Varuvan Vadivelan
Institute of Technology

Dharmapuri – 636 703

LAB MANUAL

Regulation : **2013**

Branch : ***B.E.* - Mechanical Engineering**

Year & Semester : **III Year / VI Semester**

ME6611- CAD/CAM LAB

ANNA UNIVERSITY : CHENNAI
REGULATION – 2013

ME6611- CAD/CAM LAB

1. 3D GEOMETRIC MODELLING **24 PERIODS**

List of Experiments

Creation of 3D assembly model of following machine elements using 3D Modeling software

1. Introduction of 3D Modeling software
2. Flange Coupling
3. Plummer Block
4. Screw Jack
5. Lathe Tailstock
6. Universal Coupling
7. Machine Vice
8. Stuffing box
9. Safety Valves
10. Swivel bearing
11. Connecting rod

* Students may also be trained in manual drawing of some of the above components

2. MANUAL PART PROGRAMMING. **21 PERIODS**

Part Programming - CNC Machining Centre

a) Linear Cutting.
b) Circular cutting.
c) Cutter Radius Compensation.
d) Canned Cycle Operations.

Part Programming - CNC Turning Centre

a) Straight, Taper and Radius Turning.
b) Thread Cutting.
c) Rough and Finish Turning Cycle.
d) Drilling and Tapping Cycle.

3. COMPUTER AIDED PART PROGRAMMING

CL Data and Post process generation using CAM packages.
Application of CAPP in Machining and Turning Centre.

TOTAL: 45 PERIODS

INDEX

I. COMPUTER AIDED DESIGN (CAD)

Sl. No.	Date	Name of the Machine Element	Signature of the Staff	Remarks
01		Flange Coupling		
02		Machine Vice		
03		Plummer Block		
04		Universal Joint		
05		Screw Jack		
06		Stuffing Box		
07		Tail Stock		
08		Swivel Bearing		
09		Connecting Rod		
10		Safety Valve		

II. COMPUTER AIDED MANUFACTURING (CAM)

Sl. No.	Date	Name of the CNC Operation (Manual Part Programming)	Signature of the Staff	Remarks
		TURINING		
01		Linear Interpolation		
02		Taper Turning		
03		Canned Cycle		
04		Thread Cutting		
05		Peck Drilling-Grooving		
06		Linear with Circular Interpolation – CNC Milling		
		MILLING		
07		Linear Interpolation		
08		Circular Interpolation		
09		Peck Drilling		
10		Mirror Image		
11		Square Pocketing with Cutter Compensation		

Introduction about CATIA V5

It is important to understand the format of the manual in order to use it effectively. This manual is designed to be used along with an instructor; however, you will need to do a lot of reading as well, in order to fully understand CATIA Version 5(computer aided three dimensional integrated application). The exercises in this book will list steps for you to complete, along with explanations that inform you what you have just done and what you are getting ready to do. The actual steps are in bold type and the information that follows the steps is for your benefit. Anything that appears in italics refers to a message CATIA provides—this includes information in pull-down menus, pop-up windows and other messages.

Select a location to the right of the origin. This specifies the other end point of the line. You will continue specifying locations in order to complete your profile. It should appear similar to the diagram shown below.

As you can see, the desired action blends in with the text except that it appears in bold. The information following the step explains what that step accomplished and where you are going next. It is important to read this information in order to better your understanding of CATIA Version 5.

Also, you will find that the exercises build upon themselves. Later exercises often assume you know how to do certain steps which have been covered earlier in the course. If you did not quite pick up what you needed to know from an exercise, you will probably want to review it several times before moving onto more advanced sections. The advanced sections assume that you have a good understanding of the previous sections therefore fewer steps will be provided. Eventually, you are expected to be able to create parts without any steps.

As mentioned before, the different workbenches contain different toolbars. Although some toolbars will appear in almost every workbench. This section will discuss how to customize your toolbars.

In most of the workbenches, there are too many toolbars to display in one column or row along the right side of the screen and along the bottom of the screen. In this case, it is important to move the toolbars around so that you can view all of them. If you can't view a toolbar, then you can't use the toolbar.

Having toolbars hidden off the screen isn't very helpful since you can't pick any of the icons on them. Therefore, it is a good idea to move the toolbars so you can see them completely.

You can drag toolbars around by selecting and holding the handler bar.

Select the handler bar as shown below on the Sketch-Based Features toolbar.

Drag the toolbar out into the display. It should appear as shown.

While holding down on the handler bar, select the Shift key on the keyboard. Notice that the toolbar toggles to be horizontal instead of vertical. If you let go of the handler bar while the shift key is pressed, the toolbar will remain in the horizontal position.

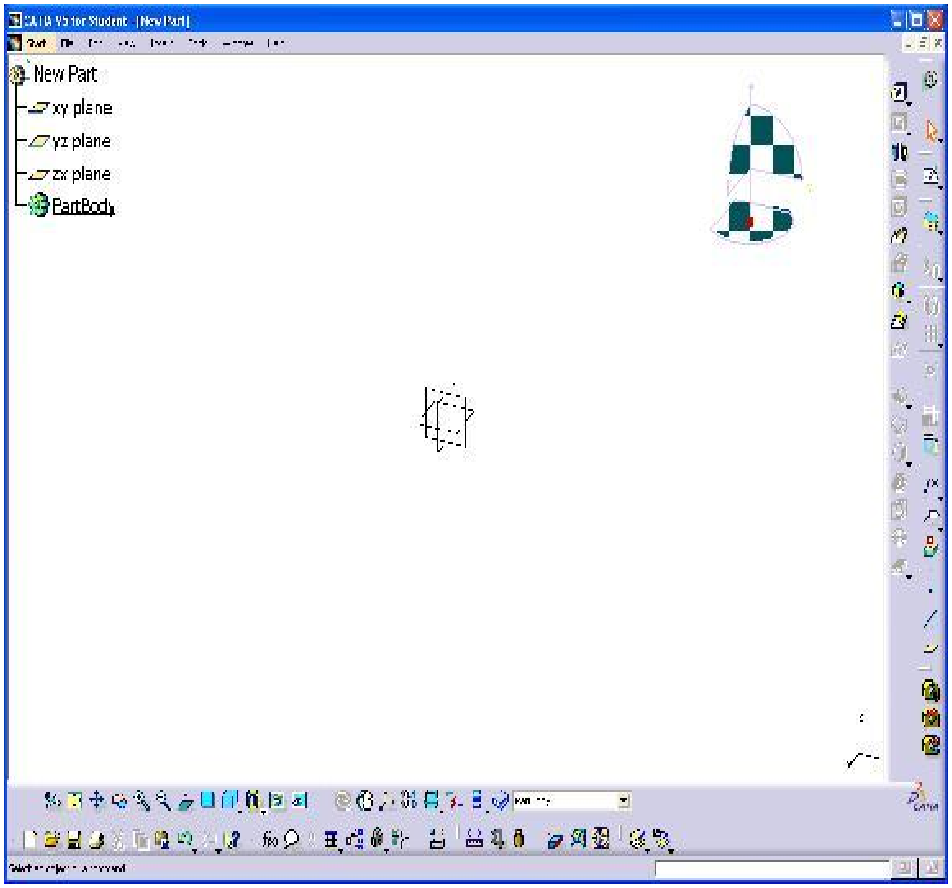
Select the "x" in the corner of the toolbar to turn the toolbar off. The toolbar disappears. Select the View pull down menu and move your cursor over the Toolbars option. This provides a list of all of the available toolbars within this workbench.

Select Sketch-Based Features to turn the toolbar back on. Every toolbar that has a checkmark beside it in the list is turned on.

This is a good place to look if you can't find a toolbar. We will discuss a way to set all of the toolbars back to the default later on.

Select the handler for the toolbar and drag it over to the right side and drop it next to the column of toolbars. If you get it close enough, it will create a second column as shown. If you get it too close, it will add it back into the first column.

MAIN SCREEN OF CATIA V5



Position the toolbars as shown so that all of them can be seen.

This will make finding toolbars much more convenient later on.

Tools customization contains a number of options allowing you to customize certain aspects of the CATIA workbenches.

Select the Tools pull down menu and select Customize. The Customize window appears.

This allows you to specify what workbenches you want available to you on the start menu. You can also access these workbenches by selecting the workbench icon on each workbench. Whatever is listed in the Favorites area will be available when you select the workbench icon and they will be listed at the top of the pull down menu Start.

Available List of all of the available workbenches within CATIA V5

START MENU WORK BENCN SCREEN:

Favorites List of workbenches that you will use on an everyday basis

Moves workbenches from one side to the other.

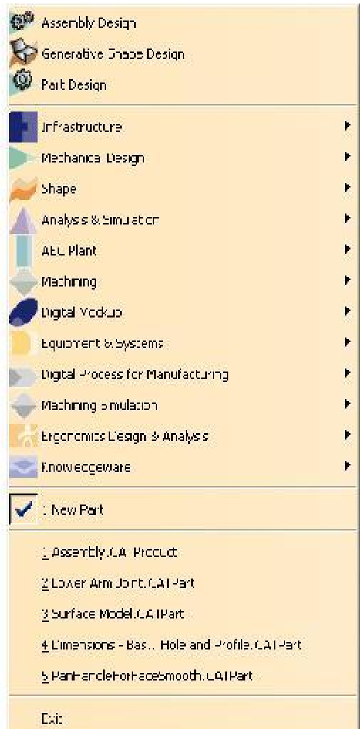
Move the Assembly Design, Generative Shape Design and Part Design workbenches from the Available frame to the Favorites frame as shown. You may select the

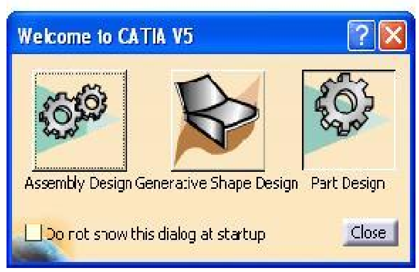
Workbenches and then select the arrows () to move them back and forth or you may drag and drop the workbenches within the two frames.

Select the Start pull down menu. The menu displays. Notice that the three workbenches that you chose as your favorites display at the top of the menu.

Select the Part Design workbench icon. The Welcome to CATIA V5 window displays. This icon is located in the toolbars along the right side of the display, near the top.

ASSEMBLY WORK BENCH SCREEN:





Notice that the three favorite workbenches that were chosen display here as well. This provides another method for switching between workbenches.

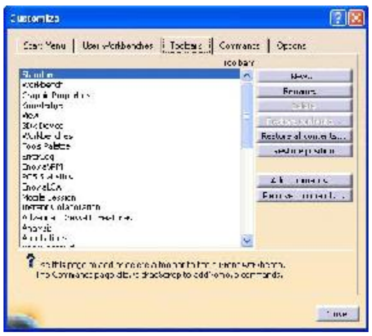
Select Generative Shape Design. You are switched into the Generative Shape Design workbench.

Select the Start pull down menu and select Part Design from the favorites at the top of the menu. You are switched back into the Part Design workbench.

Select the Tools pull down menu and select Customize again. The Customize window appears.

Select the User Workbenches tab.

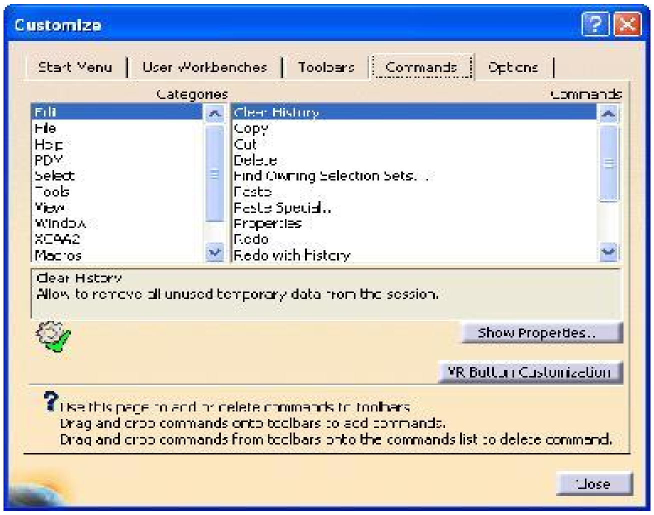
USER WORKBENCHES SCREEN:



This allows you to modify existing toolbars or create your own toolbars. You can also restore the position of the toolbars if you move them around and want to get them back to their default position. This can be very useful if you lose a toolbar and you cannot find it.

New	Allows you to create a new toolbar
Rename	Allows you to rename a toolbar
Delete	Allows you to delete a toolbar that you created
Restore contents	Allows you to restore an icon to a toolbar that you created
Restore contents	Allows you to restore a toolbar back to its default if it had previously had an icon added to it or deleted from it
Restore position	Turns all of the default toolbars back on for the workbench that you are in and restores the default position of those toolbars as well
Add commands	Allows you to add icons to an existing toolbar
Remove commands	Allows you to remove icons that were added to a toolbar

COMMAND WORK BENCH SCREEN:



Select the Commands tab.

Allows you to add or delete keyboard commands from toolbars. You can also set up hot keys for various options if you Show Properties.

Large Icons	
Icon Size Ratio	Allows you to adjust the size of the icons
Tooltips	Allows you to toggle the tooltips on or off
User Interface Language	Selects the language that will be displayed
Lock Toolbar Position	Allows you to lock the toolbars in place so that they cannot be moved
Center the display	Select and release the middle mouse button on the location that you want to be centered and it will move to the center of the display
Pan	Select and hold the middle mouse button and you can move your display around by moving the mouse
Rotate	Select and hold the middle mouse button and then select and hold either the first or third mouse buttons and you can rotate the display around by moving the mouse. You should see a rotational ball appear for reference. Both buttons will be held down simultaneously.
Zoom	Select and hold the middle mouse button and then select and

	release either the first or third mouse buttons and you can zoom in or out by moving the mouse up or down. Only the middle mouse button will be held down.
Rotate and Zoom	While on an geometrical entity you can press and hold the Shift key and then press the middle mouse button to perform a rotation and zoom using a viewpoint control.

I. COMPUTER AIDED MANUFACTURING (CAM)

CNC LATHE HYTECH

FANUC CONTROLLER - TECHNICAL SPECIFICATION CNC LATHE TRAINER

1. Model	: HE-100-CNC-PC
2. Distance between centers	:230mm
3. Maximum machining diameter	:40mm
4. Spindle speed	:50 to 3000rpm
5. Swing over carriage	:100mm
6. Spindle motor	:AC Motor, 3HP
7. Rapid feed rate X	:150mm/min
8. Rapid feed rate Z	:150mm/min
9. Type of bearing for spindle	:Angular contact bearing

FANUC CONTROLLER - TECHNICAL SPECIFICATION CNC MILL TRAINER

1. Model	: MT-250
2. Table dimensions	:600*225mm
3. Clamping area	:450*160mm
4. “T” slot	:10mm
5. Axes: i) Longitudinal traverse	:250mm
ii) Cross traverse	:150mm
iii) Vertical traverse	: 200mm
6. Milling head:	
i) Spindle inside taper	: ISO 30
ii) Spindle speed	: 200 to 2500mm
7. Spindle motor	:DC Motor, 1.5HP
8. Rapid feed rate X	:250mm/min
9. Rapid feed rate Y	:250mm/min
10. Rapid feed rate Z	:250mm/min

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.

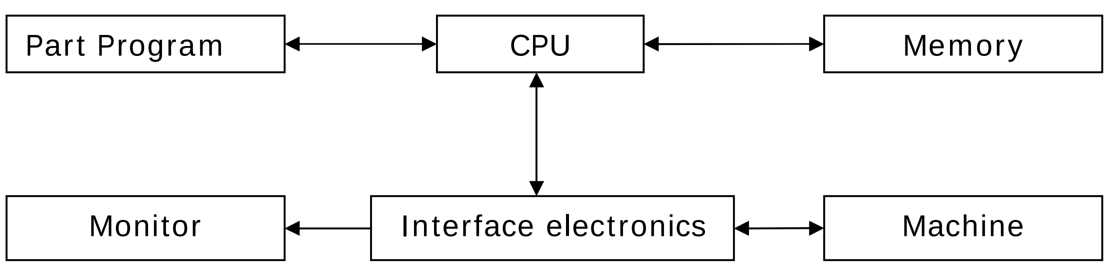
DISADVANTAGE OF NC MACHINE

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.

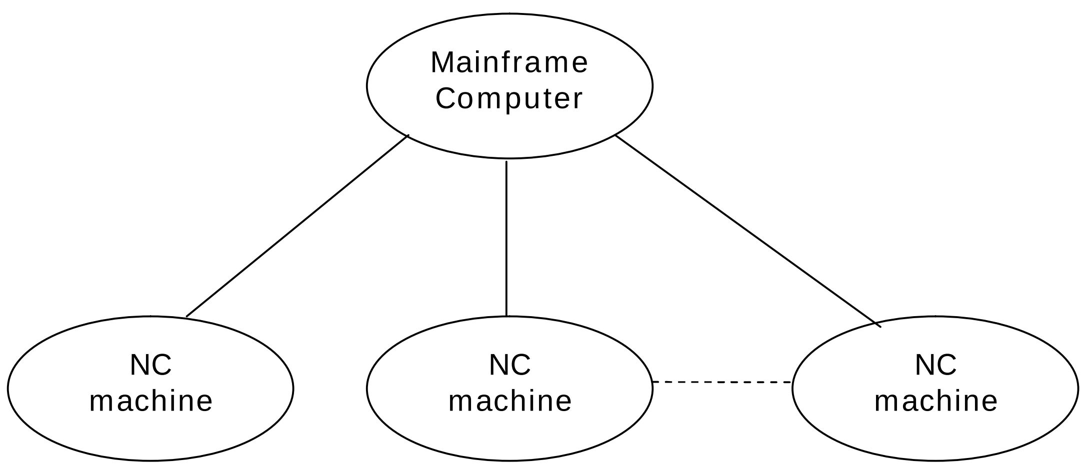


Fig.3. Direct Numerical Control system

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.

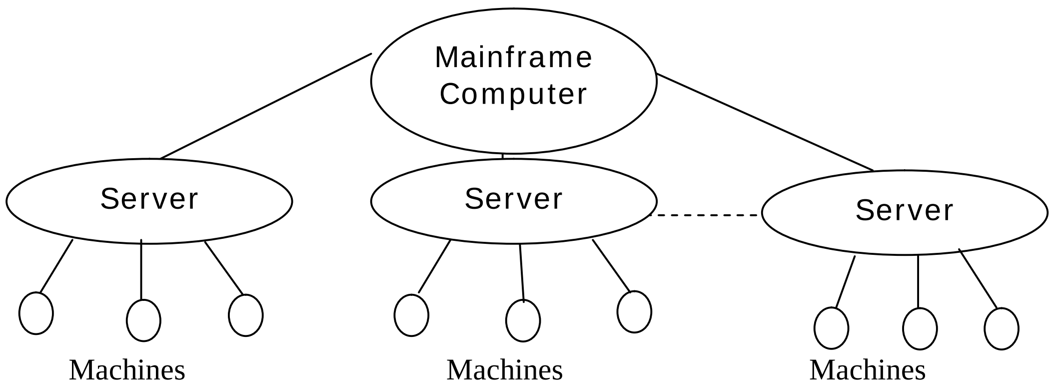


Fig.4. Distributive Numerical Control system

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.

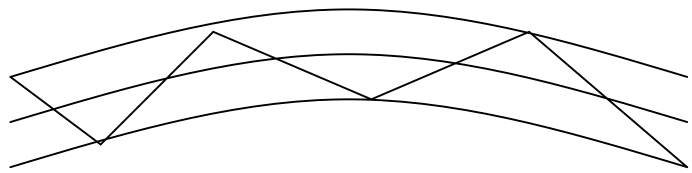


Fig.5. Straight line approximation of a curve

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.

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.

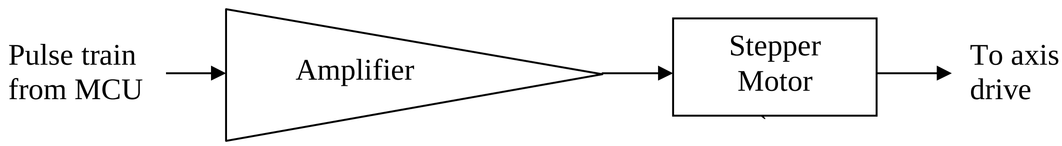


Fig.6. Open-loop control system

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.

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

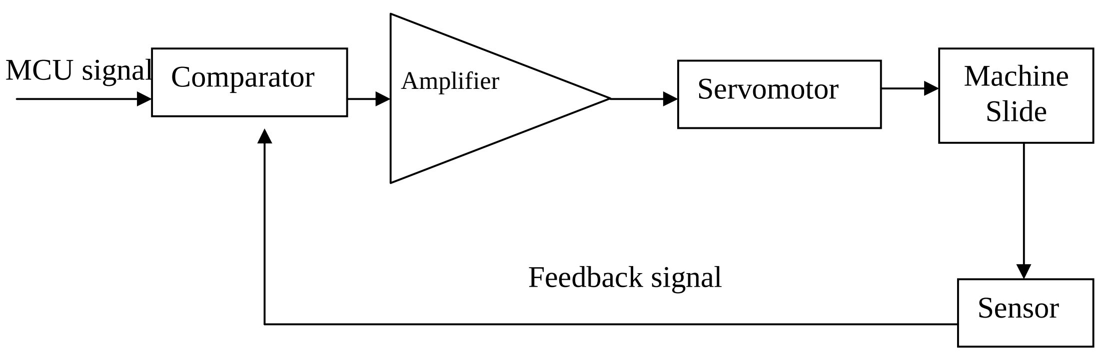


Fig.7. Closed-loop control system

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
M08	-	Coolant off
M10	-	Vice /chuck open
M11	-	Vice/chuck close

PREPARATORY FUNCTION (G FUNCTION)

G codes are instructions describing machine tool movement.

G00	-	Rapid 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
G79	-	Canned 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. 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

LIST OF G – CODES FOR MILLING

G CODES	FUNCTION
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	2nd reference point return
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G43	Tool length compensation
G52	Work coordinate system 1
G54	Work coordinate system 2

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

LIST OF M CODES FOR MILLING

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

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 G02 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.

<u>G84 RIGHT HAND TAPPING CYCLE:</u>

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.

Ex No: 01

Date :

FLANGE COUPLING

AIM:

To model the parts of FLANGE COUPLING and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

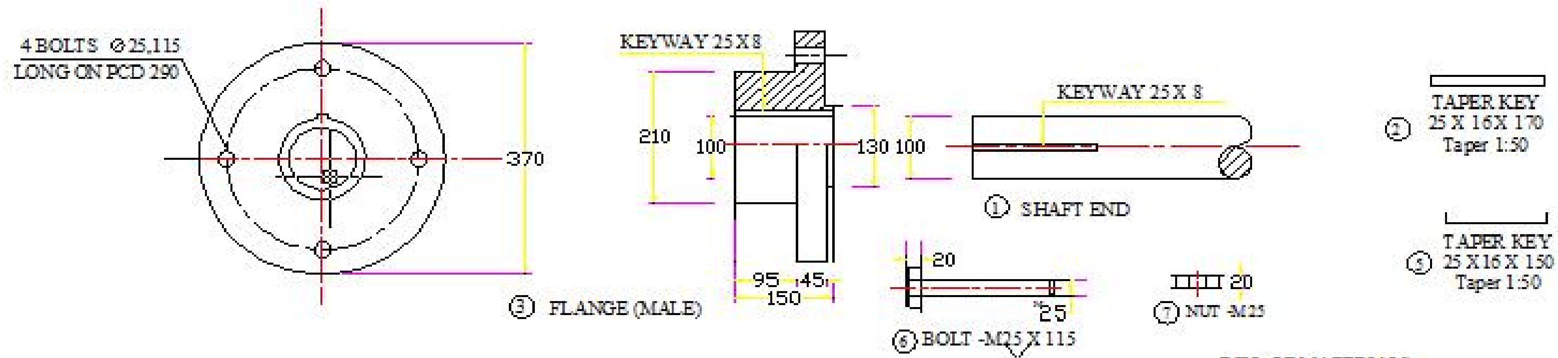
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut, Chamfer, Mate, Insert, Move, etc.

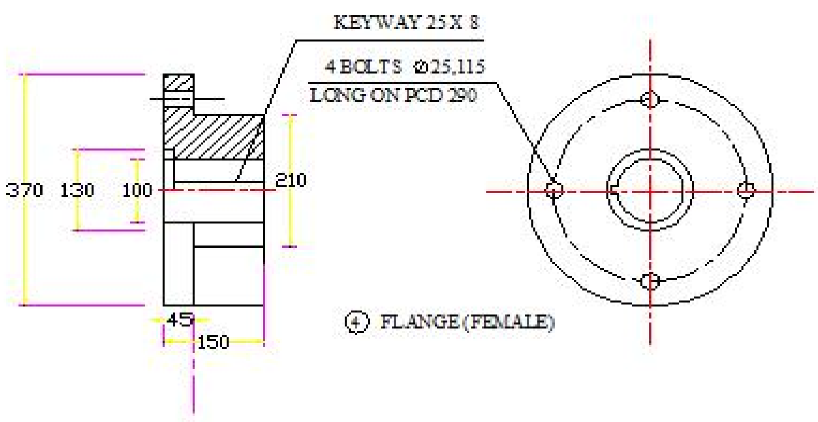
PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:01

FLANGED COUPLING - PART DRAWING





BILL OF MATERIALS

ITEM NO.	DESCRIPTION	MATERIAL	NO. OFF
1	SHAFT END	STEEL	2
2	KEY (LONG)	MILD STEEL	1
3	FLANGE (MALE)	CAST IRON	1
4	FLANGE (FEMALE)	CAST IRON	1
5	KEY (SHORT)	MILD STEEL	1
6	BOLT	MILD STEEL	4
7	NUT	MILD STEEL	4

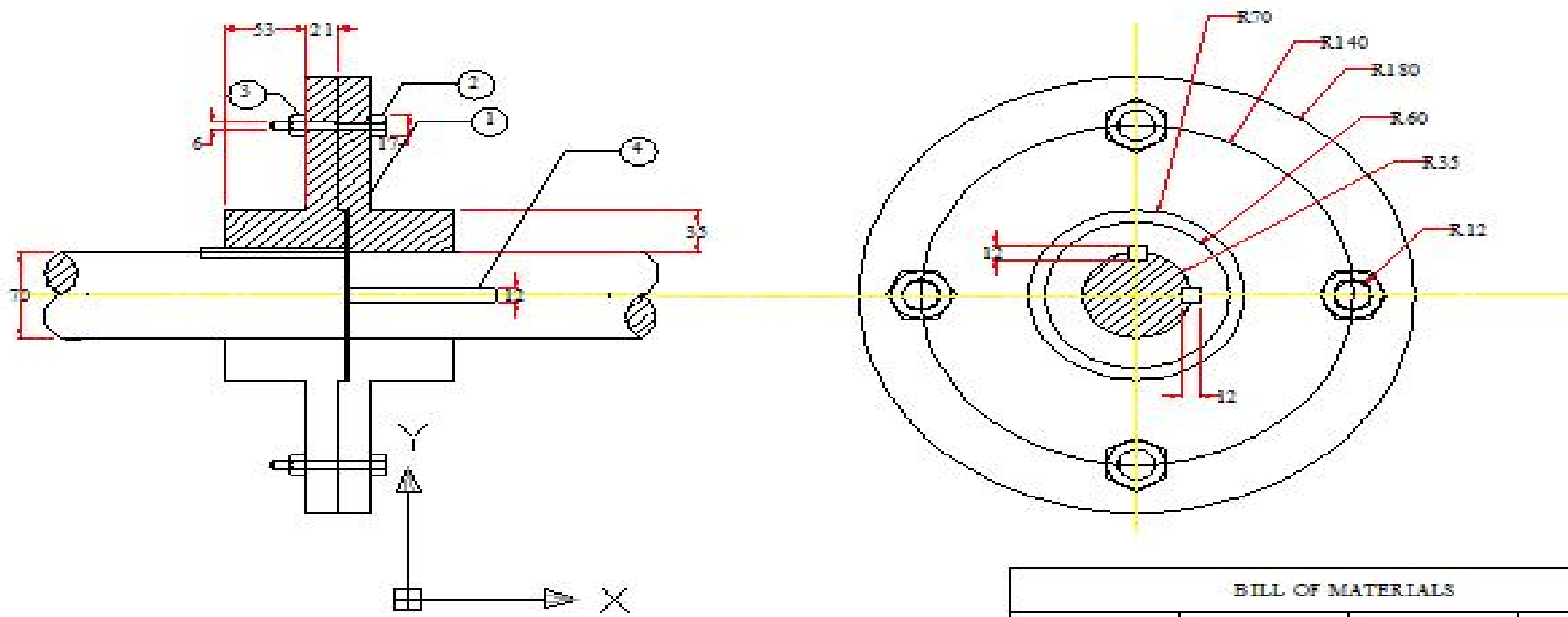
ALL DIMENSIONS ARE IN "MM"

RESULT:

Thus the assembly drawing FLANGE COUPLING is drawn taken the printout.

EX.NO:01

FLANGED COUPLING - ASSEMBLY DRAWING



BILL OF MATERIALS			
s.no	part name	material	no.off
1	flange	cast iron	2
2	bolts	mild steel	4
3	nuts	mild steel	4
4	keys	mild steel	2

Ex.No: 02

Date:

MACHINE VICE

AIM:

To model the parts of MACHINE VICE and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

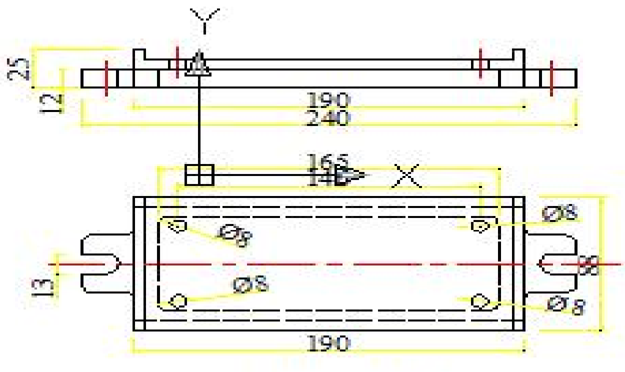
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut, Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

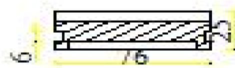
- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

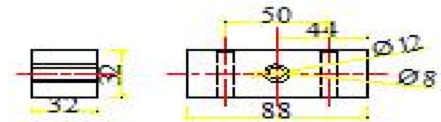
EX.NO:2

MACHINE VICE - PART DRAWING



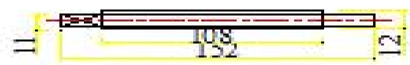
BASE C.I-1 OFF



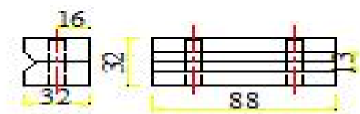


BLOCK C.I-1 OFF

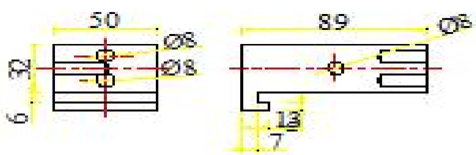
CSK HEAD SCREW



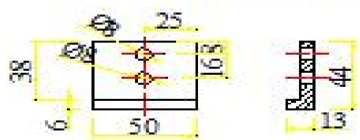
SCREW M.S-1 OFF



FIXED JAW C.I-1 OFF



SLIDING JAW C.I-1 OFF



END PLATE

BILL OF MATERIALS			
PART NO	NAME	MATERIAL	NO.OFF
1	BASE	CI	1
2	SCREW	MS	1
3	BLOCK	CI	1
4	SLIDING JAW	CI	1
5	FIXED JAW	CI	1
6	SET SCREWS	MS	6
7	END PLATE	MS	1

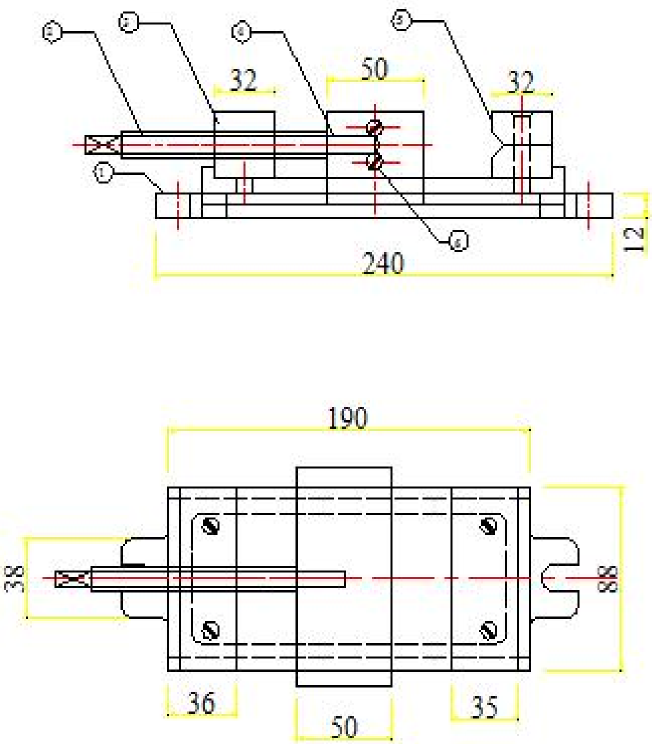
ALL DIMENSIONS ARE IN MM

RESULT:

Thus the assembly drawing MACHINE VICE is drawn taken the printout.

EX.NO:02

MACHINE VICE - ASSEMBLY VIEW



BILL OF MATERIALS			
PART NO	**NAME**	**MATERIAL**	**NO.OFF**
1	BASE	CI	1
2	SCREW	MS	1
3	BLOCK	CI	1
4	SLIDING JAW	CI	1
5	FIXED JAW	CI	1
6	SET SCREWS	MS	6
7	END PLATE	MS	1

ALL DIMENSIONS ARE IN MM

Ex.No: 03

Date:

PLUMMER BLOCK

AIM:

To model the parts of PLUMMER BLOCK and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

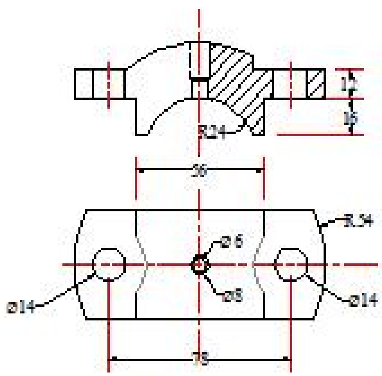
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut, Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

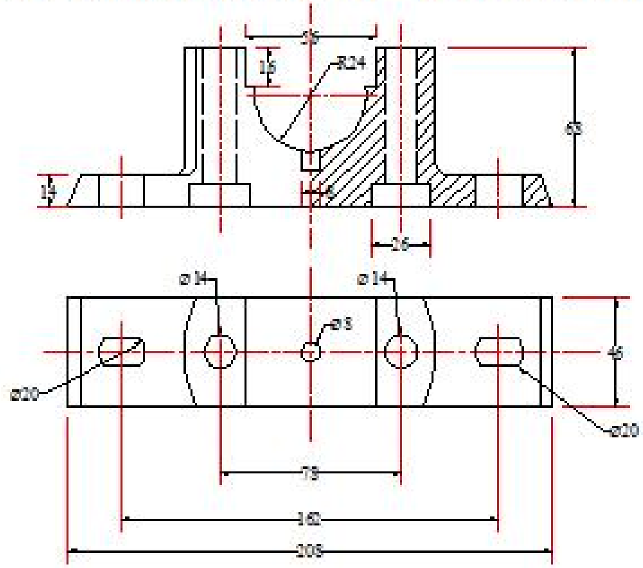
- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:03

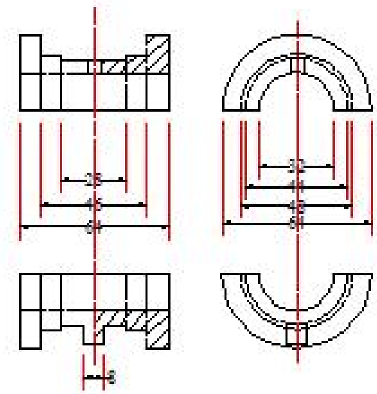
PLUMMER BLOCK - PATR DRAWING



CAP CI-1 OFF



BODY C.I-1 OFF



BRASSES G.M-2 OFF

NUTS & LOCK NUTS M.S-2 OFF

BILL OF MATERIALS

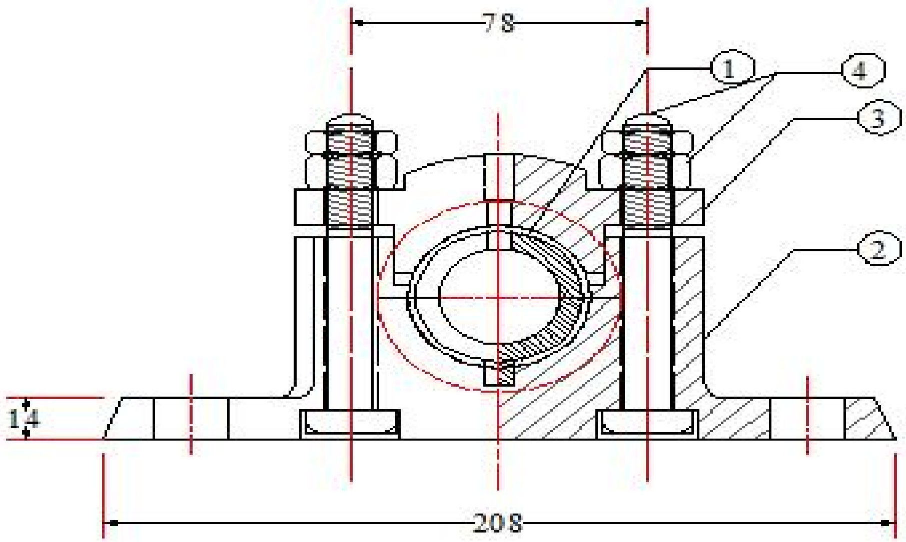
S.NO	NAME	MATERIALS	NO.OFF
1	BRASSES	GUN METAL	2
2	BODY	CAST IRON	1
3	CAP	CAST IRON	1
4	BOLT&N UTS	MILD STEEL	2

*ALL DIMENSIONS ARE IN MM

RESULT:

Thus the assembly drawing PLUMMER BLOCK is drawn taken the printout.

EX.NO:03 PLUMMER BLOCK - ASSEMBLY DRAWING



BILL OF MATERIALS			
S.NO	NAME	MATERIALS	NO.OFF
1	BRASSES	GUN METAL	2
2	BODY	CAST IRON	1
3	CAP	CAST IRON	1
4	BOLT&NUTS	MILD STEEL	2

ALL DIMENSIONS ARE IN MM

Ex. No: 04

Date :

UNIVERSAL COUPLING

AIM:

To model the parts of UNIVERSAL COUPLING and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

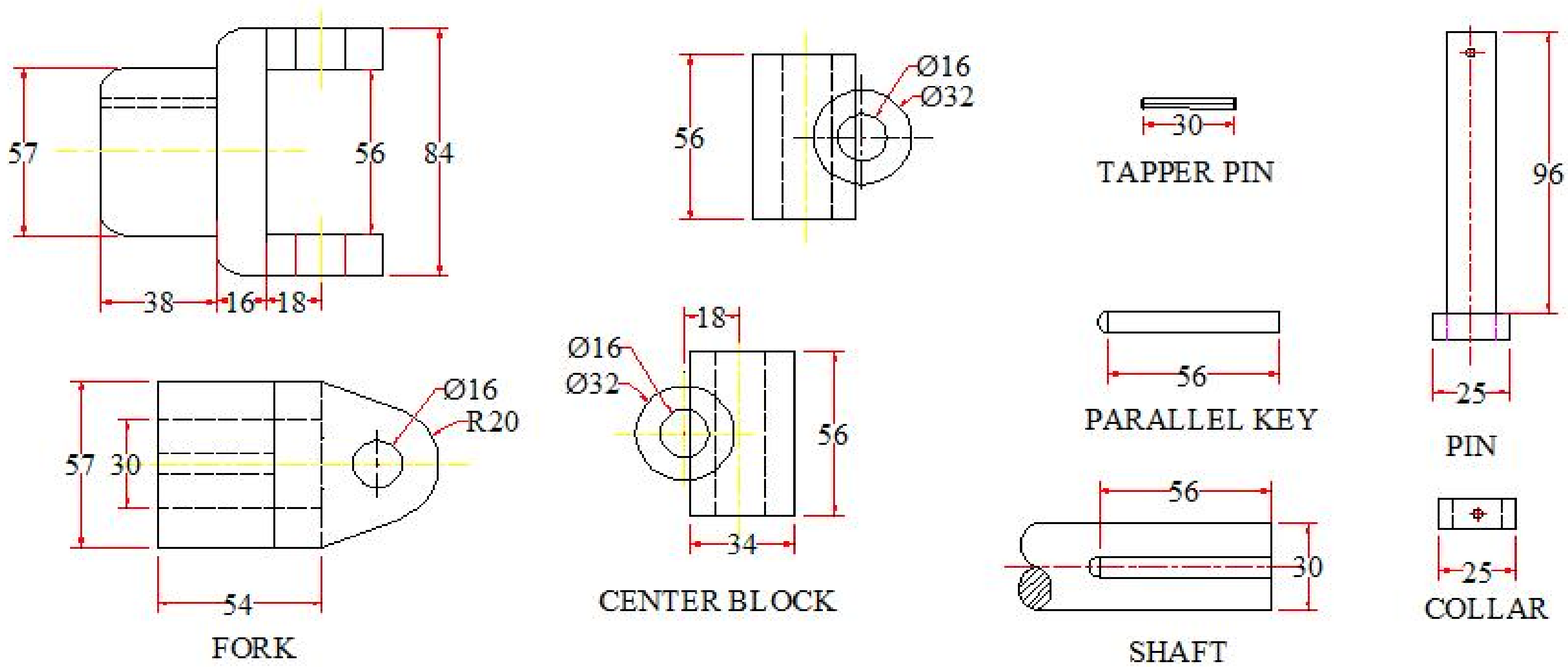
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:04

UNIVERSAL COUPLING PART DRAWING

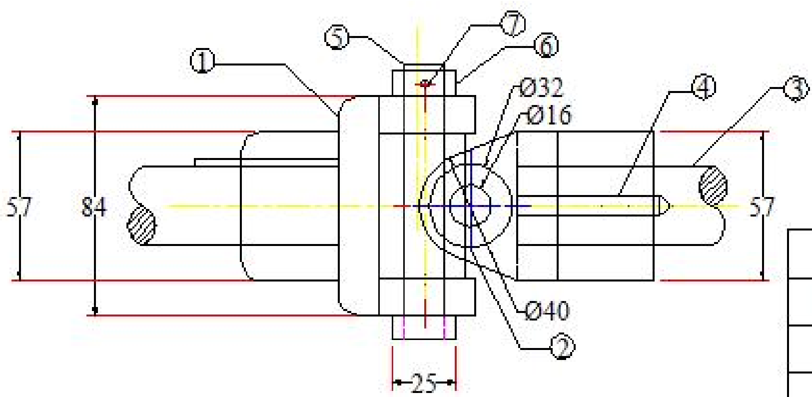


*ALL DIMENSIONS ARE IN MM

RESULT:

Thus the assembly drawing UNIVERSAL COUPLING is drawn taken the printout.

EX.NO:04 UNIVERSAL COUPLING- ASSEMBLY VIEW



BILL OF MATERIALS			
PART NO	**NAME**	**MATERIAL**	**NO.OFF**
1	FORK	CI	2
2	CENTER BLOCK	CI	1
3	SHAFT	MS	2
4	PARALLEL KEY	MS	2
5	PIN	MS	2
6	COLLAR	MS	2
7	TAPER PIN	MS	2

*ALL DIMENSIONS ARE IN MM

Ex.No: 05

Date :

SCREW JACK

AIM:

To model the parts of SCREW JACK and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

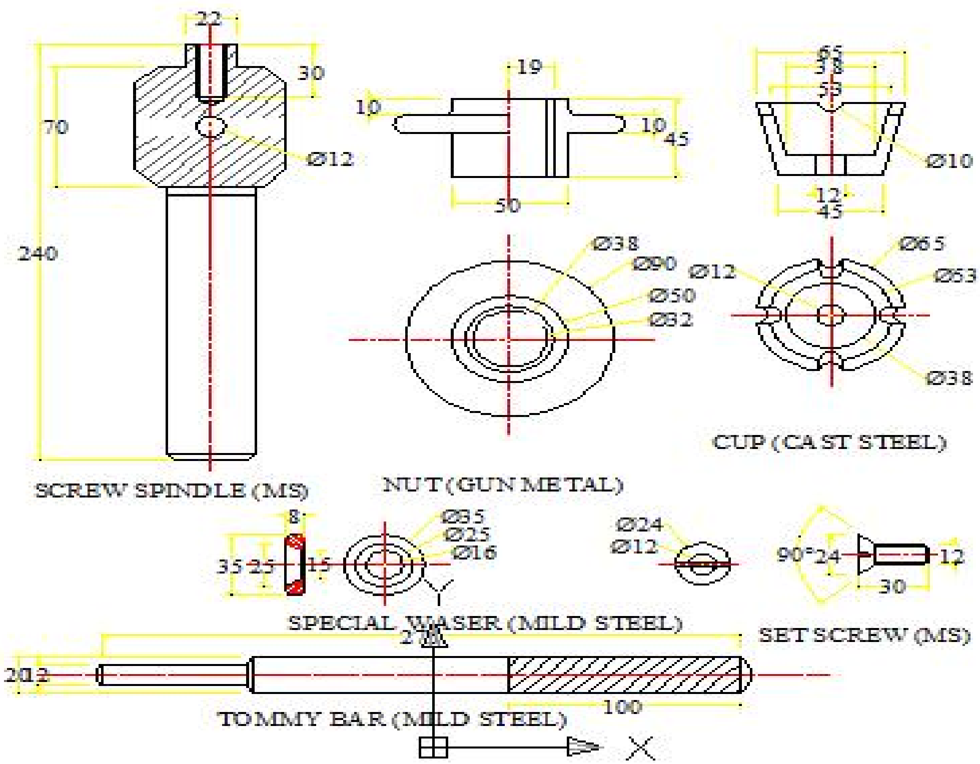
COMMANDS USED:

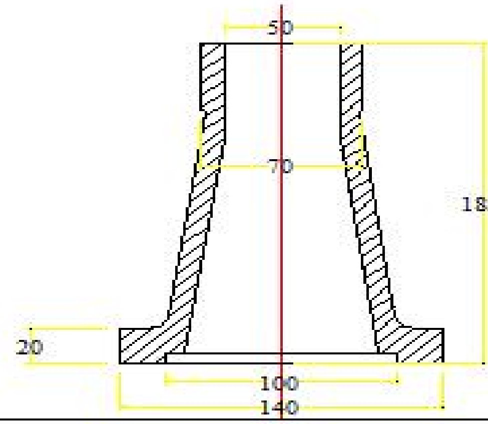
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:05 SCREW JACK - PART DRWING





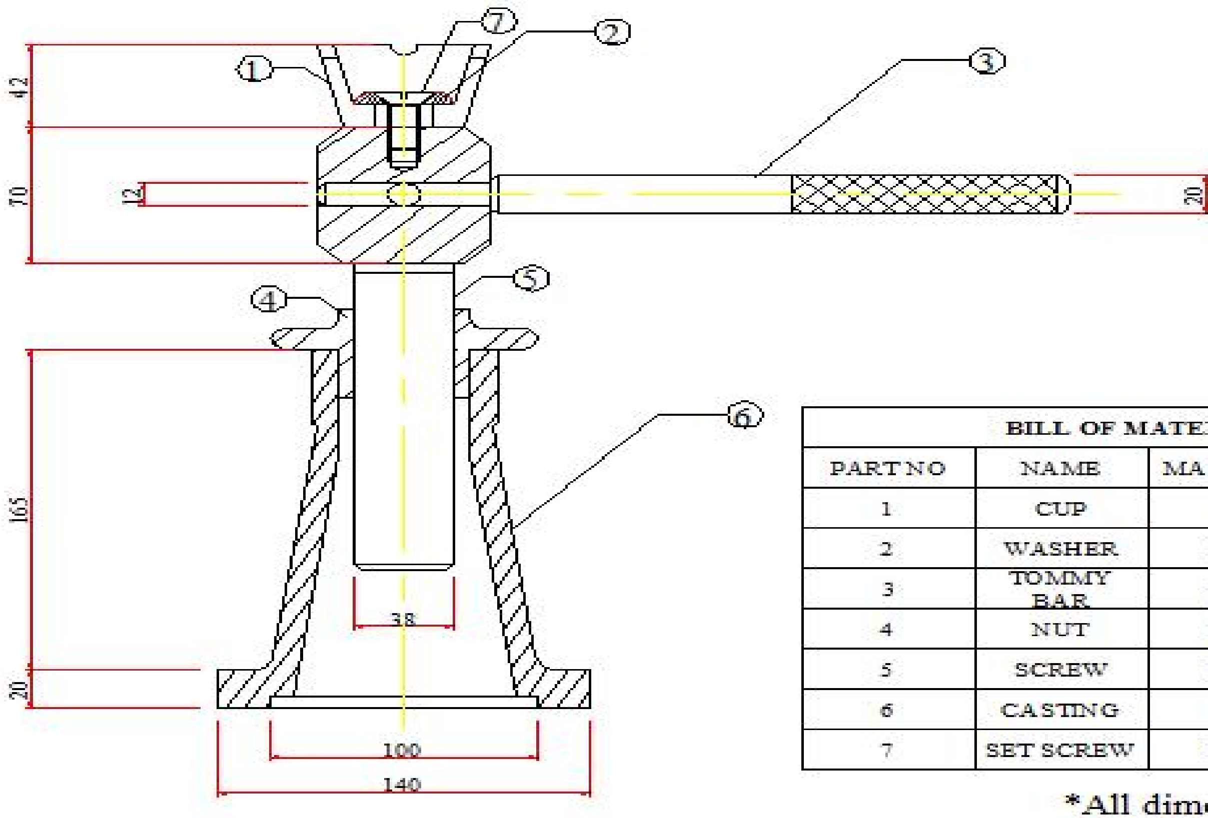
BILL OF MATERIALS			
PART NO	NAME	MATERIAL	NO;OFF
1	CUP	CS	1
2	WASHER	MS	1
3	TOMMY BAR	MS	1
4	NUT	GM	1
5	SCREW	MS	1
6	CASTING	CI	1
7	SET SCREW	MS	1

*All dimensions are in mm

RESULT:

Thus the assembly drawing SCREW JACK is drawn taken the printout.

EX.NO:05 SCREW JACK - ASSEMBLY DRAWING



BILL OF MATERIALS			
PART NO	NAME	MATERIAL	NO.OFF
1	CUP	CS	1
2	WASHER	MS	1
3	TOMMY BAR	MS	1
4	NUT	GM	1
5	SCREW	MS	1
6	CASTING	CI	1
7	SET SCREW	MS	1

*All dimensions are in mm

Ex.No : 06

Date :

STUFFING BOX

AIM:

To model the parts of STUFFING BOX and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

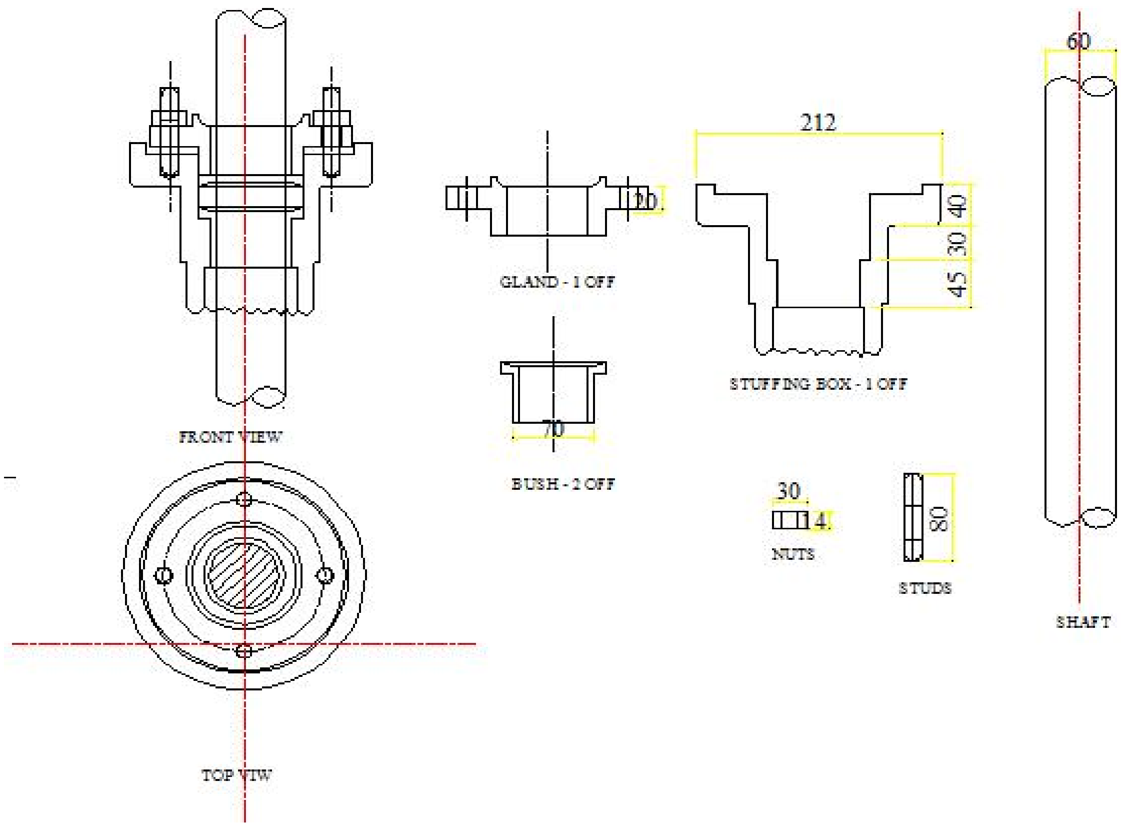
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:06

VERTICAL STUFFING BOX PART DRAWING



BILL OF MATERIALS

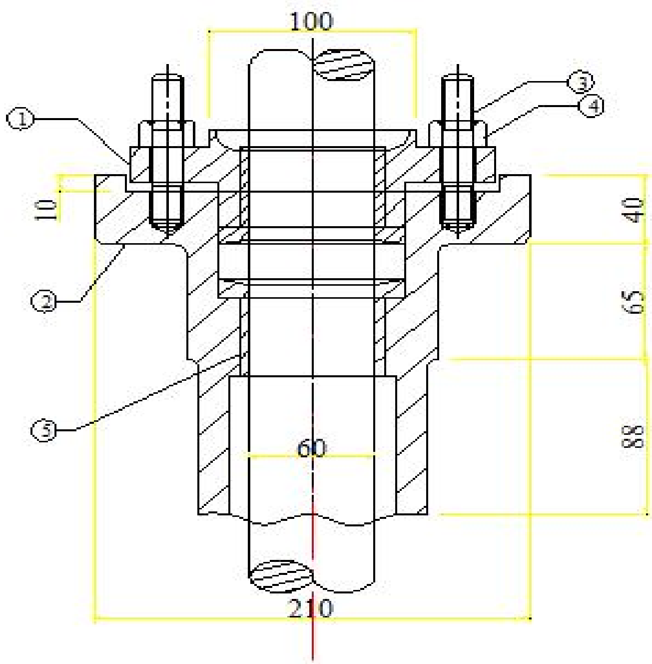
PART NO	NAME	MATERIAL	NO OFF
01	GLAND	CI	1
02	STUFFING BOX	CI	1
03	STUDS	MS	4
04	BOLTS	MS	4
05	BUSHES	BRASS	2

ALL DIMENSION ARE IN MM

RESULT:

Thus the assembly drawing STUFFING BOX is drawn taken the printout.

EX.NO:06 VERTICAL STUFFING BOX



BILL OF MATERIAL			
PART NO	NAME	MATERIAL	NO.OFF
1	GLAND	CI	1
2	STUFFING BOX	CI	1
3	STUDS	MS	4
4	BOLTS	MS	4
5	BUSHES	BRASS	2

*ALL DIMENSIONS ARE IN MM

Ex.No: 07

Date:

TAIL STOCK

AIM:

To model the parts of TAIL STOCK and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

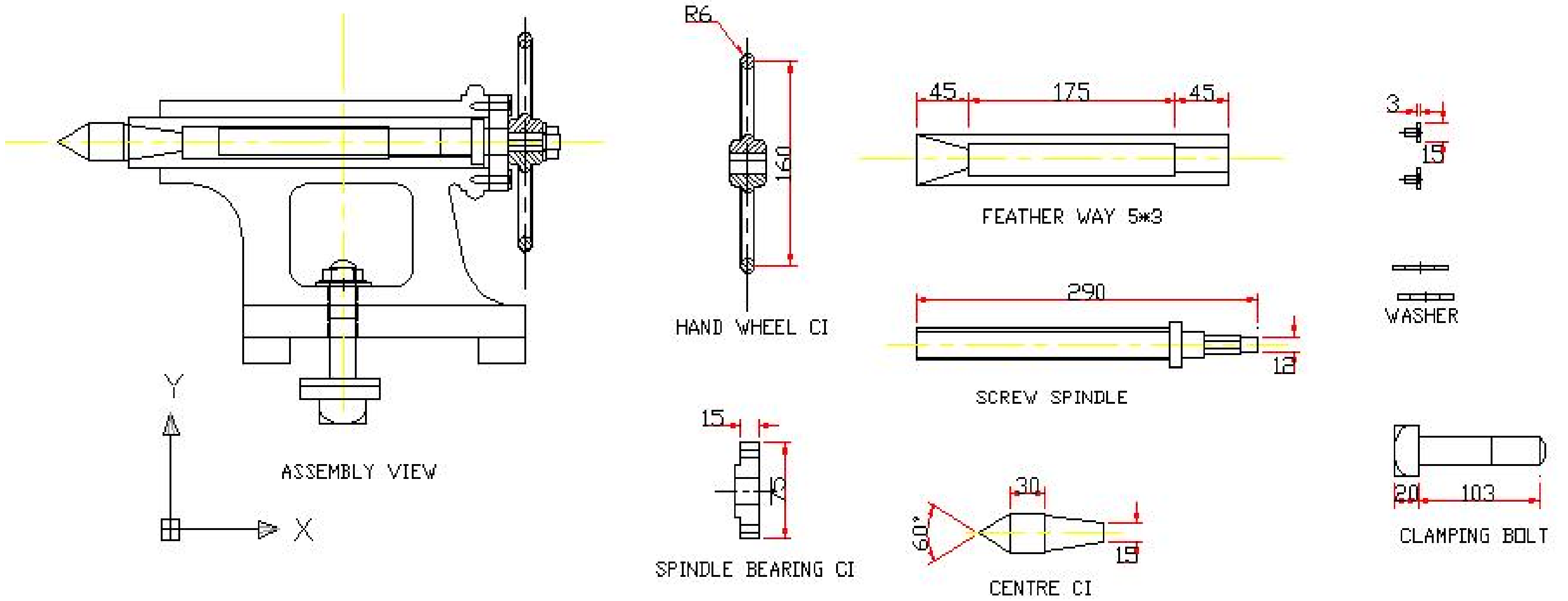
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:07

TAIL STOCK PART DRAWING



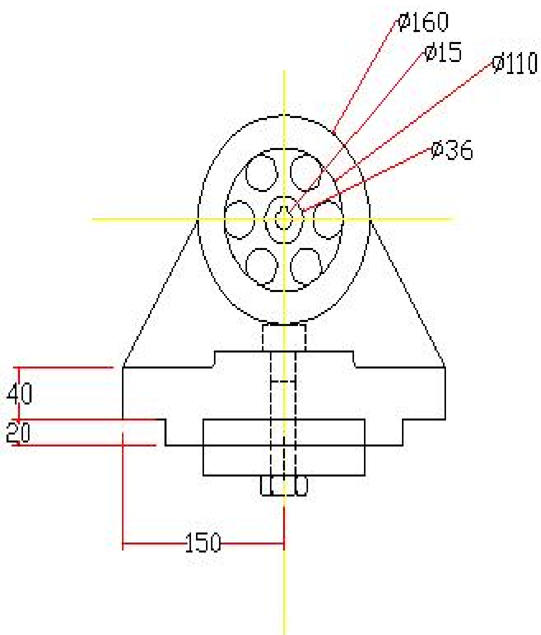
ALL DIMESIONS ARE IN mm

RESULT:

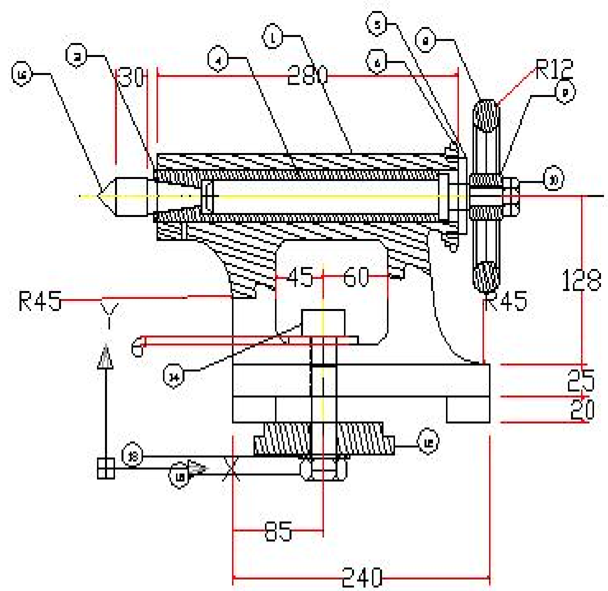
Thus the assembly drawing TAIL STOCK is drawn taken the printout.

EX.NO:07

TAIL STOCK - ASSEMBLY DRAWING



SIDE VIEW



FRONT VIEW

ALL DIMENSIONS ARE IN MM

BILL OF MATERIAL

PART NAME	DESCRIPTION	MATERIAL	NO OFF
1	BODY	CI	1
2	FEATHER	MS	1
3	BARREL	CI	1
4	SCREW SPINDLE	MS	1
5	FLANGE	CI	1
6	SCREW	MS	4
7	FEATHER KEY	MS	1
8	HAND WHEEL	CI	1
9	WASHER M12 STD	MS	1
10	HEXAGONAL NUT M12	MS	1
11	STUD	MS	1
12	CLAMPING PLATE	CI	1
13	SQ HEADED BOLT	MS	1
14	WASHER M-20STUD	MS	1
15	HEXAGONAL NUT M22	MS	1
16	CENTRE	CS	1

Ex.No: 08

Date :

SWIVEL BEARING

AIM:

To model the parts of SWIVEL BEARING and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

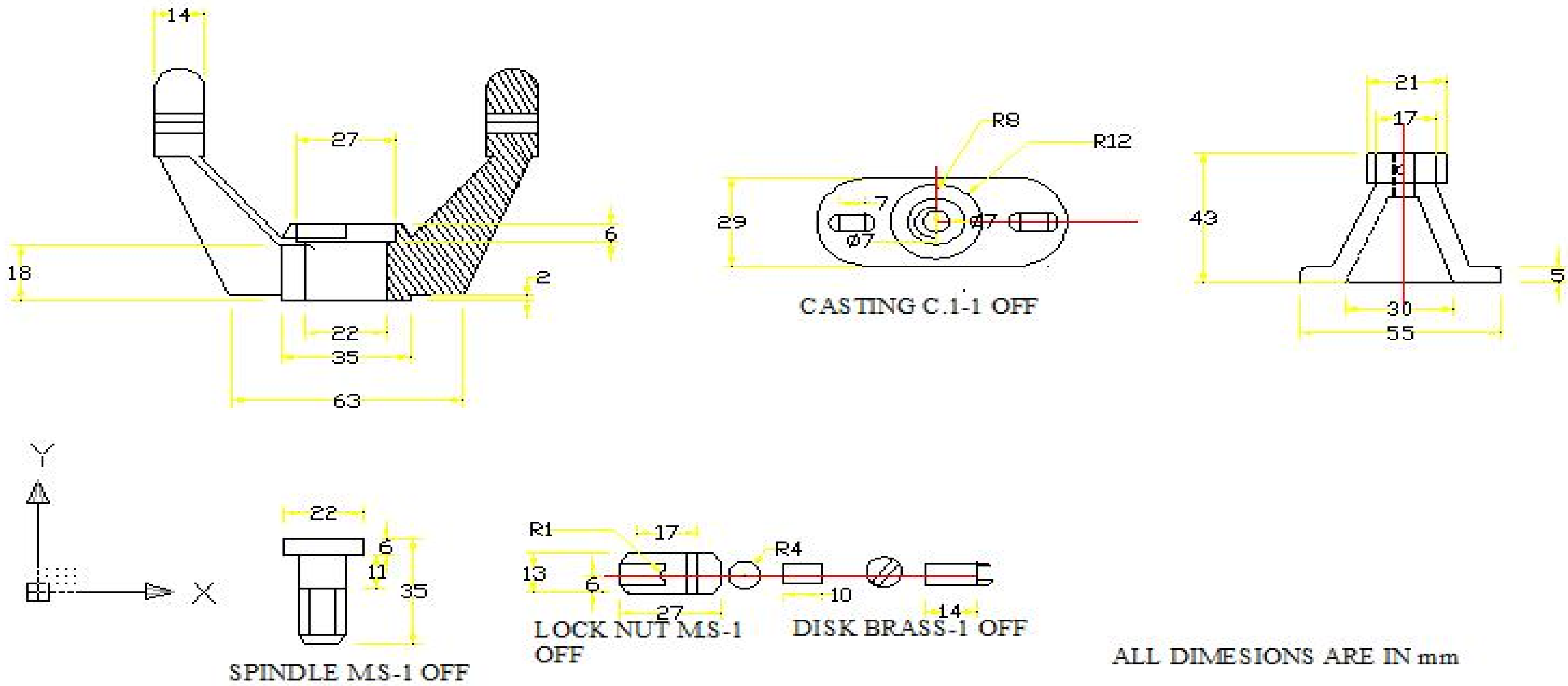
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:08

SWIVEL BEARING - PART DRAWING



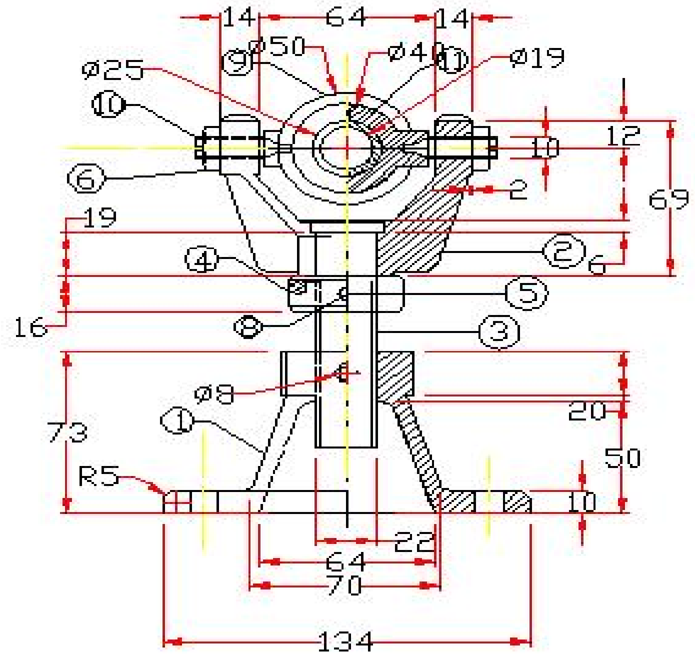
ALL DIMESIONS ARE IN mm

RESULT:

Thus the assembly drawing SWIVEL BEARING is drawn taken the printout.

EX.NO:08

SWIVEL-BEARING



ALL DIMENSIONS ARE IN MM

BILL OF MATERIAL			
PART NO	PART NAME	MATERIAL	NO. OFF
1	CASTING	C.I	1
2	FORK	MS	1
3	SPINDLE	MS	1
4	LOCK NUT	NS	1
5	SET SCREW	MS	2
6	NUT	MS	2
7	DISC	BRASS	1
8	DISC	BRASS	1
9	BEARING	C.I	1
10	SET SCREW	MS	1
11	BUSH	BRASS	1
12	SET SCREW	MS	1

Ex.No:09

Date :

CONNECTING ROD

AIM:

To model the parts of CONNECTING ROD and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

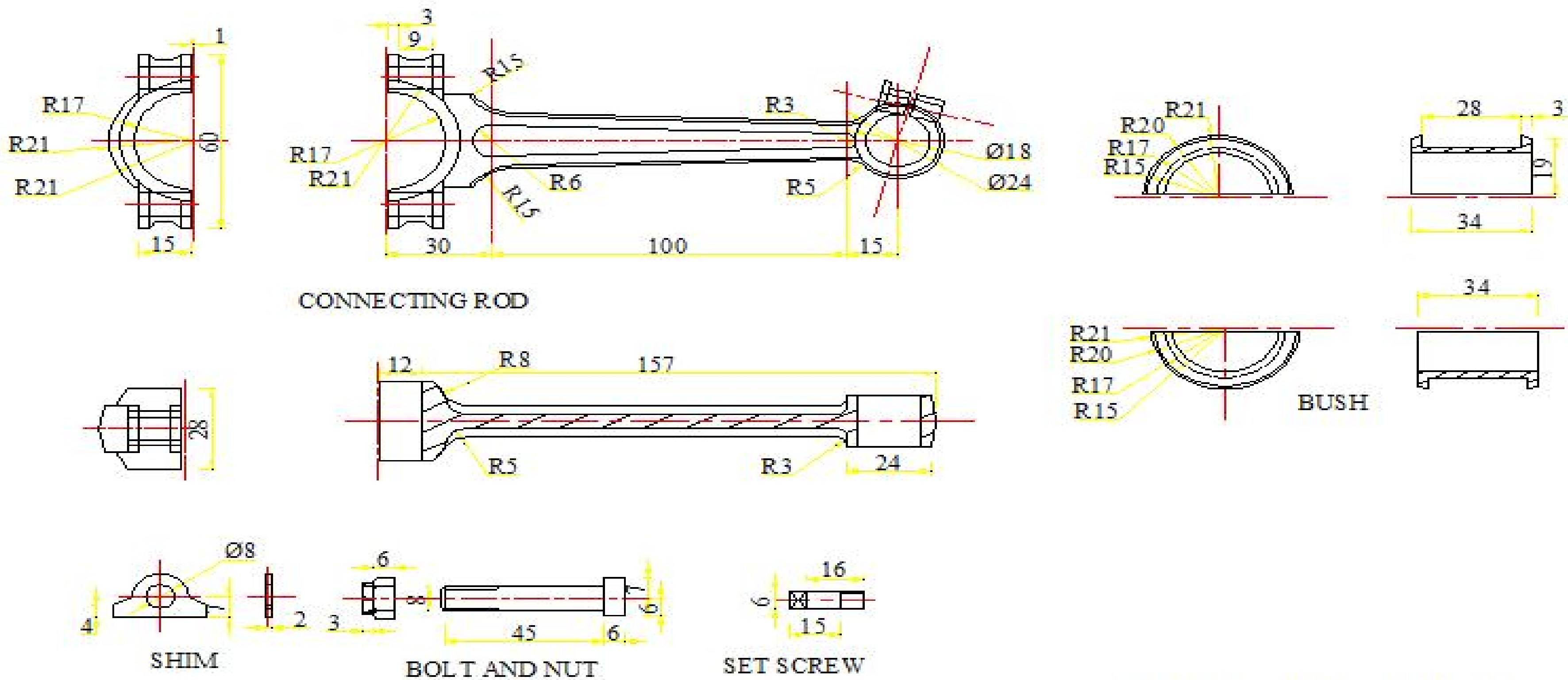
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:09

CONNECTING ROD-PART DIAGRAM

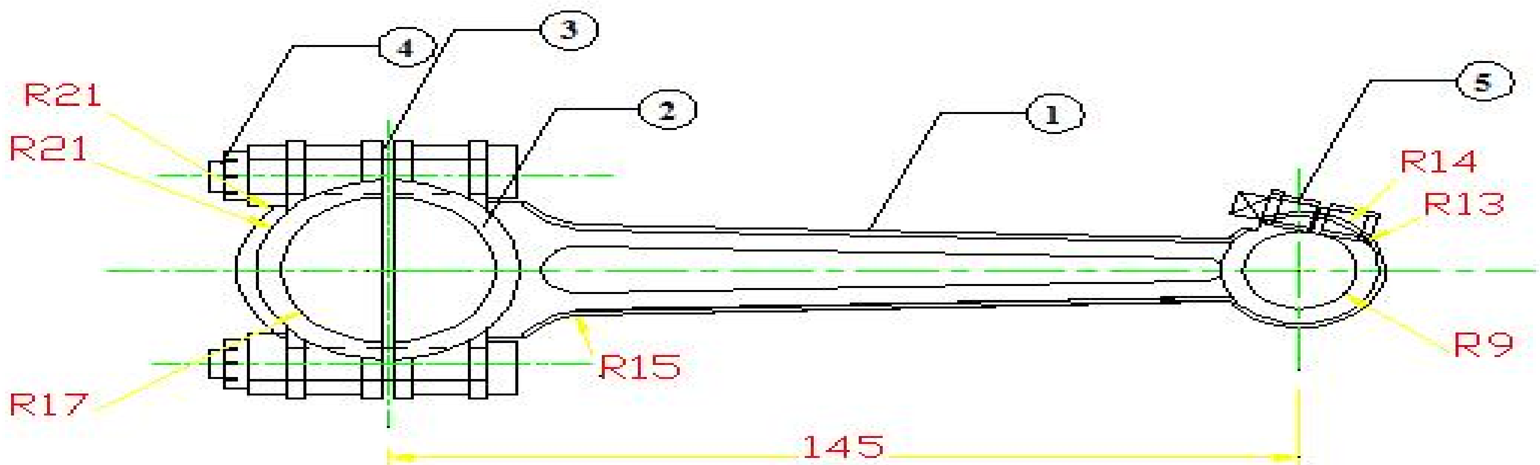


ALL DIMENSIONS ARE IN "MM"

RESULT:

Thus the assembly drawing CONNECTING ROD is drawn taken the printout.

EX.NO:09 CONNECTING ROD ASSEMBLY DRAWING



ALL DIMENSIONS ARE IN "MM"

BILL OF MATERIAL			
ITAM NO	DESCRIPTION	MATERIAL	NO OFF
1	CONNECTING ROD	CI	1
2	BUSH	WHITE METAL	1
3	SHIM	MS	2
4	BOLT AND NUT	FE 410W	2
5	SET SCREW	FE 410 W	1

Ex.No: 10

Date:

SAFETY VALVE

AIM:

To model the parts of SAFETY VALVE and assemble the parts.

SOFTWARE REQUIRED:

CATIA-V5

HARDWARES REQUIRED:

Operating system	: Windows xp
Processor	: Pentium IV
Hard disk	: 80GB
RAM	: 512 MB

COMMANDS USED:

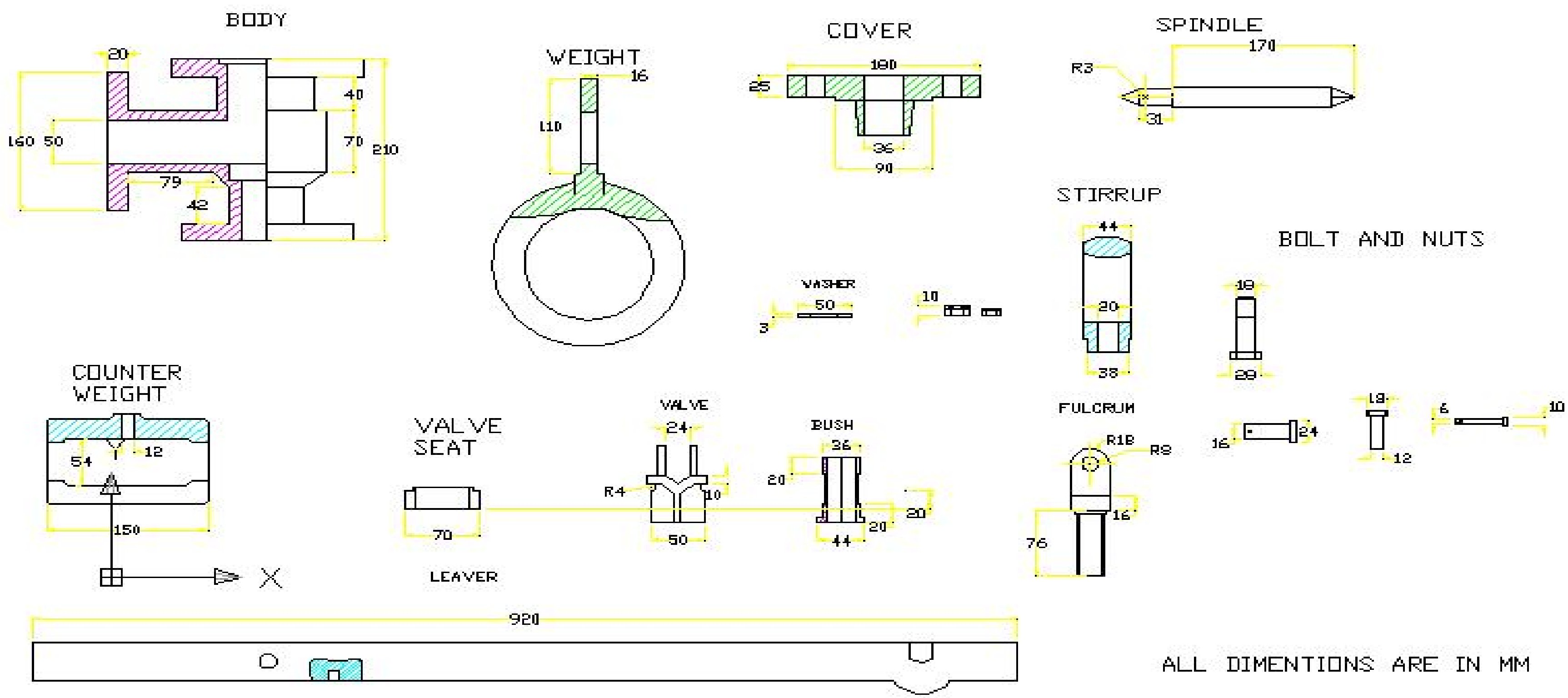
Sketch, Trim, Smart dimension, Revolve, Extrude, Extrude cut,Chamfer, Mate, Insert, Move, etc.

PROCEDURE:

- Read the part drawing thoroughly.
- Choose proper scale.
- Open new solid works document.
- Click the part drawing and model the components as per the given dimensions in part drawing by using above mentioned commands.
- Save all the parts models and mark the file name.
- Now open the assembly window and insert the parts as per the given assembly drawing.
- Save the final assembly of components.

EX.NO:10

SAFETY VALVE

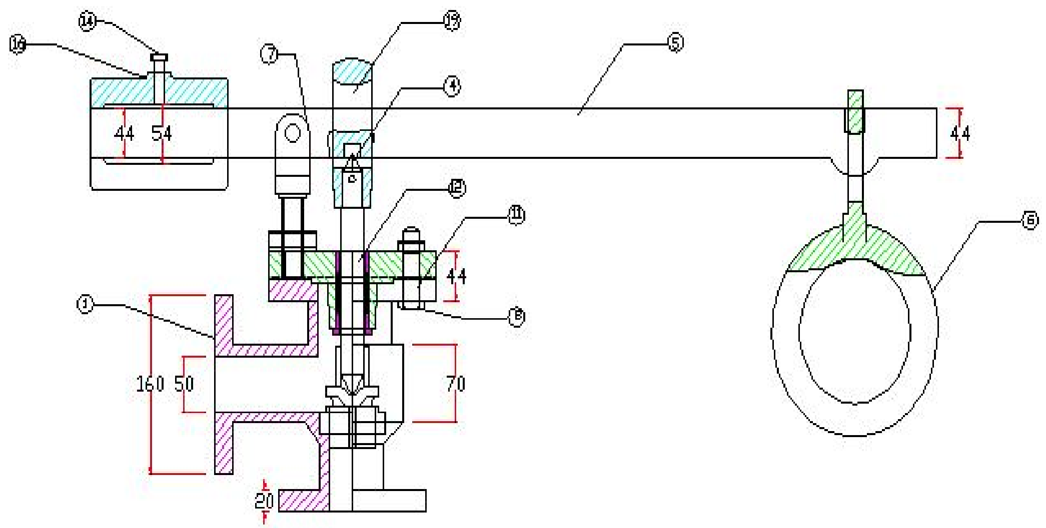


RESULT:

Thus the assembly drawing SAFETY VALVE is drawn taken the printout.

EX.NO:10

LEAVER SAFETY VALVE

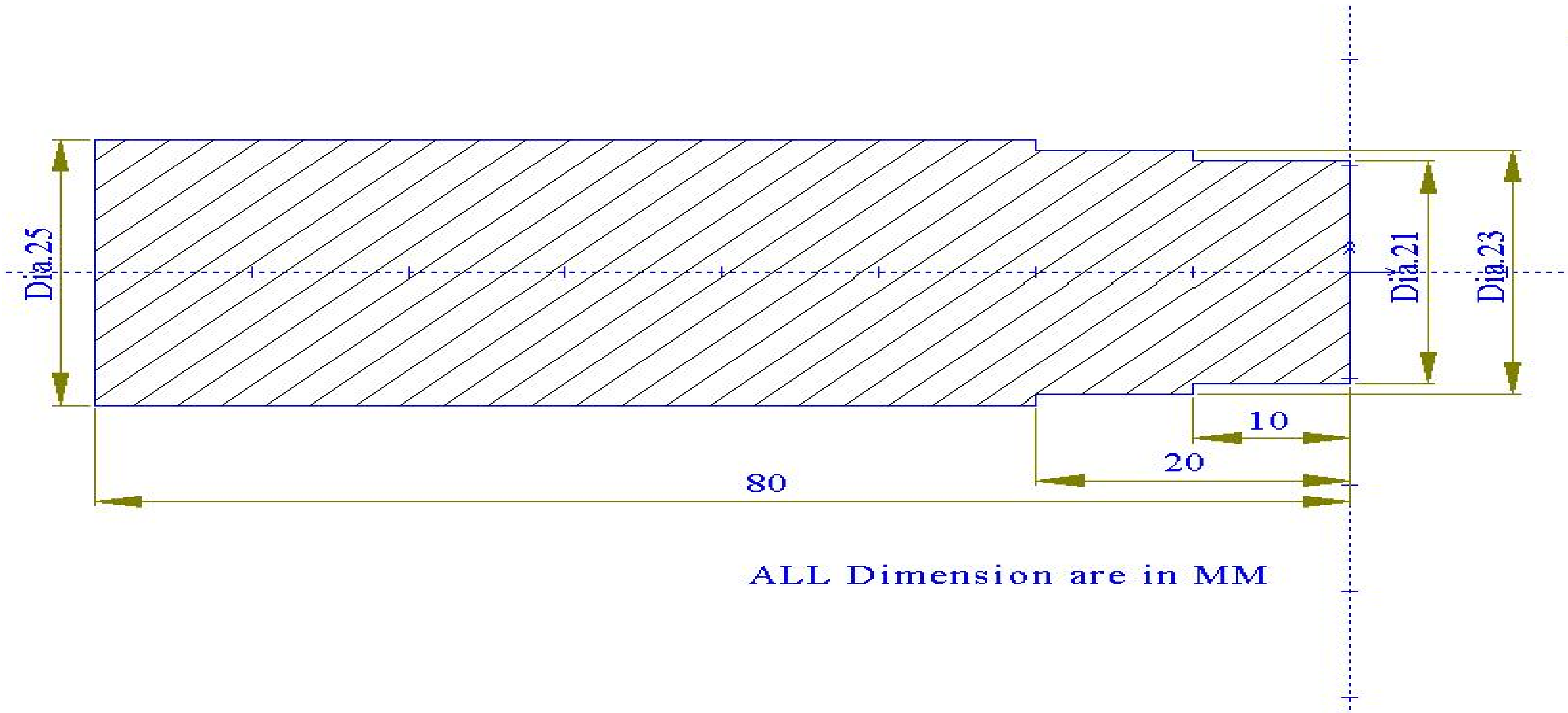


BILL OF MATERIALS

LEAVERLEAVER SAFETY VALVE			
ITEM	DESCRIPTION	MATERIAL	NO OFF
1	BODY	CAST IRON	1
2	VALVE SEAT	GUN METAL	1
3	VALVE	GUN METAL	1
4	SPINDLE	STEEL	1
5	LEAVER	MILD STEEL	1
6	WEIGHT	CAST IRON	1
7	FULCRUM	MILD STEEL	1
8	LOCK NUT	MILD STEEL	1
9	WASHER	MILD STEEL	1
10	NUT	MILD STEEL	4
11	BOLT	MILD STEEL	4
12	BUSH	GUN METAL	1
13	COVER	CAST IRON	1
14	BOLT(M12)	MILD STEEL	1
15	PIN	MILD STEEL	1
16	COUNTER WEIGHT	CAST IRON	1
17	FULCRUM PIN	STEEL	1
18	SPLIT PIN	MILD STEEL	2
19	STIRRUP	CAST IRON	1
20	BEARING INSERT	STEEL	1

ALL DIMENTIONS ARE IN MM

LINEAR INTERPOLATION



RAW MATERIAL SIZE : Dia. 25 X80 MM MATERIAL: Aluminum

EX.NO : 01

DATE :

LINEAR INTERPOLATION

AIM:

To write the CNC part program for given component drawing using G& M codes, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G01 – linear interpolation

PROGRAM:

TOOL/STANDARD,15,55,0,10,3) Tool Definition

(STOCK/80,25,0,0) Stock Definition

G90 G21 Absolute mode, Metric Units

N01 M03 S2000 Spindle Start Clockwise With Spindle

Speed 2000N02 G00 X25 Z2 Rapid Positioning Up to the

Reference Point

N03 G01 X24 Linear interpolation with 1mm cut to the diameter

N04 Z-20 F80 Up to the Length -20 with Feed Rate 80

N05 G01 X25 Linear interpolation with 1mm along

diameter

N06 G00 Z2 Rapid Movement Up to Initial Point

N07 G01 X23 Linear interpolation with 1mm cut to the diameter

N08 Z-20 F80 Up to the Length -10 with Feed 80 mm/min

N09 G01 X25 Linear interpolation with 1mm along

diameter

N10 G00 Z10 Rapid Movement Up to Initial Point

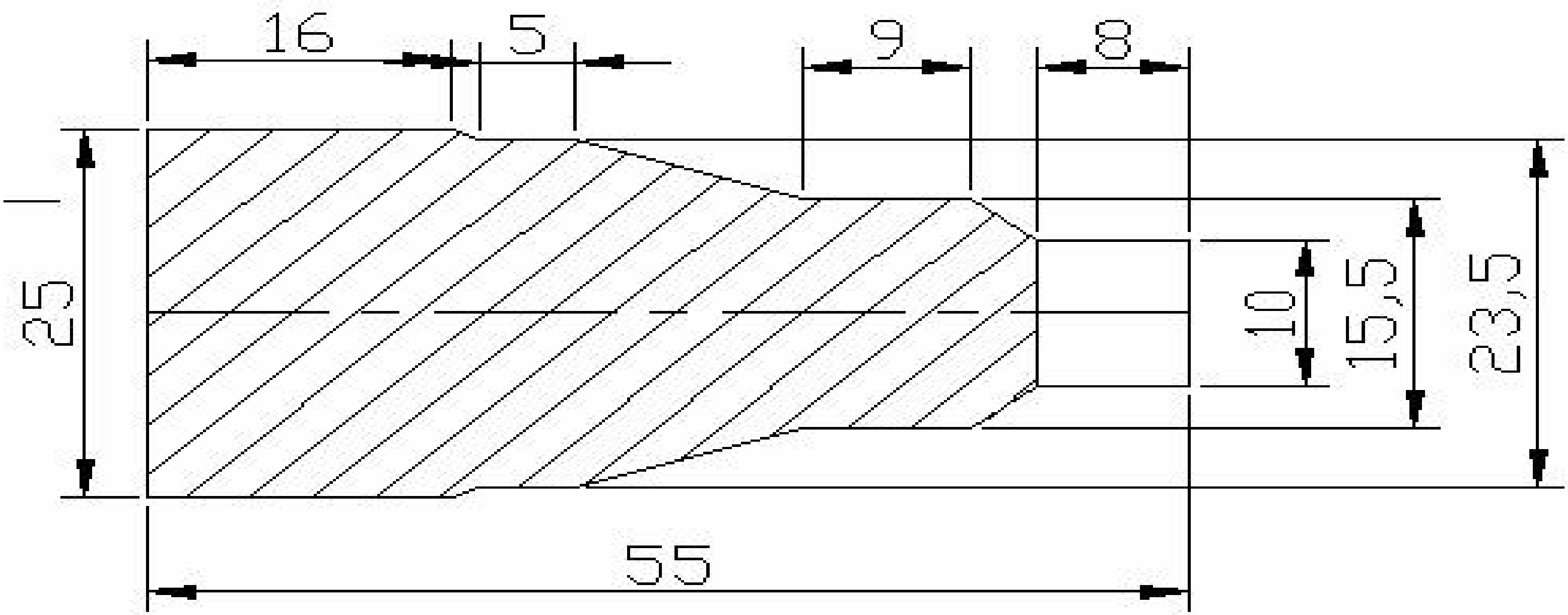
N11 G01 X22 F60 Linear interpolation with 1mm cut to the diameter

N12 G01 Z-10 F60 Up to the Length -10 with Feed 60mm/min

N13 G01 X25 F60 diameter	Linear interpolation with1mm along
N14 G00 Z2	Rapid Movement Up to Initial Point
N15 G01 X21 F60	Linear interpolation with 1mm cut to the diameter
N16 G01 Z-10 F60	Up to the Length -10 with Feed 60mm/min
N17 G01 X25 F60	Linear interpolation with 1mm along diameter
N18 G00 Z2	Rapid Movement Up to Initial Point
N19 M30	Program End & Rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC lathe.



RAW MATERIAL:- Ø25X60MM

EX.NO. : 02

DATE :

TAPER TURNING

AIM:

To write the CNC part program for given component drawing using G& M codes, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G01 – linear interpolation
G71 - Stock Removal Operation in turning

PROGRAM:

(TOOL/STANDARD, 40, 40, 0, 10, 3)	Tool Definition
(STOCK/55, 25, 0, 0)	Stock Definition
G90 G21	Absolute mode, Metric Units
N10 M03 S2000	Spindle Start Clockwise With Spindle Speed 2000
N20 G00 X30 Z5	Rapid Positioning Up to the Reference Point
N30 G71 U1.0 R0.5	Stock Removal Operation
N40 G71 P50 Q120 U0.5 W0.5 F80 S2000	Stock Removal Operation for Turning
N50 G01 X10	
N60 G01 Z-8 F80	
N70 G01 X15 Z-12	Programmed Block
N80 G01 Z-21	
N90 G01 X23 Z-33	
N100 G01 Z-38	
N110 G01 X25 Z-40	
N120 G01 Z-45	

N130 G70 P50 Q120 -----------Finishing Cycle for the previously Defined Block

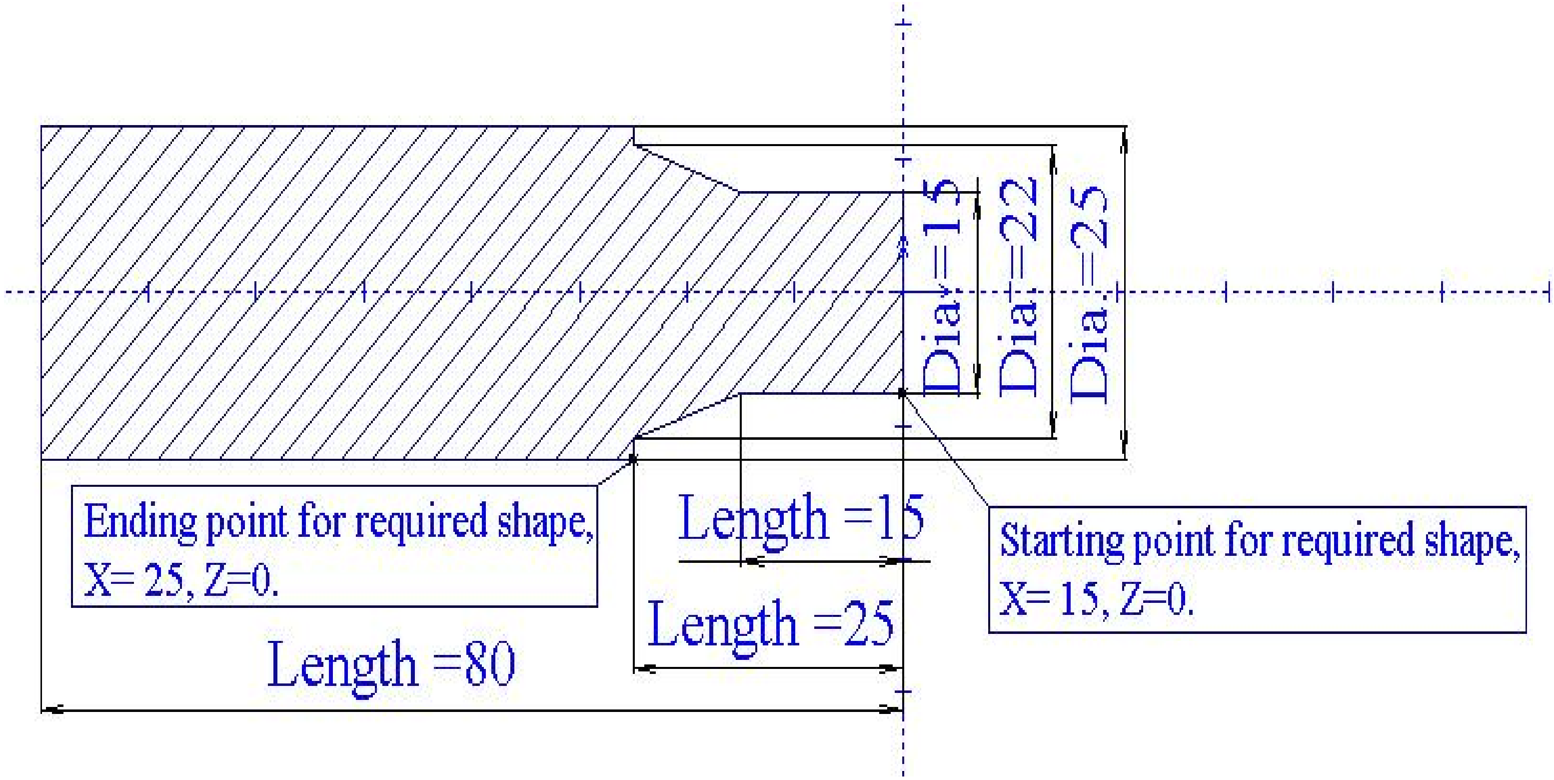
N140 G00 X30 Z5------------- Rapid Positioning Up to the Reference Point

N150 M30------------------------ Program End & Rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC lathe.

CANNED CYCLE - PATTERN REPEATING CYCLE



RAW MATERIAL SIZE: Dia. 25 X80 MM MATERIAL: Aluminium

EX.NO. : 03
DATE :

CANNED CYCLE - PATTERN REPEATING CYCLE

AIM:

To write the CNC part program for given component drawing using Canned Cycle code, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G73 - Pattern Repeating Cycle
G70 - Finishing Cycle

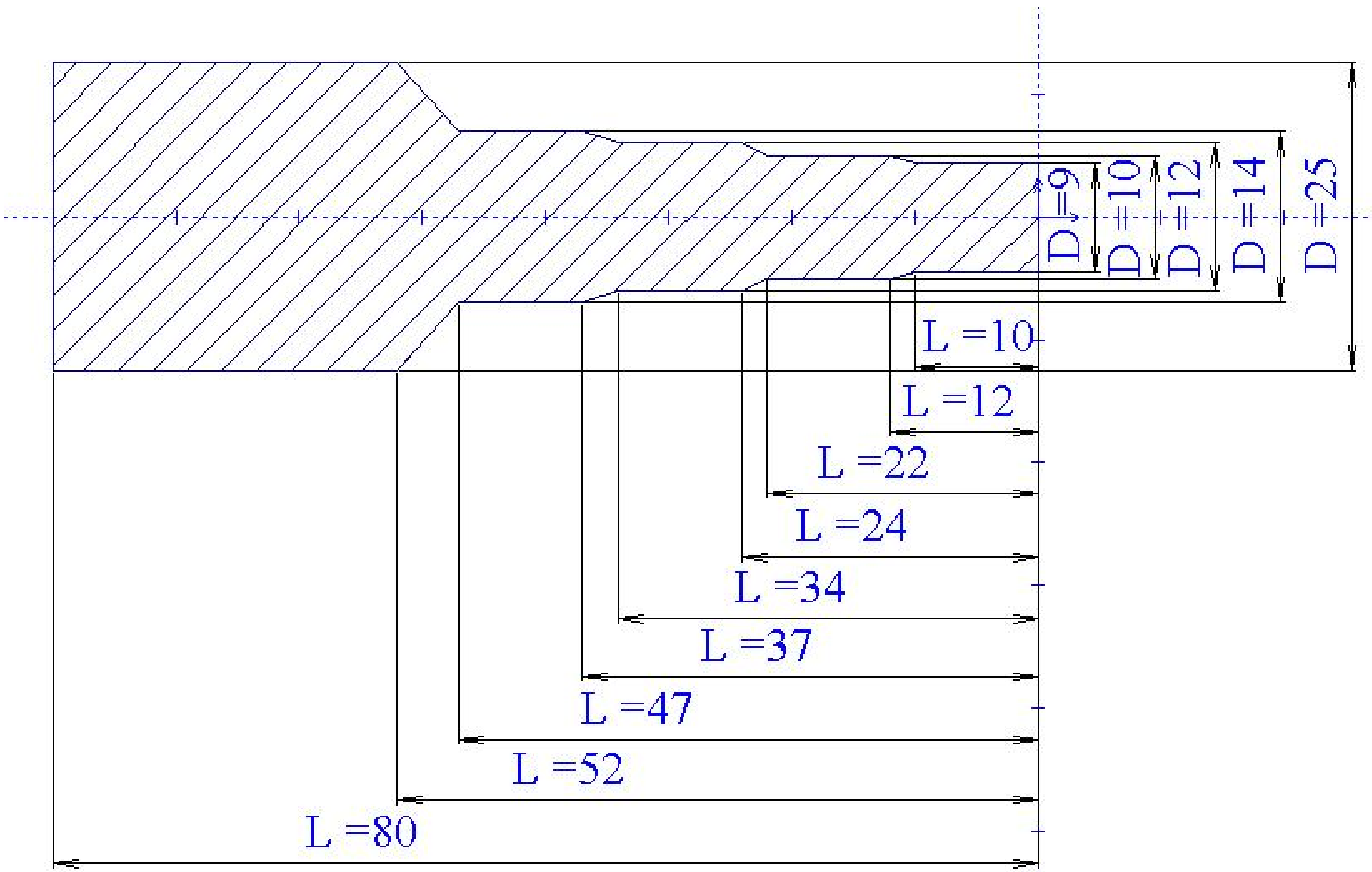
PROGRAM:

(TOOL/STANDARD,15,45,0,10,3)	Tool Definition
(STOCK/80,25,0,0)	Stock Definition
G90 G21	Absolute mode, Metric Units
N00 M03 S1800	Spindle start clockwise with spindle speed 1800
N01 G00 X25 Z2	Rapid positioning up to reference point
N02 G73 U5.0 W0.5 R10	Pattern Repeating Cycle
N00 G73 P04 Q08 U0.5 W0.5 F60	Pattern Removal Operation
N04 G00 X15.000	Starting block no. of the program for the required shape
N05 G01 Z-15.000	
N06 G01 X22.000 Z-25.000	
N07 G01 X25	
N08 G00 Z0.000	Final block no. of the program for the required shape
N11 G70 P04 Q08	Finishing Cycle for the previously Defined Block
N13 G00 X28 Z2	Rapid Positioning Up to the Reference Point
N14 M30	Program End & Rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC lathe.

THREAD CUTTING



EX.NO.04

DATE:

THREAD CUTTING

AIM:

To write the CNC part program for given component drawing using Canned Cycle code, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 - Absolute mode
G00 - Rapid travel
G92 - Thread Cutting Canned Cycle
G71 - Stock Removal Operation
G70 - Finishing Cycle

PROGRAM:

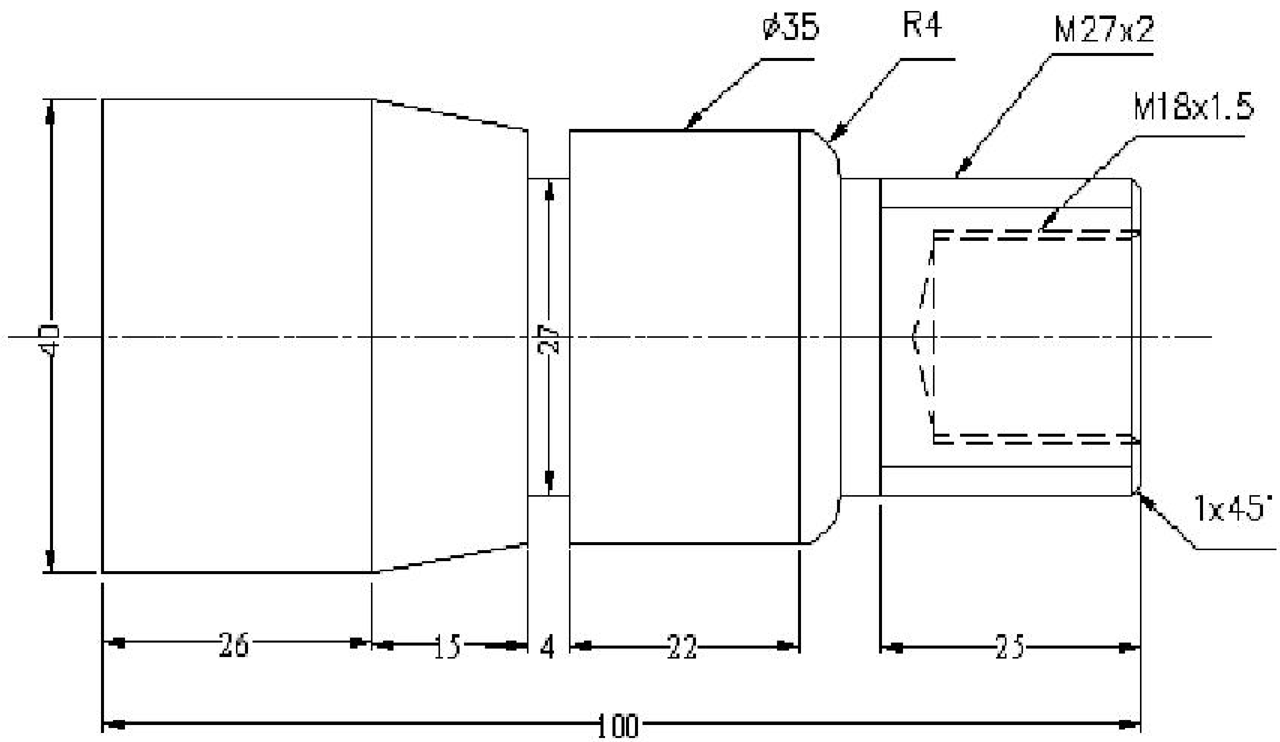
```
 (STOCK/80, 25, 0, 0)                         Stock Definition
(TOOL/STANDARD, 15, 45, 0, 10, 3)              Tool Definition
G90 G21                                       Absolute mode, Metric
                                               Units
M03 S2500                                      Spindle start clockwise
                                               with spindle speed
2500
N00 G00 X25 Z0
N01 G71 U0.5 R0.5                             Stock Removal
Operation
N02 G71 P03 Q11 U0.2 W0 F80                   Stock Removal
Operation
                                               for Turning
N03 G00 X9 Z0                                 Programmed Block
N04 G01 X9  Z -10
N05 G01 X10  Z-12
N06 G01 X10  Z-22
N07 G01 X12  Z-24
N08 G01 X12  Z-34
N09 G01 X14 Z-37
N10 G01 X14 Z-47
N11 G01 X25 Z-52
N12 G70 P03 Q11                               Finishing Cycle for the
```

	previously Defined Block
N13 G00 X26 Z40	
(TOOL/THREAD, 45, 25, 5, 90)	Tool Definition
N14 T3	Tool Change
N15 G00 X9 Z0.5	
N16 G92 X8.9 Z-10 F1	Thread Cutting Canned
Cycle	
N17 G92 X8.7 Z-10 F1	
N18 G92 X8.6 Z-10 F1	
N19 G92 X8.5 Z-10 F1	
N20 G92 X8.4 Z-10 F1	
N21 G92 X8.3 Z-10 F1	
N22 G92 X8.2 Z-10 F1	
N23 G92 X8.1 Z-10 F1	
N24 G92 X8.0 Z-10 F1	
N25 G92 X7.9 Z-10 F1	
N26 G92 X7.8 Z-10 F1	
N27 G00 X10 Z0	Rapid Positioning Up to the Reference Point
N28 M30	Program End & Rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC lathe.

PECK DRILLING - GROOVING



ALL DIMENSIONS ARE IN MM

EX.NO : 05

DATE :

PECK DRILLING - GROOVING

AIM:

To write the CNC part program for given component drawing using Canned Cycle code, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G74 - Peck drilling cycle
G75- Grooving Cycle

PROGRAM:

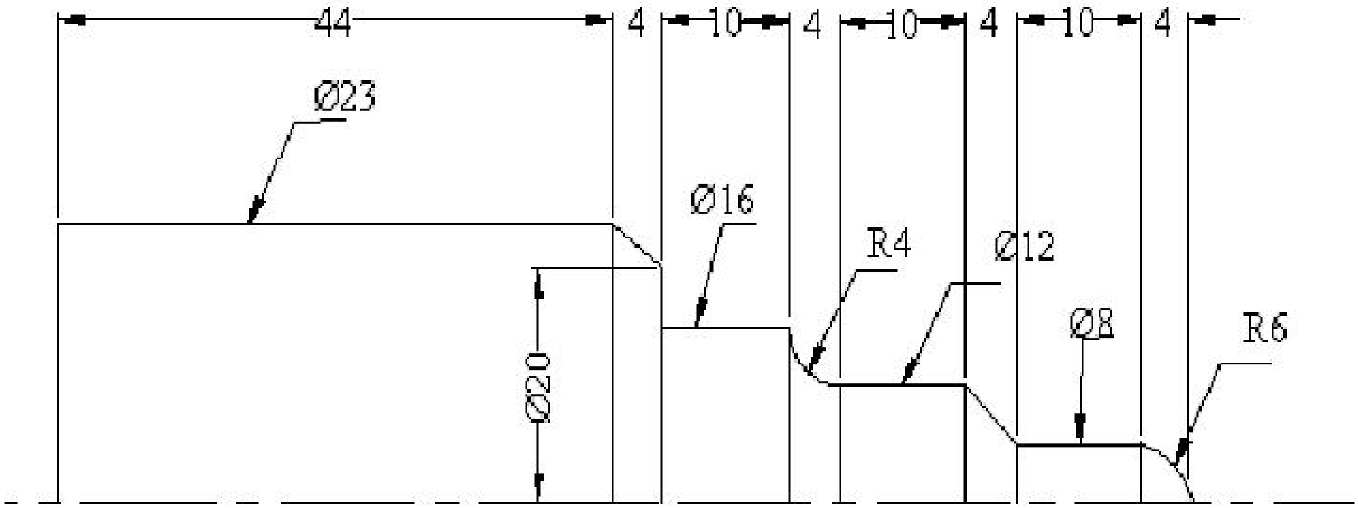
```
O 0014;
[BILLET X22 Z70;
G21 G97 G98;
G28 U0 W0;
M06 T01;
M03 S1000;
G00 X22 Z1;
G01 Z0 F50;
G71 U0.5 R1;
G71 P100 Q200 U0.2 W0.2;
N100 G01 X18;
Z-30;
N200 X22 Z-35;
M05;
G28 U0 W0;
M06 T02;
M03 S1200;
G00 X22 Z1;
G70 P100 Q200;
G28 U0 W0;
M05;
G28 U0 W0;
```

```
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 is simulated successfully in CAM simulation software and executed in CNC lathe.

CIRCULAR INTERPOLATION



ALL DIMENSION ARE IN MM

EX.NO : 06

DATE :

CIRCULAR INTERPOLATION

AIM:

To write the CNC part program for given component drawing using G& M codes, execute the program in CNC simulation software and CNC lathe machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G01 – linear interpolation
G02 – circular interpolation (Clockwise)
G03 - circular interpolation (Anti - Clockwise)

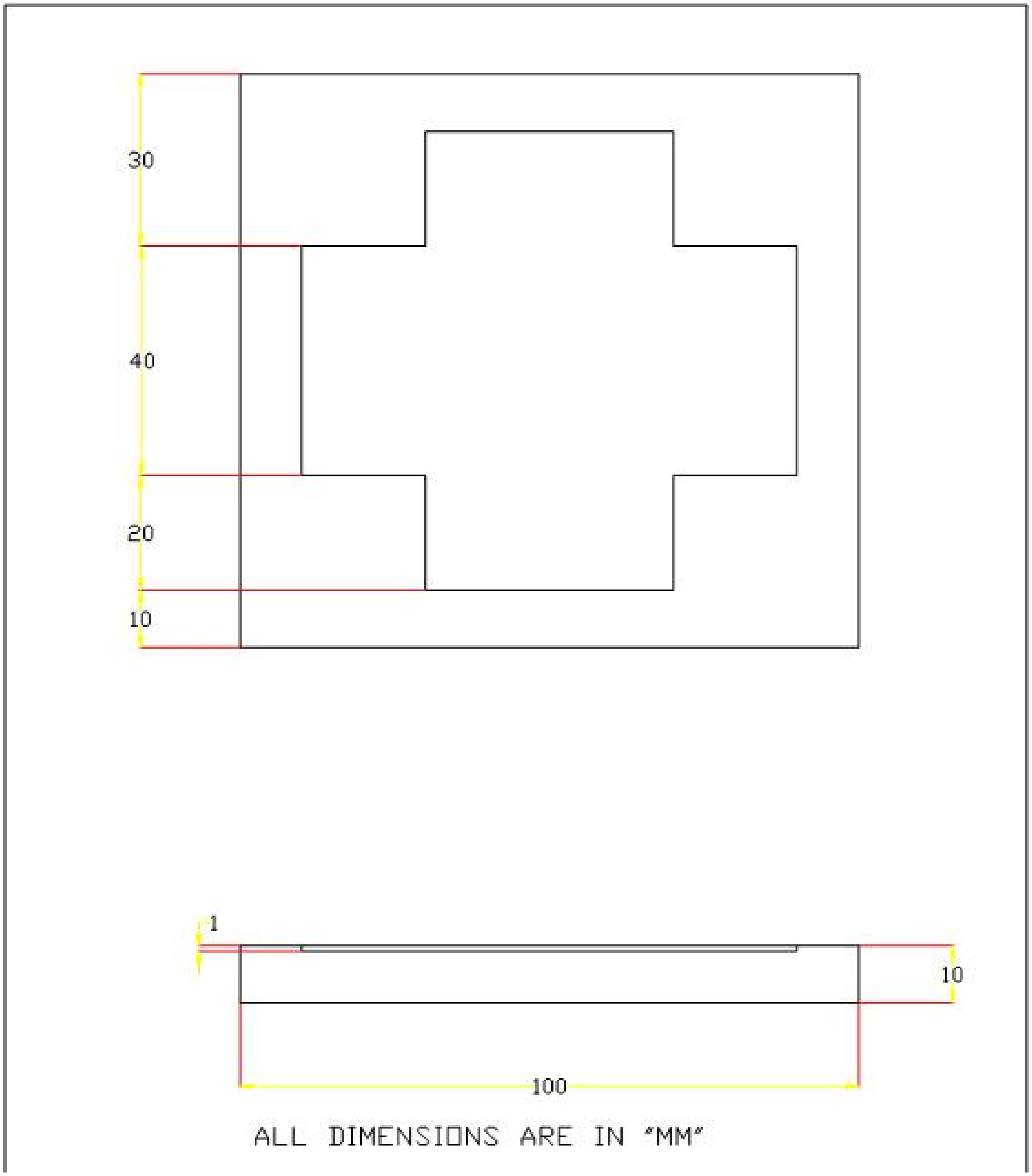
PROGRAM:

```
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 is simulated successfully in CAM simulation software and executed in CNC lathe.

LINEAR INTERPOLATION



EX.NO : 07

DATE :

LINEAR INTERPOLATION - ABSOLUTE AND INCREMENTAL MODE

AIM:

To write the CNC part program for given contour drawing using G& M codes, execute the program in CNC simulation software and CNC milling machine

CODES:

G90 – Absolute mode

G91- Incremental Program Mode

G00 – rapid travel

G01 – linear interpolation

PROGRAM:

ABSOLUTE METHOD

(TOOL/MILL, 6, 0, 50, 0)	Flat End Mill Cutter of 6 mm Dia.
(STOCK/BLOCK, 100, 100 10, 0, 0, 10)	
N00 G90 M03 S2000	Spindle start clockwise with speed 2000
N01 G90 G00 X0	Absolute program mode, Rapid positioning
N02 Y0	Rapid positioning
N03 Z2	Rapid positioning up to 2mm along Z axis
N04 G00 X30 Y10 Z2	Rapid positioning
N05 G01 Z-1 F50	linear interpolation
N06 G01 X70 Y10	
N08 G01 X70 Y30	
N09 G01 X90 Y30	
N10 G01 X90 Y70	
N11 G01 X70 Y70	

N12 G01 X70 Y90	
N13 G01 X30 Y90	
N14 G01 X30 Y70	
N15 G01 X10 Y70	
N16 G01 X10 Y30	
N17 G01 X30 Y30	
N18 G01 X30 Y10	
N16 G00 Z5	Rapid positioning up to 5mm along z axis
N17 X00 Y00 Z10	
N18 M30	program end and rewind

INCREMENTAL MODE

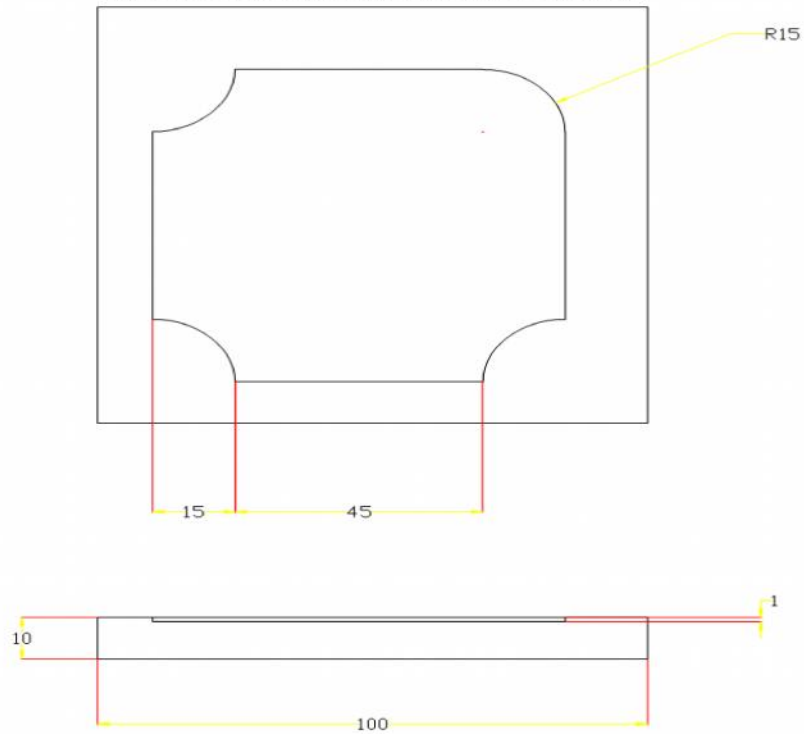
(TOOL/MILL, 6, 0, 50, 0)	Cutter Flat End Mill Cutter of 6 mm
(STOCK/BLOCK,100,100,10,0,0,10)	
(STOCK/BLOCK, 100, 100 10, 0, 0, 10)	
N00 M03 S2000	Spindle start cw with speed 2000
N01 G90 G00 X0	Absolute program mode, Rapid positioning
N02 Y0	Rapid positioning
N03 Z2	Rapid positioning up to 2mm along Z axis
N04 G00 X30 Y10 Z2	Rapid positioning
N05 G01 Z-1 F50	
G91	Incremental program mode
N06 G01 X40 Y0	
N08 G01 X00 Y20	
N09 G01 X20 Y0	
N10 G01 X0 Y40	
N11 G01 X-20 Y0	
N12 G01 X0 Y20	
N13 G01 X-40 Y0	
N14 G01 X0 Y-20	
N15 G01 X-20 Y0	

N16 G01 X0 Y-40	
N17 G01 X20 Y0	
N18 G01 X0 Y-20	
N19 G90	Absolute program mode
N20 G00 Z5	Rapid positioning up to 5mm along z axis
N21 X00 Y00 Z10	
N22 M30	program end and rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC milling machine.

CIRCULAR INTERPOLATION



EX.NO : 08
DATE :

CIRCULAR INTERPOLATION

AIM:

To write the CNC part program for given contour drawing using G& M codes, execute the program in CNC simulation software and CNC milling machine

CODES:

G90 – Absolute mode
G00 – rapid travel
G01 – linear interpolation
G02 - clockwise circular interpolation
G03 - anticlockwise circular interpolation

PROGRAM:

(TOOL/MILL, 4, 0, 50, 0) Cutter Flat End Mill Cutter of 4 mm Dia.

(STOCK/BLOCK, 100, 100, 10, 0, 0, 10)

N00 G90 M03 S2000 Spindle start clockwise with speed 2000

N01 G90 G00 X0 Absolute program mode, Rapid positioning

N02 Y0 Rapid positioning

N03 Z2 Rapid positioning up to 2mm along Z axis

N04 G00 X25 Y10 Z2 Rapid positioning

N05 G01 Z-1 F50 linear interpolation

N06 G03 X10 Y25 R15 F50 Anticlockwise circular interpolation

N08 G01 X10 Y70 linear interpolation

N09 G03 X25 Y85 R15 F50 Anticlockwise circular interpolation

N10 G01 X70 Y85 linear interpolation

N11 G02 X85 Y70 R15 F50 Clockwise circular interpolation

N12 G01 X85 Y25 linear interpolation

N13 G03 X70 Y10 R15	Anticlockwise circular interpolation
N14 G01 X25 Y10	linear interpolation
N16 G00 Z5	Rapid positioning up to 5mm along z axis
N17 X00 Y00 Z10	
N18 M30	program end and rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC milling machine.

EX.NO : 09

DATE :

PECK DRILLING CYCLE WITH SUBROUTINE

AIM:

To write the CNC part program for given part drawing using G& M codes, execute the program in CNC simulation software and CNC milling machine

CODES:

G90 – Absolute mode

G91- Incremental Program Mode

G00 – rapid travel

G01 – linear interpolation

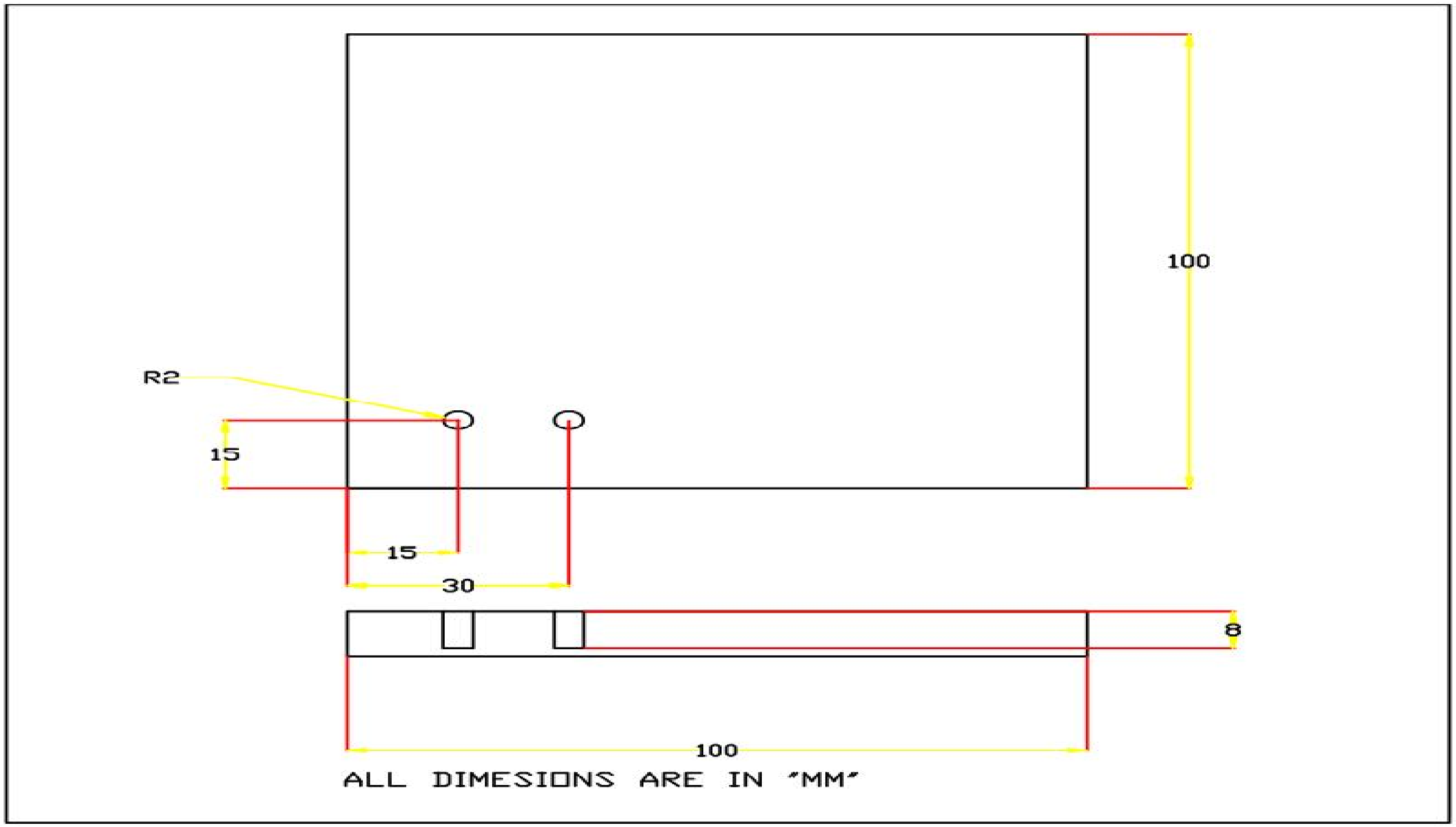
M98 - Subroutine call

M99 - Subroutine end

PROGRAM:

(TOOL/DRILL,6,120,50)		Cutter Flat End Mill Cutter of 4 mm
(STOCK/BLOCK,100,100,10,0,0,10)		
N10	M03 S2000	Spindle start cw with speed 2000
N20	G00 X00	Rapid positioning
N30	G00 Y00	Rapid positioning
N40	G00 Z00	Rapid positioning
N50	G00 Z2	
N60	M98 P27 L5	Subroutine call 5 times
N90	G00 Z2	
N100	G90	absolute program mode
N110	G00 X10 Y90	
N120	M98 P28 L5	Subroutine call 5 times
N130	G00 Z5	
N140	M30	program end and rewind

PECK DRILLING



Subroutine 1

O27

N1 G91 incremental mode

N2 G83 X15 Y15 Z-8 R2 Q2 F80. Peck drilling cycle

N3 M99 Subroutine end

Subroutine 2

O28

NI G91 incremental mode

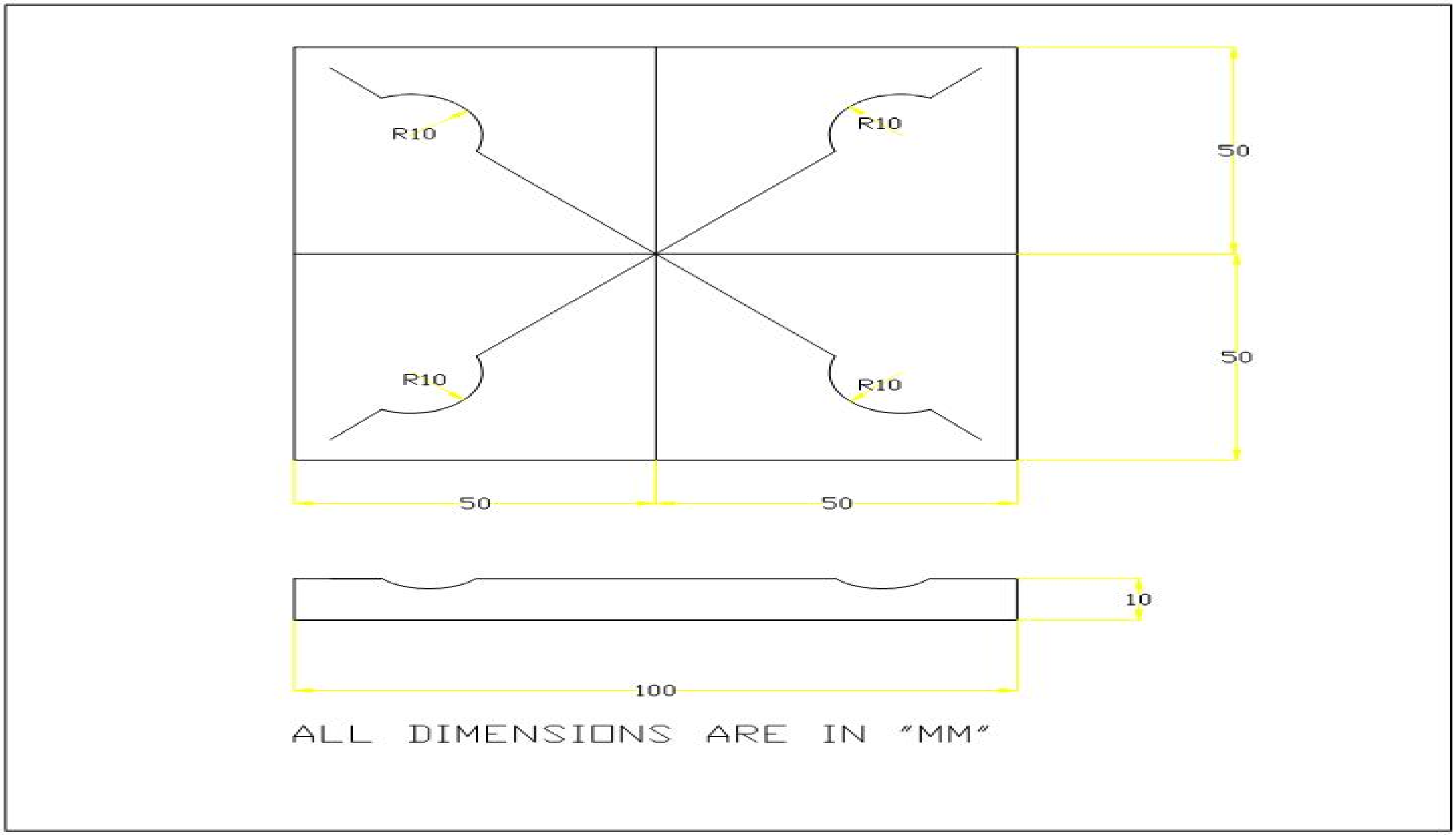
N2 G83 X15 Y00 Z-8 R2 Q2 F80 Peck drilling cycle

N3 M99 Subroutine end

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC milling machine.

MIRROR IMAGE



EX.NO : 10

DATE :

INTERPOLATION WITH MIRROR IMAGE

AIM:

To write the CNC part program for given part drawing using G& M codes, execute the program in CNC simulation software and CNC milling machine

CODES:

M21- Mirror Image along X axis

M22- Mirror Image along Y axis

M23 -Mirror Image cancel

M98 - Subroutine call

M99 - Subroutine end

PROGRAM:

```
O2123
(STOCK/BLOCK,100,100,10,50,50,10)
(TOOL/MILL,3,0,50,0)
N10   G01 Z0 F100
N20   M98 P123 L1           Subroutine call 1times
N30   M21                   Mirror Image along X axis
N40   M98 P123 L1           Subroutine call 1times
N50   M22                   Mirror Image along Y axis
N60   M98 P123,L1           Subroutine call 1 times
N70   M23
N80   M22                   Mirror Image along Y axis
N90   M98 P123    L1        Subroutine call 1 times
N100  M30                   program end and rewind
```

Subroutine

```
O123
N1 G01 X5 Y5
N2 G01 Z-0.5
N3 G01 X15 Y15
N4 G02 X35 Y35 R10 F100
N6 G01 X45 Y45
N7 G00 Z4
```

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC milling machine.

EX.NO : 11

DATE :

SQUARE POCKETING WITH CUTTER RADIUS COMPENSATION

AIM:

Write the CNC part program to make square pocketing using G& M codes with cutter radius compensation, execute the program in CNC simulation software and CNC milling machine

CODES:

G41-Cutter Radius Compensation Left

G42-Cutter Radius Compensation Right

PROGRAM:

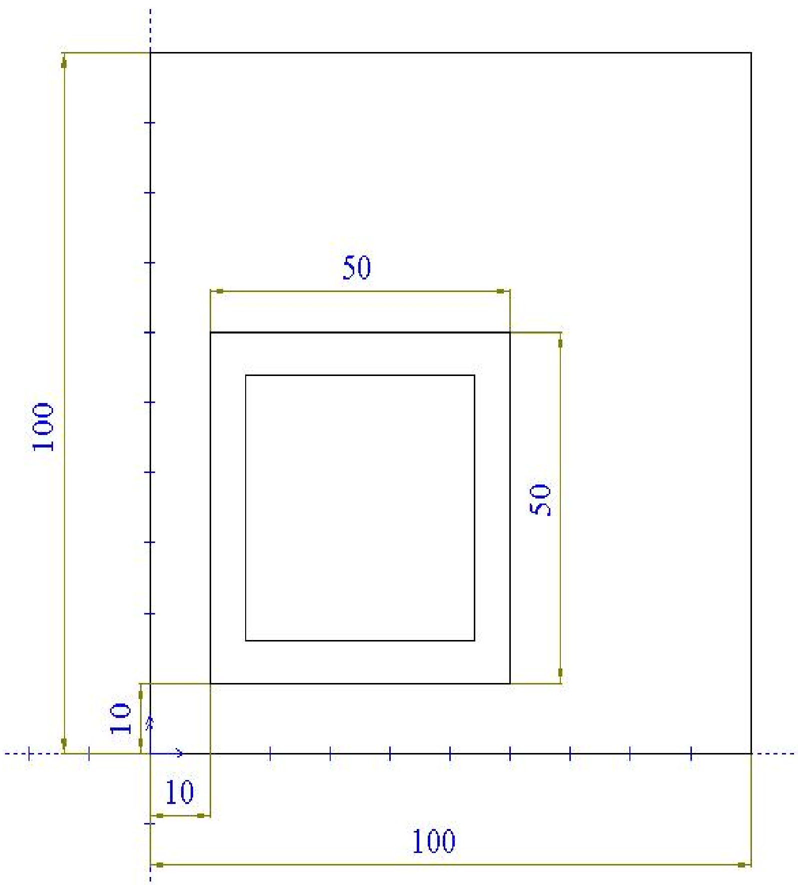
CUTTER RADIUS COMPENSATION LEFT (G41)
(MACHINING FROM INSIDE OF CONTOUR)

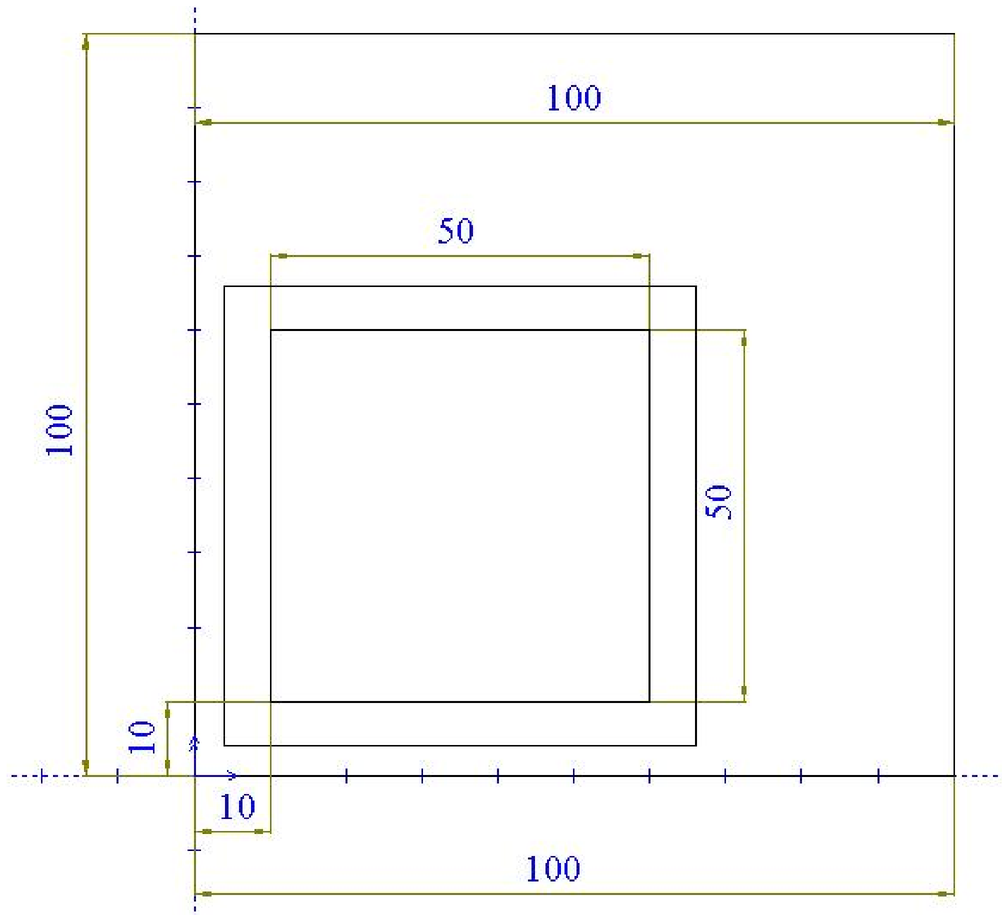
(STOCK/BLOCK,100,100,10,0,0,10)

(TOOL/MILL,6,0,50,0)

N10 M03 S2000	Spindle start clockwise with speed 2000rpm
N20 G00 X0 Y0 Z1	Rapid positioning up to specified point
N30 G00 X13	Rapid positioning up to specified point
N40 G00 Y40	Rapid positioning up to specified point
N50 G01 Z-1 F20	
N60 G41 X10	Cutter radius compensation start
N70 G01 Y10	
N80 G01 X60	
N90 G01 Y60	
N100 G01 X10	
N110 G01 Y30	
N120 G40	Cutter radius compensation Cancel)
N130 G00 Z5	
N140 M30	program end & rewind

CUTTER RADIUS COMPENSATION LEFT (G41)





CUTTER RADIUS COMPENSATION RIGHT -G42
(MACHINING FROM OUTSIDE OF CONTOUR)

(STOCK/BLOCK,100,100,10,0,0,10)	
(TOOL/MILL,6,0,50,0)	
N10 M03 S2000	
N20 G00 X0 Y0 Z1	Rapid positioning up to specified point
N30 G00 X7	Rapid positioning up to specified point
N40 G00 Y40	Rapid positioning up to specified point
N50 G01 Z-1 F20	Depth of cut
N60 G42 X10	Cutter radius compensation start
N70 G01 Y10	
N80 G01 X60	
N90 G01 Y60	
N100 G01 X10	
N110 G01 Y30	
N120 G40	Cutter radius compensation Cancel
N130 G00 Z5	
N140 M30	program end & rewind

RESULT:

Thus the part program is simulated successfully in CAM simulation software and executed in CNC milling machine.