K.L.N. College of Engineering

(An Autonomous Institution, Affiliated to Anna University, Chennai)

Pottapalayam – 630 612

Accredited by National Assessment and Accreditation Council (NAAC)

Department of Mechanical Engineering

Accredited by NBA, New Delhi

Approved Research Center by Anna University, Chennai

REGULATIONS 2020

20ME4L1

MANUFACTURING TECHNOLOGY LABORATORY MANUAL

Lab In charge

Dr. N. NAGASUBRAMANIAN, M.E., Ph.D.

Prepared by

Dr. N. Nagasubramanian, M.E., Ph.D.

Assistant Professor / Mech. Engg.

Approved by

Dr. P. Udhayakumar

HOD / Mech. Engg.

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

To become a Centre of excellence for Education and Research in Mechanical Engineering.

MISSION

- Attaining academic excellence through effective teaching learning process and state of the art infrastructure.
- Providing research culture through academic and applied research.
- Inculcating social consciousness and ethical values through co-curricular and extra-curricular activities.

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

PEO I	Graduates will have successful career in Mechanical Engineering and service industries.
PEO II	Graduates will contribute towards technological development through academic research and industrial practices.
PEO III	Graduates will practice their profession with good communication, leadership, ethics and social responsibility.
PEO IV	Graduates will adapt to evolving technologies through lifelong learning.

PROGRAM SPECIFIC OBJECTIVES (PSOs)

PSO I	Derive technical knowledge and skills in the design, develop, analyze and manufacture of mechanical systems with sustainable energy, by the use of modern tools and techniques and applying research based knowledge.
PSO II	Acquire technical competency to face continuous technological changes in the field of mechanical engineering and provide creative, innovative and sustainable solutions to complex engineering problems.
PSO III	Attain academic and professional skills for successful career and to serve the society needs in local and global environment.

K.L.N. College of Engineering

(An Autonomous Institution, Affiliated to Anna University, Chennai)

Pottapalayam – 630 612

Accredited by National Assessment and Accreditation Council (NAAC)

Department of Mechanical Engineering

Accredited by NBA, New Delhi

Approved Research Center by Anna University, Chennai

REGULATIONS 2020

20ME4L1

MANUFACTURING TECHNOLOGY LABORATORY MANUAL

Lab In charge

Dr. N. NAGASUBRAMANIAN, M.E., Ph.D.

Name : ***Roll No. :***

Year / Sem. / Sec : ***Reg. No :***

20ME4L1 MANUFACTURING TECHNOLOGY LABORATORY L T P C

0 0 3 1.5

OBJECTIVES:

- **To practice the various operations that can be performed in Lathe.**
- **To gain practical knowledge about shaper, drilling, milling machines etc.**
- **To understand the various grinding processes.**
- **To measure the cutting forces in Turning/ Milling Process.**
- **To write CNC programs for Machining processes.**

PREREQUISITE: NIL

LIST OF EXPERIMENTS

1. **External Thread cutting in lathe**
2. **Eccentric Turning in lathe**
3. **Square Head Shaping**
4. **Spur gear cutting in milling machine**
5. **Helical gear cutting in milling machine**
6. **Contour milling in vertical milling machine**
7. **Angular drilling in Radial drilling machine**
8. **Gear generation in gear hobbing machine**
9. **Gear generation in gear shaping machine**
10. **Surface grinding and Cylindrical grinding**
11. **Measurement of cutting forces in Milling / Turning Process**
12. **Simple CNC Programming – Lathe and Milling**

TOTAL: 45 PERIODS

OUTCOMES:

AT THE END OF THE COURSE, LEARNERS WILL BE ABLE TO:

- **Perform various operations in Lathe.**
- **Perform shaping, drilling and milling operations.**
- **Generate gear profile using milling, gear hobbing and gear shaping machines.**
- **Use grinding machine for surface finishing operations on simple parts.**
- **Calculate cutting forces using cutting tool dynamometer in Turning/ Milling Process.**
- **Develop CNC programming for the simple components produced in CNC lathe and CNC milling.**

LIST OF EQUIPMENT FOR A BATCH OF 30 STUDENTS

S.No.	NAME OF THE EQUIPMENT	Qty.
1	Centre Lathes	7
2	Shaper	1
3	Radial Drilling Machine	1
4	Horizontal Milling Machine	1
5	Vertical Milling Machine	1
6	Surface Grinding Machine	1
7	Cylindrical Grinding Machine	1
8	Centerless grinding machine	1
9	Gear Hobbing Machine	1
10	Gear Shaping machine	1
11	Lathe Tool Dynamometer	1
12	Milling Tool Dynamometer	1
13	CNC Lathe	1
14	CNC Milling machine	1

Contents

General Instructions for Laboratory Classes

- Students must attend the lab classes with ID cards.
- Boy should "**TUCK IN**" the shirts.
- Students should wear uniform only.
- **LONG HAIR** should be protected.
- Any other website **should not be operated** other than the prescribed one for that day.
- POWER SUPPLY to your test table should be obtained only through the **LAB TECHNICIAN.**
- Any damage to any of the equipment / instrument / machine caused due to carelessness, the **cost** will be fully recovered from the individual (or) group of students.

Safety Precautions

1. Do not wear watch, ring etc in your hand while machining
2. Wear shoe and lab coat
3. Attention to be paid for clamping the job, tool, tool holders or supporting items
4. Care should be taken for avoiding accidental contact with revolving cutters.
5. Do not handle chips with bare hands, use brush or hand gloves.
6. Pay attention while selecting tools or blades for the proposed use to avoid accidents.
7. Do not remove chip while machine is running
8. Care should be taken while selecting rapid feed
9. Follow safety precautions while approach with cutter to avoid tool damage
10. Use coolants for heat dissipation
11. Avoid sharp edge tools
12. Select proper speed or feed or depth of cut

Index

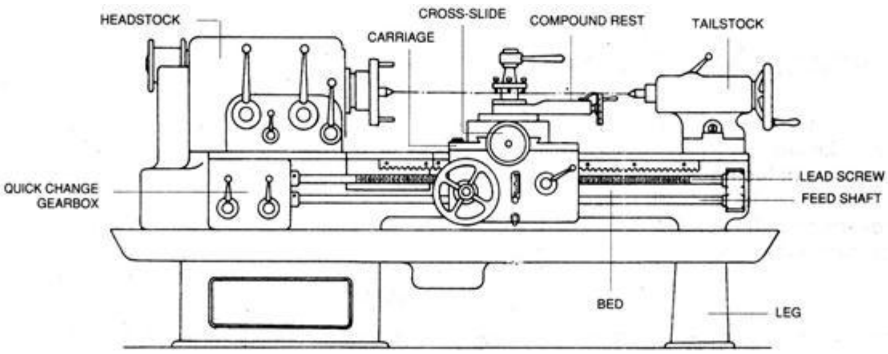
S. No.	Date	Name of the Experiment	Page	Marks	Staff Signature
1.					
2.					
3.					
4.					
5.					
6.					
7.					
8.					
9.					
10.					
11.					
12.					
13.					
14.					
15.					
16.					
17.					
18.					
19.					
20.					

Completed date:

Average Mark: **Staff - in – charge**

Introduction to Lathe

Introduction:

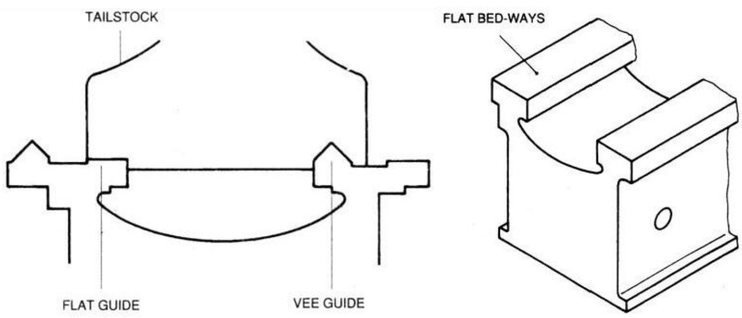


Lathe

The main function of a lathe is to remove metal iron a piece of work to give it the required shape and size. The lathe is used to machine cylindrical shapes. In a lathe, work piece is held and rotated about its axis. Generally single point cutting tool is used as the cutting tool. The tool is moved parallel to the axis of rotation of work piece to produce a cylindrical surface. The tool is moved perpendicular to the work piece axis to produce a flat surface. The tool is moved at an angle to the axis of work piece to produce a tapered surface. Straight turning, taper turning, eccentric turning, chamfering, facing, parting off, drilling, boring, reaming, tapping, knurling, forming, grooving, polishing, spinning and thread cutting are the main operations done on lathe. When the operations are done automatically, then the lathe is called automatic lathe.

Parts of a Lathe

The various parts and their functions are given below:



Bed

It is the base of the machine. The headstock is mounted on the left end, the carriage in the middle and the tailstock at the right end of bed. The carriage and the tailstock move over the bed. The bed has guide ways. The bed is very strong to resist the cutting forces and vibrations. The bed has ribbed construction. The guide ways are very accurate for getting accuracy in jobs. The bed is made of cast iron alloyed with nickel and chromium. The guide ways of the bed may be flat or inverted 'V'.

Headstock

It is mounted on the bad at the left end. It carries a hollow spindle. A long bar may be passed through the hole of the spindle. The front end of the hole is tapered for holding tapered shanks (Morse Taper). A live center can be attached into the spindle. The spindle nose is threaded. Chucks and face plates can be attached to the nose of the spindle. The headstock has the driving and speed changing mechanisms. Headstock may be of back geared type or all geared type. There are speed changing and feed changing levers attached to the headstock.

Tailstock

It is mounted on the bed on the right end. It is used for supporting the right end of work. It is also used for holding drill, reamer or tape for drilling, reaming or tapping operations. To support different lengths of work, body of the tailstock can be moved along the bed and clamped at any position. The upper body of the tailstock can be moved towards or away from the operator for taper turning. The tailstock body is bored and the tailstock spindle moves through it. The spindle can be moved axially by means of a hand wheel. A dead center can be fixed into the taper hole of the spindle for supporting the right end of work.

Carriage

The carriage is used for giving various movements to .the tool by hand and by power. The carriage has the following parts;

i) Saddle

It is a H shaped casting fitted over the bed. It moves along the guide way. It carries the cross slide and tool post. It can be moved to the required position and locked to the bed.

ii) Cross Slide

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

ii) Compound Rest

The compound rest is marked in degrees. It is used during taper turning to set the tool for angular cuts. There is no power feed to the compound rest. It can be operated only by hand to feed the tool longitudinally or at an angle to the lathe axis. There is micrometer dial for showing the depth of cut. The compound slide should be locked strongly with its base after any setting.

iv) Tool Post

The tool is clamped in the tool post. The tool post is fitted over the compound rest.

Apron

The apron is attached to the saddle and hangs in front of the hand and automatically. There is a split nut for engaging the carriage with the lead screw for thread cutting. The apron hand wheel is used to move the carriage parallel to the lathe axis.

Specification of a Lathe

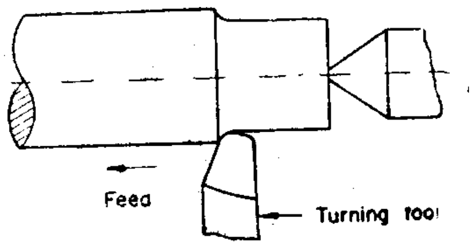
A lathe of specified by:

1. Length of bed
2. Length between centers
3. Height of centers from the bed
4. Swing diameter of work over bed
5. Swing diameter of work over the carriage
6. Maximum bar diameter which will pass through the hole of headstock spindle.

Lathe Operations

1. Straight turning	2. Facing	3.Forming
4. Shoulder turning	5. Chambering	6.Drilling
7. Boring	8. Thread cutting	9. Knurling
10. Taper turning	11. Reaming	12.Parting off
13. Grooving	14. Spinning	15.Eccentric turning

Straight Turning



STRAIGHT TURNING

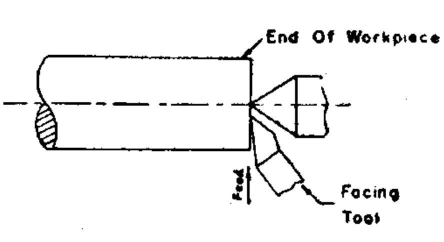
In straight turning, cylindrical surface is produced. The work piece is rotated about the axis. The tool is fed parallel to the axis .The work piece is held between the lathe centre or held in chuck. A right hand turning tool is clamped on the tool post .For light cut, the tool inclined towards the head stock. For heavy cuts the tool is inclined towards the tailstock Hand feed or automatic feed can be used for straight turning.

For rough turning the rate of feed of the tool is fast and depth of cut is very heavy, (Feed is rate of travel of tool. It is expressed in mm per revolution of the job)For rough turning tool is used .In rough turning the depth of cut will be from 2 mm to 5 mm. The feed rate will be from 0.3 to 1.5 mm per revolution of work.

For finish turning the cutting speed will be high, feed will be small and depth of cut will be very small. A finish turning tool is used in finish turning. The depth of cut will be from 0.5 to1 mm. The feed will be from 0.1to 0.3mm per revolution of the work piece.

Facing

In facing operation, the end of the work piece is made flat. It is sometimes called squaring. The work may be held between centers or in a chuck. The work piece is rotated about lathe axis. A facing tool is fed perpendicular to the axis of lathe. Tool is slightly, inclined towards the work piece end. The feed can be given by hand or it can be automatic



FACING

Taper Turning

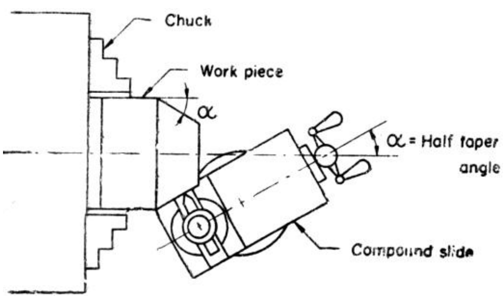
Taper

A taper is defined as the uniform change in the diameter of a work piece measured along its length. It is expressed as ratio of the difference in diameter to the length. Taper is also expressed in degrees of half the included angle. Taper turning produces a conical surface on the work piece on lathe.

Taper turning can be done by the following methods:

1. Form Tool
2. Tailstock Set Over
3. Swiveling the Compound Rest
4. Taper Turning Attachment
5. Combining Longitudinal Feed and Cross Feed

Compound Rest Method



In this method, the work piece is rotated parallel to the lathe axis and the tool is moved at an angle to the lathe axis. The compound rest is mounted on a circular base. The base can be swiveled and fixed at an angle. The compound base is swiveled and set at half taper angle. The tool is moved by the compound rest hand wheel. The work is rotated in a chuck or face plate or between centres. This method is used for turning steep tapers on small lengths of job. The compound rest can be swiveled up to 45° on both sides. The tool should be moved only by hand.

Drilling

For drilling, the work piece is rotated in a chuck or face plate. The tailstock spindle has a standard taper. The drill is fitted into the tailstock spindle directly or through drill chuck of socket. The tailstock is moved over the bed and clamped on the bed near the work. When the job rotates, the drill bit is moved into the work by turning the tailstock hand wheel. Enough coolant is supplied during drilling operations. Tailstock spindle has markings to know the length of hole drilled.

Calculation of the machining time

The amount of time which is assigned for the completion of a work order (e.g. Thread cutting) is called total time. It is the sum of setting time, machining time, auxiliary time, and delay time.

Setting time

Setting time is the time needed for preparing for a certain operation and after operation, bringing it to its original state; this includes also the study of drawings, the time required for adjusting tools, getting tools out of store and returning them.

Auxiliary time

Auxiliary time is the time required regularly for progress of the operation such as positioning and removing of the work piece, setting the depth of cut, measuring and tool sharpening.

Machining time

Machining time is the time during which operations are performed which contribute directly to the completion of the work order (e.g. time in which the work piece is machined, operating time of the machine, cutting time).

Delay time

Delay time is the time allowed for personal needs, overcoming fatigue, and unavoidable delays. Delay time occurs irregularly .It comprises of such conditions as walking to the lavatory, rest periods, waiting for material, etc.

Machining time for turning and facing can be determined by calculation.

$$\text{Machining time} = \frac{\text{Turning length}}{\text{Feed/min}}$$

$$t_m = \frac{L}{s \times n}$$

Longitudinal turning

L = length of the work piece (L_1)+starting allowance (l_a)+allowance after turning (l_u)

$L = L_1 + l_a + l_u$

Where

t_m - Machining time ; L - Turning length;

s - feed in mm/rev; n - revolutions per min

$\mathbf{s'} = \mathbf{s} \times n$ s' - feed per min

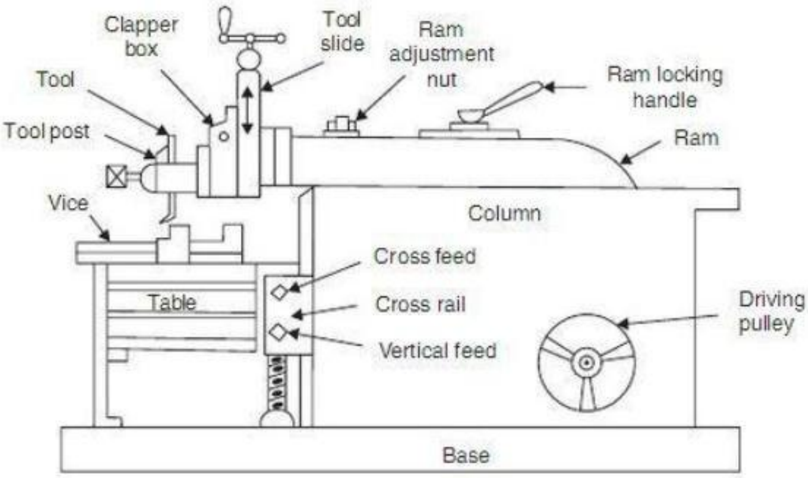
Facing:

The turning length L corresponds with the radius plus starting allowance

$L = r + l_a$

Introduction to Shaper

Introduction:



Shaping Machine

Base

It is rigid and heavy cast iron body to resist vibration and takes up high compressive load. It supports all other parts of the machine, which are mounted over it. The base may be rigidly bolted to the floor of the shop or on the bench according to the size of the machine.

Column

The column is a box shaped casting mounted upon the base. It houses the ram-driving mechanism. Two accurately machined guide ways are provided on the top of the column on which the ram reciprocates.

Cross rail

Cross rail of shaper has two parallel guide ways on its top in the vertical plane that is perpendicular to the rai1 axis. It is mounted on the front vertical guide ways of the column. It consists of mechanism for raising and lowering the table to accommodate different sizes of jobs by rotating an elevating screw which causes the cross rail to slide up and down on the vertical face of the column. A horizontal cross feed screw is fitted within the cross rail and parallel to the top guide ways of the cross rail. This screw actuates the table to move in a crosswise direction

Saddle

The saddle is located on the cross rail and holds the table on its top. Crosswise movement of the saddle by rotation the cross feed screw by hand or power causes the table to move sideways.

Table

The table is a box like casting having T -slots both on the top and sides for Clamping the work. It is bolted to the saddle and receives crosswise and vertical movements from the saddle and cross rail.

Ram

It is the reciprocating part of the shaper, which reciprocates on the guide ways provided above the column. Ram is connected to the reciprocating mechanism contained within the column.

Tool head

The tool head of a shaper performs the following functions.

It holds the tool rigidly,

It provides vertical and angular feed movement of the tool, and

It allows the tool to have an automatic relief during its return stroke.

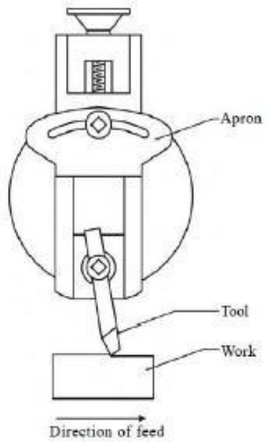
The various parts of tool head of shaper are apron clamping bolt, clapper box, tool post, down feed, screw micrometer dial, down feed screw, vertical slide, apron washer, apron swivel pin, and swivel base. By rotating the down feed screw handle, the vertical slide carrying the tool gives down feed or angular feed movement while machining vertical or angular surface. The amount of feed or depth of cut may be adjusted by a micrometer dial on the top of the down feed screw. Apron consisting of clapper box, clapper block and tool post is clamped upon the vertical slide by a screw. The two vertical walls on the apron called clapper box houses the clapper block, which is connected to it by means of a hinge pin. The tool post is mounted upon the clapper block. On the forward cutting stroke the clapper block fits securely to the clapper box to make a rigid tool support. On the return stroke a slight frictional drag of the tool on the work lifts the block out of the clapper box a sufficient amount preventing the tool cutting edge from dragging and consequent wear. The work surface is also prevented from any damage due to dragging.

Specification of a Shaper

The size of a shaper is specified by the maximum length of stroke or cut it can make. Usually the size of shaper ranges from 175 to 900 mm. Besides the length of stroke, other particulars, such as the type of drive (belt drive or individual motor drive), floor space required, weight of the machine, cutting to return stroke ratio, number and amount of feed, power input etc. are also sometimes required for complete specification of a shaper.

Operations performed in a shaping machine

1. Different types of operations are performed in a shaping machine. They are broadly classified as Regular operations
2. Special operations

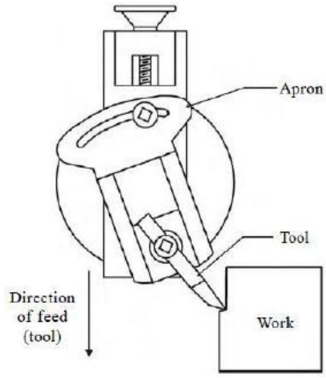


Machining horizontal surfaces

A shaper is mostly used to machine a flat, true surface on a work piece. Horizontal surfaces are machined by moving the work mounted on the machine table at a cross direction with respect to the ram movement. The clapper box can be set vertical or slightly inclined towards the uncut surface. This arrangement enables the tool to lift automatically during the return stroke. The tool will not drag on the machined surface

Machining vertical surfaces

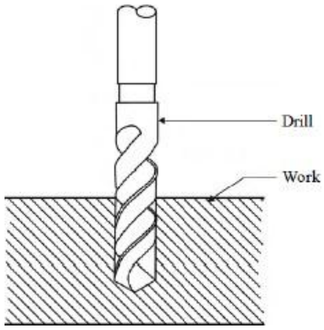
A vertical cut is made while machining the end of a work piece, squaring up a block or machining a shoulder. The feed is given to the tool by rotating the down feed screw of the vertical slide. The table is not moved vertically for this purpose. The apron is swiveled away from the vertical surface being machined as shown in the diagram.



Types of Drilling Operations

Though drilling is the primary operation performed in a drilling machine, a number of similar operations are also performed on holes using different tools. The different operations that can be performed in a drilling machine are:

1. Drilling
2. Reaming
3. Boring
4. Counterboring
5. Countersinking
6. Spotfacing
7. Tapping
8. Trepanning

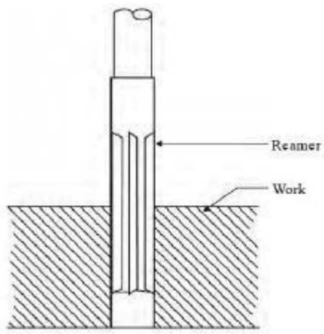


Drilling

Drilling is the operation of producing a cylindrical hole of required diameter and depth by removing metal by the rotating edge of a cutting tool called drill. Drilling is one of the simplest methods of producing a hole. Drilling does not produce an accurate hole in a work piece. The internal surface of the hole generated by drilling becomes rough and the hole is always slightly oversize due to vibration of the spindle and the drill. A hole made by a drill of size 12mm will measure approximately up to 12.125mm and by a drill ofsize22mm will measure up to22.5mm.

Reaming

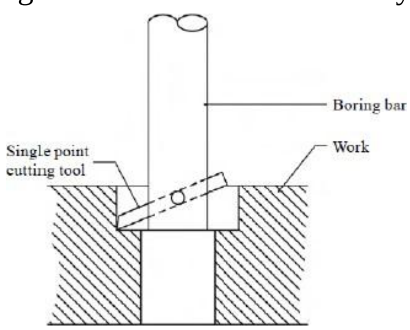
The size of hole made by drilling may not be accurate and the internal surface will not be smooth. Reaming is an accurate way of sizing and finishing a hole which has been previously drilled by a multi-point cutting tool known as reamer. The surface obtained by reaming will be smoother and the size accurate. The speed of the spindle is made half that of drilling. Reaming removes very small amount of metal (appr ox 0.375mm). In order to finish a hole and bring it to the accurate size, the hole is drilled slighly undersize.



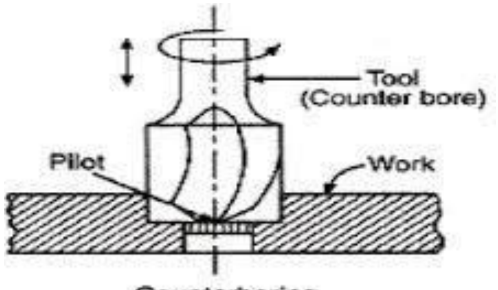
Boring

Boring is the operation enlarging the diameter of the previously made hole. Itis done for the following reasons.

1. To enlarge a hole by means of an adjustable cutting tool. This is done when a suitable sized drill is not available or the hole diameter is so large that is cannot be ordinarily drilled.
2. To finish a hole accurately and bring it to the required size
3. To machine the internal surface of the hole already produced in casting
4. To correct out of roundness of the hole



To correct the location of the hole as the boring tool follows independent path with respect to the hole. Boring tool is a tool with only one cutting edge. The tool is held in a boring bar which has a taper shank to fit in to the spindle or a socket. For perfectly finishing a hole, the job is drilled undersize slightly. Boring operation in some precise drilling machine is performed to enlarge the holes to an accuracy of 0.00125mm. The spindle speed during boring should be adjusted to be lesser than that of reaming

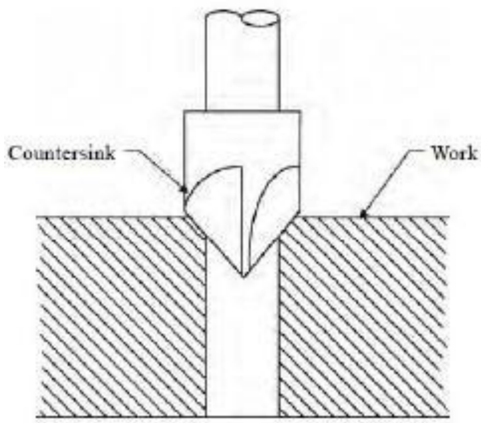


Counter boring

Counter boring is the operation of enlarging the end of the hole cylindrically. The enlarged hole forms a square shoulder with the original hole. This is necessary in some cases to accommodate the heads of bolts, studs and pins. The tool used for counter boring is known as counter bore. The counter bores are made with cutting edges which may be straight or spiral. The cutting speed for Counter boring is at least 25% lesser than that of drilling

Countersinking

Countersinking is the operation of making a cone shaped enlargement at the end of the hole. The included angle of the conical surface may be in the range of 60° to 90°. It is used to provide recess for a flat headed screw or a counter sunk rivet fitted into the hole. The tool used for counter sinking is known as a countersink. It has multiple cutting edges on its conical surface. The cutting speed for counter sinking is 25% lesser than that of drilling.



Calculation of the machining time for shaping

L = length of stroke;

$L = l + l_a + l_u$

Time for the working stroke $t_A = \frac{L}{V_A} = \frac{Length\ of\ stroke(m)}{Cutting\ Spee(\frac{m}{min})}$

Time for the working stroke $t_R = \frac{L}{V_R} = \frac{Length\ of\ stroke\ (m)}{Return\ spped(\frac{m}{min})}$

V_R - Return speed m/min

V_A - Cutting speed m/min

Time cycle for t = time for working stroke + time for cutting stroke

$t = t_A + t_R$

For shaping a work piece, a certain number of cycles is necessary, depending on the amount of feed and the width of the work piece, plus allowance. The shaping width consists of the width of the work piece plus an allowance of 5mm on both sides of the work piece.

Shaping width B=width of work piece+ allowance from both sides (B=b+2× 5mm)

If the shaping width is divided by the feed, the required cycles are obtained.

Thus number of required cycles $Z = \frac{\text{Shaping width}}{\text{Feed}} = \frac{B}{s}$

The machining time t_m = Number of cycle × time per cycle

$t_m = Z \times t$

Introduction to Gears and Machines

Bevel Gear

Bevel gears are used mostly in situations that require power to be transmitted at right angles (or applications that are not parallel). Bevel gears can have different angles of application but tend to be 90°

Helical Gears

Helical gears are very similar to spur gears except the teeth are not perpendicular to the face. The teeth are at an angle to the face giving helical gears more tooth contact in the same area. Helical gears can also be used on non-parallel shafts to transmit motion. Helical gears tend to run quieter and smoother than spur gears due to the increased number of teeth in constant contact at any one period of time

Herringbone Gear

Herringbone gears resemble two helical gears that have been placed side by side. They are often referred to as "double helicals". One benefit of herringbone gears is that it helps to avoid issues related to side thrust created with the use of helical gears

Worm Gears

Worm gears are used to transmit power at 90° and where high reductions are required. The worm resembles a thread that rides in concaved or helical teeth

Spur Gears

Spur gears are by far the most common type of gear and with the exceptions of the "cog" the type of gear that has been around the longest. Spur gears have teeth that run perpendicular to the face of the gear

Internal Gears

Internal gears typically resemble inverted spur gears but are occasionally cut as helical gears

Racks Gear

A rack is basically a straight gear used to transmit power and motion in a linear movement.

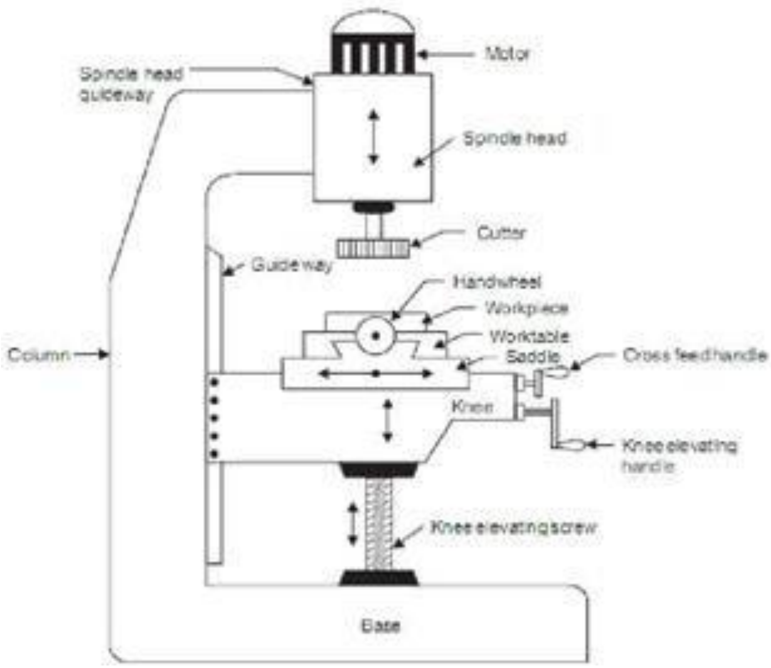
Face Gears

Face gears transmit power at (usually) right angles in a circular motion. Face gears are not very common in industrial application

Sprockets Gear

Sprockets are used to run chains or belts. They are typically used in conveyor systems

Introduction - Milling Machine



Milling Machine

Milling is the process of removing metal by feeding the work piece through a rotating multipoint cutter. Milling machine can be used for machining flat surfaces, complex and irregular areas, surface of revolution, external and internal threads, gear cutting, helical surface of cross sections.

1. Base	2.Column	3.Knee
4.Saddle	5.Table	6.Spindle
7.Arbor		

Base

It is the foundation of the machine and is that part upon which all parts are mounted. It gives the machine rigidity and strength.

Column

It is the main supporting frame. The motor and other driving mechanisms are contained within it.

Knee

The knee projects from the column and slides up and down on its face. It supports the saddle and table and partially supported by the elevating screw which adjusts its height.

Saddle

The saddle supports and carries the table and is adjustable transversely on ways on top of the knee. It is provided with graduations for exact movement and operated by power or hand.

Table

The table rests on ways on the saddle and travels longitudinally in a horizontal plane. It supports the works piece, fixtures and all other equipments.

Spindle

The spindle obtains its power from the motor through motors. Cutters are mounted directly in the spindle nose.

Arbor

The arbor is an accurately machined shaft for holding and driving the arbor cutter. It is tapered at one end to fit the spindle nose and two slots to fit the nose keys for locating and driving it

Introduction - Gear Hobbing Machining

Gear Hobbing

Gear Hobbing is a technique that is employed to create gear teeth configurations that are ideal for use in a wide range of machinery components. In cases where the gear hobbing takes place in a mass producing environment, gear hobbing is accomplished through the use of precision gear hobbing machines that ensure that the cut of each tooth on each gear produced meets the specifications set by the producer.

Generally, a gear hobbing machine will make use of a series of customized bits that help to create the specific types of cutting and shaping necessary to create gears those possess exactly the right pitch and circle to work in various types of equipment. A customized bit is used for a particular size and type of gear hobbing, which helps to ensure that the cuts that are made into the blank surface of the circle of metal are relatively smooth and uniform.

CNC Machines

G00 – Positioning at rapid speed; Milling and Turning

G01 – Linear interpolation (machining a straight line); Milling and Turning

G02 – Circular interpolation clockwise (machining arcs); Milling and Turning

G03 – Circular interpolation, counter clockwise; Milling and Turning

G04 – Milling and Turning, Dwell

G09 – Milling and Turning, Exact stop

G10 – Setting offsets in the program; Milling and Turning

G12 – Circular pocket milling, clockwise; Milling

G13 – Circular pocket milling, counterclockwise; Milling

G17 – X – Y plane for arc machining; Milling and Turning with live tooling

G18 – Z – X plane for arc machining; Milling and Turning with live tooling

G19 – Z – Y plane for arc machining; Milling and Turning with live tooling

G20 – Inch units; Milling and Turning

G21 – Metric units; Milling and Turning

G27 – Reference return check; Milling and Turning

G28 – Automatic return through reference point; Milling and Turning

G29 – Move to location through reference point; Milling and Turning (slightly different for each machine)

G31 – Skip function; Milling and Turning

G32 – Thread cutting; Turning

G33 – Thread cutting; Milling

G40 – Cancel diameter offset; Milling. Cancel tool nose offset; Turning

G41 – Cutter compensation left; Milling. Tool nose radius compensation left; Turning

G42 – Cutter compensation right; Milling. Tool nose radius compensation right; Turning

G43 – Tool length compensation; Milling

G44 – Tool length compensation cancel; Milling (sometimes G49)

G50 – Set coordinate system and maximum RPM; Turning

G52 – Local coordinate system setting; Milling and Turning

G53 – Machine coordinate system setting; Milling and Turning

G54~G59 – Work piece coordinate system settings #1 t0 #6; Milling and Turning

G61 – Exact stop check; Milling and Turning

G65 – Custom macro call; Milling and Turning

G70 – Finish cycle; Turning

G71 – Rough turning cycle; Turning

G72 – Rough facing cycle; Turning

G73 – Irregular rough turning cycle; Turning

G73 – Chip break drilling cycle; Milling

G74 – Left hand tapping; Milling

G74 – Face grooving or chip break drilling; Turning

G75 – OD groove pecking; Turning

G76 – Fine boring cycle; Milling

G76 – Threading cycle; Turning

G80 – Cancel cycles; Milling and Turning

G81 – Drill cycle; Milling and Turning

G82 – Drill cycle with dwell; Milling

G83 – Peck drilling cycle; Milling

G84 – Tapping cycle; Milling and Turning

G85 – Bore in, bore out; Milling and Turning

G86 – Bore in, rapid out; Milling and Turning

G87 – Back boring cycle; Milling

G90 – Absolute programming

G91 – Incremental programming

G92 – Reposition origin point; Milling

G92 – Thread cutting cycle; Turning

G94 – Per minute feed; Milling

G95 – Per revolution feed; Milling

G96 – Constant surface speed control; Turning

G97 – Constant surface speed cancel

G98 – Per minute feed; Turning

G99 – Per revolution feed; Turning CNC M Codes

M00 – Program stop; Milling and Turning

M01 – Optional program stop; Turning and Milling

M02 – Program end; Turning and Milling

M03 – Spindle on clockwise; Turning and Milling

M04 – Spindle on counterclockwise; Turning and Milling

M05 – Spindle off; Turning and Milling

M06 – Tool change; Milling

M08 – Coolant on; Turning and Milling

M09 – Coolant off; Turning and Milling

M10 – Chuck or rotary table clamp; Turning and Milling

M11 – Chuck or rotary table clamp off; Turning and Milling

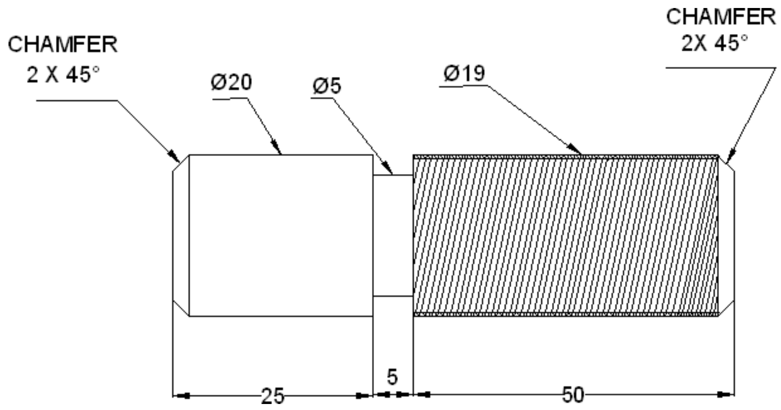
M19 – Orient spindle; Turning and Milling

M30 – Program end, return to start; Turning and Milling

M97 – Local sub-routine call; Turning and Milling

M98 – Sub-program call; Turning and Milling

M99 – End of sub program; Turning and Milling



External Thread Cutting

All Dimensions are in mm

Calculation:

Depth of the cut = 0.64 x pitch x 0.254

= 0.64 x 10 x 0.254

= 1.6256 ≅ 2mm.

5 mm reach to 100 divisions

$\therefore 2\,mm reaches = \frac{100 \times 2}{5}$ = 40 divisions

(Driver teeth) / (Driven teeth) = (Pitch of the work)/ (Pitch of the lead screw)

= (10 TPI) / (4 TPI) = 100 / 40

Operation Plan

S.No.	***Operations***	***Tools***
1.	Turning and Facing	Single point turning tool
2.	Drilling	Drill bit
3.	Boring	Boring tool
4.	External thread cutting	Single point 'v' shaped tool
Measuring Instruments: Vernier Caliper and Steel rule.		

Ex No : 1 ***Date :***

External Thread Cutting

Aim:

To perform external thread cutting on the given work piece by using a lathe.

Materials Supplied:

Mild Steel Rod Ø25x 80 mm

Tool Material:

High Speed Steel (H.S.S)

Equipment & Tools required:

1. Lathe
2. Parting tool
3. 3 jaw chuck & Chuck key
4. Vernier caliper.
5. Steel rule.
6. Cleaning brush.
7. Single point V shaped high speed thread cutting tool

Procedure:

1. The work piece is held in the three jaw chuck
2. The single point turning tool is fixed in the tool post and the machine is switched on.
3. The facing and turning operations are carried out up to the dimensions required.
4. The turning tool is replaced by a parting tool and an undercut is made at the end of the thread length for tool relieving purpose.
5. Then the Norton gear mechanism is set for gear pitch
6. Tool zero is set and initial depth of cut 0.5mm is given.
7. When cutting a thread (RHS), the carriage must move towards the head stock.
8. The job moves always in the anticlockwise direction when it is viewed from the tail stock.
9. As previously mentioned the direction at which the carriage moves in relation to the lathe head stock is controlled by means of the tumbler gear feed reversing mechanism.
10. The depth of the cut is gradually increased up to the required root diameter is obtained.
11. Both ends of the work piece are chamfered by using turning tool
12. The machine is switched off and the chips are removed by cleaning brush.

Calculation of machining time for External Threading:

Threads can be cut on lathe with the help of a single point cutting tool. The time for cutting threads is calculated as follows:

$$Time\ T = \frac{Length\ of\ cut}{feed\ per\ revolution\ x\ r.p.m}$$

where

Feed/rev. = lead of thread

Full depth of the thread cannot be obtained in a single cut when cutting threads by single point cutting tool on a lathe. A number of cuts have to be taken to get the full depth.

The number of cuts for external threads may be calculated with the help of following relation:

Number of cuts = 25 / Thread per cm

Result:

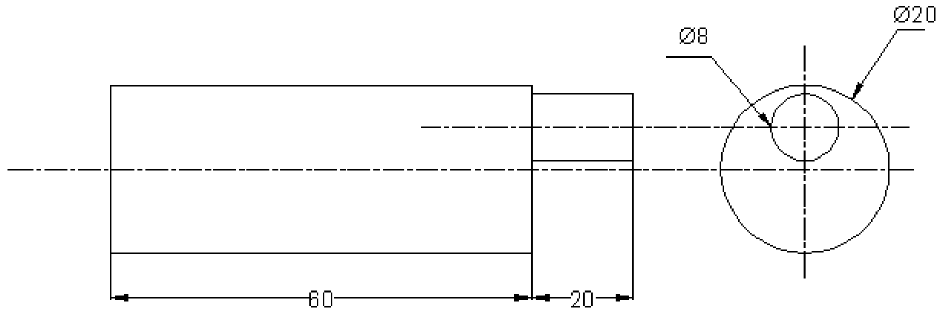
External thread cutting operation was performed on the given job as per the dimensions.

Inference:

I have gained hands on experience on the mechanism of External thread cutting.

Application:

- Bolts and nuts used in the machinery
- Conveyers
- Taps, couplings and pipes



Eccentric Turning

All Dimensions are in mm

Operation Plan

S.No.	***Operations***	***Tools***
1.	Marking	Scriber, Divider and dot punch
2.	Turning and Facing	Single point turning tool
3.	Eccentric turning	Single point turning tool
4.	Finishing	Smooth file
Measuring Instruments: Vernier Caliper and Steel rule.		

Ex No : 2 ***Date :***

Eccentric Turning

Aim:

To perform an eccentric turning operation on the given work piece by using a lathe.

Material Supplied:

Mild Steel Rod Ø20 x 80 mm

Tool Material:

High Speed Steel (H.S.S)

Equipment & Tools required:

1. Lathe
2. 4 jaw chuck & Chuck key
3. Single point turning tool
3. Vernier Caliper
4. Divider
5. Steel rule
6. Smooth file
7. Scriber
8. Dot punch
9. Cleaning brush

Procedure:

1. Eccentric marking is done on the given job using scriber, dot punch and divider.
2. The marked job is held on the chuck. The center of the job is adjusted by changing the position of the jaws on the four jaw chuck.
3. The single point turning tool is fixed in the tool post and the machine is switched on.
4. The turning and facing operations are performed for the required length. Now the eccentric center coincides with the center of the lathe by changing the position of the jaw.
5. Calculation of machining time for turning and facing
6. The depth of cut is given to the job in small rate.
7. The job is finished by Filing by using Smooth file
8. The machine is switched off and the dimensions are checked using vernier caliper and steel rule.
9. The chips are removed by cleaning brush.

$$\text{Machining time} = \frac{\text{Turning length}}{\text{Feed/min}}$$

$$\mathbf{t_m} = \frac{L}{s \times n}$$

Longitudinal turning

L = length of the work piece (L_1) +starting allowance (l_a) +allowance after turning(l_u)

L = $L_1 + l_a + l_u$

Where

T_m	–	Machining time	;	L	–	Turning length;
5	–	feed in mm / rev	;	n	–	revolutions per min
s'	**=**	**s**×n		s'	–	feed per min

Facing

The turning length L corresponds with the radius plus starting allowance

L = $r + l_a$

Result:

The eccentric turning is made on the given job as per the dimensions.

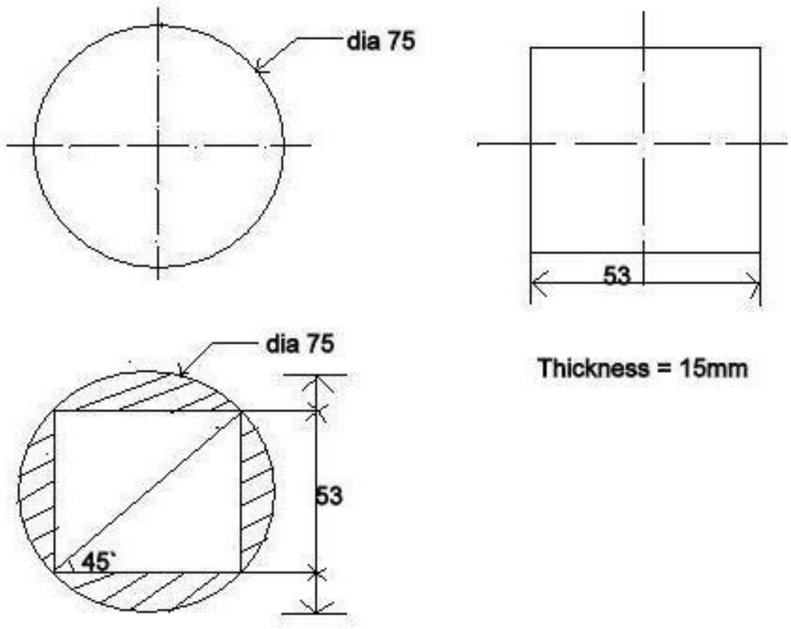
Inference

I have gained hands on experience about machining a rod with eccentricity.

Applications

Eccentric turning has the following applications:

1. In Automobile engineering:
   a. For crankshaft turning.
   b. For turning eccentric rods for opening and closing of inlet and exhaust valves.
2. Marine steam engines
3. Aircraft bushings.
4. Self-lubricating bushings.
5. Corrosion resistance bushings.
6. Alloy bushings.
7. Metric bushings.



Square Head Shaping

All Dimensions are in mm

Operation Plan

S.No.	***Operations***	***Tools***
1.	Marking	Scriber, Dot punch and hammer
2.	Mounting of work piece	Machine vice
3.	Successive shaping of longitudinal sides	Pointed Shaping tool
4.	Reclamping	Machine vice
5.	shaping of other faces	Pointed Shaping tool
Measuring Instruments: Vernier Caliper, Try square and Steel rule.		

Calculation:

Side of the square	=	$D \times \sin 45^o$	=	$75 \times \sin 45^o$	=	53
Dia of the circle	=	$a/\sin 45^o$	=	$53/\sin 45^o$	=	74.95

Ex No : 3 ***Date :***

Square Head Shaping

Aim:

To machine a square shape for the given dimension of the cylindrical lock using the shaper machine

Materials Supplied:

Cast iron Ø75 x 15 mm

Tool Material:

High Speed Steel (H.S.S)

Equipment & Tools required:

1. Shaper	2. Machine vice	3.Hammer
4. Try square	5. Scriber	6.Steel rule
7. Dot punch	8. Vernier caliper	9.Cleaning brush

Procedure:

1. Mark a square of given dimension on one of the faces with the help of the steel rule and scriber.
2. Then make punches on the boundary of the square using dot punch and hammer.
3. The work piece is fitted on the table such that one of the sides of the square punched is parallel to the top edge of the machine.
4. The shaping machine is switched on and the surface of the work piece is machined by increasing the depth of cut by lowering the tool post.
5. Continue till all the faces become a square.
6. After finishing the job, the shaping machine is switched off and the job is removed from the table and the dimensions are checked using vernier caliper and try square.
7. The chips are removed by cleaning brush.

Operation Plan

Calculation of the machining time for shaping

L = length of stroke;

L = $l + l_a + l_u$

Time for the working stroke t_A = $\frac{L}{V_A}$ = $\frac{Length\ of\ stroke(m)}{Cutting\ Spee(\frac{m}{min})}$

Time for the working stroke t_R = $\frac{L}{V_R}$ = $\frac{Length\ of\ stroke\ (m)}{Return\ spped\ (\frac{m}{min})}$

V_R – Return speed m/min

V_A – Cutting speed m/min

Time cycle for

t = time for working stroke + time for cutting stroke

$t = t_A + t_R$

For shaping a work piece, a certain number of cycles is necessary, depending on the amount of feed and the width of the work piece, plus allowance. The shaping width consists of the width of the work piece plus an allowance of 5mm on both sides of the work piece.

Shaping width

B = width of work piece+ allowance from both sides (B=b+2× 5mm)

If the shaping width is divided by the feed, the required cycles are obtained.

Thus number of required cycles Z = $\frac{\text{Shaping width}}{\text{Feed}} = \frac{B}{s}$

The machining time t_m = Number of cycle × time per cycle

t_m = Z× t

Result:

The given block is machined to a square block of given dimension.

Inference:

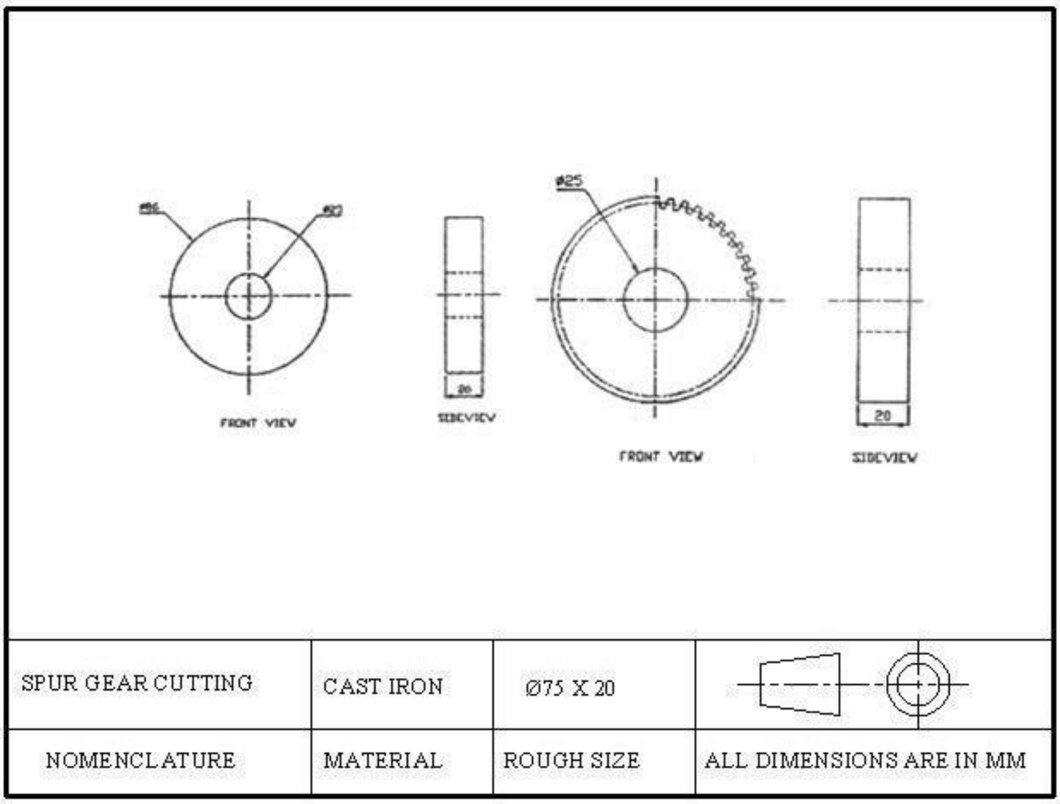
I have gained hands on experience in the working of shaping process and the quick return mechanism of shaper.

Application:

1. Metric bolts - Available in Hex/Standard/Flange type
2. Shoulder bolts - Allow slide or pivot
3. Hex tap bolts - Trimmed hex head with no flange
4. Square neck bolts - Square or domed top with square under head

Operation Plan

S.No	***Operation***	***Tools***
1.	***Turning and Facing***	***Single point cutting tool***
2.	***Drilling***	***Φ 15 mm drill bit***
3.	***Boring***	***Boring tool***
4.	***Gear cutting***	***3mm module cutter***
Measuring Instruments : Vernier Caliper and Steel Rule		



Ex No : 4 **Date :**

Spur Gear Cutting in Milling Machine

Aim:

To perform spur gear cutting on the given work piece using vertical milling machine.

Introduction:

Spur gears or straight-cut gears are the simplest type of gear. They consist of a cylinder or disk with the teeth projecting radially, and although they are not straight-sided in form (they are usually of special form to achieve constant drive ratio, mainly involute), the edge of each tooth is straight and aligned parallel to the axis of rotation. These gears can be meshed together correctly only if they are fitted to parallel shafts.

Material used:

Cast iron blank

Tools required:

1. Horizontal Milling machine
2. Vernier caliper
3. Vice
4. Milling Tools
5. Mandrel

Calculation:

Z = No. of teeth = 20

m = module = 3 mm

pitch circle diameter = M x Z

= 3 x 20 = 60 mm

Blank Diameter = 2m + pcd

= 2 x 3 + 60 = 66 mm

Tooth Depth = m(addendum + Dedendum)

= 3(1+1.25)

= 6.75mm

= 2.25 × 2 = 4.5 mm

Indexing Calculation = $\frac{40}{Z} = \frac{40}{20}$ = 2 revolution

Procedure:

1. Turning and facing operations are performed on the given gear blank using single point cutting tool in the Lathe.
2. A hole is drilled on the gear blank using 16 mm dia drill bit and boring operation is carried out using boring tool.
3. The dividing head and the tail stock are bolted on the machine table. Their axis must be set parallel to the machine table.
4. The gear blank is held between the dividing head and tailstock using a mandrel. The mandrel is connected with the spindle of dividing head by a carrier and catch plate.
5. The cutter is mounted on the arbor. The cutter is centered accurately with the gear blank. Set the speed and feed for machining.
6. For giving depth of cut, the table is raised till the periphery of the gear blank just touches the cutter.
7. The micrometer dial of vertical feed screw is set to zero in this position.
8. Then the table is raised further to give the required depth of cut.
9. The machine is started and feed is given to the table to cut the first groove of the blank.
10. After the cut, the table is brought back to the starting position. Then the gear blank is indexed for the next tooth space.
11. This is continued till all the gear teeth are cut.

Inference:

- I am able to understand the various types of gear.
- I am able to understand the principle of gear making.
- I am able to understand the gear tooth calculation.
- These gears have straight teeth and must be mounted on parallel shafts for their teeth to mesh with those of other gears. Spur gears can be used to increase or decrease the torque,

Applications:

- Used in mechanical clocks to adjust the relative speeds of the second, minute and hour hands.
- In hand-held eggbeaters, spur gears are used to increase the speed of the eggbeater so it can be used more effectively.
- Spur gears are used in washing machines, blenders, clothes dryers, construction equipment, fuel pumps and mills.

Result:

The given work piece as is subjected to gear generating operation to become a finished work piece

Ex No: 5 ***Date:***

Helical Gear Cutting In Milling Machine

Aim:

To perform Helical Gear Cutting on the given work piece using milling machine.

Introduction:

Helical or "dry fixed" gears offer a refinement over spur gears. The leading edges of the teeth are not parallel to the axis of rotation, but are set at an angle. Since the gear is curved, this angling causes the tooth shape to be a segment of a helix. Helical gears can be meshed in parallel or crossed orientations.

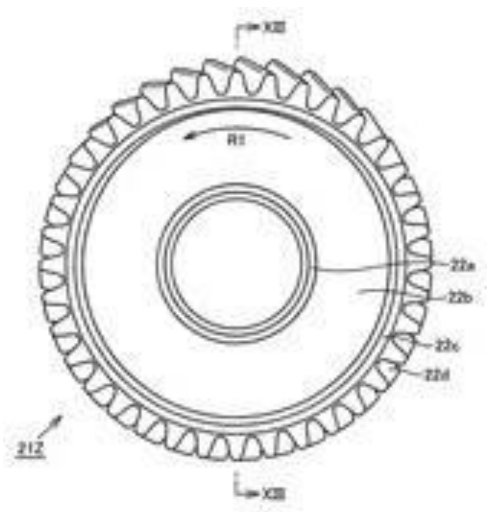
Material used

Cast iron

Tools required

1. Vertical Milling machine
2. Vernier caliper
3. Holding Materials
4. Milling Tools

Diagram



Helical Gear Formulas

Parameter	***Formula***
Normal module m_n	$m_n = (2a / (z_1 + z_2)).\ \text{Cos}\beta$
Transverse module m_t	$m_t = (2a / (z_1 + z_2))$
Bottom clearance c	$c = 0.25\ m_n$
Tooth depth h	$h = 2.25\ m_n$
Transverse circular pitch p_t	$p_t = \pi.m_t$
Normal circular pitch p_n	$p_n = \pi.m_n$

Calculation:

Z = No. of teeth = 20

m = module = 3 mm

pitch circle diameter = M x Z

= 3 x 20 = 60 mm

Blank Diameter = 2m + pcd

= 2 x 3 + 60 = 66 mm

Tooth Depth = m(addendum + Dedendum)

= 3(1+1.25)

= 6.75mm

= 2.25 × 2 = 4.5 mm

Indexing Calculation = $\frac{40}{Z} = \frac{40}{20}$ = 2 revolution

Operation Plan

S. No	***Operation***	***Tools***
1.	***Turning and Facing***	***Single point cutting tool***
2.	***Drilling***	***Φ 15 mm drill bit***
3.	***Boring***	***Boring tool***
4.	***Chamfering***	***"V" tool***
5.	***Gear cutting***	***3 mm module cutter***
Measuring Instruments : Digital Vernier Caliper, Bevel Protractor and Steel Rule		

Procedure:

1. Turning and facing operations are performed on the given gear blank using single point cutting tool in the Lathe.
2. A hole is drilled on the gear blank using 15 mm dia drill bit and boring operation is carried out using boring tool.
3. The dividing head and the tail stock are bolted on the machine table.
4. Their axis must be set parallel to the machine table.
5. The gear blank is held between the dividing head and tailstock using a mandrel. The mandrel is connected with the spindle of dividing head by a carrier and catch plate.
6. The cutter is mounted on the arbor. The cutter is centered accurately with the gear blank.
7. Set the speed and feed for machining.
8. For giving depth of cut, the table is raised till the periphery of the gear blank just touches the cutter.
9. The micrometer dial of vertical feed screw is set to zero in this position.
10. Then the table is raised further to give the required depth of cut.
11. The machine is started and feed is given to the table to cut the first groove of the blank.
12. After the cut, the table is brought back to the starting position.
13. Then the gear blank is indexed for the next tooth space.
14. This is continued till all the gear teeth are cut.

Inference:

I am able understand the different formulas to obtain helical gear of desired pitch diameter, lead etc.

Applications:

1. Fertilizer industry
2. Railway industry
3. Printing industry
4. Earth moving industry

Result:

The given work piece as is subjected to gear generating operation to become a finished work piece

Operation Plan

S. No	Operation	Tools
1.	Profile Milling	End mill cutter
2.	Finishing	Smooth file
Measuring Instruments : Vernier Caliper and Steel Rule		

Ex No : 6 ***Date :***

Contour Milling Using Vertical Milling Machine

Aim:

To study the contour milling on the given work piece using vertical milling machine.

Introduction:

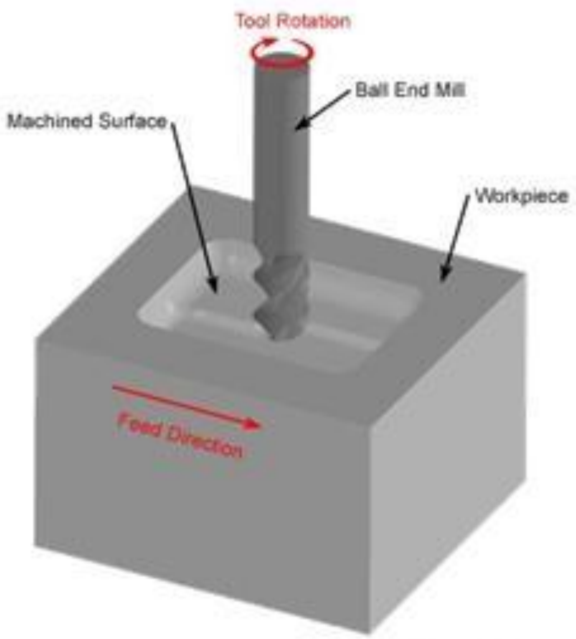
Machining of Irregular contour on the surface of the given workpiece.

Material used

1. Cast Iron

Surface contouring

The end mill, which is used in surface contouring has a hemispherical end and is called ball-end mill. The ball-end mill is fed back and forth across the work piece along a curvilinear path at close intervals to produce complex three-dimensional surfaces. Similar to profile milling, surface contouring require relatively simple cutting tool but advanced, usually computer-controlled feed control system.



Inference:

I am able to understand the various milling process.

I am able to understand the influence of rotating speed of tool.

Applications:

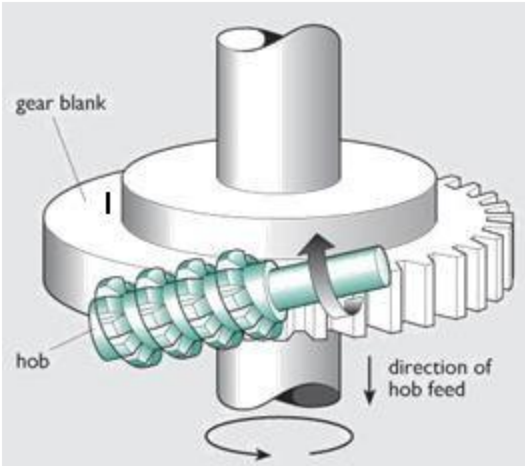
Industrial applications

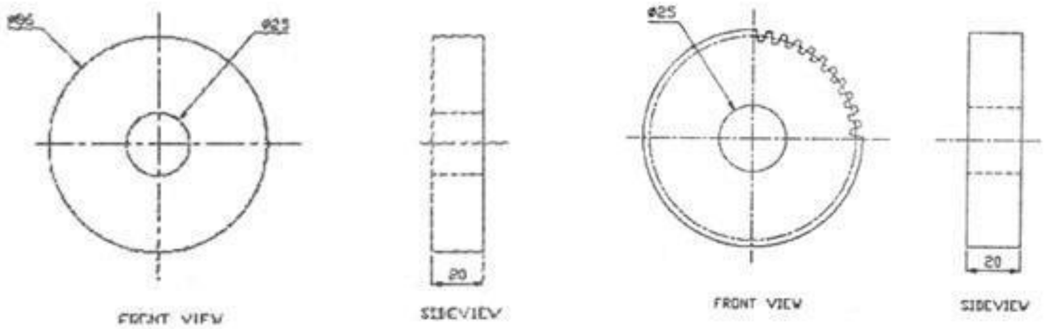
Automobile and Aeronautical applications.

Result:

Thus the contour milling using vertical milling machine is studied.

Gear Hobbing





SPUR GEAR CUTTING	CAST IRON	Ø75 X 20	
NOMENCLATURE	MATERIAL	ROUGH SIZE	ALL DIMENSIONS ARE IN MM

Ex No: 7 ***Date:***

Gear Generation In Hobbing Machine

Introduction:

Hobbing is a machining process for gear cutting, cutting splines, and cutting sprockets on a hobbing machine, which is a special type of milling machine. The teeth or splines are progressively cut into the work piece by a series of cuts made by a cutting tool called a hob. Compared to other gear forming processes it is relatively inexpensive but still quite accurate, thus it is used for a broad range of parts and quantities.

Aim:

To machine a Spur Gear on the given work piece using a gear Hobbing machine.

Materials required:

Cast iron Blank

Tools required:

1. Gear Hobbing machine
2. Hob
3. Gear tooth vernier
4. Spanners

Calculation:

Z = No. of teeth = 79

m = module = 1 mm

PCD = m x z

= $1 \times 79 = 79\, mm$

Blank Diameter = 2m + PCD

= $2 \times 1 + 79 = 81$mm

Tooth Depth = m (addendum + Dedendum)

= $1(1 + 1.25)$ = 2.25 mm

Operation Plan

S.No	Operation	Tools
1.	Turning and Facing	Single point cutting tool
2.	Drilling	Φ 19 mm drill bit
3.	Boring	Boring tool
4.	Chamfering	"V" tool
5.	Gear cutting	1 mm module hobb cutter
Measuring Instruments : Digital Vernier Caliper and Steel Rule		

Procedure:

1. Turning and facing operations are performed on the given gear blank using single point cutting tool in the Lathe.
2. A hole is drilled on the gear blank using 19 mm dia drill bit and boring operation is carried out using boring tool.
3. The gear blank is held firmly on the spindle of the gear hobbing machine.
4. The Hob is set at an angle to the hob helix angle for cutting spur gear.
5. The change gears are set for the desired speed and feed of work piece and Hob
6. The machine is switched on.
7. The work piece and Hob are allowed to rotate at the desired speed.
8. The hob or work piece is given full depth of cut equals to the tooth depth.
9. The cutter is given feed for the full width of the work.
10. After machining all gear teeth on the blank the machine is switched off.
11. The gear teeth are checked using a gear tooth Vernier.

Inference:

Compared to other gear forming processes it is relatively inexpensive but still quite accurate.

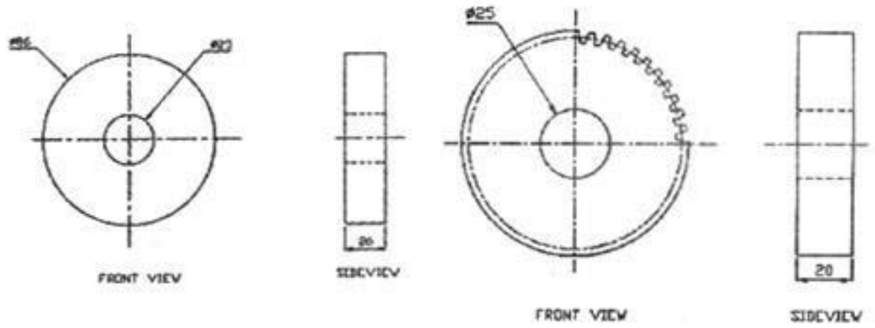
I am able to understand the different types of gear Hobbs.

Applications:

1. Cycloid gears
2. Helical gears
3. Involute gears
4. Ratchets
5. Splines
6. Sprockets
7. Spur gears
8. Worm gears

Result:

The given work piece as shown in fig (1) is subjected to gear generating operation to become a finished work piece as shown in fig (2) in gear Hobbing machine.



SPUR GEAR CUTTING	CAST IRON	Ø75 X 20	
NOMENCLATURE	MATERIAL	ROUGH SIZE	ALL DIMENSIONS ARE IN MM

Ex No: 8 ***Date:***

Gear Generation in Shaping Machine

Aim:

To machine a Spur Gear on the given work piece using a gear Hobbing machine.

Materials required:

Cast iron blank

Tools required:

1. Gear Shaping machine
2. Gear tooth Vernier
3. Spanners

Procedure:

1. Turning and facing operations are performed on the given gear blank using single point cutting tool in the Lathe.
2. A hole is drilled on the gear blank using 10 mm dia drill bit and boring operation is carried out using boring tool.
3. The gear blank is held firmly on the spindle of the gear shaping machine
4. The work piece is set at an angle to shaping tool angle for cutting spur gear.
5. The change gears are set for the desired speed of work piece and
6. The machine is switched on.
7. The work piece and Shaper are allowed to remove the metal at the desired speed.
8. The work piece is given full depth of cut equals to the tooth depth.
9. The cutter is given feed for the full width of the work.
10. After machining all gear teeth on the blank the machine is switched off.
11. The gear teeth are checked using a gear tooth Vernier.

Calculation:

Z = No. of teeth = 23

m = module = 2 mm

Blank Diameter = (Z + 2) m

= (23 + 2) * 2 = 50 mm

Tooth Depth = 2.25 m

= 2.25 × 2 = 4.5 mm

Indexing Calculation = $\frac{40}{Z} = \frac{40}{23} = 1\frac{17}{23}$

Operation Plan

S.No	***Operation***	***Tools***
1.	***Turning and Facing***	***Single point cutting tool***
2.	***Drilling***	***Φ 10 mm drill bit***
3.	***Boring***	***Boring tool***
4.	***Chamfering***	***"V" tool***
5.	***Gear cutting***	***2 mm module cutter***
Measuring Instruments : Digital Vernier Caliper and Steel Rule		

Inference:

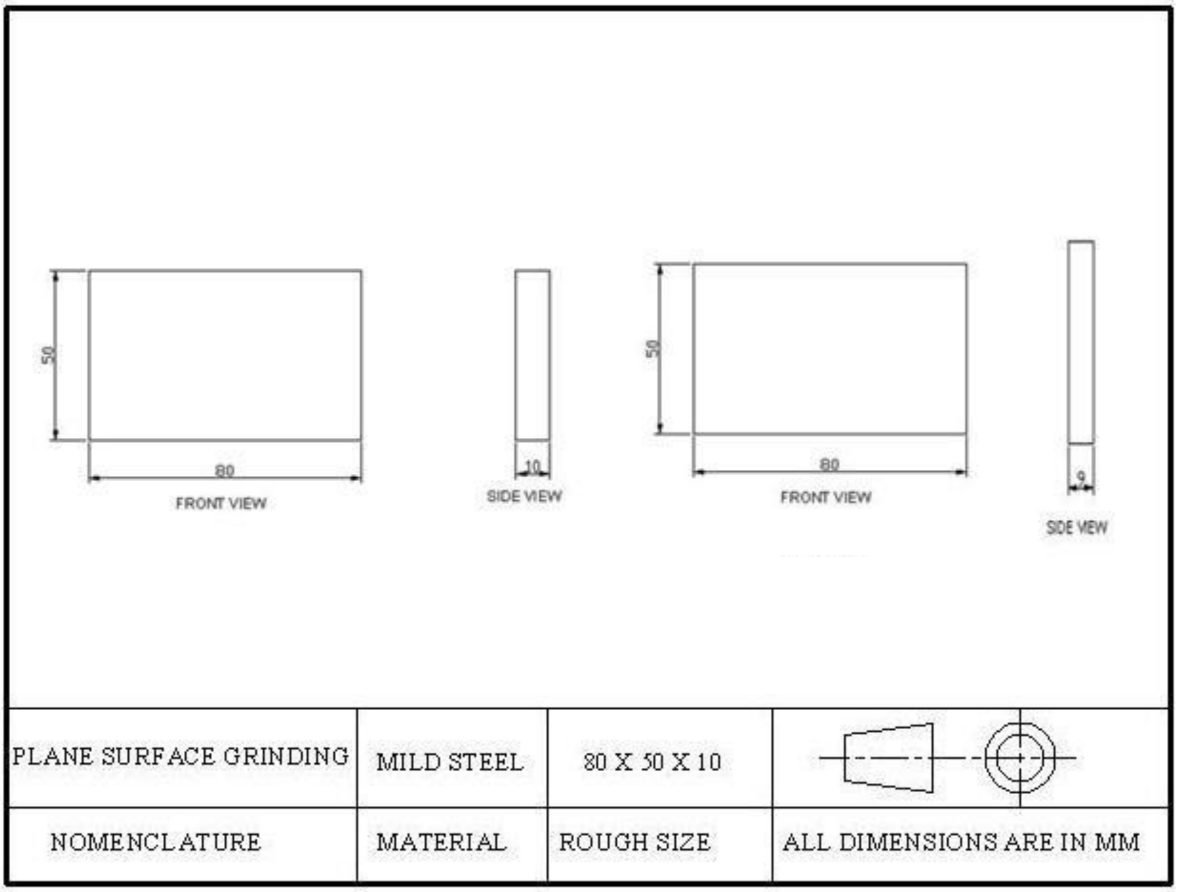
A shaper machine is a cutting machine that cuts a linear tool path using a linear relative motion between a single-point cutting tool and the piece of work. This type of machine is usually used to machine flat, straight surfaces, although it is also able to perform more complex tasks including the machining of dovetail slides, gear teeth and internal spline, keyways in the boss of either gears or pulleys and many other forms of work that take advantage of the machines linear relative motion.

Application:

1. Cycloid gears
2. Helical gears
3. Involute gears
4. Ratchets
5. Splines
6. Sprockets

Result:

The given work piece is subjected to gear generating operation to become a finished work piece in gear Shaping Machine.



Operation Plan

S.No	***Operation***	***Tools***
1.	***Surface grinding***	***Grinding wheel***
2.	***Deburring***	***Smooth file***
Measuring Instruments : Digital Vernier Caliper and Steel Rule		

Ex No: 9 ***Date:***

Plain Surface Grinding

Aim:

To perform a Plain surface grinding operation on the given work piece for the given dimensions.

Principle:

The principle involved in this process is to make flat surface on the given work piece. The cutter is moved perpendicular to the work piece and the grinding is done.

Requirements:

1. Grinding Machine
2. Work Piece 100x50x6 mm MS Plate
3. Grinding Wheel

Procedure:

1. At first work piece is placed in the magnetic chuck.
2. The work piece should be light weight so that it cannot be removed from the magnetic chuck easily.
3. Various arrangements regarding the positions of work piece is done.
4. Grinding wheel and grinding spindle are kept in position with the work piece.
5. Before switching on the motor, necessary steps should be taken.
6. For proper grinding process wheel speed, work speed, transverse speed of the wheel in feed, area of contact is to be noted.
7. While running the area of contact is adjusted accordingly to the spindle in order to remove the surface.
8. It is done slowly to remove the materials on the both sides.
9. In surface grinding the stock removal rate is given by Q = bdy Where d = depth of cut (m) b = width of cut (m) y = work velocity (m/s) q = rate of stroke (m^3/s)

Inference:

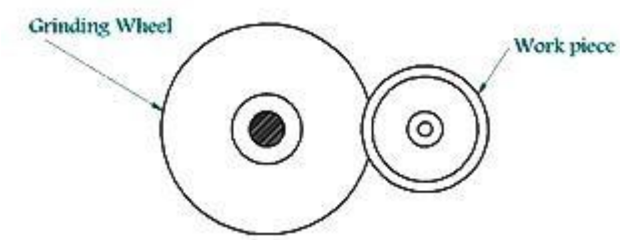
1. Surface grinding is used to produce a smooth finish on flat surfaces.
2. The typical precision of a surface grinder depends on the type and usage, however ±0.002 mm (±0.0001 in) should be achievable on most surface grinders.
3. Surface grinding or grinding flat surfaces, is characterized by a large contact area of the wheel with the work piece, as opposed to cylindrical grinding where a relatively small area of contact is present. As a result, the force of each abrasive grain against the work piece is smaller than that applied to each grain in cylindrical grinding. In surface grinding the grinding wheel should be generally softer in grade and wider in structure than for cylindrical grinding.

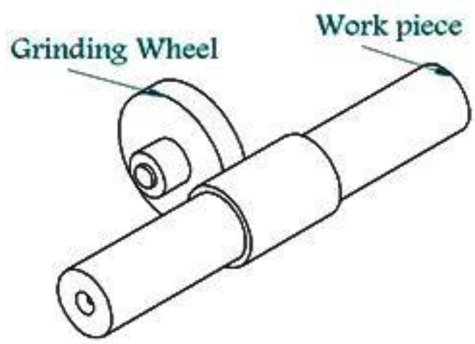
Applications:

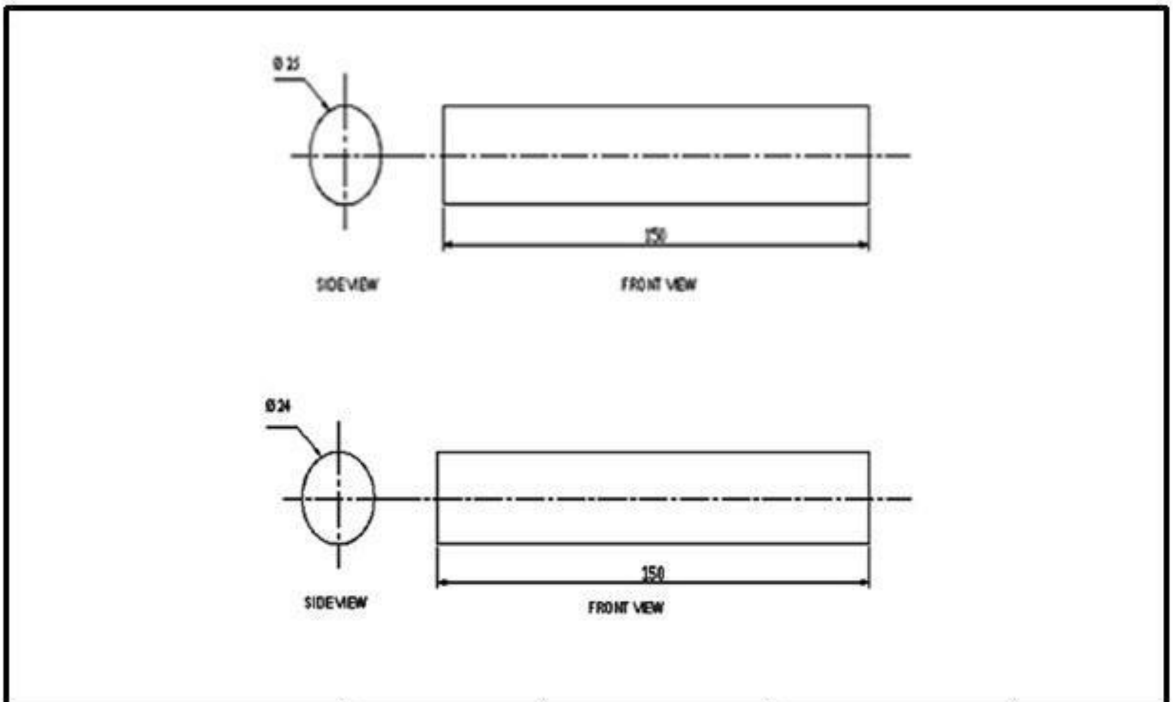
1. Polishing
2. Buffing
3. Lapping

Result:

Thus the surface grinding is done for the given dimensions.







CYLINDRICAL GRINDING	MILD STEEL	Ø20 X 100	
NOMENCLATURE	MATERIAL	ROUGH SIZE	ALL DIMENSIONS ARE IN MM

Operation Plan

S.No	***Operation***	***Tools***
1.	***Facing***	***Single point cutting tool***
2.	***Cylindrical grinding***	***Grinding wheel***
Measuring Instruments : Digital Vernier Caliper and Steel Rule		

Ex No: 10 **Date:**

Cylindrical Grinding

Aim:

To grind the cylindrical surface of the given materials as per the given dimensions

Requirements:

1. Grinding Machine
2. Grinding Wheel
3. Work Piece
4. Steel rule.
5. Outside calipers.
6. Cutting tool.

Procedure:

1. The given work piece is first fitted in the chuck of the lathe.
2. By fitting the tool in tool post the work piece will be reduced to given dimensions.
3. First reduce the diameter to 23mm size then reduced the diameter to 15mm and 18mm at the middle.
4. By facing the work piece to the tool work piece is reduced to 70mm.
5. After the preliminary lathe operation, the work piece is held in the ends of the cylindrical grinder.
6. The grinding wheel is turned on and it is moved towards the work piece such that the surfaces of the cylindrical position are grinded to +-0.2mm.

Inference:

Cylindrical grinding is divided into three general operations: plain cylindrical, conical grinding (taper grinding), and internal grinding. The work piece and wheel are set to rotate in opposite directions at the point of contact.

Application:

- Polishing
- Buffing
- Lapping

Result:

Thus the required dimension of cylindrical surface is obtained.

Operation Plan

S.No	***Operation***	***Tools***
1.	***Milling operation to measure the cutting forces***	***Tool dynamometer, End mill cutter***
2.	***Turning operation to measure the cutting forces***	***Tool dynamometer, sensor tool, Carbide tool insert***
Measuring Instruments : Vernier Caliper and Steel Rule		

Ex No: 11 ***Date:***

Measurement of Cutting Forces in Milling / Turning Process

Aim:

To measure the cutting forces for the given cutting conditions.

Tools And Equipment's required:

1. Lathe
2. Milling machine
3. Tool Dynamometer

Procedure:

1. The Lathe Tool Dynamometer is initially set to zero reading.
2. The known depth of cut is given and take the readings of Px and Pz force components from the Lathe Tool Dynamometer.
3. Calculate the resultant cutting force
4. $P = Sqrt (Px^2 + Pz^2)$
5. Repeat the same procedure to get few more readings and calculate the mean cutting force.
6. Repeat the same procedure for different depth of cuts.

Inference:

Significant cutting force and spindle or drilling torque data is very important in ensuring the process optimization involved in metal cutting machining. Analyzing cutting forces prior to the start of production increases process capability and boosts productivity. The detection of overloads, tool collisions and tool breakage can also be monitor.

Applications:

- Measurement of cutting forces.
- Optimization of cutting process

Result:

Thus the cutting forces are measured for different depth of cuts.

Ex No: 12 ***Date:***

CNC Part Programming

Aim:

To Study about the CNC part programming

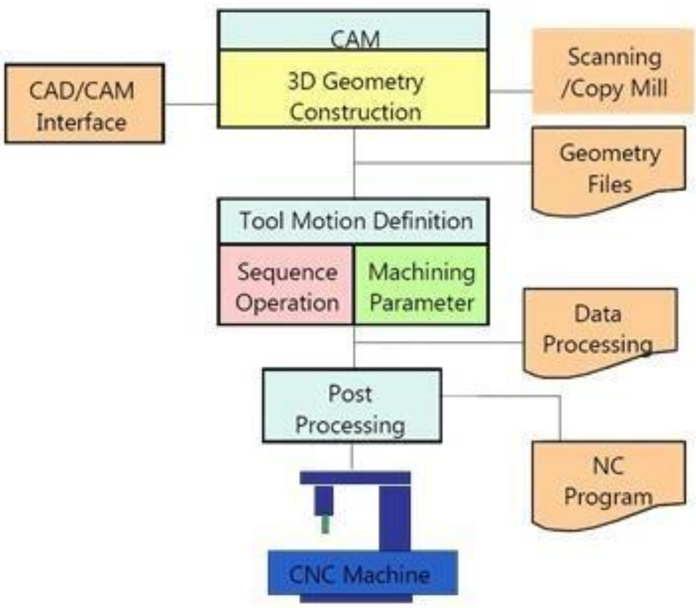
Introduction:

A Part program is a set of instructions given to a computerized numerical control (CNC) machine.

If the complex-shaped component requires calculations to produce the component are done by the programming software contained in the computer. The programmer communicates with this system through the system language, which is based on words. There are various programming languages developed in the recent past, such as APT (Automatically Programmed Tools), ADAPT, AUTOSPOT, COMPAT-II, 2CL, ROMANCE, SPLIT is used for writing a computer programme, which has English like statements. A translator known as compiler program is used to translate it in a form acceptable to MCU.

The programmer has to do only following things:

1. Define the work part geometry.
2. Defining the repetition work.
3. Specifying the operation sequence.



Standard G and M Codes:

The most common codes used when programming NC machines tools are G-codes (preparatory functions), and M codes (miscellaneous functions). Other codes such as *F*, *S*, *D*, and *T* are used for machine functions such as feed, speed, cutter diameter offset, tool number, etc. G-codes are sometimes called cycle codes because they refer to some action occurring on the *X*, *Y*, and/or *Z*-axis of a machine tool. The G-codes are grouped into categories such asGroup 01, containing codes G00, G01, G02, G03, which cause some movement of the machine table or head. Group 03 includes either absolute or incremental programming. A G00 code rapidly positions the cutting tool while it is above the workpiece from one point to another point on a job. During the rapid traverse movement, either the *X* or *Y*-axis can be moved individually or both axes can be moved at the same time. The rate of rapid travel varies from machine to machine.

G – Codes (Preparatory Functions)

Code Function

G00 Rapid positioning

G01 Linear interpolation

G02 Circular interpolation clockwise (CW)

G03 Circular interpolation counterclockwise (CCW)

G20 Inch input (in.)

G21 Metric input (mm)

G24 Radius programming

G28 Return to reference point

G29 Return from reference point

G32 Thread cutting

G40 Cutter compensation cancel

G41 Cutter compensation left

G42 Cutter compensation right

G43 Tool length compensation positive (+) direction

G44 Tool length compensation minus (-) direction

G49 Tool length compensation cancels

G 53 Zero offset or M/c reference

G54 Settable zero offset

G84 canned turn cycle

G90 Absolute programming

G91 Incremental programming

M – Codes (Miscellaneous Functions)

M or miscellaneous codes are used to either turn ON or OFF different functions, which control certain machine tool operations. M-codes are not grouped into categories, although several codes may control the same type of operations such as M03, M04, and M05, which control the machine tool spindle. Some of important codes are given as under with their function s:

Code Function M00 Program stop

M02 End of program

M03 Spindle start (forward CW)

M04 Spindle start (reverse CCW)

M05 Spindle stop

M06 Tool change

M08 Coolant on

M09 Coolant off

M10 Chuck - clamping

M11 Chuck - unclamping

M12 Tailstock spindle out

M13 Tailstock spindle in

M18 Tool post rotation reverse

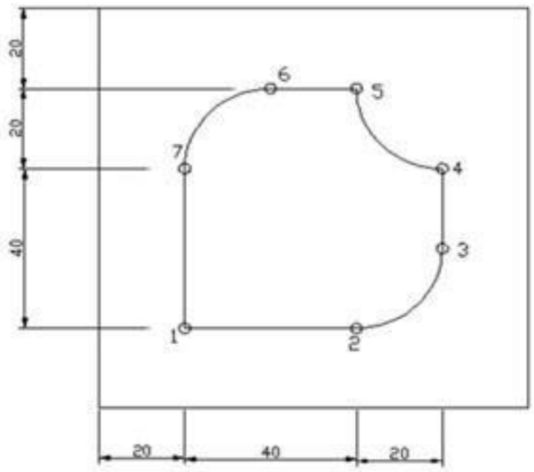
M30 End of tape and rewind or main program end M98 Transfer to subprogram

M99 End of subprogram

Note: On some machines and controls, some may be differ.

Result:

Thus the Study about the CNC part programming has been completed



CNC MILLING	MILD STEEL	100 X 100 X 10	
NOMENCLATURE	MATERIAL	ROUGH SIZE	ALL DIMENSIONS ARE IN MM

Ex No: 13 **Date:**

CNC Machining – For Milling

Aim:

To write a CNC Milling Program and machine the given profile using CNC Milling Machine.

Requirements:

CNC Milling With Fanuc Software

Procedure:

Feed the CNC Program as per the requirement of the given profile using G Codes & M Codes

Program:

```
O2000;
[BILLET X100.0 Y100.0 Z10.0;
[TOOLDEF T1 D5;
[EDGEMOVE X0.0 Y0.0;
G21 G40 G94;
G50 S2000;
G91 G28 Z0.0;
G28 X0.0 Y0.0;
M06 T0101;
M03 S1000;
G90 G00 X20.0 Y20.0 Z1.0;
G01 Z-1 F35.0;
X60.0;
G03 X80 Y40.0 R20.0;
G01 Y60.0;
G02 X60.0 Y80.0 R20.0;
G01 X40.0;
G03 X20.0 Y60.0 R20.0;
G01 Y20.0;
G91 G28 Z2.0
G28 X0.0 Y0.0;
M05;
M30;
```

Inference:

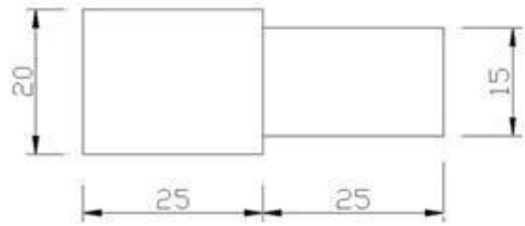
Milling is the machining process of using rotary cutters to remove material from a work piece advancing (or *feeding*) in a direction at an angle with the axis of the tool. It covers a wide variety of different operations and machines, on scales from small individual parts to large, heavy-duty gang milling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

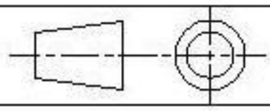
Applications:

- Automobile components.
- Mechanical components
- Industrial applications

Result:

Thus the given profile was machined by using the CNC Milling Machine.



CNC LATHE	MILD STEEL	Ø25.4 X 100	
NOMENCLATURE	MATERIAL	ROUGH SIZE	ALL DIMENSIONS ARE IN MM

Ex No: 14 ***Date:***

CNC Machining – For Lathe

Aim:

To write a CNC Lathe Program and machine the given profile using CNC Lathe Machine.

Requirements:

CNC Lathe with Fanuc Software

Procedure:

Feed the CNC Program as per the requirement of the given profile using G Codes & M Codes

Program:

```
O1010;
[BILLET X20 Z50;
G21 G98 G40;
G28 U0 W0;
M06 T0101;
M03 S1000;
G00 X20 Z01;
G90 X19 Z-25 F45;
X18;
X17;
X16;
X15;
G28 U0 W0;
M05;
M30;
```

Inference:

Turning is an engineering machining process in which a cutting tool, typically a non-rotary tool bit, describes a helical tool path by moving more or less linearly while the work piece rotates. The tool's axes of movement may be literally a straight line, or they may be along some set of curves or angles, but they are essentially linear (in the nonmathematical sense).

Applications:

- Automobile components.
- Industrial Application
- Mechanical components

Result:

Thus the given profile was machined by using the CNC Lathe Machine

PROGRAM OUTCOMES (POs)

Mechanical Engineering Graduates will be able to

1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to solution of complex engineering problems.
2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3	**Design / development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5	**Modern tool usage:** Create, select and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects in multidisciplinary environments.
12	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

K.L.N. COLLEGE OF ENGINEERING

VISION

To become a Centre of Excellence in Technical Education and Research in producing Competent and Ethical professionals to the society.

MISSION

To impart Value and Need based curriculum to the students with enriched skill development in the field of Engineering, Technology, Management and Entrepreneurship and to nurture their character with social concern and to pursue their career in the areas of Research and Industry.

Principal **Secretary & Correspondent** **President**