IJERST

International Journal of Engineering Research and Science & Technology

ISSN : 2319-5991
Vol. 4, No. 4
November 2015

www.ijerst.com

Email: editorijerst@gmail.com or editor@ijerst.com

International Journal of Engineering Research and Science & Technology

ISSN 2319-5991 www.ijerst.com
Vol. 4, No. 4, November 2015

Research Paper

STUDY ON VERTICAL MACHINING CENTER

V V Prathibha Bharathi[1]*, A Kalyan Charan[1], V Poojitha[1] and V V Naga Deepthi[1]

Corresponding Author: **V V Prathibha Bharathi ✉ prathibhaseenu@gmail.com*

In the present study in view of the latest development and revolutionary changes taking place in CNC field through the world, Mechanical elements have to be designed and manufactured to precision, which is perfectly and easily possible through these modern CNC machines. This work is based on the capacity and capability of vertical machining Centre (VTC) with auto tool changer. The top slide which was part programmed can be machined using VTC. And Machining Time is compared in between carbide and hardened tools. The "Top slide" oflathe's called for powerful NC programming technique were used absolute position type data input system using G codes, M codes, polar coordinate programs, circular and linear interpolation, canned cycles, etc. The above mentioned component – top slide being manufactured by using various Conventional machine tools like horizontal milling, vertical milling, surface grinding, boring machine and slotting machines. This involved a considerable lead time and usually delayed the assembly schedule. it has been modified and adopted for regular production on this machine, in two setups there by boosting their productivity and ensuring quality in each and every piece. Finally, we can establish for regular production

Keywords: Text data compression, Lossless data compression

INTRODUCTION

Vertical Machining Center

Milling is the machining process of using rotary cutters to remove material from a work piece advancing (or feeding) in a direction at an angle with the axis of the tool. It covers a wide variety of different operations and machines, on scales from small individual parts to large, heavy-duty gang milling operations. It is one of the most commonly used processes in industry and machine shops today for machining parts to precise sizes and shapes.

Milling can be done with a wide range of machine tools. The original class of machine tools for milling was the milling machine (often called a mill). After the advent of Computer Numerical Control (CNC), milling machines evolved into machining centers (milling machines with automatic tool changers, tool magazines or carousels, CNC control, coolant systems, and

[1] Department of Mechanical Engineering, Matrusri Engineering College, #16-1-486, Saidabad, Hyderabad, Telangana 500059, India.

enclosures), generally classified as Vertical Machining Centers (VMCs) and Horizontal Machining Centers (HMCs). The integration of milling into turning environments and of turning into milling environments, begun with live tooling for lathes and the occasional use of mills for turning operations, led to a new class of machine tools, Multitasking Machines (MTMs), which are purpose-built to provide for a default machining strategy of using any combination of milling and turning within the same work envelope. In the vertical mill the spindle axis is vertically oriented. Milling cutters are held in the spindle and rotate on its axis. The spindle can generally be extended (or the table can be raised/lowered, giving the same effect), allowing plunge cuts and drilling. There are two subcategories of vertical mills: the bed mill and the turret mill.

- A turret mill has a stationary spindle and the table is moved both perpendicular and parallel to the spindle axis to accomplish cutting. The most common example of this type is the Bridgeport, described below. Turret mills often have a quill which allows the milling cutter to be raised and lowered in a manner similar to a drill press. This type of machine provides two methods of cutting in the vertical (Z) direction: by raising or lowering the quill, and by moving the knee.
- In the bed mill, however, the table moves only perpendicular to the spindle's axis, while the spindle itself moves parallel to its own axis.

A third type also exists, a lighter machine, called a mill-drill, which is a close relative of the vertical mill and quite popular with hobbyists. A mill-drill is similar in basic configuration to a small drill press, but equipped with an X-Y table. They also typically use more powerful motors than a comparably sized drill press, with potentiometer-controlled speed and generally have more heavy-duty spindle bearings than a drill press to deal with the lateral loading on the spindle that is created by a milling operation. A mill drill also typically raises and lowers the entire head, including motor, often on a dovetailed vertical, where a drill press motor remains stationary, while the arbour raises and lowers within a driving collar. Other differences that separate a mill-drill from a drill press may be a fine tuning adjustment for the Z-axis, a more precise depth stop, the capability to lock the X, Y or Z axis, and often a system of tilting the head or the entire vertical to allow angled cutting. Aside from size and precision, the principal difference between these hobby-type machines and larger true vertical mills is that the X-Y table is at a fixed elevation; the Z-axis is controlled in basically the same fashion as drill press, where a larger vertical or knee mill has a vertically fixed milling head, and changes the X-Y table elevation. As well, a mill-drill often uses a standard drill press-type Jacob's chuck, rather than an internally tapered arbor that accepts collets. These are frequently of lower quality than other types of machines, but still fill the hobby role well because they tend to be benchtop machines with small footprints and modest price tags.

Vertical Machining Center

Objective of the Present Work

The success of any milling operation depends, Before setting up a job, be sure that the to a great extent, upon judgment in setting up the job, workpiece, the table, the taper in the spindle, selecting the proper milling cutter, and holding the cutter by the best means under the circumstances. Some fundamental practices have been proved by experience to be necessary for and the arbor or cutter shank are all clean and good results on all jobs. Some of these practices are mentioned below:

- Before setting up a job, be sure that the work piece, table, the taper in the spindle, and the arbour or cutter shank are free from chips, nicks, or burrs.
- Do not select a milling cutter of larger diameter than is necessary.
- Check the machine to see if it is in good running order and properly lubricated, and that it moves freely, but not too freely in all directions.
- Consider direction of rotation. Many cutters can be reversed on the arbor, so be sure you know whether the spindle is to rotate clockwise or counter clockwise.
- Feed the work piece in a direction opposite the rotation of the milling cutter (conventional milling).
- Do not change feeds or speeds while the milling machine is in operation.
- When using clamps to secure a work piece, be sure that they are tight and that the piece is held so it will not spring or vibrate under cut.

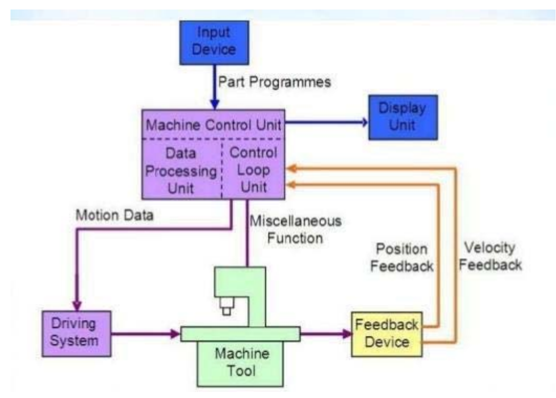
WORKING PROCESS OF VMC

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 workpiece 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 revolution.
- The axial depth of cut, which is the distance the tool is set below an unmachined 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.

Block Diagram of CNC Machine



Milling Machine Basics

The most basic milling machine is the vertical spindle, ram-type "knee" mill. Though not well adapted to production milling, it is ideal for toolmaking and prototype machining. Knee mills are primarily used for manual operations, but their capabilities can be expanded.

The knee travels vertically, up and down the column, and supports the saddle and table. The saddle moves in and out from the column, while the table moves side to side of the column. Additionally, the ram, at the top of the column, supports the milling head which contains the motor, toolhead, speed and feed controls, quill and spindle.

Cutting tools are secured in collets or drill chucks held in the spindle. Work is usually secured to the table using bolts and clamps, or by using vises or fixtures bolted to the table. The work table contains longitudinal "T" slots to facilitate the attachment of these devices. The knee mill's capabilities are expanded by the use of digital readout displays and CNC technology. CNC technology provides three-axis capability to the mill.

Machining Center Accessories and Programming

Machining centers can incorporate two very useful accessories. One is the touch-trigger probe which, with its computer software, will dimensionally check workpiece measurements before removal from the machining center. The probe is stored with other tooling for quick application. The second accessory is the tool presetting machine, which allows the technician to assemble the tooling according to the programmed part requirements before placing tools in the machining center's tool storage. The choice of toolholder itself can be critical. Chucks, collets and other mechanical-interference toolholders use applied clamping pressure to set tools for common milling situations. In recent years, shrink-fit toolholders have become more popular, especially for high-speed machining operations. Here, the toolholder develops uniform clamping pressure as it contracts around the tool shank, after first being heated and expanded. Shrink-fit systems require extra equipment, but the cost may be justified by the higher cuttingspeeds and feeds that they allow.

Basic Process

Preparation

Programming Competitors must know and understand: The different methods and techniques to generate a program Competitors must be able to: Select the best methods according to the production type and part specification. Create a program using G-Codes. Create a program using a CAD/CAM system taking into account the format of the initial data:

Start with a drawing in paper format - creation of the geometry in wireframe and/or surface and/or solid. Start with a file in wireframe and/or surface and/or solid and import it to CAD/CAM system Using the parametric programming system.Ensure that the x-y table is level and aligned.

Milling Operations

Milling operations may be classified under four general headings as follows:

- Face milling: Machining flat surfaces which are at right angles to the axis of the cutter.
- Plain or slab milling: Machining flat surfaces which are parallel to the axis of the cutter.

- Angular milling: Machining flat surfaces which are at an inclination to the axis of the cutter.
- Form milling: Machining surfaces having an irregular outline.

SPECIAL OPERATIONS

Explanatory names, such as sawing, slotting, gear cutting, and so forth have been given to special operations. Routing is a term applied to milling an irregular outline while controlling the workpiece movement by hand feed. Grooving reamers and taps is called fluting. Gang milling is the term applied to an operation in which two or more milling cutters are used together on one arbor. Straddle milling is the term given to an operation in which two milling cutters are used to straddle the workpiece and mill both sides at the same time.

Speeds for Milling Cutters

The speed of milling is the distance in FPM at which the circumference of the cutter passes over the work. The spindle RPM necessary to give a desired peripheral speed depends on the size of the milling cutter. The best speed is determined by the kind of material being cut and the size and type of cutter used, width and depth of cut, finish required, type of cutting fluid and method of application, and power and speed available are factors relating to cutter speed.

Factors Governing Speed

There are no hard and fast rules governing the speed of milling cutters; experience has shown that the following factors must be considered in regulating speed:

- A metal slitting saw milling cutter can be rotated faster than a plain milling cutter having a broad face.
- Cutters having undercut teeth (positive rake) cut more freely than those having radial teeth (without rake); hence, they may run at higher speeds.
- Angle cutters must be run at slower speeds than plain or side cutters.
- Cutters with inserted teeth generally will stand as much speed as a solid cutter.
- A sharp cutter may be operated at greater speeds than a dull one.
- A plentiful supply of cutting oil will permit the cutter to run at higher speeds than without cutting oil.

Selecting Proper Cutting Speeds

The approximate values A may be used as a guide for selecting the proper cutting speed. If the operator finds that the machine, the milling cutter, or the workpiece cannot be handled suitably at these speeds, immediate readjustments should be made.

Table lists speeds for high-speed steel milling cutters. If carbon steel cutters are used, the speed should be about one-half the recommended speed in the table. If carbide-tipped cutters are used, the speed can be doubled.

If a plentiful supply of cutting oil is applied to the milling cutter and the workpiece, speeds can be increased 50 to 100%. For roughing cuts, a moderate speed and coarse feed often give best results; for finishing cuts, the best practice is to reverse these conditions, using a higher speed and lighter feed.

Speed Computation

The formula for calculating spindle speed in revolutions per minute is as follows:

$$RPM = \frac{CS \times 4}{D}$$

where RPM = Spindle speed (in revolutions per minute).

CS = cutting speed of milling cutter (in SFPM)

D = diameter of milling cutter (in inches)

For example, the spindle speed for machining a piece of steel at a speed of 35 SFPM with a cutter 2 inches in diameter is calculated as follows:

$$RPM = \frac{CS \times 4}{D} = \frac{35 \times 4}{2} = \frac{xxx}{xx} = xxxx$$

Therefore, the milling machine spindle would be set for as near 70 RPM as possible.

DIFFERENT TYPES OF TOOLS USED

Features of A Milling Cutter

Features of a milling cutter Milling cutters come in several shapes and many sizes. There is also a choice of coatings, as well as rake angle and number of cutting surfaces.

Shape: Several standard shapes of milling cutter are used in industry today, which are explained in more detail below.

Flutes / Teeth: The flutes of the milling bit are the deep helical grooves running up the cutter, while the sharp blade along the edge of the flute is known as the tooth. The tooth cuts the material, and chips of this material are pulled up the flute by the rotation of the cutter. There is almost always one tooth per flute, but some cutters have two teeth per flute Often, the words flute and tooth are used interchangeably. Milling cutters may have from one to many teeth, with 2, 3 and 4 being most common. So, a 4-tooth cutter can remove material at twice the rate of a 2-tooth cutter.

Helix Angle: The flutes of a milling cutter are almost always helical. If the flutes were straight, the whole tooth would impact the material at once, causing vibration and reducing accuracy and surface quality.

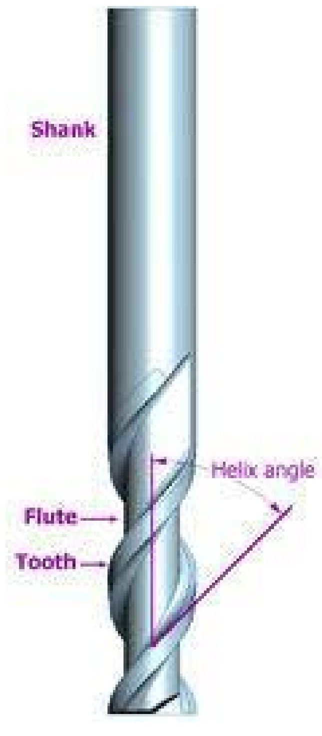
Center Cutting: Some milling cutters can drill straight down (plunge) through the material, while others cannot. This is because the teeth of some cutters do not go all the way to the center of the end face. However, these cutters can cut downwards at an angle of 45^o or so.

Roughing Or Finishing: Different types of cutter are available for cutting away large amounts of material, leaving a poor surface finish (roughing), or removing a smaller amount of material, but leaving an end mill cutter with two flutes a good surface finish (finishing). A roughing cutter may have serrated teeth for breaking the chips of material into smaller pieces. These teeth leave a rough surface behind. A finishing cutter may have a large number (4 or more) teeth for removing material carefully. However, the large number of flutes leaves little room for efficient swarf removal, so they 1 2 2 types are less appropriate for removing large amounts of material.

Coatings: The right tool coatings can have a great influence on the cutting process by increasing cutting speed and tool life, and improving the surface finish. Polycrystalline Diamond (PCD) is an exceptionally hard coating used on cutters which must withstand high abrasive wear. A PCD coated tool may last up to 100 times longer than an uncoated tool. However the coating cannot be used at temperatures above 600^oC, or on ferrous metals. Tools for machining aluminium are sometimes given a coating of TiAlN. Aluminium is a relatively sticky metal, and can weld itself to the teeth of tools, causing them to appear blunt. However it tends not to stick to TiAlN, allowing the tool to be used for much longer in aluminium

Shank: The shank is the cylindrical (non-fluted) part of the tool which is used to hold and locate it in the tool holder. A shank may be perfectly round, and held by friction, or it may have a Weldon flat, where a set screw, also known as a grub screw, makes contact for increased torque without the tool slipping. The diameter may be different from the diameter of the cutting part of the tool, so that it can be held by a standard tool holder.

Figure 4.1: An End Mill Cutter with Two Flutes



END MILL

End mills (middle row in image) are those tools which have cutting teeth at one end, as well as on the sides. The words end mill are generally used to refer to flat bottomed cutters, but also include rounded cutters (referred to as ball nosed) and radiused cutters (referred to as bull nose, or torus). They are usually made from high speed steel or cemented carbide, and have one or more flutes. They are the most common tool used in a vertical mill.

Roughing end mill Roughing end mills quickly remove large amounts of material. This kind of end mill utilizes a wavy tooth form cut on the periphery. These wavy teeth form many successive cutting edges producing many small chips, resulting in a relatively rough surface finish. During cutting, multiple teeth are in contact with the workpiece reducing chatter and vibration. Rapid stock removal with heavy milling cuts is sometimes called hogging. Roughing end mills are also sometimes known as ripping cutters.Ball nose cutter Ball nose cutters or ball end mills (lower row in image) are similar to slot drills, but the end of the cutters are hemispherical. They are ideal for machining 3-dimensional contoured shapes in machining centres, for example in moulds and dies. They are sometimes called ball mills in shop-floor slang, despite the fact that that term also has another meaning. They are also used to add a radius between perpendicular faces to reduce stress concentrations.

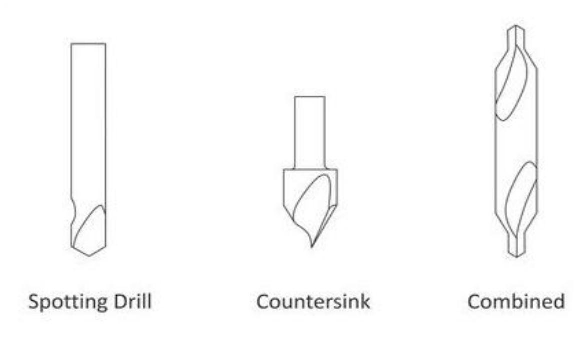
FACE MILL

A face mill is a cutter designed for facing as opposed to e.g., creating a pocket (end mills). The cutting edges of face mills are always located along its sides. As such it must always cut in a horizontal direction at a given depth coming from outside the stock. Multiple teeth distribute the chip load, and since the teeth are normally disposable carbide inserts, this combination allows for very large and efficient face milling.

HOLE-MALING TOOLS

Center (spotting) drills are short and very rigid drills used to create a conic on the face of the part. Because they come to a sharp point and resist bending, they locate the hole precisely. The conic helps prevent the subsequent drill from wobbling and ensure the drill is located precisely and drills straight down.Countersink drills are used to create the conical face for a machine screw. Combined spotting-countersinks are used to create a screw clearance hole and countersink in one operation. There are many different sizes and tip angles of center, countersink, and combined drills. Be sure the tip angle of the countersink matches the included angle of the machine screw, and that the drill diameter is greater than the screw head diameter.

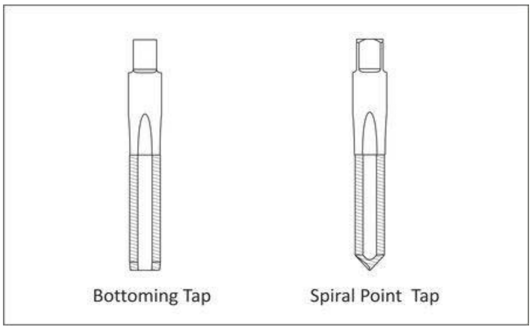
Countersink and Center Drill



TAPS

Cutting taps form threads by shearing material away. Form taps (roll taps) form the thread by forming the metal to shape. Form taps produce no chips and are used for soft materials including aluminum, copper, brass and plastics. Bottoming taps are used to tap blind holes. Spiral point taps push the chip ahead and out the bottom of a through hole. Taps require a hole drilled to the correct size to ensure the thread is formed properly. For example, a ¼-20 cutting tap requires drilling a .201 (#7) hole. Refer to the drill chart in to find the correct drill size for a specified thread size and fit. Most CNC machines support rigid tapping, which means the tap can be held in a rigid holder. The tap is advanced at a feed rate that matches the thread lead into the hole. The spindle then stops, reverses, and backs out of the hole. Machines without rigid tapping require special tapping attachments. Always refer to the manufacturers' instructions as the speed, feed, and other machining parameters for tapping attachments may be different that those for rigid tapping.

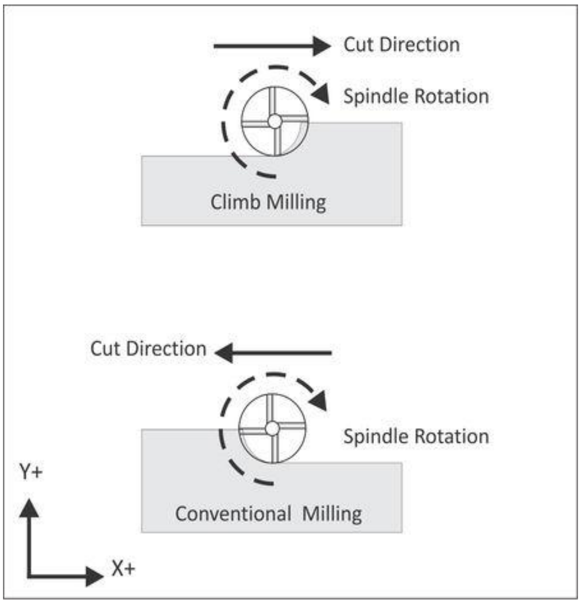
Figure 4.4: Taps



CUTTING TOOL FUNDAMENTALS

Milling tools can advance through the material so that the cutting flutes engage the material at maximum thickness and then decreases to zero. This is called Climb Milling. Cutting in the opposite direction causes the tool to scoop up the material, starting at zero thickness and increasing to maximum. This is called Conventional Milling. Conventional milling is used often on manual machines because backlash in the machine lead screws causes the tool to lurch when climb cutting. Raking chips across the finished surface also produces a poorer surface finish.Unless specifically recommended by the tool manufacturer for the material being milled, always use climb milling on a CNC. Climb milling produces far less cutting pressure and heat, leaves a better surface finish, and results in longer tool life.

Figure 4.5: Climb vs. Conventional Milling



CUTING DATA

Tables on the following pages provide basic speed, feed and cutting data for some of the materials commonly used for prototypes. Use the tool manufacturer's data instead whenever it is available.

VERTICAL MILLING CENTER MACHINE MOTION

CNC machines use a 3D Cartesian coordinate system. Figure 4.4 shows a typical Vertical Milling Center (VMC) with the sheet metal covers removed to expose the movable parts. Material to be machined is fastened to the machine table. This table moves in the XY-plane. As the operator faces the machine, the X-axis moves the table left-right. The Y-axis moves the table forward-backward. The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

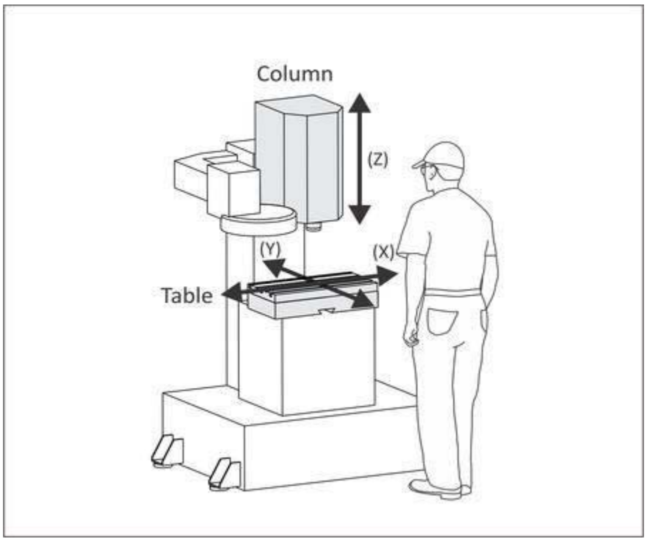
Table 4.1: Milling Speed Data (SFM)

Mill Cutting Speeds (SFM) SURFACE FT/MIN		
Material	**HSS**	**CARBIDE**
Aluminum	600	800
Brass	175	175
Delrin	400	800
Polycarbonate	300	500
Stainless Steel (303)	80	300
Steel (4140)	70	350

Drill Cycles Speed Data (SFM)				
Drill Cutting Speeds (SFM) SURFACE FT/MIN				
Material	**Drilling**	**C-Sink**	**Reamer**	**Tap**
Aluminum	300	200	150	100
Brass	120	90	66	100
Delrin	150	100	75	100
Polycarbonate	240	160	120	100
Stainless Steel (303)	50	35	25	35
Steel (4140)	90	60	45	35

Feed Data (IPR)					
Cutting Feeds (IPR) IN/REV					
Operation	**Tool Diameter Range (IN)**				
Milling	**<.125**	**.125-.25**	**.25-.5**	**.5-1.**	**>1.**
Aluminum	.002	.002	.005	.006	.007
Brass	.001	.002	.002	.004	.005
Delrin	.002	.002	.005	.006	.007
Polycarbonate	.001	.003	.006	.008	.009
Stainless Steel (303)	.0005	.001	.002	.003	.004
Steel (4140)	.0005	.0005	.001	.002	.003

VMC Machine Motion



CNC MOTION CONTROL

Most CNC machines can position each axis within .0002 inches or less over the entire machining envelope. This accuracy is achieved in part by the use of a closed-loop servo mechanism, The machine control sends a motion signal, via a controller board, to a servomotor attached to each machine axis. This causes the servomotor to rotate a ball screw attached to the table or column, causing it to move. The actual position of the axis is continuously monitored and compared to the commanded position with feedback from a servo transmitter attached to the ball screw. Ball screws have almost no backlash, so when the servo reverses direction there is almost no lag between a commanded reversing motion and corresponding change in table direction. CNC controls employ electronic compensation to adjust for any minor backlash that may exist.

WORKING COORDINATE SYSTEM

Obviously it would be difficult to write a CNC program in relation to machine coordinates. The home position is far away from the table, so values in the CNC program would be large and have no easily recognized relation to the part model. To make programming and setting up the CNC easier, a Work Coordinate System (WCS) is established for each CNC program.

The WCS is a point selected by the CNC programmer on the part, stock or fixture. While the WCS can be the same as the part origin in CAD, it does not have to be. While it can be located anywhere in the machine envelope, its selection requires careful consideration.

- The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.
- It must be located with high precision: typically plus or minus .001 inches or less.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should take into account how the part will be rotated and moved as different sides of the part are machined.

The outside dimensions of the part have already been milled to size on a manual machine before being set on the CNC machine. The CNC is used to make the holes, pockets, and slot in this part. The WCS is located in the upper-left corner of the block. This corner is easily found using an Edge Finder

TOOL LENGTH OFFSET

Every tool loaded into the machine is a different length. In fact, if a tool is replaced due to wear or breaking, the length of its replacement will likely change because it is almost impossible to set a new tool in the holder in exactly the same place as the old one. This value is entered in the TLO register for that tool. Problems with this method include the need to face mill the part to the correct depth before setting tools. Also, if the Z-datum is cut away (typical of 3D surfaced parts) it is impossible to set the datum should a tool break or wear and need to be replaced. All tools must be reset whenever a new job is set up. When this method is used, the Fixture Offset Z is not used, but set to zero. The method shown in the center is much better and used in this book. All tools are set to a known Z-position, such the top of a precision 1-2-3 block resting on the machine table. This makes it very easy to reset tools if worn or broken. A tool probe is very similar to the 1-2-3 block method, except the machine uses a special cycle to automatically find the TLO. It does this slowly lowering the tool until the tip touches the probe and then updates the TLO register. This method is fast, safe and accurate but requires the machine be equipped with a tool probe. Also, tool probes are expensive so care must be taken to never crash the tool into the probe.

Both the 2nd and 3rd methods also require the distance from the tool setting position (the top of the 1-2-3 block or tool probe) to the part datum to be found and entered in the Fixture Offset Z. The machine adds the two values together to determine the total tool length offset.

CODES AND STRUCTURE

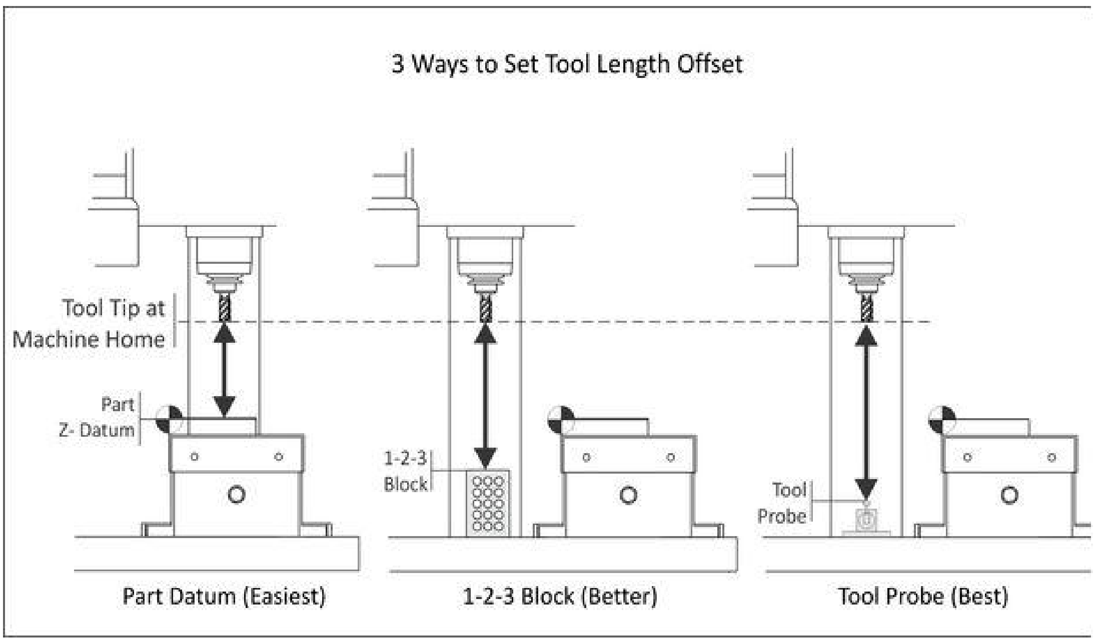
CNC programs list instructions to be performed in the order they are written. They read like a book, left to right and top-down. Each sentence in a CNC program is written on a separate line, called a Block. Blocks are arranged in a specific sequence that promotes safety, predictability and readability, so it is important to adhere to a standard program structure.

Typically, blocks are arranged in the following order:

- Program Start
- Load Toole
- Spindle On
- Coolant On
- Rapid to position above part
- Machining operation
- Coolant Off
- Spindle Off
- Move to safe position
- End program

The steps listed above represent the simplest type of CNC program, where only one tool is used and one operation performed. Programs that use multiple tools repeat steps two through nine for each.on section G and M codes show the most common G and M codes that should be memorized if possible.Like any language, the

Ways to Set TLO



G-code language has rules. For example, some codes are modal, meaning they do not have to be repeated if they do not change between blocks. Some codes have different meanings depending on how and where there are used.

While these rules are covered in this chapter, do not concern yourself with learning every nuance of the language. It is the job of the job of the CAD/CAM software Post Processor to properly format and write the CNC program.

PROGRAM FORMATE

G and M CODES

G CODES

Codes that begin with G are called preparatory words because they prepare the machine for a certain type of motion. The most common G-codes are shown in Table 5.3.

M CODES

Codes that begin with M are called miscellaneous words. They control machine auxiliary options like coolant and spindle direction. Only one M-code can appear in each block of code.The table below lists the most common M codes and their meaning. A complete list of M-codes is included in.

ALPHABETIC AND SPECIAL ADDRESS CODES

Every letter of the alphabet is used as a machine address code. In fact, some are used more than once, and their meaning changes based on which G-code appears in the same block.Codes are either modal, which means they remain in effect until cancelled or changed, or non-modal, which means they are effective only in the current block.

The table below lists the most common address codes.

Table 5.2: Common G-Codes

Code	Meaning
G0	Rapid motion. Used to position the machine for non-milling moves.
G1	Line motion at a specified feed rate.
G2	Clockwise arc.
G3	Counterclockwise arc.
G4	Dwell.
G28	Return to machine home position.
G40	Cutter Diameter Compensation (CDC) off.
G41	Cutter Diameter Compensation (CDC) left.
G42	Cutter Diameter Compensation (CDC) right.
G43	Tool length offset (TLO).
G54	Fixture Offset #1.
G55	Fixture Offset #2.
G56	Fixture Offset #3.
G57	Fixture Offset #4.
G58	Fixture Offset #5.
G59	Fixture Offset #6.
G80	Cancel drill cycle.
G81	Simple drill cycle.
G82	Simple drill cycle with dwell.
G83	Peck drill cycle.
G84	Tap cycle.
G90	Absolute coordinate programming mode.
G91	Incremental coordinate programming mode.
G98	Drill cycle return to Initial point (R).
G99	Drill cycle return to Reference plane (last Z Height)

ADVANTAGES

- CNC machines can be used continuously 24 h a day, 365 days a year and only need to be switched off for occasional maintenance.

Table 5.3: Common M-Codes

Code	Meaning
M0	Program stop. Press Cycle Start button to continue.
M1	Optional stop. Only executed if Op Stop switch on the CNC control is turned ON.
M2	End of program.
M3	Spindle on Clockwise.
M4	Spindle on Counterclockwise.
M5	Spindle stop.
M6	Change tool.
M8	Coolant on.
M9	Coolant off.
M30	End program and press Cycle Start to run it again.

- CNC machines are programmed with a design which can then be manufactured hundreds or even thousands of times. Each manufactured product will be exactly the same.
- Less skilled/trained people can operate CNCs unlike manual lathes / milling machines, etc., which need skilled engineers.

DISADVANTAGES

- CNC machines are more expensive than manually operated machines, although costs are slowly coming down.
- The CNC machine operator only needs basic training and skills, enough to supervise several machines. In years gone by, engineers needed years of training to operate centre lathes, milling machines and other manually operated machines. This means many of the old skills are been lost.
- Less workers are required to operate CNC machines compared to manually operated machines. Investment in CNC machines can lead to unemployment.

Table 5.4: Common Alphanumeric Address Codes

Code	Meaning
A	Rotation about X-axis.
B	Rotation about Y-axis.
C	Rotation about Z-axis.
D	Cutter diameter compensation (CDC) offset address.
F	Feed rate.
G	G-Code (preparatory code).
H	Tool length offset (TLO).
I	Arc center X-vector, also used in drill cycles.
J	Arc center Y-vector, also used in drill cycles.
K	Arc center Z-vector, also used in drill cycles.
M	M-Code (miscellaneous code).
N	Block Number.
O	Program Number.
P	Dwell time.
Q	Used in drill cycles.
R	Arc radius, also used in drill cycles.
S	Spindle speed in RPM.
T	Tool number.
X	X-coordinate.
Y	Y-coordinate.
Z	Z-coordinate.

CONCLUSION

The machine tool industry, ably lead by HMT, continuously grew both in content and quality upto the seventies. The industry not only achieved a high degree of self reliance, but also made a good beginning in entering export markets. With the advent of CNC technology, though initial efforts at development were made by HMT, the Indian industry could not match the international competition by way of features and reliability of systems. Hence, not only did Indian Machine Tool Industry lose its growing export market, but it became inevitable for the user industry to resort to import of machine tools.

THE FOLLOWING CONCLUSIONS ARE OBSERVED AS FOLLOWS

1. The interaction of machine/work axes and work handling devices, have enabled upto five faces of a work to be finished without setting changes.
2. Tool changers, holding upto 120 tools and chip to chip tool change time of 2 s, have been achieved.
3. Higher spindle speed and higher feed rates have enabled use of ceramic tools, for optimum material removal rate, with very good surface finishes.
4. The latest CNC controls are capable of acceptance of program in conversational language, in addition to normal machine tool programming lanuage, improving thus the programming ease.

BIBILOGRAPHY

1. Noble David F (1984), *Forces of Production: A Social History of Industrial Automation*, New York, New York, USA: Knopf.
2. Pease William (1952), "An automatic machine tool", *Scientific American,* Vol. 187, No. 3.
3. Roe Joseph Wickham (1916), *English and american tool builders New Haven*, Connecticut: Yale University Press, . Reprinted by McGraw-Hill, New York and London, and by Lindsay Publications, Inc., Bradley, Illinois.
4. Roe Joseph Wickham (1918), *The mechanical equipment,*Vol. 3, Factory Management Course, New York, USA: Industrial Extension Institute, Inc.
5. Usher John T (1896), *The modern Machinist* (2nd ed.), N. W. Henley. Retrieved 2013-02-01.

International Journal of Engineering Research and Science & Technology
Hyderabad, INDIA. Ph: +91-09441351700, 09059645577
E-mail: editorijlerst@gmail.com or editor@ijerst.com
Website: www.ijerst.com

ISSN 2319-5991