Machining

Level-III

Based on October 2023, Curriculum Version II



Module Title: Perform Advanced CNC Milling Operations

Module Code: IND MAC3 03 1023
Nominal Duration: 120 Hours

Prepared by: Ministry of Labor and Skill

October 2023

Addis Ababa, Ethiopia

Acknowledgment

The Ministry of Labor and skill wishes to thank and appreciation to MoLS leaders and experts, Regional Labor and skill/training Bureaus leader, experts, TVT College Deans, Instructors and industry experts who contribute their time and professional experience to the development of this Training Module.

Acronym

Al_2O_3 ----------Coating - Aluminum (oxide)

CNC ----------Computer Numerical Control

CrC ----------Coating - Chromium carbide

CrN ----------Coating - Chromium nitride

CVD ----------Chemical vapor deposition

CVD ----------Chemical vapor deposition

DNC ----------Direct Numerical Control

FPT ----------Feed per tooth

HSS: ----------HIGH Speed Steels

IPR ----------Inches per rev

PCBN ----------Polycrystalline cubic boron nitrides

PCD ---------- Polycrystalline diamond

PVD). ----------physical vapor deposition

PVD. ----------physical vapor deposition

SOPs ----------Standard operating procedures

TiC coating ----------Titanium carbide

TiCN coating - ----------Titanium carbon nitride

TiN coating - ----------Titanium nitride

VMC ----------Vertical Machining Centre

የሥራና ክህሎት ሚኒስቴር
Ministry of Labor and Skills

Table of Contents

Acknowledgment ii

Acronym iii

Introduction to Module 1

Unit one: Fundamentals of Milling CNC ii

Sequence of operation 19

Cutting tools. 22

Cutting parameters 30

Processing/job/adjustment sheets 36

Jigs and fixture 38

Tool and die type 43

Safety procedures and environmental protections 44

Self-check 1. 45

Lap Test 1. 47

Unit Two: CNC milling machine program 48

CNC Milling Coordinates system 48

CNC milling operation program, Simulating, and editing Program 55

CNC Milling Machine Operations 62

Document and save Program to the machine 68

Upload the Program 69

Self-check 2 71

Operation Sheet: -2 72

Lap test 2 74

Unit three: - CNC milling operations 75

Standard operating procedures 76

Machine zero and work zero 78

Dry run with the desired tool path movements 82
CNC milling DNC 84
Corrective measures/adjustments 84
Self-Check 3 88
Operation Sheet: -3 90
Lap Test 3 92
Unit four: - Select materials 93
Introduction to material selection 93
Manufacturing process plan 94
Materials test 96
Self-check 4.1. 99
Operation Sheet: -4.1 100
Lap Test 4 102
CNC milling Machining process 103
Test run. 104
Self- check **5**.1 106
Operation Sheet **5**.1 107
Lap Test 5. 108
Unit six: - Quality assurance 109
Methods, measuring tools and equipment's 110
Defective works 116
Product quality of CNC production 116
Self-check 6**.1.** 118
Operation Sheet: -6.1. 119
Lap Test 6.1. 120

Reference 121

Introduction to Module

In machining field, Performing Advanced CNC Milling Operations is especially important for the overall health of the machine. It helps to know basic Advanced CNC Milling Operations concepts, perform advanced CNC milling operations conforming to the required specifications, how to write programing on the machine.

This module is designed to meet the industry requirement under the Machining occupational standard, particularly for the unit of competency: Advanced CNC Milling Operations

This module covers the units:

- **Fundamentals of Milling CNC**
- **CNC milling machine program**
- **CNC milling operations**
- **Select materials**
- **Produce and assemble components**
- **Quality assurance**

Learning Objective of the Module

- Determine job requirements
- Write or load CNC milling machine program
- Perform milling operations
- Select materials
- Produce and assemble components
- Comply with Quality assurance

Module Instruction

For effectively use these modules trainee are expected to follow the following module instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the “LAP test” giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

Unit one: Fundamentals of Milling CNC

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Information of CNC milling machine
- Sequence of operation
- Cutting tools.
- Cutting parameters
- Processing/job/adjustment sheets.
- Jigs and fixture
- Tool and die type
- Safety procedures and environmental protections

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Interpret drawings to produce components.
- Determine Sequence of operation to produce component.
- Select cutting tools.
- Calculate cutting speed and feed rate
- Fill up process/job/adjustment sheets.
- Select Standardized jigs and fixture, and tool and die components
- Determine and design jigs and fixture
- Visualize and interpret jigs and fixtures.
- Conceptualize and plan tool and die type

What is CNC Milling?

CNC milling is a machining process that uses computerized controls and cutting tools to selectively remove material from a work piece and produce a custom-designed part. A CNC milling machine features a table that automatically moves (or rotates) the work piece on different planes so the cutting tool can work on it.

Unlike manual lathes, a CNC milling machine operates by converting a 3D CAD model into a set of computer instructions, which is then used to dictate the tooling's actions and movements automatically.

What are the Advantages of CNC Milling?

1 **Precision**

One of the biggest advantages of using CNC milling machines is that they can create parts exactly to specification. Because CNC mills rely on computer instructions to fabricate parts, they eliminate the possibility of human error common in manually operated machines. This means you can accurately create complex parts while achieving tolerances as tight as 0.004mm.

2 **Fast and Efficient**

Conventional milling machines often require an operator to manually change cutting tools depending on the cutting operation to be performed. Not only is this time-consuming, but it is also inefficient as end results are based on the operator's judgment.

CNC mills feature rotating carousels that can hold up to 30 different tools. These tools are automatically exchanged on the spindle during machining operations, allowing cutting operations to be performed quickly and efficiently. With CNC mills, you can have your parts ready in a matter of hours, whereas conventional methods may take days (or weeks) to complete.

3. Wide Range of Material Options

CNC mills boast compatibility with a wide range of materials, including plastics, metals, and composites. So long as you've got blocks of the material, CNC mills will have no problem machining it.

4. Affordability

We can talk about precision, speed, and a ton of other advantages all day, but nothing beats affordability, especially for businesses looking to rely on third-party manufacturers. CNC milling is among the most affordable modern-day manufacturing processes. In fact, a CNC milled component would cost you less than a comparable 3D printed part.

What are the Disadvantages of CNC Milling?

1 **Material wastage is more in CNC Milling**

CNC milling is a subtractive manufacturing method-it starts with a block of material and cuts away portions of it to form the finished part. As a result, there is more material wastage in CNC milling than in additive manufacturing processes, such as 3D printing.

2 **There are not Enough Qualified Technicians**

Although pretty much the whole milling process is automated, CNC milling still requires highly trained engineers or technicians to program mills, calculate requirements and supervise the mills so that high-quality parts are always produced.
Not so many machine shops can boast of highly qualified CNC mill operators; as a result, there have been many horror stories about low-quality machined parts.

CNC stands for Computerized Numerical Control. It is a computerized manufacturing process in which pre-programmed software and code controls the movement of production equipment. CNC machining controls a range of complex machinery, such as grinders, lathes, and turning mills, all of which are used to cut, shape, and create different parts and prototypes. On the day to day, CNC machinists combine elements of mechanical design, technical drawings, mathematics, and computer programming skills to produce a variety of metal and plastic parts. CNC operators can take a sheet of metal and turn it into a critical airplane or automobile part.

Milling machines can perform a wide range of operations, including face milling, peripheral milling, slotting, drilling, and boring. They can create various shapes, contours, and features on a work piece, such as flat surfaces, grooves, threads, and complex 3D geometries. The ability to perform multiple operations in one setup makes milling machines highly efficient and productive.
CNC milling is a manufacturing process that uses a cutting tool mounted on a rotating spindle to selectively remove material from a block of raw substrate. The work piece is rigidly mounted to a table, and this table moves or rotates on different planes so the tool can work at many angles.

This type of code instructs machine tools to cut and shape materials using precise production measurements such as feed rate, speed, location, and coordination. Typical tolerances range between 0.001 to 0.005 inches, with some milling machines achieving tolerances of up to 0.0005 inches.

Computer Numerical control may be considered to be a means of operating a machine through the use of discrete numerical values fed into the machine, where the required 'input' technical information is stored on a kind of input media such as floppy disk, hard disk, CD ROM, DVD, USB flash drive, or RAM card etc. the machine follows a predetermined sequence of machining operations at the predetermined speeds necessary to produce a work piece of the right shape and zise and thus according to completely predictable results. A different product can be produced through reprogramming and a low-quality production run of different products is justified

1.1.1. Drawing component

A. A technical CNC drawing is a detailed and precise representation of a design created using Computer-aided design (CAD) software. CNC, or computer numerical control, refers to the process of using computers to control machines that create the design. The technical drawings for CNC machining typically include all the necessary measurements, materials, and dimensions, and serve as a blueprint for the manufacturing or production process. The software also allows for complex shapes, multiple angles, and intricate details to be incorporated into the design, making it extremely precise and accurate. Technical drawings for CNC machining are essential in industries such as engineering, manufacturing, and architecture, where precision and accuracy are critical for creating high-quality products.

Technical drawing involves the use of various tools and techniques to accurately and precisely create diagrams, blueprints, and other visual representations of mechanical or architectural designs. Here, we will take a brief look at some key components of technical drawings for CNC machining.

Coordinates

Coordinates are a set of numbers or letters that correspond to specific points on a grid or plane. In technical drawing for CNC machining, designers use two primary types of coordinates to create accurate and detailed designs. Those are

- Cartesian coordinates
- Polar coordinates.

Cartesian coordinates use horizontal and vertical lines to determine the position of objects, while polar coordinates use angles and distances to specify the location of objects. Both types of coordinates are necessary components of technical drawing, allowing designers to create precise and detailed representations of their designs

Drawings show the machinist what to make and identify the standards that must be followed so the various parts will fit together properly. The resulting parts will also be interchangeable with similar components on equipment already in service.

An especially important skill needed for success in the machining field is the ability to interpret engineering drawings, or prints. Engineering drawings show the sizes and shapes of components and their specific features, such as holes, slots, or surfaces. No matter how skilled you are at performing machining operations, if you are unable to properly interpret these drawings, you will not be able to produce machined components independently or efficiently within required specifications.

Engineering drawings can range from simple hand-drawn sketches to complex, computer-generated prints. They are produced in a standard format that enables machinists anywhere to understand them. Printed drawing sizes range from (A4) 210 × 297up to (A0) 841 × 1189 millimeters. Drawings for overly complex components may contain several sheets..

- **Title Block**

One major component of a drawing is the title block. The title block includes information such as the part name and number, tolerances, scale, material that the part should be made from, and any required heat treatment. Engineer and draftsperson names and drawing creation date are also usually included. A history of revisions, or changes, can be shown above the title block as well in a box often called the revision block. A revision block may also be shown in the upper-right corner of a print.

- ***Scale*** is the size of an actual object related to its size drawn on a print.

In all cases, regardless of scale, drawing dimensions shown are actual part sizes.

- **Bill of materials**: Sometimes a print may have a bill of materials. A bill of materials can either list the raw materials used to make the machined part or list components that are assembled to produce the part specified on the print.

A sample engineering drawing with the title block, revision block, and bill of materials labeled is shown in figure below.

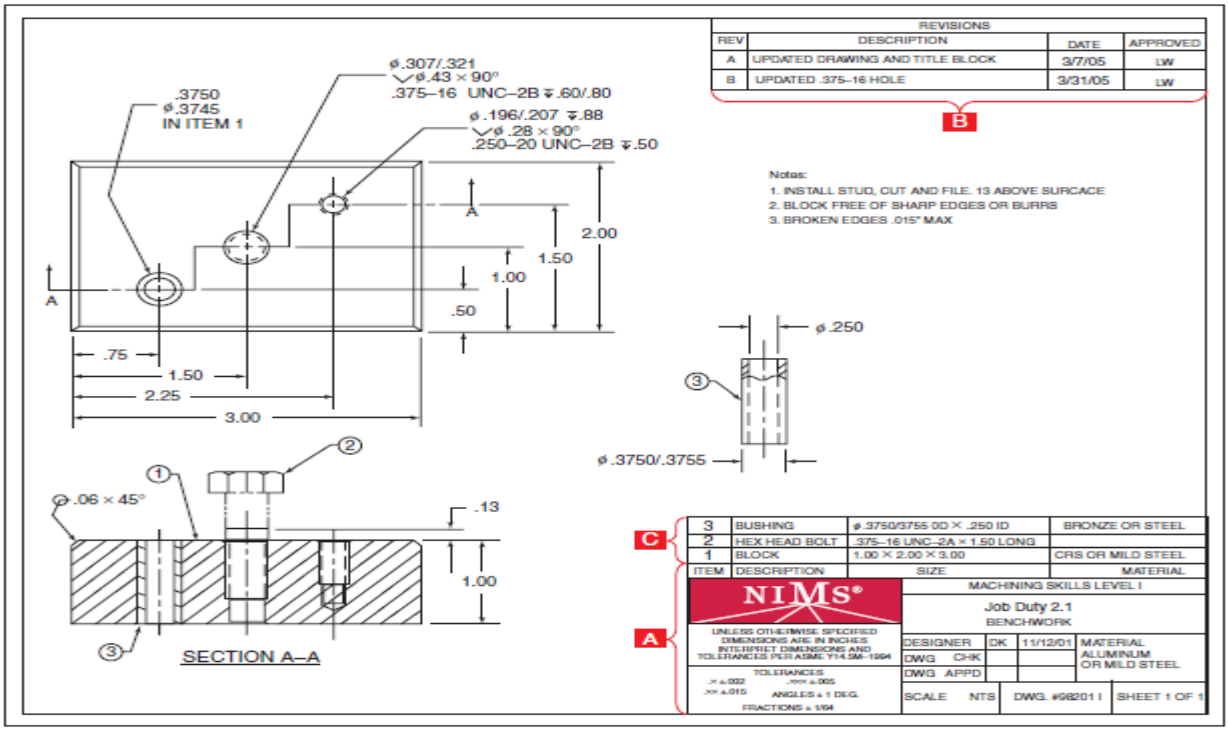


Figure1.1.The title block (A) of an engineering drawing, or print, contains information including part name, tolerances, and scale. The revision block (B) shows a list of changes that have been made over time. The bill of materials (C) lists raw material or components used to produce the part.

- **Orthographic Projection**

These views show the top, front, and side views of a three-dimensional object and are necessary for accurately creating a design for CNC machining. Without these views, it would be difficult for designers and engineers to communicate critical dimensions and shapes necessary for manufacturing the object with precision. Orthographic views also provide valuable information on the placement of holes, slots, and other features, as well as the overall dimensions of the object. With detailed orthographic views, CNC machines can produce complex parts with a high degree of accuracy and consistency.

Section Views

These views allow for a better understanding of the internal workings of a component, which is essential for ensuring accurate and efficient machining. It shows cross-sections of the component at key points. This can help identify any design flaws or issues that may affect the functionality of the component. This view can also help to optimize machining processes and ensure that the finished product meets all the necessary specifications. By including section

views in technical drawings for CNC machining, engineers and machinists can work together to produce high-quality, precise components that meet or exceed expectations.

Detail Views

They allow for a closer look at specific features of a part or assembly and provide important information on dimensions, tolerances, and surface finishes. With CNC machining, precision is key, and detail views enable engineers and machinists to ensure that every detail of a part is accurate and up to standard. Detail views also allow for clear communication between designers and manufacturers, ensuring that everyone is on the same page and working towards the same goal. By incorporating detailed views into technical drawings, designers can ensure that their parts are made exactly as they are intended, with no room for error..

What is the Need of Technical Drawings for CNC Machining?

Technical drawings are an essential aspect of CNC machining, as it allows engineers and technicians to precisely manufacture complex and detailed parts.

- ✓ These drawings act as blueprints for CNC machines, providing the necessary information on the dimensions, tolerances, and materials required for the final product to be created.
- ✓ Without technical drawings, CNC machining would be an unreliable and difficult process, as there would be too much room for error and uncertainty.
- ✓ They allow for clear and concise communication between the designer, the engineer, and the machinist, ensuring that everyone involved in the process is on the same page.
- ✓ Technical drawings can be stored and referred to for future use or modification, making them an important part of any manufacturing operation. Drawings can be created in a variety of different formats and styles, depending on the specific needs of the project.
- ✓ Technical drawings for CNC machining can be hand-drawn or scanned into digital files, and they may either be scaled down to a smaller size for use on CNC machines or enlarged for easier viewing. Regardless of their format, technical drawings must always include accurate information regarding dimensions, tolerances, material types, and specifications, as well as other necessary details

B. Interpret tolerance, limits and fit

- **Tolerance**

Recall, from the measurement section, that a tolerance is an allowable variation from a given size. A dimension shown on a print is called a basic size. A tolerance is applied to the basic size to determine the largest and smallest acceptable size for a dimension. The largest acceptable size

is often called the high limit, or upper limit. The smallest acceptable size is often called the low limit, or lower limit. The difference between the upper limit and the lower limit is the total tolerance. The amount of a tolerance can vary greatly depending on the intended use of a machined component, but the principle of interpreting any tolerance is the same.

- ✓ **Bilateral Tolerances**

A bilateral tolerance allows a dimension to vary both above and below basic size. A bilateral tolerance can take two forms. It can vary by equal amounts above and below basic size or by different amounts above and below basic size. The allowable amount above basic size is shown with a "+" symbol. The allowable amount below basic size is shown with a "-" symbol. When both amounts are equal, a "±" symbol is used.

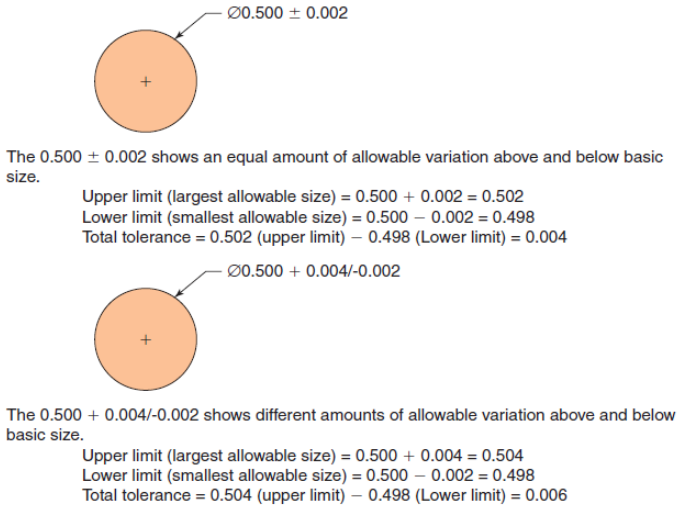
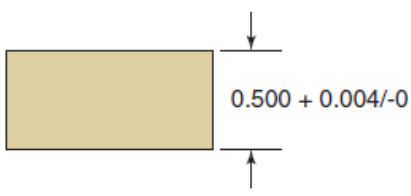


Figure1.1. Bilateral tolerance examples

- ✓ **Unilateral Tolerances**

A unilateral tolerance allows a dimension to vary either above or below basic size, but not both. A unilateral tolerance is also shown using a "+" or "-" symbol, but one amount is "0," indicating no variation is allowable in that direction.



Upper limit (largest allowable size) = 0.500 + 0.004 = 0.504
Lower limit (smallest allowable size) = 0.500 - 0 = 0.500
Total tolerance = 0.504 (upper limit) - 0.500 (lower limit) = 0.004

Figure1.2. Unilateral tolerance examples

- ✓ **Limit Tolerances**

A limit tolerance does not use the "+," "−," or "±" symbol. Instead of a basic dimension being listed, the upper and lower limits are shown. They are usually separated by a bar or a slash. To determine total tolerance from a limit tolerance, simply subtract the lower limit from the upper limit.

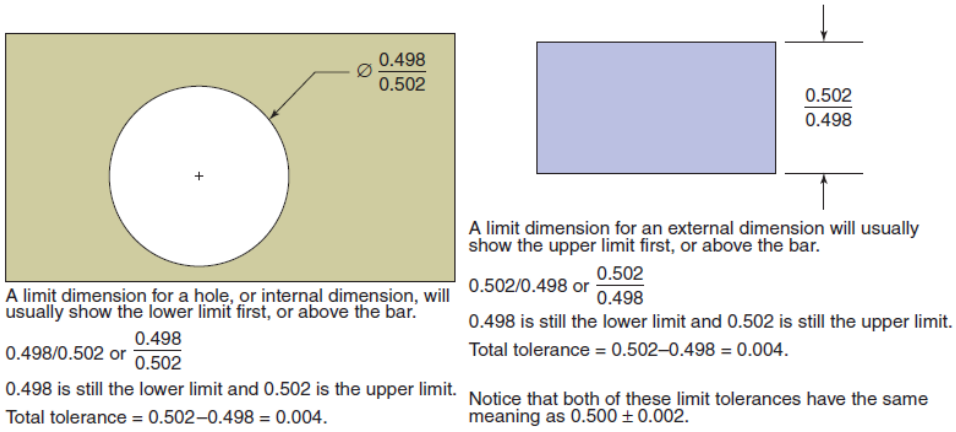
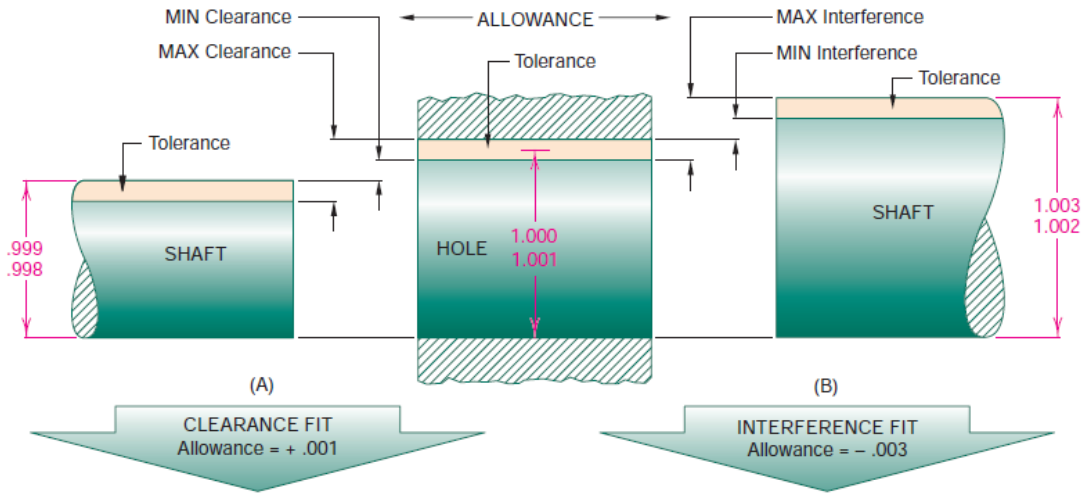


Figure1.3. Limit tolerance examples

- **Classes of Fit**

Sometimes machining operations produce two mating parts, such as a shaft that fits inside a hub. If the print doesn't specifically call out the dimensions or tolerances, use of technical reference material may be required to determine the proper size ranges for those two mating parts. This relationship between the sizes of the two mating parts is called the class of fit.



Allowance always equals smallest hole minus largest shaft

x

Figure1.4. Clearance and interference fits between two shafts and a hole

Shaft A is a clearance fit, and shaft B is interference fit.

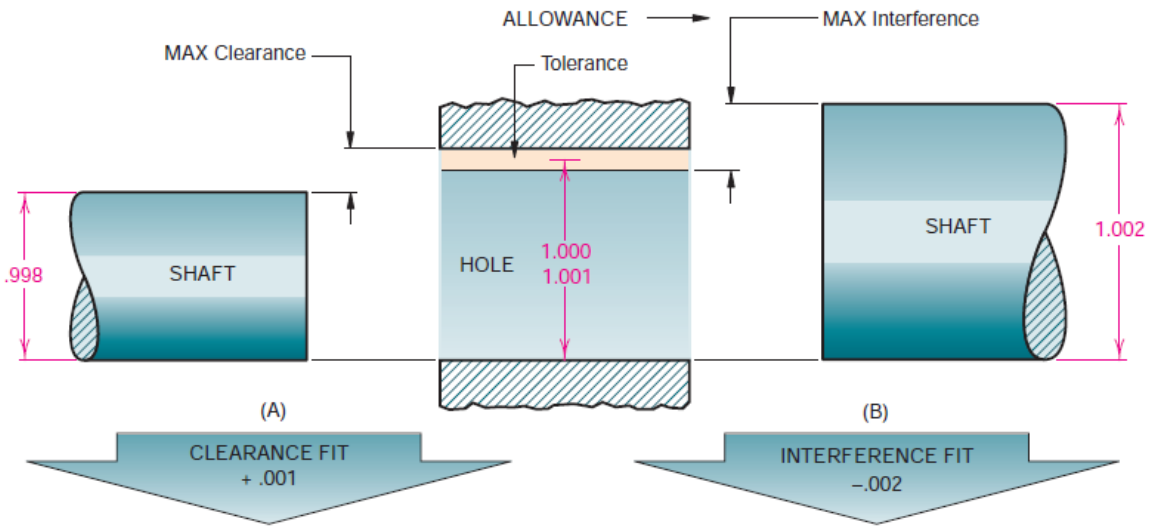


Figure1.5. Transition fit between a shaft and a hole

When the shaft is machined to its smallest diameter (.998), there is a clearance fi t with the hole. When the shaft is machined to its largest diameter (1.002), there is an interference fi t with the hole.

- ✓ **Clearance fit** occurs when two tolerance mating parts will always leave a space or clearance when assembled.

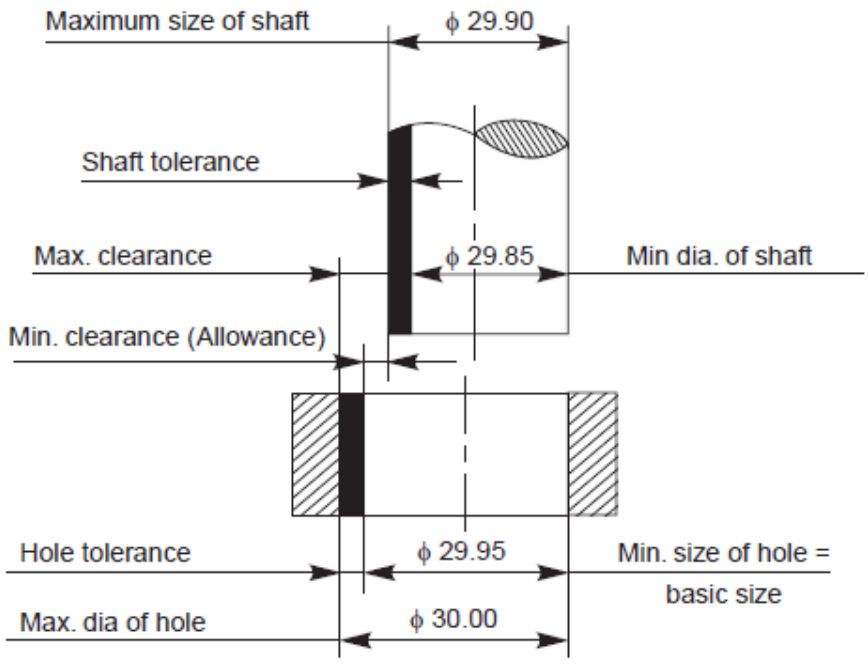


Figure1.6. Clearance fit

- ✓ **Interference fit**. This means that the shaft will always be larger than the hole. In order to assemble the parts under this condition, it would be necessary to stretch the hole or shrink the shaft or to use force to press the shaft into the hole.

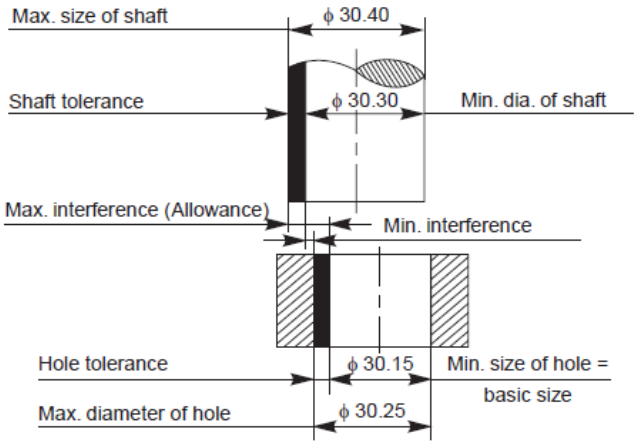


Figure1.7. Interference fit

- ✓ **Transition fit** occurs when two tolerances mating parts are sometimes an interference fit and sometimes a clearance fi t when assembled.

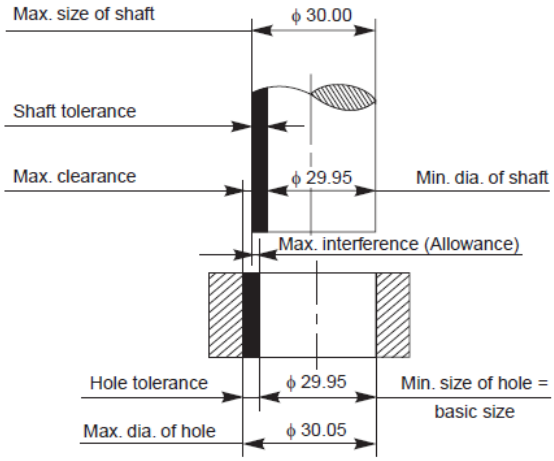


Figure1.8. Transition fit

a. Surface finish

Surface finish refers to the roughness, waviness, lay, and flaws of a surface. Surface finish is the specified smoothness required on the finished surface of a part that is obtained by machining, grinding, honing, or lapping. Figure below shows the drawing symbol associated with surface finish.



Figure1.9. Surface finish symbol

- **Surface finish symbol**

Some of the surfaces of an object are machined to certain specifications. When this is done, a surface finish symbol is placed on the view where the surface or surfaces appear as lines, which is an edge view. The finish symbol on a machine drawing alerts the machinist that the surface must be machined to the given specification.

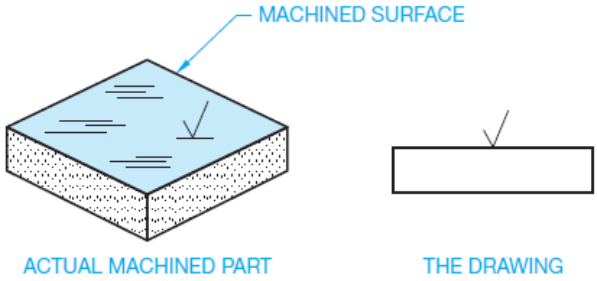


Figure1.15. Standard surface finish symbol placed on the edge view. The symbol should always be placed horizontally when unidirectional dimensioning is used.

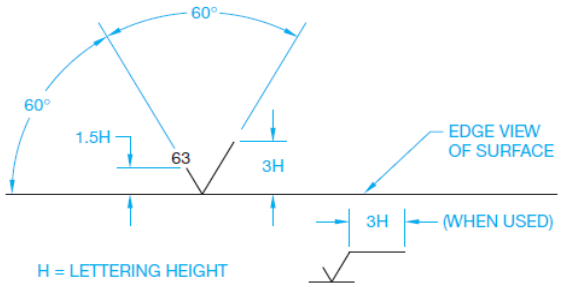


Figure1.10. Properly drawn surface finish symbol

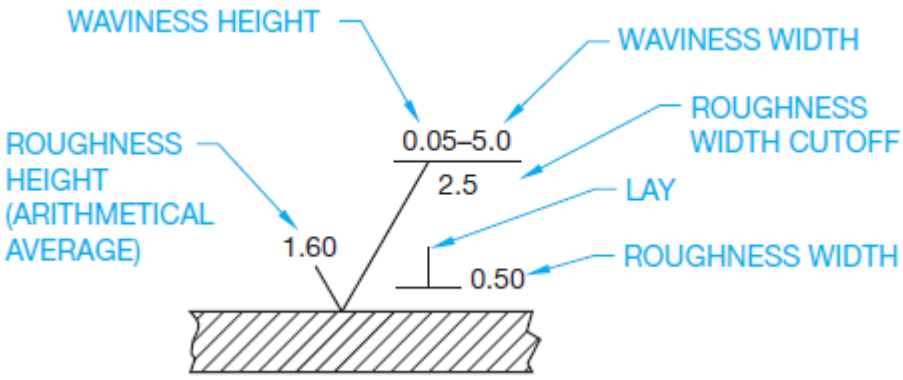


Figure1.11. Elements of a complete surface finish symbol

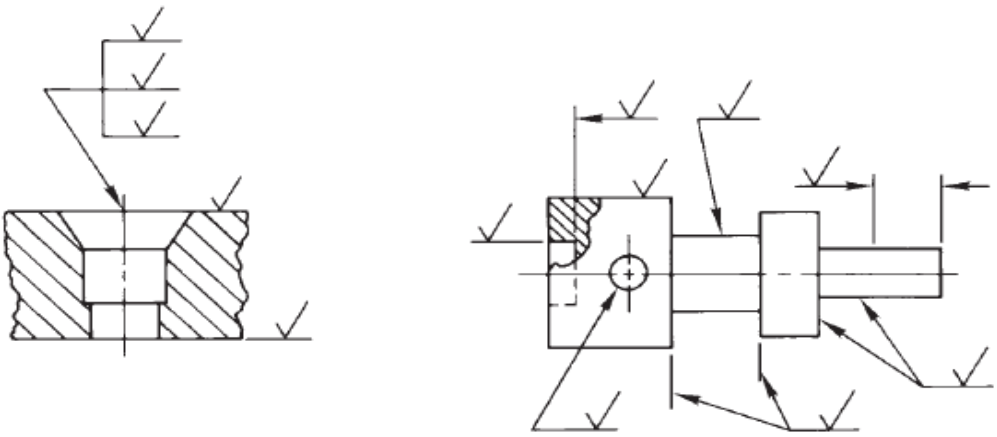


Figure1.12. Proper placement of surface finish symbol

1.1.2. Milling types and specifications

Milling Introduction

Milling machine is one of the most versatile conventional machine tools with a wide range of metal cutting capability. Many complicated operations such as indexing, gang milling, and straddle milling etc. can be carried out on a milling machine.

This training module is intended to give you a good appreciation on the type of milling machines and the various types of milling processes. Emphasis is placed on its industrial applications, operations, and the selection of appropriate cutting tools.

On completion of this module, you will acquire some of these techniques from the training exercises as illustrated in figure1.19

However, to gain maximum benefit, you are strongly advised to make yourself familiar with the following notes before undertaking the training activities, and to have a good interaction between yourself and the staff in charge of your training.

Assessment of your training will be based on a combination of your skill and attitude in getting the work done.

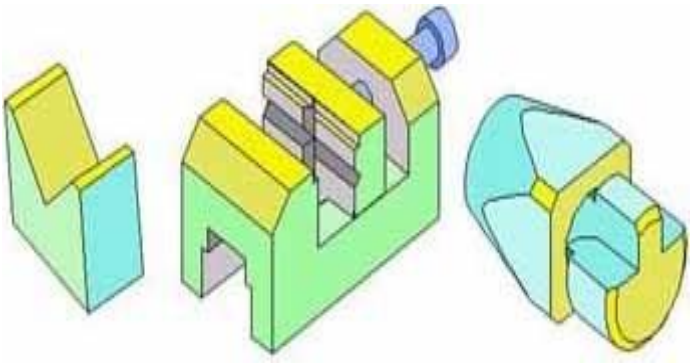


Figure1.13. Milling Products

A. Types of Milling Machine

Most of the milling machine is constructed of ¡¥column and knee¡¦ structure and they are classified into two main types namely Horizontal Milling Machine and Vertical Milling Machine. The name Horizontal or Vertical is given to the machine by virtue of its spindle axis. Horizontal machines can be further classified into Plain Horizontal and Universal Milling Machine. The main difference between the two is that the table of an Universal Milling Machine can be set at an angle for helical milling while the table of a Plain Horizontal Milling Machine is not.

- **Horizontal Milling Machine**

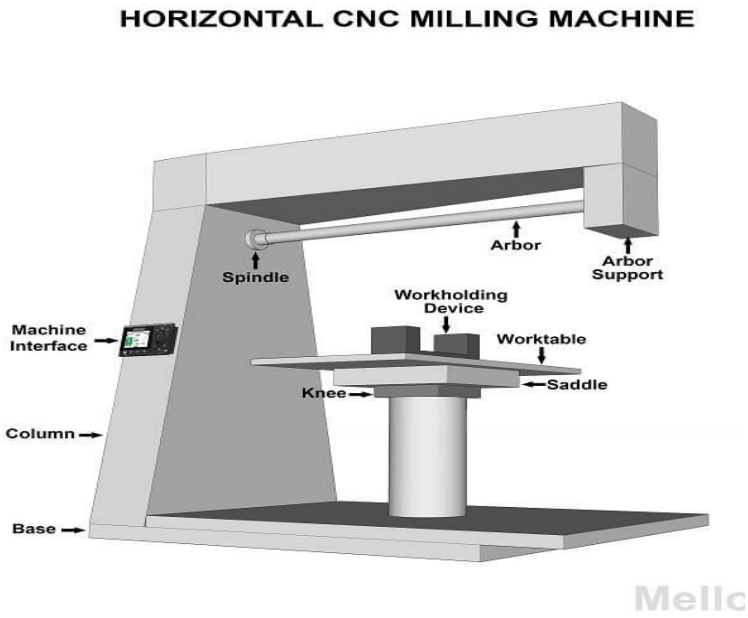


Figure1. 14shows the main features of a Plain Horizontal Milling Machine.

- **Vertical Milling Machine**

Figure 1.22 shows a vertical milling machine which is of similar construction to a horizontal milling machine except that the spindle is mounted in the vertical position.

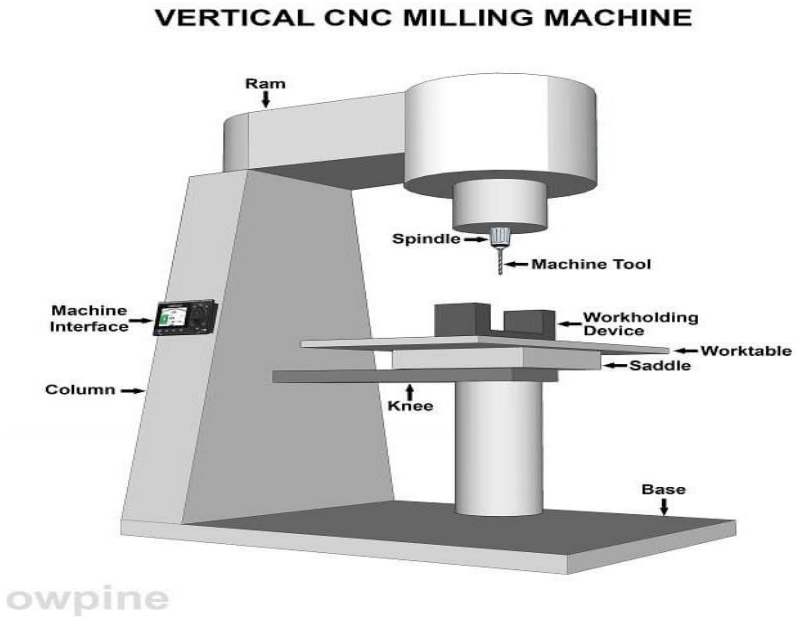


Figure1.15. Vertical Milling Machine

1.1.3. Milling parts and functions

CNC milling machines are one of the most popular tools for machining metals to get the desired work piece.

These rigid and powerful machines consist of various parts that work in synchronization to perform the desired milling operation.

Parts of a CNC milling machine include a column, base, knee, power feed mechanism, a work table, ram, spindle, milling tool, milling chuck, CNC controller, and an overhanging arm along with arbor support in case of horizontal milling machines.

a. Column

The column houses the spindle, the bearings, the gear box, the clutches, the shafts, the pumps, and the shifting mechanisms for transmitting power from the electric motor to the spindle at a selected speed.

b. Knee

The knee mounted in front of the column is for supporting the table and to provide an up or down motion along the Z axis.

c. Saddle

The saddle consists of two slide ways, one on the top and one at the bottom located at 90º to each other, for providing motions in the X or Y axes by means of lead screws.

d. Table

The table is mounted on top of the saddle and can be moved along the X axis. On top of the table are some T-slots for the mounting of work piece or clamping fixtures.

e. Arbor

The arbor is an extension of the spindle for mounting cutters. Usually, the thread end of an arbor is of left hand helix.

f. Base

The base of the milling machine, along with the column is the major structural components. They hold, align, and support the rest of the machine.

g. Spindle

The spindle holds the tool and provides the actual tool rotation.

h. Spindle Reverse Lever

The position of this lever determines the spindle direction. The three positions of the handle are; In, Middle, and Out. The middle position is the neutral position. Never move the spindle reverse lever when the spindle is turning.

ı. Spindle Speed Selection Lever

The spindle speed selection lever is used to change the spindle R.P.M. setting. This type of machine has a geared head so the spindle speed can only be changed when the spindle is stopped.

j. Spindle Clutch Lever

The spindle clutch lever engages the spindle clutch to the motor. By manipulating the spindle clutch lever the operator can start and stop the spindle.

k. Feed Rate Selection Lever

The feed rate selection lever is used to change the feed rate setting. The feed rate settings are expressed in inches per minute.

m. Motor Start and Stop Buttons-

The motor start and stop buttons control the power to the main motor for the machine

In Vertical Milling Machine Its additional features are

a. Milling head

The milling head consisting the spindle, the motor, and the feed control unit is mounted on a swivel base such that it can be set at any angle to the table.

b. Ram

The ram on which the milling head is attached can be positioned forward and back ward along the slide way on the top of the column

Sequence of operation

All CNC machining begins with a part program, which is a sequential instructions or coded commands that direct the specific machine functions. The part program may be manually generated or more commonly generated by computer aided part programming systems.

It is a sequence of machining operation plan. Which is prepared based on the information provided on the drawing of the component to be manufactured. in a sequence of CNC milling machining operation plan. Which is prepared based on the information provided on the job sheet or verbal instructions of the component to be machined has to ready for producing expected parts / components on its required specification. Study the sequences of operations in machining and see that all finish marks are indicated

Plan includes

I- Which consequence of machining operation is efficient.

II- Tools and equipment's required for each machining operation.

III- Methods of chucking and examine the component in each machining operation.

- **Plan the Sequence of Machining Operations**

The sequence of machine operations used to machine the work piece will have a dramatic impact on the success of the program. If the process is correct, the work piece will be machined efficiently and pass inspection. If the process is poor, the work piece will not be machined correctly no matter how well the program is written. If you are new to developing a sequence of machining operations, you should seek help whenever there is a question as to whether your planned machining process will work.

The last reason we will give to plan the process first is to simply help you remember the operations to perform during programming. Remember, beginners tend to make mistakes of omission. You will have enough to think about when it comes to remembering the various commands needed in the program. The process planning form is a step-by-step set of instructions to machine the work piece. It can be used as a checklist. Without this form, you will be prone to omitting important operations from the CNC program.

Develop the Cutting Conditions Before a program can be written, cutting conditions must be determined for all the cutting tools used in the program. Each cutting tool will need a spindle speed in revolutions-per-minute (rpm) and a feed rate in inches-per-minute (ipm). For roughing tools, like rough milling cutters and rough boring bars, you must also determine a depth-of-cut for the tool–as well as how much finishing stock you will leave for the finishing tool. You must also determine whether to use coolant and, if so, what kind–flood, mist, through-the-tool, or high pressure, based upon the work piece and cutting tool materials.

For machining center programs, you must of course, calculate the speed in rpm and feed rate in rpm. For rpm, you must know the recommended speed in rpm and the cutting tool diameter. For rpm, you must know the rpm and rpm feed rate.

Here is an example of a filled-in process planning form.

Part no.:		Date:		Machining Process Planning Form		
Part name: exercise part		Programmer: your name				
Machine: machining center		Material: mild steel				
Seq.	Operation Description	Tool	Station	Speed	Feed	Note
1	Rough face mill the top surface	4.0" face mill		382	12.224	
2	Rough mill the 2 ends	1.0" end mill		267	5.34	
3	Finish face mill the top surface	4.0" face mill		477	9.54	
4	Finish mill the 2 ends	1.0" end mill		325	5.2	
5	Center drill 5 holes	#3 center drill		1430	5.72	
6	Drill 2 1/4-inch diameter holes	1/4 drill		1070	4.28	
7	Drill 2 1/2-inch diameter holes	1/2 drill		535	3.74	

Fig1.16. Example machining process planning form with feeds & speeds

In general, using the part drawing, the programmer determines the sequence of operations, speeds, and feeds for various operations and determines the magnitude of various motions decided. He prepares the planning sheet and writes the instructions in a coded format for the MCU. The part programs are written in the blocks using the standard codes and following the sequence.

- Sequencing of planning the program for CNC part programs should consider the following:

1. Part drawing. This drawing includes the shape, tolerances, material requirements, surface finish, and product quality. There are several methods of part dimensioning. In the center line method, the part centerline is used as reference point for dimensioning the component. The coordinate or base/ datum line method enables easy transcription of the various features of the part. The dimensions should be specified in imperial (inch) or metric (millimeters) units. An alternative method of dimensioning that is convenient for ease of programming is to label with letters and numbers all the relative features of the component.

2. Machine tool used. The part programmer should be aware of the machine size, power, accuracy, tooling, capacity, and number of axes. Specifically, he should be aware of the following points:
   a. Power of the drive motors, which decide the material removal rates and area of cut
   b. The possible movement for all the machine axes.
   c. The possibility of machining the whole part in once setup
   d. The spindle speeds available
   e. Machine rigidity and accuracy capabilities.

- **Operations sequence for a sample flat part**

The part shown in Figure below is used only as an example to illustrate a sequence of operations that should be followed when machining similar parts. These are not meant to be hard-and-fast rules, only guides.

- ✓ The part is relatively thin and has a large surface area.
- ✓ Since at least .125 in. (3 mm) of work should be above the vise jaws, it would be difficult to use a round bar between the work and movable jaw for machining the large flat surfaces.
- ✓ A small inaccuracy (out-of- squareness) on the narrow edge would create a greater error when the large surface was machined.

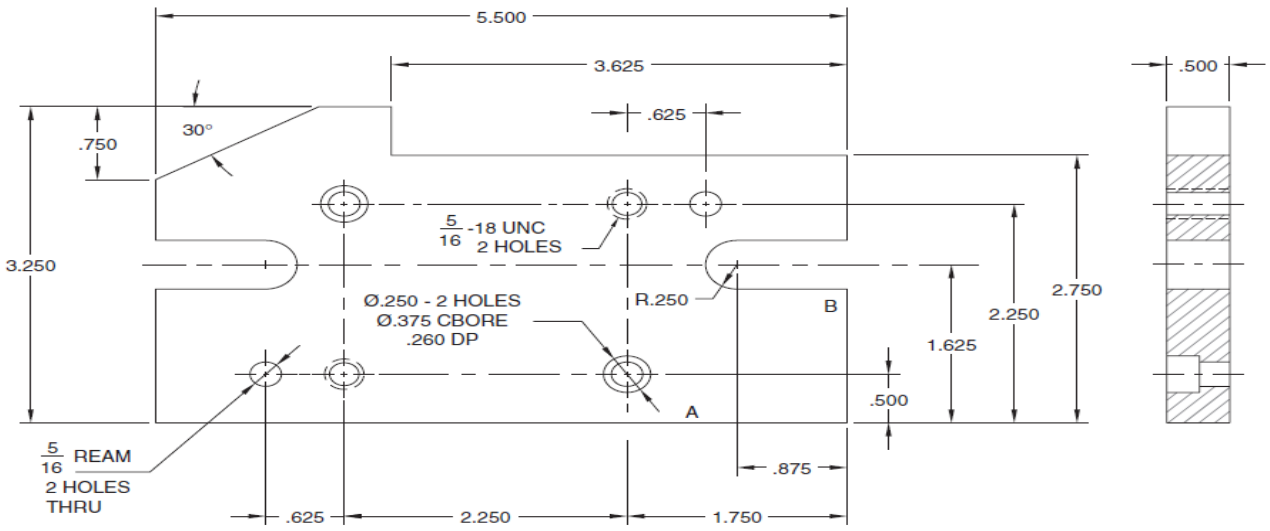


Figure1.17. A typical flat part that must be laid out and machined

Procedure

1. Cut off a piece of steel .625 in. (16 mm) × 3.375 in. (86 mm) × 5.625 in. (143 mm) long.
2. In a milling machine, finish one of the larger surfaces (face) first.

Note: Leave .010 in. (0.25 mm) on each surface to be ground.

3. Turn the work piece over and machine the other face to .500 in. (13 mm) thick.
4. Machine one edge square with the face.
5. Machine an adjacent edge square (at 90^0) with the first edge.
6. Place the longest finished edge (A) down in the machine vise and cut the opposite edge to 3.250 in. (83 mm) wide.
7. Place the narrower finished edge (B) down in the machine vise and cut the opposite edge to 5.500 in. (140 mm) long.
8. With edge A as a reference surface, lay out all the horizontal dimensions with an adjustable square, a surface gage, or a height gage (Figure below).

9. With edge B as a reference surface, lay out all the vertical dimensions with an adjustable square, a surface gage, or a height gage.
10. Use a bevel protractor to lay out the 30° angle on the upper right-hand edge.
11. With a divider set to .250 in. (6 mm), draw the arcs for the two center slots.
12. With a sharp prick punch, lightly mark all the surfaces to be cut and the centers of all hole locations.
13. Center-punch and drill 1/2-in. (13-mm) diameter holes for the two center slots.
14. On a vertical band saw, cut the 30° angle to within 0.30 in. (0.79 mm) of the layout line.
15. Place the work piece in a vertical mill and machine the two .500-in. (13-mm) slots.
16. Machine the step on the top edge of the work piece.
17. Set the work to 30° in the machine vise and finish the 30° angle.
18. 18. Prick-punch the holes locations, scribe reference circles, and then center-punch all hole centers.
19. Center-drill all hole locations.
20. Drill and counter bore the holes for the 1⁄4 in.–20 NC screws.
21. Tap drill the 5⁄16 in.-18 thread holes (F drill or 6.5 mm).
22. Drill the 1⁄4-in. (6-mm) ream holes to 15⁄64 in. (5.5 mm).
23. Countersink all holes to be tapped slightly larger than their finished size.
24. Ream the 1⁄4-in. (6-mm) holes to size.
25. Tap the 5⁄16 in.–18 UNC holes.

Cutting tools.

Selecting tool holders and cutting tools is another overly critical part of part program development process. Tool holders are generally used with different cutting tools and remain unchanged for extended periods of time. Tools, such as drills, reamers, taps, end mills, car bide inserts, etc. are perish able tools.

Some tool holders are used for a specific tool group, such as a collet holder is a better choice for end mills, Weld on TM type tool holders prevent tool spinning, Jacob site chucks are used for drills, etc. Some tools, such as taps, require special holders, designed for a single purpose.

Tooling selection is always closely related to the setup and cutting conditions. When selecting tools, always keep in mind these other two considerations. When the part setup method has been established, the tools can be selected based on drawing dimensions and machining operations required. Tools are always selected based on machining operations required.

A wide range of tool types and configurations are available for CNC milling machines. They are face milling; count our milling, circular pocket milling, slot milling, spot drilling, and drilling, tapping, reamer.

CNC machining requires the use of the best possible cutting tools. Cutting tool technology is changing rapidly with the development of new tool materials and tool coatings. With these new materials come changes in tool geometries that were previously not possible. More-positive rake tools have allowed higher metal removal rates with the same horsepower machine tools.

1.3.1. Types of Cutting Tool

- **End Mills:** End mills are the sharp spinning tools used in CNC machining



Figure1.18. Common end mills

Drill Bits versus End Mills: Although similar in appearance to each other an end mill and a drill bit are not the same thing, nor do they perform the same functions.



Figure 1.19. Comparison of a drill bit (top) and an end mill (bottom) note the different tips

- **Drill bits:** Drill bits are intended to be used to drill holes, plunging directly into the material at a 90-degree angle, as depicted in. That's why the tip of a drill bit is pointed it's designed to bore straight down into the material. The flutes on a drill bit are designed to guide the material being removed up and out of the hole created by the bit.

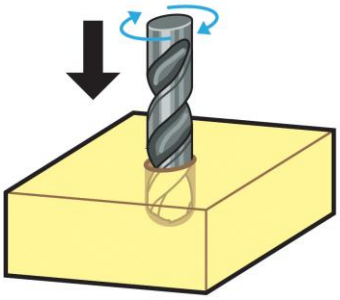


Figure1.20. Drill bits action

End mills: end mills are designed to cut from their edges as they move laterally through the material. A center-cutting end mill can both plunge into the material (like a drill bit) and move laterally across the material. End mills are designed to use their entire cutting edge length to cut, not just the tip.

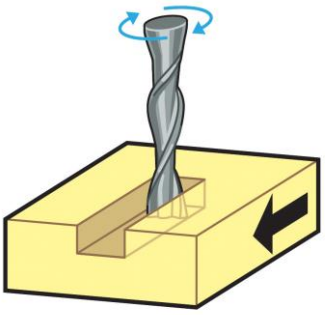


Figure 1.21. End mills action

End Mill Anatomy:

The parts of an end mill include

Overall length: Measuring your end mill from one end to the other will yield the overall length of the bit.

Shank diameter: This is the diameter of the part of the end mill that is not doing the cutting. In other words, this is the diameter that will go into your spindle. You'll need to know about your spindle's work holding capacity before selecting the Shank diameter. Common shank sizes are 1/8 inch (3.175 mm) and ¼ inch (6.35 mm).

Cutting edge diameter: This is the diameter of the cutting end of your end mill. If you were to plunge straight down into your material, and then measure the hole created, it would match the cutting-edge diameter of your end mill.

Cutting length: End mills have flutes that cover only a portion of the overall length of the bit. The cutting length determines how deep of a pass can be taken with a single plunge. Reach Many end mills have a neck—a portion of the end mill between the shank and the cutting length that is not meant for cutting and is not meant for clamping. The reach length is the maximum distance your tool can go into the material.

Flutes: The grooves that spiral around your cutting tool. Each flute has a single cutting tooth that runs along the edge of the flute, from bottom to top.

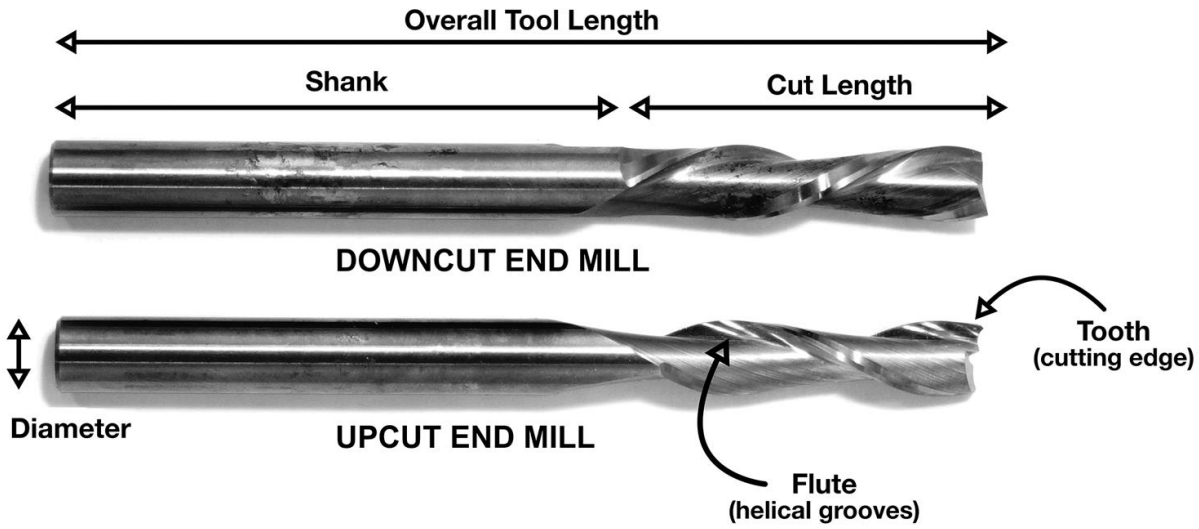


Figure1.22.End Mill Anatomy

Tool Materials

To select the most appropriate tool for a machining operation, an understanding of tool materials is critical. The following outline provides a generic look at tool materials. Each tool manufacturer has a complete description and application guidelines for their specific grades.

HSS: HSS steels, used mostly for drills and reamers, can be used on less rigid machine tools, because the tool material is relatively ductile.

Cobalt: Premium HSS steels such as M-42 remain considerably harder at temperatures up to 1000°F than tools made from conventional HSS steels such as M-2 or M-7. These HSS steels are much more abrasion resistant, making them a good choice to use on scaly materials such as cast iron, cast steels, and heat-treated steels. Cutting speeds for these tools usually can be increased by 10 percent over standard HSS steels.

Cermet: Cermet is a combination of titanium carbide and titanium nitride with a metallic binder such as nickel and cobalt. Sometimes, molybdenum carbide and niobium carbide are added to improve toughness. Cermet are used to obtain high-quality surface finishes in carbon and alloy steels, stainless steels, and ductile irons. Most machining with cermet is done dry because of their relatively poor thermal shock resistance.

Carbides Uncoated carbides combine toughness and good wear resistance for finishing nonferrous materials and cast iron. Uncoated inserts can be used where a built-up edge is not a problem. Fine-grain carbides have a toughness and ductility that approach that of HSS steel, which allows their use on less rigid machine tools.

Ceramics Aluminum oxide tool materials are used in roughing and finishing carbon steels, cast iron, and nickel alloys.

PCBN Polycrystalline cubic boron nitrides (PCBNs) are used for finishing heat-treated alloy steels (45– 65RC), tool steels, cast iron, powdered metal, cobalt based alloys, and nickel-based super alloys.

PCD Polycrystalline diamond (PCD) gives outstanding wear resistance and surface finish when machining highly abrasive nonferrous materials such as aluminum, aluminum alloys, copper alloys, carbon-and epoxy composites, hard rubber, wood, fiberglass, and plastics. Use positive geometry and a small nose radius (.008 in. or less).

Tool Coatings

Tool coatings are applied by the use of chemical vapor deposition (CVD) or physical vapor deposition (PVD). Each coating process has distinct advantages in the proper application. Tool coatings are useful because of their high hardness and low coefficient of friction. Coatings improve tool performance whether the substrate is HSS or carbide.

CVD coatings are produced in a high-temperature (1850°F) process. CVD is a gaseous process that coats every surface exposed to the gasses during the process. CVD coatings are chemically and metallurgical bonded to a base material such as carbide or high-speed steel. Because of the high coating temperature, these tools need to be heat-treated again to restore their metallurgical properties. During this second heat treatment, the tool may change size. CVD-coated tools generally are not made with tight dimensional tolerances. CVD coatings normally are between .0002 and .0003 in. thick. CVD coatings are used where severe compressive stresses are encountered, such as in heavy roughing cuts.

PVD coatings are applied in a vacuum chamber at less than 750°F. At this low temperature, there are usually no dimensional changes or loss of hardness during the coating process. PVD-coated tools usually retain sharper cutting edges, preferred for light cuts. PVD coatings can be directed to a specific surface because the process uses a directed ion beam that affects only line-of-sight areas. Coating thickness is usually from .0001 to .00015 in.



Figure 1.23. Tool coating

- **TiN coating -** Titanium nitride gives high lubricity to the tool surface, which facilitates chip flow and resists built-up edge during machining. It is used on ferrous materials and cast iron and has excellent abrasion resistance, edge strength, and resistance to cratering.
- **TiC coating -** Titanium carbide adds abrasion resistance.
- **TiCN coating -** Titanium carbonitride has the same lubricity properties as TiN but is 30 percent harder. This increased hardness gives maximum tool life.
- **Al_2O_3 coating -** Aluminum (oxide) and carbon dioxide coatings on tools are used to machine cast irons (nodular and malleable). This gives resistance to abrasive wear and cratering and adds shock resistance.
- **CrN coating -** Chromium nitride is used to machine many nonferrous materials. This coating also has a high oxidation resistance.
- **CrC coating -** Chromium carbide coatings can be used effectively when machining titanium or aluminum.
- **Multilayered coatings -** These combine the best properties of single coatings. Multiple coatings are made up of layers of TiN>TiCN>Al_2O_3 and repetitions of these.

Understanding Cutting Tool Numbers

Tool numbers in the CNC document must be correctly identified in the control. If the machine is a lathe, the task is simple: tool 01 goes in turret position number 1. Tool 02 goes in position 2, and

so on. Some mills require the same: T01 goes in storage drum hole 1, T02 in hole 2, and so on. These are generally older mills lacking random tool storage ability.

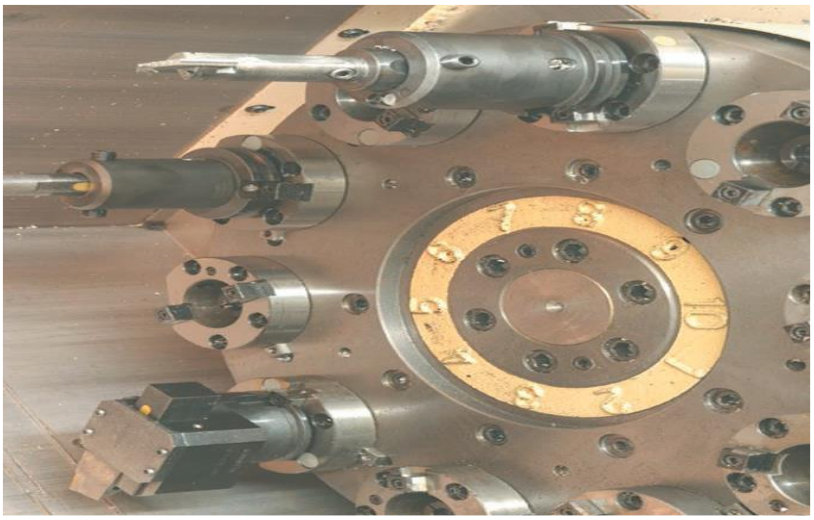


Figure1.24. Understanding cutting tool numbers

More modern CNC mills allow the storing of any tool in any drum hole. But the tool-loading task is bit more complex. For example, the mill operator is storing T01 in open drum position 08 in. His next action will be to immediately go to the control and enter tool 1 at position 8 on the tool tracking page. Tool numbers are arranged in sets of two-digit data fields, in one, two, or three columns. Various controls use different tool number sets. Some take up the cutter offset information when reading the tool number, while others require a separate code to get and use offsets.

Séparâtes Code Exemple

T01 H43 D1

Use tool 1 and apply length offset (H43) to the number stored in D1 (diameter 1)

- Face mills flatten large surfaces quickly and with an excellent finish. Notice the engine block being finished in one pass with a large cutter.



Figure1.25. Face mills

- Drills, Taps, and Reamers

Common HSS tools such as drills, taps, and reamers are commonly used on CNC machining centers. Note that a spot drill is used instead of a center drill. Also, spiral point or gun taps are used for through holes and spiral flute for blind holes. Rarely are hand taps used on a machining center



All cutting tools must be held in a holder that fits in the spindle. These include end mill holders (shown), collet holders, face mill adapters, etc. Most machines in the USA use a CAT taper which is a modified NST 30, 40, or 50 taper that uses a pull stud and a groove in the flange. The machine pulls on the pull stud to hold the holder in the spindle, and the groove in the flange gives the automatic tool changer something to hold onto. HSK tool holders were designed several years ago as an improvement to CAT tapers, but they are gaining acceptance slowly.

- **Angle cutters**: Angle cutters differ from other cutters in that the cutting edges are neither parallel, nor at right angles to the cutter axis.

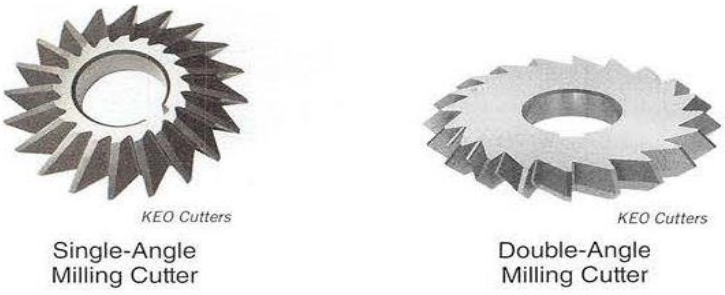


Figure 1.26.Angle cutters.

- **Metal slitting saws**: Metal slitting saws are thin milling cutters that resemble circular saw blades. They are employed for narrow slotting and cutoff operations. Slitting saws are available in diameters as small as 2 1/2" (60 mm) and as large as 8" (200 mm).

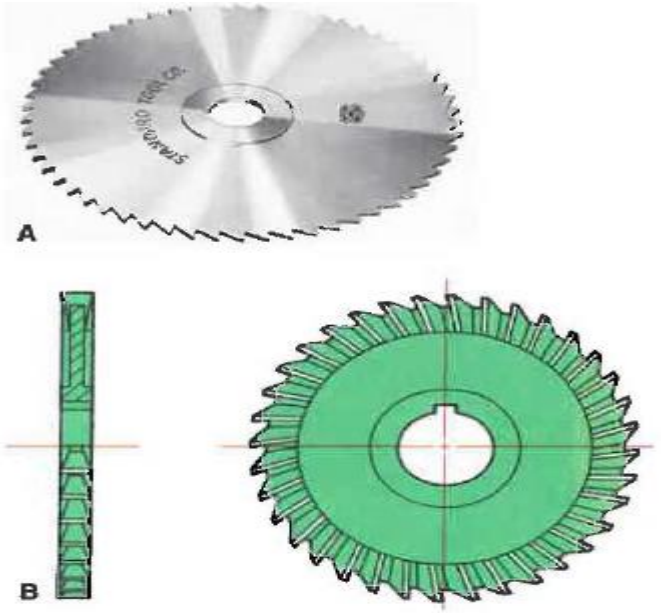


Figure1.27 slitting saws A-Plain metal slitting saw is used for slotting and cutoff operations.

Cutting parameters

The tool moves through the material at a specified rotational speed, defined in revolutions per minute (RPM), and feed rate, defined in inches per minute (IPM). Probably the most vexing problem for the beginning CNC machinist is selecting proper cutting speeds and feeds.

This selection is more difficult on a CNC than a manual mill because, with a manual mill, the operator can feel the cutting pressure and alter the feed based in part on the cutting force.

CNC mills require calculating speeds and feeds in advance. These speeds and feeds can, and often are, adjusted at the machine based on chip shape and color, cutting sound, and machine horsepower meter readings.

The best source of data about cutting speeds and feeds for a specific tool, application, and material is the tool supplier. Much of this data is found on manufacturer's web sites or printed tooling catalogs. Tool sales representatives can be a valuable resource, so if you do a lot of machining, develop a good relationship with a knowledgeable representative.

Another source of speeds and feeds data is CAD/CAM software. These have become increasingly sophisticated and often provide good cutting data.

Yet even the best speed and feed data is just a starting point. Speeds and feeds require adjustment due to many factors including the maximum spindle speed or horsepower of the machine, rigidity of work holding, and the quality and condition of the machine tool itself.

The following pages provide cutting data for the most commonly machined materials and a methodology for calculating speeds and feeds. As always, use common sense. If the part is held by double sided tape, feeds based on vise work holding are probably too high. If the tool is very long and thin, speeds and feeds will likely require reduction.

Cutting Speed, Feed, and Depth of Cut

- Cutting speed
  - Too slow, time wasted
  - Too fast, time lost in replacing/regrinding cutters
- Feed
  - Too slow, time wasted and cutter chatter
  - Too fast, cutter teeth can be broken
- Depth of cut
  - Several shallow cuts waste time

1.3.2. Speeds and Feeds

The phrase “speeds and feeds” refers to how quickly the spindle is turning, combined with how fast the tool is moving through the material. These measurements are communicated in units over time, commonly in inches per minute or millimeters per minute.

Feed rate: Refers to how quickly the machine is moving through your material.

- When any feed move is programmed, a feed rate must be assigned to it. Feed rates are modal and if none is programmed in a block, then the previously programmed rate will still be active.
- An F character precedes the desire feed rate in the program. Feed rates can be expressed in either inches per minute (IPM) using a G94 code or inches per revolution (IPR) using a G95 code. This setting is frequently defined in the safe-start portion of the CNC program but can be activated or changed at any time.
- For milling operations, use of IPM is most common, but IPR is sometimes used for hole making operations. If the feed mode for IPM is added to the previous safe-start block, the result would be: **G90 G20 G17 G94**.

It is important to remember that whatever feed rate is programmed, the MCU will command the machine to move at that value without regard to whether it is an IPM or IPR value. If the machine is in G94 (IPM) and feed rate of 0.001 is programmed, the axis will attempt to move at only 0.001 IPM.

Spindle speed: Refers to the revolutions per minute (RPM) of your spindle. Spindle speeds for CNC milling are expressed in RPM and defined with the combination of an M- and S-command. The S-command provides the spindle RPM and the following M-commands are used to turn the spindle on or off:

- **M3** Spindle on clockwise (forward)
- **M4** Spindle on counterclockwise (reverse)
- **M5** Spindle off

For example, **M3 S2000** means clockwise spindle rotation at 2000 RPM, and **M4 S500** means counterclockwise spindle rotation at 500 RPM. An M5 will stop the spindle.

Depth pass: A key factor in determining your speeds and feeds is depth pass, or how deep the bit plunges into the material in a single pass. Another term for depth pass is depth of cut, or DOC for short.

- When determining speeds and feeds, a good rule of thumb is to base the depth pass on the diameter of your cutter.
- For soft materials (like plywood), your depth pass is typically equal to the diameter of your cutter. For denser materials (like hardwood), the depth pass should be about half the diameter of your bit.
- When cutting really hard materials (like aluminum), depth pass should be a fraction of the diameter of your tool.

Chip load: As the flute's cutting tooth bites into the substance being machined, it removes a small chip, with each pass. Chip load is the thickness of that chip. The following formulas give you a starting point for determining the best speeds-and-feeds settings for any given material

.

Calculating Chip Load

Chip Load = Feed Rate / (RPM × Number of Flutes)

Calculating Feed Rate

Feed Rate = RPM × Number of Flutes × Chip load

Calculating Spindle Speed

Speed = Feed Rate / (Number of Flutes × Chip load)

Cutting Speed

- Speed, in surface feet per minute (sf/min) or meters per minute (m/min) at which metal may be machined efficiently
- Work machined in a lathe, speed in specific number of revolutions per min (r/min) depending on its diameter to achieve proper cutting speed
- In milling machine, cutter revolves r/min depending on diameter for cutting speed

In general, Cutting Speed; Excessive Speed will result in increased tool wear,
Increasing cutting speed by 50% can result in a decrease of tool-life by 90%

Important Factors in Determining Cutting Speed

- Work piece material
- **Cutting Tool material**
- **Operation performed**
- **Surface finish required**

The Big Three

- How Speed Formula is Derived
- Because cutting tools are defined by their diameter in inches, this formula is rewritten and simplified as follows:

$$Speed\left(\frac{rev}{min}\right) = \frac{SFM\left(\frac{ft}{min}\right)}{\pi\, x \left(\frac{1ft}{12in}\right) x\, Dia(in)}$$

$$Speed\left(\frac{rev}{min}\right) = \frac{SFM\left(\frac{ft}{min}\right)}{\frac{\pi}{12} x Dia(in)}$$

$$Speed\left(\frac{rev}{min}\right) = \frac{SFM\left(\frac{ft}{min}\right) x\, 3.82}{Dia(in)}$$

Fig.1.28. Speed formula

- DIA - is the tool diameter in inches.
- 3.82 - is a constant derived from $12/\pi$ which convert the tool circumference in feet to diameter in inches

Cutting Speed Rules for Best Results

1. For longer cutter life, use lower CS in recommended range
2. Know hardness of material to be machined
3. When starting, use lower range of CS and gradually increase to higher range
4. Reduce feed instead of increase cutter speed for fine finish
5. Use of coolant will generally produce better finish and lengthen life of cutter

Milling Machine Feed

- Defined as distance in inches (or mm) per minute that work moves into cutter
  - Independent of spindle speed
- Feed: rate work moves into revolving cutter
  - Measured in in/min or mm/min
- Milling feed: determined by multiplying chip size (chip per tooth) desired, number of teeth in cutter, and r/min of cutter
- Chip, or feed, per tooth (CPT or (FPT): amount of material that should be removed by each tooth of the cutter

In general, Milling Machine Feed; Excessive feed can increase tool wear,

Increase of feed rate by 50% can result in a 60% decrease in tool-life

Factors in Feed Rate

1. Depth and width of cut
2. Design or type of cutter
3. Sharpness of cutter
4. Work piece material
5. Strength and uniformity of work piece
6. Type of finish and accuracy required
7. Power and rigidity of machine, holding device and tooling setup

Recommended Feed Per Tooth
(High-speed Cutters)

Sample Table
See Table 60.2 in Text

Material	Face Mills		Helical Mills		Slotting and Side Mills	
	in.	mm	in.	mm	in.	mm
Alloy steel	.006	0.15	005	0.12	.004	0.1
Aluminum	.022	0.55	.018	0.45	.013	0.33
Brass and bronze (medium)	.014	0.35	.011	0.28	.008	0.2
Cast iron (medium)	.013	0.33	.010	0.25	.007	0.18

Table shows feed per tooth for roughing cuts — for finishing cut, the feed per tooth would be reduced to1/2 or even 1/3 of value shown

Recommended Feed Per Tooth
(Cemented-Carbide-Tipped Cutters)

Sample Table
See Table 60.3 in Text

Material	Face Mills		Helical Mills		Slotting and Side Mills	
	in.	mm	in.	mm	in.	mm
Aluminum	.020	0.5	.016	0.40	.012	0.3
Brass and bronze (medium)	.012	0.3	.010	0.25	.007	0.18
Cast iron (medium)	.016	0.4	.013	0.33	.010	0.25

Table shows feed per tooth for roughing cuts — for finishing cut, the feed per tooth would be reduced to1/2 or even 1/3 of value shown

Ideal Rate of Feed

- Work advances into cutter, each successive tooth advances into work equal amount
  - Produces chips of equal thickness
    - Feed per tooth

Feed = no. of cutter teeth x feed/tooth x cutter r/min
Feed (in./min) = N x CPT x r/min

Feed Formula
Cutting feeds are in (IPM) and use the following formula:

$$Feed\left(\frac{in}{min}\right) = Speed\left(\frac{rev}{min}\right) xCL\left(\frac{\frac{in}{rev}}{flute}\right) x\ NumFlutes$$

Fig.1.29. Feed formula

Feed - is the linear feed of the tool through the material in inches per minute.

Speed - is the result of the speed formula (Figure 15) in revolutions per minute.
CL - is the chip load, or how much material each cutting edge of the tool removes per revolution. Chip load is sometimes referred to as feed per tooth (FPT) or inches per rev (IPR).
Num Flutes - is the number of cutting flutes. (For a twist drill, this value is one.

Drilling Tapping

Tapping speed was calculated by multiplying the spindle speed by the thread pitch.

Tap Feed Formula
For tapping operations, feed rate is based on the number of threads per inch and feed rate:

$$Feed\left(\frac{in}{min}\right) = \frac{Speed\left(\frac{rev}{min}\right)}{TPI}$$

Fig.1.29 Tap feed formula

Feed - is the linear feed of the tool through the material in inches per minute.

Speed- is the result of the previous formula in revolutions per minute.

TPI- is the threads per inch of the tap. For example, the TPI of a ¼-20 tap is 20.
Tap Speed/Feed Example Problem: Calculate the cutting speed and feed for a ¼-24tap operation given the following values:

Table.1.4.1.Tap speed/Feed example

Parameter	Value
Tool Diameter	.25*in*
SFM	100*ft/min*
TPI	24

Solution:

Step 1: Calculate RPM

$$Speed\left(\frac{rev}{min}\right) = \frac{\left(SFM\left(\frac{ft}{\min}\right) x\ 3.82\right)}{Dia(in)}$$

$$Speed\left(\frac{rev}{min}\right) = \frac{100\left(\frac{ft}{min}\right) x\ 3.82}{.25in}$$

$$\boldsymbol{Speed\ \left(\frac{rev}{min}\right) = 1528\ \left(\frac{rev}{min}\right)}$$

Step 2: Calculate IPM

$$Feed\left(\frac{in}{min}\right) = \frac{\left(Speed\left(\frac{rev}{\min}\right)\right)}{TPI}$$

$$Feed\left(\frac{in}{min}\right) = \frac{\left(1528\left(\frac{rev}{\min}\right)\right)}{24}$$

$$\boldsymbol{Feed\ \left(\frac{in}{min}\right) = 63.667\left(\frac{in}{\min}\right)}$$

Processing/job/adjustment sheets.

- **Setup Sheet**

In many shops, setup sheets are luxury. It is a simple statement of fact, but many setup sheets are quite poorly prepared if they are prepared at all. Often, they do not reflect the latest program changes and adjustments; they are not consistent between individual machines and even programmers. Although the time spent on preparing a setup sheet is considered nonproductive from the cost angle, it is a time far from being wasted. Setup process can be organized, certain rules can be set and adhered to and they can be applied to the preparation of a good setup sheet.

The golden rule of a good setup sheet is to always make it in scale. Setup sheet using an outline of the material, fixtures layout, finished shape, tool path, etc., should always be done in scale. Scale, even an approximate scale, is very important for visual comparison. Clamps and other mounting devices should be drawn in positions corresponding to the actual setup. Tool change location should be marked accurately, different views shown, if necessary. Critical positions should be dimensioned, indicating the maximum or minimum distances.

If cutter radius offset is used, speeds and feeds reflect a certain nominal cutter radius. At the discretion of machine operator, cutter radius may be changed within a reasonable range. This range should appear in the setup sheet, including a note on the adjustment of speeds and feeds.

A simple setup sheet is shown in Figure 1.5.1. Feel free to improve it as necessary. A well-designed setup sheet should also include information about the material (stock) used for

machining, stock the program is based on. Not only the type of stock, also its rough dimensions, amount left for machining, its condition, and other features that are important to include in program documentation. This information is valuable at its conception and will be even more valuable in the future, mainly for repeated jobs. Many times, a program is made when the stock is not yet available.

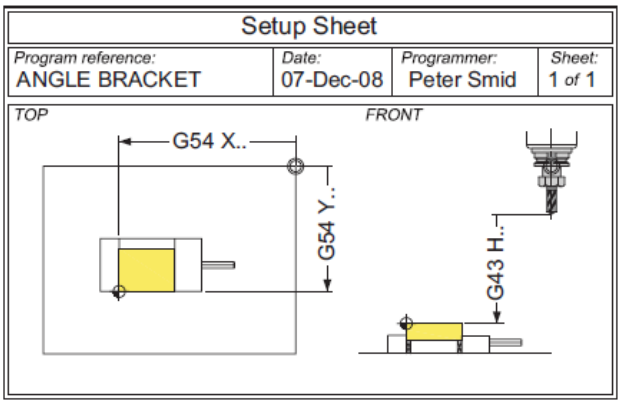


Figure 1.30 set up sheet

- **Tooling Sheet**

Although tooling is really part of the setup, it requires a separate set of data that may or may not fit on the setup sheet itself. If setups and tools used are constantly simple, it may be more convenient to have only one sheet, describing them both. However, for large or complex setups, making a separate tooling sheet is more practical.

Both, the setup sheet and tooling sheet, are part of the same documentation and complement - not replace - each other. Machine unit and its CNC system influence the contents of a tooling sheet. Tooling sheet for a lathe will be different than tooling sheet for a machining center. Data gathered for either machine will have some similarities and some unique items.

In addition to the most common items, also include any unique information in the tooling sheet, for example, to inform the operator about non-standard tools, tools that require modification, pre machined condition of the material, etc. An example of a simple tooling sheet is shown in Figure 1.5.1.

To this example, the spindle speed and feed rate calculations will be made based on the following surface speeds and chip load for each machining group (higher rates are possible):

Table 1.5.1 Tooling sheet

Tool Number	Description	Size in mm	Surface speed in m/min	Chipload per tooth in mm	Spindle r/min	Feedrate mm/min
T01	Face mill	∅100	150	0.35	477	501.0
T02	Center-cutting end mill	∅12	55	0.06	1459	175.0
T03	Center-cutting end mill	∅8	55	0.06	2188	263.0
T04	Spot drill ∅10 mm	∅4.7	25	0.08	1693	135.0
T05	Tap drill	∅3.2	25	0.07	2487	174.0
T06	Plug tap	M4×0.75	10	N/A	796	597.0

The feed rate for the face mill was calculated for three cutting edges in the material, and the feed rate for both end mills, for two flutes cutting. Tapping speed was calculated by multiplying the spindle speed by the thread pitch.

The cutting data presented here are only examples - always evaluate the actual work conditions
The main goal of explain the steps required to complete this task
Tool 1 - Face Milling
Tool 2 - Outside Con tour
Tool 2 - Circular Pocket
Tool 3 - Slot Milling
Tool 4 - Spot Drilling
Tool 5 – Drilling
Tool 6 - Tap ping

Raw material data

- Materials and related science may include:
  - Classification and mechanical properties of engineering materials

Jigs and fixture

A jig may be defined as a device which holds and located a work piece and guides and controls one or more cutting tools. The holding of the work and guiding of the tool is such that they are in true positions relative to each other.

A fixture is a device which holds and locates a work piece during an inspection or for a manufacturing operation. The fixture does not guide the tool.

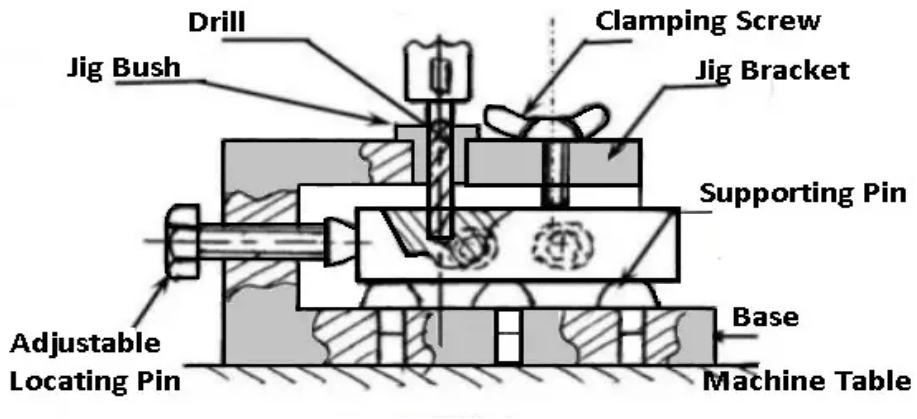


Fig 1.31.Elements of gigs

- PURPOSE OF JIGS AND FIXTURE

The fundamental motivation behind creating and utilizing reasonable jigs and installations for bunch generation in machine shops are:

- To dispense with checking, punching, situating, arrangements and so on.
- Easy, brisk, and reliably exact finding, supporting, and clasping the clear in arrangement of the cutting device
- Guidance to the cutting device like penetrates reamer and so on.
- Increase in profitability and keep up item quality reliably [3].

- ADVANTAGE OF USING JIGS AND FIXTURE

The principal elements of jigs or installation are: -

- Griping a work piece in the foreordained way of solidness and area.
- Holding segments inflexible and forestall development amid working keeping in mind the end goal to grant more noteworthy profitability and part exactness.
- Supporting and finding each segment (part) to guarantee that each is bored or machined inside as far as possible.
- Positioning parts precisely and keep up relationship and arrangement between the instrument and the work piece accurately to perform on the work piece an assembling operation.

1.6.1. Design & Manufacturing

- STEPS OF FIXTURE DESIGN

Successful fixture outlines start with a coherent and orderly arrangement. With a total investigation of the fixture's useful prerequisites, not very many plan issues happen. When they do, odds are some outline prerequisites were overlooked or belittled. The work piece, preparing, tooling and accessible machine fixtures may influence the degree of arranging required. Preparatory examination may take from a couple of hours up to a few days for more convoluted fixture outlines [4].

Fixture configuration is a five stages critical thinking process. The accompanying is a point by point investigation of each progression

Step 1:- Define requirements.

Step 2: Gather/Analyze Information

Step3. Develop Several Options

Step 4: Choose the Best Option

Step 5: Implement the Design

1.6.2. Manufacturing

- Basic Design Principle for Jig and Fixture

Basic principle while designing of jig and fixture is supporting, locating, and clamping and work holding. From the final design of semi-auto insertion this principle can be shown in Fig.1.6.1.

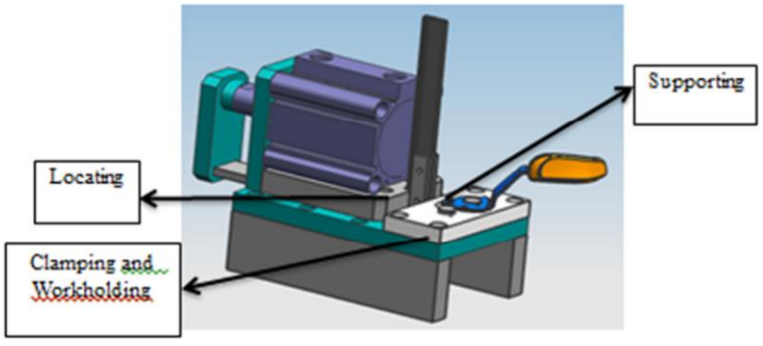


FIGURE 1.32.Classification for Basic Principle

1.6.3. Select jigs and fixture

When selecting jigs and fixtures for CNC milling, there are several factors to consider ensuring efficient and accurate machining. Here are some steps to guide you through the process:

Understand the Work piece: Begin by thoroughly analyzing the work piece and its design requirements. Consider the shape, size, and complexity of the part, as well as any critical features that need to be machined. This analysis will help you determine the specific needs for jigs and fixtures.

Determine Machining Process: Identify the specific machining operations that will be performed on the work piece. This could include operations such as face milling, contouring, drilling, tapping, or any other relevant processes. Understanding the required operations will help determine the type of jigs and fixtures needed.

Consider Work holding Requirements: Evaluate the work holding requirements based on the work piece and machining operations. Determine the points of contact and the forces involved during machining. Consider the material of the work piece, its stability, and any required access for cutting tools. These factors will influence the selection of appropriate jigs and fixtures.

Review Available Options: Research and review the available types of jigs and fixtures that are commonly used in CNC milling. Some common options include V-blocks, angle plates, clamps, vises, modular featuring systems, and custom-designed fixtures. Each type has its advantages and is suitable for specific applications.

Assess Accuracy and Repeatability: Consider the required accuracy and repeatability of the machining process. Some parts may require higher precision and tighter tolerances, necessitating fixtures with better clamping and alignment capabilities. Ensure that the selected jigs and fixtures can consistently hold the work piece securely in the desired position throughout the machining process.

Evaluate Flexibility and Versatility: Assess the flexibility and versatility of the jigs and fixtures. Ideally, they should be adjustable and adaptable to accommodate different work pieces or future design iterations. Modular fixturing systems or adjustable fixtures can provide the flexibility needed for various machining requirements.

Cost Considerations: Evaluate the cost-effectiveness of the jigs and fixtures. Consider the overall cost, including acquisition, maintenance, and any necessary customization. Balancing cost with performance and productivity is essential to maximize the return on investment.

Seek Expert Advice: If you are new to CNC milling or unsure about the selection process, seek advice from experienced machinists, engineers, or fixture specialists. They can provide valuable insights and help you choose the most suitable jigs and fixtures for your specific application.

By following these steps and carefully considering the work piece, machining processes, work holding requirements, accuracy, flexibility, and cost, you can select jigs and fixtures that will optimize the CNC milling process and ensure accurate and efficient machining operations.

Types of Jigs

There are seven types of CNC jigs that are commonly used in machining operations. They are:

Template Jig: The simplest of all the models, which uses a template to guide the tool.

Plate Jig: A plate jig improves the template jig by incorporating drill bushes on the template.

Channel Jig: A channel jig is a simple type of jig having a channel-like cross-section.

Diameter Jig: A diameter jig is used to drill radial holes on a cylindrical or spherical work piece.

Leaf Jig: A leaf jig consists of a base plate and a hinged leaf that can be opened and closed to hold the work piece.

Ring Jig: A ring jig is similar to a diameter jig, but it has a circular or ring-shaped base plate that surrounds the work piece.

Box Jig: A box jig is a complex type of jig that has a box-like structure with multiple faces and openings to hold and locate the work piece.

Types of Fixtures

Following are the 10 different types of fixtures:

1. Turning fixtures.
2. Milling fixtures.
3. Broaching fixtures.
4. Grinding fixtures.
5. Boring fixtures.
6. Indexing fixtures.
7. Tapping fixtures.
8. Duplex fixtures.
9. Welding fixtures.
10. Assembly fixtures

Difference between Jigs and Fixtures

The following are the fundamental difference between fixtures with a jig:

1. A fixture holds and positions the work but does not guide the tool, whereas a jig holds, locate and as well as guide the tool.
2. The fixtures are heavier in construction and are bolted rigidly on the machine table, whereas the jigs are made lighter for quicker handling and clamping with the table is often unnecessary.
3. The fixtures are employed for holding work in milling, grinding, planning or turning operations, whereas the jigs are used for holding the work and guiding the tool, particularly in drilling reaming or tapping operations.

Tool and die type

Ever wondered how intricate parts and components are crafted with precision in the manufacturing world? The secret lies in the power of CNC milling tools. These high-tech cutting tools create a vast array of shapes, slots, and holes, enabling manufacturers to bring their designs to life. As you embark on this captivating journey to explore the world of CNC milling tools, you'll discover the different types of cutting tools, their materials, and coatings, and the factors to consider when choosing the best tool for your project.

Unleash your creativity and empower your projects with the knowledge of CNC milling tools, their applications, and how to select the right tool for every job. From end mills to specialty tools, dive into a world where technology and craftsmanship collide to produce impeccable results.

Every artist has a set of specialty tools for unique applications, and CNC milling is no exception. Specialty CNC milling tools are designed for specific purposes, catering to projects that require a touch of finesse or a unique approach. These tools are employed in CNC milling machines to selectively remove material from a block of raw substrate, ensuring the highest level of precision and accuracy. Tools may also include

- Punch
- Die
- Die holder and punch holder
- Stripper and strip guide
- Upper bolster and lower bolster
- Pillar
- Bushing
- shank
- Die set

Safety procedures and environmental protections

Milling machines and computer-numerical-controlled (CNC) mills use moving cutters and/or move stock materials to cut shapes in materials such as metal, wood, or plastic. Follow safety precautions when using milling and CNC machines.

HAZARDS

Milling and CNC machines cut away material using rotating blades, which can throw or eject dust and chips at high speed.

- Flying chips present an **eye injury** hazard.
- Fine dust can be a **respiratory hazard**.
- Mills can also be very **loud**, which can impact hearing as well as drown out voices and alarms.

Rotating machinery presents a serious hazard, as gloves, clothing, jewelry, or loose hair can be caught, and body parts drawn into the running machine. Mills have guards to prevent some exposure; some are completely enclosed when running.

SAFETY PRECAUTIONS

- Users must be trained on the operation of the specific mill they are going to use..
- Securely clamp the stock material in place.
- Secure guards, shields, doors in place prior to starting.
- Keep hands, tools, and clothing at least 12 inches away from the moving mill and do not attempt to adjust the mill while operating.
- Wear safety glasses or goggles, and hearing protection if needed. Do not wear gloves near operating equipment.
- Keep the mill surfaces and shop floor clean of cuttings and dust. Metal filings can combust spontaneously and require a Class D fire extinguisher.

Machinery must be completely disconnected from power (and de-energized) if it is to be repaired or adjusted (refer to the Hazardous Energy Control page on the EH&S website for more information

- **Personal protective equipment (PPE)**

To protect themselves when working with machine, workers need to use the proper personal protective equipment (PPE)..

Self-check 1.

Part I:- Directions: Answer all the questions listed below & circle the answer on the later

1. Drawings are used to:
   a) Show, in multi view, what an object looks like before it is made.
   b) Standardize parts.
   c) Show what to make and the sizes to make it.
   d) All the above.
   e) None of the above.
2. Tolerances are:
   a) The different materials that can be used.
   b) Allowances in either oversize or undersize that a part can be made and still be acceptable.
   c) Dimensions.
   d) All the above.
   e) None of the above.
3. Print reading falls into two general categories. They are
   a) Visualization and tolerance
   b) Interpretation and dimensioning
   c) Visualization and interpretation
   d) Projection and orthographic
4. when tolerances are plus and minus, it is called a_____ tolerance
   a) Bilateral tolerance b) Unilateral tolerance
5. When tolerances are only plus or only minus, it is called a_____ tolerance
   a) Bilateral tolerance
   b) Unilateral tolerance

PART **II:-Directions true or false**

1. Projection means reading and understanding the information given by the drawing about the component to be manufactured. (1 point)
2. Orthographic Drawings are convectional projections of a three-dimensional object on a Two dimensional plane (1 point)
3. Profile projection plane is used to locate front view? (1point)
4. The Object lines should be drawn with thin lines? (1 point),
5. Rotating machinery presents a serious hazard in decimals

Lap Test 1.

Directions: Answer all the questions listed below.

Use the Answer sheet provided in the next page:

Date: _____Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools, and materials you are required to perform the following tasks within --- hour.

1. Explain about CNC machine?
2. Write the six CNC machine system elements
3. What are the two-machine control units in CNC?
4. _____system is also referred to as the measuring system.
5. Which key is used to cancel alarm of CNC?

Unit Two: CNC milling machine program

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- CNC Milling Coordinates system
- CNC milling operation program Simulating and editing Program.
- Document and save Program to the machine.

Upload the Program This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

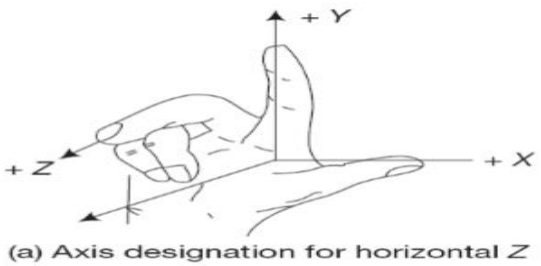
- Calculate coordinates for simple and advanced tool path machining functions
- Write program in standard CNC milling operation code format in 2D and 3D procedures
- Simulate and edit program
- Document and save program to the machine
- Download program to the machine

CNC Milling Coordinates system

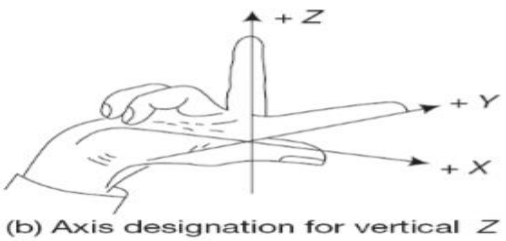
CNC motion is based on the Cartesian coordinate system. A CNC machine cannot be successfully operated without an understanding of the how coordinate systems are defined in CAM and CNC machine and how the systems work together.

The Cartesian coordinate system and then explains in detail how the coordinate systems between CAM and the CNC machine are related to each other. It also describes how the machine work coordinate system (WCS) is set on the CNC machine so the machine knows where the part is located within the work space

Finding directions in a Right hand Co-ordinate System



(a) Axis designation for horizontal Z



(b) Axis designation for vertical Z

Cartesian coordinate system

CNC motion is based on a 3D Cartesian coordinate system.

Number Line

The basis of this system is the number line marked at equal intervals. The axis is labelled (X, Y or Z). One point on the line is designated as the Origin. Numbers on one side of the line are marked as positive and those to the other side marked negative.

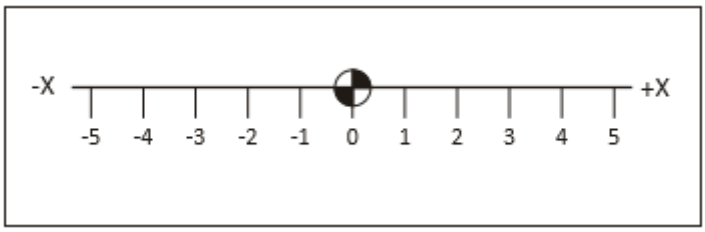


Fig 2.1.X axis number line

3D Cartesian coordinate system

The Cartesian coordinate system consists of three number lines, labelled X, Y and Z, set at 90 degree angles to each other as shown in Figure 2 below. The origin, or Datum, is where the three axes cross each other. The labels, orientations, and directions of the Cartesian coordinate system in Figure 2.2 are typical of most Vertical Machining Centre (VMC).

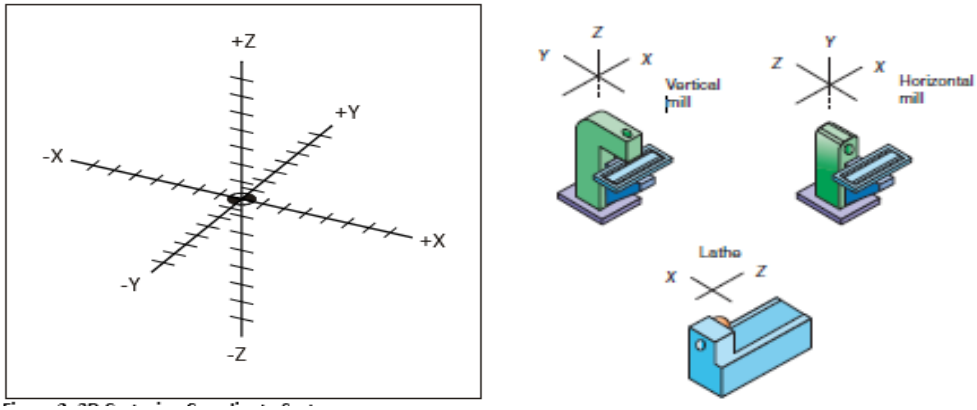


Fig 2,2 3D Cartesian coordinate system

Quadrants

Any two axes form a plane. Planes are named by the axes that define them. For example, Figure 3 shows the XY plane, which is the primary work plane for machining on a VMC. A plane can be divided into four quadrants, labeled I, II, III and IV with axes designations as shown in the illustration below.

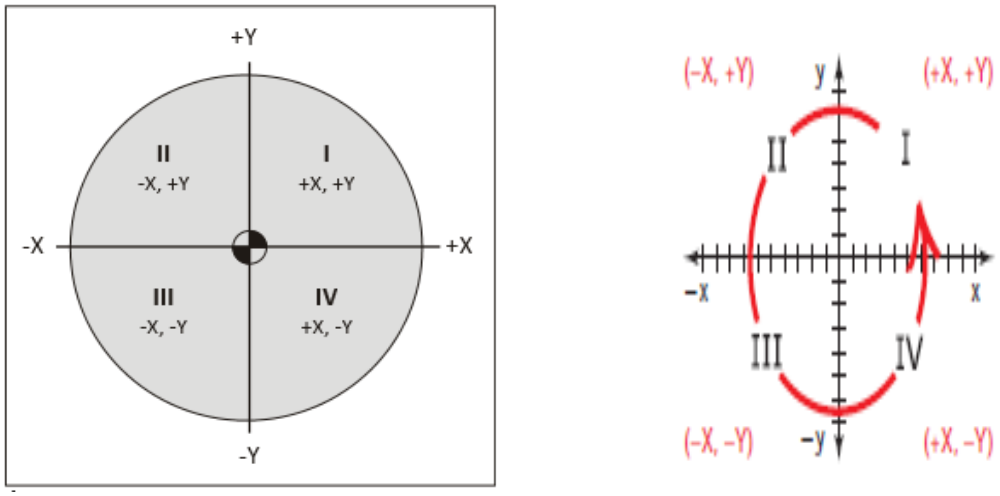


Fig 2.3 Four Quadrants of Cartesian coordinate system

Polar coordinate system

The Cartesian coordinate system is the most commonly used coordinate identifying system in CNC programming, but is not the only coordinate system used in CNC programming. The polar coordinate system is also sometimes used to identify locations on a work piece. Polar coordinates require that positions be identified by defining both an angle and a distance (like a vector in mathematics) from the origin to a specific location. The ideal use for polar coordinates is any application where a programmer may need to position the machine at multiple angular locations.

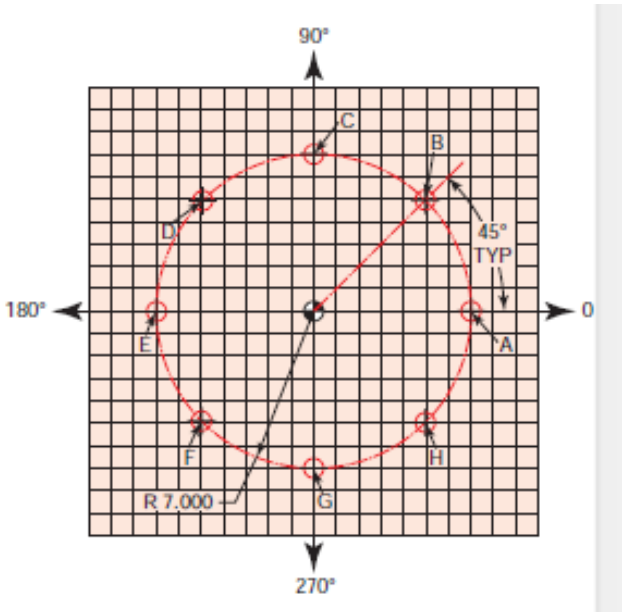


Fig.2.4 Polar coordinate system

Units

CNC Programs can be written in either Inch or Metric units. The machine can be switched with a single code to accept either.

In the United States, most programming is using inch units because most tooling is in inches and machinists are more familiar with the inch measurement system. Even if the part is designed in metric, it is usually converted to inch units for machining and metric tools are used only when no inch equivalent is available (for example when creating metric tapped holes).

CNC Machine Coordinates

The CNC Machine Coordinate System is illustrated in Figure 6. The control point for the Machine Coordinate System is defined as the centre-face of the machine spindle.

The Origin point for the machine coordinate system is called Machine Home. This is the position of the centre face of the machine spindle when the Z-axis is fully retracted and the table is moved to its limits near the back left corner.

Vertical Milling Centre (VMC) Machine Motion

CNC machines use a 3D Cartesian coordinate system. Figure 4 shows a typical 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.

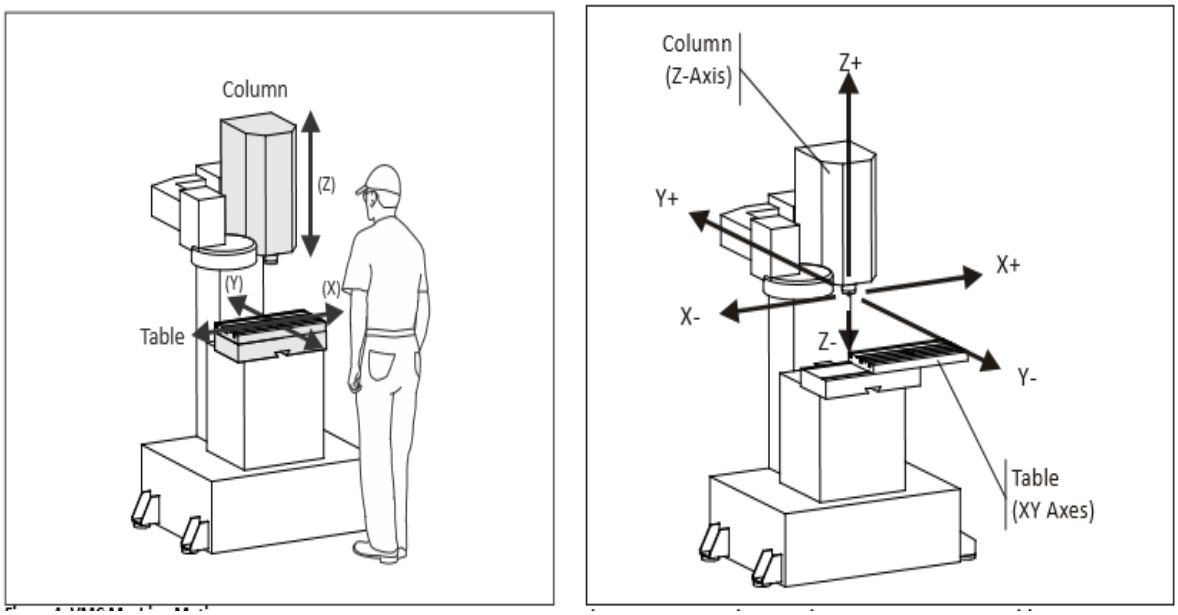


Fig 2.5.VMC system machine motion and at home position

- **Absolute positioning systems**

When using a coordinate system for programming, there are two methods available to reference work piece positions. One method is known as the **absolute positioning system**. When using absolute positioning, the coordinates of all positions will be referenced from the origin (X0, Y0, Z0). **(See Figure 2.6.)** NOTE: It can often be helpful to think of absolute coordinates as positions or locations instead of distances.

A **G90 code** sets the CNC to absolute positioning mode. This means that the CNC will interpret all location values as relative to a single zero location. That single zero location is usually the work piece zero location (set by a work offset such as G54) or the CNC machine zero location (home position)

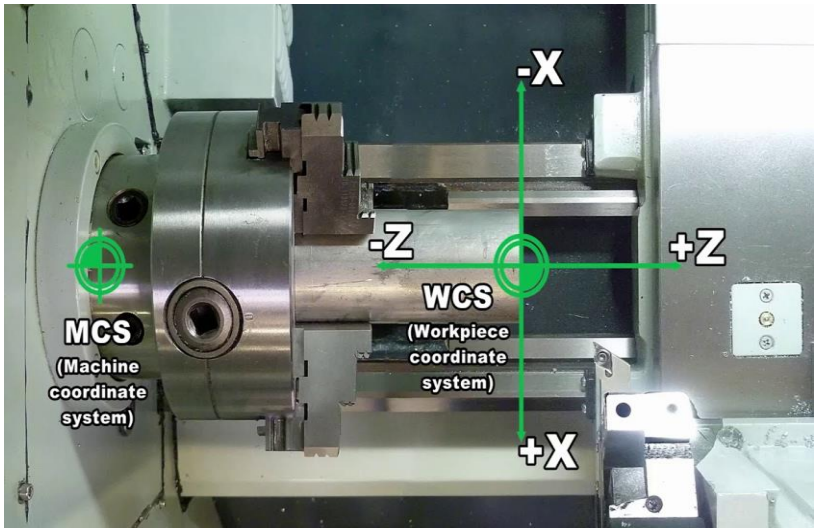


Fig. 2.6.Assume that each block on the grid equals 1mm

Using the absolute programming method, the coordinates for each identified location are: (A) X6.0 Y6.0; (B) X-6.0 Y6; (C) X-6.0 Y-6; (D) X6.0 Y-6.0. © Cen gage Learning 2012

- **Incremental positioning systems**

The other method for referencing coordinates is the incremental positioning system, which specifies a distance from the current position to the next position instead of a location related to the origin. It is important that the correct sign (+/-) be given to indicate the direction of the incremental distance.

Overall, **incremental positioning** in CNC machining allows for efficient and flexible control over tool or work piece movements by specifying incremental distances or increments relative to the current position. It is a widely used method in CNC programming for its convenience and ease

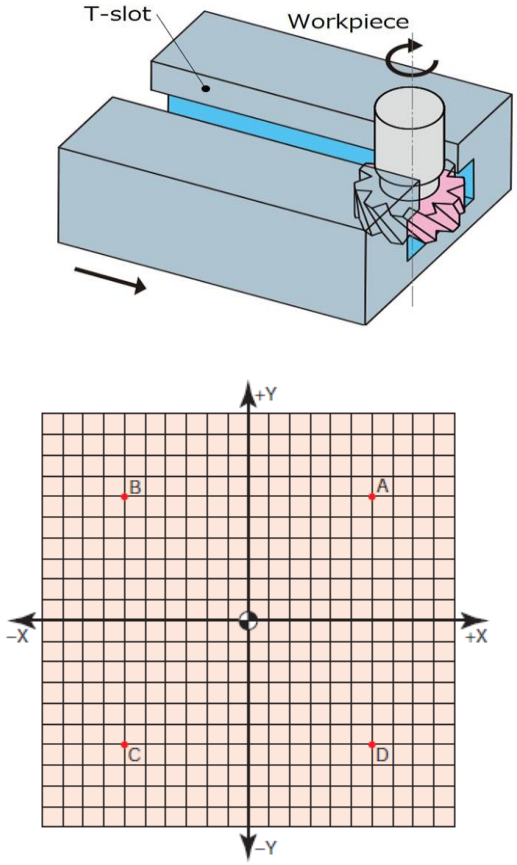


Fig 2.7 Assume that each block on the grid equals 1mm

Using the incremental programming method, assuming the cutting tool is already at position A, the coordinates for each following location are: (B) X-12.0 Y0; (C) X0.0 Y-12; (D) X12.0 Y0.0

There are advantages and disadvantages to either method of positioning depending on the situation. One disadvantage to using the incremental system is that any errors will accumulate. In other words, if a programming position error is made, that position and all positions after it will also be wrong since they all are all referenced from an incorrect position. While programming, it can also be difficult to keep track of the current position and know the next incremental position to program.

Troubleshooting a positioning mistake in an incremental program may also be very difficult. The advantage to using the incremental method is that parts requiring repetitive motions can be easily computed and programmed (example: a series of 1mm motions for drilling holes all in a row).

All things considered, absolute programming is typically less confusing and programming errors are less likely to be made. Absolute positioning is the most commonly used system in industry.

Coordinate calculation example

Calculate the positions of the points shown in figure below using

I) Absolute positioning system

II) Incremental positioning system

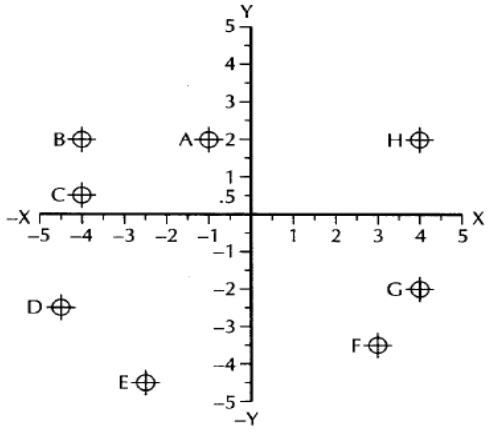


Table1. Coordinate data sheet

Points	Absolute positioning		Incremental positioning	
	X	Y	X	Y
A	-1	2	-1	2
B	-4	2	-3	0
C	-4	0.5	0	-1.5
D	-4.5	-2.5	-0.5	-3
E	-2.5	-4.5	2	-2
F	3	-3.5	5.5	1
G	4	-2	1	1.5
H	4	2	0	4

CNC milling operation program, Simulating, and editing Program

Writing program

Based on the component drawing, a lot of information is needed to be fed into the control unit for processing and machining. The information includes dimensions, shape, cutting speed, feed, depth of cut, sequence of operations, tool and work material. The said information should be fed into the machine control unit in an acceptable form. Preparation of the instructions in the above form is known as programming.

Part program

A series of instructions describing the part to be produced in an acceptable form to the control unit or computer is known as part program. Generally part programming is done by the following methods.

1. Manual part programming
2. Computer assisted part programming.

APT (Automatically Programmed Tools), there are some of the high level languages used for writing CNC programs.

How CNC Works

- Controlled by G and M codes.
- These are number values and co-ordinates.
- Each number or code is assigned to a particular operation.
- Typed in manually to CAD by machine operators.
- G&M codes are automatically generated by the computer software.

For CNC milling operations with the G - codes

May include, but is not limited to:

- face milling
- cutting shoulders, grooves, fillets and chamfers, drilling, boring
- a variety of cutters including angle, gang, end, shell, slot, form, slitting, core, cavity

A group of commands given to the CNC for operating the machine is called the program. It consists of:

Information about part geometry

- Motion statements to move the cutting tool

- Cutting speed
- Feed
- Auxiliary functions such as coolant on and off, spindle direction

The basics of CNC milling programming It uses the same standard G and M codes, can be programmed in inch or metric dimensions in both absolute and incremental programming. Many machines also are equipped with canned cycle processing and canned thread-cutting cycles.

CNC program structure

There are four basic terms used in CNC programming Character -> Word -> Block -> Program

• Character is the smallest unit of CNC program. It can have Digit / Letter / Symbol.

• Word is a combination of alpha-numerical characters. This creates a single instruction to the CNC machine. Each word begins with a capital letter, followed by a numeral. These are used to represent axes positions, federate, speed, preparatory commands, and miscellaneous functions.

• A program block may contain multiple words, sequenced in a logical order of processing.

• The program comprises of multiple lines of instructions, „blocks" which will be executed by the machine control unit (MCU).

FIXED ZERO v/s FLOATING ZERO

Fixed zero:

• Origin is always located at some position on M/C table (usually at south west corner/Lower left-hand) of the tables & all tool location are defined W.R.T. this zero

Floating Zero:

• Very common with CNC M/C used now a days.

• Operator sets zero point at any convenient position on M/C table.

• The Coordinate system is knows as work coordinate system (WCS)

Modal and Non modal commands

Commands issued in the CNC program may stay in effect indefinitely (until they explicitly cancelled or changed by some other command), or they may be effective for only the one time that they are issued.

The former are referred as Modal commands. Examples include feed rate selection and coolant selection.

Commands that are effective only when issued and whose effects are lost for subsequent commands are referred to as non-modal commands.

A dwell command, which instructs the tool to remain in a given configuration for a given amount of time, is an example of a non-modal command.

❖ **Structure of a CNC part program**

- An NC part program is made up of a series of commands that are input into the MCU in a serial manner.
- The MCU interprets these commands and generates the necessary signals to each of the drive units of the machine to accomplish the required action.
- The CNC program is required to have a particular structure that the controller can understand and it must follow a specific syntax.
- Commands are inputs into the controller in units called blocks or statements.
- Each block is made up of one or more machine commands
- In general, several commands are grouped together to accomplish a specific machining operation, hence the use of a block of information for each operation.
- Each command gives a specific element of control data, such as dimension or a feed rate. Each command within a block is also called a word.
- The way in which words are arranged within the block is called block format.
- Three different block formats are commonly used, (Fixed sequential format, Tab sequential format and Word address format)

Programming Key Letters (commonly used word addresses)

N-CODE: Sequence number, used to identify each block with in an NC program and provides a means by which NC commands may be rapidly located. It is program line number. It is a good practice to increment each block number by 5 to 10 to allow additional blocks to be inserted if future changes are required.

G-CODE: Preparatory Word, used as a communication device to prepare the MCU. The G-code indicates that a given control function such as G01, linear interpolation, is to be requested.

Most common CNC Milling G Code List

G- Code	Function	G-code	function
G00	Rapid Linear Positioning	G55	Work Coordinate System 2 Selection
G01	Linear Feed Interpolation	G56	Work Coordinate System 3 Selection
G02	CW Circular Interpolation	G57	Work Coordinate System 4 Selection
G03	CCW Circular Interpolation	G58	Work Coordinate System 5 Selection
G04	Dwell	G59	Work Coordinate System 6 Selection
G07	Imaginary Axis Designation	G60	Single Direction Positioning
G09	Exact Stop	G61	Exact Stop Mode
G10	Offset Value Setting	G64	Cutting Mode
G17	XY Plane Selection	G65	Custom Macro Simple Call
G18	ZX Plane Selection	G66	Custom Macro Modal Call
G19	YZ plane Selection	G67	Custom Macro Modal Call Cancel
G20	Input In Inches	G68	Coordinate System Rotation On
G21	Input In Millimeters	G69	Coordinate System Rotation Off
G22	Stored Stroke Limit On	G73	Peck Drilling Cycle
G23	Stored Stroke Limit Off	G74	Counter Tapping Cycle
G27	Reference Point Return Check	G76	Fine Boring
G28	Return To Reference Point	G80	Canned Cycle Cancel
G29	Return From Reference Point	G81	Drilling Cycle, Spot Boring
G30	Return To 2nd, 3rd and 4th Ref. Point	G82	Drilling Cycle, Counter Boring
G31	Skip Cutting	G83	Peck Drilling Cycle
G33	Thread Cutting	G84	Tapping Cycle
G40	Cutter Compensation Cancel	G85	Boring Cycle
G41	Cutter Compensation Left	G86	Boring Cycle
G42	Cutter Compensation Right	G87	Back Boring Cycle
G43	Tool Length Compensation + Direction	G88	Boring Cycle
G44	Tool Length Compensation - Direction	G89	Boring Cycle
G45	Tool Offset Increase	G90	Absolute Programming
G46	Tool Offset Double	G91	Incremental Programming
G47	Tool Offset Double Increase	G92	Programming Of Absolute Zero
G48	Tool Offset Double Decrease	G94	Feed Per Minute
G49	Tool Length Compensation Cancel	G95	Feed Per Revolution
G50	Scaling Off	G96	Constant Surface Speed Control
G51	Scaling On	G97	Constant Surface Speed Control Cancel
G52	Local Coordinate System Setting	G98	Return To Initial Point In Canned Cycles
G54	Work Coordinate System 1 Selection	G99	Return To R Point In Canned Cycles

X, Y & Z-CODES : Coordinats. These give the coordinate positions of the tool.

F-CODE: Feed rate. The F code specifies the feed in the machining operation.

S-CODE: Spindle speed. The S code specifies the cutting speed of the machining process.

T-CODE: Tool selection. The T code specifies which tool is to be used in a specific operation.

M-CODE: Miscellaneous function. The M code is used to designate a particular mode of operation for an NC machine tool.

Most common CNC Milling M codes List

M codes vary from machine to machine depending on the functions available on it. They are decided by the manufacturer of the machine. The M codes listed below are the common ones.

M-codes	Function
M00	Optional program stop automatic
M01	Optional program stop request
M02	Program end
M03	Spindle ON clock wise (CW)
M04	Spindle ON counter clock wise (CCW)
M05	Spindle stop
M06	Tool change
M07	Mist coolant ON (coolant 1 ON)
M08	Flood coolant ON (coolant 2 ON)
M09	Coolant OFF
M30	End of program, Reset to start
M98	Sub program call
M99	Sub program end

I, J & K-CODES: They specify the centre of arc coordinates from starting.

O	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T

- These are the headings used on the programming sheet, where:

O = Program number (Used for program identification) eg. **O0012**

N = Block or sequence number. (optional)

G = Preparatory function.

M = Miscellaneous function.

X = X co-ordinate (absolute or incremental depending on programming mode).

Y = Y co-ordinate (absolute or incremental depending on programming mode).

Z = Y co-ordinate (absolute or incremental depending on programming mode).

I = Incremental move in X from start of arc to arc center when circular interpolating.

J = Incremental move in Y from start of arc to arc center when circular interpolating.

K = Incremental move in Z from start of arc to arc center when circular interpolating.

F = feed rate in mm/min.

S = spindle Speed in RPM

P = Time dwell (in milling seconds) or a Canned cycle parameter.

Q = Canned cycle parameter.

R = Radius value when used with circular interpolation or a canned cycle parameter.

T = Tool Change.

Word Sequential Format: Used on virtually all modern controllers

N50 G00 X50Y25 Z0 F0

N60 G01 Z-1 F50 M08

N70 Z0 M09

- With this type of format, each type of word is assigned as address that is identified by a letter code within the part program.
- Thus the letter code specifies the type of word that follows and then its associated numeric data is given.
- For example, the code T represents a tool number. Thus a word of the form T01 would represent tool number 1.

- Theoretically, with this approach, the words in a given block can be entered in any sequence and the controller should be able to interpret them correctly.
- With the word address format only the needed words for a given operation have to be included within the block.
- The command to which the numeric data applies is identified by the preceding address code.
- Word format has the advantage of having more than one command in one block something that would be impossible in the other two formats.

NOTE. Each block may not necessarily contain all these items.

Most CNC machine tools use a language set by the Electronics Industry Association (EIA) in the 1960's. The official name of this language is RS-274D, but everyone refers it "G-code" or "G&M Code" because many of the words of this language begin with the letters G or M.

While many of the words used by different CNC machines are the same, there are differences between makes and models. This is due in part to machines having different configurations and options. For example, a machine with a chip conveyor will have words to turn the conveyor on and off, while a machine with outa conveyor does not. So, while RS-274D is a standard, it is not rigid or enforced. Always refer to the machine documentation for the exact words and syntax for your CNC machine

Example

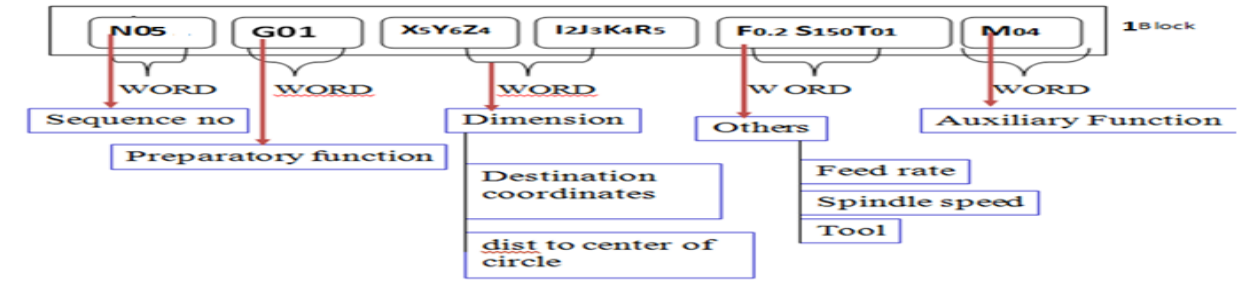
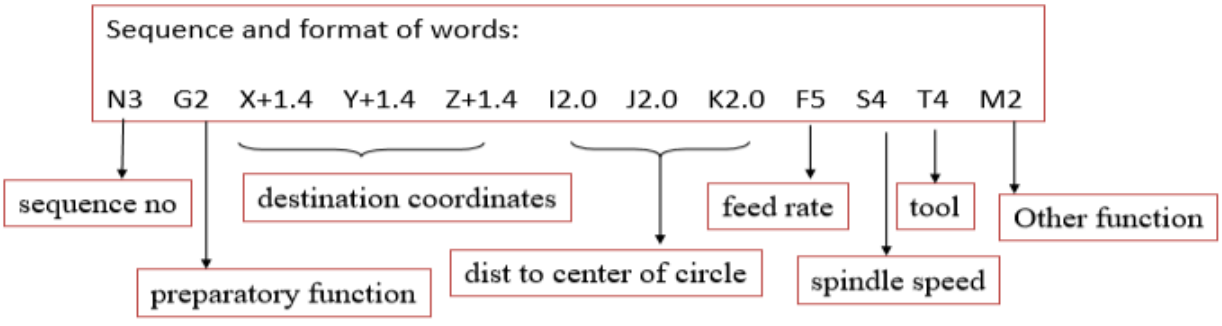
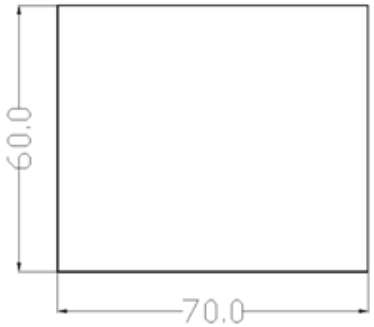


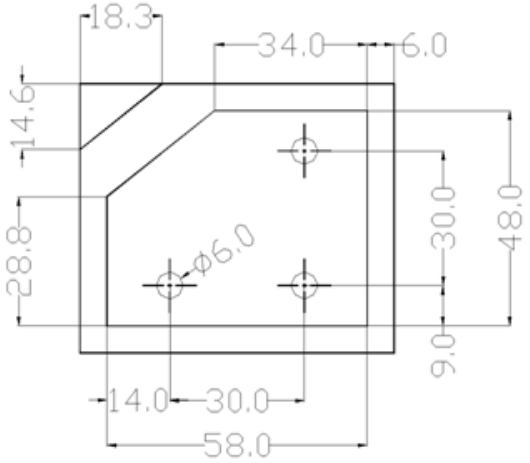
Fig 2.8.word address format



Programing Examples

1. Write a G-code program for the part shown below





A. Raw material B. Finished part

Note: Work Zero is at the center of the work, we use absolute positioning system we write a program for counter milling and drill milling (two programs) for counter milling the depth is 5mm and for drilling is 10mm

0001(program for counter)

G90 G54 G00 X0.0 y0.0 S100 m03;

Z100.0;

Z2.0;

X28.0;

G01 Z-5.0 F20.0;

G41 Y-24.0 D01 F100;

X -28.0;

Y4.8;

X-6.0 Y24.0;

x28.0;

y0.0;

G00 Z00;

0002(program for Drilling)

G90 G54 G00 X0.0 y0.0 S100 m03;

Z100.0;

Z2.0;

G98 G81 x15.0 y15.0 R5.0 Z-10.0 F100;

y-15.0

x-15.0

G00 Z100.0;

G80 x0 y0 m05 ;

G40 X0 Y0 M05;

M30;

CNC Milling Machine Operations

Face milling: Face milling is a machining operation that controls the height of a machined part. For most applications, face milling is a relatively simple operation, at least in the sense that it usually does not include any special contouring motions. Cutting tool used for face milling is typically a multi tooth cutter, called a **face mill**, although end mills may also be used for certain face milling operations, usually within small areas. Top surfaces machined with a face mill are generally perpendicular to the facing cutter axis. In CNC programming, face milling operations are simple, although two important considerations are rather critical:

- Face mill diameter selection
- Tool initial starting position in relation to the part

It helps to have some experience and knowledge of face milling principles, such as the right cutter and insert selection, distribution of cuts, machine power consumption, and several other technical considerations. Some of the basic ones are covered in this chapter, but manufacturers' tooling catalogues and various technical references will be a more in-depth source.

- Programming for face milling is so effortless that, in fact, many programmers do not pay sufficient attention to proper selection of the face milling cutter, proper inserts, and they do not even consider machine requirements and capabilities.

Single Face Mill Cut: The first face mill programming example (in G20mode), uses a 5_3 plate (1 inch thick) that has to be face milled along the whole top surface to the final thickness of 0.800. Figure 5-6 shows this simple drawing.

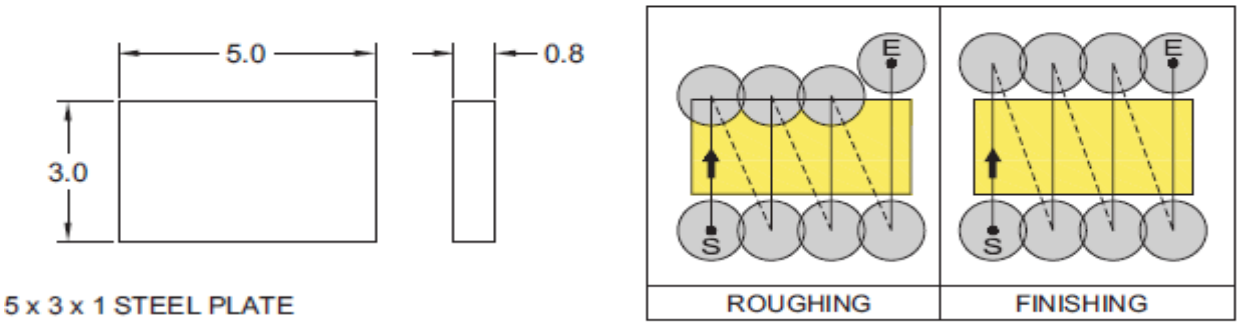


Figure 2.9 steel plates

O2801

(SINGLE FACE MILLING CUT)

N1 G20

N2 G17 G40 G80

N3 G90 G54 G00 X7.75 Y1.0 S344 M03

N4 G43 Z1.0 H01

N5 G01 Z0 F50.0 M08

N6 X-2.75 F21.0

N7 G00 Z1.0 M09

N8 G28 X-2.75 Y1.0 Z1.0

N9 M30

Contour Milling: Even with the ever-increasing use of carbide cutters for metal removal, the traditional HSS (high-speed steel) end mills still enjoy a great popularity for a variety of milling operations and even on lathes. These venerable cutters offer several benefits - they are relatively inexpensive, easy to find, and do many jobs quite well. End mills are the most common tools used for peripheral milling. There is a wide selection of end mills available for just about any conceivable machining application. Traditional end mills come in metric and imperial sizes, variety of diameters, styles, number of cutting flutes, numerous flute designs, special corner designs, shanks, and tool material compositions.

Pockets and slots: Many applications for a CNC machining center, the material has to be removed from the inside of a certain area, bounded by a contour and a flat bottom. This process is generally known as **pocketing**. To have a true pocket, the contour that defines the pocket boundary must be closed. However, there are many other applications, where the material has to be removed from an open area, with only a partial contour defined. An **open slot** is a good example of this type.

Slot milling: Slots are often considered as special types of grooves. These 'grooves' usually have one or two radial ends. If there are two ends, they are joined by a straight groove. A slot can be either open or closed, with the same size radius on either ends, two different radii, or one radius only. Atypical slot that has only one end radius is called a keyway.

Slots can be open or closed, straight, angular, circular, with straight walls or shaped walls (using a tapering end mill).

Programming slots with accuracy in mind usually requires a roughing operation and a finishing operation. Both operations can be made with the same tool or with two or more tools, depending on the part material, required dimensional tolerances, surface finish, and other conditions. Certain slots, for example keyways, can be done with special cutters, called slotting cutters, rather than an end mill. To program a slotting cutter is usually a simple process of a linear motion - in and out. More complex – and more accurate - slots are machined with end mills, and the walls of the slot are contoured under program control. Figure 2.10 shows a drawing of a typical open slot. This drawing will be used to illustrate the programming techniques of an open slot.

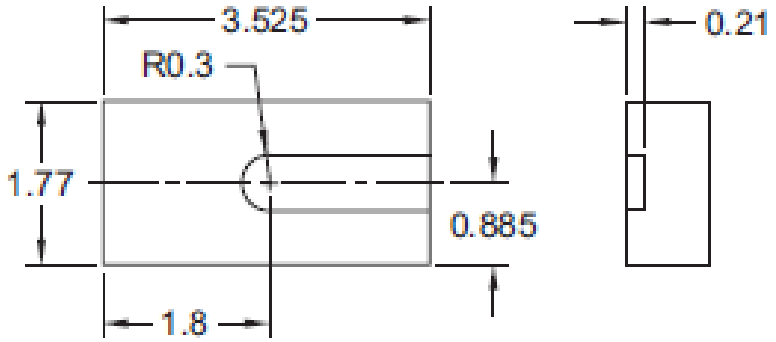


Figure 2.10 open slot

```
o3301 (open slot)
n1 g20                                  (inch mode)
n2 g17 g40 g80                          (startup settings)
n3 g90 g54 g00 x3.875 y0.885 s950 m03   (start)
n4 g43 z0.1 h01 m08                     (start position above)
n5 g01 z-0.2 f50.0                      (0.01 left on bottom)
n6 x1.8 f8.0                            (cut to slot radius cntr)
n7 g00 z0.1                             (retract above work)
n8 x3.875                               (return to start)
n9 g01 z-0.21 f50.0                     (feed to full depth)
n10 g41 y1.185 d01 f8.0                 (approach contour)
n11 x1.8                                (cut top wall)
n12 g03 y0.585 r0.3                     (cut slot radius)
n13 g01 x3.875                          (cut bottom wall)
n14 g00 g40 y0.885                      (return to start point)
n15 z1.0 m09                            (retract above work)
n16 g28 x3.875 y0.885 z1.0 m05          (m© zero)
n17 m30                                 (end of program)
```

Pocket Milling: Pocket milling is also a typical and common operation on CNC machining centers. Milling a pocket means to remove material from an enclosed area, defined by its boundary. This bounded area is further defined by its walls and bottom, although walls and bottom could be tapered, convex, and concave, rounded, and have other shapes. Walls of a pocket create the boundary contour. Pockets can have square, rectangular, circular or undefined shape, they can be empty inside or they may have islands (areas that are not machined).

Programming pockets manually is usually efficient only for simple pockets, pockets of regular shapes, such as rectangular or circular pockets. For pockets with more complex shapes and pockets with many islands, the assistance of a computer is usually required.
The most common pockets are also the easiest to program. They all have a regular shape, without any islands:

- Square pocket
- Rectangular pocket
- Circular pocket

Square and rectangular pockets are fundamentally the same and apart from their different length-
to Program example for rectangular pocket

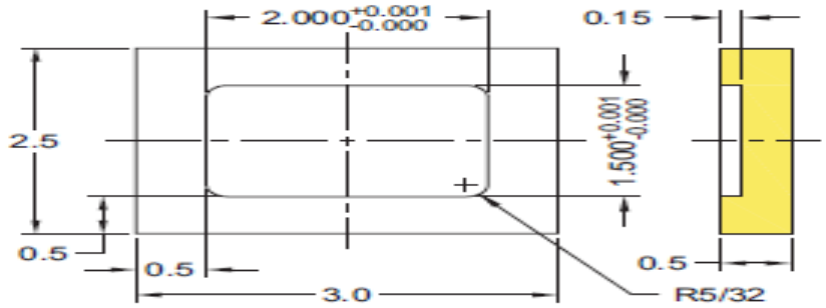


Figure 2.11 Rectangular pocket

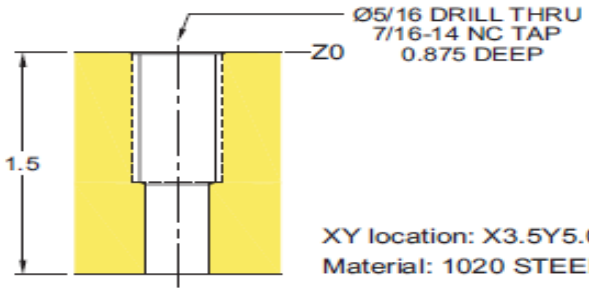
- O3303 (RECTANGULAR POCKET)
- N1 G20
- N2 G17 G40 G80 T01 (0.250 ROUGHING SLOT DRILL)
- N3 M06
- N4 G90 G54 G00 X0.66 Y0.66 S1250 M03 T02
- N5 G43 Z0.1 H01 M08
- N6 G01 Z-0.15 F7.0
- N7 G91 X1.68 F10.0 (CUT 1)
- N8 Y0.236 (STEPOVER 1)
- N9 X-1.68 F12.0 (CUT 2)
- N10 Y0.236 (STEPOVER 2)
- N11 X1.68 (CUT 3)
- N12 Y0.236 (STEPOVER 3)
- N13 X-1.68 (CUT 4)

- N14 Y0.236 (STEPOVER 4)
- N15 X1.68 (CUT 5)
- N16 Y0.236 (STEPOVER 5)
- N17 X-1.68 (CUT 6)
- N18 G40 G00 Z0.1 M09
- N19 G28 Z0.1 M05
- N20 X-2.0 Y10.0
- N21 M30

Machining Holes: When programming CNC machining centers, there is always a hole or two that must be drilled, bored, reamed, taped, etc. From a simple spot drill to reaming, tapping and a complex back boring, the field of holes machining is exceptionally large. This topic looks at several available programming methods for machining holes and presents techniques used.

Before machining even, a single hole on a CNC machine, all required tool paths must be programmed. Before that, cutting tools must be selected, speeds and feeds applied, the best setup determined, and many other related issues must be resolved. Regardless of the exact approach, always start with a thorough evaluation of the given hole.

Evaluation step number one relates to part drawing. It will define the material to be machined, exact holes location and its dimensions. Holes are often described, rather than dimensioned and CNC programmer has to supply the missing details. Figure 2.12 shows a medium complexity hole that can be machined using a CNC machine.



- Figure 2.12 drilling single hole

O2601 (SINGLE HOLE EXAMPLE)

- (T01 - 5/8 DIA - 90 DEGREE SPOT DRILL)
- N1 G20
- N2 G17 G40 G80 T01
- N3 M06
- N4 G90 G54 G00 X3.5 Y5.0 S900 M03 T02
- N5 G43 Z0.1 H01 M08
- N6 G99 G82 R0.1 Z-0.2338 P250 F4.0
- N7 G80 Z1.0 M09
- N8 G28 Z1.0 M05
- N9 M01
- (T02 - LETTER U DRILL - 0.368 DIA DRILL)
- N10 T02
- N11 M06

- N12 G90 G54 G00 X3.5 Y5.0 S1100 M03 T03
- N13 G43 Z0.1 H02 M08
- N14 G99 G83 R0.1 Z-1.085 Q0.5 F8.0
- N15 G80 Z1.0 M09
- N16 G28 Z1.0 M05
- N17 M01
- (T03 - 5/16 DRILL THROUGH - 0.3125 DIA)
- N18 T03
- N19 M06
- N20 G90 G54 G00 X3.5 Y5.0 S1150 M03
- N21 G43 Z0.1 H03 M08
- N22 G98 G81 R-0.985 Z-1.644 F8.0
- N23 G80 Z1.0 M09
- N24 G28 Z1.0 M05
- N25 M01
- (T04 - 7/16-14 TAP)
- N26 T04
- N27 M06
- N28 G90 G54 G00 X3.5 Y5.0 S750 M03 T01
- N29 G43 Z0.4 H04 M08
- N30 G99 G84 R0.4 Z-0.95 F53.57 (F = S x LEAD)
- N31 G80 G00 Z1.0 M09
- N32 G28 Z1.0 M05
- N33 G00 X-1.0 Y10.0 (PART CHANGE POSITION)
- N34 M30

2.2.1. Program Simulation

- Use simulation software: Many CNC machines have simulation capabilities built into their control systems or provide separate simulation software. This software allows you to visualize and verify the program's actions before running it on the actual machine.
- Load the program into the simulation software: Import the CNC program into the simulation software for analysis and visualization.
- Review the simulation: The simulation software will display a virtual representation of the machining process, showing the tool's path, material removal, and any potential collisions or errors. This step enables you to identify and correct any issues before running the program on the actual machine.

2.2.2. Editing the Program:

Open the program for editing: Access the CNC program either through the control systems Built-in editor or by using the software used to create the program.

Make necessary changes: Edit the program's code to modify parameters, tool paths, cutting parameters, or any other required adjustments. This can be done manually or through the software's graphical interface.

Validate the changes: After making edits, it's important to review and verify them to ensure they are correct and won't cause any issues during machining.

- Transferring the Edited Program to the Machine:

Save the edited program: Save the modified CNC program with the necessary changes.

- Transfer the program to the machine: Use the same method as in step 1 to transfer the edited program to the machine's control system.

Document and save Program to the machine

To document and save a program to a CNC milling machine, you typically follow these steps:

1. Prepare the Program: Create or obtain the CNC program using a software application compatible with your CNC machine. The program should include all the necessary instructions for the desired machining operations.
2. Connect to the CNC Machine: Establish a connection between your computer and the CNC milling machine. This can be done using various methods such as USB, Ethernet, or serial communication.
3. Access the Control System: Use the appropriate software or interface to access the control system of the CNC milling machine. This can be a dedicated software provided by the machine manufacturer or a built-in control panel on the machine itself.
4. Import the Program: Locate the option to import or load a program into the control system. Select the program file you prepared earlier and import it into the machine's memory.
5. Review the Program: Once the program is loaded, verify that it is correctly displayed on the control system. Review the code to ensure it matches your intended machining operations. This is an important step to avoid any potential errors or discrepancies.
6. Save the Program: Once you have confirmed the program's accuracy, you need to save it to the CNC milling machine's memory. Look for the option to save the program or store it in the machine's program library. Follow the prompts to assign a name or identifier to the program for future reference.
7. Document the Program: Documenting the program is crucial for future reference and troubleshooting. Create a record of the program, noting important details such as the program name, version, date created, and any specific instructions or parameters. You can maintain this documentation electronically or in a physical format, depending on your preference.

It is important to consult the user manual or documentation provided by the CNC milling machine manufacturer for specific instructions on how to document and save programs. The exact steps and terminology used may vary depending on the machine's make and model.

Remember to follow proper file management practices and maintain backups of your CNC programs to prevent any loss of data or accidental modifications.

If you have access to the specific documentation or software for your CNC milling machine, it is recommended to refer to those resources for detailed instructions tailored to your machine's requirements.

Upload the Program

To upload a program to a CNC milling machine, follow these general steps:

1. Prepare the Program: Create or obtain the CNC program using a compatible software application. Ensure that the program is saved in a format supported by your CNC milling machine (e.g., G-code).
2. Connect to the CNC Machine: Establish a connection between your computer and the CNC milling machine using a suitable method such as USB, Ethernet, or serial communication.
3. Access the Control System: Use the appropriate software or interface to access the control system of the CNC milling machine. This can be a dedicated software provided by the machine manufacturer or a built-in control panel on the machine itself.
4. Select the Program Transfer Option: Locate the program transfer or file upload option within the control system software or interface. This option may be named differently depending on the specific machine and software.
5. Choose the Transfer Method: Depending on the CNC milling machine and its capabilities, you may have multiple options to transfer the program. Common methods include:
    - USB: If your machine supports USB connectivity, you can transfer the program by connecting a USB storage device (e.g., flash drive) to the machine or computer and selecting the appropriate files for transfer.
    - Ethernet or Network: If your machine is connected to a network, you may be able to transfer the program directly from your computer to the machine using network sharing or file transfer protocols.

- Serial Communication: Some older machines use serial communication, where you connect your computer to the machine using a serial cable and use a terminal emulator software to transfer the program.

6. Select the Program File: In the transfer option, browse your computer's file system to locate the CNC program file that you want to upload to the machine. Select the file for upload.
7. Initiate the Transfer: Follow the prompts within the control system software or interface to initiate the transfer process. This may involve confirming the file selection and starting the upload process.
8. Monitor the Transfer: During the file transfer, monitor the progress to ensure that it completes successfully without any errors or interruptions. The control system software or interface may provide a progress bar or status indicator.
9. Confirm the Upload: Once the transfer is complete, verify that the uploaded program is correctly displayed on the CNC milling machine's control system. Review the program code to ensure it matches your intended machining operations.
10. Safely Disconnect: Once the program is successfully uploaded, safely disconnect your computer from the CNC milling machine. Follow any recommended procedures to ensure a proper disconnection.

Remember to consult the user manual or documentation provided by the CNC milling machine manufacturer for specific instructions on how to upload programs. The exact steps and terminology used may vary depending on the machine's make and model.

If you have access to the specific documentation or software for your CNC milling machine, it is recommended to refer to those resources for detailed instructions tailored to your machine's requirements

Self-check 2

part I

Direction: Choose the best answer for the following questions. Use the Answer sheet provided in the next page: Each question worth one point

1. In the third quadrant of Cartesian coordinate system
   A. The X and Y values are positive
   B. The X value is negative and Y value is positive
   C. The X and Y values are negative
   D. The X value is positive and Y value is negative
2. Benefits of using the absolute programming system versus the incremental system?
   A. In absolute programming an error will accumulate
   B. In absolute programming errors are less likely to be made
   C. It is difficult to program using absolute system
   D. A and C.
3. Which coordinate system uses an angle and a distance to identify a position?
   A. Cartesian coordinate system
   B. polar coordinate system
   C. A and B
4. Which positioning system specifies a distance from the current position to the next position?
   A. Absolute
   B. Incremental
   C. Polar
   D. All
5. In _____positioning system the coordinates of all positions will be referenced from the origin (X0, Y0, Z0).
   A. Cartesian
   B. Polar
   C. Incremental
   D. Absolute.

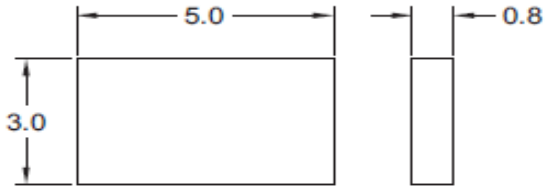
part II

Part II. Direction: Say true if the statement is true and false if the statement is false for the following questions.

1. Suppose you want to program using absolute positioning system, which G-code is used G91
2. Horizontal machine center is popularly used to machine components with doubly curved surfaces like aerospace components and dies and molds
3. A CNC milling machine operation that controls the height of a machined part is. Face milling
4. G01 and G00 two codes are used to activate automatic cutter radius compensation
5. M08command would be given to turn on the spindle in a clockwise direction at 1500 RPM?
6. For program errors, the operator will try to take corrective measures to eliminate the problem

Operation Sheet: -2

Instruction:-Programing the given project



Operation Title: - single face milling operation

Purpose: - Write single face milling cut & produce the above project

Conditions or situations for the operations: - 5x3x1 steel plate

- ✓ Safe working area
- ✓ Properly operated tools and equipment
- ✓ Effectively use of design soft ware
- ✓ Good understanding of programming

Precautions: - Safe working, free from accidents

O2801
(SINGLE FACE MILLING CUT)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X7.75 Y1.0 S344 M03
N4 G43 Z1.0 H01
N5 G01 Z0 F50.0 M08
N6 X-2.75 F21.0
N7 G00 Z1.0 M09
N8 G28 X-2.75 Y1.0 Z1.0
N9 M3

Equipment Tools and Materials: -

- ✓ Computer/pc
- ✓ Drawing instruments
- ✓ Designee soft wares/ solid work, Catia, AutoCAD,
- ✓ CNC milling machine

Quality Criteria: - Accurate, Precise & Exact

Lap test 2

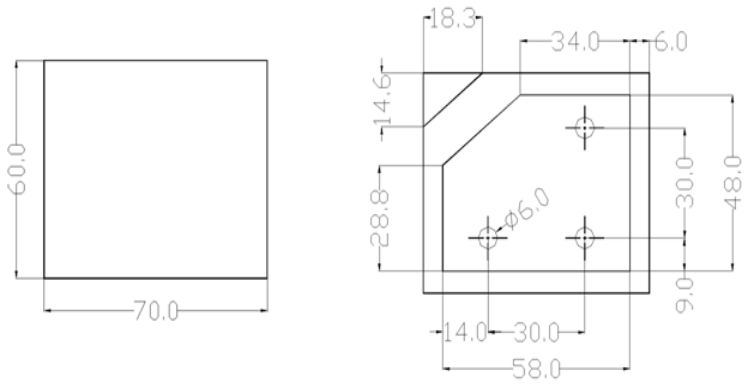
Directions: Answer all the questions listed below.

Use the Answer sheet provided in the next page:

Date: _____Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools, and materials you are required to perform the following tasks within --- hour

Task1. Writing a G-code program for the part shown below



Unit three: - CNC milling operations

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Standard operating procedures.
- Machine zero and work zero.
- Dry run with the desired tool path movements.
- CNC milling DNC
- Corrective measures/adjustments

This guide will also assist you to attain the learning outcomes stated in the cover page.

Specifically, upon completion of this learning guide, you will be able to:

- Mount or set up work piece
- Perform machine zero and work zero
- Perform Dry run
- Perform DNC
- Perform CNC milling operations
- Perform corrective measures/adjustments
- Observe safety procedures and environmental protection
- Use personal protective devices

Standard operating procedures

Standard operating procedures (SOPs) for advanced CNC milling machines typically encompass a wide range of operations and may vary depending on the specific machine and workplace requirements. Here are some key elements that are often included in SOPs for advanced CNC milling machines:

Safety Precautions:

- Emphasize the importance of safety and ensure that operators are trained in proper safety procedures.
- Follow all safety guidelines and regulations, including wearing appropriate PPE and using machine guards.
- Conduct regular inspections of the machine for any potential safety hazards and address them promptly.
- Establish emergency stop procedures and ensure operators are familiar with them.

Machine Startup:

- Perform a comprehensive machine inspection to ensure all components are in good working condition.
- Check and replenish lubricants, coolants, or other necessary fluids.
- Power on the machine following the manufacturer's instructions and allow it to complete the startup sequence.
- Verify that all systems, including axes, spindle, and tool changers, are functioning correctly.

Work piece Setup:

- Select the appropriate work piece and secure it properly to the machine's worktable or fixture using suitable methods such as clamps, vises, or vacuum systems.
- Utilize advanced work holding techniques, such as zero-point clamping systems or pallet changers, for efficient work piece setup.
- Align the work piece accurately using advanced measuring tools like touch probes or laser alignment systems.
- Establish the coordinate system based on the part design and machining requirements.

Tool Setup and Management:

- Choose the optimal cutting tools based on the machining operation, material, and part geometry.
- Utilize advanced tool management systems for tool identification, organization, and tracking.
- Incorporate automatic tool changers and tool length/diameter compensation systems for efficient tool setup.
- Implement tool life management strategies, including tool wear monitoring and replacement protocols.

Program Loading and Verification:

- Load the CNC program into the machine's control system using the appropriate method (e.g., USB, Ethernet, network transfer).
- Utilize advanced CAM software for generating CNC programs and ensuring accuracy.
- Verify the program using simulation software or advanced features in the control system, such as 3D visualization or collision detection.
- Conduct a comprehensive review of the program code to ensure it aligns with the intended machining operations and parameters.

Machine Operation:

- Execute the CNC program, allowing the machine to perform the advanced machining operations on the work piece.
- Monitor the machine during operation, paying attention to any alarms, warnings, or abnormal conditions.
- Optimize machining parameters, such as feed rates, spindle speeds, tool paths, and coolant flow, for improved performance and quality.
- Utilize advanced features like high-speed machining, adaptive control, or probing routines for enhanced efficiency and accuracy.

Post-Machining Tasks:

- Carefully remove the finished work piece from the machine, ensuring proper handling and avoiding damage.
- Conduct thorough inspections of the machined part, utilizing advanced measuring equipment such as coordinate measuring machines (CMMs) or optical inspection systems.
- Document relevant information, including machine settings, tool changes, quality inspection results, and any deviations or issues encountered.

- Clean the machine, work area, and tools, removing chips, coolant, or debris, and store tools in designated locations.

Shutdown:

- Follow the manufacturer's instructions for proper machine shutdown procedures, including turning off power, cooling systems, and any auxiliary equipment.
- Conduct routine maintenance tasks according to the manufacturer's recommendations.
- Leave the work area clean and organized for the next user or shift.

It is important to note that the SOPs for advanced CNC milling machines should be tailored to the specific machine's capabilities and features, as well as any industry or workplace requirements. Always refer to the manufacturer's guidelines and consult with your organization's safety and operational procedures when creating or following SOPs for advanced CNC milling machines.

Machine zero and work zero

In advanced CNC milling machines, the terms "machine zero" and "work zero" refer to specific reference points used for positioning and programming the machine.

- **Machine Zero**

Machine zero is a fixed position on a CNC machine that can be reached repeatedly, on request, through the control panel, MDI (manual data input), or program code execution any reference to the Z-axis in the description was quite intentional. Z-axis machine zero position for a vertical machining center is always where the Automatic Tool Change (ATC) takes place. This is a built-in location, normally placed at a safe distance from the machine table and the work area. For most machines, standard machine zero of CNC machining centers is at the extreme travel limit of each axis in the positive direction. There are exceptions, as may be expected. In both illustrations, the arrows indicate tool motion direction towards the work area. Moving the tool from machine zero into the opposite direction will result in a condition known as over travel - compare the two possibilities:

Tool motion from machine zero, if machine zero is located at the upper right corner:
X+ Y+ Z+ ... tool motion will over travel
_ Tool motion from machine zero, if machine zero is located at the upper left corner:
X- Y+ Z+ ... tool motion will over travel

The other two corners (lower left and lower right of the XY view) are not used as machine zero.

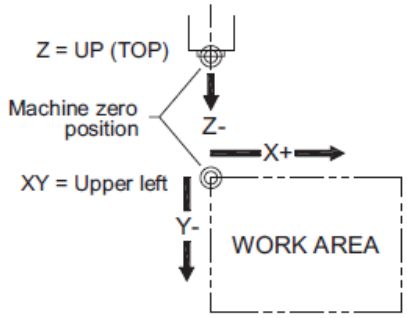


Figure3.2.1 Machine zero

- **Work holding setup**

When using a vise or fixture, the base along with the machine's table should be cleaned and inspected for debris and defects. Once the device has been inspected and cleaned, it is placed on the table, aligned using a dial indicator, and clamped when oriented properly. If parallels are to be used, they must be chosen to allow the proper height of the work in the vise jaws. The width and placement of the parallels within the bed of the vise should also be planned so that tools that will machine all the way through the work piece will not hit with the parallels. When clamps are used, special attention needs to be paid to their overall height. Place clamps strategically to prevent collision during machine motion. The same procedures for work piece alignment used for manual milling may also be used for CNC milling. No matter which device is used, when multiple parts are to be machined it is important to use the same torque on clamp fasteners and vise screws on each one so that the positioning of the part is consistent. This is especially important to prevent distortion of soft or thin parts.

- **Machine and Work Coordinate Systems**

The Cartesian coordinate system that identifies the location of the origin on the work piece is called the work coordinate system (WCS). The WCS is movable and its origin can be placed anywhere on the work piece for ease of programming. The machine has a coordinate system of its own called the machine coordinate system (MCS). This coordinate system is in a fixed location and its origin cannot be changed or moved. The MCS is used for the machine's own reference purposes and helps it to keep track of how far each axis can move before it runs out of travel. These two coordinate systems play an essential role in machine setup. **See Figure3.2.2**

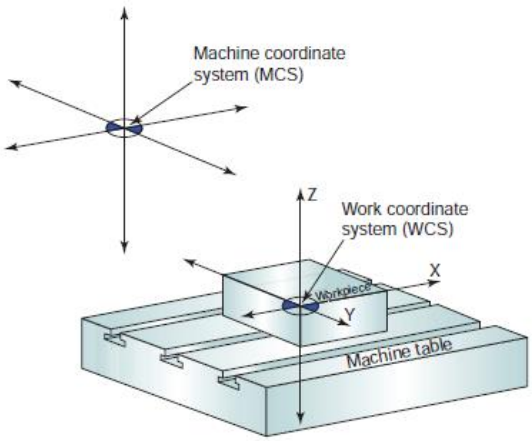


Figure3.2.2. Relationship between machine coordinate system to work coordinate system

- **Power-Up and Homing**

The very first step to operating CNC milling equipment is to power up the machine properly. Since there are many different machine variations, refer to a specific machine's manual for the correct procedure. After the machine has been properly turned on, most machines require a reference return to the machine's origin, or machine home position, of the MCS. This process is called a homing procedure. Recall that the machine moves its axes by rotating a ball screw with a servo motor. Again, this motor can monitor and adjust axis position by how far its shaft rotates. When the machine power is switched off and the MCU is no longer able to monitor and adjust axis position, it simply loses track of its position. Therefore, every time a CNC is powered up from a total shutdown, it must be homed. The homing procedure is used to re zero and establishes MCS location so that it may begin keeping track of position again. By homing, the machine is also able to recall the position of the WCS that was active before the machine was turned off. This prevents having to reset the location of the work coordinate system origin each and every time the machine is powered up. The other reason why homing is so important is that once the machine knows where each axis is positioned, it will also know the limits of the slide's travel. Severe machine damage can occur if a machine overruns its axis travel limits! Each machine requires specific steps to perform the homing procedure. These steps can also be found in a specific machine's operating manual, but the basic steps are as follows:

- Select "zero-return" or "home" mode on the machine control panel.
- Jog each axis in the direction of the machine's home position (usually the same position as the machine's coordinate system [MCS] origin) with the jog arrow keys. If the

machine axes were already at home position they must be jogged away and then sent back toward home position.

- As each axis is sent in the appropriate direction, most machines will automatically complete the procedure by moving the axis and then slowing as a sensor or switch is approached.
- When the switch is tripped, the machine will zero the machine coordinate system and will be ready to be set up or start machining using the previous setup.

Work offset setting

The work piece setup procedure establishes the work offset or origin location for the WCS. All work piece coordinates for programming will be referenced from this origin. This is established by finding the offset or "shift" from the MCS origin to the intended WCS origin. A milling work offset is usually established for the X-, Y-, and Z-axes

Recall that the MCS never changes position. It is reset by zeroing-out at the same position each time the machine is homed. Consider this point an unchanging reference point. Since the position of this machine origin never changes, but the work origin changes with each new work piece setup, the work offset for the part is defined as a distance referenced from the MCS origin. Some controls call this offset a work shift because it essentially shifts the machine origin to the location of the work origin. **Figure 3.2.3** shows an illustration depicting work shift for CNC milling. Milling work pieces will change in shape, location, and origin placement from job to job. Therefore the origin must be reset for each newly set-up job by establishing a new WCS. Some machines can automatically set the work offset value by pressing "set X," "set Y," or "set Z" when in position using a locating device (edge finder, indicator, or gage block).

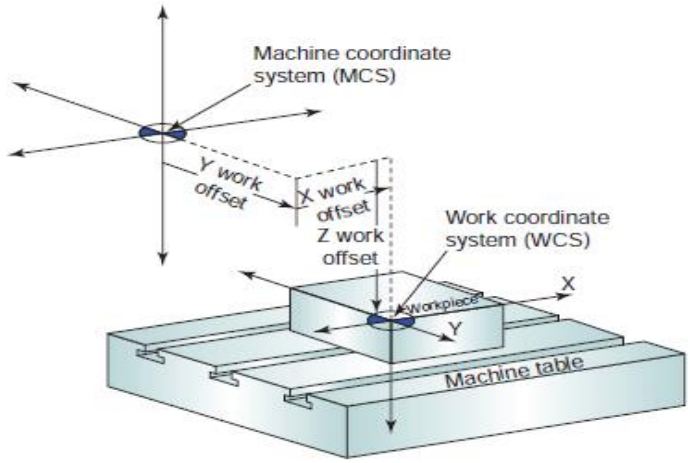


Figure **3.2.3** Work piece offset, or work shift, is the distance from the MCS origin to the desired location of the WCS origin.

- Work Zero:

Work zero, also called the work piece origin or work piece reference point, is a user-defined reference position that is specific to each work piece or part being machined. It represents the desired starting point for milling operations relative to the work piece's geometry. The work zero is typically set at a specific location on the work piece using a variety of methods, such as edge finders, touch probes, or manual measurement techniques

When programming a CNC milling machine, the machine zero and work zero are used to establish the relationship between the machine's coordinate system and the work piece being machined. By defining the work zero position, the machine can accurately position the tool relative to the work piece based on the programmed coordinates.

During the setup process, the operator will establish the work zero by manually moving the machine's spindle or tool to the desired location on the work piece and setting the corresponding coordinates in the machine's control system. The machine zero and work zero coordinates are typically specified in a specific coordinate system (e.g., Cartesian coordinates) and can be adjusted and referenced throughout the machining process as needed.

It's important to note that the specific methods and procedures for setting and referencing machine zero and work zero can vary depending on the CNC milling machine and the machining software being used. Therefore, it's always recommended to consult the machine's documentation and follow the manufacturer's guidelines for proper setup and programming.

Dry run with the desired tool path movements

Dry Run: Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program. And the machine runs by a constant speed other than the speed specified by the program.

In Auto mode, the methods for turning on the Dry run switch are as follows.

- Press DRY key to make the dry run indicator in State area to light up, it means that the dry run function is selected.
- The speed specified by the program is invalid in Dry run, and actual feed rate is set by the DATA parameter No.174.

By pressing the DRY RUN button during automatic cycle, all of the rapid and work feeds are changed to the rapid traverse feed set in the parameters instead of the programmed feed. Consult the manufacturer manual for specific directions on the use of this function. DRY RUN is also used to check a new program on the machine without any work actually being performed by the

tool. This is particularly useful on programs with long cycle times so the operator can progress through the program more quickly.

To perform a dry run with desired tool path movements on an advanced CNC milling machine, you would typically follow these steps:

1. Prepare the CNC Program: Create or obtain the CNC program that contains the tool paths you want to simulate. This program is usually generated using Computer-Aided Design (CAD) or Computer-Aided Manufacturing (CAM) software.

2. Load the Program: Load the CNC program into the control software of the milling machine. This is typically done by transferring the program file to the machine's control unit through a network connection, USB, or other file transfer methods.

3. Set up the Machine: Ensure that the milling machine is properly set up for the dry run. This includes securing the work piece, mounting the appropriate cutting tools, and verifying that all required machine parameters and settings are correctly configured.

4. Enable Simulation Mode: Many advanced CNC milling machines offer a simulation mode that allows you to visually simulate the tool path movements without actually cutting the material. Enable this simulation mode in the machine's control software.

5. Start the Dry Run: Initiate the dry run by executing the CNC program in simulation mode. The machine will simulate the tool movements based on the programmed instructions while providing a visual representation on the machine's display or a connected computer.

6. Observe the Tool Movements: Monitor the simulated tool movements to ensure they match your expectations. Pay attention to the sequence of operations, tool paths, cutting depths, and any other relevant parameters.

7. Verify Clearance and Safety: During the dry run, observe whether the tool paths maintain sufficient clearance and avoid collisions with the work piece, fixtures, or machine components. Ensure that the machine operates safely without any unexpected issues.

8. Analyze Results and Make Adjustments: After the dry run, evaluate the simulation results. If any issues or deviations are observed, you may need to make adjustments to the tool paths, cutting parameters, or machine settings in the CNC program or CAM software.

By performing a dry run with desired tool path movements, you can verify the accuracy and safety of the CNC milling operation before actually cutting the material. This helps minimize the risk of errors, optimize machining processes, and ensure efficient production.

CNC milling DNC

Performing advanced CNC milling DNC (Direct Numerical Control) involves utilizing a DNC system to control multiple CNC machines from a central computer

DNC: Some CNC system has a DNC function, by the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system.

In Auto mode the method for turning on the Dry run switch are as follows.

- Press the DNC key, it enters the DNC mode.
- Then press the CYCLE START key to start the program DNC machining under the condition that the PC is get ready

Occasionally, it will be necessary to run the program from a remote storage device (Floppy Disc, PCMCI card or a computer hard disk). This is typical for instances when the program is very large and will not fit in the control system memory. Direct Numerical Control (DNC) allows the program to be executed from the offline storage location. When using the computer hard drive method, the offline Personal Computer (PC) is required to have the necessary communications software and RS232 cabling hardware connected or have an Ethernet configuration. Please consult the manufacturer manual for specific instructions. To execute a DNC operation, follow these steps:

1. Call the program number to be executed by one of the search methods.
2. Press the RMT button on the Operator Panel to set REMOTE execution mode.
3. Note that this button is not shown on the controller we depict.
4. Press CYCLE START to begin automatic operation.

By utilizing an advanced CNC milling DNC system, you can effectively manage and control multiple CNC machines, streamline production, and optimize the machining process for improved efficiency and productivity.

Corrective measures/adjustments

In advanced CNC milling machines, there are various corrective measures and adjustments that can be made to ensure accurate and optimal machining results.

If an error is discovered at the control, the preventive check doses not uncover it. An error that is found at the machine slows down production. It forces the machine operator to take corrective measures and eliminate such error. Operator can take one of two actions. One will be to return the program to the programmer; another action will be to correct the error at the machine. Which choice is better depends on the error seriousness.

An error can be soft or hard. A soft error is one that does not require stopping the program from being processed by the CNC system. For example, a missing coolant function M08 in the program can be switched on manually at the machine, without interrupting program processing. That is an example of a soft error - it is still an error, but classified as a minor error.

A hard error occurs when the program processing must be stopped by the operator, as the only available choice, and without doing damage to the machine, cutting tool, part, or all of them. A common example of a hard error is a programmed tool motion that cuts in the wrong direction. Program itself is wrong and must be corrected. This is an example of a hard error, classified as a major error. Most CNC operators do not like delays, especially delays caused by somebody else. A dedicated machine operator will do anything possible to correct a problem without any assistance. For program errors, the operator will try to take corrective measures to eliminate the problem. Not every operator is qualified to do even a simple change to the program.

Such training, if it is designed and delivered in a professional manner, is always a worthwhile investment. It may be a relatively short training that will pay for itself very quickly. Time delays on CNC machines are costly and the sooner the program is made functional, the less damage to the production control has been done.

Whenever a program has been changed at the machine, the program documentation must reflect these changes, particularly if they are permanent. Even a small permanent change should be always be documented in all copies of program documentation

Common corrective measures and adjustments are**:-**

- Calibration:
- Tool Length Compensation:
- Federate Adjustments:
- Spindle Speed Adjustments:
- Tool Selection:
- Work holding Optimization:
- Surface Speed Compensation:
- Tool Path Optimization:
- Spindle Run out Compensation:
- Machine Maintenance:

It's important to note that specific adjustments and corrective measures may vary depending on the CNC milling machine model, software, and the machining requirements. Referring to the machine's user manual, consulting with machine operators, or seeking assistance from technical support can provide additional guidance for making appropriate adjustments and corrections**.**

3.4.1. Replacement of cutting tools

When replacing cutting tools in a CNC milling machine, there are several corrective measures that can be taken to ensure optimal performance. By following corrective measures during the

replacement of cutting tools, you can minimize errors, optimize machining results, and maintain the accuracy and quality of the CNC milling process.

3.4.2. Adjustment of tool offset

On the other hand, some qualified operators may not be authorized to do program changes as a matter of policy. Every company benefits greatly, if the CNC operator has at least a basic training in CNC programming. The purpose of such training is not to make the machine operator a fully qualified CNC programmer. Its purpose is to highlight how a part program influences CNC machining, the setup, tooling and all the other relationships between programming and machining. Its purpose is to offer the operator tools that can be used for minor program changes, etc.

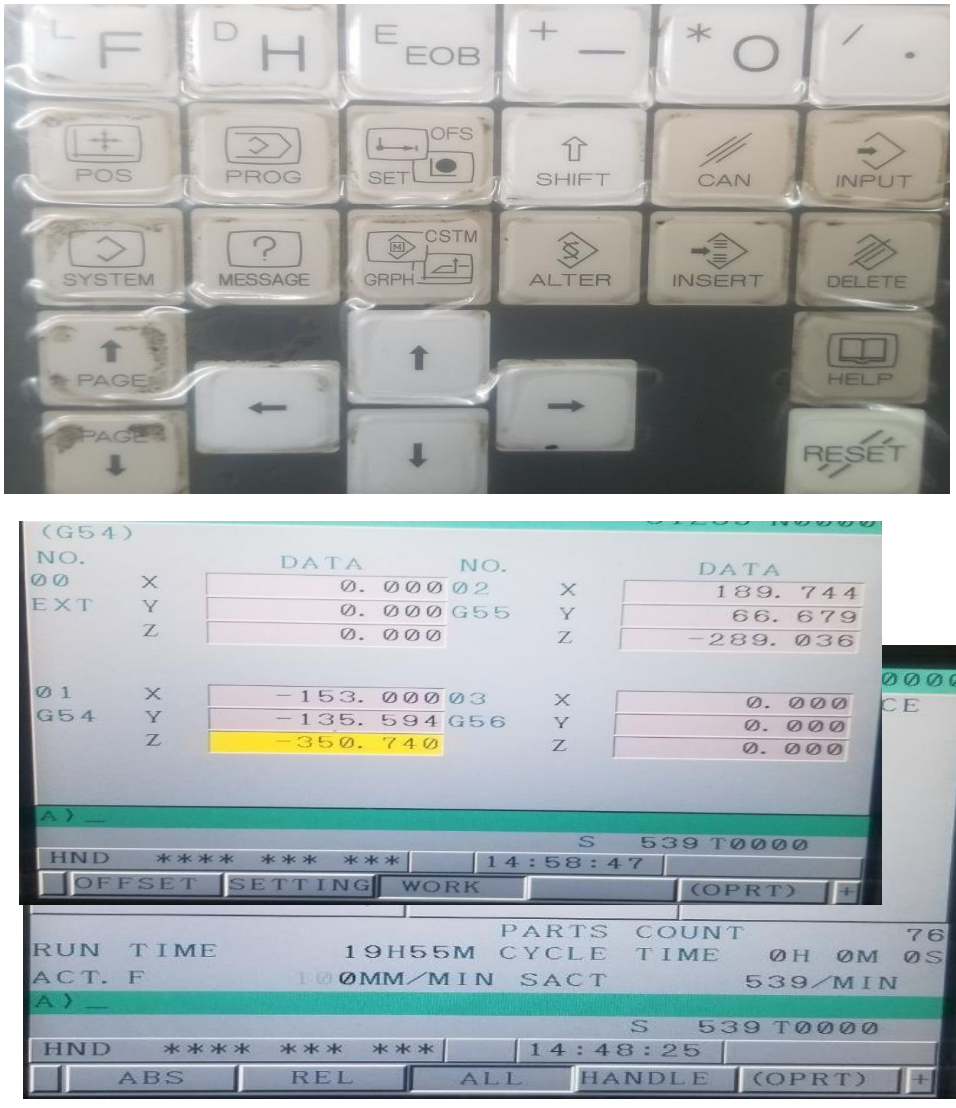


Fig 3.3.1.Tool offset setup

Keep in mind that the specific steps and procedures for adjusting cutting speed and feed rate may vary depending on the CNC milling machine and control system being used. It's important to consult the machine's documentation, programming manual, or seek guidance from experienced operators or technicians for precise instructions related to your specific machine and application.

Self-Check 3

Part I

Direction: Choose the best answer for the following questions. Use the Answer sheet provided in the next page: Each question worth one point

1. A fixed position on a CNC machine that can be reached repeatedly, on request
   A. Work Zero
   B. Machine Zero
   C. Work coordinate system
   D. work offset
2. Z-axis machine zero position for a vertical machining center is
   A. where the Automatic Tool Change (ATC) takes place
   B. where work piece is set
   C. At the extreme travel limit of each axis in the positive direction.
   D. A and C.
3. Which one is attached to machine control unit (MCU?)
   A. Control panel B. Automatic tool exchange C. Edit mode screen D. None
4. The Cartesian coordinate system that identifies the location of the origin on the work piece is called
   A. Machine Zero
   B. Work coordinate system
   C. Machine coordinate system
   D. Work offset
5. One is not true regarding work offset

A. A milling work offset is usually established for the X-, Y-, and Z-axes

B. Established by finding the offset or “shift” from the MCS origin to the intended WCS origin

C. Some controls call it a work shift

D. None of the above

Part II

Direction: Say true if the statement is true and false if the statement is false for the following questions. Use the Answer sheet provided in the next page: Each question worth one point

1. Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program

2. By pressing the DRY RUN button during automatic cycle, all of the rapid and work feeds are unchanged

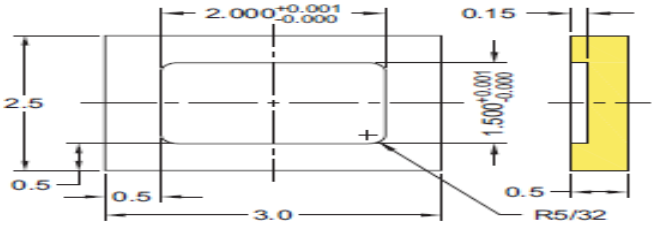
3. In auto mode pressing the DNC key, enters the DNC mode.

4. By the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system

5. For program errors, the operator will try to take corrective measures to eliminate the problem

Operation Sheet: -3

Instruction:- Program the given project according to the given design



Operation Title: - Programing Rectangular pocket

Purpose: - project work exercising

Conditions or situations for the operations: -

- ✓ Safe working area
- ✓ Properly operated tools and equipment
- ✓ Effectively use of design soft ware
- ✓ Good understanding of programming

Precautions: - Safe working, free from accidents

Procedure:-

- O3303(RECTANGULAR POCKET)
- N1 G20
- N2 G17 G40 G80 T01 (0.250 ROUGHING SLOT DRILL)
- N3 M06
- N4 G90 G54 G00 X0.66 Y0.66 S1250 M03 T02
- N5 G43 Z0.1 H01 M08
- N6 G01 Z-0.15 F7.0
- N7 G91 X1.68 F10.0 (CUT 1)
- N8 Y0.236 (STEPOVER 1)
- N9 X-1.68 F12.0 (CUT 2)
- N10 Y0.236 (STEPOVER 2)
- N11 X1.68 (CUT 3)
- N12 Y0.236 (STEPOVER 3)
- N13 X-1.68 (CUT 4)
- N14 Y0.236 (STEPOVER 4)
- N15 X1.68 (CUT 5)
- N16 Y0.236 (STEPOVER 5)
- N17 X-1.68 (CUT 6)
- N18 G40 G00 Z0.1 M09
- N19 G28 Z0.1 M05
- N20 X-2.0 Y10.0
- N21 M30

Equipment Tools and Materials: -

- ✓ Computer/pc
- ✓ Drawing instruments
- ✓ Designee soft wares/ solid work, Catia, AutoCAD,

Quality Criteria: - Accurate, Precise & Exact according to the given design

Lap Test 3.

Directions:

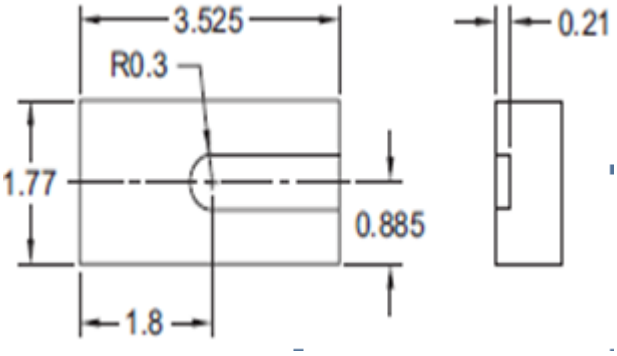
Answer all the questions listed below.

Use the Answer sheet provided in the next page:

Date: _____Time started: _____ Time finished: _____

Instructions: According to the given project design & materials you are required to perform the following tasks within --- hour

Task1. Perform a G-code program for the part shown below



Unit four: - Select materials

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Introduction to material selection
- Manufacturing process plan.
- Materials test

Upload the Program This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Select and obtain appropriate materials
- Develop comprehensive plan to sequence and to perform manufacturing process requirements
- Taste Selected materials for hardness
- Develop plan to sequence and stage manufacturing process

Introduction to material selection

Material selection is a critical aspect of the design and manufacturing process. Choosing the right materials for a particular application is crucial to ensure the desired functionality, performance, and durability of a product. Material selection involves evaluating and comparing various properties and characteristics of different materials to determine which one is most suitable for a specific application

Material selection is a multidisciplinary process that requires collaboration between designers, engineers, material scientists, and other stakeholders. It often involves conducting material testing, consulting material databases, and leveraging expertise to make informed decisions. Computer-aided tools and software can also assist in material selection by providing databases, material property charts, and simulations to compare and analyze different materials.

Ultimately, the goal of material selection is to choose a material that meets the functional requirements, performance criteria, and constraints of the specific application, leading to a successful and optimized product design.

The Material Selection Problem Design of an engineering component involves three interrelated problems: (i) selecting a material, (ii) specifying a shape, and (iii) choosing a manufacturing process. Getting this selection right the first time by selecting the optimal combination your design has enormous benefits to any engineering-based business. It leads to lower product costs, faster time-to-market, a reduction in the number of in-service failures and, sometimes, significant advantages relative to your competition.

Manufacturing process plan

Process planning in CNC machining is the process of transforming an engineering design into a work instruction to manufacture a physical product. It involves selecting the best sequence of operations, machines, tools, cutting parameters, and other factors to optimize the efficiency and quality of the machining process. Process planning can be done manually or with the help of computer-aided tools such as CAD/CAM, CAPP, and PLM[1]. Process planning is an important function that influences many of the company objectives, such as cost, time, quality, and environmental impact[2]

Creating a manufacturing process plan for CNC milling involves several steps to ensure efficient and accurate production. Here's a general outline of the process:

1. Design and CAD Modeling: Begin by designing the part or component you want to manufacture using computer-aided design (CAD) software. Create a 3D model of the part, including all necessary dimensions and features.
2. Material Selection: Determine the appropriate material for the part based on its intended use and requirements. Consider factors such as strength, durability, and cost.
3. Tool Selection: Choose the appropriate milling tools based on the material and part design. Consider factors such as tool diameter, flute type, coating, and cutting parameters.
4. Fixture Design: Design a fixture or work holding system that securely holds the work piece during the milling process. The fixture should provide stability and ensure accurate positioning and alignment.
5. Process Planning: Analyze the part design and identify the milling operations required to create the desired features. Consider the sequence of operations, tool changes, and any special considerations like roughing and finishing passes.

6. Tool path Generation: Use CAM (Computer-Aided Manufacturing) software to generate the tool paths based on the part geometry and milling operations. The tool paths define the precise movement of the milling machine to remove material and create the desired features.
7. Simulation and Verification: Simulate the milling process using CAM software to check for any errors, collisions, or issues that may arise during machining. Verify that the tool paths are correct and will produce the desired results.
8. Machine Setup: Prepare the CNC milling machine for production. Install the appropriate cutting tools, load the work piece into the fixture, and secure it properly. Set up the machine parameters, including spindle speed, feed rate, and cutting depths.
9. Machining Operations: Run the CNC milling machine according to the planned tool paths. Monitor the process to ensure the machine is operating correctly and the part is being machined accurately. Make any necessary adjustments as needed.
10. Inspection and Quality Control: After the machining is complete, inspect the finished part to ensure it meets the required specifications and tolerances. Use measuring tools such as calipers, micrometers, or coordinate measuring machines (CMM) to verify dimensions and features.
11. Finishing Operations: If required, perform additional finishing operations such as de burring, polishing, or surface treatments to achieve the desired final appearance and functionality.

Documentation and Record Keeping: Document the manufacturing process plan, including tooling details, machine settings, inspection results, and any modifications or adjustments made during production. This information will be valuable for future reference or process improvement.

Remember that the specifics of the manufacturing process plan will depend on the part design, material, and machining capabilities of your CNC milling machine. It's essential to consider these factors and adapt the plan accordingly for each unique project

Materials test

A material testing is often the last step in the manufacturing process. Yet quality cannot be put in after the fact since it is the result of both the materials and the process. In forming materials, it's helpful to understand their properties in order to better predict the manufacturing outcome. For stamping and forming operations and for many products it is also helpful to have a profile of the material in order to detect variations in materials from suppliers over time.

A material testing procedure is a set of steps followed to measure the properties of a material. The specific steps in a procedure may vary depending on the type of material being tested and the properties being measured. However, there are some general steps that are common to most material testing procedures:

1. Prepare the sample. This may involve cutting the sample to a specific size and shape, cleaning the sample, and drying the sample.
2. Condition the sample. This may involve exposing the sample to a specific temperature, humidity, or other environmental conditions for a period of time.
3. Load the sample into the testing apparatus. The specific apparatus used will depend on the type of test being performed.
4. Apply the load. The load may be applied in different ways, such as pulling on the sample, pushing on the sample, or bending the sample.
5. Measure the response of the sample. The response of the sample may be measured in different ways, such as the amount of force required to deform the sample, the amount of deformation that occurs, or the time it takes for the sample to fail.
6. Record the data. The data collected from the test is recorded and analyzed to determine the properties of the material.

Some common types of material testing procedures include:

- Mechanical testing: Mechanical testing procedures measure the strength, stiffness, and toughness of materials. Common mechanical tests include tensile tests, compression tests, flexural tests, and shear tests.

- Thermal testing: Thermal testing procedures measure the properties of materials related to heat, such as thermal conductivity, specific heat, and melting point.
- Electrical testing: Electrical testing procedures measure the properties of materials related to electricity, such as resistivity, conductivity, and dielectric constant.
- Chemical testing: Chemical testing procedures measure the composition of materials and their susceptibility to chemical attack.
- Nondestructive testing: Nondestructive testing procedures measure the properties of materials without damaging them. Common nondestructive tests include ultrasonic testing, X-ray testing, and magnetic particle testing.

Material testing procedures are used in a wide variety of industries, including construction, manufacturing, and aerospace. By testing materials, engineers can ensure that they are using the right materials for the job and that the materials will perform as expected.

Brinell hardness testing procedure with picture:



Step 1: Prepare the sample.

- ✓ Make sure the surface of the sample is clean and smooth.
- ✓ If the sample is too small or thin, it may need to be embedded in a harder material to prevent it from moving during the test.

Step 2: Select the appropriate indenter and load.

- ✓ The Brinell hardness test uses a hardened steel ball as the indenter. The size of the ball and the load used will depend on the material being tested.
- ✓ For softer materials, a smaller ball and lower load will be used to avoid excessive indentation. For harder materials, a larger ball and higher load will be used to obtain a measurable indentation.

Step 3: Position the sample and indenter.

- ✓ Place the sample on the testing platform and position the indenter directly over the surface of the sample.
- ✓ Make sure the indenter is perpendicular to the surface of the sample.

Step 4: Apply the load and dwell.

- ✓ Apply the load to the indenter and hold it for a specified period of time, typically 10 to 15 seconds.

Step 5: Remove the load and measure the indentation.

- ✓ Remove the load from the indenter and measure the diameter of the indentation.

Step 6: Calculate the Brinell hardness number (BHN).

- ✓ The BHN is calculated using the following formula:

$BHN = 2P / \pi D^2$

Where:- P is the load in kilograms-force (kgf)

D is the diameter of the indentation in millimeters (mm

Self-check 4.1.

Directions: Answer all the questions listed below & circle the answer on the later

1. One of the following is not work holding device

   A. Vise B. Clamps C. chucks D. CNC End mill holders

2. Machine vises are the most commonly used work holding devices for milling because

   A. They are highly versatile, accurate, and simple to use

   B. Can holding extremely large or oddly shaped work pieces

   C. Has movable jaws

   D. A and C.

3. Machining center requires _____to adapt the machine's spindle to the cutting to

   A. Tool holder B. Work holding device C. Work piece D. Cutting tool

4. Which one is advantage of end mill holders?

   A. They are inexpensive, rugged, and simple

   B. Different sized holder is necessary for each shank diameter

   C. Balance can be affected by the setscrew

   D. All

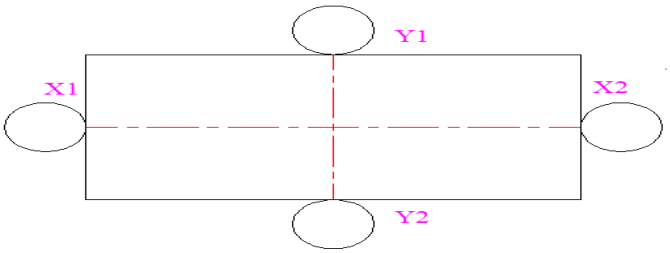
5. _____are very versatile and are often used for holding many different types of straight-shank milling tools

   A. Shrink-Fit Tool holders B. end mill holders C. CNC collet chucks

Operation Sheet: -4.1

Instructions:-According to the program produce the given project

Operation Title: - Tool off set setup



Purpose: - Setting tool off set

Conditions or situations for the operations: -

- ✓ Safe working area
- ✓ Properly operated tools and equipment
- ✓ Effectively use of design soft ware
- ✓ Good understanding of programming

Precautions: - Safe working, free from accidents

Procedures:-

Steps 1 (*X, Y and Z coordinate zero at the center of the work piece*)

1. Press **"*Position*"** button
2. Press ***All***
3. Touch the tool slightly to the workpiece both the left and right direction
4. X = (X1 + X2)/2or

   X = (123.657 + 84.234)/2……..example

   X = - <u>103.945</u>
5. Press "offset" button
6. Press "*Work"*
7. Under "G54" Write the average X value in the place of X coordinate and press "input"

8. -103.94 will be recorded in the X coordinate
9. Press position and do the same steps for Y coordinate
10. For Z coordinate touch the work piece slightly on the top
11. Press offset
12. Press *Work*
13. Under "G54" Write "Z0 (zero)" in the place of Z coordinate and press "measure"
14. Now work coordinate will be 0 zero in all direction

Steps 2 (X, Y and Z coordinate zero at the first quadrant)

A. For X coordinate

1. Touch the tool slightly to the work piece from the left direction of X
2. Press "Off set" button
3. Press "*Work"*
4. Under "G54" Write "X – the radius of the cutter" in the place of X coordinate and press "Measure"

B. For Y coordinate

1. Touch the tool slightly to the work piece from the front direction of Y
2. Press "Offset" button
3. Press "*Work"*
4. Under "G54" Write "Y – the radius of the cutter" in the place of Y coordinate and press "Measure"

C. For Z coordinate

1. For Z coordinate touch the work piece slightly on the top
2. Press offset
3. Press *Work*
4. Under "G54" Write "Z0 (zero)" in the place of Z coordinate and press "measure"
5. Now work coordinate will be 0 zero in all direction

Equipment Tools and Materials: -

- ✓ Computer/pc
- ✓ Drawing instruments

- ✓ Designee soft wares/ solid work, Catia, AutoCAD,
- ✓ CNC milling machine

Quality Criteria: - Accurate, Precise & Exact, According to the given design

Lap Test 4.

Directions: Answer all the questions listed below.

Use the Answer sheet provided in the next page:

Date: _____Time started: _____ Time finished: _____

Instructions: Given necessary information to Cutter composition Setup perform the following tasks within --- hour

Task1. Make sure ***"D"*** value is the same as the cutter number

⬋ If the cutter number is ***04*** the D value should be *D04 on the program of G42*

Task2. Press ***offset*** button

Task3. Press *Offset* option

Task4. Select the correct cuter number

Task5. Then write the radius of the cutter

Unit five: -. Produce and assemble components
This unit is developed to provide you the necessary information regarding the following content coverage and topics:
• CNC milling Machining process. • Test run. Upload the Program This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to: • Select appropriate machines and machining process . • Produce where practical, prototype or section for testing • Observe Occupational health and safety procedures and environmental protection guidelines

CNC milling Machining process

Milling is typically used to produce parts that are not axially symmetric and have many features, such as holes, slots, pockets, an even three dimensional surface con- tours. Parts that are fabricated completely through milling often include components that are used in limited quantities, perhaps for prototypes, such as custom designed fasteners or brackets. Another application of milling is the fabrication of tooling for other processes. For example, three-dimensional molds are typically milled. Milling is also commonly used as a secondary process to add or refine features on parts that were manufactured using a different process. Due to the high tolerances and surface finishes that milling can offer, it is ideal for adding precision features to a part whose basic shape has already been formed.

PROCESS CYCLES

The time required to produce a given quantity of parts includes the initial setup time and the cycle time for each part. The setup time is composed of the time to setup the milling machine, plan the tool movements (whether performed manually or by machine), and install the fixture device into the milling machine. The cycle time can be divided into the following four times:

- Load/Unload time - The time required to load the work piece into the milling machine and secure it to the fixture,
- Cut time - The time required for the cutter to make all the necessary cuts in the work piece for each operation. The cut time for any given operation is calculated by dividing the total cut length for that operation by the feed rate, which is the speed of the cutter relative to the workpiece.as well as the time to unload the finished part. The load time can depend on the size, weight, and complexity of the work piece, as well as the type of fixture.
- Idle time - Also referred to as non-productive time, this is the time required for any tasks that occur during the process cycle that do not engage the work piece and therefore remove material. This idle time includes the tool approaching and retracting from the work piece, tool movements between features, adjusting ma- chine settings, and changing tools.
- Tool replacement time - The time required to replace a tool that has exceeded its lifetime and therefore become to worn to cut effectively. This time is typically not performed in every cycle, but rather only after the lifetime of the tool has been reached. In determining the cycle time, the tool replacement time is adjusted for the production of a single part by multiplying by the frequency of a tool replacement, which is the cut time divided by the tool lifetime.

Following the milling process cycle, there is no post processing that is required. How- ever, secondary processes may be used to improve the surface finish of the part if it is required. The scrap material, in the form of small material chips cut from the work piece, is propelled away from the work piece by the motion of the cutter and the spraying of lubricant. Therefore, no process cycle step is required to remove the scrap material, which can be collected and discarded after the production.

Test run.

If you want to test run your CNC milling machine, you will need to prepare a G code program that instructs the machine how to move and cut the work piece. You can either write your own program using CAD/CAM software, or download some free G code files from the internet. Here are some sources that offer G code examples for CNC milling:

DIY Machining has a simple G code example that will draw a 1" diameter circle. It also provides a step by step guide on how to load and run the program on your machine.

CNC Philosophy has three G code files that will engrave a spider, a skull, and a dragon on your work piece. It also has a 7-step beginner's guide to CNC milling that explains the workflow and processes involved.

Helman CNC has several CNC quizzes and tests that challenge your CNC programming skills. You can try to complete the programs and run them on your machine to see the results.

Before you run any G code program, make sure you set your zero position correctly and adjust the spindle speed, feed rate, and depth of cut according to your machine and material specifications. You can also use a marker or a pen instead of a cutting tool to test the program without damaging your work piece. Always be careful and follow the safety precautions when operating your CNC milling machine

Self- check **5**.1

Direction: Choose the best answer for the following questions. Use the Answer sheet provided in the next page: Each question worth one point

1. Milling is typically used to produce parts that are not axially symmetric and have many features, such as

   A. Holes, B. slots, C. pockets D. All

2. The time required to load the work piece into the milling machine and secure it to the fixture

   is called

   A. Tool replacement time B. Idle time

   C. Cut time D. Load/Unload time

3. Idle time includes the tool approaching and retracting from the work piece, tool movements between

   A. features, B. adjusting ma- chine settings, C. Changing tools. D. all

4. I n Edit mode, the programs can be

   A. created, selected, modified, copied and delete
   B. Bidirectional communication between CNC and CNC, or CNC and PC can also be achieved.
   C. A and B
   D. None

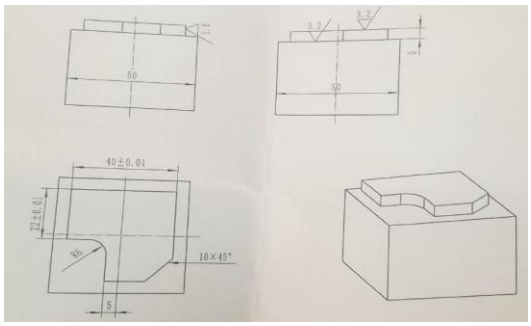
5. To set up the new program
   A. Press program key B. Press Edit key
   C. Press EOB key D. Press address key

6. By pressing address key, symbol key numerical key . . . by sequence
   A. all program are deleted B. Single program is deleted C. Both are deleted

Operation Sheet 5.1.

:- Instruction:- Program the given project according to the given design

Operation Title: Setting program



Purpose: To set proper CNC milling program

Conditions or situations for the operations:

- ✓ Safe working area
- ✓ Properly operated tools and equipment
- ✓ Well-equipped machine work shop
- ✓ Appropriate working cloths fit with the body

Equipment Tools and Materials:

- ✓ CNC milling machine
- ✓ PC/ computer
- ✓ Drawing instruments
- ✓ Designee soft wares/ solid work, Catia, AutoCAD

Procedure:- Exercise 1 (*X, Y and Z coordinate is 0 (zero) ate the first quadrant***)**

O0034	Y45.
G90 G17 G80 G49 G40 G55	X45.
M06T02	Y15.
M03 S1000	X35.Y5.X20.
G00 Z10.	Y17.
G00 X0Y0	G03X14.Y23.R6.G01X-10
G00 X5.Y-20.	G00Z100
G01 G41X5. Y-20. D02 F100	M05

Quality Criteria: Assured performing of all the activities according to the procedures

Precautions:

- ✓ Wearing proper clothes, eye glass, glove
- ✓ Make working area hazard free
- ✓ Read and interpret manual which guide you how to use tools and equipment

Lap Test 5.

Directions: Answer all the questions listed below.

Use the Answer sheet provided in the next page:

Date: _____Time started: _____ Time finished: _____

Instructions: Given necessary information about CNC Machine operation perform the following tasks within --- hour

Task1. Dry-runs the machine

Task2. Test-run the machine

Unit six: - Quality assurance

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Checking and measuring work piece.
- Methods, measuring tools and equipment's
- Defective works.
- Product quality of CNC production.

Upload the Program This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Check and measure work piece
- Utilize appropriate methods, measuring tools and equipment
- Mark, record and report defective work pieces
- Compare product quality of the CNC production with conventional production

Checking and measuring work piece**.**

To check and measure a work piece in milling machine operation, you can follow these steps:

Select the appropriate measuring tools: Depending on the dimensions and characteristics of the work piece, you may need different measuring tools such as calipers, micrometers, depth gauges, height gauges, or dial indicators.

Prepare the work piece: Ensure that the work piece is securely clamped in the milling machine vise or fixture and properly aligned with the milling cutter.

Initial visual inspection: Visually inspect the work piece to ensure there are no visible defects, burrs, or irregularities that might affect the measurements.

Measure length and width: Use calipers or micrometers to measure the overall length and width of the work piece. Take multiple measurements at different points to ensure accuracy, and record the readings.

Measure depth or height: Depending on the specific milling operation, you may need to measure the depth of a groove or hole, or the height of a raised feature. Use depth gauges, height gauges, or dial indicators to make these measurements.

Check tolerances: Compare the measurements with the required tolerances specified in the engineering drawings or work instructions. Ensure that the work piece dimensions fall within the acceptable range.

Verify perpendicularity or parallelism: If the milling operation requires specific perpendicular or parallel surfaces, use a square or surface plate to check for these characteristics. Ensure that the work piece meets the required specifications.

Surface finish inspection: If surface finish is important, visually inspect the milled surfaces for roughness, burrs, or any other imperfections. Use appropriate measuring tools such as surface roughness testers if necessary.

Document the measurements: Record all the measurements taken, including dimensions, tolerances, and any other relevant information. This documentation is essential for quality control purposes and for future reference.

Remember, different milling operations may have specific measurement requirements, so it's important to consult the engineering drawings, work instructions, or any applicable standards to ensure you are measuring the work piece accurately and according to the specific requirements of the job**.**

Methods, measuring tools and equipment's

Principle and use Vernier caliper

To measure smaller distances can measure up to 0.02mm.

- **Vernier caliper**
  - ✓ **Main Scale**
    - Main scale is graduated in cm and mm.
  - ✓ **Vernier Scale**
    - It slides on the main scale.
    - On Vernier scale 0.9cm is divided into 10 equal parts.
  - ✓ **Jaws**
    - Two inside jaws (Upper)
    - Two outside jaws (Lower)

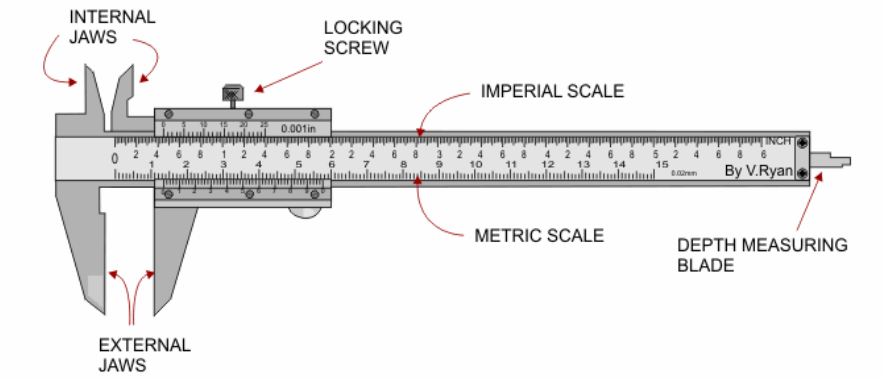


Figure 6.2.1 parts of Vernier caliper

- **Dial Caliper:** An outgrowth from the vernier caliper is the **dial caliper**. However, this instrument does not employ the principle of the vernier. The beam scale on the dial caliper is graduated into only .10-in. increments. The caliper dial is graduated into either 100 or 200 divisions. The dial hand is operated by a pinion gear that engages a rack on the caliper beam. On the 100-division dial, the hand makes one complete revolution for each .10-in. movement of the sliding jaw along the beam. Therefore, each dial graduation represents 1100 of .10 in., or .001 in. maximum discrimination. On the 200-division dial the hand makes only half a revolution for each .10 in. of movement along the beam. Discrimination is also .001 in.

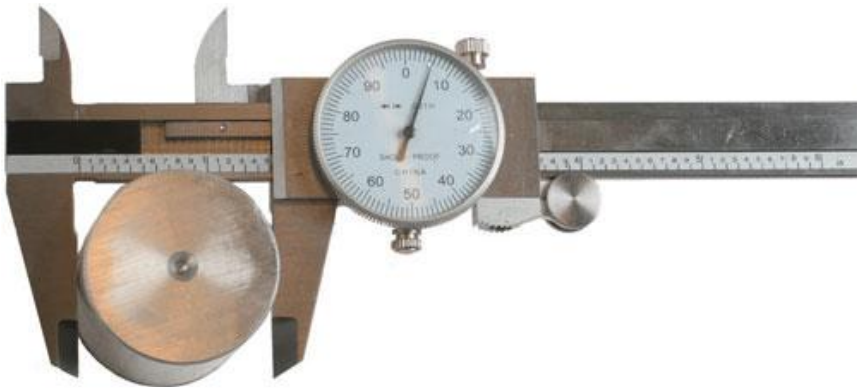


Figure 6.2.2 Dial caliper.

Steps to know the diameter external and internal work piece

Steps 1- Apply safety precaution.
Steps 2- Move the sliding jaw to be in contact with the faces being measured.

Steps 3- Tighten the locking screw on the clamp.

Steps 4- Make fine adjustment using the fine-setting screw.

Steps 5- Move the jaws so that they just touch the work; do not apply any force.

Steps 6-Tighten the head lock

For example, to know the diameter of a piece of round section steel

- The main metric scale is read first and this shows that there are 13 whole divisions before the 0 on the Vernier (hundredths) scale. Therefore, the first number is 13.
- The Vernier (' hundredths of mm') scale is then read. Only one division on the main metric scale lines up with a division on the Vernier (hundredths) scale below it, whilst others do not. In the example below, 21 divisions on the Vernier scale lines up exactly with a division on the main metric scale above.
- Therefore 21 is multiplied by 0.02 giving 0.42 as the answer (each division on the Vernier scale is equivalent to 0.02mm).
- 13 and 0.42 are added together to give the final measurement of 13.42mm (the diameter of the piece of round section steel).

.

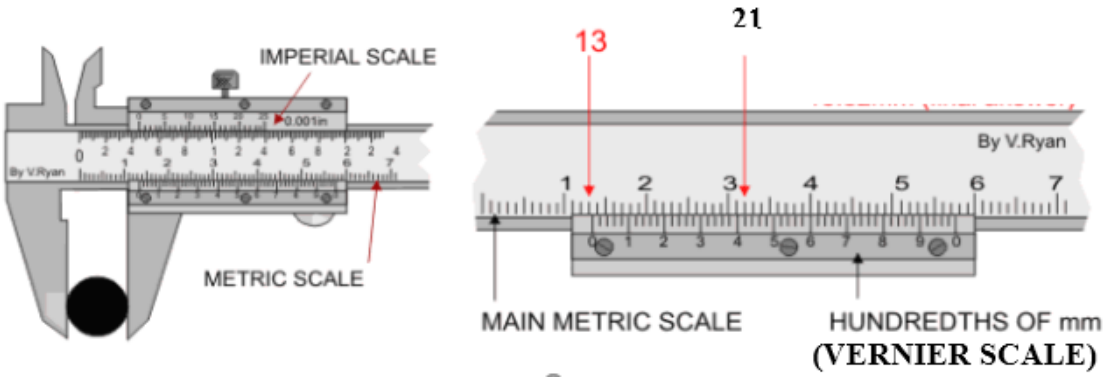


Figure6. 2.3 reading the Vernier scale

External (outside) micrometer

A micro meter is a precision instrument used to measure a job, generally within an accuracy of 0.01mm. Micrometers used to take the outside measurements are known as outside micrometers**.**

- **Parts of outside micrometer:**

✓ **Frame:** The frame is made of drop forge steel or malleable cast iron. All other part of the micrometer is attached to this.

✓ **Barrel/sleeve**: The barrel or sleeve is fix to the frame. The datum line and graduation are marked on this.

✓ **Spindle**: One end of the spindle is the measuring face. The other end is threaded and passes through a nut. The threaded mechanism allows for the forward and backward movement of the spindle.

✓ **Spindle lock nut**: The spindle lock nut is used to lock the spindle at a desired position.

- ✓ **Ratchet stop**: The ratchet stop ensures a uniform pressure betw-een the measuring surface.
- ✓ **Thimble**: On the beveled surface of the thimble also, graduation is marked. The spindle is attached to this.
- ✓ **Anvil**: The anvil is one of the measuring faces which is fitted on the micrometer frame. It is made of alloy steel and finished to a perfectly flat surface

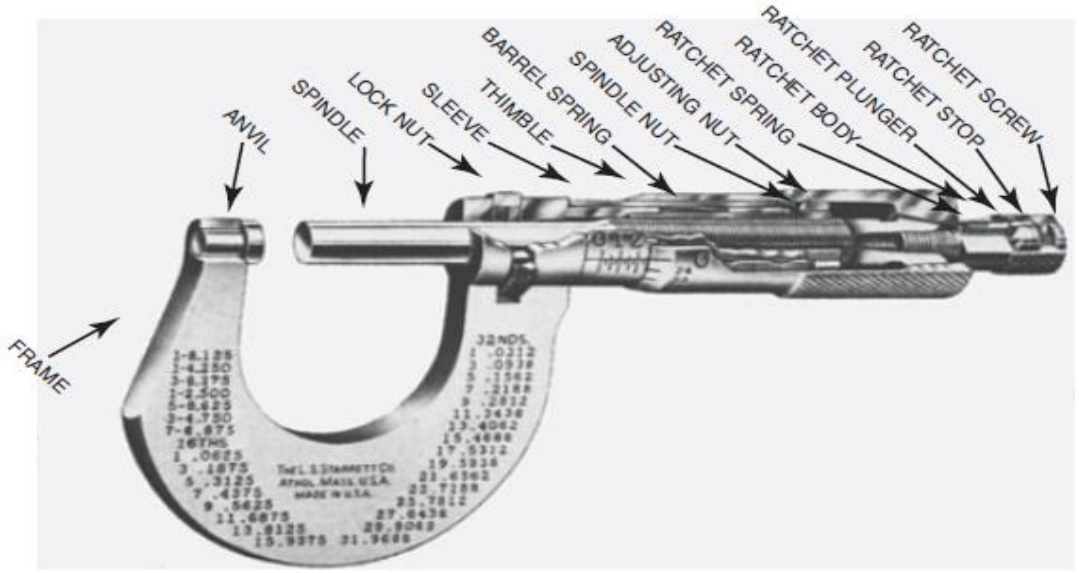


Figure 6.2.4 parts of micrometer

Procedure of micrometer reading

Steps 1- Apply safety precaution.

Steps 2- Hold the plastic insert to prevent thermal expansion.

Steps 3- Keep the measuring faces square with the surfaces that you are measuring, to ensure an accurate measurement.

Steps 4- Turn the thimble until the faces touch the work.

Steps 5- Use the ratchet (if there is one) to obtain the correct pressure when turning the thimble, and prevent the jaw from moving further when it comes into contact with the work.

Reading of out-side micrometer:

-Read the whole mm. graduation on the barrel above datum line

-Read half mm. graduation on the barrel below datum line

-Read the value of the visible line on the left of the thimble edge.

-See below fig.

Main reading = 13.0mm

Half mm. reading= 0.5mm

Thimble reading=13x0.01=0.13mm

Total reading= 13.0+ 0.5 + 0.13

= 13.63mm

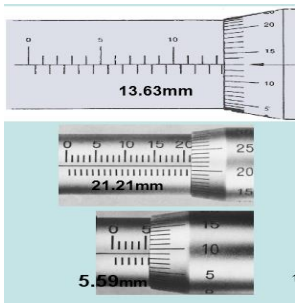


Figure2.5. reading of outside micrometer

Dial indicator and gages

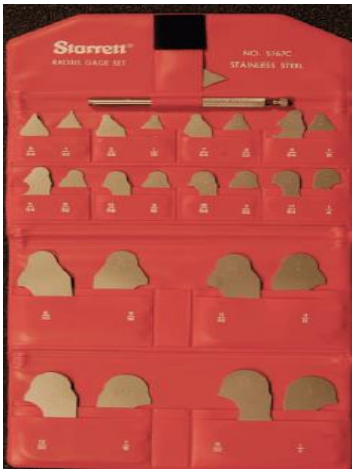
Dial indicator: Dial indicators are used to center and align work on machine tools, check for eccentricity, and inspect manufactured parts. They are designed with shockproof movements and have jeweled bearings (similar to fine watches). Dial indicators must be mounted to rigid holding devices.



Figure 6.2.6 dial indicator

Dial Indicators discrimination of dial indicators typically ranges from .00005 to .001 in. In metric dial indicators, the discrimination typically ranges from .002 to .01 mm. Indicator ranges, or the total reading capacity of the instrument, may commonly range from .003 to 2.000 in., or .2 to 50 mm for metric instruments.

Radius Gauges: Radius and Fillet Gages Radius and fillet gages are used to check outside corner and insider corner (fillet) radii. A set is shown in Figure A. Figure B shows how radius and fillet gages can be used.



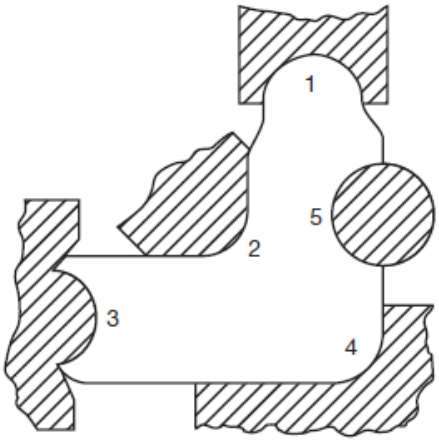


Figure6. 2.7. A. A radius and fillet gage set B. Uses of radius and fillet gages

Vernier Depth Gage: A vernier depth gage uses a sliding rod similar to a vernier caliper with a depth rod. Also using 0.001" or 0.02-mm graduations, it usually has a measuring range of 0–6" (0–150 mm) or 0–12" (0–300 mm).



Figure6.2.8. Vernier depth gage

Screw Pitch Gage: A screw pitch gage determines the distance between threads. Each leaf is used for a different size. They are available in inch and metric versions. shows a typical screw pitch gage and how it is used.



Figure6.2.9. (A) An example of a screw pitch gage. (B) Checking threads with a screw pitch gage

Defective works

A defective work record form is a document that records the details of any work that does not meet the required quality standards or specifications[1]. It is used to identify the problem, the cause, the rectification required, and the responsible party. A defective work record form helps to track and monitor the progress of fixing the defects, and to communicate with the relevant stakeholders. A defective work record form can also be used to avoid disputes or claims arising from defective work. You can find some examples of defective work record forms in the links below

If we have defective work pieces, it is important to mark, record and report them. You can use a **defective work record form** to record the details of the defective work[1]. You should describe the work in sequence of work element/location; condition as rejected; rectification required[1]. You should also check and update this record regularly with clerk of works and design team[1]. If you have concerns about a product due to its design, construction or the manner of supply, you should report the matter to the appropriate market surveillance authority (MSA)[2]. If you notice a problem or damage during your regular job safety analysis or as a result of an unexpected event, don't think twice about reporting it. Don't let any detail go unnoticed, and submit an equipment damage report

Product quality of CNC production

Once a work piece has been machined, its dimensions need to be inspected for quality control. The techniques for measurement of machined work piece dimensions may be classified into three modes:

1. **In-process measurement**. The measurement takes place during the actual machining process without any interruption to the process itself. The information produced by measurement is provided continuously. This mode is used for work piece dimension, profile, and surface roughness measurement. In-process measurement offers practical solutions for real-time measurement and quality control. It is well known in grinding

applications of cylindrical parts. It is difficult to use in turning operations where there may be massive amounts of high speed swarf and coolants disturbance

2. **In-cycle measurement**. Most machining and turning centers use the in-cycle measuring technique, which is also referred to as on machine measurement. Unlike in-process measurement, the process must be stopped while the measurement takes place during the process cycle. With this measurement method, the work piece can be assessed either between cuts or after completion, prior to removal from the set-up. It converts the machine tool into a coordinate measuring machine and adds a machining step to correct the work piece while on machine.

3. **Post-process measurement.** The machined work piece is removed from the process and transported by a suitable device to an independent inspection machine such as CMM where the work piece is inspected The major advantage of this method is that post-process measurement includes the effects of all error sources that affect the work piece in a single setup. This is usually time-consuming and results in the risk of producing a large number of defective work pieces before the inspection results are known.

Accuracy

It is a measure of control system's capacity to position the tool or work piece to a specified set of coordinates.

- ✓ Control resolution: Basic length unit (BLU)
- ✓ Hardware inaccuracies
- ✓ inaccuracies in the machine elements
- ✓ machine-tool assembly errors
- ✓ spindle run-out
- ✓ Lead-screw backlash
- ✓ tool deflection
- ✓ thermal error

Post-process measurement instrument may includes

CNC form measuring instruments

CNC surface texture measuring instruments

Self-check 6.1.

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. Which One is Common corrective measures and adjustment measure

   A. Calibration: B. Tool Length Compensation

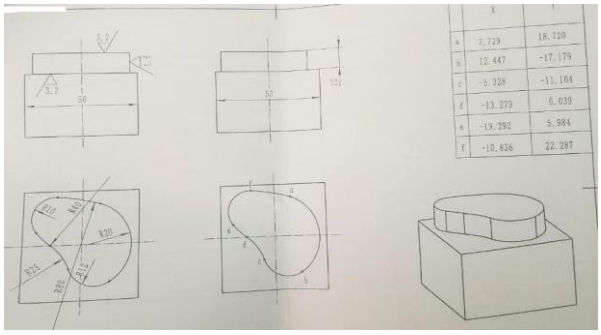
   C. Federate Adjustments: D. All

2. _____ is a measuring instrument that used to measure the diameter of work pieces
3. Which of the following measuring instrument is precise?
   a) Caliper
   b) Vernier caliper
   c) Micrometer
   d) Steel Rule
4. Which of the following CNC Milling axis represents the tool height?
   a) Z-axis
   b) X-axis
   c) Y- axis
   d) All
5. An outside micrometer would be appropriate for measuring which of the following (two correct answers):
   a) Hole depth,
   b) Hole diameter,
   c) Part length,
   d) Shaft diameter,

Operation Sheet: -6.1.

Instruction:-According to the program produce the given project

Operation Title: - (*X, Y and Z coordinate is 0 (zero) ate the work piece center*



Purpose: - programing CNC milling machine

Conditions or situations for the operations: -

- ✓ Safe working area
- ✓ Properly operated tools and equipment
- ✓ Effectively use of design soft ware
- ✓ Good understanding of programming

Procedures:-

O0034;

G90 G17 G80 G49 G40 G55;

M06T02

M03 S1000

G00 Z10

G01 G41X0Y0D02 F100;

G00X20.Y30

G01 Z-2 F100

G01X7.729Y18.720

G02X12.447Y-17.179R20.; G02

X-5.328Y-11.104R12

G03X-13.279Y0.039R25.; G02

X-19.292Y5.984R40

G02 X10.836Y22.287R10 G00

Z100

M05

Precautions: - Safe working, free from accidents

Equipment Tools and Materials: -

- ✓ Computer/pc
- ✓ Drawing instruments
- ✓ Designee soft wares/ solid work, Catia, AutoCAD,

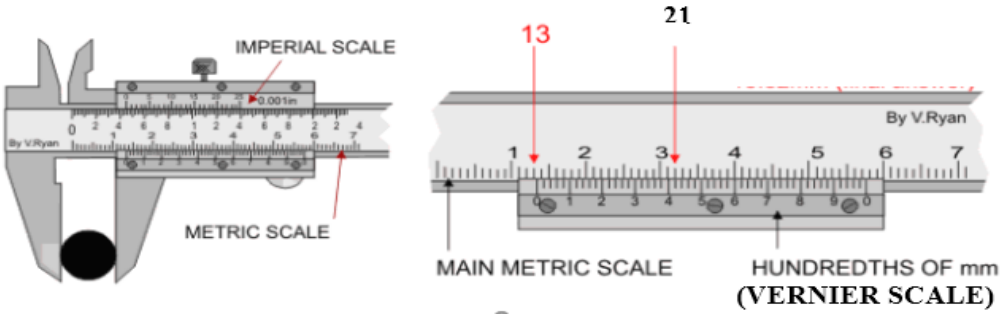
Quality Criteria: - Accurate, Precise & Exact

Lap Test 6.1.

Directions: Reading Metric Vernier Caliper

Date: _____Time started: _____ Time finished: _____

Instructions: Given necessary measuring instruments, tools, and materials you are required to perform the following tasks within --- hour



Task 1 – Look at the main scale

When taking a measurement, you should first read the value on the main scale. On a metric vernier caliper, this will be given in millimeters (mm).

The smallest value that can read from the main scale is 1mm (indicated by a single increment).

The value on the main scale is the number immediately to the left of the 0 marker of the vernier scale.

Task 2 – Look at the vernier scale

Next, look at the vernier scale. The vernier scale of an metric vernier caliper has a measuring range of 1 mm. In the example we will be looking at, the vernier scale is graduated in 50 increments.

Each increment represents 0.02mm. However, some vernier scales are graduated in 20 increments, with each one representing 0.05mm.

When reading the veneer scale, identify the increment that lines up most accurately with an increment on the main scale. This value will make up the second part of your measurement.

Task 3 – Add both values together

To get your total reading, add both the value from the main scale and the value from the vernier scale together.

Reference

1. International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 4, Vol. 4, Dec 2017
2. Design and Analysis of Jigs and Fixtures for Manufacturing Process
   To cite this article: H Radhwan *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* 551 01202
3. JIGS AND FIXTURES: TYPES, PARTS, ADVANTAGES, APPLICATIONS AND MORE. Last Updated on: October 27, 2020, by Saif M
4. MILLING MACHINES AND CNC MILLS: Safety Precautions
5. Process Planning For CNC Machining of Swedish Subcontractors – A Web Survey S. And erberg[a], T. Beno[a]*, L Pejryd[a]
6. Workshop report on milling machine and process.
   TRIBHUVAN UNIVERSITY DEPARTMENT OF MECHANICAL ENGINEERING PULCHWOK CAMPUS
7. Peter Smid (2007) CNC Programming Handbook: Comprehensive Guide to Practical CNC Programming(3th edition)
8. GEORGE HAYNES. Milling Machines and Operations (7th edition) CYBER PRESS
9. Alan Over by. (2011). CNC Machining Handbook Building, Programming, and Implementation. GrawHill
10. Graham T. Smith (1993) CNC Machining Technology Volume II Cutting, Fluids and Work holding Technologies, Springer-Verlag
11. Michael Fitzpatrick. (2014). Machining and CNC Technology (3th edition) Pearson-Prentice Hall. MC Graw Hill
12. Peter Smid, Fanuc CNC Custom Macros. (2005) Programming Resources for Fanuc Custom Macro B Users. Industrial Press

.

Developer's Profile

No	Name	Qualification	Field of Study	Organization/ Institution	Mobile number	E-mail
1)	Abebe Hailiye	(MSc)	Manufacturing engineering	M/G Mulugeta Buli Polytechnic College	0913521369	Aabebe52@gmaile.com
2)	Tsegaye Mulat	(MSc)	Manufacturing technology	A.A tegbaried poly technic college	910411511	Ttshgm97@gmail.com
3)	Boka Terefe	(MSc)	Manufacturing Technology	Ambo poly technic college	917038870	bokaterefe@gmaile.com
4)	Mesay Teshome	BSC	Manufacturing	Hawassa poly technic college	0905 456888	mesayy2018@gmail.com
5)	Mesret Woldemariam	MSC	Manufacturing technology	Wolkite poly technic college	912117416	bmesi2007@gmail.com
6)	Mohammed Bedri	MSC	Manufacturing Technology	Robe poly technic college	913897297	
7)	Mohammed Abdela	MSC	Manufacturing Technology	Ethio italy poly technic college	973794977	mehamed abdela 49@gmail.com