# COMPUTER AIDED DESIGN LAB

## LIST OF EXPERIMENTS

### CAD LAB

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### CAM LAB

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INTRODUCTION TO GRAPHICS PACKAGE

The engineering drawing has been and is an integral part of industry and it is a link between engineering design and manufacturing. Information is quickly communicated in the form of drawings prepared according to prescribed drafting standards.

What is Computer Aided Design?

The use of interactive graphics programs to develop 2D and 3D models, assemblies, part lists, and dimensional drawings of various components, structures or objects with the help of a computer is called Computer-Aided Design (CAD). The data generated by a CAD system can be directly utilized by a CAM (Computer Aided Manufacturing) system. Thus CAD and CAM are interrelated to each other.

Advantages of CAD

- Reduction of drafting labor.
- Direct cost savings.
- High accuracy (up to one millionth of a unit)
- Improvement in the general flow of information through the company.
- Evaluation of alternative designs.
- Use of common parts in multiple products.

Before we design and manufacture any component, the object must be presented in the form of drawings along with different views and dimensions. Further, the bill of materials etc. must also be supplied. With the advent of computers and relevant software packages, the drafting drifted from manual to computer aided drafting. The networking between the shop floor sections and design sections makes it easier to share the data and information. The modifications (in the drawings, materials or process of operation) if required, can be discussed on line and can be implemented without any time lag. An attempt is made here to introduce the graphics package in the following topics.
1. INITIATING THE GRAPHICS PACKAGE

**Aim:** Setting the paper size, space; setting the limits, units; use of snap and grid commands.

**Requirements:** Auto CAD software, PC with a min. of 256MB RAM.

**Description:**

LIMITS: sets and controls the drawing boundaries and grid display.

The drawing limits are two-dimensional points in the World Co-ordinate System that represent a lower-left limit and an upper-right limit. Limits cannot be imposed on the Z direction.

Turn on the drawing limits and restrict within the rectangular area. Drawing limits also determine the area of the drawing that can be displayed with or without grid dots.

Command: LIMITS

```
ON/ OFF/ Lower left corner < current> :
```

- **On:** Turns on limits checking. When the “limits checking” is on, AutoCAD rejects attempts to enter points outside the drawing limits.

- **Off:** Turns off limits checking, but maintains the current values, for the next time when the “limits checking” is turned on.

**Procedure:** Enter the command “LIMITS”.

Reset model space limits.

Specify lower left corner or [ON/OFF], < 0.0000, 0.0000 > : 0,0 (return)

Specify upper right corner < 120.0000, 120.0000 > : 120,120 ( or any value) (return)

Command: zoom (return)

**Use of Snap And Grid Commands**

GRID displays a dot grid

At the command prompt, enter ‘GRID’.

Command: GRID

```
Grid Spacing ( X ) or ON / OFF / Snap / Aspect / Current > :
```

You can turn grid on and off with F7 key or by double clicking the GRID button in the status bar.
SNAP restricts cursor movement to specified intervals
At the command prompt, enter ‘SNAP’.

The points you enter with a pointing device can be locked into alignment by SNAP. You can rotate the SNAP grid, set differing X and Y spacing, or choose an isometric format for the snap grid.

A change in the snap grid affects only the co-ordinate of new points. Objects already in the drawing retain their existing co-ordinates.
The snap grid is invisible. Use SNAP with GRID to display a separate visible grid of dots. Set the spacing of two grids to equal or related values dots.

Command: SNAP

Snap spacing or ON / OFF / Aspect / Rotate / Style < Current > :

- **Snap spacing:** Actives snap mode using the current snap grid resolution, rotation and style.
- **ON:** Actives snap mode using the current snap grid resolution, rotation and style.
- **OFF:** Deactivates snap mode but retains the values and modes.
- **Rotate:** Sets the rotation of the snap grid with respect to the drawing and the display screen. Any angle between –90 and + 90 can be specified. A positive angle rotates the grid counterclockwise about the base point.

  Base point <current > :

  Rotation angle < current >:

- **Style:** Selects the format of the snap grid, which is standard or Isometric. Standard displays a rectangular grid that is parallel to the XY plane of the current UCS. X and Y spacing may differ
- **Spacing:** Changes overall spacing.
- **Aspect:** Changes the horizontal and vertical spacing individually.
  Isometric displays an isometric grid where the grid points are initially at 30 and 150-degree angles.

  Vertical spacing < current >:
2. DRAWING OF PRIMITIVES

Aim: Drawing various graphics primitives.

Requirements: Auto CAD software, PC with a min. of 256MB RAM.

Description: The command “LINE” creates straight line segments. From the draw menu, select Line. At the command prompt, enter Line or L. Line command allows creating a line where the end points are specified in two-dimensional or three-dimensional coordinates.

Procedure:

Command: limits (return)
Reset model space limits:
Specify lower left corner or [ON/OFF], < 0.0000, 0.0000 > : 0,0 (return)
Specify upper right corner < 120.0000, 120.0000 > : 120,120 ( or any value) (return)
Command: zoom (return)
Command: line (return)
Specify first (point P1): 50,50 (return)
Specify next point (P2): 100,100 (return)

AutoCAD prompts for points, allowing to draw continuous line segments.

U (Undoing a line)
This allows erasing the most recent segment in a line sequence
C (closing a line sequence)
This allows closing a sequence of continuous line segments, resulting in a polygon.

You may specify the points randomly with the mouse or by any of the coordinate systems.
Lines can be drawn, using absolute mode (eg.100,100) or incremental mode (eg.@100,100) or polar mode(eg.100<90).

**Circle**

The command “CIRCLE” creates a circle.

From Draw menu, select Circle.

At the command prompt enter CIRCLE OR C.

Specify center point for circle: 50,50 (return)

Specify radius of circle [or diameter]: 50 (return)

The other options for drawing a circle are 2P, 3P and TTR.

With 2P option, the user defines two points. These two points are at the diameter of the circle. 3P option allows you to define 3 non-collinear points. The circle fits in these three points. TTR option allows drawing a circle by defining two tangent points and specifying a radius. AutoCAD by default asks for a centre point and radius.
**ARC**

*ARC allows to create an arc segment*

From Draw menu, select Arc
At the command prompt, enter ARC or A
Command: ARC
Center / < start point >: specify a point P1
Start point - This specifies the start point of an arc
Center / End/ < second point>:
Specify a point P2
Second point - This draws an arc using three specific points along the circumference of the arc. The first and the third points from the end points of the arc. The second point can be any point on the circumference of the arc.

![Diagram of an arc with points P1, P2, and P3]

**RECTANG**

RECTANG draws a rectangular polyline
First corner : specify a point P1
Other corner : specify a point P2

![Diagram of a rectangle with points P1, P2, P3, and P4]
POLYGON

*POLYGON allows you to create polygons with 3 to 1024 numbers of sides.*

From Draw menu, select Polygon
At the command prompt, enter POLYGON or POL

Command: POLYGON

Number of sides < current>

Edge /< centre of polygon >:

This specifies the number of sides the polygon must have. AutoCAD accepts any value between 3 to 1024

Centre of polygon

Defines the center point of the polygon. Specify a point

Inscribed in circle / Circumscribed about circle ( I/C ) < I >:

**Inscribed in circle**

This specifies the radius of a circle on which all vertices of the polygon lie.

**Circumscribed about circle**

This specifies the distance from the centre of polygon to the midpoints of the edges of the polygon.

Radius of circle:

**Edge**

Defines a polygon by specifying the endpoints of the first edge.

First endpoint of edge

Second endpoint on edge
ELLIPSE

ELLIPSE creates an ellipse or an elliptical arc

From Draw menu, select Ellipse

At the command prompt, enter Ellipse or EI

Command: ELLIPSE

Arc/ center/ <Axis endpoint 1> :

Axis endpoint 1:
This is the first step towards defining the first axis. The two end points of the axis are specified. When the first point is specified in the above option, it prompts for the second point.

Axis endpoint 2 :
< other axis distance > / Rotation :

The first axis can define either the major or the minor axis of the ellipse.

Other axis distance

Defines the second axis as the distance from the centre of ellipse in a direction perpendicular to the first axis.

![Ellipse by axis end points](image)

![Ellipse by centre point](image)
3. FLANGE

Aim: Drawing a flange

Procedure:

FIRST STEP
Draw PLINE of required cross section
PLINE creates two-dimensional poly lines
A 2D poly line is a connected sequence of a line and arc segments, and is created by AutoCAD as a single object using region command.

SECOND STEP
REVOLVE the drawn pline about required axis and a specified angle
Command: Revolve
REVOLVE creates a solid by revolving a two-dimensional object about an axis.
Select objects: Use object selection method
Axis of revolution – Object / X / Y / < Start point axis > : Specify a point or enter an option.
Start point of axis:
Specifies the first and second point of the axis if revolution.
Angle of revolution <full circle> : Specify an angle or press Enter
THIRD STEP

Draw a circle on the face of the flange and use the polar array command for making number of holes on the face.

ARRAY

ARRAY creates multiple copies of objects in a pattern.
From modify menu, select Array.
At the command prompt, enter ARRAY or AR
Select objects: select objects by any of the selection methods.
Rectangular / Polar array ( R/P) < current> :
Polar array
This creates an array about a defined center point
Center point of array:
Number point of items:
Angle to fill ( + = ccw, - = cw ) < 360 > :
Rotate objects as they are copied ? < Y > :
Angle between items:
Rotate objects as they are copied? < Y > :

Object arrayed
FOURTH STEP
Extrude all the six components to the required path and distance using EXTRUDE command. Then subtract all the components from the flange. You will get a required flange as shown below.
4. BUSHING ASSEMBLY

Aim: Drawing a bushing assembly

Procedure:

FIRST STEP
Draw PLINE of required cross section
PLINE creates two-dimensional polylines
A 2D polyline is a connected sequence of a line and arc segments, and is created by AutoCAD as a single object.

SECOND STEP
REVOLVE the drawn pline about required axis and a specified angle
Command: Revolve
REVOLVE creates a solid by revolving a two-dimensional object about an axis.
Select objects: Use object selection method
Axis of revolution – Object / X / Y / < Start point axis >: Specify a point or enter an option.
Start point of axis:
Specifies the first and second point of the axis if revolution.
Angle of revolution <full circle >: Specify an angle or press Enter

THIRD STEP

Repeat the same procedure for drawing the bearing
FOURTH STEP

Assemble the bush and bearing
5. DIMENSIONING THE DRAWING AND ADDING TEXT

Aim: To dimension the drawing and to add text.

Description:
DIMLINEAR: DIMLINEAR creates linear dimensions
From Dimension menu, select Linear
At the command prompt, enter DIMLINEAR or DLI
Command: DIMLINEAR
First extension line origin or RETURN to select: Specify a point or press Enter for automatic extension lines

Point specification
If you specify a point (P1) for the first extension line, AutoCAD prompts for the second extension line origin
Second extension line origin: specify a point (P2)
Object selection-Automatic extension lines
If you press Enter to select an object, AutoCAD automatically determines the origin points of the first and second extension lines.
If you select a circle, the endpoints of its diameter are used as the origin points. When the point used to select the circle is close to the north or south quadrant point,
AutoCAD draws a horizontal dimension. When the point used to select the circle is close to the east or west quadrant point, AutoCAD draws a vertical dimension.

Dimension line location (Text/Angle/Horizontal/Vertical/Rotated):
Specify a point (P2) or enter an option.

**Text:** Allows you to customize the text.

**Angle:** Changes the angle of the dimension text.

- Enter text angle: specify an angle
- After you specify the angle, AutoCAD prompts
- Dimension line location (Text/Angle/Horizontal/Vertical/Rotated)

**Horizontal:** Creates horizontal linear dimensions

- Dimension line location (Text/ Angle): Specify a point or enter an option

**Vertical:** Creates vertical dimensions

- Dimension line location (Text/ Angle): Specify a point or enter an option

**Rotate:** Creates rotated linear dimensions

- Dimension line angle<current>: specify an angle or press enter

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**DIMALIGNED**

DIMALIGNED creates an aligned linear dimension

From dimension menu, select Aligned.

At the command prompt, enter DIMALIGNED or DAL

Command: DIMALIGNED

First extension line origin or RETURN to select: Specify a point for manual extension lines, or press enter for automatic extension lines.

Object selection for Automatic extension lines Automatically determines the origin points of the first and second extension lines.

Select object to dimension: Select an object
DIMRADIUS
DIMRADIUS Creates radial dimensions for circles and arcs
From Dimension menu, select Radius
At the command prompt, enter DIMRADIUS or DRA
Command: DIMRADIUS
Select arc or circle: select an arc or a circle
AutoCAD measures the radius and displays the text with the letter R in the front of it
Dimension line location (Text/Angle): specify a point or enter an option
Text
Dimension text < measured value>: Enter text or press
Angle
Enter text angle: specify an angle

DIMDIAMETER
DIMDIAMETER creates diameter dimensions for circles and arcs
From Dimension menu, select Diameter
At the command: prompt, enter DIMDIAMETER or DDI
Command: DIMDIAMETER
Select arc or circle: select an arc or circle
AutoCAD measures the diameter and displays the text with symbol in front of it. The next prompt is
Dimension line location (Text/ Angle): *specify a point or enter an option*

Text

Dimension text < measured value>: *Enter text or press return*

Dimension line location:

**Angle**

Changes the angle of the dimension text

Enter text angle: *specify an angle*

After you specify the angle, AutoCAD redisplays

Dimension line location (Text/ Angle):

**DIMANGULAR**

DIMANGULAR creates angular dimensions.

From dimension menu, select Angular

At the command prompt, enter DIMANGULAR or DAN

Command: DIMANGULAR

Select arc, circle, line or RETURN

Select the appropriate object or press enter to specify three points

**Three point specification**

Angle vertex: Specify a point (P1)

First angle end point: specify a point (P2)

Second angle end point: *specify a point (P3)*
Arc selection
If you select an arc, AutoCAD determines the defining points for a three-point angular dimension. The center of the arc is the angle vertex. The arc endpoints become the origin points of the extension lines.

Circle selection
If you select a circle, the centre of the circle is the angle vertex. The selection point is used as the origin of the first extension line. AutoCAD draws the dimension line as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

DIMBASELINE
DIMBASELINE creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.
From the Dimension menu, select Baseline or DBA.
Command: DIMBASELINE
Second extension line origin or RETURN to select: specify a point or press to select a base dimension

If the previous dimension was linear, angular or ordinate, AutoCAD uses the first extension line origin of that dimension for the first extension line origin of the baseline dimension. After you specify a point, AutoCAD draws the baseline dimension and redisplayed the prompt.

Second extension line origin
If the previous dimension was linear, angular or ordinate, if you press enter at the second extension line origin prompt, AutoCAD prompts you to select linear, angular or ordinate dimension to use as the basis for the baseline dimension.
Select base dimension: *Select a linear, angular or ordinate dimension*

**DIMCONTINUE**

DIMCONTINUE continues a linear, angular or ordinate dimension from the second extension line of the previous or a selected dimension.

From Dimension menu, select Continue

At the command: prompt enter DIMCONTINUE or DCO

Command: DIMCONTINUE

If the previous dimension was linear, angular or ordinate, AutoCAD uses the origin of that dimension’s second extension line for the origin of the next dimension’s first extension line.

Select extension line origin or RETURN to select:

Specify a point or press enter to select a continued dimension

After you specify a point, AutoCAD draws the next dimension. AutoCAD redispaly the prompt

Select extension line origin or RETURN to select:

To end the command, press enter
DIMORDINATE
DIMORDINATE creates ordinate point dimensions
From Dimension menu, select Ordinate.
At the Command prompt, enter DIMORDINATE or DOR
Ordinate dimensions display the X or Y coordinate of a feature along with a simple leader line. These dimensions are also known as datum dimensions. AutoCAD uses the current UCS to determine the measurement.
Command: DIMORDINATE
Select feature: specify a point or snap to an object
Leader endpoint (X datum /Y datum /Text): Specify a point or enter an option
TEXT OBJECT

Description:

You can use TEXT to enter several lines of text that you can rotate, justify, and resize. As you type at the Enter Text prompt, the text you are typing is displayed on the screen. Each line of text is a separate object. To end a line and begin another, press ENTER after entering characters at the Enter Text prompt. To end TEXT, press ENTER without entering any characters at the Enter Text prompt.

By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

Procedure:

Draw menu: Text   Single Line Text
Command line: text
Current text style: current
Current text height: current

Specify start point of text or [Justify/Style]: Specify a point or enter an option
If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle and immediately displays the Enter Text prompt. The text is placed directly beneath the previous line of text. The point specified at the prompt is also stored as the insertion point of the text.

START POINT

Specifies a start point for the text object.
Specify height <current>: Specify a point (1), enter a value, or press ENTER
The Specify Height prompt is displayed only if the current text style does not have a fixed height.
Specify rotation angle of text <current>: Specify an angle or press ENTER
Enter text: Enter text and press ENTER to exit the command

JUSTIFY

Controls justification of the text.
Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

You can also enter any of these options at the Specify Start Point of Text prompt.
Align

Specifies both text height and text orientation by designating the endpoints of the baseline.
Specify first endpoint of text baseline: Specify a point (1)
Specify second endpoint of text baseline: Specify a point (2)
Enter text: Enter text and press ENTER to exit the command
The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.

Fit

Specifies the text that fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.
Specify first endpoint of text baseline: Specify a point (1)
Specify second endpoint of text baseline: Specify a point (2)
Specify height <current>:
Enter text: Enter text and press ENTER to exit the command
The height is the distance in drawing units that the uppercase letters extend from the baseline. Designated text height is the distance between the start point and a point you specify. The longer the text string, the narrower the characters. The height of the characters remains constant.

Center

Aligns text from the horizontal center of the baseline, which you specify with a point.
Specify center point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command
The rotation angle specifies the orientation of the text baseline with respect to the center point. You can designate the angle by specifying a point. The text baseline runs from the start point toward the specified point. If you specify a point to the left of the center point, AutoCAD draws the text upside down.
Middle

Aligns text at the horizontal center of the baseline and the vertical center of the height you specify. Middle-aligned text does not rest on the baseline.
Specify middle point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command
The Middle option differs from the MC option in that it uses the midpoint of all text, including descenders. The MC option uses the midpoint of the height of uppercase letters.

Right

Right-justifies the text at the baseline, which you specify with a point.
Specify right endpoint of text baseline: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

TL (Top Left)

Left-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.
Specify top-left point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

TC (Top Center)

Centers text at a point specified for the top of the text. Available for horizontally oriented text only.
Specify top-center point of text: Specify a point (1)
Specify height <current>: 
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

TR (Top Right)
Right-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.
Specify top-right point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

ML (Middle Left)
Left-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.
Specify middle-left point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

MC (Middle Center)
Centers the text both horizontally and vertically at the middle of the text. Available for horizontally oriented text only.
Specify middle-center point of text: Specify a point (1)
Specify height of text <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

The MC option differs from the Middle option in that it uses the midpoint of the height of uppercase letters. The Middle option uses the midpoint of all text, including descenders.

MR (Middle Right)
Right-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.
Specify middle-right point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

**BL (Bottom Left)**
Left-justifies text at a point specified for the baseline. Available for horizontally oriented text only.
Specify bottom-left point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

**BC (Bottom Center)**
Centers text at a point specified for the baseline. Available for horizontally oriented text only.
Specify bottom-center point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command

**BR (Bottom Right)**
Right-justifies text at a point specified for the baseline. Available for horizontally oriented text only.
Specify bottom-right point of text: Specify a point (1)
Specify height <current>:
Specify rotation angle of text <current>:
Enter text: Enter text and press ENTER to exit the command
6. SETTING THE LAYERS AND APPLICATION OF LAYERS

Aim: To set the layers and to learn and manage the layers.

Description:

**LAYER:** The command “LAYER” manages layers. It makes a layer current, adds new layers, deletes layers and renames layers. One can assign properties to layers, turn layers *on and off*, freeze and thaw layers globally or by view port, lock and unlock layers, set plot styles for layers, and turn plotting on and off for layers. Also, one can filter the layer names displayed in the Layer Properties Manager, and can save and restore layer states and properties settings.

By creating layers, one can associate similar types of objects by assigning them to the same layer. For example, construction lines, text, dimensions, and title blocks can be put on separate layers and can then be controlled to check:

- Whether the objects on a layer are visible in any view ports
- Whether and how the objects are plotted
- What color is assigned to all the objects on a layer
- What default line type and line weight is assigned to all objects on a layer
- Whether objects on a layer can be modified

When a new drawing is begun, AutoCAD creates a special layer named 0. By default, layer 0 is assigned colour number 7 (white or black depending upon your background colour), the **CONTINUOUS** line type, a line weight of Default (the default setting is .01 inch or .25 mm), and the **NORMAL** plot style. Layer 0 cannot be deleted or renamed.

Procedure:

From Format menu, select Layer

At the command prompt, enter LAYER or LA

The Layer & Line type properties dialog box appears

Command: LAYER

Various icons and buttons in the dialog box indicate various statuses of layers and layer management tools.
Named Layer Filters
Determines which layers to display in the list of layers. You can filter layers based on whether they're xref-dependent, or whether they contain objects. You can also filter layers based on name, visibility, color, line type, line weight, plot style name, whether they are plotted, or whether they are frozen in the current view port or in new view ports.

[...] Button: Displays the Named layer filters dialog box

Invert Filter: Displays layers based on the opposites of the criteria in a named layer filter. Layers that fit the inverse criteria are displayed in the layer name list.

New
Creates a new layer. After you choose New, the list displays a layer named LAYER1. You can edit this layer immediately. To create multiple layers more quickly, you can select a layer name for editing and enter multiple layer names separated by commas. If you create a new layer, the new layer inherits the properties of the currently selected layer in the layer list (Color, On/Off state, and so on). To create layers with default settings, make sure that there are no selected layers in the list or that you select a layer with default settings before beginning layer creation.

Current
Sets the selected layer as the current layer

Delete
Deletes selected layers from the drawing file definition. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects, the current layer, and xref-dependent layers. Layers that don't contain objects, are not current, and are not xref-dependent can be deleted.

List of Layers: Displays layers and their properties. To modify a property, click its icon. To quickly select all the layers, use the right-click and shortcut menu.

Names: Displays the names of the layers. You can select a name, and then click and enter a new name.

On/Off: Turns layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.
Freeze/ Thaw selected layers in all view ports.

You can freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings. AutoCAD does not display, plot, hide, render, or regenerate objects on frozen layers. When you thaw a frozen layer, AutoCAD regenerates and displays the objects on that layer. You can freeze layers in all view ports, in the current layout view port, or in new layout view ports as they are created.

Lock/Unlock: Locks and unlocks the layers. You cannot edit objects on a locked layer. Locking a layer is useful if you want to view information on a layer for reference but do not want to edit objects on that layer.

Color: Changes the color associated with the selected layers. Clicking the color name displays the select color dialog box.

Linetype: Changes the line type associated with the selected layers. Clicking any line type name displays the select Line type color dialog box.

Lineweight: Changes the line weight associated with the selected layers. Clicking any line weight name displays the select Line weight color dialog box.

Plot Style: Changes the plot style associated with the selected layers. If you are working with color-dependent plot styles you cannot change the plot style associated with a layer. Clicking any plot style displays the select plot style dialog box.
7. ISOMETRIC AND ORTHOGRAPHIC PROJECTIONS

**Aim:** To create isometric and orthographic projections of a drawing.

**Procedure:**

**3DFACE**

3DFACE creates a three dimensional face

From Draw menu, select Surfaces

At the command prompt, enter 3DFACE or 3F

First point: Specify a point (P1)

Second point: Specify a point (P2)

Third point: Specify a point (P3)

Fourth point: Specify a point (P4) or press Enter

AutoCAD repeats the Third point and fourth point prompts until you press Enter. Specify points 5 and 6 at these repeating prompts. When you have finished entering points, press Enter.

**Description:**

**HIDE**

HIDE regenerates a three-dimensional model with hidden lines suppressed.

From View menu, select Hide

**Procedure:** At the command prompt, enter HIDE or HI

**Description:**

**VPORTS:** VPORTS divides the graphics area into multiple tiled view ports

Command: VPORTS

**Procedure for orthographic projection:**

Save /Restore/ Delete/ Join /Single/? / 2/ 3/ 4/: Enter an option or press Enter 3.

This option divides the current view port into three view ports.

Horizontal/ Vertical/ Above/ Below /Left / < Right>:

Enter an option or press Enter

The horizontal and vertical options split the area into thirds. The other options create one large view port of half the available area and two smaller ones. The Above, Below, Left, and Right options specify where the larger view port is placed.

The option “2” divides the current view port in half.
Horizontal< Vertical>: Enter h or press Enter 4.
This option divides the current view port into four view ports of equal size.
Single: Returns the drawing to a single view port view, using the view from the active view port.

Join
Combines two adjacent view ports into one larger view port. The resulting view port inherits the view of the dominant view port.
Select dominant view port < current>: Press Enter or select a View port
Select view port to join:

Save
Saves the current view port configuration using a specified name.
? / Name for new view port configuration: Enter a name or?
Entering? Lists the saved view port configurations.

Restore
Restores a previously used view port configuration.
? / Name of view port configuration to restore: Enter a name or ?
Delete
? / Name of view port configuration to delete: Enter a name or ?

REVSURF
REVSURF creates a revolved surface about a selected axis.
From Draw menu, select Surfaces > Revolved Surfaces
At the command: prompt, enter REVSURF
REVSURF constructs a polygon mesh approximating a surface of revolution by rotating a path or profile about a selected axis.
Command: REVSURF
Select path curve: Select a line, circle, or 2D or 3D polyline
Select axis of revolution: Select a line or open 2D or 3D polyline
The path curve is swept about the selected axis to define a surface.
Start angle < 0 >: Enter a value or press Enter
Included angle (+ = ccw, - = cw) < Full circle>: Enter a value or press Enter
3D
3D creates three-dimensional polygon mesh objects

From Draw menu, select Surfaces > 3D surfaces
At the command prompt, enter 3D.

The three-dimensional polygon mesh objects 3D constructs look like wire frame objects but have surfaces. Thus, 3D objects can be shaded or rendered to appear as solid objects.

Command: 3D
Selecting 3D objects from the menu displays the 3D dialog box. Entering 3D on the command line displays the following prompt:
Box / Cone / Dish / Mesh / Pyramid / Sphere / Torus / Wedge: Enter an option

Box: Creates a 3D box polygon mesh.
Corner of Box: Specify a point
Length: Specify a distance
Cube / <Width>: Specify a distance or Enter C
Width: Defines the width of the box
Height: specify a distance
Rotation angle about Z axis: Specify an angle
The base point for the rotation is the first corner of the box.

Cube
Creates a cube using the length for the width and height of the box
Rotation angle about Z axis: Specify an angle
The base point for the rotation is the first corner of the box.
**Cone**

Creates a cone–shaped polygon mesh.

Base center point: Specify a point

Diameter < radius > of base: Specify a distance or Enter D

![Cone Diagram](image)

**Sphere**

Command: Sphere

Centre of sphere <0,0,0>: Specify a point or press Enter

Diameter / <Radius > of sphere: Specify a distance or enter d

![Sphere Diagram](image)

**Torus**

Torus creates a donut shaped solid

At the command prompt, enter TORUS or TOR

A torus is defined by two radius values, one for the tube and the other from the centre of the torus to the centre of the tube.

Command: TORUS

Center of torus <0,0,0 >= Specify a point or press Enter

Diameter / <Radius > of Torus: Specify a distance or enter d

![Torus Diagram](image)
UNION
UNION creates a composite region or solid by addition
At the command prompt, enter UNION or UNI
Command: UNION
Select objects: Use an object selection method

SUBTRACT
SUBTRACT creates a composite region or solid by subtraction
At the command: Prompt, enter SUBTRACT or SU
Command: SUBTRACT
Select objects: Use an object selection method
Select solids and regions to subtract
Select object: Use an object selection method.
Select object: press Enter
INTERSECT

INTERSECT creates composite solids or regions from the intersection of two or more solids or regions
At the command prompt, enter INTERSECT or INT
Command: INTERSECT
Select objects: Use an object selection method

Procedure for drawing Isometric Views:
1. Change the cursor position to an Isometric plane.
2. From the tool menu, choose drafting settings.
3. In the drafting setting dialogue box, put ON the Snap & Grid tab under snap type and style, and select Isometric snap.
4. Choose O.K.
5. Cursor is changed to an Isometric plane.
6. We can cycle through the three Isometric planes by pressing F5.
7. AutoCAD through the Iso-plane top, Iso-plane right and Iso-plane left.
8. VIEWING IN THREE DIMENSIONS

Aim: To view the drawings in three dimensions.

Description: VPOINT

VPOINT sets the viewing direction for a three dimensional visualization of the drawing.

Procedure:

From menu, select 3D Viewpoint

At the command prompt, enter VPOINT.

Command: VPOINT

Rotate / < View point > < Current>: Enter R, specify a point, or press to display a compass and axis tripod.

Viewpoint

AutoCAD follows some convention regarding the sides of a model. As shown in the figure assuming you are standing on top of the model facing along the +ve Y direction, the following directions are defined.

+ ve X : Right side
- ve X : left side
+ ve Y : Back side
- ve Y : Front side
+ ve Z : Top side
-ve Z : Bottom side

According to this rule, if you want to see the front side of the model, you need to walk along – ve Y-axis. Based on this rule, you can derive the VPOINT.
V Point Value | View
---|---
0, 0, 1 | Top View
0, 0, -1 | Bottom View
0, -1, 0 | Front View
0, 1, 0 | Back View
1,0, 0, | Right View
-1, 0, 0 | Left View
1,-1, -1 | Right, front, Bottom View
-1,-1, -1 | Left, front, Bottom View
1,1, -1 | Right, Back, Bottom View
-1,1, -1 | Left, Back, Bottom View
1,-1, 1 | Right, Front, Top View
-1,-1, 1 | Left, Front, Top View
1,1, 1 | Right, Back, Top View
-1,1, 1 | Left, Back, Top View

**Compass & Tripod**

Pressing enter displays a compass and axis tripod, which you use to define a viewing direction in the view port.

To select a viewing direction, move the pointing device to a location on the globe and press the pick button.

With reference to the compass, the resulted view depends on the where you select the view point on the compass.
• Any where to the right of the vertical line gives Right side view.
• Any where to the left of the vertical line gives left side view.
• Any where above the horizontal line gives Backside view.
• Any where below the horizontal line gives Front side view.
• Any where inside the small circle gives Top view.
• Any where in between small and large circles gives Bottom view.
9. REMOVAL OF HIDDEN LINES - SHADING AND RENDERING

**Aim:** To remove the hidden lines. To apply shading on the model and also to render the model.

**Description:**

**2D Wireframe**

Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible. Even if the value for the COMPASS system variable is set to 1, it does not appear in the 2D Wireframe view.

**3D Wireframe**

Displays the objects using lines and curves to represent the boundaries. Displays a shaded 3D UCS icon. Raster and OLE objects, linetypes, and lineweights are not visible. You can set the COMPASS system variable to 1 to view the compass. Material colors that you have applied to the objects are shown.

**Hidden**

Displays the objects using 3D wireframe representation and hides lines representing back faces.

**Flat Shaded**

Shades the objects between the polygon faces. The objects appear flatter and less smooth than Gouraud-shaded objects. Materials that you have applied to the objects show when the objects are flat shaded.

Controls the display of object shading in the current view port.

View menu: Shade
Command line: shademode
Enter option [2D wireframe/3D wireframe/Hidden/Flat/Gouraud/fLat+edges/Gouraud +edges] <current>: Enter an option
Gouraud Shaded

Shades the objects and smooths the edges between polygon faces. This gives the objects a smooth, realistic appearance. Materials that you have applied to the objects show when the objects are Gouraud shaded.

Flat Shaded, Edges On

Combines the Flat Shaded and Wireframe options. The objects are flat shaded with the wireframe showing through.

Gouraud Shaded, Edges On

Combines the Gouraud Shaded and Wireframe options. The objects are Gouraud shaded with the wireframe showing through.

Procedure for Removal of hidden lines:

1. On the view menu choose “hide” or enter the command “hide” or “hi”.
2. The lines remain hidden until you perform an action that regenerates the drawing and redisplays a hidden line wire frame view.

Procedure for Shading:

1. Enter the command “Shade” or from the view menu choose “shade”
2. Choose one of the following option, .e. 2D wire frame/ D wireframe/ Hidden/ flat shaded/ Gouraud shaded/ flat shaded edges on/ Gouraud shaded edges on/.
3. Select the required option to shade the object.

RENDERING

View menu: Render Render
Command line: render
Defines the scene, procedure, options, destination, sampling, and other settings for rendering.

Rendering Type
Lists Render, Photo Real, Photo Raytrace.

Scene to Render
Lists scenes, including the current view that you can select for rendering.

Rendering Procedure
Controls how RENDER behaves by default.

Query for Selections
Displays a prompt to select objects to render.

Crop Window
Creates a render area at render time. When you choose Render, AutoCAD prompts you to select an area on the screen before rendering proceeds. This option is available only when View port is selected under Destination.

Skip Render Dialog
Procedure for Rendering:

1. To the solid object apply the material using RMAT command.
2. To see that the particular applied material, we have to use “render”.
3. Enter the command render or RR.
4. Select the Current view.
5. Choose O.K.
6. Rendering command can also be applied for background and images of the object.

CAD VIVA QUESTIONS

1. What are the uses of Grid and Snap commands?
2. Write different coordinate systems and write the syntax for those coordinate systems?
3. Write diff. draw commands used for tutorial -1?
4. Write about ‘p line’ command and write the modifying command which is used to modify ‘p line’.
5. How many modes are there to draw an arc? name them.
6. Write diff modes to draw a circle.
7. Write the use of array command and name diff. types of array commands.
8. Write the difference between extend and trim commands.
9. How many times REDO command works at a time?
10. What is the use of B HATCH command?
11. Write the command which is used to dimension and object.
12. What is the difference between TEXT and M TEXT command?
13. What is the use of layers?
14. Write different layer properties and layer states and explain them.
15. Write the procedure to highlight a layer.
16. How to create a layer?
17. What is the use of Freeze?
18. How to create an isometric grid and oblique?
19. What is the difference between Orthographic, Isometric and Projection?
20. What keys are used to select the isoplanes cursor?
21. Write different 3D geometric models used in Auto CAD?
22. What is the use of Extrude command?
23. What is the use of Revolve command?
24. Write diff. modifying commands used in solid modelling.
25. What do you mean by V Port? What is the use of it?
26. Write the difference between sweep and Extension command.
27. What is meant by B-rep?
28. What are the different display controls used?
29. What are diff. Graphic standards?
30. What is the use of limits command?
31. Write diff. editing commands?
32. What are the diff. selection methods?
33. What is the diff. between copy and mirror commands?
34. Explain about ‘explode’ command?
35. Explain about ‘block’ command. What are the uses and how to create it?
36. Explain about ‘W block’ command?
37. What is the one of DDedit command?
38. Write about X ref. Command. What is the diff. between insert and X ref. commands?
39. Write about Purge command.
40. Write about Donut command.
41. What do you mean by Aperture and pick box?
42. What is the diff. between Annotation and Attributes?
43. Write about Grip, ddgrip, grip color, grip hot.
44. What are the diff. extensions used for a file?
45. Write differences between copy and copy clip commands.
1. PREPARATION OF MANUAL PART PROGRAMMING FOR CNC TURNING/MILLING

MANUAL PART PROGRAMMING FOR CNC LATHE

Program in incremental mode
Tool Offset T1 X-0.3 Z 0
G92 X0 Z0
N1 G01 X-13 F80 T1
N2 Z-4
N3 X4 Z-5
N4 Z-5
N5 X-2 Z-2
N6 Z-10
N7 X10 Z-5
N8 Z-3
N9 X-2 Z-2
N10 X2 Z-2
N11 Z-6
N12 G0 X1
N13 Z44 T0
N14 M30

EXAMPLE 1: MILLING

[ BILLET L 170 W 90 D 10
[ EDGEMOVE X-70 Y-30
[ TOOLDEF T1 L 65 D6
[ SET DATUM X 20 Y20
N1 G00X0Y-18.4 Z5
N2 MO3 S1000
N3 G01 Z-3
N4 GO1 X32
EXAMPLE 2: MILLING

[ BILLET L 170 W 90 D 10
[ EDGEMOVE X-60 Y-55
[ TOOLDEF T1 L 65 D6
[ SET DATUM X 60 Y45
N1 G00X-30Y-43 Z5
N2 MO3 S1000
N3 G01 Z-3
N4 GO1 X30
N5 GO3 X41.21Y-26.51 R13
N6 GO1 X14.78 Y11.98
N7 GO3 X-14.782 Y11.98 R18
N8 GO3 X-40.25 Y-26.51
N9 GO3 X-30 Y-43 R13
N10 GO0 X0 Y0 Z5
N11 M05
2. PART PROGRAMMING PREPARATION THROUGH AUTOCAD

Program in incremental mode
Tool Offset T1 X-0.3 Z 0
G92 X0 Z0
N1 G01 X-11 F100 T1
N2 Z-1
N3 X3 Z-3
N4 Z-7
N5 G02 X5 Z-5 15 K0
N6 G01 Z-5
N7 X-3 Z-5
N8 Z-34
N9 G2 X5 Z-5 15 K0
N10 X2 Z-5
N11 X-2 Z-2
N12 X2 Z-2
N13 Z-2
N14 G0 X11
N15 Z 53
N16 G2 X0 Z-40 10 K –20
N17 G0 X0 Z 63 T0
N18 M 30
3. APT SYSTEM

APT GEOMETRY: Any part geometry can be built up from the basic geometric elements. A few geometric elements are defined below are.

**Point definitions in APT**

Syntax: `<symbol> = POINT / <parameter listing>`

1. POINT defined by rectangular coordinates
   
   P1 = POINT / 50, 75, 70
   P2 = POINT / 50, 25

2. POINT defined by intersection of two lines
   
   P2 = POINT / INTOF, L1, L2

3. POINT by intersection of a line and a circle
   
   P3 = POINT / XSMALL, INTOF, L, C

4. POINT by intersection of two circles
   
   P4 = POINT / 25
   C1A = CIRCLE / YSMALL, INTOF, C1, C2

**Line definitions in APT**

1. LINE passing through two points
   
   L1 = LINE / PT1, PT2
LINE/ x1,y1, x2, y2
L1B = LINE / 20, 30, 70, 60
2. LINE passing through a point and parallel to a line
   LINE/point, PARLEL, line
   L3 = LINE / P3, PARLEL, L1
3. LINE passing through a point and perpendicular to a line
   LINE/ point, PERTO, line
   L4 = LINE/P4, PERPTO, L1
4. LINE passing through a point and tangent to a circle
   LINE/ point \( \{ \) LEFT
   \( \} \) RIGHT, TANTO, circle
   L8 = LINE / PT, LEFT, TANTO, CR
   L8A = LINE / PT, RIGHT, TANTO, CR
5. LINE tangential to two circles
   LINE /point LEFT \( \{ \) LEFT
   \( \} \) RIGHT, TANTO, circle1, RIGHT, TANTO, circle2
   L9 = LINE / LEFT, TANTO, C1, RIGHT, TANTO, C2
   L8A = LINE / RIGHT, TANTO, C1, RIGHT, TANTO, C2

CIRCLE DEFINITIONS IN APT

Syntax: \(<\text{symbol}> = \text{CIRCLE} / <\text{parameter string}>\)

1. CIRCLE defined by its center and radius
   \(\text{CIRCLE} / x, y, \text{radius}\)
   \(\text{CIRCLE} / \text{CENTRE}, \text{point}, \text{RADIUS}, \text{radius}\)
   \(C1 = \text{CIRCLE} / 75, 50, \text{CENTRE}, \text{PT1}, \text{RADIUS}, 45\)
2. CIRCLE defined by its centre and tangent to a line
   \(\text{CIRCLE} / \text{CENTRE}, \text{point}, \text{TANTO}, \text{line}\)
   \(C2 = \text{CIRCLE} / \text{CENTRE}, \text{PT2}, \text{TANTO}, \text{L1}\)
3. CIRCLE defined by its centre and a point on the circumference
   \(\text{CIRCLE} / \text{CENTRE}, \text{point1}, \text{point2}\)
   \(C3 = \text{CIRCLE} / \text{CENTRE}, \text{P3}, \text{PT3}\)
4. CIRCLE passing through three points
CIRCLE / point1, point2, point3
C4 = CIRCLE / PT4, PT5, PT6

5. CIRCLE of known radius tangential to two lines

\[
\begin{align*}
CIRCLE & \left\{ \begin{array}{l}
\text{XLARGE} \\
\text{XSMALL} \\
\text{YLARGE} \\
\text{YSMALL, line1, $} \\
\text{YSMALL, line2, RADIUS, radius}
\end{array} \right. \\
\end{align*}
\]

C6 = CIRCLE / YLARGE, L2, YSMALL, L2, RADIUS, 15
C6A = CIRCLE / YLARGE, L1, XSMALL, L2, RADIUS, 15

6. CIRCLE of known radius and tangent to circle

\[
\begin{align*}
CIRCLE & \left\{ \begin{array}{l}
\text{XLARGE} \\
\text{XSMALL} \\
\text{YLARGE} \\
\text{YSMALL, } \{ \begin{array}{l}
\text{IN} \\
\text{OUT, circle1, } \{ \begin{array}{l}
\text{IN} \\
\text{OUT, circle2, RADIUS, radius}
\end{array} \\
\end{array} \right. \\
\end{array} \right. \\
\end{align*}
\]

C8A = CIRCLE / YLARGE, OUT, C1, IN, C2, RADIUS, 15
C81 = CIRCLE / YLARGE, IN, C3, OUT, C4, RADIUS, 50

Plane definitions in APT

Syntax:  \(<\text{symbol}> = \text{PLANE} / <\text{parameter string}>\)

1. PLANE passing through three points

PLANE / point1, point2, point3
PL1 = PLANE / P1, P2, P3

MOTION COMMANDS
They are used to describe the actual machining sequence making use of geometric statements defined. The motion commands are divided into 3 groups.

1. Set up commands.
2. Point-to-point commands.
3. Continuous path motion commands.

SET UP COMMANDS:

The set up commands are required for initiating the machining operations.

Starting point command:
To establish the starting point of the cutter for the starting motion.

FROM / POINT
FROM / x, y or FROM / x, y, z

Cutter command:
To describe the cutting tool.

CUTTER / d or CUTTER / d, r

Surface commands:
The desired path of the 3-dimensional cutting tool is described by means of three interesting surfaces. These surfaces are

Part surface (PS)
Drive surface (DS)
Check surface (CS)

The part surface is the surface which is in continuous contact with the top of the tool. Drive surface is the plane along with the cutter is in machining contact. CS is the surface which limits the motion of the tool.

POINT TO POINT MOTION COMMANDS:
Position in APT is achieved with two statements GOTO and GODLTA

GODLTA : incremental motion :

GODLTA / dx, dy, dz if one value is given with GODLTA,
GODLTA /dz it is implied for Z value.
GODLTA / VECTOR.

GOTO : Absolute movement command
GOTO/ point
GOTO/ x,y or GOTO/ z,y,x
When no z value is specified, it takes from ZSURF.
GOTO can also be used for positioning at PATTERN
GOTO/ pattern
The type of machining is specified by POST PROCESSOR commands preceding GOTO statement.

CONTINUOUS PATH MOTION COMMANDS:

Part surface command:
Part surface is specified by PSIS

PSIS / PL (PL is a plane defined earlier)
Part surface can be established.
Parallel to X, Y plane at current Z height by AUTO PS.
NOPS (is a non-model command).
To ignore a part surface which is already in force, NOPS is used before Startup command and the original part surface is restored after the startup motion.

START UP command:
Before proceeding to start machining, the cutting tool should bring in contact with the part and drive surface within specified tolerance.

i) 3 surface startup command:

\[
\text{GO / } \begin{cases} 
\text{TO ON, ds PAST} \\
\text{TO ON, ps PAST} \\
\text{TO ON, cs PAST}
\end{cases}
\]

ii) 2 surface startup command:

\[
\text{GO / } \begin{cases} 
\text{TO ON, ds PAST} \\
\text{TO ON, ps PAST}
\end{cases}
\]

iii) 1 surface startup command:

\[
\text{GO / } \begin{cases} 
\text{TO ON, ds PAST}
\end{cases}
\]
If no terminal modifier is specified, the default is TO. The end point of tool is always in contact with part surface, so only TO is meaningful with part surface.

To avoid ambiguity (if any) direction control commands are used.

- INDIRP / point
- INDIRV / vector

Tool relationship commands:
- TLRGT (tool is on right side of drive surface)
- TLLFT (tool is on left side of drive surface)
- TLO (tool is on the drive surface)

Tool relationship should be mentioned before motion commands.

ACTION verbs:
The direction of every motion of the tool have to be specified with respective to its previous motion direction. The following six action verbs are used for this purpose.

1. GO LFT / Move left along the drive surface.
2. GO RGT / Move right along the drive surface.
3. GO FWD / Move forward from a tangent position.
4. GO BACK / Move backward from tangent position.
5. GO UP / Move up along the drive surface.
6. GO DOWN / Move down along the drive surface.

To describe any motion, one of the above six action verbs are required.

Action verbs define direction w.r.t. the previous motion.

Example of simple contour:

Check surface is implied when the check surface becomes the drive surface in the subsequent command.
**Machining outside**

**Machining inside**

**Contour**

**FROM/SP**

**TLLFT**

**GOLFT/L1, PAST, L2**

**GORGT/L2, PAST, L3**

**GORGT/L3, PAST, L4**

**GORGT/L4, PAST, L1**

---

**POST PROCESSOR COMMANDS**

**PARTNO/** is used to indicate the name of the part program.

**PARTNO/** part details.

**MACHIN/** is used to specify the machine tool.

**ACHIN/** **RILL, 4** to specify the 4th NC drill in the cell.

**COOLNT/** is used to turn coolant ON or OFF.

**OOLNT/** **MIST** to turn on the air coolant.

**OOLNT/** **FLOOD** to turn on the oil coolant.

**OOLNT/** **OFF** to turn coolant off.

**FEDRAT/** is used to specify the feed rate of cutter motion.

**PM** \( \text{f inch per minute} \)

**MPM** \( \text{f mm per minute} \)

**EDRAT/** **IPR, f** \( \text{f inch per revolution} \)

**MPR** \( \text{f mm per revolution} \)

**RAPID** is used to indicate the rapid speed (similar to G00).

**SPINDL/** is used to specify the speed of the spindle.

**SFM** \( \text{n surface feed feet per minute} \)

**SPINDL/** **RPM, n, CLW** \( \text{n revolutions per minute} \)

**SMM** **CCLW** \( \text{n surface meters per minute} \)

**CUTCOM/** is used for inclusion of cutter compensation.

**ON**

**CUTCOM/** **OFF, LENGTH, a** \( \text{a refers the register associated} \)

**RIGHT RADIUS** \( \text{with compensation} \)

**LEFT**
CYCLE/ is used to generate canned cycles for drilling/reaming/tapping, etc.

DRILL similar to G81 fixed cycle
FACE IPM similar to G82 fixed cycle

CYCLE/ TAP, z, IPR, f, r similar to G84 fixed cycle
BORE MMPR similar to G86 fixed cycle
REAM MMMP similar to G85 fixed cycle

z is the dept, f is the feed rate and r is the rapid plane.

CYCLE/ ON actuates the previous cycle.
OFF suppresses the cycle command (similar to G80)

LOADTL/ is used to load tool of specified number from the specified tool magazine.

LOADTL/ tool no., magpos

THREAD/ is used to activate the threading mode.

FACE

THREAD/ TURN, INCR, a, TPI, p, DEPTH, d
TAPER DECR MMPR

a indicates the rate that which pitch of the thread changes. For constant pitch threads this can be omitted.
p is the pitch in threads per inch or mm per revolution.

PRINT/ is used to print required information on the line printer.
PRINT/ ON or PRINT/ALL
PRINT/ OFF

CLPRINT/ is used to print the cutter location data.
CLPRINT/ ON
CLPRINT/ OFF

REMARK/ is used to add comments for clarity of the program.
REMARK/ comment

FIN/ is used to indicate the finishing of the program (similar to END command).

PROGRAM EXAMPLE

PART NO/ MILLING JOB NO.1
REMARK PROGRAMMER – B.SATYANARAYANA
MACHINE / STARMILL
REMARK / GEOMETRICAL DEFINITIONS
CLPRINT / ON $$ prints CL data.
REMARK / GEOMETRIC STATEMENTS
SP=POINT/−4.0, −4.0
P1= POINT/−2.0
Q1=POINT/0, 0
Q2=POINT/4, 0
Q3=POINT/0, 0, 6.0
CL12=LINE/XAXIS
CL13=LINE/YAXIS
L1=LINE/PARLEL, CL13, XSMALL, 2.0
L2=LINE/PARLEL, CL13, XLARGE, 2.0
L3=LINE/PARLEL, CL12, YLARGE, 2.0
L2=LINE/PARLEL, CL12, YSMALL, 2.0
C1=CIRCLE/CENTRE, Q3, RADIUS, 2.0
C2=CIRCLE/CENTRE, Q2, RADIUS, 2.0
C3=CIRCLE/CENTRE, Q1, RADIUS, 2.0
Q11=POINT/0,0,−5.0
Q21=POINT/4,0,−5.0
Q31=POINT/0,4.06,−5.0
PS=PLANE/Q11, Q21, Q31
REMARK / MOTION STATEMENTS
FROM/SP
CUTTER/16
TOLER/0.05
SPINDL/RPM, 600, CLW
COOLNT/ON
FEDRAT/MMPM, 100
GO/TO, L1, T0, PS, TANTO, C3
AUTO PS
GODLT/−10.0
TLLFT
GOFWD/L1, TANTO, C1
GOFWD/C1, TANTO, C2
GOFWD/L2, L3
GOFWD/L3, TANTO, C2
GOFWD/C2
GOFWD/L4
GOFWD/C3, TANTO, L1
GOFWD/10.0
RAPID
GOTO/SP
COOLNT/OFF
SPINDL/OFF
CLPRINT/OFF
END
FINI
4. MACHINING OF ONE JOB ON CNC MACHINE TOOL

EXAMPLE 1

[ BILLET X 20 Z 80;
G28
M06 T03 ;
M03 S1500 F90 ;
G00 X20  Z1 ;
G71 U0.2  R0.5 ;
G71 P10 Q 20 U0 W0 ;
N10 G01 X0 ;
Z0;
G03 X10 Z-5 R5

EXAMPLE NO: 2
CAM VIVA QUESTIONS

1. What is NC part programming?
2. In NC part programming “BLOCK” means?
3. In NC part programming data contains in the form of?
4. How many types NC part programming is prepared?
5. Manual part programming is prepared on?
6. Give one example to prepare NC part programming using computer language?
7. What are G, N, and M codes?
8. Explain each term given below NC program
   N001 G02 X023 Y024 R5 M03
9. For circular interpolation which preparatory code is used?
10. How many number of “dimension numbers” required for NC part programming?
11. How many digits used to represent feed word?
12. How many digits used to represent spindle speed word?
13. How many digits used to represent tool word?
14. What is magic-three code?
15. What is different between G00 and G01?
16. What is different between G90 and G91?
17. What is cutter compensation? Which codes are used for cutter compensation?
18. Explain canned cycle or fixed cycle? Which codes are used for cycle or fixed cycle?
19. Expand BLU?
20. Explain different types of tool paths used in NC part programming?
21. Explain co-ordinate system for CNC turning?
22. Explain co-ordinate system for CNC milling?
23. Expand and explain APT?
24. How many types of statements used APT programming?
25. Explain and classify motion statements in APT programming?
26. How many types point, circle and line represented in APT programming using geometrical statements?
27. Explain point-to-point motion statement and sketch?
28. Explain contour motion statement with sketch?
29. What is postprocessor in APT programming?
30. What is CNC?
31. What is DNC?
32. For plane selection which preparatory codes used?
33. Expand and explain ATC?
34. What is difference between incremental and absolute system?
35. What is difference between absolute zero and floating zero co-ordinate system?
36. Explain difference between point-to-point, linear and contour interpolation?
37. G04 represents which operation in NC part programming?
38. Explain difference between NC, CNC, and DNC?
39. Which preparatory code is used for tool nose radius compensation in turning?
40. What are different types of CNC controllers is used in now days?
41. Polar co-ordinate systems are CNC programming or not?
42. Explain High speed machining?
43. What are functions of adaptive control system in CNC machines?
44. Explain different parts in NC machines?
45. Explain different steps from design to manufacturing using NC machines?
46. What is CNC turning center?
47. What is difference between ATP and ATC?