DEPARTMENT OF MECHANICAL ENGINEERING

COMPUTER INTEGRATED MANUFACUTRING LAB (PG)

LAB MANUAL

Prepared by

P.Kumaran
ASSISTANT PROFESSOR – II / MECH

CERTIFICATE

Name of the lab : 40221C81- **CIM LAB (PG)**

Prepared by : Mr. P.KUMARAN,

Assistant Professor (Gr-II),

Department of Mechanical Engineering.

Approved By : Dr. M.PRABHAHAR,

Head of the Department

Department of Mechanical Engineering.

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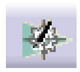
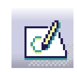
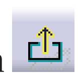
1. INTRODUCTION

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systèmes product lifecycle management software suite.

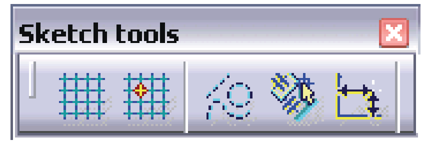
THE SKETCHER WORKBENCH 

The Sketcher workbench is a set of tools that helps you create and constrain 2D geometries. Features (pads, pockets, shafts, etc...) may then be created solids or modifications to solids using these 2D profiles. You can access the Sketcher workbench in various ways. Two simple ways are by using the top pull down menu (Start – Mechanical Design – Sketcher), or by selecting the Sketcher icon. 
When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or after you select the Sketcher icon.

To exit the sketcher, select the Exit Workbench  icon.
The Sketcher workbench contains the following standard workbench specific toolbars.

• ***<u>Profile Toolbar</u>***: The commands located in this toolbar allow you to create simple geometries (rectangle, circle, line, etc...) and more complex geometries (profile, spline, etc...).

• ***<u>Operation Toolbar</u>***: Once a profile has been created, it can be modified using commands such as trim, mirror, chamfer, and other commands located in the Operation toolbar.



•***Constraint Toolbar:*** Profiles may be constrained with dimensional (distances, angles, etc...) or geometrical (tangent, parallel, etc...) constraints using the commands located in the Constraint toolbar.

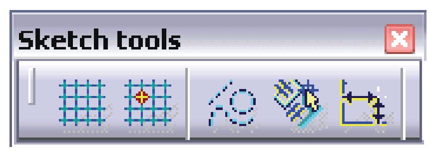
• ***Sketch Tools Toolbar***: The commands in this toolbar allow you to work in different modes which make sketching easier.

• ***User Selection Filter Toolbar***: Allows you to activate different selection filters.

• ***Visualization Toolbar***: Allows you to, among other things to cut the part by the sketch plane and choose lighting effects and other factors that influence how the part is visualized.

• ***Tools Toolbar:*** Allows you to, among others other things, to analyze a sketch for problems, and create a datum.

THE SKETCH TOOLS TOOLBAR

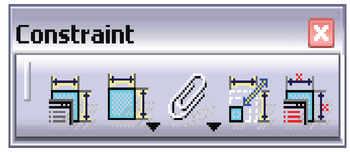


The Sketch tools toolbar contains icons that activate and deactivate different work modes. These work modes assist you in drawing 2D profiles. Reading from left to right, the toolbar contains the following work modes; (Each work mode is active if the icon is orange and inactive if it is blue.)

• ***Grid***: This command turns the sketcher grid on and off.

• ***Snap to Point***: If active, your cursor will snap to the intersections of the grid lines.

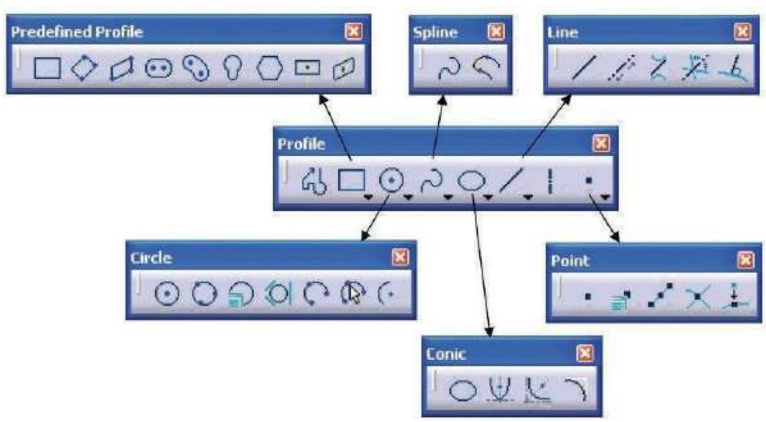
• ***Construction / Standard Elements***: You can draw two different types of elements in CATIA a standard element and a construction element. A standard element (solid line type) will be created when the icon is inactive (blue). It will be used to create a feature in the Part Design workbench. A construction element (dashed line type) will be created when the icon is active (orange). They are used to help construct your sketch, but will not be used to create features.



- ***Geometric Constraints***: When active, geometric constraints will automatically be applied such as tangencies, coincidences, parallelisms, etc...
- ***Dimensional Constraints:*** When active, dimensional constraints will automatically be applied when corners (fillets) or chamfers are created, or when quantities are entered in the value field. The value field is a place where dimensions such as line length and angle are manually entered.

PROFILE TOOLBAR

The Profile toolbar contains 2D geometry commands. These geometries range from the very simple (point, rectangle, etc...) to the very complex (splines, conics, etc...). The Profile toolbar contains many sub-toolbars. Most of these sub toolbars contain different options for creating the same geometry. For example, you can create a simple line, a line defined by two tangent points, or a line that is perpendicular to a surface. Reading from left to right, the Profile toolbar contain the following commands.



• ***Profile***: This command allows you to create a continuous set of lines and arcs connected together.

• ***Rectangle / Predefined Profile Toolbar***: The default top command is rectangle. Stacked underneath are several different commands used to create predefined geometries.

• ***Circle / Circle Toolbar***: The default top command is circle. Stacked underneath are several different options for creating circles and arcs.

• ***Spline / Spline Toolbar***: The default top command is spline which is a curved line created by connecting a series of points.

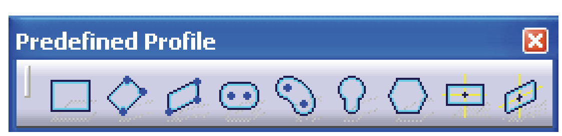
• ***Ellipse / Conic Toolbar***: The default top command is ellipse. Stacked underneath are commands to create different conic shapes such as a hyperbola.

• ***Line / Line Toolbar***: The default top command is line. Stacked underneath are several different options for creating lines.

• ***Axis***: An axis is used in conjunction with commands like mirror and shaft (revolve). It defines symmetry. It is a construction element so it does not become a physical part of your feature.

• ***Point / Point Toolbar***: The default top command is point. Stacked underneath are several different options for creating points.

PREDEFINED PROFILE TOOLBAR: Predefined profiles are frequently used geometries. CATIA makes these profiles available for easy creation which speeds up drawing time. Reading from left to right, the Predefined Profile toolbar contains the following commands.



• ***Rectangle***: The rectangle is defined by two corner points. The sides of the rectangle are always horizontal and vertical.

• ***Oriented Rectangle***: The oriented rectangle is defined by three corner points. This allows you to create a rectangle whose sides are at an angle to the horizontal.

• ***Parallelogram***: The parallelogram is defined by three corner points.

• ***Elongated Hole***: The elongated hole or slot is defined by two points and a radius.

• ***Cylindrical Elongated Hole***: The cylindrical elongated hole is defined by a cylindrical radius, two point and a hole radius.

• ***Keyhole Profile:*** The keyhole profile is defined by two center points and two radii.

• ***Hexagon***: The hexagon is defined by a center point and the radius of an inscribed circle.

• ***Centered Rectangle***: The centered rectangle is defined by a center point and a corner point.

• ***Centered Parallelogram***: The centered parallelogram is defined by a center point (defined by two intersecting lines) and a corner point.

CIRCLE TOOLBAR: The Circle toolbar contains several different ways of creating circles and arcs. Reading from left to right, the Circle toolbar contains the following commands.

• ***Circle***: A circle is defined by a center point and a radius.

• ***Three Point Circle***: The three point circle command allows you to create a circle using three circumferential points.

• ***Circle Using Coordinates***: The circle using coordinates command allows you to create a circle by entering the coordinates for the center point and radius in a Circle Definition window.

• ***Tri-Tangent Circle***: The tri-tangent circle command allows you to create a circle whose circumference is tangent to three chosen lines.

Three Point Arc: The three point arc command allows you to create an arc defined by three circumferential points.

• ***Three Point Arc Starting With Limits***: The three point arc starting with limits allows you to create an arc using a start, end, and midpoint.

• ***Arc***: The arc command allows you to create an arc defined by a center point, and a circumferential start and end point.

SPLINE TOOLBAR

Reading from left to right, the Spline toolbar contains the following commands.

• ***Spline***: A spline is a curved profile defined by three or more points. The tangency and curvature radius at each point may be specified.

• ***Connect***: The connect command connects two points or profiles with a spline.

CONIC TOOLBAR

Reading from left to right, the Conic toolbar contains the following commands.

• ***Ellipse***: The ellipse is defined by center point and a major and minor axis points.

• ***Parabola by Focus***: The parabola is defined by a focus, apex and a start and end point.

• ***Hyperbola by Focus***: The hyperbola is defined by a focus, center point, apex and a start and end point.

• ***Conic***: There are several different methods that can be used to create conic curves. These methods give you a lot of flexibility when creating above three types of curves.

LINE TOOLBAR

The Line toolbar contains several different ways of creating lines. Reading from left to right, the Line toolbar contains the following commands.

• ***Line***: A line is defined by two points.

• ***Infinite Line:*** Creates infinite lines that are horizontal, vertical or defined by two points.

• ***Bi-Tangent Line***: Creates a line whose endpoints are tangent to two other elements.

• ***Bisecting Line***: Creates an infinite line that bisects the angle created by two other lines.

• ***Line Normal to Curve:*** This command allows you to create a line that starts anywhere and ends normal or perpendicular to another element.

POINT TOOLBAR

The Point toolbar contains several different ways of creating points. Reading from left to right, the Point toolbar contains the following commands.

• ***Point by Clicking***: Creates a point by clicking the left mouse button.

• ***Point by using Coordinates***: Creates a point at a specified coordinate point.

• ***Equidistant Points:*** Creates equidistant points along a predefined path curve.

• ***Intersection Point:*** Creates a point at the intersection of two different elements.

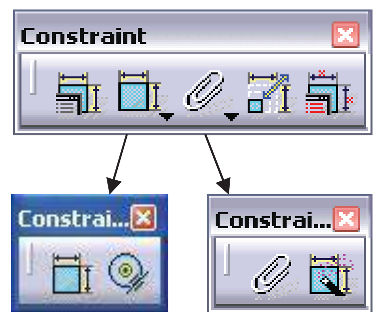
• ***Projection Point:*** Projects a point of one element onto another.

CONSTRAINT TOOLBAR

Constraints can either be dimensional or geometrical. Dimensional constraints are used to constrain the length of an element, the radius or diameter of an arc or circle, and the distance or angle between elements. Geometrical constraints are used to constrain the orientation of one element relative to another. For example, two elements may be constrained to be perpendicular to each other. Other common geometrical constraints include parallel, tangent, coincident, concentric, etc... Reading from left to right:

•***Constraints Defined in Dialoged Box:*** Creates geometrical and dimensional constraints between two elements.

• ***Constraint***: Creates dimensional constraints.



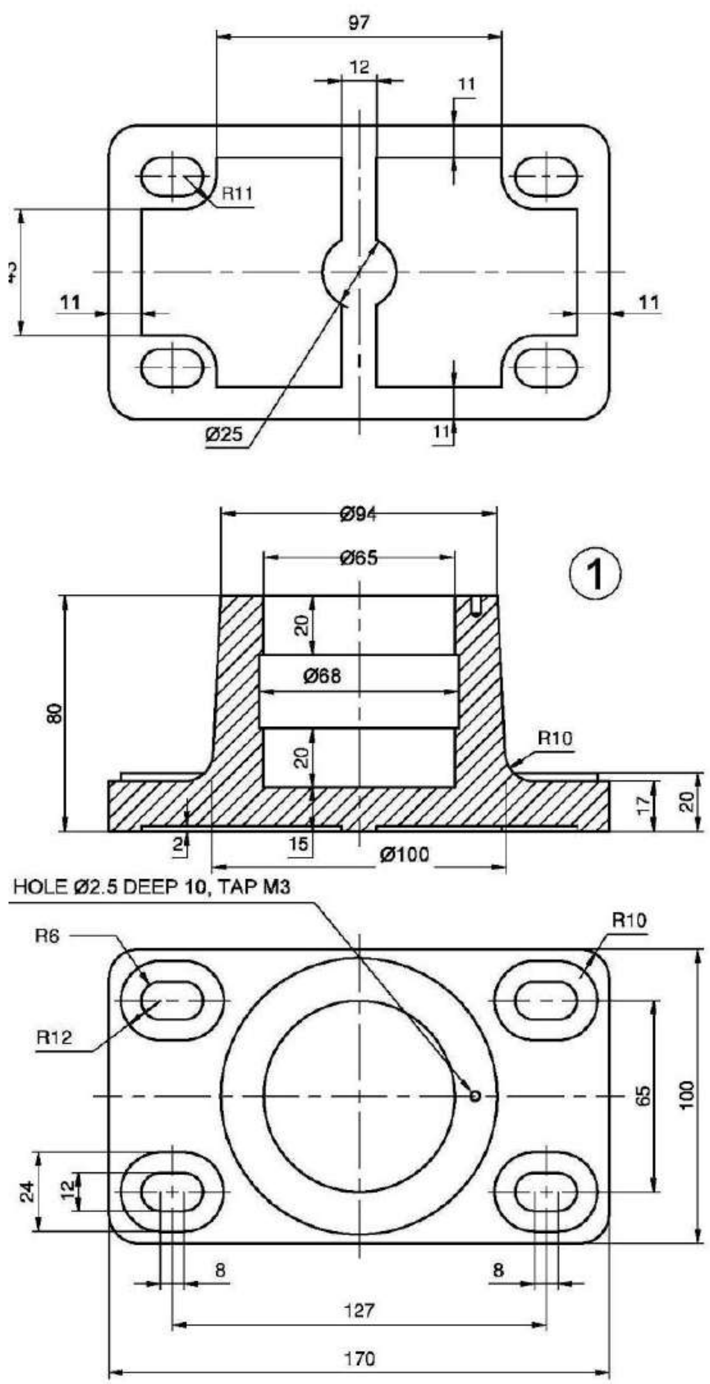
• ***Contact Constraint:*** Creates a contact constraint between two elements.

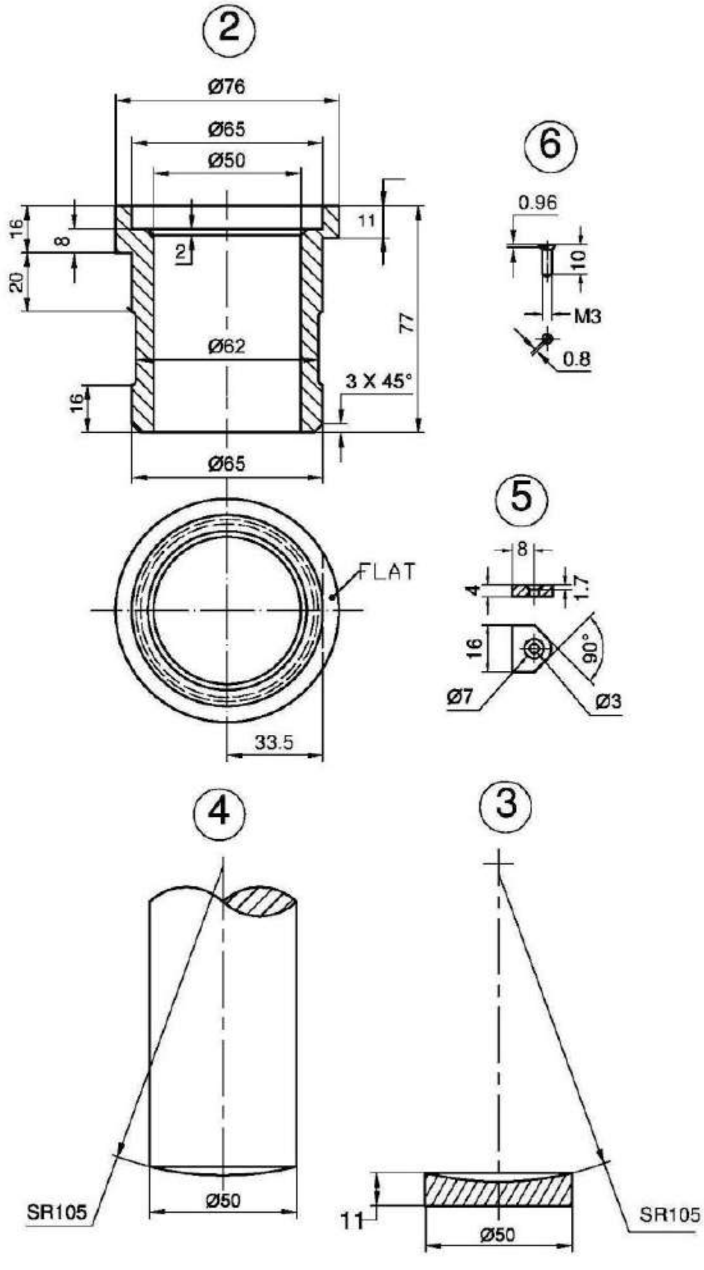
• ***Fix Together***: The fix together command groups individual entities together.

• ***Auto Constraint:*** Automatically creates dimensional constraints.

• ***Animate Constraint:*** Animates a dimensional constraint between to limits.

FOOTSTEP BEARING





All Dimensions are in mm

6	SCREW	Fe 410 W	1
5	LOCKING PLATE	Fe 410 W	1
4	SHAFT	Fe 410 W	1
3	PAD	GUN METAL	1
2	BUSH	GUN METAL	1
1	SUPPORTING BRACKET	CAST IRON	1
ITEM NO	DESCRIPTION	MATERIAL	NO OFF

EX.NO: 01 ASSEMBLY OF FOOT STEP BEARING

DATE:

AIM:

Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

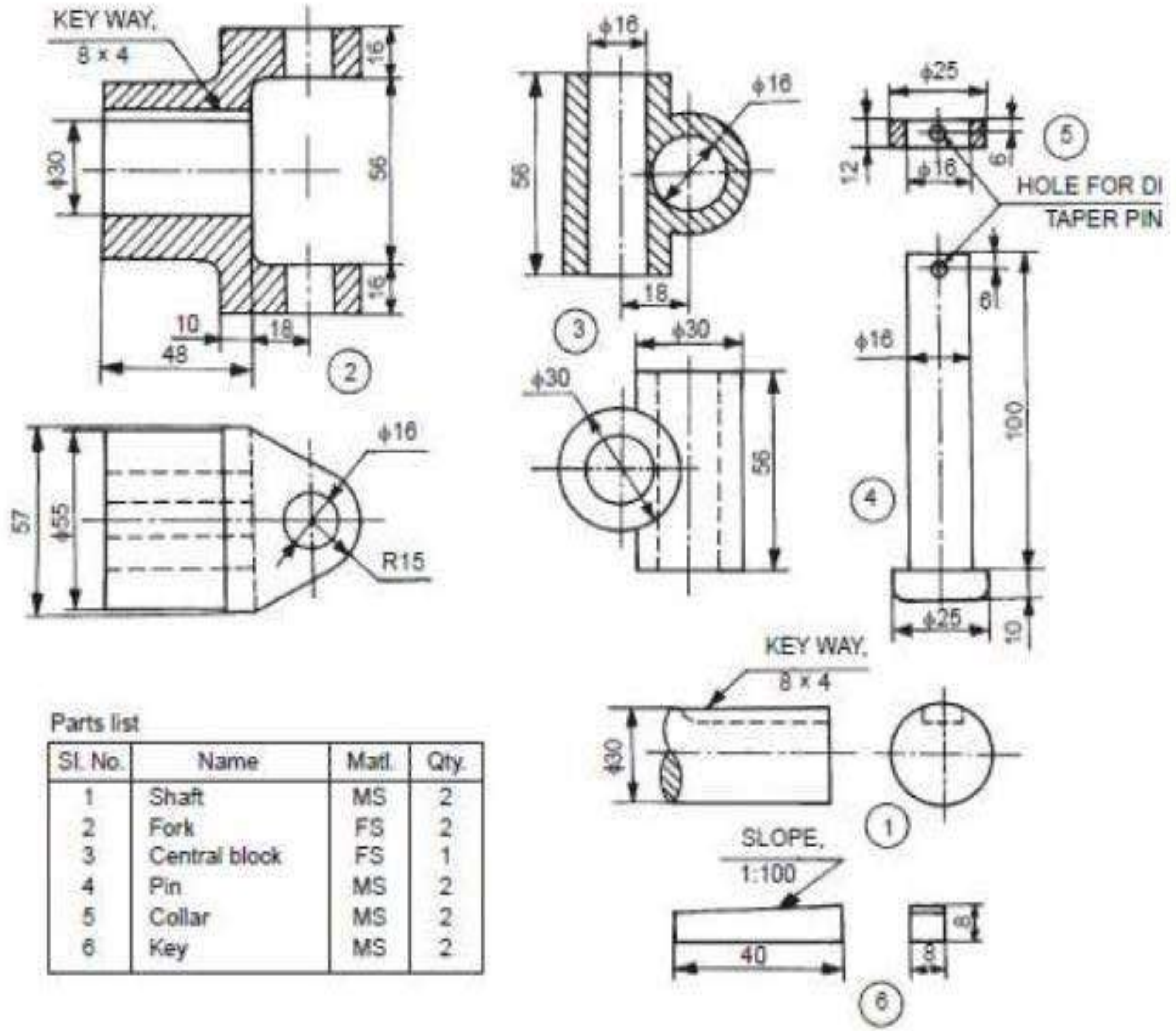
Existing Component icon, Manipulate, Constraining etc.,

PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were the required part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon, to *Fix* a part in the space.
5. Then the parts are assembled in the following order ***Block, Bush, Shaft by*** adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models are ensured that it is in constraints condition before the models are finished.

RESULT:

Thus the given 3D assembly model as per the drawing is modelled using CATIA V5.18 software.



Parts list

Sl. No.	Name	Matl.	Qty.
1	Shaft	MS	2
2	Fork	FS	2
3	Central block	FS	1
4	Pin	MS	2
5	Collar	MS	2
6	Key	MS	2

EX.NO: 02 **ASSEMBLY UNIVERSAL COUPLING**

DATE:

AIM:

Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

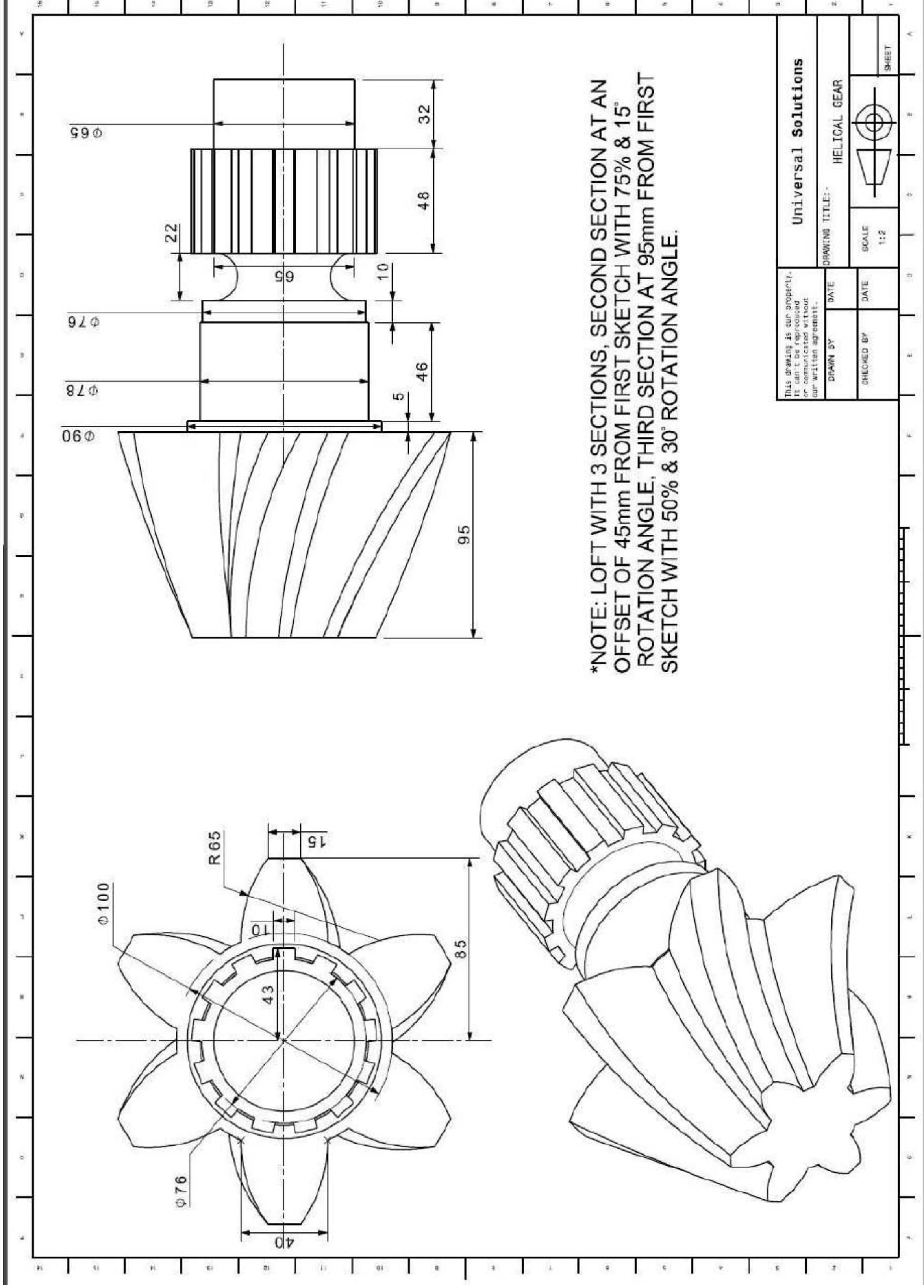
Existing Component icon, Manipulate, Constraining etc.,

PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were the required part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon, to *Fix* a part in the space.
5. Then the parts are assembled in the following order ***Fork, Centre, Shaft, Parallel Key, Pin, Collar, Taper Pin by*** adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models are ensured that it is in constraints condition before the models are finished.

RESULT:

Thus the given 3D assembly model as per the drawing is modelled using CATIA V5.18 software.


*NOTE: LOFT WITH 3 SECTIONS, SECOND SECTION AT AN OFFSET OF 45mm FROM FIRST SKETCH WITH 75% & 15° ROTATION ANGLE, THIRD SECTION AT 95mm FROM FIRST SKETCH WITH 50% & 30° ROTATION ANGLE.
Φ65
32
48
22
65
10
Φ76
46
Φ78
5
Φ90
95
R65
15
Φ100
10
43
85
Φ76
40
Universal Solutions
This drawing is our property. It can't be reproduced or communicated without our written agreement.
DRAWING TITLE:-
HELICAL GEAR
DRAWN BY
DATE
CHECKED BY
DATE
SCALE 1:2
SHEET

EX.NO: 03 **MODELLING OF HELICAL GEAR**

DATE:

AIM:

Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

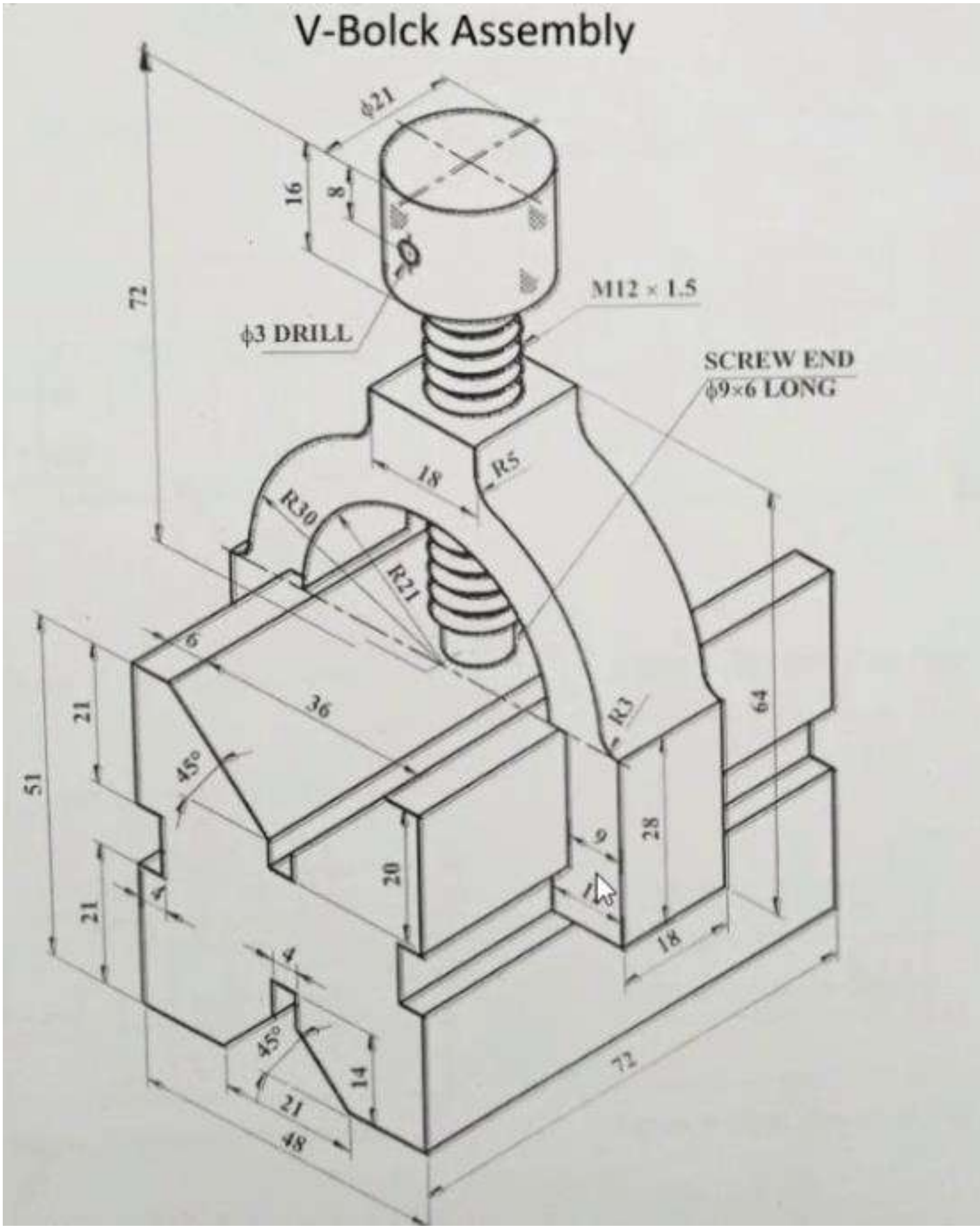
Existing Component icon, Manipulate, Constraining etc.,

PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were the required part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon, to *Fix* a part in the space.
5. Then the parts are assembled in the following order by adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models are ensured that it is in constraints condition before the models are finished.

RESULT:

Thus the given 3D assembly model as per the drawing is modelled using CATIA V5.18 software.


V-Bolck Assembly
φ21
16
8
72
M12 × 1.5
φ3 DRILL
SCREW END
φ9×6 LONG
18
R5
R30
R21
64
6
36
51
21
45°
R3
9
28
20
18
21
4
4
45°
14
21
48
72

EX.NO: 04 MODELLING OF JIG, FIXTURE & DIE ASSEMBLY

DATE:

AIM:

Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

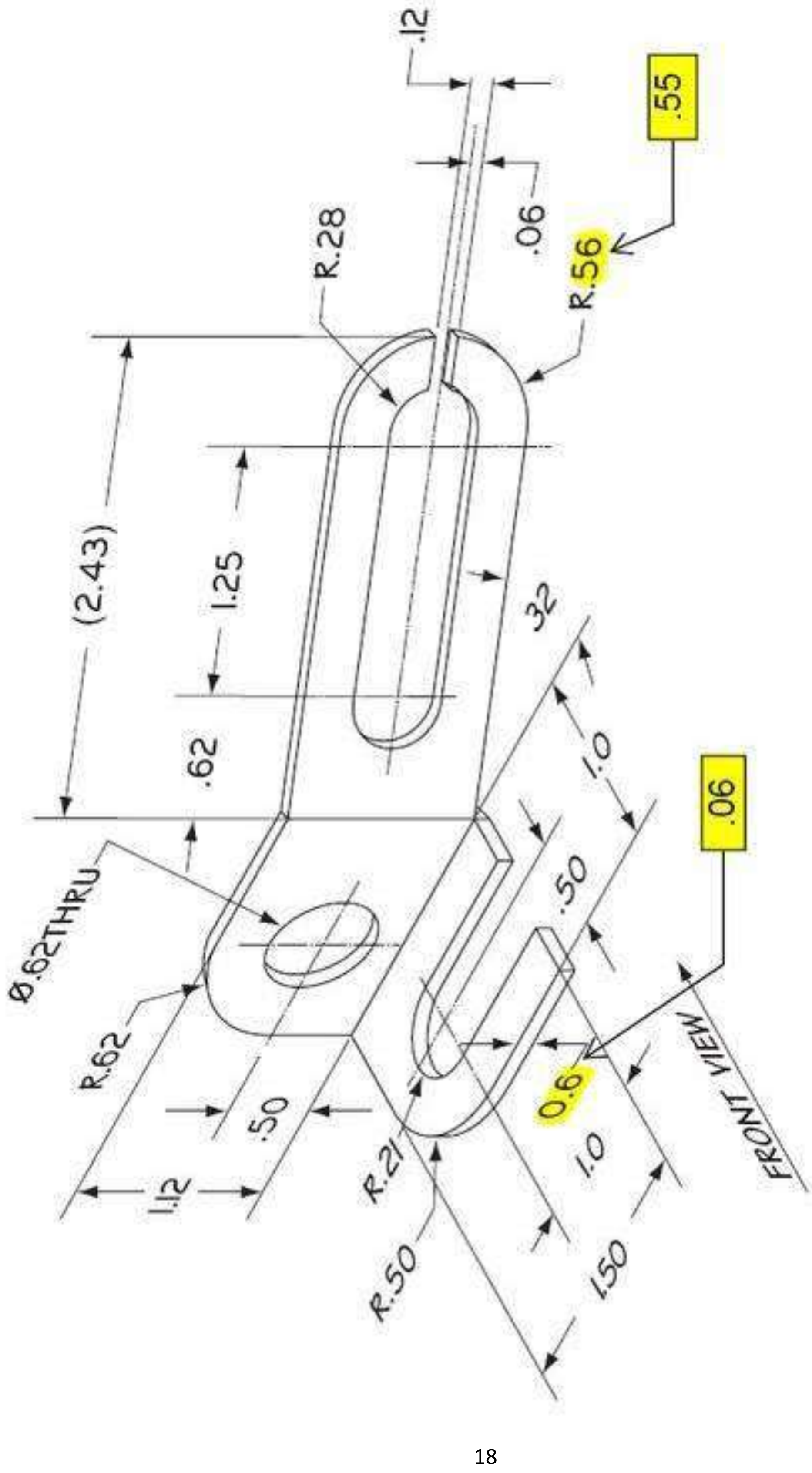
Existing Component icon, Manipulate, Constraining etc.,

PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were the required part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon, to *Fix* a part in the space.
5. Then the parts are assembled in the following order by adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models are ensured that it is in constraints condition before the models are finished.

RESULT:

Thus the given 3D assembly model as per the drawing is modelled using CATIA V5.18 software.



Mounting Clip

EX. NO.: 05 MODELLING OF SHEET METAL COMPONENT

DATE:

AIM:

Preparation of 3D model using CATIA V5.18 software

TOOLS USED:

Pad, Pocketing etc.,

PROCEDURE:

1. Click the Sketcher icon to start the Sketcher workbench.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using Constraint command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click Pad and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using Pocket command material is removed and the final model is created.

RESULT:

Thus the given 3D assembly model as per the drawing is modelled using CATIA V5.18 software.

CAM LABORATORY

INTRODUCTION TO CNC

DEFINITION OF CNC

“A system in which the actions are controlled by direct insertion of numerical data at some point .The system must automatically interpret at least some portion of this data”

WHY IT IS CALLED AS CNC?

Since the information required to actuate and control slides of the machine are coded numerically, this technology came to be known as Numerical Control.

WHAT IS CNC?

CNC is acronym for <u>Computer Numerical Control.</u>

A dedicated computer is used to perform all the basic NC functions. The complete part programme to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools. The program can be stored and used in future

AXIS IN CNC MACHINES

THE BASIS OF AXIS IDENTIFICATION IS THE 3-DIMENSIONAL CARTESIAN CO-ORDINATE SYSTEM AND THREE AXIS OF MOVEMENT ARE IDENTIFIED AS X,Y AND Z AXIS

Z AXIS.

Z- Axis The Z Axis of motion is always the axis of the main spindle of the machine.It doses not matters whether the spindle carries the work piece or the cutting tool . On vertical machining centers Z axis is vertical and on horizontal machining center and turning centers Z axis is horizontal

Positive Z Movement is away from the spindle

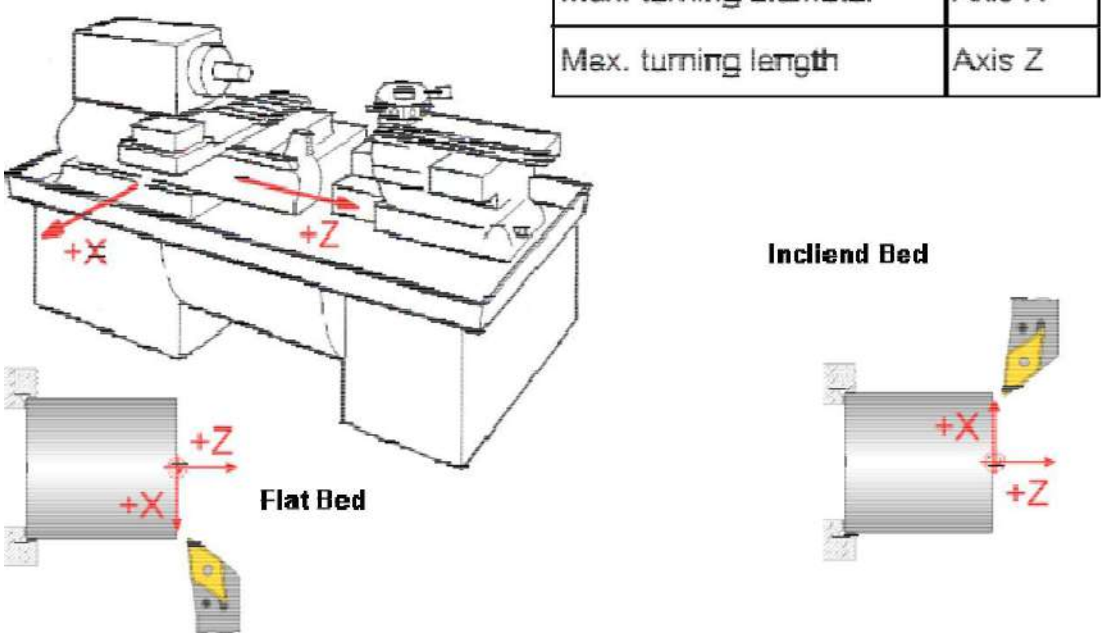
X-Axis The axis is always horizontal and is always parallel to the work holding surface. Positive X Axis movement is identified as being to the right, when looking from the spindle towards its supporting column.

Y- Axis The axis is always at right angle to both X-Axis and Z-Axis

Rotary axis .The rotary motion about the X,Y and Z-Axis are identified by A,B,C respectively .Clockwise is designated as +VE. .Positive rotation is identified looking in x ,y and z direction respectively

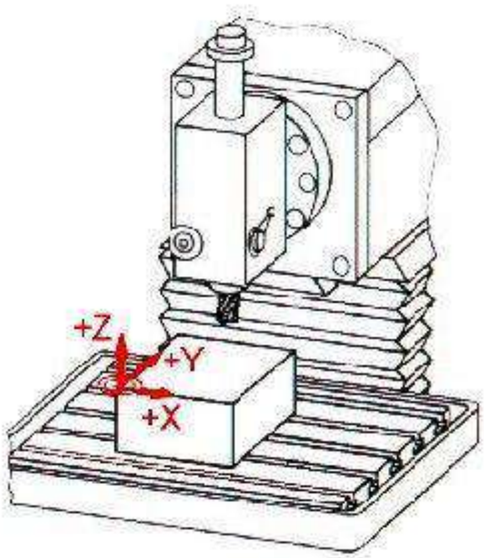
AXIS IN CNCLATHE

Max. turning diameter	Axis X
Max. turning length	Axis Z



AXIS IN MILLING MACHINE

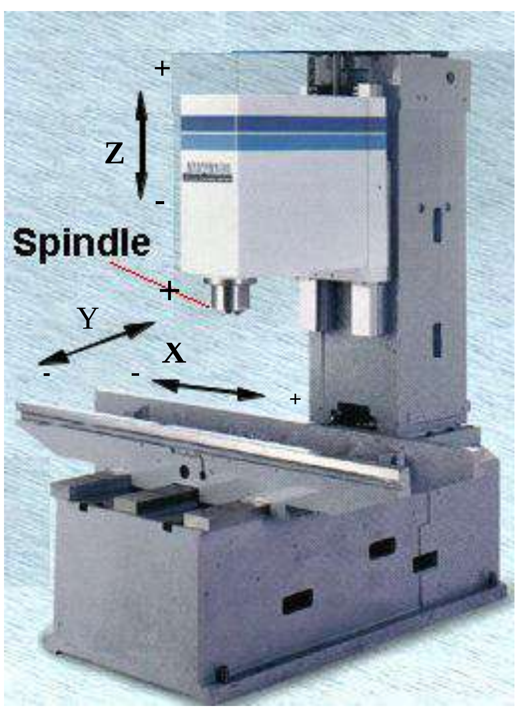
A milling machine has 3 axes of movement identified by X, Y & Z axes

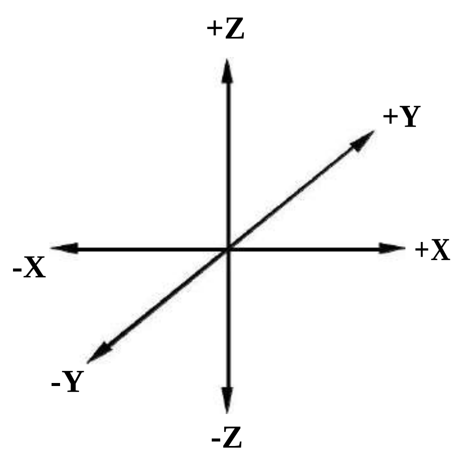


Max. workpiece length	Axis X
Max. workpiece width	Axis Y
Max. workpiece height	Axise Z

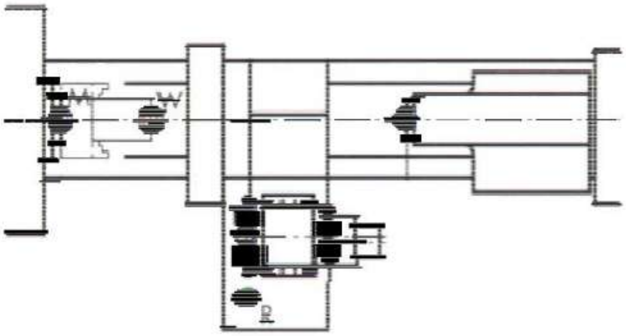
The maximum work piece dimensions correspond to the possible traversing path of the tool in the particular axis.

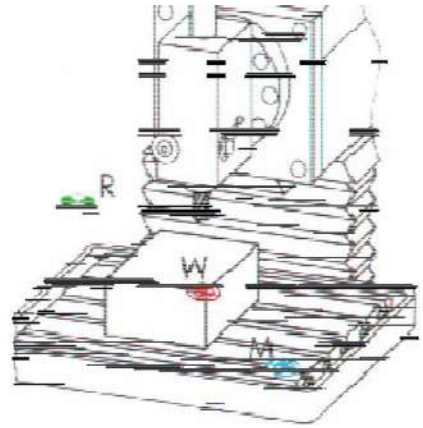
AXIS IN MILLING MACHINE





ZERO POINTS REFERENCE POINT





Machine zero M

The manufacturer defines the machine zero M and this cannot be changed.

It is located at the origin of the machine coordinate system.

Workpiece zero W

The workpiece zero W (also known as program zero) is at the origin of the workpiece coordinate system. It can be freely selected, and should be located at the point where most of the dimensions originate in the drawing.

Reference point R

The reference point R is approached to set the measuring system to zero, as the machine zero point can generally not be approached. The control starts to count in its incremental position measuring system.

The reference point R serves for calibrating and for controlling of measuring systems of the slides and tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point.

After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system.

DIMENSION SYSTEM

Dimensional information in a work piece drawing can be stated in two ways Absolute Dimension System and Incremental Dimension System.

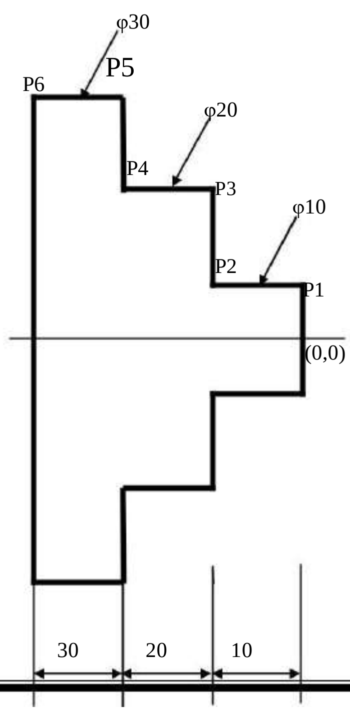
Absolute Dimension System

Data in absolute dimension system always refer to a fixed reference point. This point has the function of a coordinate zero point. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as 'Reference dimensions'

Incremental Dimension System

When using Incremental Dimension system, every measurement refers to a previously dimensioned position Incremental dimensions are distance between adjacent points. These distances are converted into incremental coordinates by accepting the last dimension point as the coordinate origin for the new point. This may be compared to a small coordinate system, i.e., shifted consequently form point to point (P1..P2..through P9). Incremental dimensions are also frequently called 'Relative dimensions' or 'Chain dimensions'.

DIMENSION SYSTEM



ABSOLUTE DIMENSIONING			INCREMENIAL DIMENSIONING		
POINTS	**X**	**Z**	**POINTS**	**U**	**W**
P1	10	0	P1	10	0
P2	10	-10	P2	0	-10
P3	20	-10	P3	10	0
P4	20	-25	P4	0	-15
P5	30	-25	P5	10	0
P6	30	-35	P6	0	-10

STUDY OF ISO CODES

CNC TURNING

AIM: To study various CNC Turning codes and addresses

G Codes

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G32: Thread Cutting

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G49: Tool length compensation cancel

G50: Work co-ordinate Change / Max Spindle speed setting

G70: Finishing cycle

G71: Stock removal in turning

G72: Stock removal in facing

G73: Pattern repeating

G74: Peck drilling in z axis

G75: Grooving in x axis

G76: Thread Cutting Cycle

G80: Canned cycle cancel

G90: Cutting cycle A

G92: Thread cutting cycle

G94: Cutting cycle B

G96: Constant surface speed control

G97: Constant surface speed control cancel

G98: Feed per minute

G99: Feed per revolution

M Codes

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle forward clockwise

M04: Spindle forward counter clockwise

M05: Spindle stop

M06: Tool change

M08: Coolant ON

M09: Coolant OFF

M10: Chuck open

M11: Chuck close

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M98: Sub program Call

M99: Sub program Exit

Common Lathe G-Codes

*G00 **Rapid linear move**. [X, Z, U, W] Moves the machine at the fastest rate possible to the X, Z location specified or incrementally U (X) W(Z) distance.

G01 **Linear feed move** [X, Z, U, W, F] Moves the machine at the specified feed rate (F) to the X, Z location specified or incrementally U (X) W(Z) distance.

G02 **Circular interpolation; CW.** [X, Z, U, W, F, R (or I, K)] Moves the machine, in a clockwise circular path to the X, Z, location (or incrementally by U,W) with radius R, or with a center point defined relative to the start point in the X,& Z axis by I, & K respectively.

G03 **Circular interpolation; CCW.** [X, Z, U, W, F, R (or I, K)] Same as G02, but opposite direction of movement.

G28 **Machine home**. Causes the machine to return to it's X0, Z0 position at a rapid rate.

*G40 **Tool nose compensation, *CANCEL.*** [X, Z, U, W, I, K, F] Cancels the G41 or G42 Cutter compensation listed below. Causes a feed move to X and/or Z at feed rate F (or at modal feed F, if not specified). The distance of the move must be greater than the radius of the tool.

G41 **Tool nose compensation, *LEFT***. [X, Z, U, W, D, F] Looking from the spindle toward the part, G41 offsets the position of the tool *left* of the programmed tool path by the value stored in the offsets register position called by the D word. Causes a feed move from the current position to the *compensated* position specified by X, Y, at feed F (or at modal feed F if not specified). The distance of this move must be greater than the radius of the tool.

G42 **Tool nose compensation, RIGHT**. Same as G41 above except that the tool is compensated to the *right* of the programmed tool path.

NOTE: Cutter compensation may be accomplished at the machine, through the tool path generated by the CAM program, or both.

G50 **Spindle speed clamp.** Specifies the maximum RPM the spindle can run during constant surface speed operation.

G70 **Finishing Cycle**

G71 **Multiple Turning Cycle.** It is used when the major direction of cut is along the Z axis. It causes the profile to be roughed out by turning.

G72 **Multiple Facing Cycle,** used when the major direction of cut is along the X axis. This cycle causes the profile to be roughed out by facing.

G73 **Pattern Repeating Cycle,** provides for roughing out of a form by repeating the desired tool path a set number of times, the tool path being incremented into the work piece until the full form is completed.

G75 **Grooving Cycle**

G92 **Single Thread cycle.** This is a Box type cycle producing a single pass of the threading tool.

G76 **Multiple Threading Cycle.** This is a box type cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip.

G74 **End Face Peck Drilling.** This cycle is designed for deep hole drilling, the drill entering the work piece by a predetermined amount then by backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes.

G90 **Single Turning Cycle**, used to produce either a parallel or tapered tool path. It performs four distinct moves with one line of information and it is equivalent to rapid to X position, feed to Z position, feed to start X position, rapid to start Z position

G94 **Facing Cycle** .This cycle is used for stock removal either parallel or at an angle to work piece face. It is equivalent of rapid to Z position, feed to X Position, and feed to start Z position and rapid to start X position. If R value is specified tapering will be performed.

Common Lathe M-Codes

M00 **Program stop**. Stops the machine, requiring the operator to restart the program to continue.

M01 **Optional stop**. Stops the machine as above only when the optional stop button has been pressed prior to this command in the program.

M03 **Spindle start**, **CW.** [S] Starts the spindle in a clockwise direction, at the RPM specified by the S word accompanying the code.

M04 **Spindle start, CCW**. [S] Similar to above, but rotation is reversed**.**

M05 **Spindle stop.**

M06 **Tool Change.** The M06 in conjunction with "T" word, is used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M08 **Coolant on**.

M09 **Coolant off.**

M10 **Chuck Open**

M11 **Chuck Close**

M13 **Spindle Forward, Coolant ON**

M14 **Spindle Reverse, Coolant ON**

M19 **Spindle orientation.**

M25 **Quill (**Tailstock**) Extend**

M26 **Quill (**Tailstock**) Retract**

M30 **Program stops and rewind.** To program start

M38 **Door Open**

M97 **Local subroutine call.** [N] Causes the program to skip to a subprogram contained inside the current program at line number N.

M98 **Subprogram call** [P, L] the program will call another program number, specified by the P word, and execute it L times.

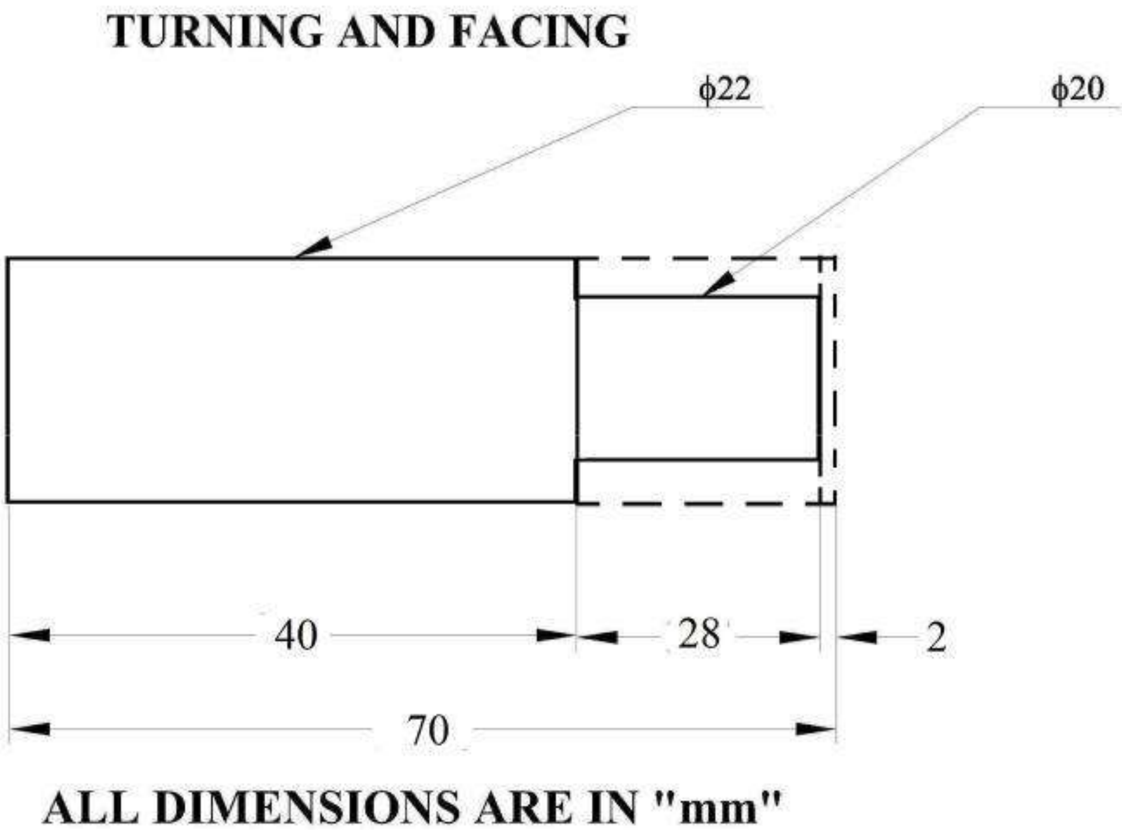
M99 **Subprogram return**. Contained at the end of the subprogram (or subroutine) will return the control to the main program.

Note: only one M Code can be used per line and will be executed after all other operations specified in the line.

RESULT:

Thus various CNC Turning codes and addresses were studied.

Write a manual part program for simple **Turning and Facing** operation for component shown in **DWG.NO.T01**



DWG.NO.T01

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1001				**DWG.NO:1**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Simple Facing	SDJCR121H11	DCMT11T304	1	1	1200	45

PLAIN TURNING AND FACING

EX.NO: L01

DATE:

AIM:

To write a manual part program for simple turning and facing operation for component shown in DWG.NO.T01

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

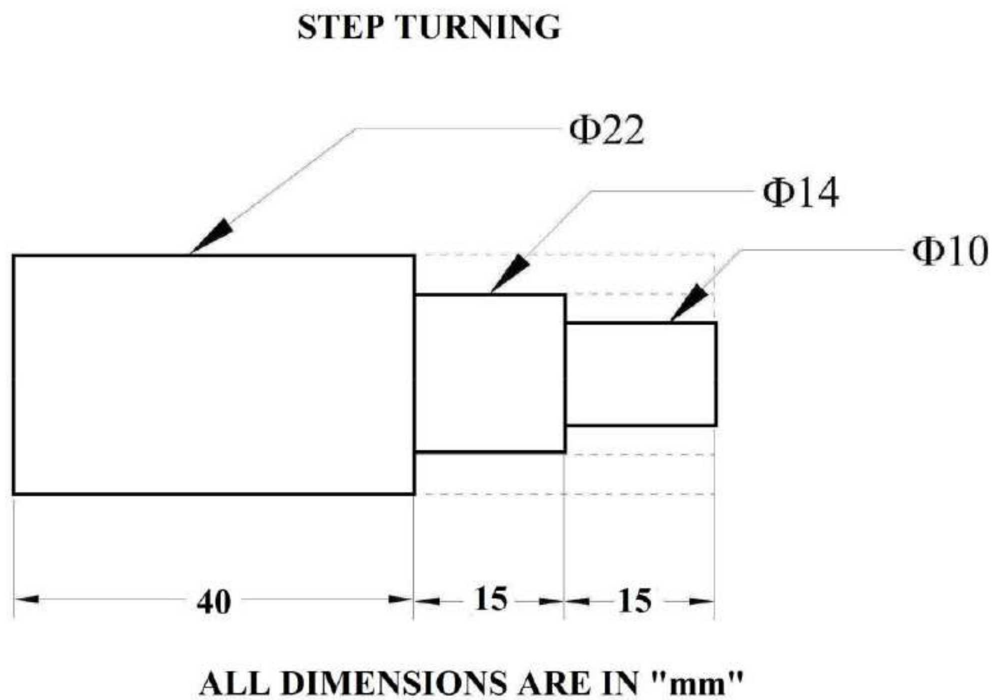
PROGRAM:

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X23 Z3
G00 Z-0.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-2
G01 X-1 F45
G00 Z3
G00 X21.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X21
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20
G01 Z-30 F45
G00 X23
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Step Turning** operation with G90 cycle for component shown in **DWG.NO.T02**



DWG.NO.T02

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1002				**DWG.NO:2**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Step Turning	SDJCR121H11	DCMT11T304	1	1	1200	30

Cutting Cycle

G90 X(U) Z(W) F(*f)

X Diameter to which the movement is being made.

U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.

W The incremental distance from the current tool position to the required Z axis Position fFeed rate

STEP TURNING

EX.NO: L02

DATE:

AIM:

To write a manual part program for step turning operation with G90 cycle for component shown in DWG.NO.T02

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

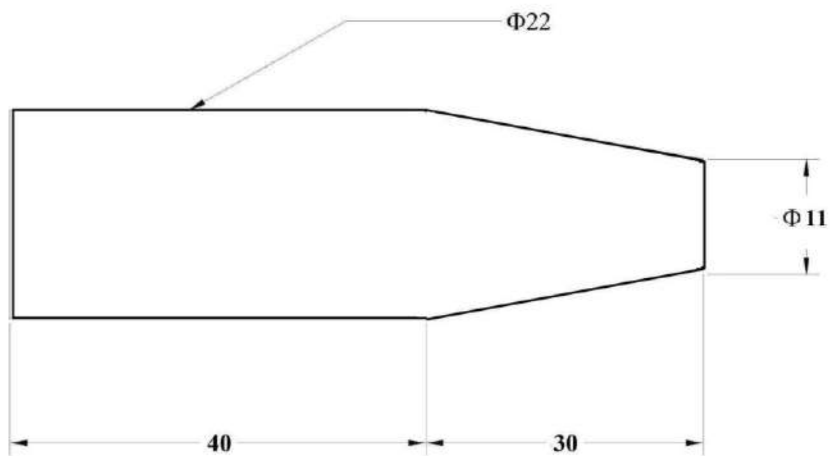
PROGRAM:

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 F30
X21
X20
X19
X18
X17
X16
X15
X14
G00 X14 Z1
G90 X14 Z-15 F30
X13
X12
X11
X10
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Taper Turning** operation for component shown in **DWG.NO.T03**



DWG.NO.T03

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1003				**DWG.NO:3**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Taper Turning	SDJCR121H11	DCMT11T302	3	3	1000	30

Cutting Cycle
G90 X(U) Z(W) R F
X Diameter to which the movement is being made.
U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.
W The incremental distance from the current tool position to the required Z axis Position R The difference in incremental of the cut start radius value and the cut finish radius value.
fFeed rate

TAPER TURNING

EX.NO:L03

DATE:

AIM:

To write a manual part program for taper turning operation for component shown in DWG.NO.T03

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

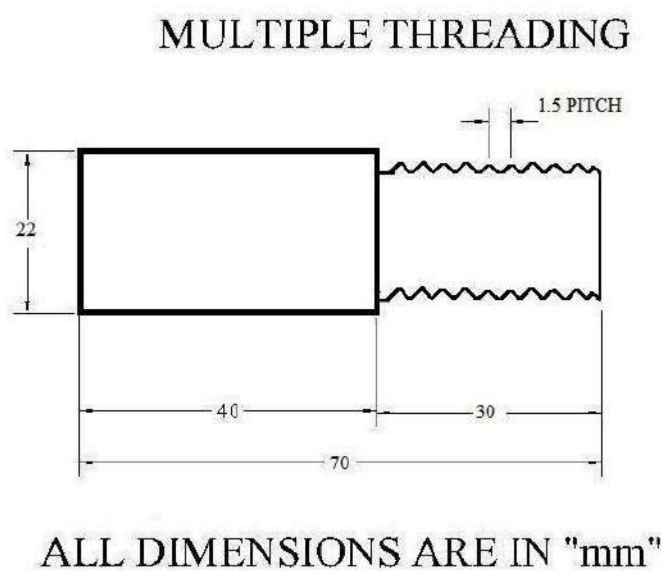
PROGRAM:

```
O0002
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 R0 F30
X22 R-0.5
X22 R-1
X22 R-1.5
X22 R-2
X22 R-2.5
X22 R-3
X22 R-3.5
X22 R-4
X22 R-4.5
X22 R-5
X22 R-5.5
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Threading** operations for component shown in **DWG.NO.T06**



DWG.NO.T06

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1006				**DWG.NO:6**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

MULTIPLE THREADING CYCLE

G76 P(m) (r)(a) Q(*q1) R(*r1)
G76 X(*x) Z(*z) U(*u2) W(*w2) F(*f) S(*s) T(*t) P(*p2) Q(*q2) F(*f)

Where

m- Repetitive count in finishing (1 to 99)
r = Pull out angle
a= Angle of tool tip
*X = Minor diameter (core diameter)
*z = End position of thread
*p2 = Height of thread as a radius value x 1000, value in microns e.g. 1.02mm becomes p1020
*q2 = depth of the first cut as a radius value x 1000 , value is microns. E.g. 0.25 mm becomes Q250 *f = Lead or pitch of thread. E.g. lead 1.5 = F1.5
*q1 = Min. cutting depth
*r1 = Finishing allowance

depth of cut in z axis
Relief Amount Starting Line Number Ending Line Number
the amount and direction of the finishing allowance left in the X Axis
the amount and direction of the finishing allowance left in the Z Axis
Feed Rate
spindle speed
tool number

THREADING

EX.NO:L04

DATE:

AIM:

To write a manual part program for multiple threading operation for component shown in DWG.NO.T06

MATERIAL REQUIRED:

Material : Aluminium
Size: : Length 70mm, Diameter 22mm

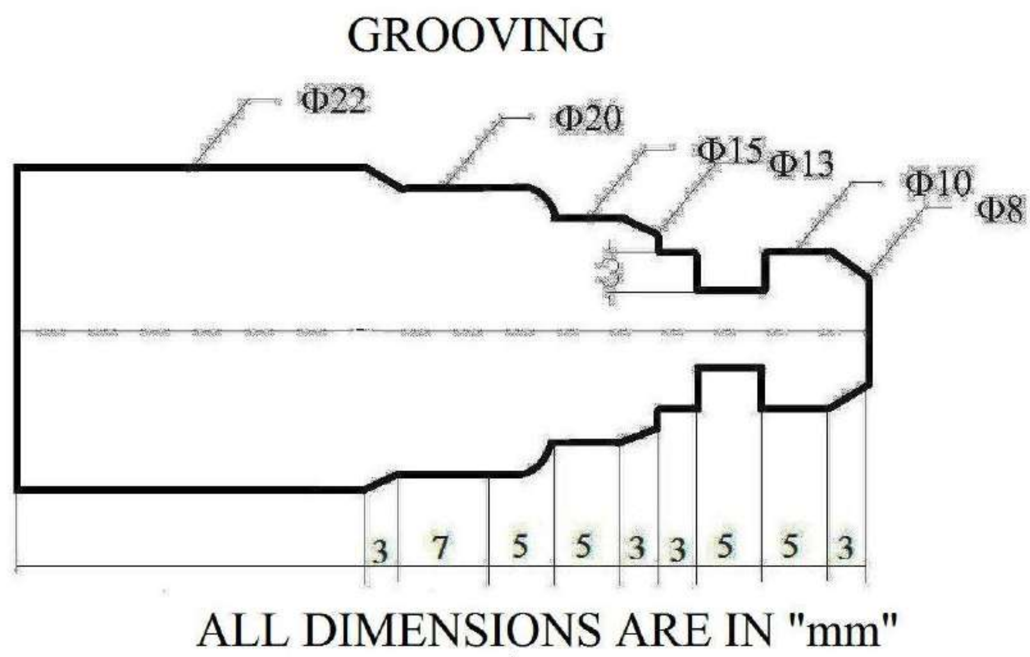
PROGRAM:

```
O0006
G21 G98
G28 U0 W0
M06 T0404
M03 S600
G00 X22.5 Z2
G76 P031560 Q050 R0.02
G76 X20.162 Z-20 P920 Q100 F1.5
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **External Grooving** operation for component shown in **DWG.NO.T05**



DWG.NO.T05

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1005				**DWG.NO:5**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

GROOVING CYCLE

G75 R (*r1)
G75 X(u) Z(w) P(*p) Q(*q) F (*f)

Where, r1-Return Amount
X-Total Depth along X Axis (Absolute)
u-Total Depth along X Axis (Incremental)
z-Total Depth along Z Axis (Absolute)
w-Total Depth along Z Axis (Incremental)
p-Peck Increment in X Axis in microns
q -Stepping distance in Z Axis in microns
f -Feed Rate in mm

GROOVING

EX.NO:L05

DATE:

AIM:

To write a manual part program for external grooving operation for component shown in DWG.NO.T05

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

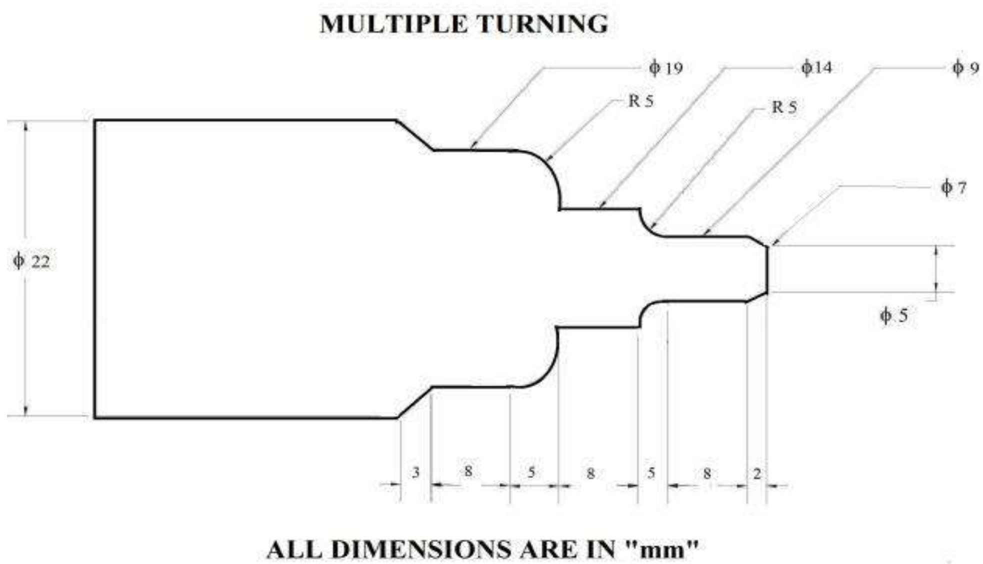
PROGRAM:

```
O0005
G21 G98
G28 U0 W0
M06 T03
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G01 X8 Z0
G01 X10 Z-3
G01 Z-16
G01 X13
G01 X15 Z-19
G01 Z-24
G03 X20 Z-29 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600
G70 P10 Q20 F30
G28 U0 W0
M06 T0707
M03 S600
G00 X10.5 Z-8
G75 R1
G75 X7 Z-10 P100 Q1000 F40
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Turning** operation for component shown in **DWG.NO.T04**



DWG.NO.T04

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25

Multiple Turning Cycle

G72 W (*w1) R (*r)

G72 P(*p) Q(*q)

where

*w1

*r

*P

*q

*u2

*w2

*f

*s

*t

CANNED CYCLE – MULTIPLE TURNING

EX.NO: L06

DATE:

AIM:

To write a manual part program for multiple turning operation for component shown in DWG.NO.T04

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

PROGRAM:

```
O000
G21 G98
G28 U0 W0
M06 T0303
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G00 X7 Z0
G01 X9 Z-2
G01 Z-10
G02 X14 Z-15 R5
G01 Z-23
G03 X19 Z-28 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600 F30
G70 P10 Q20
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

STUDY OF ISO CODES

CNC MILLING

AIM: To study various CNC Milling codes and addresses

PREPARATORY FUNCTIONS (G CODES)

A number following address G determines the meaning of the command for the concerned block. G codes are divided into the following two types

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified

LIST OF G CODES

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G43: Tool length compensation positive direction

G44: Tool length compensation negative direction

G49: Tool length compensation cancel

G50: Scaling OFF

G51: Scaling ON

G54: Datum shift

G68: Rotation ON

G69: Rotation cancel

G73: High speed Peck drilling cycle

G74: L.H Tapping cycle

G76: Fine Boring

G80: Canned cycle cancel

G81: Continuous drilling cycle, Stop boring

G82: Continuous drilling cycle, Stop boring with dwell

G83: Peck drilling cycle

G84: R.H. Tapping cycle

G85: Boring cycle with feed retraction

G86: Boring cycle with rapid retraction

G87: Back boring cycle

G88: Boring cycle

G89: Boring cycle with dwell & feed retraction

G90: Absolute coordinates

G91: Incremental coordinates

G92: Set Datum

G94: Feed per minute

G95: Feed per revolution

G98: Return to initial point in a canned cycle

G99: Return to R point in a canned cycle

G170, G171: Circular pocketing

G172, G173: Rectangular pocketing

MISCELLANEOUS FUNCTIONS (M CODES)

When a three digit figure is specified following address M, a 3-digit BCD code signal and a strobe signal are transmitted. These signals are used for ON/OFF control of a machine function such as tool change, spindle rotation change, coolant ON/OFF. One M code can be specified in one block. Selection of M codes for functions varies with the machine tool builder.

LIST OF M CODES

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle ON clockwise

M04: Spindle ON counter clockwise

M05: Spindle stop

M06: Tool change

M07: Coolant 2 ON

M08: Coolant 1 ON

M09: Coolant OFF

M10: Vice open

M11: Vice close

M12: Synchronization code

M13: Spindle clockwise and coolant ON

M14: Spindle counterclockwise and coolant ON

M19: Orientates Spindle

M20: ATC arm IN (Towards spindle)

M21: ATC arm out (Retracts from spindle)

M22: ATC arm down

M23: ATC arm up

M24: Activates ATC draw bar

M25: Releases draw bar

M30: Program stop & rewind

M32: Rotates ATC clockwise

M33: Rotates ATC counter clockwise

M38: Opens the door

M39: Closes the door

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M70: X mirror ON

M71: Y mirror ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M80: X mirror OFF

M81: Y mirror OFF

M98: Sub program Call

M99: Sub program Exit

COMMON MILLING G-CODES

(* Indicates default start-up codes)

*G00 Rapid linear move. [X, Y, Z] Moves the machine at the fastest rate possible to the X, Y, Z location specified

G01 Linear feed move [X, Y, Z, F] moves the machine at the specified feed rate (F) to the X, Y, Z location specified

G02 Circular interpolation; CW. [X, Y, Z, F, R (or I, J, K)] Moves the machine, in a clockwise circular path to the X, Y, Z, location specified with radius R, or with a center point defined relative to the start point in the X, Y, & Z axis by I, J, & K respectively.

G03 Circular interpolation; CCW. [X, Y, Z, F, R (or I, J, K)] Same as G02, but opposite direction of movement.

G28 Machine home. Causes the machine to return to its X0, Y0, and Z0 position at a rapid rate.

*G40 Cutter compensation, CANCEL. [X, Y, F] Cancels the G41 or G42 Cutter compensation listed below. Causes a feed move to X and/or Y at feed rate F (or at modal feed F, if not specified). The distance of the move must be greater than the radius of the tool.

G41 Cutter compensation, LEFT. [X, Y, D, F] Looking from the spindle toward the part, G41 offsets the position of the tool left of the programmed tool path by the value stored in the offsets register position called by the D word. Causes a feed move from the current position to the compensated position specified by X, Y, at feed F (or at modal feed F if not specified). The distance of this move must be greater than the radius of the tool.

G42 Cutter compensation, RIGHT. Same as G41 above except that the tool is compensated to the right of the programmed tool path.

NOTE: Cutter compensation may be accomplished at the machine, through the tool path generated by the CAM program, or both.

G43 Tool Length compensation. [Z,H] Compensates for the length of the tool during subsequent machine motion. Causes the spindle (Z axis) to move to designated Z location offset by the value stored in the offsets register position called by the H word. This compensation will remain in effect until cancelled during a tool change by the G28 command.

*G53 Machine coordinate system (MCS). Cancels G-54 thru G59

G54-G59 Work coordinate offsets. Allows the user to establish one or more alternative coordinate systems to correspond to individual work coordinate systems (WCS)

G80 Canned cycle cancel. This command cancels any active “canned cycle” commands below.

CANNED CYCLE

A Canned cycle is a combination of machine movements that perform machining operations like drilling, milling, boring and tapping. For example, the drilling cycle consists of the following movements of the tool.

Fast approach to work
piece. Drill at feed rate
Rapid return to initial position.

These movements can be combined to form a cycle and give a code. When this code is invoked, the machine performs all these operations. The use of canned cycle reduces programming effort. This also saves the length of the program, thus saving the space required to store the program.

G81 Drill cycle. [X, Y, Z, R, F] Will drill a hole to Z depth at a location defined by X, Y, at feed rate F. R is the rapid plane, a Z-axis dimension "above the part" that denotes the point where the machine switches from rapid rate to feed rate. Will continue to drill the same hole profile at subsequent X, Y locations until cancelled by G80.

G82 Spot drill / Counter bore cycle. [X, Y, Z, R, F, P] Similar to above, but adds the P word to establish a dwell, or pause at the end of the Z-axis stroke.

G83 Deep Hole Peck drill cycle. [X, Y, Z, R, Q, (or I, J, K), R, F, P] Drills the hole in a series of steps, or pecks, at either a constant peck depth Q, or at a decreasing peck depth where I is the initial peck depth, J is the amount of decrease per peck and K is the minimum peck depth. Other parameters are as for other drill cycles.

G84 Rigid tap cycle (right hand threads). [X, Y, Z, J, R, F] Precisely coordinates the rotation of the spindle with the feed rate of the Z-axis for use with right hand taps. The J word is a multiplying factor for the retract speed. Feed rate is computed by multiplying rpm by thread pitch and is carried to 3 decimal places.

G85 Basic boring cycle. [X, Y, Z, R, F] Similar to the G81 drill cycle, except that the tool is advanced into and out of the work at the feed rate speed, instead of a feed in and rapid out of the G81.

*G90 Absolute Programming Mode. All X, Y, Z coordinates determined from a single origin.

G91 Incremental Programming Mode Next X, Y, Z, coordinates are determined from current location

G98 Canned cycle initial point return. Traverses from hole to hole at the initial point Z height, typically 1.0 inch.

G99 Canned cycle rapid (R) plane return. Traverses from hole to at the rapid plane Z height, typically 0.1 inch.

COMMON MILL M-CODES

M00 Program stops. Stops the machine, requiring the operator to restart the program to continue.

M01 Optional stop. Stops the machine as above only when the optional stop button has been pressed prior to this command in the program.

M03 Spindle start, CW. [S] Starts the spindle in a clockwise direction, at the RPM specified by the S word accompanying the code.

M04 Spindle start, CCW. [S] Similar to above, but rotation is reversed.

M05 Spindle stops.

M06 Tool change. [T] Executes a tool change to the tool specified with the T word. Automatically stops the spindle, returns the Z-axis to machine zero, and turns off the coolant. On machines with automatic tool changers, the tools will be swapped. On the Tool room machines, an alarm light signals the operator to switch tools manually.

M08 Coolant on.

M09 Coolant off.

M19 Spindle orientation.

M30 Program stop and rewind. to program start

M97 Local subroutine call. [N] Causes the program to skip to a subprogram contained inside the current program at line number N.

M98 Subprogram call [P, L] The program will call another program number, specified by the P word, and execute it L times.

Sub Program

When a program contains certain fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into memory as a subprogram to simplify programming. A subprogram can tell another subprogram. When the main program call a subprogram, it is regarded as a one-loop subprogram call.

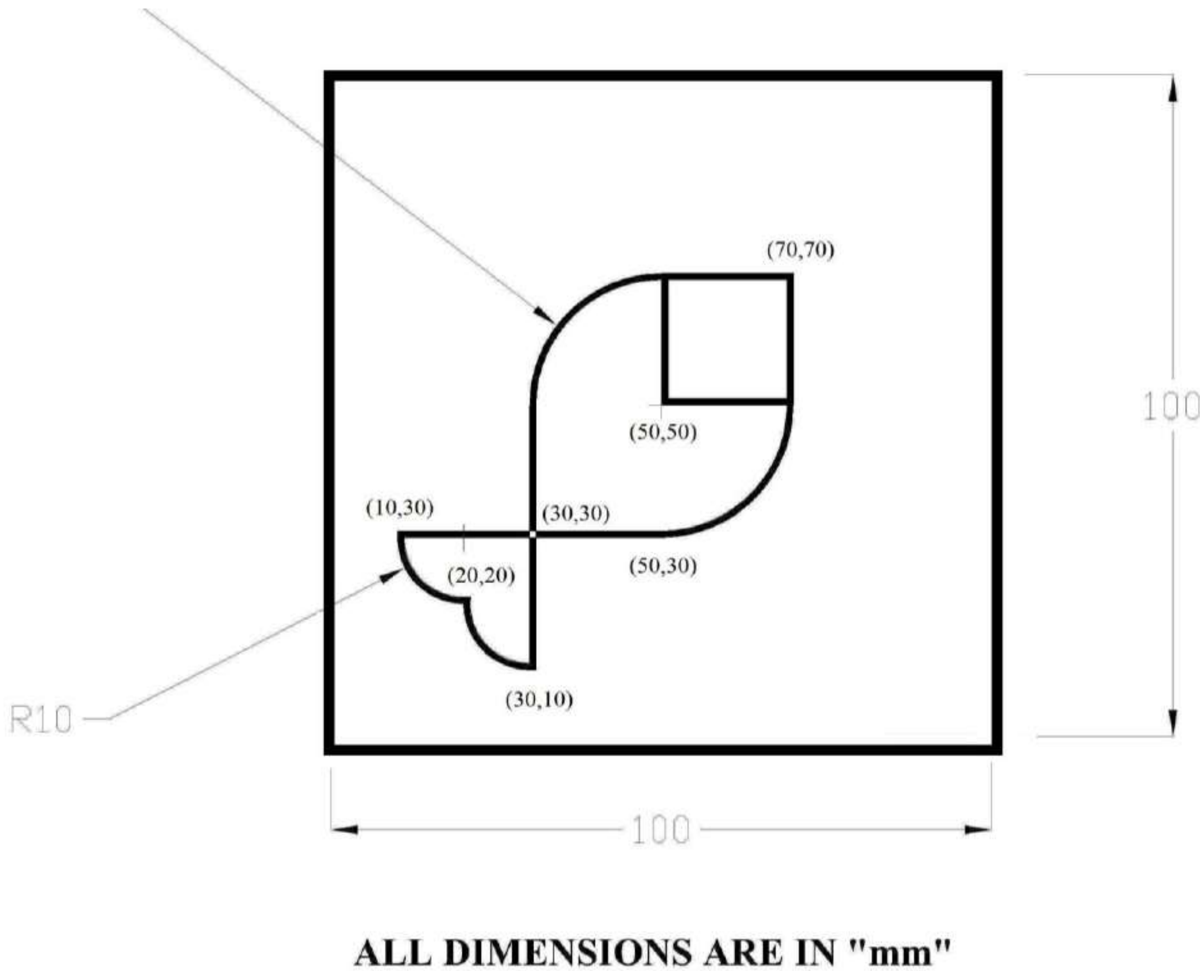
M99 Subprogram returns. Contained at the end of the subprogram (or subroutine) will return the control to the main program.

Note: only one M Code can be used per line and will be executed after all other operations specified in the line.

RESULT:

Thus various CNC Milling codes and addresses were studied.

Write a manual part program for **Profile Milling operation** for the component shown in **DWG.NO.4**. Profile depth=1mm



DWG.NO.4

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

PROFILE MILLING

EX.NO: M01

DATE:

AIM:

To write a manual part program for contouring operation for the component shown in the DWG. NO.4

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

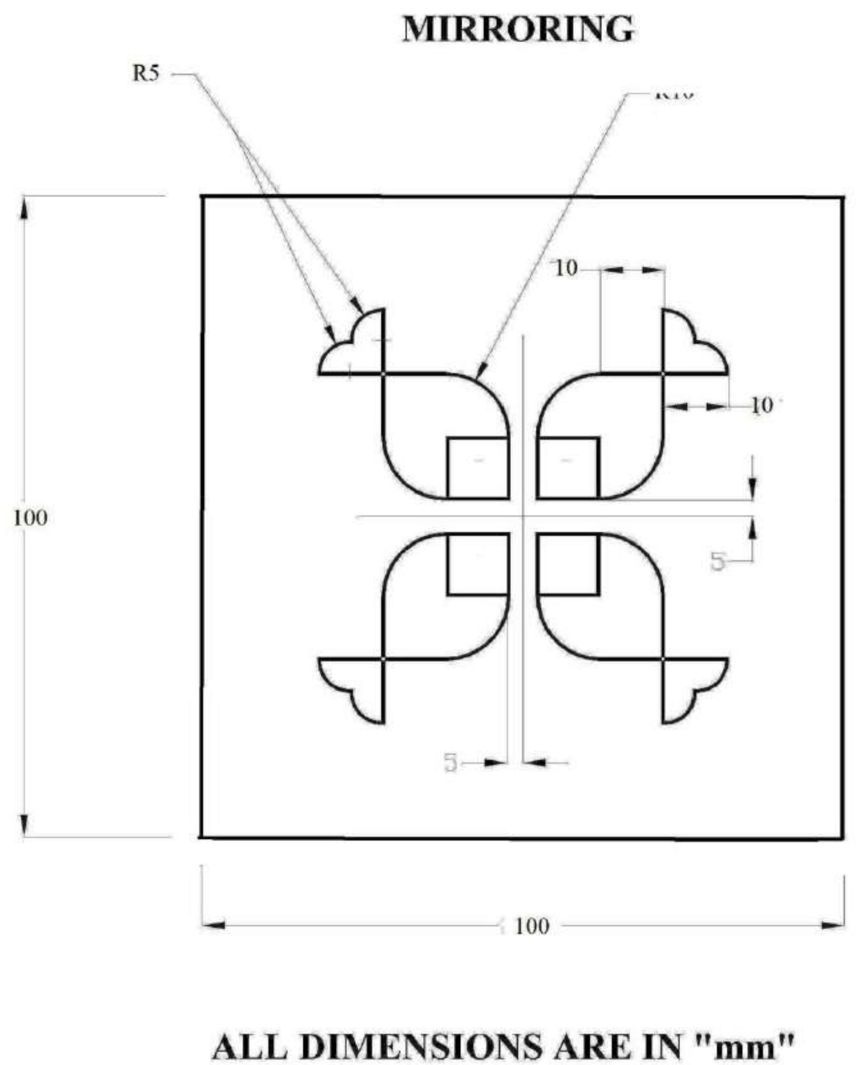
PROGRAM:

```
O0004
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X50 Y70 Z5
G01 Z-1 F30
G01 X70
G01 Y50
G01 X50
G01 Y70
G03 X30 Y50 R20
G01 X30 Y10
G02 X20 Y20 R10
G02 X10 Y30 R10
G01 X50
G03 X70 Y50 R20
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Mirroring operation** for the component shown in **DWG.NO.7**. Profile depth=1mm



DWG.NO.7

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1007				**DWG.NO:7**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

MIRRORING

EX.NO: M04

DATE:

AIM:

To write the manual part program for Mirroring Operation for the component shown in DWG.NO.7

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

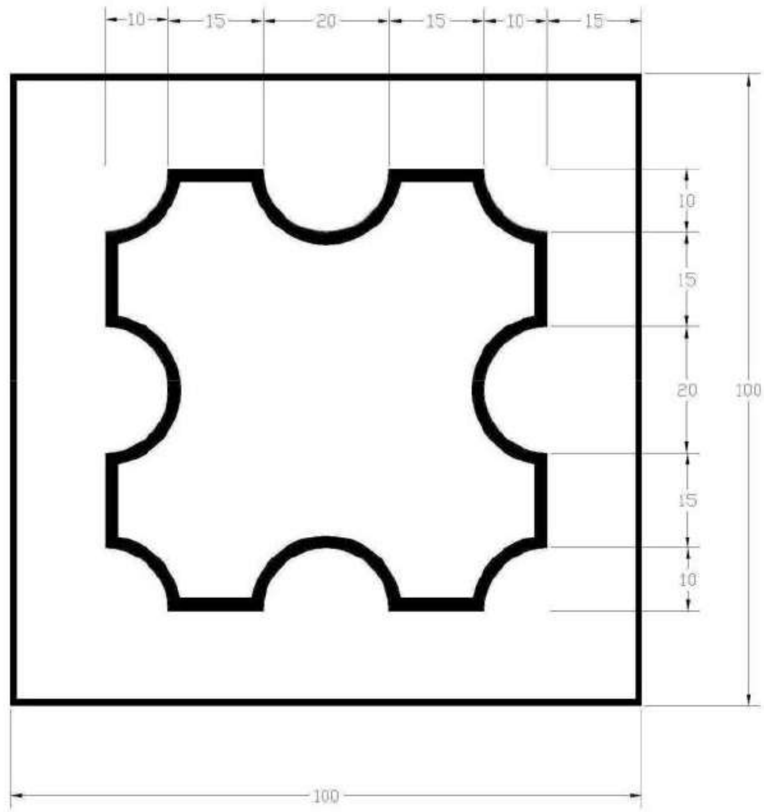
PROGRAM:

```
O0007
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1200
G90 G00 X0 Y0 Z5
M98 P001 3535
M70
M98 P001 3535
M71
M98 P001 3535
M80
M98 P001 3535
M81
G91 G28 X0 Y0 Z0
M05
M30
:3535
G00 X5 Y15
G01 Z-1 F30
G01 X15
G01 Y5
G01 X5
G01 Y15
G02 X15 Y25 R10
G01 X35
G03 X30 Y30 R5
G03 X25 Y35 R5
G01 Y15
G02 X15 Y5 R10
G00 Z5
M99
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling

Write the manual part program for **Scaling** for the component shown in **DWG.NO.6.** Profile depth: 3mm. Depth of cut for each pass=1mm.



ALL DIMENSIONS ARE IN "mm"

DWG.NO.6

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1006				**DWG.NO:6**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

SCALING

EX.NO: M03

DATE:

AIM:

To write the manual part program for contouring operation through subroutine for the component shown in DWG.NO.6

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

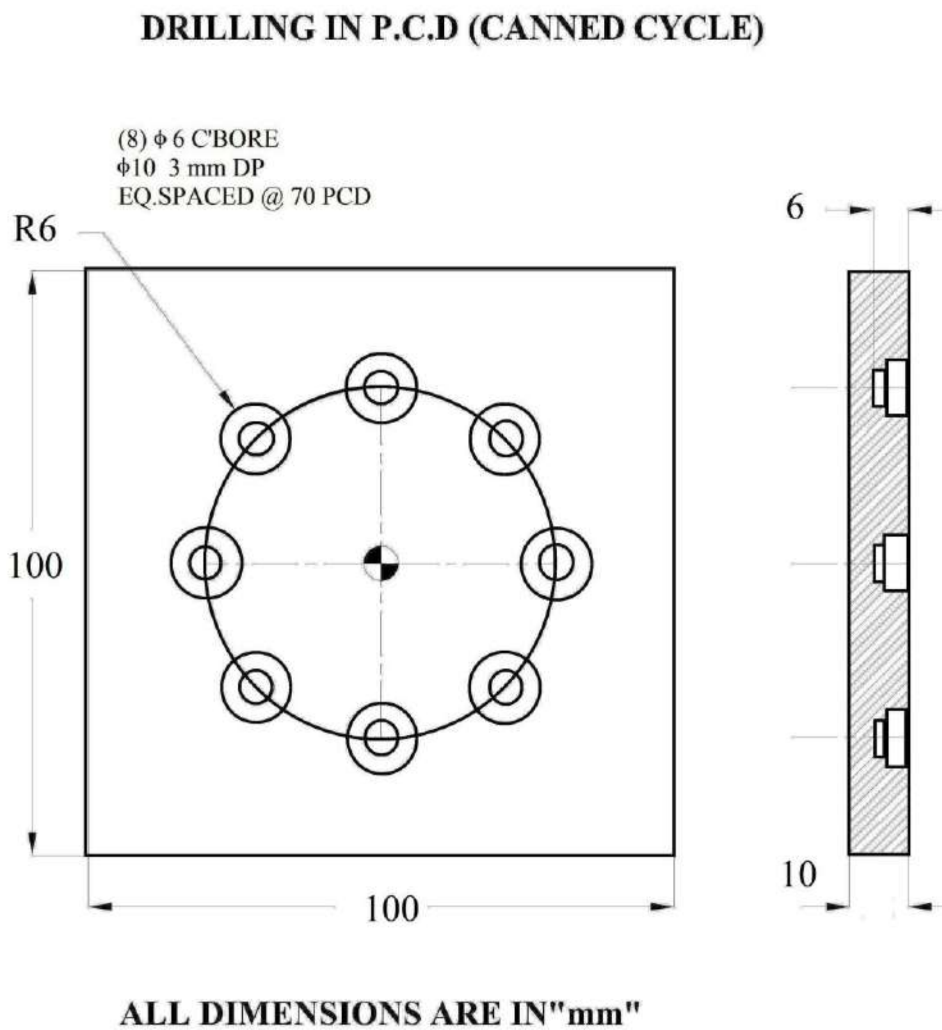
PROGRAM:

```
O0006
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X25 Y15 Z5
G01 Z0 F50
M98 P003 3242
G90 G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
:3242
G91 G01 Z-1
G90 G01 X40 Y15
G02 X60 Y15 R10 F30
G01 X75 Y15
G02 X85 Y25 R10 F30
G01 X 85 Y40
G02 X85 Y60 R10
G01 X85 Y75
G02 X75 Y85 R10
G01 X60 Y85
G02 X40 Y85 R10
G01 X25 Y85
G02 X15 Y75 R10
G01 X15 Y60
G02 X15 Y40 R10
G01 X15 Y25
G02 X25 Y15 R10
M99
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for machining the component **through Canned Cycle** for the component shown in the **DWG. NO.9**



DWG.NO.9

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1009				**DWG.NO:09**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Drilling	Slot drill	6	1	1	1500	35
2	Drilling	Slot drill	10	2	2	1500	35

DRILLING IN P.C.D (CANNED CYCLE)

EX.NO:M04

DATE:

AIM:

To write a manual part program for machining the component through canned cycle for the component shown in the DWG. NO.9

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

```
O0009
G21 G94
G91 G28 X0 Y0 Z0
M06 T11
M03 S1500
G90 G00 X0 Y35 Z5
G98 G81 X0 Y35 Z-6 R2 F35
M98 P001 2525
G91 G28 X0 Y0 Z0
M05
M06 T01
M03 S1500
G90 G00 X0 Y35 Z5
G98 G81 X0 Y35 Z-3 R2 F35
M98 P001 2525
G91 G28 X0 Y0 Z0
M05
M30
:2525
X24.74 Y24.74
X35 Y0
X24.74 Y-24.74
X0 Y-35
X-24.74 Y-24.74
X-35 Y0
X-24.74 Y24.74
G00 Z10
G80
M99
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling

5. STUDY OF SENSORS AND TRANSDUCERS

1. Sensor

It is defined as an element which produces signal relating to the quantity being measured. According to the Instrument Society of America, sensor can be defined as " *A device which provides a usable output in response to a specified measurand.* " Here, the output is usually an 'electrical quantity' and measurand is a 'physical quantity, property or condition which is to be measured'. Thus in the case of, say, a variable inductance displacement element, the quantity being measured is displacement and the sensor transforms an input of displacement into a change in inductance.

2. Transducer

It is defined as an element when subjected to some physical change experiences a related change or an element which converts a specified measurand into a usable outputby using a transduction principle.

It can also be defined as a device that converts a signal from one form of energy to another form.

A wire of Constantan alloy (copper-nickel 55-45% alloy) can be called as a sensor because variation in mechanical displacement (tension or compression) can be sensed as change in electric resistance. This wire becomes a transducer with appropriate electrodes and input-output mechanism attached to it. Thus we can say that 'sensors are transducers

3. Sensor/transducers specifications

Transducers or measurement systemsare not perfect systems. Mechatronics design engineer must know the capability and shortcoming of a transducer or measurement system to properly assess its performance. These are a number of performance related parameters of a transducer or measurement system. These parameters are called as sensor specifications.

Sensor specifications inform the user about deviations from the ideal behavior of the sensors. Following are the various specifications of a sensor/transducer system.

3.1 Range

The range of a sensor indicates the limits between which the input can vary. Thus, for example, a thermocouple for the measurement of temperature might have a range of 25-225°C.

3.2 Span

The span is difference between the maximum and minimum values of the input. Thus, the above-mentioned thermocouple will have a span of 200°C.

3.3 Error

Error is the difference between the result of the measurement and the true value of the quantity being measured. A sensor might give a displacement reading of 29.8 mm, when the actual displacement had been 30 mm, then the error is - 0.2 mm.

3.4 Accuracy

The accuracy defines the closeness of the agreement between the actual measurement result and a true value of the measurand. It is often expressed as a percentage of the full range output or full–scale deflection. A piezoelectric transducer used to evaluate dynamic pressure phenomena associated with explosions, pulsations, or dynamic pressure conditions in motors, rocket engines, compressors, and other pressurized devices is capable to detect pressures between 0.1 and 10,000 psig (0.7 KPa to 70 MPa). If it is specified with the accuracy of about±1% full scale, then the reading given can be expected to be within ± 0.7 MPa.

3.5 Sensitivity

Sensitivity of a sensor is defined as the ratio of change in output value of a sensor to the per unit change in input value that causes the output change.For example, a general purpose thermocouple may have a sensitivity of 41 µV/°C.

3.6 Nonlinearity

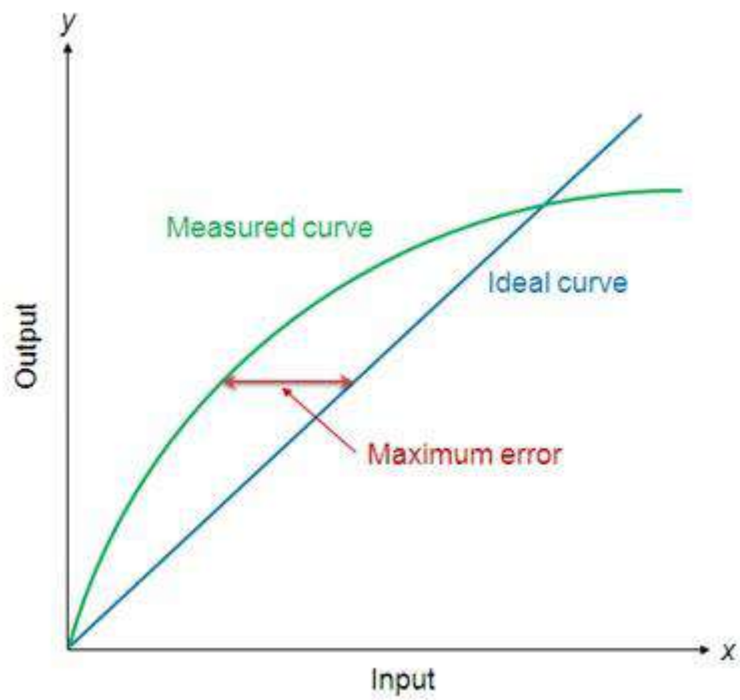


Figure 2.1.1 Non-linearity error

The nonlinearity indicates the maximum deviation of the actual measured curve of a sensor from the ideal curve. Figure 2.1.1 shows a somewhat exaggerated relationship between the ideal, or least squares fit, line and the actual measured or *calibration* line. Linearity is often specified in terms of *percentage of nonlinearity,* which is defined as:

Nonlinearity (%) = Maximum deviation in input ⁄ Maximum full scale input

The static nonlinearity defined by Equation 2.1.1 is dependent upon environmental factors, including temperature, vibration, acoustic noise level, and humidity. Therefore it is important to know under what conditions the specification is valid.

3.7 Hysteresis

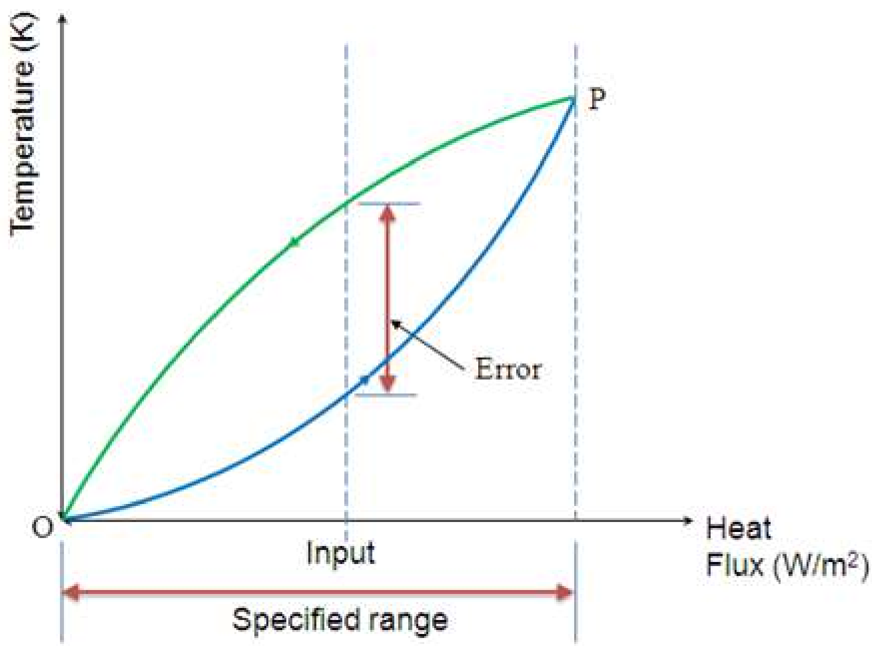


Figure 2.1.2 Hysteresis error curve

The hysteresis is an error of a sensor, which is defined as the maximum difference in output at any measurement value within the sensor's specified range when approaching the point first with increasing and then with decreasing the input parameter. Figure 2.1.2 shows the hysteresis error might have occurred during measurement of temperature using a thermocouple. The hysteresis error value is normally specified as a positive or negative percentage of the specified input range.

3.8 Resolution

Resolution is the smallest detectable incremental change of input parameter that can be detected in the output signal. Resolution can be expressed either as a proportion of the full-scale reading or in absolute terms.For example, if a LVDT sensor measures a displacement up to 20 mm and it provides an output as a number between 1 and 100 then the resolution of the sensor device is 0.2 mm.

3.9 Stability

Stability is the ability of a sensor device to give same output when used to measure a constant input over a period of time. The term 'drift' is used to indicate the change in output that occurs over a period of time. It is expressed as the percentage of full range output.

3.10 Dead band/time

The dead band or dead space of a transducer is the range of input values for which there is no output. The dead time of a sensor device is the time duration from the application of an input until the output begins to respond or change.

3.11 Repeatability

It specifies the ability of a sensor to give same output for repeated applications of same input value. It is usually expressed as a percentage of the full range output:

Repeatability = (maximum – minimum values given) X 100/full range

3.12 Response time

Response time describes the speed of change in the output on a step-wise change of the measurand. It is always specified with an indication of input step and the output range for which the response time is defined. **4. Classification of sensors**

Sensors can be classified into various groups according to the factors such as measurand, application fields, conversion principle, energy domain of the measurand and thermodynamic considerations. These general classifications of sensors are well described in the references [2, 3].

Detail classification of sensors in view of their applications in manufacturing is as follows.

A. Displacement, position and proximity sensors
    - Potentiometer
    - Strain-gauged element
    - Capacitive element
    - Differential transformers
    - Eddy current proximity sensors
    - Inductive proximity switch
    - Optical encoders
    - Pneumatic sensors
    - Proximity switches (magnetic)
    - Hall effect sensors
B. Velocity and motion
    - Incremental encoder
    - Tachogenerator
    - Pyroelectric sensors
C. Force
    - Strain gauge load cell
D. Fluid pressure
    - Diaphragm pressure gauge
    - Capsules, bellows, pressure tubes
    - Piezoelectric sensors
    - Tactile sensor
E. Liquid flow
    - Orifice plate
    - Turbine meter
F. Liquid level
    - Floats

- Differential pressure

G. Temperature
- Bimetallic strips
- Resistance temperature detectors
- Thermistors
- Thermo-diodes and transistors
- Thermocouples
- Light sensors
- Photo diodes
- Photo resistors
- Photo transistor

Principle of operation of these transducers and their applications in manufacturing are presented in the next lectures.

Displacement sensors

1. Potentiometer Sensors

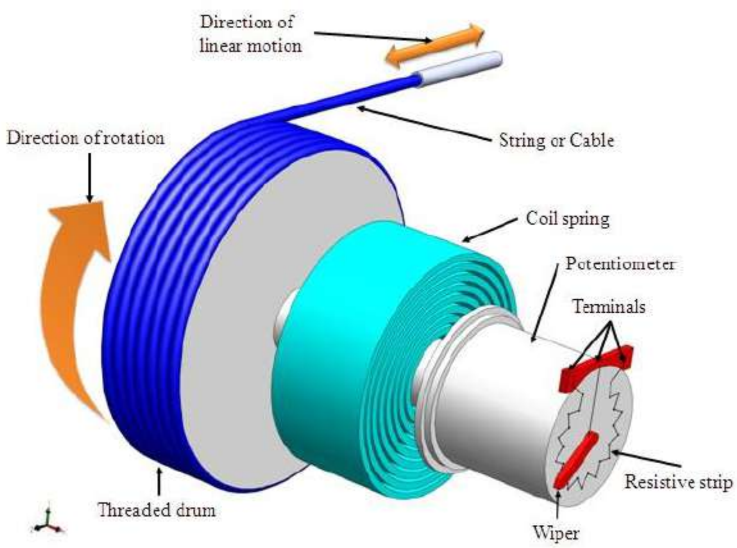


Figure 2.2.1 Schematic of a potentiometer sensor for measurement of linear displacement

Figure 2.2.1 shows the construction of a rotary type potentiometer sensor employed to measure the linear displacement. The potentiometer can be of linear or angular type. It works on the principle of conversion of mechanical displacement into an electrical signal. The sensor has a resistive element and a sliding contact (wiper). The slider moves along this conductive body, acting as a movable electric contact.

The object of whose displacement is to be measured is connected to the slider by using

- a rotating shaft (for angular displacement)
- a moving rod (for linear displacement)
- a cable that is kept stretched during operation

The resistive element is a wire wound track or conductive plastic. The track comprises of large number of closely packed turns of a resistive wire. Conductive plastic is made up of plastic resin embedded with the carbon powder. Wire wound track has a resolution of the order of ± 0.01 % while the conductive plastic may have the resolution of about 0.1 µm.

During the sensing operation, a voltage V_S is applied across the resistive element. A voltage divider circuit is formed when slider comes into contact with the wire. The output voltage (V_A) is measured as shown in the figure 2.2.2. The output voltage is proportional to the displacement of the slider over the wire. Then the output parameter displacement is calibrated against the output voltage V_A.

2. Strain Gauges

The strain in an element is a ratio of change in length in the direction of applied load to the original length of an element. The strain changes the resistance *R* of the element. Therefore, we can say,

$$\frac{\Delta R}{R} \propto \varepsilon$$

$$\frac{\Delta R}{R} = G\varepsilon; \qquad (2.2.5)$$

where G is the constant of proportionality and is called as gauge factor. In general, the value of G is consideredin between 2 to 4 and the resistances are taken of the order of 100 Ω.

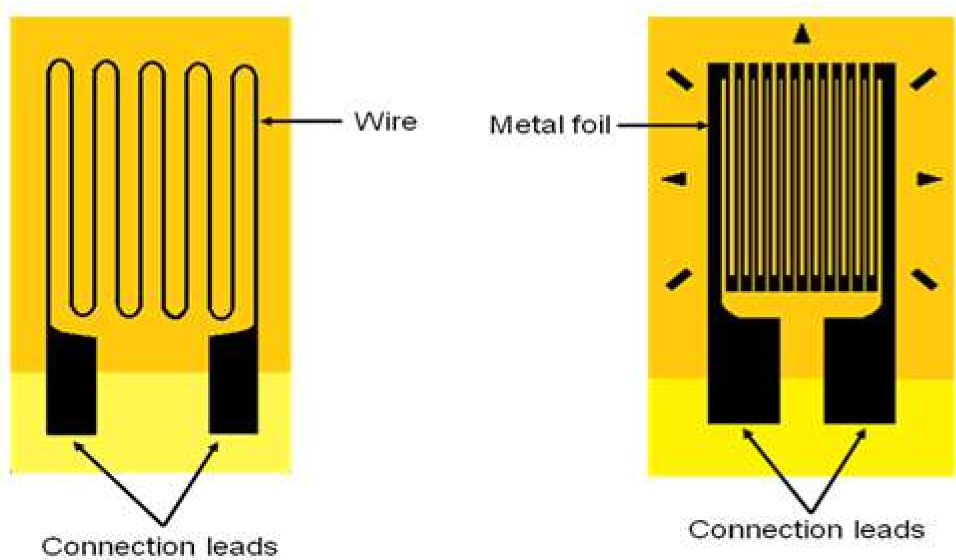


Figure 2.2.3 A pattern of resistive foils

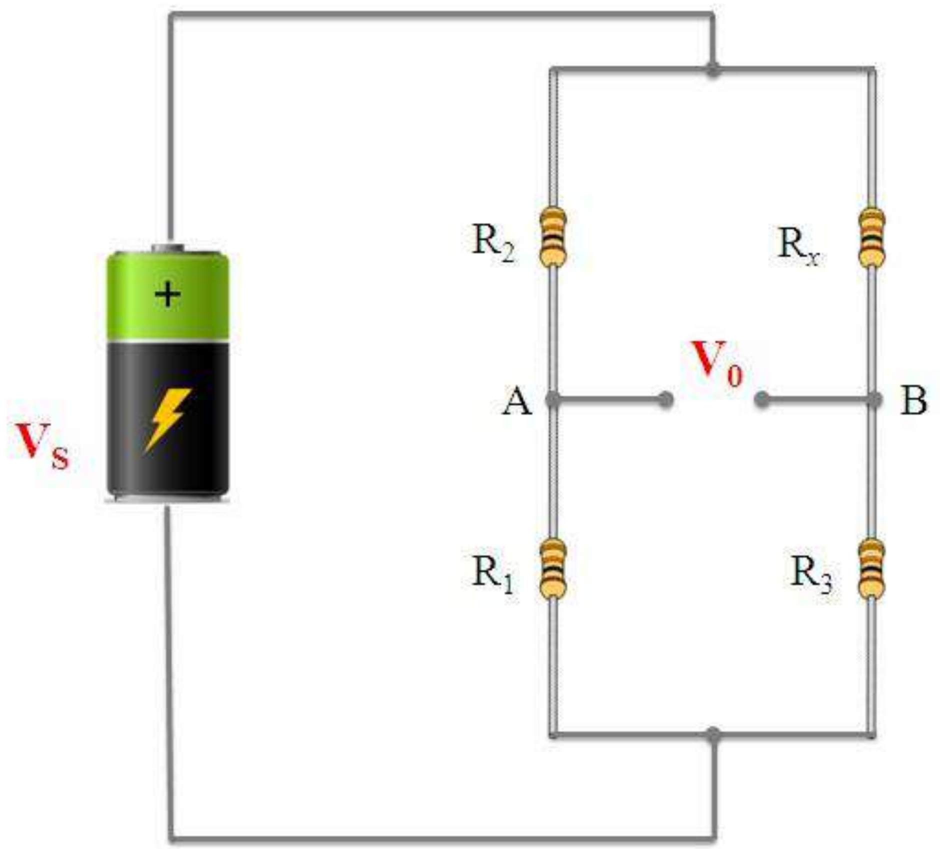


Figure 2.2.4 Wheatstone's bridge

Resistance strain gauge follows the principle of change in resistance as per the equation 2.2.5. It comprises of a pattern of resistive foil arranged as shown in Figure 2.2.3. These foils are made of Constantan alloy (copper-nickel 55-45% alloy) and are bonded to a backing material plastic (ployimide), epoxy or glass fiber reinforced epoxy. The strain gauges are secured to the workpiece by using epoxy or Cyanoacrylate cement Eastman 910 SL. As the workpiece undergoes change in its shape due to external loading, the resistance of strain gauge element changes. This change in resistance can be detected by a using a Wheatstone's resistance bridge as shown in Figure 2.2.4.In the balanced bridge we can have a relation,

$$\frac{R_2}{R_1} = \frac{R_x}{R_3} \qquad (2.2.6)$$

where R_x is resistance of strain gauge element, R_2 is balancing/adjustable resistor, R_1 and R_3 are known constant value resistors. The measured deformation or displacement by the stain gauge is calibrated against change in resistance of adjustable resistor R_2 which makes the voltage across nodes A and B equal to zero.

Applications of strain gauges

Strain gauges are widely used in experimental stress analysis and diagnosis on machines and failure analysis. They are basically used for multi-axial stress fatigue testing, proof testing, residual stress and vibration measurement, torque measurement, bending and deflection measurement, compression and tension measurement and strain measurement.

Strain gauges are primarily used as sensors for machine tools and safety in automotives. In particular, they are employed for force measurement in machine tools, hydraulic or pneumatic press and as impact sensors in aerospace vehicles.

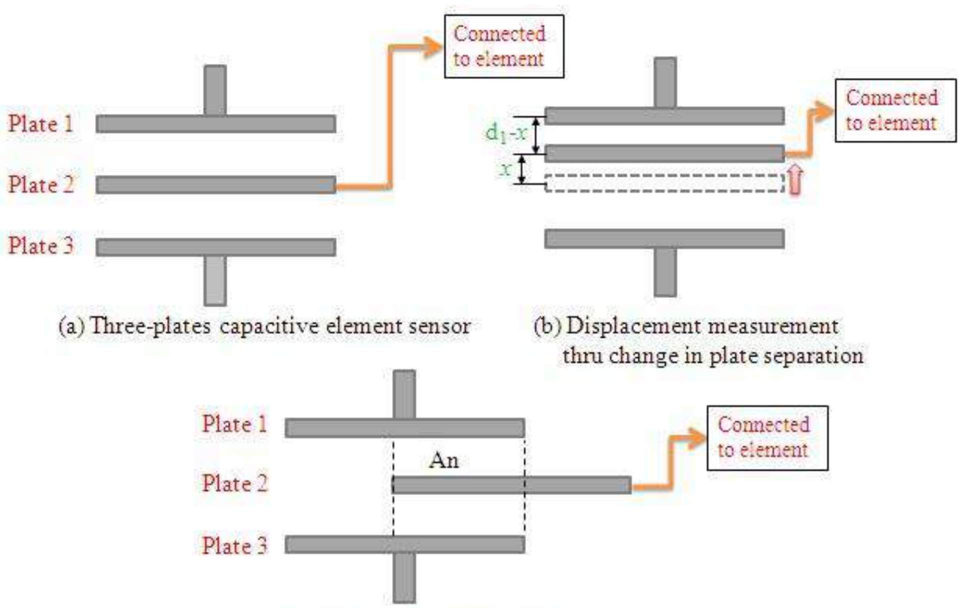
3. Capacitive element based sensor

Capacitive sensor is of non-contact type sensor and is primarily used to measure the linear displacements from few millimeters to hundreds of millimeters. It comprises of three plates, with the upper pair forming one capacitor and the lower pair another. The linear displacement might take in two forms:

(a) one of the plates is moved by the displacement so that the plate separation changes

(b) area of overlap changes due to the displacement.

Figure 2.2.5 shows the schematic of three-plate capacitive element sensor and displacement measurement of a mechanical element connected to the plate 2.



(a) Three-plates capacitive element sensor
(b) Displacement measurement thru change in plate separation
(c) Displacement measurement thru change in area of overlap

4. Linear variable differential transformer (LVDT)

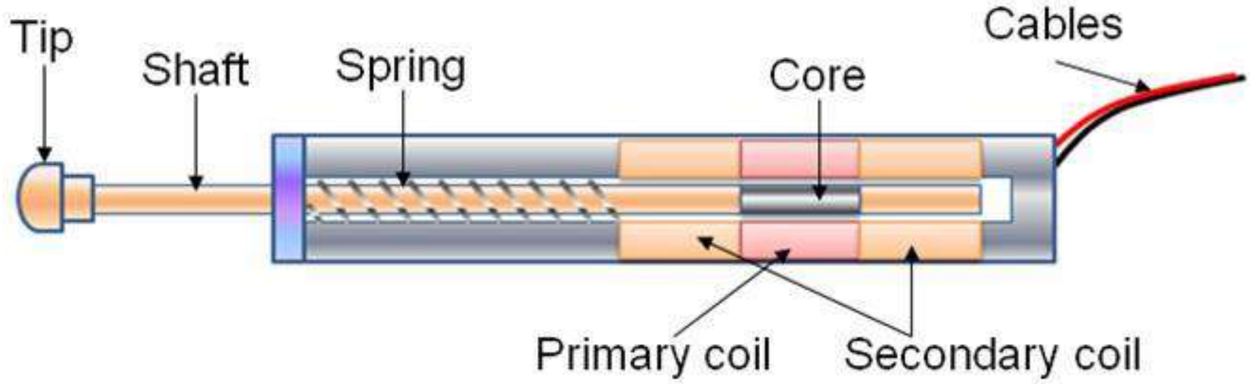
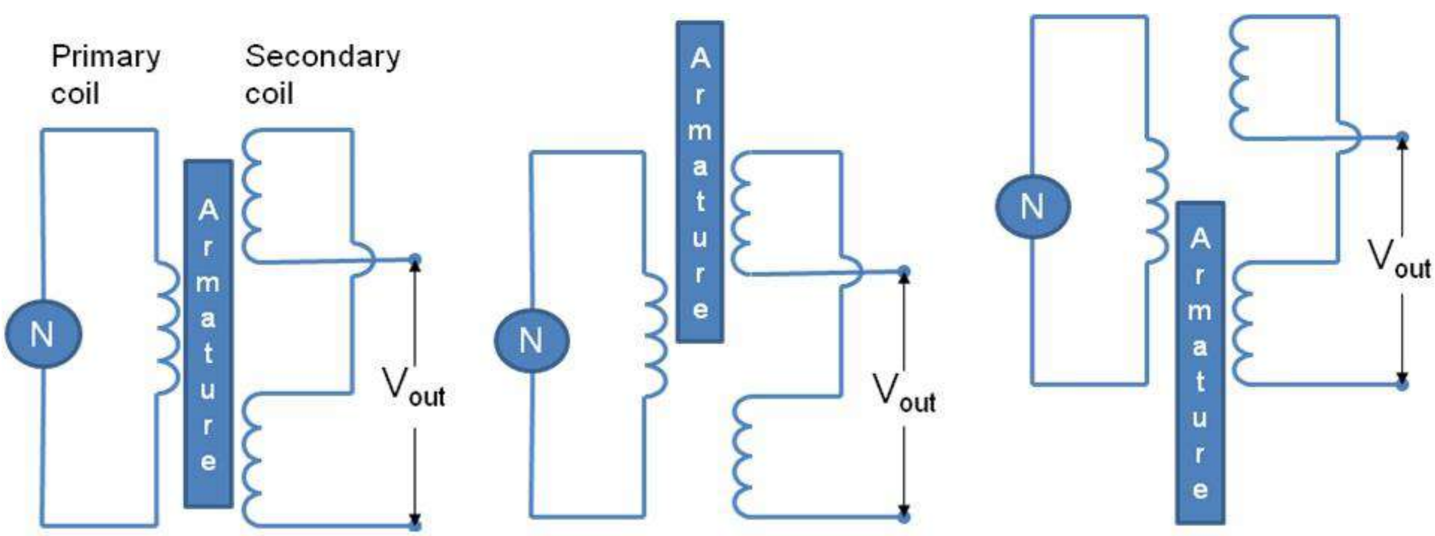


Figure 2.2.6 Construction of LVDT sensor

Linear variable differential transformer (LVDT) is a primary transducer used for measurement of linear displacement with an input range of about ± 2 to ± 400 mm in general. It has non-linearity error ± 0.25% of full range. Figure 2.2.6 shows the construction of a LVDT sensor. It has three coils symmetrically spaced along an insulated tube. The central coil is primary coil and the other two are secondary coils. Secondary coils are connected in series in such a way that their outputs oppose each other. A magnetic core attached to the element of which displacement is to be monitored is placed inside the insulated tube.



5. Bimetallic strips

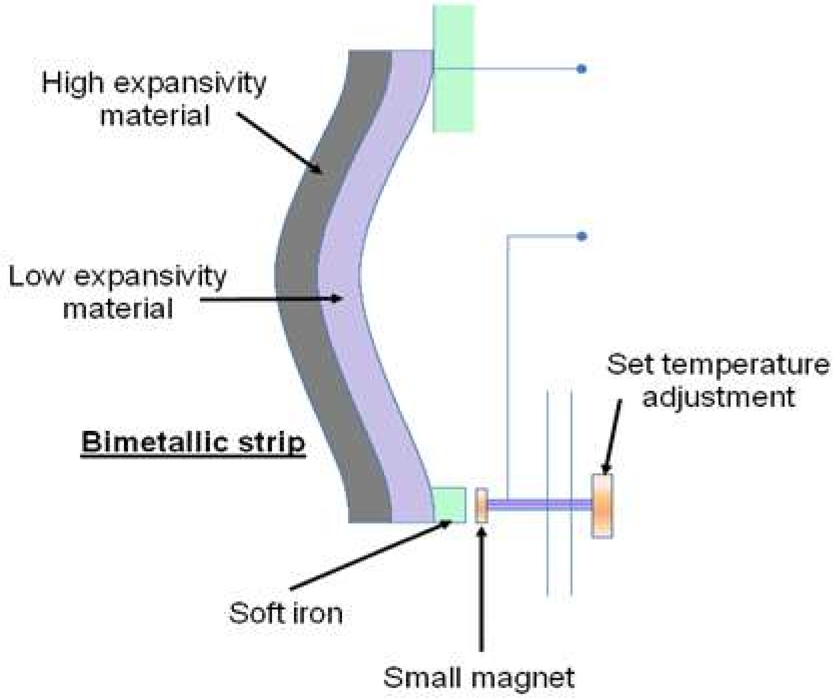


Figure 2.5.1 Construction and working of Bi-metallic strip

Bimetallic strips are used as thermal switch in controlling the temperature or heat in a manufacturing process or system. It contains two different metal strips bonded together. The metals have different coefficients of expansion. On heating the strips bend into curved strips with the metal with higher coefficient of expansion on the outside of the curve. Figure 2.5.1 shows a typical arrangement of a bimetallic strip used with a setting-up magnet. As the strips bend, the soft iron comes in closer proximity of the small magnet and further touches. Then the electric circuit completes and generates an alarm. In this way bimetallic strips help to protect the desired application from heating above the pre-set value of temperature.

6.Thermocouple

Thermocouple works on the fact that when a junction of dissimilar metals heated, it produces an electric potential related to temperature. As per Thomas Seebeck (1821), when two wires composed of dissimilar metals are joined at both ends and one of the ends is heated, then there is a continuous current which flows in the thermoelectric circuit. Figure 2.5.5 shows the schematic of thermocouple circuit. The net open circuit voltage (the Seebeck voltage) is a function of junction temperature and composition of two metals. It is given by,

$$\Delta V_{AB} = \alpha \Delta T \qquad (2.5.2)$$

Where α, the Seebeck coefficient, is the constant ofproportionality.

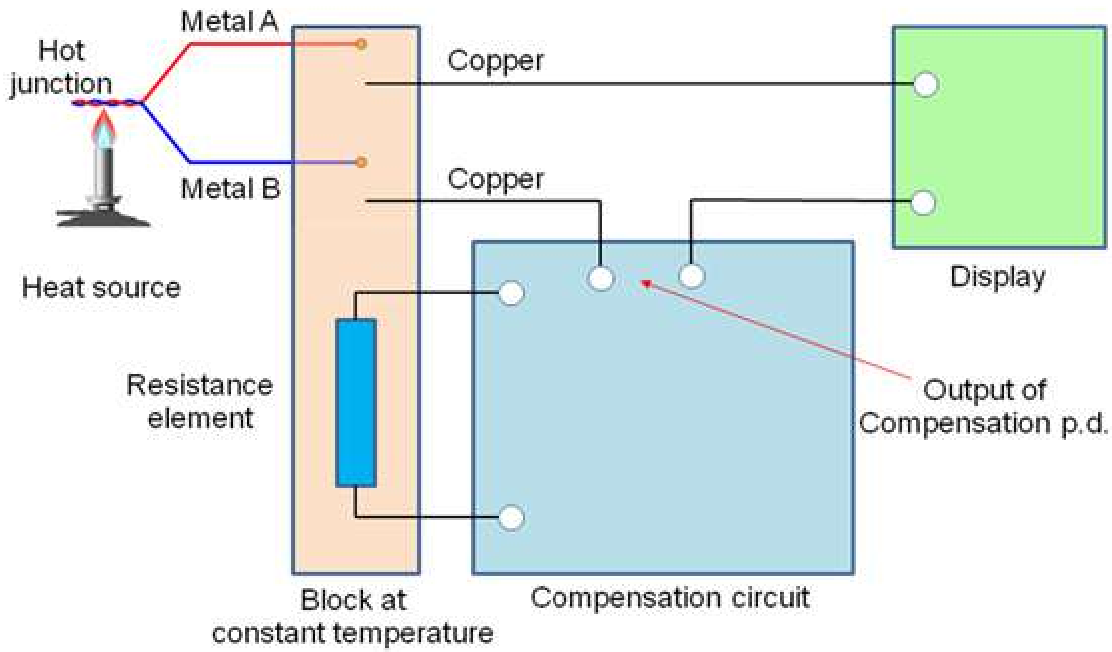


Figure 2.5.5 Schematic of thermocouple circuit

Generally, Chromel(90% nickel and 10% chromium)–Alumel(95% nickel, 2% manganese, 2% aluminium and 1% silicon) are used in the manufacture of a thermocouple. Table 2.5.1 shows the various other materials, their combinations and application temperature ranges.

Tachogenerator

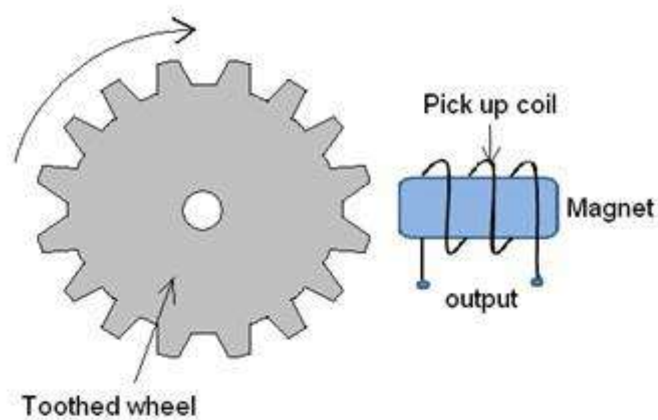


Figure 2.4.1 Principle of working of Techogenerator

Tachogenerator works on the principle of variable reluctance. It consists of an assembly of a toothed wheel and a magnetic circuit as shown in figure 2.4.1. Toothed wheel is mounted on the shaft or the element of which angular motion is to be measured. Magnetic circuit comprising of a coil wound on a ferromagnetic material core. As the wheel rotates, the air gap between wheel tooth and magnetic core changes which results in cyclic change in flux linked with the coil. The alternating emf generated is the measure of angular motion. A pulse shaping signal conditioner is used to transform the output into a number of pulses which can be counted by a counter.

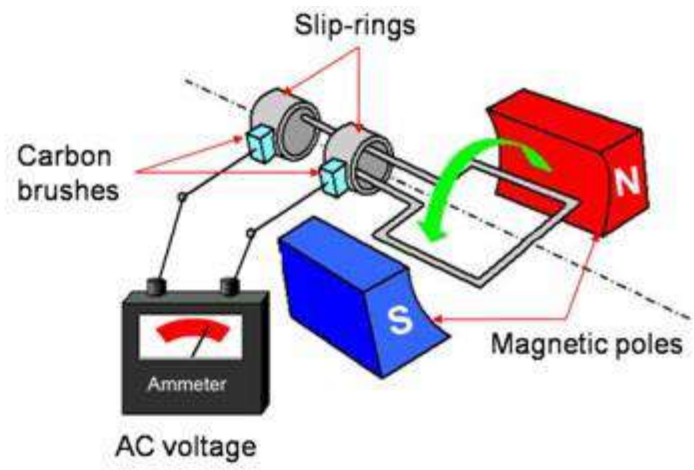


Figure 2.4.2 Construction and working of AC generator

An alternating current (AC) generator can also be used as a techognerator. It comprises of rotor coil which rotates with the shaft. Figure 2.4.2 shows the schematic of AC generator. The rotor rotates in the magnetic field produced by a stationary permanent magnet or electromagnet. During this process, an alternating emf is produced which is the measure of the angular velocity of the rotor. In general, these sensors exhibit nonlinearity error of about ± 0.15% and are employed for the rotationsup to about 10000 rev/min.

Strain Gauge as force Sensor

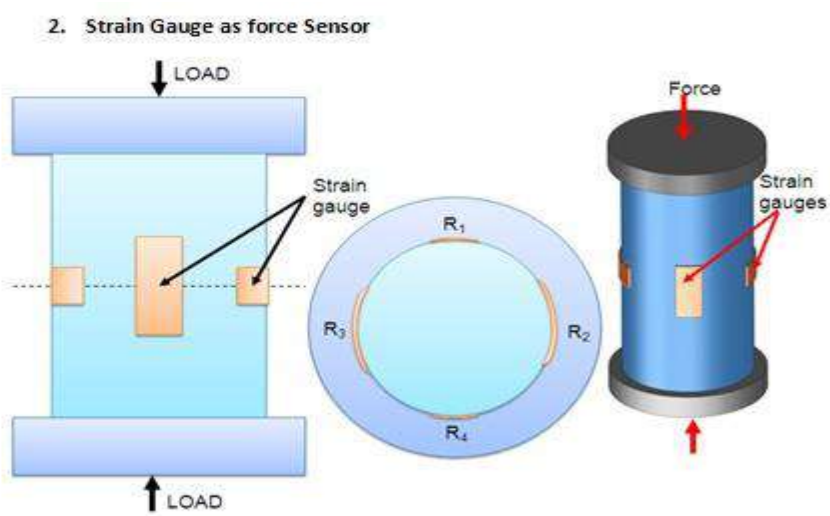


Figure 2.4.5 Strain gauge based Load cell

Strain gauge based sensors work on the principle of change in electrical resistance. When, a mechanical element subjects to a tension or a compression the electric resistance of the material changes. This is used to measure the force acted upon the element. The details regarding the construction of strain gauge transducer are already presented in Lecture 2 of Module 2.

Figure 2.4.5 shows a strain gauge load cell. It comprises of cylindrical tube to which strain gauges are attached. A load applied on the top collar of the cylinder compress the strain gauge element which changes its electrical resistance. Generally strain gauges are used to measure forces up to 10 MN. The non-linearity and repeatability errorsof this transducer are ±0.03% and ±0.02% respectively.

4. Fluid pressure

Chemical, petroleum, power industry often need to monitor fluid pressure. Various types of instruments such as diaphragms, capsules, and bellows are used to monitor the fluid pressure. Specially designed strain gauges doped in diaphragms are generally used to measure the inlet manifold pressure in applications such as automobiles. A typical arrangement of strain gauges on a diaphragm is shown in figure 2.4.6. Application of pressurized fluid displaces the diaphragm. This displacement is measured by the stain gauges in terms of radial and/or lateral strains. These strain gauges are connected to form the arms of a Wheatstone bridge.

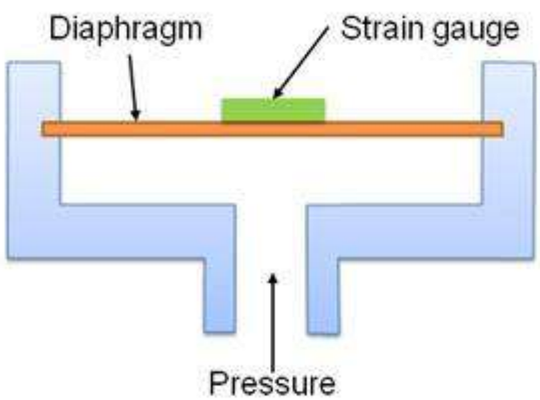


Figure 2.4.6 A diaphragm

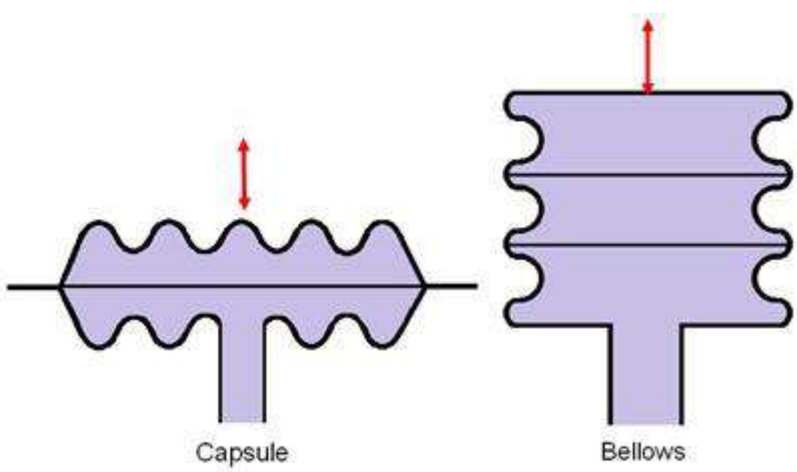


Figure 2.4.7 Schematic of Capsule and Bellow

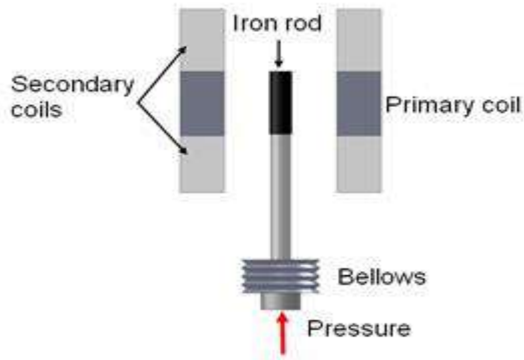


Figure 2.4.8 Bellow with a LVDT

Capsule is formed by combining two corrugated diaphragms. It has enhanced sensitivity in comparison with that of diaphragms. Figure 2.4.7 shows a schematic of a Capsule and a Bellow. A stack of capsules is called as 'Bellows'. Bellows with a LVDT sensor measures the fluid pressure in terms of change in resultant voltage across the secondary coils of LVDT. Figure 2.4.8 shows a typical arrangement of the same.

VIVA VOCE QUESTIONS & ANSWERS

1. Expand CATIAV5?
Computer Aided Three Dimensional Interactive Application.

2. What is the save extension of sketcher file?
CAT Part

3. Does CATIA V5 work on UNIX Platform?
Yes

4. Is it possible to increase the size of plane boundary representation & how?
Yes, go for Tools-Options- Infrastructure-Part structure-Display

5. Is It Possible to directly enter in to Sketcher Workbench?
No, it is not possible to enter in to sketcher workbench directly. We have to go for any workbench &form there we can enter the sketcher workbench…

6. Which is the tool used to exit from sketcher workbench to part design Workbench?
Exit Sketcher

7. What is the use of construction elements?
Construction elements assist in sketching the required profile in sketcher.

8. What are the default units of LMT (Length, Mass and Time)
mm, Kg, Second.

9. What is SKETCH TOOLS in sketcher work bench & Explain the Importance of it?
SKETCH TOOLS in sketcher workbench are the commands, which find very use in creating sketches. SKETCH TOOLS are namely geometric and dimensional constraints, construction elements/standard elements and Grid option. They play very important role in sketching, whenever we want to constrain a sketch we use these options and if we want to convert any element into a construction element once again these options come into picture.

10. Is it possible to hide specification tree?
Yes, with help of F3 button, but the option in Tools command must be checked to allow this.

11. What is SHOW/HIDE option?
Show mode enables us to see all the components presently opened and in Hide mode we can hide the desired elements from the view for time being.

12. What is the use of Cut Part by Sketch Plane?
This task shows how to make some edges visible. In other words, we are going to simplify the sketch plane by hiding the portion of the material that is not needed for sketching.

13. How do you measure arc length?
We can measure arc length by using MEASURE ITEM command. Sometimes we need to customize the option for arc length if it is not checked earlier using customization in MEASURE ITEM command.

14. What is the meaning of true dimension?
True dimension is the dimension desired after the machining. In other words, this is the value that should be attained after the machining.

15. What do you mean by ISO-Constraints?
If all of the degrees of freedom of geometry have been takes up by a consistent combination of dimensions & fixed geometry. That geometry is said to be ISO-CONSTRAINED. Geometry that still has some degrees of freedom is said to be UNDER constrained.

16. Mention the color code of ISO-constrained, under, and over con strained elements?
The color code for these elements is Green, White and Magenta respectively.

17. What for the animated constraint command is used?
This task shows how constrained sketched element reacts when we decide to vary one constraint.

18. How many dimensions are required to constrain the ellipse?
Three dimensions are required namely major axis, minor axis and the distance from the origin.
19. What are different conic sections?
Ellipse, Hyperbola and Parabola
20. What is RHO value for ellipse, Parabola and hyperbola?
Parabola has RHO values of 0.5, Ellipse has RHO value between 0 & 0.5 and Hyperbola has RHO value b/w 0.5 &1.0.
21. What is NURBS?
Nurbs are the type of curves
22. How many types of Co-ordinate systems are there?
Three namely Cartesian, Polar and Spherical co-ordinate system.
23. What are project 3D silhouette edges?
Project 3D silhouette edges in sketcher will shows how to create silhouette edges to be used in as geometry or reference elements.
24. What is use of sketch analysis?
To check whether the sketch is fully closed or not so that the sketch can be used or not so that the sketch can be used for further operations in part design.
25. Where do we use axis?
Axis is used in creating shaft (revolved) feature
26. Can we redefine the sketches?
Yes.
27. Can axis be converted into line or vice versa?
We can convert line into axis but axis cannot convert into line.
28. How many axis can be created in a single sketch?
Only one axis can be created in a sketch, if more than one axis are drawn then only one of them, the latest one, will be axis and others will be converted into reference elements.
29. How do we change, sketch's reference plane?
Right click on the sketch whose reference plane is to be changed and select the change reference plane and then select new reference.
30. What is the function of mirror command in sketch?
Mirror command in sketch will create a copy of the sketch about a reference plane.
31. If I donor want the relation b/w original and mirrored elements what should I do?
Explore and the relation b/w the original and mirror element doesn't exit.
32. What is the use of isolate in sketcher workbench?
Isolated is used when 3D geometry is projected on to a sketch in order to be modified and used as part of the sketch's profile.
33. Can we select non-planer surf ace as sketch plane?
No, we cannot select a non-planar surface as sketch.
34. What are the different options available in quick trim command?
BREAK & RUBBER in removes part of the element, which is clicked. BREAK & RUBBER out removes part of the element, which is not clicked. BREAK & KEEP keeps both parts of elements after breaking.
35. What do CATIA P1, CATIA P2 AND CATIA P3 mean?
P1, P2 and P3 will indicate the different types of platforms of CATIA. Higher the number sophisticated will be the software.

36. What is kernel?
The kernel is the basic indispensable part of an operating system that allocates resources such as low-level hardware interfaces & security.
38. What is the importance of sketch tools?
This tool bar only appears when you are in sketcher workbench. The four tools found in this tool bar are toggle tools. When tool is highlighted the tool is on. This particular tool bar changes depending on what other sketcher workbench tool is currently selected.
39 .How many degrees of freedom are there for points, lines, circles & ellipse in 2dimensions?
Degree of freedom for points & ellipse is 2 for circles it is 3 & for ellipse it is 5 in two dimensions.
40. What is the meaning of mean dimension?
Mean dimension is the dimension that should be mean of all the dimensions, which are tolerance.
41. How many types of environment are available to start CATIA?
1. From desktop (motif)
2. From console (dterm)