Varuvan Vadivelan Institute of Technology

Dharmapuri – 636 703

LAB MANUAL

Regulation : 2013

Branch : *B.E.* - Mechanical Engineering

Year & Semester : II Year / IV Semester

ME6411- MANUFACTURING TECHNOLOGY LABOROTARY- II

ANNA UNIVERSITY::CHENNAI – 600025

Regulation - 2013

ME6411 - MANUFACTURING TECHNOLOGY LABORATORY –II

List of Experiments

1. Contour milling using vertical milling machine
2. Spur gear cutting in milling machine
3. Helical Gear Cutting in milling machine
4. Gear generation in hobbing machine
5. Gear generation in gear shaping machine
6. Plain Surface grinding
7. Cylindrical grinding
8. Tool angle grinding with tool and Cutter Grinder
9. Measurement of cutting forces in Milling / Turning Process
10. CNC Part Programming.

Outcomes

1. Ability to use different machine tools to manufacturing gears.
2. Ability to use different machine tools for finishing operations
3. Ability to manufacture tools using cutter grinder
4. Develop CNC part programming

INDEX

EX. NO	DATE	NAME OF THE EXPERIMENT	SIGNATURE	REMARKS
1		STUDY OF CENTRE LATHE		
2		EXTERNAL KEYWAY MILLING		
3		SPUR GEAR CUTTING IN PLAIN MILLING MACHINE		
4		SPUR GEAR CUTTING IN GEAR HOBBING MACHINE		
5		HELICAL GEAR CUTTING IN GEAR HOBBING MACHINE		
6		PLAIN SURFACE GRINDING		
7		CYLINDRICAL GRINDING		
8		TOOL GRINDING		
9		TURNING FORCE MEASUREMENTS		
10		TURNING OPERATION IN CNC LATHE MACHINE		
11		STEP TURNING OPERATION IN CNC LATHE MACHINE		
12		MILLING OPERATION IN CNC MILLING MACHINE		
13		MILLING OPERATION IN CNC MILLING MACHINE		

GENERAL INSTRUCTIONS

1. All the students are instructed to wear protective uniform and shoes before entering into the laboratory.
2. Before starting the exercise, students should have a clear idea about the principal of that exercise
3. All the students are advised to come with completed record and corrected observation book of previous experiment.
4. Don't operate any instrument without getting concerned staff member's prior permission.
5. All the instruments are costly. Hence handle them carefully, to avoid fine for any breakage.
6. Utmost care must be taken to avert any possible injury while on laboratory work. In case, anything occurs immediately report to the staff members.
7. One student form each batch should put his/her signature during receiving the instrument in instrument issue register.

WORKSHOP

Workshop technology comprises mainly materials and manufacturing process. Machine tools being the mother of all machines, the manufacturing process have the greatest possible bearing with machine tools.

It is a place where the raw material is converted into finished product. In other words, it is a place where human efforts, machines materials and the tools together manufacture products.

SAFETY MEASUREMENTS

1. Wear uniform and shoes, Remove watch and ID when entering in to work shop.
2. Handle the machine and tools carefully.
3. Jobs should be tightly in the fitting vice.
4. Don’t measure the job while the machine is running.
5. Never operate a machine about which you are not fully aware of control/operation.
6. Tools which are not used should always be kept at their respective places.
7. Do not handle the metal chips and work by hand.
8. Don’t try to shop the lathe chuck while bind.
9. Do not give more depth to cut while the job rotating at high speed.
10. Tighten the tool in the tool post before going the work.

EX.NO: 1
DATE:

STUDY OF CENTRE LATHE

AIM

To study about the centre lathe.

INTRODUCTION

Lathe is called the father of machine tools. The main function of lathe is to remove metals from work piece to give a required shape and size. In the lathe the workpiece is held in the chuck. The tool is moved at an angle of 90^0 to the axis. Various operations such as straight turning, taper turning, and chamfering, facing, knurling, grooving, thread cutting, taper turning are carried out. When the operations above said or done automatically, then the lathe is called automatic lathe.

WORKING PRINCIPLE OF LATHE

In a lathe, the work piece is held in chuck and rotates about its axis by means of power. A single point cutting tool is mounted in tool post. When the chuck rotated the work piece also rotated. The tool moves parallel to the axis of rotation of work piece to produce a cylindrical surface, where as the tool moves perpendicular to the work piece to produce a flat surface. The tool moves at an angle to the axis of work piece to produce a turn surface. The material is removed in the form of chip from the work piece by giving proper feed and depth of cut. So, the required size and shape of the work is obtained.

MAIN PARTS OF LATHE

The lathe consists of various parts. Their parts and functions are discussed below.

1. Bed

Bed is the base of the lathe. The headstock is mounted on the left end; the carriage is in the middle and the tailstock at the right end of bed. The bed is made up of cast iron, alloyed with nickel, chromium. The bed is made up of cast iron to observe shock and vibration created during machining. The guide ways of the bed may be flatter inverted ‘V’ shape.

2. Headstock

It is mounted on the left end of the bed. It carries a hollow spindle. The live center can be attached in the spindle. The spindle nose is threaded. In chuck faceplates can be attached to the spindle. The headstock may be back threaded type. The headstock has two types of driving mechanism.

1. Break geared mechanism
2. Belt driven mechanism

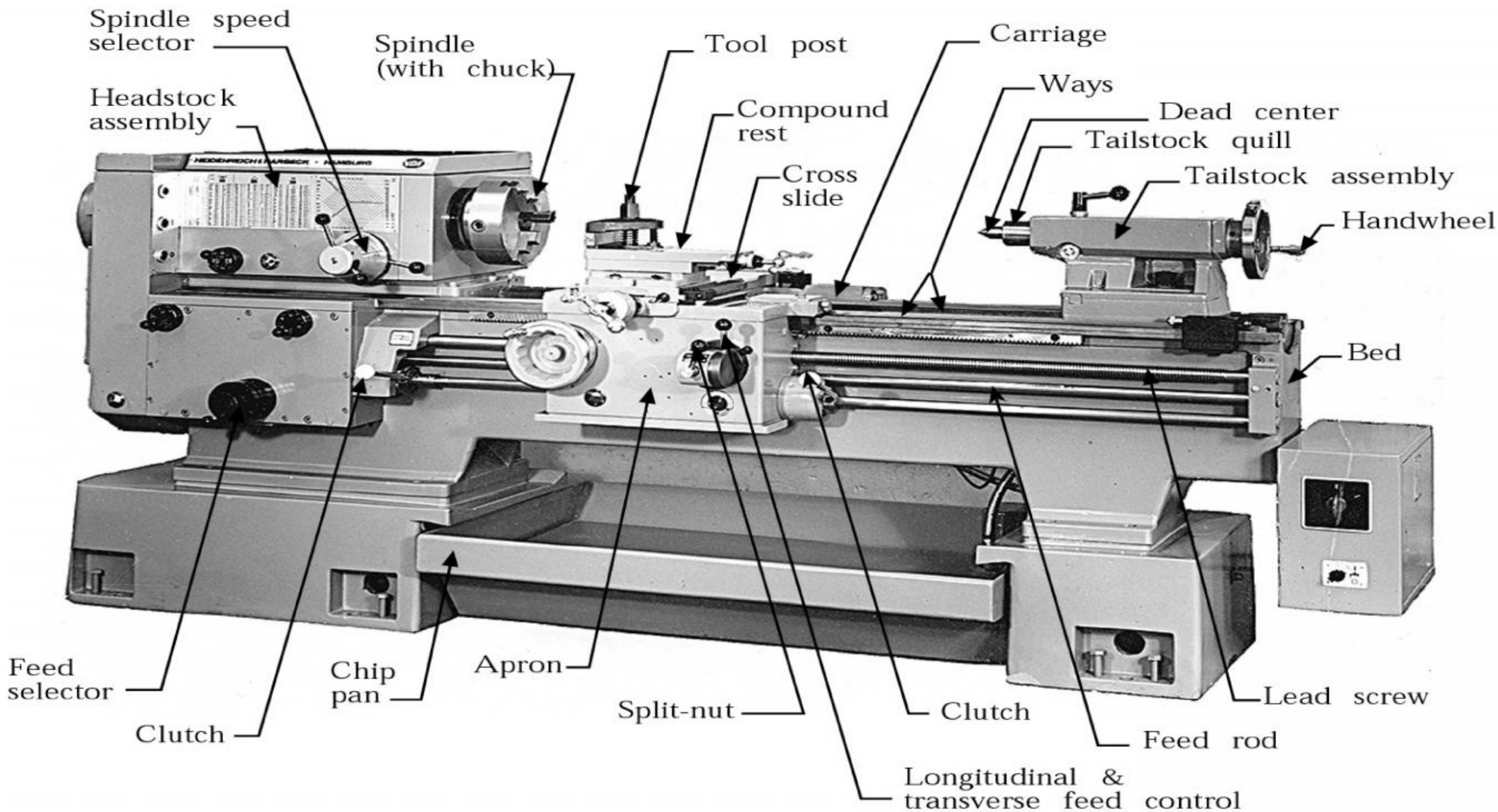


Fig.1 Centre lathe

3. Tailstock

It is located on the bed at the right end. It is used for supports right end of work and also for holding drills, reamer tools for drilling, reaming and such other operations. The tailstock can be moved along the bed and clamped at any position, to support the different length work.

4. Carriage

Carriage is used for giving various feed to the tool by hand or by power. The carriage is attached with the saddle.

5. Saddle

It is an H shaped casting fitted on the bed and moves along the guide ways. It carries the cross slide, compound rest and a tool post.

i) Cross slide

It is attached to the upper side of saddle and carries compound slide and tool post. The cross slide can be moved cross wise by hand or power. The micrometer dial is mounted on the cross slide hand wheel, with an accuracy of 0.05mm.

ii) Compound Rest

It is attached over the cross slide. It is used during the taper turning opening operations to set the tool for angular cuts. Here the micrometer dial is mounted to show the depth of cut.

iii) Tool Post

The tool is clamped over the tool post. It is fixed over the compound rest. There are four types of tool post.

- Single screw tool post
- Open side tool post
- Four bolt tool post
- Four way tool post

6. Apron

Apron is attached to the saddle and hangs in front of the bed. It has gears, levers clutches for moving the carriage automatically. A split nut is attached for engaging and disengaging the carriage from the lead screw. It is used in thread cutting work.

7. Lead Screw

It is a longer screw with standard ACME square threads and used for transmitting power for automatic feed for thread cutting operation.

8. Feed rod

The feed rod is the long shaft used for the movement of carriage along the axis of bed. It is used for operations like facing, turning and boring.

RESULT

Thus the centre lathe was studied.

EX.NO: 2
DATE:

EXTERNAL KEYWAY MILLING

AIM

To make the external keyway on the given work piece by using vertical milling machine.

TOOLS REQUIRED

1. End mill cutter
2. Marker
3. Vernier caliper
4. File

MATERIAL SUPPLIED

1. M.S Flat (55x50x10)
2. Tolerance ± 0.02

All dimensions are in "**mm**"

SEQUENCE OF OPERATION

1. Checking
2. Marking
3. Fitting
4. Milling
5. Checking

PROCEDURE

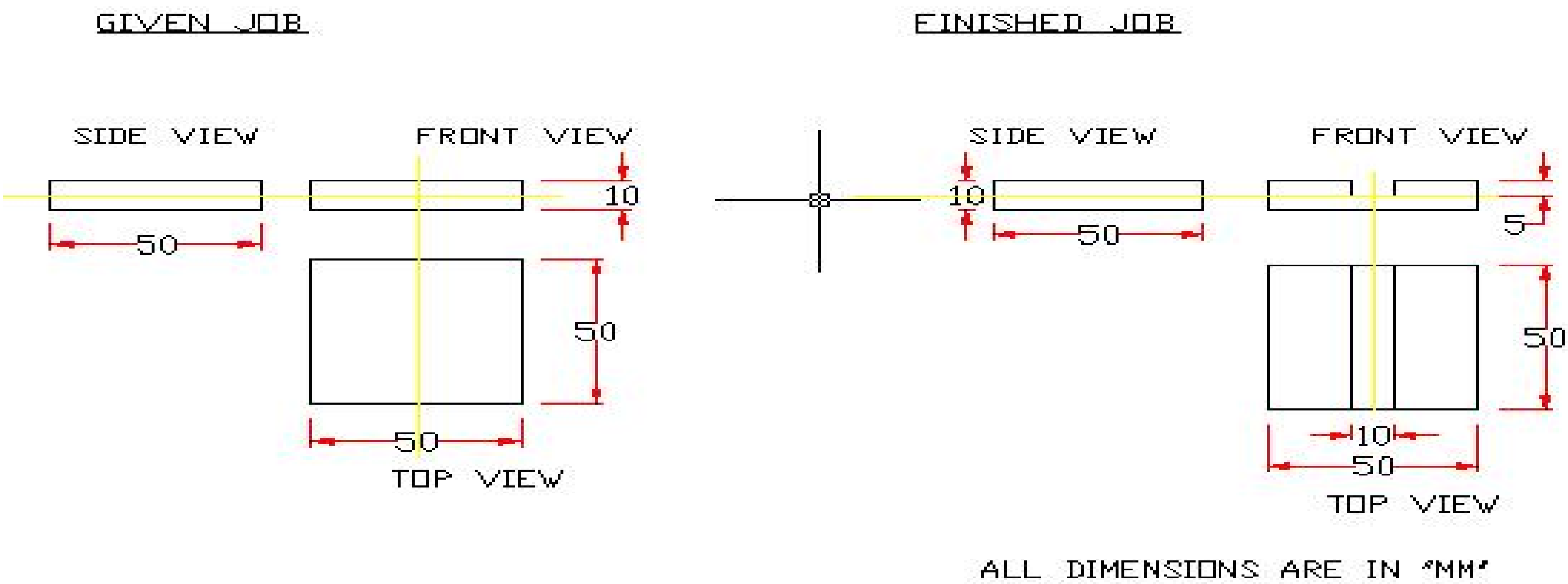
1. The given work piece is checked for the dimensions accurately by using steel rule
2. The work piece is accurately fixed and required end mill cutter is fixed.
3. The feed is given slowly at initial stage in vertical motion
4. The work piece is done and required output is obtained
5. Finally the dimension is checked.

RESULT

Thus the given work piece was made as External Keyway done by vertical milling machine.

EX.NO:02

EXTERNAL KEYWAY MILLING



EX.NO: 3
DATE:

SPUR GEAR CUTTING IN PLAIN MILLING MACHINE

AIM

To cut a spur gear of module 2.5 mm and 23 teeth in given gear blank by using plain milling machine.

MACHINE TOOL REQUIRED

1. Plain or Universal milling machine
2. Lathe machine

TOOLS REQUIRED

1. Gear cutter/ form cutter (module 2.5 mm & cutter no.5)
2. Single point cutting tool
3. Drill chuck
4. Drill bits (ϕ10, 15, 20 mm)
5. Mandrel
6. Dog carrier

MATERIAL SUPPLIED

1. C.I blank (ϕ 75x25)
2. Tolerance ± 0.02

All dimensions are in "**mm**"

PROCEDURE

1. Turn the required diameter of gear blank by using lathe machine and required cutting tool.
2. Make the drilled hole on the gear blank center is the size of the work mandrel.
3. Gear blank with mandrel is fitted between the centers on the machine table.
4. Start machine and rotates the cutter in anti clockwise direction with respect to the cutting speed.
5. Give the depth of cut by using vertical feed in the knee by using handle.
6. Down the indexing head with respect to the number of teeth.
7. Repeatedly indexing for cut the each and every teeth and depth of cut remains constant.

CALCULATION

Addendum	= module (m) in **mm**
Dedendum	= (1.25 * m) in **mm**
Working depth	= (2 * m) in **mm**
Tooth depth	= (2.5 * m) in **mm**
P.C.D (d)	= (z * m) in **mm**
Outer diameter	= d + 2m +2c
	= m (z + 2) in **mm**
Circular pitch	= (п * m) in **mm**
Tooth thickness	= (1.5708 * m) in **mm**
Clearance	= (0.25 * m) in **mm**

Where, z is number of tooth

INDEXING

It is used to split the equal spared division of the circumference of the work piece.

INDEXING USED

Simple indexing is used.

Index crank moment = 40/z

Where,

40 = no of teeth on worm wheel in the inside of the indexing head.
z = no of division to split up
= 40/23
= 1(17/23)

Therefore each time we have to rotate the crank pin is 1 full revolution and 17 holes in a 23 whole circle for complete cut tooth.

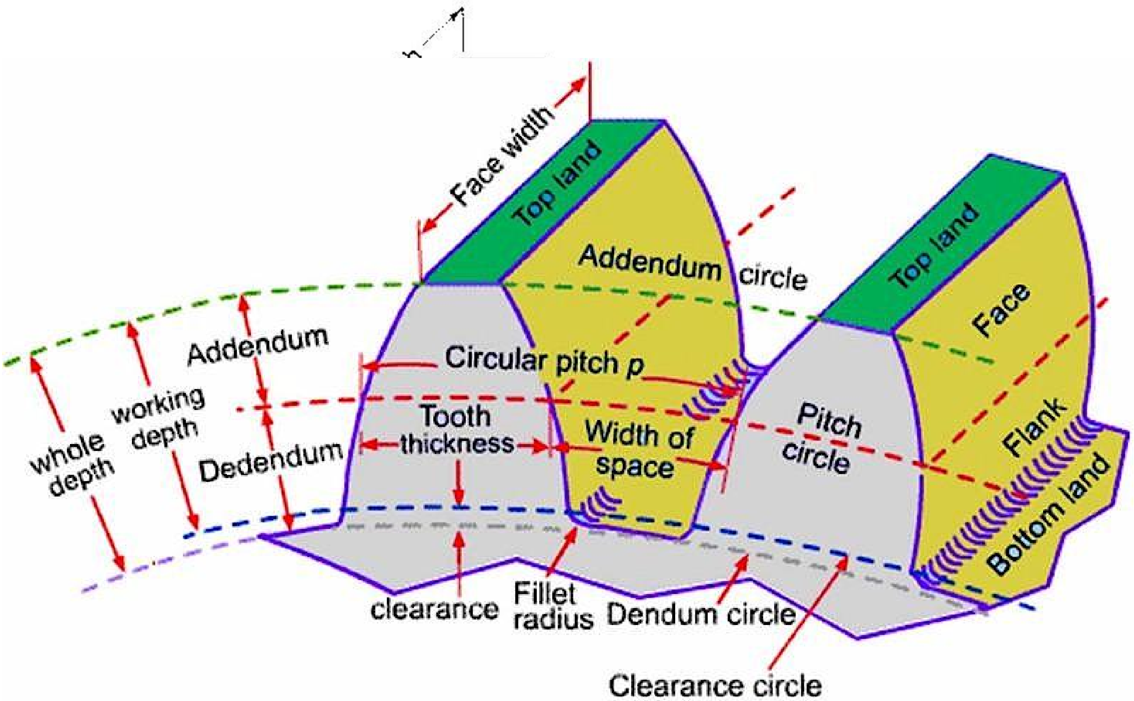


Fig.3 spur gear

RESULT

Thus the mentioned module of gear and number of tooth were cut in plain milling machine.

EX.NO: 4
DATE:

SPUR GEAR CUTTING IN GEAR HOBBING MACHINE

AIM

To cut a spur gear of module 3 mm and 40 teeth in a given gear blank by using gear hobbing machine.

MACHINE TOOL REQUIRED

1. Gear hobbing machine
2. Lathe machine

TOOLS REQUIRED

1. Hob cutter (module 3 mm)
2. Vernier caliper
3. Drill chuck
4. Drill bits (ϕ10, 15, 20 mm)
5. Turning tool

MATERIAL SUPPLIED

1. C.I blank (ϕ 135x25)
2. Tolerance ± 0.02

All dimensions are in “**mm**”

PROCEDURE

1. Turn the given gear blank for the required dimension.
2. Make drill hole on the centre of the gear blank respect to the mandrel diameter.
3. Gear blank is fitted the work mandrel of the rotary table in the machine.
4. The hob cutter is fitted on the hob arbor in the cross slide of the machine.
5. Start the machine and rotates hob cutter with respect to the cutting speed.
6. The required depth of cut is given by the cross feed of the machine.
7. The gear blank moves towards the hob cutter and cuts the tooth.
8. Cut the spur gear while disengaging the differential change gears and engaging feed change gears.

CALCULATION

Addendum	= module (m) in **mm**
Dedendum	= (1.25 * m) in **mm**
Working depth	= (2 * m) in **mm**
Tooth depth	= (2.5 * m) in **mm**
P.C.D (d)	= (z * m) in **mm**
Outer diameter	= d + 2m +2c
	= m (z + 2) in **mm**
Circular pitch	= (π * m) in **mm**
Tooth thickness	= (1.5708 * m) in **mm**
Clearance	= (0.25 * m) in **mm**

Where, z is number of tooth

INDEXING

It is used to split the equal spared division of the circumference of the work piece.

INDEXING USED

Automatic indexing

INDEXING CALCULATION

Automatic indexing = **8 * (g/z)**

Where,

8 = indexing constant
g = start of hob (single start)
z = no of teeth to be cut
= 8 * (1/40)
= 8/40
= 4/20
= 32/160 = (32/40)*(40/160)

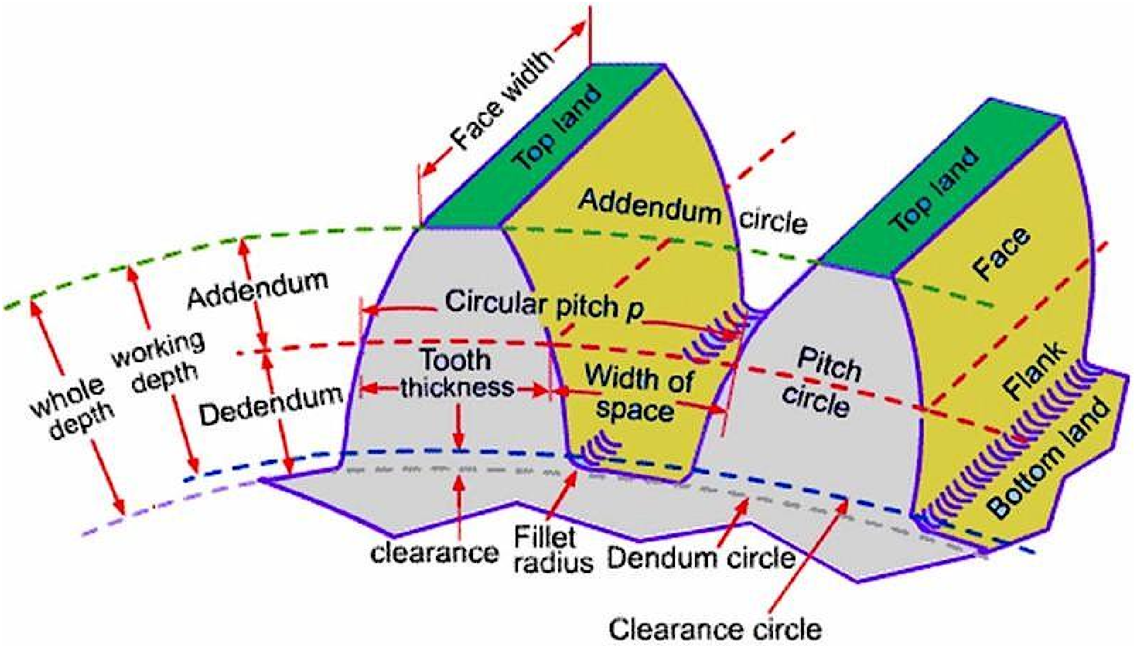


Fig.4. spur gear

RESULT

Thus the spur gear was cut at the given module and no of tooth by using gear hobbing machine.

EX.NO: 5
DATE:

HELICAL GEAR CUTTING IN GEAR HOBBING MACHINE

AIM

To cut a helical gear of module 3 mm and 40 teeth in a given gear blank by using gear hobbing machine.

MACHINE TOOL REQUIRED

1. Gear hobbing machine
2. Lathe machine

TOOLS REQUIRED

1. Hob cutter (module 3 mm)
2. Vernier caliper
3. Drill chuck
4. Drill bits (ɸ10, 15, 20 mm)
5. Turning tool
6. D.E. spanner (10-11, 20-22)

MATERIAL SUPPLIED

1. C.I blank (ɸ 135x25)
2. Tolerance ± 0.02

All dimensions are in "**mm**"

PROCEDURE

1. Turn the given gear blank for the required dimension.
2. Make drill hole on the centre of the gear blank respect to the mandrel diameter.
3. Gear blank is fitted the work mandrel of the rotary table in the machine.
4. The hob cutter is fitted on the hob arbor in the cross slide of the machine.
5. Start the machine and rotates hob cutter with respect to the cutting speed.
6. The required depth of cut is given by the cross feed of the machine.
7. The gear blank moves towards the hob cutter and cuts the tooth.
8. Cut the helical gear while disengaging the differential change gears and engaging feed change gears.
9. The vertical slide is to be fixed as per helix angle and tightening.

CALCULATION

Addendum	= module (m) in **mm**
Dedendum	= (1.25 * m) in **mm**
Working depth	= (2 * m) in **mm**
Tooth depth	= (2.5 * m) in **mm**
P.C.D (d)	= (z * m) in **mm**
Outer diameter	= d + 2m +2c
	= m (z + 2) in **mm**
Circular pitch	= (π * m) in **mm**
Tooth thickness	= (1.5708 * m) in **mm**
Clearance	= (0.25 * m) in **mm**

Where, z is number of tooth

INDEXING

It is used to split the equal spared division of the circumference of the work piece.

INDEXING USED

Automatic indexing

INDEXING CALCULATION

Automatic indexing = **8 * (g/z)**

Where,

8 = indexing constant
g = start of hob (single start)
z = no of teeth to be cut
= 8 * (1/40)
= 8/40
= 4/20
= 32/160 = (32/40)*(40/160)

DIFFERENTIAL INDEXING CALCULATION

Ratio of differential change gears $= \frac{5.96831 \times \sin\alpha}{m}$

$$= \frac{5.96831 \times \sin 15}{m}$$

$$= \frac{36}{66}$$

$$= \frac{36 \times 40}{40 \times 66}$$

where,

Driver gear = 36 teeth
Driven gear = 66 teeth
Ideal gear or intermediate gears are 40 teeth.

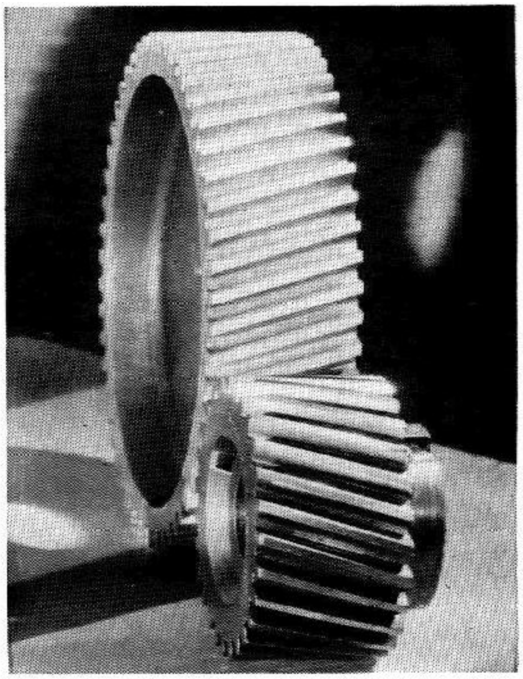


Fig.5 Helical Gear

RESULT

Thus the spur gear was cut at the given module and no of tooth by using gear hobbing machine.

EX.NO: 6
DATE:

PLAIN SURFACE GRINDING

AIM

To grind a plain surface on the given workpiece by the plain surface grinding machine.

TOOLS REQUIRED

1. Flat file
2. Try square
3. Vernier caliper

MATERIAL SUPPLIED

1. M.S Flat (55x50x10)
2. Tolerance ± 5 µm

All dimensions are in "**mm**"

SEQUENCE OF OPERATION

1. Filing
2. Checking
3. Front side grinding
4. Checking
5. Back side grinding
6. Checking

PROCEDURE

1. The filing operation is carried out to the required dimensions.
2. The work piece is clamped on the table reciprocates under the rotating grinding wheel. The work may be held by means of a magnetic chuck or fixture.
3. The trip dogs at the side of the table are adjusted for getting the correct stroke length of the table.
4. The machine is switched ON. The periphery of the grinding wheel does the grinding. Cross-feed is given to the work piece at the end of the every stroke.
5. After the full width of the work is ground, the wheel head is lowered downwards, or the table is raised upwards to give depth of cut.
6. By repeating the same procedure, the remaining surface of the workpiece is ground.

RESULT

Thus the given M.S plate was ground to obtain the required dimensions and surface finishing.

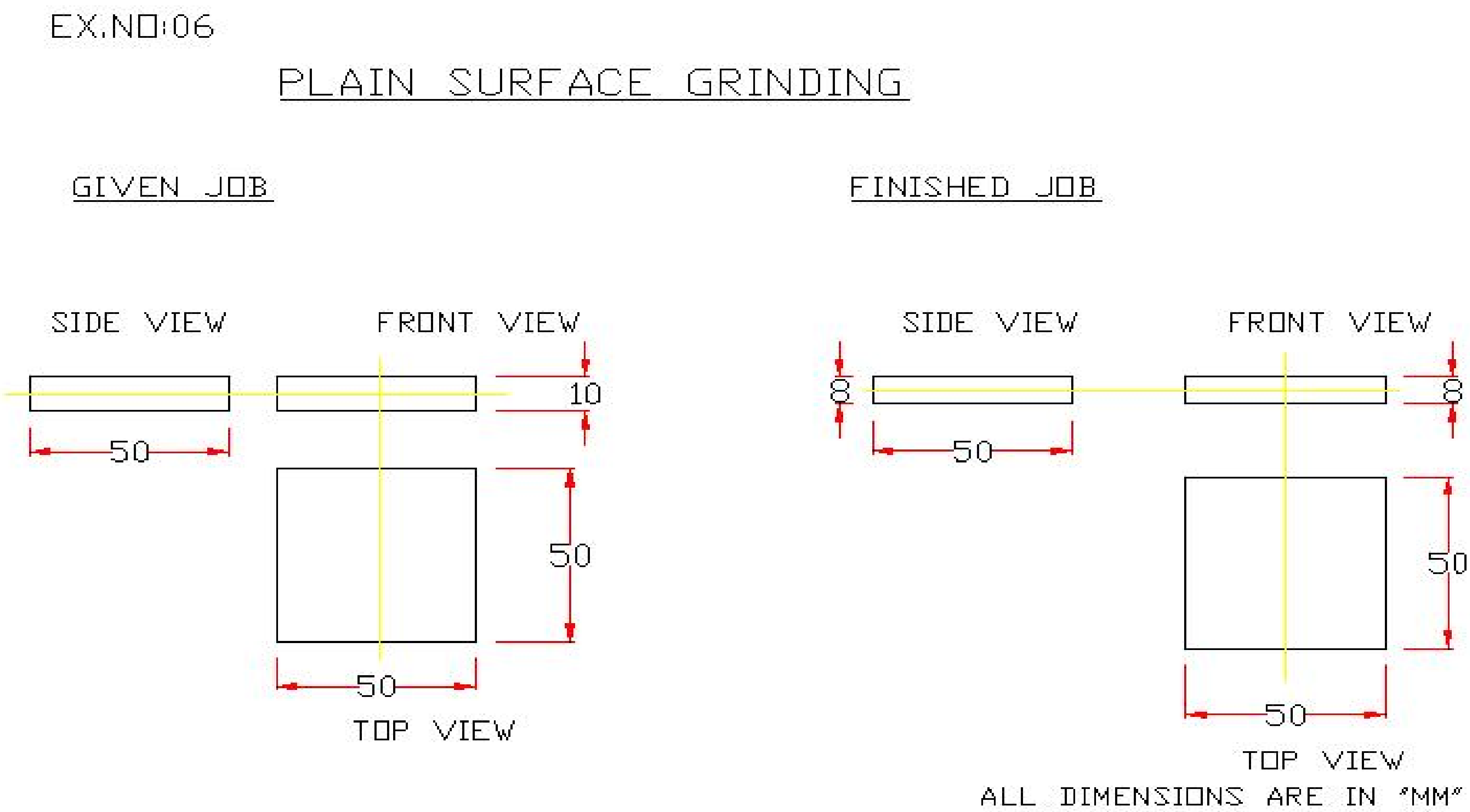


Fig.6 Plain surface grinding

EX.NO: 7
DATE:

CYLINDRICAL GRINDING

AIM

To make cylindrical grinding on a circular rod by using lathe, then use the cylindrical grinding machine to grind.

TOOLS REQUIRED

1. Turning tool
2. Vernier caliper
3. Micro meter

MATERIAL SUPPLIED

1. M.S Round rod (ф25x110)
2. Tolerance ± 5 µm

DESCRIPTION

1. Grinding is metal cutting process performed by means of a rotating abrasive wheel, which act as a cutting tool. This is used to finish work pieces which must show a high surface quality, accuracy of shape and dimensions.
2. Mostly grinding is the finishing operation, it removes comparatively little metal. Grinding is also done to machine materials which are tools used for other machining methods in the case of cutting tools.

PROCEDURE

1. In the given job, the facing and turning operations are carried out in the lathe.
2. By using counter sink drill bit, both ends are drilled for the purpose of holding the job between the centers.
3. The dimensions of the job are measured before the grinding operation.
4. The work piece is held between the centers. It is rotated by a dog and a face plate. The trip dogs at the side of the table are adjusted for setting the correct stroke length of the table.
5. The proper depth of cut is selected. The grinding wheel is fed by hand or automatically towards for successive cuts.
6. The longitudinal feed is given to the workpiece. This feed should not be more than the width of the wheel.
7. By repeating the same procedure, the grinding operation is carried out in the work to obtain the surface finish.

RESULT

Thus the given work was ground to the required dimension by using the cylindrical grinding machine.

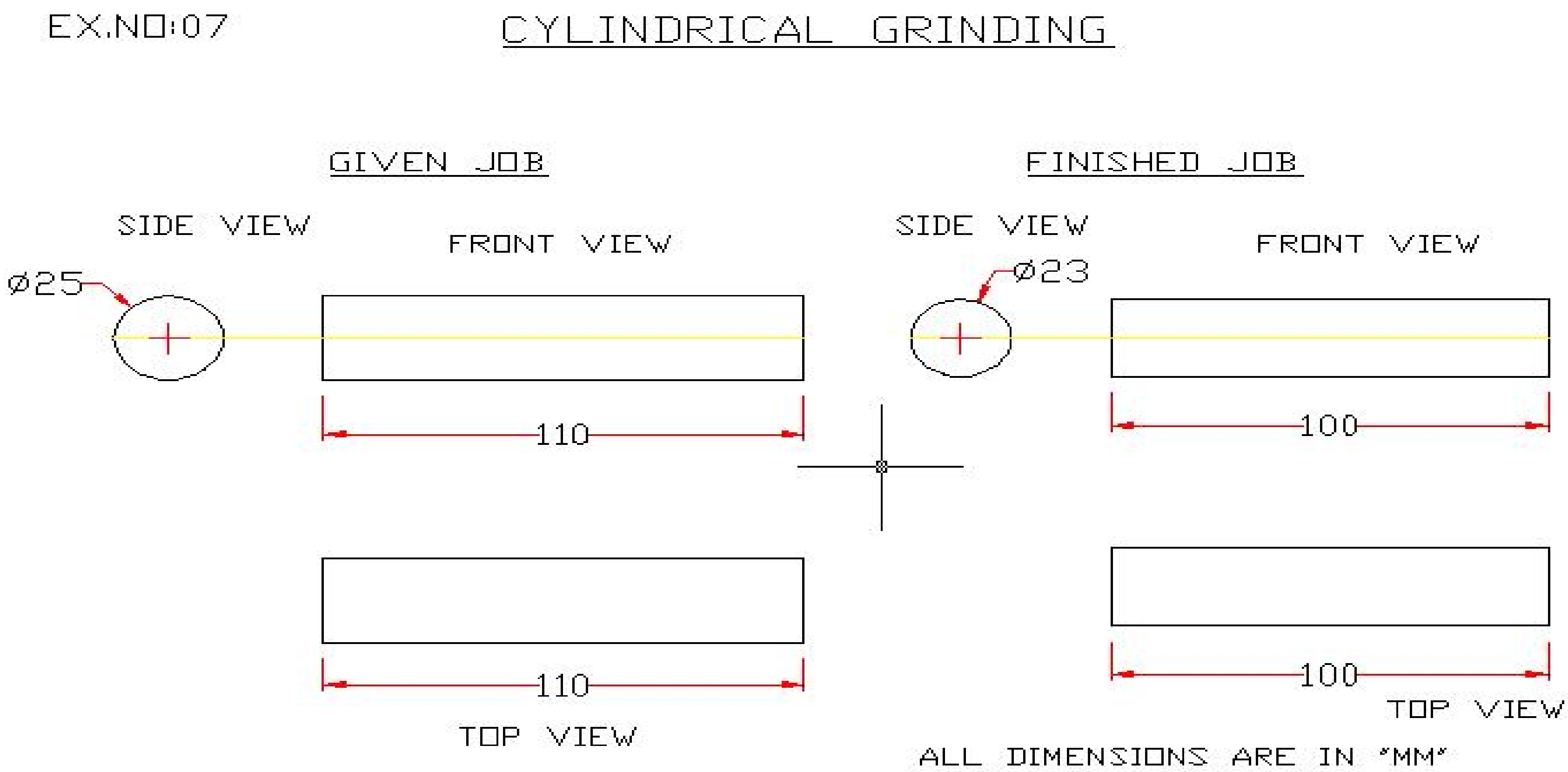


Fig.7. Cylindrical grinding

EX.NO: 8
DATE:

TOOL GRINDING

AIM

To grind a single point cutting tool (Turning tool) by using bench grinding machine.

TOOLS REQUIRED

1. Gloves
2. Goggles
3. Spanner
4. Vernier caliper
5. Steel rule.

MATERIAL SUPPLIED

1. M.S Stock (55x10x10)
2. Tolerance ± 0.02

All dimensions are in "**mm**"

PROCEDURE

1. The correct size tool bit of High speed steel is selected and initial measurements are taken and noted.
2. Proper cutting fluid is selected. The Gloves and goggles are worn for the safety purpose.
3. The job (Tool bit) is held on the grinding machine in proper position.
4. By adjusting the table, the following angles are ground on the tool bit.

Rake angle	- 12^0
End relief angle	- 10^0
Side relief angle	- 10^0
Side rake angle	- 12^0
End clearance angle	- 30^0
Side clearance angle	- 15^0

5. After machining the portions is ground in the tool and cutter grinder as per the figure shown.

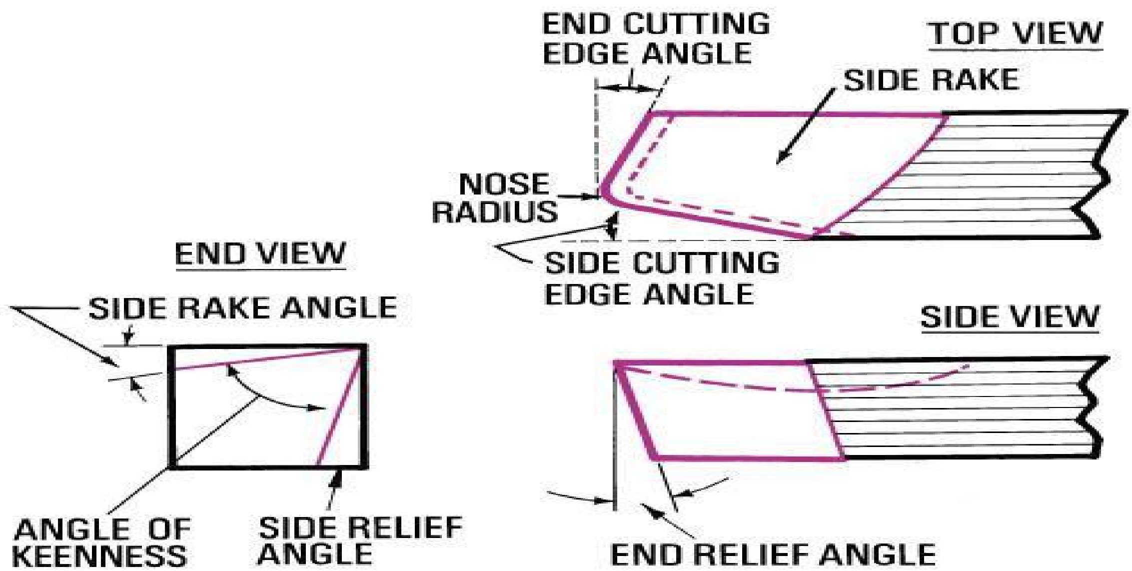


Fig.8. Single point cutting tool nomenclature

RESULT

Thus the single point cutting tool is ground in the tool and cutter grinder as per the fig shown

EX.NO: 9
DATE:

TURNING FORCE MEASUREMENTS

AIM

To measure the cutting forces on turning the job in a lathe machine by using lathe tool dynamometer.

TOOLS REQUIRED

1. Dynamometer with tool setup
2. Strain gauge setup
3. Vernier caliper
4. Turning tool

MATERIAL SUPPLIED

1. M.S Round rod (ϕ25x200)
2. Tolerance ± 5 µm
   All dimensions are in "**mm**"

PROCEDURE

1. Tool dynamometer with setup is fitted on the lathe machine tool post.
2. Three props are fitted (x, y, z) from tool dynamometer to strain gauge.
3. The electrical connection given to strain gauge.
4. Power supply on the strain gauge.
5. Switch ON the lathe machine and to rotates chuck with workpiece with respect to cutting speed.
6. The three force values have to be set zero position in the strain gauge before machining the workpiece.
7. To given depth of cut and turn the workpiece noted on the all forces(x, y,z) in kgf.

RESULT

Thus the measure the cutting forces on turning the job in a lathe machine by using lathe tool dynamometer.

<u>TABULATION</u> (Turning Force Measurements)

S.No	Depth of cut ***mm***	Speed ***rpm***	Feed force (x) ***kgf***	Main force (y) ***kgf***	Thrust force (z) ***kgf***
1					
2					
3					
4					

Note:

- Main force (y) is higher than other two forces (x,z) in **kgf**
- Thrust force (z) is 2 or 3 times less than main force (y) in **kgf**
- Feed force (x) is 4 to 10 times less than main force (y) in **kgf**.

EX.NO:09

TURNING FORCE MEASUREMENT

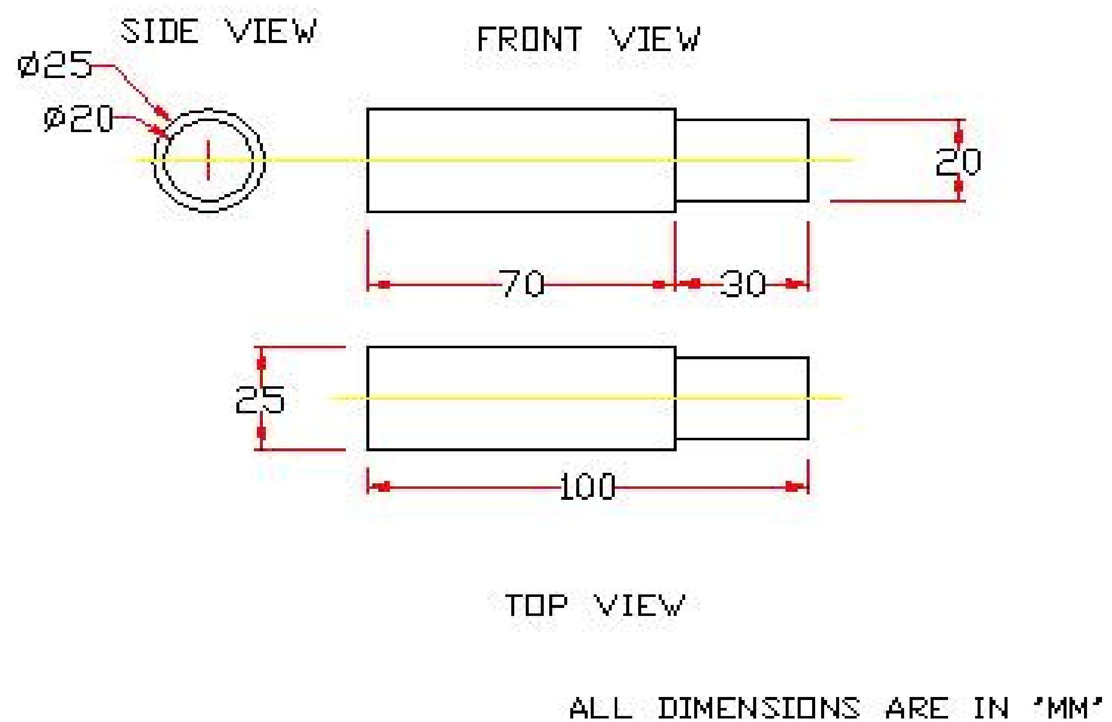


Fig.9. Turning force measurement

INTRODUCTION TO NC (NUMERICAL CONTROL

Numerical Control is a technique of automatically operating a productive facility, based on a code of letters, numbers and special characters. Numerical control has been developed out of the need for higher productivity, lower cost and most precise manufacturing. Numerical control is essentially an application of the digital technology to control a machine.

INTRODUCTION TO NC MACHINE AND ITS COMPONENTS

NC Machine responds to a series of coded instructions by actuating various drives to required extents in desired sequence with pre-set speed, feed, etc., without human intervention. Such instructions are called *part programs*.

A ***part program*** needs to be written for every job to be produced. It instructs the machine to operate in a particular manner. This type of programming is also called *manual part programming* since it is performed manually without the help of a computer.

Numerical control programming with the help of some software is called *computer-aided part programming (CAPP)* or simply *computer-aided manufacturing*. Today several software's are available which automatically generate the codes for a given part.

Fig.10 A typical NC system

The ***Machine Control Unit (MNC)*** is the brain of an NC machine. The information contained in the part program is read by the MNC which, in turn, converts the coded information in the part program into voltage or current pulses of varying frequency or magnitude. These generated electrical signals control the tool movement and also controls miscellaneous operations such as flow of coolant, tool changes, door opening/closing and gripping / un-gripping the job.

NC machines usually have their own ***memory*** where they can store a program when it is read by the machine for the first time. For subsequent production of the same part, the machine need not read the part program again. It uses the program stored in its memory for subsequent execution. A machine without any memory must read the part program every time. This slows the production process considerably.

The primary types of memory are RAM (Random Access Memory) and ROM (Read One Memory). RAM is a volatile memory. It gets washed out the moment the machine is switched off, unless a battery back-up is provided for the RAM. ROM, on the other hand, is a non-volatile memory. It stores information permanently which can be read any number of times unless the information is deliberately erased or overwritten. It does not need any power supply to retain the information fed to it. In addition to memory, the MCU also contains *hardware and software* necessary to read and interpret the coded program for obtaining the desired movements in the machine.

Since an NC Machine does not have an on-board computer, a separate computer is required for preparing codes for machining a part. The coded program is usually transferred to the machine through a punched tape which the machine reads by passing light through it. Presence and absence of a hole is taken as 1 and 0 binary signals respectively.

ADVANTAGES OF NC MACHINE OVER CONVENTIONAL MACHINE

The principal advantage of an NC machine is the increased and accurate control of the cutting tool which would be manually very difficult or even impossible in some cases.

A simple example is circular motion of the tool where movements along both X and Y axes need to be simultaneously controlled while it is virtually impossible to do this manually on a conventional machine, an NC machine can easily perform this task within the accuracy of microns, that too any number of times.

DISADVANTAGE OF NC MACHINE

Since an NC machine does not have an on-board computer, a separate computer is required for preparing codes for machining a part. Besides this, the machine has to read the coded tape every time a part is to be produced even if the same part is to be reproduced. This results in loss of time and sometimes error in reading. The tape is usually made of paper, may also get damaged after repeated use. Moreover, even for a small change the whole tape has to be re-made.

INTRODUCTION TO COMPUTER NUMERICAL CONTROL (CNC) MACHINE

A CNC machine is essentially an NC machine with a dedicated computer being its integral part. It has got more flexibility compared to an NC machine.

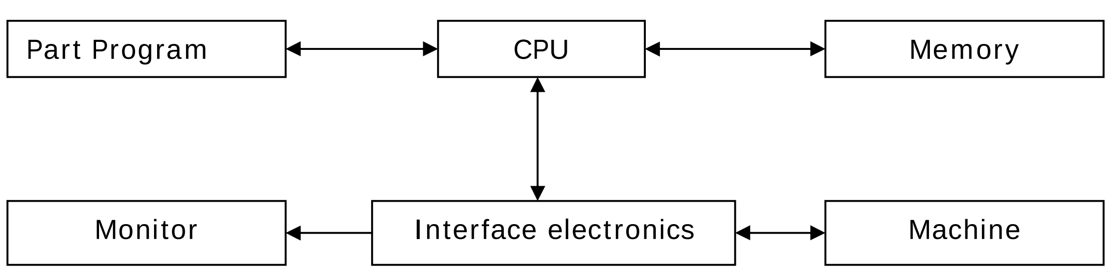


Fig 11. CNC system

Numerous types of CNC machines have been manufactured. Out of these, CNC Lathe / Turning Centre and CNC Milling / Machining Center are very commonly used.

ADVANTAGES OF CNC SYSTEM OVER NC SYSTEM

1. In conventional NC machine, the control is hardwired which makes any change in the controller very difficult because of limitations of its basic configuration. A CNC machine does not have such limitations which are inherent to an NC machine. A bare of minimum of electronic hardware is used for control. Software is used for obtaining the basic function leads to increased productivity and flexibility in manufacturing.

2. Compared to NC machines, CNC machines have the added advantage of reading, storing and editing the part programs. They also provide graphical capabilities, diagnostic procedures and system troubleshooting. This simplifies the operation and maintenance of CNC machines to a great extent.

INTRODUCTION TO DIRECT NUMERICAL CONTROL (DNC) MACHINE

If a large capacity computer directly controls a number of NC machines, such a system is called DNC machine. This is useful because in present age of computer-aided manufacturing, centralized data handling and control is desirable. The main frame computer stores programs and after processing, sends the control signals to respective NC machines.

LIMITATIONS OF DIRECT NUMERICAL CONTROL SYSTEM

1. It is expensive because a mainframe computer with a large memory is required.
2. Extensive cabling work is involved for interlinking the machines to the main computer.
3. All the machines should be compatible with the computer being used, and in case of any problem with the computer, the whole system stops functioning.

INTRODUCTION TO DISTRIBUTIVE NUMERICAL CONTROL (DNC) MACHINE

DNC is also the abbreviation for Distributive Numerical Control, which uses a network of computers to coordinate the operations of several machines.

Though expensive, such a system can control the entire manufacturing operation of a company, and thus, it is a step towards automation of the manufacturing system.

TOOL MOVEMENT MODES

In an NC / CNC machine, usually the tool moves with respect to the workpiece which remains at the same place.

There are three types of motion control used in an NC / CNC machine:

1. Point-to-point placement
2. Axial cut
3. Contour cutting

1. Point-to-point placement

Such a control simply places the tool over desired locations in desired sequence. There is no control over the speed of the tool movement between selected points, which is always a fast traverse.

This type of control can be used in drilling, punching or similar machines where only the location of the tool at the time of machining is important.

2. Axial Cut

This control allows the tool to move along any major axis with desired speed. Therefore, cutting along X, Y, or Z axis is possible. The limitation being simultaneous motion along two axes is not possible. So, it cannot make an angular cut. That is why, it is also called *straight cut* control.

For an angular cut, the job will have to be reoriented so as to make the cutting direction parallel to one of the axes. A machine, which is capable of performing axial cuts, also provides point-to-point control.

3. Contour Cutting

This is the most flexible but the most expensive type of control. It permits simultaneous control of more than one axis movement of the tool. So, it is possible to make any complex contour which is approximated by several small straight line segments within permissible tolerance band.

The contour cutting or contouring control also permits point-to-point and axial cut movements. Milling and turning operations are common examples of contouring control.

INTERPOLATION SCHEMES

In contouring control, the tool is made to move along a contour such as a circle or other smooth curves. Some of these curves can be exactly defined mathematically using simple formulae, whereas more complex ones can only be represented approximately. In any case, the fundamental problem is that the curves are continuous whereas control is digital. Hence, interpolation is a very important aspect in contour cutting.

To cut along a curve, the curve must be divided into a series of small straight line segments. The tool is made to trace these straight lines. For obtaining good accuracy, the number of straight lines must be extremely large.

Interpolation schemes have been developed which calculate the intermediate points automatically for a given curve. The MCU locates the intermediate points and instructs the tool to follow the path defined by joining these points by straight lines. These straight lines are so small that the resulting contour quite a smooth curve for all practical purposes.

A number of interpolation schemes are available on various types of machine. They include:

1. Linear interpolation
2. Circular interpolation
3. helical interpolation
4. Parabolic interpolation
5. Cubic interpolation

Out of these, linear and circular interpolations are the most common and are available on most of the machines.

TOOL CHANGING DEVICES

In a CNC machine, tools are changed through program instructions. The tools are fitted in a tool magazine or drum. When a tool needs to be changed, the drum rotates to an empty position, approaches the old tool and pulls it. Then it again rotates to position the new tool, fits it and then retracts. This is a typical tool changing sequence of an automatic tool changer (ATC) on a milling machine.

On a lathe machine, the tool magazine only need to rotate to a new position to allow the new tool to come in the cutting position. There is no need to change the tool physically. Tool changing time is of the order of a few seconds. This saves time and thus, increases productivity.

CNC APPLICATIONS

Computer numerical control has been used in a wide variety of machine tools. In fact, whenever good accuracy and repeatability is desired and frequent changes in component type are expected, a CNC machine becomes an ideal choice.

Some of the machines where computer numerical control is used are listed below:

1. Lathe
2. Turning centre
3. Miller
4. Machining centre
5. Drilling machine
6. Gear hobbing machine
7. Grinding machine
8. Electro-discharge machine
9. Welding and cutting

10. Coordinate measuring machine, etc.

CO-ORDINATES (X, Y, AND Z WORD)

These give the coordinates positions of the tool. In a two axis system, only two of the word would be used. In a four or five axis machine, additional a - words and/or b - words would specify the angular positions.

Although different NC systems use different formats for expressing a coordinate, we will adopt the convention of expressing it in the familiar decimal form. For examples X+7.325 or Y-0.500. Some formats do not use the decimal point in writing the coordinates. The positive sign to define positive coordinate locations is mandatory.

FEED RATE (F WORD)

This specifies the feed in a machining operation. Units are inches per minute (ipm).

CUTTING SPEED (S WORD)

This specifies the cutting speed of the process, the rate at which the spindle rotates.

TOOL SELECTION (T WORD)

This word would be needed only for machines with a tool turret or automatic tool changes. The t-words specifies which tool is to be used in the operation .For example T05 might be the designation of a 1/2 –in drill bit in turret position 5 on a NC turret drill.

MISCELLANEOUS FUNCTION (M WORD)

This M - word is used to specify certain miscellaneous or auxiliary function which may be available on the machine tool of course, the machine must possess the function that is being called an example would be M03 to start the spindle rotation. The miscellaneous function is the last word in the block. To identify the end of the instruction, an end of block (EOB) symbol is punched on the tape.

LIST OF G – CODES

G CODES	FUNCTION
G00	Positioning rapid transverse
G01	Linear interpolation
G02	Circular interpolation clockwise
G03	Circular interpolation anticlockwise
G04	Dwell
G20	Inch unit
G21	Metric unit
G28	Automatic zero return
G30	2^{nd} reference point return
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G43	Tool length compensation
G52	Work coordinate system 1
G54	Work coordinate system 2
G55	Work coordinate system 3
G56	Work coordinate system 4
G57	Work coordinate system 5
G58	Work coordinate system 6
G74	Left hand tapping cycle
G76	Fine boring cycle
G80	Canned cycle
G81	Drilling cycle
G82	Drilling cycle with dwell
G83	Peck drilling cycle /deep drilling cycle
G84	Tapping cycle
G85	Boring cycle/ reaming cycle
G86	Boring cycle
G87	Back boring cycle
G90	Absolute command
G91	Incremental command
G94	Feed per minute
G95	Feed per revolution
G98	Return to initial position in canned cycle
G99	Return to R point in canned cycle

LIST OF M CODES FOR MILLING

M CODES	FUNCTIONS
M00	Optional program stop automatic
M01	Optional program stop request
M02	Program end
M03	Spindle ON CW
M04	Spindle OFF CCW
M05	Spindle stop
M06	Tool change
M07	Mist coolant ON(coolant 1 ON)
M08	Flood coolant ON(coolant 1ON)
M09	Coolant OFF
M19	Spindle orientation
M30	End program
M98	Sub program call
M99	Sub program end

EX.NO: 10
DATE:

STEP TURNING OPERATION IN CNC LATHE MACHINE

AIM

To write a manual part program to step turn the given component by using CNC lathe machine.

TOOLS REQUIRED

1. Single point cutting tool
2. Vernier caliper

MATERIAL SUPPLIED

1. Aluminum Round rod (ϕ25x110)
2. Tolerance ± 0.02
   All dimensions are in "**mm**"

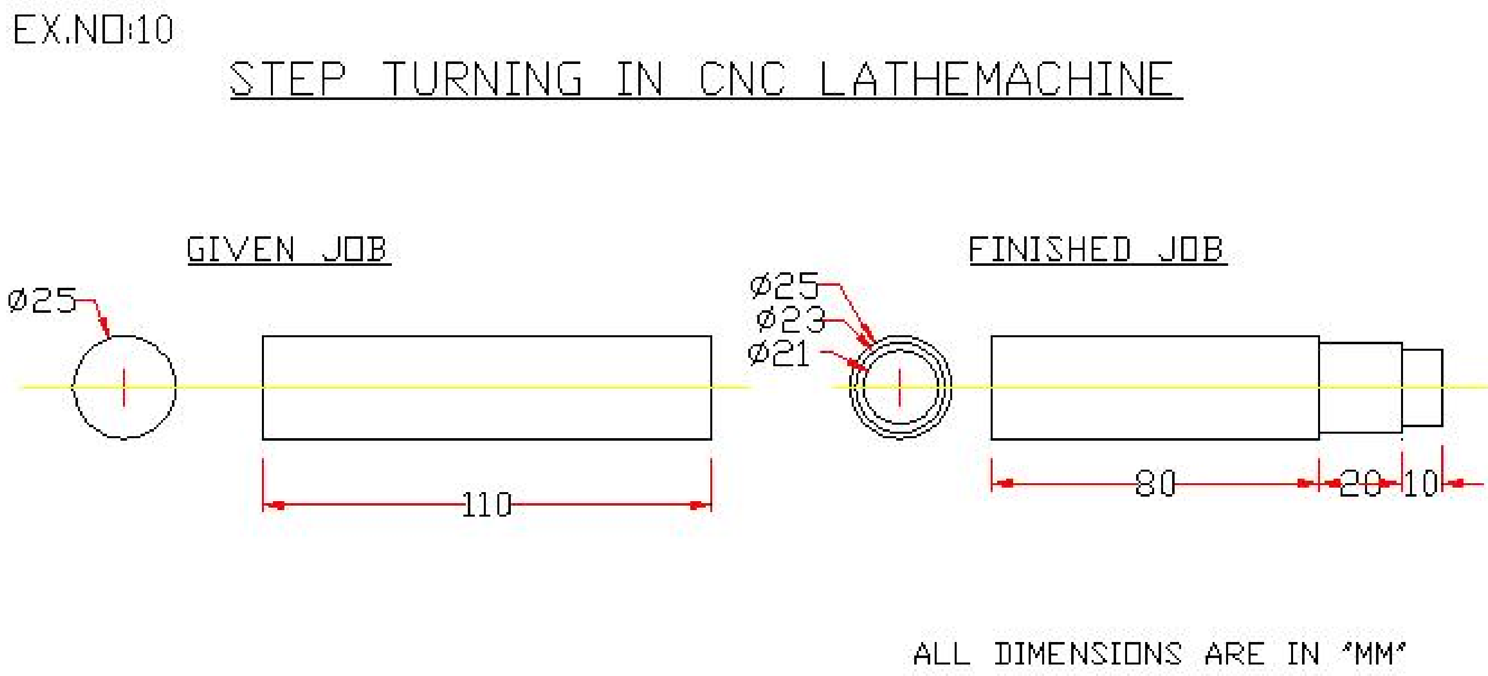


Fig: 12 Step turning operation in CNC Lathe

PROGRAM

Program code	Description
TOOL/STANDARD,15,55,0,10,3)	Tool Definition
(STOCK/80,25,0,0)	Stock Definition
G90 G21	Absolute mode, Metric Units
N01 M03 S2000	Spindle Start Clockwise With Spindle Speed 2000
N02 G00 X25 Z2	Rapid Positioning Up to the Reference Point
N03 G01 X24	Linear interpolation with 1mm cut to the diameter
N04 Z-20 F80	Up to the Length -20 with Feed Rate 80
N05 G01 X25	Linear interpolation with 1mm along diameter
N06 G00 Z2	Rapid Movement Up to Initial Point
N07 G01 X23	Linear interpolation with 1mm cut to the diameter
N08 Z-20 F80	Up to the Length -10 with Feed 80 mm/min
N09 G01 X25	Linear interpolation with 1mm along diameter
N10 G00 Z10	Rapid Movement Up to Initial Point
N11 G01 X22 F60	Linear interpolation with 1mm cut to the diameter
N12 G01 Z-10 F60	Up to the Length -10 with Feed 60mm/min
N13 G01 X25 F60	Linear interpolation with1mm along diameter
N14 G00 Z2	Rapid Movement Up to Initial Point
N15 G01 X21 F60	Linear interpolation with 1mm cut to the diameter
N16 G01 Z-10 F60	Up to the Length -10 with Feed 60mm/min
N17 G01 X25 F60	Linear interpolation with 1mm along diameter
N18 G00 Z2	Rapid Movement Up to Initial Point
N19 M30	Program End & Rewind

RESULT

Thus the manual part program was step turned the given component by using CNC lathe machine.

EX.NO: 11
DATE:

STEP TURNING OPERATION IN CNC LATHE MACHINE

AIM

To write a manual part program to step turning in the given component by using CNC lathe machine.

TOOLS REQUIRED

1. Single point cutting tool
2. Vernier caliper

MATERIAL SUPPLIED

1. Aluminum Round rod (ϕ25x110)
2. Tolerance ± 0.02
   All dimensions are in "**mm**"

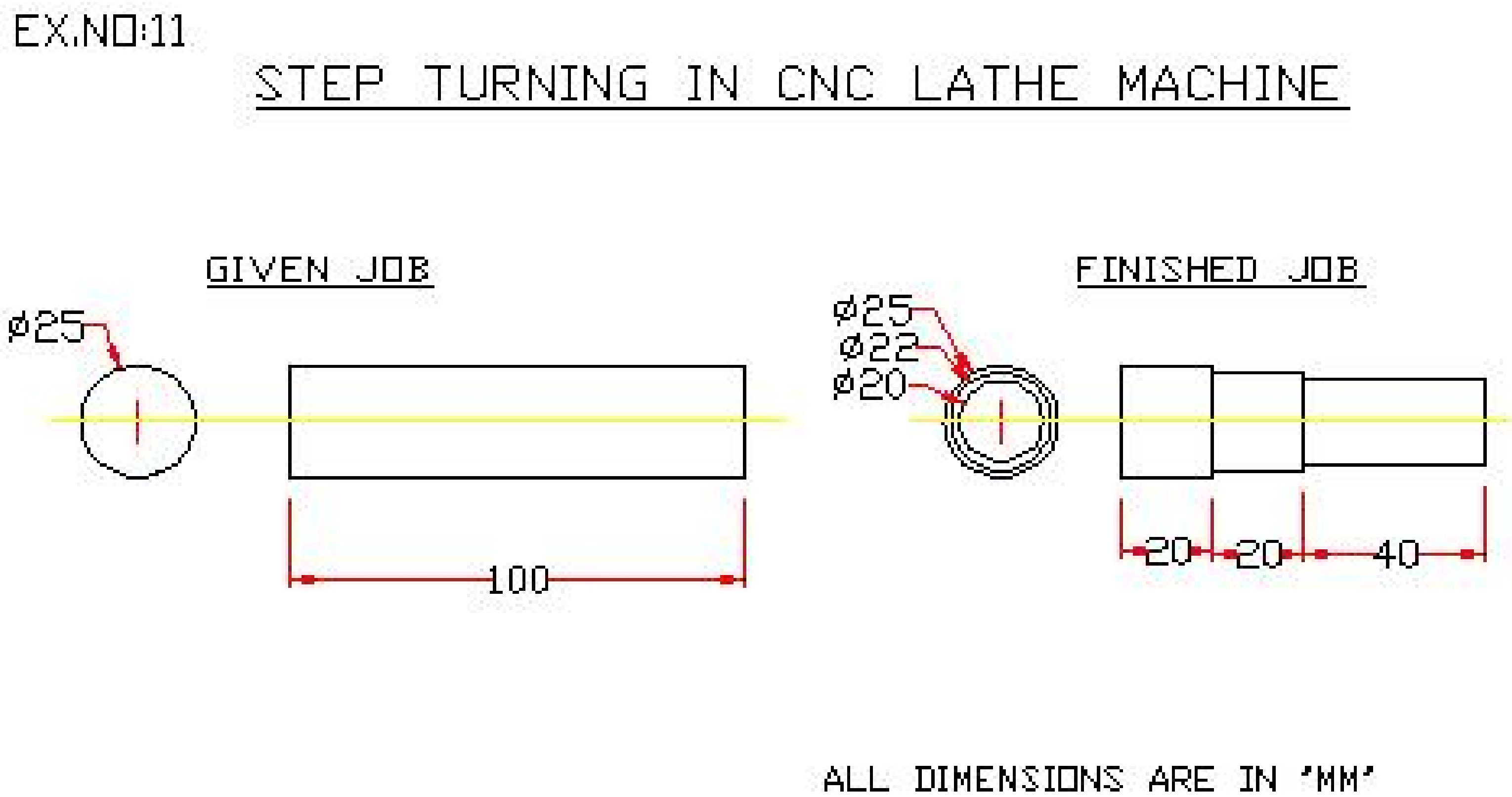


Fig: 13 Step turning operation in CNC Lathe

PROGRAM

Program code	Description
G21 G97 G98	Metric mode, constant RPM and feed in mm/min selected
G28 U0 W0	Tool goes to its home if already not there
M03 S600	Clockwise rotation 600 rpm
G00 X30 Z1	Fast positioning to position A, which becomes the start point for the parting cycle
G75 X15 Z1 P2000 Q0 F200	Grooving cycle selected (G75), (X15, Z1) is the target point. P is the depth of cut in µm and Q is the Z axis stepping distance in µm. F is the feed in mm/min.
G00 X30 Z1 Z-1	Tool position back to A Tool shifting to face 1 mm thick
G01 X15 Z-1 F200	Facing done at feed of 200 mm/min
G00 X30 Z1	Fast positioning to position A, which becomes the start point for the parting cycle
G90 X26 Z40 F200	Turning to 26 mm dia. At the distance Z= -40mm
X24	Turning to 24 mm dia. from 40 mm
X22	Turning to 22 mm dia. from 40 mm
X20	Turning to 20 mm dia. from 40 mm
G00 X30 Z1	Fast positioning to position A, which becomes the start point for the parting cycle
G90 X26 Z-100 F200	Turning to 26 mm dia. At the distance Z= -100 mm, at feed in 200 mm/min
G00 X30 Z1	Return to position A
G00 U0 W0	Tool goes to its home
G28	Spindle stops
M05	Door open
M38	Chuck open
M10	Program reset
M30	End of the program

RESULT

Thus the manual part program was step turned in the given component by using CNC lathe machine.

EX.NO: 12
DATE:

MILLING OPERATION IN CNC MILLING MACHINE

AIM

To write a manual part program to mill the given component by using CNC lathe machine.

TOOLS REQUIRED

1. End mill cutter
2. Vernier caliper

MATERIAL SUPPLIED

1. Aluminum Flat (55x55x10)
2. Tolerance ± 0.02
   All dimensions are in "**mm**"

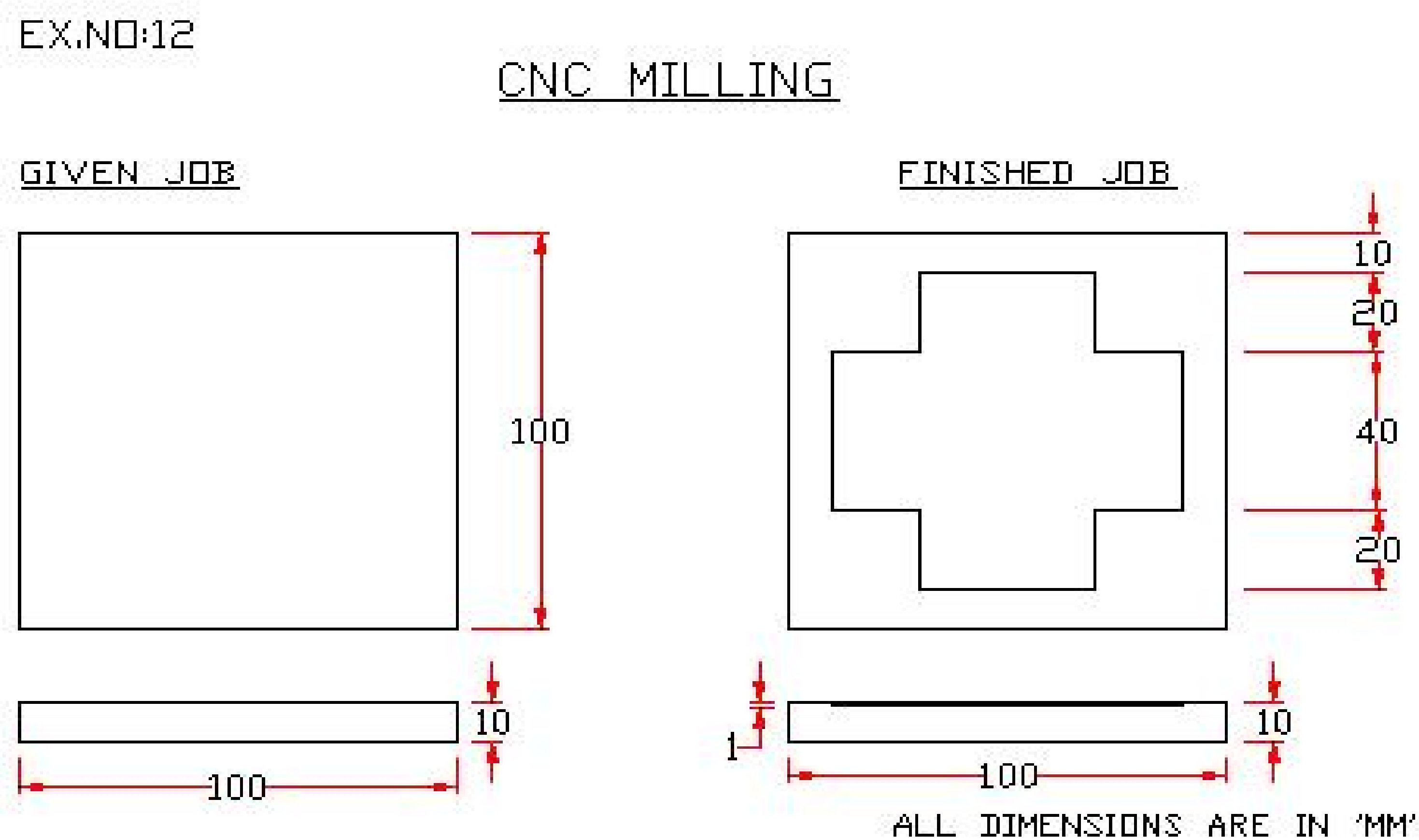


Fig: 14 Milling operation in CNC milling machine

PROGRAM

Program code	Description
(TOOL/MILL, 6,0,50,0) *(STOCK/BLOCK 100,100,10,10)*	Flat end mill cutter of 6 mm dia
N00 G90 M03 S2000	Spindle start clockwise with speed 2000
N01 G90 G00 X0	Absolute program mode, rapid positioning
N02 Y0	Rapid positioning
N03 Z2	Rapid positioning up to 2 mm along Z axis
N04 G00 X30 Y10 Z2	Rapid positioning
N05 G01 Z-1F50	Linear interpolation
N06 G01 X70 Y10	X axis movement by 70mm Y axis movement by 10 mm
N07 G01X70 Y30	Y axis movement by 30 mm
N08 G01 X90 Y30	X axis movement by 90 mm
N09 G01 X90 Y70	Y axis movement by 70 mm
N10 G01 X70 Y70	X axis movement by 70 mm
N11 G01 X70 Y90	Y axis movement by 90 mm
N12 G01 X30 Y90	X axis movement by 30 mm
N13 G01 X30 Y70	Y axis movement by 70 mm
N14 G01 X10 Y70	X axis movement by 10 mm
N15 G01 X10 Y30	Y axis movement by 30 mm
N16 G01 X30 Y30	X axis movement by 30 mm
N!7 G01 X30 Y10	Y axis movement by 10 mm
N18 G00 Z5	Rapid positioning up to 5mm along Z axis
N19 X00 Y00 Z10	Z axis movement by 10 mm
N20 M30	Program end and rewind
N21 G01 X0 Y-40	Y axis movement by -40 mm
N22 G01 X20 Y0	X axis movement by 20 mm
N23 G01 X0 Y-20	Y axis movement by -20 mm
N24 G90	Absolute program mode
N25 G00 Z5	Rapid positioning up to 5mm along Z axis
N26 X00 Y00 Z10	Z axis movement by 10 mm
N27 M30	Program end and rewind

RESULT

Thus the manual part program was milled the given component by using CNC milling machine.

EX.NO: 13
DATE:

MILLING OPERATION IN CNC MILLING MACHINE

AIM

To write a manual part program to mill the given component by using CNC lathe machine.

TOOLS REQUIRED

1. End mill cutter
2. Vernier caliper

MATERIAL SUPPLIED

1. Aluminum Flat (55x55x10)
2. Tolerance ± 0.02
   All dimensions are in "**mm**"

PROGRAM

Program code	Description
(TOOL/MILL,4,0,50,0) (STOCK/BLOCK 100,100,10,0,0,10)	Cutter flat end mil cutter of 4 mm Dia.
N00 G90 M03 S2000	Spindle starts clockwise with speed 2000
N01 G90 G00 X0	Absolute program mode, rapid positioning
N02 Y0	Rapid positioning
N03 Z2	Rapid positioning up to 2 mm along Z axis
N04 G00 X25 Y10 Z2	Rapid positioning
N05 G01 Z-1 F50	Linear interpolation
N06 G03 X10 Y25 R15 F50	Anticlockwise circular interpolation
N07 G01 X10 Y70	Linear interpolation
N08 G03 X25 Y85 R15 F50	Anticlockwise circular interpolation
N09 G01 X70 Y85	Linear interpolation
N10 G02 X85 Y70 R15 F50	Clockwise circular interpolation
N11 G01 X85 Y25	Linear interpolation
N12 G03 X70 Y10 R15	Anticlockwise circular interpolation
N13 G01 X25 Y10	Linear interpolation
N!4 G00 Z5	Rapid positioning up to 5 mm along Z axis
N15 X00 Y00 Z10	Z axis movement by 5m m
N16 M30	Program end and rewind

EX.NO:13

CNC MILLING

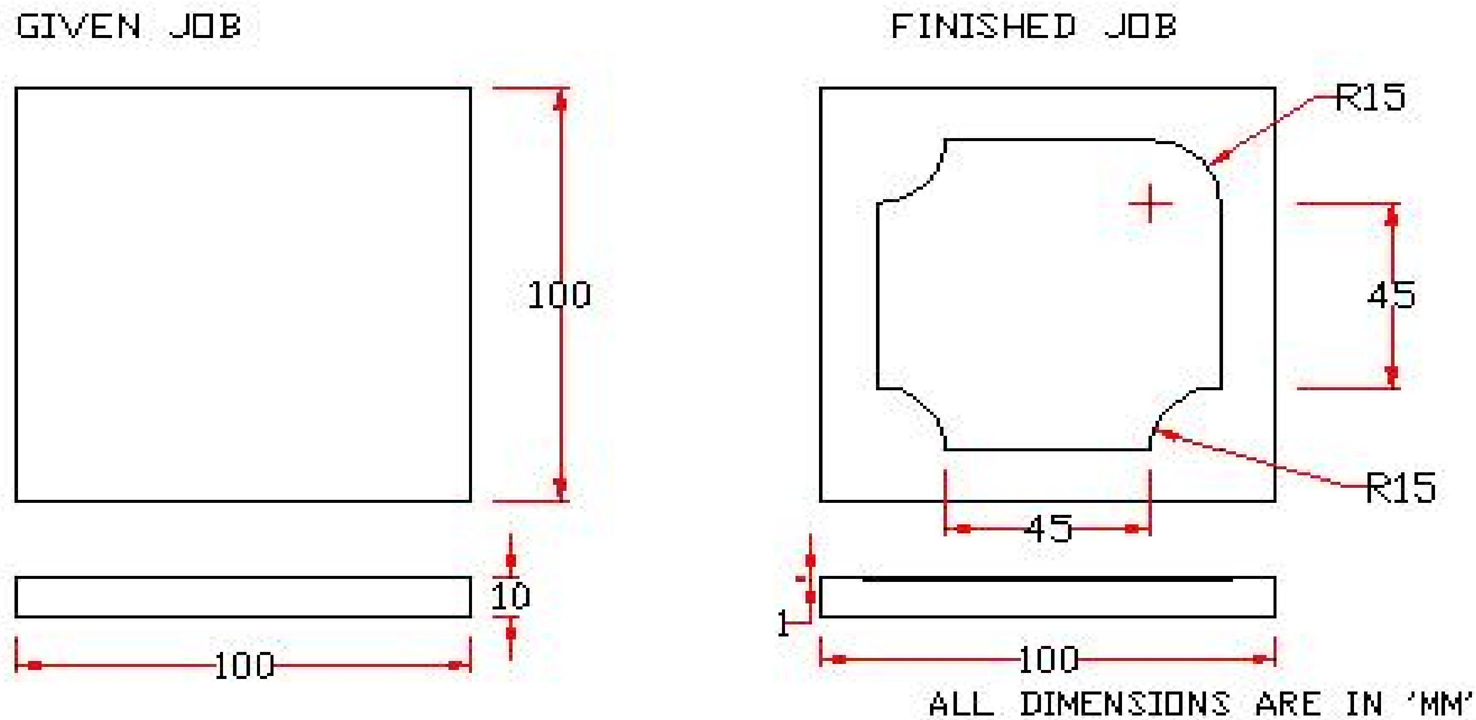


Fig.15 Milling operation in CNC milling machine

RESULT

Thus the manual part program was milled the given component by using CNC milling machine.