



# LABORATORY MANUAL

## ME3382- MANUFACTURING TECHNOLOGY LABORATORY

<b>YEAR / SEMESTER</b>	<b>:</b>	<b>II / 03</b>
<b>DEPARTMENT</b>	<b>:</b>	<b>Mechanical Engineering</b>
<b>ACADEMIC YEAR</b>	<b>:</b>	<b>2023-24</b>
<b>REGULATION</b>	<b>:</b>	<b>2021</b>

**OBJECTIVE:**

To Study and practice the various operations that can be performed in lathe, shaper, drilling, milling machines etc. and to equip with the practical knowledge required in the core industries.

**LIST OF EXPERIMENTS**

Machining and Machining time estimations for:

1. Taper Turning
2. External Thread cutting
3. Internal Thread Cutting
4. Eccentric Turning
5. Knurling
6. Square Head Shaping
7. Hexagonal Head Shaping
8. Fabrication of simple structural shapes using Gas Metal Arc Welding
9. Joining of plates and pipes using Gas Metal Arc Welding/ Arc Welding /Submerged arc welding
10. Preparation of green sand moulds
- 11 Manufacturing of simple sheet metal components using shearing and bending operations.
12. Manufacturing of sheet metal components using metal spinning on a lathe

**TOTAL: 60 PERIODS**

**OUTCOMES:**

**Upon the completion of this course the students will be able to**

**CO1** Demonstrate the safety precautions exercised in the mechanical workshop.

**CO2** Make the work piece as per given shape and size using Lathe.

**CO3** Join two metals using arc welding.

**CO4** Use sheet metal fabrication tools and make simple tray and funnel.

**CO5** Use different moulding tools, patterns and prepare sand moulds.

**LIST OF EQUIPMENT FOR A BATCH OF 30 STUDENTS**

S. NO.	NAME OF THE EQUIPMENT	Qty.
1	Centre Lathes	7 Nos.
2	Horizontal Milling Machine	1No.
3	Vertical Milling Machine	1 No.
4	Shaper	1 No.
5	Arc welding transformer with cables and holders	2 Nos.
6	Oxygen and acetylene gas cylinders, blow pipe and other welding outfit	1 Nos.
7	Moulding table, Moulding equipments	2 Nos.
8	Sheet metal forming tools and equipments	2 Nos.

**LIST OF EXPERIMENTS:**

<b>Ex.No</b>	<b>Name of the Experiment</b>	<b>Hours</b>	<b>PAGE NO</b>
1	Taper Turning Using Compound Rest Method	4	8
2	External Thread cutting	4	10
3	Drilling, Boring and Internal Thread Cutting	4	12
4	Eccentric Turning	4	14
5	Knurling	4	16
6	Square Head Shaping	4	20
7	Hexagonal Head Shaping	4	24
8	Double V-Butt Joint	4	29
9	Lap Joint	4	31
10	T- Fillet Joint	4	33
11	Gear Pattern	4	37
12	Stepped Cone Pulley	4	39
13	Rectangular Tray	4	44
14	Funnel	4	46
15	Metal Spinning	4	48

1	Taper Turning by Tail stock set over method	4	50
2	SLOTING - KEY WAY	4	52
3	SHAPER-V SURFACE	4	54

## **GENERAL SAFETY PRECAUTION**

1. Always wear uniform, shoes and gloves for safety
2. Never operate any machines until you have been instructed properly
3. Always wear goggles to protect your eyes from flying chips.
4. Never hold the job when the machine runs.
5. Never give heavy cut on longer job
6. Never change the gear and belt when the machine is running
7. Always use the correct size spanner and tool for fitting and removing of tool
8. Never try to clear the chips when the machine is running
9. To prevent accident, clean the spilled oil and grease immediately
10. Always clear the area around the machine and machine tool surface
11. No open toed shoes or sandals are allowed in the shop.
12. Do not wear loose clothing. Cotton clothes are best to wear.
13. Remove all jewelry and tie back long hair.
14. Do not operate any machine equipment unless you have been instructed on its proper use and the safety risks involved with the machining operation.
15. Do not leave any machinery or power tools running and unattended.
16. Clean up metal shavings, oil, etc. from machine tools after use; pick up after yourself and return tools to their proper storage area.
17. Don't run, push or surprise other students. No horse play will be tolerated
18. Always listen carefully to the teacher and follow instructions.
19. Know where the emergency stop buttons are positioned in the workshop. If you see an accident at the other side of the workshop you can use the emergency stop button to turn off all electrical power to machines.
20. Always wear an apron as it will protect your clothes and hold loses clothing such as ties in place.
21. Bags should not be brought into a workshop as people can trip over them.
22. Always be patient, never rush in the workshop.
23. Always use a guard when working on a machine.
24. Keep hands away from moving/rotating machinery.
25. Use hand tools carefully, keeping both hands behind the cutting edge.
26. Report any damage to machines/equipment as this could cause an accident.

# I. STUDY OF LATHE MACHINE

Lathe is the machine tool which is used to perform several operations on the work piece. Lathe is useful in making several parts which is further assembled to make new machine. Hence lathe is known as “mother of machines”.

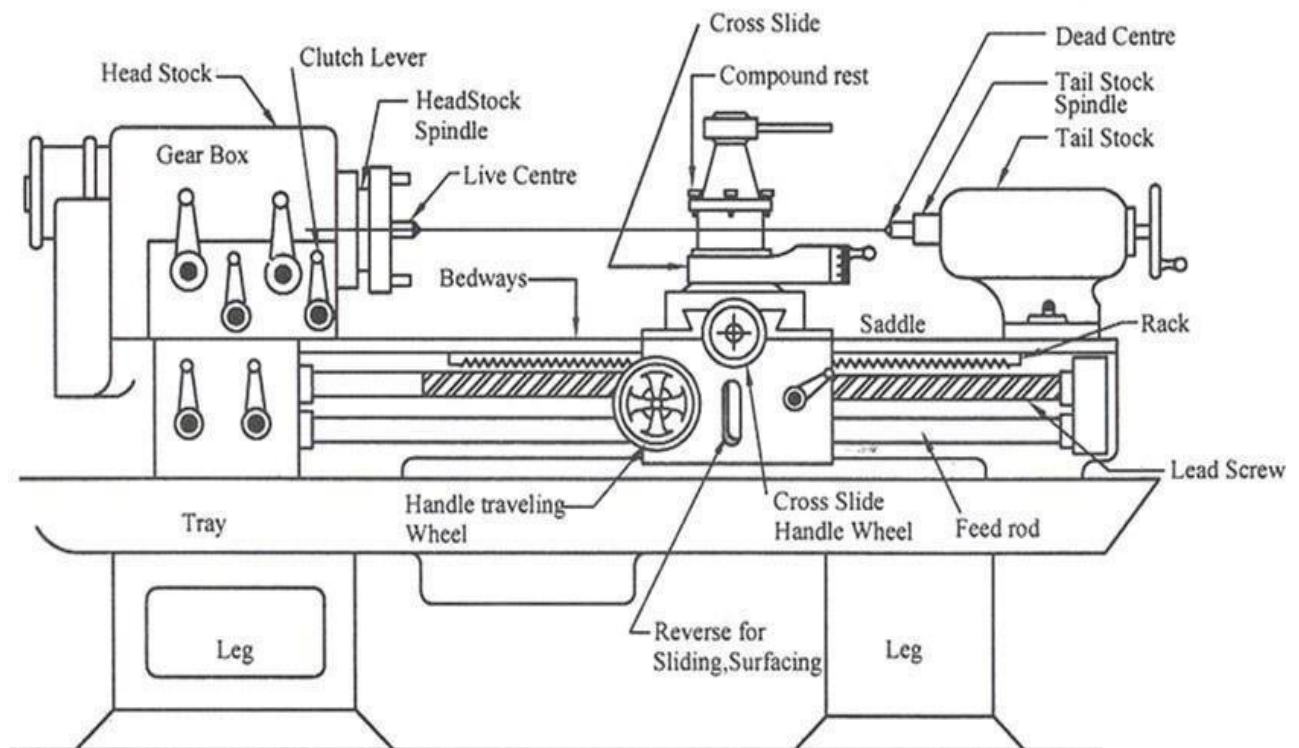
## BASIC WORKING PRINCIPLE OF LATHE

In lathe, the work piece is held in the chuck, a work holding device. The cutting tool is mounted in the tool post. The chuck is rotated by means of power. When the chuck rotates, the work piece also rotates. The tool is moved against the rotating work piece by giving small amount of depth of cut. The material is removed in the form of chips. Continuous feed and depth of cut is given until the required dimensions are obtained in the work piece.

## TYPES OF LATHE MACHINES

There are different types of lathe machines,

1. Centre lathe
2. Tool room lathe
3. Bench lathe
4. Capstan lathe
5. Turret lathe
6. Automatic lathe



## DESCRIPTION OF LATHE

Lathe is a machine tool which has several parts in it. They are

### 1. Bed

It is the base of the machine. On its left side, the head stock is mounted and on its right it has movable casting known as tailstock. Its legs have holes to bolt down on the ground.

### 2. Head stock

It consists of spindles, gears and speed changing levers. It is used to transmit the motion to the job. It has two types one is the headstock driven by belt and the other one is the gear driven.

### 3. Carriage

Carriage is used to carry a tool to bring in contact with the rotating work piece or to withdraw from such a contact. It operates on bed ways between the headstock and tail stock.

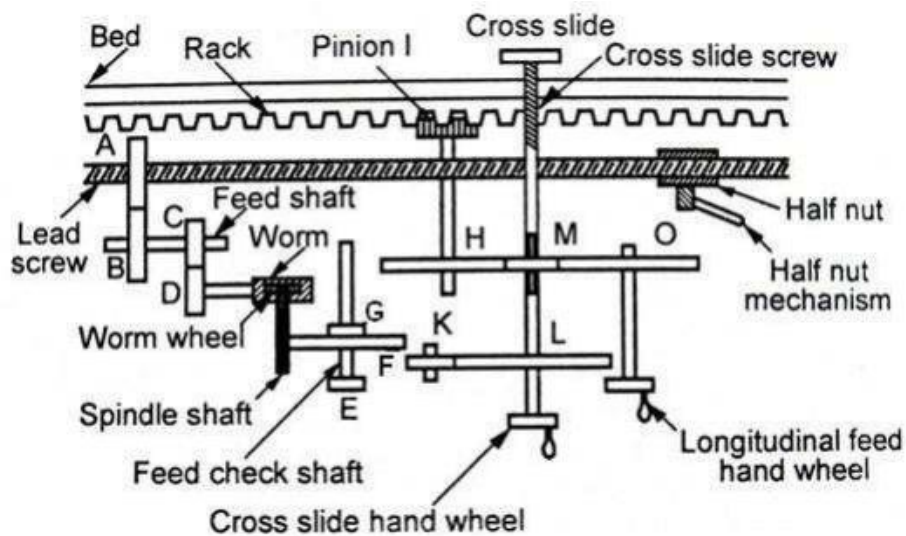
### 4. Saddle

It is an 'H' shaped part fitted on the lathe bed. There is a hand wheel to move it on the bed way. Cross slide, compound rest, tool post is fitted on this saddle.

- a) **Cross slide:** It is on the upper slide of saddle in the form of dove tail. A hand wheel is provided to drive the cross slide. It permits the cross wise movement of the tool (i.e.) movement of tool towards or away from the operator
- b) **Compound rest:** It is fitted over the cross slide on a turn table. It permits both parallel and angular movements to cutting tool.
- c) **Tool post:** It is fitted on the top most part of the compound rest. Tool is mounted on this tool post. Cutting tool is fixed in it with the help of screws.

### 5. Apron

It is the hanging part in front of the carriage. It accommodates the mechanism of hand and power feed to the cutting tool for carrying out different operations.



## 6. Lead screw

It is a long screw with ACME threads. It is used for transmitting power for automatic feed or feed for thread cutting operation.

## 7. Tail stock

It is located at the right end of the lathe bed and it can be positioned anywhere in the bed. It is used for supporting lengthy jobs and also carries tool to carry out operations such as tapping, drilling, reaming.

## WORK HOLDING DEVICES

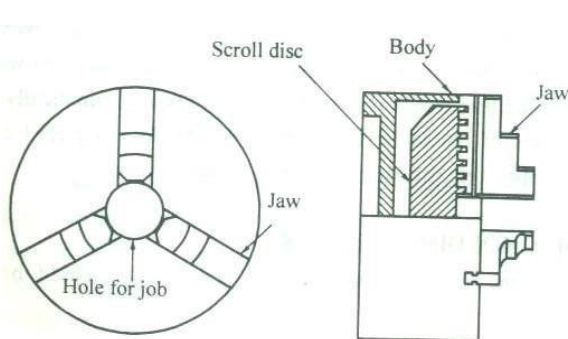
### 1. Lathe centers

They are used to support work. It has two categories of centers. Live center is one which is fitted in the headstock spindle. Dead is another one which is fitted in the tail stock.

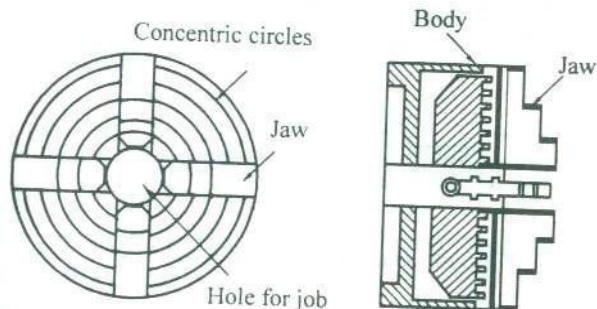
### 2. Chuck

It is a device used to hold a job. It is easily fitted on the thread cut on the end of head stock spindle. Various types of chuck are

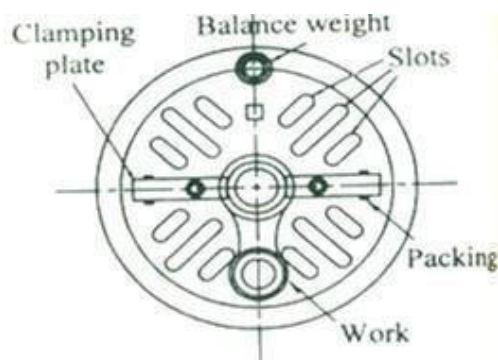
- Three jaw chuck
- Four jaw chuck
- Magnetic chuck



Three Jaw Universal self-centering chuck



Four Jaw Independent chuck

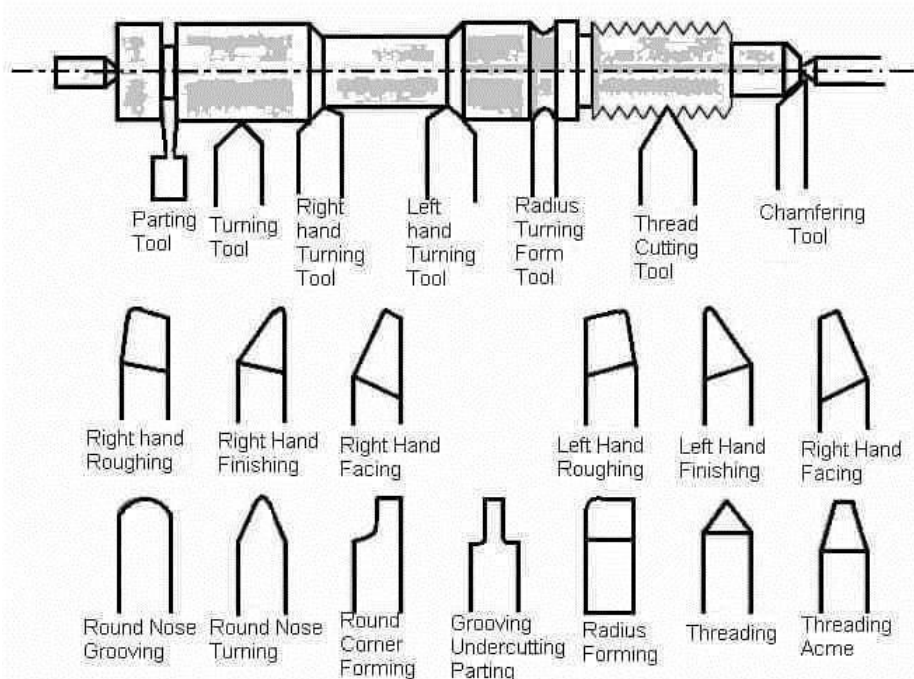


Face Plate

3. Face plate
4. Catch plate
5. Lathe carriers or dog's
6. Steady rest
7. Mandrel
8. Follower rest

### CUTTING TOOLS USED

For making a finished job on lathe machine, various types of cutting tools are used. One of them is single point cutting tool which is used to perform several operations on the work piece. Various types of cutting tools are



### Lathe Machine Cutting Tools

#### 1. Facing Tool

It is used for facing the longitudinal ends of the job. Its shape is like a knife.

#### 2. Rough Turning Tool

It is used to remove excess material from the work piece in quick time. It can be used to give large depth of cut and work at coarse feed.

#### 3. Finishing Tool

It is used for getting smooth finish on the work piece. Its point is a little more round.

#### 4. Radius Tool

Jobs which need round cutting are done with this tool. Its type is

- a) Convex radius tool
- b) concave radius tool



**5. Parting Tools**

It is used to cut the jobs into two parts. It is also used for grooving.

**6. Form Turning Tool**

It is used for jobs which require both convex and concave turning.

**7. Thread Cutting Tool**

It is used for making internal or external threads on the work piece. The tool nose is designed with a definite profile for taking threads.

**8. Drill Tool**

It is used for making holes of various diameters on the job. Drill bit of various sizes of diameter are available.

**9. Boring Tool**

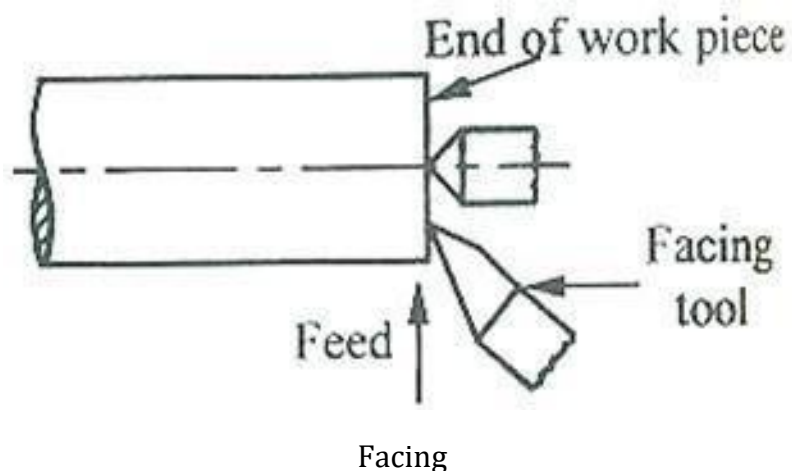
It used for enlarging the drill hole.

**10. Knurling Tool**

Drawing slanting or square projecting lines on the surface of a job is known as knurling. It is used for making better grip on the surface of a job.

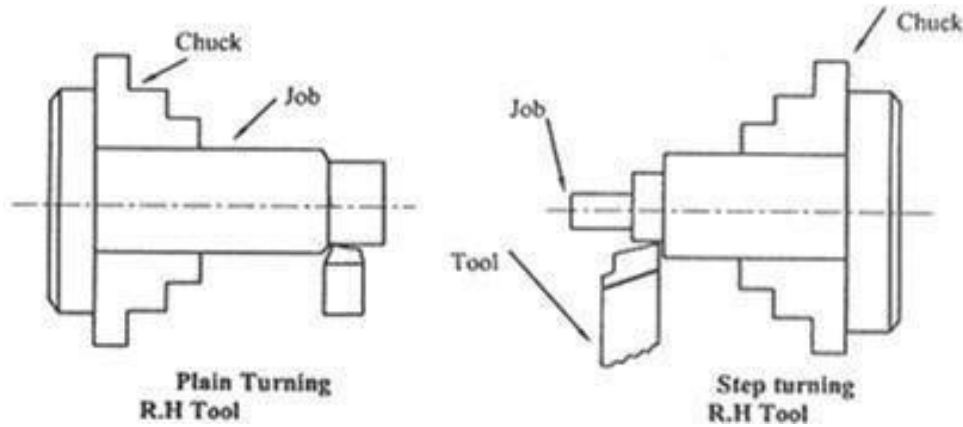
**LATHE OPERATIONS****1. Facing**

- ❖ It is done for getting fine finish (good surface finish) on the face of the job.
- ❖ Facing tool is set at an angle to the work piece.
- ❖ The tool is fed from the centre of work piece towards the outer surface against the rotating work piece.
- ❖ Depth of cut is low for the facing operation.



## 2. PlainTurning

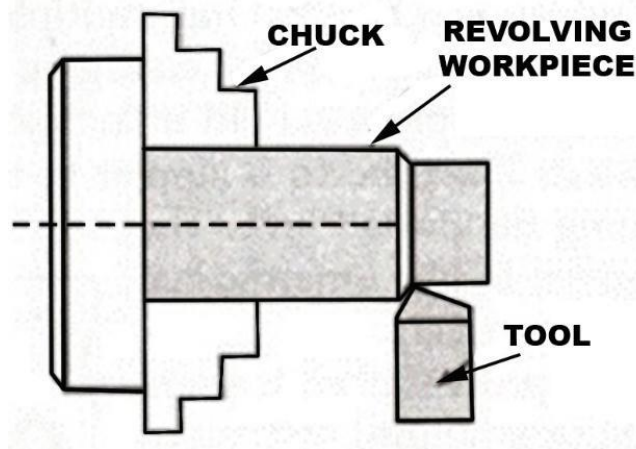
- ❖ It is done for reducing the diameter of the work piece.
- ❖ A cutting tool with  $70^\circ$  setting angle is used for roughing operation.
- ❖ More feed is given for rough turning while less feed is given for finishing.
- ❖ Work piece is held in chuck and tool is set to center height of the work piece.



Plain turning

## 3. Step Turning:

It is similar to the process of turning but in this case different diameter in step of various sizes is taken on the work piece.



Step Turning

**TAPER TURNING USING COMPOUND REST METHOD**

EX.NO:

DATE :

**AIM:**

To produce the component by performing Taper Turning using lathe.

**MATERIALS REQUIRED**

Mild steel  $\Phi$  32 mm X 103 mm long

**TOOLS REQUIRED**

1. Chuck key
2. Steel Rule
3. Tool Post Spanner
4. Single point Cutting tool
5. Vernier Caliper

**FORMULA**

The taper angle is calculated using the following formula:

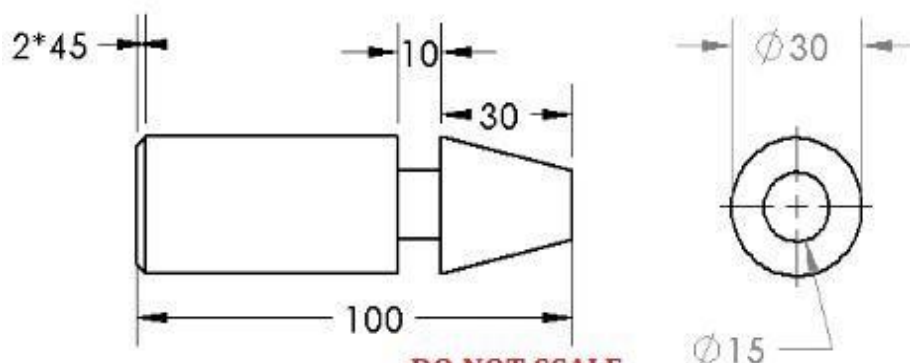
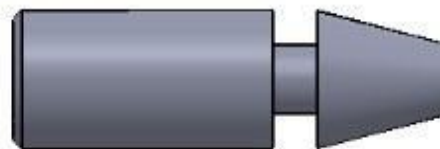
$$\text{Taper angle } (\alpha) = \tan^{-1} \left( \frac{D-d}{2l} \right)$$

Where D = large diameter of taper in mm

d = small diameter of taper in mm

l = length of tapered part in mm

$\alpha$  = angle of taper

**SPECIMEN DRAWING:****TAPER TURNING USING COMPOUND REST**

**DO NOT SCALE  
ALL DIMENSION ARE IN mm**

## PROCEDURE

1. The given work piece is held firmly in a lathe chuck.
2. The cutting tool is set in a tool post such that the point of the Cutting tool coincides with the lathe axis.
3. The machine is switched on to revolve the work piece at the selected speed.
4. By giving Cross feed and longitudinal feed to the cutting tool, the facing and turning operations are done respectively.
5. The compound rest is swiveled for the calculated taper angle.
6. By giving angular feed to the cutting tool through the compound slide the taper turning operation is done.
7. The machine is switched off.
8. The work piece is removed from the chuck and all the dimensions are measured and checked.

**RESULT:** Thus the component is produced as per the given drawing using lathe

**EXTERNAL THREAD CUTTING**

EX.NO

DATE:

**AIM:**

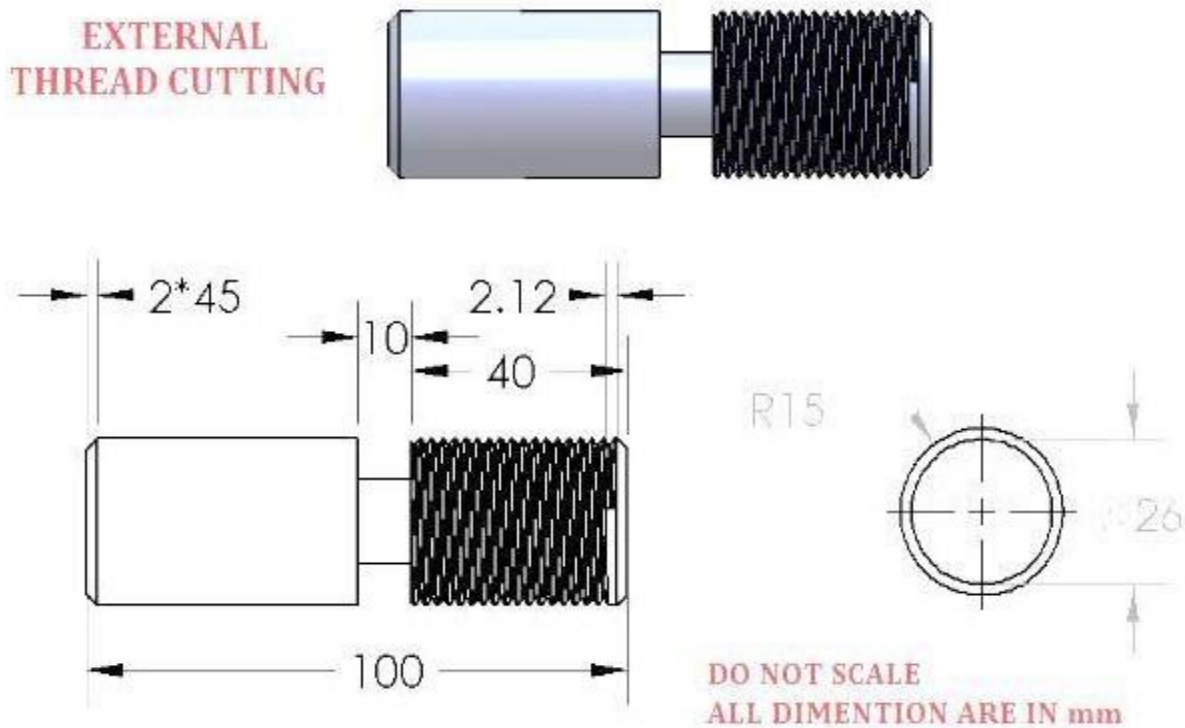
To produce the component by performing threads cutting operation using lathe

**MATERIAL REQUIRED:**

Mild steel  $\Phi$  32 mm X 103 mm long

**TOOLS REQUIRED:**

1. Chuck key
2. Tool post spanner
3. Vernier caliper
4. Steel rule
5. Single point cutting tool
6. V thread cutting tool
7. Pitch gauge
8. Grooving tool
9. Drill chuck with centre drill bit

**SPECIMEN DRAWING:**

**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. Hold the work pieces in the chuck and tight it properly.
3. Clamp the tool on tool post properly and check the centre of work piece with cutting tool tip.
4. Do the facing operation by giving cross feed to the cross slide and reduce the length.
5. Do the turning operation by giving longitudinal feed to the carriage and reduce the length.
6. Two or three pass can be given for the turning operation.
7. Remove the work piece and hold in chuck on the other side and do the turning operation for the remaining work piece.
8. Engage the back gear mechanism to reduce the speed of the workpiece.
9. Check the pitch required, calculate the tumbler gear, and engage it with head stock.
10. Move the carriage towards tail stock and make a mark on the dial.
11. Give small depth of cut and engage the half nut so that the carriage will move automatically towards head stock and cut the v-thread on the work piece.
12. After reaching the required length of thread, release the tool by rotating cross slide in opposite to the depth of cut by counting the number of rotation.
13. Move the carriage towards tail stock, give depth of cut and engage the half nut.
14. Repeat the procedure to produce complete form of v-thread. After finishing remove the work piece from the chuck.

**RESULT:** Thus the component is produced as per the given drawing using lathe.

## DRILLING, BORING AND INTERNAL THREAD CUTTING

EX.NO

DATE:

### AIM:

To produce the component by performing drilling, boring and thread cutting using lathe

### MATERIAL REQUIRED:

Cast iron  $\Phi$  50 mm X 20 mm long

### TOOLS REQUIRED:

1. Chuck key
2. Tool post spanner
3. Vernier caliper & Steel rule
4. Single point cutting tool & Boring tool
5. Drill chuck with drill bits
6. Internal thread cutting tool

### SPECIMEN DRAWING:



**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. Hold the work piece in the chuck and tight it properly.
3. Clamp the tool on tool post properly and check the Centre of work piece with cutting tool tip.
4. Do the facing operation by giving cross feed to the cross slide and reduce the length.
5. Hold the drill bit on the tail stock and unclamp the tail stock, then move it towards the head stock and ensure the Centre coincidence with work piece Centre.
6. Start the motor and give depth of cut to the drill bit by rotating the tail stock wheel and make complete hole.
7. Clamp the boring tool on the tool post to perform the boring operation.
8. Clamp the internal thread tool on the tool post to perform internal threading operation,
9. Engage the back gear mechanism to reduce the speed of the workpiece.
10. Check the pitch required, calculate the tumbler gear and engage it with head stock.
11. Move the carriage towards tailstock and make marking on the dial.
12. Give small depth of cut and engage the half nut so that the carriage will move automatically towards headstock and cut the v thread on the workpiece.
13. Release the tool by rotating cross slide in opposite to the depth of cut by counting number of rotation after reaching the required length of thread.
14. Move the carriage toward the tail stock and give depth of cut and engage the half nut.
15. Repeat the procedure to produce complete form of v thread. After finishing remove the work piece from the chuck.

**RESULT:** Thus the component is produced as per the given drawing using lathe



## ECCENTRIC TURNING

EX.NO

DATE:

### AIM:

To turn a given work piece eccentrically in a lathe machine as per the given dimensions

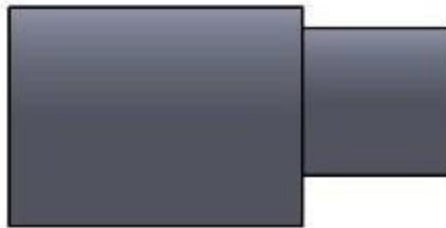
### MATERIAL REQUIRED:

Mild steel  $\Phi$  32 mm X 62 mm long

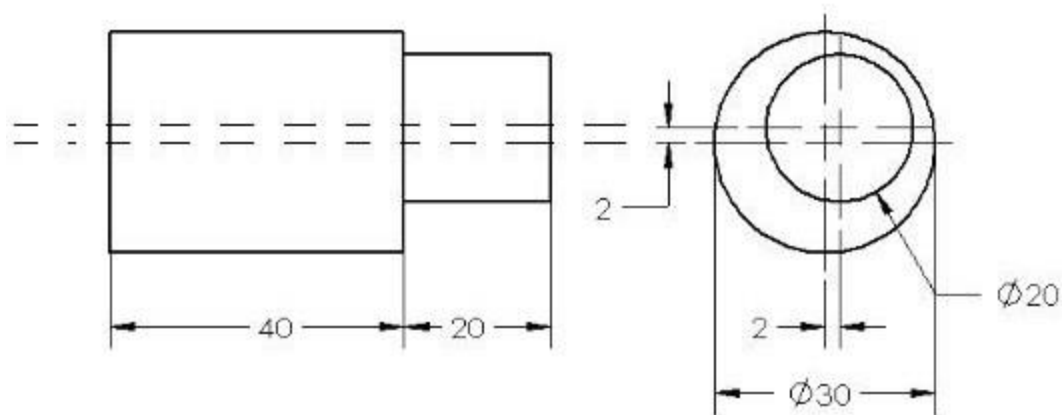
### TOOLS REQUIRED:

1. Steel rule
2. Dot punch
3. Ball pen hammer
4. Single point cutting tool
5. Surface gauge
6. Drill bit
7. Drill chuck
8. Boring tool

### SPECIMEN DRAWING:



### ECCENTRIC TURNING



DO NOT SCALE  
ALL DIMENSION ARE IN mm

**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. The work piece is fixed in the four jaw chuck using surface gauge.
3. The cutting tool is fixed in the tool post to the lathe axis.
4. Facing operation has been done on both sides of given work pieces.
5. Marking and punching is to be done on the given work pieces as per the dimension.
6. Then the work piece is fixed in the chuck such that the offset point coincides with spindle axis.
7. Using the turning tool, turning operation has been performed.
8. The same procedure is also carried out to the other side of the work piece for the given offset distance.
9. Thus the eccentric turning would be finished.

**RESULT:** Thus the work piece is turned eccentrically as per the given dimensions.

**KNURLING**

EX.NO:

DATE

**AIM:**

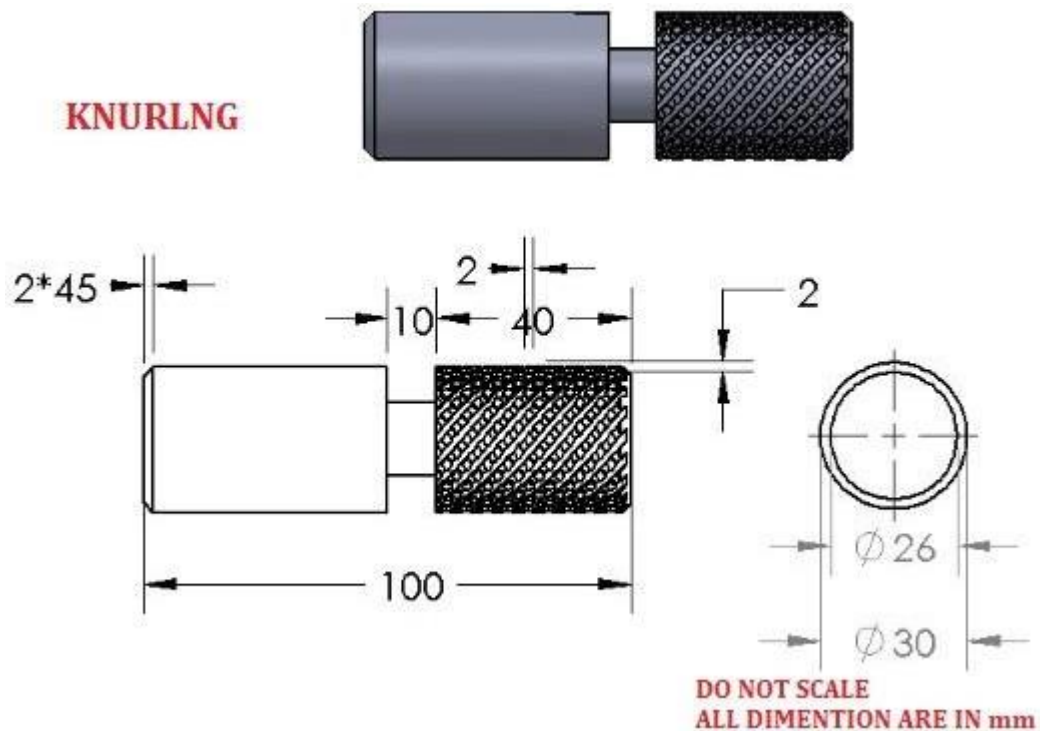
To machine a work piece by facing, plain turning and knurling operation using a lathe.

**MATERIALS REQUIRED**

Mild steel  $\Phi$  32 mm X 103 mm long

**TOOLS REQUIRED**

1. Knurling Tool
2. Chuck Key
3. Steel Rule
4. Tool Post Spanner
5. Single point Cutting tool
6. Vernier Caliper

**SPECIMEN DRAWING:**

**PROCEDURE**

1. The given work piece is held firmly in a lathe chuck.
2. The cutting tool is set in a tool post such that the point of the Cutting tool coincides with the lathe axis.
3. The machine is switched on to revolve the work piece at the selected speed.
4. By giving Cross feed and longitudinal feed to the cutting tool, the facing and turning operations are done respectively.
5. The speed of the work piece is reduced.
6. The knurling operation is done using knurling tool.
7. The machine is switched off.
8. The work piece is removed from the chuck and all the dimensions are measured and checked.

**RESULT:** Thus the component is produced as per the given drawing using lathe

## II. STUDY OF SHAPER MACHINE

### INTRODUCTION:

Shaper is a versatile machine which is primarily intended for producing flat surfaces. These surfaces may be horizontal, vertical or inclined. This machine involves the use of a single point tool held in a properly designed tool box mounted on a reciprocating ram. It is almost an indispensable machine in tool rooms, die making shops and general repair shops, where only a very few job of identical shapes are produced. If properly handled, it can be safely adopted for producing and irregular surfaces also.

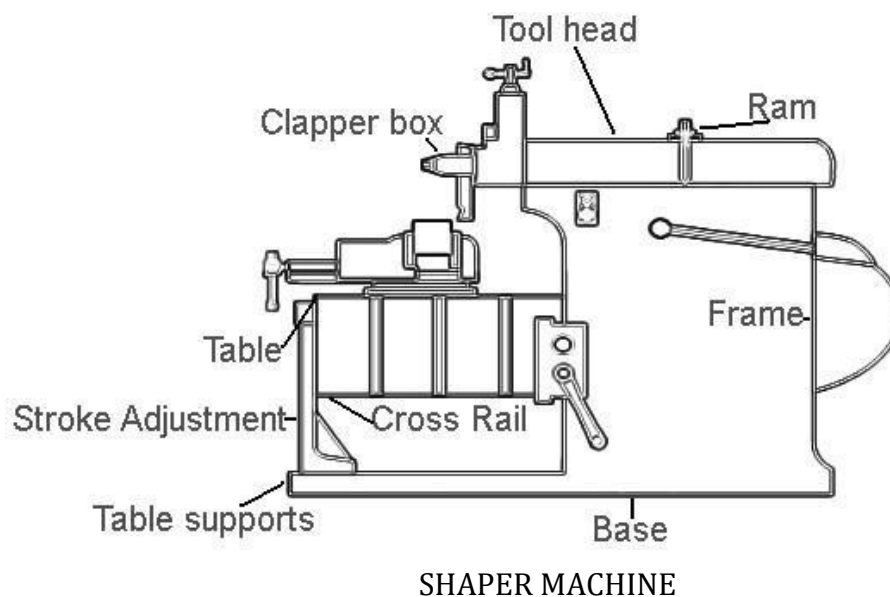
The shaper is a machine tool used primarily for:

1. Producing a flat or plane surface which may be in a horizontal, a vertical or an angular plane.
2. Making slots, grooves and keyways
3. Producing contour of concave/convex or a combination of these.

### WORKING PRINCIPLE OF A SHAPER:

The job is rigidly fixed on the machine table. The single point cutting tool held properly in the tool post is mounted on a reciprocating ram. The reciprocating motion of the ram is obtained by a quick return motion mechanism. As the ram reciprocates, the tool cuts the material during its forward stroke. During return, there is no cutting action and this stroke is called the idle stroke. The forward and return strokes constitute one operating cycle of the shaper.

In case of shaper, the job is rigidly held in a suitable device like a vice or clamped directly on the machine table. The tool is held in the tool post mounted on the ram of the machine. This ram reciprocates to and fro and is doing so, makes the tool to cut the material in the forward stroke. No cutting of material takes place during the return stroke of the ram. Hence, it is termed as „Idle“ stroke. However, in case of a draw-cut shaper, the cutting takes place in the return stroke and the forward stroke in an idle stroke. The job is given an indexed feed (equal amount after each cut) in a direction normal to the line of action of the cutting tool.



**PARTS OF A SHAPER:**➤ **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 rail axis. It is mounted on the front vertical guide ways of the column. It consists 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.

➤ **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**

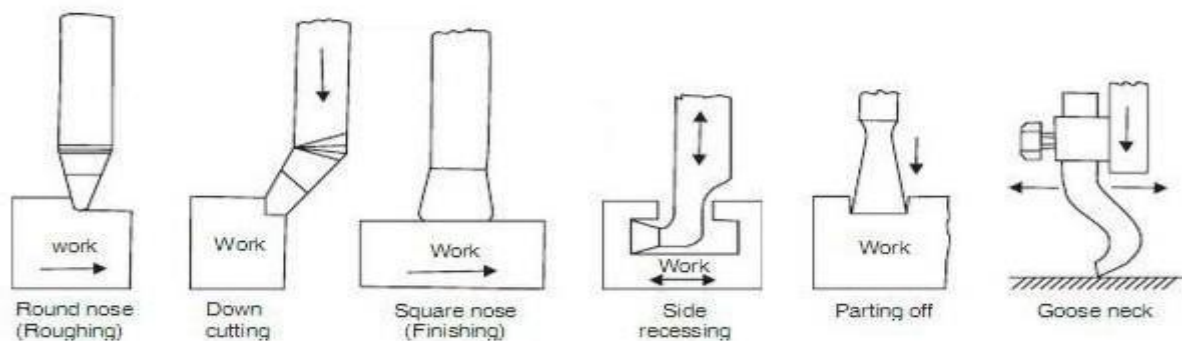
The table is a box like casting having T -slots both on the top and sides for clamping the work. It is bolted to the saddle and receives crosswise and vertical movements from the saddle and cross rail.

➤ **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.

**CUTTING TOOLS USED IN SHAPER:**

The cutting tools for shaper are generally made up of H.S.S or with braze tips. Due the interrupted cuts, tungsten carbide tools are not preferred for shaping work. These tools are made sturdy with fairly generous size for shank and tip. Various types of tools useful for shaping are show in Fig.



CUTTING TOOLS

## SQUARE HEAD SHAPING

EX.NO:

DATE :

**AIM:**

To make round work piece to square by using shaper.

**MATERIALS REQUIRED**

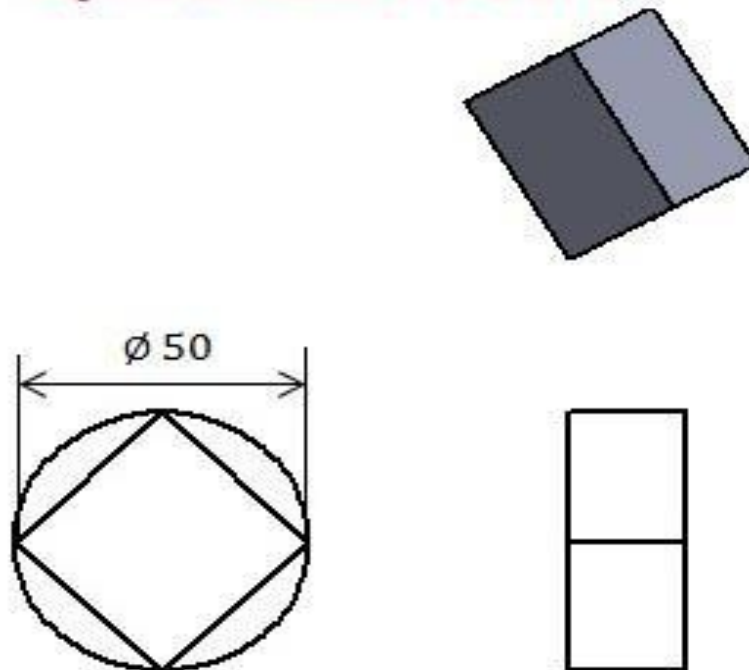
Cast iron  $\Phi$  50 mm X 15 mm long

**TOOLS REQUIRED**

1. Steel rule
2. Dot punch
3. Hammer
4. Vernier height gauge
5. T-square
6. Scriber
7. Single point cutting tool
8. Tool holder.

**SPECIMEN DRAWING:**

### SQUARE HEAD SHAPING



DO NOT SCALE  
ALL DIMENSION ARE IN mm

**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. The work piece is fixed in the three-jaw chuck.
3. The cutting tool is fixed in the tool post to the lathe axis.
4. Facing operation has been done on both sides of given work pieces.
5. Chalk is applied uniformly on the face of the work piece.
6. Marking and punching is to be done on the given work pieces as per the dimension.
7. The cutting tool is fixed in the tool holder in shaper machine.
8. The work piece is fixed in the machine vice, then switch on the machine.
9. At the end of each stroke give the feed to the work piece.
10. Shaping operation has been done on four sides in the work piece till the round work piece is turned into square shaped as per the given dimension.

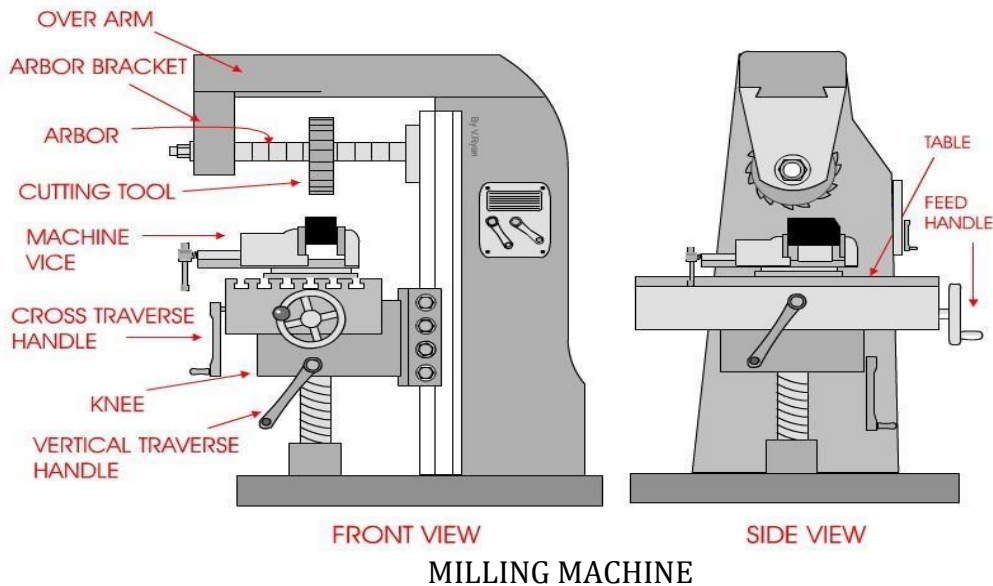
**RESULT:** Thus the round work piece is machined to square work piece by shaper machine.



### III. STUDY OF MILLING MACHINE

#### INTRODUCTION:

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. Milling is a cutting process that uses a milling cutter to remove material from the surface of a work piece. The milling cutter is a rotary cutting tool, often with multiple cutting points.



#### Two major classes of milling process:

1. In **face milling**, the cutting action occurs primarily at the end corners of the milling cutter. Face milling is used to cut flat surfaces (faces) into the work piece, or to cut flat-bottomed cavities.
2. In **peripheral milling**, the cutting action occurs primarily along the circumference of the cutter.

#### Milling Machine Parts and their Function:

##### 1. BASE:

It is a heavy casting provided at the bottom of the machine. It is accurately machined on both the top and bottom surfaces. It actually acts as a load bearing member for all other parts of the machine. Also it carries the screw jack which supports and moves the knee. In addition to this it also serves as reservoir for the coolant.

##### 2. COLUMN:

It is a very prominent part of the milling machine and is produced with enough care. On the front face of the column are made the vertical parallel ways in which the knee slides up and down. It carries the enclosed motor drive. Top of the column carries dovetail horizontal ways for the over arm.

##### 3. KNEE:

It is a rigid casting, which is capable of sliding up and down along the vertical ways on the front face of the column. This enables the adjustment of the table height. The adjustment is provided by operating the elevating jack, provided below the knee, by means of hand wheel or application of power feed. For efficient operation of the machine, rigidity of the knee and accuracy of its ways play an important role.

## 4. SADDLE:

It is the intermediate part between the knee and the table and acts as a support for the latter. It can be adjusted crosswise, along the ways provided on the top of the knee, to provide cross feed to the table. As its top, it carries horizontal ways, along which moves the table during longitudinal traverse.

## 5. TABLE:

It acts as a support for the work. It is made of cast iron, with its top surface accurately machined. Its top carries longitudinal cross T-slots to accommodate the clamping bolts for fixing the work or securing the fixtures. Also, the cutting fluid, after it is used, drains back to the reservoir through these slots and then the pipe fitted for this purpose. Longitudinal feed is provided to it by means of hand wheel fitted on one side of the feed screw.

## 6. OVERARM:

It is the heavy support provided on the top of both plain and universal milling machine. It can slide horizontally, along the ways provided on the top of the column, and adjusted to a desired position in order to provide support to the projecting arbor.

### Milling processes

The various milling processes may be grouped under two headings:

1. Peripheral milling
2. Face milling

**1. Peripheral milling:** The machining is performed by the cutting edges on the periphery of the milling cutter. It is classified under two headings

- a. Up milling
- b. Down milling

➤ **Up milling:** In this method, the work piece mounted on the table is fed against the direction of rotation of the milling cutter. The cutting force is minimum during the beginning of the cut and maximum at the end of cut. The thickness of chip is more at the end of the cut. As the cutting force is directed upwards, it tends to lift the work piece from the fixtures.

➤ **Down milling:** The work piece mounted on the table is moved in the same direction as that of the rotation of the milling cutter. The cutting force is maximum at the beginning and minimum at the end of cut. The chip thickness is more at the beginning of the cut.

### 3. Face milling

Face milling is the milling of surfaces that are perpendicular to the cutter axis. Face milling produces flat surfaces and machines work to the required length. In face milling, the feed can be either horizontal or vertical. In face milling, the teeth on the periphery of the cutter do practically all of the cutting. However, when the cutter is properly ground, the face teeth actually remove a small amount of stock which is left as a result of the springing of the work piece or cutter, thereby producing a finer finish.

**HEXAGONAL HEAD SHAPING**

EX.NO:

DATE :

**AIM:**

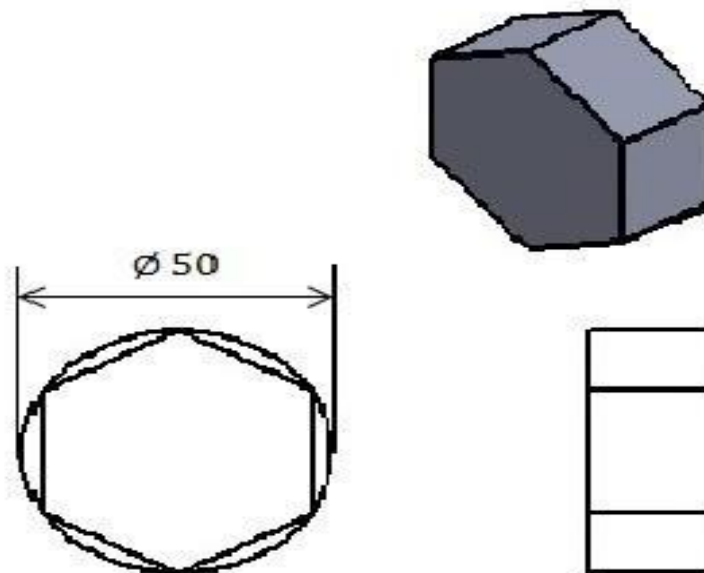
To make a round work piece to hexagonal shape by using milling.

**MATERIALS REQUIRED**

Cast iron  $\Phi$  50 mm X 15 mm long

**TOOLS REQUIRED**

1. Steel rule
2. Dot punch
3. Hammer
4. Verneir height gauge
5. T-square
6. Scriber
7. Surface plate
8. Single point cutting tool
9. Tool holder.

**SPECIMEN DRAWING:****HEXAGONAL HEAD SHAPING**

**DO NOT SCALE**  
**ALL DIMENTION ARE IN mm**

**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. The work piece is fixed in the three jaw chuck.
3. The cutting tool is fixed in the tool post to the lathe axis.
4. Facing operation has been done on both sides of given work pieces.
5. Chalk is applied uniformly on the face of the work piece.
6. Marking and punching is to be done on the given work pieces as per the dimension.
7. The end mill cutter is mounted in the spindle
8. The work piece is fixed in the machine vice, then switch on the machine.
9. The spindle is switched on and the material is removed on one face.
10. Similarly by changing the face, the other sides of the work piece is machined.
11. The job is removed from the machine vice and checked for dimensions using Vernier.
12. Shaping operation has been done on six sides in the work piece till the round work piece is turned into hexagonal shaped as per the given dimension.

**RESULT:** Thus the round work piece is machined to hexagonal shape by milling machine.

## IV. WELDING

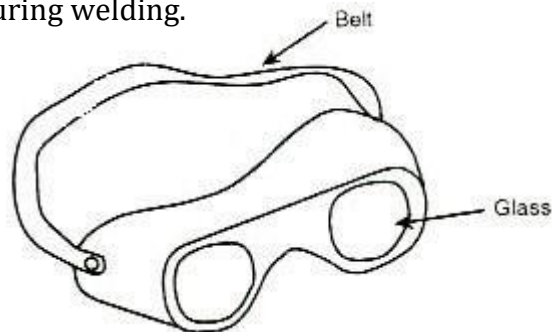
### INTRODUCTION:

Welding is a metal joining process wherein localized coalescence is produced either by heating the metal to a suitable temperature, with or without the use of filler metal, with or without application of pressure. The filler material has similar compositional melting point temperature as that of the base metal. It is used to fill gap between the join surfaces

### SAFETY EQUIPMENTS AND TOOLS:

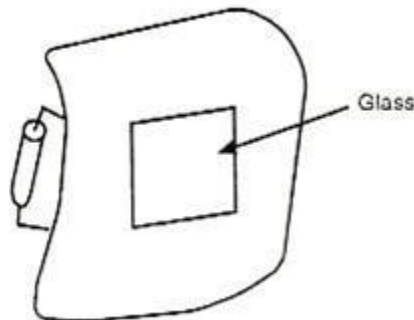
#### 1 Welding goggles:

Goggles with glasses are used to protect the eyes of the eyes of the welder from the light sparks produced during welding.



#### 2 Face shield:

A face shield is also used to protect the eyes of the welder from the light sparks produced during welding. It is normally held in hand

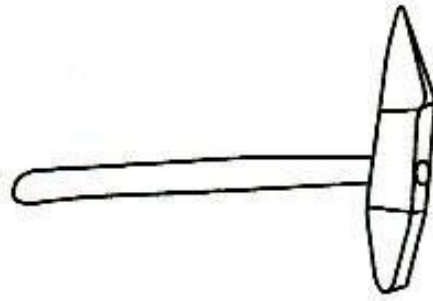


#### 3 Hand gloves:

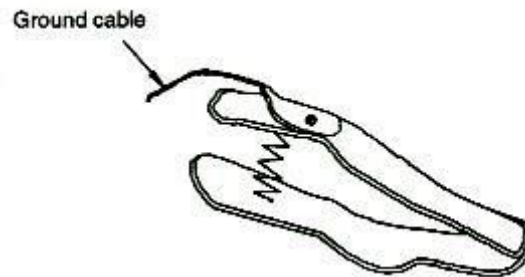
It is used to protect the hands of the welder from the effect of ultra violet rays, infra red rays, heat and sparks.

**4 Chipping Hammer:**

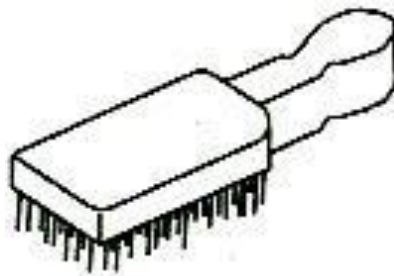
A chipping hammer is used to remove slags which from during welding.

**5 Ground Clamp:**

It is connected to the end of the ground cable. It is normally clamped to the welding table or the job itself to complete the electric circuit.

**6 Wire brush:**

The wire brush is used to clean the surface to be welded

**Types of flames:**

1. Neutral flame ( oxygen, acetylene in equal proportions )
2. Oxidizing flame ( excess of oxygen )
3. Reducing flame (excess of acetylene )

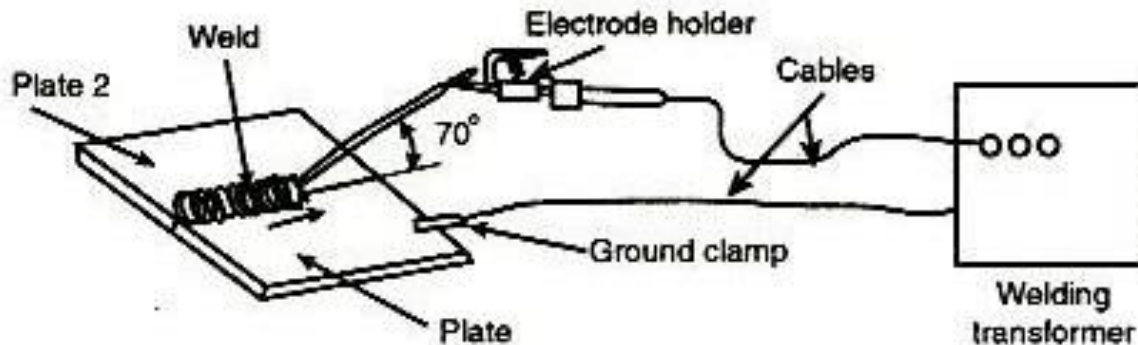
**TYPES OF WELDING:**

The two types of welding most prevalently in use are

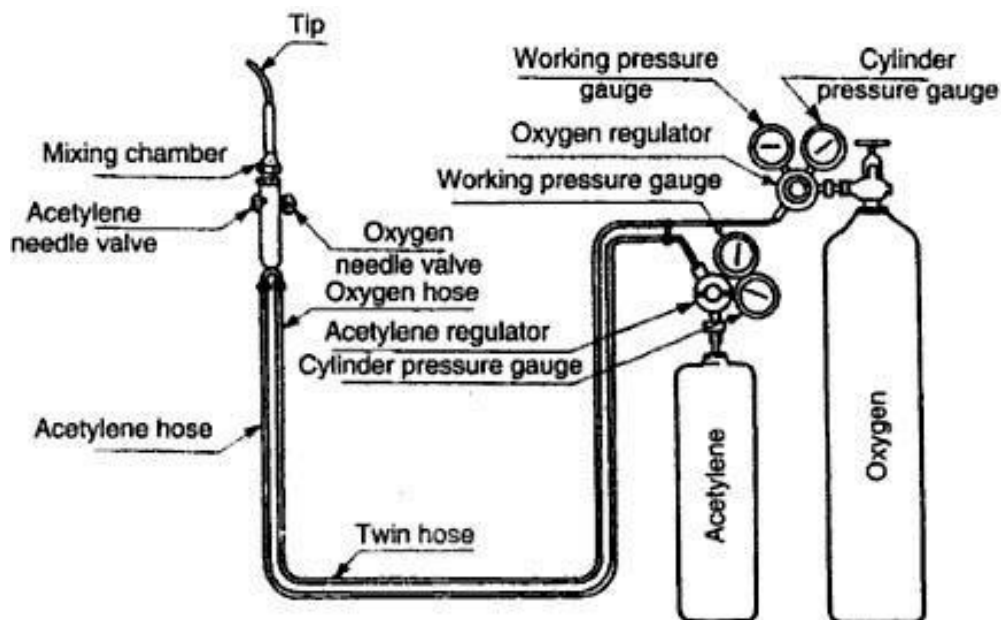
- a. Arc welding
- b. Gas Arc welding

**a. Arc welding:**

Electric arc welding is widely used to join metal plates using a filler rod. The filler rod (welding rod) melts on the electric arc produced and welds the metal plates.

**b. Gas welding:**

Gas welding is a process in which the required heat to melt the surfaces is supplied by a high temperature flame obtained by a mixture of two gases. Usually the mixture of oxygen and acetylene is used for welding purpose. The filler rod and parent metal plates are melted by the heat of the flame produced using oxygen and acetylene gas mixture. Gas welding is also widely used to join metal plates.



oxy-acetylene gas welding.

## DOUBLE V-BUTT JOINT

EX.NO:

DATE :

**AIM:**

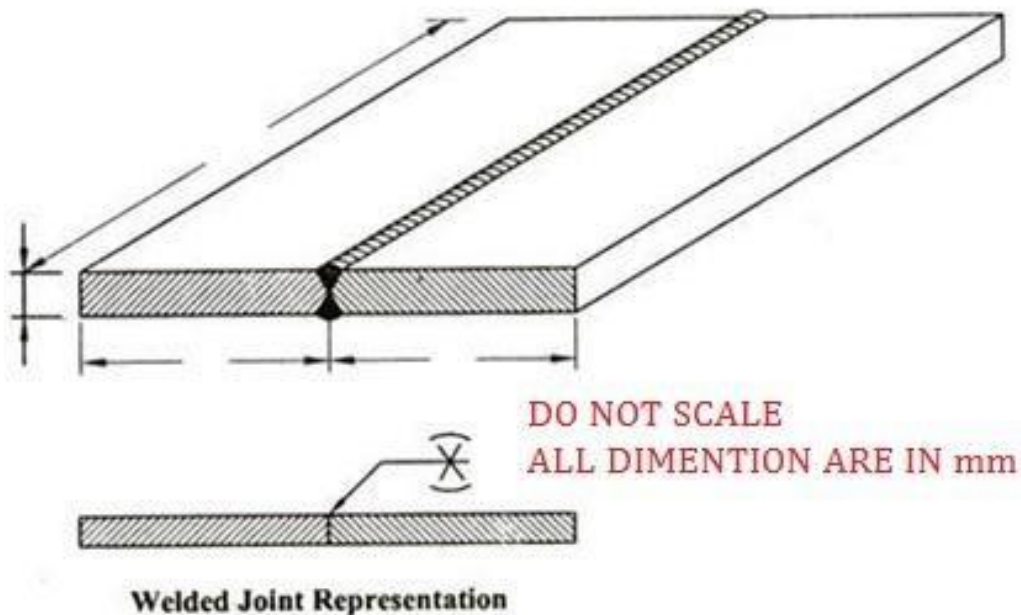
To make a Double V-Butt joint using arc welding on the given work pieces.

**MATERIALS REQUIRED**

Mild steel plate of size 100 x 50 x 6mm - 2 Nos.

**TOOLS REQUIRED**

1. Power supply (AC or DC)
2. Welding Torch
3. Electrodes
4. Tongs
5. Chipping hammer
6. Wire brush
7. Gloves
8. Apron
9. Shield or Safety goggles
10. Earthing clamps

**SPECIMEN DRAWING:**



**SEQUENCE OF OPERATIONS:**

1. Edge preparation (Removal of rust, scale etc.,)
2. Tacking
3. Welding
4. Cooling
5. Chipping
6. Cleaning

**PROCEDURE:**

1. First of all, the work pieces must be thoroughly cleaned to remove rust, scale and other foreign materials.
2. Then the given work pieces are placed on the table in such a way that two work pieces are brought close to each other so that it forms a '**V-Shape**' bough side when the plates butt each other.
3. Appropriate power supply should be given to the electrode and work pieces.
4. Now the welding current output may be existed.
5. When current is passed, arc is produced between the electrode and work pieces.
6. Now set the two work pieces in correct position and maintain the gap 3mm and drack at both ends of the work pieces as shown in the figure.
7. Then the welding is carried out throughout the length.
8. As soon as the welding process is finished, switched off the current supply and allow the work piece to cool.
9. Slags are removed by chipping process with the help of chipping hammer.
10. Finally using wire brush, welded portions are cleaned.

**RESULT:** Thus the desired Double V-Butt joint is obtained using arc welding.

## LAP JOINT

EX.NO:

DATE :

AIM:

To make a lap joint using arc welding on the given work pieces..

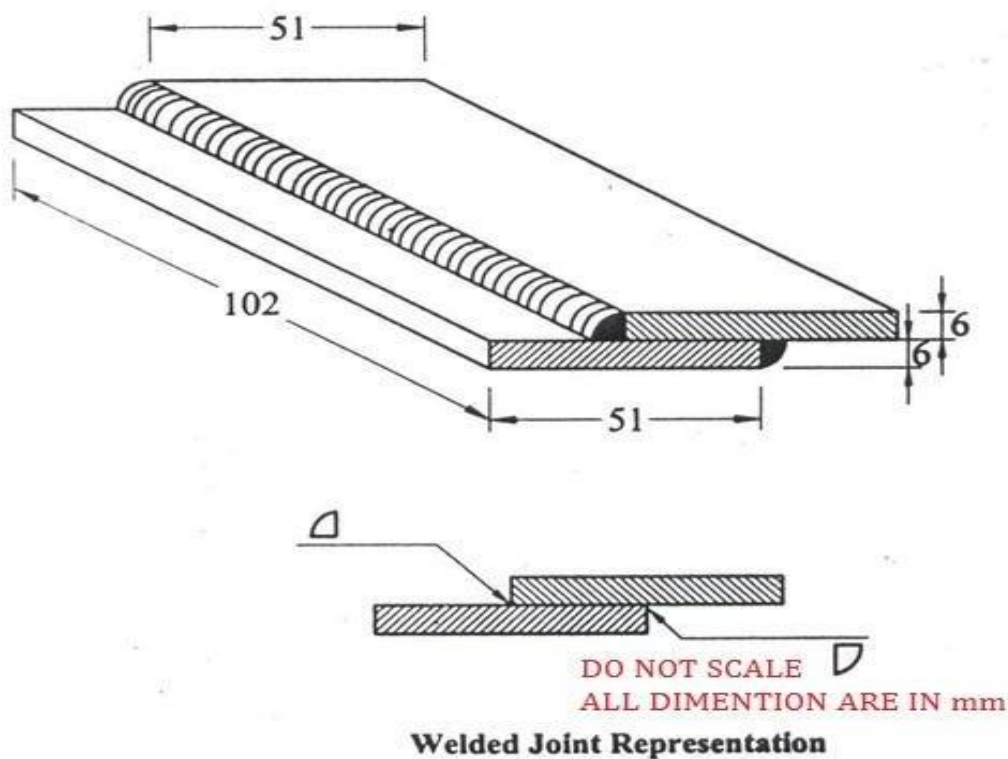
### MATERIALS REQUIRED

Mild steel plate of size 100 x 50 x 6mm - 2 Nos.

### TOOLS REQUIRED

1. Power supply (AC or DC)
2. Welding Torch
3. Electrodes
4. Tongs
5. Chipping hammer
6. Wire brush
7. Gloves
8. Apron
9. Shield
10. Safety goggles
11. Earthing clamps

### SPECIMEN DRAWING:



**SEQUENCE OF OPERATIONS:**

1. Edge preparation (Removal of rust, scale etc.,)
2. Tacking
3. Welding
4. Cooling
5. Chipping
6. Cleaning

**PROCEDURE:**

1. First of all, the work pieces must be thoroughly cleaned to remove rust, scale and other foreign materials.
2. Then the given work pieces are placed on the table in such a way that two work pieces are overlapped one over another as shown in fig.
3. Appropriate power supply should be given to the electrode and work pieces.
4. Now the welding current output may be existed.
5. When current is passed, arc is produced between the electrode and work pieces.
6. Set the two work pieces in correct position like lap joint and tack at both ends of the work pieces as shown in the figure.
7. Then the welding is carried out throughout the length of the work piece.
8. As soon as the welding process is finished, switch off the current supply and allow the work piece to cool.
9. Slags are removed by chipping process with the help of chipping hammer.
10. Finally using wire brush, welded portions are cleaned.

**RESULT:** Thus the desired lap joint is obtained using arc welding.

## T-FILLET JOINT

EX.NO:

DATE :

AIM:

To make a T-Fillet joint using arc welding on the given work pieces..

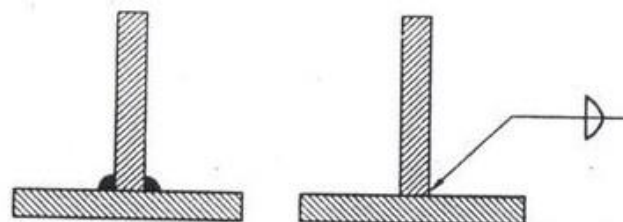
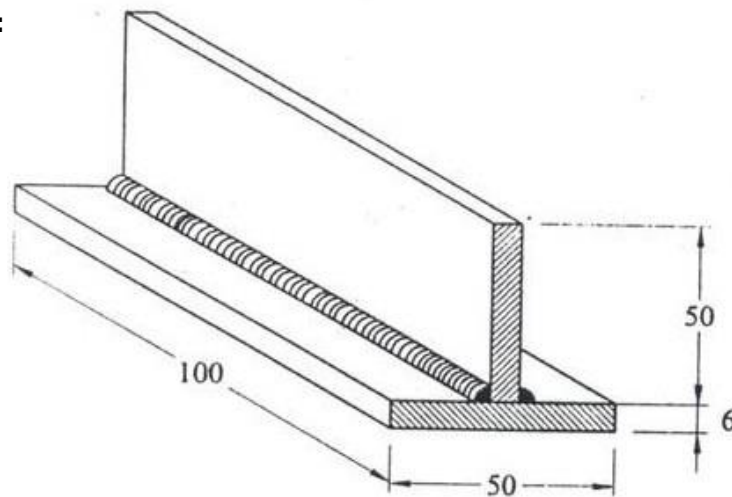
### MATERIALS REQUIRED

Mild steel plate of size 100 x 50 x 6mm - 2 Nos.

### TOOLS REQUIRED

1. Power supply (AC or DC)
2. Welding Torch
3. Electrodes
4. Tongs
5. Chipping hammer
6. Wire brush
7. Gloves
8. Apron
9. Shield
10. Safety goggles
11. Earthing clamps

### SPECIMEN DRAWING:



Welded Joint Representation

DO NOT SCALE  
ALL DIMENTION ARE IN mm

**SEQUENCE OF OPERATIONS:**

1. Edge preparation (Removal of rust, scale etc.,)
2. Tacking
3. Welding
4. Cooling
5. Chipping
6. Cleaning

**PROCEDURE:**

1. First of all, the work pieces must be thoroughly cleaned to remove rust, scale and other foreign materials.
2. Then the given work pieces are placed on the table in such a way that two work pieces are brought close to each other so that it forms a 'T' shape is shown in the fig.
3. Appropriate power supply should be given to the electrode and work pieces.
4. Now the welding current output may be existed.
5. When current is passed, arc is produced between the electrode and work pieces.
6. Set the two work pieces in correct position like T-Fillet joint and tack at both ends of the work pieces as shown in the figure.
7. The joint is placed on a welding table in a flat position by keeping the tack side down.
8. Then the welding is carried out throughout the length.
9. As soon as the welding process is finished, switched off the current supply and allow the work piece to cool.
10. Slags are removed by chipping process with the help of chipping hammer.
11. Finally using wire brush, welded portions are cleaned.

**RESULT:** Thus the desired T-Fillet joint is obtained using arc welding.

**INTRODUCTION:****V. FOUNDRY**

Producing components by casting has been since the earliest day of civilization. Lot of shape and sizes can be prepared in a casting process. To make the casting of components, a cavity of desired shape made in moulding sand or other material. The process of moulding consists of all operations done to make a mould.

**Pattern:** Pattern is the model used to get required casting. It is used to produce to mould cavity in sand.

**Foundry:** The place where moulding and castings are done is known as foundry.

**Moulding sand or Green sand:** It is mixture of sand and additives such as water, bentonite, inoculant, sodium silicate etc., used to create mould cavity.

**Components required for moulding**

The following components are essential for producing mould, Moulding Sand (Green sand), Moulding Boxes, Pattern, Moulding tools.

**I. Moulding Sand**

It is a special type of sand used for making mould. Moulding sand has three constitues. They are Sand, Binder, Additive.

**Properties of moulding sand**

A good moulding sand must have the following properties.

- Porosity or permeability
- Plasticity
- Adhesiveness
- Cohesiveness
- Refractoriness

**II. Moulding Boxes**

Moulding box is also called moulding flask. It is a frame or box of wood or metal. Wood is cheaper boxes made quickly. Wood wears out quickly. It is destroyed by contact with hot metal. Metal boxes in steel, cast iron and aluminum alloys are used in mass production. Moulding boxes are used for making sand moulds. Moulding flasks may have two or more parts. The main types of flasks are:

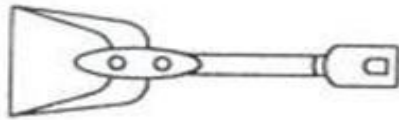
- a) Snap Flask
- b) Tight or Box Flash

**III. Patterns**

A pattern is the replica of the desired casting used to produce a mould cavity into which liquid metal is poured. When patter packed in a suitable material produces a cavity called the mould. This cavity when filled with molten metal produces the desired casting.

**IV. Moulding tools**

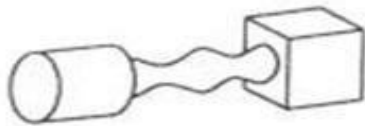
- |                   |                 |
|-------------------|-----------------|
| 1) Shovel         | 8) Bellows      |
| 2) Riddle         | 9) Swab         |
| 3) Rammer         | 10) Gate cutter |
| 4) Trowels        | 11) Draw spike  |
| 5) Slick          | 12) Lifter      |
| 6) Strike off bar | 13) Vent rod    |
| 7) Sprue pin      |                 |



Shovel



Riddle



Rammer



Trowel



Slick



Lifter



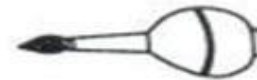
Strike-off Bar



Sprue Pin



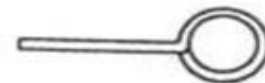
Bellow



Brush



Gate Cutter



Vent Rod



Draw Spike

## GEAR PATTERN

EX.NO:

DATE :

**AIM:**

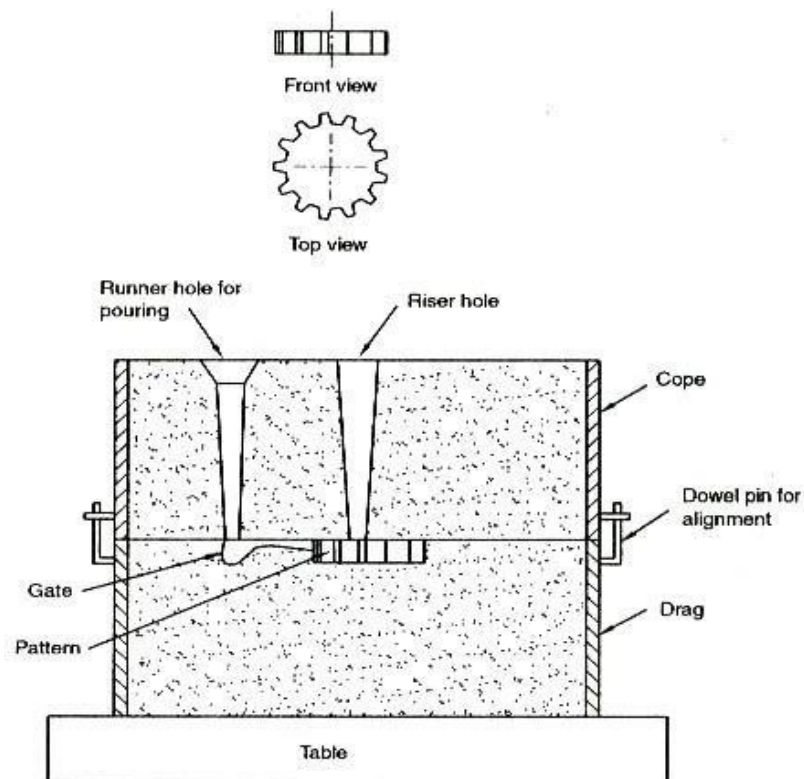
To prepare a sand mould using a gear pattern

**MATERIALS REQUIRED**

Moulding sand, Parting sand, Facing sand, Gear pattern, Moulding boxes etc.

**TOOLS REQUIRED**

1. Showel
2. Sieve
3. Trowel
4. Rammers
5. Spure pin
6. Strike off bar
7. Lifter
8. Gate cutter
9. Runner
10. Riser
11. Vent rod
12. Draw spike
13. Bellow

**SPECIMEN DRAWING:**



**SEQUENCE OF OPERATIONS:**

1. Sand preparation.
2. Core preparation.
3. Placing the pattern on moulding board.
4. Ramming of drag.
5. Placing runner and riser.
6. Ramming of cope.
7. Removal of pattern, runner, riser.
8. Gate cutting.

**PROCEDURE:**

1. The gear pattern is placed on the moulding board.
2. A suitable core is prepared and placed in the hole of gear pattern.
3. Clay washing is done inside the drag surface.
4. Parting sand is applied over the pattern.
5. Box is filled with smooth moulding sand and proper ramming is done using flat and peen rammers.
6. Excess sand is removed using the strike off bar.
7. The drag is turned upside down.
8. The cope is placed on the drag after doing clay wash.
9. The runner and riser are placed over the pattern.
10. After applying parting sand, moulding sand is filled over the pattern.
11. Ramming is done to get a rigid mould.
12. Using strike off bar excess sand is removed.
13. Vent holes are made using vent rod.
14. Runner and raiser are removed a funnel shape is made on the runner hole.
15. Cope is kept aside and the pattern is removed using draw spike.
16. The cope is placed on the drag.

**RESULT:** Thus the gear mould is prepared and ready for casting

## STEPPED CONE PULLEY

EX.NO:

DATE :

AIM:

To prepare a sand mould using a flange pattern

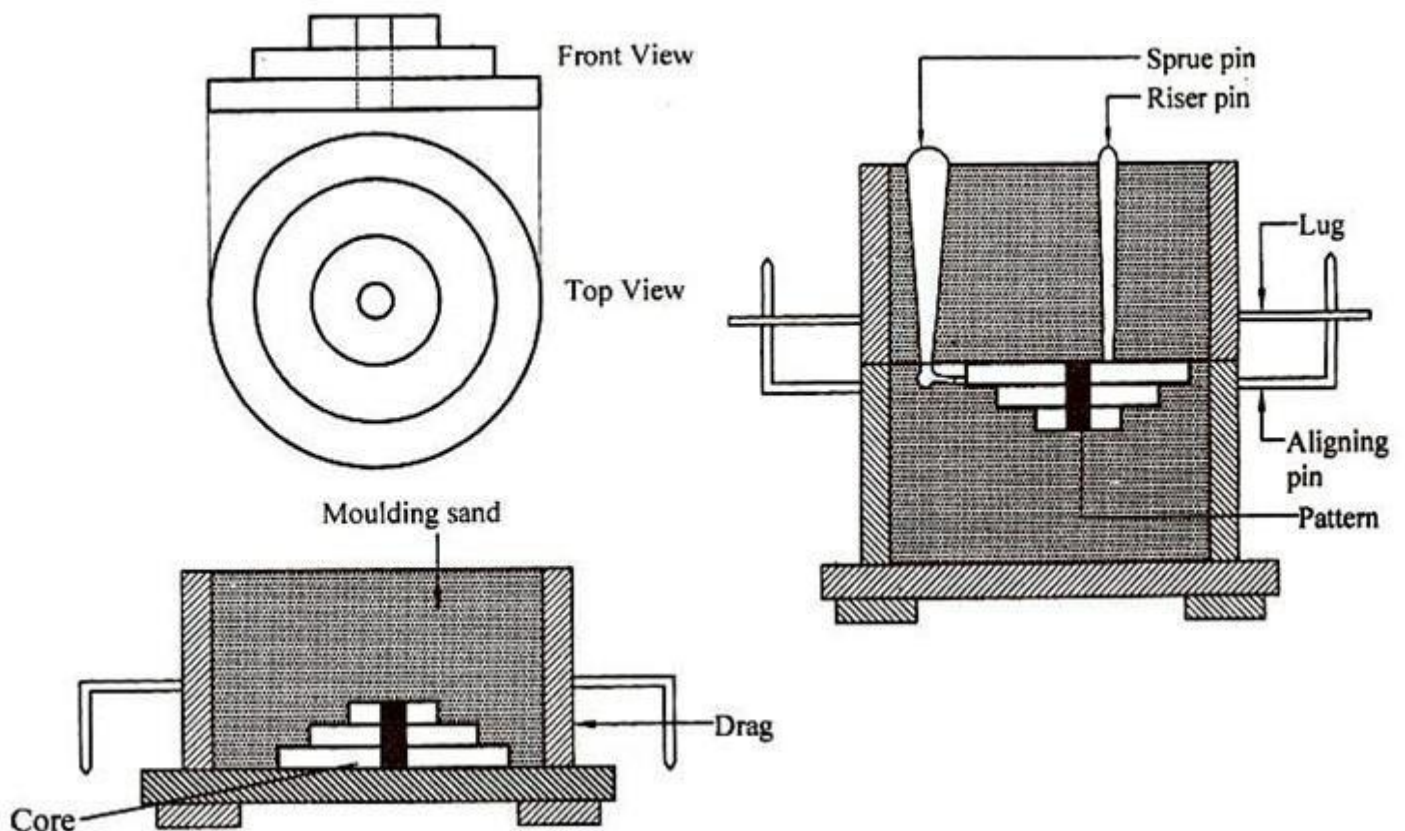
### MATERIALS REQUIRED

Moulding sand, Parting sand, Facing sand, Gear pattern, Moulding boxes etc.

### TOOLS REQUIRED

1. Showel
2. Sieve
3. Trowel
4. Rammers
5. Spure pin
6. Strike off bar
7. Lifter
8. Gate cutter
9. Runner
10. Riser
11. Vent rod
12. Draw spike
13. Bellow

### SPECIMEN DRAWING:



**SEQUENCE OF OPERATIONS:**

1. Sand preparation.
2. Core preparation.
3. Placing the pattern on moulding board.
4. Ramming of drag.
5. Placing runner and riser.
6. Ramming of cope.
7. Removal of pattern, runner, riser.
8. Gate cutting.

**PROCEDURE:**

1. The stepped pattern is placed on the moulding board.
2. A suitable core is prepared and placed in the hole of gear pattern.
3. Clay washing is done inside the drag surface.
4. Parting sand is applied over the pattern.
5. Box is filled with smooth moulding sand and proper ramming is done using flat and peen rammers.
6. Excess sand is removed using the strike off bar.
7. The drag is turned upside down.
8. The cope is placed on the drag after doing clay wash.
9. The runner and riser are placed over the pattern.
10. After applying parting sand, moulding sand is filled over the pattern.
11. Ramming is done to get a rigid mould.
12. Using strike off bar excess sand is removed.
13. Vent holes are made using vent rod.
14. Runner and raiser are removed a funnel shape is made on the runner hole.
15. Cope is kept aside and the pattern is removed using draw spike.
16. The cope is placed on the drag.

**RESULT:** Thus the stepped mould is prepared and ready for casting

**INTRODUCTION:****VI. SHEET METAL**

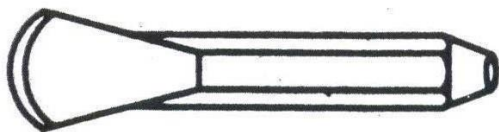
Sheet metal work is working on the metal of 16 gauge to 30 gauge with hand tools and simple machines into different forms by cutting, forming into shape and joining. Sheet metal work is one of the major applications in the engineering industry. It has its own significance as useful trade in engineering work.

**Application of sheet metal:**

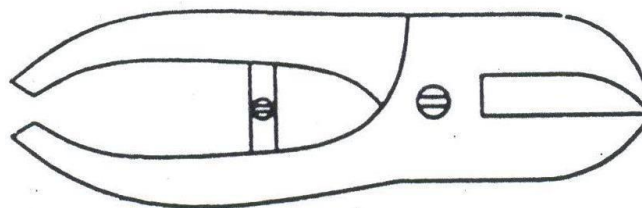
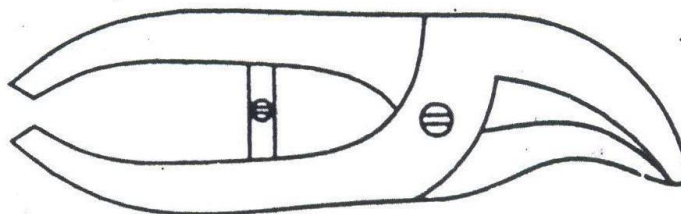
Sheet metal work is used for making hoppers, funnels, various ducts, chimneys, ventilating pipes, machine tool guards, boilers, etc., It is also used in major industries like aircraft manufacturing, ship building, automobile body building and fabricating ducts in air conditioning equipments etc.,

**Tools used:****Cutting tools (chisels, snips or shears):****Chisels:**

Chisels are used for cutting sheets, rivets, bolts and clipping operations. Round nose chisel and flat chisel are the often used ones.

**Flat Chisel****Round Nose Chisel****Snips or shears:**

Snips are hard shears, varying in length from 200mm to 600mm. 200mm and 250mm length is commonly used. Straight snips are used for cutting along outside curves and straight lines. Curved snips or bent snips are used for trimming along inside curves.

**Straight Snip****Bend Snip**

**Striking tools:****Hammer:**

Hammers are used in sheet metal work for hollowing, stretching, levelling, revetting, strengthening of sheet metal joints etc., the following joints are used in sheet metal. Ball peen hammer, Straight peen hammer, Rivetting hammer, Mallet

**Punches:**

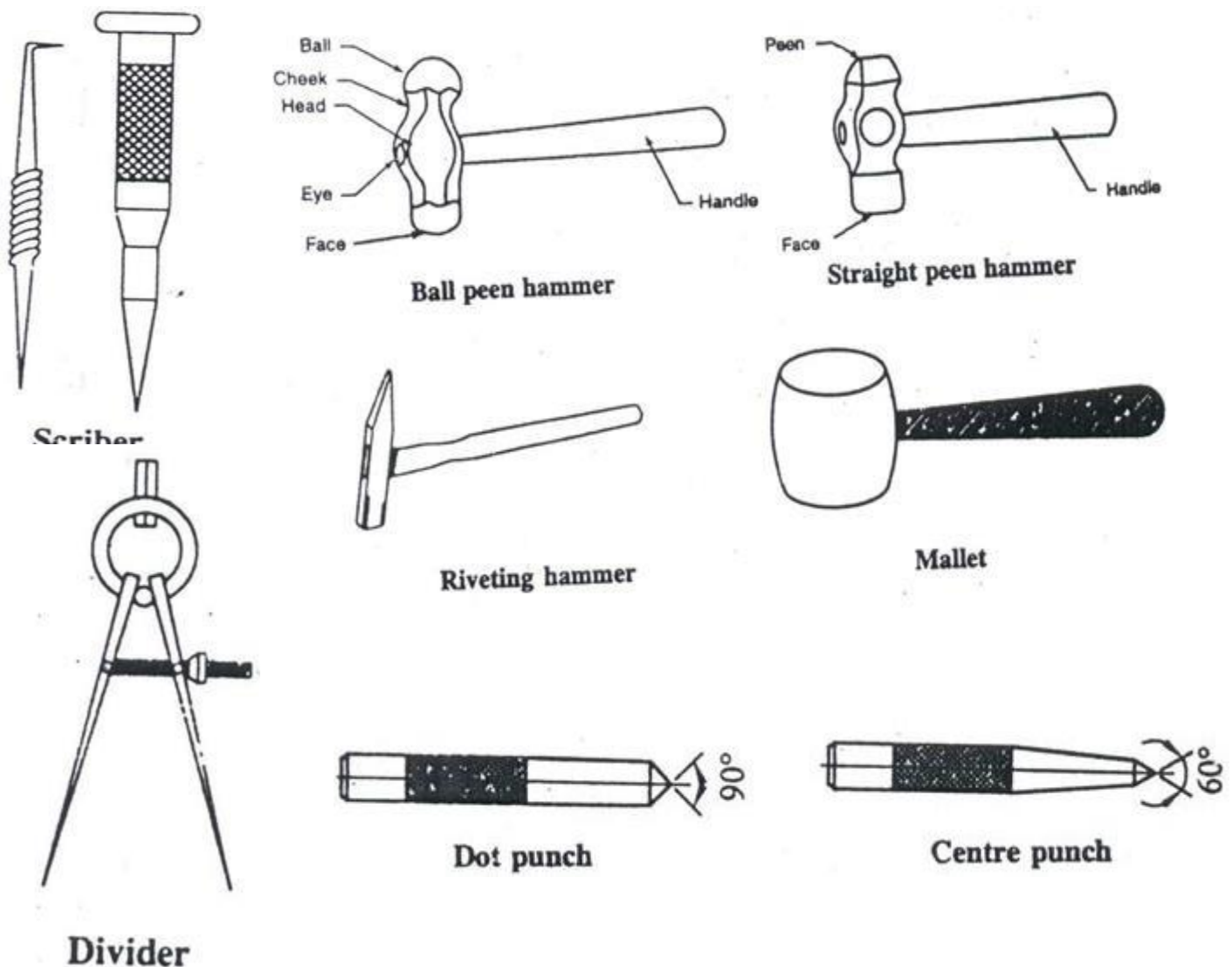
In sheet metal work, punch is used for making out work locating centers etc., the following two types of punches are widely used. Dot punch & Centre punch.

**Supporting tools:****Stakes:**

Stakes are nothing but sheet metal worker's anvils used for bending, hemming, seaming, forming etc., using hammers and mallet.

**Bending tools:****Pliers:**

Pliers are used for bending the sheet metal to the required shape. It is also used for holding and cutting the sheet metal. Flat nose pliers and round nose pliers are used in sheet metal work for forming and holding work.



**Layout tools:****Steel rule:**

It is used for measuring and laying out small work it can measure with as accuracy of up to 0.5mm.

**Scriber:**

It is a long wire of steel with its one end sharply pointed and hardened to scratch line on sheet metal for laying out patterns.

**Dividers:**

Dividers are used for drawing circles or arcs on sheet metal.

**Trammels:**

It is used for marking areas and circles. Maximum size of the arc that can be scribed depends on the length of the beam in scriber.

**Sheet metal gauge:**

It is used to find the thickness of the sheet metal. The various types of gauges are: Standard Wire Gauge (SWG), Birham Wire Gauge, Birmingham Wire Gauge and American Wire Gauge.

**Other tools:****Grooving:**

In order to join the sheet metal joints, their ends are grooved with the help of grooving tools. This process is called grooving. It is a most widely used tool made up of hardened and tempered carbon steel. These are used for making locked joints in sheet metal works such as wired edges, slots etc., These are available in different sizes.

**Bench plate:**

A special type of plate made of carbon steel. Types of bench plate: fixed bench plate, revolving bench plate.

**Hand dolly**

It is a steel block rectangular in shape and fitted with a handle in the bottom of the block.

**Sheet metal joints:**

Sheet metal working incorporates a wide variety of hems and seams.

**Hem:**

A hem is an edge or border made by folding. Types of hem are Single hem, Double hem, wired edge.

## RECTANGULAR TRAY

EX.NO:

DATE :

**AIM:**

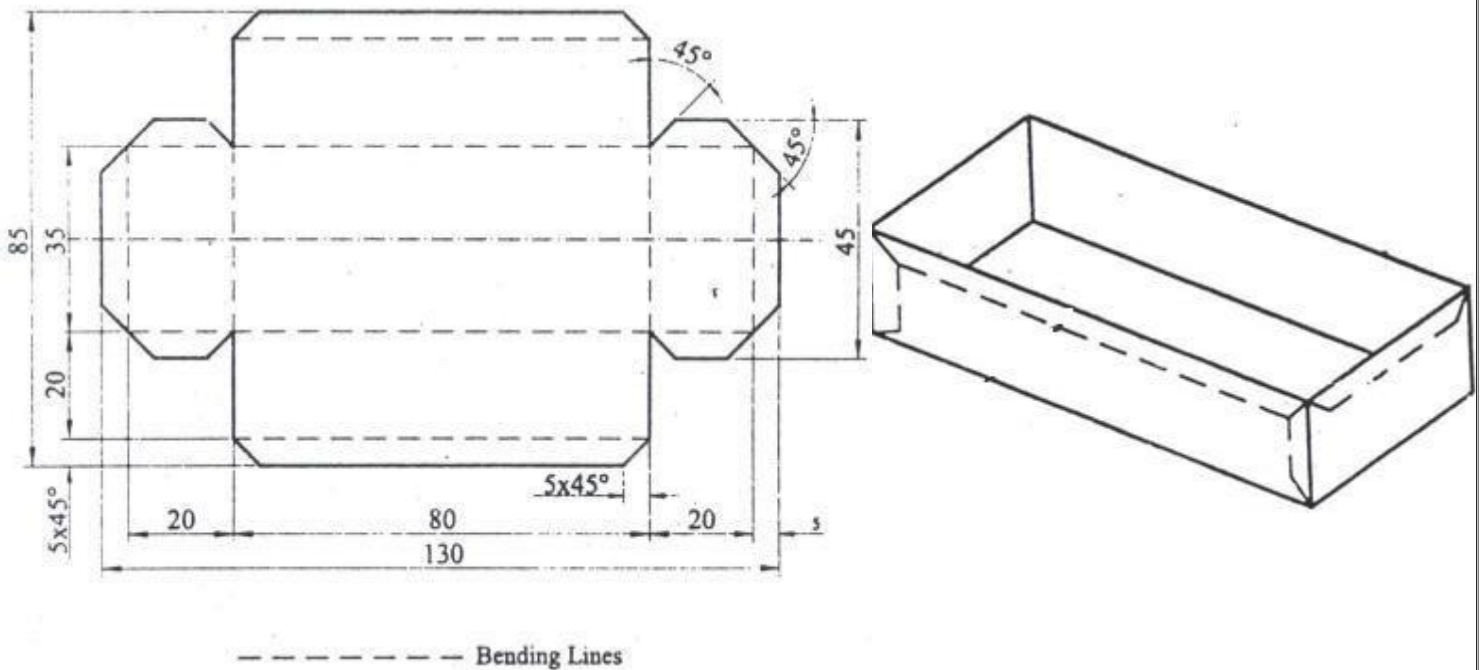
To make a Rectangular Tray from the given sheet metal.

**MATERIALS REQUIRED**

22 gauge Galvanized Iron (G.I) sheet.

**TOOLS REQUIRED**

1. Steel rule
2. Mallet
3. Scriber
4. Divider
5. Protractor
6. Snips
7. Stakes
8. Rivet set
9. Ball peen hammer

**SPECIMEN DRAWING:**

**SEQUENCE OF OPERATIONS:**

1. Checking
2. Levelling
3. Marking on paper
4. Marking on sheet metal
5. Cutting
6. Folding
7. Hemming
8. Soldering

**PROCEDURE:**

1. The gear pattern is placed on the moulding board.
2. The size of the given sheet metal is checked for its dimensions using a steel rule.
3. The required development of surface is being made on the white paper which is overlapped on the sheet metal.
4. The marking is done on the sheet metal as per the development being done on the paper.
5. Now using straight snips, unwanted materials are removed.
6. Now fold and bend the work piece to make the funnel shape and joint is made on the work piece.
7. Then using groover, locked grooved joint is made for about 5mm. Also, hemming is done in the bottom of the funnel.
8. In between top face and bottom face, butt joints is made using solder.
9. Finally, trimming and finishing operations are being carried out.

**RESULT:** Thus the desired rectangular tray is made from the given sheet metal.



## FUNNEL

EX.NO:

DATE :

**AIM:**

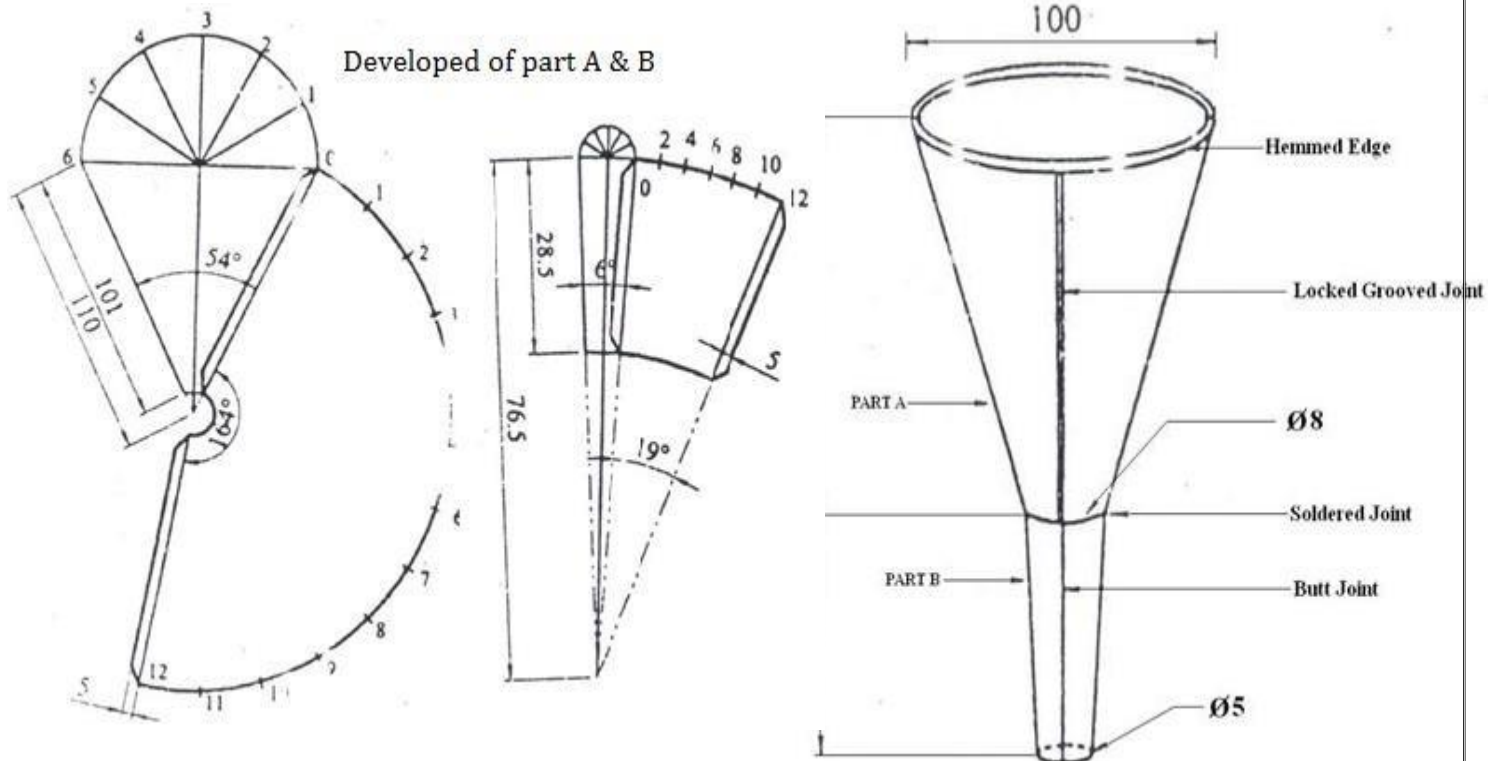
To make a funnel from the given sheet metal.

**MATERIALS REQUIRED**

22 gauge Galvanized Iron (G.I) sheet.

**TOOLS REQUIRED**

1. Steel rule
2. Mallet
3. Scriber
4. Divider
5. Protractor
6. Snips
7. Stakes
8. Rivet set
9. Ball peen hammer

**SPECIMEN DRAWING:**

**SEQUENCE OF OPERATIONS:**

1. Checking
2. Levelling
3. Marking on paper
4. Marking on sheet metal
5. Cutting
6. Folding
7. Hemming
8. Soldering

**PROCEDURE:**

1. The gear pattern is placed on the moulding board.
2. The size of the given sheet metal is checked for its dimensions using a steel rule.
3. The required development of surface is being made on the white paper which is overlapped on the sheet metal.
4. The marking is done on the sheet metal as per the development being done on the paper.
5. Now using straight snips, unwanted materials are removed.
6. Now fold and bend the work piece to make the funnel shape and joint is made on the work piece.
7. Then using groover, locked grooved joint is made for about 5mm. Also, hemming is done in the bottom of the funnel.
8. In between top face and bottom face, butt joints is made using solder.
9. Finally, trimming and finishing operations are being carried out.

**RESULT:** Thus the funnel of the required dimensions is made from the given sheet metal

## METAL SPINNING

EX.NO:

DATE :

### AIM:

To make a component using Lathe machine by metal spinning process

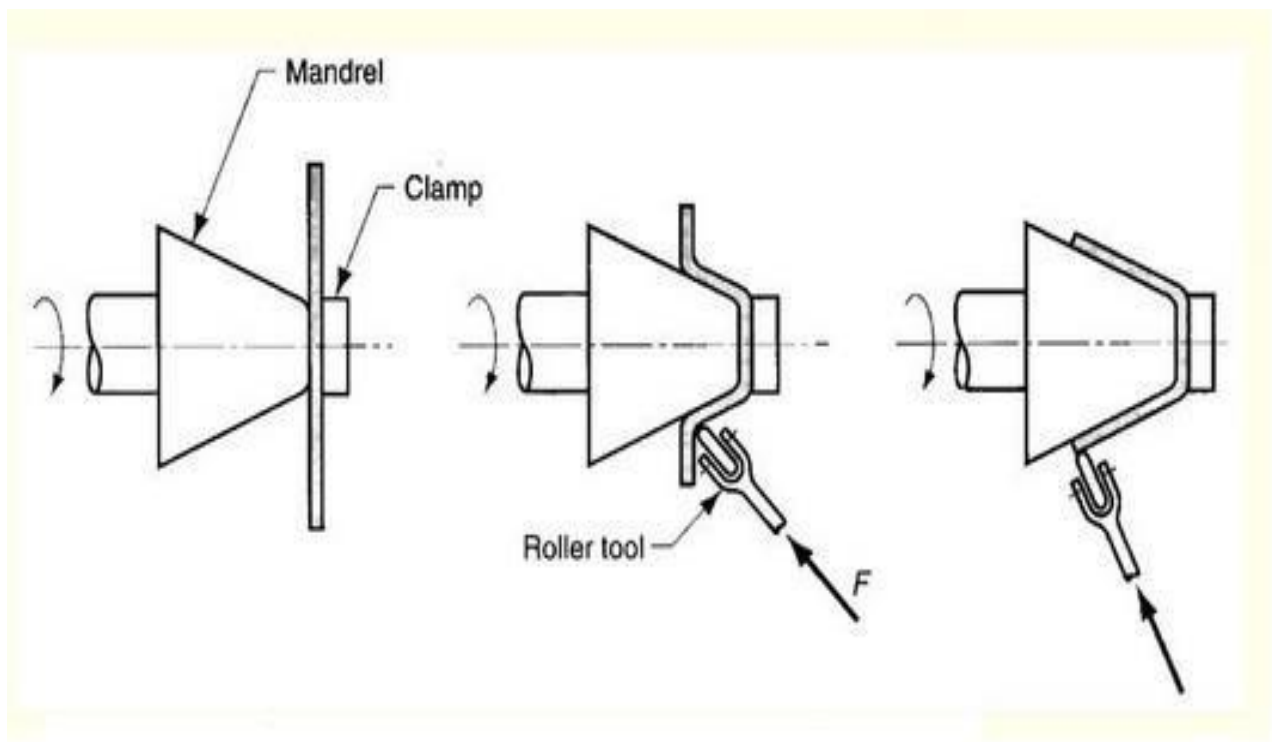
### MATERIALS REQUIRED

26 gauge Galvanized Iron (G.I) sheet.

### TOOLS REQUIRED

1. Steel rule
2. Mallet
3. Scriber
4. Divider
5. Snips
6. Lubrication
7. Mandrel
8. Roller tool
9. Clamp

### SPECIMEN DRAWING:



**PROCEDURE:**

1. First the work piece must be thoroughly cleaned to remove rust, scale and other foreign material.
2. The size of the given sheet is checked for its dimensions using a steel rule.
3. The required shape is first sniped from the metal sheet using snipping tool
4. Then the sheet is clamped to mandrel between head stock and tail stock.
5. The roller tool is placed over the jig of the lathe machine is turned on.
6. Using the roller tool force is applied manually over the metal sheet.
7. The metal sheet attains the external shape of the mandrel.
8. In general metal spinning process is achieved in two to three stages.
9. Edges of the metal sheet are then trimmed to avoid sharp points.

**RESULT:** Thus the metal spinning process is performed using lathe machine.

## TAPER TURNING BY TAIL STOCK OFFSET METHOD

EX.NO:

DATE :

### AIM:

To Produce the component by performing Taper Turning using lathe.

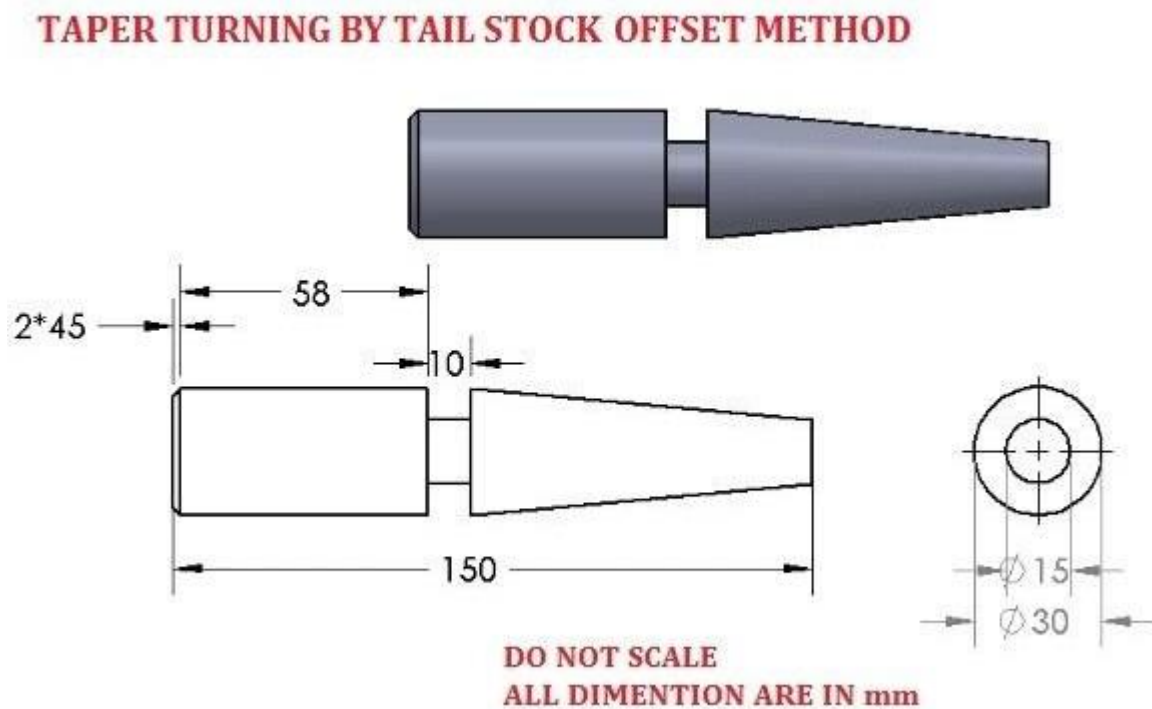
### MATERIALS REQUIRED

Mild steel  $\Phi$  32 mm X 153 mm long.

### TOOLS REQUIRED

1. Chuck key
2. Tool post spanner
3. Vernier caliper
4. Steel rule
5. Single point cutting tool
6. Revolving centre
7. Drill chuck with centre drill bit
8. Catch plate with dog carrier

### SPECIMEN DRAWING:



**PROCEDURE:**

1. Check the size of raw material given and ensure the size.
2. Held the work piece in the chuck and tight it properly.
3. Clamp the tool on tool post properly and check the center of work piece with cutting tool tip.
4. Do the facing operation by giving cross feed to the cross slide and reduce the length.
5. Do the turning operation by giving longitudinal feed to the carriage and reduce the diameter of the work piece.
6. Two or three passes can be given for turning operation.
7. Remove the work piece and hold in chuck on other side. Does the turning operation for the remaining portion of the work piece also do the facing operation and get the required length.
8. Make a center drilling operation on both sides by tail stock.
9. Calculate the offset distance and offset the tail stock from the lathe axis.
10. Hold the work piece between lathe centers on both side using catch plate with dog carrier.
11. Do the taper turning operation by giving longitudinal feed to the carriage as per the dimension.
12. Remove the component carefully and check the dimension.

**RESULT:** Thus the component is produced as per the given drawing using lathe.

## KEY WAY SLOTTING

EX.NO:

DATE :

### AIM:

To cut a keyway on the given specimen as per the dimensions using slotting machine.

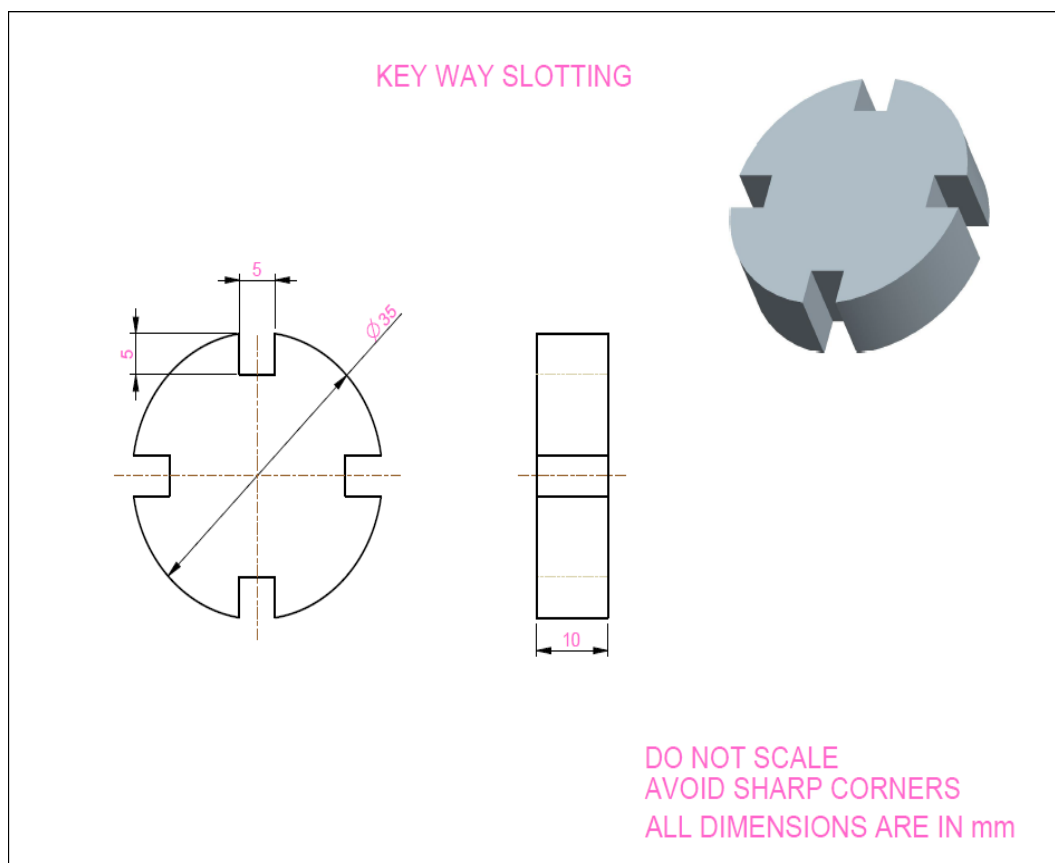
### MATERIALS REQUIRED

Cast iron blank of  $\Phi$  50mm x 10mm.

### TOOLS REQUIRED

1. Steel rule
2. Vernier caliper (0-150mm)
3. T bolt strap
4. Spanner
5. punch

### SPECIMEN DRAWING:



**PROCEDURE:**

1. First the given work piece is fitted on chuck and it is turned along the diameter to the required dimension by using lathe.
2. Then the facing operation is done on both the sides of the work piece.
3. Then the work piece is fixed in the slotting machine.
4. Indexing was done on the slotting machine.
5. By adjusting the feed hand wheel the key way are cut outer diameter of the work piece.
6. Then the same procedures are repeated for next key way.

**RESULT:** Thus the key way is made on the work piece by slotting machine.



## SHAPING AN ANGULAR SURFACE

EX.NO:

DATE :

### AIM:

To make an Angular surface by using a shaping machine.

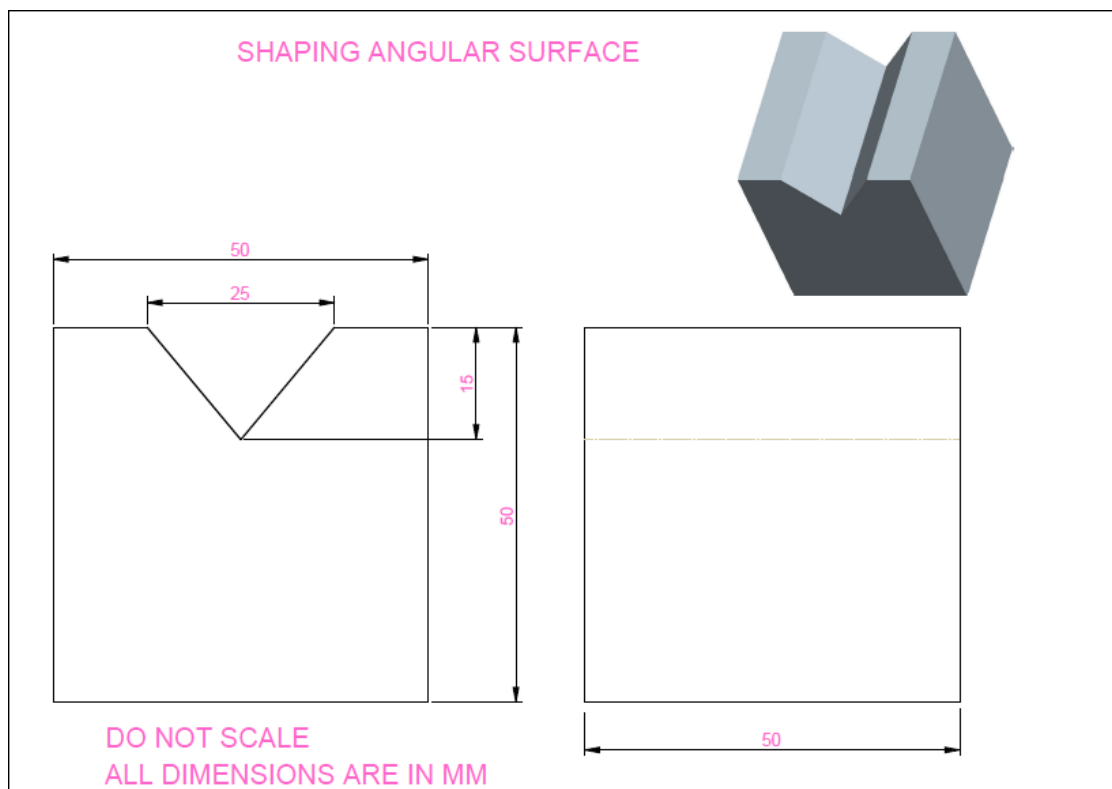
### MATERIALS REQUIRED

Cast iron blank of  $\Phi 50 \times \Phi 50$

### TOOLS REQUIRED

1. Machine handle
2. Vice handle
3. Round nose tool
4. Vernier caliper
5. Vernier height gauge
6. Tri-square
7. surface plate
8. Hammer and punch.

### SPECIMEN DRAWING:



**PROCEDURE:**

1. Check the machine is in proper condition.
2. Check the work piece given in suitable size.
3. Select and set the correct stroke length, speed and feed.
4. The job is marked and punched. The punched job is set over the vice.
5. Hold the work piece in the vice and fix the tool in tool holder in correct position and start the machine.
6. Remove the material as per the size.
7. Remove the job and also remove the burrs with the help of files.

**RESULT:** Thus the shaping of angular surface operation is performed on the given work piece as per the dimension.

## LAB VIVA QUESTIONS AND ANSWERS

**1. What is a lathe?**

Lathe is a machine, which removes the metal from a piece of work to the required shape & size

**2. what are the various operations can be performed on a lathe?**

- |               |                   |              |
|---------------|-------------------|--------------|
| 1. Turning    | 6. Thread cutting | 11. Grooving |
| 2. Facing     | 7. Drilling       |              |
| 3. Forming    | 8. Boring         |              |
| 4. Knurling   | 9. Recessing      |              |
| 5. Chamfering | 10. Tapping       |              |

**3. What are principle parts of the lathe?**

Headstock, tailstock, carriage, cross slide, tool post

**4. What are the types of headstock?**

Back geared type, all geared type

**5. State the various parts mounted on the carriage?**

Saddle, compound rest, cross slide, tool post

**6. What is an apron?**

The integral part of several gears, levers clutches mounted with the saddle for moving the carriage along with lead screw while thread cutting

**7. List any four types of lathe?**

1. Engine lathe
2. Bench lathe
3. Tool room lathe
4. Semi automatic lathe
5. Automatic lathe

**8. What is a semi-automatic lathe?**

The lathe in which all the machining operations are performed automatically and loading and unloading of work piece, coolant on or off is performed manually

**9. State the various feed mechanisms used for obtaining automatic feed?**

1. Tumbler gear mechanism
2. Quick change gearbox
3. Tumbler gear- Quick change gearbox
4. Apron mechanism

**10. List any four holding devices?**

1. Chucks
2. Centers
3. Face plate
4. Angle plate

**11. What are the different operations performed on the lathe?**

Centering, straight turning, rough turning, finish turning, shoulder turning, facing, chamfering, knurling, etc

**12. What is tooling?**

Planning of operations sequence & preparation of turret or capston lathe are termed as tool-layout or tooling

**13. What are the three stage of a tool-layout?**

1. Planning & scheduling
2. Detailed sketching of various machining operation sequence
3. Sketching the plan showing various tools

**14. What is shaper?**

The machine, which is having a reciprocating type of machine tool with a single point cutting tool, used to produce flat surfaces called as Shapers

**15. List any four important parts of a Shaper?**

Table, Tool head, Ram, Cross rail

**16. How the feed & depth of cut is given to the shaper?**

Feed is given by rotating the down feed screws of tool head depth of cut is given by rotating by raising or elevating the table

**17. Mention any four-shaper specification?**

1. Maximum length of stroke
2. Type of driving mechanism
3. Power of the motor
4. Speed & feed available

**18. How the planer differs from the shaper?**

In planer-the work reciprocate while the tool is stationary

In shaper-the tool reciprocate while the work is stationary

**19. State the use of planer?**

The planer is used for machining heavy & large casting Ex. lathe bed ways, machine guide ways

**20. What is meant by drilling?**

Drilling is the process of producing hole on the work piece by using a rotating cutter called drill

**21. What is gang -drilling machine?**

When a number of single spindles with essential speed & feed are mounted side by side on one base and have common worktable is known as gang -drilling machine

**22. Mention any four specification of drilling machine?**

1. Maximum size of the drill in mm that the machine can operate
2. Table size of maximum dimensions of a job can mount on a table in square meter
3. Maximum spindle travel in mm
4. Number of spindle speed & range of spindle speeds in r.p.m.

**23. List any four machining operations that can be performed on a drilling machine?**

1. Drilling
2. Counter sinking
3. Tapping
4. Trepanning

**24. What are the different ways to mount the drilling tool?**

1. Fitting directly in the spindle
2. By using a sleeve
3. By using a socket
4. By means of chucks

**25. what is broaching?**

Broaching is a process of machining a surface with a special multipoint cutting tool called broach which has successfully higher cutting edges in a fixed path

**26. Indicate any two specification of a broaching machine?**

1. Maximum length of stroke in mm
2. Maximum force developed by the slide in tones

**27. What are the different operations that can be performed on a broaching machine?**

1. Broaching splines
2. Broaching a key way

**28. What is boring?**

Boring is a process of enlarging & locating previously drilled holes with a single point cutting tool

**29. What are the application of boring?**

The boring machine is designed for machining large & heavy work piece in mass production work of engine frame, cylinder, machine housing etc

**30. What is the function of cutting fluids?**

1. It is used to cool the cutting tool & the work piece.
2. It improves the surface finish as stated earlier.
3. It causes the chips to break up into small parts.
4. It protects the finish surface from corrosion.
6. It prevents the corrosion of work & machine.

**31. What are causes of wear?**

The tool is subjected to three important factors such as force, temperature and sliding action due to relative motion between tool and the work piece. So the tool is wear easily.

**32. What are the specifications of the milling machine?**

1. The table length & width.
2. Number of spindle speeds & feeds.
3. Power of driving motor.

**33. Mention the various movements of universal milling machine table?**

1. Vertical movement-through the knee.
2. Cross vise movement-through the saddle.

**34. What are the cutter holding devices?**

1. Arbors
2. Adaptors
3. Collets

**35. List the various type of milling attachment?**

1. Vertical milling
2. Universal milling
3. High speed milling

**36. Write any ten nomenclature of plain milling cutter?**

Body of cutter, cutting edge, face, fillet, gash, lead, land, outside dia, root dia, cutter angles.

**37. What are the down milling processes?**

1. Cutter with higher rake angle can be used. This reduces power requirements.
2. Cutter wear is less because chip thickness is maximum at the start of cut.

**38. List out the various milling operations?**

1. Plain or slab milling.
2. Face milling.
3. Angular milling.
4. Gang milling.
5. End milling.
6. Gear cutting.

**39. What does term indexing mean?**

Indexing is the process of dividing the periphery of a job into equal number of divisions.

**40. Mention the various parts of single point cutting tool?**

1. Shank
2. Face
3. Flank
4. Base
5. Nose
6. Cutting edge

**41. What is tool signature?**

The various angles of tools are mentioned in a numerical number in particular order. that order is known as tool signature.

**42. What is effect of back rake angle & mention the types?**

Back rake angle of tool is increases the strength of cutting tool & cutting action. It can be classified in to two types

Positive rake angle

Negative rake angle

**43. What is side rake angle & mention its effects?**

The angle between the tool face & the line parallel to the base of the tool. it is used to control chip flow

**44. Define orthogonal & oblique cutting?**

Orthogonal cutting: The cutting edge of tool is perpendicular to the work piece axis.

Oblique cutting: The cutting edge is inclined at an acute angle with normal to the cutting velocity vector is called oblique cutting process

**45. What is cutting force?**

The sheared material begins to flow along the cutting tool face in the form of small pieces. The compressive force applied to form the chip is called cutting force

**46. Define machinability of metal?**

Machinability is defined as the ease with which the material can be satisfactorily machine.

**47. What are the factors affecting the machinability?**

1. Chemical composition of work piece material.
2. Microstructure of work piece material

**48. What is machinability index?**

It is the comparison of machinability different material to standard material. US material standards for 100% machinability are sae1112 hot rolled steel.

Machinability index  $I = \frac{\text{Cutting speed of metal investigated for 20 minutes tool life}}{\text{Cutting speed of standard steel for 20 minutes tool life}}$

**49. What are the factors affecting tool life?**

1. Cutting speed
2. Feed & depth of cut
3. Tool geometry
4. Tool material

**50. Express the Taylor's tool life equation?**

Taylor's tool life equation:  $V T^n = C$

V=Cutting speed in m/min T=Tool life in minutes

C=Constant

n=Index depends upon tool & work.



# LABORATORY MANUAL

## **ME3381- COMPUTER AIDED MACHINE DRAWING**

<b>YEAR / SEMESTER</b>	<b>:</b>	<b>II / 03</b>
<b>DEPARTMENT</b>	<b>:</b>	<b>Mechanical Engineering</b>
<b>ACADEMIC YEAR</b>	<b>:</b>	<b>2023-24</b>
<b>REGULATION</b>	<b>:</b>	<b>2021</b>

# INDEX

## LIST OF EXPERIMENTS

### I. COMPUTER AIDED DESIGN (CAD)

SL. NO.	DATE	NAME OF THE MACHINE ELEMENT	PAGE NO	STAFF SIGN
01		Flange Coupling – Unprotected Type		
02		Flange Coupling – Protected Type		
03		Screw Jack		
04		Machine Vice		

### II. COMPUTER AIDED MANUFACTURING (CAM)

SL. NO.	DATE	NAME OF THE CNC OPERATION (MANUAL PART PROGRAMMING)	PAGE NO	STAFF SIGN
05		Multiple Turning – CNC Lathe		
05A		Multiple Turning using Multiple Turning Cycle (G71) – CNC Lathe		
06		Grooving & Thread Cutting – CNC Lathe		
06A		Grooving & Thread Cutting using Cycle – CNC Lathe		
07		Turning, Taper-Turning, Grooving & Tapping – CNC Lathe		
08		Linear with Circular Interpolation – CNC Milling		
09		Rectangular & Circular Pocketing – CNC Milling		
09A		Rectangular & Circular Pocketing using Cycle – CNC Milling		
10		Profile Milling – CNC Milling		



### **III. SIMULATION AND NC CODE GENERATION**

SL. NO.	DATE	NAME OF THE OPERATION (SIMULATION AND NC CODE GENERATION)	PAGE NO	STAFF SIGN
11		Grooving using CAPSturn		
12		Thread Cutting using CAPSturn		
13		Grooving & Thread Cutting using CAPSturn		
14		Circular & Rectangular Pocketing using CAPSmill		
15		Mirroring using CAPSmill		

**THEORY**

# **I. COMPUTER AIDED DESIGN (CAD)** **OF MACHINE ELEMENTS**

## **1. FLANGED COUPLING - UNPROTECTED TYPE**

### **Introduction:**

Couplings are used for connecting shafts of two machines to function as a single unit. The shafts are coupled for transmitting power and motion from driver unit to driven unit. Flange coupling is extensively used. It consists of male and female flange, key and bolt & nuts.

### **Flange:**

It is made up of cast iron. For correct alignment of shaft, a projection is made in the centre of the flange. This projection is made to fit accurately into a corresponding recess provided in the other flange.

### **Keys:**

Keys are used to fix the flange and transmit the power from shaft to flange.

### **Bolt & Nut:**

It is used to connect the two flanges (Male & Female), and make it two machines as one machine. And also it transmits the power from driver end flange to driven end flange. The number of bolt and nuts are depends on size of the coupling.

## **2. FLANGED COUPLING - PROTECTED TYPE**

### **Introduction:**

Couplings are used for connecting shafts of two machines to function as a single unit. The shafts are coupled for transmitting power and motion from driver unit to driven unit. Protected type flange coupling is extensively used. It consists of male and female protected flange, key and bolt & nuts.

### **Protected Flange:**

It is made up of cast iron. In this each flange is provided with annular projection. It shelters the bolt heads or the nuts. Hence it prevents them from catching cloths of workmen. For correct alignment of shaft, a projection is made in the centre of the flange. This projection is made to fit accurately into a corresponding recess provided in the other flange.

### **Keys:**

Keys are used to fix the flange and transmit the power from shaft to flange.

### **Bolt & nut:**

It is used to connect the two protected flange (Male & Female), and make it two machines as one machine. And also it transmits the power from driver end flange to driven end flange. The number of bolt and nuts are depends on size of the coupling.

### 3. SCREW JACK

#### **Introduction:**

Screw jack is a manually operated device. It is used to lift the automobile vehicle and any heavy objects through a small height by applying small effort. The effort is given by a Tommy bar. It consists of cast iron body, gunmetal nut, screw spindle, cup and Tommy bar.

#### **Body:**

The body is made up of cast iron. This cast iron body with a circular base, which provides a large bearing area.

#### **Nut:**

It is made up of gun metal, mounted at the top of the body at tight fitted. Internal thread of the nut is square threaded which guided the threaded spindle.

#### **Spindle:**

The spindle is made up of mild steel. One end of the spindle has knurled head; the remaining portion of the spindle is square threaded. The square threaded screw spindle is screwed through gun metal nut. The head of the screw has holes to insert the Tommy bar.

#### **Cup:**

A load bearing cup is mounted at the top of the screw spindle. The cup is loosely held in position by a washer and counter shank screw, so that when the spindle is rotated cup moves only up and down but will not rotate with it.

#### **Tommy Bar:**

It is made up of mild steel. It is to turn the spindle screw by applying minimum effort. It will be inserted at the head of the screw spindle hole.

## 4. MACHINE VICE

Machine vice is a work holding device used in machine shop. Machine vices are fixed to the bed or table of milling, planing, shaping, drilling machines etc. There are different types of machine vice.

A typical machine vice contains one *fixed jaw* and one *movable jaw*. Fixed jaw is fixed to the body of the vice by means of two set screws. A 'V' – cut is provided in the fixed jaw to facilitate holding of round rod.

A movable jaw is mounted on the guide ways of the body and secured to it by means of clamping plate using two set screws. A *screw rod* passes through the threaded hole in the movable jaw.

Therefore when the screw rod is rotated, the movable jaw slides over the bed ways of the body. The screw rod is supported by a spindle bearing block which is fitted on the body by means of set screws.

## **II. COMPUTER AIDED MANUFACTURING (CAM)**

### **CNC LATHE STARTURN - MTAB DENFORD**

#### **FANUC CONTROLLER - STANDARD FEATURE & MACHINE SPECIFICATION**

##### **A. CABINET**

Length	:	600 mm (26'')
Width	:	425 mm (16.25'')
Height	:	430 mm (17'')

##### **B. CAPACITY**

Distance between centers	:	250 mm (10'')
Swing over crossslide	:	38 mm (1.5'')
Swing over bed	:	140 mm (5.5'')
Spindle taper	:	No .1MT
Spindle bore	:	10 mm (0.375'')
'X' Axis balls crew	:	8 mm * 2.5 mm pitch
'Z' Axis ball screw	:	10 mm* 4 mm pitch
Bed	:	Ground

## INTRODUCTION TO NC (NUMERICAL CONTROL)

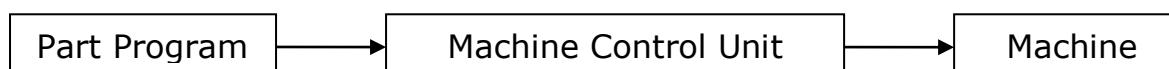
Numerical Control is a technique of automatically operating a productive facility, based on a code of letters, numbers and special characters. Numerical control has been developed out of the need for higher productivity, lower cost and most precise manufacturing. Numerical control is essentially an application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

## INTRODUCTION TO NC MACHINE AND ITS COMPONENTS

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

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

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



**Fig.1. A typical NC system**

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

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

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



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

### **ADVANTAGES OF NC MACHINE OVER CONVENTIONAL MACHINE**

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

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

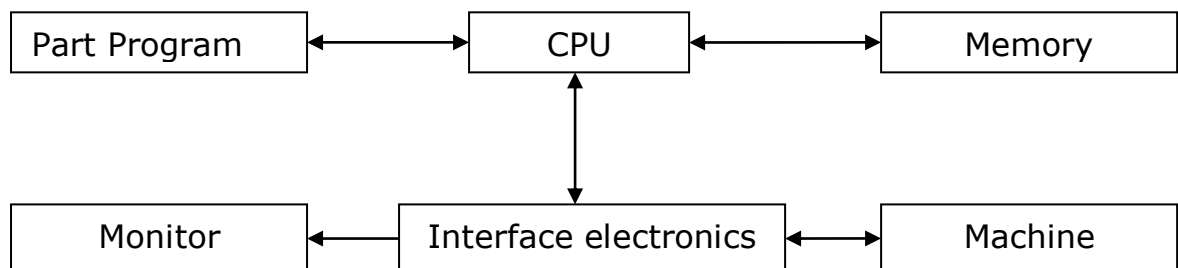
### **DISADVANTAGE OF NC MACHINE**

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

### **INTRODUCTION TO CNC (COMPUTER NUMERICAL CONTROL) MACHINE**

A CNC machine is essentially an NC machine with a dedicated computer being its integral part. It has got more flexibility compared to an NC machine. From outside, NC and CNC machines are not very different. The only apparent difference is the presence of a monitor (CRT) on a CNC machine which an NC machine does not have. The monitor continuously displays the machine status to enable us to communicate with the machine.

**Fig.2. A CNC system**



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

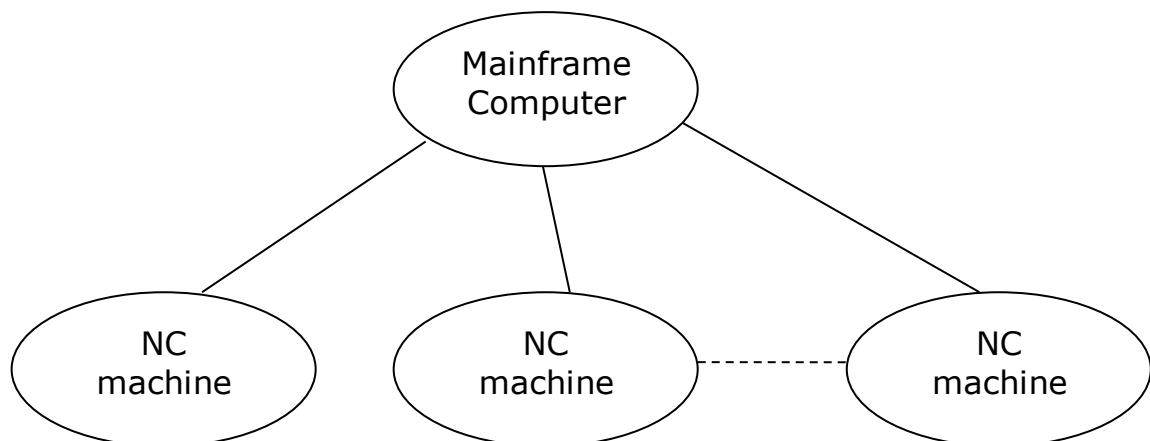
## ADVANTAGES OF CNC SYSTEM OVER NC SYSTEM

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

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

## INTRODUCTION TO DIRECT NUMERICAL CONTROL (DNC) MACHINE

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



**Fig.3. Direct Numerical Control system**

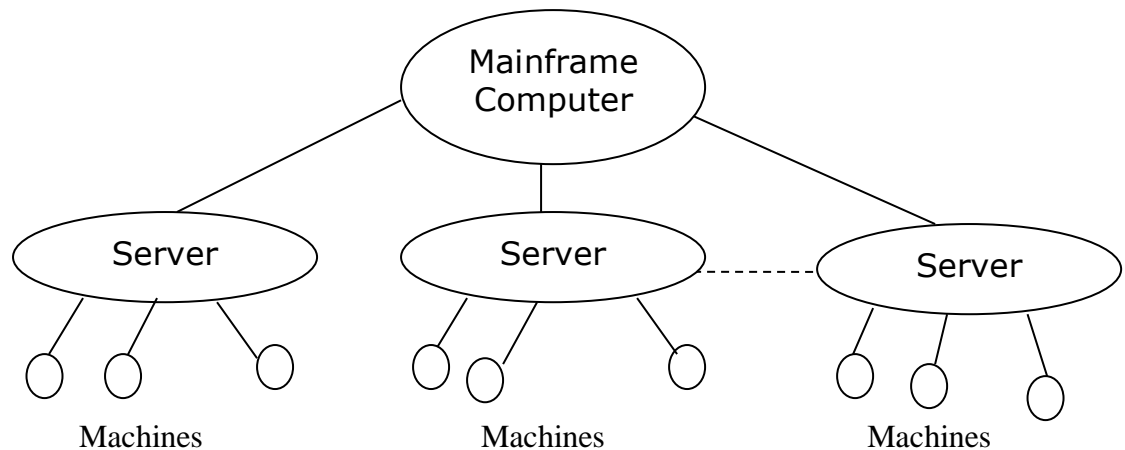
## LIMITATIONS OF DIRECT NUMERICAL CONTROL SYSTEM

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

## INTRODUCTION TO DISTRIBUTIVE NUMERICAL CONTROL (DNC) MACHINE

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

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



**Fig.4. Distributive Numerical Control system**

## TOOL MOVEMENT MODES

In an NC / CNC machine, usually the tool moves with respect to the workpiece which remains at the same place. In some cases, e.g., in a milling machine, the workpiece moves with respect to the tool.

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

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

### **1. Point-to-point placement**

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

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

### **2. Axial Cut**

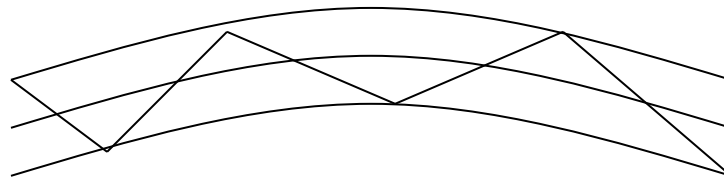
This control allows the tool to move along any major axis with desired speed. Therefore, cutting along X, Y, or Z axis is possible. The limitation being simultaneous

motion along two axes is not possible. So, it cannot make an angular cut. That is why, it is also called *straight cut* control.

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

### 3. Contour Cutting

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



**Fig.5. Straight line approximation of a curve**

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

## DRIVES AND CONTROL LOOPS

The drive motors which control the movement of various axes are of four basic types:

1. Stepper motor
2. AC servo motor
3. DC servo motor
4. Hydraulic servo motor

### 1. Stepper Motor

Stepper motor is driven by an electrical pulse train generated by the machine control unit. Rotation of the motor shaft is proportional to the number of pulses it receives, and its angular velocity is proportional to the frequency of pulses.

Stepper motors are used on light duty machinery where high precision is not required. Since it is possible to regulate the angular position and the angular velocity of a stepper motor, such motors are used in open-loop control system.

### 2. AC / DC Servo Motor

The servo motor can be of AC / DC type. They are used on small to medium-sized CNC machines. These are variable speed motors that rotate in response to the applied voltage.

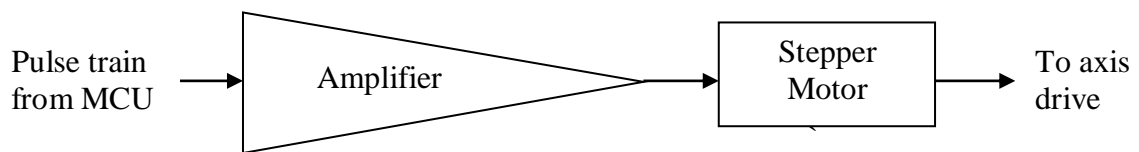
DC servo motors are controlled by varying the voltage magnitude. AC servo motors are regulated by varying the voltage frequency. MCU develops required magnitude and frequency of the voltage to control the speed of these motors. AC servos can develop more power than DC servos. Larger machines use servos rather than stepper motors.

### 3. Hydraulic Servo Motor

Hydraulic servos also are variable speed motors. They produce more power compared to electric servo motors. Such motors are used on large capacity machines.

### OPEN-LOOP CONTROL SYSTEM

In open-loop control system arrangement, there is no feedback from the output, and accuracy of the machine depends fully on the response of the stepper motor. Open-loop control involves less hardware and is relatively inexpensive compared to closed-loop system.

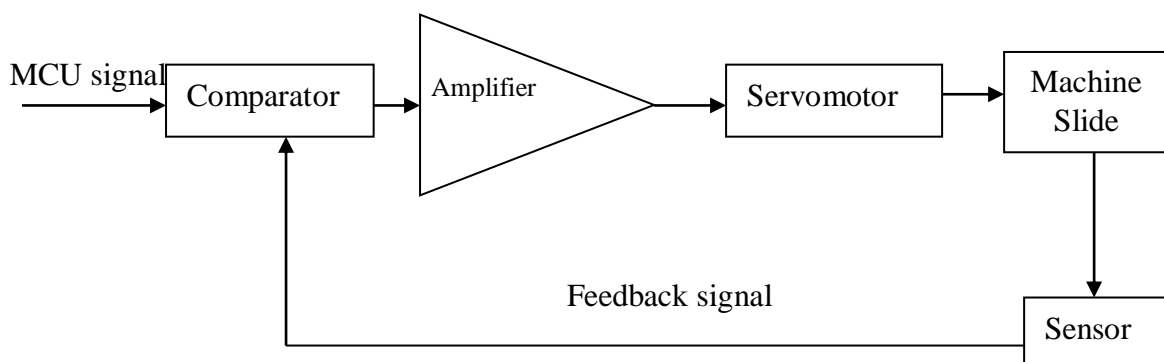


**Fig.6. Open-loop control system**

### CLOSED-LOOP CONTROL SYSTEM

A closed-loop control system measures the actual position and velocity of axes and compares them with the desired values. The difference between the two is the error which is used to regulate the drive motor.

The control is designed in such a way as to eliminate or reduce the error to a minimum value. Fig.7 shows a closed-loop control system using negative feedback. The sensor is usually a rotary or a linear transducer. Rotary transducers may be connected to the lead screw either directly or through gears. In case of linear transducers, one part of the device is fixed to the machine tool structure while the other part is attached to the moving slide.



**Fig.7. Closed-loop control system**

## **COMPARISON BETWEEN OPEN AND CLOSED LOOP CONTROL SYSTEM**

Advances in the manufacture and control of stepper motors have made the open-loop systems fairly accurate and dependable. However, there is always a possibility of some error in the output.

The advantage is that since there is no feedback circuit, less hardware is needed, which not only reduces the cost of the machine but also faces less maintenance problems. However, if accuracy cannot be compromised, one has to go in for closed loop control systems using servomotors.

### **ACCURACY AND REPEATABILITY**

Accuracy is the ability of a machine to produce desired dimensions. Repeatability is the ability to produce the same part for the same dimensions every time. Accuracy of a machine depends mainly on its control resolution which is the minimum distance between two points which the machine can differentiate.

Repeatability of MCU to differentiate between closely spaced points is a function of factors such as the controller's bit storage capacity, drive motor and the type of feedback sensor.

### **INTERPOLATION SCHEMES**

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

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

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

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

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

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

## **TOOL CHANGING DEVICES**

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

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

## **WORK HANDLING DEVICES**

Some machining centres provide more than one separate pallets which can be of linear or rotary types. These pallets simply move or rotate for interchanging their positions on the machine table.

While machining is being done on a job kept on one pallet, the other pallets are accessible to the operator for clamping/unclamping raw material/ finished product. This saves lot of material handling and set up time, resulting in higher productivity.

## **OTHER APPLICATIONS**

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

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

- Lathe
- Turning centre
- Miller
- Machining centre
- Drilling machine
- Gear hobbing machine
- Grinding machine
- Electro-discharge machine
- Welding and cutting
- Coordinate measuring machine, etc.

## **RECENT ADVANCES**

- Most of today's CNC controllers, such as FANUC, SINUMERIK, HEIDENHAIN etc., include software that greatly simplify the programming and diagnostic processes.

- The graphic display in these controllers shows the tool movement for the program. This is very useful as one can verify the tool paths even before actually machining the job.

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

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

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

## **FEED RATE (F WORD)**

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

## **CUTTING SPEED (S WORD)**

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

## **TOOL SELECTION (T WORD)**

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

## **MISCELLANEOUS FUNCTION (M WORD)**

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

## **MISCELLANEOUS FUNCTIONS (M CODES)**

M codes are instructions describing miscellaneous functions like calling the tool, spindle rotation, coolant on etc.,

M00	-	Program stop
M03	-	Spindle forward
M05	-	Spindle stop
M06	-	Automatic tool change
M98	-	Sub program call
M99	-	Sub program end
M08	-	Coolant on
M09	-	Coolant off
M10	-	Vice /chuck open
M11	-	Vice/chuck close



## **PREPARATORY FUNCTION (G FUNCTION)**

G codes are instructions describing machine tool movement.

G00	-	Fast traverse
G01	-	Linear interpolation
G02	-	Circular interpolation (CW)
G03	-	Circular interpolation (ACW)
G21	-	Metric (input in metric)
G28	-	Go to reference
G70	-	Finishing cycle
G72	-	Multiple facing cycle
G76	-	Multiple threading cycle
G81	-	Drilling cycle
G90	-	Turning cycle
G94	-	Facing cycle
G98	-	Feed per minute
G99	-	Feed per rev

## **VARIOUS CYCLES USED IN THE CNC LATHE**

### **A. FACING**

#### **1. G94 (or) G79: CANNED CYCLE**

Format: G79 X---Z---F---;  
          Z---;Z---;Z---

X	-	Depth of cut in x axis
Z	-	Depth of cut in z axis
F	-	Feed rate

#### **2. G72: STOCK REMOVAL MULTIPLE REPETITIVE CYCLE**

Format: G72 W(1) --- R---;  
          G72 P---Q---U---W(2) ---F---;

W(1)	-	Depth of cut in z axis
R	-	Relief amount in x axis
P	-	Sequence number of the first block of the program
Q	-	Sequence number of the last block of the program
U	-	Finish allowance in x axis
W(2)	-	Finish allowance in z axis
F	-	Feed rate

### **B. TURNING**

#### **1 G77 (or) G90 : CANNED CYCLE**

Format: G77 X---Z---F---;

X---X---X---;

X	-	Depth of cut in x axis
Z	-	Depth of cut in z axis
F	-	Feed rate

## 2 G71: STOCK REMOVAL MULTIPLE REPETITIVE CYCLE

Format: G71 W(1) --- R---;  
G71 P---Q---U---W (2) ---F---;

W(1)	-	Depth of cut in x axis
R	-	Relief amount in y axis & x axis
P	-	Sequence number of the first block of the program
Q	-	Sequence number of the last block of the program
U	-	Finish allowance in x axis
W(2)	-	Finish allowance in z axis
F	-	Feed rate

## C. THREAD CUTTING

### 1. G 32: THEARD CUTTING OPERATION

Format: G32Z---F---;  
Z - Depth of cut in z axis  
F - Pitch of the thread

### 2. G 78 (or) G92: CANNED CYCLE THREADING

Format: G78 X---Z---F---;  
X---X---X---X---;

X	-	Depth of cut in x axis
Z	-	Depth of cut in z axis
F	-	Pitch of the thread

### 3. G76: MULTIPLE THREAD CYCLE

Format: G76P\_\_ \_Q(1) \_ R \_;  
G76X\_Z\_P\_Q(2)\_R\_F\_;

P020000 (eg.)	-	02→Number of ideal pass (after cutting thread) 00→Retract angle 00→Thread angle
Q(1)	-	Radial depth of cut in regular pass
R	-	Finishing allowance in the last pass
X	-	Thread minor dia value (in case of ext.)

		thread) (or) Thread major dia. value (in case of int. thread)
Z	-	Thread length
P	-	Thread depth (one side depth)
Q(2)	-	Radial depth of cut in the first pass
R	-	Taper value
F	-	Pitch of the thread

### **D.G33 TAPPING CYCLE**

Format: G33Z---F---;

Z	-	Depth of cut in z axis
F	-	Pitch of the thread

### **E. G74DRILLING CYCLE**

Format: G74R--;  
G74Z---Q---F---;

R	-	Return amount in z axis
Z	-	Drill hole depth
Q	-	Incremental depth of cut in z axis
F	-	Feed rate

### **F. G75 GROOVING CYCLE**

Format: G75R---;  
G75X---Z---P---Q---F---;

R	-	Relief amount in x axis
X	-	Groove diameter
Z	-	Groove length /Groove end point
P	-	Depth of cut in x axis
Q	-	Incremental depth of cut in z axis
F	-	Feed rate

### **G70 FINISHING CYCLE:**

A G70 causes a range of blocks to be executed/ then control passes to the block after the G70. This will be used after the completion of the roughing cycle. The P and Q values specify the “N” block numbers at the start and end of the profile.

Example: G70P10Q20

P- First block of cycle  
Q- Last block of cycle

### **G71 MULTIPLE TURNING CYCLE:**

A G71 causes the profile to be roughed out by turning. Control passes on to after the last block of the profile. Two G71 blocks are needed to specify all the values.

Example: i) G71U2.0R1.5  
ii) G71P10Q20U0.1W0.1F25

i) G71U2.0R1.0

U → Depth of cut in mm

R → Retraction (or) Retardation amount in mm

ii) G71P10Q20U0.1W0.1F25

P → Starting block number (i.e.) first block of the cycle.

Q → End block number

U → Finishing allowance along X axis in mm

W → Finishing allowance along Z axis in mm

F → Feed rate

### **CNC MILLING XLMILL – MTAB**

#### **SPECIFICATION:**

##### **A. CABINET**

Length	:	550 mm (21.654'')
Width	:	540 mm (21.25'')
Overall height (with guards)	:	880 mm (34.65'')

##### **B. CAPACITY**

Maximum cross travel	:	90 mm (3.54'')
Maximum longitudinal travel	:	170 mm (11.5'')
Maximum head travel	:	115 mm (4.52'')
Spindle nose to table top	:	190 mm (7.5'')
Spindle to column	:	110 (4.33'')
Work table surface	:	360 * 130 mm (14.17'' * 0.12'')
Z –axis ball screw	:	16 mm dia. * 5 mm pitch
Y –axis ball screw	:	16 mm dia. * 5 mm pitch

#### **LIST OF G – CODES FOR MILLING**

<b>G CODES</b>	<b>FUNCTION</b>
G00	Positioning rapid transverse
G01	Linear interpolation
G02	Circular interpolation CC
G03	Circular interpolation CCW
G04	Dwell
G20	Inch unit
G21	Metric unit
G28	Automatic zero return
G30	2 <sup>nd</sup> reference point return
G40	Tool nose radius compensation cancel

G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G43	Tool length compensation
G52	Work coordinate system 1
G54	Work coordinate system 2
G55	Work coordinate system 3
G56	Work coordinate system 4
G57	Work coordinate system 5
G58	Work coordinate system 6
G74	Left hand tapping cycle
G76	Fine boring cycle
G80	Canned cycle
G81	Drilling cycle
G82	Drilling cycle with dwell
G83	Peck drilling cycle /deep drilling cycle
G84	Tapping cycle
G85	Boring cycle/ reaming cycle
G86	Boring cycle
G87	Back boring cycle
G90	Absolute command
G91	Incremental command
G94	Feed per minute
G95	Feed per revolution
G98	Return to initial position in canned cycle
G99	Return to R point in canned cycle

### **LIST OF M CODES FOR MILLING**

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

## **DESCRIPTION OF G CODES**

### **G00 FAST TRAVERSE**

A G00 causes linear motion to the given position at the maximum feed rate from the current position that is predefined in the option file.

Examples: G00X0.0Y0.0

### **G01 LINEAR INTERPOLATION:**

A G01 causes linear motion to the position at the last specified feed rate from the current position. The feed rate for the linear motion should be mentioned in the part program.

Example: G01X30.0Y10.0F100.0

### **G02 CIRCULAR INTERPOLATION (CW)**

A G02 causes a clockwise arc to the specified position.

Example: G02X30.0Y20.0R10.0

### **G03 CIRCULAR INTERPOLATION (CCW)**

A G03 causes a counter clockwise arc to the specified position.

Example: G03X30Y20R20

### **G21 METRIC:**

A G21 cause positions to be interpreted as being in metric units (mm). This can only be at the main program. By default metric units will be taken for programming.

### **G28 GOTO REFERENCE POINT:**

A G28 causes a fast traverse to the specified position and then to the machine datum.

Example: G28U0.0W0.0

### **G90 ABSOLUTE MOVEMENT:**

All future movement will be absolute until overridden by a G91 instruction. This is the default setting.

Example:     G90  
              G01X30Y0

The position becomes (X30, Y0), irrespective of the previous position.

### **G91 INCREMENTAL MOVEMENT:**

All future movement will be incremental (i.e. relative to the current position of the tool) until overridden by a G90 instruction.

Example:     G90  
              G01X15Y10  
              G91  
              G01X2

The position becomes (X17, Y10).

### **G170 - G171 CIRCULAR POCKETING:**

It creates a circular pocket on the surface of the work piece

Example: G170 R0 P0 Q1 X0 Y0 Z-5 I0.2 J0.1 K10

R is the Z value of the top surface of the uncut pocket.

P = 0, for roughing cycle and  
= 1, for finishing cycle

Q - is the depth of cut in each run.

X, Y, Z - are the coordinates of the bottom centre point of the pocket.

I - is side finishing allowance.

J - is base finishing allowance.

K - is radius of pocket (a negative value effects CCW cutting).

Example: G71 P90 S200 R10 F60 B4000 J30

P - is cutter moment (lateral) in percent of tool diameter for next cut (P90 implies 10% tool overlap during subsequent cuts).

S - is roughing spindle speed.

B - is finishing spindle speed.

R - is roughing feed in Z direction.

F - is roughing feed in XY plane.

J - is finishing feed.

All the parameters must be specified even if they are not required for machining. For example, in a finishing operation, roughing parameters are not relevant, but they must be specified.

### **G172 - G173 RECTANGULAR POCKETING:**

It makes rectangular pocket on the surface of the work piece.

Example: G172 I50 J50 K0 P0 Q3 R0 X25 Y25 Z-6

I - is the length of pocket in X direction.

J - is the length of pocket in Y direction.

K - is the corner radius (always zero, or greater than the cutter radius. Zero corner radius is not possible. K=0 gives the minimum possible corner radius, i.e., equal to the radius of the cutter).

P = 0 for roughing cycle and 1 for finishing cycle.

Q - is the depth of cut for each pass.

R - is the absolute depth of the start of the pocket from the surface (=0 for pocket on a flat surface).

X, Y - are the coordinates of the lower left hand pocket corner.

Z - is the Z coordinate of the base of the pocket.

Example: G173 I0.5 K0.1 P75 T1 S2500 R75 F250 B3500 J200 Z5

I - is the side finish allowances.

K - is the base finish allowance.

P - is the percentage lateral shift of the tool for the next cut.

T - is the tool number.

S, B - are respectively roughing and finishing spindle speeds.

R - is the roughing feed in Z.

F - is roughing feed along XY.

J - is the finishing feed.

Z - is the safe Z position.

All the parameters must be specified, even if they are not needed.

### **G84 RIGHT HAND TAPPING CYCLE:**

Example: G84X1.0 Y1.0 Z-6.0 R1.0 P750 K1 F200

X and Y - specify the hole position.

Z - is the vase of the thread.

R - is the R point level.

P - is the delay in milliseconds.

K - is the number of repetitions (it defaults to 1).

F - is the feed.

G84 can be used with G98orG99 in the same block.



# LAB EXERCISES

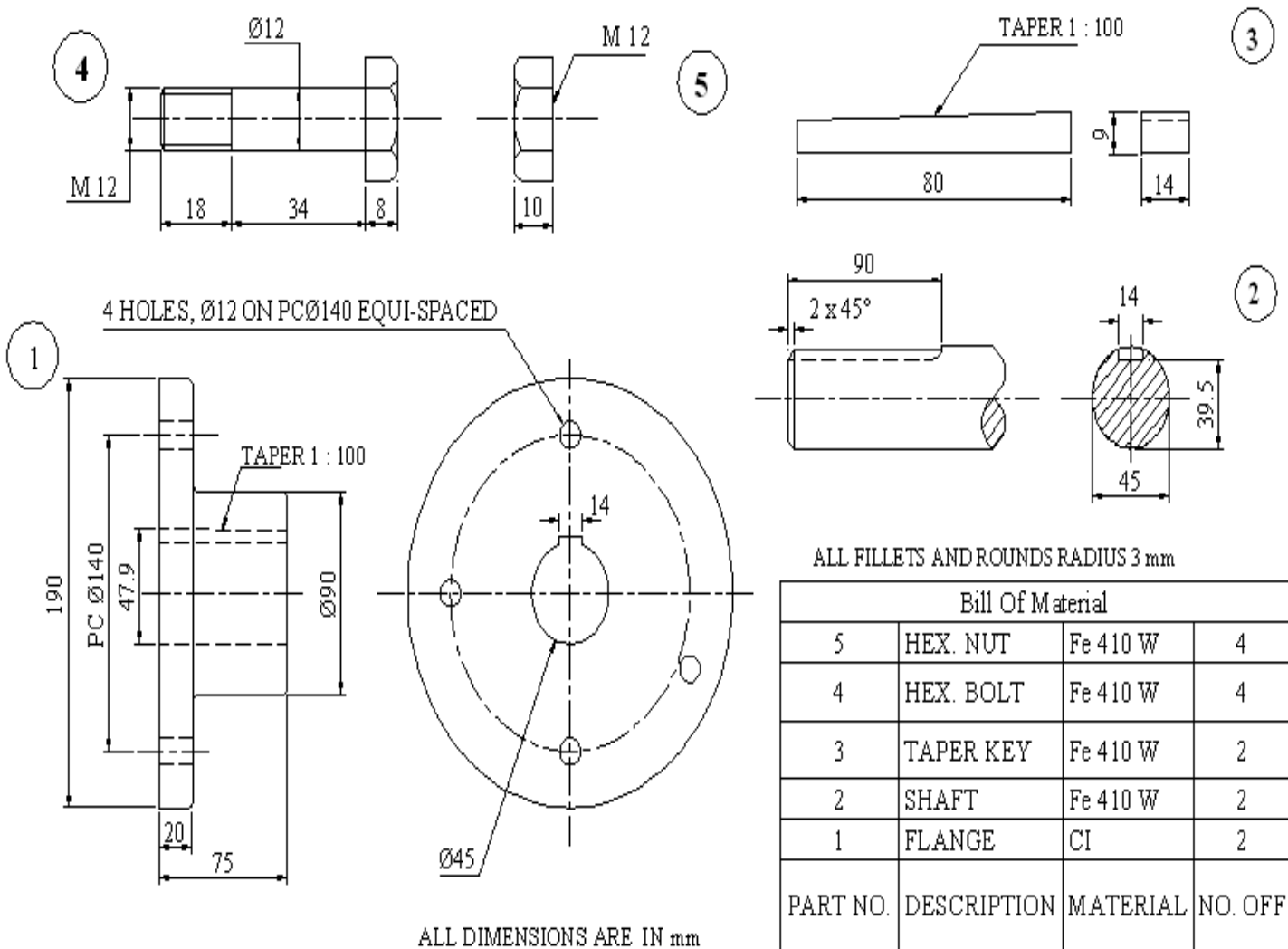
# I. COMPUTER AIDED DESIGN (CAD)

EX. NO. : 01

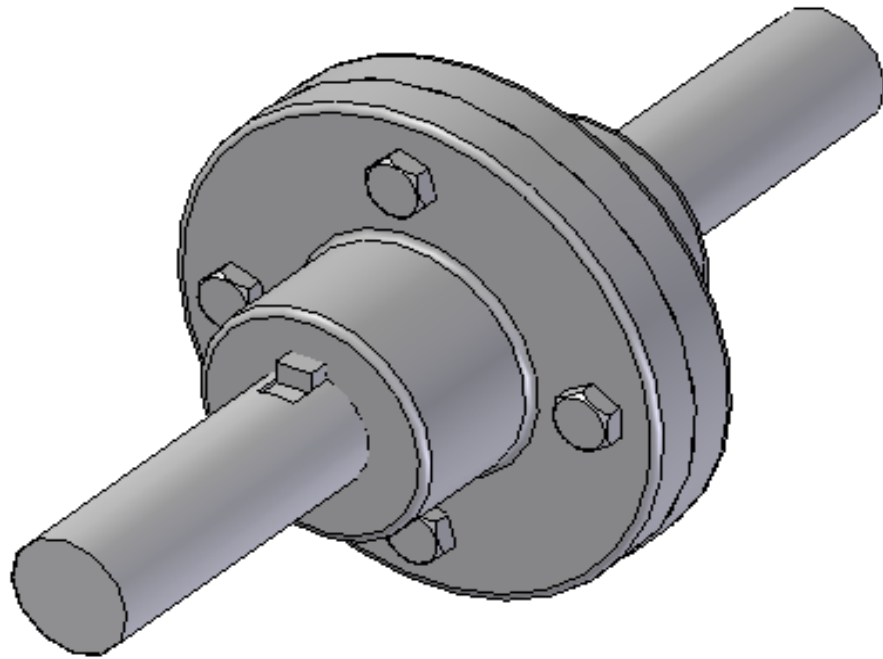
DATE :

Figure shows the details of a **FLANGED COUPLING - UNPROTECTED TYPE**.  
 Create the 3D model and Assemble the parts of the Flanged Coupling - Unprotected type using Autodesk Inventor Professional 2008.

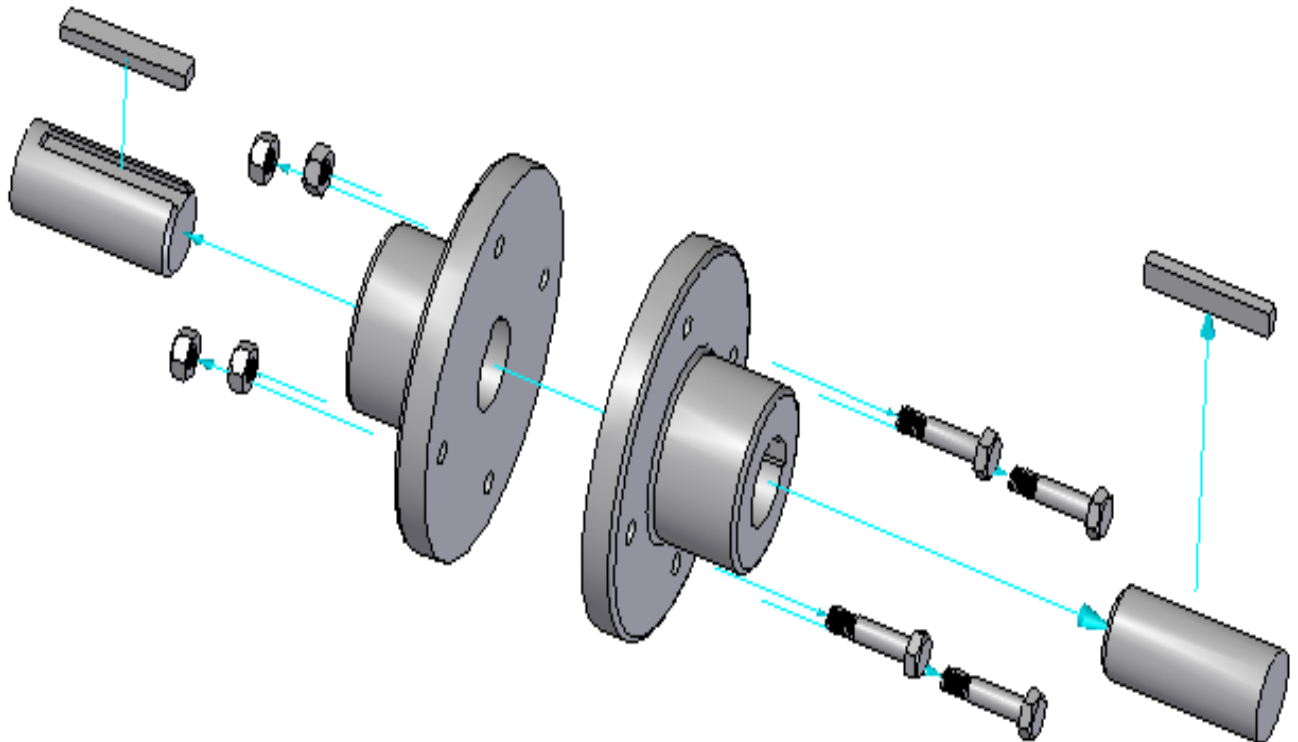
## Details of Flanged Coupling - Unprotected Type



## Assembled View



## Exploded View

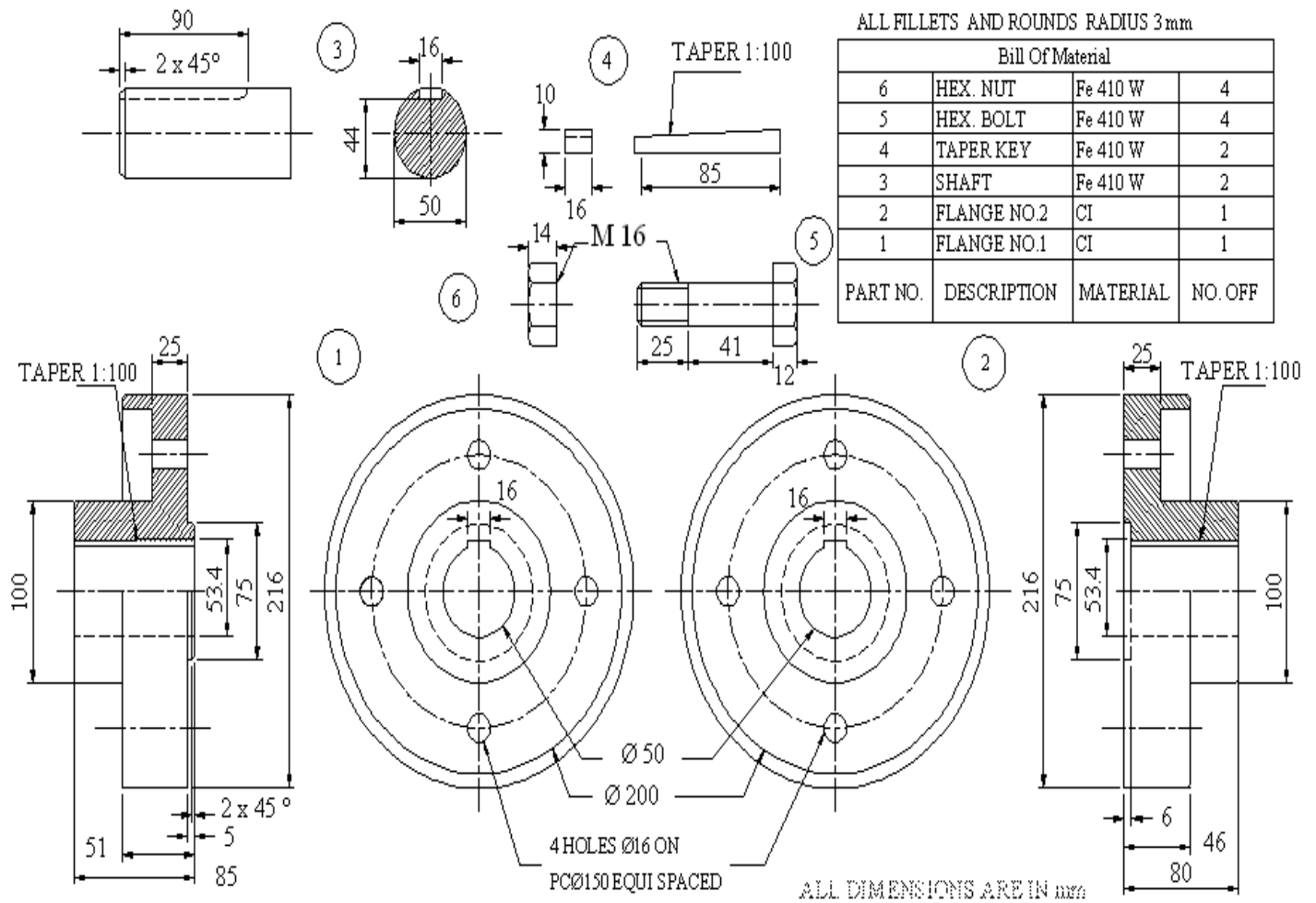


EX. NO. : 02

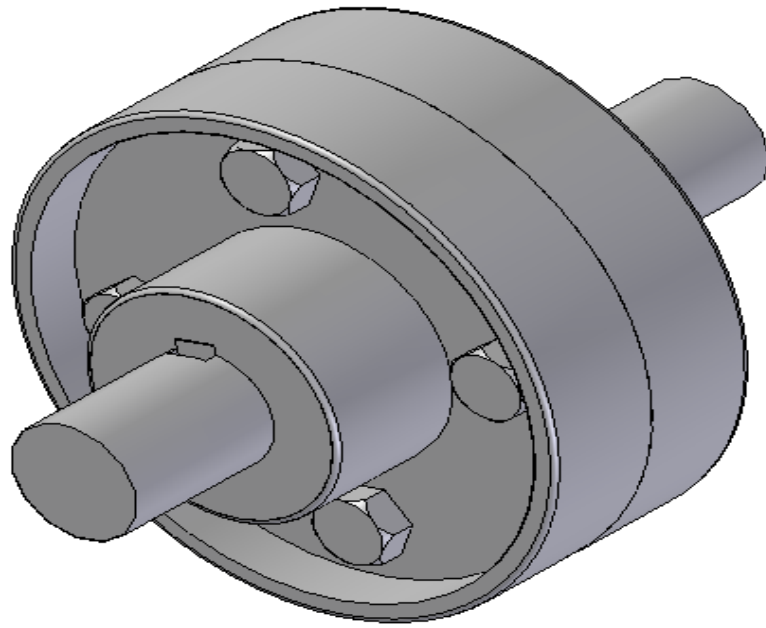
DATE :

**Figure shows the details of a FLANGED COUPLING - PROTECTED TYPE. Create the 3D model and Assemble the parts of the Flanged Coupling - Protected type using Autodesk Inventor Professional 2008.**

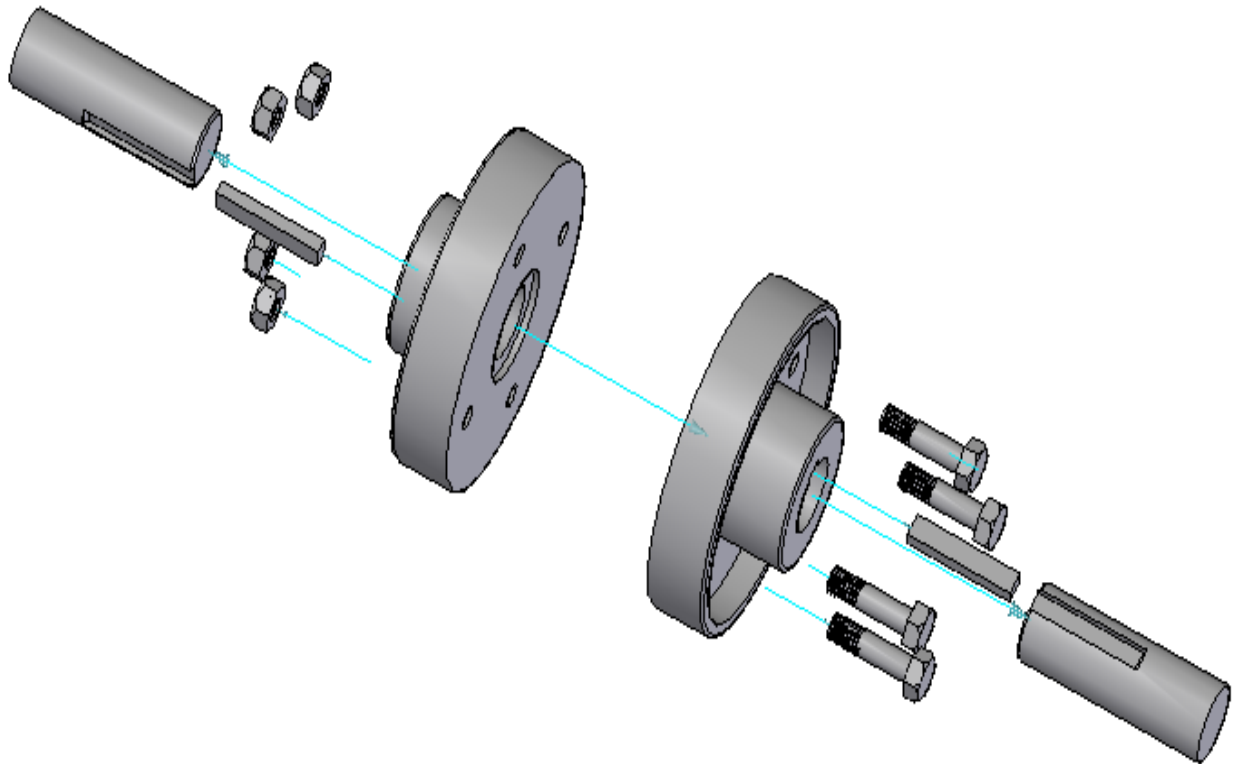
**Details of Flanged Coupling Protected Type**



## Assembled View



## Exploded View

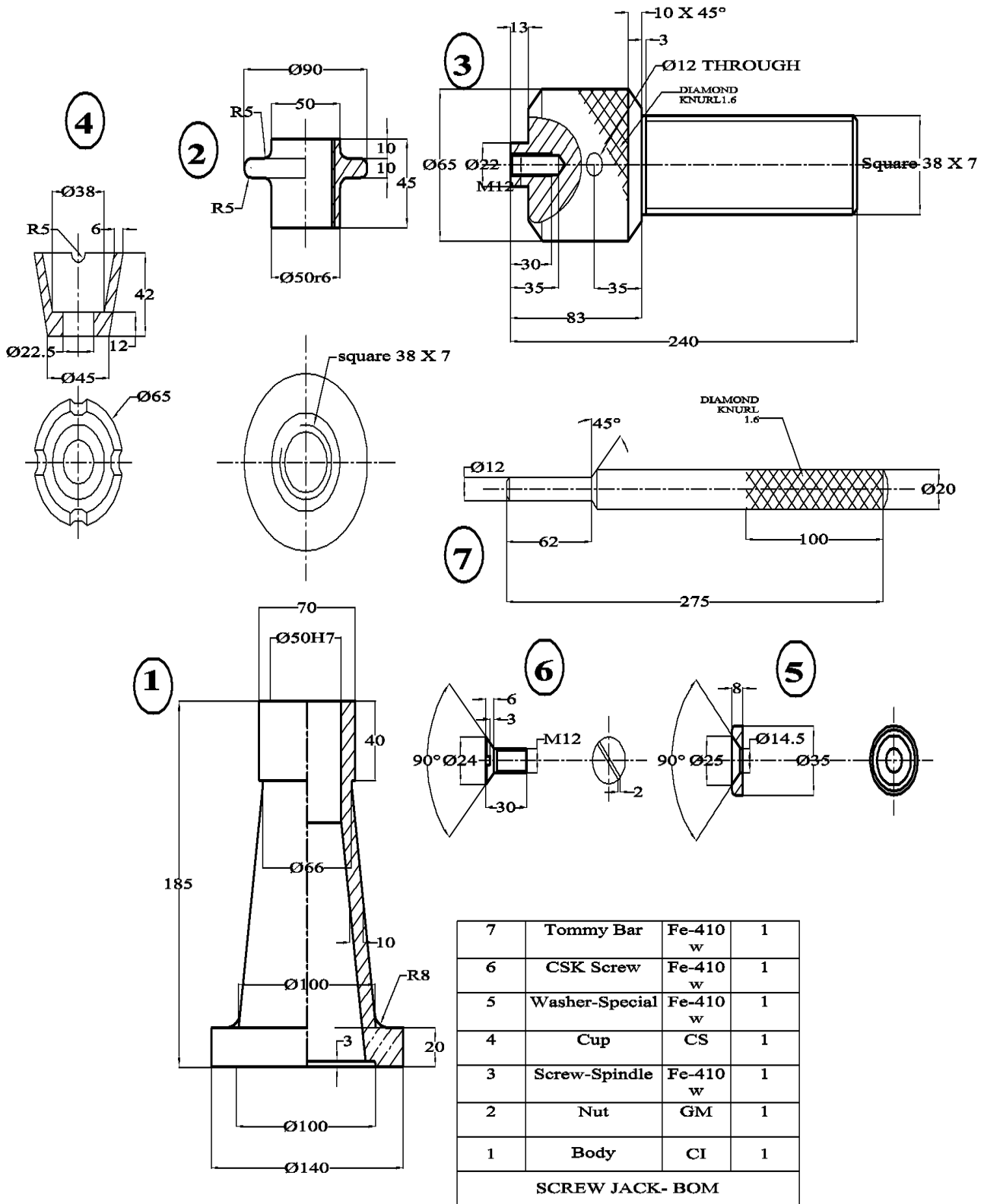


EX. NO. : 03

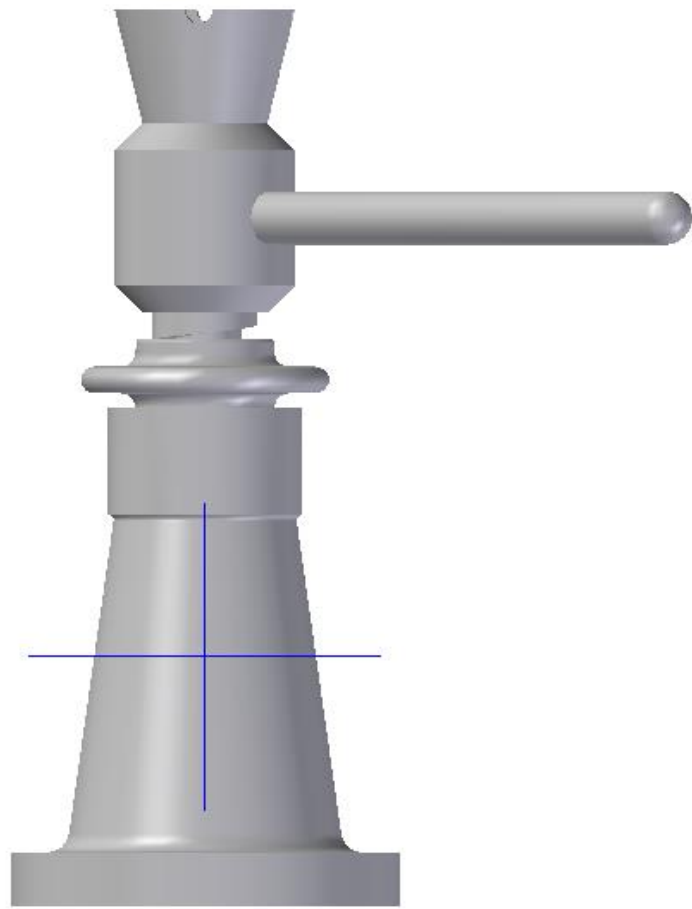
DATE :

Figure shows the details of SCREW JACK. Create the 3D model and Assemble the parts of the Screw Jack using Autodesk Inventor Professional 2008.

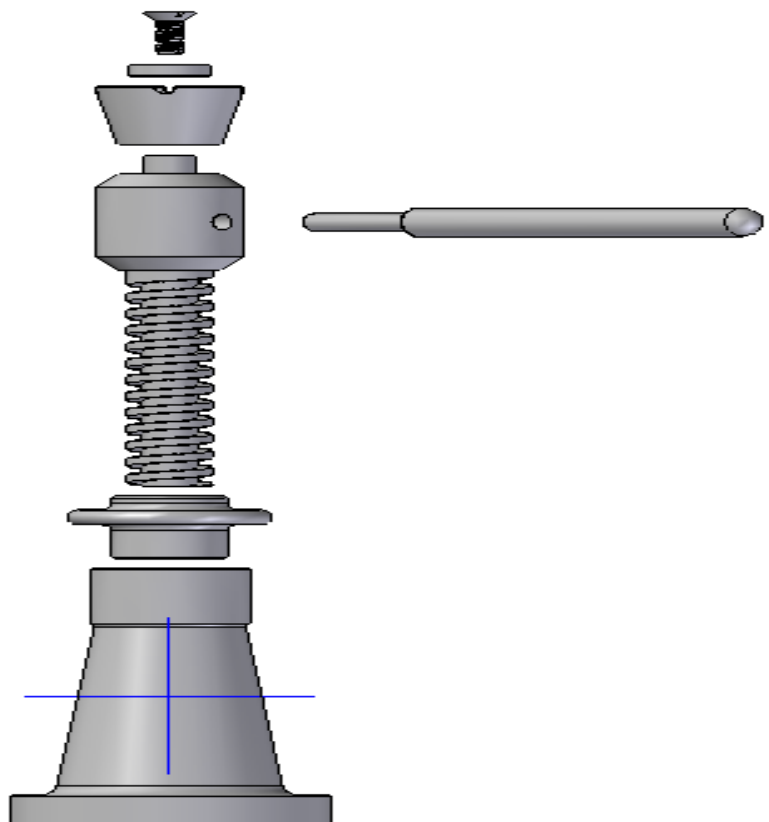
Details of Screw Jack



## Assembled View



## Exploded View

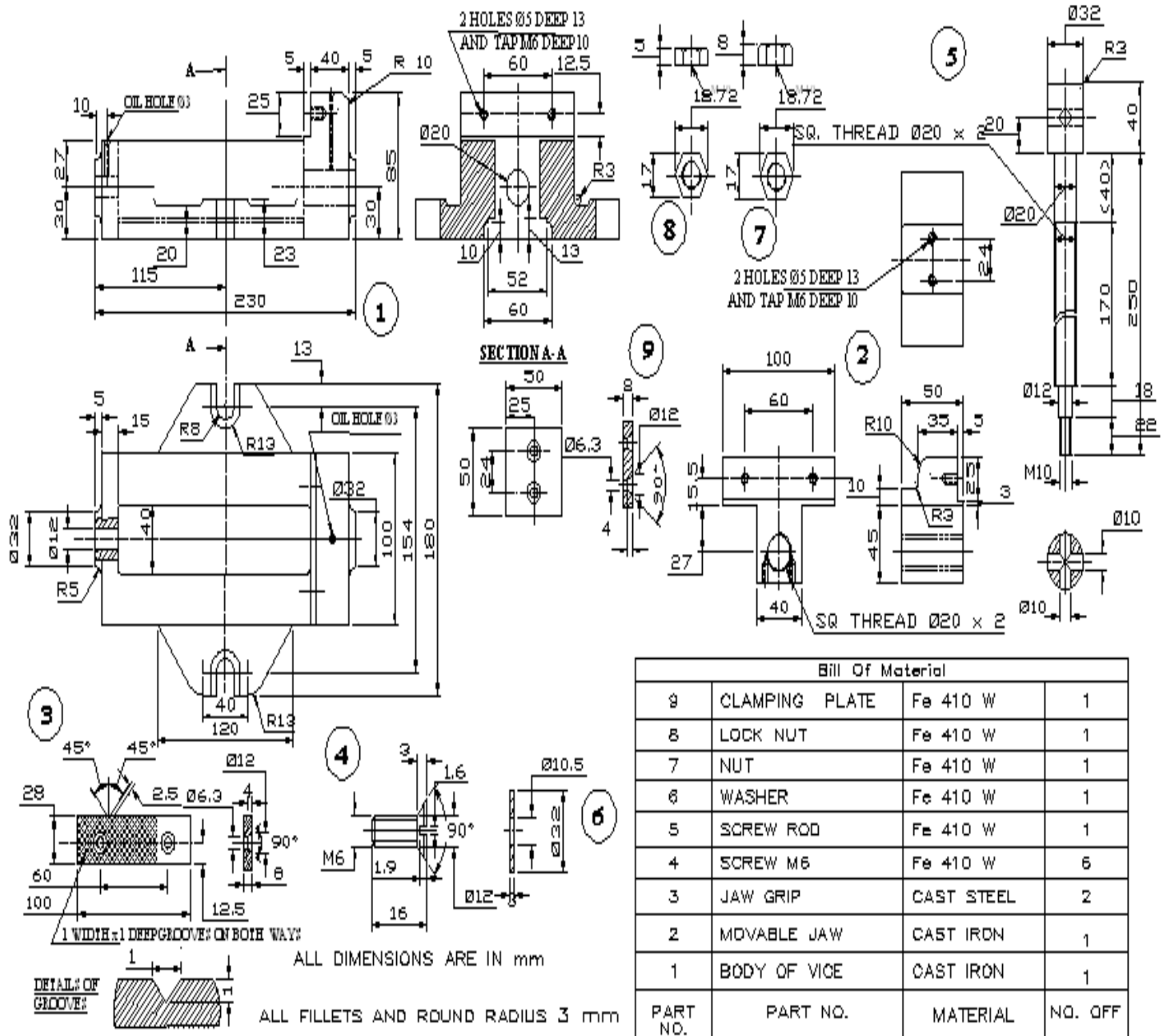


EX. NO. : 04

DATE :

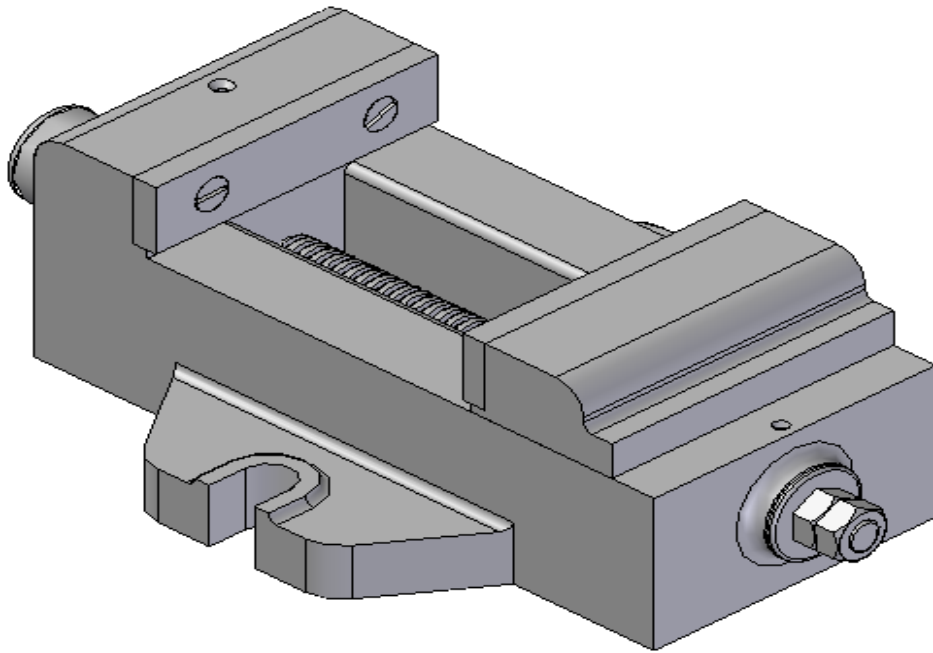
Figure shows the details of MACHINE VICE. Create the 3D model and Assemble the parts of the Machine Vice using Autodesk Inventor Professional 2008.

Details of Machine Vice

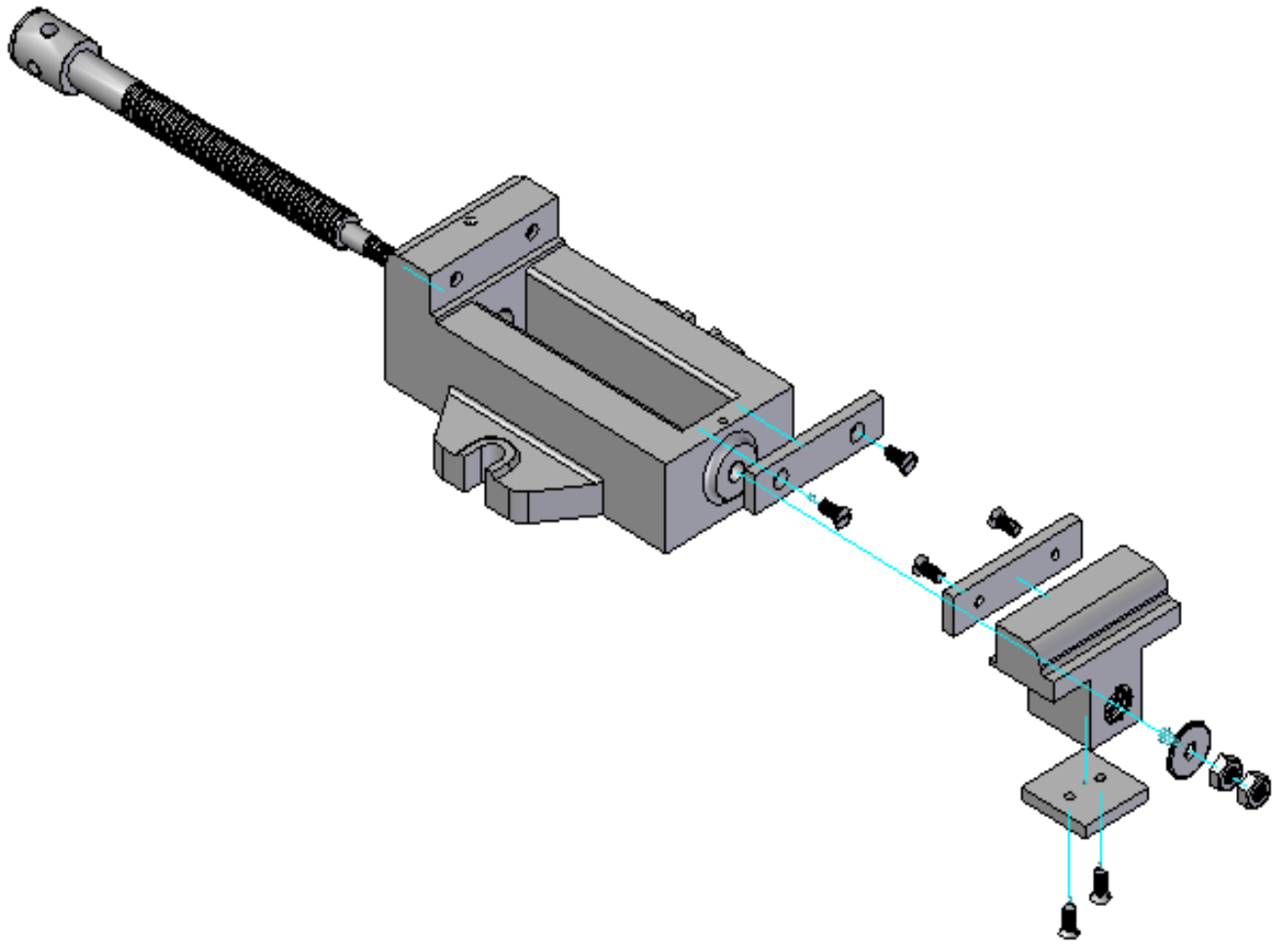




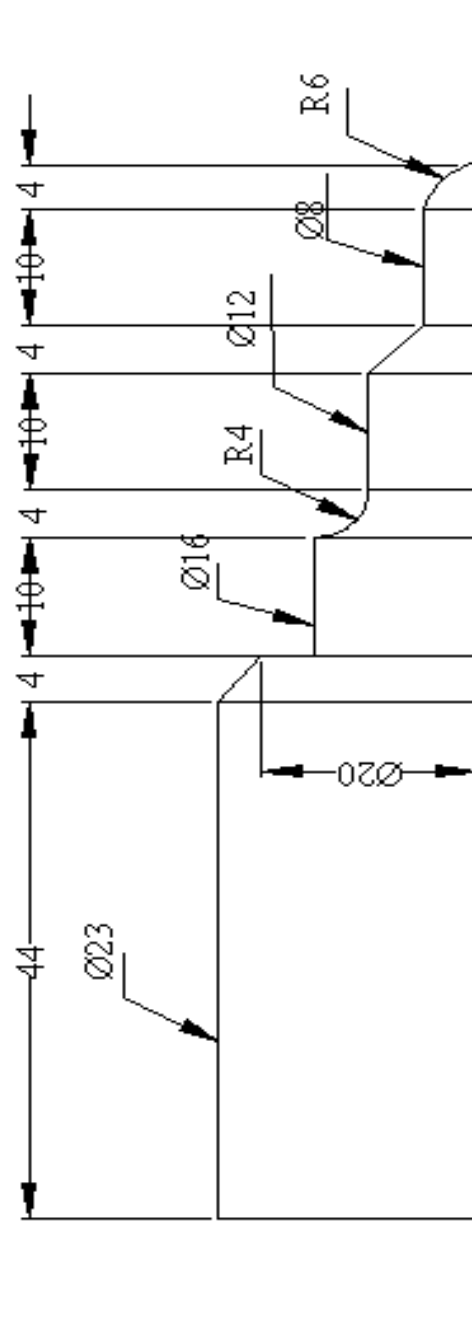
## Assembled View



## Exploded View



## 5. MULTIPLE TURNING



ALL DIMENSION ARE IN MM

## CNC – MANUAL PART PROGRAMMING

EX. NO. : 05

DATE :

### MULTIPLE TURNING

#### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

#### **PROGRAM:**

```
o0011;
[BILLET X22 Z70;
G21 G97 G98;
G28 U0 W0;
M06 T01;
M03 S1000;
G00 X21 Z1;
G01 Z-42 F50;
X22;
G00 Z1;
X20;
G01 Z-42;
X22;
G00 Z1;
X19;
G01 Z-42;
X22;
G00 Z1;
X18;
G01 Z-42;
X22;
G00 Z1;
X17;
G01 Z-42;
X22;
G00 Z1;
X16;
G01 Z-42;
X22;
G00 Z1;
X15;
G01 Z-28;
X22;
G00 Z1;
X14;
G01 Z-28;
X22;
G00 Z1;
X13;
G01 Z-28;
X22;
G00 Z1;
X12;
G01 Z-28;
X22;
G00 Z1;
X11;
G01 Z-14;
X22;
G01 Z1;
X10;
G01 Z-14;
X22;
G00 Z1;
X9;
G01 Z-14;
X22;
G01 Z1;
X8;
G01 Z-14;
X22;
G01 Z1;
X21 Z-41;
G01 Z-42;
X22 Z-45;
G00 Z1;
X20 Z-41;
G01 Z-42;
X22 Z-48;
G00 Z1;
X11 Z-13;
G01 Z-14;
X12 Z-15;
G00 Z1;
X10 Z-13;
G01 Z-14;
```

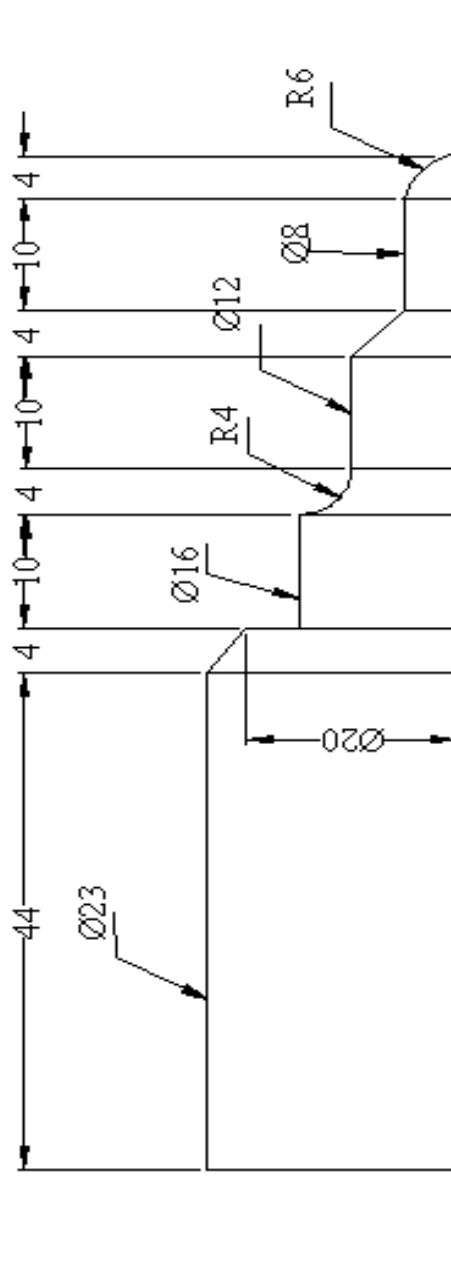
X12 Z-16;  
G00 Z1;  
X9 Z-13;  
G01 Z-14;  
X12 Z-17;  
G00 Z1;  
X8 Z-13;  
G01 Z-14;  
X12 Z-18;  
G00 Z1;  
X15 Z-27;  
G01 Z-28;  
G02 X16 Z-29 R4;  
G00 Z1;  
X14 Z-27;  
G01 Z-28;  
G02 X16 Z-30 R4;  
G00 Z1;  
X13 Z-27;  
G01 Z-28;  
G02 X16 Z-31 R4;  
G00 Z1;

X12 Z-27;  
G01 Z-28;  
G02 X16 Z-32 R4;  
G00 Z1;  
X6;  
G01 Z0;  
G03 X8 Z-1 R6;  
G00 Z1;  
X4;  
G01 Z0;  
G03 X8 Z-2 R6;  
G00 Z1;  
X2;  
G01 Z0;  
G03 X8 Z-3 R6;  
G00 Z1;  
X0;  
G01 Z0;  
G03 X8 Z-4 R6;  
G28 U0 W0;  
M05;  
M30;

## **RESULT:**

Thus the part program for the given model is written and verified through simulation.

5A. MULTIPLE TURNING  
USING MULTIPLE TURNING CYCLE (G71)



ALL DIMENSION ARE IN MM

EX. NO. : 05A  
DATE :

## **MULTIPLE TURNING BY USING MULTIPLE TURNING CYCLE (G71)**

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

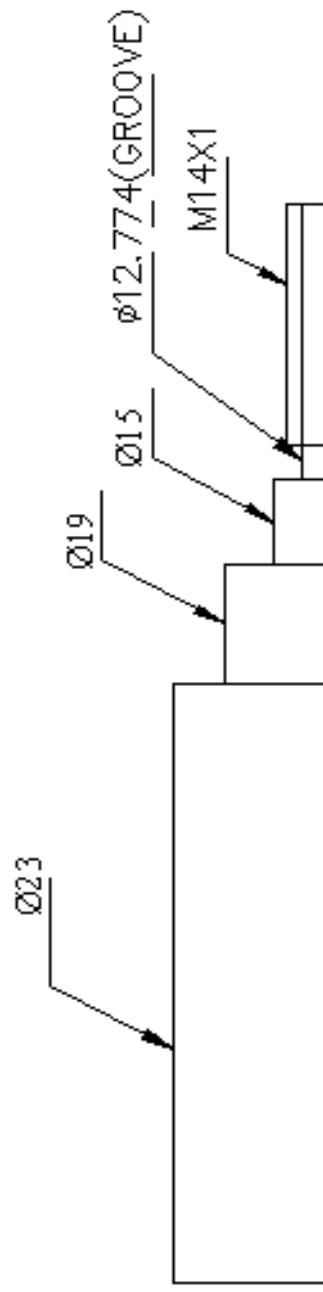
### **PROGRAM:**

```
O0011;
[BILLETX23.0Z90.0;
G28U0.0W0.0;
M06T01;
G00X23.0Z10.0;
S1300M03;
G00X23.0Z1.0;
G01Z0.0F10.0;
G71U0.5R1.0;
G71P100Q120U0.2W0.2F50;
N100G01X0.0;
G03X8.0Z-4.0R6;
G01Z-14.0;
X12.0Z-18.0;
Z-28.0;
G02X16.0Z-32.0R4.0;
G01Z-42.0;
X20.0;
X23.0Z-46.0;
X24.0;
N120Z10.0;
G70P100Q120S3000F10.0;
S0;
G28U0.0W0.0;
M05;
M30;
```

### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

## 6. GROOVING AND THREAD CUTTING





EX. NO. : 06  
DATE :

## GROOVING AND THREAD CUTTING

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

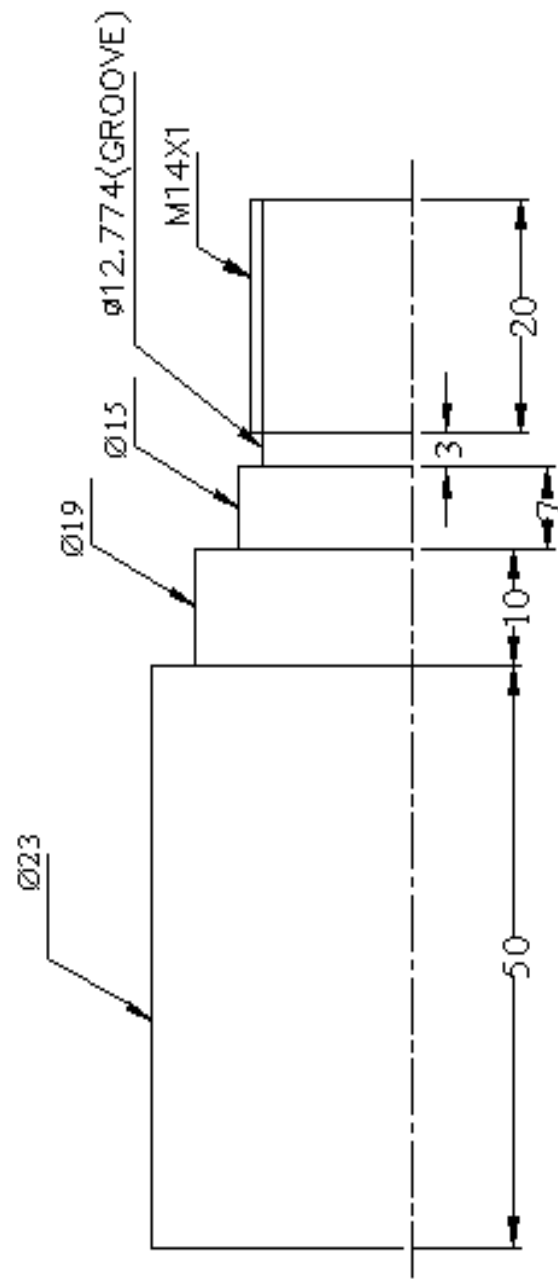
### **PROGRAM:**

```
O 0012;
[BILLET X22 Z70;
G21 G97 G98;
G28 U0 W0;
M06 T01;
M03 S1000;
G00 X22 Z1;
G01 Z0 F50;
G71 U0.5 R1;
G71 P100 Q200 U0.2 W0.2;
N100 G01 X14;
Z-23;
X15;
Z-30;
X19;
Z-40;
N200 X22;
M05;
G00Z10.0
S0
G28U0.0W0.0
M05
M30;
G28 U0 W0;
M06 T02;
M03 S1200;
G00 X22 Z1;
G70 P100 Q200;
G28 U0 W0;
M05;
M06 T03;
M03 S250;
G00 X14 Z1;
G01 Z0;
G92 X13.8 Z-20 F1;
X13.6;
X13.4;
X13.2;
X13;
X12.8;
X12.774;
G28 U0 W0;
M05;
M30;
```

### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

## 6A. GROOVING AND THREAD CUTTING USING CYCLE



ALL DIMENSIONS ARE IN MM

EX. NO. : 06A

DATE :

## GROOVING AND THREAD CUTTING USING CYCLES

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

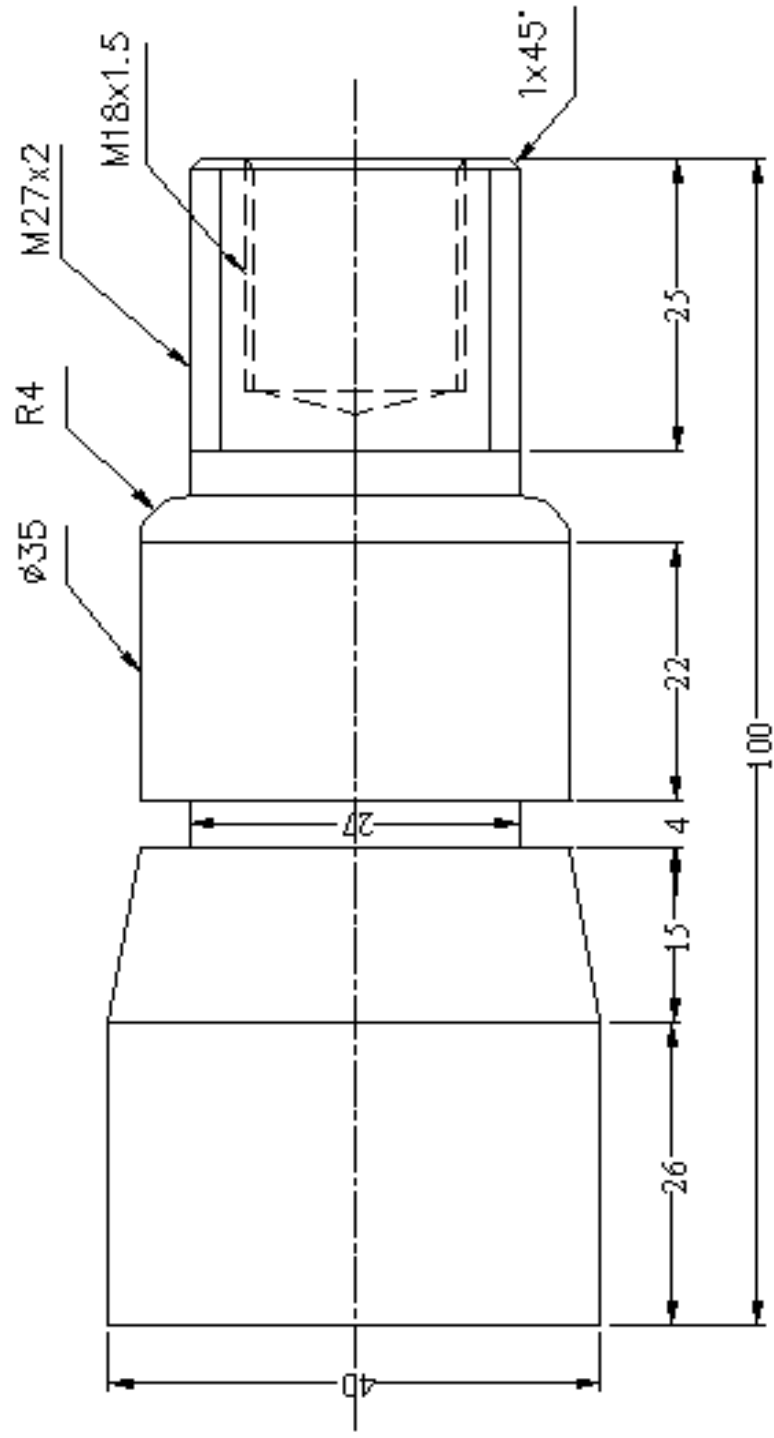
### **PROGRAM:**

```
O 0013;
[BILLET X22 Z70;
G21 G97 G98;
G28 U0 W0;
M06 T01;
M03 S1000;
G00 X22 Z1;
G01 Z0 F50;
G71 U0.5 R1;
G71 P100 Q200 U0.2 W0.2;
N100 G01 X14;
Z-23;
X15;
Z-30;
X19;
Z-40;
N200 X22;
M05;
G28 U0 W0;
M06 T02;
M03 S1200;
G00 X22 Z1;
G70 P100 Q200;
G28 U0 QW0;
M05;
G28 U0 W0;
M06 T03;
M03 S250;
G00 X14 Z1;
G76 P031560 Q150 R0.15;
G76 X12.774 Z-20 P0613 Q300 F1;
G00 X14 Z10;
G28 U0 W0;
M05;
M30;
```

### **RESULT**

Thus the part program for the given model is written and verified through simulation.

7. TURNING, TAPER-TURNING, GROOVING & TAPPING



ALL DIMENSIONS ARE IN MM

EX. NO. : 07

DATE :

### **TURNING, TAPER-TURNING, GROOVING & TAPPING – CNC LATHE**

#### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

#### **PROGRAM:**

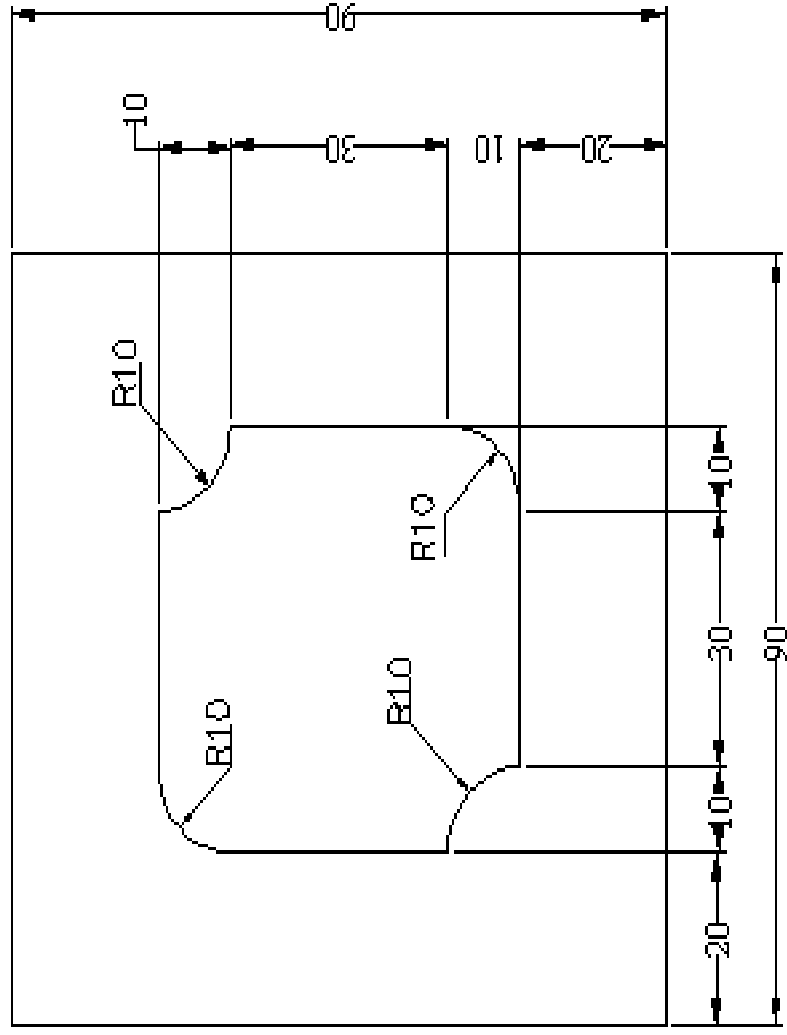
```
O 0014;
[BILLET X22 Z70;
G21 G97 G98;
G28 U0 W0;
M06 T01;
M03 S1000;
G00 X22 Z1;
G01 Z0 F50;
G71 U0.5 R1;
G71 P100 Q200 U0.2 W0.2;
N100 G01 X18;
Z-30;
N200 X22 Z-35;
M05;
G28 U0 W0;
M06 T02;
M03 S1200;
G00 X22 Z1;
G70 P100 Q200;
G28 U0 W0;

M05;
G28 U0 W0;
M06 T03;
M03 S1000;
G00 X0 Z1;
G74 R1;
G74 Z-25 Q5000 F50;
G00 Z1;
G28 U0 W0;
M05;
G28 U0 W0;
M06 T04;
M03 S1000;
G00 X0 Z1;
G01 X12 Z1;
G90 X12.5 Z-20 F50;
X13;
X13.5;
G28 U0 W0;
M05;
M30;
```

#### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

8. LINEAR WITH CIRCULAR INTERPOLATION



ALL DIMENSIONS ARE IN MM

EX. NO. : 08

DATE :

### **LINEAR WITH CIRCULAR INTERPOLATION**

#### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

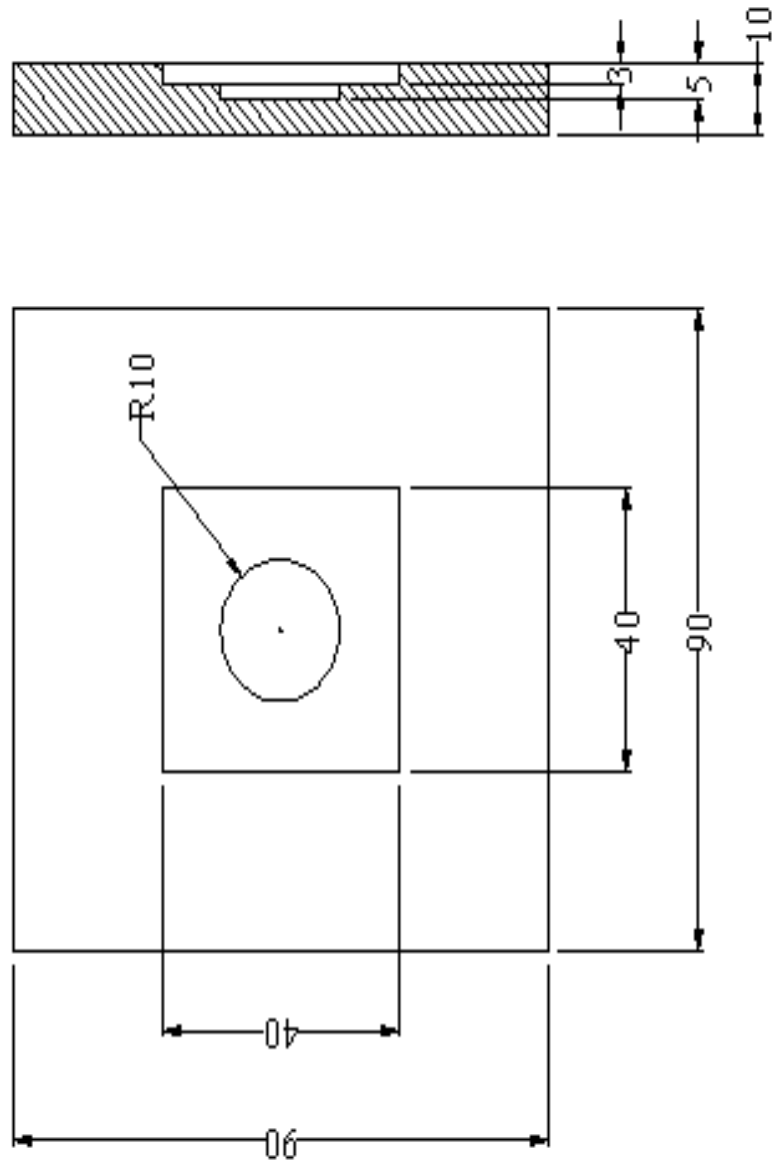
#### **PROGRAM:**

```
O0010;  
[BILLET X90 Y90 Z12;  
G21 G94 G90;  
G28 X0. Y0. Z0.;  
M06 T01 D06;  
M03 S1000;  
G00 X-15. Y-25. Z50.;  
G01 Z-3. F50.;  
X15.;  
G03 X25. Y-15. R10.;  
G01 Y15.;  
G02 X15. Y25. R10.;  
G01 X-15.;  
G03 X-25. Y15. R10.;  
G01 X-25. Y-15.;  
G02 X-15. Y-25. R10.;  
G00 Z50.;  
G28 X0. Y0. Z0.;  
M05;  
M30;
```

#### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

9. RECTANGULAR AND CIRCULAR POCKETING



ALL DIMENSIONS ARE IN MM



EX. NO. : 09

DATE :

## RECTANGULAR AND CIRCULAR POCKETING

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

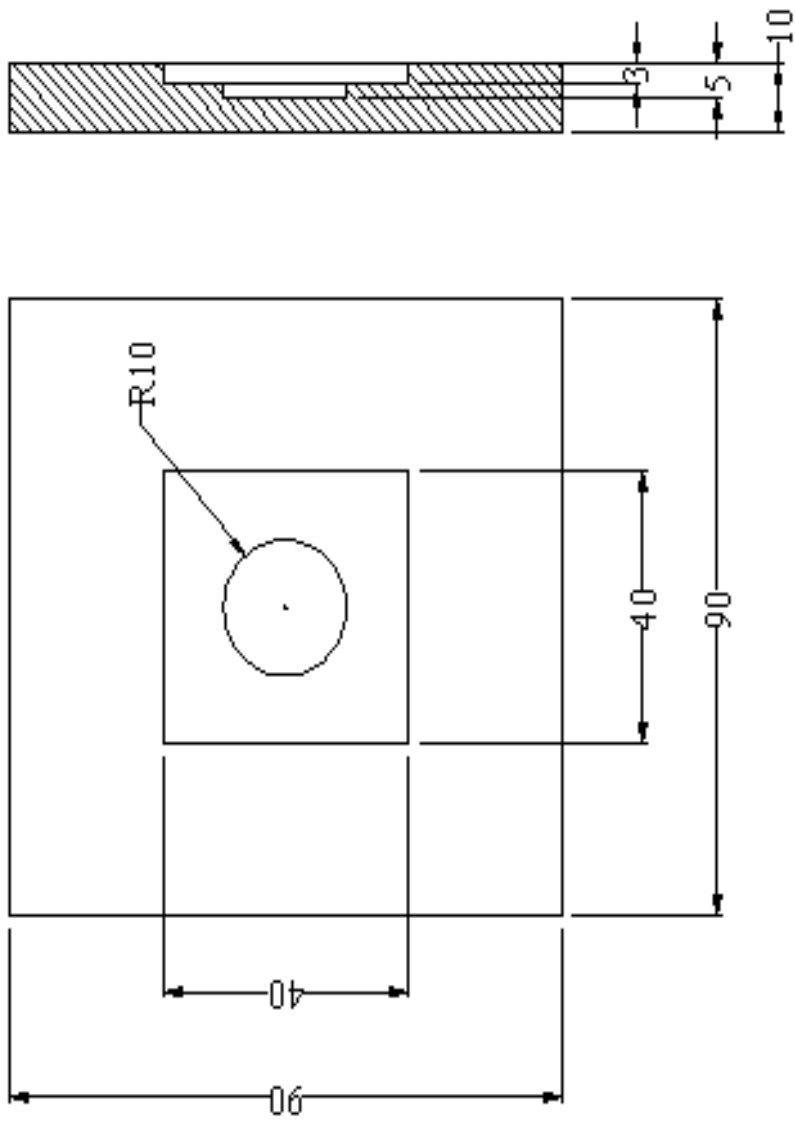
### **PROGRAM:**

```
O 0011;  
[BILLET X90 Y90 Z12;  
G21 G94 G90;  
G28 X0. Y0. Z0.;  
M06 T01 D06;  
M03 S1000;  
G00 X-20. Y-20. Z50.;  
G01 Z-3. F50.;  
X20.;  
Y20.;  
X-20.;  
Y-20.;  
G00 Z50.;  
G28 X0. Y0. Z0.;  
M05;  
M06 T02 D06;  
M03 S1000;  
G00 X0. Y0. Z50.;  
G01 Z-5. F50.;  
G00 Z50.;  
G28 X0. Y0. Z0.;  
M05;  
M30;
```

### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

9A. RECTANGULAR AND CIRCULAR POCKETING USING CYCLE



ALL DIMENSIONS ARE IN MM

EX. NO. : 09A

DATE :

## **RECTANGULAR AND CIRCULAR POCKETING USING CYCLE**

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

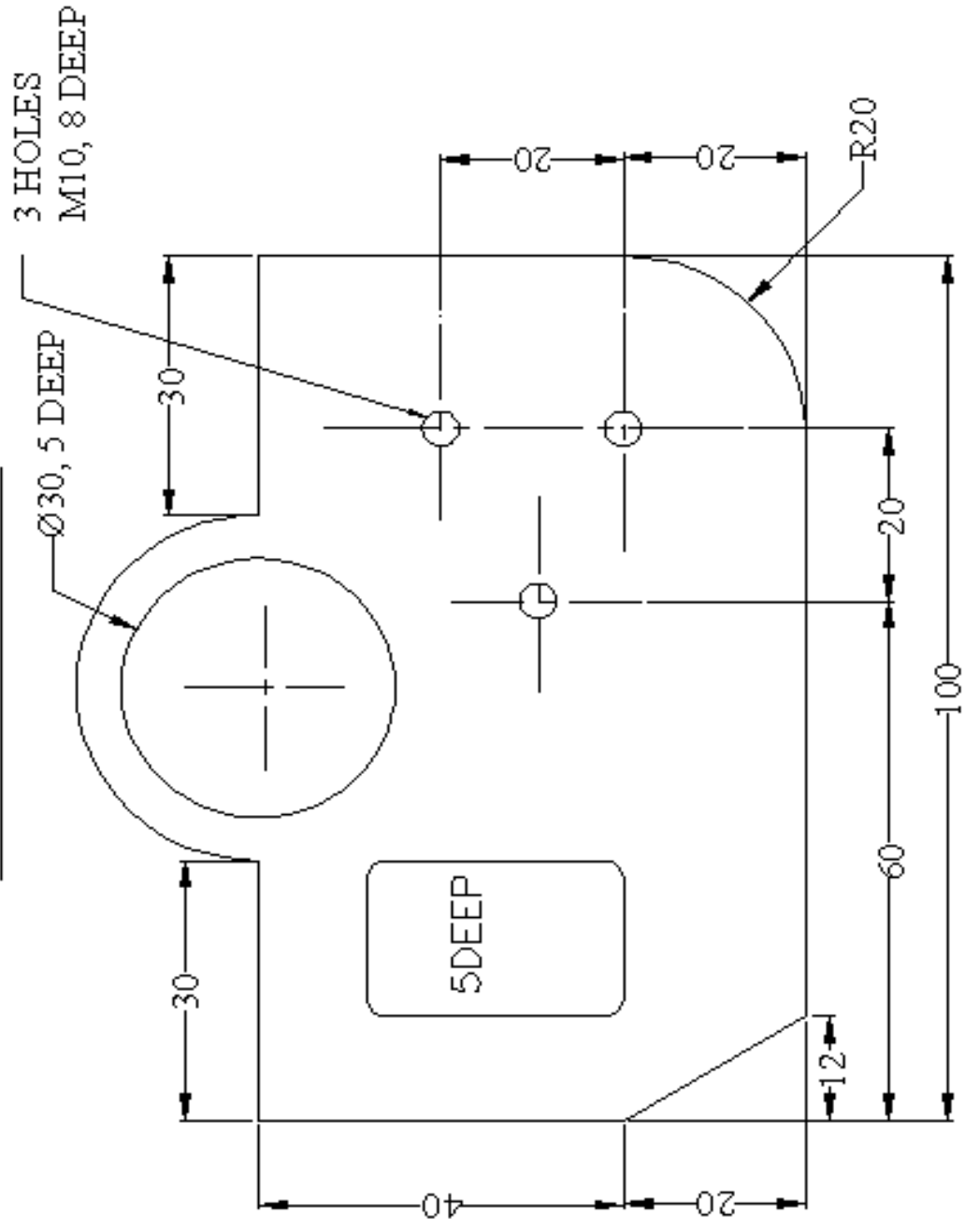
### **PROGRAM:**

```
O 0012;
[BILLET X90 Y90 Z12;
G21 G94 G90;
G28 X0. Y0. Z0.;
M06 T01 D06;
M03 S1000;
G00 X-20. Y-20. Z1.;
G172 I40. J40. K0. P0 Q0.5 R0. X-20. Y-20. Z-3.;
G173 I0.1 K0.1 P50 T1 S1000 R40 F60 B1200 J50 Z5;
G172 I40. J40. K0. P1 Q0.5 R0. X-20. Y-20. Z-3.;
G173 I0.1 K0.1 P50 T1 S1000 R40 F60 B1200 J50 Z5;
G00 Z50.;
G170 R-3. P0 Q0.5 X0. Y0. Z-5. I0.1 J0.1 K10.;
G171 P50 S1000 R10 F50 B1200 J50;
G170 R-3. P1 Q0.5 X0. Y0. Z-5. I0.1 J0.1 K10.;
G171 P50 S1000 R10 F50 B1200 J50;
G00 Z50.;
G28 X0. Y0. Z0.;
M05;
M30;
```

### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

### 10. PROFILE MILLING



ALL DIMENSIONS ARE IN MM

EX. NO. : 10

DATE :

## PROFILE MILLING

### **AIM:**

To write a part program and simulate the tool path for the given model using fanuc controller.

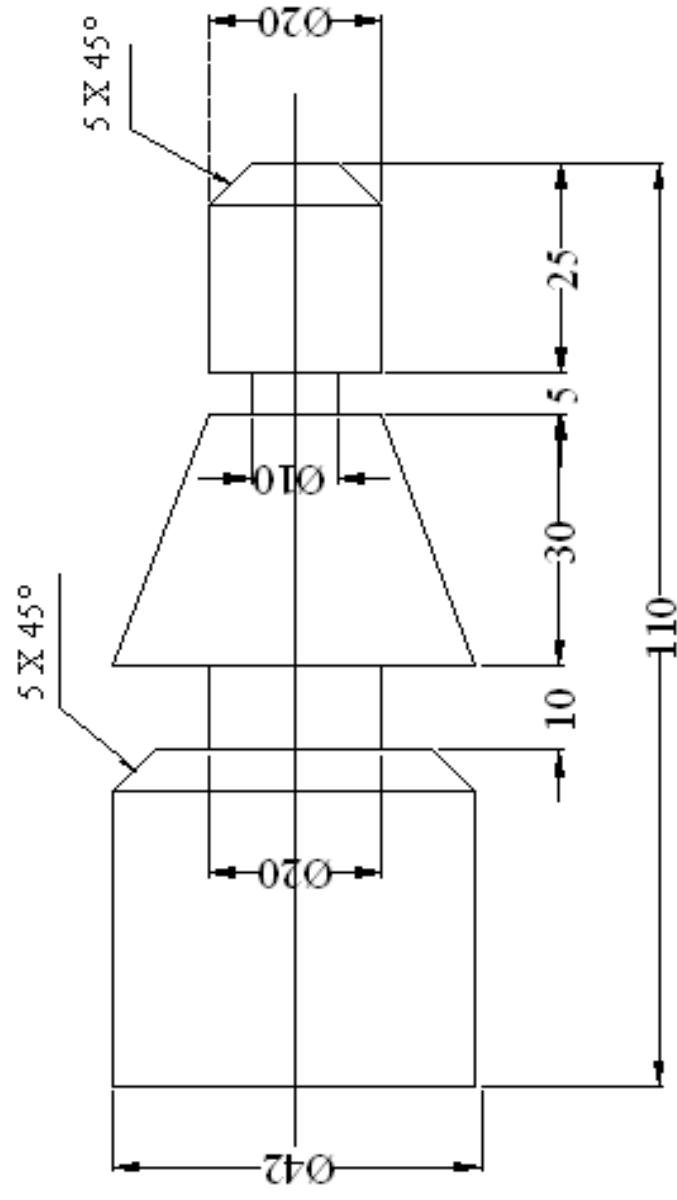
### **PROGRAM:**

```
O 0013;
[BILLET X90 Y90 Z12;
G21 G94 G90;
G28 X0. Y0. Z0.;
M06 T01 D06;
M03 S1000;
G00 X-23. Y-35. Z50.;
G01 Z-3. F50.;
X15.;
G03 X35. Y-15. R20.;
G01 Y15.;
X20.;
G03 X-20. Y15. R20.;
G01 X-35.;
Y-15.;
X-23. Y-35.;
G00 Z50.;
G28 X0. Y0. Z0.;
M05;
M06 T02 D12;
M03 S1000;
G00 X0. Y0. Z100.;
G83 G99 X-5. Y0. Z-8. Q2 R2. F50.;
X15. Y-10.;
G98 Y10.;
M05;
G28 X0. Y0. Z0.;
M06 T03 D06;
M03 S1000;
G00 X-23. Y-15. Z10.;
G172 I20. J30. K0. P0 Q0.5 R0. X-23.
Y-15. Z-5.;
G173 I0.1 K0.1 P50 T1 S1000 R40
F60 B1200 J50 Z5;
G172 I20. J30. K0. P1 Q0.5 R0. X-23.
Y-15. Z-5.;
G173 I0.1 K0.1 P50 T1 S1000 R40
F60 B1200 J50 Z5;
G00 Z50.;
G170 R0. P0 Q0.5 X0. Y15. Z-5. I0.1
J0.1 K15.;
G171 P50 S1000 R10 F50 B1200 J50;
G170 R0. P1 Q0.5 X0. Y15. Z-5. I0.1
J0.1 K15.;
G171 P50 S1000 R10 F50 B1200 J50;
G00 Z50.;
G28 X0. Y0. Z0.;
M05;
M30;
```

### **RESULT:**

Thus the part program for the given model is written and verified through simulation.

## 11. GROOVING



BLANK :  $\varnothing 60 \times 120$

MATERIAL : ALUMINIUM

ALL DIMENSIONS ARE IN mm

## C. SIMULATION AND NC CODE GENERATION

EX. NO. : 11

DATE :

### GROOVING

#### **AIM:**

To generate NC code for a given component using capsturn.

#### **PROCEDURE:**

1.New Program → 2.Work Setup→3.Geometry → 4.Machining → 5.Machine Selection  
→ 6.Toolpath Generation → 7.NC Code Generation

#### **1. FILE - NEW**

#### **2. WORK SETUP**

##### **a) SETUP DATA-1 (Compulsory)**

Units	:	mm
W/p material	:	Al
Holding	:	External
Part load +un load time	:	30 sec
Limiting spindle speed	:	3000 rpm
Jaw position absolute	:	-115 mm
Clearance (z)	:	5 mm
Clearance ID	:	0 mm
Clearance OD	:	70 mm

##### **b) SETUP DATA -2 (Compulsory)**

Tool turret position	:	Rear turret X = 200; Z = 100
Program mode	:	Diameter
Default turret position	:	Rear turret
Default tool mounting	:	Face up
Jaw size	:	Length = 20; Height = 20

##### **c) DOCUMENTATION (Optional)**

Part number	:	
Part name	:	
Set up number	:	
Fixture	:	
Programmer	:	
Remark	:	

Save the file as EX11.

### **3. GEOMETRY**

#### **A. Select the geometry mode**

Click geometry mode → Draw a part → Define a part → Define a blank.

#### **B. To draw a part**

Click draw → Choose multiline → Enter the values of X, Z in the input box.

Enter the following data in the input box:

Specify the first point	:	0,0
Specify the next point	:	20,0
Specify the next point	:	20,-25
Specify the next point	:	10,-25
Specify the next point	:	10,-30
Specify the next point	:	20,-30
Specify the next point	:	42,-60
Specify the next point	:	20,-60
Specify the next point	:	20,-70.
Specify the next point	:	42,-70
Specify the next point	:	42,-110

Give done to complete the line & Select zoom all.

#### **To draw chamfer**

Click draw → Select chamfer → Select chamfer type → Angle & Distance  
Enter the values in the input box (Distance = 5 & Angle = 45°) → Select done →  
Select the lines by using the mouse → select done to complete the chamfer.

#### **C. To define the part**

Click draw → Define part → Select the object by using mouse → Select YES.

#### **D. To define the blank**

Click draw → Define blank → Enter the values for blank in the standard blank box  
(Length = 120; Diameter = 60; Start Z = 0) → Enter OK.

### **4. MACHINING**

#### **Select the machining mode**

Machining operations involved:

- A. Turning the job using PTGNL 1616 H11 R0.4 tool
- B. Grooving the job using 32\*32, 3.00W, 0.10R, 20DEPTH, LH tool.

#### **A. Turning**

Click machining → Turning → Contour turn → New tool



Enter the following data in the select tool page:

Tool library	:	mm
Selected tool	:	PTGNL 1616 H11 R.4
Selected tool material	:	Coated carbide
Tool number	:	1
Tool offset number	:	1

Enter **ok** to select the tool.

Select the first point on the part by using the mouse → Select the second point on the part by using the mouse.

Enter the following data in the contour turn page:

Operation name	:	Contour turn
Process	:	Roughing & finishing
Depth of cut	:	2 mm
Finish allowance	:	X = 0.1, Z = 0.1

Leave the other data as it is & Enter **ok**.

## **B. Grooving**

Click machining → Turning → Groove → Radial → New tool

Enter the following data in the select tool page:

Tool library	:	General mm
Selected tool	:	32*32,3.00W, 0.10R, 20DEPTH, LH
Selected tool material	:	Coated carbide
Tool number	:	2
Tool offset number	:	2

Select the groove at the first end → Select the other end of the groove

Enter the following data in the external groove page:

Operation name	:	External groove
Process	:	Roughing & Finishing
Finish allowance	:	0.1
Start location	:	Right
Cutting method	:	Plunge cut → Enter <b>ok</b>

Repeat the same procedure for machining the second groove in the part.

## **5. MACHINE SELECTION**

Click machine → Choose the **Fanuc OiT** m/c type from machine list → Click select.

## **6. TOOLPATH GENERATION**

Click toolpath → Click solid mode → Click start to see the toolpath simulation.

## **7. TO GENERATE NC CODE**

Click NC program → Enter the following data in the program number page

Program file name	:	EX11
Program number	:	11
Output directory	:	My document\2k6230 (Folder

name)

Enter **ok**. To read program, click edit and read the program.

**PROGRAM:**

%  
O11  
T0 G40 G50 S50  
T0303  
G97 M4 S2274  
G0 X70. Z2. M8  
X64.  
G1 X56. F0.2  
Z-110.  
X60.  
G0 X64.  
Z2.  
G1 X52. F0.2  
Z-110.  
X56.  
G0 Z2.  
G1 X48. F0.2  
Z-110.  
X52.  
G0 Z2.  
G1 X44. F0.2  
Z-110.  
X48.  
G0 Z2.  
G1 X40. F0.2  
Z-57.311  
X42.139 Z-60.228  
G3 X42.198 Z-60.431 R0.5  
G1 X41.04 Z-69.9  
X41.2  
G3 X42.2 Z-70.4 R0.5  
G1 Z-110.  
X44.  
G0 Z2.  
G1 X36. F0.2  
Z-51.857  
X40. Z-57.311  
G0 Z2.  
G1 X32. F0.2  
Z-46.402  
X36. Z-51.857  
G0 Z2.  
G1 X28. F0.2  
Z-40.948  
X32. Z-46.402  
G0 Z2.  
G1 X24. F0.2  
Z-35.493  
X28. Z-40.948  
G0 Z2.  
X20.  
G1 Z-5.1 F0.2  
G3 X20.2 Z-5.4 R0.5  
G1 Z-25.4  
G3 Z-25.43 R0.5  
G1 X20. Z-27.05

Z-30.1  
G3 X20.139 Z-30.228 R0.5  
G1 X24. Z-35.493  
G0 Z-27.05  
F0.14  
G1 X20. F0.14  
X19.649 Z-29.919  
F0.2  
Z-29.953  
G3 X20. Z-30.1 R0.5  
G0 X28.  
Z2.  
G1 X16. F0.2  
Z-3.093  
X19.907 Z-5.046  
G3 X20. Z-5.1 R0.5  
G0 Z2.  
G1 X12. F0.2  
Z-1.093  
X16. Z-3.093  
G0 Z2.  
G1 X10.049 F0.2  
Z-0.117  
X19.907 Z-5.046  
G3 X20.2 Z-5.4 R0.5  
G1 Z-25.4  
G3 Z-25.43 R0.5  
G1 X19.645 Z-29.952  
G3 X20.139 Z-30.228 R0.5  
G1 X42.139 Z-60.228  
G3 X42.198 Z-60.431 R0.5  
G1 X41.04 Z-69.9  
X41.2  
G3 X42.2 Z-70.4 R0.5  
G1 Z-110.  
X60.  
G0 X64.  
G42 Z2.  
G1 X10. F0.2  
S2728  
Z0. F0.16  
X20. Z-5.  
Z-25.  
X19.388 Z-30.  
X20.  
X42. Z-60.  
X40.777 Z-70.  
X42.  
Z-110.  
X64.  
G0 G40 X70.  
X200. Z100. M9  
T0 M01  
T0404  
G97 M4 S245  
G0 X70. Z-63.1 M8

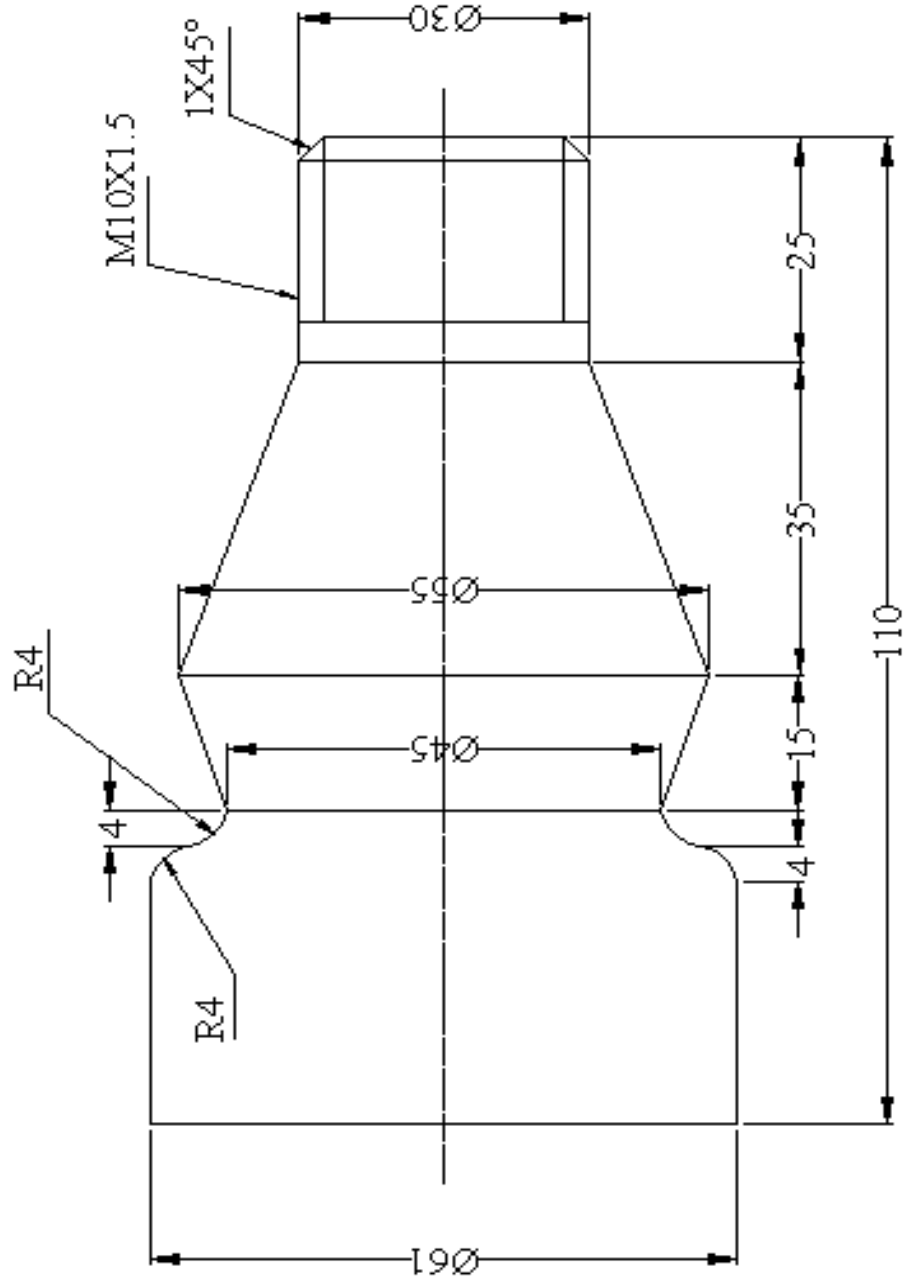
X46.  
Z-65.5  
F0.02  
G1 X20.2 F0.02  
F0.04  
G0 X46.  
Z-67.9  
G1 X20.2 F0.04  
G0 X46.  
Z-63.1  
G1 X42. F0.04  
X20.2  
Z-66.5  
G0 X46.  
Z-69.9  
G1 X42. F0.04  
X40.394  
X20.2  
Z-66.5  
G0 X46.  
S295  
Z-63.  
G1 X20. F0.032  
Z-66.5  
G0 X46.  
Z-70.  
G1 X20. F0.032  
Z-66.5  
G0 X46.  
X70.

X200. Z100. M9  
T0 M01  
T0505  
G97 M4 S505  
G0 X70. Z-27.1 M8  
X24.  
F0.02  
G1 X10.2 F0.02  
F0.04  
G0 X24.  
Z-28.7  
G1 X10.2 F0.04  
G0 X24.  
Z-29.9  
G1 X18.806 F0.04  
G0 X24.  
S611  
Z-27.  
G1 X10. F0.032  
Z-28.5  
G0 X24.  
Z-30.  
G1 X10. F0.032  
Z-28.5  
G0 X24.  
X70.  
X200. Z100. M9  
T0 M5  
M30  
%

**RESULT:**

Thus the NC code was generated for the component using capsturn.

## 12. THREAD CUTTING



BLANK :  $\varnothing 70 \times 120$   
MATERIAL : ALUMINIUM

ALL DIMENSIONS ARE IN mm

EX. NO. : 12  
DATE :

## THREAD CUTTING

### **AIM:**

To generate NC code for a given component using capsturn.

### **PROCEDURE:**

1.New Program → 2.Work Setup → 3.Geometry → 4.Machining → 5.Machine Selection → 6.Toolpath Generation → 7.NC Code Generation

#### **1. FILE- NEW**

#### **2. WORK SETUP**

##### **a) SETUP DATA-1 (Compulsory)**

Units	:	mm
W/p material	:	Al
Holding	:	External
Part load + un - load time	:	30 sec
Limiting spindle speed	:	3000 rpm
Jaw position absolute	:	-115 mm
Clearance z	:	5 mm
Clearance ID	:	0 mm
Clearance OD	:	80 mm

##### **b) SETUP DATA -2 (Compulsory)**

Tool turret position	:	Rear turret X = 200; Z = 100
Program mode	:	Diameter
Default turret position	:	Rear turret
Default tool mounting	:	Face up
Jaw size	:	Length = 20; Height = 20

##### **c) DOCUMENTATION (Optional)**

Part number	:	
Part name	:	
Set up number	:	
Fixture	:	
Programmer	:	
Remark	:	

Save the file as EX12.

### 3. GEOMETRY

#### Select the Geometry mode

Click geometry mode → Draw a part → Define a part → Define a blank.

#### A. To draw a part

Click draw → Choose multiline → Enter the values of X, Z in the input box

Enter the following data in the input box:

Specify the first point	:	0,0
Specify the next point	:	28,0
Specify the next point	:	30,-1
Specify the next point	:	30,-25
Specify the next point	:	55,-60
Specify the next point	:	45,-75 & Give done to complete multiline.

#### To draw CW Arc

Click draw → Arc → Start, end, radius → Give start point (53,-79) → Give end point (45,-75) → Enter radius value (4mm) → Enter done.

#### To draw CCW Arc

Click draw → Arc → Start, end, radius → Give start point (53,-79) → Give end point (61,-83) → Enter the radius value (4mm) → Enter done.

#### To draw Line

Click draw → Line → Point – point

Specify the start point	:	61,-83
Specify the end point	:	61,-110 → Enter done & Select zoom all.

#### B. To define the part

Click draw → Define part → Select the object by using mouse → Select YES

#### C. To define the blank

Click draw → Define blank → Enter the values for blank in the standard blank box (Length = 120; Diameter = 70; Start Z = 0) → Enter OK.

### 4. MACHINING

#### Select the Machining mode

Machining operations involved:

A. Turning the job using PTGNL 1616 H11 R0.4 tool

B. Thread cutting using 16\*16, 60deg, depth3.0 tool.

## A. Turning:

Click machining → Turning → Contour turn → New tool

Enter the following data in the select tool page:

Tool library	:	mm
Selected tool	:	PTGNL 1616 H11 R0.4
Selected tool material	:	Coated carbide
Tool number	:	1
Tool offset number	:	1

Enter **ok** to select the tool.

Select the first point on the part by using the mouse → Select the second point on the part by using the mouse.

Enter the following data in the contour turn page:

Operation name	:	Contour turn
Process	:	Roughing & Finishing
Depth of cut	:	1 mm
Finish allowance	:	X = 0.1, Z = 0.1

Leave the other data as it is & Enter **ok**.

## B. Thread Cutting

Click machining → Turning → Thread → New tool

Enter the following data in the select tool page:

Tool library	:	General mm
Selected tool	:	16*16, 60deg, depth3.0
Selected tool material	:	Coated carbide
Tool number	:	2
Tool offset number	:	2

Enter **ok** → Select a line for threading.

Enter the following data in the threading page:

Operation name	:	Threading	
Safe approach distance	:	2 mm	
Thread type	:	} → Select from standard → Metric 60°	
Thread pitch	:		} → External → Choose pitch=1.5 → Finish
Thread depth	:		
No. of cutting passes	:		
Number of starts	:	1	
Hand of thread	:	Left	
No. of ideal pass	:	1	
Entry distance	:	0	
Exit distance	:	-5 → Enter <b>ok</b>	

## 5. MACHINE SELECTION

Click machine → Choose the FANUC OiT m/c type from machine list → Click select.

## **6. TOOLPATH GENERATION**

Click toolpath → Click solid mode → Click start to see toolpath simulation.

## **7. TO GENERATE NC CODE**

Click NC program → Enter the data in the program number page

Program file name	:	EX12
Program number	:	12
Output directory	:	My document\2k6230 (Folder name)

Enter **ok**. To read program, click edit and read the program.

### **PROGRAM:**

%	X54.
O1234	G1 Z-58.804 F0.2
T0 G40 G50 S50	X55.128 Z-60.214
T0101	G3 X55.149 Z-60.558 R0.5
G97 M4 S1624	G1 X54. Z-62.281
G0 X80. Z2. M8	Z-78.991
X74.	G3 X56. Z-79.321 R4.5
G1 X68. F0.2	G0 Z-62.281
Z-109.995	F0.14
X69.207 Z-110.	G1 X54. F0.14
X70.	X52. Z-65.281
G0 X74.	F0.2
Z2.	Z-78.899
G1 X66. F0.2	G2 X52.047 R3.5
Z-109.986	G1 X52.279 Z-78.9
X68. Z-109.995	G3 X54. Z-78.991 R4.5
G0 Z2.	G0 Z-65.281
G1 X64. F0.2	F0.14
Z-109.977	G1 X52. F0.14
X66. Z-109.986	X50. Z-68.281
G0 Z2.	F0.2
G1 X62. F0.2	Z-78.723
Z-109.969	G2 X52. Z-78.899 R3.5
X64. Z-109.977	G0 Z-68.281
G0 Z2.	F0.14
G1 X60. F0.2	G1 X50. F0.14
Z-81.155	X48. Z-71.281
G3 X61.2 Z-83.4 R4.5	F0.2
G1 Z-109.965	Z-78.2
X62. Z-109.969	G2 X50. Z-78.723 R3.5
G0 Z2.	G0 Z-71.281
G1 X58. F0.2	F0.14
Z-79.959	G1 X48. F0.14
G3 X60. Z-81.155 R4.5	X46. Z-74.281
G0 Z2.	F0.2
G1 X56. F0.2	Z-77.025
Z-79.321	G2 X48. Z-78.2 R3.5
G3 X58. Z-79.959 R4.5	G0 Z-74.281
G0 Z2.	F0.14



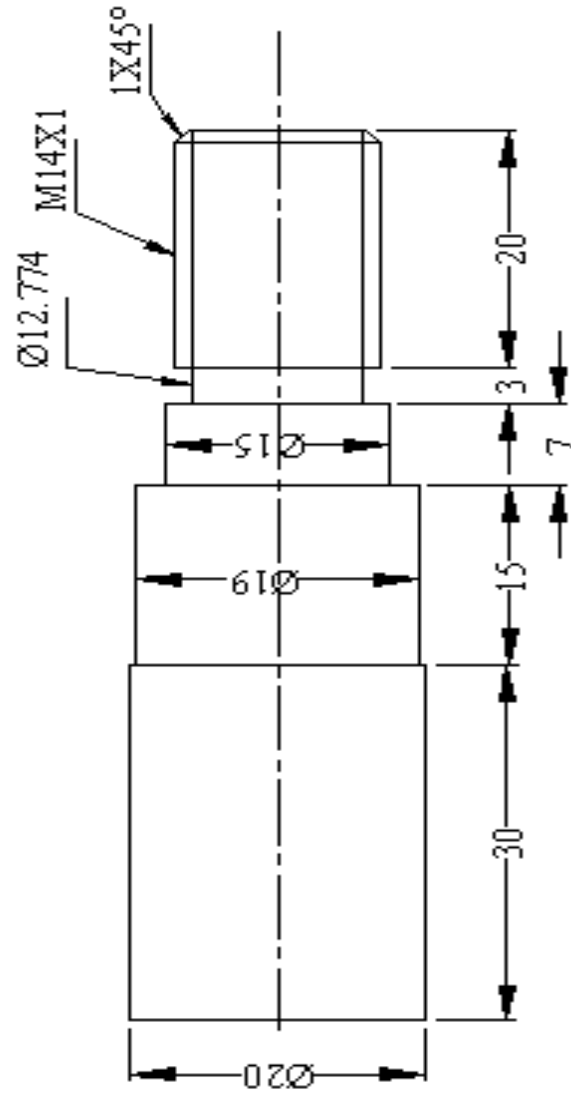
G1 X46. F0.14  
X45.206 Z-75.472  
F0.2  
Z-75.542  
G2 X46. Z-77.025 R3.5  
G0 X60.  
Z2.  
G1 X52. F0.2  
Z-56.304  
X54. Z-58.804  
G0 Z2.  
G1 X50. F0.2  
Z-53.804  
X52. Z-56.304  
G0 Z2.  
G1 X48. F0.2  
Z-51.304  
X50. Z-53.804  
G0 Z2.  
G1 X46. F0.2  
Z-48.804  
X48. Z-51.304  
G0 Z2.  
G1 X44. F0.2  
Z-46.304  
X46. Z-48.804  
G0 Z2.  
G1 X42. F0.2  
Z-43.804  
X44. Z-46.304  
G0 Z2.  
G1 X40. F0.2  
Z-41.304  
X42. Z-43.804  
G0 Z2.  
G1 X38. F0.2  
Z-38.804  
X40. Z-41.304  
G0 Z2.  
G1 X36. F0.2  
Z-36.304  
X38. Z-38.804  
G0 Z2.  
G1 X34. F0.2  
Z-32.938  
X35.154 Z-35.246  
X36. Z-36.304

G0 Z2.  
G1 X32. F0.2  
Z-28.938  
X34. Z-32.938  
G0 Z2.  
G1 X30. F0.2  
Z-1.1  
G3 X30.2 Z-1.4 R0.5  
G1 Z-25.338  
X32. Z-28.938  
G0 Z2.  
G1 X28.049 F0.2  
Z-0.117  
X29.907 Z-1.046  
G3 X30. Z-1.1 R0.5  
G0 X31. Z-0.6  
X74.  
G42 Z2.  
G1 X28. F0.2  
S1949  
Z0. F0.16  
X30. Z-1.  
Z-25.  
X35. Z-35.  
X55. Z-60.  
X45. Z-75.  
G2 X52.831 Z-78.999 R4.  
G1 X53.07 Z-79.  
G3 X61. Z-83. R4.  
G1 Z-110.  
X74.  
G0 G40 X80.  
X200. Z100. M9  
T0 M01  
T0202  
G97 M4 S2653  
G0 X80. Z-1. M8  
X34.  
G76 P060060 Q0 R0  
G76 X28.12 Z-20. R0. P940 Q383  
F1.5  
X80.  
X200. Z100. M9  
T0 M5  
M30  
%

**RESULT:**

Thus the NC code was generated for the component using capsturn.

### 13. GROOVING AND THREAD CUTTING



BLANK:  $\varnothing 30 \times 100$

MATERIAL: ALUMINIUM ALL DIMENSIONS ARE IN mm

EX. NO. : 13

DATE :

## GROOVING AND THREAD CUTTING

### **AIM:**

To generate NC code for a given component using capsturn.

### **PROCEDURE:**

1.New Program → 2.Work Setup → 3.Geometry → 4.Machining → 5.Machine Selection  
→ 6.Toolpath Generation → 7.NC Code Generation

#### **1. FILE- NEW**

#### **2. WORK SETUP**

##### **a) SETUP DATA-1 (Compulsory)**

Units	:	mm
W/p material	:	Al
Holding	:	External
Part load + un - load time	:	30 sec
Limiting spindle speed	:	3000 rpm
Jaw position absolute	:	-90
Clearance z	:	5
Clearance ID	:	0
Clearance OD	:	40

##### **b) SETUP DATA -2 (Compulsory)**

Tool turret position	:	Rear turret X = 200; Z = 100
Program mode	:	Diameter
Default turret position	:	Rear turret
Default tool mounting	:	Face up
Jaw size	:	Length = 20; Height = 20

##### **c) DOCUMENTATION (Optional)**

Part number	:	
Part name	:	
Set up number	:	
Fixture	:	
Programmer	:	
Remark	:	

Save the file as EX13.

### 3. GEOMETRY

#### Select the Geometry mode

Click geometry mode → Draw a part → Define a part → Define a blank.

#### A. To draw a part

Click draw → Choose multiline → Enter the values of X, Z in the input box.

Enter the following data in the input box:

Specify the first point	:	0, 0
Specify the next point	:	12,0
Specify the next point	:	14,-1
Specify the next point	:	14,-20
Specify the next point	:	12.774,-20
Specify the next point	:	12.774,-23
Specify the next point	:	15,-23
Specify the next point	:	15,-30
Specify the next point	:	19,-30
Specify the next point	:	19,-45
Specify the next point	:	20,-45
Specify the next point	:	20,-75

Give done to complete the line & Select zoom all.

#### B. To define the part

Click draw → Define part → Select the object by using mouse → Select YES.

#### C. To define the blank

Click Draw → Define blank → Enter the values for blank in the standard blank box (Length = 100; Diameter = 30; Start Z = 0) → Enter **ok**.

### 4. MACHINING

#### Select the Machining mode

Machining operations involved:

- Turning the job using PTGNL 1616 H11 R0.4 tool
- Grooving the job using 12\*12, 2.00W, 0.20R, 08DEPTH, LH tool.
- Thread cutting using 16\*16, 60DEG, DEPTH3.0 tool.

#### A. Turning;

Click machining → Turning → Contour turn → New tool

Enter the following data in the select tool page:

Tool library	:	mm
Selected tool	:	PTGNL 1616 H11 R.4
Selected tool material	:	Coated carbide

Tool number : 1  
Tool offset number : 1

Enter **ok** to select the tool.

Select the first point on the part by using the mouse → Select the second point on the part by using the mouse.

Enter the following data in the contour turn page:

Operation name : Contour turn  
Process : Roughing & Finishing  
Depth of cut : 1 mm  
Finish allowance : X = 0.1, Z = 0.1

Leave the other data as it is & Enter **ok**.

### **B. Grooving:**

Click machining → Turning → Groove → Radial → New tool

Enter the following data in the select tool page:

Tool library : General mm  
Selected tool : 32\*32,3.00W, 0.10R, 20DEPTH,  
LH  
Selected tool material : Coated carbide  
Tool number : 2  
Tool offset number : 2

Select the groove at the first end → Select the other end of the groove

Enter the following data in the external groove page:

Operation name : External groove  
Process : Roughing & Finishing  
Finish allowance : 0.1  
Start location : Right  
Cutting method : Plunge cut → Enter **ok**

### **C. Thread Cutting**

Click machining → Turning → Thread → New tool

Enter the following data in the select tool page:

Tool library : General mm  
Selected tool : 16\*16, 60deg, depth3.0  
Selected tool material : Coated carbide  
Tool number : 3  
Tool offset number : 3

Enter **ok** → Select a line for threading.

Enter the following data in the external groove page:

Operation name : Threading  
Safe approach distance : 2 mm  
Thread type : }  
Thread pitch : } → Select from standard → Metric  
60° : }  
Thread depth : } External → Choose pitch=1 →  
Finish : }  
No. of cutting passes :  
Number of starts : 1

Hand of thread	:	Right
No. of ideal pass	:	1
Entry distance	:	0
Exit distance	:	0 → Enter <b>ok</b>

## **5. MACHINE SELECTION**

Click machine → Choose the **Fanuc OiT** m/c type from machine list → Click select.

## **6. TOOLPATH GENERATION**

Click toolpath → Click solid mode → Click start to see toolpath simulation.

## **7. TO GENERATE NC CODE:**

Click NC program → Enter the following data in the program number page

Program file name	:	EX13
Program number	:	13
Output directory	:	My document\2k6230 (Folder name)

Enter **ok**. To read program, click edit and read the program.

### **PROGRAM:**

%	Z-45.1
O13	G3 X20.2 Z-45.4 R0.5
T0 G40 G50 S50	G1 Z-74.961
T0101	X22. Z-74.969
G97 M4 S2107	G0 Z2.
G0 X40. Z2. M8	G1 X18. F0.3
X34.	Z-29.899
G1 X28. F0.3	X18.205
Z-74.995	G3 X19.2 Z-30.4 R0.5
X29.207 Z-75.	G1 Z-44.9
X30.	X19.205
G0 X34.	G3 X20. Z-45.1 R0.5
Z2.	G0 Z2.
G1 X26. F0.3	G1 X16. F0.3
Z-74.986	Z-29.89
X28. Z-74.995	X18. Z-29.899
G0 Z2.	G0 Z2.
G1 X24. F0.3	X14.
Z-74.977	G1 Z-1.1 F0.3
X26. Z-74.986	G3 X14.2 Z-1.4 R0.5
G0 Z2.	G1 Z-20.4
G1 X22. F0.3	G3 X14.087 Z-20.631 R0.5
Z-74.969	G1 X14. Z-20.714
X24. Z-74.977	Z-22.899
G0 Z2.	X14.205
G1 X20. F0.3	G3 X15.2 Z-23.4 R0.5

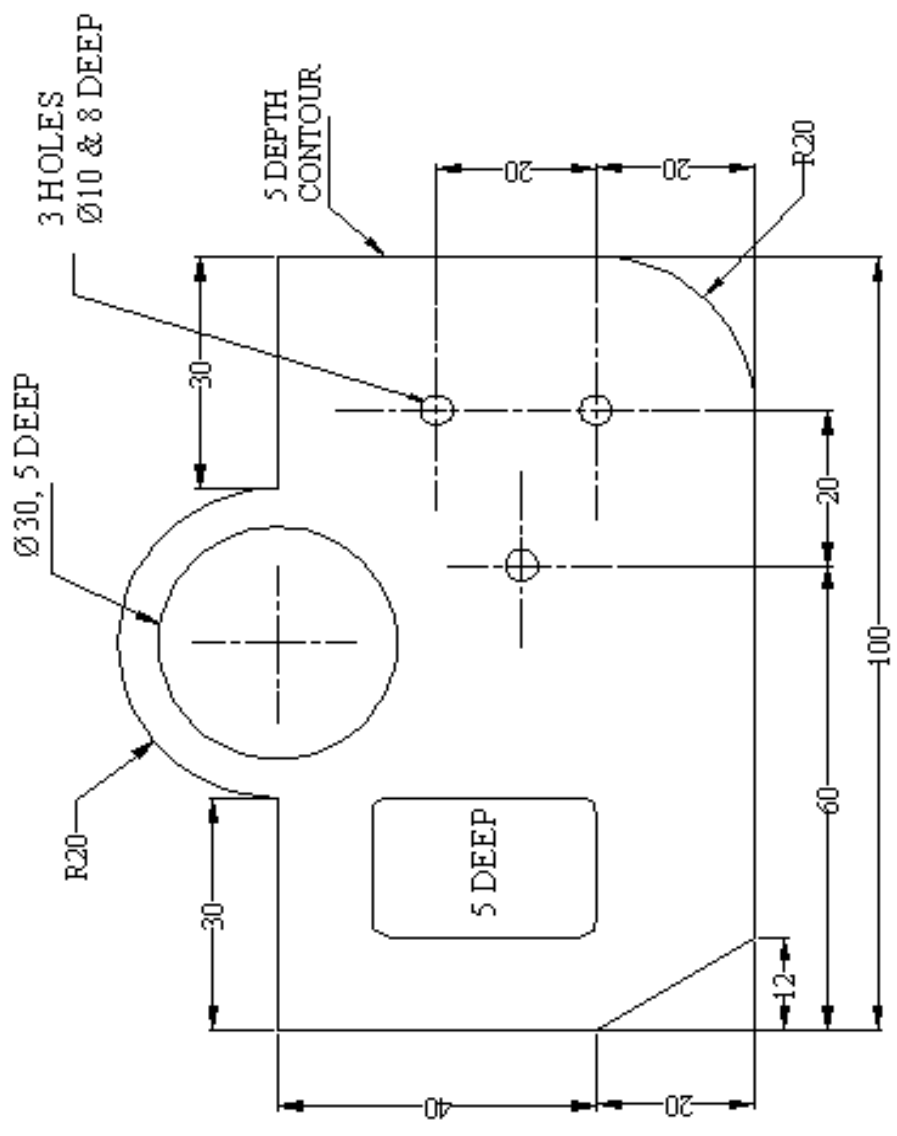
G1 Z-29.887  
X16. Z-29.89  
G0 X20.  
Z2.  
G1 X12.049 F0.3  
Z-0.117  
X13.907 Z-1.046  
G3 X14. Z-1.1 R0.5  
G0 X15. Z-0.6  
X34.  
G42 Z2.  
G1 X12. F0.3  
S2528  
Z0. F0.24  
X14. Z-1.  
Z-20.  
X12.774 Z-21.178  
Z-22.99  
X15. Z-23.  
Z-29.983  
X19. Z-30.  
Z-44.996  
X20. Z-45.  
Z-75.  
X34.  
G0 G40 X40.  
X200. Z100. M9  
T0 M01  
T0202  
G97 M4 S3000  
G0 X40. Z-22.1 M8  
X19.  
Z-22.5

F0.125  
G1 X12.974 F0.125  
F0.25  
G0 X19.  
Z-22.9  
G1 X12.974 F0.25  
G0 X19.  
Z-22.1  
G1 X12.974 F0.25  
G0 X19.  
Z-22.  
G1 X12.774 F0.25  
Z-22.5  
G0 X19.  
Z-23.  
G1 X12.774 F0.25  
Z-22.5  
G0 X19.  
X40.  
X200. Z100. M9  
T0 M01  
T0303  
G97 M4 S2956  
G0 X40. Z-20. M8  
X18.  
G76 P050060 Q0 R0  
G76 X12.66 Z-1. R0. P670 Q299  
F1.  
X40.  
X200. Z100. M9  
T0 M5  
M30  
%

**RESULT:**

Thus the NC code was generated for the component using capsturn.

**14. RECTANGULAR & CIRCULAR POCKETING**



**BLANK: 120 X 100 X 23**

**MATERIAL: CI**

**ALL DIMENSIONS ARE IN mm**



EX. NO. : 14

DATE :

## **RECTANGULAR AND CIRCULAR POCKETING**

### **AIM:**

To generate NC code for a given component using capsmill.

### **PROCEDURE:**

1.New Program → 2.Work Setup → 3.Geometry → 4.Machining → 5.Machine Selection →  
6.Toolpath Generation →7.NC Code Generation

#### **1. FILE- NEW**

#### **2. WORK SETUP**

##### **a) SETUP DATA (Compulsory)**

Units	:	mm
Work piece materials	:	cast iron, alloy, medium hardness
Part load + un-load time	:	30 sec
Clearance Z	:	100
Tool change position	:	X=100; Y=100; Z=100.

##### **b) TABLE SETUP (Compulsory)**

Select the first table setup type.

##### **c) DOCUMENTATION (Optional)**

Part number	:	
Part name	:	
Set up number	:	
Fixture	:	
Programmer	:	
Remark	:	

Save the file as EX14.

#### **3. GEOMETRY**

##### **Select the Geometry mode**

Click geometry mode → Draw a part → Define contour → Define a blank.

##### **A. To draw a part**

Click draw → Choose multiline → Enter the values of X, Y in the input box.

Enter the following data in the input box:

Specify the first point	:	0, 0
Specify the next point	:	100, 0
Specify the next point	:	100, 60
Specify the next point	:	0, 60
Specify the next point	:	0, 0

Give done to complete the line & Select zoom all.

### **To draw the chamfer**

Click draw → Select chamfer → Select chamfer type → Distance 1 & Distance 2. Enter the chamfer distances (12&20) → Select the corresponding lines by using mouse → Select done.

### **To draw the radius**

Click draw → Select the radius → Enter the value of radius (20mm) in the dialog box → Select the first and second object by using mouse → Select done.

### **To draw the arc of radius 20mm**

Click draw → Arc → Start, end, radius → Specify the start point (70, 60) → Specify the end point (30,60) → Enter the radius value (20mm) → Select done.

### **To trim the line**

Click Edit → Choose trim → Select the boundary objects by using mouse (top line and 20 mm radius arc) → Select done → Select the in-between line to trim → Select done.

### **To draw the centre point for $\phi$ 10 holes**

Click draw → Point pattern → Points - random → Specify the three random points: [(80,20),(60,30),(80,40)] → Select done.

### **To draw the centre point for $\phi$ 30 circular pocket**

Click draw → Point pattern → Points - random → Specify the random points (50, 60) → Select done.

### **To draw a rectangle for the rectangular pocket**

Click draw → Shapes → Rectangle (Corner - length - width) → Enter the following data in the dialog box appeared (Length = 20, Breadth = 30, Rotation angle = 0°, Radius = 2) → Select **ok** → Enter lower left corner of the rectangle (12, 20) & Select done.

### **B. To define the contour**

Select draw → Define contour → Auto Contour → Select the contour by mouse → Give YES → Select done.

### C. To define the blank

Click Draw → Define blank → Enter the following data in the dialog box:

<u>Minimum</u>	<u>Maximum</u>		
X = -10	X = 110	Blank thickness	= 23
Y = -10	Y = 90	Z coordinate at the bottom	= -20

Select **ok** & Select zoom all.

## 4. MACHINING

### Select the Machining mode

Machining operations involved:

- A. Face milling
- B. Contour milling
- C. Drilling (3 holes, dia. 10 mm, 8 mm deep)
- D. Circular pocketing (dia. 30 mm)
- E. Rectangular pocket milling

### A. To perform face milling

i) Select Machining → Milling → Face mill → Click New tool

To select the tool, fill the dialog box that appeared

Tool library	:	General mm
Tool type	:	Face mill - round insert
Select tool	:	40 mm dia. round insert - Face

mill

Tool material	:	Carbide → Enter <b>ok</b> .
---------------	---	-----------------------------

ii) Select the 'blank contour' by clicking a point on the contour

iii) Enter the following data in the Face mill dialog box that appears:

Operation name	:	Face mill
Process	:	Roughing & Finishing
Area to be cleared	:	<b>Surface</b>
Work surface Z	:	0
Material thickness	:	3
Return to Z	:	200
Depth of cut	:	1 on cutting parameters tab menu

&

Give **ok**.

iv) Specify the machining start point by clicking on any one of the circle.

### B. To perform contour milling

i) Select Machining → Milling → Face mill → Click New tool

To select the tool, fill the dialog box that appeared

Tool library	:	General mm
Tool type	:	End mill finish - 2 flute
Select tool	:	2 mm dia. End mill finish - 2 flute

Tool material	:	Carbide → Enter <b>ok</b> .
---------------	---	-----------------------------

ii) Select the 'inner contour' by clicking a point on the contour

- iii) Enter the following data in the Face mill dialog box that appears:
- |                    |   |                                  |
|--------------------|---|----------------------------------|
| Operation name     | : | Face mill                        |
| Process            | : | Roughing & Finishing             |
| Area to be cleared | : | <b>Contour</b>                   |
| Work surface Z     | : | -5                               |
| Material thickness | : | 5                                |
| Return to Z        | : | 200                              |
| Depth of cut       | : | 1 on cutting parameters tab menu |
- &
- Give **ok**.
- iv) Enter the machining start point by clicking on any one of the corner & Select YES & Select the same point as the tool entry and exit point.

### **C. To perform drilling operation (3 holes, dia. 10 mm, 8 mm deep)**

- i) Select Machining → Hole → Drilling → Click New tool  
To select the tool, fill the dialog box that appeared
- |               |   |                             |
|---------------|---|-----------------------------|
| Tool library  | : | General mm                  |
| Tool type     | : | Drill - Twist drill         |
| Select tool   | : | 10 mm dia. Twist drill      |
| Tool material | : | Carbide → Enter <b>ok</b> . |
- ii) Select the three points for drilling, Select done & Give 'NO'
- iii) Enter the following data in the drill dialog box:
- |                |   |                       |
|----------------|---|-----------------------|
| Operation name | : | Drill                 |
| Work surface   | : | 0                     |
| Hole depth     | : | 8                     |
| Return to z    | : | 200 → Enter <b>ok</b> |

### **D. To perform circular pocketing operation (diameter 30mm)**

- i) Select Machining → Hole → Drilling → Click New tool  
To select the tool, fill the dialog box that appeared
- |                    |   |   |
|--------------------|---|---|
| Tool library       | : | General mm                                    |
| Tool type          | : | Hole mill - Hole mill                         |
| Select tool        | : | 32 mm dia. Hole mill                          |
| Tool material      | : | Carbide                                       |
| Edit tool geometry | : | Change dia. from 32 to 30 → Enter <b>ok</b> . |
- ii) Select the point for hole milling, Select done & Give 'NO'
- iii) Enter the following data in the drill dialog box:
- |                |   |                       |
|----------------|---|-----------------------|
| Operation name | : | Drill                 |
| Work surface   | : | 0                     |
| Hole depth     | : | 5                     |
| Return to z    | : | 200 → Enter <b>ok</b> |

### **E. To perform the rectangular pocket milling**

- i) Select Machining → Milling → Pocket → Click New tool  
To select the tool, fill the dialog box that appeared
- |              |   |            |
|--------------|---|------------|
| Tool library | : | General mm |
|--------------|---|------------|

Tool type : End mill – End mill finish - 2 flute  
 Select tool : 5 mm dia. End mill finish - 2 flute  
 Tool material : Carbide → Enter **ok**.

ii) Select the contour for pocket milling

iii) Enter the following data in the Pocketing milling dialog box:

Work surface : 0  
 Pocket depth : 5 mm  
 Side allowance : 0  
 Bottom allowance : 0  
 Return to Z : 200  
 Cutting method : Zigzag  
 Entry into the material : Plunge  
 Depth of cut : 1 mm (in cutting parameters) → Enter  
**ok**

## 5. MACHINE SELECTION

Click Machine → Choose the **Fanuc OiM** m/c type from machine list → Click select.

## 6. TOOLPATH GENERATION

Click toolpath → Click solid mode → Click start to see toolpath simulation.

## 7. TO GENERATE NC CODE

Click NC program → Enter the following data in the program number page

Program file name : EX14  
 Program number : 14  
 Output directory : My documents \ 2k6230 (Folder name)

Enter **ok**. To read program, click edit and read the program.

### **PROGRAM:**

%	M98 P00010055
O14	G90 G00 X-10. Y78.6
G17 G21	Z4.
G0 G49 G40 G80 G90	X135. Y1.4
(40.00 MM. DIA. ROUND INSERT -	G01 Z0. F457.361
FACE MILL)	X-10. F653.373
M3 S1088 G43 H1 G54 T1	G00
G90 G00 X135. Y1.4 M54	Z4.
Z100.	X135. Y31.8
Z6.	G01 Z0. F457.361
G01 Z2. F457.361	X-10. F653.373
M98 P00010055	G00
G90 G00 X-10. Y78.6	Z4.
Z5.	X135. Y62.2
X135. Y1.4	G01 Z0. F457.361
G01 Z1. F457.361	X-10. F653.373

G00  
Z4.  
X135. Y78.6  
G01 Z0. F457.361  
X-10. F653.373  
G00  
Z3.  
X135. Y1.4  
S1256  
G01 Z0. F316.635  
X-35. F452.335  
G00  
Z3.  
X135. Y31.8  
G01 Z0. F316.635  
X-35. F452.335  
G00  
Z3.  
X135. Y62.2  
G01 Z0. F316.635  
X-35. F452.335  
G00  
Z3.  
X135. Y78.6  
G01 Z0. F316.635  
X-35. F452.335  
G00  
Z200.  
M5  
(2.00 MM. DIA. END MILL-FINISH-  
2 FLUTE)  
M3 S6000 G43 H2 G54 T2  
X100.055 Y59.989 M54  
Z100.  
Z3.  
G01 Z-1. F557.042  
X100. Y60.  
M98 P00010056  
G90 X100.055 Y59.989  
G00  
Z3.  
G01 Z-2. F557.042  
X100. Y60.  
M98 P00010056  
G90 X100.055 Y59.989 F557.042  
G00  
Z3.  
G01 Z-3.  
X100. Y60.  
M98 P00010056

G90 X100.055 Y59.989 F557.042  
G00  
Z3.  
G01 Z-4.  
X100. Y60.  
M98 P00010056  
G90 X100.055 Y59.989 F557.042  
G00  
Z3.  
G01 Z-5.  
X100. Y60.  
M98 P00010056  
G90 X100.055 Y59.989 F557.042  
G00  
Z3.  
S6000  
G01 Z-5. F267.38  
X100. Y60. F381.972  
X70.  
G03 X30. I-20. J0.  
G01 X0.  
Y20.  
X12. Y0.  
X80.  
G03 X100. Y20. I0. J20.  
G01 Y60.  
X100.055 Y59.989  
G00  
Z200.  
M5  
(10.00 MM. DIA. TWIST DRILL)  
M3 S3183 G43 H3 G54 T3  
X80. Y20. M54  
Z100.  
Z200.  
G98 G81 Z-11.004 R3. F477.465  
Y40.  
X60. Y30.  
G80  
M5  
(32.00 MM. DIA. HOLE MILL)  
M3 S583 G43 H4 G54 T4  
X50. Y60. M54  
Z100.  
Z200.  
G98 G81 Z-5. R3. F116.714  
G80  
M5  
(5.00 MM. DIA. END MILL-FINISH-  
2 FLUTE)

M3 S3183 G43 H5 G54 T5  
X29.5 Y43.5 M54  
Z100.  
Z3.  
G01 Z-1. F222.817  
X14.5 F318.31  
Y39.5  
X29.5  
Y35.5  
X14.5  
Y31.5  
X29.5  
Y27.5  
X14.5  
Y23.5  
X29.5  
G00  
Z3.  
X14.5 Y22.5  
G01 Z-1. F222.817  
X29.5 F318.31  
Y47.5  
X14.5  
Y22.5  
G00  
Z3.  
X29.5 Y43.5  
Z-0.5  
G01 Z-2. F222.817  
X14.5 F318.31  
Y39.5  
X29.5  
Y35.5  
X14.5  
Y31.5  
X29.5  
Y27.5  
X14.5  
Y23.5  
X29.5  
G00  
Z3.  
X14.5 Y22.5  
G01 Z-2. F222.817  
X29.5 F318.31  
Y47.5  
X14.5  
Y22.5  
G00  
Z3.

X29.5 Y43.5  
Z-1.5  
G01 Z-3. F222.817  
X14.5 F318.31  
Y39.5  
X29.5  
Y35.5  
X14.5  
Y31.5  
X29.5  
Y27.5  
X14.5  
Y23.5  
X29.5  
G00  
Z3.  
X14.5 Y22.5  
G01 Z-3. F222.817  
X29.5 F318.31  
Y47.5  
X14.5  
Y22.5  
G00  
Z3.  
X29.5 Y43.5  
Z-2.5  
G01 Z-4. F222.817  
X14.5 F318.31  
Y39.5  
X29.5  
Y35.5  
X14.5  
Y31.5  
X29.5  
Y27.5  
X14.5  
Y23.5  
X29.5  
G00  
Z3.  
X14.5 Y22.5  
G01 Z-4. F222.817  
X29.5 F318.31  
Y47.5  
X14.5  
Y22.5  
G00  
Z3.  
X29.5 Y43.5  
Z-3.5

G01 Z-5. F222.817  
X14.5 F318.31  
Y39.5  
X29.5  
Y35.5  
X14.5  
Y31.5  
X29.5  
Y27.5  
X14.5 Y23.5  
X29.5  
G00 Z3.  
X14.5 Y22.5  
G01 Z-5. F222.817  
X29.5 F318.31  
Y47.5  
X14.5  
Y22.5  
G00  
Z200.  
M5  
G80 M56  
G59 X0 Y0 Z100  
M16  
M30  
O55 (SUB PROGRAM)  
G91 X-145. Y-0. F653.373

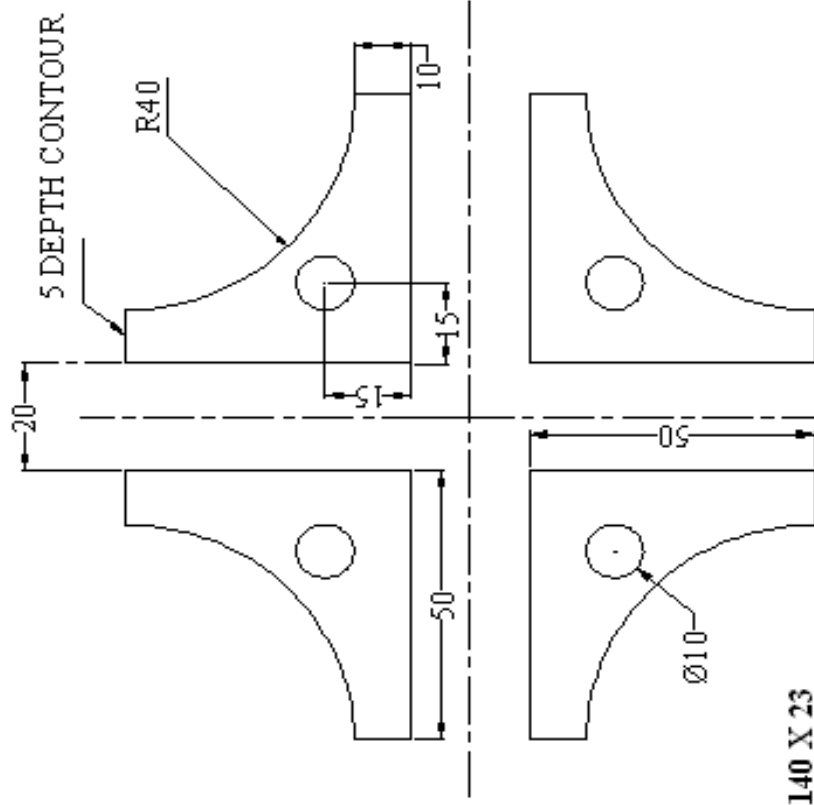
G00  
Z4.  
X145. Y30.4  
G01 Z-4. F457.361  
X-145. Y-0. F653.373  
G00  
Z4.  
X145. Y30.4  
G01 Z-4. F457.361  
X-145. Y-0. F653.373  
G00  
Z4.  
X145. Y16.4  
G01 Z-4. F457.361  
X-145. Y-0. F653.373  
M99  
O56 (SUB PROGRAM)  
G91 X-30. F795.775  
G03 X-40. Y-0. I-20. J0.  
G01 X-30. Y0.  
X-0. Y-40.  
X12. Y-20.  
X68.  
G03 X20. Y20. I0. J20.  
G01 Y40.  
M99  
%

**RESULT:**

Thus the NC code was generated for the component using capsmill.



## 15. MIRRORING



**BLANK: 140 X 140 X 23**

**MATERIAL: CI**

**ALL DIMENSIONS ARE IN mm**

EX. NO. : 15  
DATE :

## MIRRORING

### AIM:

To generate NC code for a given component using capsmill.

### PROCEDURE:

1.New Program → 2.Work Setup → 3.Geometry → 4.Machining → 5.Machine Selection →  
6.Toolpath Generation → 7.NC Code Generation

#### 1. FILE- NEW

#### 2. WORK SETUP

##### a) **SETUP DATA** (Compulsory)

Units	:	mm
Work piece materials	:	Cast iron, alloy, medium
hardness	:	
Part load + un-load time	:	30 sec
Clearance Z	:	100
Tool change position	:	X = 100; Y = 100; Z = 100

##### b) **TABLE SETUP** (Compulsory)

Select the first table setup type.

##### c) **DOCUMENTATION** (Optional)

Part number	:	
Part name	:	
Set up number	:	
Fixture	:	
Programmer	:	
Remark	:	

Save the file as EX15.

#### 3. GEOMETRY

##### Select the Geometry mode

Click geometry mode → Draw a part → Define contour → Define a blank.

##### A. To draw a part

Click Draw → Choose multiline → Enter the values of X, Y in the input box

Enter the following data in the input box:

Specify the first point	:	20, 60
Specify the next point	:	10, 60
Specify the next point	:	10, 10
Specify the next point	:	60, 10
Specify the next point	:	60, 20

Give done to complete the line.

### **To draw the arc of radius 40mm**

Click Draw → Arc → Start, end, radius → Give the start point (20, 60) → Give the end point (60, 20) → Enter the radius value (40mm) → Enter done.

### **To draw the centre point for the hole**

Click Draw → Point pattern → Points - random → Specify the random point (25, 25) → Enter done.

### **To mirror the object**

Click Edit → Mirror → Select the object through window → Specify the first point of the mirror line (using the mouse) → Specify the second point of the mirror line → Give YES to retain the original object → Give done.

(NOTE: Use the same procedure to mirror the remaining elements)

### **B. To define contour**

Select Draw → Contour → Window → Select the contour → Select done.

### **C. To define the blank**

Click Draw → Define blank → Enter the following data in the dialog box:

<u>Minimum</u>	<u>Maximum</u>		
X = -70	X = 70	Blank thickness	= 23
Y = -70	Y = 70	Z coordinate at the bottom	= -20

Select **ok** & Select zoom all.

## **4. MACHINING**

### **Select the machining mode**

Machining operations involved:

- A. Face milling
- B. Contour milling
- C. Drilling (4 holes, dia. 10)

### **A. To perform face milling**

- i) Select Machining → Milling → Face mill → Click New tool  
To select the tool, fill the dialog box that appeared

- |              |   |                                |
|--------------|---|--------------------------------|
| Tool library | : | General mm                     |
| Tool type    | : | Face mill - round insert       |
| Select tool  | : | 40 mm dia. round insert - Face |
- mill
- |               |   |                            |
|---------------|---|----------------------------|
| Tool material | : | Carbide → Enter <b>ok.</b> |
|---------------|---|----------------------------|
- ii) Select the 'blank contour' by clicking a point on the contour
- iii) Enter the following data in the Face mill dialog box that appears:
- |                    |   |                                  |
|--------------------|---|----------------------------------|
| Operation name     | : | Face mill                        |
| Process            | : | Roughing & Finishing             |
| Area to be cleared | : | <b>Surface</b>                   |
| Work surface Z     | : | 0                                |
| Material thickness | : | 3                                |
| Return to Z        | : | 200                              |
| Depth of cut       | : | 1 on cutting parameters tab menu |
- &
- Give **ok.**
- iv) Specify the machining start point by clicking on any one of the circle.

### **B. To perform contour milling**

- i) Select Machining → Milling → Face mill → Click New tool  
To select the tool, fill the dialog box that appeared
- |               |   |  |
|---------------|---|--|
| Tool library  | : | General mm                             |
| Tool type     | : | End mill finish - 2 flute              |
| Select tool   | : | 2 mm dia. End mill finish - 2<br>flute |
| Tool material | : | Carbide → Enter <b>ok.</b>             |
- ii) Select the 'inner contour' by clicking a point on the contour
- iii) Enter the following data in the Face mill dialog box that appears:
- |                    |   |                                  |
|--------------------|---|----------------------------------|
| Operation name     | : | Face mill                        |
| Process            | : | Roughing & Finishing             |
| Area to be cleared | : | <b>Contour</b>                   |
| Work surface Z     | : | -5                               |
| Material thickness | : | 5                                |
| Return to Z        | : | 200                              |
| Depth of cut       | : | 1 on cutting parameters tab menu |
- &
- Give **ok.**
- iv) Enter the machining start point by clicking on any one of the corner & Select YES & Enter the same point as the tool entry and exit point.

(NOTE: Follow the same procedure to mill the other three contours)

### **C. To perform drilling operation (4 holes dia.10 mm, 10 depth)**

- i) Select Machining → Hole → Drilling → Click New tool  
To select the tool, fill the dialog box that appeared
- |              |   |                     |
|--------------|---|---------------------|
| Tool library | : | General mm          |
| Tool type    | : | Drill - Twist drill |

- |               |   |                             |
|---------------|---|-----------------------------|
| Select tool   | : | 10 mm dia. Twist drill      |
| Tool material | : | Carbide → Enter <b>ok</b> . |
- ii) Select the four points for drilling, Select done & Give 'No'
- iii) Enter the following data in the drill dialog box:
- |                |   |                       |
|----------------|---|-----------------------|
| Operation name | : | Drill                 |
| Work surface   | : | 0                     |
| Hole depth     | : | 10                    |
| Return to z    | : | 200 → Enter <b>ok</b> |

## **5. MACHINE SELECTION**

Click Machine → Choose the **Fanuc OiM** m/c type from machine list → Click select.

## **6. TOOL PATH GENERATION**

Click toolpath → Click solid mode → Click start to see toolpath simulation.

## **7. TO GENERATE NC CODE**

Click NC program → Enter the following data in the program number page:

Program file name	:	EX15
Program number	:	15
Output directory	:	My document \ 2k6230 (Folder name)

Enter **ok**. To read program, click edit and read the program.

### **PROGRAM:**

%	G00
O15	Z4.
G17 G21	X95. Y-28.2
G0 G49 G40 G80 G90	G01 Z0. F457.361
(40.00 MM. DIA. ROUND INSERT -	X-70. F653.373
FACE MILL)	G00
M3 S1088 G43 H1 G54 T1	Z4.
G90 G00 X95. Y-58.6 M54	X95. Y2.2
Z100.	G01 Z0. F457.361
Z6.	X-70. F653.373
G01 Z2. F457.361	G00
M98 P00010055	Z4.
G90 G00 X-70. Y58.6	X95. Y32.6
Z5.	G01 Z0. F457.361
X95. Y-58.6	X-70. F653.373
G01 Z1. F457.361	G00
M98 P00010055	Z4.
G90 G00 X-70. Y58.6	X95. Y58.6
Z4.	G01 Z0. F457.361
X95. Y-58.6	X-70. F653.373
G01 Z0. F457.361	G00
X-70. F653.373	Z3.

X95. Y-58.6  
S1256  
G01 Z0. F316.635  
X-95. F452.335  
G00  
Z3.  
X95. Y-28.2  
G01 Z0. F316.635  
X-95. F452.335  
G00  
Z3.  
X95. Y2.2  
G01 Z0. F316.635  
X-95. F452.335  
G00  
Z3.  
X95. Y32.6  
G01 Z0. F316.635  
X-95. F452.335  
G00  
Z3.  
X95. Y58.6  
G01 Z0. F316.635  
X-95. F452.335  
G00  
Z200.  
M5  
(5.00 MM. DIA. END MILL-FINISH-2  
FLUTE)  
M3 S3183 G43 H2 G54 T2  
X60. Y-10. M54  
Z100.  
Z3.  
G01 Z-1. F222.817  
M98 P00010056  
G90 G00 X60. Y-10.  
Z3.  
G01 Z-2. F222.817  
M98 P00010056  
G90 G00 X60. Y-10.  
Z3.  
G01 Z-3. F222.817  
M98 P00010056  
G90 G00 X60. Y-10.  
Z3.  
G01 Z-4. F222.817  
M98 P00010056  
G90 G00 X60. Y-10.  
Z3.  
G01 Z-5. F222.817

M98 P00010056  
G90 G00 X60. Y-10.  
Z3.  
S3819  
G01 Z-5. F106.952  
Y-20. F152.789  
G03 X20. Y-60. I0. J-40.  
G01 X10.  
Y-10.  
X60.  
G00  
Z200.  
S3183  
X-10.  
Z3.  
G01 Z-1. F222.817  
X-10.05  
M98 P00010057  
G90 X-10. Y-10.  
G00  
Z3.  
G01 Z-2. F222.817  
X-10.05  
M98 P00010057  
G90 X-10. Y-10. F222.817  
G00  
Z3.  
G01 Z-3.  
X-10.05  
M98 P00010057  
G90 X-10. Y-10. F222.817  
G00  
Z3.  
G01 Z-4.  
X-10.05  
M98 P00010057  
G90 X-10. Y-10. F222.817  
G00  
Z3.  
G01 Z-5.  
X-10.05  
M98 P00010057  
G90 X-10. Y-10. F222.817  
G00  
Z3.  
S3819  
G01 Z-5. F106.952  
X-60.05 F152.789  
Y-20.  
G02 X-20.05 Y-60. I0. J-40.

G01 X-10.05  
Y-10.  
X-10.  
G00  
Z200.  
S3183  
Y10.  
Z3.  
G01 Z-1. F222.817  
X-10.05  
M98 P00010058  
G90 X-10. Y10.  
G00  
Z3.  
G01 Z-2. F222.817  
X-10.05  
M98 P00010058  
G90 X-10. Y10. F222.817  
G00  
Z3.  
G01 Z-3.  
X-10.05  
M98 P00010058  
G90 X-10. Y10. F222.817  
G00  
Z3.  
G01 Z-4.  
X-10.05  
M98 P00010058  
G90 X-10. Y10. F222.817  
G00  
Z3.  
G01 Z-5.  
X-10.05  
M98 P00010058  
G90 X-10. Y10. F222.817  
G00  
Z3.  
S3819  
G01 Z-5. F106.952  
X-60.05 F152.789  
Y20.  
G03 X-20.05 Y60. I0. J40.  
G01 X-10.05  
Y10.  
X-10.  
G00  
Z200.  
S3183  
X10.

Z3.  
G01 Z-1. F222.817  
M98 P00010059  
G90 G00 X10. Y10.  
Z3.  
G01 Z-2. F222.817  
M98 P00010059  
G90 G00 X10. Y10.  
Z3.  
G01 Z-3. F222.817  
M98 P00010059  
G90 G00 X10. Y10.  
Z3.  
G01 Z-4. F222.817  
M98 P00010059  
G90 G00 X10. Y10.  
Z3.  
G01 Z-5. F222.817  
M98 P00010059  
G90 G00 X10. Y10.  
Z3.  
S3819  
G01 Z-5. F106.952  
X60. F152.789  
Y20.  
G02 X20. Y60. I0. J40.  
G01 X10.  
Y10.  
G00  
Z200.  
M5  
(10.00 MM. DIA. TWIST DRILL)  
M3 S891 G43 H3 G54 T3  
X25. Y25. M54  
Z100.  
Z200.  
G98 G81 Z-13.004 R3. F89.127  
Y-25.  
X-25.05  
Y25.  
G80  
M5  
G80 M56  
G59 X0 Y0 Z100  
M16  
M30  
O55 (SUB PROGRAM)  
G91 X-165. Y-0. F653.373  
G00  
Z4.

X165. Y30.4  
G01 Z-4. F457.361  
X-165. Y-0. F653.373  
G00  
Z4.  
X165. Y30.4  
G01 Z-4. F457.361  
X-165. Y-0. F653.373  
G00  
Z4.  
X165. Y30.4  
G01 Z-4. F457.361  
X-165. Y-0. F653.373  
G00  
Z4.  
X165. Y26.  
G01 Z-4. F457.361  
X-165. Y-0. F653.373  
M99  
O56 (SUB PROGRAM)  
G91 X-0. Y-10. F318.31  
G03 X-40. Y-40. I0. J-40.  
G01 X-10. Y0.  
X0. Y50.  
X50. Y-0.

M99  
O57 (SUB PROGRAM)  
G91 X-50. Y0. F318.31  
X-0. Y-10.  
G02 X40. Y-40. I0. J-40.  
G01 X10. Y-0.  
X0. Y50.  
M99  
O58 (SUB PROGRAM)  
G91 X-50. Y0. F318.31  
X0. Y10.  
G03 X40. Y40. I0. J40.  
G01 X10. Y0.  
X-0. Y-50.  
M99  
O59 (SUB PROGRAM)  
G91 X50. F318.31  
Y10.  
G02 X-40. Y40. I0. J40.  
G01 X-10. Y0.  
Y-50.  
M99  
%

## **RESULT:**

Thus the NC code was generated for the component using capsmill.