GOVERNMENT CO-ED POLYTECHNIC COLLEGE RAIPUR

DIPLOMA
3rd YEAR – V (SEM)

Department of Mechanical Engineering

MACHINE TOOLS TECHNOLOGY
LAB MANUAL
2037573(037)

LIST OF EXPERIMENTS

PART-A MACHINE TOOLS

1. To Perform Various Lathe Operations Such As Plain Turning, Step Turning, Taper Turning Knurling And Chamfering On A Given Material Made Of Mild Steel.

2. To Perform V-Thread Cutting On A Lathe Forming Right Hand And Left Hand Metric Threads.

3. To Perform Milling Operation On The Given Specimen (Mild Steel) & Get To Its Correct Dimensions.

4. To Perform V And Dovetail Machining & U-Cut On The Given Work Piece.

5. To Drill The Given Work Piece As Required And Then To Perform Counter Drilling ,Counter Sinking, Tapping Operations On The Given Work Piece.

6. To Perform Surface Grinding Operation On The Given Work Piece.

7. To Perform Cylindrical Grinding Operation On The Given Work Piece.

8. To Make A Slot On The Given Aluminum Work Piece

SAFETY PRECAUTIONS:

1. Attention to be paid for clamping the job, tool, tool holders or supporting items.
2. Care should be taken for avoiding accidental contact with revolving cutters.
3. Break the sharp edges in jobs
4. Do not handle chips with bare hands, use brush or hand gloves.
5. Pay attention while selecting tools or blades for the proposed use to avoid accidents.
6. Do not remove chip while machine is running.
7. Ensure proper bucking of m/c slides or pay attention or alertness.
8. Care should be taken while selecting rapid or feed .
9. Follow safety precautions while approach with cutter to avoid tool damage.
10. Use coolants for heat dissipation.
11. Use goggles for sparks, spatters, avoid the watch clearly with bare eyes.
12. Avoid sharp edge tools.
13. Ensure clamping on surface grinding m/c before take a cut.
14. Select proper speed or feed or depth of cut.
15. Aim for easy chip disposal system.

PROBABLE ACCIDENTS:

1. Before switching on any machine tool, work piece, tool or tool holder or any supporting assembly like tailstock in lathe to be clamped properly.
2. The chief hazard associated is accidental contact with moving cutter
3. Hazard of sharp edge contact with chips while machining.
4. Selection of no. of teeth or blade size on primer hacksaw machine.
5. Ramming of chips when machine in motion viz. shaping or slotting.
6. Locking of tables and ensure the feed.
7. Switch on the connection selection of lever (rapid/feed).
8. Approach the tool to the work piece while machining at slow pace to avoid cutting tool damage.
9. Flying sparks in welding.
10. Holding of heated parts after machining, welding or spot welding.
11. Magnetic clamping refines starting the surface grinding.
12. Selection of proper depth cut or feeds or any machine.
13. Chip disposal system to the accident free.

PART-A MACHINE TOOLS

EXP:1 STEP TURNING AND TAPER TURNING ON LATHE

AIM: To perform Step turning and Taper turning operations on the given work piece

MATERIAL REQUIRED: Mild steel rod of 25 mm diameter and 100 mm long.

TOOLS REQUIRED: Vernier calipers, steel rule, spanner, chuck spanner, and H.S.S. single point cutting tool.

SPECIFICATION OF LATHE:

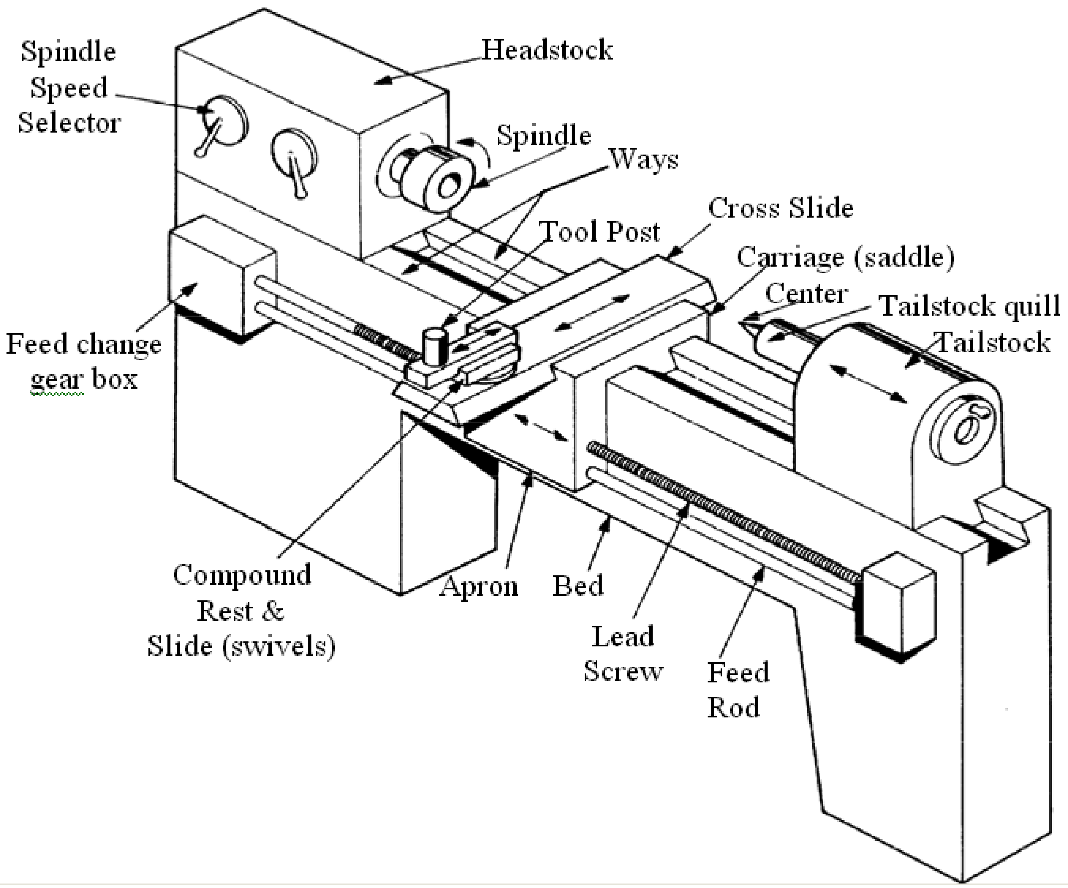
Length of bed	1390 mm
Width of bed	200 mm
Height of centers	165 mm
Admit between centers	700 mm
Lead screw pitch	4TPI
Power of the motor	1 h.p.

THEORY:

Lathe removes undesired material from a rotating work piece in the form of chips with the help of a tool which is traversed across the work and can be fed deep in work. The tool material should be harder than the work piece and the later help securely and rigidly on the machine. The tool may be given linear motion in any direction. A lathe is used principally to produce cylindrical surfaces and plane surfaces, at right angles to the axis of rotation. It can also produce tapers and bellows etc.

OPERATION CHART:

S NO.	SEQUENCE OF OPERATIONS	CUTTING TOOL USED
1.	Facing	H.S.S Single Point tool
2.	Rough turning	H.S.S Single Point tool
3	Finish turning	H.S.S Single Point tool
4	Step turning	Parting tool
5	Taper turning	H.S.S Single Point tool
6	Knurling	Knurling tool
7	Chamfering	H.S.S Single Point tool



PROCEDURE:

1. The work piece and HSS single point cutting tool are securely held in the chuck and tool post respectively.

2. Operations such as facing, rough turning and finish turning are performed on a given mild steel bar one after the other in sequence up to the dimensions shown. Then the step turning is performed using parting tool.

3. Then the compound rest is swiveled by calculated half taper angle and taper is generated on the work piece. Rotation of the compound slide screw will cause the tool to be fed at the half-taper angle.

4. HSS single point cutting tool is replaced by the knurling tool and knurling operation is performed at the slowest speed of the spindle.

5. The knurling tool is replaced by the HSS single point tool again; the work piece is removed from the chuck and re fixed with the unfinished part outside the chuck. This part is also rough turned, finish turned and facing is done for correct length.

6. Finally, the chamfering is done at the end of the work piece.

OBSERVATIONS: (a) Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
lathe							

Speed and Feed Data (Table B)

No.	Lathe	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		
6.		

PRECAUTIONS:

1. Operate the machine at optimal speeds

2. Do not take depth of cut more than 2 mm.

3. Knurling should be done at slow speeds and apply lubricating oil while knurling

4. Care should be taken to obtain the required accuracy.

RESULT:

APPLICATIONS:

Applications	**Description**
Woodturning	Used to make wooden objects such as ornate table legs, baseball bats, wooden bowl, and platters; operators use a variety of tools to form
Metalworking	Used to create precision parts; most often associated with a multistep process requiring different tools for each step
Metal Spinning	A process where metal spins on a spindle, while the operator works it with tools; an automated process
Acrylic Spinning	Involves spinning acrylic on a spindle to form items from acrylic; most often used in the making of the top pieces for trophies
Thermal Spraying	Combines the rotating spindle with the painting process; the paint sticks to the stock via processes involving heating the paint materials

ADVANTAGES:
One advantage of a lathe machine is that it can perform very detailed and intricate designs.

DIS-ADVANTAGES:
One disadvantage of a lathe machine is that these machines are more expensive than other types of machines used to produce this type of work.

EXP.2: THREAD CUTTING AND KNURLING ON LATHE

AIM: To perform V-thread cutting on a lathe forming right hand and left hand metric threads.

MATERIAL REQUIRED

Mild steel bar of 24 mm diameter and 100 mm length

TOOLS AND EQUIPMENT

H.S.S. single point cutting tool, Grooving tool, Threading tool thread gauge, Outside caliper, Chuck key, Tool post key, Steel rule.

OPERATION CHART

S no.	Sequence of Operations	Cutting tool used
1.	Facing	H.S.S Single Point cutting tool
2.	Rough turning	H.S.S Single Point cutting tool
3	Finish turning	H.S.S Single Point cutting tool
4	Step turning	H.S.S Single Point cutting tool
5	Grooving	Grooving tool
6	Thread cutting	Threading tool
7	Chamfering	H.S.S Single Point cutting tool

PRINCIPLE OF THREAD CUTTING

The principle of thread cutting is to produce a helical groove on a cylindrical or conical surface by feeding the tool longitudinally when the job is revolved between centers or by a chuck. The longitudinal feed should be equal to the pitch of the thread to be cut per revolution of the work piece. The lead screw of the lathe, through which the saddle receives its traversing motion, has a definite pitch. A definite ratio between the longitudinal feed and rotation of the head stock spindle should therefore be found out so that the relative speeds of rotation of the work and the lead screw will result in the cutting of a screw of the desired pitch. This is affected by change gears arranged between the spindle and the lead screw or by the change gear mechanism or feed box used in a modern lathe.

Calculation of change-wheels, metric thread on English lead screw:

To calculate the wheels required for cutting a screw of certain pitch, it is necessary to know how the ratio is obtained and exactly where the driving and driven wheels are to be placed. Suppose the pitch of a lead screw is 12 mm and it is required to cut a screw of 3 mm pitch, then the lathe spindle must rotate 4 times the speed of the lead screw that is

$$\frac{\text{Spindle turn}}{\text{Lead screw turn}} =$$ Means that we must have

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} =$$ Since a small gear rotates faster than a larger one with which it is connected.

Hence we may say,

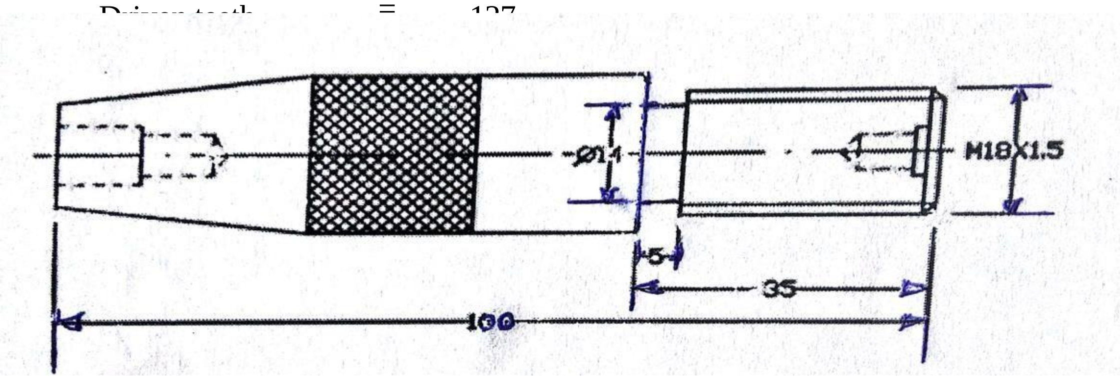
$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{lead screw turn pitch of the screw to be cut}}{\text{spindle turn pitch of the lead screw}}$$

In BRITISH SYSTEM

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Threads per inch on lead screw}}{\text{Threads per inch on work}}$$

Often engine lathes are equipped with a set of gears ranging from 20 to 120 teeth in steps of 5 teeth and one translating gear of 127 teeth. The cutting of metric threads on a lathe with an English pitch lead screw may be carried out by a translating gear of 127 teeth.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{5\,p\,n}{127}$$



..

This is derived as follows:

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{pitch of the work}}{\text{pitch of the lead screw}} = \frac{p}{(1/n) \times (127/5)} = \frac{pn}{127}$$

$$\text{Since, pitch} = \frac{1}{\text{No. of threads per inch}}$$

THREAD CUTTING OPERATION:

In a thread cutting operation, the first step is to remove the excess material from the work piece to make its diameter equal to the major diameter of the screw thread. Change gears of correct size are then fitted to the end of the bed between the spindle and the lead screw. The shape or form of the thread depends on the shape of the cutting tool to be used. In a metric thread, the included angle of the cutting edge should be ground exactly 60^0.The top of the tool nose should be set at the same height as

the center of the work piece. A thread tool gauge is usually used against the turned surface to check the cutting tool, so that each face of the tool may be equally inclined to the center line of the work piece as shown.

The speed of the spindle is reduced by one half to one – fourth of the speed require for turning according to the type of the material being machined and the half – nut is then engaged. The depth of cut usually varies from 0.05 to 0.2 mm is given by advancing the tool perpendicular to the axis of the work.

After the tool has produced a helical groove up to the desired length of the work, the tool is quickly withdrawn by the use of the cross slide, the half-nut disengaged and the tool is brought back to the starting position to give a fresh cut. Before re-engaging the half-nut it is necessary to ensure that the tool will follow the same path it has traversed in the previous cut, otherwise the job will be spoiled. Several cuts are necessary before the full depth of thread is reached arising from this comes the necessity to "pick-up" the thread which is accomplished by using a chasing dial or thread indicator.

Chasing dial or thread indicator

The chasing dial is a special attachment used in modern lathes for accurate "picking up" of the thread. This dial indicates when to close the split of half nuts. This is mounted on the right end of the apron. It consists of a vertical shaft with a worm gear engaged with the lead screw. The top of the spindle has a revolving dial marked with lines and numbers. The dial turns with the lead screw so long the half nut is not engaged.

If the half-nut is closed and the carriage moves along the dial stands still. As the dial turns, the graduations pass a fixed reference line. The half-nut is closed for all even threads when any line on the dial coincides with the reference line. For all odd threads, the half-nut is closed at any numbered line on the dial determined from the charts. If the pitch of the thread to be cut is an exact multiple of the pitch of the lead screw, the thread is called even thread, if otherwise the thread is odd thread.

In a chasing dial, the rule for determining the dial division is: In case of metric threads, the product of the pitch of lead screw and the no. of teeth on the worm wheel must be an exact multiple of the pitch of the threads to be cut. In case of English threads, the product of the threads per inch to be cut and the number of teeth on the worm wheel must be an exact multiple of the number of threads per inch of the lead screw. For example, if the pitch of the lead screw is 6 mm and the worm wheel has 15 teeth.

The product will be 90. so any pitch which is exactly divisible by 90, such as 1, 1.25, 2.25,3,3.75,4.5,5,6,7.5,9,10,15,30,45,90 may be picked up when any line of the dial coincides with the reference line.

Right hand and left-hand thread:

If the bolt advances into the nut when rotated in clockwise direction, the thread is called right-hand thread. When cutting a right-hand thread the carriage must move towards the head stock.

If the bolt advances into the nut when rotated in counter-clockwise direction, the thread is called left-hand, for a left hand thread the carriage moves away from the head stock and towards the tail stock. The job moves as always in the anti-clock wise direction when viewed from the tail stock end. The direction at which the carriage moves in relation to lathe head stock is controlled by means of the tumbler gears or bevel gear feed reversing mechanism.

PROCEDURE:

The work piece and HSS single point cutting tool are fixed in chuck and tool post respectively.

1. Operations such as facing, rough turning finish turning and step turning are performed on the given mild steel bar one after the other in sequence up to the dimensions shown.

2. Single point cutting tool is replaced by a grooving tool and grooving operation is performed at half of the normal spindle speed.

3. The grooving tool is replaced by a threading tool. Right hand and left hand metric threads are cut on the work piece up to the required length at $1/4^{th}$ of the normal speed of the spindle.

4 Threading tool replaced by a single point cutting tool again and finally chamfering is done at right end of the work piece at normal spindle speed.

PRECAUTIONS:

1. Low spindle speeds should be used for accurate threads in thread cutting operation.
2. Ensure correct engage and dis-engage of half-nut.

   Plenty of oil should be flowed on the work and tool during thread cutting

OBSERVATIONS: Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
lathe							

Speed and Feed Data (Table B)

No.	Lathe	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		
6.		
7.		
8.		

RESULT:

ADVANTAGES:
One advantage of a lathe machine is that it can perform very detailed and intricate designs

DISADVANTAGES:
One disadvantage of a lathe machine is that these machines are more expensive than other types of machines used to produce this type of work.

VIVA QUESTIONS:

1. What is a lathe?
2. What are the various operations can be performed on a lathe?
3. What are principle parts of the lathe?
4. What are the types of headstock?
5. State the various parts mounted on the carriage?
6. What are the four types of tool post?
7. What is an apron?

EXP 3: MANUFACTURING OF SPUR GEAR USING MILLING MACHINE

AIM: To perform plane milling operation on the given specimen (mild steel) & get to its correct dimensions.

MATERIALS REQUIRED: mild steel work piece.

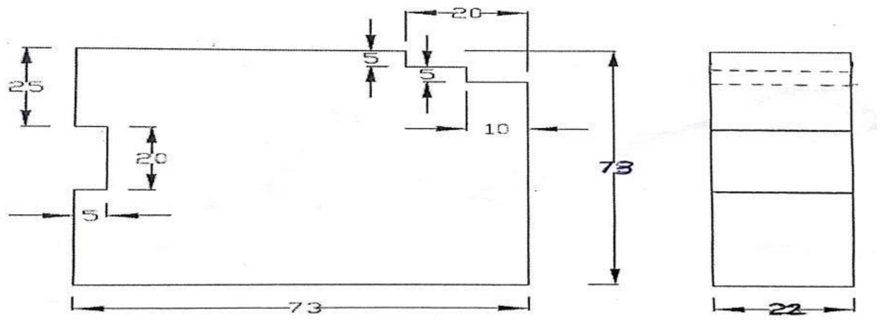
MACHINE REQUIRED: milling machine

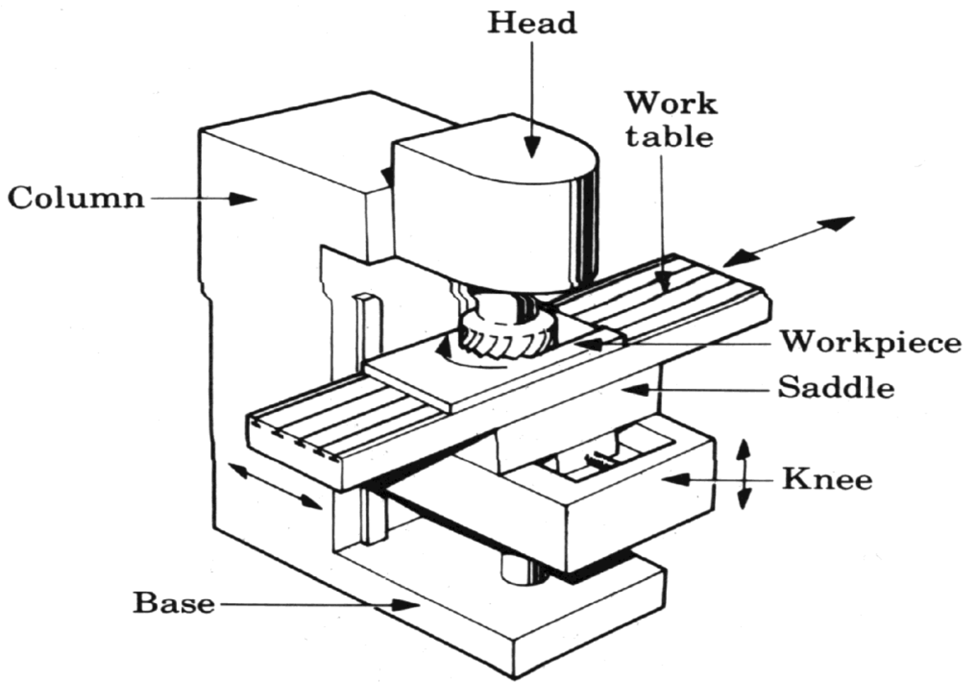
MEASURING INSTRUMENTS: Vernier calipers

CUTTING TOOLS: Plane (face) milling cutter.

MARKING TOOLS: steel rule, scriber

a. Work holding fixtures: work piece supporting fixtures
b. Miscellaneous tools: Hammer, brush, Allen keys





OBSERVATION: Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Milling m/c							

Speed and Feed Data (Table B)

No.	Milling m/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PROCEDURE:

1. The dimensions of the given rod are checked with the steel rule.
2. The given rod is fixed in the vice provided on the machine table such a, one end of it is projected outside the jaws of the vice.
3. A face milling cutter is mounted on the horizontal milling machine spindle and one end of the rod is face milled, by raising the table so that the end of the rod faces the cutter.
4. The rod is removed from the vice and fitted in the reverse position.
5. The other end of rod is face milled such that, the length of the job is exactly 100 mm.
6. The table is lowered and the rod is removed from the vice and refitted in it such that, the top face of the rod is projected from the vice jaws.
7. The face milling cutter is removed from the spindle and the arbor is mounted in the spindle; followed by fixing the plain milling cutter.
8. The top surface of the job is slab milled; first giving rough cuts followed by a finish cut.
9. The job is removed from the vice and refitted in it such that, the face opposite to the above, comes to the top and projects above the vice jaws.
10. The top surface of the job is milled in stages; giving finish cuts towards the end such that, the height of the job is exactly 40 mm.
11. The burrs if any along the edges are removed with the help of the flat file.

12. Learn the names of the major units and the components of each machine. Record these details (Table A). Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.
13. Record the obtainable speed and feed values (Table B).
14. Note down the special features of the speed and feed control on each machine

PRECAUTIONS:

1. The milling machine must be stopped before setting up or removing a work piece, cutter or other accessory.

2. Never stop the feeding of job when the cutting operation is going on, otherwise the tool will cut deeper at the point where feed is stopped.

3. All the chips should be removed from the cutter. A wiping cloth should be placed on the cutter to protect the hands. The cutter should be rotated in the clockwise direction only for right handed tools.

4. The work piece and cutter should be kept as cool as possible (i.e. coolant should be used where necessary to minimize heat absorption).

5. The table surface should be protected with a wiping cloth.

6. Tool must be mounted as close to the machine spindle as possible.

RESULT:

ADVANTAGE:

1. Both flat and formed surface can be produced.

DISADVANTAGES:

(i) Quality of surface generated will be slightly wavy

(ii) Lubrication is difficult.

(iii) Needs heavy fixture since the cutting force results in lifting the work piece.

(iv) Results in vibration.

(v) Cutting force is not uniform.

APPLICATIONS:

Milling machines are widely used in the tool and die making industry and are commonly used in the manufacturing industry for the production of a wide range of components .Typical examples are the milling of flat surface, indexing, gear cutting, as well as the cutting of slots and key-ways.

VIVA QUESTIONS:

1. What are the specifications of the milling machine?
2. Mention the various movements of universal milling machine table?
3. State any two comparisons between plain &universal milling machine?
4. What are the cutter holding devices?
5. List the various type of milling attachment?
6. Write any ten nomenclature of plain milling cutter?
7. What are the advantages of milling process?
8. what are the down milling processes?
9. List out the various milling operations?
10. What does term indexing mean?
11. What are the three types dividing heads?
12. What is cam milling?
13. What are the different types of thread milling?
14. Gear cutting by single point form tool.
15. List the gear generating process?
16. What is a semi-automatic lathe?
17. What is copying lathe?
18. State the various feed mechanisms used for obtaining automatic feed?
19. List any four holding devices?
20. What are the different operations performed on the lathe?

EXP.4 MACHINING FLAT SURFACE USING SHAPER MACHINE

AIM: To perform V and Dovetail machining & U-cut on the given work piece.

MATERIALS REQUIRED: Mild steel / Cast iron / Cast Aluminum.

MACHINE REQUIRED: Shaping machine

MEASURING INSTRUMENTS:

Vernier calipers,

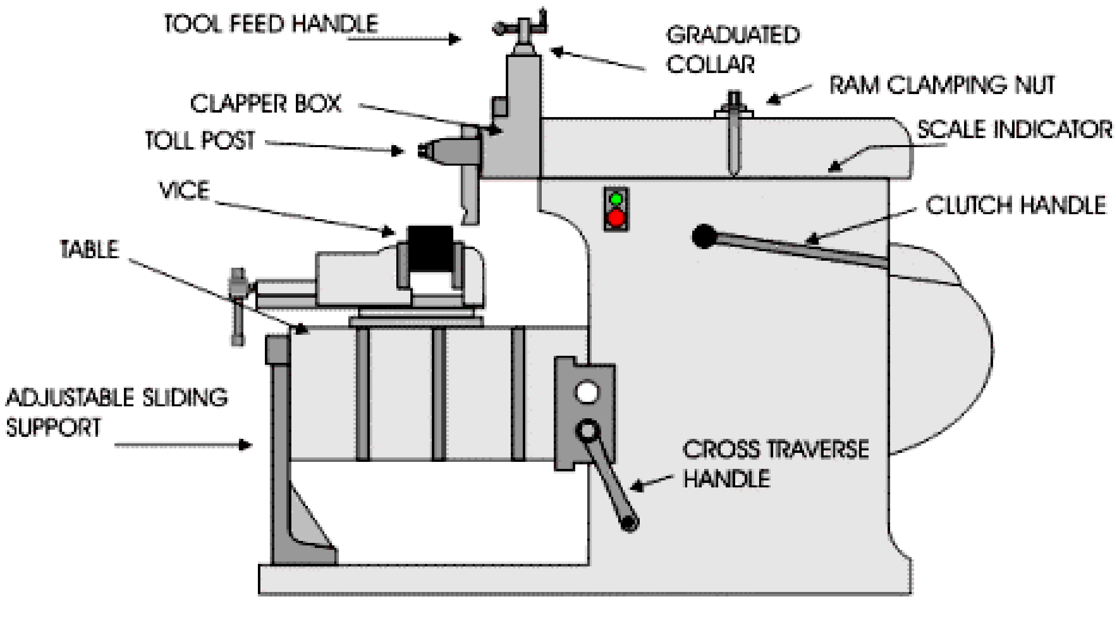
Vernier height gauge,

Dial indicator,

Required steel ball.

CUTTING TOOLS

H.S.S tool bit, V-tool, Plain tool, Grooving tool.



OBSERVATION Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	

Shaper M/c.							

Speed and Feed Data (Table 2)

No.	Shaper M/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PROCEDURE:

1. Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

2. Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

3. Record the obtainable speed and feed values (Table B).

4. Note down the special features of the speed and feed control on each machine.

5. Measuring of specimen.

6. Fixing of specimen in the machine vice of the shaping machine

7. Giving the correct depth and automatic feed for the slot is to be made.

8. Check the slot with the Vernier calipers & precision measurement by slip gauges at the end.

PRECAUTIONS:

1. The shaping machine must be stopped before setting up or removing the work piece
2. All the chips should be removed from the cutter.

RESULTS:

ADVANTAGES:

1. Single point cutting tools used in shaper are expensive these tools can be easily grounded to any desirable shape.

2. Shaper set-up is very quick and easy and can be readily changed from one job to another.

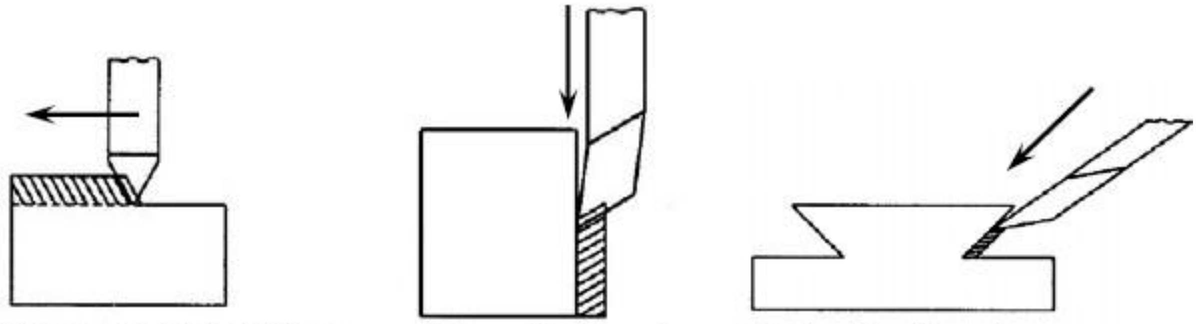
DIS ADVANTAGES:

1. The shaper is unsuitable for generating the flat surfaces on very large parts because of limitations on the stroke and overhang the ram.

2. The primary motion is accomplished by rack and pinion drive using a variable speed motor.

APPLICATIONS:

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.



(a) horizontal surface (b) vertical surface (c) inclined surfaces

VIVA QUESTIONS:

1. Mention the applications of gear shaping process?
2. What are the limitations of gear hobbing?
3. What is shaper?
4. List any four important parts of a Shaper?
5. How the feed & depth of cut is given to the shaper?
6. Mention any four-shaper specification?
7. How the planer differs from the shaper?

EXP:5 DRILLING AND TAPPING

AIM: To drill the given work piece as required and then to perform to make, counter boring, countersinking and tapping operations

MATERIALS REQUIRED: mild steel specimen, coolant (oil and water mixture), lubricant oil, nut and bolt.

MACHINE REQUIRED: Drilling machine

MEASURING INSTRUMENTS: Vernier calipers

CUTTING TOOLS:

Button pattern stock,
Dies,
Drill bits,
Hand taps,
Tap wrench.

MARKING TOOLS: Dot punch

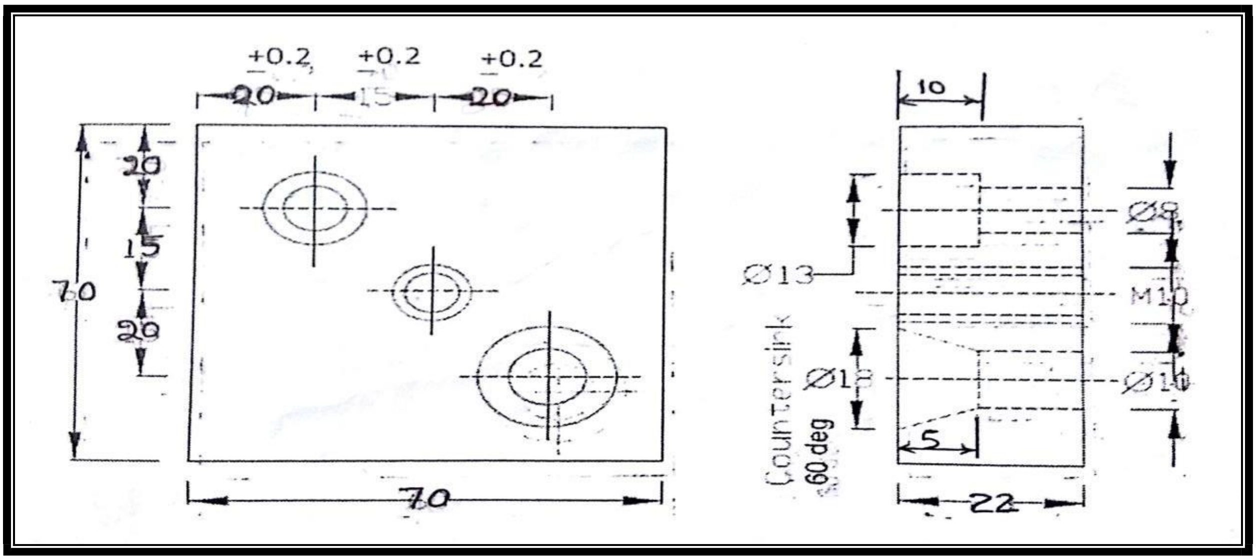
Work holding fixtures:

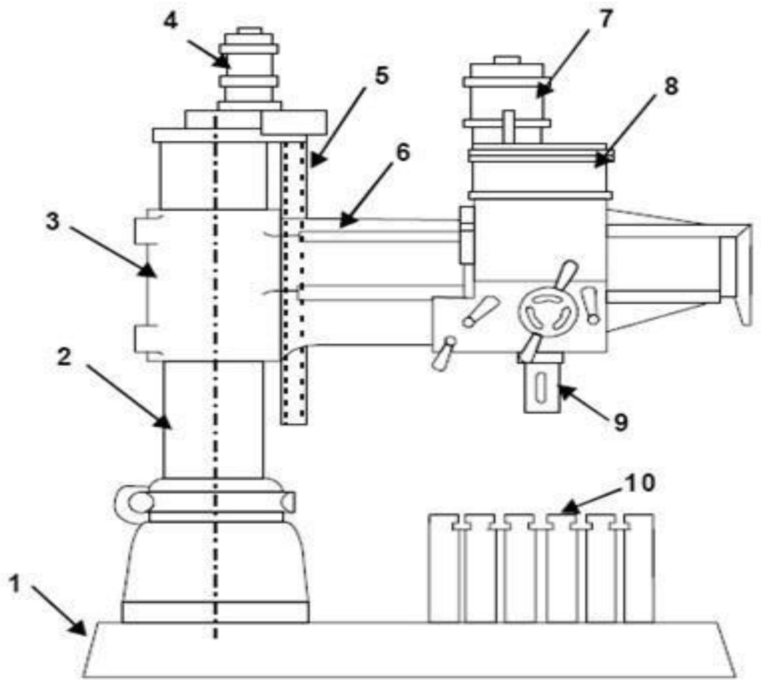
Bench vice, V-Block

Miscellaneous tools:

Brush, Allen Keys

DIAGRAM:





PARTS OF RADIAL DRILL:

1. Base
2. Column
3. Radial arm
4. Motor for elevating screw
5. elevating screw
6. Guide ways
7. Motor for driving spindle
8. Drill head
9. Drill spindle
10. Table

OBSERVATION

Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Drilling m/c							

Speed and Feed Data (Table B)

No.	Drilling M/c.	
	Speed	Feed
1		
2		
3		
4		

SEQUENCE OF OPERATIONS:

1. Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced.
2. Note particularly the features, which control the geometrical form of the surface.
3. Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.
4. Record the obtainable speed and feed values (Table B).
5. Note down the special features *of* the speed and feed control on each machine,
6. Mark the center of hole and center punching
7. Drill bid

   $$\mathbf{D_d = d_h - p}$$

   Where,

   1. D_h - dia. of the hole,
   2. d_d – dia. of drill bit,
   3. p = pitch
8. Use the suitable drill size for required tapping
   D=Dia. of tap
   Tap Drill size = (D-1.3p)+0.2 – for metric threads
9. Chamfering of specimen
10. Use the sequential tapping as tap set 1, 2, and 3
11. Internal taping of drilled specimen
12. Filling of specimen on which external threading to be done
13. Measuring the diameter of the specimen & choosing of dies according to it
14. Dying operation (external threading) of the specimen.

PRECAUTIONS:

1. Coolant has to be sued while drilling
2. Lubricating oil has to be used to get smooth finish while tapping.

RESULT:

ADVANTAGES:

1. The universal movements of the tool head permit the drill tool located at any desired position over the stationary work piece.
2. Possible to work on odd shaped jobs and to drill larger diameter holes.
3. Accurate precision drilling is possible.

DISADVANTAGES:

1. A skilled worker is a must.
2. Only small size holes can be drilled.

APPLICATIONS:

1. Origination and / or enlargement of existing straight through or stepped holes of different diameter and depth in wide range of work materials – this is the general or common use of drilling machines
2. Making rectangular section slots by using slot drills having 3 or four flutes and 1800 cone angle
3. Boring, after drilling, for accuracy and finish or prior to reaming
4. Counter boring, countersinking, chamfering or combination using suitable tools.

VIVA QUESTIONS:

1. What is meant by drilling?
2. What is gang -drilling machine
3. Mention any four specification of drilling machine?
4. List any four machining operations that can be performed on a drilling machine?
4. What are the different ways to mount the drilling tool?

EXP.6: PRECISION SURFACE GRINDING

AIM: To perform surface grinding operation on the given (50*50*20) work piece.

MATERIALS REQUIRED: mild steel specimen.

MACHINE REQUIRED: surface grinding machine

MEASURING INSTRUMENTS:

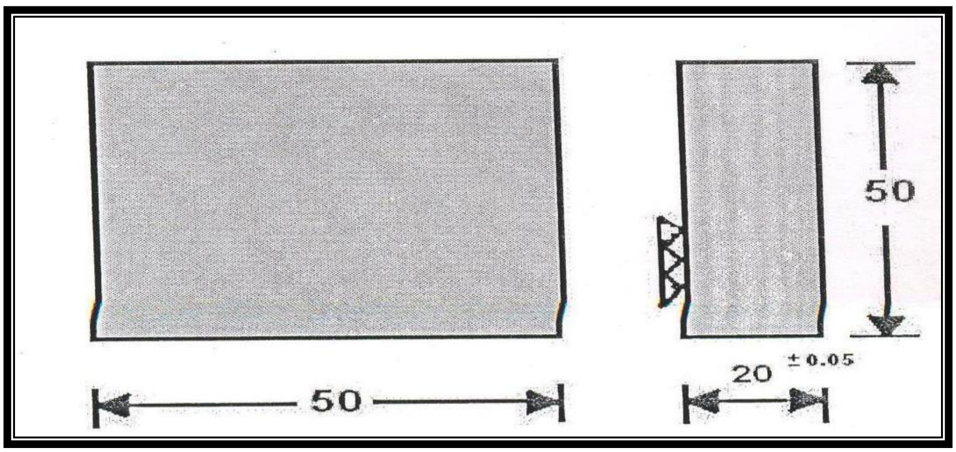
Vernier calipers,

Micrometer.

CUTTING TOOLS: Diamond point dressing block

WORK HOLDING FIXTURES: Magnetic chuck

Experimental Diagram Surface Grinding:



All Dimensions are In mm

MISCELLANEOUS TOOLS:

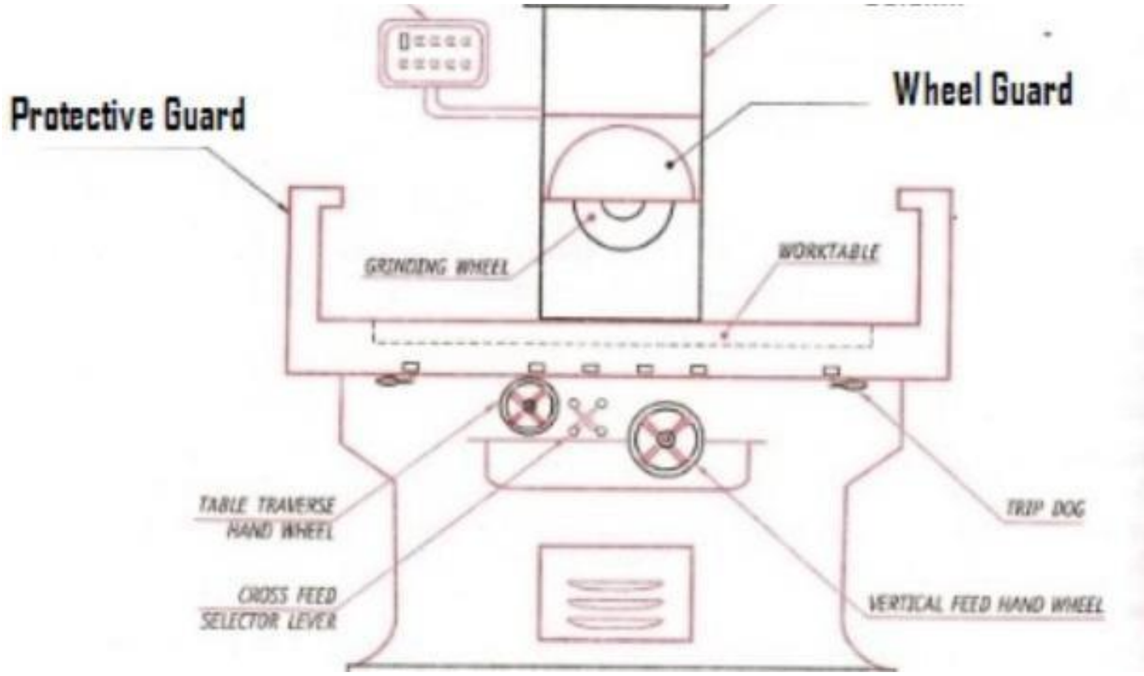
Wire brush (for cleaning the formed chips),

Lubricant (coolant),

PROCEDURE:

1. Work piece is mounted on magnetic table, so that the line along face of grinding wheel coincides
   with the edge of work piece.
2. Depth of cut is given to work piece by down feed hang wheel.
3. The work piece is reciprocates under wheel and the table feeds axially between passes to

produce flat surface and to get required size of work piece.



PRECAUTIONS:

1. Coolant usage is compulsory as the speeds employed are very high and continuous application of coolant is necessary for ductile materials like-steel etc.
2. The grinding tools are first dressed properly.
3. Care has to be taken so as to maintain the right feed of the material.
4. Work-wheel interface zone is to be flooded with coolant
5. Dressing of grinding wheel to be done before commencement of cutting action, intermittent dressing also to be done if wheel is loaded.

RESULT:

APPLICATIONS :

1. Cylindrical grinding process is used for grinding the outer surface of cylindrical object
2. Center less grinding process is used for preparing the transmission bushing, shouldered pins and ceramic shafts for circulator pumps.
3. Internal grinding process is used for finishing the tapered, straight and formed holes precisely.
4. There are few special grinders used for sharpen the milling cutters, taps, other various machine cutting tool cutter and reamers.

ADVANTAGES:

1. Investment is less
2. Working principle and operation is simple
3. It does not require additional skills
4. Surface finishing will be approximate 10 times better as compared to milling and turning process of machining.
5. Dimensional accuracy will be quite good
6. Grinding process could be performed on hardened and unhardened workpiece also

VIVA QUESTIONS:

1. State the purpose of grinding?
2. What is the function of cutting fluids?
3. What are the properties of cutting fluid?
4. What are causes of wear?

Experiment:7 CYLINDRICAL GRINDING

Aim: To Perform Cylindrical Grinding Operation On The Given Work Piece.
MATERIALS REQUIRED: mild steel specimen.

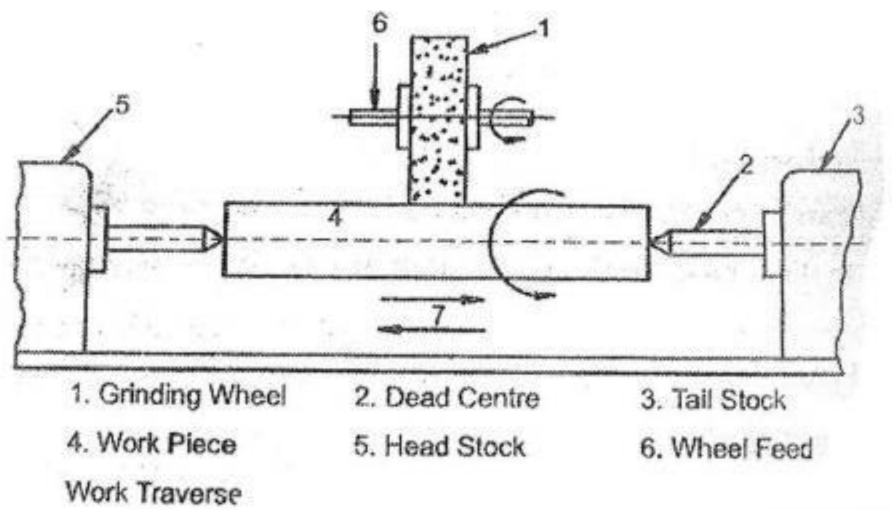
MACHINE REQUIRED: cylindrical grinding machine

MEASURING INSTRUMENTS: Vernier calipers, Micrometer.
CUTTING TOOLS: Diamond point dressing block

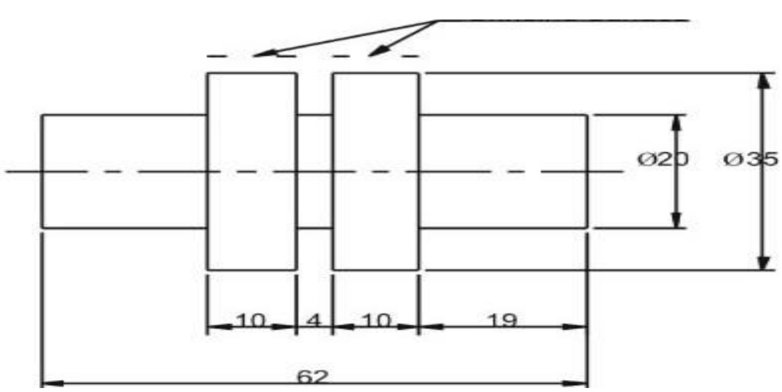
WORK HOLDING FIXTURES: Magnetic chuck

MISCELLANEOUS TOOLS:

Wire brush (for cleaning the formed chips), Lubricant (coolant)



Experimental Diagram Cylindrical Grinding:



All dimensions are in imm

PROCEDURE:

1. Work piece is mounted on magnetic table, so that the line along face of grinding wheel coincides with the edge of work piece.
2. Depth of cut is given to work piece by down feed hang wheel.
3. The work piece is reciprocates under wheel and the table feeds axially between passes to produce flat surface and to get required size of work piece.

PRECAUTIONS :

1. Coolant usage is compulsory as the speeds employed are very high and continuous application of coolant is necessary for ductile materials like-steel etc.
2. The grinding tools are first dressed properly.
3. Care has to be taken so as to maintain the right feed of the material.
4. Work-wheel interface zone is to be flooded with coolant
5. Dressing of grinding wheel to be done before commencement of cutting action, intermittent dressing also to be done if wheel is loaded.

RESULT

APPLICATIONS :

1. Cylindrical grinding process is used for grinding the outer surface of cylindrical object
2. Center less grinding process is used for preparing the transmission bushing, shouldered pins and ceramic shafts for circulator pumps.
3. Internal grinding process is used for finishing the tapered, straight and formed holes precisely.
4. There are few special grinders used for sharpen the milling cutters, taps, other various machine cutting tool cutter and reamers.

Experiment:8 MAKING INTERNAL SPLINES USING SLOTTING MACHINE

AIM: To make a slot in cast iron pulley as per the sketch given dimensions.

MATERIALS REQUIRED: mild steel, aluminum.

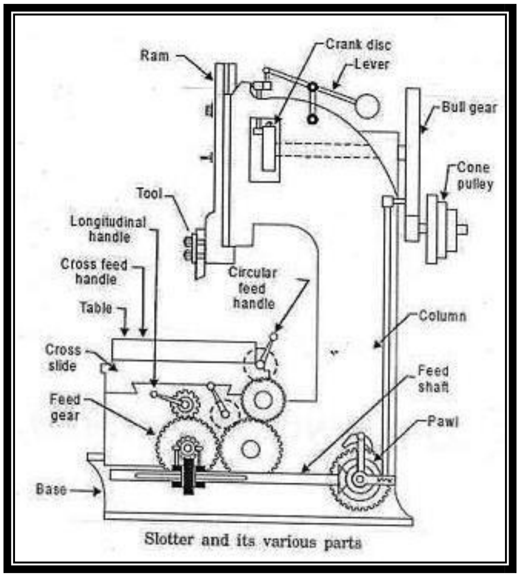
MACHINE REQUIRED: slotting machine

MEASURING INSTRUMENTS: Vernier calipers slip gauges.

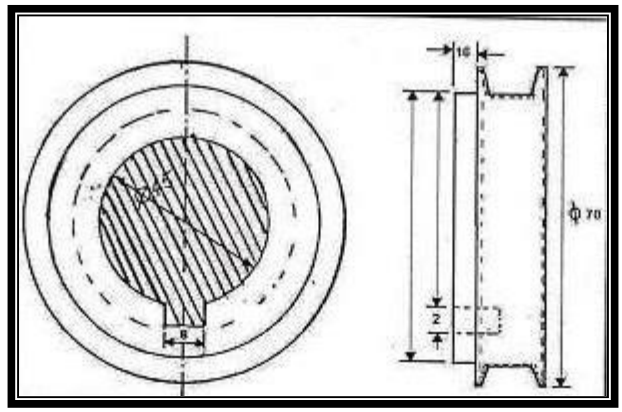
CUTTING TOOLS: H.S.S.Tool bit of the required slot size.

PROCEDURE:

1. Fix the work piece in the head stock chuck firmly
2. Turning tool is fixed in the tool post and centering is to be done
3. Turn the job to get a Diameter of required length
4. Facing is to be done on one side of the job
5. Drill bit of 8 mm diameter is fixed on tail stock and centering of work piece is to be done by feeding through tail stock.
6. Drill bit of 25 mm diameter is fixed in tail stock
7. Drill through a hole of 25 mm diameter in the work piece feeding the tail stock.
8. Boring tool is the fixed in tool post to perform boring operation to get a hole of required diameter
9. Fit the job in reverse position in the chuck
10. Facing of other side of the work piece is to be done to get the required length of the job
11. Drilled work piece is fixed on slotting machine.
12. A slot of required depth is made



Slotter and its various parts



Parts of slotting Machine **Experimental Diagram:**

All Dimensions are in mm

PRECAUTIONS:

1. Choose proper feed and depth of cut.

2. Feed should be controlled to avoid any damage to the cutting tool

3. Lock the index table before starting the operation.

4. Care has to be taken so as to maintain the right feed of the material.

5. Work-wheel interface zone is to be flooded with coolant

6. Dressing of grinding wheel to be done before commencement of cutting action, intermittent dressing also to be done if wheel is loaded.

RESULT:

APPLICATIONS:

Slotting machines are used to cut grooves and slots in shapes and holes while additionally smoothing the

worked surface. Because they are more economical at high production rates due to reproducibility and
consistency, slotters are generally used in high volume operations. They are used in steel rolling mills,
paper mills, power plants, ship building, textile factories, tool rooms, and repair shops.

PLANER MACHINE:

INTRODUCTION

Planning is one of the basic operations performed in machining work and is primarily intended for machining.
These surfaces may be horizontal, vertical or inclined. In this way, the function of a planning machine is quite similar to that of a shaper except that the former is basically designed to undertake machining of such large and heavy jobs which are almost impractical to be machined on a shaper or milling, etc. It is an established fact that the planning machine proves to be most economical so far as the machining of large flat surfaces is concerned. However, a palning machine differs from a shaper in that for machining, the work, loaded on the table, reciprocates past the stationary tool in a planer, whereas in a shaper the tool reciprocates past the stationary work.

A planer is a type of metalworking machine tool that uses linear relative motion between the workpiece and a single-point cutting tool to machine a linear tool path. Its cut is analogous to that of a lathe, except that it is linear instead of helical. (Adding axes of motion can yield helical tool paths; see "Helical planing" below.) A planer is analogous to a shaper, but larger, and with the entire workpiece moving on a table beneath the cutter, instead of the cutter riding a ram that moves above a stationary workpiece. The table is moved back and forth on the bed beneath the cutting head either by mechanical means, such as a rack and pinion drive or a leadscrew, or by a hydraulic cylinder.

SPECIFICATIONS:

Horizontal distance between two vertical housings:

Vertical distance between table top and the cross rail: 800mm

Maximum length of table travel: 1350mm

Length of bed: 2025mm

Length of table: 1425mm

Method of driving – Individual

Method driving table – Geared

H.P. of motor: 3 H.P. & 1 H.P.

STANDARD OR DOUBLE HOUSING PLANER:

This is the most commonly used type of planer. It consists of two vertical housings or columns, one on each side of the bed. The housings carry vertical or scraped ways. The cross-rail is fitted between the two housings and carries one or two tool heads. The work table is mounted over the bed. Some planers may fit with side tool heads fitted on the vertical columns.

MAIN PARTS OF A PLANER

A planer consists of the following main parts as illustrated by means of a block diagram in fig.

- Bed , Table , Housings or columns, Cross – rail, Tool head, Controls

These machines are heavy duty type and carry a very rigid construction. They employ high speeds for cutting but the size of work they can handle is limited to the width of their table i.e. the horizontal distance between the columns.

Extremely large and heavy castings, like machine beds, tables, plates, slides, columns, etc., which normally carry sliding surfaces like guide ways or dovetails on their longitudinal faces, are usually machined on these machines. Also because of long table and larger table travel, on either side of the columns, it is possible to hold a number of work pieces in a series over the bed length and machine them together. This will effect a substantial saving in machining time. Further because of no.of tool heads the surfaces can be machined simultaneously. This effects further reduction in machining time. Also because of high rigidity of high rigidity of the machineand robust design of the cutting tools heavier cuts can be easily employed, which leads to quicker metal removal and reduced machining time. Thus an overall picture emerges that the employment of this type of machine apart from its capacity to handle such heavy and large jobs which are difficult to be handled on other machines, leads to faster machining and reduced machining time and hence to economical machining. However considerable time is used in setting up a planer.

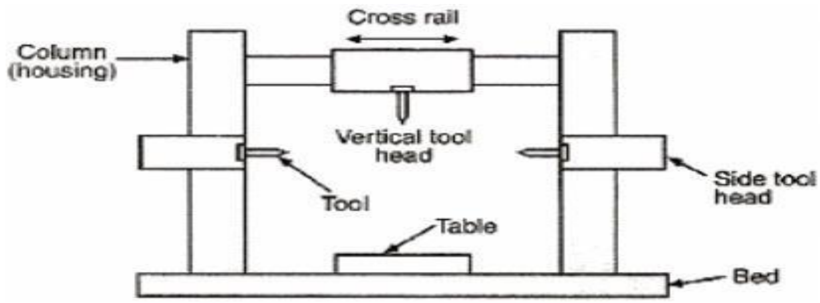
DRIVE MECHANISMS:

Four different methods are employed for driving the table of a planer.

They are:

- Crank drive

- Belt drive
- Direct reversible drive
- Hydraulic drive



ELECTRICAL DISCHARGE MACHINING

(Cutting Metal to Precise Shapes using Electricity)

.