Work Instruction

Subject	**MACHINE DESIGN**
Subject Code	**PC-REP601**

Department of Mechanical Engineering
THE NEOTIA UNIVERSITY

ज्ञानम् आत्म प्रदीपाय

Contents

NAME OF THE EXPERIMENT: 2-DIMENSIONAL DRAFTING PRACTICE BY AUTOCAD

EXPERIMENT NO.: PC-REP601/01

OBJECTIVE

Accuracy: The drawings are not useful to the maximum extent if they are not accurate.
Speed: "Convert Time into Money" in industry. There is no place for the slow technician, or engineer. Speed is not attained in a hurry; it should be with intelligent and continuous work. It comes with practice.
Legibility: Drawing is a means of communication to others, and that it should be clear and legible to serve its purpose. Care should be taken especially in dimensioning and lettering.
Neatness: If a drawing is to be acceptable it should be clean and neat because even small dust particle can act as smallest entity as point.

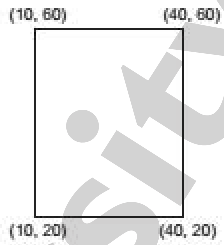
PRINCIPLE

Basic Geometric Command:

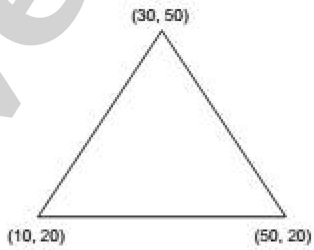
Drawing Entity-Point
The point command locates a point in the drawing.
Command: POINT (one has to give the location)
POINT: 25, 45 location of the point. Thus, a point is placed at the given location (25, 45).
After setting the limits of the drawing, the following drawing aids/tools may be used to locate specific points on the screen (electronic drawing sheet).
ORTHO Command—this is orthogonal drawing mode. This command constrains the lines drawn in horizontal and vertical direction only.
Command: ORTHO
ON/OFF <current>:
SNAP Command—this command is used to set increments for cursor movement. If the screen is on SNAP mode, the cursor jumps from point to point only. The cursor movement can be effectively controlled using the SNAP command. This is useful for inputting the data through digitizer/mouse.
Command: SNAP
Snap spacing or ON/OFF/Aspect/Locate/Style <current>: 0.1 (default)
GRID Command-working on a plain drawing area is difficult since there is no means for the user to understand or correlate the relative positions or straightness of the various objects made in the drawing. The command enables to draw dotted lines on the screen at pre-defined spacing. These lines will act as graph for reference lines in the drawing. The grid spacing can be changed at will. The grid dots do not become part of the drawing.
Command: GRID
Grid spacing or ON/OFF/Snap/Aspect <0>: 0.5 (default)
Function keys may create drawing aids/tools also. The function keys F7, F8 and F9 act as toggle keys for turning ON or OFF of GRID, ORTHO and SNAP tools respectively.

Drawing Entity-Line
Lines can be constrained to horizontal/ vertical by the ORTHO commands. CLOSE option uses the starting point of the first line segment in the current LINE command as the next point.

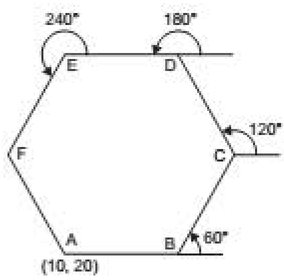
1. Lines can be drawn using co-ordinate system (rectangular cartesian co-ordinates). To draw a rectangle:
Command: LINE
From point: 10, 20 ↵
To point: 40, 20 ↵
To point: 40, 60 ↵
To point: 10, 60 ↵
To point: ↵



2. It is also possible to specify the co-ordinates in the incremental format as the distances from the current cursor position in the drawing area. The distance is specified by using the @ parameter before the actual value. To construct a triangle of given altitude (30) and base (40):
Command: LINE
From Point: 10, 20 ↵
To point: @ 40, 0 ↵
To point: @ – 20, 30 ↵
To point: ↵



3. It is also possible to specify the point co-ordinate using the ploar co-ordinate format. To construct a hexagon of side 30:
Command: LINE
From point: 10, 20 ↵(A)
To point: @ 30<0 ↵ (B)
To point: @ 30<60 ↵ (C)
To point: @ 30<120 ↵ (D)
To point: @ 30<180 ↵ (E)
To point: @ 30<240 ↵(F)
To point: close

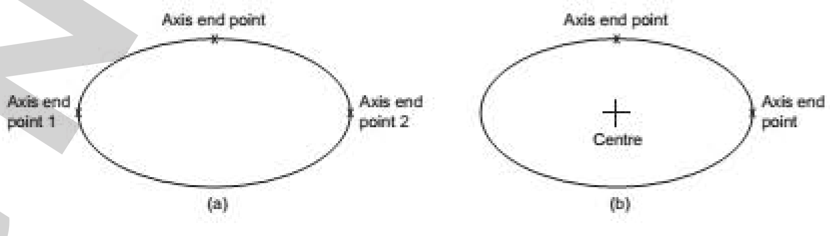


Drawing Entity-Ellipse

This command allows one to draw ellipses or egg shaped objects. From Release 13 onwards, ellipse is treated as a separate entity. The methods available for making ellipses are:

1. By means of axis end points: (Fig.a)

Command: ELLIPSE <axis end point 1>/ center: point ↵
Axis end point 2: (point)

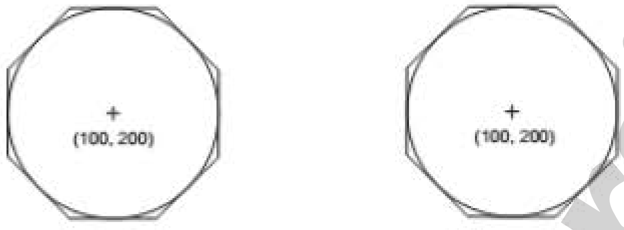


<other axis distance>/ Rotation:
Now, if the distance is entered, AutoCAD interprets it as half the length of the other axis.
2. By means of centre, axis end points (Fig.b)
Command: ELLIPSE <axis end point 1>/ centre: C ↵
Centre point and one end point of each axis should be provided for the response of the AutoCAD.

Drawing Entity-Polygon

This option permits to make/draw polygons from 3 to 24 sides in a number of ways:

1. For making inscribed/circumscribed polygon with a side parallel to x-axis: (Fig.a, b)



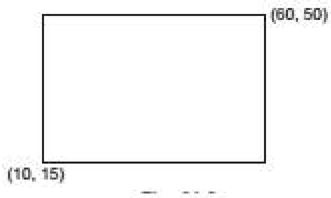
(a) Inscribed (b) Circumscribed

Command: POLYGON
Number of sides: 8
Edge/ <centre of polygon>: 100, 200 ↵
Inscribed / circumscribed about a circle (I/C): I or C ↵
Radius of circle: 80

2. With edge option, specifying the size of the edge and orientation:

Drawing Entity-Rectangle

A rectangle is a polygon based on two opposite corner points, known as diagonal points.

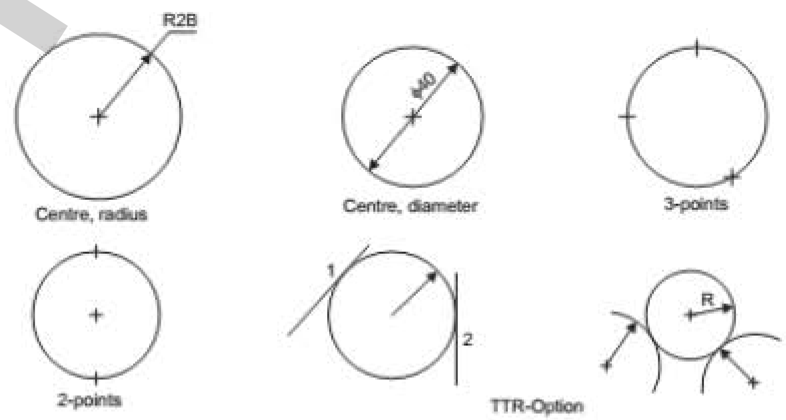


Command: RECTANGLE
First corner: 10, 15 ↵
Second corner: 60, 50 ↵
Or from the tool bar menu icon, the pointing device can drag the rectangle and therectangle can be completed.

Drawing Entity-Circle

Command: CIRCLE

1. 3P/ 2P/ TTR/ <centre point>:
Pick a centre point or enter an option
2. Diameter/ <Radius><current default>: select D or R
3. 3P (3 point) option: one is prompted for a first, second and third point. The circle willmbe drawn to pass through these points.
4. 2p (2 point) option: one is prompted for the selection of two points which form them opposite ends of the diameter.
5. TTR option: allows one to define a circle based on two tangent points and aradius. The tangent points can be on lines, arcs or circles.



Drawing Entity-Arc

Arc command permits to draw an arc, using a variety of methods.

Command: ARC

1. Centre/ <start point>: pick a start point using mouse or select C for more options.
2. Centre/End/ <second point>: pick a second point of the arc or select C, if option is C.
3. Angle/length of chord/end point: pick end point of the arc, if option is E.
4. Angle/Direction/Radius/ <centre point>: pick end point of the arc or specify the option.

Angle — "included angle" prompt appears, to enter the value.

Centre — enter the location of an arc's centre point-at the prompt centre-pick a point,

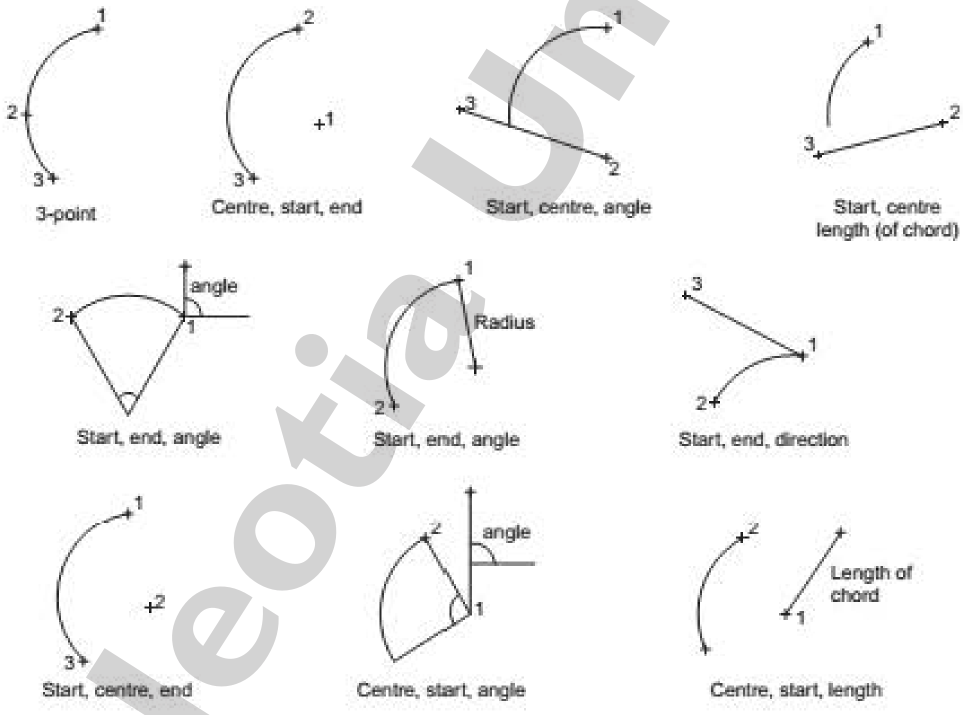
Direction —enter a tangent direction from the starting point of an arc. At this prompt, pick a point with cursor.

End — at this prompt, pick the end point of the arc.

Length — enter the length of a arc's chord. At this prompt, enter a length or drag and pick a length with cursor.

Radius — at the prompt "radius", enter a radius value.

Start point — enter the beginning point of an arc.



Edit Commands:

The commands used for modifying the drawings fall under this category. Using these commands, the objects may be erased, retrieved, moved to another location, made into multiple copies, rotated, enlarged, mirror imaged, part of a drawing may be moved and the above effects can also be reversed (undo).

ERASE Command—this lets the entities to be permanently removed from the drawing. The command format is

Command: ERASE

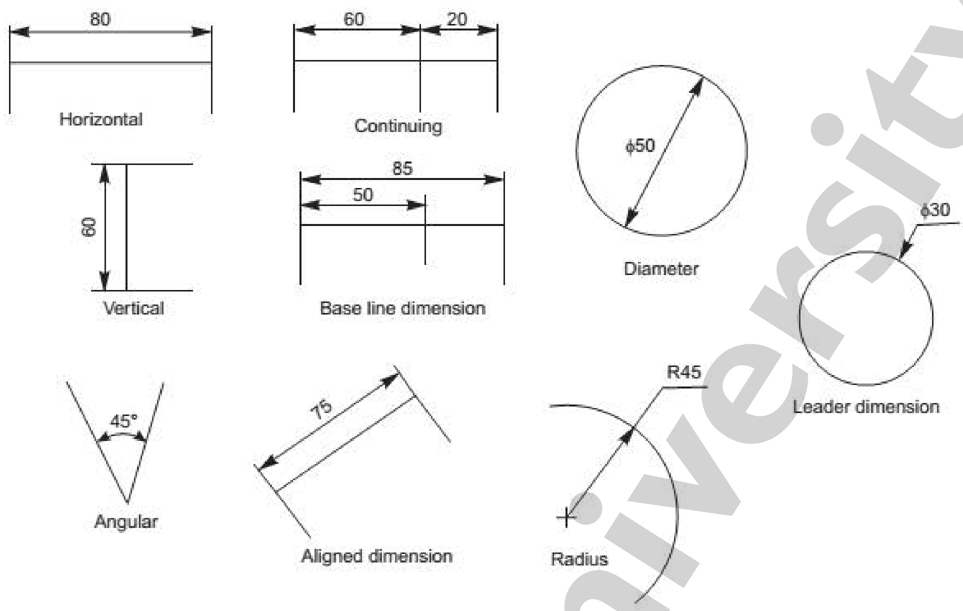
Select objects: (desired objects) once it is entered, the objects/portion of the object is erased/ deleted.

OOPS Command—this restores the entities that have been inadvertently ERASED. Whenever ERASE command is used, a list of entities erased is retrieved by this command.

Command: OOPS

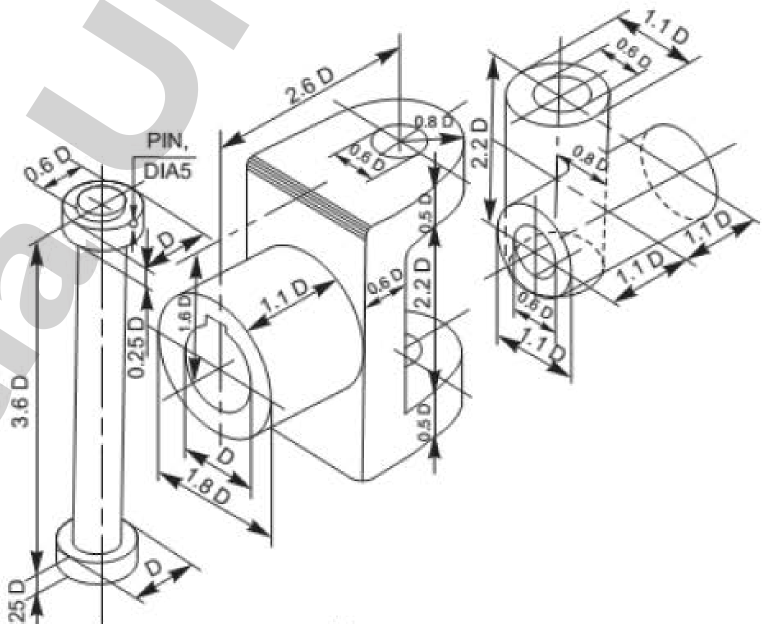
Once it is entered, it restores all the entities erased by the recent ERASE command. Once another ERASE is done, the list of entities erased by the previous ERASE command is discarded. OOPS cannot be used to restore them.
AutoCAD allows backup step by step to an earlier point in an editing session, using the UNDO command. This stores all the sequences made by the user in the current drawing session.
UNDO Command—this command allows to undo several commands at once. This command is used for correcting any errors made in the editing process. When a SAVE option is used, then the UNDO cannot do anything before that.
Command: UNDO
If the response contains a number, that many number of preceding operations will be undone.
REDO Command—if REDO is entered immediately after a command that undoes something, it will undo the UNDO.
Command: REDO
An UNDO after REDO will redo the original UNDO.
OFFSET Command—this constructs an entity parallel to another entity at either a specified distance or through a specified point.
MIRROR Command—this allows to mirror the selected entities in the drawing. The original objects can be deleted (like a move)/retained (like a copy).
MOVE Command—the move command is used to move one/more existing drawing entities from one location in the drawing to another.
COPY Command—this is used to duplicate one or more existing drawing elements at another location without erasing the original.

Basic Dimensioning:
In many applications, a drawing should contain annotations showing lengths or distances or angles between objects to convey the desired information. Dimensioning is the process of adding these annotations to a drawing. AutoCAD provides four basic types of dimensioning; linear, angular, diameter and radius.
DIM and DIMI Commands—DIMI command allows executing one dimensioning command and then returns to the normal command mode. If several dimensioning commands are to be executed, DIM command should be used. In this mode, the normal set of AutoCAD commands is replaced by a special set of dimensioning commands. To end the process of dimensioning, EXIT command has to be used.
The dimensioning commands can be grouped into six categories:
1. Linear — is done with a horizontal, vertical, aligned and rotated command. However, rotated command requires specifying the dimension line angle explicitly.
2. Angular — is used to dimension angles. Here, one has to select two non-parallel lines to introduce the angular dimension.
3. Diameter — this can be invoked for dimensioning arcs and circles.
4. Radius — it is almost identical to diameter dimensioning, except that only a radius line is drawn. This line has only one arrow.
5. Associative — used to make various changes to associative dimension entities.
6. Dimensioning utility commands — to draw a centre line or centre mark for a circle/arc, this command is used. AutoCAD generally uses same type of dimensions and dimension label components as standard draughting. Figure 21.19 gives examples of types of dimensions possible: linear, angular, diametric, radial and aligned. A number of variables such as extension lines, text location, tolerance specifications, arrow styles and sizes, etc., actually control the way in which the dimensions may appear in the drawings.

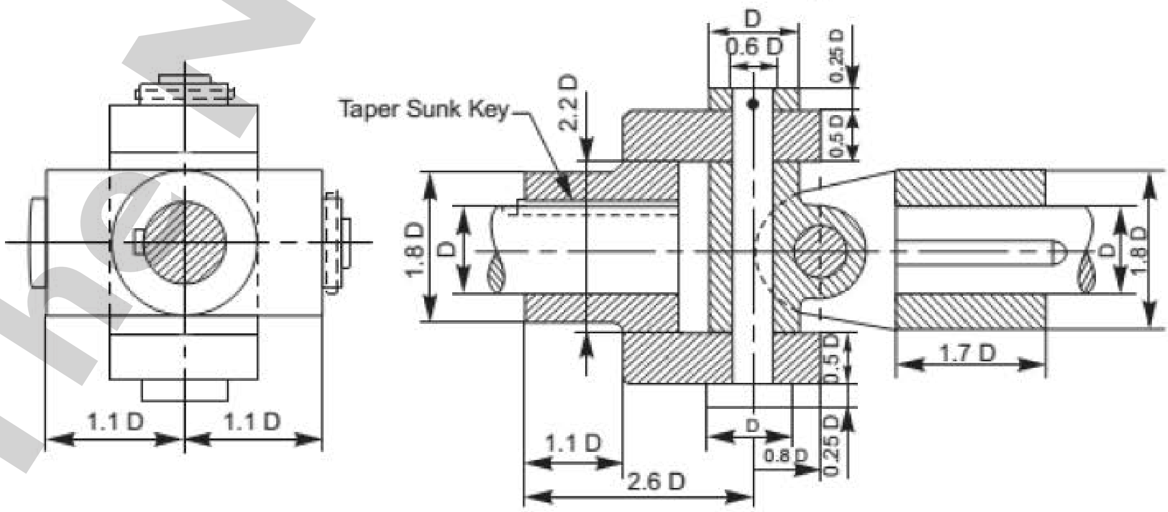


Problem: Draw the sectional front and top view of all spare parts and the assemblage drawing of the Universal Coupling for shaft diameter d = 50 mm. in AutoCAD

It is a rigid coupling that connects two shafts, whose axes intersect if extended. It consists of two forks which are keyed to the shafts. The two forks are pin joined to a central block, which has two arms at right angle to each other in the form of a cross (Fig. 7.11). The angle between the shafts may be varied even while the shafts are rotating



(a)



Problem 1: Draw the isometric views of Figs. 1.1, 1.2, 1.3 and 1.4.

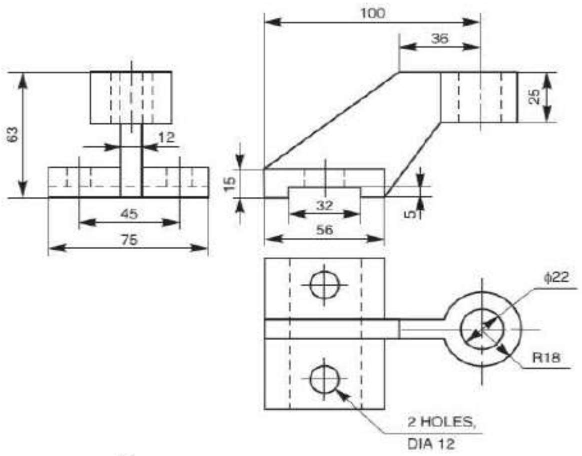


Fig. 1.1

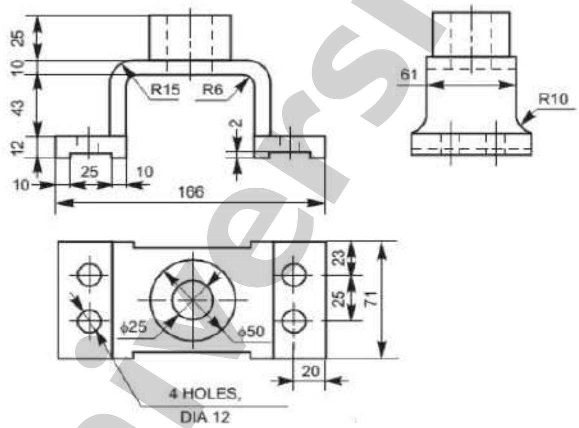


Fig. 1.2

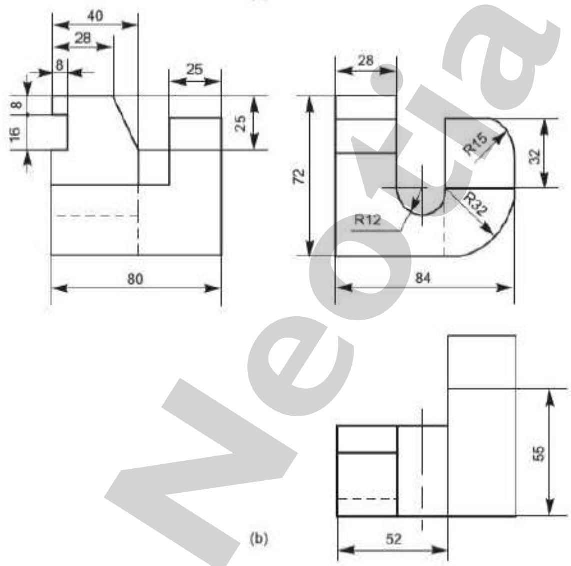


Fig. 1.3

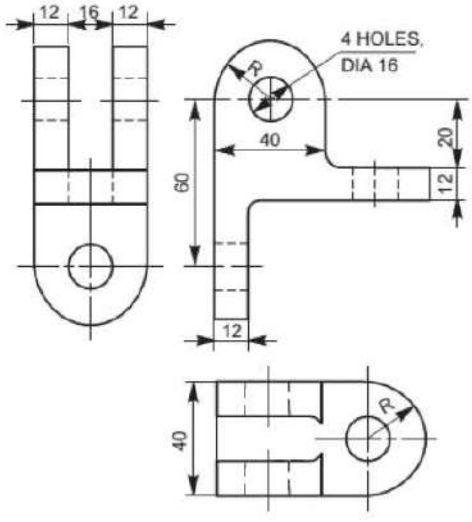


Fig. 1.4

NAME OF THE EXPERIMENT: VELOCITY ANALYSIS IN MECHANISMS BY AUTOCAD

EXPERIMENT NO.: PC-REP601/02

OBJECTIVE

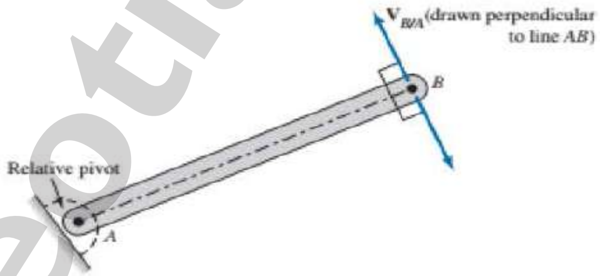
Velocity analysis involves determining "how fast" certain points on the links of a mechanism are traveling. Velocity is important because it associates the movement of a point on a mechanism with time. Often the timing in a machine is critical. The determination of velocity in a linkage is the purpose of this experiment. Two common analysis procedures are examined: the relative velocity method and the instantaneous center method. In this section, graphical techniques are followed.

PRNCIPLE

Graphical velocity analysis will determine the velocity of mechanism points in a single configuration. It must be emphasized that the results of this analysis correspond to the current position of the mechanism. As the mechanism moves, the configuration changes, and the velocities also change. The basis of the relative velocity method of analysis is derived from the following fact:

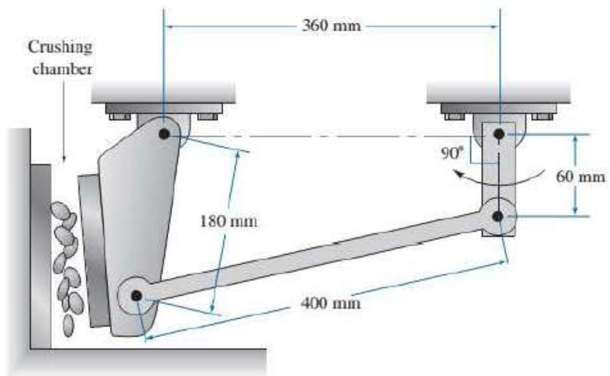
Two points that reside on the same link can only have a relative velocity that is in a direction perpendicular to the line that connects the two points.

This fact is an extension of the definition of relative velocity. Figure 6.9 illustrates two points, *A* and *B*, that are on the same link. Recall that $v_{B/A}$ is the velocity of *B* "as observed" from *A*. For an observer at *A*, it appears that *B* is simply rotating around *A*, as long as both *A* and *B* are on the same link. Thus, the velocity of *B* with respect to *A* must be perpendicular to the line that connects *B* to *A*. With this fact and vector analysis techniques, the velocity of points on a mechanism can be determined.



EXAMPLE PROBLEM 1

Figure below shows a rock-crushing mechanism. It is used in a machine where large rock is placed in a vertical hopper and falls into this crushing chamber. Properly sized aggregate, which passes through a sieve, is discharged at the bottom. Rock not passing through the sieve is reintroduced into this crushing chamber. Determine the angular velocity of the crushing ram, in the shown configuration, as the 60-mm crank rotates at 120 rpm, clockwise.



Solution:

1. ***Draw a Kinematic Diagram and Calculate Degrees of Freedom***

Figure below shows a kinematic diagram of this mechanism. Notice that this mechanism is the familiar four-bar linkage, having a single degree of freedom. With one degree of freedom, this mechanism is fully operated with the one input motion. Of course, this motion is the rotation of link 2, at a rate of 120 rpm.

2. ***Decide on an Appropriate Relative Velocity Equation***

The objective of the analysis is to determine the angular velocity of link 4. Link 2 contains the input motion (velocity). Point *B* resides on both links 2 and 3. Point *C* resides on both links 3 and 4. Because points *B* and *C* reside on link 3, the relative velocity method can be used to relate the input velocity (link 2) to the desired velocity (link 4). The relative velocity equation for this analysis becomes **V***C* = **V***B* + > **V***C/B*

3. ***Determine the Velocity of the Input Point***

The velocity of point *B* is calculated as

$$\omega_2\left(\text{rad/s}\right) = \frac{\pi}{30}(120\ \text{rpm}) = 12.56\ \text{rad/s, cw}$$

$$\mathbf{V}_B = \omega_2\, r_{AB} = \left(12.56\ \text{rad/s}\right)(60\ \text{mm}) = 754\ \text{mm/s}\ \leftarrow$$

4. ***Determine the Directions of the Desired Velocities***

Because link 4 is fixed to the frame at *D*, link 4 is limited to rotation about *D*. Therefore, the velocity of point *C* must be perpendicular to the line *CD*. Also, as earlier stated, points *B* and *C* reside on link 3. Therefore, the relative velocity of *C* with respect to *B* must lie perpendicular to the line *BC*.

5. ***Draw a Velocity Polygon***

In the relative velocity equation, only the magnitudes of **V***C* and **V***C/B* are unknown. The vector polygon used to solve this problem is shown in Figure. The magnitudes can be determined by observing the intersection of the directed lines of **V***C* and **V***C/B*. The completed vector polygon is shown in the Figure.

6. ***Measure the Velocities from the Velocity Polygon***

The velocities are scaled from the velocity diagram to yield

$$\mathbf{V}_C = 784.0\ \text{mm/s}\ \underline{7.0^\circ}\nwarrow$$

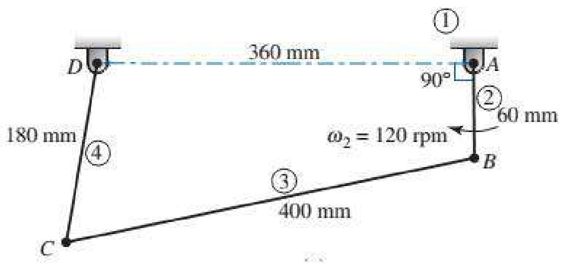
$$\mathbf{V}_{C/B} = 101.1\ \text{mm/s}\ \underline{72.7^\circ}\nwarrow$$

7. ***Calculate Angular Velocities***

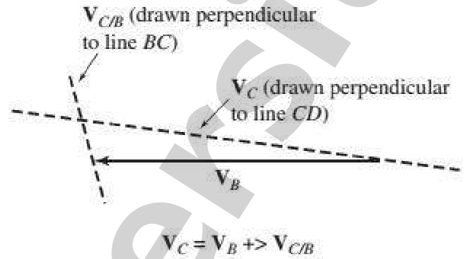
Ultimately, the angular velocities of link 4 is desired. The angular velocities of both links 3 and 4 are:

$$\omega_4 = \frac{v_C}{r_{CD}} = \frac{(789.4\ \text{mm/s})}{(180\ \text{mm})} = 4.36\ \text{rad/s, cw}$$

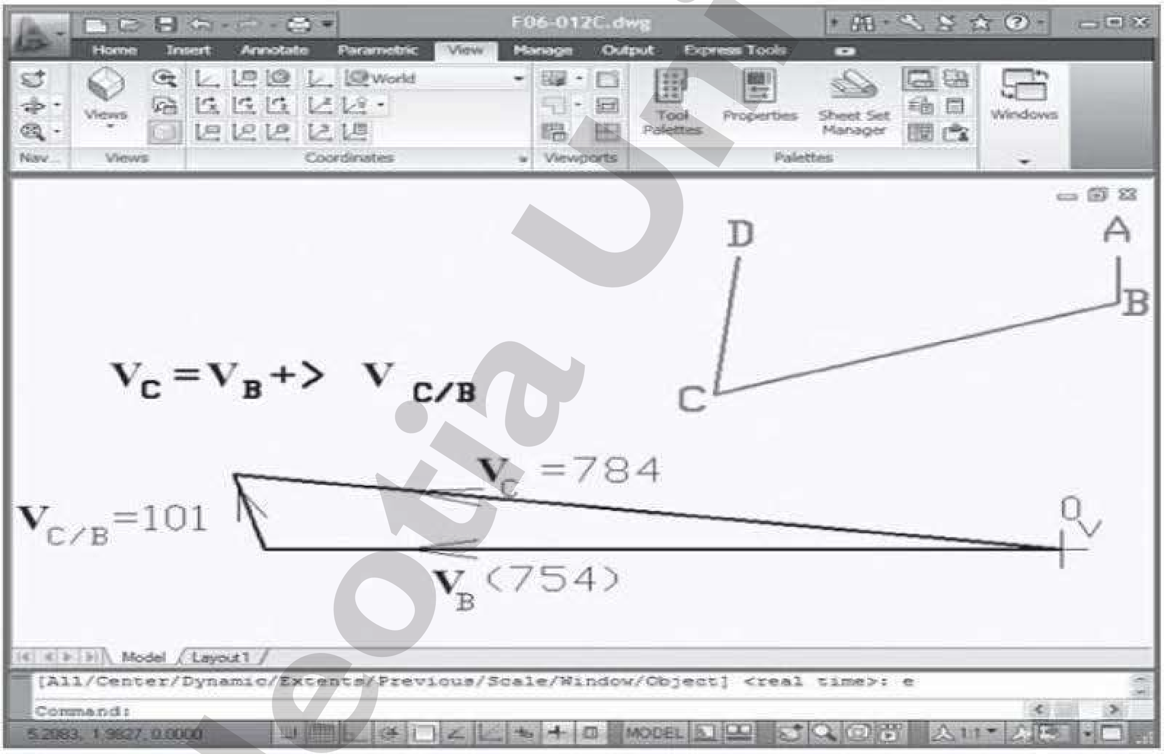
$$\omega_3 = \frac{v_{C/B}}{r_{BC}} = \frac{101.1\ \text{mm/s}}{(400\ \text{mm})} = 0.25\ \text{rad/s, cw}$$



①
360 mm
D
A
90°
②
60 mm
180 mm
④
ω_2 = 120 rpm
B
③
400 mm
C



$V_{C/B}$ (drawn perpendicular to line BC)
V_C (drawn perpendicular to line CD)
V_B
$V_C = V_B +> V_{C/B}$



F06-012C.dwg
Home
Insert
Annotate
Parametric
View
Manage
Output
Express Tools
World
Views
Tool Palettes
Properties
Sheet Set Manager
Windows
Nav...
Coordinates
Viewports
Palettes
D
A
B
C
$V_C = V_B +> V_{C/B}$
V_C =784
$V_{C/B}$=101
O
V_B (754)
Model
Layout1
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: e
Command:
MODEL

NAME OF THE EXPERIMENT: ACCELERATION ANALYSIS BY AUTOCAD

EXPERIMENT NO.: PC-REP601/03

OBJECTIVE

Acceleration analysis involves determining the manner in which certain points on the links of a mechanism are either "speeding up" or "slowing down." Acceleration is a critical property because of the inertial forces associated with it. In the study of forces, Sir Isaac Newton discovered that an inertial force is proportional to the acceleration imposed on a body. This phenomenon is witnessed anytime you lunge forward as the brakes are forcefully applied on your car. The determination of accelerations in a linkage is the purpose of this section. The primary procedure used in this analysis is the relative acceleration method, which utilizes the results of the relative velocity method introduced in the last experiment.

PRNCIPLE

Analysis can proceed throughout a mechanism by usingpoints that are common to two links. For example, a point that occurs on a joint is common to two links. Therefore, determining the acceleration of this point enables one to subsequently determine the acceleration of another point on either link. In this manner, the acceleration of any point on a mechanism can be determined by working outward from the input link. As with velocity, the following notation is used to distinguish between absolute and relative accelerations:

$\mathbf{A}_A$ = absolute acceleration (total) of point A

$\mathbf{A}_B$ = absolute acceleration (total) of point B

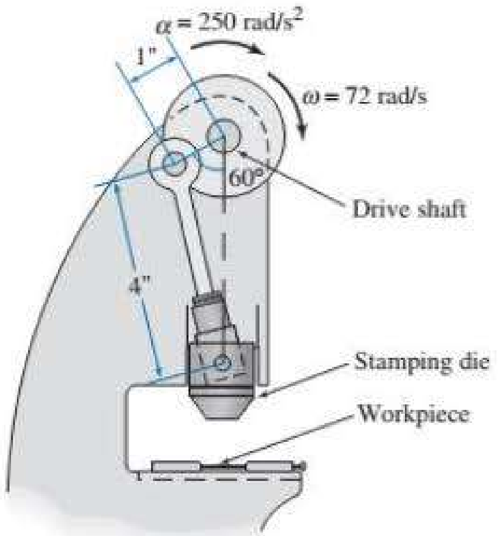
$\mathbf{A}_{B/A}$ = relative acceleration (total) of point B = acceleration (total) of point B with respect to A "as observed" from point A

Typically, it is more convenient to separate the total accelerations into normal and tangential components. Thus, each acceleration is separated into its two components, yielding the following:

$$\mathbf{A}_B^n +> \mathbf{A}_B^t = \mathbf{A}_A^n +> \mathbf{A}_A^t +> \mathbf{A}_{B/A}^n +> \mathbf{A}_{B/A}^t$$

EXAMPLE PROBLEM 1

The mechanism shown in Figure 7.13 is a common punch press designed to perform successive stamping operations. The machine has just been powered and at the instant shown is coming up to full speed. The driveshaft rotates clockwise with an angular velocity of 72 rad/s and accelerates at a rate of 250 rad/s2. At the instant shown, determine the acceleration of the stamping die, which will strike the workpiece.



Solution:

1. ***Draw the Kinematic Diagram and Identify the Degrees of Freedom***

The portion of the mechanism that is under consideration includes the driving wheel, the stamping die, and the link that connects the two. Notice that this is the common slider-crank mechanism, having a single degree of freedom. A scaled kinematic diagram is shown in the Figure.

2. ***Decide on a Method to Achieve the Desired Acceleration***

The acceleration of the die (link 4) is strictly translational motion and is identical to the motion of point *A*. The acceleration of point *A*, which also resides on link 3, can be determined from knowing the acceleration of point *B*. Point *B* is positioned on both links 2 and 3. Therefore, the acceleration of point *B* can be determined from knowing the motion of the input link, link 2.

3. ***Determine the Velocity of Points* A *and* B**

Calculating the magnitude of the velocity of point *B* is as follows:

$$\mathbf{V}_B = \omega_2 r_{AB} = (72\ \text{rad/s})(1.0\ \text{in.}) = 72\ \text{in./s} \quad 60^\circ$$

The direction of $\mathbf{V}B$ is perpendicular to link 2 and consistent with the direction of $\omega 2$, up and to the left. Using CAD, a vector can be drawn to scale, from the velocity diagram origin, to represent this velocity. The next step is to construct a velocity diagram that includes points *A* and *B*. The relative velocity equation for points *A* and *B* can be written as $\mathbf{V}_A = \mathbf{V}_B +> \mathbf{V}_{A/B}$. Thus, at the origin of the velocity diagram, a line can be drawn to represent the direction of vector $\mathbf{V}_A$. This is parallel to the sliding surface because link 4 is constrained to vertical sliding motion. At the end of the vector $\mathbf{V}_B$, a line is drawn to represent the direction of $\mathbf{V}_{A/B}$. As with all relative velocity vectors between two points on the same line, the direction is perpendicular to the line that connects points *A* and *B*. The intersection of the $\mathbf{V}_A$ and $\mathbf{V}_{A/B}$ direction lines determines the magnitudes of both vectors. The completed velocity diagram is shown in the Figure. Scaling the vector magnitudes from the diagram is determined as follows:

$$\mathbf{V}_A = 70.3\ \text{in./s} \uparrow$$

$$\mathbf{V}_{A/B} = 36.8\ \text{in./s} \quad 13^\circ$$

4. ***Calculate the Acceleration Components***

The next step is to construct an acceleration diagram that includes points *A* and *B*. Calculating the magnitudes of the known accelerations is accomplished by the equations:

$$\mathbf{A}_B^n = \frac{(v_B)^2}{r_{BC}} = \frac{(72\ \text{in./s})^2}{1.0\ \text{in.}} = 5184\ \text{in./s}^2 \quad 30^\circ$$

(directed toward the center of rotation, point *C*)

$$\mathbf{A}_B^t = \alpha_2 r_{AB} = (250\ \text{rad/s}^2)(1.0\ \text{in.}) = 250\ \text{in./s}^2 \quad 60^\circ$$

(directed perpendicular to *BC* in the direction of rotational acceleration)

$$\mathbf{A}_{A/B}^n = \frac{(v_{A/B})^2}{r_{AB}} = \frac{(36.8\ \text{in./s})^2}{4.0\ \text{in.}} = 338\ \text{in./s}^2 \quad 77^\circ$$

(directed from *A* toward *B*, measured from CAD)

Note that point *A* does not have a normal acceleration because the motion is strictly translational.

5. ***Construct an Acceleration Diagram***
The relative acceleration equation for points A and B can be written as

$$\mathbf{A}_B^n +> \mathbf{A}_B^t = \mathbf{A}_A^n +> \mathbf{A}_A^t +> \mathbf{A}_{B/A}^n +> \mathbf{A}_{B/A}^t$$

In forming the acceleration diagram, vector construction will arbitrarily start on the right side of the equation. At the origin of the acceleration diagram, a line can be drawn to represent the vector A_B^n which is known. At the end of A_B^n, a line can be drawn to represent vector A_B^t which is also known. At the end of vector A_B^t another line can be drawn to represent vector $A_{A/B}^n$ which is also known. At the end of this vector, a line can be drawn to represent the direction of vector $A_{A/B}^t$. This is perpendicular to the normal component $A_{A/B}^n$, but has an unknown magnitude.
Focusing on the left side of the equation, a new series of vectors will begin from the origin of the acceleration diagram. The vector A_A^n has zero magnitude and is ignored. A line can be drawn to represent the direction of vector A_A^t; however, the magnitude is unknown. The line is directed parallel to the sliding motion of link 4. Finally, the intersection of the A_A^t and $A_{A/B}^t$ direction lines determines the magnitudes of both vectors. The completed acceleration diagram is shown in Figure.

6. ***Measure the Desired Acceleration Components***
Scaling the vector magnitudes from the diagram is done with the following:

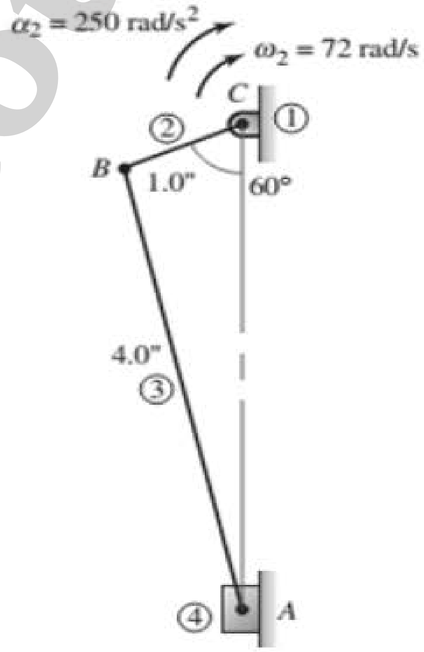
$$\mathbf{A}_{A/B}^t = 4404 \text{ in./s}^2 \quad \overline{13^\circ \swarrow}$$

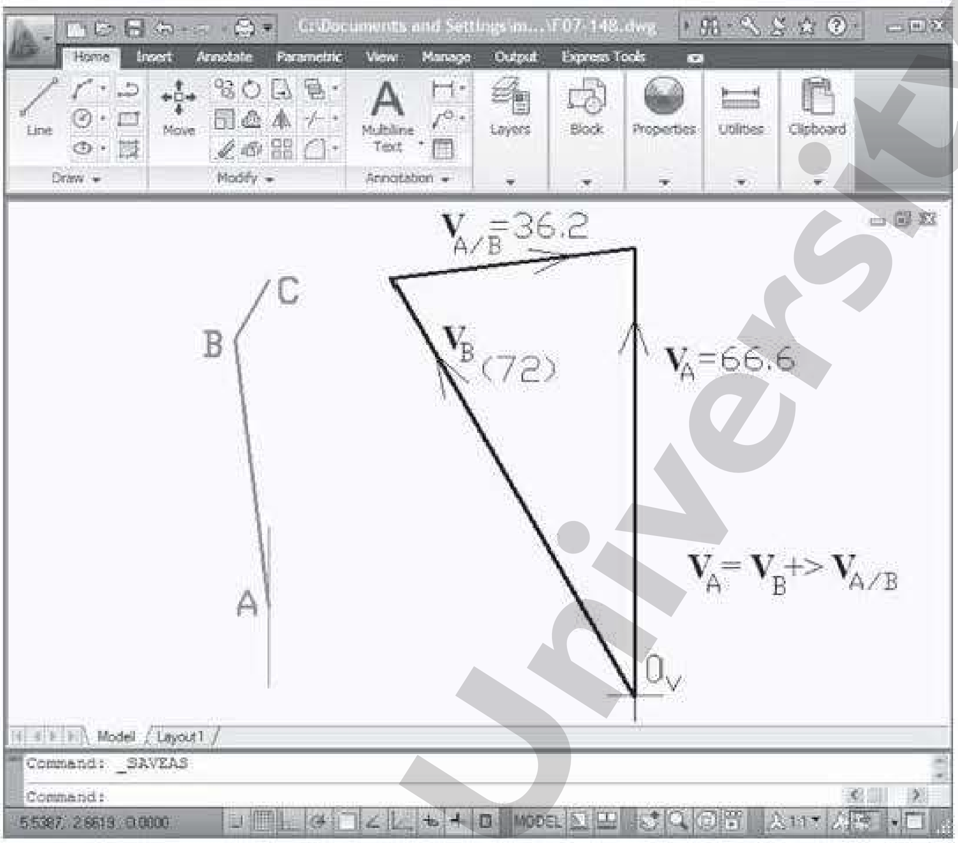
$$\mathbf{A}_A^t = 2138 \text{ in./s}^2 \uparrow$$

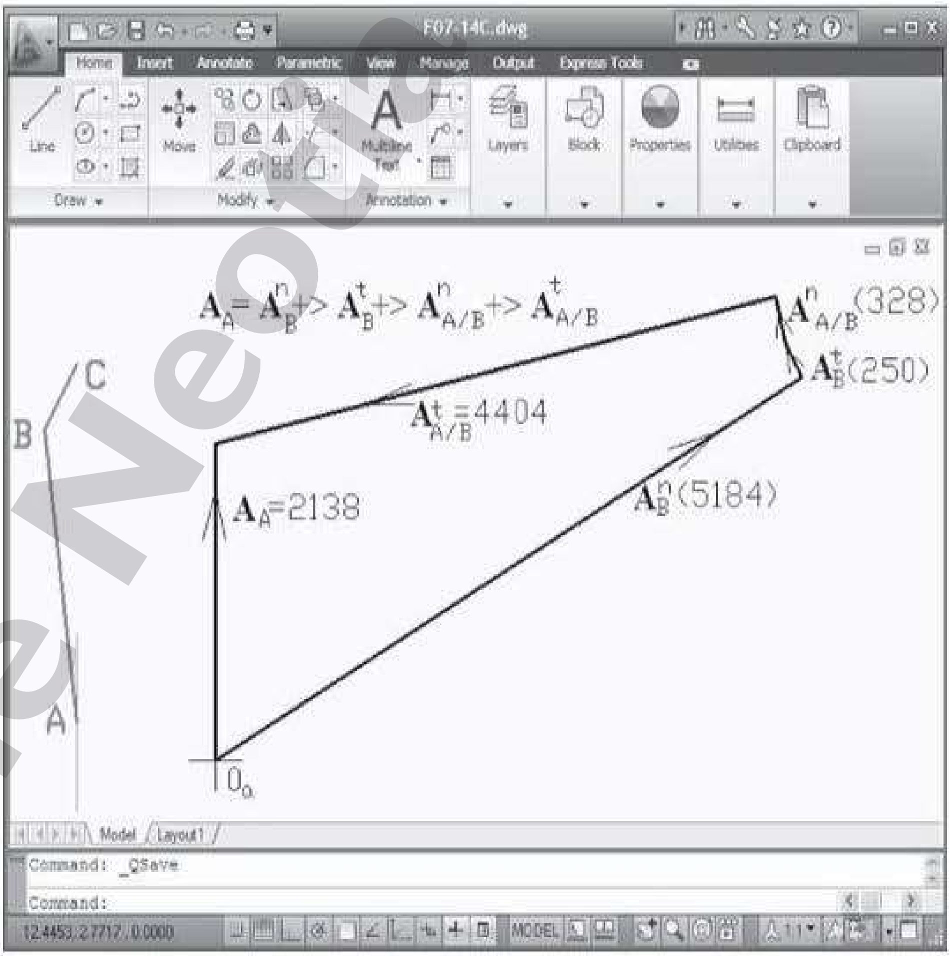
Thus, the total acceleration of point A is

$$\mathbf{A}_A = \mathbf{A}_A^t = 2138 \text{ in./s}^2 = 178 \text{ ft/s}^2 = 5.53 \text{ g} \uparrow$$

Notice that the tangential acceleration of point A is in the same direction as the velocity. This indicates that point A is accelerating (speeding up), not decelerating.




C:\Documents and Settings\m...\F07-14B.dwg
$V_{A/B}=36.2$
V_B (72)
$V_A=66.6$
$V_A = V_B +> V_{A/B}$
0_V
Command: _SAVEAS
Command:


F07-14C.dwg
$A_A = A_B^n +> A_B^t +> A_{A/B}^n +> A_{A/B}^t$
$A_{A/B}^n$ (328)
A_B^t (250)
$A_{A/B}^t = 4404$
$A_A = 2138$
A_B^n (5184)
0_a
Command: _QSave
Command:

NAME OF THE EXPERIMENT: CAM DESIGN BY AUTOCAD

EXPERIMENT NO.: PC-REP601/04

OBJECTIVE

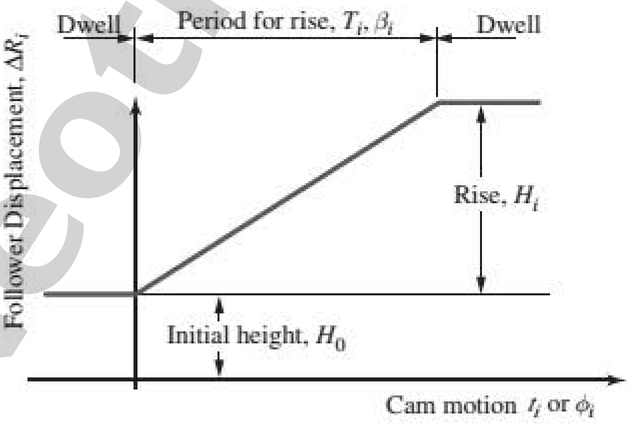
A cam is a common mechanism element that drives a mating component known as a follower. From a functional viewpoint, a cam-and-follower arrangement is very similar to the linkages discussed throughout this book. The cam accepts an input motion similar to a crank and imparts a resultant motion to a follower. The unique feature of a cam is that it can impart a very distinct motion to its follower. In fact, cams can be used to obtain unusual or irregular motion that would be difficult, or impossible, to obtain from other linkages. Because the motion of cams can be prescribed, they are well suited for applications where distinct displacements and timing are paramount

PRNCIPLE

In designing a cam, the objective is to identify a suitable shape for the cam. The primary interest is to ensure that the follower will achieve the desired displacements. Of course, these displacements are outlined in the displacement diagram. The shape of the cam is merely a means to obtain this motion. The follower, during its travel, may have one of the following motions. 1. Uniform velocity, 2. Simple harmonic motion, 3. Uniform acceleration and retardation, and 4. Cycloidal motion.

Displacement diagram of follower moves with Uniform Velocity

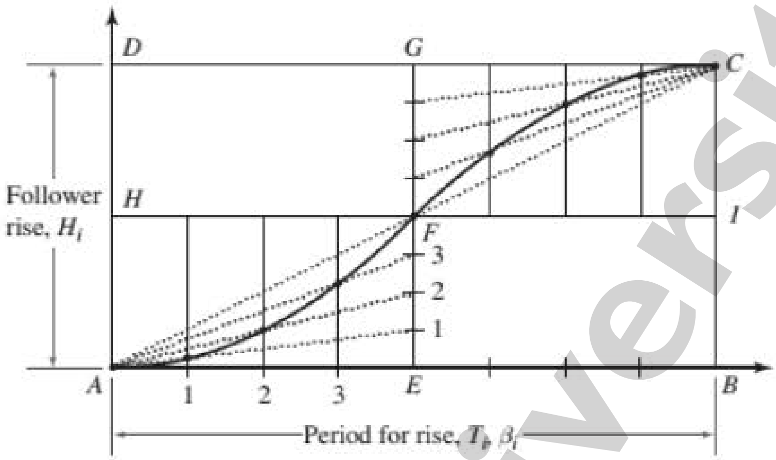
Since the follower moves with uniform velocity during its rise and return stroke, therefore the slope of the displacement curves must be constant. A little consideration will show that the follower remains at rest during part of the cam rotation. The periods during which the follower remains at rest are known as dwell periods, as shown in Fig.



Displacement diagram of follower moves with Uniform Acceleration and Deceleration

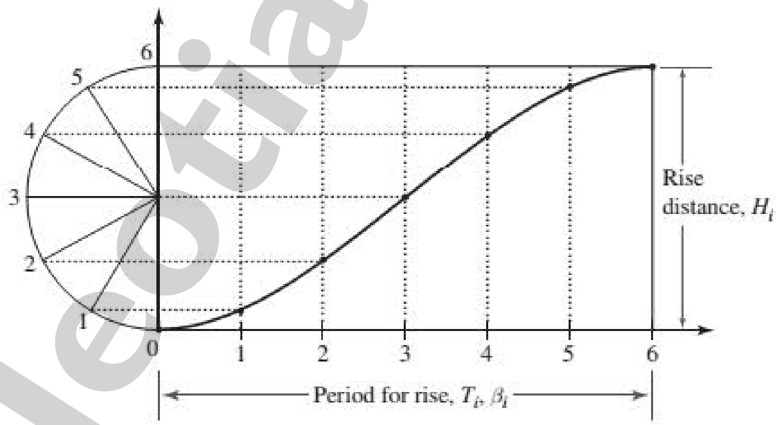
1. Divide the follower rise (or fall) sequence into two halves. *AE* represents the time period and *EF* the magnitude of rise for the first half of this motion scheme.
2. Divide both the horizontal and vertical axes of the quadrant *AEFH* into equal parts.
3. Construct vertical lines from the horizontal divisions.
4. Construct straight lines from corner *A* to the vertical divisions.
5. Draw a smooth curve through the points of intersection of the vertical lines and the lines drawn from corner *A*.

6. Repeat steps 2 through 5 for the remaining half of the curve as shown in quadrant *FICG* in the figure



Displacement diagram of follower moves with Simple Harmonic Motion

1. Draw a semi-circle on the follower stroke as diameter.
2. Divide the semi-circle into any number of even equal parts (say eight).
3. Divide the angular displacements of the cam during out stroke and return stroke into the same number of equal parts.
4. The displacement diagram is obtained by projecting the points as shown in Fig.



GRAPHICAL DISK CAM PROFILE DESIGN

In-Line Roller Follower

Using the displacement diagram from the above, a cam profile to be used with an in-line roller follower has been constructed and shown in Figure below. The following general procedure is used to construct such a profile:

1. Draw the base circle of radius *Rb*. The size is typically a function of the spatial constraints of the application.

2. Draw the follower of radius *Rf* in the home position, tangent to the base circle.

3. Draw radial lines from the center of the cam, corresponding to the cam angles identified on the displacement diagram. For construction purposes, the cam will remain stationary and the follower will be rotated in a direction opposite to the actual cam rotation.

4. Identify the trace point at the home position. For a roller follower, this is the point at the center of the roller.

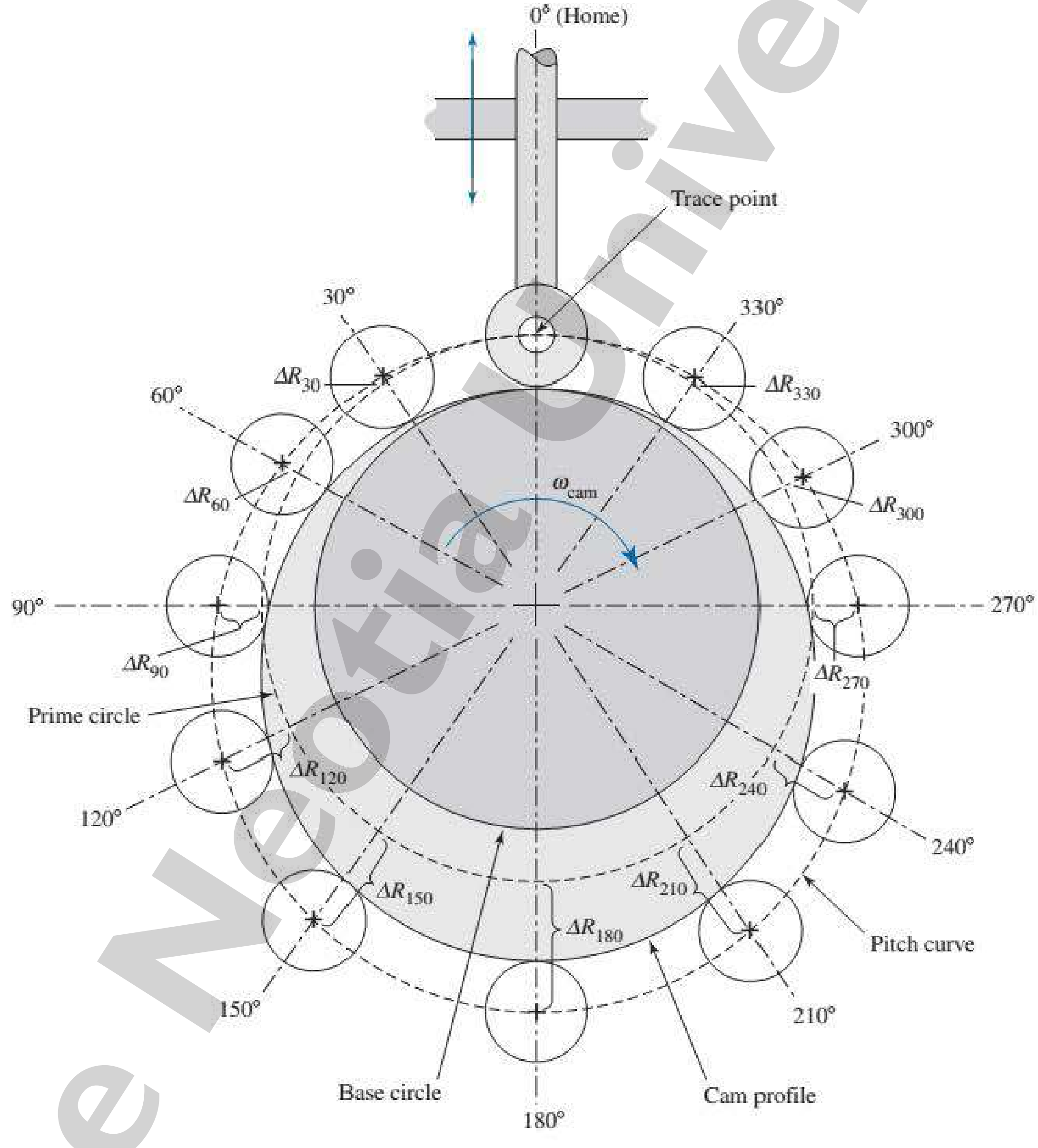
5. Draw the prime circle through the trace point at its home position.

6. Transfer the displacements from the displacement diagram to the radial lines. Measure these displacements from the prime circle.

7. Draw the roller outline of radius *Rf*, centered at the prescribed displacements identified in the previous step.

8. Draw a smooth curve tangent to the roller at these prescribed displacements.

9. To accurately construct a profile consistent with the displacement diagram, it may be necessary to transfer additional intermediate points from the rise and fall intervals.

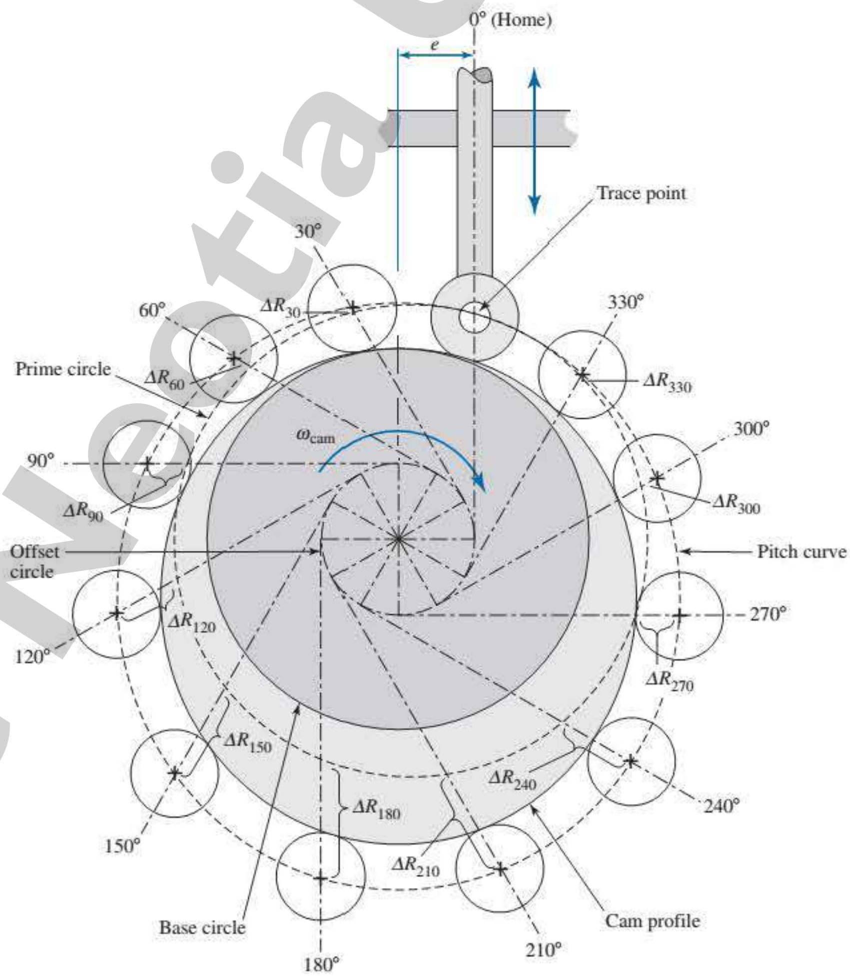


Offset Roller Follower

The most efficient manner of describing the construction of a cam with an offset roller follower is through an actual construction. Using the displacement diagram a cam profile to be used with an offset roller follower has been constructed and shown in Figure below. The following general procedure is used to construct such a profile.

1. Draw the base circle of radius *Rb*. The size is typically a function of the spatial constraints

of the application.
2. Draw the follower centerline in the home position.
3. Draw the prime circle, whose radius is equal to the sum of the base and roller follower radii
4. Draw the follower in the home position of radius *Rf*, centered where the follower centerline intersects the prime circle. (*Rb* 6 *Rf*).
5. Identify the trace point at the home position. For a roller follower, this is the point that is at the center of the roller.
6. Draw an offset circle of radius *e*, centered at the cam rotation axis. It will be tangent to the follower centerline.
7. Draw lines tangent to the offset circle, corresponding to the reference cam angles on the displacement diagram. For construction purposes, the cam will remain stationary and the follower will be rotated in a direction opposite to the actual cam rotation.
8. Transfer the displacements from the displacement diagram to the offset lines. Measure these displacements from the prime circle.
9. Draw the roller outline of radius *Rf*, centered at the prescribed displacements identified in the previous step
10. Draw a smooth curve tangent to the roller at these prescribed displacements.
11. To accurately construct a profile consistent with the displacement diagram, it may be necessary to transfer additional intermediate points from the rise and fall intervals.



NAME OF THE EXPERIMENT: SINGLE-STAGE GEAR BOX

EXPERIMENT NO.: PC-REP401/05

OBJECTIVE

Gears are machine elements, which are used for power transmission between shafts, separated by small distance. Irrespective of the type, each gear is provided with projections called teeth and intermediate depressions called tooth spaces. While two gears are meshing, the teeth of one gear enter the spaces of the other. Thus, the drive is positive and when one gear rotates, the other also rotates; transmitting power from one shaft to the other.

PRINCIPLE

Gears are classified on the basis of the shape of the tooth profile and the relative position of the shafts between which, power transmission takes place. The pictorial views of some of the most commonly used gear trains, are shown in Figure: 5.1. A number of curves may be used for the tooth profile. However, from a commercial stand point, cycloidal and involute curves are used. Of these two, involute form is extensively used because of its advantages from manufacturing and operational points of view. Involute is a curve traced by a point on a straight line when it rolls without slipping, on the circumference of a circle.

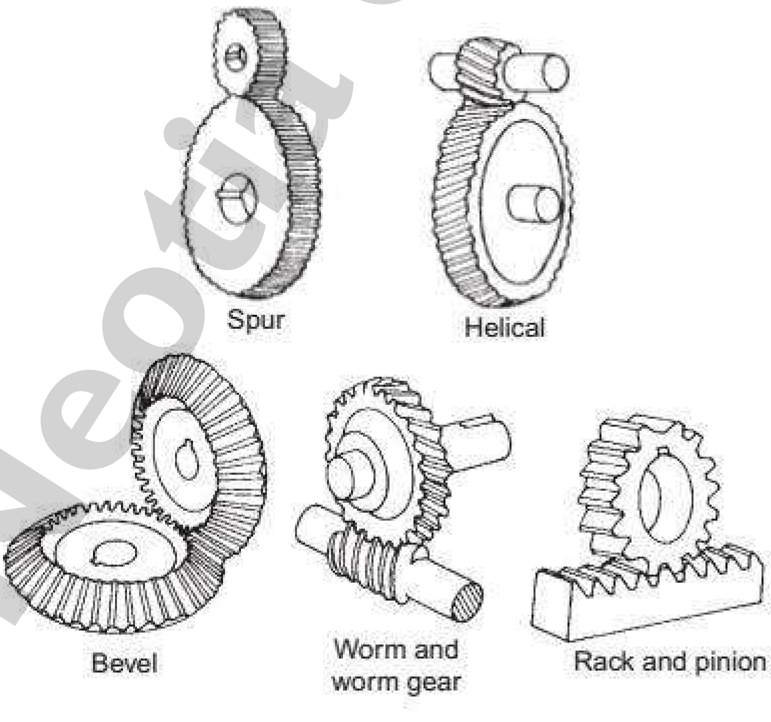


Figure: 5.1 Types of Gears

A gear has 30 teeth of involute profile, pitch circle diameter of 180mm and pressure angle of 20°. Draw the profile of four complete teeth for the gear. Sketch the views of two gears, a pinion and gear in mesh. Indicate the required parameters.

1. With O as centre and radius equal to the pitch circle radius, draw an arc.
2. At any point P on it, drawn a line T-T, tangential to the above arc.
3. Through the point P, draw the line of action N-N, making an angle equal to the pressure angle φ, with the tangent line T-T.
4. From the centre O, draw the line OQ, perpendicular to the line of action (it will make an angle φ with OP).
5. With O as centre and radius equal to OQ, draw an arc, representing the base circle.
6. With O as centre, draw arcs, representing addendum and dedendum circles.
7. Starting from any point on the base circle, construct an involute curve, as shown at X.
8. Trace the curve and a part of the base circle, on a piece of tracing paper, as shown at Y.
9. On the pitch circle, mark points 1, 2, 3, 4, etc., separated by a distance equal to half of the circular pitch.
10. Place the tracing paper, such that the arc AB coincides with the base circle and the curve passes through the point 1.
11. Prick a few points on the curve, lying between the addendum and base circles.
12. Join these points by a smooth curve.
13. Draw a radial line below the base circle and join it with the bottom land, by means of fillet of radius r, which may be taken as 0.125 pc.
14. Reverse the tracing paper, follow the steps 11 to 13 and complete the curve through the point 2; obtaining one tooth profile.
15. Repeat the steps 11 to 14 and construct the other tooth profiles.

Spur Gearing:
Gears with pitch circle diameters less than 10cms are produced from solid blanks, with uniform thickness. When pitch circle diameters lie between 10 to 25 cms, gears are produced with a web connecting the hub and rim. The web thickness (Tw) may be taken as equal to the circular pitch of the gear. Still larger gears are produced with arms; the number being dependent upon the pitch circle diameter. The rim thickness Tr, i.e., the thickness of the metal under the teeth may be taken as equal to the depth of the tooth. Two spur gears in mesh is known as spur gearing. In all gearings except worm gearing, the smaller of the two gears is called the pinion and the larger one, the gear or gear wheel. Figure: 4.3 shows the views of spur gearing, indicating the required parameters.

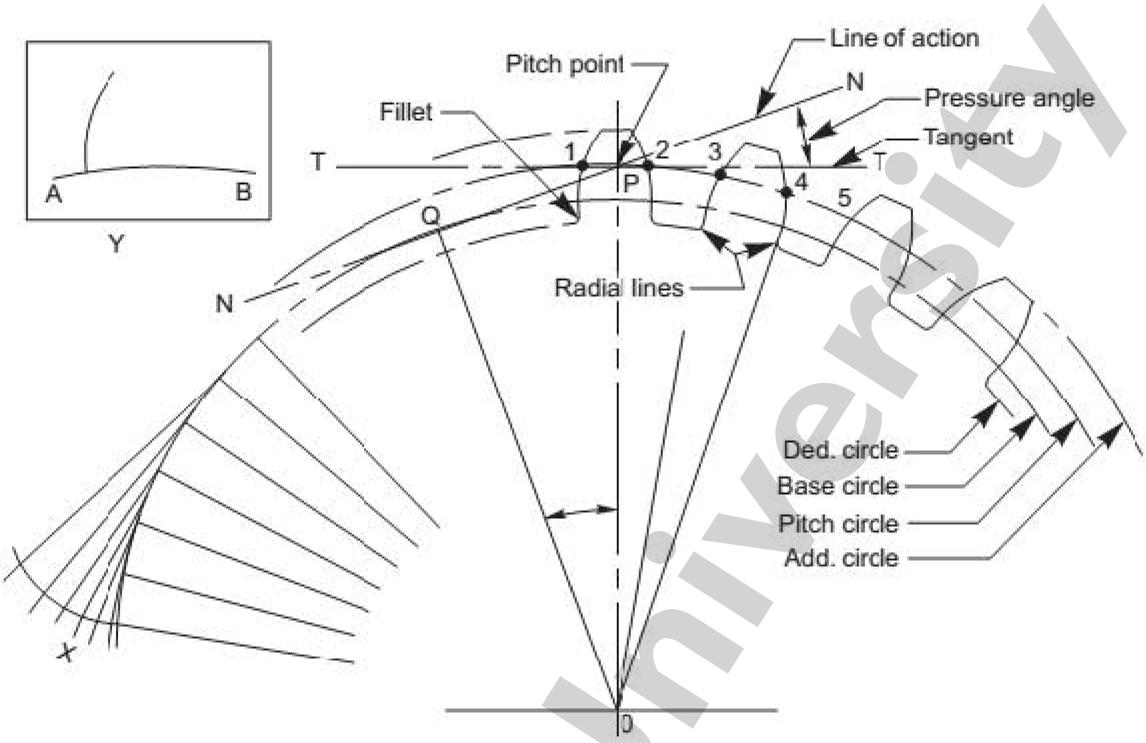


Figure: 4.2 Method of drawing involute tooth profile

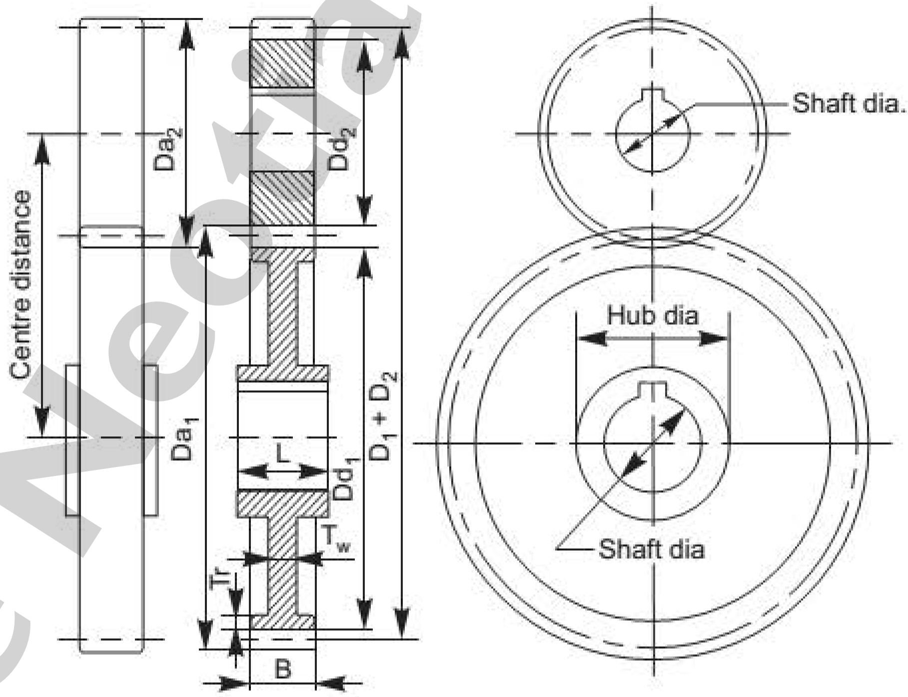


Figure: 4.3 Spur gearing

NAME OF THE EXPERIMENT: SOLID MODELLING IN Auto CAD

EXPERIMENT NO.: PC-REP401/06

OBJECTIVE

AutoCAD software has helped professionals around the world continue to innovate in the architecture, engineering, and design fields. In this guide to Autodesk's newest version of AutoCAD software, you'll discover improvements that have made it easier to use than ever– from new 3D design tools to enhanced PDF support and improved documentation, you'll have access to an arsenal of robust features. As an added benefit, this state-of-the-art software makes it easy to connect and collaborate with other AED professionals, improving efficiency and accuracy where it matters most.

Creating Solids

1. Solid Primitives

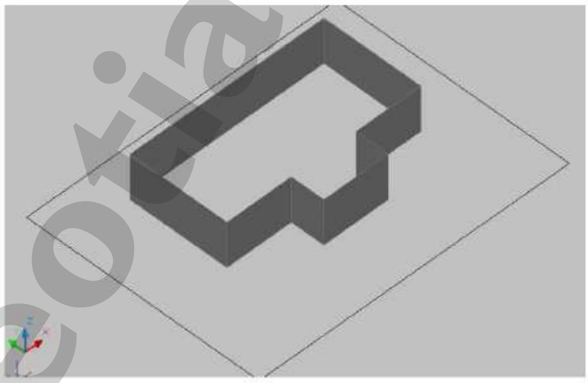
Solid primitives can easily be drawn from from the 3D Modeling panel, Solid Panel.

2. Polysolid Command

With the POLYSOLID command, you can convert an existing line, 2D polyline, arc, or circle to a solid with a rectangular profile. A polysolid can have curved segments, but the profile is always rectangular by default.

1. Open a drawing with a closed 2D polyline and display in a 3D view.

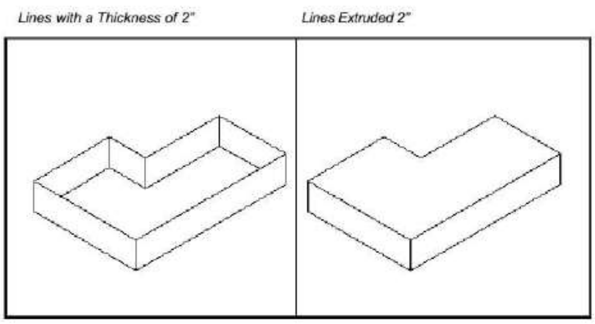
2. Type POLYSOLID at the command prompt. Command: **polysolid** Specify start point or [Object/Height/Width/Justify] <Object>: **h** Specify height <0'-4">: **10** Specify start point or [Object/Height/Width/Justify] <Object>: **o** Select object: Pick polygon



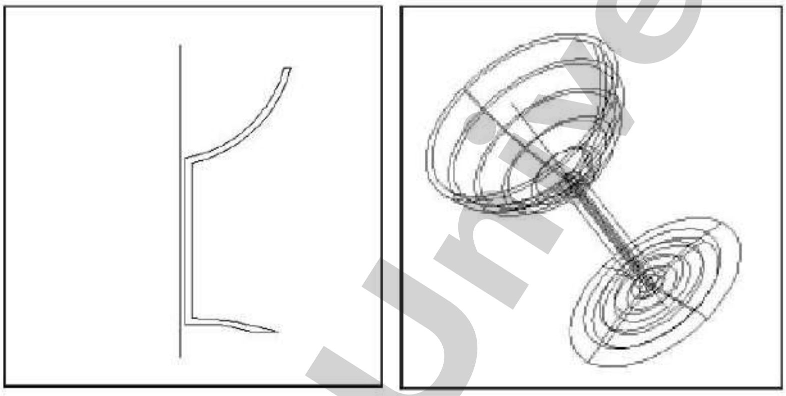
3 Extrude

Creates unique solid primitives by extruding existing two-dimensional objects. You can extrude multiple objects with EXTRUDE.

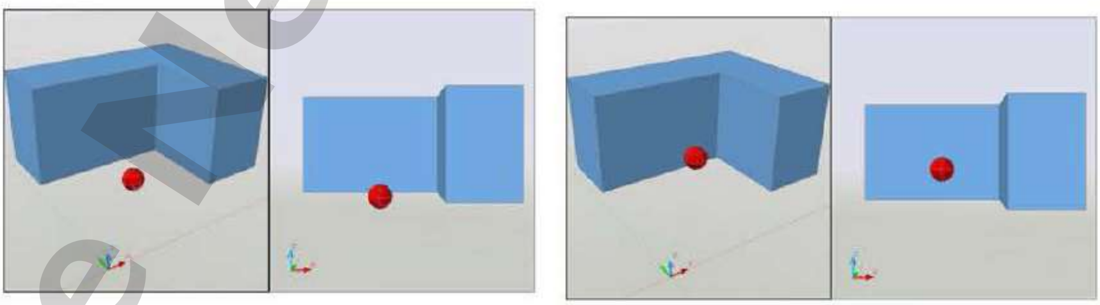
1. Type EXTRUDE at the command prompt. Command: **extrude** Current wire frame density: ISOLINES=4 Select objects: **pick objects** Select objects: **enter** Specify height of extrusion or [Direction/Path/Taper angle]: **2**



4. Revolve Command
1. Open a drawing with 3D objects and display in a 3D view.
2. Type REVOLVE at the command prompt.
Command: **revolve**
Current wire frame density: ISOLINES=4
Select objects: **pick profile**
Select objects: **enter**
Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]: **o**
Select an object: **pick axis**
Specify angle of revolution <360>: **enter**

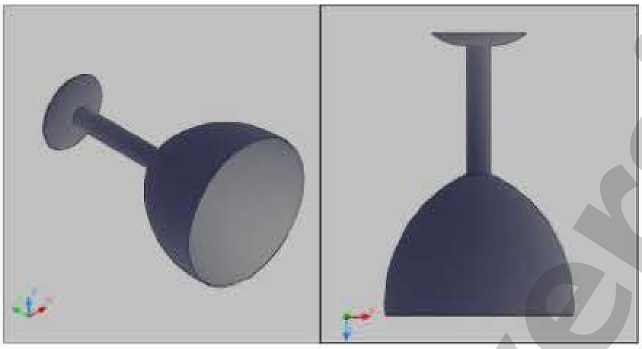


3D Edits
1. 3D Move
Displays the move grip tool in a 3D view and moves objects a specified distance.
1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, 3D Operations, 3DMove. or
3. Type 3DMOVE at the command prompt.
Command: **3Dmove**
Select objects: pick object to move 1 found
Select objects: enter Specify base point or [Displacement] <Displacement>: D
Specify displacement <0.0000, 0.0000, 0.0000>: **0,0,2**

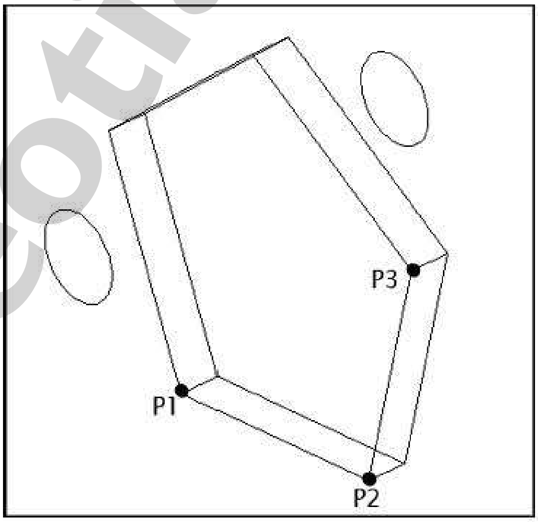


3D Rotate
1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, 3D Operations, 3DRotate. or
3. Type 3DROTATE at the command prompt.
Command: **3DROTATE** Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0 Select objects: pick object and press enter Specify base point: **pick point**

Pick a rotation axis: **select X axis**
Specify angle start point: -**90**

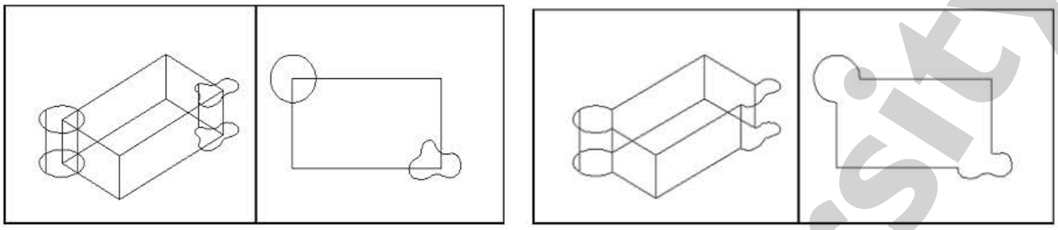


3D Mirror
1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, 3D Operations, 3DMirror. or
3. Type MIRROR3D at the command prompt.
Command: **mirror3D**
Select objects: **pick the circle**
Select objects: **enter**
Specify first point of mirror plane (3 points) or [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: **P1**
Specify second point on mirror plane: **P2**
Specify third point on mirror plane: **P3**
Delete source objects? [Yes/No] <N>: **enter** *CircleMirroredaround3Points*



Union
1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, Solids Editing, Union. or
3. Type UNION at the command prompt.
Command: **UNION**
Select objects: pick objects to union

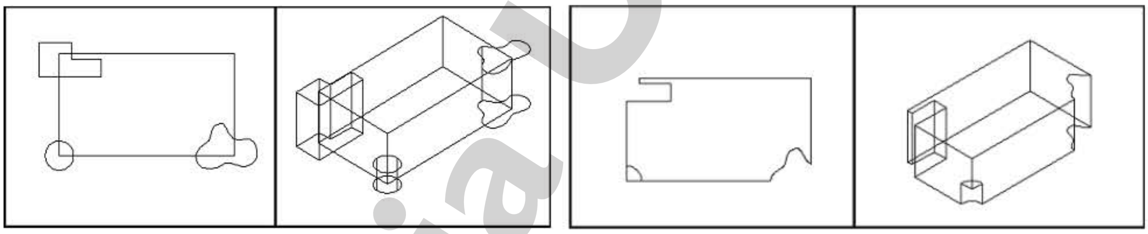
Select objects: **ENTER**



Subtract

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, Solids Editing, Subtract. or
3. Type SUBTRACT at the command prompt.

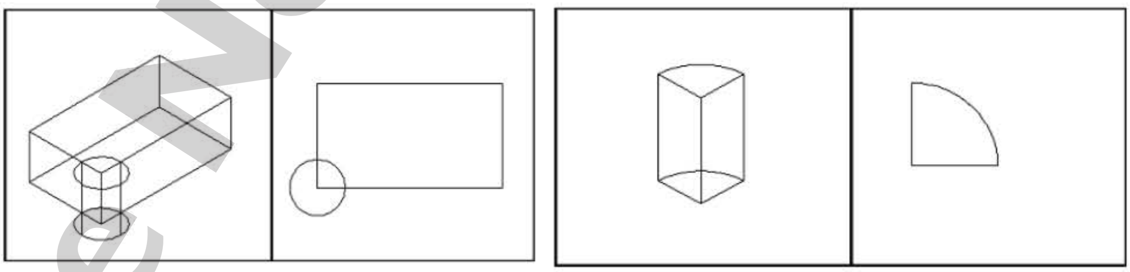
Command: **SUBTRACT** SUBTRACT
Select solids and regions to subtract from...
Select objects: **pick the main box**
Select objects: **(press enter)** Select solids and regions to subtract...
Select objects: **pick the other solids**
Select objects: **enter**



Intersect

1. Choose Modify, Solids Editing, Intersect or
2. Type INTERSECT at the command prompt.

Command: **INTERSECT**
Select objects: pick objects
Select objects: **enter**



Exercise: Create 3-D solids of Figs. 1.5, 1.6, 1.7 and 1.8. in AutoCAD

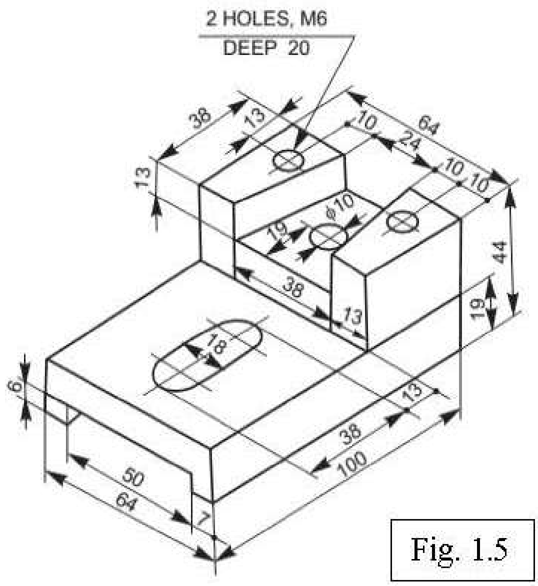


Fig. 1.5

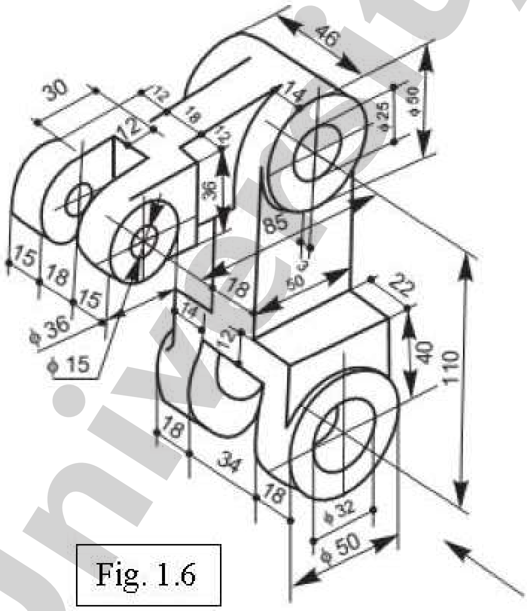


Fig. 1.6

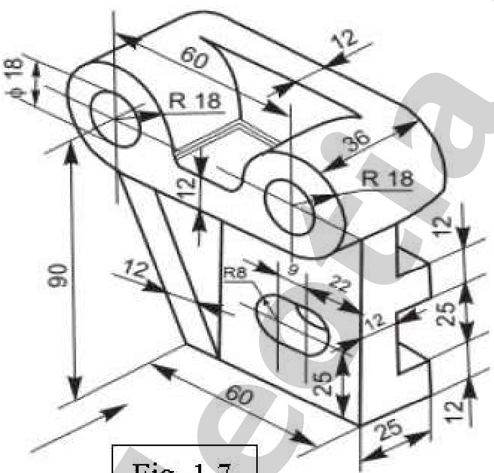


Fig. 1.7

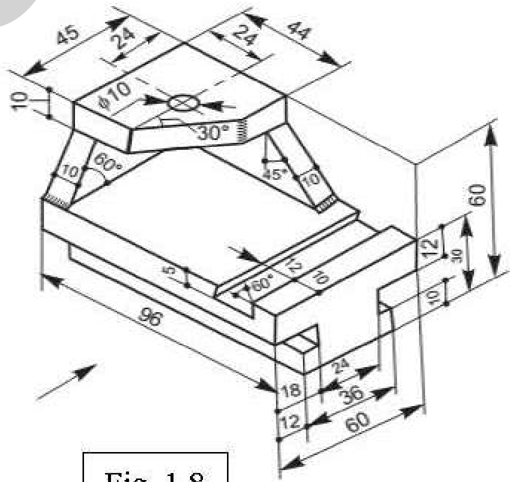


Fig. 1.8