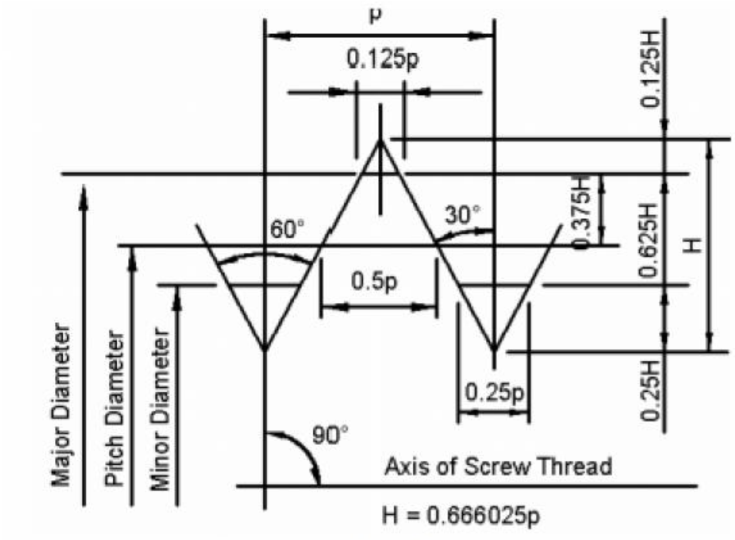
Z = End position of the thread in Z-axis

F = Threading feed rate in in/rev (Lead)

G92 Threading Cycle Format for Taper Threading

G92 X__Z__R__F__;   OR   G92 X__Z__I__F__;

The **R** or **I** parameter in G92 threading cycle is the tapered value.
Note: that R or I is given as Radius value.

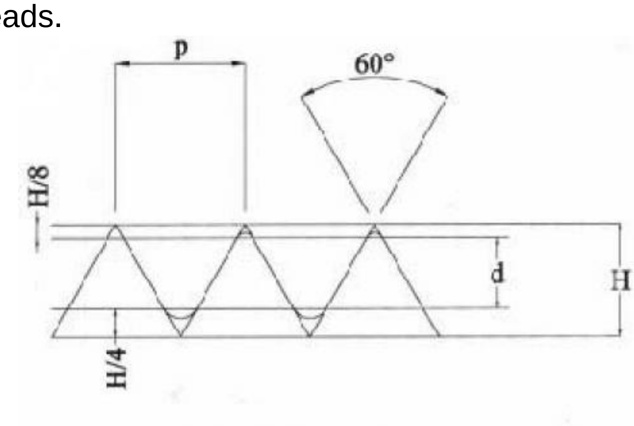


ISO Metric Screw Thread

STANDARD FORMS OF THREAD:

Metric thread:

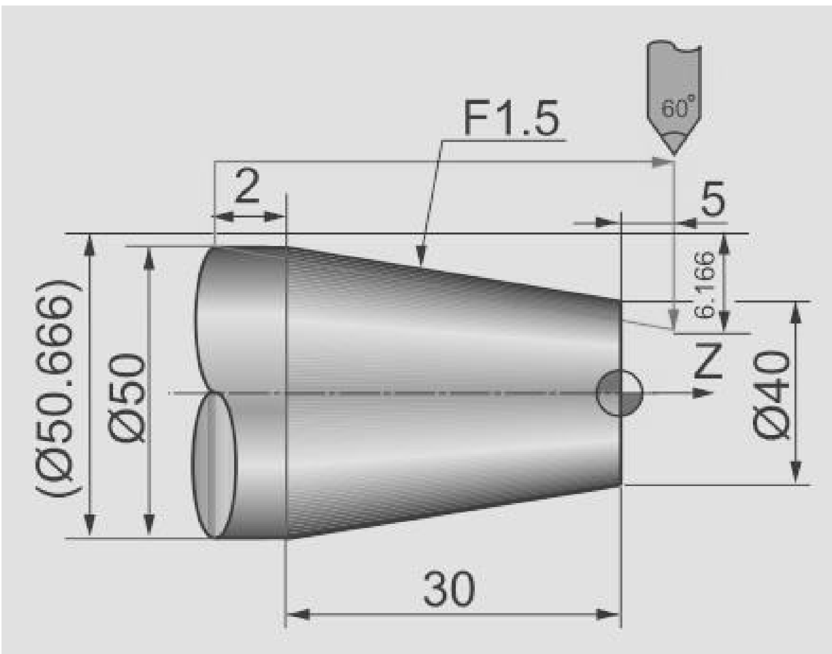
The basic profile of ISO metric screw thread is BIS (Bureau of Indian Standard) has adopted the same thread form as in the practice in several other countries.
The thread is characterized by angle of 60 degree between the flanks & pitch denoted by p.
The theoretical depth, H is related to p as H = 0.866025 p
Certain practical changes are introduced in manufactured threads according to design profiles of threads.



Basic Thread Form
ISO Metric Threads

P = Pitch = 1/Number of threads per inch (TPI)
H = Angular Depth = 0.866025 x P
H/8 = Shortening of major dia = 0.108253 x P
H/4 = Shortening of minor dia = 0.216506 x P
d = Actual Depth = 0.541266 x P
r = Radius at the Root = 0.1443 x P
Hn = Basic height of Internal Thread = 0.54127 x P
Hs = Basic height of External Thread = 0.61344 x P

<u>G92 Threading Cycle Format for Taper Threading</u>




```
T0303;
G00 X52 Z5;
G97 S400 M03;
G92 X49.5 Z-32 I-6.166 F1.5;
       X49.3;
       X49.1;
       X48.9;
       X48.7;
       X48.5;
       X48.3;
       X48.1;
       X48.05;
G00 X100 X150;
```

G70 (FINISHING CYCLE IN TURNING)

G70 P… Q….F… :

P : Start sequence block no of Finished Part.

Q : Final sequence block no of Finished Part..

F: Cutting federate For Finish Cut.

G71 (STOCK REMOVAL CYCLE IN TURNING)

G71 U….. R….. :

G71 P… Q… U… W… F… :

U: Cut volume of one time OR Depth per Cut on X (Designate the radius).

R: Escape volume(Always 45 degree escape) OR Relief Amount.

P : Start sequence block no of Finished Part.

Q : Final sequence block no of Finished Part..

U: Finishing tolerance in X axis or Finishing allowances on X.

W : Finishing tolerance in Z axis or finishing allowances on Z.

F: Cutting federate

G72 (STOCK REMOVAL CYCLE IN FACING)

G72 W…. R…:

G72 P…. Q….U….. W…..F….. :

W: Cut volume of one time OR Depth per Cut on Z Direction.

R: Escape volume(Always 45 degree escape) OR Relief Amount.

P : Start sequence block no of Finished Part.

Q : Final sequence block no of Finished Part..

U: Finishing tolerance in X axis or Finishing allowances on X.

W : Finishing tolerance in Z axis or finishing allowances on Z.

F: Cutting federate

G76 (MULTIPLE THREAD CUTTING CYCLE)

G76 P(m) (r) (a) Q… R…;

G76 X…. Z…. R… P… Q… F…;

P (m) : Repeating time before the final thread OR No. of Dry Run.
(r): Chamfering at the end part of thread. (a): Angle between threads.

Q : Minimum cut volume OR Depth per Cut.

R : Finishing allowance.

X : Core diameter of thread OR Minor dia. of Thread.

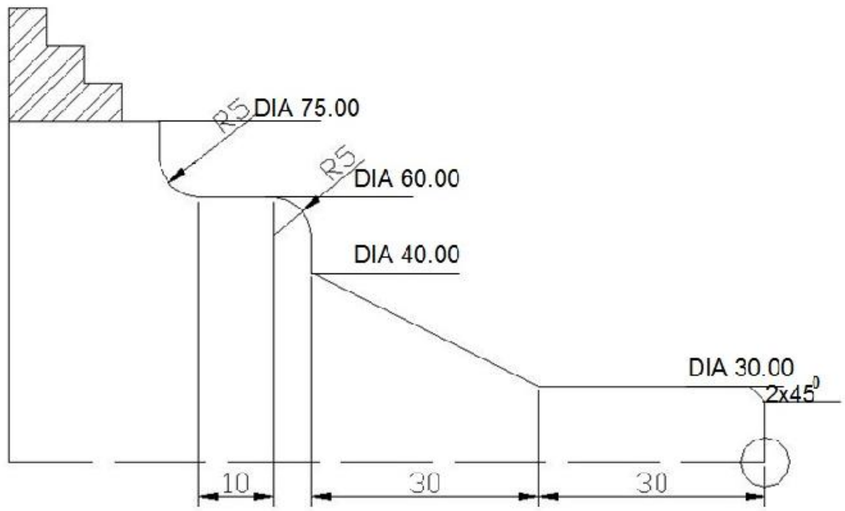
Z : Z--coordinate at the end point of thread process or Thread length.

R : Taper deflection of Taper Thread.

P : Height of thread (Omit the decimal point)(0.65 x Pitch).(Total Thread Depth)

Q: Initial cut volume OR Depth of first Cut. F: Cutting feed rate (Lead).

EXAMPLES G70, G71, G72, G73, G75 AND G76.



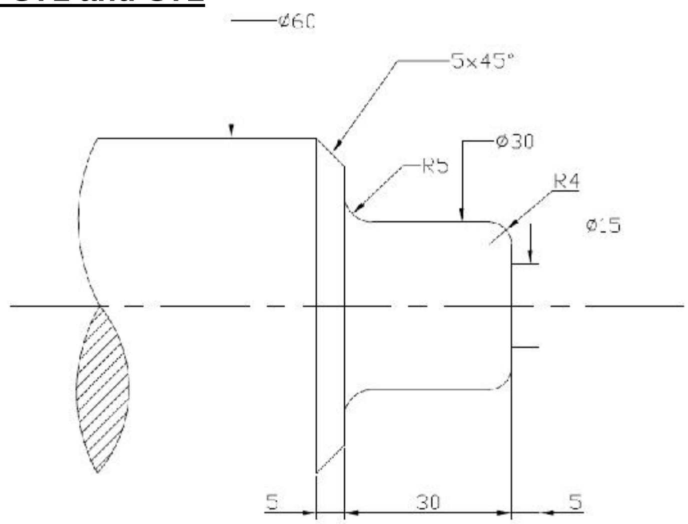
BY TURNING CYCLE

```
O0001;
T0000;
G28 U0 W0;
T0404;
G00 X80.0 Z2.0;
G96 S100;
G50 S1000 M04;
M08;
G71 U0.3 R0.4;
G71 P10 Q20 U0.1 W0.05 F0.2;
N10  G01 X0.0;
G01 Z0.0;
G01 X26.0;
G01 X30.0 Z-2.0;
G01 Z-30.0;
G01 X40.0 Z-60.0;
G01 X50.0;
G03 X60.0 Z-65.0 R5.0;
G01 Z-75.0;
G02 X70.0 Z-80.0 R5.0;
G01 X75.0;
N20 G01 Z-82.0;
G70 P10 Q20 F0.1;
G00 X100.0 Z100.0;
M30;
```

BY FACING CYCLE

```
O0001;
T0000;
G28 U0 W0;
T0404;
G00 X80.0 Z2.0;
G96 S100;
G50 S1000 M04;
M08;
G72 W0.3 R0.4;
G72 P10 Q20 U0.1 W0.05 F0.2;
N10  G01 Z-82.0;
G01 X75.0;
G01 Z-80.0;
G01 X70.0 ;
G03 X60.0 Z-75.0 R5.0;
G01 Z-65.0 ;
G02 X50.0 Z-60.0 R5.0;
G01 X40.0;
G01 X30.0 Z-30.0;
G01 Z-2.0;
G01 X26.0 Z0;
N20 G01 X0.0;
G70 P10 Q20 F0.1;
G00 X100.0 Z100.0;
M30;
```

EXAMPLE G71 and G72



BY TURNING CYCLE	**BY FACING CYCLE**
O1111;	O1111;
T0000;	T0000;
G28 U0 W0;	G28 U0 W0;
T0404;	T0404;
G00 X62.0 Z2.0;	G00 X62.0 Z2.0;
G96 S100;	G96 S100;
G50 S1000 M04;	G50 S1000 M04;
M08;	M08;
G71 U0.3 R0.4;	G72 W0.3 R0.4;
G71 P10 Q20 U0.1 W0.05 F0.2;	G72 P10 Q20 U0.1 W0.05 F0.2;
N10 G01 X0.0;	N10 G01 Z-50.0;
G01 Z0.0;	G01 X60.0;
G01 X15.0;	G01 Z-40.0;
G01 Z-5.0;	G01 X50.0 Z-35.0;
G01 X22.0;	G01 X40.0;
G03 X30.0 Z-9.0 R4.0;	G03 X30.0 Z-30.0 R5.0;
G01 Z-30.0;	G01 Z-9.0;
G02 X40.0 Z-35.0 R5.0;	G02 X22.0 Z-5.0 R4.0;
G01 X50.0;	G01 X15.0;
G01 X60.0 Z-40.0;	G01 Z0;
G01 Z-50.0;	G01 X0;
N20 G01 X62.0;	N20 G01 Z2.0;
G70 P10 Q20 F0.1;	G70 P10 Q20 F0.1;
G00 X100.0 Z100.0;	G00 X100.0 Z100.0;
M30;	M30;

G73 (STOCK REMOVAL BY PATTERN REPEATING CYCLE)

G73 U….W…. R…:

G73 P…. Q….U….. W…..F….. :

First Program Block

U : Distance and direction of relief in x-axis (radius value). This is the amount of material which will be cut in x-axis.

W : Distance and direction of relief in z-axis. This is the amount of material which will be cut in z-axis.

R : Number of divisions. The number the contour will be repeated.

Second Program Block.

P : The contour start block number.

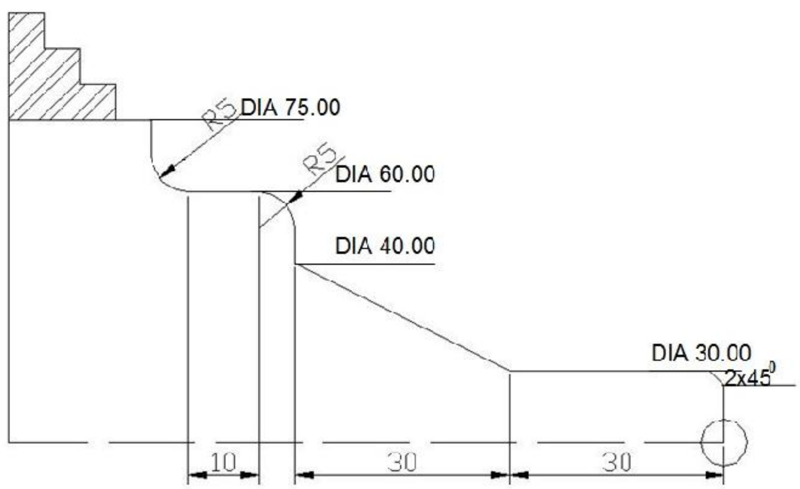
Q : The contour end block number.

U : Finishing allowance in x-axis.

W: Finishing allowance in z-axis.

F : Feed

EXAMPLE

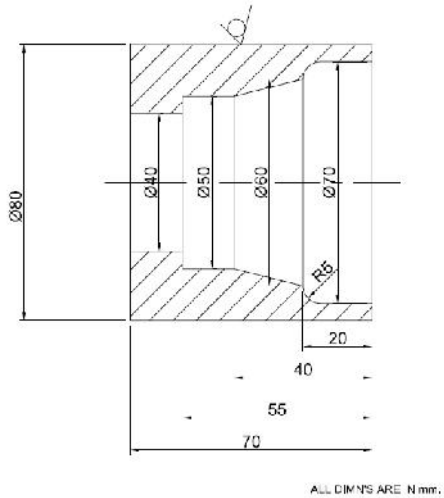


PATTERN REPEATING CYCLE

O4321;

```
G28 U0 W0;
T0;
T0404;
G00 X80.0 Z2.0;
G96 S100;
G50 S1000 M04;
M08;
G73 U3.0 W3.0 R5;
G73 P10 Q20 U0.1 W0.05 F0.2;
N10  G01 X0.0;
G01 Z0.0;
G01 X26.0;
G01 X30.0 Z-2.0;
G01 Z-30.0;
G01 X40.0 Z-60.0;
G01 X50.0;
G03 X60.0 Z-65.0 R5.0;
G01 Z-75.0;
G02 X70.0 Z-80.0 R5.0;
G01 X75.0;
N20 G01 Z-82.0;
G70 P10 Q20 F0.1;
G00 X100.0;
G00 Z100.0;
M30;
```

<u>Internal Turning (Boring)</u> Example:-



O4321;
G28 U0 W0;
T0;
T0303 (**Drill dia. 20mm**);
G00 X0.0 Z2.0;
G97 S600;
M03;
G74 R5.0;
G74 Z75.0 Q5000 F0.05;
G00 X150.0 Z150.0;
G28 U0 W0;
T0404 (**Int. Turning Tool**);
G00 X20.0 Z2.0;
G96 S100;
G50 S1200 M04;

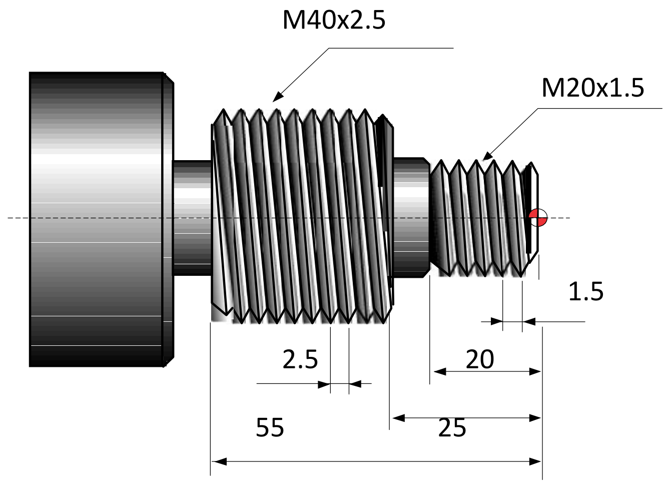
G71 U1.0 R0.5
G71 P10 Q20 U-0.2 W0.05 F0.2;
N10 G01 X70.0;
G01 Z-15.0;
G03 X60.0 Z-20.0 R5.0;
G01 X50.0 Z-40.0;
G01 Z-55.0;
G01 X40.0;
N20 G01 Z-72.0;
G70 P10 Q20 F0.1;
G00 X200 Z200; M30;

;

<u>OR</u> (BY FACING CYCLE)
G28 U0 W0;
T0;
T0505 (**Int. Facing Type Tool**);
G00 X18.0 Z2.0;
G96 S100;
G50 S1200 M04;
M08;
G72 W1.0 R0.5;
G72 P10 Q20 U-0.2 W0.05 F0.2;
N10 G01 Z-72.0;
G01 X40.0;

G01 Z-55.0;
G01 X50.0;
G01 Z-40;
G01 X60.0 Z-20.0 ;
G02 X70.0 Z-15.0 R5.0;
N20 G01 Z2.0;
G70 P10 Q20;
M30;

PROGRAM {Multiple Threading Cycle}



Grooving Cycle

O0010;
G28 U0;
G28 W0;
T0101;
G00 X51 Z2;
G96 S100;
G50 S1000 M04;
M08;
G71 U0.3 R0.4;
G71 P10 Q20 U0.1 W0.05 F0.2;
N10 G00 X0;
G01 Z0;
G01 X16;
G01 X20 Z-2;
G01 Z-25;
G01 X36;
G01 X40 Z-27;
G01 Z-60;
G01 X46;
G01 X50 Z-62;
N20 G01 X51;
G70 P10 Q20 F0.1;
G97 S0;
M05;
M09;

G75 R……….
G75 X….Z….Q…P….F….
R- Relief amount.
X- Minor Dia. Of Groove.
Z- Final Groove Length.
Q- Sifting amount in Z-axis.
P- Depth per Cut, F- Feed.
T0303; (Grooving Tool)
G00 X41 Z-60;
G97 S400 M04;
M08;
G75 R0.5;
G75 X20 Z-58 Q1000 P500 F0.05;
M05;
M09;
G28 U0 W0;
T0505; (THREADING TOOL 60°)
G00 X22.0 Z2.0;
G97 S800 M04;
G76 P021060 Q100 R100;
G76 X18.05 Z-20.0 P975 Q500 F1.5;
G00 X42.0 Z-22.0;
G76 P021060 Q100 R100;
G76 X36.75 Z-57.0 P1625 Q500 F2.5;
G00 X200.0 Z200.0; M30;

<u>G74 = Face Grooving Cycle / Drilling Cycle.</u>

G74 R__ ;

G74 X__ Z__ P__ Q__ F__;

R = Retraction value.

X = Minor diameter (Groove end position in x-axis).

Z = Groove Length (End Position in z-axis).

Q = Peck increment in Z-axis (depth of each cut in Z-axis in micron).

P = Stepping (Sleepover) in X-axis (in Micron).

F = Feed Rate.

<u>G75 = Grooving Cycle.</u>

G75 R__;

G75 X__ Z__ P__ Q__F__;

R= Return/Retract amount,

X= Minor Dimeter,

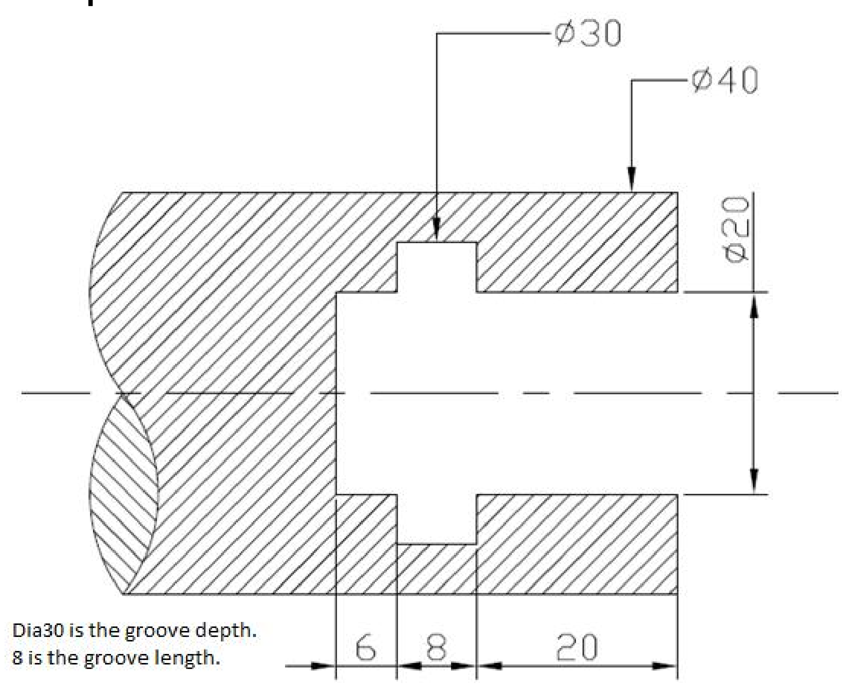
Z= Last groove position in Z- axis,

P= Peck increment in X- axis (in micron),

Q= Stepping in Z-axis (in micron),

F= Feed,

Example:-



O2222;

G28 U0 W0;
T0101; (**Borning Tool**)
G96 S120;

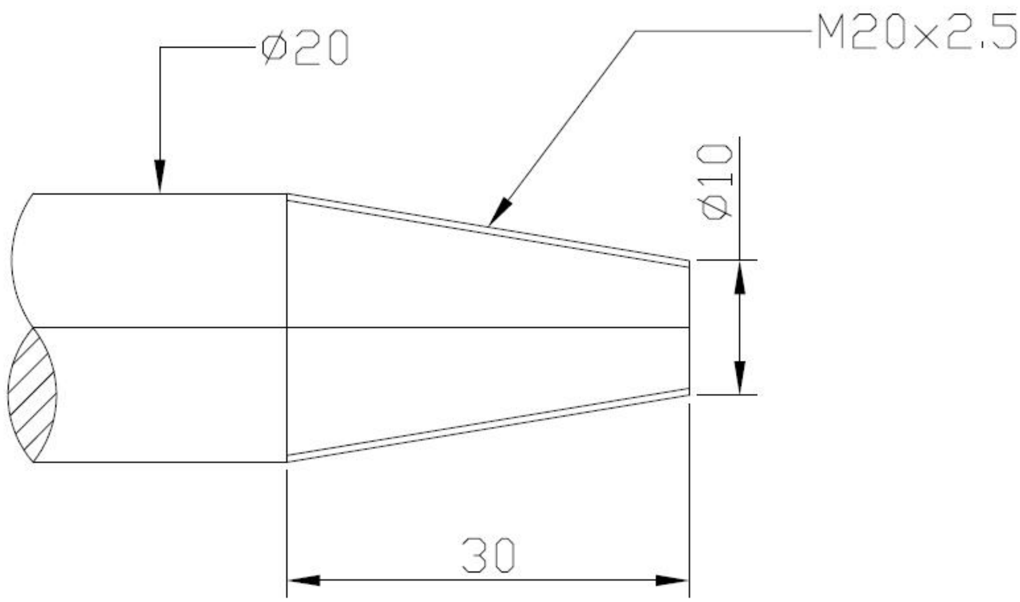
G28 U0 W0;
T0202; (**Grooving Tool 5mm**)
G96 S60;

G92 S1200 ;
G00 Z2.0 X16.0;
M04;
G71 U0.2 R0.3;
G71 P10 Q20 U0.1 W0.1 F0.2;
N10 G01 X20.0;
G01 Z-34.0;
N20 G01 X16.0;
G70 P10 Q20 F0.1;
G00 X150.0 Z200.0;
M30;

G92 S300 ;
G00 X16.0 Z2.0;
M04;
G00 X16.0 Z-28.0;
G75 R0.5;
G75 X30.0 P500 F0.02;
G00 X16.0 Z-25.0;
G75 R0.5;
G75 X30.0 P500 F0.02;
G00 X16.0 Z2.0;
M30:

EXAMPLE

EXTERNAL TAPER THREAD WITH G76



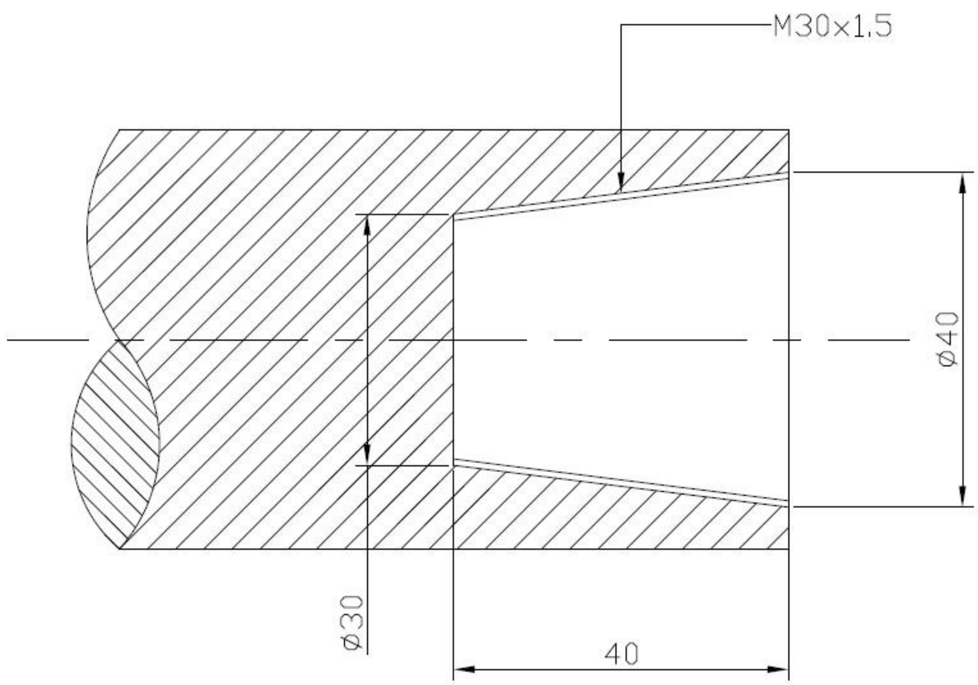
O6644;
T0000;
G28 U0 W0;
T0707;
G97 S1000 M04;
G00 Z0.0;
G00 X20.0;
M07;
G76 P040060 Q20 R0.02;
G76 X16.75 Z-30.0 P1625 Q20 R-5.0 F2.5;

G00 Z2.0;
G97 T0000 M09;
G28 U0 W0;
M30;

Taper turning must be done with correct calculation for the external threading to get a proper taper thread.
The R parameter in second block of G76 is the tapered value. Note that R is given as Radius value and to calculate R parameter for Tapered Threading is flowing-
R = (Start Diameter – End Diameter) / 2

<u>EXAMPLE</u>

<u>INTERNAL TAPER THREAD WITH G76</u>

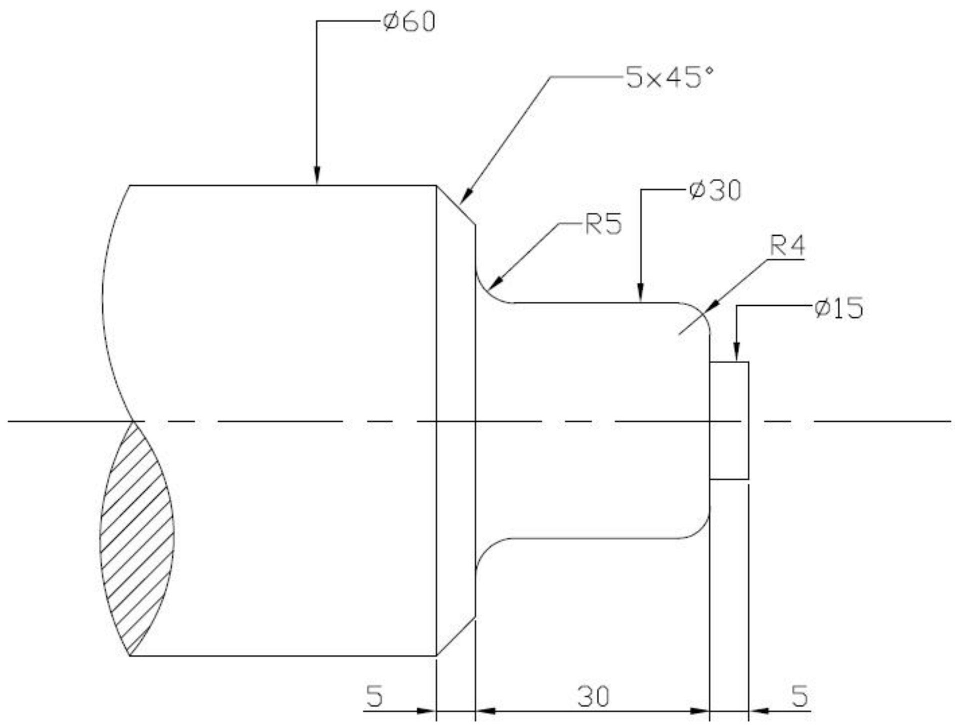


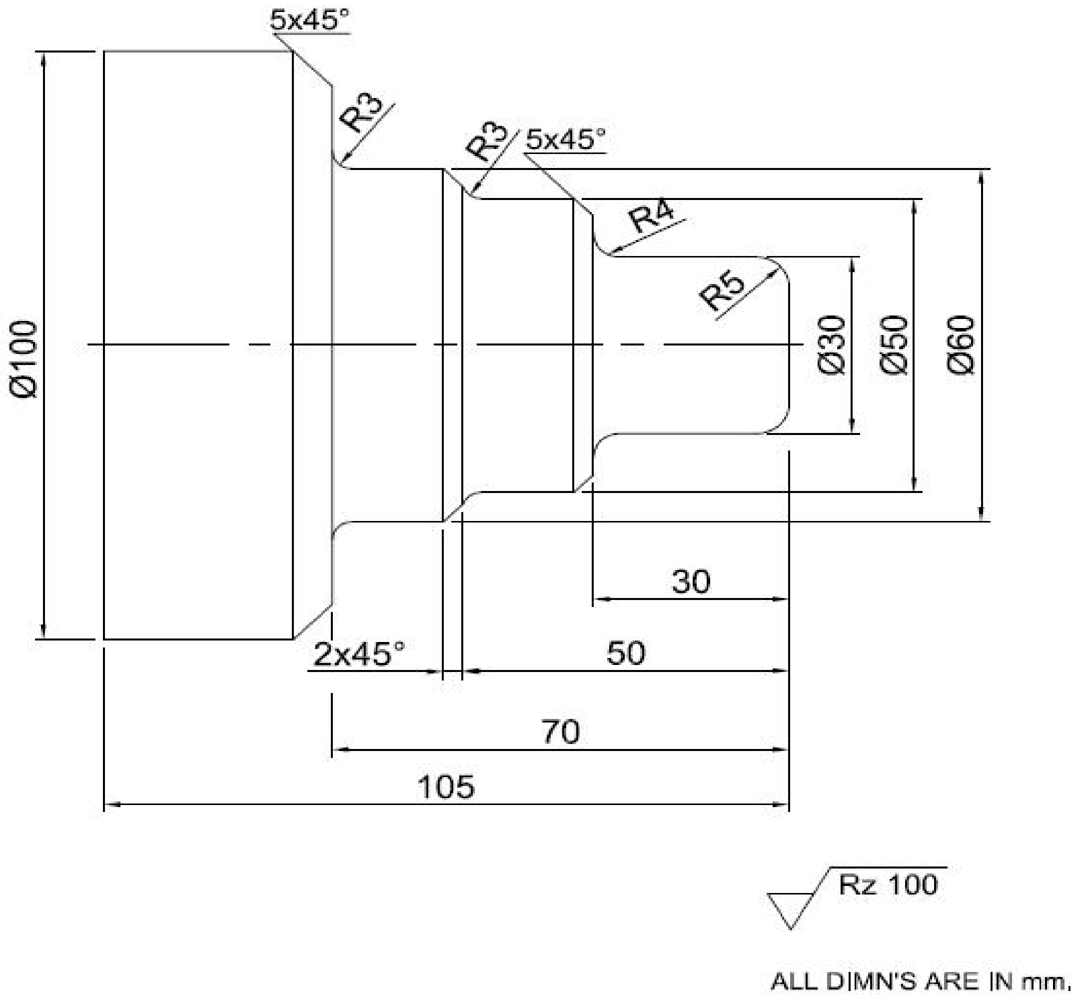
O6644;
T0000;
G28 U0 W0;
T0707;
G97 S500 M04;
G00 Z2.0;
G00 X28.0;
G01 Z0.0 F0.5; M07;

```
G76 P040060 Q20 R0.02;
G76 X30.0 Z-40.0 P975 Q20 R5.0 F1.5;
G00 Z2.0;
G97 T0000 M09;
G28 U0 W00
 M30;
```

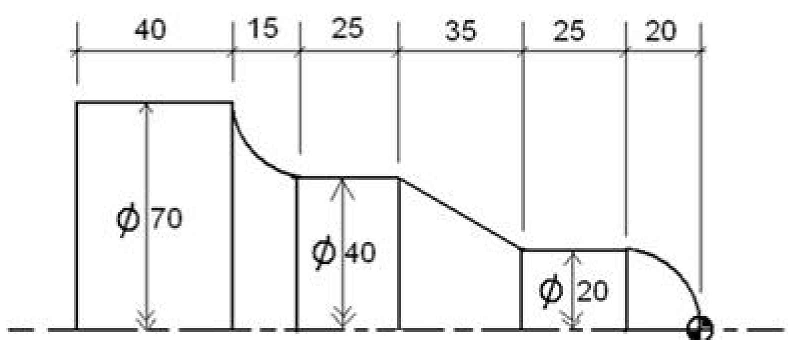
Boring done for internal taper threading must be perfectly calculated or else the thread would not form properly. Safety height also must be calculated properly or there would be a chance of hitting tool with the small diameter of the job.

EXERCISE

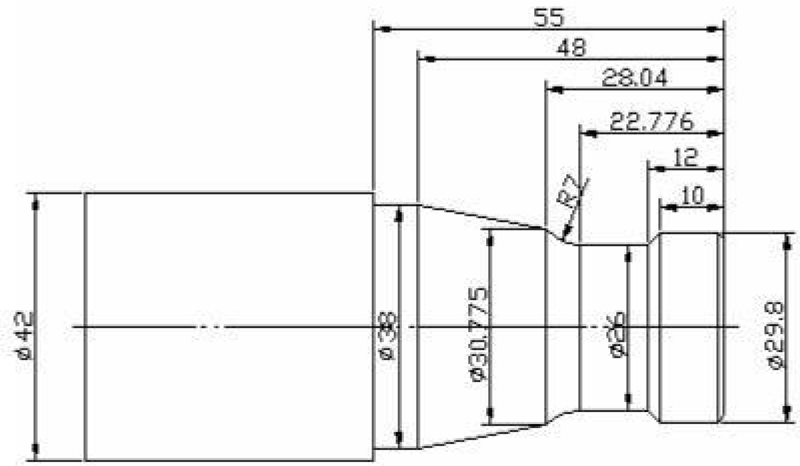


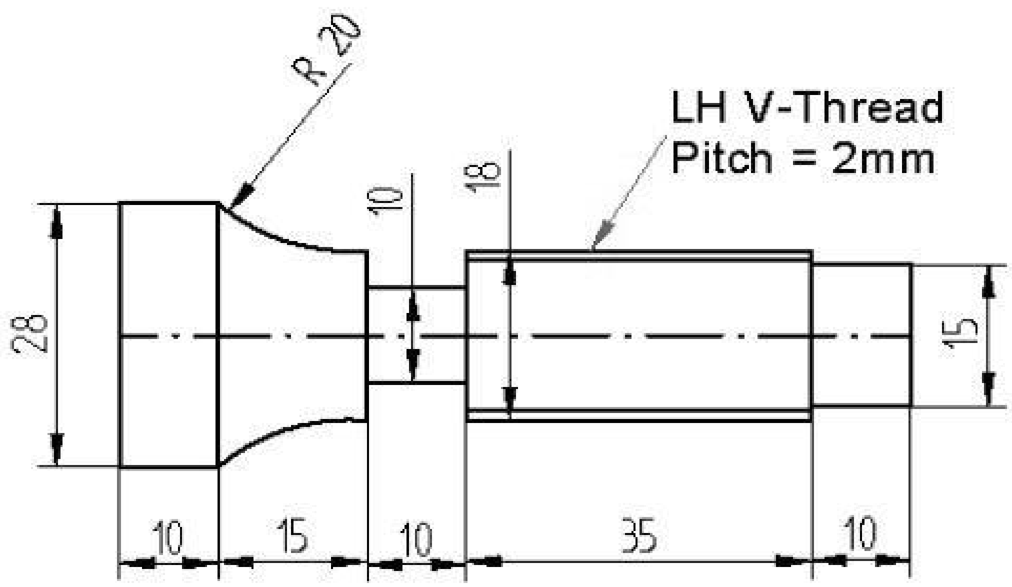


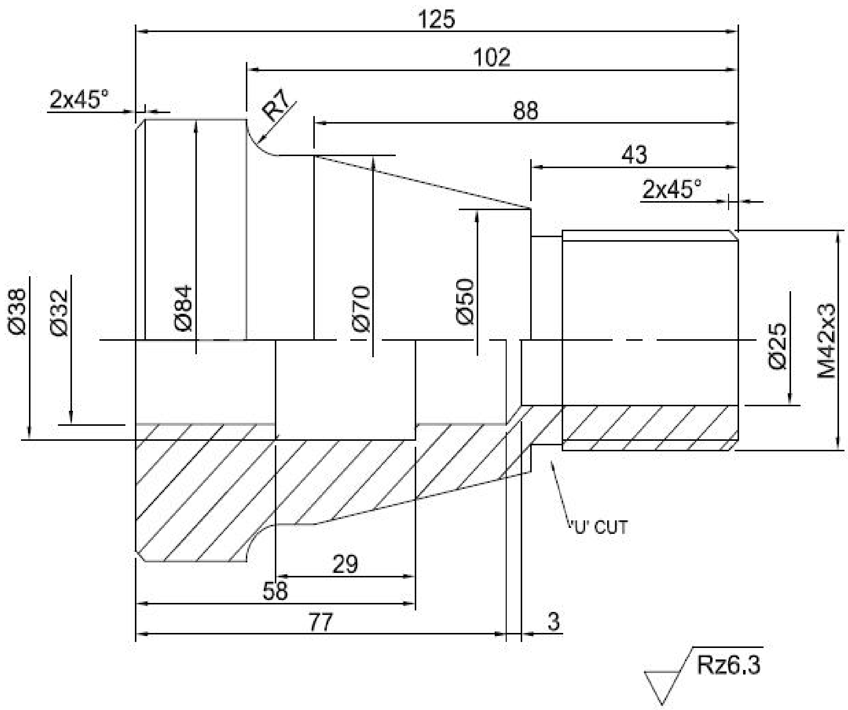
EXCERCISE



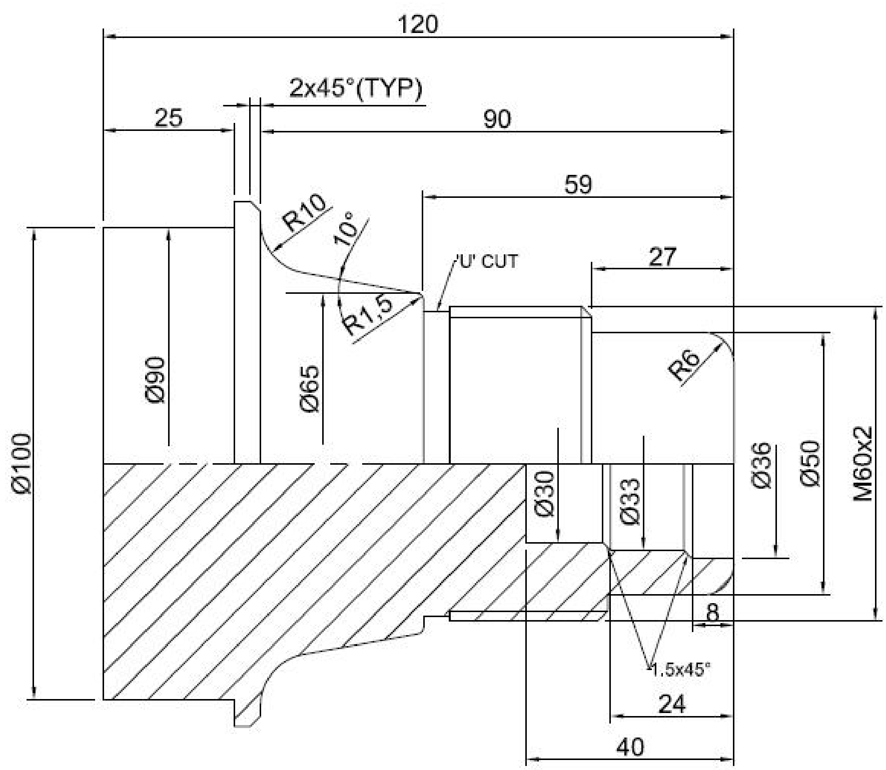
RAW MATERIAL: MS BAR OF DIAMETER 70 MM
AND LENGTH 160 MM
DIAGRAM NOT TO SCALE
ALL DIMENSIONS ARE IN MM


55
48
28.04
22.776
12
10
R2
ø42
ø38
ø30.775
ø26
ø29.8


R 20
LH V-Thread
Pitch = 2mm
10
18
28
15
10
15
10
35
10



ALL DIMN'S ARE IN mm.



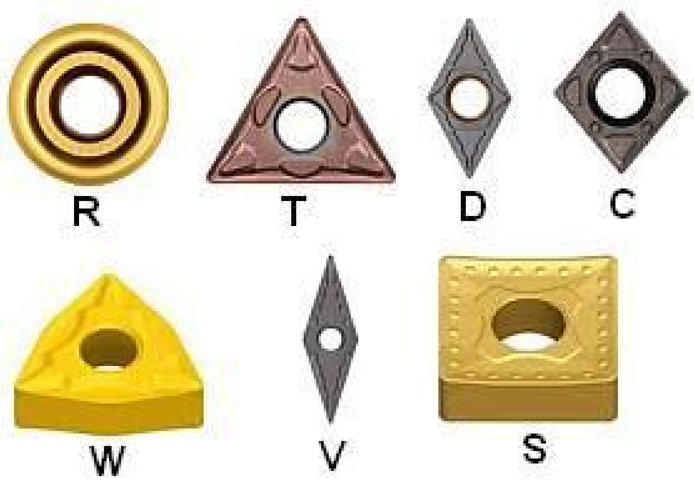
ALL DIMN'S ARE IN mm.

SPECIFICATION OF INDEXABLE TIP INSERTED CUTTING TOOL

INSERTS

Identification will consist of up to ten positions; each position defines a characteristic of the insert in the following order: Shape, Clearance, Tolerance class, Type, Size, Thickness, Cutting- point configuration, Edge preparation, Hand, Facet size.

1. SHAPE



A letter symbol is used to identify the **SHAPE**. Insofar as possible, this letter is descriptive of the shape, as follows: A – Parallelogram 85° , B – Parallelogram 82°, C – Diamond 80° (Rhombic)
D – Diamond 55° (Rhombic) E – Diamond 75° (Rhombic),
H – Hexagon 120°, K – Parallelogram 55°, L – Rectangle 90°, M – Diamond 86° (Rhombic), O – Octagon 135°, P – Pentagon 108°, R – Round, S – Square 90°, T – Triangle 60°, V – Diamond 35° (Rhombic), W – Trigon 80°

SHAPE Identification chart -

Identification chart

S	N	C	N	4
1	2	3	4	5
SHAPE	CLEARANCE	TOLERANCE CLASS	TYPE	SIZE (I.C.)

1 SHAPE

- A Parallelogram
- B Parallelogram
- C Diamond
- D Diamond
- E Diamond
- H Hexagon
- K Parallelogram
- L Rectangle
- M Diamond
- O Octagon
- P Pentagon
- R Round
- S Square
- T Triangle
- V Diamond
- W Trigon

2 CLEARANCE

- N 0°
- A 3°
- B 5°
- C 7°
- P 11°
- D 15°
- E 20°
- F 25°
- G 30°
- O Ω°

3 TOLERANCE CLASS

Tolerance on Dimensions
(± from nominal)

Tolerance Letter	Dimension B	Dimension A	Dimension T
A	0.0002	0.0010	0.001
B	0.0002	0.0010	0.005
C	0.0005	0.0010	0.001
D	0.0005	0.0010	0.005
E	0.0010	0.0010	0.001
F	0.0002	0.0005	0.001
G	0.0010	0.0010	0.005
H	0.0005	0.0005	0.001
J	0.0002	●	0.001
K	0.0005	●	0.001
L	0.0010	●	0.001
M	●	●	0.005
U	●	●	0.005
N	●	●	0.001

● see charts below

VALID FOR SHAPES: C, E, H, M, O, P, S, T, R, W

I.C.	A Class J, K, L, M, N	A Class U	B Class M, N	B Class U
3/16	0.002	0.003	0.003	0.005
7/32	0.002	0.003	0.003	0.005
1/4	0.002	0.003	0.003	0.005
5/16	0.002	0.003	0.003	0.005
3/8	0.002	0.003	0.003	0.005
1/2	0.003	0.005	0.005	0.008
5/8	0.004	0.007	0.006	0.011
3/4	0.004	0.007	0.006	0.011
1	0.005	0.010	0.007	0.015
1-1/4	0.006	0.010	0.008	0.015

VALID FOR SHAPED D ONLY

I.C.	A Class J, K, L, M, N	B Class M, N
7/32	0.002	0.004
1/4	0.002	0.004
5/16	0.002	0.004
3/8	0.002	0.004
1/2	0.003	0.006
5/8	0.004	0.007
3/4	0.004	0.007

VALID FOR SHAPE V ONLY

I.C.	A Class J, K, L, M, N	B Class M, N
7/32	0.002	0.004
1/4	0.002	0.004
5/16	0.002	0.004
3/8	0.002	0.004
1/2	0.003	0.006
5/8	0.004	0.007
3/4	0.004	0.007

4 TYPE

- A
- B
- C
- F
- G
- H
- J
- M
- N
- Q
- R
- T
- U
- W
- X With Dimensions

5 SIZE (I.C.)

For equal sided inserts this indicates the inscribed circle (I.C.) in 1/8 of an inch.

Examples:

- 1/8" = 1
- 5/32" = 1.2
- 3/16" = 1.5
- 7/32" = 1.8
- 1/4" = 2
- 5/16" = 2.5
- 3/8" = 3
- 1/2" = 4
- 5/8" = 5
- 3/4" = 6
- 7/8" = 7
- 1" = 8
- 1-1/4" = 10

For rectangles and parallelograms two digits are necessary.

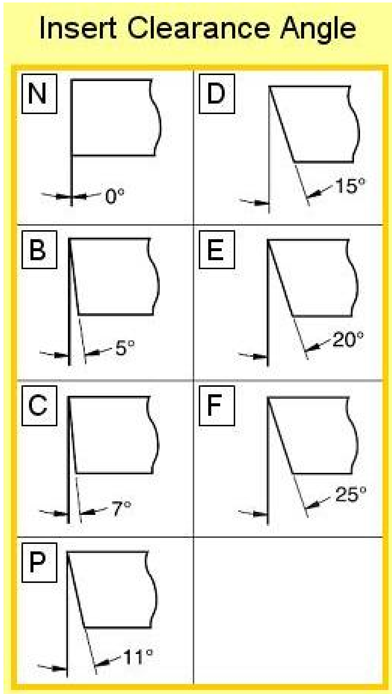
1st digit = number of 1/8" in width

2nd digit = number of 1/4" in length

(continued)

2. CLEARANCE ANGLE (Relief Angle)

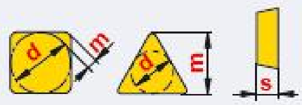
The second position is a letter denoting the relief angles. These angles are the difference from 90° measured in a plane normal to the cutting edge generated by the angle between the flank and top surface of the insert.



A=3°, B=5°, C=7°, D=15°, E=20°,
F=25°, G=30°, N=0°, P=11°, O= Others,

3. TOLERANCE CLASS

Tolerances



	d ±	m ±	s ±
A	0,025	0,005	0,025
F	0,013	0,005	0,025
C	0,025	0,013	0,025
H	0,013	0,013	0,025
E	0,025	0,025	0,025
G	0,025	0,025	0,13
J	0,05-0,15*	0,005	0,025
K	0,05-0,15*	0,013	0,025
L	0,05-0,15*	0,025	0,025
M	0,05-0,15*	0,08-0,20	0,13
N	0,05-0,15*	0,08-0,20	0,025
U	0,08-0,25*	0,13-0,38	0,13

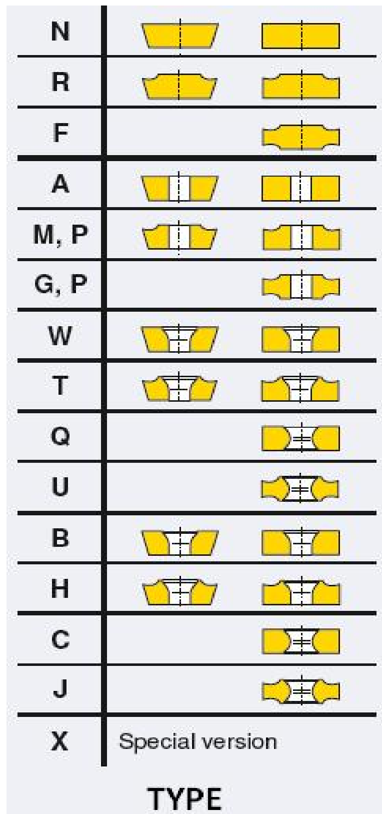
The third position is a letter that indicates the tolerances that control the indexability of the insert. Tolerances specified do not imply the method of manufacture. Dimensions are established prior to supplemental edge or coating modification.

4. <u>TYPE</u>

The fourth position is a letter to show differences in design not specifically provided for in the other sequence positions. The most common differences are the existence of fixing holes, countersinks, and special features on rake surfaces.

A – With hole, without chip grooves
B – With hole, without chip grooves, and one countersink (70°-90°)
C – With hole, without chip grooves, and two countersinks (70°-90°)
F – Without hole with chip grooves on two rake faces.
G – With hole and chip grooves on two rake faces
H – With hole, one countersink (70°-90°) and chip groove on one rake face.
J – With hole, two countersinks (70°-90°) and chip grooves on two rake faces.
M – With hole and chip groove on one rake face
N – Without hole, without chip grooves
Q – With hole, without chip grooves, and two countersinks (40°-60°).
R – Without hole with chip groove on one rake face

T – With hole, one countersink (40°-60°) and chip groove on one rake face
U – With hole, two countersinks (40°-60°) and chip grooves on two rake faces
W – With hole, without chip grooves and one countersink (40°-60°)
X – With dimensions or details requiring detailed explanation,
a sketch or additional Specifications



******* A space may be used after the fourth position to separate the shape-describing portion from the following dimensional description of the insert and is not to be considered a position in the standard description.

5. INSERT SIZE

A. Regular Polygon and Diamond (Rhombic) Inserts –

The fifth position is a significant one- or two-digit number indicating the size of the inscribed circle (I.C.) for all inserts having a true I.C. such as Round, Square, Triangle, Trigon, Pentagon, Hexagon, Octagon, and Diamond. This position designates the number of eighths of an inch in the nominal size of the I.C. and will be a one- or two-digit number when the number of eights of an inch in the I.C. is a whole number. It will be a two-digit number carried to one decimal place when it is not a whole number.

	d mm	
	06 08 10 12	16 20 25 32

L	d	d	L
mm	Inch	mm	mm
06	5/32	3,96	03
09	7/32	5,56	05
11	1/4	6,35	06
16	3/8	9,52	09
22	1/2	12,7	12
27	5/8	15,8	15
33	3/4	19,0	19
44	1	25,4	25

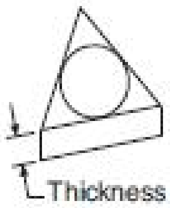
B. Rectangles or parallelograms

On rectangle and parallelogram inserts, the width and length dimensions are used in place of the I.C. A two-digit number is necessary because these do not have an inscribed circle. The first digit indicates the number of eighths of an inch in the width and the second digit indicates the number of fourths of an inch in the length of the insert.

Identification chart (concluded)

3	2	E	R	4
6	7	8	9	10
THICKNESS	CUTTING POINT	EDGE PREP.	HAND	FACET SIZE

6 – THICKNESS



This indicates the insert thickness in 1/16 of an inch.

Measured From:

Cutting edge to opposite pad on insert types F, G, J, & U

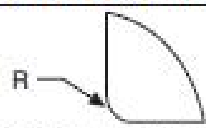
Cutting edge to bottom on types H, M, R, & T

Top to bottom on types A, B, C, N, O, & W

Examples:

1/16" = 1
5/64" = 1.2
3/32" = 1.5
1/8" = 2
5/32" = 2.5
3/16" = 3
7/32" = 3.5
1/4" = 4
5/16" = 5
3/8" = 6
7/16" = 7
1/2" = 8
9/16" = 9
5/8" = 10

7 – CUTTING POINT



This indicates the form on the cutting point in 1/64 of an inch for those with a radius.

Examples:

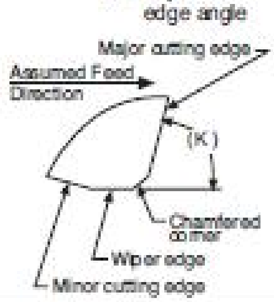
0.002" = 0
0.004" = 0.2
0.008" = 0.5
1/64" = 1
1/32" = 2
3/64" = 3
1/16" = 4
5/64" = 5
3/32" = 6
7/64" = 7
1/8" = 8
5/32" = 10
3/16" = 12
7/32" = 14
1/4" = 16
any other = x

DP

For those with a facet

Facet Angle (K) 1st letter

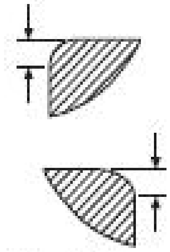
A = 45°
D = 60°
E = 75°
G = 87°
P = 90°
Z = any other edge angle



Facet Clearance (primary facet) 2nd letter:

A = 3°
B = 5°
C = 7°
D = 15°
E = 20°
F = 25°
G = 30°
N = 0°
P = 11°
Z = any other

8 – EDGE PREP.



Honed Edge
(Rounded Corner)

A = 0.0005" to less than 0.003"

B = 0.003" to less than 0.005"

C = 0.005" to less than 0.007"

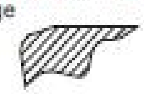
E = Rounded edge

F = Sharp edge

J = Polished to 4 microfinish AA. Rake face only.

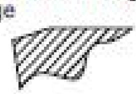
K = Double chamfered cutting edge

P = Double chamfered and rounded cutting edge

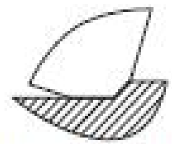


S = Chamfered and rounded cutting edge

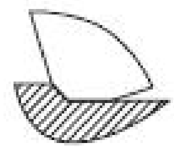
T = Chamfered cutting edge



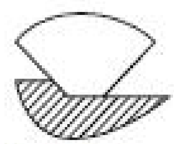
9 – HAND



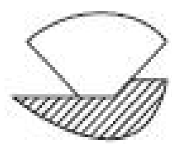
R = Right Hand



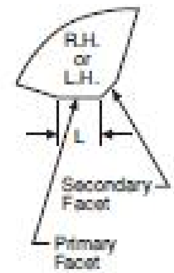
L = Left Hand

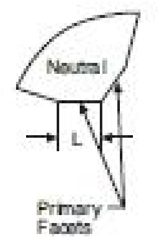


N = Neutral



10 – FACET SIZE





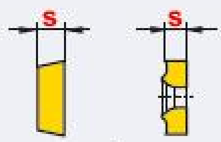
This indicates the length of the primary facet in aproximately 1/64 of an inch.

Used only following a double letter in the 7th position.

Examples:

1/64" = 1
1/32" = 2
3/64" = 3
1/16" = 4
5/64" = 5
3/32" = 6
7/64" = 7
1/8" = 8
9/64" = 9
5/32" = 10

6. THICKNESS



Inch	mm	Index
1/16	1,59	01
3/32	2,38	02
1/8	3,18	03
5/32	3,97	T3
3/16	4,76	04
7/32	5,56	05
1/4	6,35	06
5/16	7,94	07
3/8	9,52	09

Insert thickness

Code	Corner radius mm
00	≤ 0,05
01	0,1
02	0,2
04	0,4
08	0,8
12	1,2
16	1,6
24	2,4
32	3,2

RN **00**
RC **MO**

The sixth position is a significant one- or two-digit number indicating the number of sixteenths of an inch in the thickness of the insert. This position will be a one-digit number when the number of sixteenths of an inch in the thickness is a whole number. It will be a two-digit number carried to one decimal place when it is not a whole number.

7. CUTTING-POINT CONFIGURATION

The cutting point configuration, indicated by the seventh position, will be shown by either a significant number indicating a radius, tangent to the adjacent sides, or two letters indicating the details of the primary facet. In the case of a radius, the number designates the number of sixty-fourths of an inch in the radius as shown below.

0 – Sharp corner (0.002 inch max. radius)

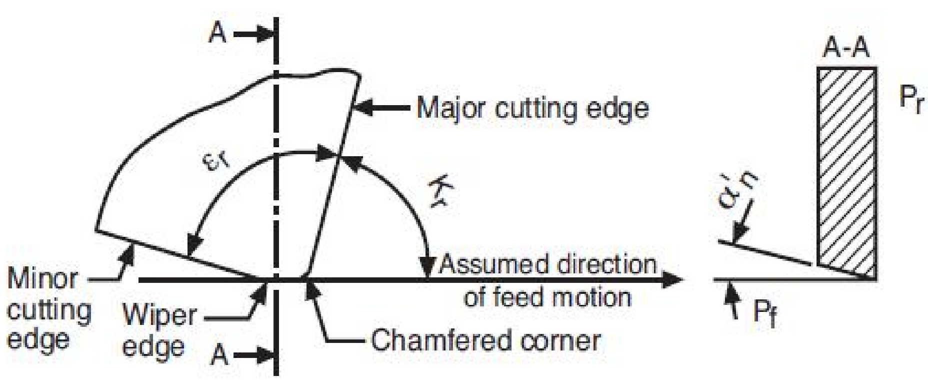
0.2 – 0.004 inch radius
0.5 – 0.008 inch radius
1 – 1/64-inch radius, 2 – 1/32-inch radius
3 – 3/64-inch radius, 4 – 1/16-inch radius
5 – 5/64-inch radius, 6 – 3/32-inch radius
7 – 7/64-inch radius, 8 – 1/8-inch radius
X – Any other corner radius

*** In the case of a facet, two letters will be used. The first designates the facet angle and the second designates the facet clearance angle as shown above. Secondary facets and radii in place of secondary facets are not defined by this standard and shall be manufacturer's standard. Primary facet angle Kr clearance α′n.

Facet angle K_r	Primary facet clearance α'_n
A – 45° facet	A – 3°
D – 60° facet	B – 5°
E – 75° facet	C – 7°
F – 85° facet	D – 15°
G – 87° facet	E – 20°
P – 90° facet	F – 25°
Z – Any other cutting edge angle	G – 30°
	N – 0°
	P – 11°
	Z – Any other wiper edge normal clearance

NOTES-

1. The wiper edge is a part of the minor cutting edge.



2. Inserts with wiper edge may or may not have chamfered corner depending on their type. The designation for indexable inserts gives no information as to whether the inserts have or do not have chamfered corners. For standardized inserts, this information is given in dimensional standards; for nonstandardized inserts, it is

given in suppliers' catalogues. It is intended that all nose radii be essentially tangent to the included angle on both sides of the form.

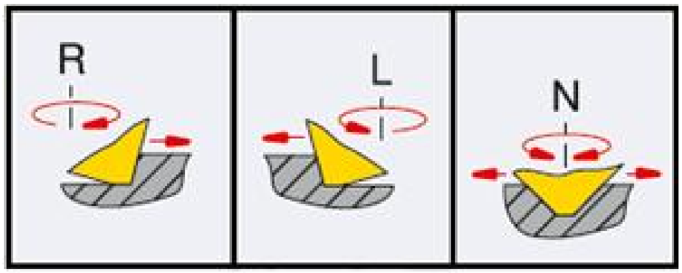
8. EDGE PREPARATION-

The eighth position shall be a letter. It shall define special conditions, such as edge treatment and surface finish, as follows

A – Honed – 0.0005 to less than 0.003 inch
B – Honed – 0.003 to less than 0.005 inch
C – Honed – 0.005 to less than 0.007 inch
E- Rounded Cutting Edges
F – Sharp Cutting Edges
J – Polished- 4-microinch arithmetic average (AA) - Rake face only
K – Double Chamfered Cutting Edges
P – Double Chamfered and Rounded Cutting Edges
S – Chamfered and Rounded Cutting Edges
T– Chamfered – Manufacturer's standard negative land–Rake face only.

9. HAND

When the geometry dictates a handed condition, the letters "R", "L", or "N" must be used in the ninth position to indicate right hand, left hand, or neutral.



10. FACET SIZE

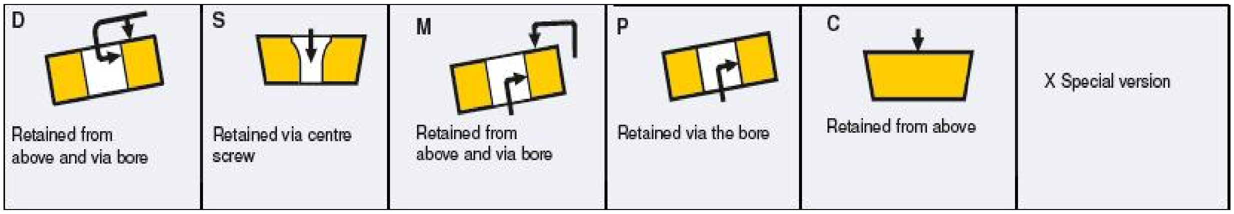
The tenth position is only used if there are letters in the seventh position. It shall be a significant number representing the nominal sixty-fourths of an inch in length of the primary facet.

<u>HOLDER</u>

Tool holders are normally used either according to the job shape or according to the machine specification. The holder holds the insert in it so the specification of tool holders is almost same as the insert. The tool holder also have ten positions of identification, each position defines a characteristic of the tool holder in the following order: Clamping method, Insert shape, Style, Clearance angle, Cutting direction, Shank height, Shank width, Tool length, Cutting edge length, Special tolerances.

1. <u>CLAMPING METHOD</u>

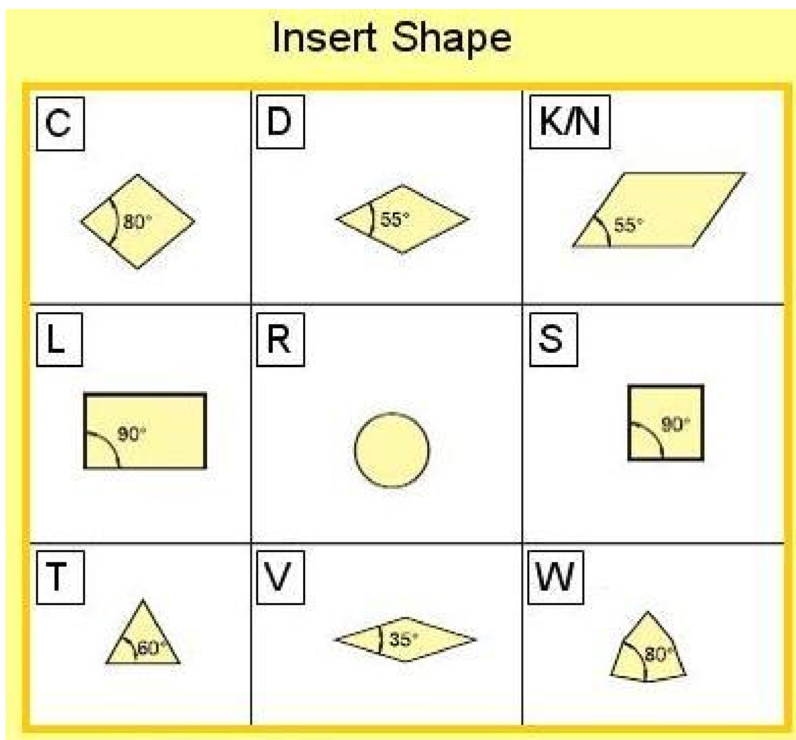
The letter in the clamping method defines, how the insert is mounted on or clamped on the tool holder.



2 <u>INSERT SHAPE</u>

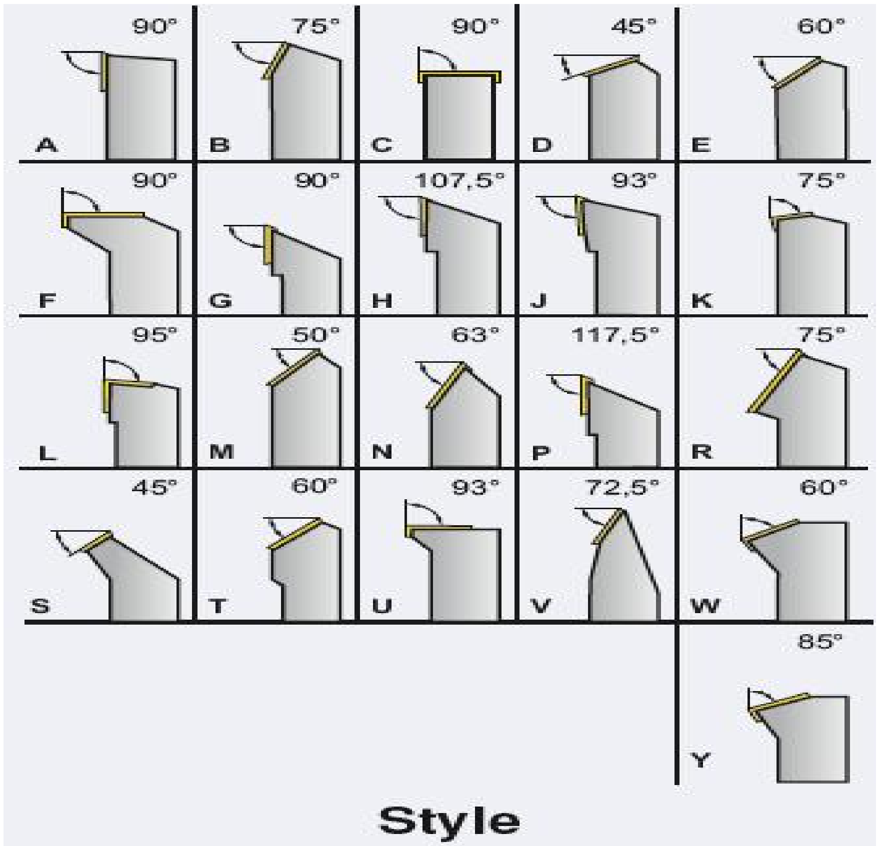
A letter symbol is used to identify the shape and the angle of the insert, this letter is descriptive of the shape of the insert that going to be use in the holder and the shape identification is same as the identification of the insert.

.



3. <u>CUTTING TOOL STYLE</u>

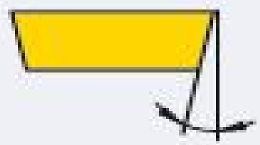
This position give define the style of the tool holder and a letter to show differences in design in the angle on the tool.



4. CLEARANCE ANGLE

The fourth position is a letter denoting the relief angles. These angles are the difference from 90° measured in a plane normal to the cutting edge generated by the angle between the flank and top surface of the insert.

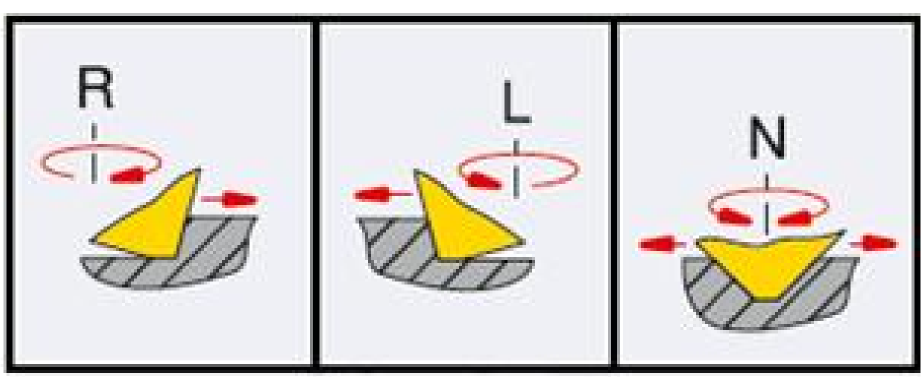
Clearance angle



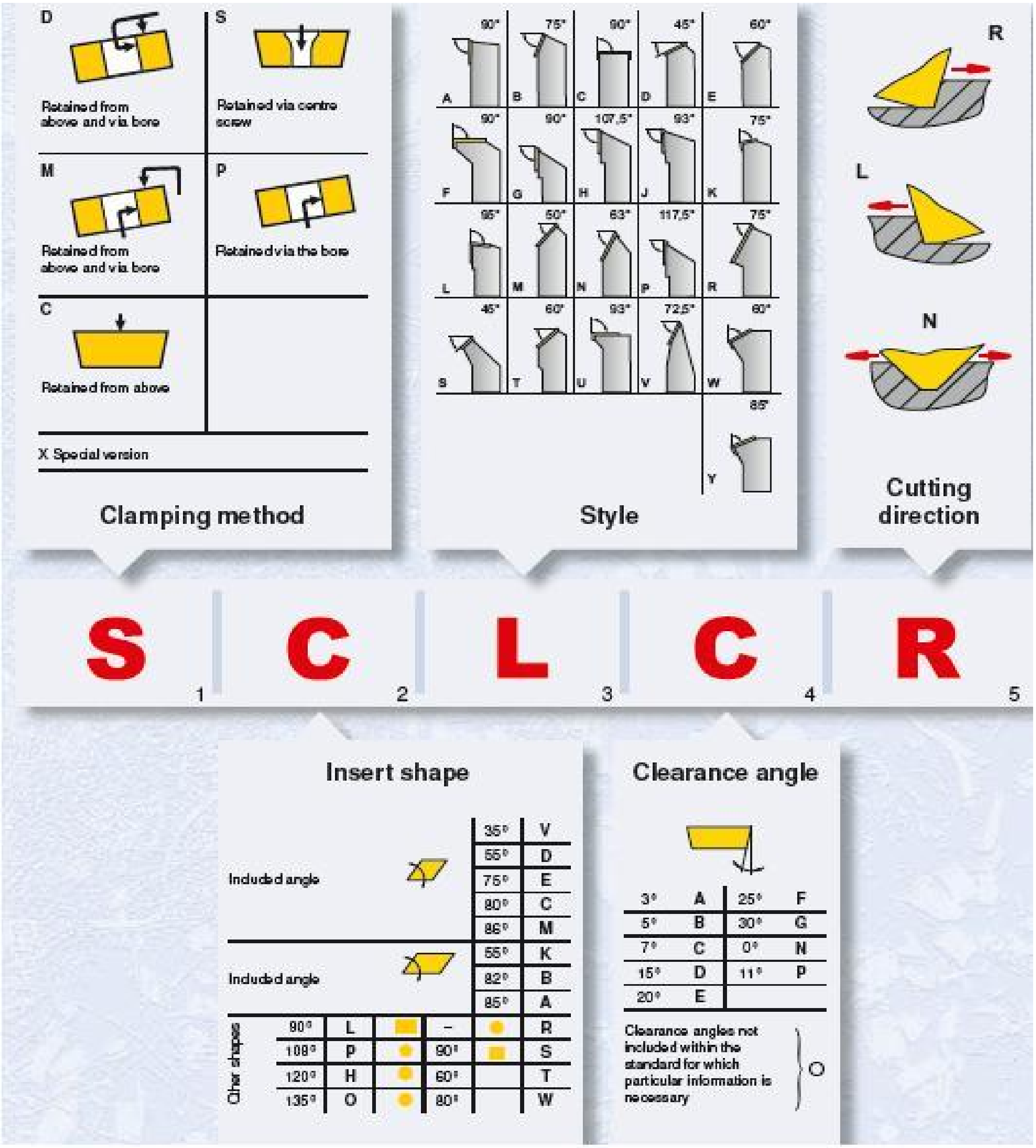
3°	A	25°	F
5°	B	30°	G
7°	C	0°	N
15°	D	11°	P
20°	E		

5. <u>CUTTING DIRECTION</u>

This geometry say about the hand direction of the tool, the letters “R”, “L”, or “N” must be used in this position to indicate right hand, left hand, or neutral.



<u>CUTTING TOOL HOLDER SELECTION</u>

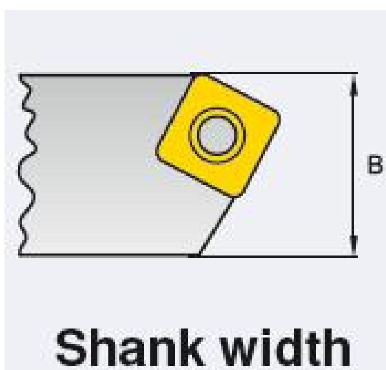


6. SHANK HEIGHT

This position will give the height of the shank of the tool holder. Normally the shank height is given in mm.

7. SHANK WIDTH

This position will give the width of the shank of the tool holder. Normally the shank height is given in mm.

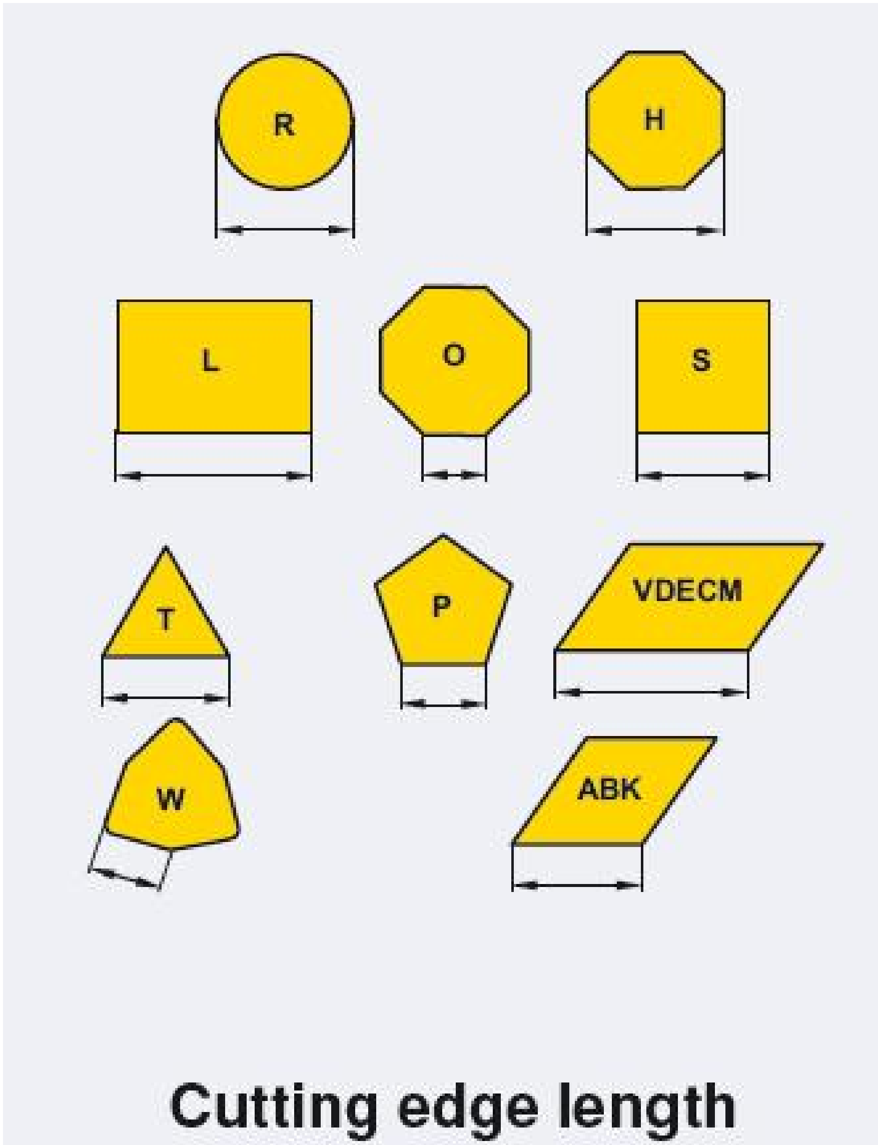


8. TOOL LENGTH

The position eight will give the total length of the shank of the tool holder. That is from the tip to the end of the tool holder.

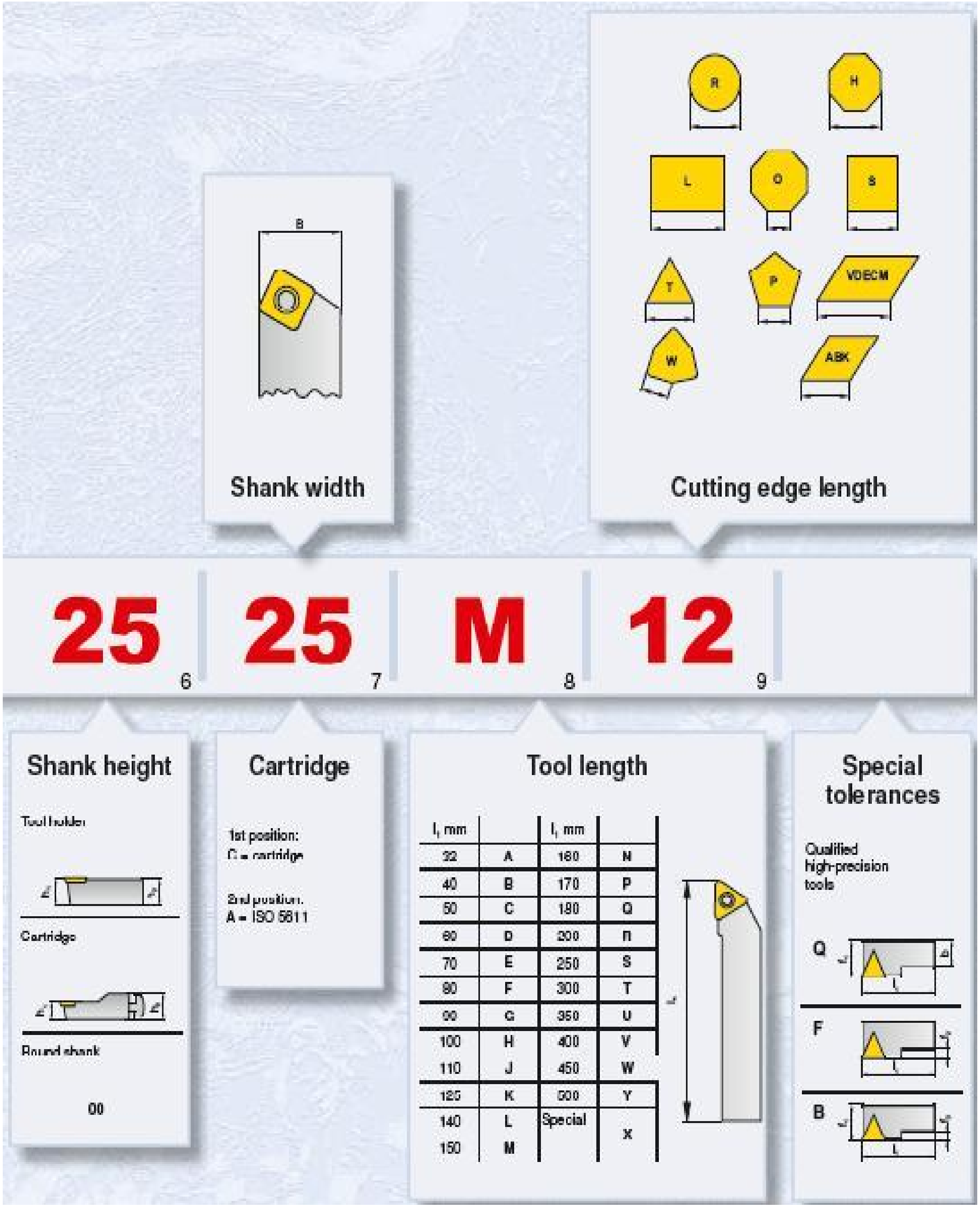
9. CUTTING EDGE LENGTH

The length of the cutting edge of the insert is been defined by using letters in this position.



10. SPECIAL TOLERANCE

This position is reserved for the special tolerances which are normally defined as per the requirement.



l_1 mm		l_1 mm	
32	A	160	N
40	B	170	P
50	C	180	Q
60	D	200	R
70	E	250	S
80	F	300	T
90	G	350	U
100	H	400	V
110	J	450	W
125	K	500	Y
140	L	Special	X
150	M		

ASSIGNMEMT

01. What are the causes of workshop accidents?
02. What are the minimum first aid facilities must be in workshop?
03. What is a soft hammer made up of?
04. What is the use of ‘C’ clamp?
05. What is measurement?
06. Name different type of measuring tools?
07. What is the use of try square?
08. What are the different parts of an outside micrometer?
09. What are the advantages of a vernier caliper over a micrometer?
10. What is a measuring error?
11. What screw and motor is used in X and Z axis.
12. Air unit consist of what?
13. Name five parts of CNC lathe?
14. How many tools can be mounted on turret?
15. Which three units are operated by hydraulic?
16. What are tail stock consist of?
17. Classify NC system?
18. What are the advantages of NC systems over conventional system?
19. Write shot notes on memories used in NC system?
20. Write the limitations of NC &CNC system?
21. What is the function of circulating ball screws? Explain with a neat sketch.
22. Name four programme languages?
23. What is machining center?
24. Write programming codes for FANUC Control program?
25. What is Machine Datum and Programme Zero Point?
26. Give example of for NC word.
27. Write the functions and example of absolute, incremental coordinates?
28. Write the functions of ‘T’, ‘S’, ‘F’ and ‘LF’?
29. How many types of block are there?
30. Create an example of block?

31. How many methods are there to do ‘program test’?
32. Explain safety distance program testing?
33. What ‘TNRC’ stand for?
34. Write the format and parameter for ‘Rough turning cycle’?
35. Write the format and parameter for ‘Finishing turning cycle’?
36. Write the format and parameter for ‘Facing cycle’?
37. What is “U0.05 and W0.1” in following example-
    “G71 U0.2 RO.3;
    G71 P10 Q20 U0.05 W0.1 F0.2;
38. What is “P5 and Q15” in following example- G70 P5 Q15 F0.1;
39. What is “W0.5 and W0.2” in following example-
    “G72 W0.5 RO.3;
    G72 P10 Q20 W0.2 F0.2;
40. Write the format and parameter for ‘Drilling cycle’?
41. Write the format and parameter for ‘Boring cycle’?
42. What is ‘U’ in Boring cycle?
43. What is the difference between Boring cycle and Rough turning cycle?
45. Write the format and parameter for ‘Profile turning cycle’?
46. Write the format and parameter for ‘Profile Boring cycle’?
47. What is G40?
48. Explain difference between G41 and G42?
49. Define thread?
50. What are pitch, lead and angle of the thread?
51. If pitch is 2.5mm in a double start thread then what is the lead of the thread?
52. What is the basic different between left hand and right hand thread?
53. Explain all the parameters of G76 cycle?
54. What is ‘F’ in threading cycle?
55. Explain “P040060” in G76 cycle?
56. What is ‘R’ in the second block of G76 cycle?
57. What is the different in ‘R’ in int. and external taper thread?
58. What is meant by canned cycle?
59. What is a DNC machine?
60. What is meant by VMC machine?

<u>Fill up the blanks</u>

(a) A group of commands at each step of the sequence is called……,
(b) The origin of co-ordinate system of the machine is called……………

(c) The full form of ISO is……………………………………….
(d) The CNC ….should have practical experience as m/c tool operator.
(e) The motion axis of the main spindle of the m/c is called ………axis,
(f) The facility of repetitive programming techniques are called…………,
(g) The main advantage of Abs. programming is.…..by the programmer.
(h) Address of CNC program relate to the Tool position is called…word.
(i) Actually the ……. is defined in the part program for CNC machining.
(j) In CNC Milling, the axis of motion should be programmed with respect to ……………. of the cutter,
(k) The number for discriminating each block is called…………….. .
(l) NC machine Tool is not allowed to the automatic tool ……………..,
If there is any change of tool dimension.
(m) The EOB symbol identifies the ………….…………..,
(n) In motion the distance to be traveled by the Tool in differences from its existing position is called …………….,
(o) The Tool Offset function is addressed by…………….. .
(p) The most versatile control system has been…….……… .,
(q) The command G04 is use for, but it cannot function without………,
(r) The coordinate measuring system of the Work is called …………...…,
(s)The system are used to control the position of the Tool is called …….,
(t) The spindle speed on CNC m/c can specified either in …….……. .
(u) Card reader can eliminate by use of ---------------------- Control.
(v) In block format the symbol (C) is used for ……………………..,
(w). In block format the sign (/) is stands for ……………………..,
(x). The preparatory function G04 is use for ……………………...
(y). The word (M) used to specify ……………………..,
(z). In the block format XP, YP & ZP are used for …………………….. .

ANSWERS-

a. Word, **b.** Machine Coordinate System, **c.** International Standard Organization, **d.** Programmer, **e.** Z axis, **f.** Sub-Program facility, **g.** Easy modification, **h.** Co-ordinate Word, **i.** Tool Path, **j.** Centre Line, **k.** Block Number, **l.** Compensation, **m.** End of Block, **n.** Incremental Movement, **o.** 'D' Letter, **p.** Man, **q**. Given Time, **r.** Work Coordinate System, **s.** Coordinate System, **t.** G96/G97 **u.** Computer Numerical **v.** Rotational axis parallel to Z-axis, **w.** Block Skip, **x.** Pause time or Dwell time, **y.** Miscellaneous Functions, **z.** Coordinate of a position.

Choose the Correct answer .

1. In CNC m/c tool, the part program entered into the computer memory

a. can be used only once
b. can be used again and again
c. can be used again but it has to be modified every time
d. cannot say

2. Which of the following statements are correct for CNC machine tool?

a. CNC control unit does not allow compensation for any changes in the dimensions of cutting tool
b. CNC machine tool are suitable for long run applications
c. It is possible to obtain information on machine utilization which is useful to management in CNC machine tool
d. CNC machine tool has greater flexibility
e. CNC machine can diagnose program and can detect the machine defects even before the part is produced

3. Several machine tools can be controlled by a central computer in

a. NC (Numerical Control) machine tool
b. CNC (Computer Numerical Control) machine tool)
c. DNC (Direct Numerical Control) machine tool)
d. CCNC (Central-Computer Numerical Control) machine tool

4. Part-programming mistakes can be avoided in

a. NC (Numerical Control) machine tool
b. CNC (Computer Numerical Control) machine tool
c. Both a. and b.
d. None of the above

5. Which machine tool reduces the number of set-ups in machining operation, time spent in setting machine tools and transportation between sections of machines?

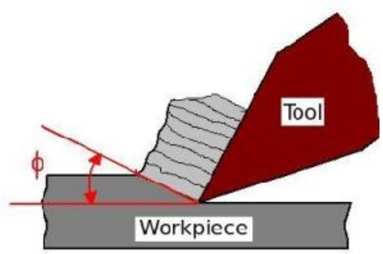
a. Computer Numerical Control machine tool
b. Direct Numerical Control machine tool

c. Adaptive Control Systems
d. Machining centre

6. The point at which the cutting tool reaches, beyond which it will not function satisfactorily until it is reground, is called as
a. tool wear
b. tool failure
c. too diffusion
d. none of the above

7.In metal cutting operation, maximum heat (i.e.80-85%) is generated in
a. the shear zone
b. the chip-tool interface zone
c. the tool-work interface zone
d. none of the above

8. What is the angle Φ shown in the below diagram of basic mechanism of chip formation?



a. Shear angle
b. Tool rake angle
c. Chip angle
d. Cutting angle

9. The forces required for metal cutting operation
a. increase with increase in the feed of the tool and decreases with increase in the depth of cut
b. decrease with increase in the feed of the tool and increases with increase in the depth of cut
c. increase with increase in both the feed of the tool & the depth of cut
d. decrease with increase in both the feed of the tool & the depth of cut

10. Which fixtures are used for machining parts which must have machined details evenly spaced?

a. Profile fixtures
b. Duplex fixtures
c. Indexing fixtures
d. None of the above

11. The angle between side cutting edge & end cutting edge is called as

a. approach angle
b. nose angle
c. side relief angle
d. end relief angle

12. Lead angle in the single point cutting tool is the angle between

a. the end cutting edge and the normal to the tool shank
b. the portion of side shank immediately below the side cutting edge and the line perpendicular to the base of the tool
c. the tool face and the parallel to the base of the tool
d. side cutting edge and the side of the tool shank

13. The surface of the single point cutting tool on which the chips formed in cutting operation slide is called as

a. flank
b. heel
c. face
d. shank

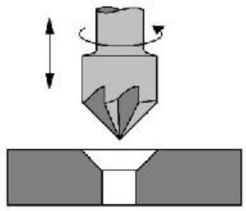
14. The thickness of layer material removed in one pass of workpiece under the cutter is called as

a. single pass cut
b. depth of cut
c. width of cut
d. face cut

15. How is the workpiece fed of in down milling process?

a. In down milling process, the workpiece is fed in the same direction as that of cutter's tangential velocity
b. In down milling process, the workpiece is fed in the opposite direction as that of cutter's tangential velocity

16. Which type of drilling operation is shown in below diagram?



a. Boring
b. Counter boring
c. Sinking
d. Countersinking

17. Arrange the below operations in operator controlled machine tool in correct order.

(A) Operator
(B) Process planning
(C) Machine tool
(D) Component drawing
(E) Completed component
a. (A) – (D) – (B) – (C) – (E)
b. (D) – (B) – (C) – (A) – (E)
c. (B) – (D) – (C) – (A) – (E)
d. (D) – (B) – (A) – (C) – (E)

18. What is the reason for using unconventional or advanced machining processes?

a. Complex surfaces
b. High accuracy and surface finish
c. High strength alloys
d. All of the above

19. Continuous chips are formed during metal cutting operation due to

a. ductile work materials
b. large rake angle
c. high cutting speed
d. all of the above

20. The boring operation means a process of-

a. making a hole in an object
b. enlarging a hole which is already in an object
c. finishing an existing hole very smoothly and accurately in size
d. none of the above

21. Encoder is used in CNC machine tool, to sense and control

(A) Spindle speed
(B) Spindle position
(C) Table position
(D) All of these

22. The codes do not directly help in machining, but required for proper functioning of an NC/CNC machine tool are called

a. miscellaneous codes
b. ‘S’ words
c. preparatory codes
d. ‘T’ words

23. Program coordinates that are based on a fixed origin are called

a. absolute
b. Cartesian
c. relative
d. incremental

24. The correct NC block for this motion is

a. N10 G03 X70.0 Y20.0 I-50.0 J20.0;
b. N10 G02 X70.0 Y20.0 I0.0 J-20.0;
c. N10 G03 X70.0 Y20.0 I-50 J-20.0;
d. N10 G03 X70.0 Y20.0 I0.0 J20.0;

25. The setting of tools to a specific length is called

a. presetting
b. post setting
c. specific setting
d. tool on setting

26. Constant surface speed facility is useful during

a. taper turning
b. thread cutting
c. drilling
d. turning

27. What is the full form of Fanuc?

a. Fuji Automatic NUmerical Control
b. Fuji Automatic Numerical Unit Control
c. French Automatic Numerical Unit Control
d. Fuji Association Numeric Universal Control

28. Full form of DNC is-

a. Distributed Numerical Control
b. Data Numerical Control
c. Differential Numerical Control
d. None of above

29. What Is Apt Language?

a. automated programming language
b. automatic programming language
c. alternative programming language
d. automated processing language

30. What is ATC?

a. alternative tool collector
b. automatic tool changer
c. automatic tool collector
d. alternative tool changer

ANSWERS

1. B, 2. C, D, E, 3. C, 4. B, 5. D, 6. B, 7. A, 8. A, 9. C, 10. C, 11. B, 12. D,
13. C, 14. D, 15. A, 16. D, 17. D, 18. D, 19. D, 20. B, 21. C, 22. A, 23. A,

24. A, 25. A, 26. A, 27. A, 28. A, 29. A, 30. B.

<u>Question for Cutting tool selection</u>

01. How many positions are there to identify a Tool Tip Insert?
02. Define the characteristics of the inserts Identification position in order.
03. If ‘D’ in 2nd place in insert. Then what is the clearance angle?
04. If ‘M’ is in the 1st place of insert, then what 1st place defines and what is ‘M’?
05. What is ‘N’ in the 9th place of insert?
06. What is the clamping method position in the holder?
07. What is 25 25 means, in the 6th and 7th position in the holder?
08. Where is the ‘cutting direction’ position in the holder?
09. Write the specification mark of a turning cum facing tool with insert.
10. Write the specification mark of a radius form cutting tool with insert.