ARCHITECTURAL ASSISTANT

2nd Semester

TRADE PRACTICAL

SECTOR: Construction, Construction Material & Real Estate

Directorate General of Training

DIRECTORATE GENERAL OF TRAINING
MINISTRY OF SKILL DEVELOPMENT & ENTREPRENEURSHIP
GOVERNMENT OF INDIA

NATIONAL INSTRUCTIONAL MEDIA INSTITUTE, CHENNAI

Post Box No. 3142, CTI Campus, Guindy, Chennai - 600 032

Sector : Construction, Construction Material & Real Estate

Duration : 2 - Years

Trades : Architectural Assistant 2nd semester - Trade Practical

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FOREWORD

The National Instructional Media Institute (NIMI) is an autonomous body under the Directorate General of Employment and Training (DGE&T) Ministry of Labour and Employment has been developing, producing and disseminating Instructional Media Packages (IMPs are extensively used in the Industrial Training Institutes/Training centres in Industries to impart practical training and develop work-skills for the trainees and the trainers

The Ministry of Labour & Employment constituted Mentor Councils (MCs) to revampcourses run / to be run under National Council of Vocational Training (NCVT) in 25 sectors. The MCs have representatives from thought leaders among various stakeholders viz. one of the top ten industries in the sector innovative entrepreneurs who have proved to be game-changers, academic/professional institutions (IITs etc.), experts from field institutes of DGE &T, champion ITIs for each of the sectors and experts in delivering education and training through modern methods like through use of IT, distance education etc. The technical support to the MCs is provided by Central Staff Training and Research Institute (CSTARI), Kolkata and National Instructional Media Institute (NIMI), Chennai. Some of the MCs are also supported by sector-wise Core Groups which were created internally in the Ministry (in 11 sectors).

A Steering Committee to provide overall coordination and guidance to Mentor Councils has also been constituted and has representation from the MCs, Chair positions to be endowed by the Ministry, trade unions, and experts on distance education and training. The MCs are mandated to work towards revamping/suggesting new courses, improving assessment systems, overall learning etc. for subjects under the purview of the NCVT.

Accordingly NIMI with the support and assistance of MC has developed **Architectural Assistant Trade Practical 2nd Semester** in **Construction, Construction Material & Real Estate sector** to enhance the employability of ITI trainees across the country and also to meet the industry requirement.

I have no doubt that the trainees and trainers of ITIs & Training centres in industries will derive maximum benefit from these books and that NIMI's effort will go a long way in improvement of Vocational Training.

I complement Director, Mentor Council members, Media Development Committee (MDC) members and staff of NIMI for their dedicated and invaluable contribution in bringing out this publication.

ALOK KUMAR, I.A.S.,
Director General of Employment &
Training/ Joint Secretary
Ministry of Labour and Employment
Government of India

New Delhi - 110 001

PREFACE

This National Instructional Media Institute (NIMI) was set up at Chennai by the Directorate General of Employment and Training (DGE&T) Ministry of Labour and Employment, Government of India with technical assistance from the Govt. of the Federal Republic of Germany. The prime objective of this institute is to develop and disseminate instructional materials for various trades as per the prescribed syllabi under the Craftsmen and Apprenticeship Training Schemes.

The instructional materials are developed and produced in the form of Instructional Media Packages (IMPs). An IMP consists of Trade Theory book, Trade Practical book, Test and Assignment book, Instructor guide, Wall Charts and Transparencies.

Hon'ble Union Minister of Finance during the budget speech 2014-2015 mentioned about developing **Skill India** and made the following announcement

"A national multi-skill programme called Skill India is proposed to be launched. It would skill the youth with an emphasis on employability and entrepreneur skills. It will also provide training and support for traditional professions like welders, carpenters, cobblers, masons, blacksmiths, weavers etc. Convergence of various schemes to attain this objective is also proposed."

The Ministry of Labour & Employment constituted Mentor Councils (MCs) to revamp courses run / to be run under National Council of Vocational Training (NCVT) in 25 sectors which will give a sustained skill based employability to the ITI trainees as the main objective of Vocational training. The ultimate approach of NIMI is to prepare the validated IMPs based on the exercises to be done during the course of study. As the skill development is progressive the theoretical content on a particular topic is limited to the requirement in every stage. Hence the reader will find a topic spread over a number of units. The test and assignment will enable the instructor to give assignments and evaluate the performance of a trainee. If a trainee possesses the same it helps the trainee to do assignment on his own and also to evaluate himself. The wall charts and transparencies are unique, as they not only help the instructor to effectively present a topic but also helps the trainees to grasp the technical topic quickly. The instructor guide enables the instructor to plan his schedule of instruction, plan the raw material requirement ,

Thus the availability of a complete Instructional Media Package in an institute helps the trainer and management to impart an effective training. Hence it is strongly recommended that the Training Institutes/Establishments should provide at least **one IMP** per unit. This will be small, one time investment but the benefits will be long lasting.

The **Architectural Assistant Trade Practical 2nd Semester in Construction, Construction Material & Real Estate sector** is one of the book develop by the core group members of the Mentor Councils (MCs). The 1st semester book includes **Module 1 - , Module 2 - , Module 3 - .**

The **Architectural Assistant Trade Practical 2nd semester** is the outcome of the collective efforts of Members of Mentor Council which includes academic/professional institutions (IITs etc.), experts from field institutes of DGE&T, champion ITIs for each of the sectors, and also Media Development Committee (MDC) members and staff of NIMI.

NIMI wishes that the above material (Trade Practical & Trade Theory) will fulfil to satisfy the long needs of the Trainees and Instructor and helps the trainees for their employability in vocational training.

NIMI would like to take this opportunity to convey sincere thanks to all the Mentor Council members and Media Development Committee (MDC) members.

A.MAHENDIRAN
Director, NIMI.

Chennai - 600 032

ACKNOWLEDGEMENT

National Instructional Media Institute (NIMI) sincerely acknowledges with thanks for the co-operation and contribution extended by the following Media Developers and their sponsoring organisation to bring out this IMP **(Trade Practical)** for the trade of **Architectural Assistant** under the **Construction, Construction Material & Real Estate** Sector for Craftsman Training Scheme. This Book is prepared as per Revised Syllabus.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Dr. N. Dhang	-	Professor. D/O Civil Engineering, Indian Institute of Technology, Kharagpur Chairman, Mentor council.
Shri. M.C. Sharma	-	Joint Director (TTC), DGE&T, New Delhi, Mentor, Mentor council.
Smt. Arpana Singh	-	Training Officer, NVTI(W),Noida. Team leader, Mentor council.
Smt. P. Kavitha Shankar	-	Vocational Instructor, NVTI (w), Noida, Member, Mentor Council.
Shri. R.N.Manna	-	Training Officer CSTARI, Kolkatta. Co-ordinator, Mentor Council.
Shri.V.Gopalakrishnan	-	Training Officer NIMI, Chennai-32. Co-ordinator, NIMI, Chennai.

NIMI records its appreciation of the Data Entry, CAD, DTP Operators for their excellent and devoted services in the process of development of this Instructional Material.

NIMI also acknowledges with thanks, the invaluable efforts rendered by all other staff who have contributed for the development of this Instructional Material.

NIMI is grateful to all others who have directly or indirectly helped in developing this IMP.

INTRODUCTION

TRADE PRACTICAL

The trade practical manual is intented to be used in workshop . It consists of a series of practical exercies to be completed by the trainees during the Second Semester course of the Architectural Assistant trade supplemented and supported by instructions/ informations to assist in performing the exercises. These exercises are designed to ensure that all the skills in the prescribed syllabus are covered.

The manual is divided into Three modules. The distribution of Exercises for the practical in the four modules are given below.

Module 1	21 Exercises
Module 2	19 Exercises
Module 3	06 Exercises
Total	46 Exercises

The skill training in the shop floor is planned through a series of practical exercises centred around some practical project. However, there are few instance where the individual exercise does not form a part of project.

While developing the practical manual a sincere effort was made to prepare each exercise which will be easy to understand and carry out even by below average traninee. NIMI, looks forward to the suggestions from the experienced training faculty for improving the manual.

TRADE THEORY

The manual of trade theory consists of theoretical information for the Second Semester course of the Architectural Assistant Trade. The contents are sequenced according to the practical exercise contained in the manual on Trade practical. Attempt has been made to relate the theoretical aspects with the skill covered in each exercise to the extent possible. This co-relation is maintained to help the trainees to develope the perceptional capabilities for performing the skills.

The Trade theory has to be taught and learnt along with the corresponding exercise contained in the manual on trade practical. The indicating about the corresponding practical exercise are given in every sheet of this manual.

It will be preferable to teach/learn the trade theory connected to each exercise atleast one class before performing the related skills in the shop floor. The trade theory is to be treated as an integrated part of each exercise.

The material is not the purpose of self learning and should be considered as supplementary to class room instruction.

CONTENTS

Starting procedure of CAD

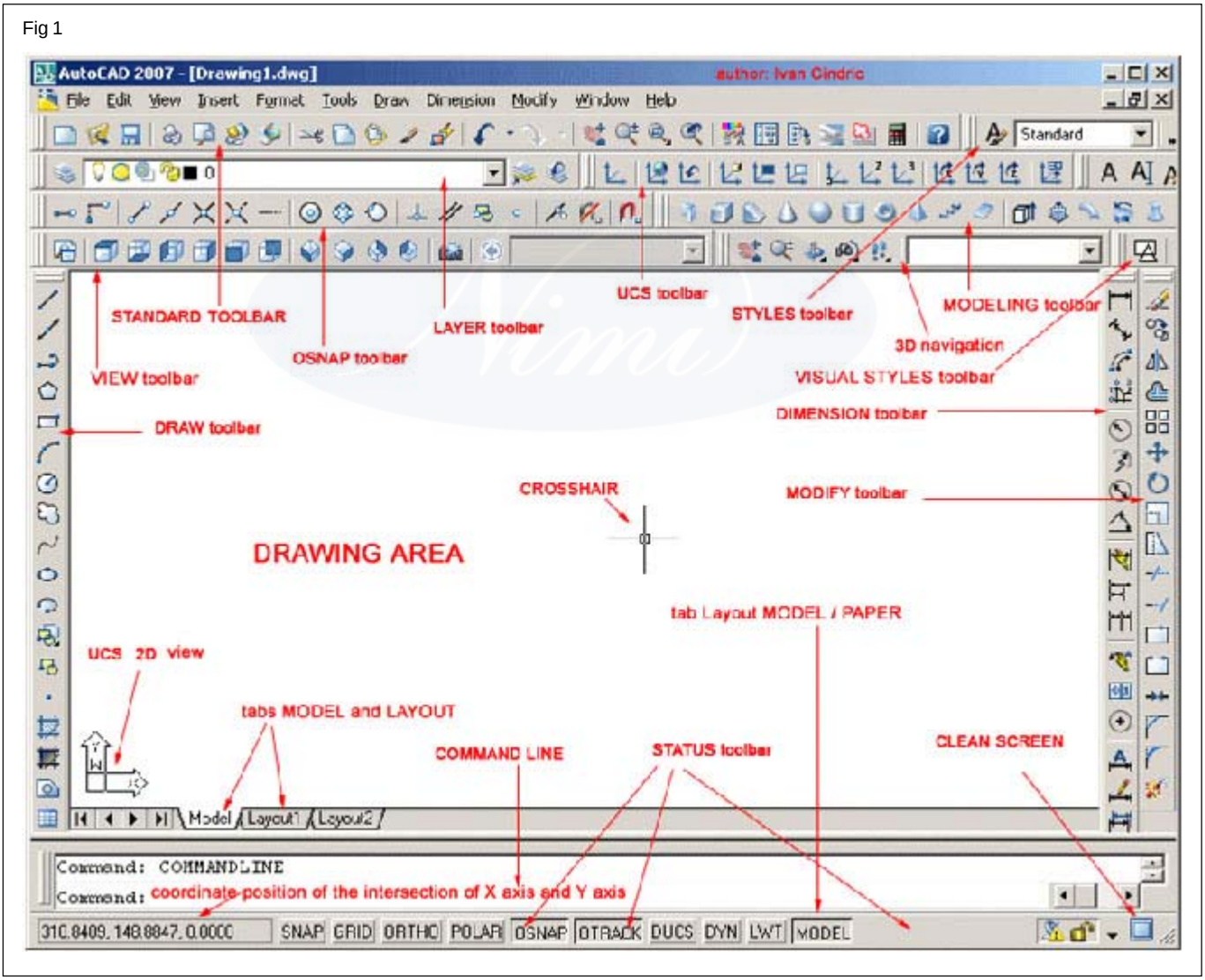
Objectives: At the end of this exercise you shall be able to
- **screen appearance of CAD**
- **work spaces**
- **tool baras in AUTOCAD**
- **loading toolbars**
- **application button**
- **toolbars**
- **setting of units.**

PROCEDURE

Screen appearance

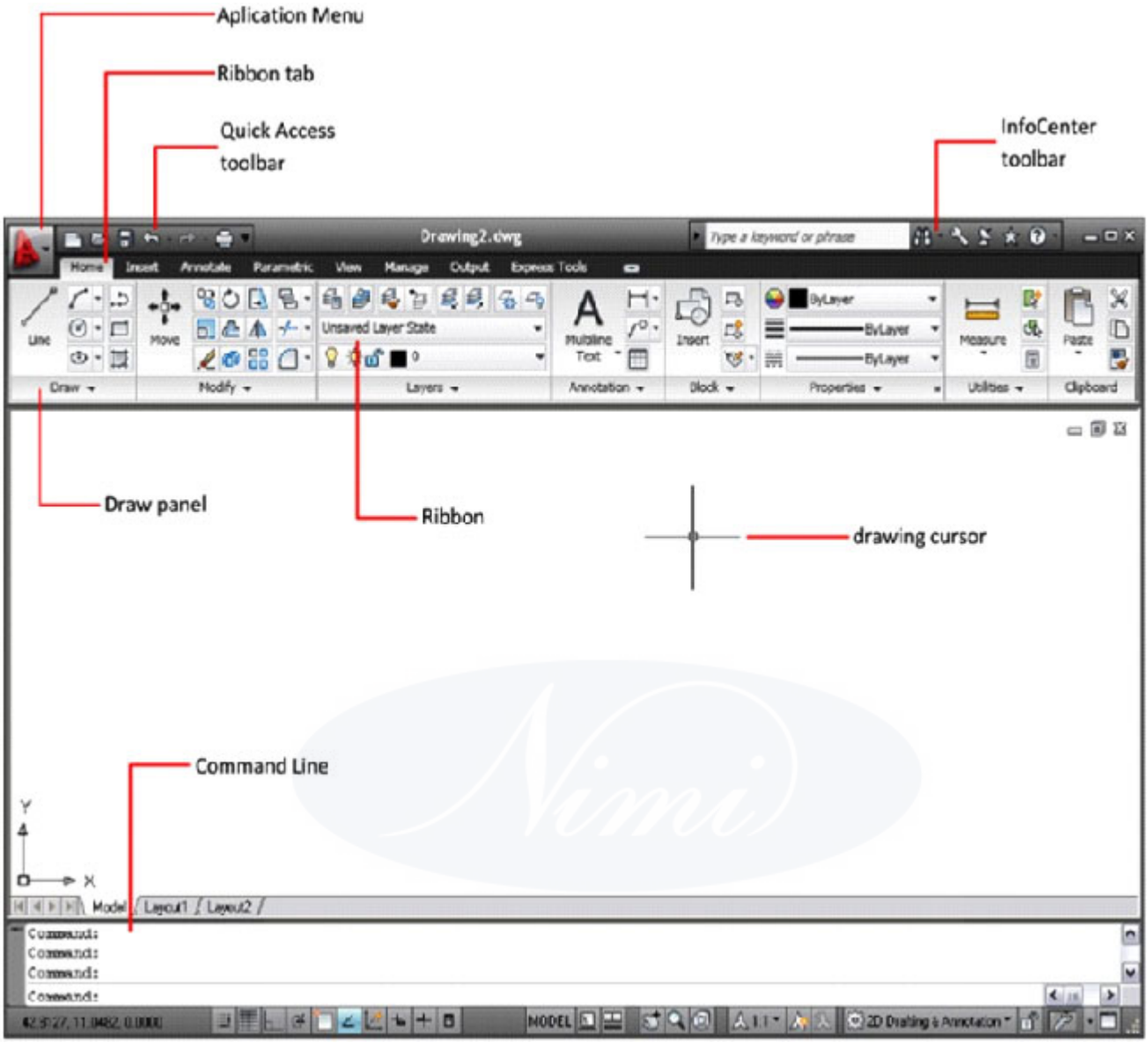
1 2007 AutoCAD screen appearance (Screen is in AutoCAD Classic)

Fig 1



2 2010 AutoCAD appearance (Screen is in 2D and annotation)

Fig 2



Point to be noted

1 Do not get confused with the screen appearance.

2 Screen appearance change according to the work space selection.

Workspaces

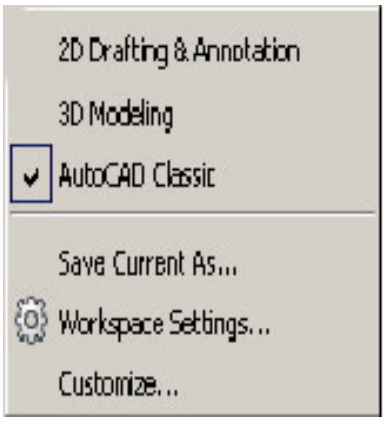
You can switch between the workspaces from the menu browser.

1 Click the workspace switching icon in the lower left corner of the screen. (Fig 3)

Fig 3

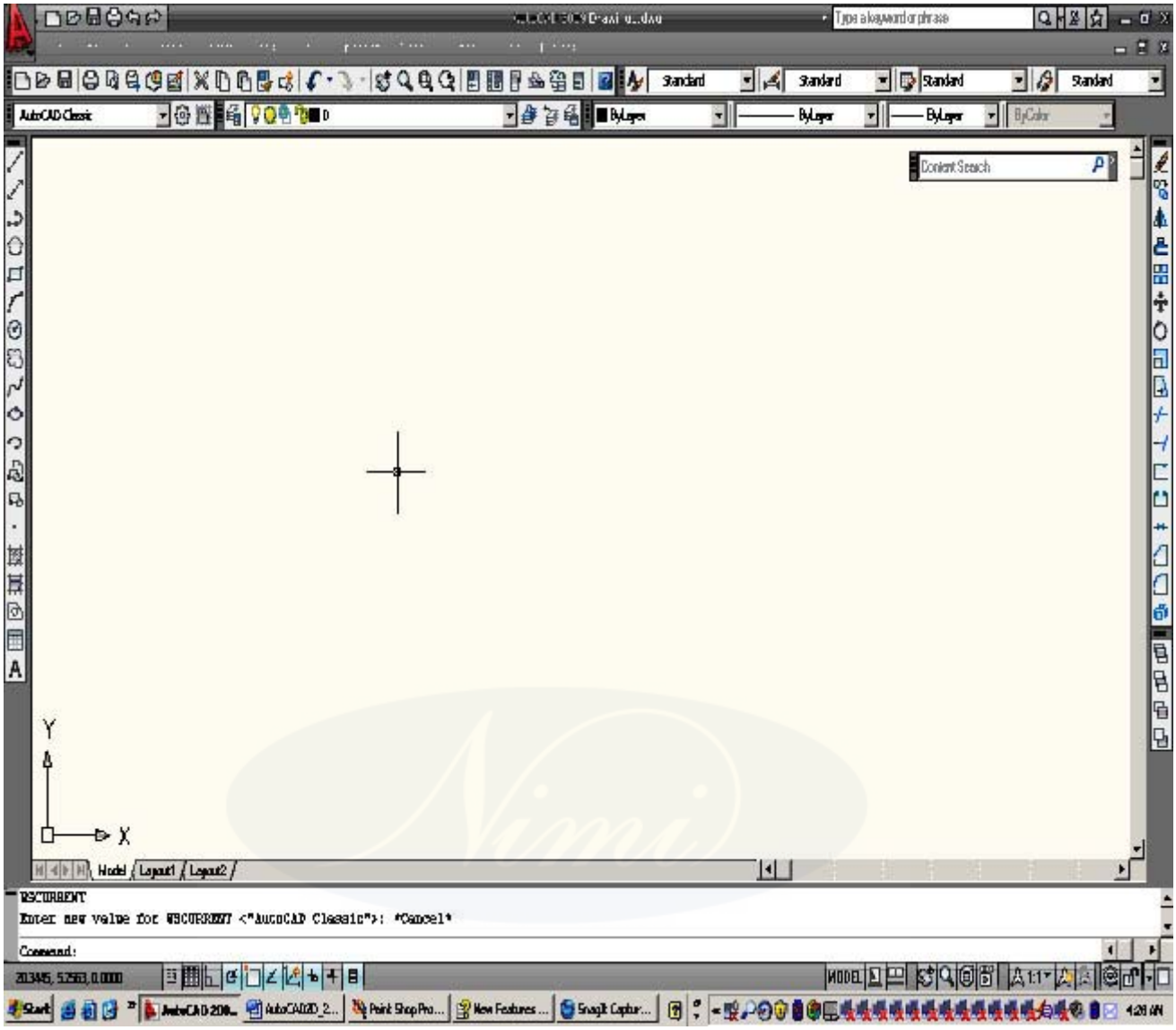
2 Click on one of the following workspace options. (Fig 4)

Fig 4



AutoCAD classic workspace (Fig 5)

Fig 5



Toolbars in AutoCAD

Activating Toolbars:

Toolbars activate by clicking DTM (right-click) in the territory where the Text menu, just below the zone where the other toolbars, or via a text menu.

To set the Toolbar click on View => Toolbars on the text menu.

The most widely used toolbars (toolbars) markup are bold.

CAD Standard

Camera Adjustment

Dimension

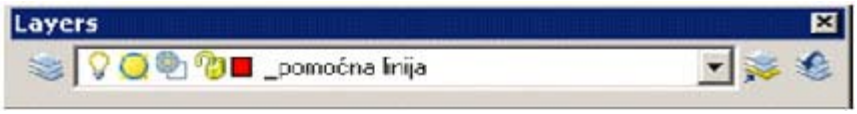
Draw

Order

Inquiry

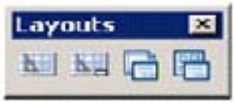
Insert

Layers



Layers II

Layouts

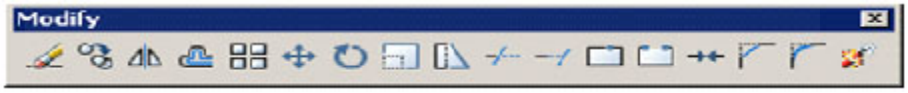


Lights

Mapping

Modeling

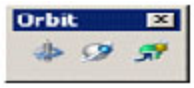
Modify



Modify II

Object snap

Orbit



Properties

Refedit

Reference

Render

Solid Editing

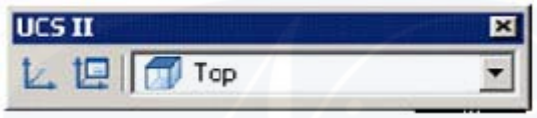
Standard

Styles

Text

UCS

UCS II



View

View ports

Visual Styles

Walk and Fly

Web

Workspaces

Zoom

Loading Toolbars

Right - clicking on an icon in any toolbar

This will show a list of all available toolbars.

Note: To bring in tool bars to the screen. When no toolbar appears on your screen do not panic, just go to "tools option" which you can see on the top at your screen (maintool bar). In that click "toolbar" now you can see an image like the image shown in the figure, now select whatever tool bars you need.

(or)

Just type in 'tool' in the command bar and follow the instruction given in it. (Fig 6)

Fig 6

Help Tooltips

1 Move the mouse to the toolbar but do not pick the button. (Fig 7)

The workspace in AutoCAD is divided into 2 distinct areas. The drawing area covers most of the screen and tool bars are anchored above and below the drawing area. They include:

Fig 7

Application button (or) menu bar (or) popup menu bar:

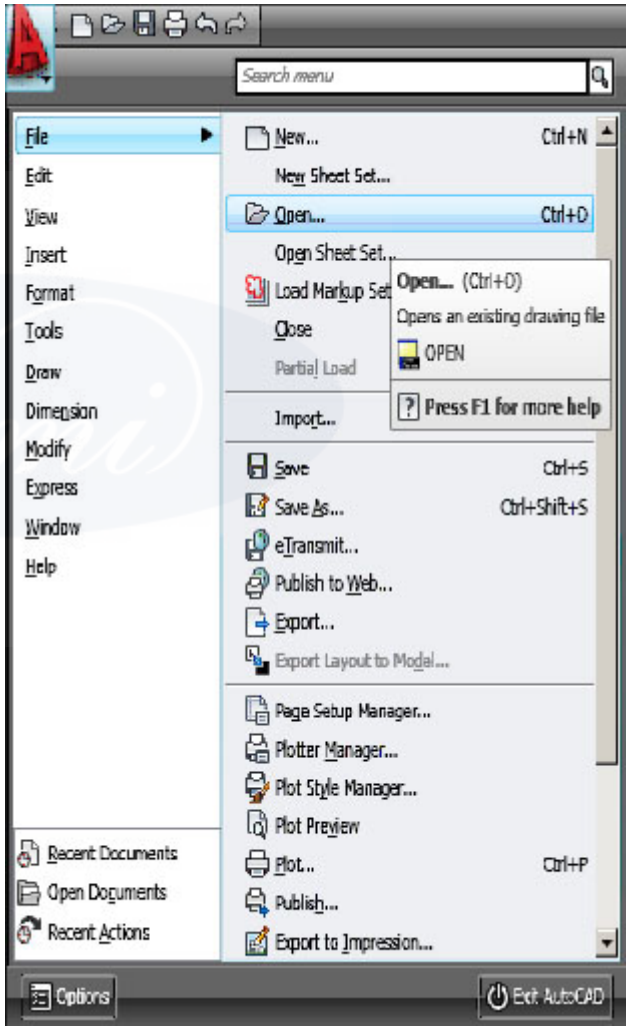
The large, red A at the top, left-hand corner of the screen is the application button. Use it to print files, to exit the program etc.

1 Click on the A icon.

2 Click the desired pulldown menu.

3 Click on the command to be executed from the pulldown. (Fig 8)

Fig 8



2 Quick Access Toolbar

i) Click on one of the following icons for quick access to commands QNEW, OPEN, SAVE, PLOT, and UNDO/REDO. (Fig 9)

Right-click the Quick Access toolbar and click Customize Quick Access Toolbar. The Customize User Interface dialog opens and displays the list of commands available.

Fig 9

Drag commands you want to add from the command list pane in the Customize User Interface dialog box to the Quick Access toolbar.

Info Center

Quickly search for a variety of information sources, access product updates and announcements, and save topics with InfoCenter. (Fig 10)

Fig 10

3 Ribbon

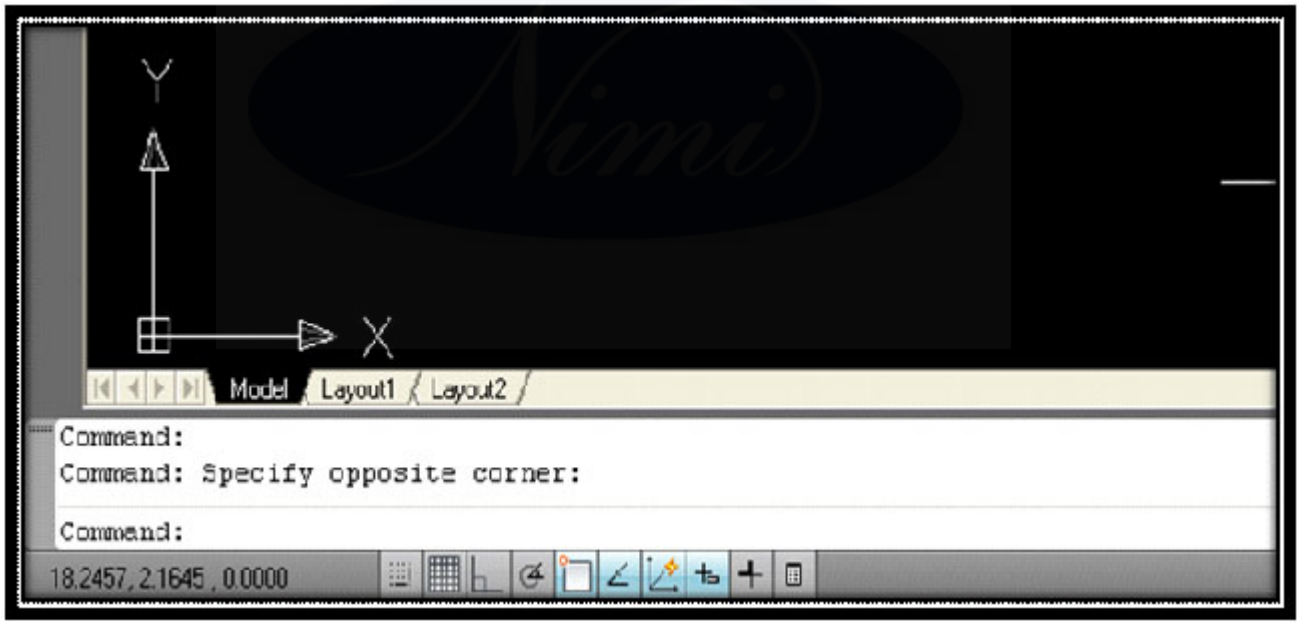
The ribbon provides a single, compact placement for operations that are relevant to the current workspace. It eliminates the need to display multiple toolbars, reducing clutter in the application window. The ribbon maximizes the area available for work using a single compact interface. The ribbon can be displayed horizontally, vertically, or as a floating palette. The horizontal ribbon is displayed at the top of the drawing window by default when you create or open a drawing. You can create your own panels to display on the ribbon; you can also modify the commands and controls on existing ribbon panels.(Fig 11)

4 Status bar

The toolbar across the bottom of the screen is the status bar. It mostly consists of control settings that allow you to monitor changes in the drawing. (Fig 12)

Fig 11

Fig 12



5 Command/prompt tool bar

Directly above the status bar is the command /prompt toolbar. This is where the software communication with you. (Fig 13)

6 Create a new drawing

From the quick-access toolbar, click "File," and then "New." The new drawing will use 1 of 2 default settings, either imperial or metric. If you want to use specific settings for a drawing, select "Templates" under the Options dialog box. (Fig 14)

Fig 13

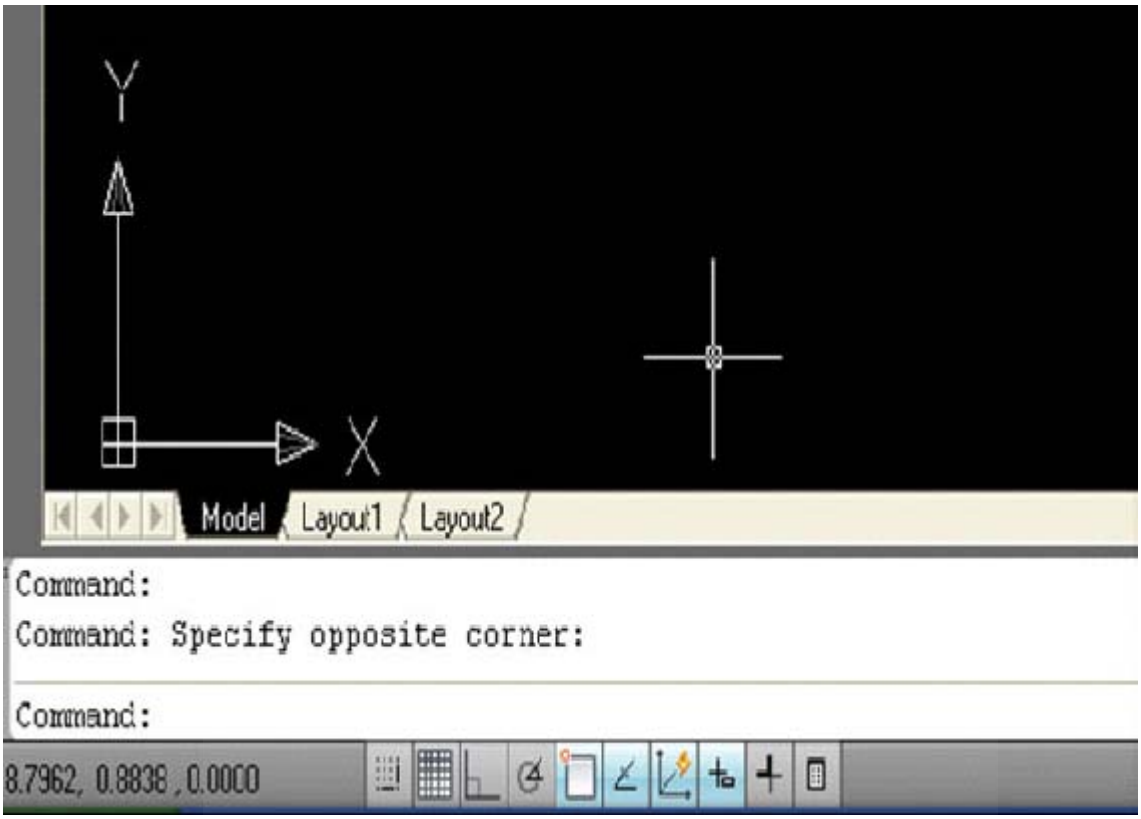
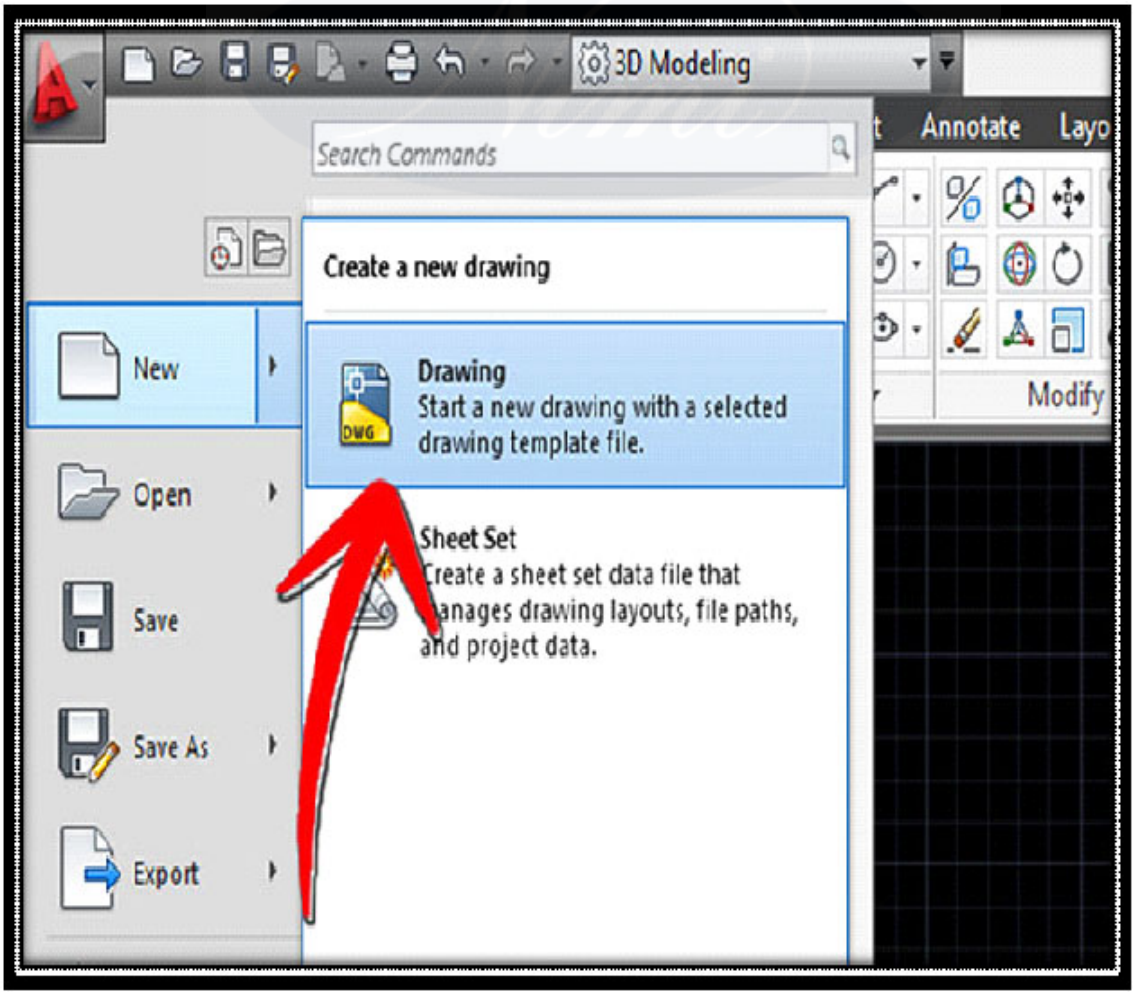

Y
X
Model
Layout1
Layout2
Command:
Command: Specify opposite corner:
Command:
8.7962, 0.8838 , 0.0000

Fig 14


3D Modeling
Search Commands
Annotate
Layo
Create a new drawing
New
Drawing
Start a new drawing with a selected drawing template file.
Open
Sheet Set
Create a sheet set data file that manages drawing layouts, file paths, and project data.
Save
Save As
Export
Modify

7 Locate the small gear icon at the bottom of the screen.

This is the Workspace icon. Click it and select "AutoCAD Classic". AutoCAD classic is user friendly. (Fig 15)

8 Plotting

When you have done the assignment, the next step is to print (or plot) it out. To do this, bring up the plot dialog box using any method explained above (plot <enter> will work). Set it up to print as shown below. Follow these steps for a successful plot (see diagram below) (Fig 16)

Fig 15

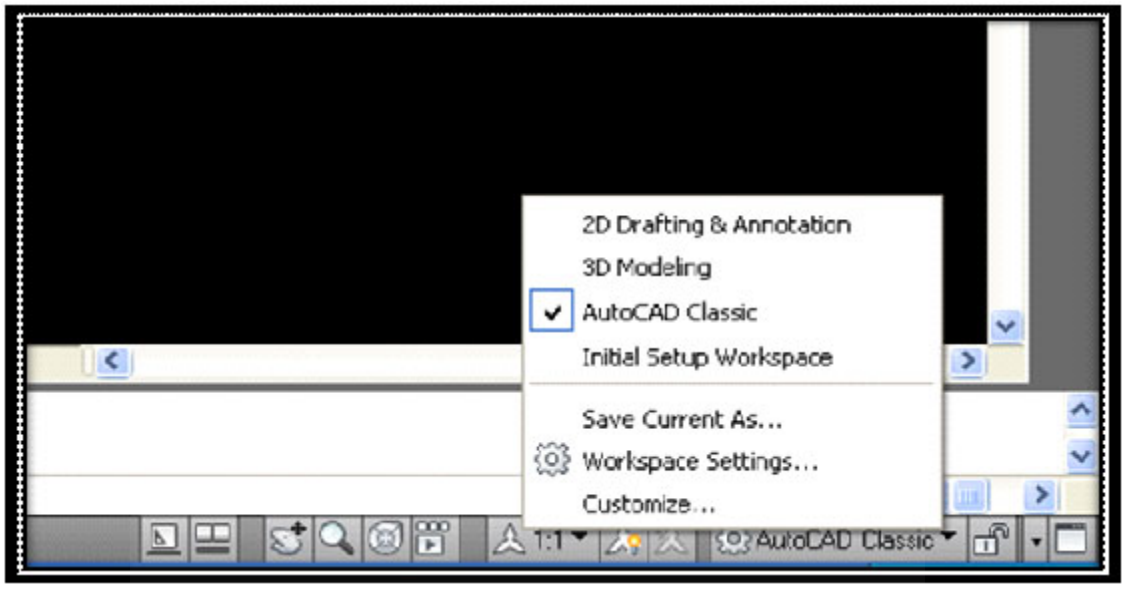
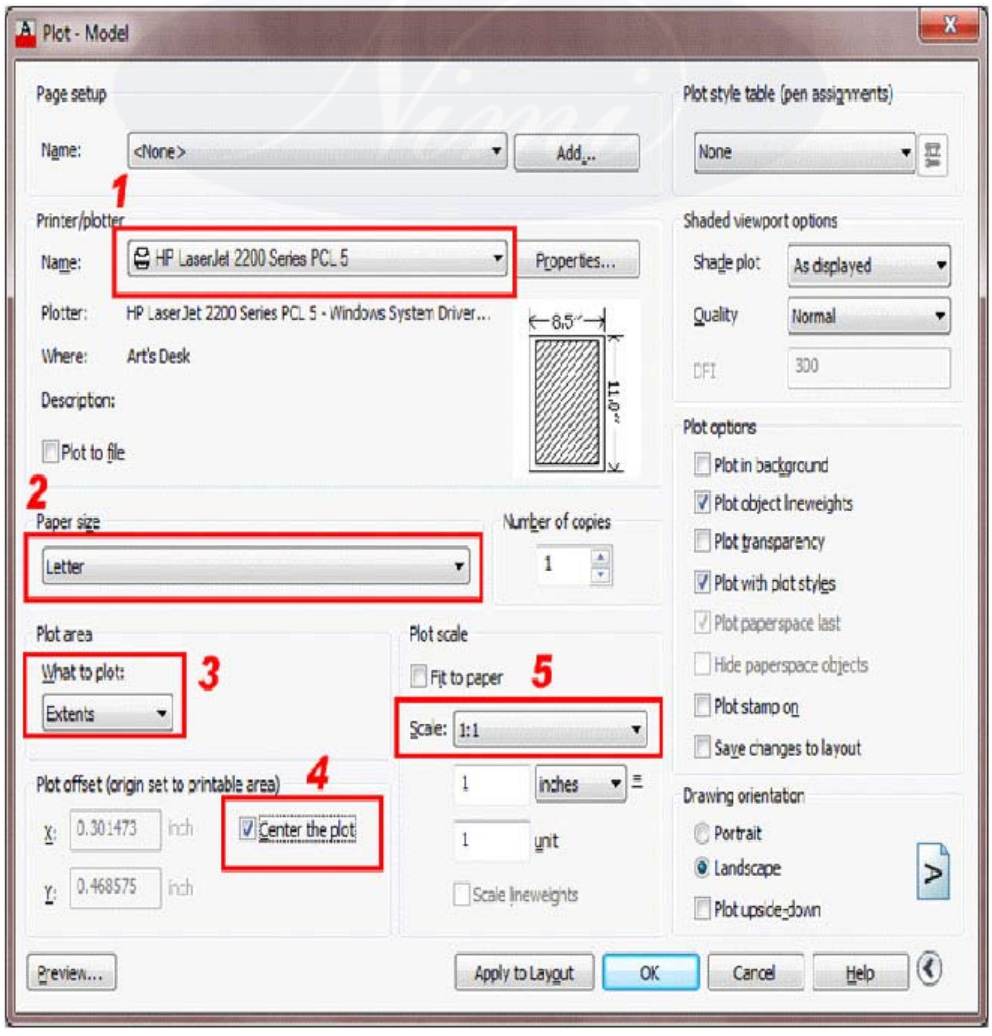


Fig 16



1 Select your printer - laser or inkjet will work fine.

2 Select the paper size - "Letter" (8-1/2" x 11") is needed in this case.

3 For the "Plot Area", select "Extents" - that will plot everything you drew.

4 Select the checkbox to "Center the Plot" on your sheet of paper (looks better).

5 If "Fit to Paper" is selected by default, uncheck it and select a scale of 1 inch to 1 paper unit (1:1). This will make your printout 'life-sized'.

6 Now Preview your drawing. I strongly recommend that you preview EVERY drawing you will ever draw in AutoCAD - a lot can go wrong, so you don't want to waste paper (especially when you're using expensive 3' x 4' sheets!). If your preview looks good, Cancel out of the preview by pressing ESC.

7 If you're sure that everything's ok (this is where good habits begin), press the OK button.

Note : You may have to change the paper size in your printer (Use the Windows printer settings to do this.) You may also have to change the rotation or origin of the plot. Check the Landscape radio button in the Drawing Orientation section.

If everything worked out, you should be able to measure your drawing and have it exactly the way you drew it (a couple of 2" squares, an angled 1" square and a 1-1/2"circle).

Save your drawing as you would any other Windows file. CTRL+S will bring up the Save or Save as dialog box.

Status tool bar mod's work

As part of the window AutoCAD is its status toolbar that contains buttons for ON/OFF function groups during the drawing. When the button is pressed then the active group of concerned functions to their preferences. The status bar contains buttons for the function: SNAP, GRID, ORTHO, POLAR, OSNAP, OTRACK, DUCS, DYN, LWT, MODEL.

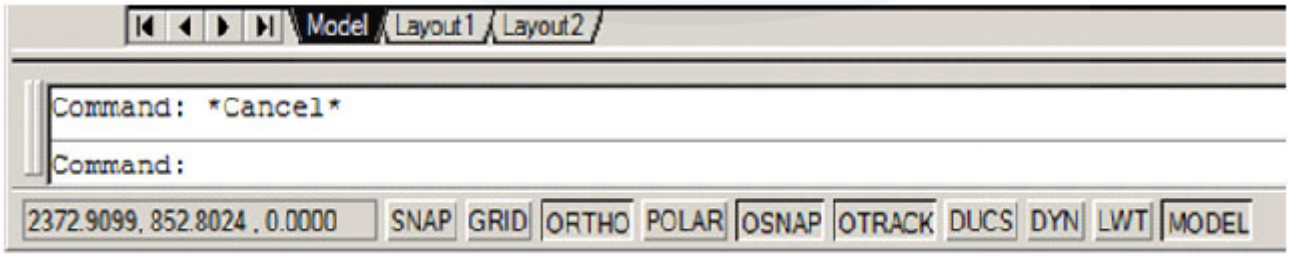
I usually start with the following function buttons including: Ortho, Osnap, Otrack and Model.

During the work started drawing a line you can freely activate and deactivate some of the buttons in the status bar, without fear that will stop the action begun drawing a line.

For example: (AutoCAD tangent to the circle) when you start to draw a line and clicking on the desktop and then you want to connect to the line tangent to the circle already drawn for which sight is not included, simply click on the button OSNAP right mouse button, select the option for the tangent sight click on OK and continue to target the tangent circles in the drawing.

Note that once you start a drawing and you clicked first click to start, no problem you can switch to another window or a function button in the status bar (Help AutoCAD, IE, OE, Photoshop, etc.) and then back into working AutoCAD window and continue with his drawing. (Fig 17)

Fig 17



Right click on the blank area of the status bar to see the tools to turn off/on. (Fig 18 , Fig 19 & Fig 20)

1 **ORTHO :** Orthogonal drawing or rectangular drawing, it means that when you turn this feature lines that you drag the mouse will be angled in relation to the UCS and you can draw only the angle that is set and it is 90 degrees. (Fig 21)

2 **POLAR :** Polar coordinates follow when you draw the line, if this function is activated then the Ortho automatically turned off. because when you draw with the POLAR function of the mouse you can draw lines at an angle to the set and you can also set up and draw it with absolute or relative coordinates when drawing lines. (Fig 22)

Fig 18

Fig 19

Fig 20

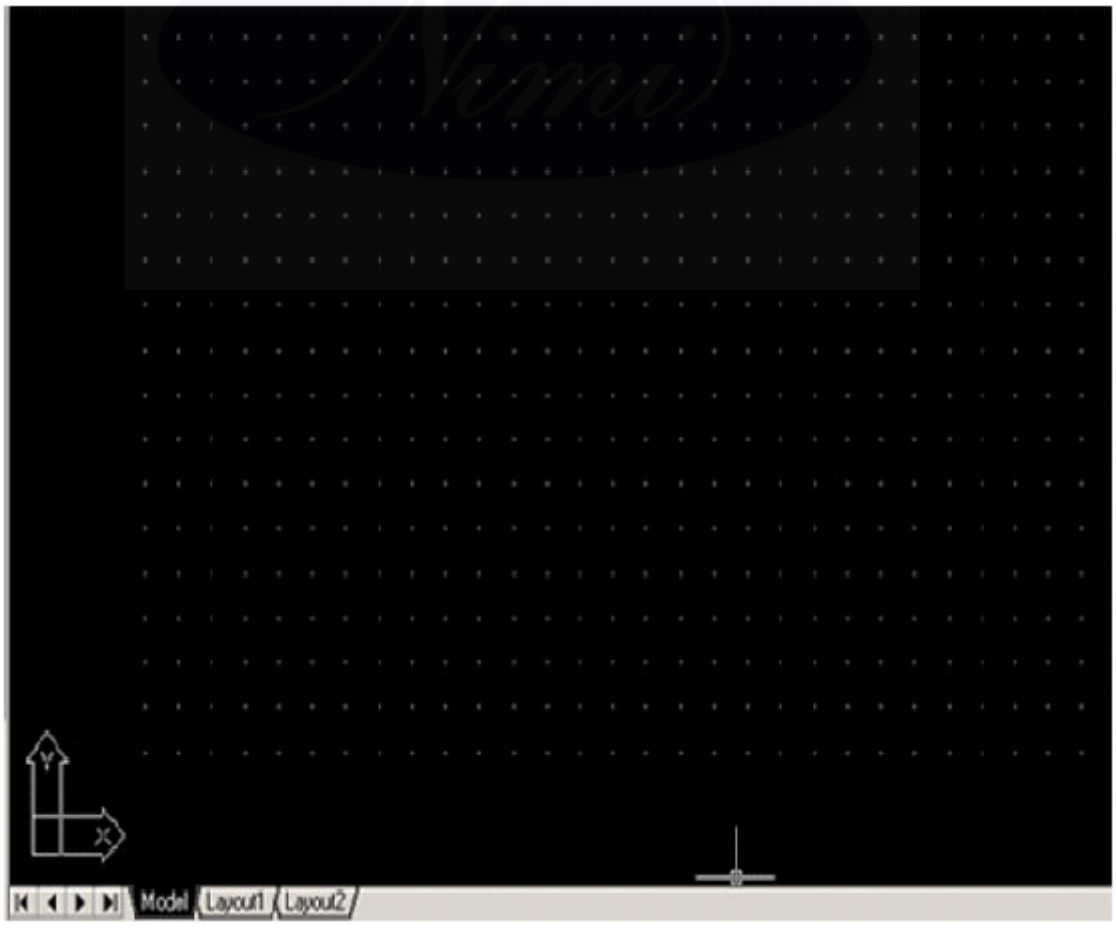


Fig 21

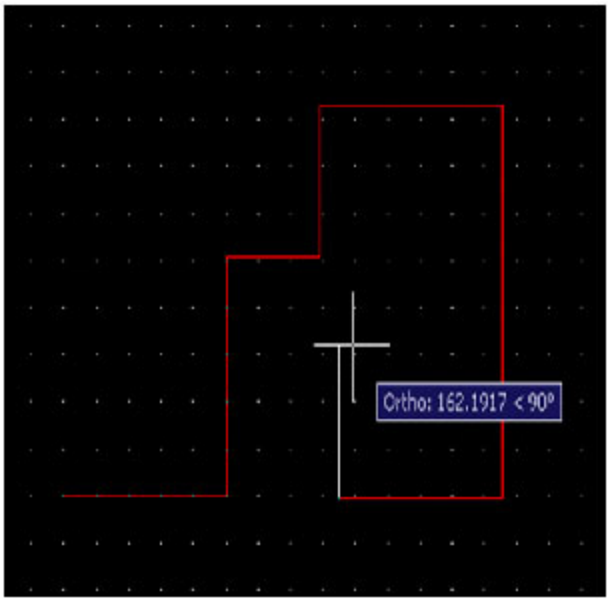
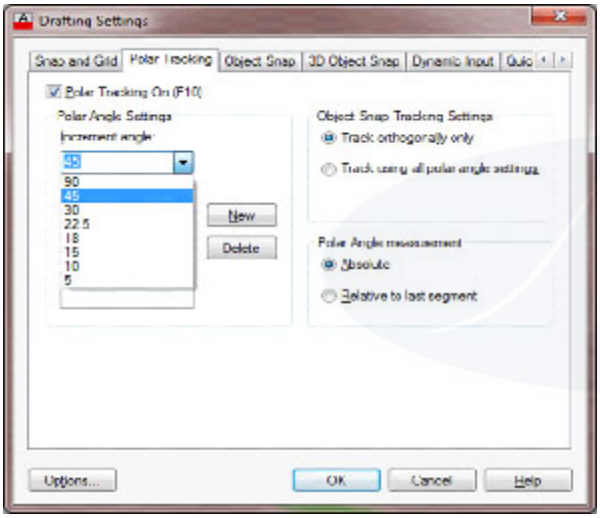


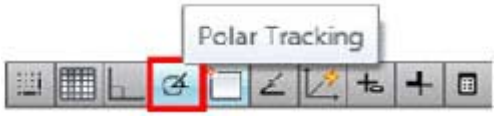
Fig 22



Now wouldn't it be cool to draw angled lines (like the short ones in the above image)? Well you can, but first you have to make change in your settings.

Type in DSETTINGS and you get the Drafting Settings dialog box, go to the "Polar Tracking" tab.

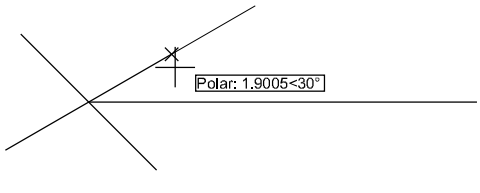
Then make sure that Polar Tracking is On



(press your F10 key to toggle it on and off) and then select the increment angle.  I recommend using polar settings with 45° increments unless you specifically need something else.

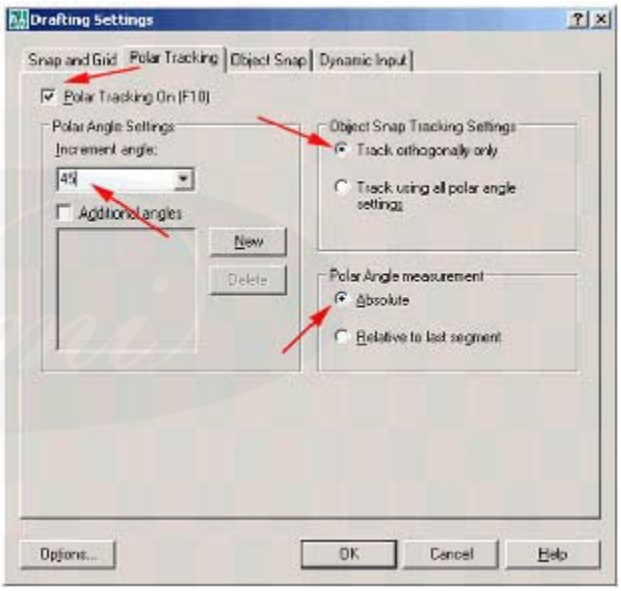
In the example above, I made 4 lines 1 unit long using Direct Distance Entry (DDE). See if you can duplicate this on the left end of the 10 unit line you just drew. The process is the same as you did for the DDE of the previous line. Make sure your increment angle is 30° and draw a line 1 unit long using DDE. (Fig 23)

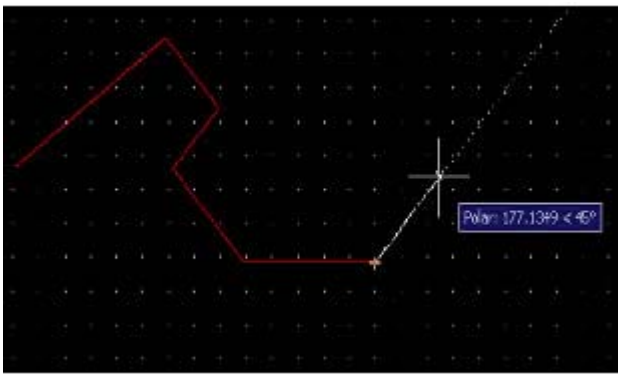
Fig 1



You cannot have Ortho and Polar Tracking on at the same time. As you start to draw more, you will see that these two features are great time savers. There will be times, though, when you have to use absolute and relative coordinate entry (especially in 3D). (Fig 24)

Fig 24





OSNAP Aiming (target) at a particular point of a line derived from activated Osnap which contains some basic positions on the line, circle, image, and the like. Endpoint, midpoint, and other sights for other points of attachment appear only.

When we turn Osnap mode (and turn on each target). All lines in the figure, circles, etc.. have a connecting point for the target, so to distinguish GRIP (grips) of the target.

BTW: Hindsight is like a magnet, when you approach a point of attachment and appears hindsight, whether you're precisely on a given point or not, just click and Autocad will bind to the target presented.

SO pay attention! when you close more tombstones (zoom in enough to see a drawing of a tombstone that bind)

OSNAP mod: F3 => Endpoint, Midpoint, Center, Node, Quadrant, Intersection, Extension.

You may select whichever points you want to 'snap' on an object. Here is a list of your options. Followed by the command entry to invoke the needed Osnap.

Endpoint - snaps to either the beginning or the end of an object such as a line - **END**

Midpoint - snaps to the exact middle of a line or an arc - **MID**

Center - snaps to the center - point of a circle or arc - **CEN**

Node - snaps to 'nodes' (not covered in this course) - **NOD**

Quadrant - snaps to any of the four quadrants of a circle - **QUA**

Intersection - snaps to the point where two object cross - **INT**

Extension - Snaps to the phantom extension of an arc or line -**EXT**

Insertion - snaps to the insertion point of an object (such as a block or text) - **INS**

Perpendicular - will snap so that the result is perpendicular to line selected - **PER**

Tangent - snaps to create a line tangent to a circle or arc - **TAN**

Nearest - will find the closest point an object and snap to that point - **NEA**

Parallel - Snaps parallel to a specified line - **PAR**

M2P - This isn't technically an 'Object Snap' as you are not snapping to specific point on an object, but it allows you to select 2 points and it will calculate the midpoint between those 2 points. This is a very handy option to have. (Fig 25)

6 OTRACK (object snap tracking) - F11 - Intersection, Perpendicular, Tangent, Nearest, Apparent intersection, Parallel. (Fig 26)

Fig 25

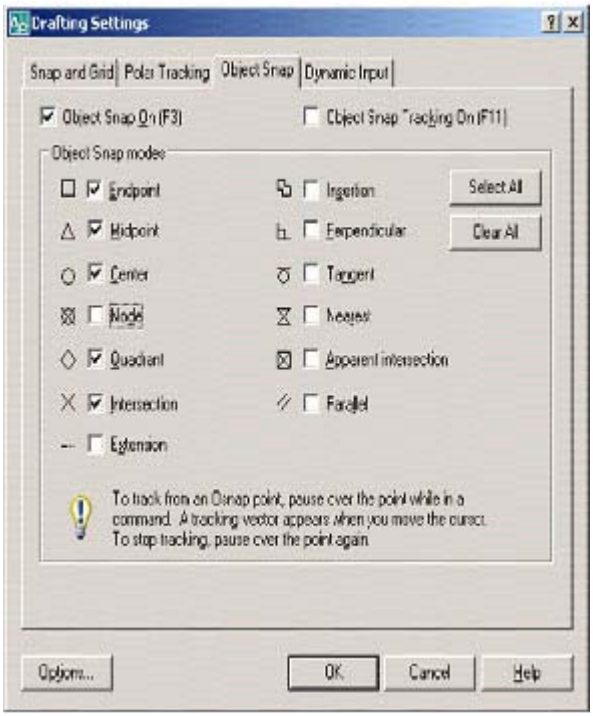
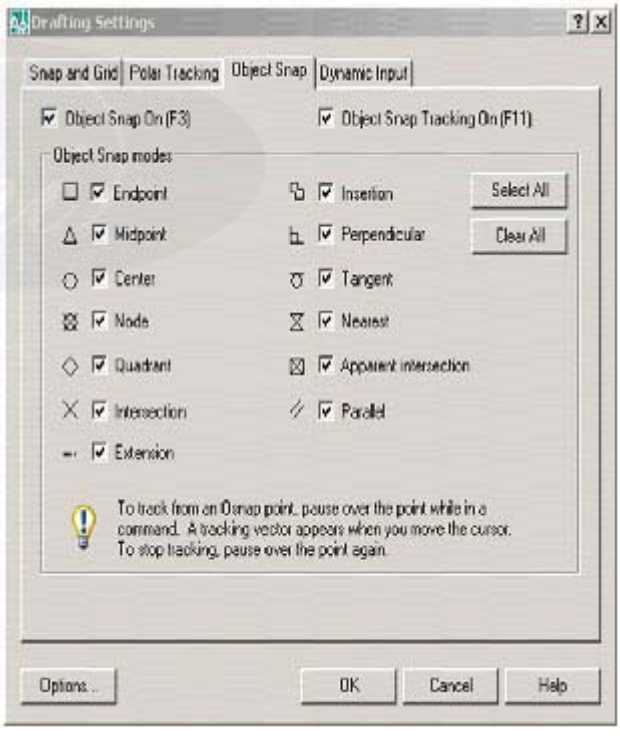


Fig 26



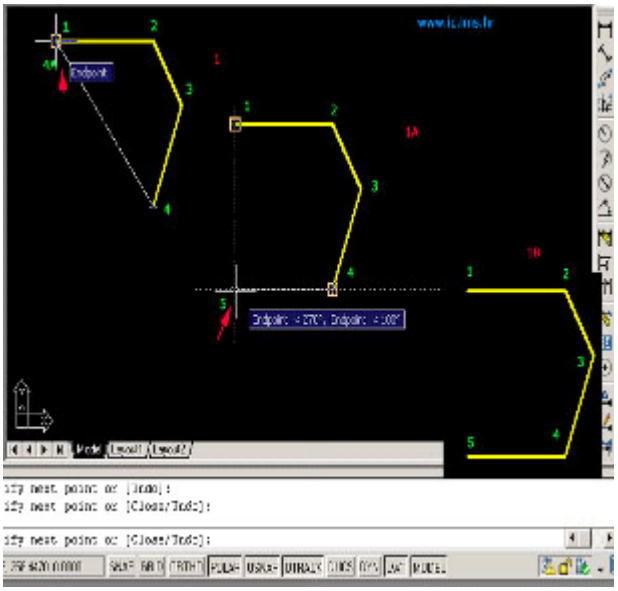
If you turn OTRACK/POLAR mode then you have an active extra dashed line that helps you draw the line to align with a line (point) to the already drawn parts of the drawings.

How does OTRACK mod?

Including button functions to incorporate the ancillary mode OTRACK a line that helps in drawing the mouse and the eventual completion of a placement - line, ie. its end point. For instance: when you draw more lines and one line you want to end up in the level of existing lines hover your

mouse on the desired line (it will be dashed shed that will create a temporary intersected lines and their intersection) and then move it to where you want to complete the intended line. See the picture below. (Fig 27)

Fig 27

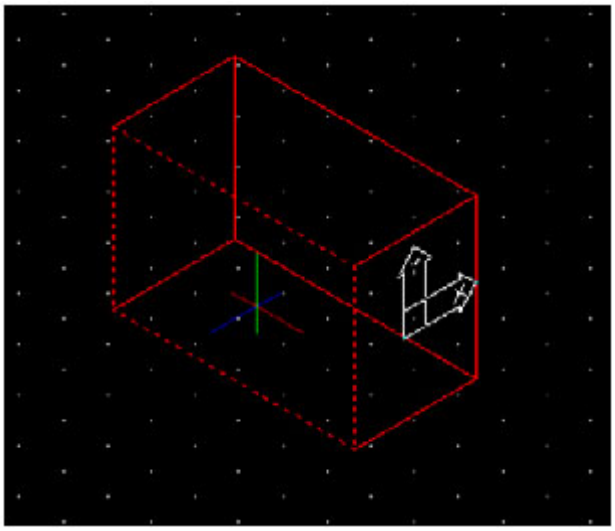


So start drawing with the mouse and at some point (item 4) when we want to end up in the vertical scale (for drawing items 1) move the cursor to the endpoint of point 1 and then shifting it down (then appearing auxiliary dashed lines) following the vertical dashed line while we mark the intersection on her second set in a vertical line and click the mouse.

Now we are confident that our 5 point within the vertical line in the direction of the line item

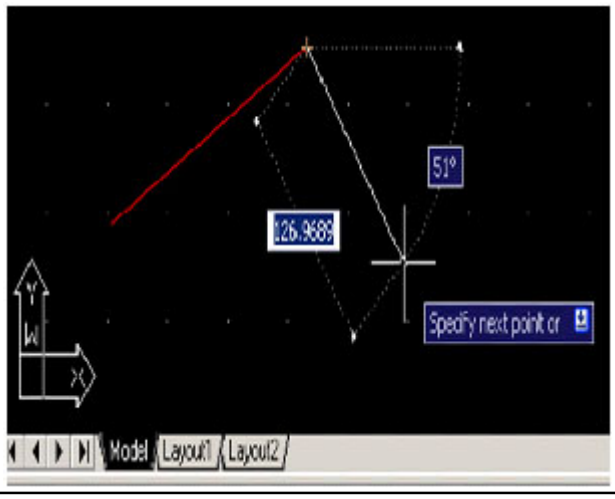
7 **DUCS** : Turning Dynamic UCS and the adhesive on a plane or surface of the body, while the included DUCS and clicking on the icon on the toolbar Origin UCS and then to position the cursor on any surface of the body, it automatically means the surface (hatched) and when clicking on it UCS for ecological surface and positioning. (Fig 28)

Fig 28



8 **DYN** : Turning Dynamic Input for information when drawing (F12) (Fig 29)

Fig 29



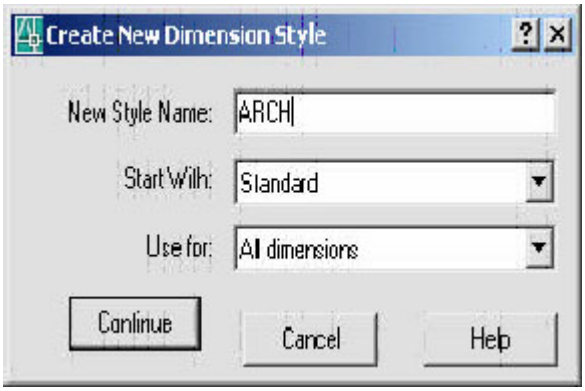
9 **LWT** : Turning Lineweight, viewed thick lines. When this button is included then we have shown with their line thicknesses, if you turn off all the lines the same thickness.

Dimension Style

5 After setting the units set the dimension with the instruction given below.

a Creating Dimension Styles

1	Choose	Format, Dimension Style...
		or
2	Choose	Dimension, Style.
		or
3	Choose	Dimension Style icon from the Dimension Style toolbar.
4	Type	DDIM at the command prompt
		Command: DDIM
5	Choose	New... from the dialog box.
6	Create	a new style from the existing styles.



7 Click the Continue button.

TIP

All dimension variables except for DIMSHO and DIMASO can be saved as a style.

UNITS Command

1 Choose Format, Units...

or

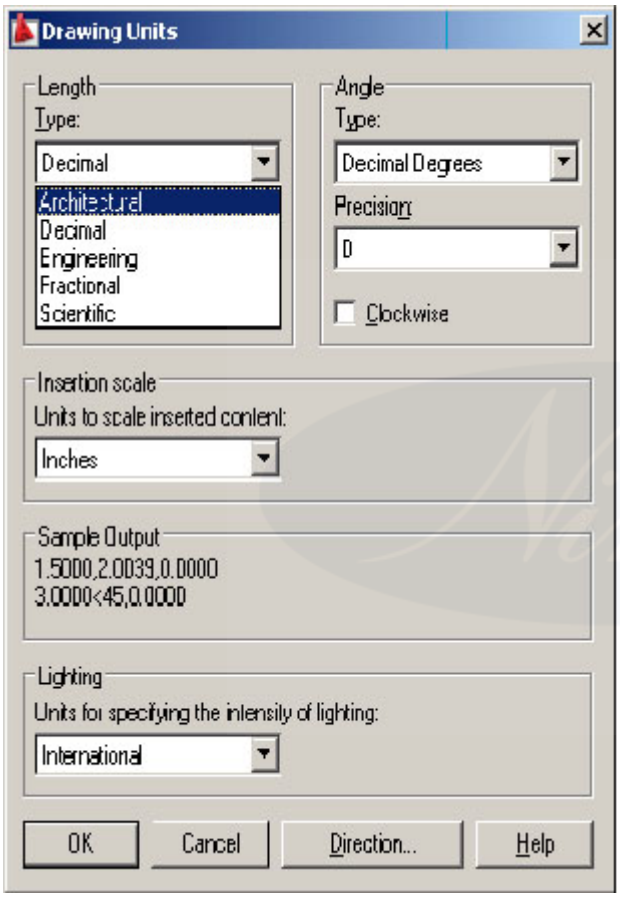
2 Type DDUNITS at the command prompt.

Command: DDUNITS or UN

3 Choose a units and angle setting.

4 Choose a precision setting. (Fig 30)

Fig 30



b Lines and Arrows

Edits Dimension Lines, Extension Lines, and Arrows.

1 Pick the Lines and Arrows tab from the Dimension Variables and Styles dialog box. (Fig 31 & Fig 32)

c Text

Edits Text Appearance, Text Placement and Text Alignment.

1 Pick the Text tab from the Dimension Variables and Styles dialog box. (Fig 33)

Fig 31

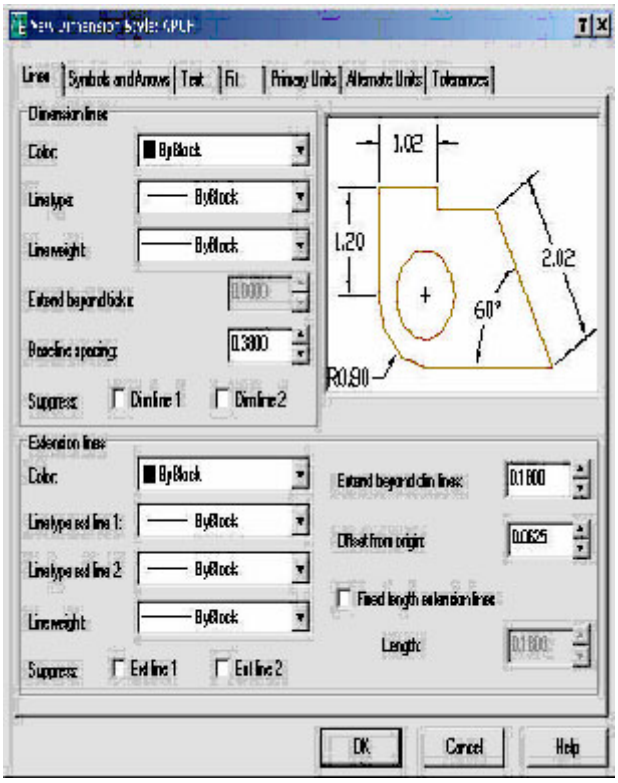


Fig 32

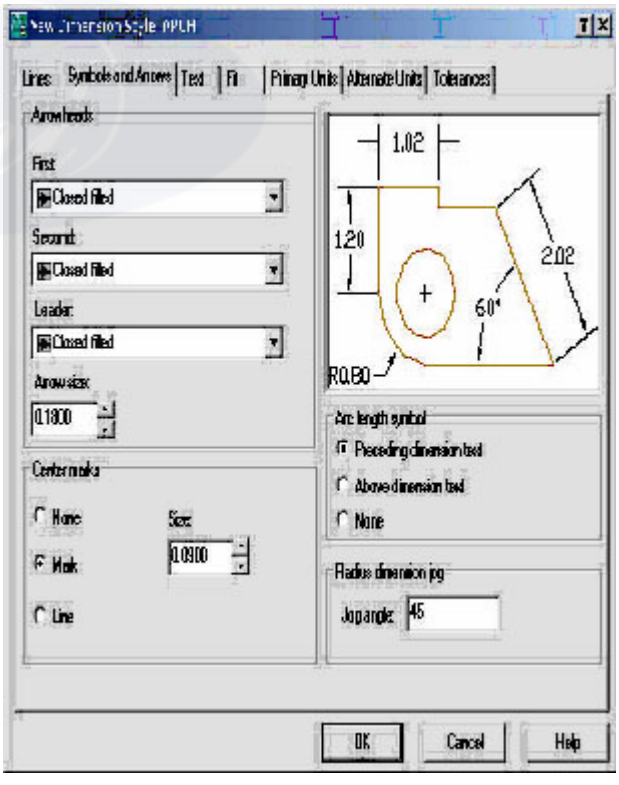
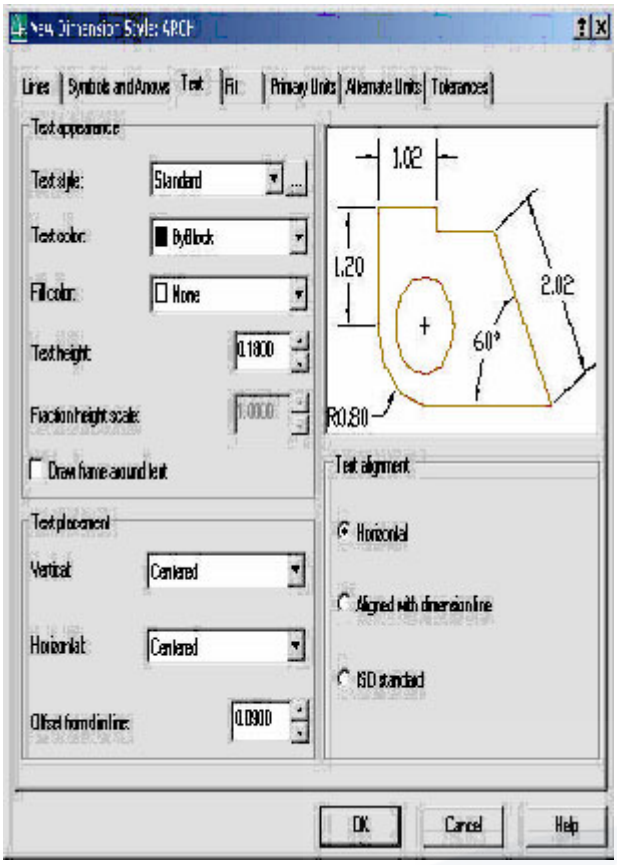


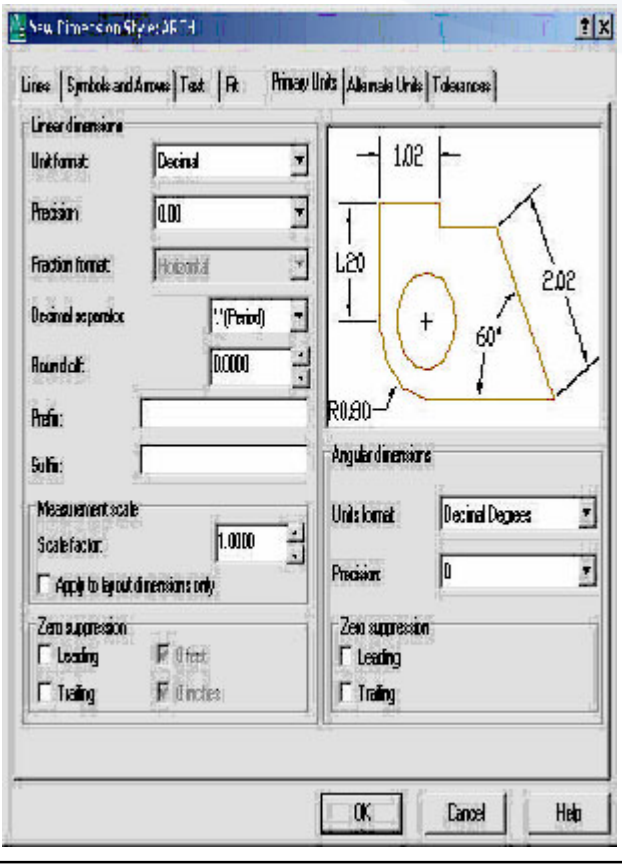
Fig 33



d Primary Units

Edits Unit options for dimension's primary units.

Fig 34



1 Pick the PRIMARY UNIT tab from the dimension Variables and Styles dialog box. (Fig 34)

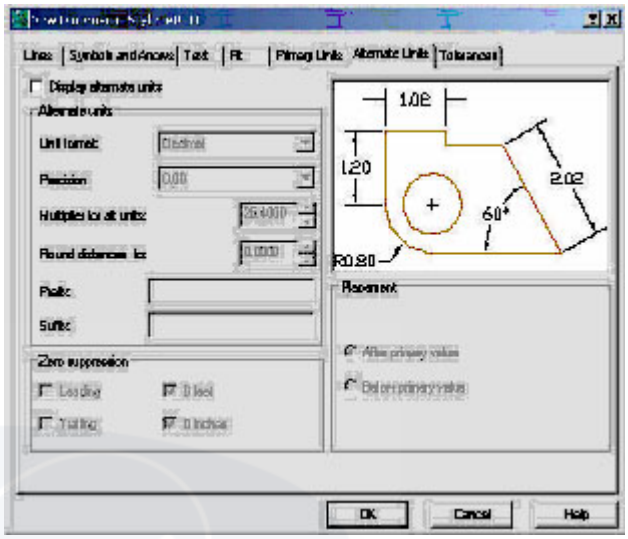
2 Whatever unit format and precision you have entered for units same thing you have to enter in primary units.

e Alternate Units

Edits Unit options for dimension's alternate units.

1 Pick the ALTERNATE UNIT tab from the Dimension Variables and Styles dialog box. (Fig 35)

Fig 35

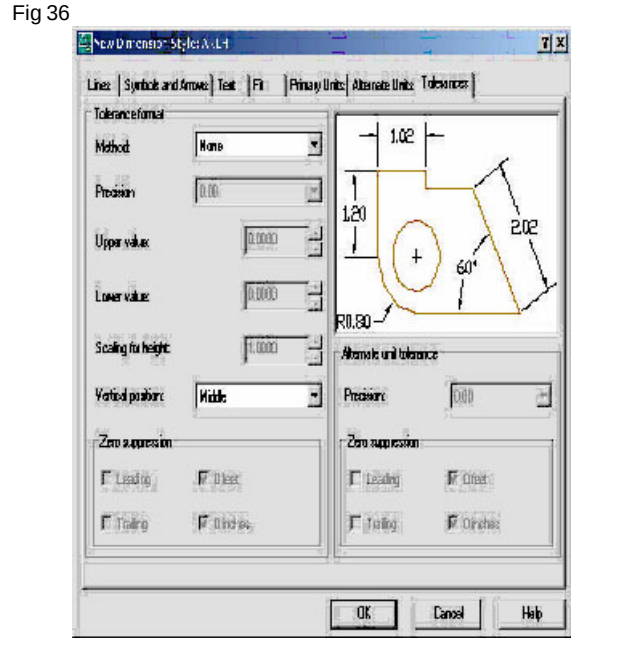


f Tolerance

Edits Unit options for tolerances.

1 Pick the TOLERANCES tab from the Dimension Variables and Styles dialog box. (Fig 36)

Fig 36



Basic CAD drafting commands - 1

Objectives: At the end of this exercise you shall be able to
- **understand the basic drafting commands**
- **line**
- **circle**
- **arc**
- **ellipse**
- **copy**
- **move**
- **rotate**
- **erase**
- **undo**
- **mirror**
- **offset.**

PROCEDURE

TASK 1 : **Line**

Command alias	Button	Classic menu	Ribbon/Application menu
L		Draw => Line	Home => Draw => Line

With LINE, you can create a series of continuous line segments. Each segment is a line object that can be edited separately.

The following prompts are displayed.

Specify first point: Specify a point or press Enter to continue from the last drawn line or arc

Specify next point or [Close/Undo]:

Continue

Continue a line from the endpoint of the most recently drawn line.

Close

Ends the last line segment at the beginning of the first line segment, which forms a closed loop of line segments. You can use Close after you have drawn a series of two or more segments.

Undo

Erases the most recent segment of a line sequence. Using this option more than once backtracks through line segments in the order you created them.

To start a new line at the endpoint of the last line drawn, start the LINE command again and press Enter at the specify start point prompt.

— — — — — — — —

TASK 2 : **Circle**

Command alias	Button	Classic menu	Ribbon/Application menu
C		Draw => Circle	Home => Circle

Drawing a circle in Auto CAD is rather simple. The default method for drawing a circle is to specify a center point and a radius. You can also choose to create a circle based on a diameter, circumference, or area.

The following options are displayed;

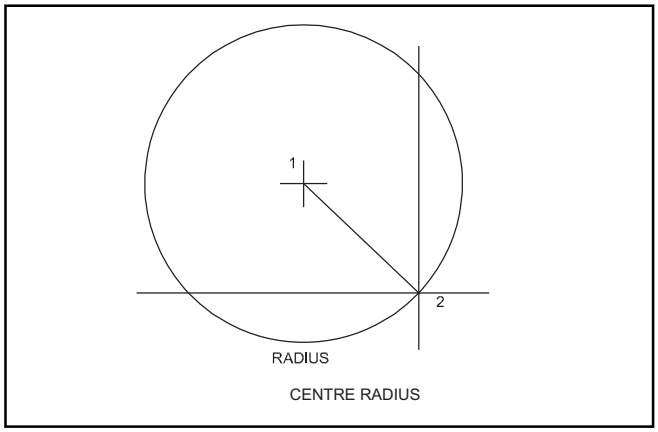
Specify center point for circle or [3p/2p/Ttr (tan tan radius)]: Specify a point or enter an option

Center Point

Draws a circle based on a center point and a diameter or a radius.

Radius

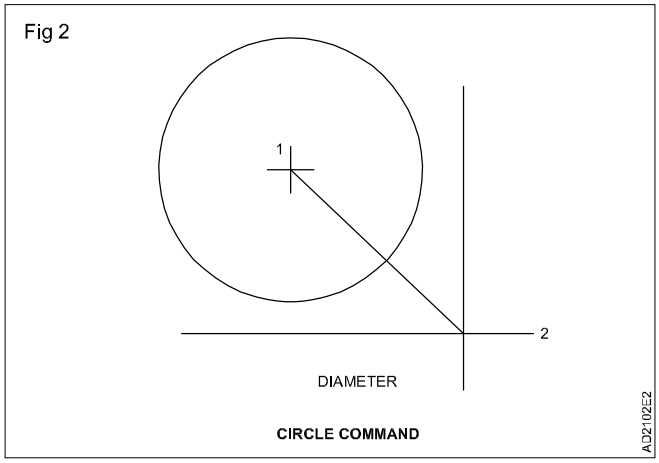
Defines the radius of the circle. For this, you are required to enter a value, or specify a point. (Fig 1)



Diameter

Defines the diameter of the circle. Enter a value, or specify a second point. (Fig 2)

Fig 2



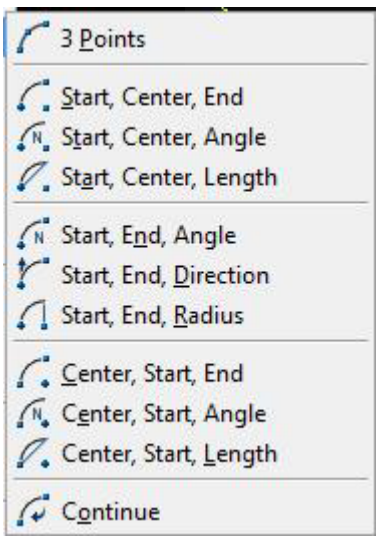
TASK 3 : **Arc**

Command alias	Button	Classic menu	Ribbon/Application menu
A		Draw => Arc	Home => Arc

To create an arc, you can specify combinations of center, endpoint, start point, radius, angle, chord length, and direction values. Except the first method, arcs are drawn counter clockwise from the start point to the endpoints.

The following options are displayed: (Fig 3)

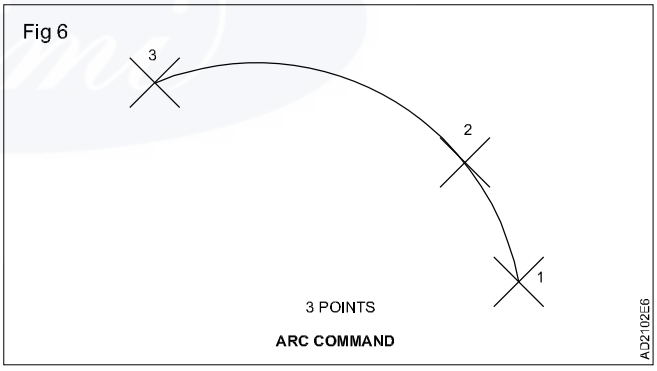
Fig 3



3 Points

This method helps to create an arc by specifying three points. (Fig 4)

Fig 6



Start, Center, End

This method helps to create an arc using a start point, center, and a third point that determines the endpoint. The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the start point first or the center point first.

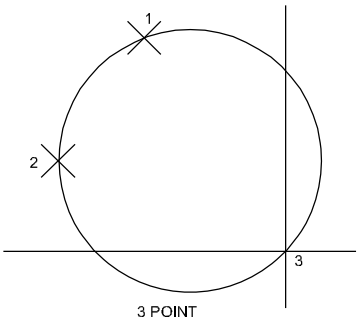
Start, Center, Angle

This method helps to create an arc using a start point, center, and an included angle. This distance between the sgtart point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the start point first or the center point first

3P (Three Points)

Draws a circle based on three points on the circumference as shown in the Fig 5.

Fig 3



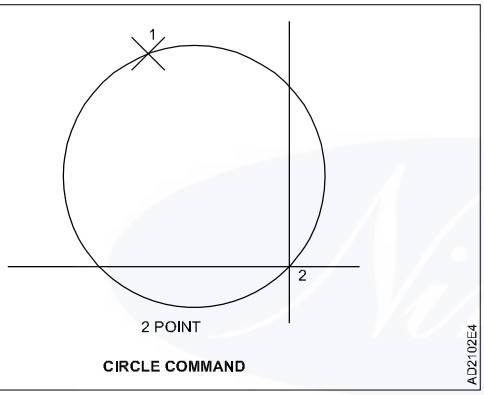
Tan, Tan, Tan

Creates a circle tangent to three objects.

2P (Two Points)

Draws a circle based on two endpoints of the diameter. As shown in the Fig 6.

Fig 4

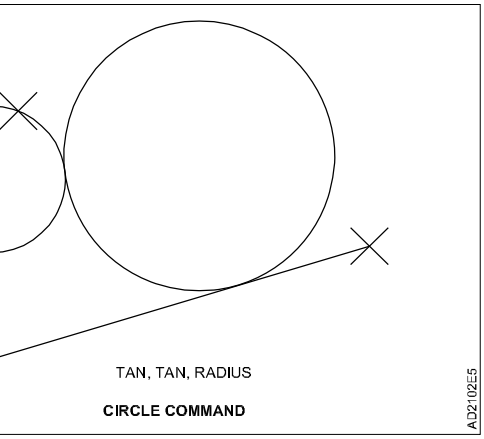


TTR (Tangent, Tangent, Radius)

Draws a circle with a specified radius tangent to two objects. As shown in Fig 7.

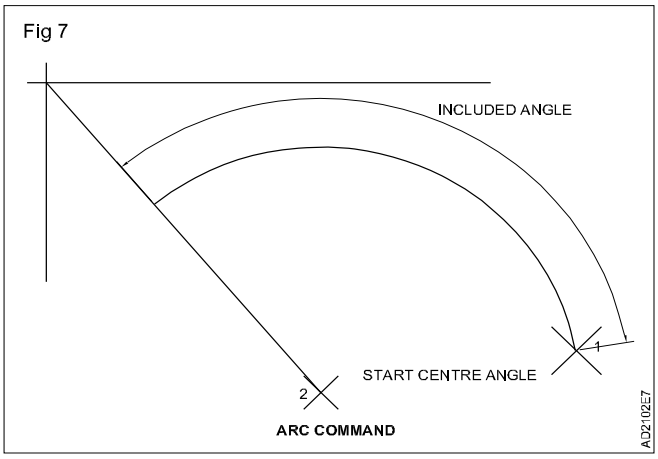
Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.

Fig 5



The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point. (Fig 8)

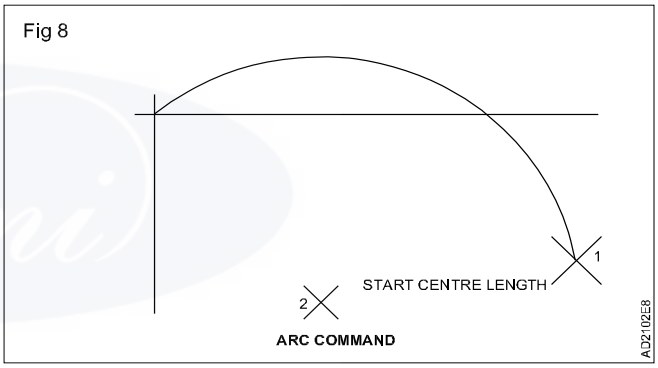
Fig 7



Start, Center, Length

This method create an arc using a start point, center, and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord between the start point and the endpoint of the arc. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the start point first or the center point first. The length of the chord in the arc determines the included angle. (Fig 9)

Fig 8



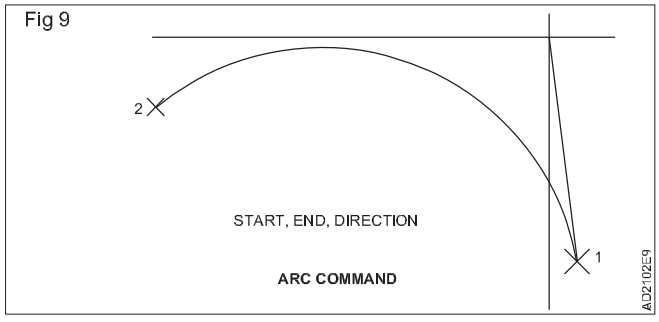
Start, End, Angle

This method creates an arc using a start point, endpoint, and an included angle. The included angle between the endpoints of the arc determines the center and the radius of the arc.

Start, End, Direction (Fig 10)

This method create an arc using a start point, endpoint, and a tangent direction at the start point. The tangent direction can be specified either by locating a point on the desired tangent line, or by entering and angle. You can determine which endpoint controls the tangent by changing the order in which you specify the two endpoints.

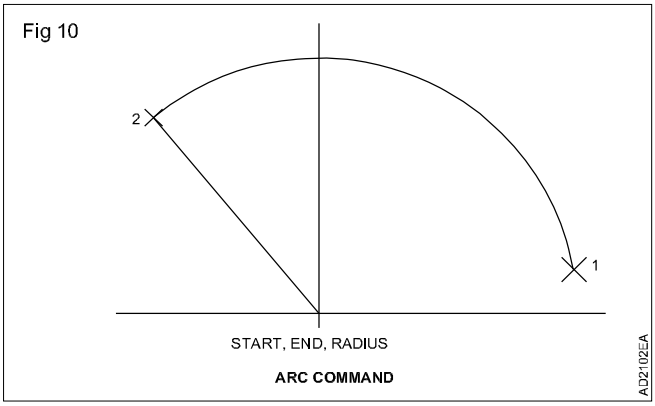
Fig 9



Start, End, Radius

This method creates an arc using a start point, endpoint, and a radius. The direction of the bulge of the arc is determined by the order in which you specify its endpoints. You can specify the radius either by entering it or by specifying a point at the desired radius distance. (Fig 11)

Fig 10



Centre, Start, End

This method helps to create an arc using a center, start point, and a third point that determines the endpoint. The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the center point first or start the point first. This method is same as the start center and end method except that the first point is specified is center instead of start.

Center, Start, Angle

This method helps to create an arc using a center, start point, and an included angel. The distance between the start point and the center determines the radius. The other end of the arc is deermined by specifying an included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the point first center or the start point first.

Start, Center, Length

This method create an arc using a center, start and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying and included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the point first center or the start point first.

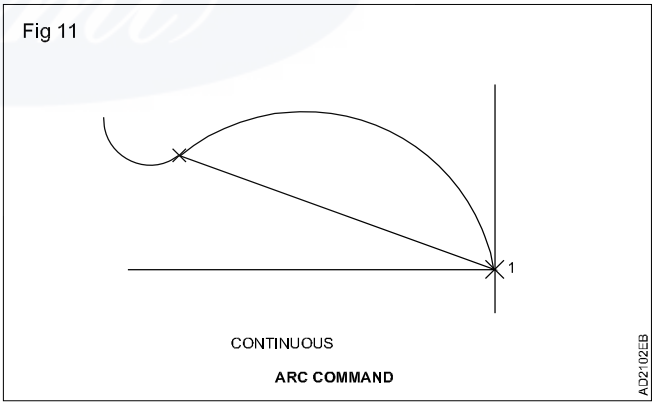
Start, Center, Length

This method create an arc using a center, start and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord between the start point and the endpoint of the arc. The resulting arc is always created counterclockwise from the start point. Using different options, you can specify either the start point first or the center point first. The length of the chord in the arc determines the included angle.

Continuous

Immediately after the creation of an arc, you can start a line that is tangent to the arc at an endpoint by starting the LINE command and pressing Enter at the specify First Point Prompt. You need to specify only the line length. Immediately after you create a line or an arc, you can start an arc that is tangent to an endpoint by starting the ARC command and pressing Enter at the Specify Start Point prompt. You need to specify only the endpoint of the new arc. (Fig 12)

Fig 11



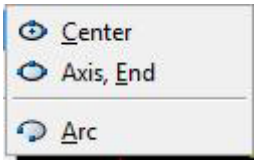
TASK 4 : **Ellipse**

Command alias	Button	Classic menu	Ribbon/Application menu
EL		Draw => Ellipse	Home => Ellipse

The shape of ellipse is just an oblong or an oval, with the major and minor axis.

The following prompts are displayed;

Specify axis endpoint of ellipse of [Arc/Center/Isocircle]:
Specify a point or enter an option



Axis End point

Defines the first axis by its two endpoints. The angle of the first axis determined the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

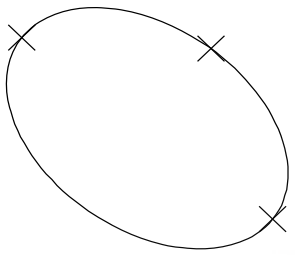
Distance to Other Axis

Defines the second axis using the distance from the midpoint of the first axis to the endpoint of the second axis.

Rotation

Create the ellipse by appearing to rotate a circle about the first axis. Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circular ellipse. (Fig 13)

Fig 12



AXIS END METHOD

ELLIPSE

AD2102EC

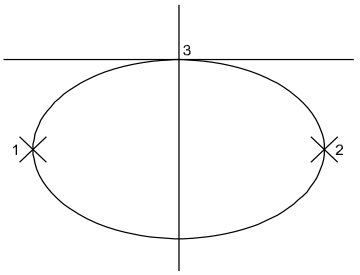
Arc

It creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis depending on is size. The first two points of the elliptical arc determine the location and length of the first axis. The third point determines the distance between the center of the elliptical arc and the endpoint of the second axis. The fourth and fifth points are the start and end angles.

Axis Endpoint

It defines the start point of the first axis. (Fig 14)

Fig 13



ELLIPSE BY AXIS ENDPOINT

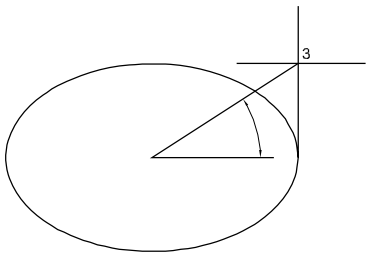
ELLIPSE

AD2102ED

Rotation

Defines the major to minor axis ratio of the ellipse by rotating a circle about the first axis. The higher the value from 0 through 89.4 degrees, the greater the ratio of minor to major axis. Values between 89.4 degrees and 90.6 degrees are invalid because the ellipse would otherwise appear as a straight line. Multiples of these angle values result in a mirrored effect every 90 degrees. (Fig 15)

Fig 14



ELLIPSE BY ROTATION

ELLIPSE

AD2102EE

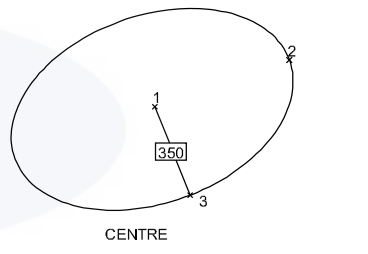
Start Angle

Defines the first endpoint of the elliptical arc. The Start Angle option toggles from Parameter mode to angle mode. The mode controls how the ellipse is calculated.

Center

Creates an ellipse using a center point, the endpoint of the first axis, and the length of the second axis. You can specify the distances by clicking a location at the desired distance or by entering a value for the length. (Fig 16)

Fig 15



CENTRE

ELLIPSE

AD2102EF

Distance to other Axis

Defines the second axis as the distance from the center of the ellipse, or midpoint of the first axis, to the point you specify.

Rotation

Creates the ellipse by appearing to rotate a circle about the first axis. Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circle, for instance.

Isocircle

Creates an isometric circle in the current isometric drawing plane.

The isocircle option is available only when you set the style option of SNAP to isometric.

Radius

Creates a circle using a radius you specify.

Diameter

Creates a circle using a diameter you specify.

TASK 5 : **Copy**

Command alias	Button	Classic menu	Ribbon/Application menu
CO		Modify => Copy	Home => Modify => Copy

The copy tool allows copying an existing object. This tool is used to make copies of the selected object and place them at specified location.

The following Prompts are displayed;

Select objects:

Specify base point or [Displacement/mode/Multiple] <Displacement>:

Specify second point or [Array] <use first point as displacement>:

Displacement

Specifies a relative distance and direction using coordinates. The two points you specify define a vector that indicates how far the copied objects are to be placed from the original and also in what direction. If you press Enter at the Specify Second Point prompt, the first point is interpreted as a relative X, Y, Z displacement.

Mode

Controls the automatic repetition of the command. (COPYMODE system variable).

Single

Creates a single copy of selected objects and ends the command.

Multiple

Overrides the single mode setting. The COPY command is set to repeat automatically for the duration of the command.

Array

Arrange a specified number of copies in a linear array.

Number of items to Array

Specifies the number of items in the array, including the original selection set.

Second point

Determines a distance and direction for the array relative to the base point. BY default, the first copy in the array is positioned at the specified displacement. The remaining copies are positioned in a linear array beyond that point using the same incremental displacement.

Fit

Positions the final copy in the array at the specified displacement. The other copies are fit in a linear array between the original selection set and the final copy.

— — — — — — — —

TASK 6 : **Move**

Command alias	Button	Classic menu	Ribbon/Application menu
CO		Modify => Copy	Home => Modify => Copy

The move tool is used to move one or more objects from their current location to new location without changing their size or orientation.

The following prompts are displayed;

Select objects:

Specify base point or [Displacement/mode/Multiple] <Displacement>:

Specify second point or [Array] <use first point as displacement>:

After selection the objects by using any one of the selection techniques, you will be prompted to specify the base point. The base point is the reference point with respect to which the object will be picked and moved. On specifying the base point, you will be prompt to specify the second point of displacement. The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.

If you press Enter at the specify Second Point prompt, the first point is interpreted as a relative X, Y, Z displacement. For example, if you specify 2,3 for the base point and press Enter at the next prompt, the objects move 2 units in the X direction and 3 units in the Y direction from their current position.

Displacement

Enter coordinates to represent a vector. The coordinate values that you enter specify a relative distance and direction.

— — — — — — — —

TASK 7 : **Rotate**

Command alias	Button	Classic menu	Ribbon/Application menu
RO		Modify => Rotate	Home => Modify => Rotate

While creating designs sometimes we have to rotate an object or group of objects. This requirement can be accomplished by using Rotate tool.

The following prompts are displayed;

Current positive angle in UCS: ANGDIR = Current ANGBASE = current.

Select objects:
Specify base point:
Specify rotation angle or [Copy/Reference]:

Rotation Angle

Determines how far an object rotates around the base point. The axis of rotation passes through the specified base point and is parallel to the Z axis of the current UCS.

Copy

Creates a copy of the selected objects for rotation.

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TASK 8 : **Erase**

Command alias	Button	Classic menu	Ribbon/Application menu
E		Modify => Erase	Home => Modify => Erase

Erase helps to remove the objects from a drawing. To erase the unwanted objects;

1) Activate the command.
2) At the Select Objects prompt, use a selection method to select the objects to be erased or enter an option.
3) Press Enter to end the command.

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TASK 9 : **Undo**

Command alias	Shortcut key	Button	Classic menu	Ribbon/Application menu
U	Ctrl + Z		Edit => Erase	Home => Modify => Erase

UNDO displays the command or system variable name at the Command prompt to indicate that you have stepped past the point where the command was used.

Undo has no effect on some commands and system variables, including those that open, close, or save a window or a drawing, display information, change the graphics display, regenerate the drawing, or export the drawing in a different format.

The command has the following options.

Number

Undoes the specified number or preceding operations. The effect is the same as entering 'U' multiple times.

Auto

Groups the commands in a macro, such as a menu macro, into a single action, making them reversible by a single U command.

Control

Limits or turns off UNDO.

All

Turns on the full UNDO command.

None

Turns off the U and UNDO commands and discards any UNDO command information saved earlier in the editing session. The Undo button on the Standard toolbar is unavailable. The Auto, Begin, and Mark options are not available when None or One is in effect.

One

Limits UNDO to a single operation. The main prompt for the UNDO command changes to show that only a control option or a single step of the UNDO command is available when that particular option is in effect.

Layer

Controls whether the layer dialog operations are combined as a single undo operation.

Begin, End

Groups a sequence of actions into a set. After you enter the Begin option, all subsequent actions become part of this set until you use the End option. Entering undo begin

while a group is already active ends the current set and begins a new one. UNDO and U treat grouped actions as a single action.

If you enter undo begin without undo end, using the Number option undoes the specified number commands but does not back up past the beginning point. If you want to go back to before the beginning point, you must choose the End option, even if the set is empty. The same applies to the U command. Mark placed by the Mark option disappears inside an UNDO group.

Mark, Back

Mark places a mark in the undo information. Back undoes all the work done back to this mark. If you undo one operation at a time, you are informed when you reach the mark. You can place as many marks necessary. Back moves back one mark at a time, removing the mark. If no mark is found, back displays the following prompt:

This will undo everything. OK? <Y>:

Enter y to undo all commands entered in the current session. Enter n to ignore the back option. When you we the Number option to undo multiple actions, UNDO stops if it encounters a mark.

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TASK 10 : **Mirror**

Command alias	Button	Classic menu	Ribbon/Application menu
E		Modify => Mirror	Home => Modify => Mirror

It creates a mirror image of the selected object. The user defines two points, along which Auto CAD generates a "line of reflection" and the reflected object is generated across this line with all components reversed.

The following prompts are displayed;

Select objects:

Specify first point of mirror line:

Specify a point:

Specify second point of mirror line:

Erase source objects? [Yes/No]<N>:

The two specified points become the endpoints of a line about which the selected objects are mirrored.

When you select 'Yes' in the prompt 'Erase source objects?', Auto CAD places the mirrored image into the drawing and erase the original objects. Otherwise places the mirrored image into the drawing and retains the original objects.

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TASK 11 : **Offset**

Command alias	Button	Classic menu	Ribbon/Application menu
O		Modify => Offset	Home => Modify => Offset

This command creates concentric circles, parallel lines and parallel curve at a specified offset distance.

The following prompts are displayed;

Current settings : Erase source = current layer = current OFFSETGAPTYPE = current

Specify offset distance of [Through/Erase/Layer] <current>

Offset distance

Creates an object at a specified distance from an existing object.

Multiple

Enters the multiple offset mode, which repeats the offset operation using the current offset distance.

Undo

Reverses the previous offset.

Through

Creates an object passing through a specified point.

Erase

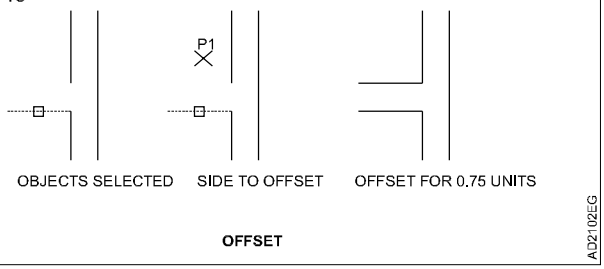
Erases the source object after it has been offer.

Layer

Determines whether offset objects are created on the current layer or on the layer of the source object. (Fig 17)

For best results when you offset a polyline with corners, specify the through point near the midpoint of a line segment, not near a corner.

Fig 16



Drafting of furnitures (Three seater sofa)

Objectives: At the end of this exercise you shall be able to
- **draw plan of a three seater sofa**
- **draw a plan of side table.**

PROCEDURE

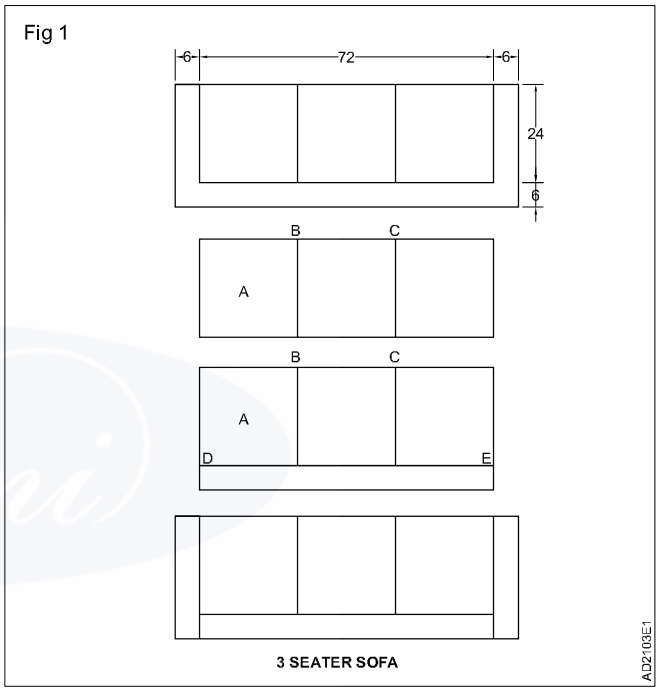
TASK 1 to 4 : **Draw the plan elevation of three seater sofa and circular side table**

TASK 1 : **Draw the plan of three seater sofa on computer**

Data : Size of sofa 84 x 36 cm.

- With line command draw a square of size 24"X24"
- Copy the square touching square 'a' at point B and C
- Draw rectangle with the help of the command of size 74x6 and move it to touch the line DE.
- Draw another rectangle with line command and move it to touch point F. Copy the rectangle to touch ling EG.

Fig 1



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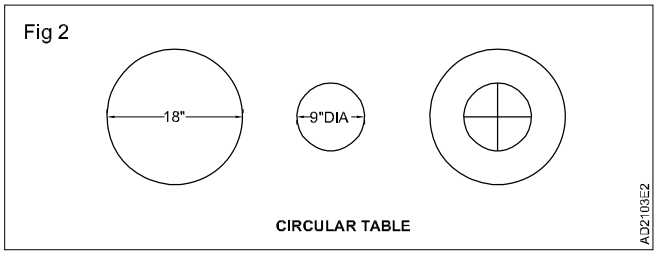
TASK 2 :

- Draw plan of circular size table.
- Data - Dia of table 18" inner circle dia - a"
- Draw a circle of dia 18"
- Draw a circle of dia 9"
- Moving the circle of a" dia from centre point to the centre point of circle of 18" dia
- Draw a line of a" (Equal to the dia of circle)

Fig 2



- Move the line from mid point to the midpoint of circle of a" dia.

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Basic drafting commands - 2

Objectives: At the end of this exercise you shall be able to

- **to understand basic drafting commands listed**
  - **rectangle**
  - **polygon**
  - **explode**
  - **array**
  - **scale**
  - **stretch**
  - **lengthen**
  - **trim**
  - **extend**
  - **break**
  - **join**
  - **chanfer**
  - **fillet.**

PROCEDURE

TASK 1 : **Explode**

Command alias	Button	Classic menu	Ribbon/Application menu
X		Modify => Explode	Home => Modify

Explodes a compound object when you want modify its components separately. Objects that can be exploded include blocks, polylines, and regions, among others.

The colour, line type, and line weight of any exploded object might change. Other results differ depending on the type of compound object you're exploding. See the following list of objects that can be exploded and the results for each.

2D Polyline

Discards any associated width or tangent information. For wide poly lines, the resulting lines and arcs are placed along the center of the polyline.

3D Polyline

Explodes into line segments. Any line type assigned to the 3D poly line is applied to each resulting line segment.

Annotative objects

Explodes the current scale representation into its constituent parts which are no longer annotative. Other scale representations are removed.

Arc

If within a non-uniformly scaled block, explodes into elliptical arcs.

Array

Explodes an associative array into copies of the original objects.

Block

Removes one grouping level at a time. If a block contains a poly line or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y and Z scales explode into their component objects. Blocks with unequal X,Y and Z scales (non-Uniformly scaled blocks) might explode into unexpected objects.

When non-uniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with as "E" prefix) and referenced with the non-uniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a non-uniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

Blocks inserted with external references (Xrefs) and their dependent blocks cannot be exploded.

Blocks insert with MINSERT cannot be exploded.

Circle

If within a non-uniformly scaled block, explodes into ellipses.

Leaders

Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), multiline text, or tolerance objects, depending on the leader.

Mesh objects

Explodes each face into a separate 3D face object. Colour and materials assignments are retained.

Multiline text

Explodes into text objects.

Multiline

Explodes into lines and arcs.

Polyface mesh

Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

Region

Explodes into lines, arcs, or splines.

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TASK 2 : **Rectangle**

Command alias	Button	Classic menu	Ribbon/Application menu
REC		Draw => Rectangle	Home => Rectangle

With this command, you can specify the rectangle parameters (length, width, rotation) and control the type of corners (fillet, chamber, or square).

The following prompts are displayed;

Current settings: Rotation = 0

Specify first corner point or [Chamfer/Elevation/Fillet/ Thickness/width]: specify a point or enter an option

First corner point

Specifies a corner point of the rectangle.

Other corner point

Creates a rectangle using the specified points as diagonally opposite corners.

Area

Creates a rectangle using the area and either a length or a width. If the Chamfer or Fillet option is active, the area includes the effect of the chamfers or fillets in the corners of the rectangle.

Dimensions

Creates a rectangle using length and width values.

Rotation

Creates a rectangle at a specified rotation angle.

Chamfer

Sets the chamfer distances for the rectangle.

Elevation

Specifies the elevation of the rectangle.

Fillet

Specifies the fillet radius of the rectangle.

Thickness

Specifies the thickness of the rectangle.

Width

Specifies the polyline width of the rectangle to be drawn.

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TASK 3 : **Array**

Command alias	Button	Classic menu	Ribbon/Application menu
AR		Modify => Array	Home => Modify => Array

It creates copies of objects arranged in both 2D as well as 3d pattern. You can create copies of objects in a regularly spaced rectangular, polar, or path array.

The following prompts are displayed;

Select objects:

Enter array type [Rectangular/PAth/Polar] < Rectangular>:

Rectangular(ARRAYRECT)

Distributes object copies into any combination of rows, columns, and levels. The following prompts are displayed.

Select objects:

Specify opposite corner for number of items or [Base point/ Angle/Count] <Count>:

Press Enter to accept or [Associative/Base point/Rows/ Columns/Levels/eXit] <eXit>:

Items

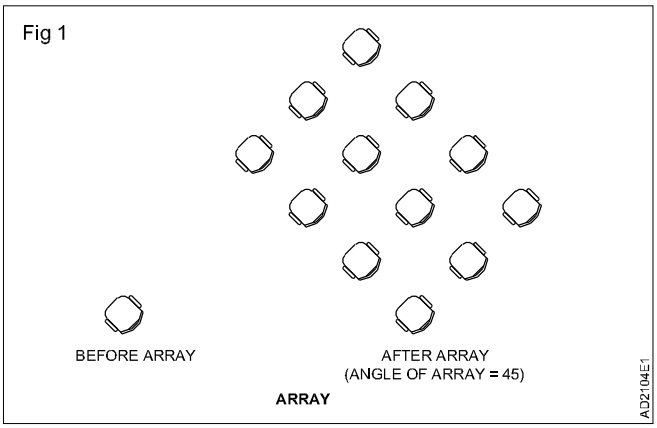
Specifies the number of items in the array.

Count

Specifies the row and column values individually.

Space Items

Specifies the row and column spacing. (Fig 1)



Spacing

Specifies the row and column spacing individually.

Base point

Specifies a base point for the array.

Key Point

For associative arrays, a valid constraint is specified (or key point) on the source objects to be used as the base point. If you edit the source objects of the resulting array, the base point of the array remains coincident with the key point of the source objects.

Rows

Edits the number and spacing of rows in the array and also the incremental elevation between them.

Expression

Derives a value using a mathematical formula or equation.

Total

Sets the total distance between the first and last rows.

Columns

Edits the number and spacing of columns.

Total

Specifies the total distance between the first and last columns.

Levels

Specifies the number and spacing of levels.

Total

Specifies the total distance between the first and last levels.

Exit

Exits the command.

Path (ARRAYPATH)

Evenly distributes object copies along a path or a portion of a path. The following prompts are displayed:

Select objects:

Select path curve:

Enter number of items along path or [Orientation/Expression] <Orientation>:

Specify base point or [Key point] <end of path curve>:

Specify direction to align with path or [2Points/Normal] <current>:

Specify the distance between items along path or [Divide/Total/Expression] <Divide evenly along path>:

Press Enter to accept or [Associative/Base point/Items/Rows/Levels/Align items/Z direction/eXit] <eXit>:

Path curve

Specifies the object to use for the path of the array. Select a line, ploy line, 3D polyline, spline, helix, arc, circle, or ellipse.

Number of Items

Specifies the number of items in the array.

Orientation

Controls whether the selected objects are reoriented (rotated) relative to the starting direction of the path before being moved to the beginning point of the path.

2. Points

Specifies two points to define the direction to be aligned with the starting direction of the path.

Normal

Objects are aligned to be normal to the starting direction of the path.

The align items option controls whether to maintain the starting orientation or continue reorienting items along the path, relative to the starting orientation.

Expression

Derives a value using a mathematical formula or equation.

Key point

For associative arrays, specifies a valid constraint point (or key point) on the source objects to use as the base point. If you edit the source objects of the resulting array, the base point of the array remains coincident with the key point of the source objects.

Distance between Items

Specifies the distance between items.

Divide

Divide items evenly along the entire length of the path.

Total

Specifies the total distance between the first and last items.

Associative

Specifies whether to create items in the array as an associative array object, or as independent objects.

Yes: Contains array items in a single array object, similar to a block. This allows you to quickly propagate changes by editing the properties and source objects of the array.

No: Creates array items as independent objects. Changes made on one item do not affect the other items.

Items

Edits the number of items in the array. If the Method property is set to Measure, you are prompted to redefine the distribution method (Distance between Items, Divide, and Total options).

Rows

Specifies the number and spacing of rows in the array, and the incremental elevation between them.

Total

Specifies the total distance between the first and last rows.

Align items

Specifies whether to align each item to be tangent to the path direction. Alignment is relative to the first item's orientation (Orientation option).

Z. Direction

Controls whether to maintain the items' direction or to naturally bank the items along a 3D path.

Exit

Exits the command.

Polar (ARRAYPOLAR)

Evenly distributes object copies in a circular pattern around a center point or axis of rotation. The following prompts are displayed;

Select objects:

Specify center point of array or [Base point/Axis of rotation]:

Enter number of items or [Angle between/Expression] <last count>:

Specify the angle to fill (+= ccw, -= cw) or [Expression]:

Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate items/eXit] <eXit>:

Center Point

Specifies the point around which to distribute the array items. The axis of rotation axis of the current UCS.

Base Point

Specifies a base point for the array.

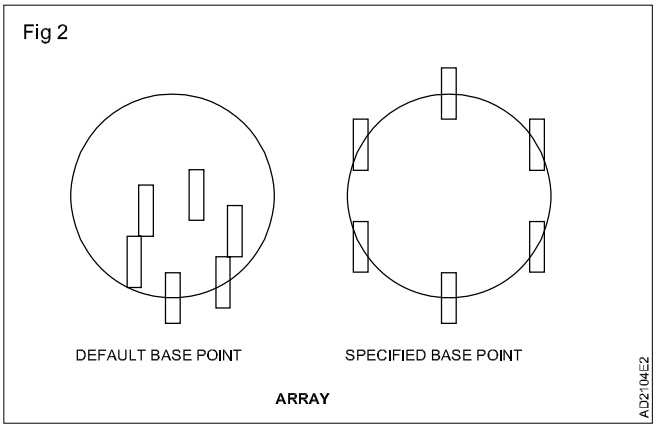
Key point

For associative arrays, a valid constraint is specified (or key point) on the source objects to be used as the base point. If you edit the source objects of the resulting array, the base point of the array remains coincident with the key point of the source objects.

Axis of rotation

Specifies a custom axis of rotation defined by two specified points.(Fig 2)

Fig 2



Items

Specifies the number of items in the array.

Expression

Derives a value using a mathematical formula or equation.

When defining the fill angle in an expression, the (+ or -) mathematical symbol in the resultant value does not affect the direction of the array.

Angle between

Specifies the angle between items.

Fill angle

Specifies the angle between the first and last item in the array.

Associative

Specifies whether to create items in the array as an associative array object or as independent objects.

Yes: Contains array items in a single array object, similar to a block. This allows you to quickly propagate changes by editing the properties and source objects or the array.

NO: Creates array items as independent objects. Changes made on one item do not affect the other items.

Rows

Edits the number and spacing of rows in the array and the incremental elevation between them.

Total

Sets the total distance between the first and the last rows.

Levels

Specifies the number and spacing of levels in the array.

Total

Specifies the total distance between the first and last levels.

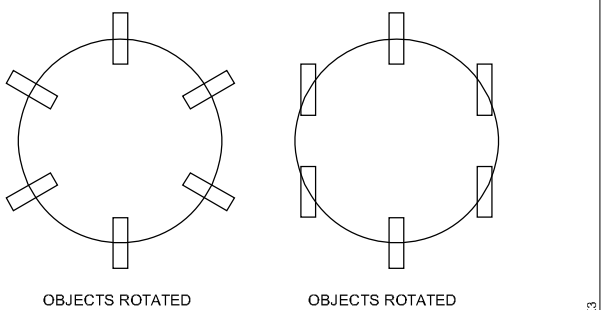
Rotate Items

Controls whether items are rotated as they are arrayed.

Exit

Exits the command. (Fig 3)

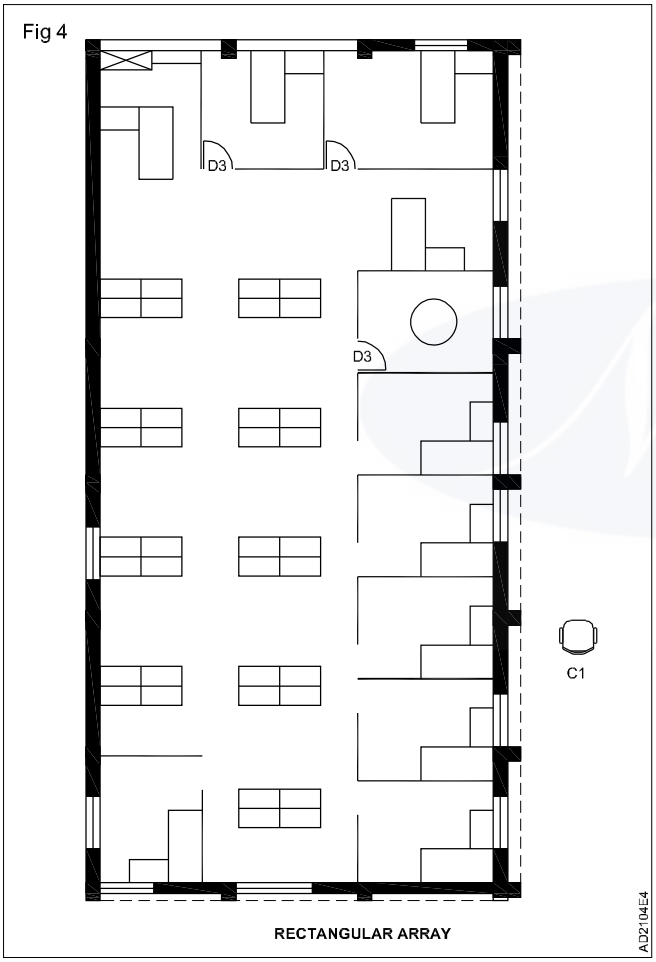
Fig 3



TASK 4 : **Draft using rectangular array**

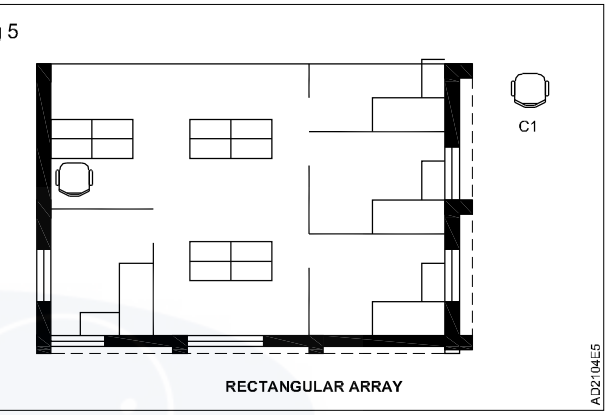
1 Open array drawing (Fig 4)

Fig 4



2 Copy the C1 chair and place it in front of the table as shown in Fig 5

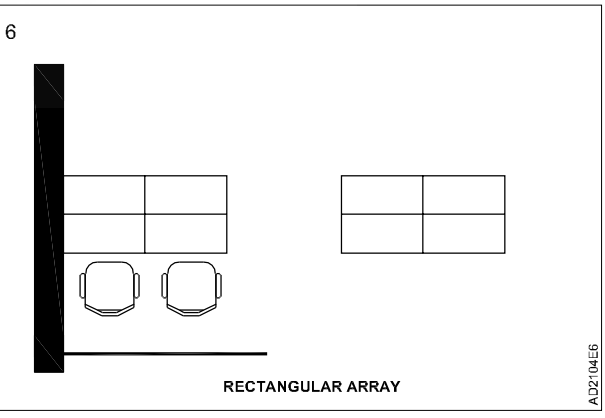
Fig 5



3 Move the chair 3” down

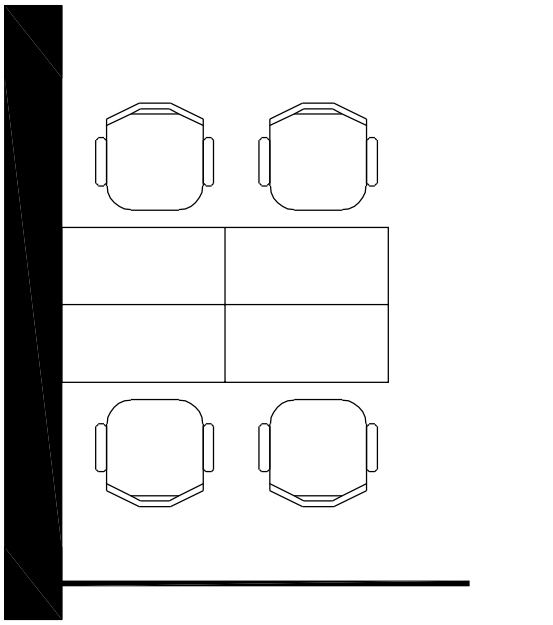
4 Copy the chair towards X axis at the distance of 4’2” (Fig 6)

Fig 6



5 Mirror both the chairs on the other side. (Fig 7)

Fig 7

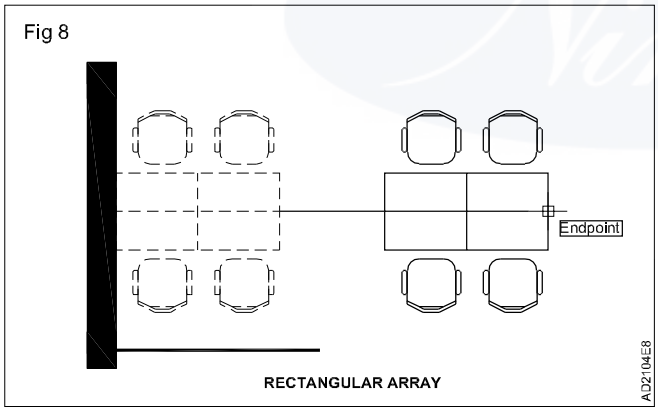


6 In command prompt type “Array”

i. Select the 4 chairs and enter

ii. Choose rectangaluar option and click the right side mid-point of the other table to specify the number of items as shown in Fig 8.

Fig 8



iii. Similarly click the bottom mid-point to specify spacing between the items as shown below.

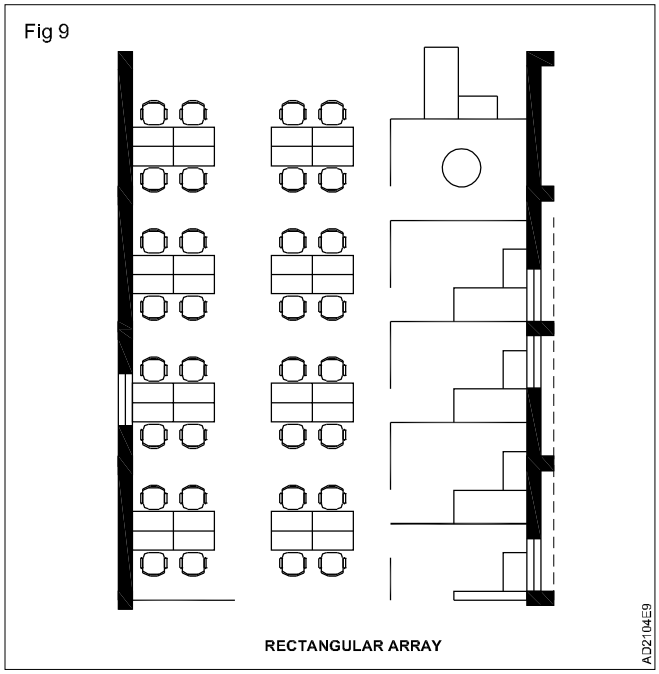
iv Now press R to create rows, Enter number of rows as 2.

v. Enter value as 12’2” and press enter twice to exit the command.

vi. Now you view the arrayed chairs.

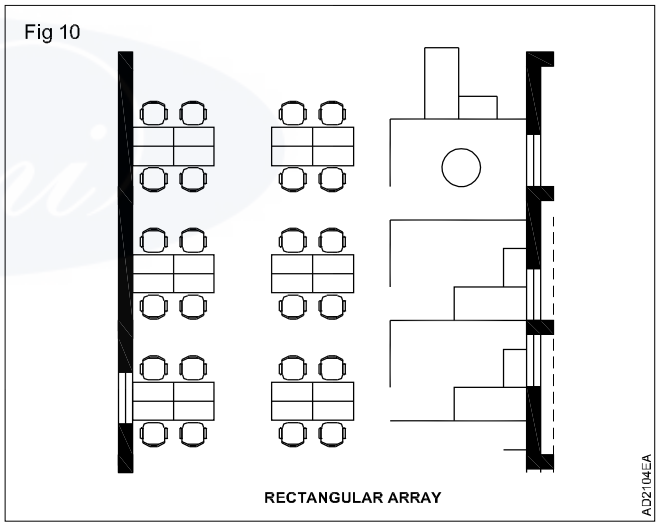
7 Copy the arrayed chairs and place it on the other set of table as shown in Fig 9.

Fig 9



8 To set the chair properly, pick the arrayed chairs and change the row spacing as 9’8”. (Fig 10)

Fig 10

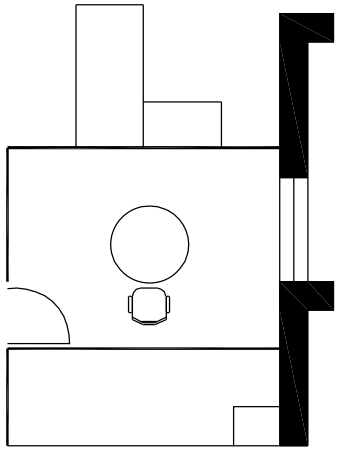


TASK 5 : **Draft using polar array (array 6 chairs around circular table)**

1 Let us do the polar array

i. Copy the C1 chair and place near the round table as shown in Fig 11.

Fig 11



POLAR ARRAY

AD2104EB

ii. Type array command and enter “PO” to choose polar array.

iii. Select the chair and choose the base point as table center.

iv. Move the cursor around the table to array the chair.

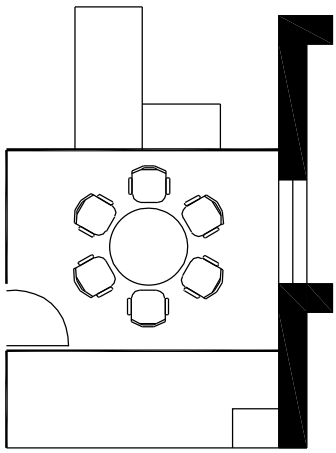
v. Pick on the bottom left corner; press enter to accept 360 as sngle to fill. (Fig 12)

vi. Once again press enter to exit the command.

vii. By default items will arrayed. To modify the array pick the arrayed objects.

viii. Change the number of items as 4 in ribbon and view the result. (Fig 13)

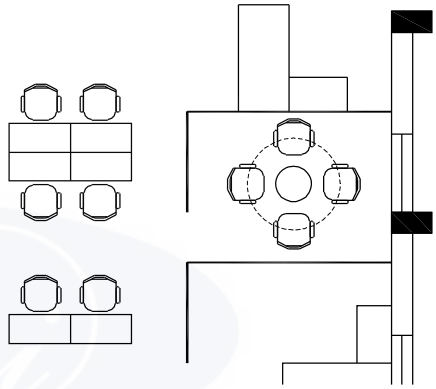
Fig 12



POLAR ARRAY

AD2104EC

Fig 14



RECTANGULAR ARRAY

AD2104EDA

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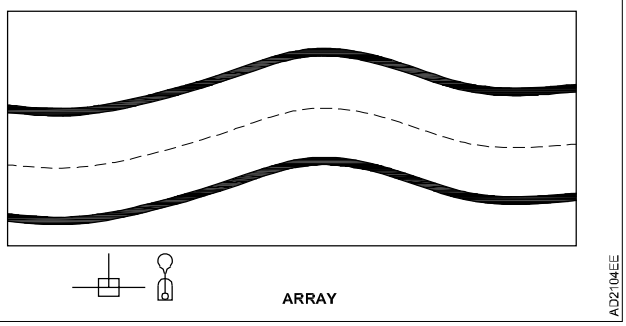
TASK 6 : Array lamp posts or a curved road

1 Open the file road drawing. (Fig 14)

Fig 14



ARRAY

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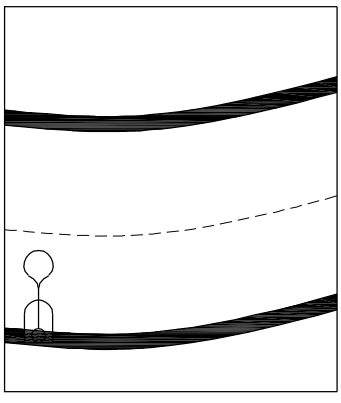
2 Copy the rotate the lamp post as shown in the Fig 15.

Fig 15



ARRAY

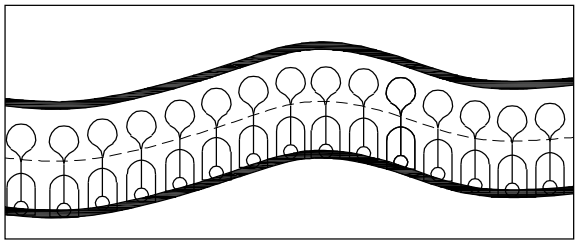
AD2104EF

3 Type array command and enter “PA” for the path array option

i. Select the lamp post object and press enter to select the path.

ii. Select the path curve and pick the right end point of the path as shown in Fig 16.

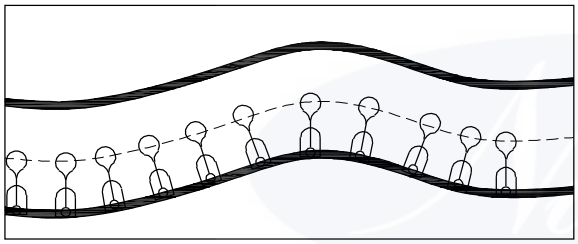
Fig 16



iii. Press enter twice to exit the command

iv. To reduce the number of object pick the arrayed object and enter value 12 in number of items. (Fig 17)

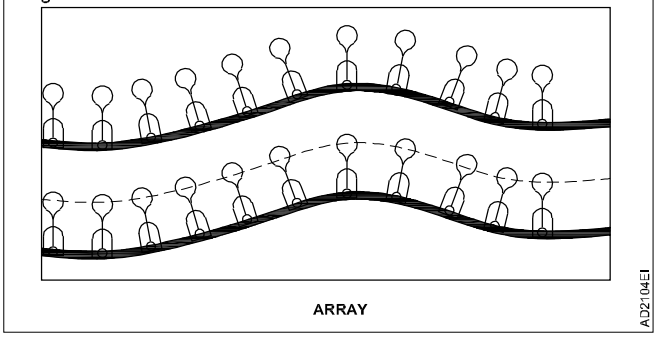
Fig 17



4 Let us place the lamps on the other side of the road,

i. Copy the arayed object and place it on the other side of the road. (Fig 18)

Fig 18



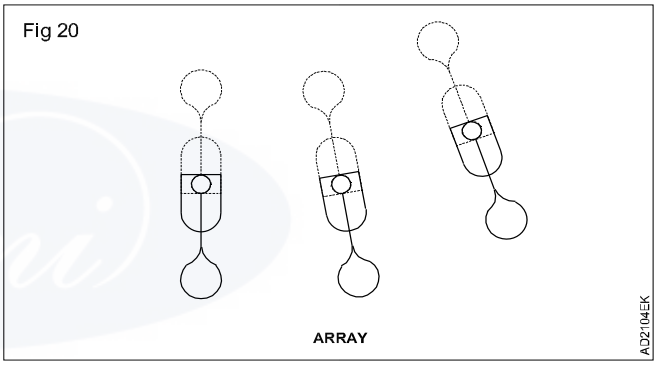
ii. Select the arrayed object and click the eidt source from the ribbon as shown.

iii. Select the first object to edit and click OK to the warning dialog box. (Fig 19)

iv. Rotate the object by 180. Now all the objects will rotate.

5 Enter zoom all to view all the objects.

Fig 20



— — — — — — — —

TASK 7 : **Scale**

Command alias	Button	Classic menu	Ribbon/Application menu
SC		Modify => Scale	Home => Modify => Scale

It Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling. The object(s) are selected, and a base point and scaling factor are defined. An easy way of changing a drawing from metres to millimetres is to scale everything up by a factor of 1000.

The following prompts are displayed;

Select objects:

Specify base point:

Specify scale factor or [Copy/Reference]:

The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

When you use the SCALE command with annotative objects, the position or location of the object is scaled relative to the base point of the scale operation, without changing the size of the object.

Scale Factor

Multiplies the dimensions of the selected objects by a specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects. You can also drag the cursor to make the object larger or smaller.

Copy

Creates a copy of the selected objects for scaling.

References

Scales the selected objects based on a reference length and a specified new length.

TASK 5 : **Stretch**

Command alias	Button	Classic menu	Ribbon/Application menu
S		Modify => Stretch	Home => Modify => Stretch

This command uses to lengthen object, shorten them and to alter their shapes by a selection window or polygon. STRETCH moves only the vertices and endpoints that lie inside the crossing selection, leaving those outside unchanged. STRETCH does not modify 3D solids, poly line width, tangent, or curve-fitting information.

The following prompts are displayed;

Select objects to stretch by crossing-window or crossing-polygon...

Select objects:

Specify base point or [Displacement] <last displacement>;

Specify second point or <use first point as displacement>:

Specify displacement <last value>:

If you enter a second point, the objects are stretched over the vector distance from the base point to the second point. If you press Enter at the Specify Second Point of Displacement prompt, the first point is treated as an X, Y, Z displacement.

— — — — — — — —

TASK 6 : **Lengthen**

The Lengthen command can often be used instead of either the Trim or Extend commands. Indeed the end result is exactly the same. The lengthen command can be used to either lengthen or shorten lines, Arcs, open polylines, elliptical Arcs and open splines without the use of cutting or boundary edges.

Select an object or [Delta/Percent/Total/Dynamic]:

Delta

This option is used if you want to add or remove extra length to or from current length. If you input as negative value, the lengthen command will shorten the line. Following options are displayed;

Select an object or [DElta/Percent/Total/Dynamic]:de

Enter delta length or [Angle] <0.0000>: 10

Select an object to change or [Undo]:

Percent

If you want to add or remove from the length, specify a percentage of the current length. The number should be positive, and non-zero number. If it is > 100 the object will lengthen, if the value is < 100, the object will shorten. The following prompts are displayed;

Select an object or [Delta/Percent/Total/Dynamic]: p

Enter percentage length<100.0000>:50

Select an object to change or [Undo]:

Total

Use the total option if you want the new total length of the line to be equal to the number you input. If the new length is greater than current length the line will lengthen. If the new length is lesser than the current length, then the line will shorten. The following prompts are displayed;

Select an object or [DElta/Percent/Total/Dynamic]: t

Specify total length or [Angle] <1.0000>:30

Select an object to change or [Undo]:

— — — — — — — —

TASK 7 : **Trim**

Command alias	Button	Classic menu	Ribbon/Application menu
TR		Modify => Trim	Home => Modify => Trim

The trim command shortens an object so that it ends precisely at a certain boundary.

The following prompts are displayed;

Current settings: Projection = current, Edge = current

Select cutting edges....

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Select the objects that define the cutting edges to which you want to trim an object, or press Enter to select all Displayed objects as potential cutting edges. TRIM projects the cutting edges and the objects to be trimmed onto the XY plane of the current user coordinate system (UCS).

Specify an object selection method to select the objects to be trimmed. If more than one trim result is possible, the location of the first selection point determines the result.

Object to Trim

Specifies the object to be trimmed.

Shift select to extend

Extends the selected objects rather than trimming them. This option provides an easy method to switch between trimming and extending.

Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Crossing

Selects objects within and crossing a rectangular area defined by two points.

Some crossing selections of objects to be trimmed are ambiguous. TRIM resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

Project

Specifies the projection method used while trimming objects.

View

Specifies projection along the current view direction. The command objects that intersect the boundary in the current view.

Edge

Determines whether an object is trimmed at another object's extrapolated edge or only to an object that intersects it in 3D space.

Extend

Extend the cutting edge along its natural path to intersect an object in 3D space.

No extend

Specifies that the object is trimmed only at a cutting edge that intersects it in 3D space.

When trimming hatches, do not set Edge to Extend. If you do, gaps in the trim boundaries will not be bridged when trimming hatches, even when the gap tolerance is set to a correct value.

Erase

Deletes selected objects. This option provides a convenient method to erase unneeded objects without leaving the TRIM command.

Undo

Reverses the most recent change made by TRIM.

— — — — — — — —

TASK 8 : **Extend**

Command alias	Button	Classic menu	Ribbon/Application menu
EX		Modify => Extend	Home => Modify => Extend

The Extend command allows increasing the length of an object within the limits of object definition.

The following prompt are displayed;

Current settings: Projection = current, Edge = current

Select boundary edges...

Select objects or <select all>:

Select object to extend or shift-select to trim or [Fence/ Crossing/Project/Edge/Undo]:

— — — — — — — —

TASK 9 : **Break**

Command alias	Button	Classic menu	Ribbon/Application menu
BR		Modify => Break	Home => Modify => Break

This command erases the part of an object.

The following prompts are displayed;

Select object:

The prompts that are displayed next depend on how you select the object. If you select the object using your pointing device, the program both selects the object and treats the selection point as the first break point. At the next prompt, you can continue by specifying the second point or overriding the first point.

Specify second break point or [First point]:

Second break point

Specifies the second point to use to break the object.

First point

Overrides the original first point with the new point that you specify.

The portion of the object is erased between the two points specified by you. If the second point is not on the object,

the nearest point on the object is selected; therefore, to break off one end of a line, arc, or polyline, specify the second point beyond the end to be removed. To split an object in two without erasing a portion, enter the same point for both the first and second points. You can do this by entering @ to specify the second point.

Lines, arcs, circles, polylines, ellipses, splines, donuts, and several other object types can be split into two objects or have one end removed.

The program converts a circle to an arc by removing a piece of the circle starting counter clockwise from the first to the second point.

You can also break selected objects at a single point with the break at point tool.

Valid objects include lines, open polylines, and arcs. Closed objects such as circle cannot be broken at a single point.

— — — — — — — —

TASK 10 : **Join**

Command alias	Button	Classic menu	Ribbon/Application menu
JO		Modify => Join	Home => Modify => Join

This command joins the endpoints of linear and curved objects to create a single object. The following prompts are displayed.

Select sources object or multiple objects to join at once;

Source object

Specifies a single source object to which you can join other objects. Press enter after selecting the source object to begin selecting the objects to join. The following rules apply for each type of source object.

Line

Only line objects can be joined to the source line. The line objects must all be collinear, but can have gaps between them.

Polyline

Lines, polylines, and arcs can be joined to the source polyline. All objects must be contiguous and coplanar. The resulting object is a single polyline.

3D polyline

Any linear or curved object can be joined to the source 3D polyline. All the objects must be contiguous, but can be non-coplanar. The resulting object is either a single 3D polyline or a single spline, depending on whether you are joining to a linear or a curved object respectively.

— — — — — — — —

TASK 11 : **Chamfer**

Command alias	Button	Classic menu	Ribbon/Application menu
CHA		Modify => Chamfer	Home => Modify => Chamfer

The chamfer command draws a straight line segment between two given lines. Chamfer is the name for machining process of flattening a sharp corner to create a bevelled edge.

The following prompts are displayed;

(Trim mode) Current chamfer Dist1 = current, Dist2 = current select first line or [Undo/Polyline/Distance/Angle/Trim/Method/Multiple]:

First line

Specifies the first of two edges required to define a 2D chamfer, If you select lines or polylines, their lengths adjust to accommodate the chamfer line. You can hold down shift while selecting the objects to override the current chamfer distances with a value of 0.

If the selected objects are line segments of a 2D polyline, they must be adjacent or separated by no more than one segment. If they're separated by another polyline segment, CHAMFER deletes the segment that separates them and replaces it with the chamfer.

Edge

Selects an individual edge to chamfer.

Loop

Switches to Edge Loop mode.

Edge Loop

Selects all edges on the base surface.

Undo

Reverses the previous action in the command.

Polyline

Chamfers an entire 2D polyline.

The intersecting polyline segments are chamfered at each vertex of the polyline. Chamfers become new segments of the polyline. If the polyline includes segments that are too short to accommodate the chamfer distance, those segments are not chamfered.

Distance

Sets the distance of the chamfer from the endpoint of the selected edge. If you set both distances to zero, CHAMFER extends or trims the two lines so they end at the same point.

Angle

Sets the chamfer distances using a chamfer distance for the first line and an angle for the second line.

Trim

Controls the trimming of selected edges to the chamfer line endpoints, by CHAMFER.

Method

Controls whether CHAMFER uses two distances or a distance and an angle to create the chamfer.

Multiple

Chamfers the edges of more than one set of objects.

Expression

Determines the chamfer distance with a mathematical expression. See control Geometry with the parameters manager for a list of operators and functions allowed.

— — — — — — — —

TASK 12 : **Fillet**

Command alias	Button	Classic menu	Ribbon/Application menu
F		Modify => Fillet	Home => Modify => Fillet

This command fits a smooth arc between any combination of lines, arcs or circles. Once the radius value is identified the direction of the fillet is determined by the cursor location which is used to identify the two objects.

The following prompts are displayed;

Current settings: Mode = current, Radius = current

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

First object

Selects the first of two objects required to define a 2D fillet or fillet the edge.

Select second object or shift- select to apply corner:

Enter fillet radius <current>:

Select an edge or [Chain/Loop/Radius]:

Edge

Selects a single edge. You can continue to select single edges until you press Enter. If you select three or more edges that converge at a vertex to form the corner of a box, FILLET computes a vertex blend that is part of a sphere if the three incident fillets have the same radii.

Chain

Changes from selection of single edges to selection of sequential tangent edges, called a chain selection.

Edge chain

Selects a tangential sequence of edges when you select a single edge. For example, if you select an edge on the top of a 3D solid box, FILLET also selects the other tangential edges on the top.

Edge

Switches to a single-edge selection mode.

Loop

Specifies a loop of edges on the face of a solid. For any edge, there are two possible loops. After selecting a loop edge, you are prompted to Accept the current selection, or choose the Next loop.

Radius

Defines the radius of the rounded edge.

Undo

Reverse the previous action in the command.

Polyline

Inserts fillet arcs at each vertex of a 2D polyline where two line segments meet. If one arc segment separates two line segments that converge as they approach the arc segment, FILLET removes the arc segment and replaces it with a fillet arc.

Radius

Defines the radius of the fillet arc. The value you enter becomes the current radius for subsequent FILLET commands. Changing this value does not affect existing fillet arcs.

Trim

Controls whether FILLET trims the selected edges to the fillet arc endpoints.

Multiple

Rounds the edges of more than one set of objects.

Skill Sequence - 2

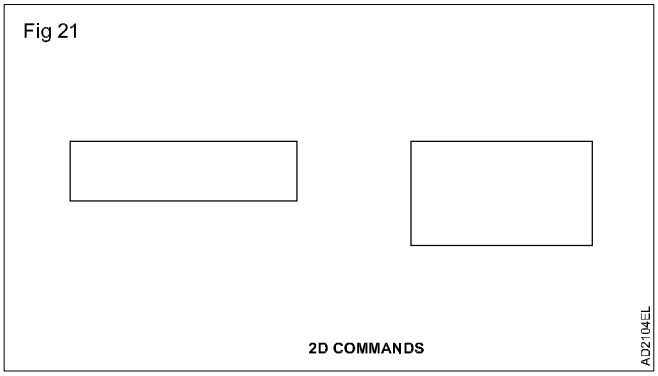
Objective : This shall help you to
• draft plan of the chair using 2d commands.

1 Create two tectangles with the following dimensions.

I Length = 15; Width = 4

II Length = 12; Width = 7 (Fig 20)

Fig 21



2 Use the MOVE command to assemble the rectangles as follows. (Fig 21)

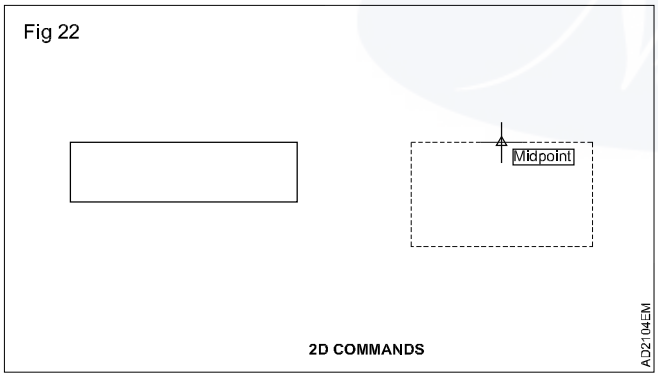
Command: Move

Select object: select the second rectangle

Select object: <Null response>

Base point: <Pick the top midpoint>

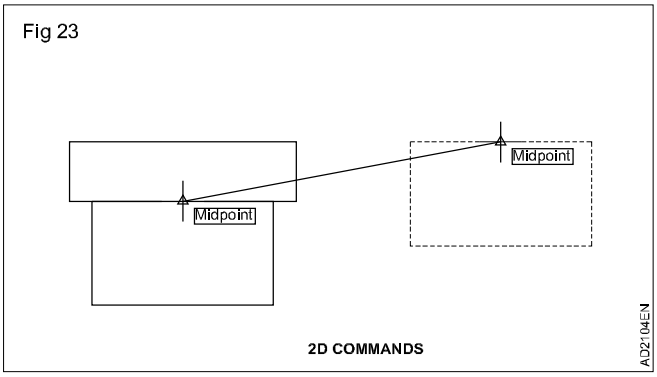
Fig 22



Second point: from

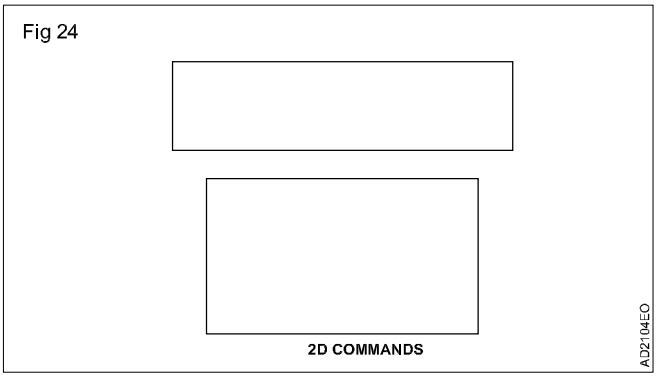
Base point: <Pick the bottom midpoint of the first rectangle> (Fig 22)

Fig 23



Offset: @2.5<270 (Fig 23)

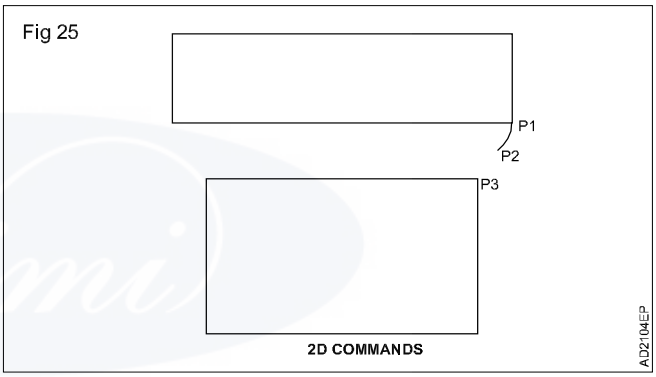
Fig 24



3 Use the ARC and MIRROR commands to connect the rectangles. (Fig 24)

Arc between P2 and P1

Fig 25



Command: ARC

Start Point: <Pick P2: Use 'Mid between two points' in object snap>

Second point [Centre/End]: E

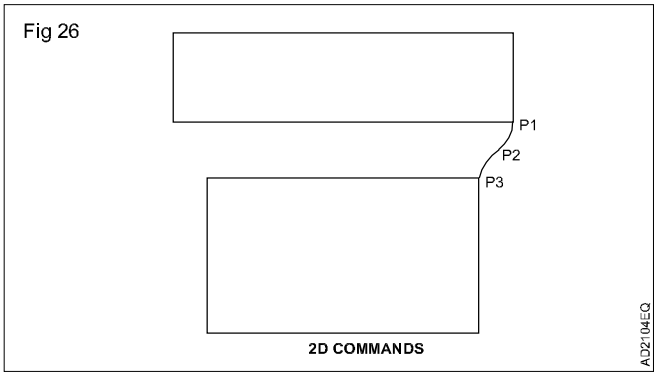
End Point: <Pick P1>

Centre Point [Angle/Direction/Radius]: R

Radius: 1.5

Create one more ARC as like as above which connects P2 and P3. (Fig 25)

Fig 26



MIRROR both the arcs to the opposite side.

Command: MIRROR

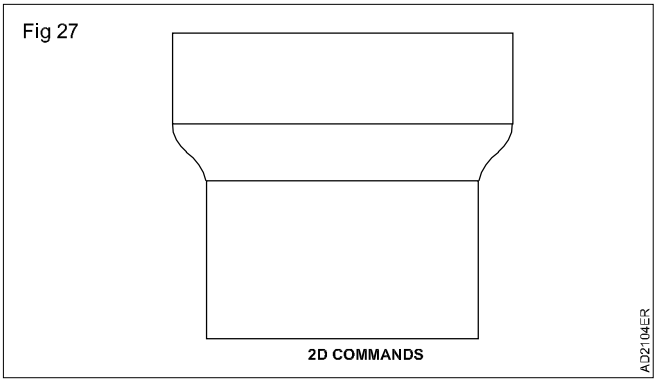
Select objects: <select both the arcs>

Select objects: <Null response>

First point of mirror line: <Pick the top middle point of bottom rectangle>

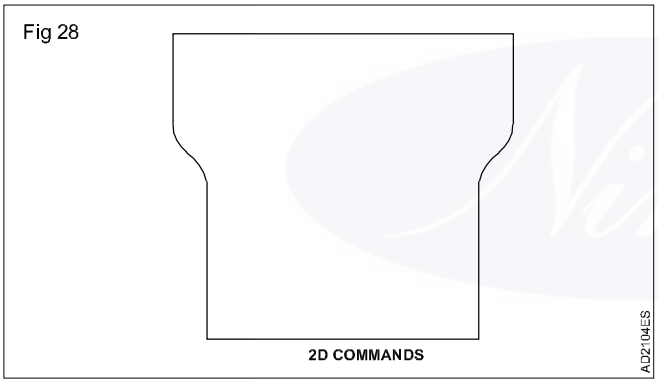
Second point of mirror line: <Pick the bottom middle point of the top rectangle> (Fig 26)

Fig 27



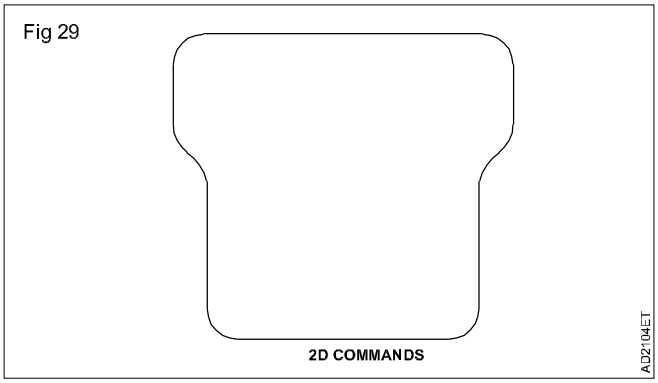
4 Trim the unwanted edges to get the following output.
Command: TRIM
Select cutting edges: <select all> (Fig 27)
Select object to trim: <select the unwanted edges>

Fig 28



5 Fillet the corners with radius 2. (Fig 28)

Fig 29



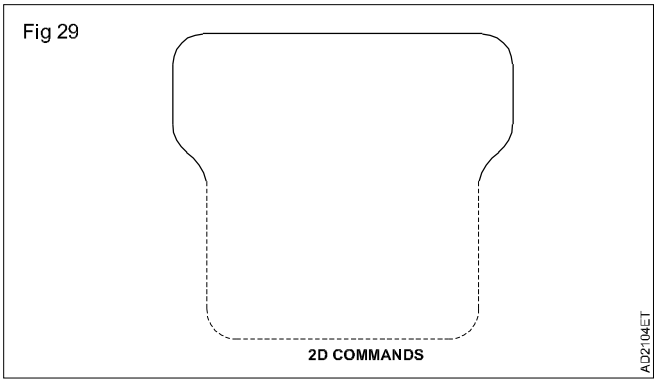
6 Create the back rest. (Fig 29)

Command: OFFSET

Offset distance: 1

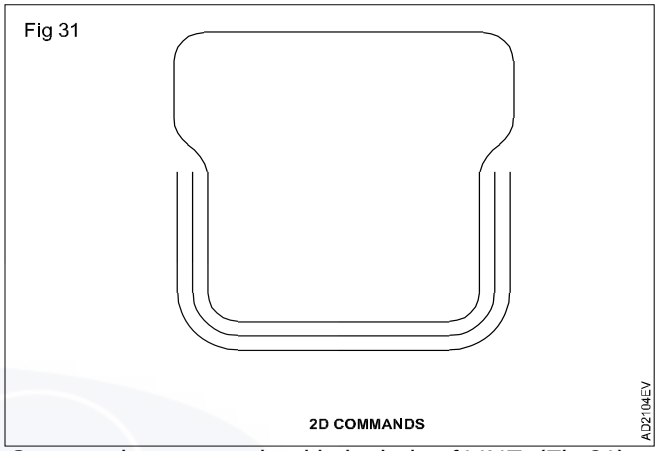
Object to offset " <select teh bottom rectangle of teh created seat >

Fig 29



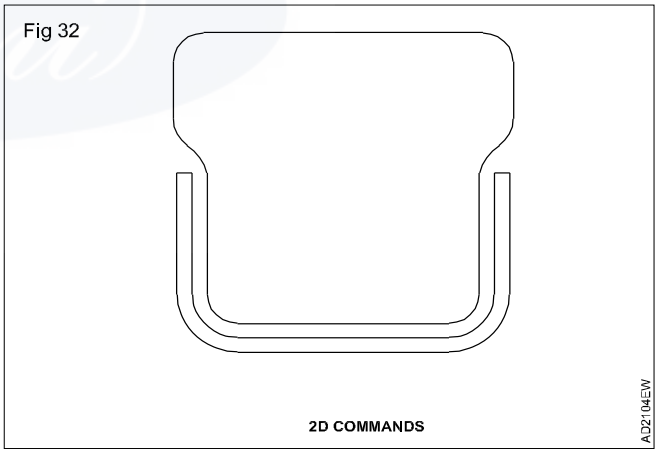
Side to offset: <pick in the outside of the seat>
Repeat the above two steps to create one more offset. (Fig 30)

Fig 31



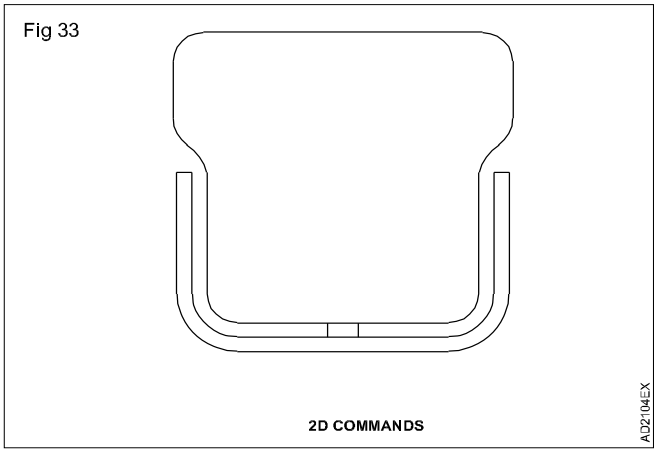
Connect the open ends with the help of LINE. (Fig 31)

Fig 32



Create the connection between the seat to the back rest with the help of LINE. (Fig 32)

Fig 33



Drafting of furniture (Door elevation -2)

Objectives: At the end of this exercise you shall be able to
- **draft the door elevation using basic CAD commands.**

PROCEDURE

TASK 1 :

Data : Size of the door

Height = 2100

Width = 100

1 Create a new metric file.

2 Draw a rectangle with the following dimensions.

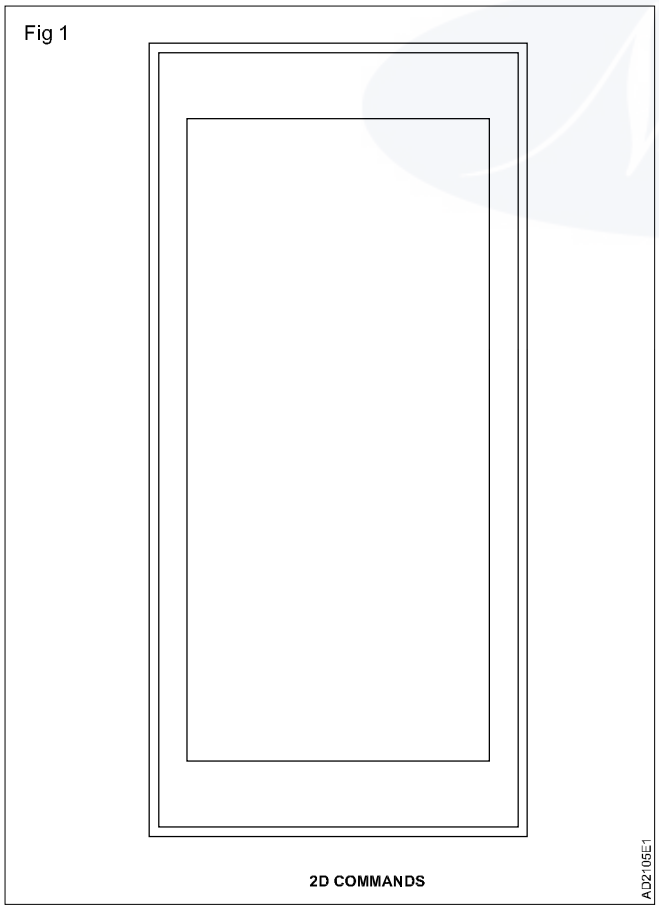
i. Length = 100

ii. Width = 210

3 Offset the rectangle towards the inside for 2.5 units.

4 Create another rectangle with the dimension of 80, 170 as shown in Fig 1.

Fig 1

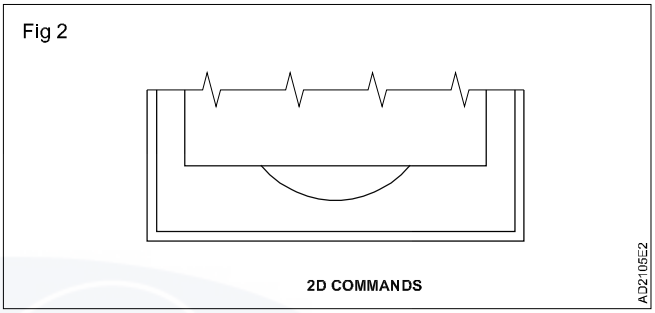


5 Create an ARC as follows with the help of start, End and Direction method. (Fig 2)

i. Chord length = 40 units

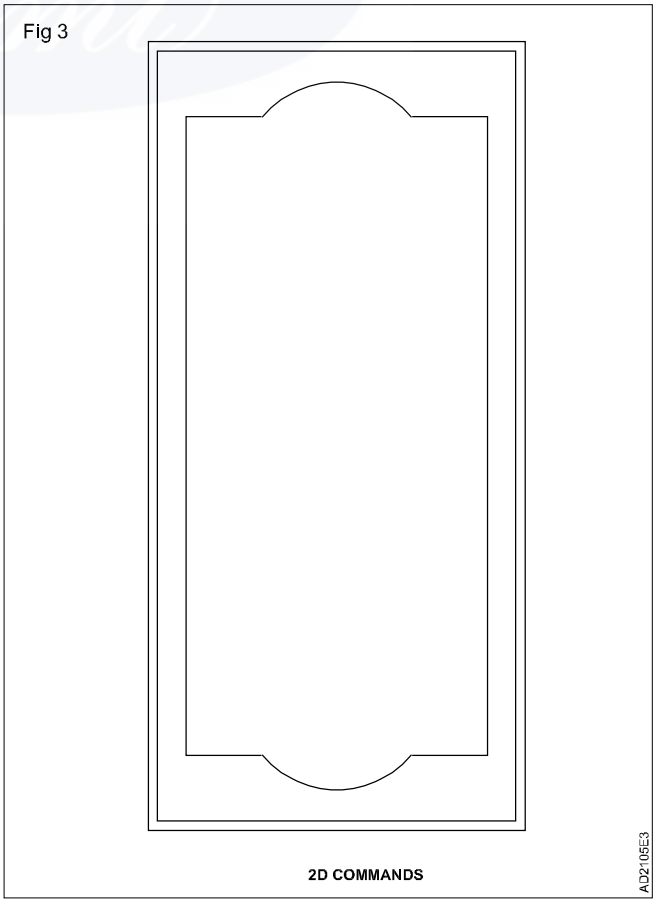
ii. Tangent angle = -60 deg

Fig 2



6 Mirror the arc to the opposite side and trim the unwanted portion to get the following result. (Fig 3)

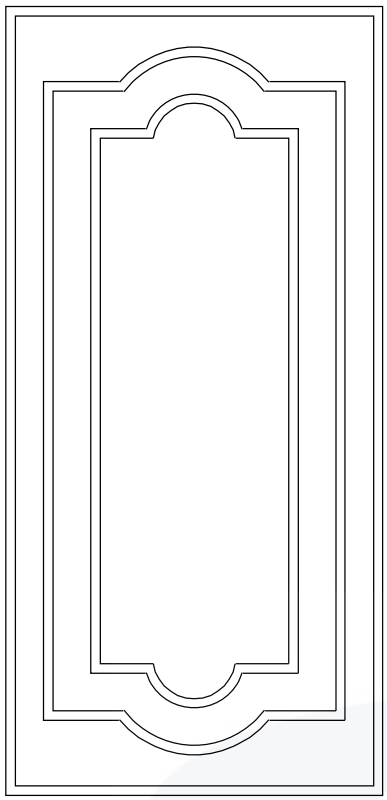
Fig 3



7 Convert the inner loop as a single element with the help of PEDIT.

8 Offset the inner loop to 2.5, 10 & 2.5 units. (Fig 4)

Fig 4

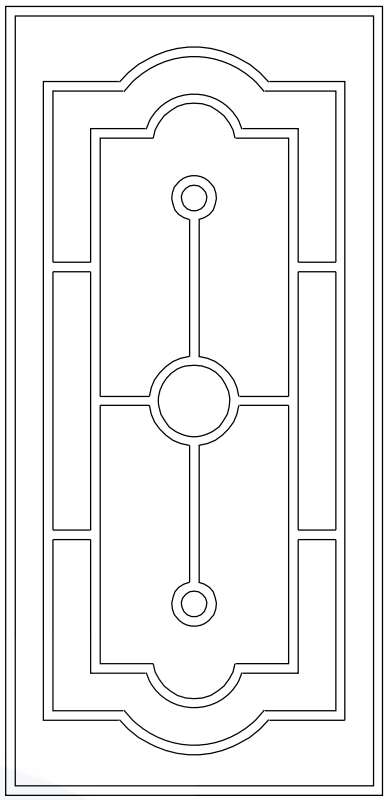


2D COMMANDS

AD2105E4

9 Complete the model as follows with the help of Circle, Line, Offset and Trim. (Fig 5)

Fig 5



2D COMMANDS

AD2105E5

Basic drafting commands - 3

Objectives: At the end of this exercise you shall be able to

- **understand following drafting commands**
  - **mine**
  - **polyline**
  - **spline**
  - **xline**
  - **ray**
  - **rev cloud**
  - **pan**
  - **redraw**
  - **reglx**
  - **colours**
  - **line type**
  - **line type manager**
  - **line weight**
  - **properties**
  - **match properties.**

PROCEDURE

TASK 1 : **Mline**

It creates multiple parallel lines.

Command alias	Button	Classic menu	Ribbon/Application menu
ML		Draw => Multiline	

The following prompts are displayed.

Specify start point or [] Justification/Scale/Style]:

Specify next point:

Specify next point or [Undo]:

Specify next point or [Close/Undo]:

Next point

Draws a multiline segment to the specified point using the current multiline style and continues to prompt for points.

Undo

Undoes the last vertex point on the multiline.

Close

Closes the multiline by joining the last segments with the first segments.

Justification

Determines how the multiline is drawn between the points you specify.

Top: Draws the multiple below the cursor, so that the line with the most positive offset is at the specified points.

Zero: Draws the multiline with its origin centered at the cursor, so that the MLSTYLE element Properties offset of 0.0 is at the specified points.

Bottom: Draws the multiline above the cursor, so that the line with the most negative offset is at the specified points.

Scale

Controls the overall width of the multiline. This scale does not affect line type scale. This scale factor is based on the width established in the multiline style definition. A scale factor of 2 produces a multiline twice as wide as the style definition. A negative scale factor flips the order of the offset line-the smallest on top when the multiline is drawn from left to right. A negative scale value also alters the scale by the absolute value. A scale factor of 0 collapses the multiline into a single line.

Style

Specifies a style to use for the multiline.

Style name

Specifies the name of a style that has already been loaded or that has been defined in a multiline library (MLN) file that you have created.

?-List styles

List the loaded multiline styles.

TASK 2 : **Polyline**

Command alias	Button	Classic menu	Ribbon/Application menu
PL		Draw => Polyline	Home => Draw => Polyline

The PLINE command allows you to create "grouped" objects that can be extruded, given line width on the screen, and can contain arc segments. A polyline is an object in AutoCAD that consists of one or more line (or arc) segments.

The following prompts are displayed.

Specify start point: Specify a point

Current line-width is <current>

Specify next point or [Arc/Close/Halfwidth/Length/Undo/ Width]: Specify a point or enter an option.

Next point

Draws a line segment. The previous prompt is repeated.

Arc

Adds arc segments to the polyline.

Specify endpoint or arc or

[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/ Secondpt/Undo/Width]: Specify a point (2) or enter an option

For the center option of the PLINE command, enter ce; for the center object snap, enter cen or center.

Endpoint of Arc

Draws an arc segment. The arc segment is tangent to the previous segment of the polyline. The previous prompt is repeated.

Angle

Specifies the included angle of the arc segment from the start point.

Specify included angle:

Entering a positive number creates counterclockwise arc segments. Entering a negative number creates clockwise arc segments.

Specify endpoint of arc or [Center/Radius]: specify a point or enter an option

Endpoint of Arc

Specifies the endpoint and draws the arc segment.

Center

Specifies the center of the arc segment.

Specify center point of arc:

Radius

Specifies the radius of the arc segment.

Specify radius of arc: specify a distance

Specify direction of chord for arc <current>: specify a point or press ENTER

Center

Specifies the center of the arc segment.

Specify center point or arc: specify a point

Specify endpoint of arc or [Angle/Length]:specify a point or enter an option

End point or Arc

Specifies the endpoint and draws the arc segment.

Angle

Specifies the included angle of the arc segment from the start point

Specify included angle:

Length

Specifies the chord length of the arc segment. If the previous is an arc, the new arc segment is drawn tangent to the previous arc segment.

Specify length of chord:

Close

Draws an arc segment from the last point specified to the starting point, creating a closed polyline. At least two points must be specified to use this option.

Direction

Specifies a starting direction for the arc segment.

Specify the tangent direction from the start point of arc: specify a point

Specify endpoint of arc: specify a point

Halfwidth

Specifies the width from the center of a wide polyline segment to one of its edges.

Specify starting half-width <current>: Enter a value or press ENTER

Specify ending half-width <starting width>: Enter a value or press ENTER

The starting half-width becomes the default ending half-width. The ending half-width becomes the uniform half-width for all subsequent segments until you change the half-width again. The starting and ending points of wide

line segments are at the center of the line.

Typically, the intersections of adjacent wide polyline segments are bevelled. No bevelling is performed for non-tangent are segments or very acute angles or when a dot-dash linetype is used.

Line

Exits the Arc option and returns to the initial PLINE command prompts.

Radius

Specifies the radius of the arc segment.

Specify radius of arc: specify a distance

Specify endpoint of arc or [Angle]: specify a point or enter a

Endpoint of Arc

Specifies the endpoint and draws the arc segment.

Angle

Specifies the included angle for the arc segment.

Specify included angle:

Specify direction of chord for arc <current>:Specify an angle or press ENTER

Second Pt

Specifies the second point and endpoint of a three-point arc.

Specify second point on arc: specify a point

Specify end point of arc: specify a point

Undo

Removes the most recent arc segment added to the polyline.

Width

Specifies the width of the next arc segment.

Specify starting width<current>: Enter a value or press ENTER

Specify ending width<starting width>: Enter a value or press ENTER

The starting width becomes the default ending width. The ending width becomes the uniform width for all subsequent segments until you change the width again. The starting and ending points of wide line segments are at the center of the line.

Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for nontangent arc segments, very acute angles, or when a dot-dash linetype is used.

Close

Draws a line segment from the last point specified to the starting point, creating a closed polyline. At least two points must be specified to use this option.

Halfwidth

specifies the width from the center of a wide polyline segment to one of its edges.

Specifify starting half-width <current>: Enter a value or press ENTER

Specify ending half-width <current>: Enter
a value or press ENTER

The starting half-width becomes the default ending half-width. The ending half-width becomes the uniform half-width for all subsequent segments until you change the half-width again. The starting and ending points of wide line segments are at the center of the line.

Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for non-tangent arc segments or very acute angles or when a dot-dash linetype is used.

Length

Draws a line segment of a specified length at the same angle as the previous segment. If theprevious segment is an arc, the new line segment is drawn tangent to that arc segment.

Specify length of line: Specify a distance

Undo

Removes the most recent line segment added to the polyline.

Width

Specifies the width of the next line segment.

Specify starting width <current>: Enter a value or press ENTER

Specify ending width <starting width>: Enter a value or press ENTER

The starting width becomes the default ending width. The ending width becomes the uniform width for all subsequent segments until you change the width again. The starting and ending points of wide line segments are at the center of the line.

Typically, the intersections of adjaceng wide polyline segments are beveled. No beveling is performed for non-tangent arc segments or very acute angles or when a dot-dash linetype is used.

TASK 3 : **Spline**

Command alias	Button	Classic menu	Ribbon/Application menu
SPL		Draw => Spline	Home => Draw => Spline

To create curves in AutoCAD you can use spline command. Spline can be drawn by specifying series of fit data poins through which the curve passes. The curves are quadratic or cubic curves. The curves can be more accurate than the spline fitted polylines because you control the tolerance to which the spline curve is fit. The prompts that display depend on whether you create a spline with fit points or with control vertices.

For splines created with the fit point method:

Specify first point or [Method/Degree/Object]:

For splines creted with the control vertices method:

Specifies first point or [Method/Knots/Object]:

First Point

Specifies the first point of the sline, either the first fit point or the first control vertex, depending on the current method.

Method

Controls whether the spline is created with fit points or with control vertices. (SPLMETHOD system variable)

Fit

Creates a degree 3 (cubic) B-spline by specifying fit points that the spline must pass through. When the tolerance value is greater than 0, the spline must be within the specified tolerance distance from each point.

Changing the Method updates the SPLMETHOD system variable.

Control vertices

Creates a spline by specifying control vertices. Use this method to create splines of degree 1 (linear), degree 2 (quadratic), degree 3 (Cubic), and so on up to degree 10. Adjusting the shape of a spline by moving control vertices often provides better results than moving the fit points.

This is the preferred method if you are creating geometry to be used with 3D NURBS surfaces.

Object

Converts 2D or 3D quadratic or cubic spline-fit polylines to equivalent splines. The originl polyline is retained or discarded depending on the setting of the DELOBJ system variable.

Next point

Creates additional spline segments until you press Enter.

Undo

Removes the last specified point.

Close

Close the spline by defining the last point to be coincident with the first. By default, closed splines are periodic, maintaining curvature continuity (C2) along the entire loop.

Options for splines with fit points

The following options are specific to the fit point method.

Knots

Specifies the knot parameterization, one of several computational methods that determines how the component curves between successive fit points within a spline are blended. (SPLKNOTS system variable)

1 **Chord or Chord-length method :** Spaces the knots connecting each component curve to be proportional to the distances between each associated pair of points. An example is the green curve inthe illustration.

2 **Square Root (or centripetal method):** This method spaces the knots connecting each component curve to be proportional to the square root of the distance between each associated pair of fit points. This method usually produces “gentler”curves. An example is the blue curve in the illustration.

3 **Uniform (or Equidistant method):** this method spaces the knots of each component curve to be equal, regardless of the spacing of the fit points. This method often produces curves that overshoot the fit points. An example is the magenta curve in the illustration.

Start targency

specifies a tangent condition on the starting point of the spline.

End tangency

Specifies a tangent conditon on the ending point of the spline.

Tolerance

Specifies the distance by which the spline is allowed to deviate from the specified fit points. A tolerance value of 0 requires the resulting spline to pass directly through the fit points. The tolerance value applies to all fit points except the starting and ending fit points, which always have a tolerance of 0.

Options for Splines with Control Vertices

The following option is specific to the control vertices (CV) method. (SPLMETHOD system variable)

Degree

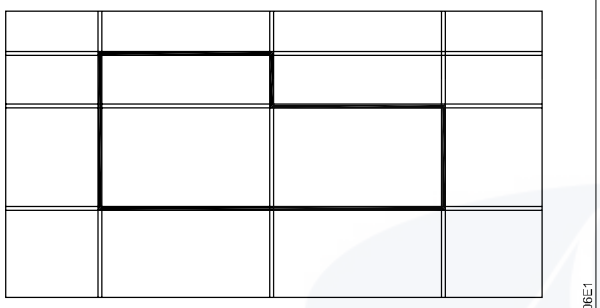
sets the polynomial degree of the resulting spline. Use this option to create splines of degree 1 (linear), degree 2 (quadiatic), degree 3 (Cubic), and so on up to degree 10.

TASK 4 : **Xline**

Command alias	Button	Classic menu	Ribbon/Application menu
XL		Draw => Xline	Home => Draw => Xline

Lines that extend to infinity in one or both directions, known as rays and construction lines, respectively, can be used as references for creating other objects. For example, you can use construction lines to find the center of a triangle, prepare multiple views of the same item, or create temporary intersections to use for object snaps. Infinite lines do not change the total area of the drawing. Therefore, their infinite dimensions have no effect on zooming or viewpoints. Hence they are ignored by commands that display the drawing extends. you can move, rotate, and copy infinite lines just as you can move, rotate, and copy other objects. You may want to create infinite lines on a construction line layer that can be frozen or turned off before plotting. (Fig 1)

Fig 1



The following prompts are displayed.

Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

Point

Specifies the location of the infinite line using two points through which it passes. The xline is created through the specified point.

Hor

It creates a horizontal x-line passing through a specified point. The xline is created parallel to the X axis.

Ver

It cretes a vertial X-line passing through a specified point. The xline is created parallel to the Y axis.

Ang

It creates an xline at a specified angle.

Angle of Xline

Specifies the angle at which to place the line.

Reference

Specifies the angle from a selected reference line. The angle is measured counterclockwise from the reference line.

Bisect

It creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line. The X-line lies in the plane determined by the three poins.

Offset

Creates an xline parallel to another object.

Offset distance

Specifies the distance the xline is offset from the selected object.

Through

Creates an xline offset from a line and passing through a specified point.

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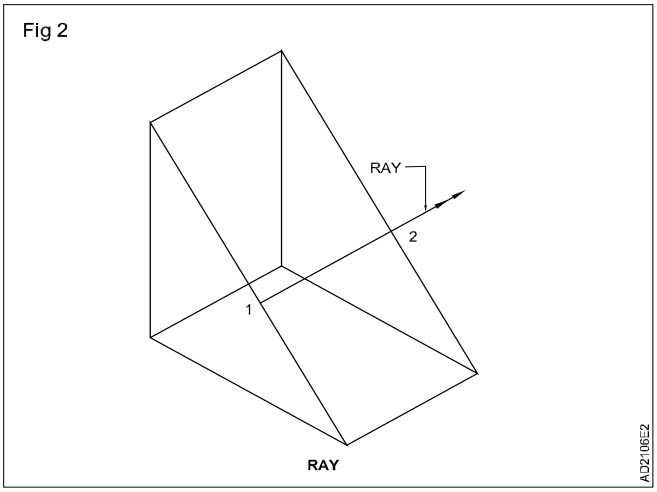
TASK 5 : **Ray**

Command alias	Button	Classic menu	Ribbon/Application menu
RAY		Draw => Xline	Home => Draw => Xline

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents. (Fig 2)

Fig 2



TASK 6 : **Pan**

Command alias	Button	Classic menu	Ribbon/Application menu
Z		View=> Pan	View => Navigate 2D => Pan

Often you cannot see the entire drawing on the screen. So you need a way to see that part of the drawing which is currently not visible. The pan means to move the display without changing the magnification.

The following prompts are displayed.

Press Esc or Enter to exit, or right-click to display a short-cut menu.

If you enter-pan at the command prompt, PAN displays command prompts, and you can specify a displacement to pan the drawing display. Position the cursor at the start location and press the mouse button down. Drag the cursor to the new location. You can also press the mouse scroll wheel or middle button down and drag the cursor to pan.

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TASK 7 : **Redraw**

Removes temporary graphics left by VSLIDE and some operations from the current viewpoint. To remove stray pixels, use the REGEN command.

— — — — — — — —

TASK 8 : **Regen**

Command alias	Classic menu
REGEN	View => Regen

By using this command all objects in the drawing are re-calculated and re-drawn in the current viewport. One of the advantages of this command is that the drawing is refined by smoothening circles and arcs out.

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TASK 9 : **Color**

Command alias	Button	Classic menu	Ribbon/Application menu
COLOR		Format => Color	Home => Properties => Object color

In AutoCAD the default color of object is white or dark grey depending on the background color of the view port. If the background color is black object will appear as white if light then AutoCAD automatically changes objects to dark (In drawing begun with acad3d.dwt the default color is pale greyblue). In case, You don't want to confine to black, white or pale greyblue, you can select from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

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TASK 10 : **Revcloud**

Command alias	Button	Classic menu	Ribbon/Application menu
REVCLOUD		Draw => Revision cloud	Home => Draw=> Revision cloud

This command creates a revision cloud using a polyline. This tool can be used to highlight the details of a drawing. It is customary in many industries to submit a set of drawings at different milestones of the project or stages of completion and then submit them again later with revisions-corrections, clarifications and requested changes. Usually the recipients like to locate changed/corrected stuff easily, and a common drafting convention is to call attention to revised items by drawing freedom clouds around them.

The following prompts are displayed

Minimum arc length : 0.5000 Maximum arc length : 0.5000

Specify start point or [Arc length/Object/Style]<Object>

Arc length

Specifies the length of the arcs in a revision cloud. The maximum arc length cannot be set to more than three times the minimum arc length.

Object

Specifies an object to be converted to a revision cloud.

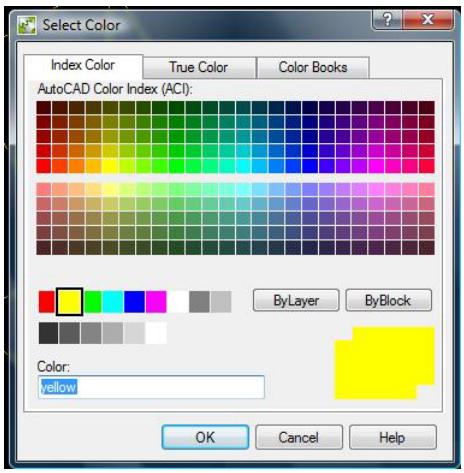
Style

Specifies the style of the revision cloud.

Index color tab

Specifies color settings using the 255 AutoCAD color Index (ACI) colors. (Fig 3)

Fig 3



The following options are displayed.

AutoCAD color index (ACI) palettes

Specifies a color from the AutoCAD color Index. If you hover over a color, the number of the color and its red, green,blue value are displayed below the palette. Click a color to select it, or enter the color number or name in the Color box.

The large palette displays colors 10 through 249. The second palette displays colors 1 through 9; these colors have names as well as numbers. The third palette displays colors 250 through 255; these colors are in shades of gray.

Index color

Indicates the ACI color number when you hover a color.

Red, Green, Blue

Indicates the RGB color value when you hover over a color.

By layer

Specifies that new objects assume the color assigned to the layer on which you create them. When BYLAYER is selected, the color of the current layer is displayed in the Old and New color swathces.

Byblock

Specifies that new objects use the default color (While or black, depending on your background color) until you group the objects into a block and insert the block. When you insert the block into a drawing, the objects in the block inherit the current Color setting.

The BYLAYER and BYBLOCK options do not apply to the LIGHT command.

Color

Specifies a color name, BYLAYER or BYBLOCK color or an AutoCAD Color Index (ACI) number of 1 through 255. The New color swatch shows the most recently selected color.

Old color swatch

Displays the previously selected color.

New color swatch

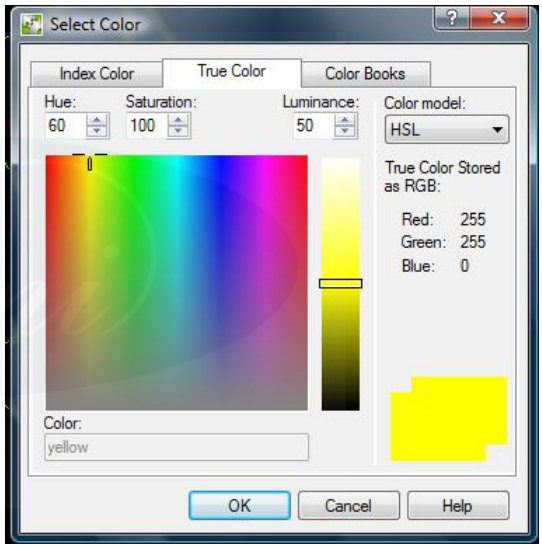
Displays the currently selected color.

True color tab

Specifies color settings using true colors (24-bit color) with the Hue, saturation, and Luminance (HSL) color model or the Red, Green, and Blue (RGB) color model. Over sixteen million colors are available when using true color functionality. The options available on the True Color tab are dependent on whether the HSL or RGB color model is specified.

The following options are displayed. (Fig 4)

Fig 4



Color book

This option specifies the color book to be used when selecting colors. The list consists of all the color books that are found in the Color Book Locations specified in the Options dialog box, Files tab. This Displays the pages of the selected color book and the colors and color names on each page. Color books containing up to ten colors per page are supported. If a color book is not paginated, the colors are organized into pages containing seven colors per page. To view color book pages, select an area on the color slider or use the up and down arrows to browse.

Color

Indicates the currently selected color book color. You can searach for a specific color in a color book by entering the number of the color swatch and pressing Tab. This action updates the new color swatch with the requested color number. if the specified color is not found in the color book, the closest number match is displayed.

Old color swatch

Displays the previously selected color.

New color swatch

Displays the currently selected color.

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TASK 11 : **Linetype**

Command alias	Button	Classic menu	Ribbon/Application menu
LT		Format => Line type	Home => Properties => Line type

A non -continuous line type is a repetition pattern of dots, dashes, and spaces. Line type can also include repeating texts or shapes. You may find that the line type you are using in your drawing are too long or short or requires clarity. The line type scale may even be so big or small that the line type looks continuous. How often the pattern is repeated is affected by three factors,

1) The line type definition
2) The global line type scale. Objects
3) The individual objects line type scale.

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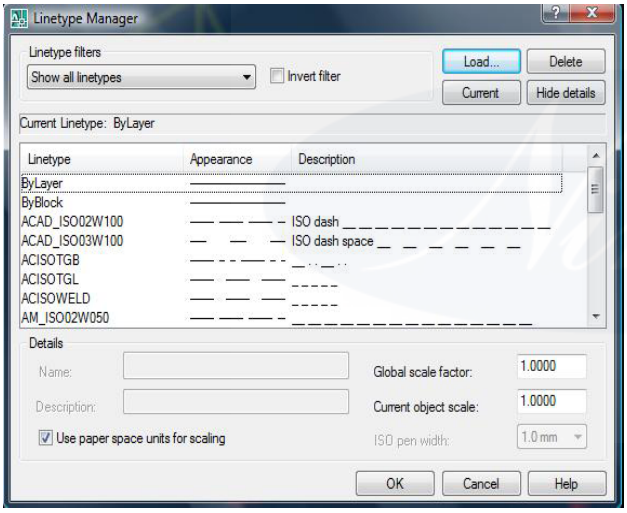
TASK 12 : **Linetype manager**

The following options are displayed.

Linetype filters (Fig 5)

Fig 5



Determines which linetype to display in the linetype list. You can filter linetype based on wherther they are x ref-dependent, or whether they are referred to by objects.

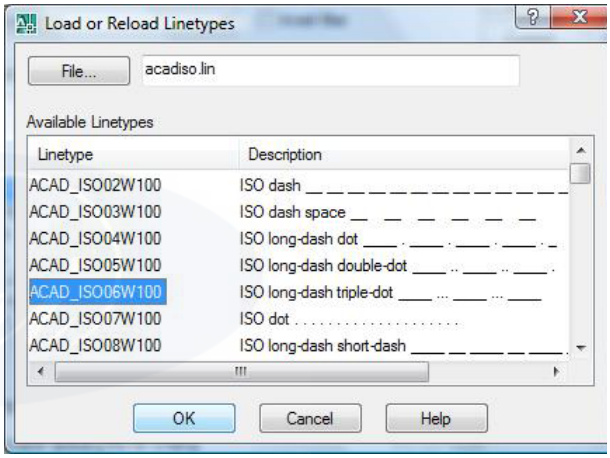
Invert filter

Displays linetypes based on the opposites of the criteria you select. Linetypes that fit the inverse filter criteria are displayed in the linetype list.

Load

Displays the Load or Reload Linetypes dialog box, in which you can load into selected linetypes the drawing from the acad.linfile and add them to the linetypelist. It also loads a linetype whose definition exists in a linetype library (LIN) file. (Fig 6)

Fig 6



The following options are displayed.

File button

Displays the select Linetype File dialog box, in which you can select a different linetype (LIN) file.

File name

Displays the name of the current LIN file. You can enter the name of another LIN file or click the File button to select a file from the select Linetype File dialog box.

Available linetypes

Displays the linetypes available to load. To select or clear all of the linetypes on the list, right-click and choose select All or Clear All.

Current

Sets the selected linetype to be the current linetype. Setting the current linetype to BYLAYER means that an object assumes the linetype that is assigned to a

particular layer. Setting the linetype to BYBLOCK means that an object assumes the CONTINUOUS linetype until it is grouped into a block. Whenever the block is inserted, all objects inherit the block's linetype. The CELTYPE system variable stores the linetype name.

Delete

Deletes selected linetypes from the drawing. You can delete only unused linetypes. The BYLAYER, BYBLOCK, and CONTINUOUS linetypes cannot be deleted.

Show details or hide details

Controls whether the Details section of the Line type Manager is displayed.

Current linetype

Displays the current linetype name.

List of linetypes

Displays the loaded linetypes according to the option specified in Linetype Filters. To quickly select all or clear all linetypes, right-click in the linetype list for the shortcut menu to be displayed.

Linetype

Displays names of loaded linetypes. To rename a linetype, select it and then click it again and enter a new name. BYLAYER, BYBLOCK, CONTINUOUS, and xref-dependent linetypes cannot be renamed.

Appearance

Displays a sample of selected linetypes.

Description

Displays descriptions of the linetypes, which can be edited in the Details area.

Details

Provides alternative access to properties and additional settings.

Name

Displays the selected linetype name, which can be edited.

Use paper space units for scaling

Scales linetypes in paper space and model space identifically. Useful when working with multiple viewports. (PSLTSCALE system variable)

Global scale factor

Displays the global scale factor for all linetypes. (LTSCALE system variable)

Current object scale

Sets linetype scale for newly created objects. The resulting scale is the global scale factor multiplied by the object's scale factor. (CELTSCALE system variable)

ISO pen width

Sets the linetype scale to one of a list of standard ISO values. The resulting scale is the global scale factor multiplied by the object's scale factor.

TASK 13 : **Lineweight**

Command alias	Button	Classic menu	Ribbon/Application menu
LW		Format => Line weight	Home => Properties => Line weight

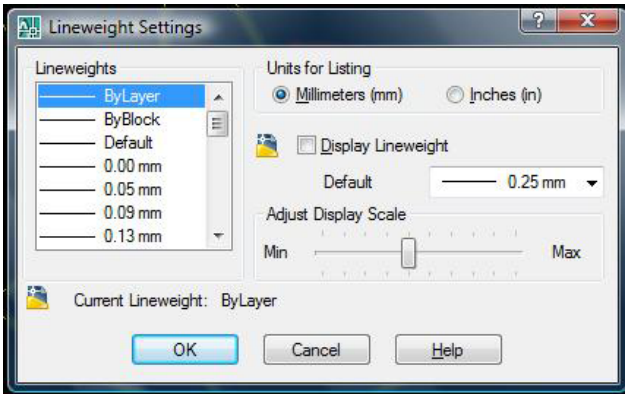
Line weight let you represent objects with varying line width. The width can represent teh width of a pen. Lineweight is also used to distinguish certain type of objects just as colors and line types do. Finally you can use line weight to represent original property of an object such as the width of wire in electrical schematics.

The following options are displayed.

Line weights

It displays the available lineweight values. Lineweight values consist of standard settings including BYLAYER, BYBLOCK, and DEFAULT. The DEFAULT value is set by the LWDEFAULT system variable, which has an initial value of 0.01 inches or 0.25 mm. All new layers use the default setting. The lineweight value of 0 plots at the thinnest lineweight is available on a specified plotting device and is displayed at one pixel wide in model space. (Fig 7)

Fig 7



Current lineweight

Displays the current lineweight. To set the current lineweight, select a lineweight from the lineweight list and choose OK.

Display lineweight

Controls whether lineweight are displayed in the current drawing. If this option is selected, lineweights are displayed in model space and paper space. You can also set Display Lineweight by using the LWDISPLAY system variable. Regeneration time increases with lineweights that are represented by more than one pixel. Clear Display Lineweight if performance slows down when working with lineweights turned on in a drawing. This option does not affect how objects are plotted.

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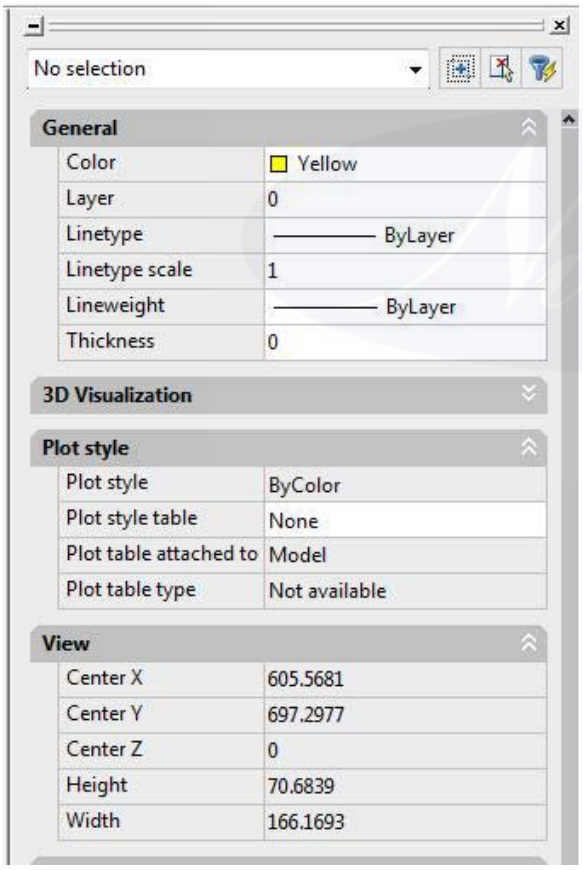
TASK 14 : **Properties**

Command alias	Button	Classic menu	Ribbon/Application menu
PR		Modify => Properties	View => Pallettes => Properties

As mentioned earlier each object has properties associated with color, Line type, Line weight and so on. You can modify these properties by using properties pallette. It displays the properties of the selected object or a set of objects. (Fig 8)

Fig 8



The following options are displayed.

Object type

Displays the type of object that is selected.

Select objects

Selects desired objects using any selection method. The properties common to the selected objects are displayed in the properties palette. You can then modify the properties of the selected objects in the properties palette, or you can make other changes to the selected objects by entering an editing command.

Quick select

Displays the Quick Select dialog box. Use Quick Select to create selection sets based on filtering criteria.

Shortcut menu

The following shortcut menu options are availabel when you right-click the title bar.

Move

Displays a four-headed arrow cursor that you can use to move the palette. The palette does not dock.

Size

Displays a four-headed arrow cursor that you can use to drag an edge or a corner to make the palette smaller or larger.

Close

Closes the properties palette.

Auto-hide

Causes a floating palette to roll open and closed as the cursor moves across it. When this option is cleared, the palette stays open.

Transparency

Displays the Transparency dialog box.

Line weight

Sets the lineweight to be used for borders that are displayed. If you use a heavy lineweight, you may have to change the cell margins.

Linetype

Sets the linetype to be used for borders that are displayed.

Color

Sets the color to be used for borders that are displayed.

Double line

When checked, a double line border will be added to the selected cells.

Spacing

Determining the spacing of double-line borders. The default value is .1800.

Note Line type will not display in the preview.

All borders

Applies the border properties settings to all borders of the selected table cells.

Outside borders

Applies the Border properties settings to the outside borders of the selected table cells.

Inside borders

Applies the border properties settings to the inside borders of the selected table cells.

No borders

Applies the border properties settings to none of the borders of the selected table cells.

Top borders

Applies the border properties setting to the top borders of the selected table cells.

Inside horizontal border

Applies the border Properties setting to the inside horizontal borders of the seleted table cells.

Bottom border

Applies the border proerties setting to the bottom borders of the selected table cells.

Left border

Applies the border properties setting to the left borders of the selected tabel cells.

Inside vertical border

Applies the border properties setting to the inside vertical border of the selected table cells.

Right border

Applies the border properties setting to the right border of the selected table cells.

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TASK 15 : **Matchprop**

Command alias	Button	Classic menu	Ribbon/Application menu
MA		Modify => Match Properties	Home => Clipboard => Match Properties

The match properties tool is used to apply properties like color, line type, layer and line type scale of a source object to selected objects. The properties of the destination object will be changes as the properties of the source object. This is a transparent tool and can be used when another tool is active.

The following prompts are displayed.

Current active settings: Currently selected matchprop settings

Select destinaion Object(s) or [Settings]: Enter s or select one or more objects to copy properties to

Destination object(s)

Specifies the objects to which you want to copy the properties of the source object.

Settings

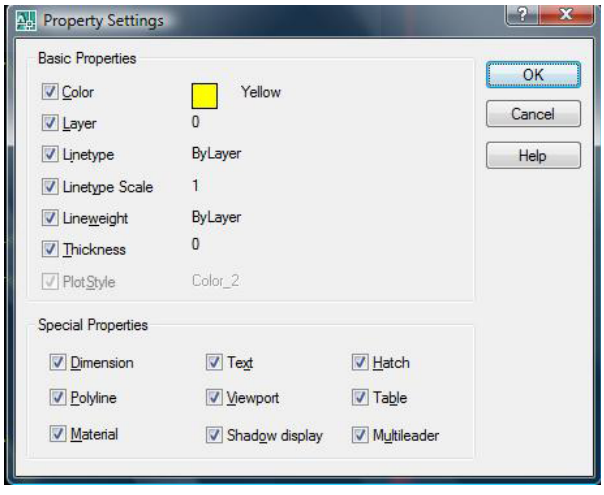
Displays the Property Settings dialog box, in which you can control which object properties to copy to the destination objects. By default, all object properties are selected for copying.

Property settings

It specifies the properties that are copied from the select source object to be destination objects.

Layer

Fig 9



The following options are displayed.

Basic properties

Color

Changes the color of the destination object to that of the source object. Available for all objects.

Changes the layer of the destination object to that of the source object. Available for all objects.

Linetype

Changes the linetye of the destination object to that of the source object. Available for all objects except attributes, hatches, multiline text, points, and viewports.

Lineweight

Changes the lineweight of the destination object to that of the source object. Available for all objects.

Transparency

Shifts are transparency of the destination object to the source object. Available for all objects.

Thickness

Shifts the thickness of the destination object to the source object. Available only for arcs, attributes, Circles, lines, points, 2D polylines, regions, text, and traces.

Plot style

Shifts the plot style of the destination object to the source object. If you are working in color-dependent plot style mode (PSTYLEPOLICY is set to1), this option is unavailable. Available for all objects, except those with the jitter edge modifier applied.

Special properties

Dimension

In addition to basic object properties, this property when availed changes the dimension style and annotative properties from the destination object to the source object. Available only for dimension, leader, and tolerance objects.

Draw bedroom interiors with help of CAD

Objectives: At the end of this exercise you shall be able to
- **draw the layout design with help of CAD.**

PROCEDURE

TASK 1

1 Draw bed room size of 6400 x 6200 with help of line comand,offset, hatch. dimension in different layers.

2 Draw ward robe with sliding shutters of 600mm width at the entry of toilet with line command. (Fig 1)

— — — — — — — —

TASK 2 :

Draw study table, chain with help of rectangle, design centre. (Fig 1)

— — — — — — — —

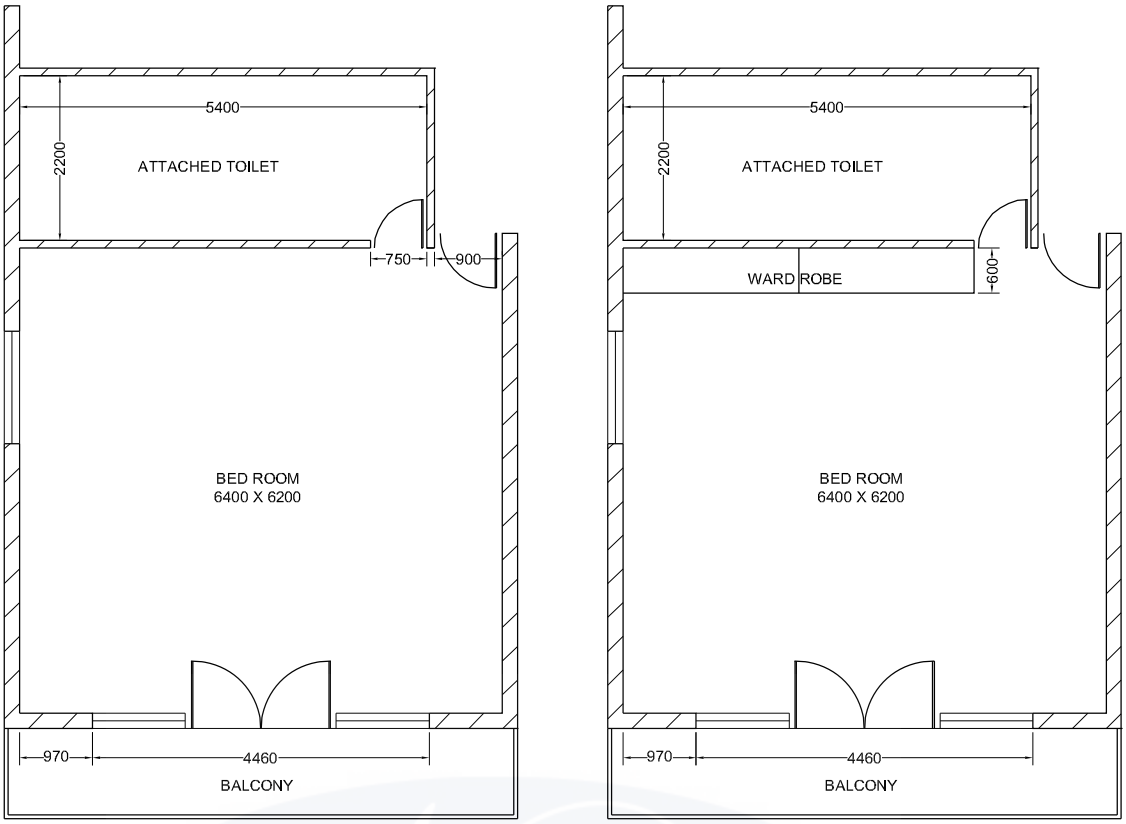
TASK 3 :

1 Draw cot, side table, carpet in specified dimensions with help of rectangle, hatch commands.

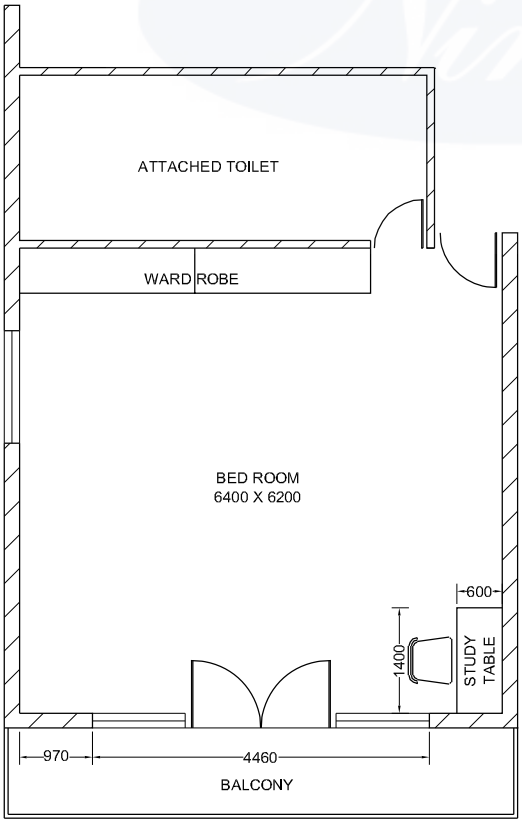
2 Mention dimension of moving spaces. (Fig 1 & Fig 2)

— — — — — — — —

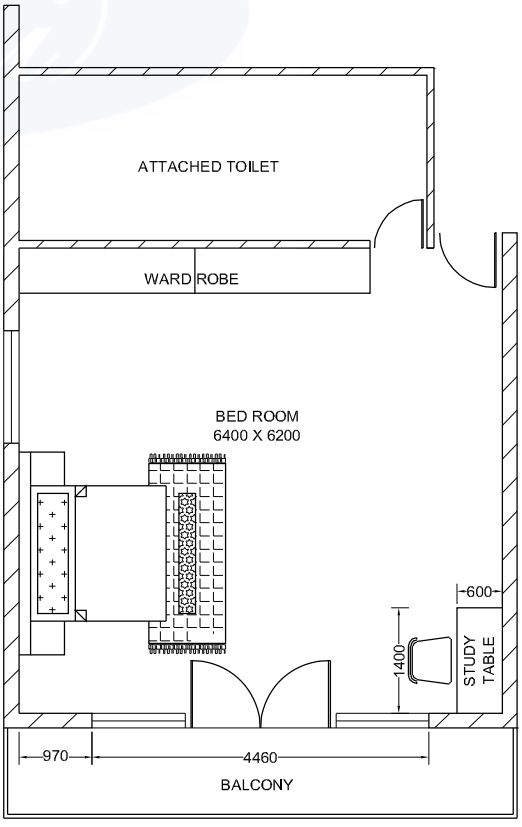
Fig 1



TASK 1



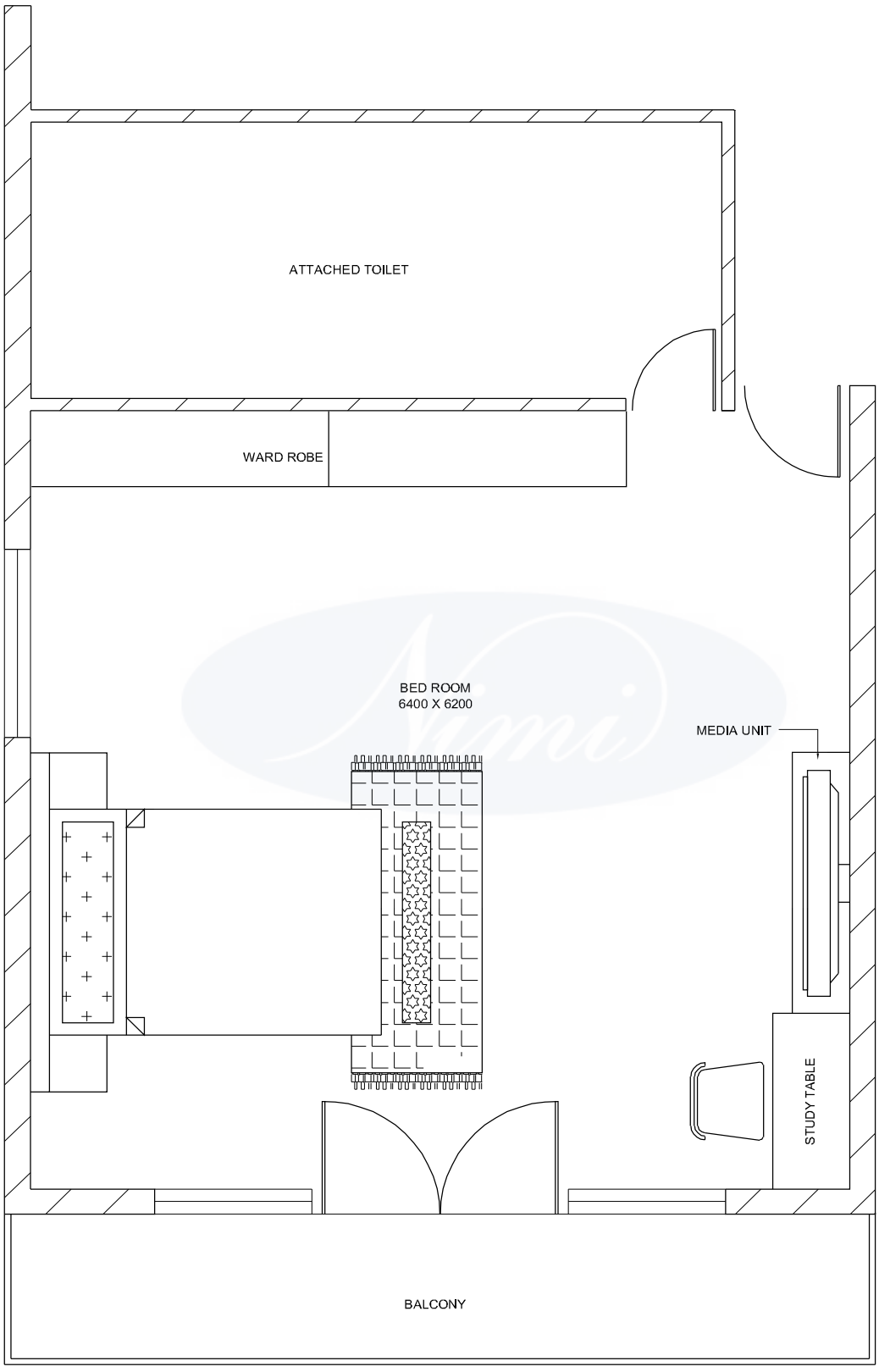
TASK 2



TASK 3

AD2107E1

Fig 2



TASK 3

INTERIOR

AD2107E2

Drafting settings in CAD applications

Objectives: At the end of this exercise you shall be able to
- **do the editing/changes in the settings according to the requirement.**

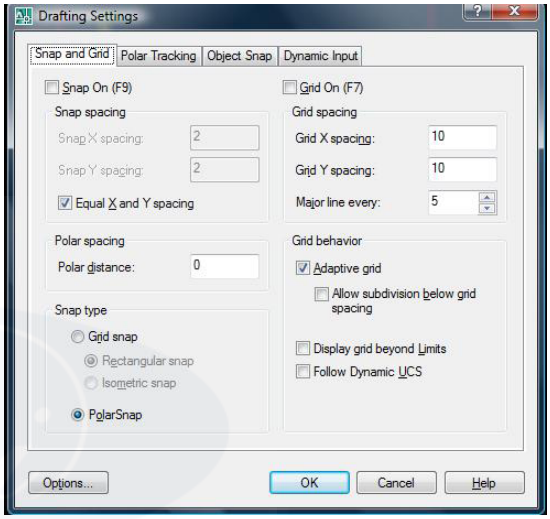
PROCEDURE

TASK 1 : **Data**

Command alias	Classic menu	Ribbon/Application menu
DS	Tools => Drafting settings	Command entry

Drafting settings contains set of traditional drafting tools, that help draw easily. (Fig 1)

Fig 1

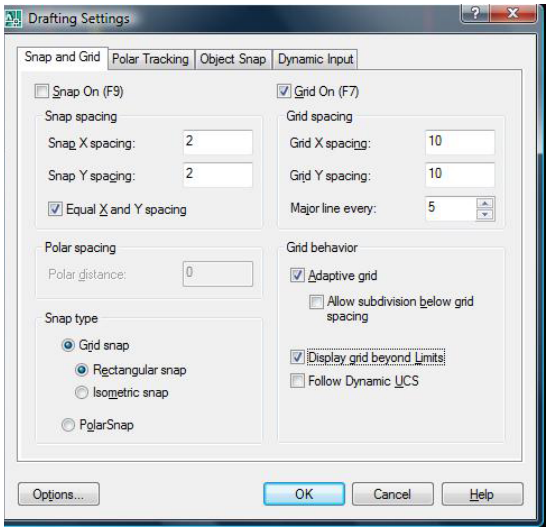


Grind and snap

Function key	Classic menu	Ribbon/Application menu
Grid F7, Snap F9	Status bar	Status bar

To enhance drawing speed and efficiency, you can display and snap the design to a rectangular grid. You can also control its spacing, angle, and alignment. (Fig 2)

Fig 2



Snap on

Turns snap mode on or off. You can also turn snap mode on or off by clicking snap on the statur bar pressing F9, or by using the SNAPMODE system variable.

Snap spacing

Controls an invisible, rectangular grid of snap locations that restricts cursor movement to specified XY intervals.

Snap X spacing

Specifies the snap spacing in the X direction. The value must be a positive real number (SNAPUNIT system variable).

Equal X and Y spacing

Forces the X and Y spacing to the same values for snap spacing and/or grid spacing.

Polar spacing

Controls the polar snap increment distance.

Polar distance

Sets the snap increment distance when polar snap is selected under snap type & style. If this value is 0polar snap distance assuemes the value for snap X spacing. The polar distance setting is used in conjunction with polar tracking and/or object snap tracking. If neither of the tracking feature is enabled, the I distance setting has no effect(*POLARDIST* system variable)

Snap type

Sets the snap style and snap type.

Grid snap

Sets the snap type to grid. When you specify points, the cursor snaps along vertical or horizontal points. (SNAPTYPE system variable).

Block editor

Sets the grid style to dotted grid for the block editor(GRIDSTYLE system variable).

Sheet/Layout

Sets the grid style to dotted grid for sheet and layout (GRIDSTYLE system variable).

Grid spacing

Controls the display of a grid that heps you visualize the distances.

The limits of the grid are controlled by the LIMITS command and the GRID DISPLAY system variable.

Grid X spacing

Specifies the grid spacing in the X driection. If this value is 0, the grid assumes the value set for snap X spacing(GRIDUNIT system variable).

Grid Y spacing

Specifies the grid spacing the Y direction. If this value is 0, the grid assumes the value set for snap Y spacing.(GRIDUNIT system variable)

Major line every

Specifies the frequency of major grid lines relative to minor grid lines. Grid lines rather than grid dots are displayed when GRIDSTYLE is set to 0 (GRIDMAJOR system variable).

Grid behaviour

Controls the appearance of the grid lines that are displayed when GRIDSTYLE is set to 0.

Adaptive grid

Limits the density of the grid when zoomed out (GRIDDISPLAY system variable).

Allow subdivision below grid spacing

Generates additional, more closely spaced grid lines when zoomed in. The frequency of these grid lines is determined by the frequency of the major grid lines (GRIDDISPLAY and GRIDMAJOR system variables).

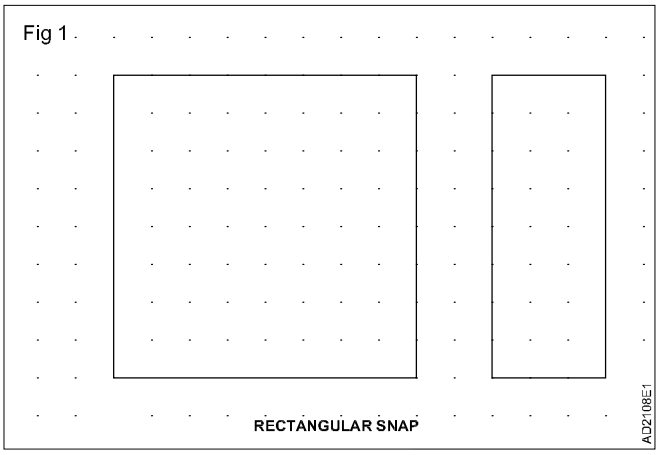
Display grid beyond limits

Display the grid beyond the area specified by the LIMITS command (GRIDDISPLAY system variable).

Rectangular snap

Sets the snap style to standard rectangular snap mode. When the snap type is set to grid snap and the snap mode is on, the cursor snaps to a rectangular snap grid. (Fig 3)

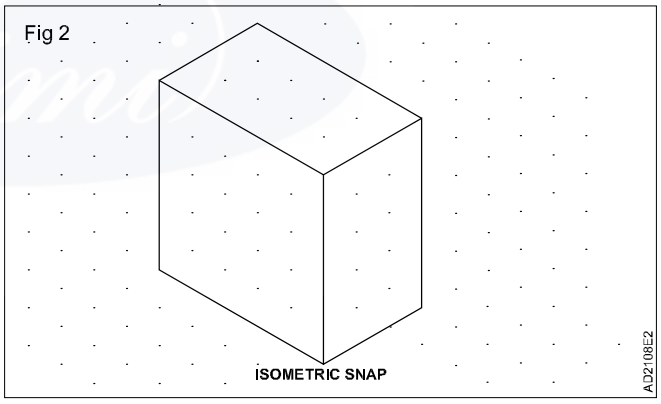
Fig 1



Isometric snap

Sets the snap style to isometric snap mode. When the snap type is set to grid snap and snap mode is on, the cursor snaps to an isometric snap grid. (Fig 4)

Fig 2



Polar snap

Sets the snap type to polar. When snap mode is on and you specify points with polar tracking turned on, the cursor snaps along polar alignment angles set on the polar tracking tab relative to the starting polar tracking point.

Grid on

Turns the grid on or off. You can also turn grid mode on or off by clicking grid on the status bar, by pressing F7.

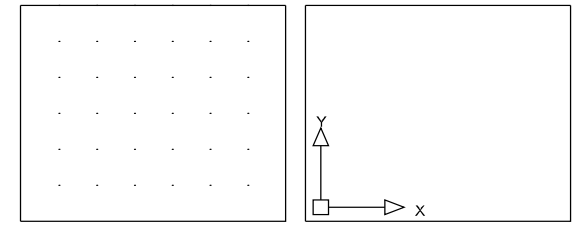
Grid style

Sets the grid style in 2D contexts. You can also set grid style.

2D model space

Sets the grid style to dotted grid for 2D model space. (Fig 5)

Fig 3



GRID ON AND OFF IN 2D MODEL SPACE

AD2108E3

Follow dynamic UCS

Changes the grid plane to follow the XY plane of the dynamic UCS(GRIDDISPLAY system variable).

Ortho

Function key	Classic menu	Ribbon /Application menu
F8	Status bar	Status bar

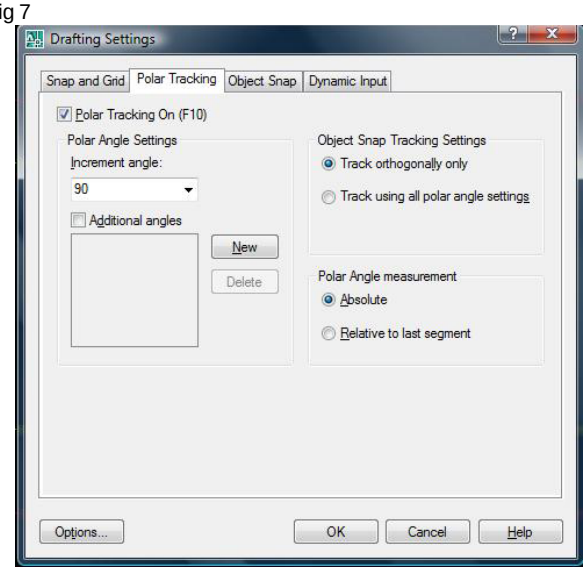
Orthos is short for orthogonal, which means either vertical or horizontal. It is a drawing mode which can either be turned on or off. When ortho is turned on, the ortho button appears pressed in. You can see how this appears by looking at the status bar illustration below. In the illustration below. In the illustration, ortho is turned on. (Fig 6)

Fig 6

Polar tracking

Function key	Classic menu	Ribbon /Application menu
Polar tracking : F10 Otrack : F11	Status bar	Status bar

Fig 7



Polar tracking restricts cursor movement to specified angles. Polar Snap restricts cursor movement to specified increments along a polar angle. (Fig 7)

Polar tracking on

Turns polar tracking on and off. You can also turn polar tracking on or off by using the AUTOSNAP system variable.

Polar angle settings

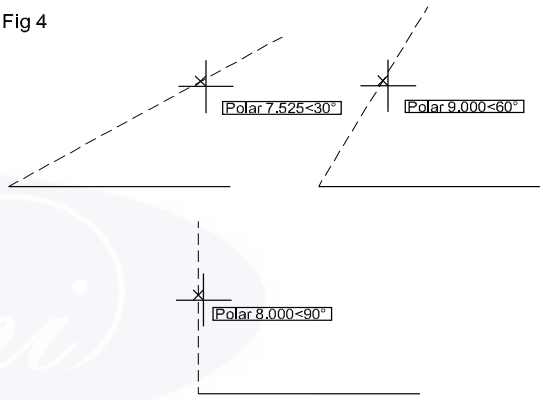
Set the alignment angles for polar tracking (POLARANG system variable).

Increment angle

Sets the polar increment angle used to display polar tracking alignment paths. You can enter any angle, or select a common angle of 90,45,30,22.5,18,15,10 or 5 degrees from the list (POLARANG system variable).

For example, if you set an increment angle of 30, it will show tracking vector in all multiple angles of 30 as shown in Fig 8.

Fig 4



POLAR TRACKING IN 30, 60 AND 90 DEGREES

AD2108E4

Additional angles

Makes any additional angles in the list available for polar tracking. The additional angles check box is also controlled by the POLARMODE system variable, and the list of additional angles is also controlled by the POLARADDANG system variable.

Additional angles are absolute, not incremental.

List of angles

If Additional Angles is selected, lists the additional angles that are available. To add new angles, click New. To remove existing angles, click Delete. (POLARADDANG system variable)

New

Displays the Add New Angle dialog box. Up to 10 additional polar tracking alignment angles can be added.

Before adding fractional angles, you must set the AUPREC system variable to the appropriatedecimal precision to avoid undesired rounding. For example, if the value of AUPREC is 0 (the default value), all fractional angles you enter are rounded to the nearest whole number.

Delete

Deletes selected additional angles.

Track orthogonally only

Displays only orthogonal (horizontal/vertical) object snap tracking paths for acquired object snap points when object snap tracking is on. (POLARMODE system variable)

Track using all polar angle settings

Applies polar tracking settings to object snap tracking. When you use object snap tracking, the cursor tracks along polar alignment angles from acquired object snap points. (POLARMODE system variable)

Polar angle measurement

Sets the basis by which polar tracking alignment angles are measured.

Absolute

Bases polar tracking angles on the current user coordinate system (UCS).

Relative to last segment

Bases polar tracking angles on the last segment drawn.

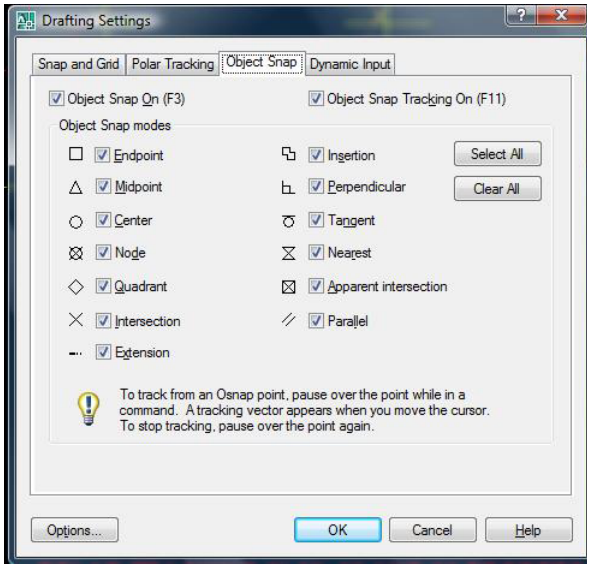
Object snap and object snap tracking

Function key	Classic menu	Ribbon /Application menu
F3	Status bar	Status bar

The object snaps (Osnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately. Osnaps allow you to *snap* onto a specific object location when you are picking a point. Press TAB to cycle through the options. (Fig 9)

The following options are displayed;

Fig 9



Object snap on

Turns running object snaps on and off. The object snaps selected under object snap modes are active while object snap is on (OSMODE system variable)

Object snap tracking on

Turns object snap tracking on and off. With object snap tracking, the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps (AUTOSNAP system variable).

Object snap modes

Lists object snaps that you can turn on as running object snaps.

Object snap on

Turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. (OSMODE system variable)

Object snap tracking on

Turns object snap tracking on and off. With object snap tracking, the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps. (AUTOSNAP system variable)

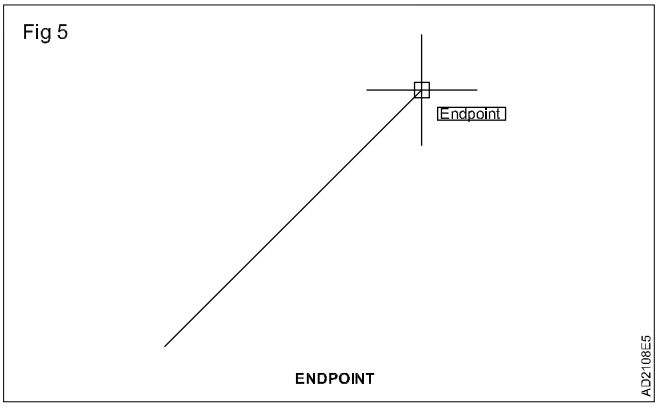
Object snap modes

Lists object snaps that you can turn on as running object snaps.

Endpoint

Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray, or to the closest corner of a trace, solid, or 3D face. (Fig 10)

Fig 5



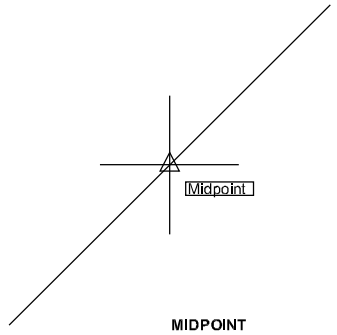
Midpoint

Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, region, solid, spline or xline. (Fig 11)

Center

Snaps to the center of an arc, circle, ellipse, or elliptical arc. The cursor must pass over the circumference of the circle or the arc so that the center can be found. (Fig 12)

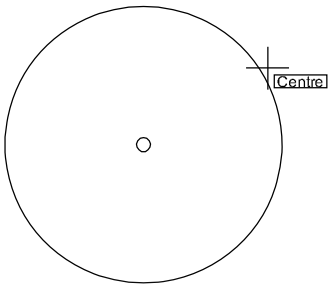
Fig 6



MIDPOINT

AD2108E6

Fig 7



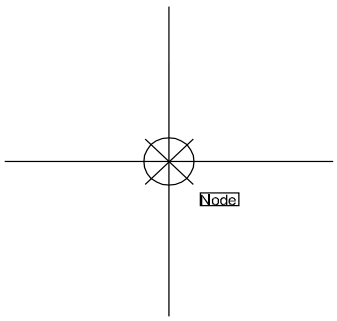
CENTRE

AD2108E7

Node

Snaps to a point object, dimension definition point, or dimension text origin. The node osnap snaps to the center of a point object. This osnap can be useful if you have created a number of points with the measure or divide commands. You could, for example insert a number of regularly spaced tree symbols(blocks) along a line using the node osnap for the insertion point of each block. (Fig 13)

Fig 8



NODE

AD2108E8

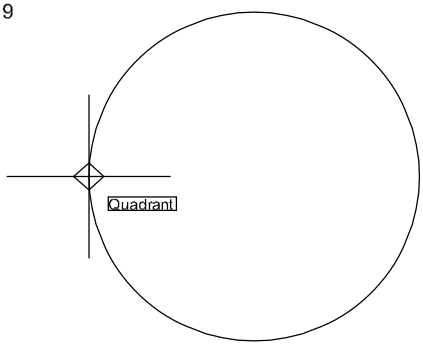
Quadrant

The quadrant osnap snaps to one of the four circle quadrant points located at north, south, east and west or 90,270,0 and 180 degrees respectively. (Fig 14)

Intersection

The intersection osnap snaps to the physical intersection of any two drawing objects (ie. where lines, arcs or circles etc. cross each other) and to polyline vertices. However, this osnap can also be used to snap to intersection points which do not physically exist. This feature is called the extended. (Fig 15)

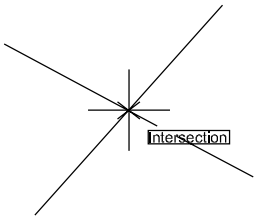
Fig 9



QUADRANT

AD2108E9

Fig 10



INTERSECTION & EXTENDED INTERSECTION

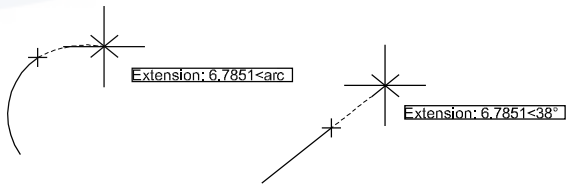
AD2108EA

You might get varying results if you have both the intersection and apparent intersection running object snaps turned on at the same time. Intersection and extended intersection do not work with edges or corners of 3D solids.

Extension

Causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension. (Fig 16)

Fig 11



EXTENSION IN LINE AND ARC

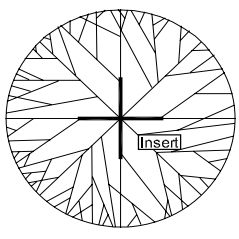
AD2108EB

When working in perspective view, you cannot track along the extension line of an arc or elliptical arc.

Insertion

Snaps to the insertion point of an attribute, a block, a shape, or text. (Fig 17)

Fig 12



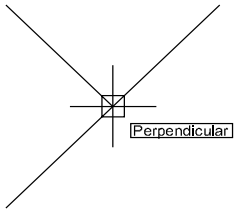
INSERTION

AD2108EC

Perpendicular

Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, solid, spline, or xline. As with the tangent osnap, perpendicular can be used to draw a line to a perpendicular point (as in the illustration), or from a perpendicular point, known as the “Deferred perpendicular” snap mode. (Fig 18)

Fig 13



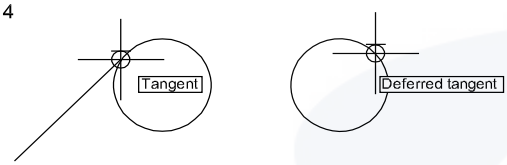
PERPENDICULAR

AD2108ED

Tangent

Snaps to the tangent of an arc, circle, ellipse, elliptical arc, or spline. This osnap works in two ways. You can either draw a line from a point to the tangent point(See illustration) or you can draw a line from a tangent point, the latter is referred to as the “Deferred tangent” snap mode.(Fig 19)

Fig 14



TANGENT AND DEFERRED TANGENT

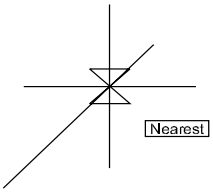
AD2108EE

When you use the from option in conjunction with the tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.

Nearest

Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, ray, spline, or x-line. Osnap is useful in making sure that a pick point lies on a drawing object but you don;t necessarily mind exactly where it is located. (Fig 20)

Fig 15



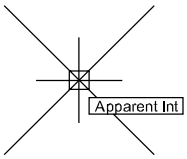
NEAREST

AD2108EF

Apparent intersection

Apparent intersection snaps to the point where objects appear to intersect in the current view. For example you may be looking at a drawing in plan view where two lines cross, as in the illustration. However, since AutoCAD is a 3 dimensional drawing environment, the two line may not physically intersect. One line may be at ground level and teh other may be 10 meters or more above or below ground level. As with the intersection osnap, apparent intersection also has an “Extended” mode. (Fig 21)

Fig 16



APPARENT INT

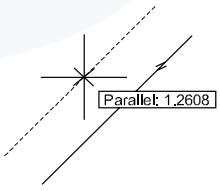
AD2108EG

You might get varying results if you have both the intersection and apparent intersection running object snaps turned on at the same time.

Parallel

Constrains a line segment, polyline segment, ray or xline to be parallel to another linear object. After you specify the first point of a linear object, specify the parallel object snap. Unlike other object snap modes, you move the cursor and *hover* over another linear object until the angle is acquired. Then, move the cursor back toward the object that you are creating. When the path of the object is parallel to the previous linear object, an alignment path is displayed, which you can use to create the parallel object. (Fig 22)

Fig 17



PARALLEL

AD2108EH

Turn off ORTHO mode before using the parallel object snap. Object snap tracking and polar snap are turned off automatically during a parallel object snap operation. You must specify the first point of a linear object before using the parallel object snap.

Select All

Turns on all object snap modes.

Clear All

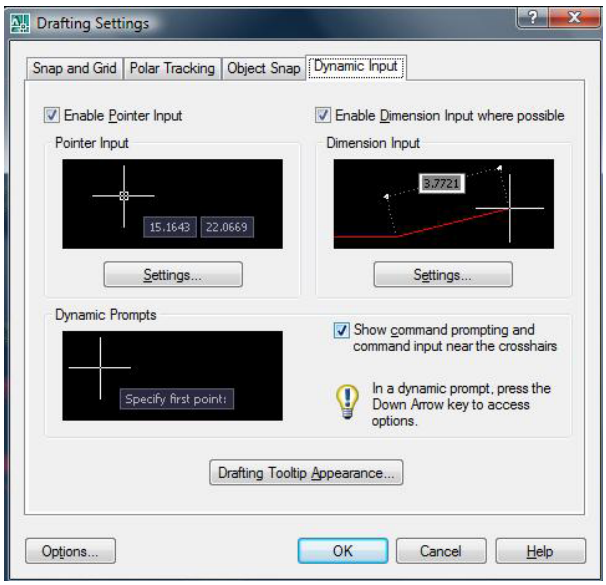
Turns off all object snap modes.

Dynamic input

Function key	Classic menu	Ribbon/Application menu
F12	Status bar	Status bar

It controls pointer input, dimension input, dynamic prompting, and the appearance of drafting tooltips. (Fig 23)

Fig 23



Enable pointer input

Turns on pointer input. When pointer input and dimensional input are both turned on, dimensional input supersedes pointer input when it is available. (DYNMODE system variable) (Fig 24)

Fig 24

Pointer input

Displays the location of the crosshairs as coordinate values in a tooltip near the cursor. When a command prompts you for a point, you can enter coordinate values in the tooltip instead of in the Command window.

Preview area

Shows an example of pointer input.

Settings

Displays the pointer input settings dialog box. (Fig 25)

Format

Controls coordinate format in the tooltips that are displayed when pointer input is turned on.

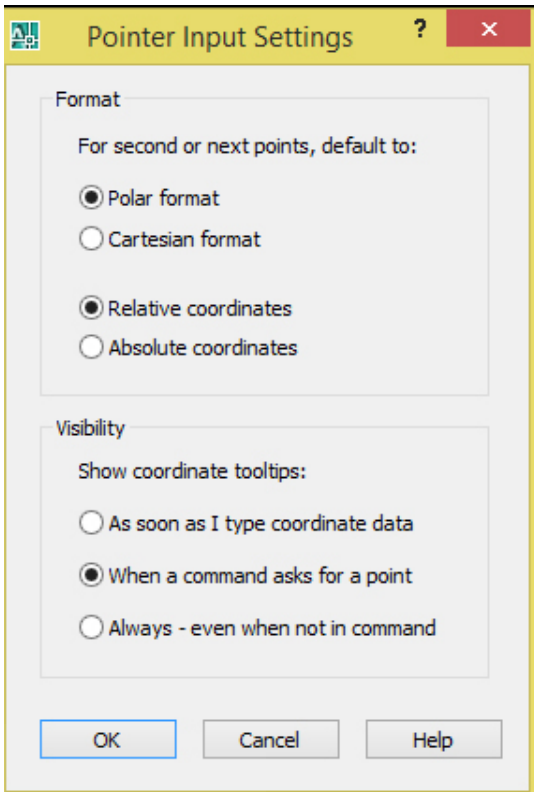
Polar format

Displays the tooltip for the second or next point in polar coordinate format. Enter a comma (,) to change to Cartesian format. (DYNPIFORMAT system variable)

Cartesian format

Displays the tooltip for the second or next point in Cartesian coordinate format. Enter an angle symbol (<) to change to polar format. (DYNPIFORMAT system variable)

Fig 25



Relative coordinates

Displays the tooltip for the second or next point in relative coordinate format. Enter a pound sign (#) to change to absolute format. (DYNPICOORDS system variable)

Absolute coordinates

Displays the tooltip for the second or next point in absolute coordinate format. Enter an at sign ( ) to change to relative format. Note that you cannot use the direct distance method when this option is selected. (DYNPICOORDS system variable)

Visibility

Controls when pointer input is displayed. (DYNPIVIS system variable)

As soon as I Type coordinate data

When pointer input is turned on, displays tooltips only when you start to enter coordinate data. (DYNPIVIS system variable)

When a command asks for a point

When pointer input is turned on, displays tooltips whenever a command prompts you for a point. (DYNPIVIS system variable)

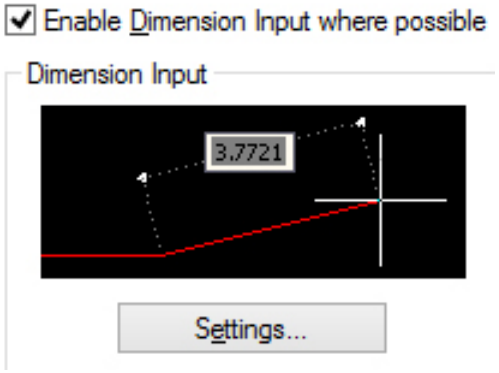
Always—Even when not in a command

Always displays tooltips when pointer input is turned on. (DYNPIVIS system variable)

Enable dimension input

Turns on dimension input. Dimensional input is not available for some commands that prompt for a second point. (DYNMOD system variable) (Fig 26)

Fig 26



Dimension input

Displays a dimension with tooltips for distance value and angle when a command prompts for a second point or a distance. The values in the dimension tooltips change as you move the cursor. You can enter values in the tooltip instead of on the command line.

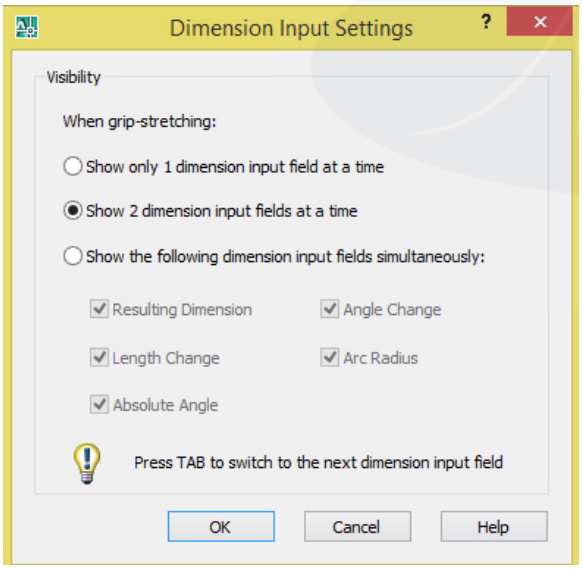
Preview area

Shows an example of dimensional input.

Settings

Displays the dimension input settings dialog box. (Fig 27)

Fig 27



Visibility

Controls which tooltips are displayed during grip stretching when dimensional input is turned on. (DYNDIVIS system variable)

Show only 1 dimension input field at a time

Displays only the length change dimensional input tooltip when you are using grip editing to stretch an object. (DYNDIVIS (page 911) system variable)

Show 2 Dimension input fields at a time

Displays the length change and resulting dimensional input tooltips when you are using grip editing to stretch an object. (DYNDIVIS (page 911) system variable)

Show the following dimension input fields simultaneously

When you are using grip editing to stretch an object, displays the dimensional input tooltips that are selected below. (DYNDIVIS and DYNDIGRIP system variables)

Resulting dimension

Displays a length dimensional tooltip that is updated as you move the grip.

Length change

Displays the change in length as you move the grip.

Absolute angle

Displays an angle dimensional tooltip that is updated as you move the grip.

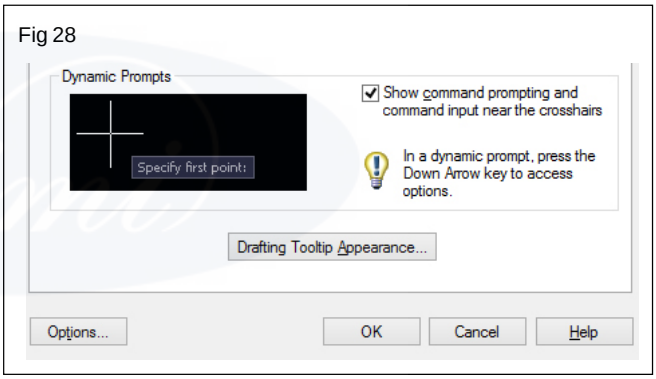
Angle change

Displays the change in the angle as you move the grip.

Arc radius

Displays the radius of an arc, which is updated as you move the grip. (Fig 28)

Fig 28



Dynamic prompts

Displays prompts in a tooltip near the cursor when necessary in order to complete the command. You can enter values in the tooltip instead of on the command line.

Preview area

Shows an example of dynamic prompts.

Show command prompting and command input near crosshairs

Displays prompts in Dynamic Input tooltips. (DYNPROMPT system variable)

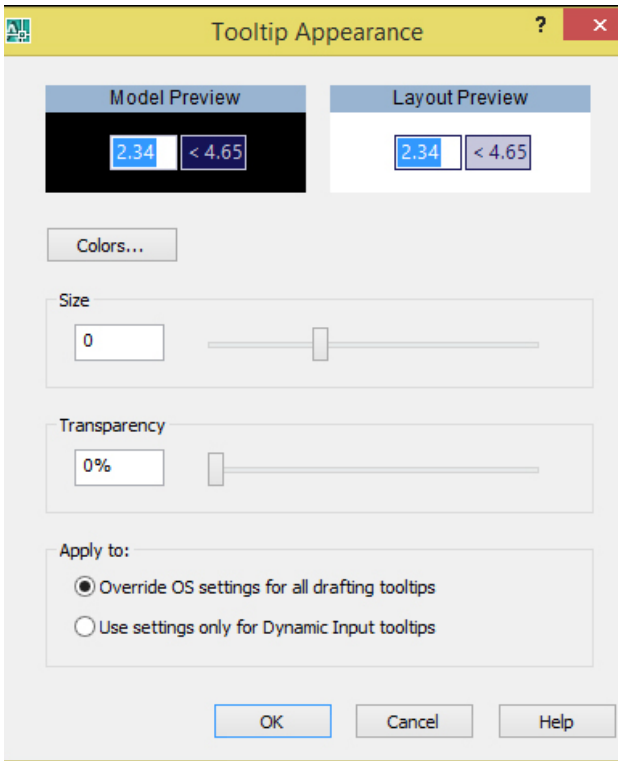
Show additional tips with command prompting

Controls whether tips for using Shift and Ctrl for grip manipulation are displayed. (DYNINFOTIPS system variable)

Drafting tooltip appearance

Displays the Tooltip appearance dialog box. (Fig 29)

Fig 29



Previews

Displays an example of the current tooltip appearance settings.

Size

Specifies a size for tooltips. The default size is 0. Use the slider to make tooltips larger or smaller.

Transparency

Controls the transparency of tooltips. The lower the setting, the less transparent the tooltip. A value of 0 sets the tooltip to opaque.

Apply to

Specifies whether the settings apply to all drafting tooltips or only to dynamic input tooltips. (DYNTOOLTIPS system variable)

Override OS settings for all drafting tooltips

Applies the settings to all tooltips, overriding the settings in the operating system.

Use settings only for dynamic input tooltips

Applies the settings only to the drafting tooltips used in Dynamic Input.

Advanced drafting commands - I

Objectives: At the end of this exercise you shall be able to

- **understand following advanced commands**
  - **layers**
  - **Q select**
  - **filter**
  - **group**
  - **block**
  - **insert**
  - **write block**
  - **divide**
  - **measure.**

PROCEDURE

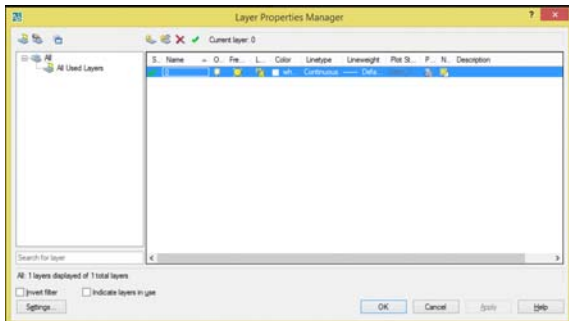
TASK 1 : **Layers**

Layer is a concept that allows grouping of drawn geometry in distinct and separate categories according to similar features and a common theme. This allows control over drawing, by applying properties to the layers such as assigning colors and linetypes. You can also manipulate each individual layer making it visible and invisible for clarity as well as being able to lock them to prevent editing.

For example, all the walls on floor plan will exist in “wall” layer, all the doors on “door” layer, windows on “window” layer and so on. Layer is like transparent sheets with each sheet having unique and distinct data. But when viewed at once, they show the complete picture. (Fig 1)

The following options are displayed.

Fig 1



New property filter

Displays the layer filter properties dialog box, where you can create a layer filter based on one or more properties of the layers.

New group filter

Creates a layer filter that conatins layers that you select and add to the filter

Layer states manager

Displays the layer states manager, in which you can save the current property settings for layers in a named layer state and then restore these settings later.

New layer

Creates a new layer. The list displays a layer named LAYER1. The name is selected so that you can enter a new layer name immediately.

Set current

Sets the selected layer as the current layer. Objects you create are drawn on the current layer.

Current layer

Displays the name of the current layer.

Search for layer

Filters the layer list by name quickly as you enter characteristics.

Status line

Shows the name of the current filter, the number of layers displayed in the list view, and the number of layers in the drawing.

Invert filter

Displays all layers that do not meet the criteria in the selected layer property filter.

Refresh

Refreshes the layer usage information by scanning all the entities in the drawing.

Settings

Displays the layer settings dialog box, in which you can set new layer notification settings, if layer filter changes are applied to the layer toolbar, and change the background color for layer property overrides.

Apply

Applies changes that have been made to layers and filters but does not close the dialog box.

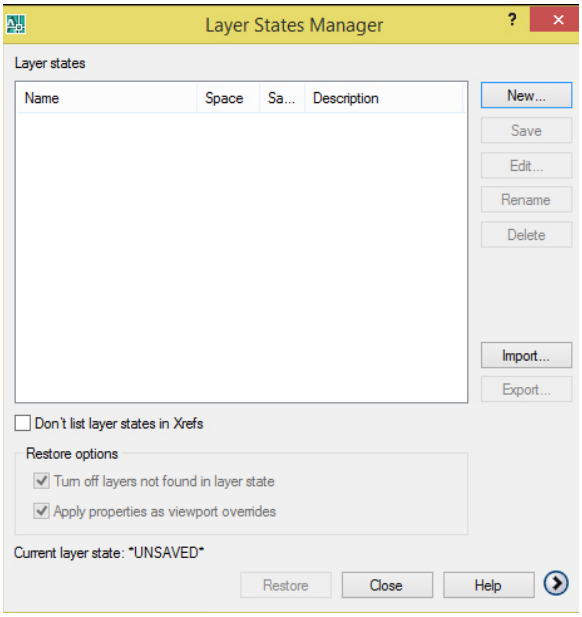
Layer states manager

While working on a drawing, you can save the properties of all layers under one name and restore the properties

later. The properties you saved can be imported to other drawings. This saves times in setting properties for layers in new drawing.

The following options are displalyed. (Fig 2)

Fig 2



Layer states

Lists the named layer states that have been saved in the drawing, the space in which they were saved (model space, layout, or xref), whether the layer list is the same as in the drawing, and an optional description.

Don't list layer states in Xrefs

Controls whether layer states in xrefs are displayed.

New

Displays the new layer state to save dialog box, where you can provide a name and a description for the new named layer state.

Save

Saves the selected named layer state.

Edit

Displays the edit layer state dialog box, where you can modify a selected named layer state.

Rename

Allows in place editing of the layer state name.

Delete

Removes the selected named layer state.

Import

Displays a standard file selection dialog box, where you can load a previously exported layer state (LAS) file into the current drawing. Layer states in files (DWG, DWS, or DWT) can be imported.

Export

Displays a standard file selection dialog box, where you can save the selected named layer state to a layer state (LAS) file.

Restore

Restore state and property settings of all layers in a drawing to their previously saved settings.

Close

Closes the layer states manager and saves changes.

Restore options

Turn off layers not found in layer state

Where a layer state is restored, turns off new layers for which settings were not saved so that the drawing looks the same way it did when the named layer state was saved.

Apply properties as viewport overrides

Applies layer property overrides to the current viewport. This option is available when the layer states manager is accessed when a layout viewport is active.

More options

Controls the display of additional options in the layer states manager dialog box.

Layer properties to restore

Restores the selected layer settings and properties to their previously saved state. The visibility in current viewport option is available only for layout viewports, while the On/Off and frozen/thawed options are available only for model space viewports.

Select all

Selects all the settings.

Clear all

Removes selection from all the settings.

— — — — — — — —

TASK 2 : **Select**

Procedure

It places selected objects in the previous selection set. A small box, called the object selection target or pickbox, replaces the crosshairs on the graphics cursor. At the select objects prompt in a subsequent command, use the previous option to retrieve the previous selection set. You can also press and hold the ctrl key to select original individual forms that are part of composite solids or vertices, edges, and faces on 3D solids. You can select one of these subobjects, or create a selection set of more than one subobject. Your selection set can include more than one type of subobject. To view all options, enter? at the command prompt.

Window

Window all objects completely inside a rectangle defined by two points. Specifying the corners from left to right creates a window selection (specifying the corners from right to left creates a crossing selection).

Last

Selects the most recently created visible object. The object must be in the current space, that is, model space or paper space, and its layer must not be set to frozen or off.

Crossing

Selects objects within and crossing an area defined by two points. A crossing selection is displayed as dashed or otherwise highlighted to differentiate it from window selection.

Box

Selects all objects inside or crossing a rectangle specified by two points. If the rectangle's points are specified from right to left, box is equivalent to crossing. Otherwise, box is equivalent to window.

All

Selects all objects in either model space or the current layout, except those objects on frozen or on locked layers.

Fence

Selects all objects crossing a selection fence. The fence method is similar to Cpolygon except that the fence is not closed and a fence can cross itself. Fence is not affected by the pickadd system variable.

Wolygon

Selects objects completely inside a polygon defined by points. The polygon can be of any shape but can neither cross nor touch itself. The last segment of the polygon is drawn so that it remains closed at all times. Wpolygon is not affected by the pickadd system variable.

Cpolygon

Selects objects within and crossing a polygon defined by specifying points. The polygon can be of any shape but cannot cross or touch itself. The last segment of the polygon is drawn so that it remains closed at all times. Cpolygon is not affected by the Pickadd system variable.

Group

Selects all objects within a specified group.

Add

Switches to the add method: selected objects can be added to the selection set by using any of the object selection methods. Auto and add are the default methods.

Remove

Switches to the remove method: objects can be removed from the current selection set using any object selection method. An alternative to remove mode is to hold down shift while selecting single objects or use the automatic option.

Multiple

Selects objects individually without highlighting them during object selection. This speeds up object selection for highly complex objects.

Previous

Selects the most recent selection set. The previous selection set is cleared by operations that delete objects from the drawing.

> **Note The previous selection set is ignored, if you switch spaces.**

Undo

Cancels the selection of the object most recently added to the selection set.

Auto

Switches to automatic selection: pointing to an object, this option selects the object. Whereas, pointing to a blank area inside or outside an object forms the first corner of a box defined by the box method. Auto and add are default methods.

Single

Switches to the single method: selects the first object or set of objects designated rather than continuing to prompt for further selections.

Subobject

Allows you to select original individual forms that are part of composite solids or vertices, edges, and faces on 3D solids. You can select one of these subobjects or create a selection set of more than one subobject. Your selection set can include more than one type of subobject. Pressing and holding the ctrl key is the same as selecting the select command's subobject option.

Object

Ends the ability to select subobjects. Allows you to use object selection methods.

— — — — — — — —

TASK 3 : **Qselect**

The quick selection tool is used to create a new selection set that will include or exclude all objects whose object type and property criteria matches as specified for the selection set. The quick select can be used to select the entities in the entire drawing or in the existing selection set.

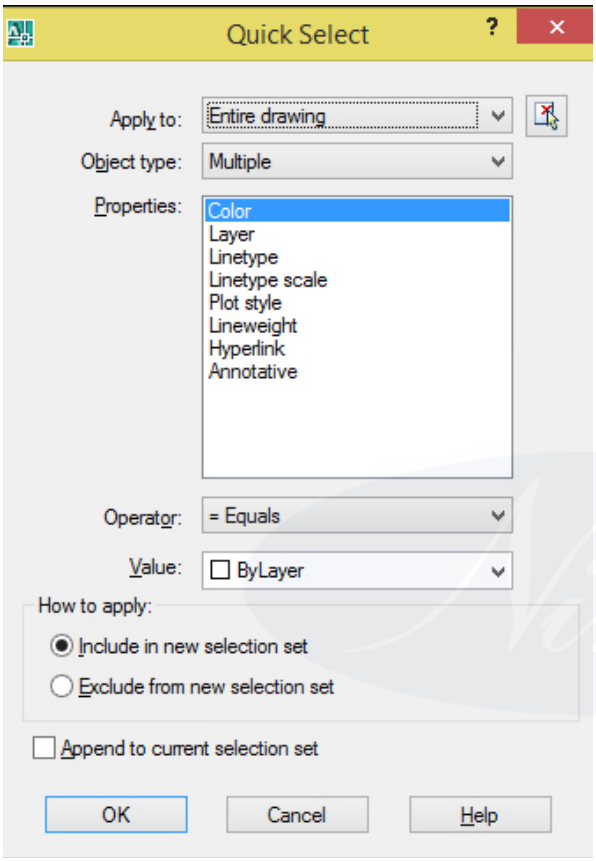
Data

Command alias	Short key	Button	Classic menu	Ribbon/Application menu
qselect	End any active commands, right-click in the drawing area, and choose quick select.		Tools=> Quick select	Home => Utilities => Quick select

Procedure

The quick select dialog box is displayed. (Fig 3)

Fig 3



The following options are displayed;

Apply to

Applies the filtering criteria to the entire drawing or to the current selection set (if one exists)

If append to current selection set is selected, the filtering criteria is applied to the entire drawing.

Select objects

Temporarily closes the quick select dialog box so that you can select the objects to which you want to apply the filter criteria.

Object type

Specifies the type of objects to be included in the filtering criteria. If the filtering criteria are being applied to the entire drawing, the object type list includes all object types, including custom. Otherwise, the list includes only the object types of the selected objects.

Properties

Specifies the object property for the filter. This list includes all searchable properties for the selected object type. The property you select determines the options available in operator and value.

Operator

Controls the range of the filter. Depending on the selected property, options can include equals, not equal to, greater than, less than, and wildcard match. Wildcard match is available only for text fields that can be edited. Use the select all option to ignore all properties filters.

Value

Specifies the property value for the filter.

How to apply

Specifies whether you want the new selection set to include or exclude objects that match the specified filtering criteria. Select include in new selection set to create a new selection set composed only of objects that match the filtering criteria. Select exclude from new selection set to create a new selection set composed only of objects that do not match the filtering criteria.

Append to current selection set

Specifies whether the selection set created by Qselect replaces or is appended to the current selection set.

Note : Qselect supports custom objects (objects created by another application) and their properties. If a custom object uses properties other than AutoCAD properties, the custom object's source application must be running in order for the properties to be available to qselect.

— — — — — — — —

TASK 4 : **Filter**

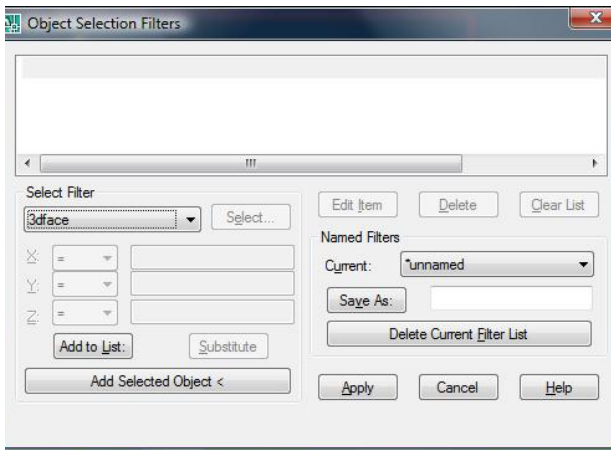
Procedure

We had already discussed using quick select before. Qselect is very useful if you just use it occasionally with not many properties to filter. These is one more utility that we can use, very powerful even it's not as pretty as qselect.

1 You can set multiple conditional filters at once. Qselect allows you to apply only one conditional filter at a time. Then you need to apply another filter to your selection.

2 You can save the filter setting and use it again later. (Fig 4)

Fig 4



To create filter, type 'Fi' on the command line to open the object selection filter dialogue box. If you have already chosen a command, type 'filter' at the select object: prompt to create a filter transparently.

Note : The selection filter finds only colors and linetypes of objects that have been specifically set as such, rather than part of the layer definition.

The following options are displayed.

Filter property list

Displays a list of the filter properties that compose the current filter. The current filter is the filter that you select in current at the named filters area.

Select filter

Adds filter properties to the current filter.

Object types and logical operators

Object types that you can filter and logical operators (AND, OR, XOR, and NOT) that could be used for grouping the filter expressions are listed when this option is availed.

Logical operators

Starting	Encloses operators	Ending operator
Begin AND	One or more operands	End AND
Begin OR	One or more operands	End OR
Begin XOR	Two operands	End XOR
Begin NOT	One operand	End NOT

If you use logical operators, make sure that you pair and balance them correctly in the filter list. The number of operands you can enclose depends on the operation.

For example, the following filter selects all circles except the ones with a radius greater than or equal to 1.0:

Object = circle

**Begin NOT

Circle radius >= 1.00

**End NOT

X, Y, Z parameters

Define additional filter parameters depending on the object. For example, if you select line start, you can enter the X, Y, and Z coordinate values that you want to filter.

In the filter parameters, you can use relative operators such as < (less than) or > (greater than). For example, the following filter selects all circles with center points greater than or equal to 1,1,0 and radii greater than or equal to 1:

Object = circle

Circle center X > = 1.0000 Y > = 1.0000 Z > = 0.0000

Circle radius > = 1.0000

Select

Displays a dialog box listing all items of the specified type in the drawing and selects the items to be filtered. For example, if you select the object type color, 'select' displays a list of colors to choose to be filtered.

Add to list

Adds the current select filter property to the filter list. Filter properties that you add to the unnamed filter remain available during the current work session unless you manually delete them.

Substitute

Replaces the filter property selected in the filter property list with the one displayed in select filter.

Add selected object

Adds one selected object in the drawing to the filter list.

Edit item

Moves the selected filter property into the select filter area for editing. The edited filter replaces the selected filter property.

Delete

Deletes a selected filter property from the current filter.

Clear list

Deletes all the listed properties from the current filter.

Named filters

Displays, saves, and deletes filters.

Current

Displays saved filters. Select a filter list to make it current. The named filter and its list of properties are loaded from the default file, filter, nfl.

Save as

Saves a filter and its list of properties. The filter is saved in the filter.nfl file. Names can contain up to 18 characters.

Delete current filter list

Deletes a filter and all its properties from the default filter file.

Apply

Exits the dialog box and displays the select objects prompt, where you create a selection set. The current filter is used on the objects you select.

— — — — — — — —

TASK 5 : **Group**

Data

Command alias	Button	Classic menu	Ribbon/application menu
G		Tools => Group	Home => Groups => Group

Procedure

The following prompts are displayed.

Enter a group option [?- List Groups/Order/Add/Remove/ Explode/Rename/Selectable/Create] <Create>:

? - List groups

Lists names and descriptions of groups defined in the drawing.

Order

Changes the numerical order of objects within a group. Re-ordering is useful while creating tool paths. For example, you can change the cut order for the horizontal and vertical lines of a tool path pattern.

Position number

Specifies the position number of the object to reorder. To reorder a range of objects, specify the first object's position number.

Reverse order

Reverses the order of all members in a group.

Add

Adds objects to a group.

Remove

Removes objects from a group.

If you remove all the group's objects, the group remains defined. You can remove the group definition from the drawing by using the explode option.

Explode

Deletes a group definition by exploding the group into its component objects.

Rename

Assigns a new name to an existing group.

Selectable

Specifies whether a group is selectable. When a group is selectable, selecting one object in the group selects the whole group. Objects on locked or frozen layers are not selected.

Create

Creates a group.

Note : Group names can be up to 31 characters long and can include letters, numbers, and special characters like dollar sign ($), hyphen (-), and underscore (_) but not spaces. The name is converted to uppercase characters.

— — — — — — — —

TASK 5 : **Block**

Data

Command alias	Button	Classic menu	Ribbon/application menu
G		Draw => Block => Make	Draw => Block => Create

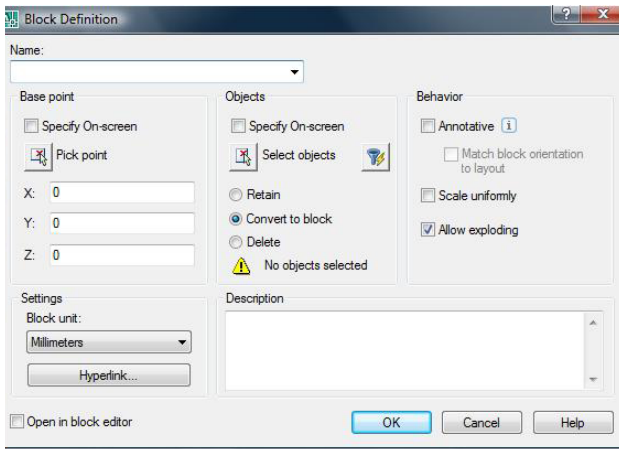
Procedure

A block is a collection of objects (lines, arcs, circles, text, etc.) that form a more complex entity that normally represents an object in the real world, e.g. a door, a chair, a window, or a computer. (Fig 5)

There are many advantages of using blocks, the major ones being described as follows:

- Blocks are a single entity. This means that you can modify (move, copy, rotate) a block by selecting only one object in it.
- You can build up a library of blocks consisting of the parts that you require many times in your workday. These blocks can be stored in a separate folder and

Fig 5



even on a network so that all drafters have access to them. Think of plumbing parts, valves, elbows, etc.

- Using blocks can help keep your file size down. AutoCAD stores block definitions in its database. When you insert a block, AutoCAD only stores the name of the block, its location (insertion point), scale and rotation. This can be very noticeable in large drawings.
- If you wish to change something, you can redefine a block. For instance, you draw a chair and turn it into a block, in the diagram. Later, you're told that the size of the chair has changed. Since you used a block you can re - define the block and all the chairs are updated automatically. This saves a lot of time and labour, as in other cases, if you had placed (or copied) 100 chairs in your drawing, you would have to manually change each one.
- Blocks can also contain non-graphical information, which are text objects called attributes. For instance, you have made blocks of different chairs. You can add information to the block such as the manufacturer, cost, weight, etc. This information stays with the block, but can also be extracted to a database or spreadsheet. This would be useful for things such as a bill of materials. Attributes can also be visible or invisible in your drawing. Another good use of attributes could be a title block.
- You can also add internet hyperlinks to blocks with ease so you can connect a block to a page onto a supplier's online catalog.
- There are two types of blocks you can create: blocks that are internal to your current drawing and those that are external, or saved as a separate file. To create the different types, different commands are used. Many companies use a template that will include a number of blocks for use in the project.

The following options are displayed.

Name

It names the block. The name can have up to 255 characters and can include letters, numbers, blank spaces, and/or any special character not used by the operating system or the program for other purposes.

The block name and definition are saved in the current drawing.

Preview

If an existing block is selected under Name, this option displays a preview of the block.

Base point

Specifies an insertion base point for the block. The default value is 0, 0, 0.

Specify on-screen

Prompts you to specify the base point when the dialog box is closed.

Pick insertion base point

Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

X

Specified the X coordinate value.

Y

Specifies the Y coordinate value.

Z

Specifies the Z coordinate value.

Objects

Specifies the objects to be included in a new block and also whether to retain or delete the selected objects or convert them to a block instance after you create the block.

Specify on-screen

Prompts you to specify the objects when the dialog box is closed.

Select objects

Closes the block definition dialog box temporarily while you select the objects for the block. When you finish selecting the objects, press enter to return to the dialog box.

Quick select

Displays the quick select dialog box, which defines a selection set.

Retain

Retains the selected objects as distinct objects in the drawing after you create the block.

Convert to block

Converts the selected objects to a block instance in the drawing after you create the block.

Delete

Deletes the selected objects from the drawing after you create the block.

Object selected

Displays the number of selected objects.

Behavior

Specifies the behavior of the block.

Annotative

Specifies that the block is annotative.

Match block orientation to layout

Specifies that the orientation of the block references in paper space viewports, matches the orientation of the layout. However, this option is unavailable if the annotative option is cleared.

Scale uniformly

Specifies whether or not the block reference is prevented from being non-uniformly scaled.

Allow exploding

Specifies whether or not the block reference can be exploded.

Settings

Specifies settings for the block.

Block unit

Specifies the insertion units for the block reference.

Hyperlink

Opens the insert hyperlink dialog box, which you can use to associate a hyperlink with the block definition.

Description

Specifies the text description of the block.

Open in block editor

Opens the current block definition in the block editor when you click ok.

— — — — — — — —

TASK 6 : **Insert**

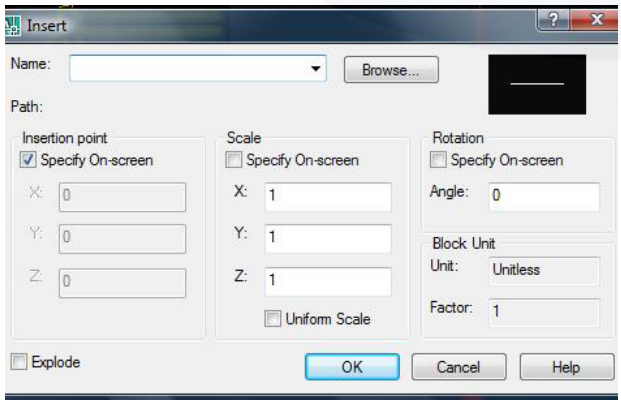
Data

Command alias	Button	Classic menu	Ribbon/application menu
I		Insert => Block => Make	Draw => Block => Insert

Procedure

The following options are displayed. (Fig 6)

Fig 6



Name

Specifies the name of a block to be inserted, or the name of a file to be inserted as a block.

Browse

Opens the select drawing file dialog box (a standard file selection dialog box) where you can select a block or drawing file to be inserted.

Path

Specifies the path to the block.

Locate using geographic data

Inserts drawing using geographic data as the reference.

Specifies if the current and attached drawing contain geographic data. This option is available only if both drawings have geographic data.

Description

Displays the description that was saved with the block.

Preview

Display a preview of the specified block to be inserted. A lightning bolt icon in the lower-right corner of the preview indicates that the block is dynamic. An icon indicates that the block is annotative.

Insertion point

Specifies the insertion point for the block.

Specify on-screen

Specifies the insertion point of the block using the pointing device.

Input coordinates

Allows you to manually enter the X, Y, and Z coordinate value for the insertion point of the block.

X

Sets the X coordinate value.

Y

Sets the Y coordinate value.

Z

Sets the Z coordinate value.

Scale

Specifies the scale for the inserted block. Specifying negative values for the Z, Y, and Z scale factors inserts a mirror image of a block.

Specify on-screen

Specifies the scale of the block using the pointing device.

Input scale factor

Allows you to manually enter a scale factor for the block.

X

Sets the X scale factor.

Y

Sets the Y scale factor.

Z

Sets the Z scale factor.

Uniform scale

Specifies a single scale value for X, Y, and Z coordinates.

Rotation

Specifies the rotation angle for the inserted block in the current UCS.

Specify on-screen

Specifies the rotation of the block using the pointing device.

Input angle

Allows you to manually enter an angle of rotation for the block.

Angle

Sets a rotation angle for the inserted block.

Block unit

Displays information about the block units.

Unit

Specifies the insunits value for the inserted block.

Factor

Displays the unit scale factor, which is calculated based on the insunits value of the block and the drawing units.

Explode

Explodes the block and inserts the individual parts of the block. When explode is selected, you can specify a uniform scale factor only.

Component objects of a block drawn on layer 0 remain on that layer. Objects having color Byblock are white. Objects with linetype byblock have the continuous linetype.

— — — — — — — —

TASK 7 : **Write block**

Data

Command alias	Button	Classic menu	Ribbon/application menu
WBLOCK			Draw => Block => Insert

Procedure

The write block dialog box displays different default settings depending on whether nothing is selected, a single block is selected, or objects other than blocks are selected. (Fig 7)

Source

Specifies blocks and objects, saves them as a file, and specifies insertion points.

Block

Specifies an existing block to be saved as a file. Select a name from the list.

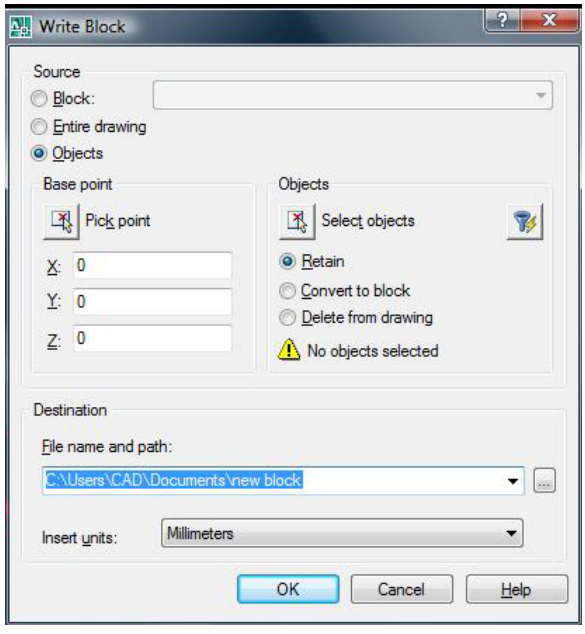
Entire drawing

Selects the current drawing to be saved as another file.

Objects

Selects objects to be saved as a file. Specify a base point and selects the objects below.

Fig 7



Base point

Specifies a base point for the block. The default value is 0,0,0.

Pick point

Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

X

Specifies the X coordinate value for the base point.

Y

Specifies the Y coordinate value for the base point.

Z

Specifies the Z coordinate value for the base point.

Objects

Sets the effect of block creation on objects used to create a block.

Retain

Retains the selected objects in the current drawing after saving them as a file.

Convert to block

Converts the selected object or objects to a block in the current drawing after saving them as a file. The block is assigned the name in file name.

Delete from drawing

Deletes the selected objects from the current drawing after saving them as file.

Select objects button

Temporarily closes the dialog box so that you can select one or more objects to save to the file.

Quick select button

Opens the quick select dialog box, which you can use to filter your selection set.

Objects selected

Indicates the number of objects selected.

Destination

Specifies the new name and location of the file and the units of measurement to be used when the block is inserted.

File name and path

Specifies a file name and path where the block or objects will be saved.

[...]

Displays a standard file selection dialog box.

Insert units

Specifies the unit value to be used for automatic scaling when the new file is dragged from design center or inserted as a block in a drawing that uses different units. Select unit less if you do not want to automatically scale the drawing when you insert it.

— — — — — — — —

TASK 8 : **Divide**

Data

Command alias	Button	Classic menu	Ribbon/application menu
DIV		Draw => Point => Divide	Home tab => Draw panel => Divide

Procedure

Creates evenly spaced point objects or blocks along the length or perimeter of an object. (Fig 7)

Select object to divide: Use an object selection method

Enter number of segments or [Block]: Enter a value from 2 through 32,767, or enter b

Number of segments

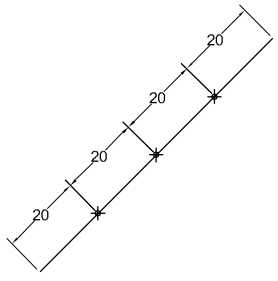
Places point objects at equal intervals along the selected objects.

Use DDPTYPE to set the style and size of all point objects in a drawing.

Block

Places blocks at equal intervals along the selected object. If the block has variable attributes, these attributes are not included.

Fig 1



Enter name of block to insert: Enter the name of a block currently defined in the drawing

Align block with object? [Yes/No] <Y>: Enter y or n or press ENTER

-Yes

Specifies that the X axes of the inserted blocks be tangent to, or collinear with, the divided object at the dividing points.

-No

Aligns the blocks according to their normal orientation.

Enter number of segments: Enter a value from 2 through 32,767 (Fig 8)

Fig 2

The illustration shows an arc divided into five equal parts using a block consisting of a vertically oriented ellipse.

TASK 9 : **Measure**

Data

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

Command alias	Button	Classic menu	Ribbon/application menu
Measure		Draw => Point => Measure	Home tab => Draw panel => Measure

Procedure

Select object to measure

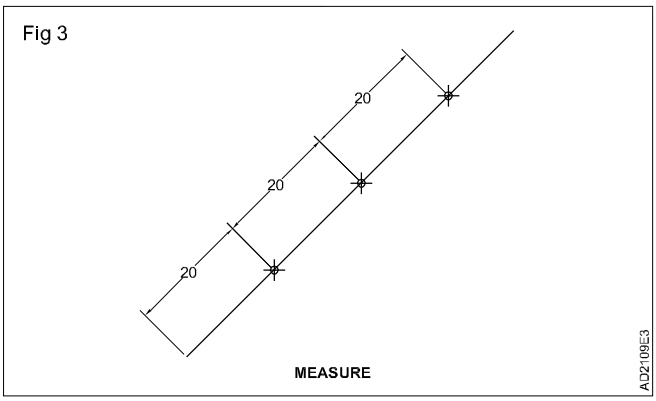
Specify length of segment of [Block]: Specify a distance or enter b

The resulting points or blocks are always located on the selected object and their orientation is parallel to the XY plane of the UCS.

Use DDPTYPE to set the style and size of all point objects in a drawing. (Fig 9)

The points or blocks are placed in the previous selection set, so you can select them all by entering p at the next select objects prompt. You can use the node object snap to draw an object by snapping to the point objects. You can them remove the points by entering erase previous.

Fig 3

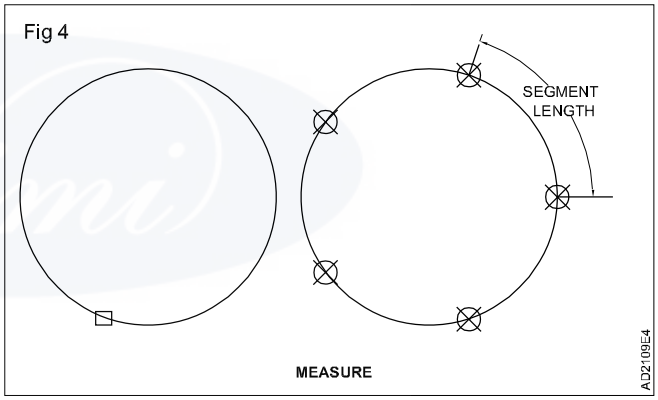


Length of segment

Places point objects at the specified interval along the selected object, starting at the endpoint closest to the point you used to select the object. Measurement of closed polylines starts at their initial vertex (the first one drawn).

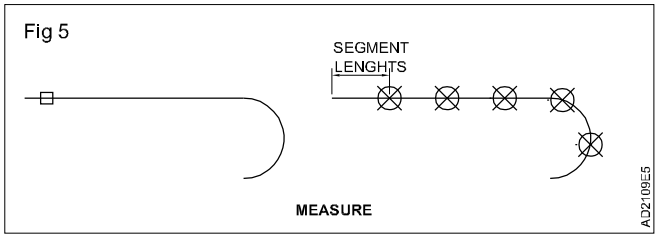
Measurement of circles starts at the angle from the center set as the current snap rotation angle. If the snap rotation angel is 0, then the measurement of the circle starts to the right of center, on its circumference. (Fig 10)

Fig 4



The illustration shows how measure marks 0.5 - unit distances along a polyline, with the PDMODE system variable set to 35. (Fig 11)

Fig 5



Block

Places blocks at a specified interval along the selected object.

Enter name of block to insert: Enter the name of a block currently defined in the drawing.

Align block with object? [Yes/No] <Y>: Enter or n or press enter

If you enter y, the block is rotated about its insertion point so that its horizontal lines are aligned with, and drawn tangent to, the object being measured. If you enter n, the block is always inserted with a 0 rotation angle.

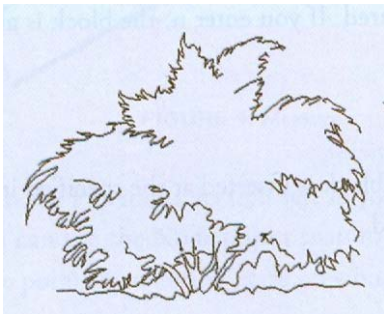
Specify length of segment

After you specify the segment length, the block is inserted at the specified interval. If the block has variable attributes, these attributes are not included.

Hands on -1

1 Open block1.dwg (Fig 12)

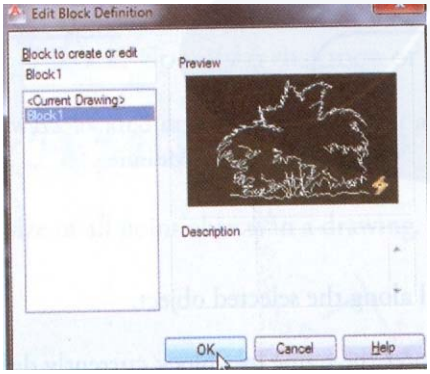
Fig 12



Command: BE

Select block: Select block 1 from list => ok (Fig 13)

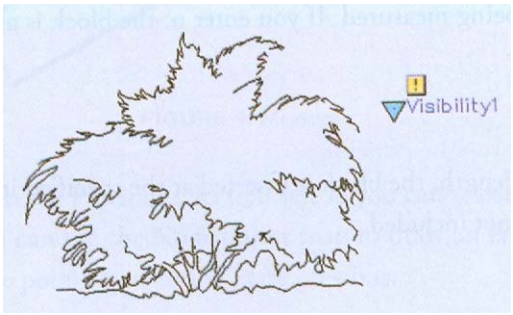
Fig 13



2 Assign visibility parameter to the block1

i. Parameter => Visibility => Place it on screen. (Fig 14)

Fig 14



ii. The visibility tab activated now. (Fig 15)

Fig 15

iii. Click on manage visibility states,

a. New => visibility state 0

b. New => visibility state 1

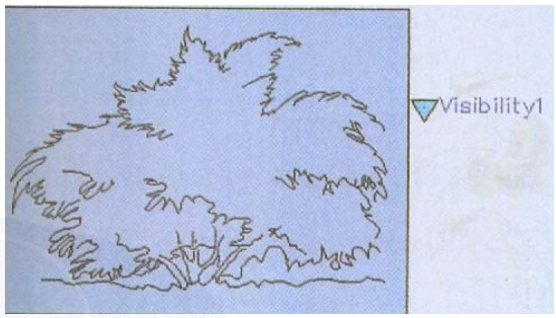
c. New => visibility state 2

d. New => visibility state 3

iv. Set the visibility state 1 as current.

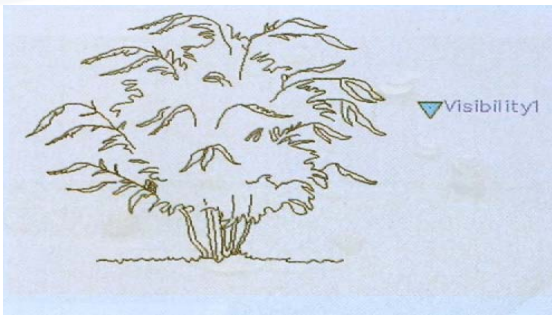
v.. Click on make invisible => Select the model shown in the screen => Enter. (Fig 16)

Fig 16



vi. Create a new drawing there. (Fig 17)

Fig 17

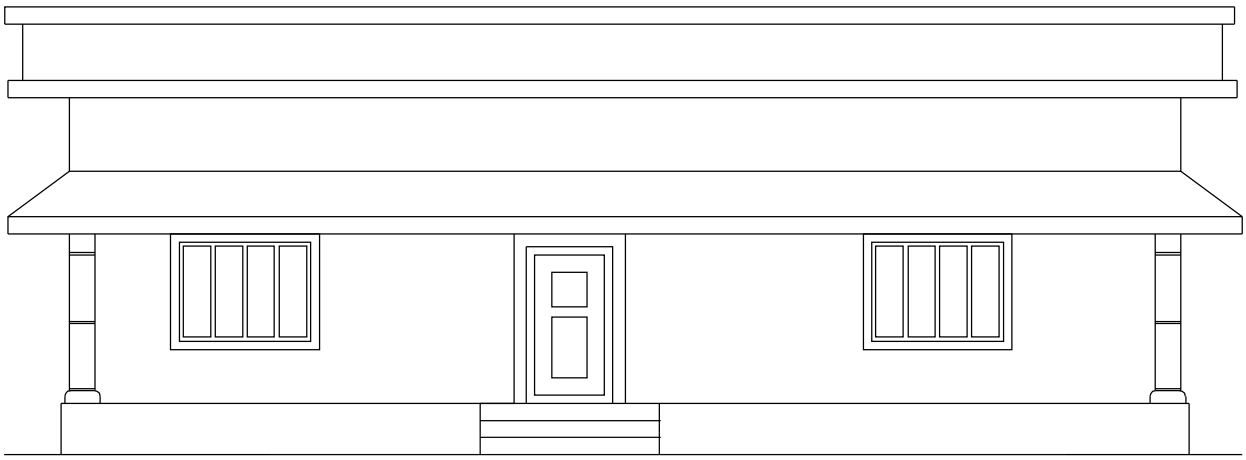


vii. Set the visibility state as visibility state 2

Hands on - 2

1 Open elevation.dwg. (Fig 18)

Fig 6



ELEVATION

AD2109E6

2 Create the parapet design with the divide command.

Command: DIVIDE

Select object to divide: <Select the 4th line from top of the parapet>

Enter the number of segments or [Block]: b

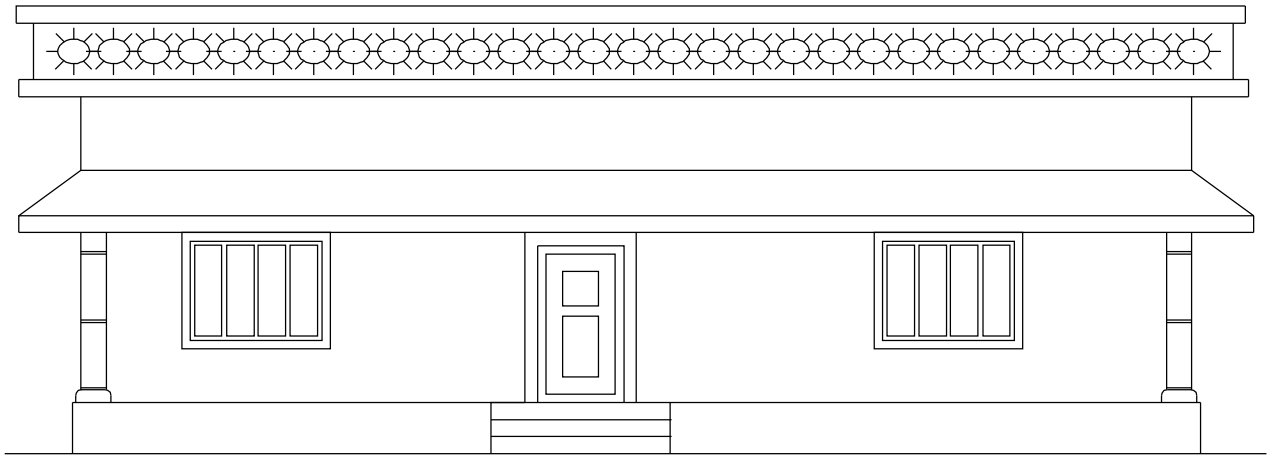
Enter name of block to insert: design1

Align block with object? [Yes/No] <Y>:

Enter the number of segments: 30 (Fig 19)

3 Save the file and exit.

Fig 7



ELEVATION

AD2109E7

Advanced drafting commands - II

Objectives: At the end of this exercise you shall be able to

- **understand following advanced commands**
  - **block attribute manager**
  - **design centre**
  - **text**
  - **text style**
  - **scale text**
  - **hatch**
  - **gradient.**

PROCEDURE

TASK 1 : **Block attribute manager**

Data

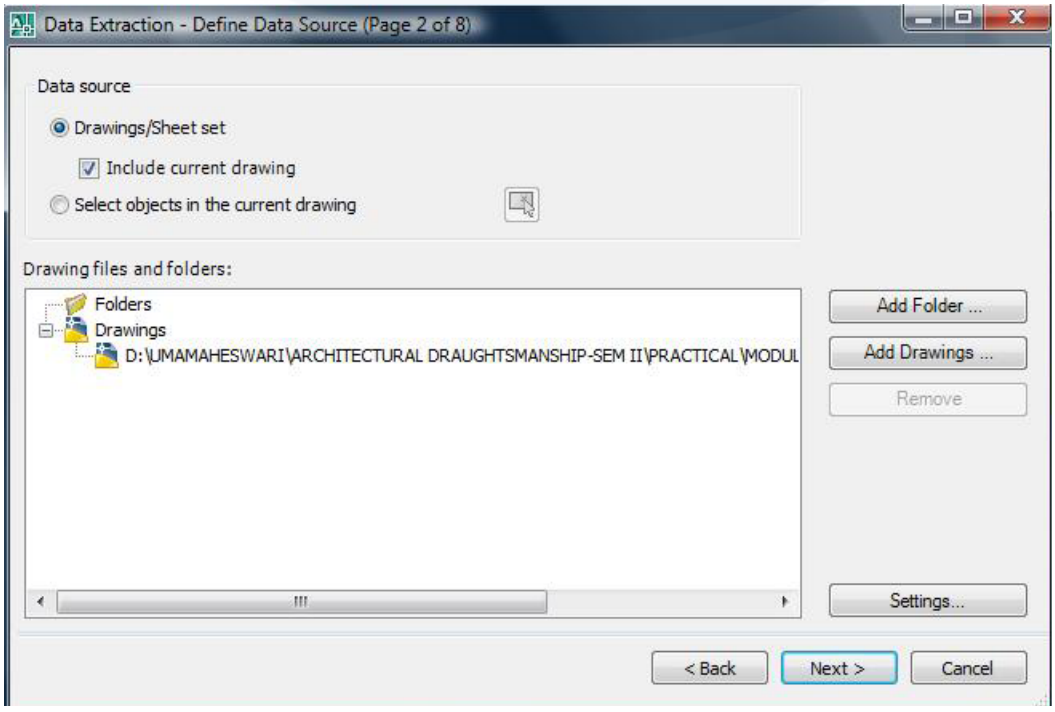
Command alias	Button	Classic menu	Ribbon/application menu
BATTMAN		Insert => Attributes panel => Manage	Modify => Object => Attribute => Block Attribute Manager

Procedure

Manages the attribute definitions for blocks in the current drawing. You can edit the attribute definitions in blocks, remove attributes from blocks, and change the order in which you are prompted for attribute values while inserting a block. (Fig 1)

Attributes of the selected block are displayed in the attribute list. By default, Tag, Prompt, Default, Mode, and annotative attribute properties are displayed in the attribute list. You can specify which attribute properties you want to be displayed in the list by choosing settings.

Fig 1



For each selected block, a description below the attribute list identifies the number of its instances in the current drawing and in the current layout.

Select block

Allows you to use your pointing device to select a block from the drawing area. When you choose select block,

the dialog box closes until you select a block from the drawing or cancel by pressing ESC.

If you modify attributes of a block and then select a new block before you save the attribute changes you have made, you will be prompted to save the changes before selecting another block.

Block

Lists all block definitions in the current drawing that have attributes. Select the block whose attributes you want to modify, first.

List of attributes

Displays the properties of each attribute in the selected block.

Blocks found in drawing

The number of instances of the selected block in the current drawing.

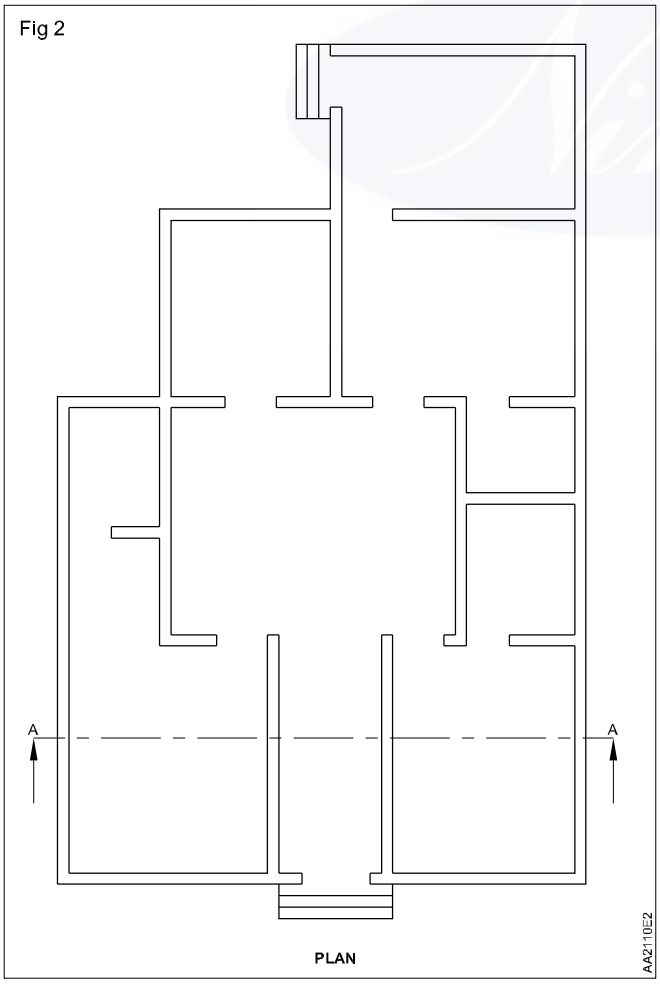
Blocks found in current space

The number of instances of the selected block in the current model space or layout.

Hands on - 1

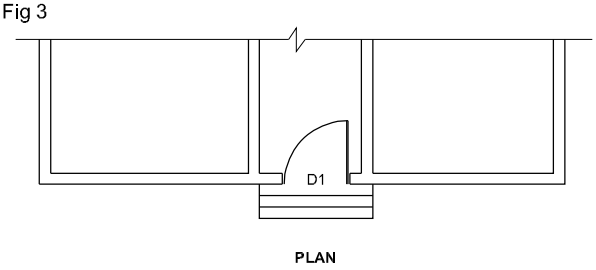
1 Open file plan.dwg. (Fig 2)

Fig 2



2 Turn off the layer DIMENSION and set '0' as the current layer. (Fig 3)

Fig 3

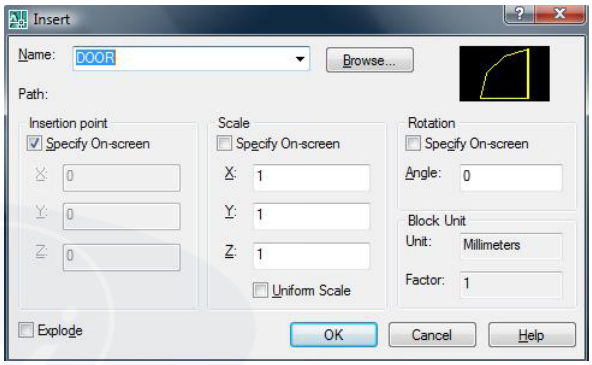


3 Command: ATTDIA

Enter new value for ATTDIA <0>:

4 Insert => Block => door => place this block on the plan. (Fig 4)

Fig 4



i. Fill with the attribute values as shown in Fig 5.

ii. Select the block and adjust the text position with the help of grip editing. (Fig 6 & Fig 7)

Fig 5

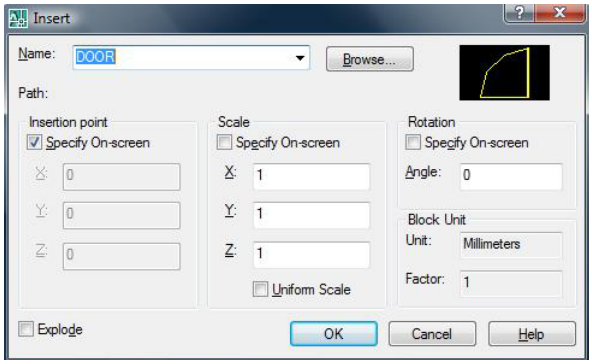
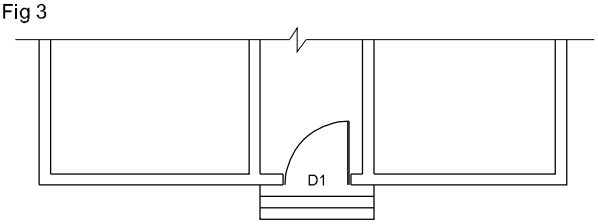


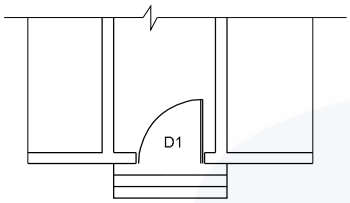
Fig 3



PLAN

AA2110E3

Fig 4



PLAN

AA2110E4

Fig 5

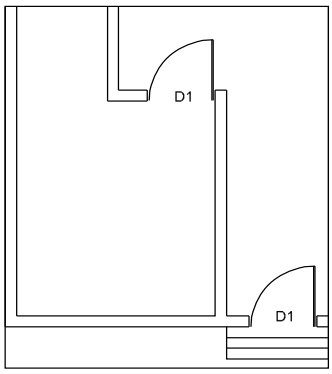


Fig 6

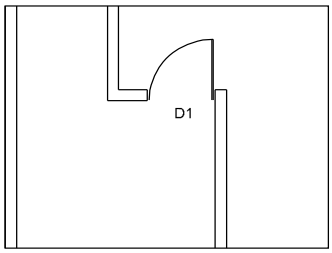
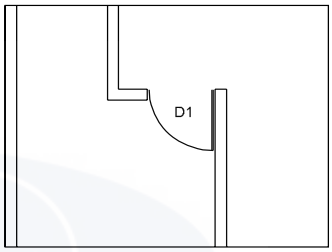


Fig 7



BLOCK

AA2110E5

5 Insert => Block => door => Place it as shown in Fig 8.

i. Select block => click on the flip button, to get the door as shown.

6 Use these methods to place all doors for this plan is shown in Fig 9.

7 Place all the windows as shown in the following table.

Name	ID	Material	Size	Count
Window	W1	TEAK	150 x 150	9
Window	W2	TEAK	60 x 90	2
Door	D1	TEAK	100 x 210	9

i. Insert => block => window (Fig 10)

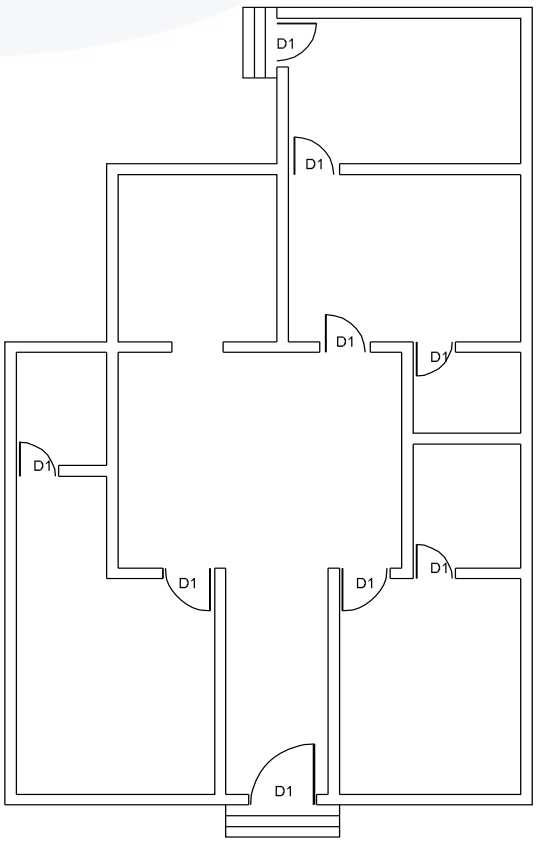
   a. Place it on the W1 position also specify attribute values as per the above table.

ii. Select the window, and use rotate icon to rotate window to correct position. (Fig 11)

iii. insert => Block => window

   a. Place it on the W2 position with suitable attribute values.

Fig 8

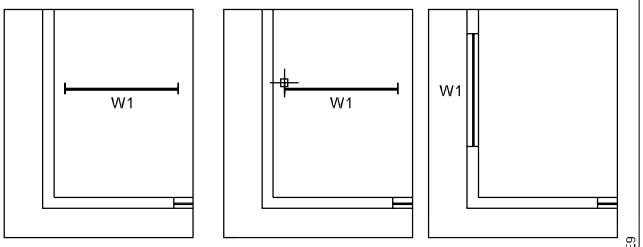


PLAN

AA2110E8

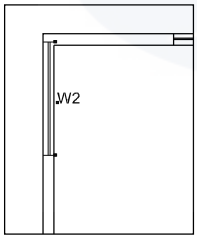
Fig 10

Fig 9 Fig 10 Fig 11



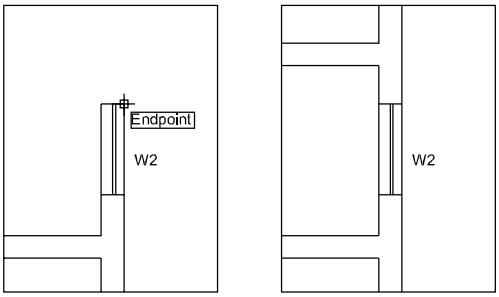
iv Select the window, and use rotate icon to rotate window to correct position. (Fig 12)

Fig 12



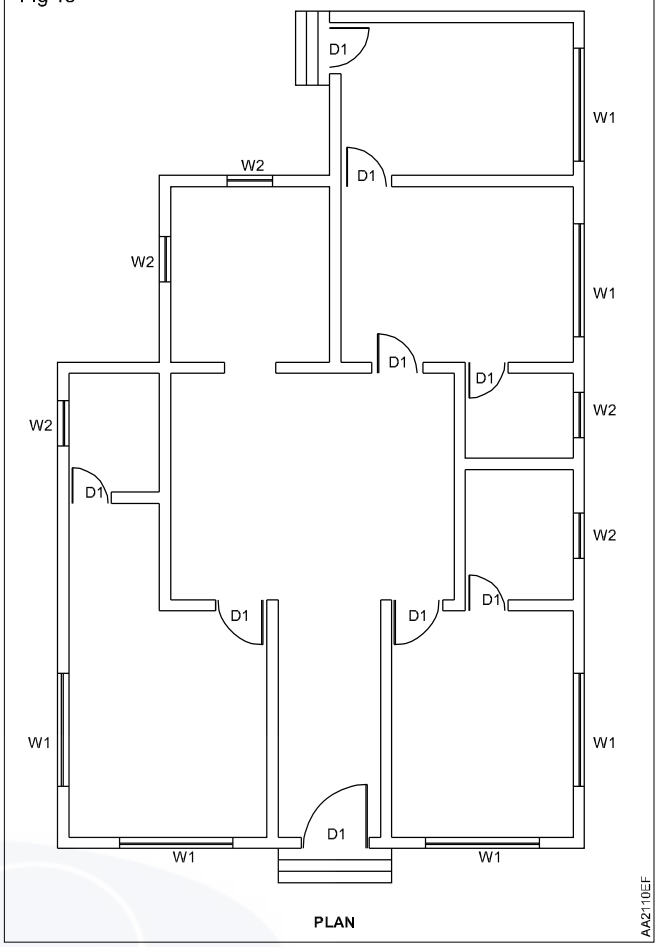
a. Click and drag on stretch icon to reduce its size to 90. (Fig 13)

Fig 13 Fig 14



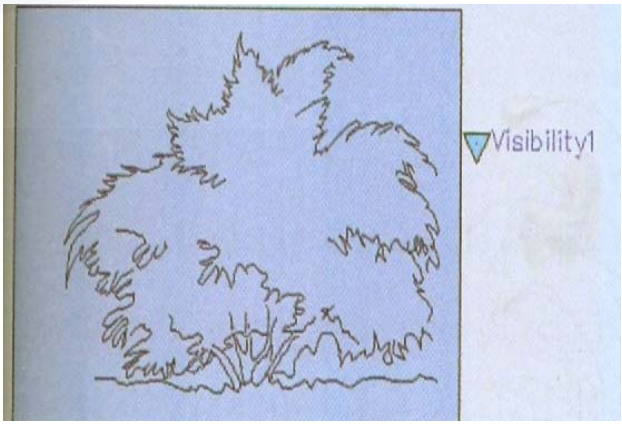
8 Repeat the steps for all windows and make the final drawing like this. (Fig 14)

Fig 15



9 Save the file and use the attribute extraction to create the joinery details. (Fig 15)

Fig 15



i. COMMAND: EATTEXT

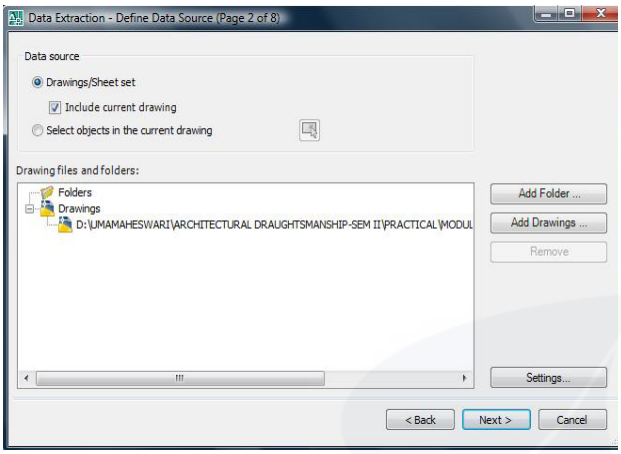
ii. Check create a new data extraction => Next

iii. Save the.dxe file with a name => Next (Fig 16)

10 Check include current drawing => Next (Fig 17)

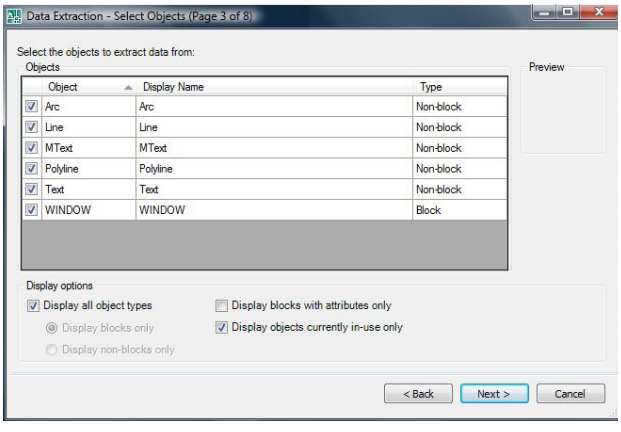
Fig 16

Fig 17



11 Check display blocks only and display blocks with attributes only as shown => Next. (Fig 18)

Fig 18



12 Choose attribute alone from the category filter => Next (Fig 19)

13 Drag the columns and arrange it in proper order => Next. (Fig 20)

Fig 19

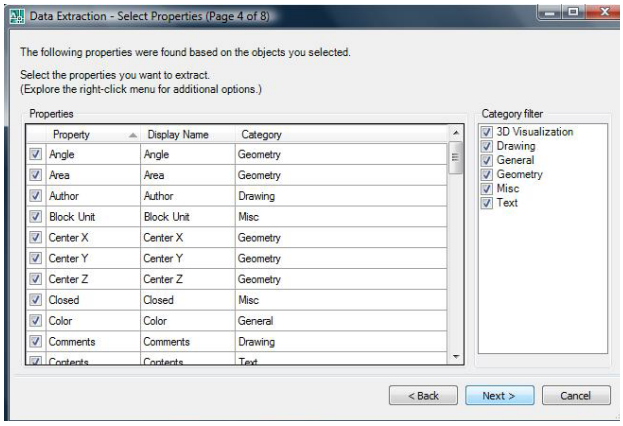
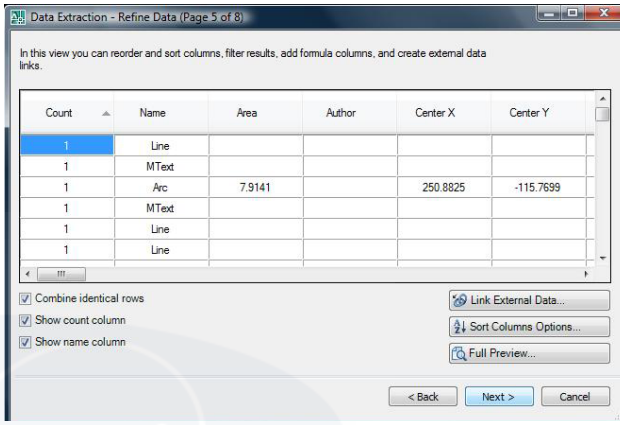


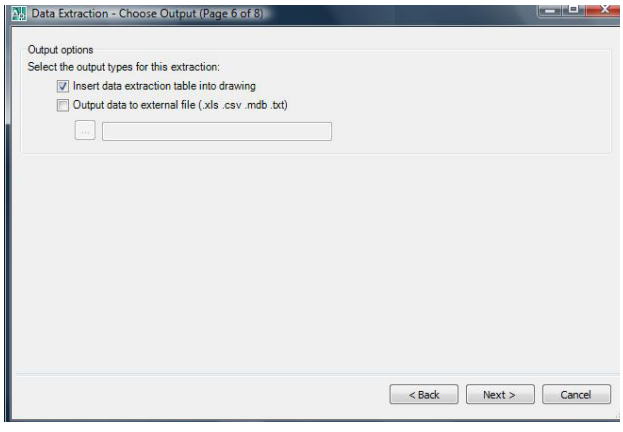
Fig 20



i. Check the option insert data extraction table into drawing => Next (Fig 21)

ii. Click on Next => Finish

Fig 21

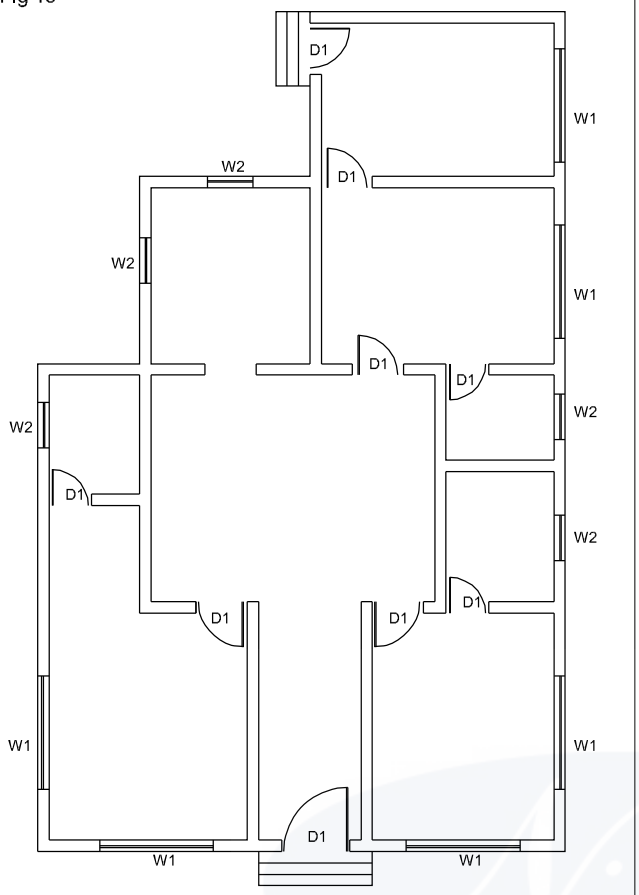


iii. Specify the insertion point on the screen to get a table as shown below

Name	ID	Material	Size	Count
Window	W1	TEAK	150 x 150	9
Window	W2	TEAK	60 x 90	2
Door	D1	TEAK	100 x 210	9

iv Click on the top row and add a title as joinery details. (Fig 21)

Fig 16



JOINERY DETAILS				
NAME	ID	MATERIAL	SIZE	COUNT
WINDOW	W1	TEAK	150X150	6
WINDOW	W2	TEAK	60X90	5
DOOR	D1	TEAK	100X210	9

PLAN

AA2110EG

When pausing for using input

Restores the view prior to the playback of an action macro when a request for user input occurs.

Once playback finishes

Restores the view prior to the playback of an action macro when playback is complete.

Check for inconsistencies when playback begins

Specifies if the action macro should be scanned for inconsistencies between the current drawing state and the drawing state when the macro was recorded.

More options

Controls the display of additional options in the action macro dialog box.

Restore pre-playback view

Check for inconsistencies when playback begins.

TASK 2 : **Design center**

Data

Command alias	Button	Classic menu	Ribbon/application menu
DC		Tools => Palettes => Design center	Insert tab => Content panel => Design center

Procedure

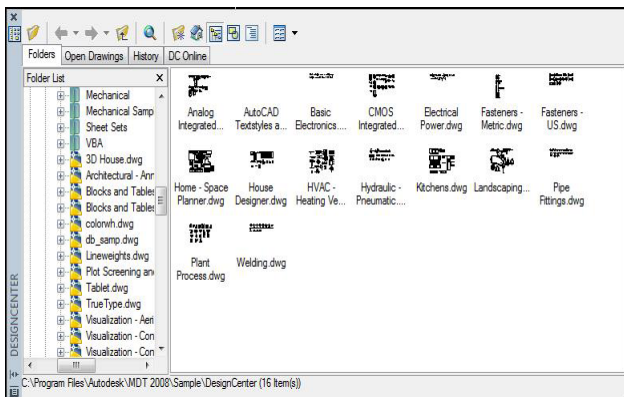
Browses, finds, and previews content, and inserts content, which includes blocks, hatches, and external references (xrefs).

Use the buttons in the toolbar at the top of design center for display and access options.

When you click the folders tab or the open drawings tab, the following two panes are displayed from which you can manage the drawing content. (Fig 22)

- Content area (right pane)
- Tree view (left pane)

Fig 22



Note The design center online (DC online tab) is disabled by default. You can enable it from the CAD manager control utility.

Content area (Design center)

Displays the content of the "container" currently selected in the tree view. A container is a network, computer, disk, folder, file, or web address (URL) containing information accessible by design center. Depending on the container selected in tree view, the content area displays the following:

- Folders containing drawings or other files
- Drawings
- Named objects contained in drawings (named objects include blocks, xrefs, layouts, layers, dimension styles, table styles, multi leader styles, and text styles)
- Images or icons representing blocks or hatch patterns.
- Web-based content
- Custom content developed by third-party applications

From the content area, you can insert blocks or hatch patterns or attach external references in a drawing by dragging, double-clicking, or right-clicking and choosing insert block, Attach Xref, or copy. You can drag or right-click to add other content to drawings, such as layers, dimension styles, and layouts. You can drag blocks and hatches from design center to tool palettes.

Note You can access relevant content area and tree view options on a shortcut menu by right-clicking in the tree view or the content area.

Load

Displays the load dialog box (a standard file selection dialog box). Use load to navigate to files on local and network drives or on the web, and then to select content to load in the content area.

Back

Returns to the most recent location in the history list.

Forward

Returns to the next latest location in the history list.

Up

Displays the contents of the container one level above the current container.

Stop (DC online tab)

Stops the current transfer.

Reload (DC online tab)

Reloads the current page.

Search

Displays the search dialog box, where you can specify search criteria to locate drawings, blocks, and nongraphical objects within drawings. Search also displays custom content saved on your desktop.

Favorites

Displays the contents of the favorites folder in the content area. The favorites folder contains shortcuts to items you often access. You can add items to favorites either by right-clicking the content area or right-clicking an item in the tree view, and then clicking add to favorites. To delete an item from favorites, use the organize favorites option on the shortcut menu and then use the refresh option on the shortcut menu.

Note : The designcenter folder is automatically added to favorites. This folder contains drawings with discipline-specific blocks which you can insert in drawings.

Home

Returns designcenter to your home folder. Upon installation, the home folder is set to ....\sample\ Designcenter. Change the home folder using the shortcut menu in the tree view.

Tree view toggle

Displays and hides the tree view. Hide the tree view if you need more space in your drawing area. When the tree view is hidden, you can use the content area to navigate to containers and to load content.

The tree view toggle button is not available while you're using the history list in the tree view.

Preview

Displays and hides a preview of the selected item in a pane below the content area. If there is no preview image saved with the selected item, the preview area is empty.

Description

Displays and hides a text description of the selected item in a pane below the content area. If a preview image is also displayed, the description is displayed below it. If there is no description saved with the selected item, the description area is empty.

Views

Provides different display formats for the content that is loaded in the content area. You can select a view from the views list or click the views button repeatedly to cycle through the display formats. The default view varies for the type of content currently loaded in the content area.

Large icon

Displays the names of the loaded content in large icon format.

Small icon

Displays the names of the loaded content in small icon format.

List view

Displays the names of the loaded content in a list.

Detail view

Displays additional information regarding the loaded content. You can sort the items by name, size, type, and other properties, depending on the type of content that is loaded in the content area.

Refresh (Shortcut menu only)

Refreshes the display in the content area to reflect any changes you have made. Right-click the content area background and click refresh on the shortcut menu.

Autodesk seek design content

Opens a web browser and displays the Autodesk seek (SEEK) home page. Product design information available on Autodesk seek depends on what content providers, both corporate partners and individual contributors, publish to Autodesk seek (SHAREWITHSEEK). Such content could include 3D models, 2D drawings, specifications, brochures, or descriptions of products or components.

Tree view (Design center)

Displays the hierarchy of files and folders on your computer and network drives, a list of open drawings, custom content, and a history of the last locations you accessed. Select an item in the tree view to display its contents in the content area.

Note : In the sample designecenter folder are drawings containing discipline-specific blocks that you can insert in drawings. These drawings are called symbol library drawings.

Use the buttons in the toolbar at the top of design center to access tree view options.

— — — — — — — —

TASK 3 : **Text**

Data

Command alias	Button	Classic menu	Ribbon/application menu
Single line - DT Multiline - T		Draw => Text	Home => Annotation => Text

AutoCAD provides two options to create notes. For simple entries you can use single line text, whereas for longer entries with internal formatting multiline text can be used. The procedures for both these commands are self-explanatory.

Single line text

The single line text is used to enter several lines of text that can be rotated and resized. The text we type can be displayed on the screen. Each line of text is treated as a separate object in AutoCAD.

The following prompts are displayed,

Current text style: <current>

Current text height: <current>

Annotative: <current>

Specify start point of text or [Justify/Style]:

When you pick the start point (also called the insertion point) for the text, the relationship between the start point and actual letters is determined by the justification.

When you want to refer text by using object snaps, we can use insertion object snap.

The specify height prompt is displayed only when the current text style is not annotative and does not have a fixed height. Whereas the specify paper text height prompt is displayed only when the current text style is annotative.

Justify

Be default the text is left justified. Hence there is no option for left justification.

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

Align

Specifies both text height as well as text orientation by designating the endpoints to the baseline. The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.

Fit

Specifies that text fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.

Center

Aligns the text from the horizontal center of the baseline, which you specify with a point. 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, the text is drawn 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. The middle option

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

Right

Text is right justified from the insertion point. The insertion point is on the base line.

TL, TC, TR

Top left, top center, top right justifications. The insertion point is at the top of the highest possible letter, and the text is either left center or right justified accordingly.

ML, MC, MR

Middle left, middle center, middle right justifications. Text is centered vertically. The vertical center point is measured from the bottom of the lower to the top of the tallest possible letter. Text is left, center or right justified accordingly.

BL, BC, BR

Bottom left, bottom center, bottom right justifications. The insertion point is below the lowest descending letter. Text is left, center or right justified accordingly.

Style

Specifies the style of the text, which determines the appearance of the text characters. The text you create uses the current text style. Entering "?" lists the current text styles, associated font files, height, and other parameters.

— — — — — — — —

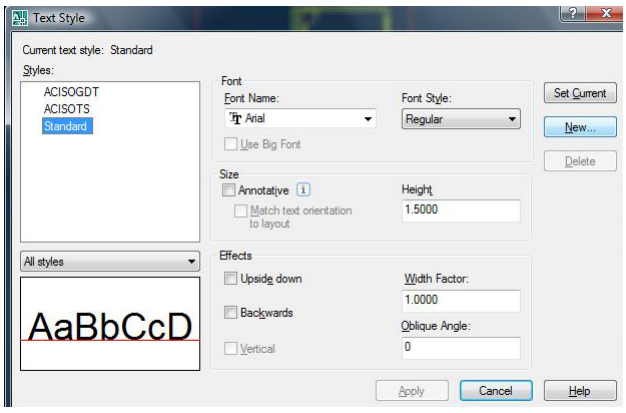
TASK 4 : **Text style**

Command alias	Button	Classic menu	Ribbon/application menu
ST		Format => Text style	Home => Annotation => Text style

The default text style in AutoCAD is standard. This text style is assigned with a default text font txt.shx. Another text style available as default is annotative. If you need to write a text using some other fonts and other parameters you need to use the text editor. It is very difficult to use text editor each and every time you write a text and change its properties. That is why AutoCAD provides an option to modify or to create a new style. After creating a style you can make it as current and the text will be using this current style to write.

The following options are displayed. (Fig 23)

Fig 23



Current text style

Lists the current text style.

Styles

Displays the list of styles in the drawing. An icon before the style name indicates that the style is annotative. Style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign ($), underscore (_), and hyphen (-).

Style list filter

The drop-down list specifies whether all styles or only the styles in use are displayed in the styles list. Preview: Displays sample text that changes dynamically as you change fonts and modify the effects.

Font

Changes the style's font.

> **Note : If you change the orientation or font file of an existing text style, all text objects with that style use the new values when the drawing is regenerated.**

Font name

Lists the font family name for all registered true type fonts and all compiled shapes (SHX) fonts in the fonts folder. When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font.

Font style

Specifies font character formatting, such as italic, bold, or regular. When use big font is selected, this option changes to big font name and is used to select a big font file name.

Use big font

Specifies an Asian language big font file. Only SHX files are valid file types for creating big fonts.

Size

Changes the size of the text.

Annotative

Specifies that the text is annotative. Click the information icon to learn more about the annotative objects.

Match text orientation to layout

Specifies that the orientation of the text in paper space viewports matches the orientation of the layout. This option is unavailable if the annotative option is cleared.

Height or paper text height

Sets the text height based on the value you enter. Entering a height greater than 0.0 sets the text height for this style automatically. If you enter 0.0, the text height defaults to the last text height used, or the value stored in the drawing template file. True type fonts might be displayed at a smaller height than SHX fonts with the same height setting. If the annotative option is selected, the value entered sets the text height in paper space.

Effects

Modifies characteristics of the font, such as its height, width factor, and oblique angle and whether it is displayed upside down, backwards, or vertically aligned.

Upside down

Displays the characters upside down.

Backwards

Displays the characters backwards.

Vertical

Displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for true type fonts.

Width factor

Sets the character spacing. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.

Oblique angle

Sets the oblique angle of the text. Entering a value between 85 and 8 italicizes the text.

Note : Truetype fonts using the effects described in this section might appear bold on the screen. On-screen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.

Set current

Sets the style selected under styles to current.

New

Displays the new text style dialog box and automatically supplies the name "stylen" (where n is the number of the supplied style) for the current settings. You can accept the default or enter a name and choose ok to apply the current style settings to the new style name.

Delete

Deletes unused text styles.

Apply

Applies style changes made in the dialog box to the current style and to the text of the current style in the drawing.

? list styles

Lists text style names and characteristics. The previous prompt is redisplayed until you specify the opposite corner of the text boundary.

Width

It specifies the width of the text boundary. If you use the pointing device to specify a point, the width is calculated as the distance between the start point and the specified point. Words within each line of the multiline text object wrap to fit the width of the text boundary. If you specify a width of 0, word wrap is turned off and the width of the multiline text object is as wide as the longest line of text. You can end a line of text at a specific point by typing the text and pressing enter. To end the command, press enter at the mtext prompt.

Columns

It specifies the column options for an mtext object.

Static

Specifies the total column width, the number of columns, the gutter width (the space between the columns), and the height of columns.

Dynamic

Specifies column width, gutter width and column height. Dynamic columns are text driven. Adjusting columns affect text flow and text flow causes columns to be added or removed.

No columns

Sets no column mode to current mtextobject. The default column setting is stored in the mtextcolumn system variable.

Note : Although the box boundary you specify controls the width of the paragraph, A single word is not broken to adjust the limits. This means that if you write a single word whose width is more than the box boundary specified autocad will write the word irrespective of the box width and therefore, will exceed the boundary limits.

TASK 5: **Scale text**

Command alias	Button	Classic menu
SCALETEXT		Modify => Object => Text => Scale

This tool enlarges or reduces selected text objects without changing their locations.

The following prompts are displayed.

Select objects

Enter a base point option for scaling [Existing/Left/Center/ Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>:

The base point is a location relative to a text object that serves as a fixed point for the resizing or scaling operation, which is applied individually to each selected text object. Specifying a base point for the operation does not change the insertion point relative to the text. The base point options shown above are described in the text command. The base point options for both single line text and multiline text are similar except that the align, fit, and left text options are equivalent to the bottom left (BL) multiline text attachment point.

Specify new model height or [Paper height/Match object/ Scale factor] <0.5000>:

Note : You can only specify a model height for non-annotative objects.

Paper height

Scales the text height depending on the annotative property.

Note : You can only specify a paper height for annotative objects.

Match object

Scales the text objects that you originally selected to match the size of a selected text object.

Note : This option only affects like objects (annotative or nonannotative).

Scale factor

Scales the selected text objects based on a reference length and a specified new length.

Reference

Scales the selected text objects relative to a reference length and a new length. The selected text is scaled by a ratio of the values that you entered for the new length and the reference length. If the new length is less than the reference length, the selected text objects are reduced in size.

— — — — — — — —

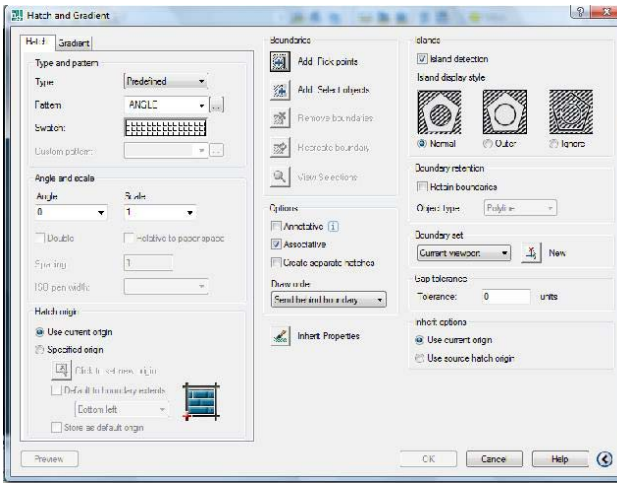
TASK 6 : **Hatch**

Command alias	Button	Classic menu	Ribbon/application menu
H		Draw => Hatch	Home => Draw => Hatch

Drafters and designers use repeating patterns called hatching to fill regions in a drawing for various purposes. In a cross section view hatch patterns helps the viewer differentiate between components of an assembly and indicate the material of each.

The following options are displayed. (Fig 24)

Fig 24



Type and pattern

Specifies the hatch's type, pattern, color, and background color.

Type

Specifies whether to create a predefined, user-defined, or custom hatch pattern.

Predefined patterns are stored in the acad. pator acadiso.pat files supplied with the program.

User-defined patterns are based on the current linetype in your drawing. A custom pattern is a pattern that is defined in any custom PAT files that you have added to the search path.

Pattern

Displays a selection of ANSI, ISO, and other industry-standard hatch patterns. Select SOLID to create solid fill. The pattern option is available only when type is set to predefined. (HPNAME system variable)

[...] Button

Displays the hatch pattern palette dialog box, in which you can preview images for all predefined patterns.

Color

Overrides the current color with a specified color for hatch patterns and solid fills. (HPCOLOR system variable)

Background color

Specifies the background color for new hatch objects. Choose none to turn off the background color. (HPBACKGROUND COLOR system variable)

Swatch

Displays a preview of the selected pattern. Click the swatch to display the hatch pattern palette dialog box.

Custom pattern

Lists the available custom patterns. The most recently used custom patterns appear at the top of the list. The custom pattern option is available only when type is set to custom. (HPNAME system variable)

[...] Button

Displays the hatch pattern palette dialog box, in which you can preview images for all custom patterns.

Angle and scale

Specifies an angle and scale for the selected hatch pattern.

Angle

Specifies an angle for the hatch pattern relative to the X axis of the current UCS. (HPANG system variable)

Scale

Expands or contracts a predefined or custom pattern. This option is available only when type is set to predefined or custom. (HPSCALE system variable)

Double

For user-defined patterns, draws a second set of lines at 90 degrees to the original lines, creating a crosshatch. This option is available only when type is set to user defined. (HPDOUBLE system variable)

Relative to paper space

Scales the hatch pattern relative to paper space units. This allows you to display hatch patterns at a scale that is appropriate for your named layout. This option is available only from a named layout.

Spacing

Specifies the spacing of lines in a user-defined pattern. This option is available only when type is set to user defined. (HPSPACE system variable)

ISO pen width

Scales an ISO predefined pattern based on the selected pen width. This option is available only when type is set to predefined and pattern is set to one of the available ISO patterns.

Hatch origin

Controls the starting location of hatch pattern generation. Some hatches, such as brick patterns, are meant to be aligned with a point on the hatch boundary. By default, all hatch origins correspond to the current UCS origin.

Use current origin

Uses the hatch origin point stored in the HPORIGIN system variable.

Specified origin

Assigns a new hatch origin using the following options.

Click to set new origin

Specifies the new hatch origin point directly.

Default to boundary extents

Calculates a new origin based on the rectangular extents of the boundary for the hatch object. Choices include each of the four corners of the extents and its center. (HPORIGINMODE system variable)

Store as default origin

Stores the value of the new hatch origin in the HPORIGIN system variable.

Boundaries

The following options are displayed.

Pick points

Determines a boundary from existing objects that form an enclosed area around the specified point. While picking internal points, you can right-click in the drawing area at any time to display a shortcut menu that contains several options.

Select

Determines a boundary from selected objects that form an enclosed area.

When you use the select objects option, interior objects are not detected automatically. You must select the objects within the selected boundary to hatch or fill those objects according to the current island detection style. Each time you click select objects, hatch clears the previous selection set. While selecting objects, you can right-click at any time in the drawing area to display a shortcut menu. You can undo the last selection or all selections, change the selection method, change the island detection style, or preview the hatch or gradient fill.

Remove

Removes from the boundary definition any of the objects that were added previously.

Recreate

Creates a polyline or region around the selected hatch or fill, and optionally associates the hatch object with it.

Display boundary objects

Displays the boundary grip controls for the selected hatch so that you can use to grip-edit both the boundary and the hatch object. When you select or use the display boundary objects option to select a non-associative hatch, the hatch boundary grips are displayed.

When you select an associative hatch, it displays only a single grip point of the hatch. To display the boundary grip controls of the objects associated with the hatch, use the display boundary objects option. You can only edit an associative hatch by grip-editing the associated boundary objects.

Options

Controls several commonly used hatch or fill options.

Annotative

Specifies that the hatch is annotative. This property automates the process of scaling annotations so that they plot or display at the correct size on the paper. (HPANNOTATIVE system variable)

Associative

Specifies that the hatch or fill is associative. A hatch or fill that is associative is updated when you modify its boundary objects. (HPASSOC system variable)

Create separate hatches

Controls whether a single hatch object or multiple hatch objects are created when several separate closed boundaries are specified. (HPSEPARATE system variable)

Draw order

Assigns a draw order to a hatch or fill. You can place a hatch or fill behind all other objects, in front of all other objects, either behind the hatch boundary, or in front of the hatch boundary. (HPDRAWORDER system variable)

Layer

Assigns new hatch objects to the specified layer, overriding the current layer. Select use current to use the current layer. (HPLAYER system variable)

Transparency

Sets the transparency level for new hatch or fills, overriding the current object transparency. Select use current to use the current object transparency setting. (HPTRANSPARENCY system variable).

Inherit properties

Hatches or fills specified boundaries using the hatch or fill properties of a selected hatch object. After selecting the hatch object whose properties you want the hatch to inherit, right-click in the drawing area and use the options on the shortcut menu to switch between the select objects and pick internal point options. The HPINHERIT system variable controls whether the hatch origin of the resulting hatch is determined by HPORIGIN or by the source object.

Islands

Specifies the methods used to hatch or fill boundaries within the outermost boundary.

Island detection

Controls detection of internal closed boundaries, also called as islands. (HPISLANDDETECTIONMODE system variable)

— — — — — — — —

TASK 7 : **Gradient**

Command alias	Button	Classic menu
Gradient		Draw => Gradient

A gradient fill is a solid hatch fill that gives the blended color effect of a surface with light on it. The color in gradient fill makes a smooth transition from light to dark or from dark to light. In a two color gradient fill the color with the transition from light color to dark and from first color to the second color. The hatch and gradient dialog box includes the same options.

The following options are only for gradient tab.

Color

Specifies whether to fill the hatch boundary with a monochromatic or two-color blend.

One color

Specifies a fill that uses a smooth transition between a color and a specified tint (the color mixed with white) or between a color and a specified shade (the color mixed with black). (GFCLRSTATE system variable)

Two color

Specifies a fill that uses a smooth transition between two colors. (GFCLRSTATE system variable)

Color swatches

Specifies the colors for the gradient fill (either one color or two colors). Click the browse button [...] to display the select color dialog box, where you can select an AutoCAD color index (ACI) color, true color, or color book color. (GFCLR1 and GFCLR2 system variables)

Shade and tint slider

Specifies the tint (the selected color mixed with white) or shade (the selected color mixed with black) of a color to

be used for a gradient fill of one color. (GFCLRLUM system variable)

Gradient patterns

Displays fixed patterns for gradient fills. These patterns include linear sweep, spherical, and parabolic.

Orientation

Specifies the angle of the gradient and also whether it is symmetrical.

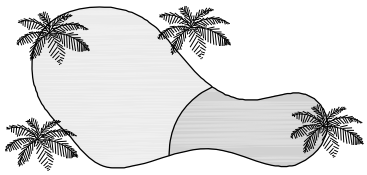
Centered

Specifies a symmetrical gradient configuration. If this option is not selected, the gradient fill is shifted up and to the left, creating the illusion of a light source to the left of the object. (GFSHIFT system variable)

Angle

Specifies the angle of the gradient fill. The specified angle is relative to the current UCS. This option is independent of the angle specified for hatch patterns. (GFANG system variable) (Fig 25)

Fig 17



GRADIENT

AA2110EH

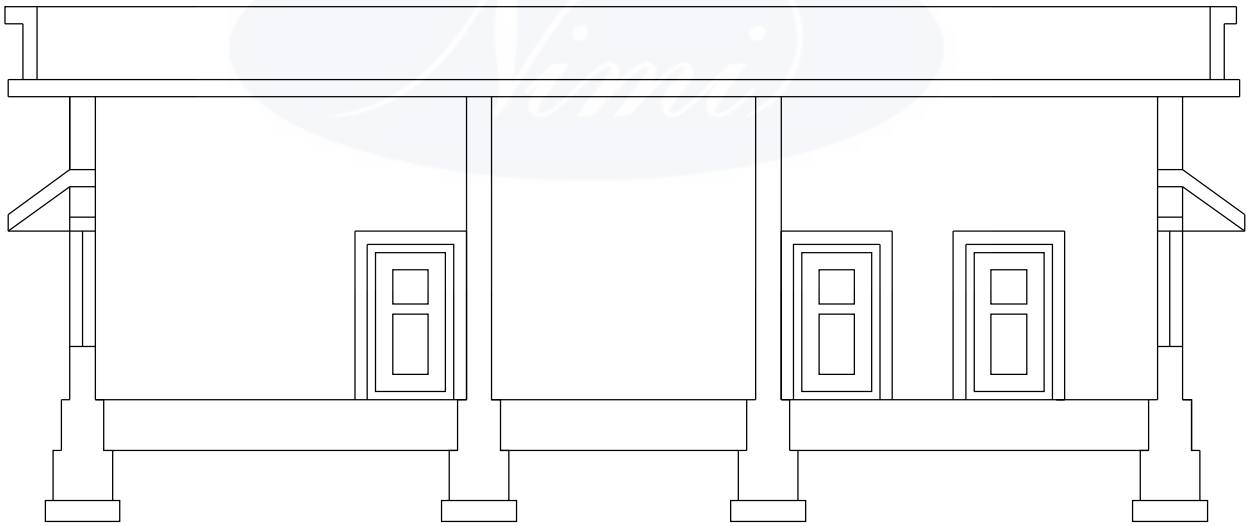
Hands on -1

1 Open the given file section.dwg. (Fig 26)

2 Apply hatch on footing.

i. Command: HATCH

Fig 18



SECTION AA

AA2110EI

ii. Select ANSI32 hatch and click ok.(Fig 27 & Fig 28)

iii. Specify the scale as 25

iv. Choose add pickpoints and click the internal points of walls. (Fig 29)

Fig 27

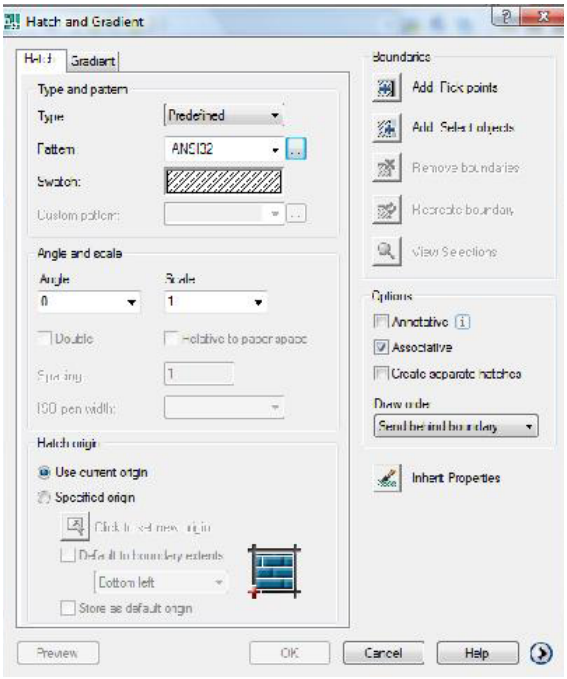


Fig 28

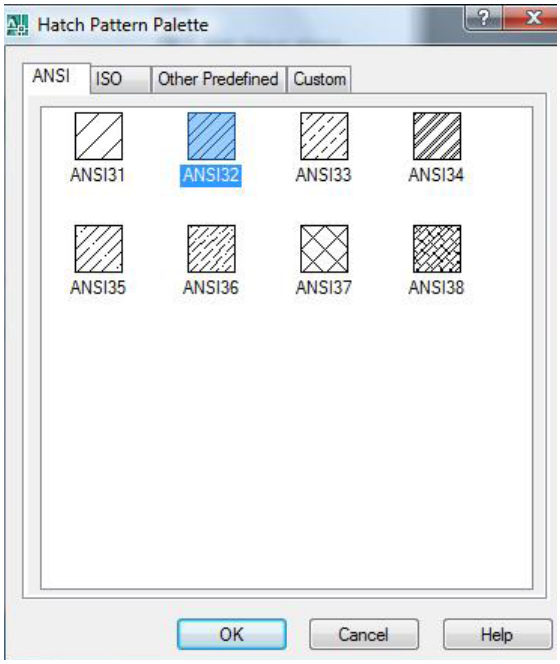
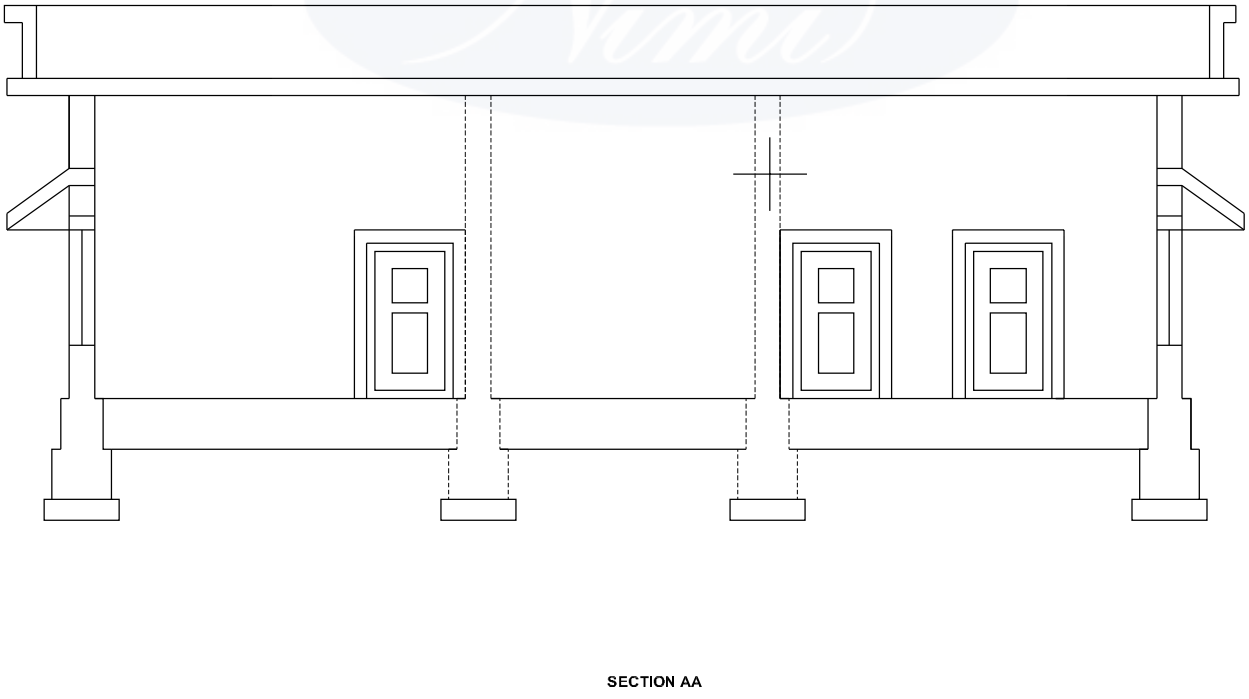


Fig 19



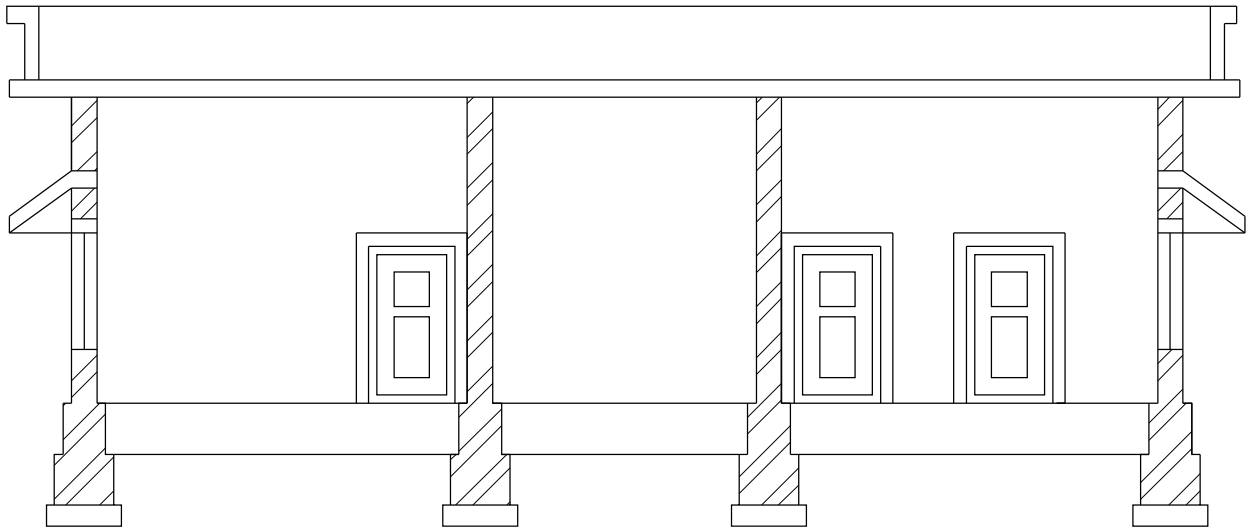
SECTION AA

AA2110EJ

v. Press enter to apply hatch the wall. (Fig 30)

3 Repeat the steps to apply hatches as shown below.

Fig 20



SECTION AA

AA2110EK

Position	Name	Scale
1	ANSI133	25
2	GRAVEL	25
3	AR-CONC	4
4	AR-SAND	10

Advanced drafting commands - III

Objectives: At the end of this exercise you shall be able to

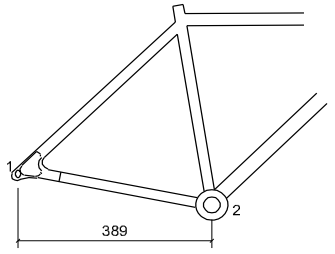
- **understand following advanced command.**
  - **Dimension linear**
  - **Dimension aligned**
  - **Dimension arc length**
  - **Dimension ordinate**
  - **Dimension radius**
  - **Dimension diameter**
  - **Dimension angular**
  - **leader**
  - **Mleader**
  - **Dimension style**
  - **Symbols and arrow tab**
  - **Primary unit tab**
  - **Alternate unit tab**
  - **Tolerance tab**
  - **Layouts**
  - **Model space view ports**
  - **Mview.**

Dimensions

Command alias	Button	Classic menu	Ribbon / Application menu
DIMLIN	⟷	Dimensions => aligned	Annotate => Dimensions => Aligned

As the name suggests the linear dimension commands are used to provide dimensions to straight lines. It measures a distance from one point to another, but it always measures along the x or y axis. To use it, clock the top button on the dimension toolbar (or go to dimension => linear), and then click your first point. After you click you first point, click the second point, and move the cursor away from the part. The length of the extension will be the distance from the actual measurement to you third click.

Fig 1



DIMENSION LINEAR

AD2111E1

The following prompts are displayed. (Fig 1)

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or [Mtext/Text/Angle/ Horizontal/Vertical/Rotated]:

Dimension text = The dimensions will be displayed

Mtext

Displays the In-place text editor, which you can use to edit the dimension text. Use control codes and unicode character strings to enter special characters or symbols. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([]). The current dimension style determines the appearance of the generated measurements.

Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

To include the generated measurement, use angle brackets (<>) which are used to represent them. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([]).

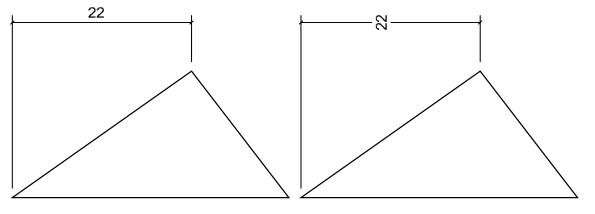
Dimension text properties are set on the text tab of the new, modify, and override dimension style dialog boxes.

Angle

Changes the angle of the dimension text. (Fig 2)

Horizontal

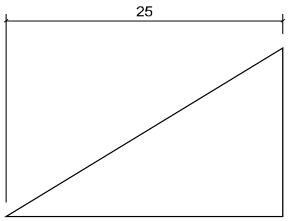
Fig 2



DIM ANGLE

AD2111E2

Fig 3



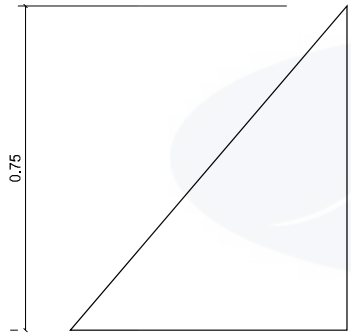
DIMENSION HORIZONTAL

AD2111E3

Creates horizontal linear dimensions. (Fig 3)

Vertical

Fig 4



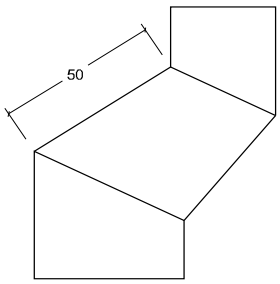
DIMENSION VERTICAL

AD2111E4

Creates vertical linear dimensions. (Fig 4)

Rotated

Fig 5



DIMENSION ROTATED

AD2111E5

Creates rotated linear dimensions. (Fig 5)

Object selection

Automatically determines the origin points of the first and second extension lines after you select an object.

For polylines and other explodable objects, only the individual line and arc segments are given dimensions. You cannot select object in a non-uniformly scaled block reference.

If you select a line or an arc, the line or arc endpoints are used as the origins of the extension lines. The extension lines are offset from the endpoints by the distance you specify in offset from origin in the lines and arrows tab of the new, modify, and override dimension style dialog boxes.

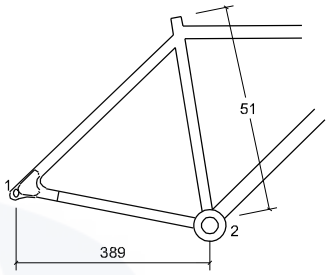
Dimension aligned

Command alias	Button	Classic menu	Ribbon / Application menu
DIMALI		Dimensions => aligned	Annotate => Dimensions => Aligned

Generally, the drawing consists of various objects that are neither parallel to the X axis nor the Y axis. Dimensioning of such objects can be done using aligned dimensioning.

The following prompts are displayed. (Fig 6)

Fig 6



DIMENSION ALIGNED

AD2111E6

Specify first extension line origin or <select object>:

After you specify either manual or automatic extension lines, the following prompt is displayed.

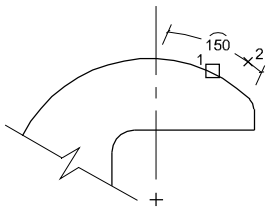
Specify dimension line location or [M text/Text/Angle]:

Dimension arc length

Command alias	Button	Classic menu	Ribbon / Application menu
DIMARC		Dimensions => Arc length	Annotate => Dimensions => Arc length

The arc length dimensioning is used to dimension the length of an arc or the polyline arc segment. You are required to select an arc or a ploy line arc segment and the dimension location. (Fig 7)

Fig 7



ARC LENGTH

AD2111E7

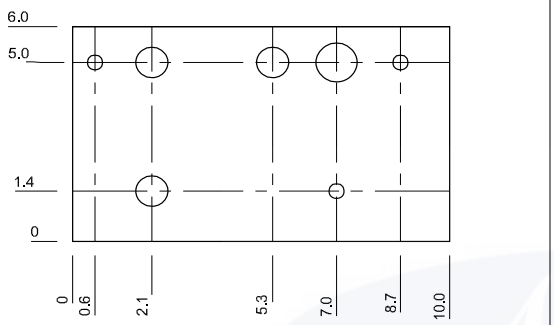
The following prompts are displayed.
Select arc or polyline arc segment:
Specify arc length dimension location or [Mtext/Text/Angle/Partial/Leader]:

Dimension ordinate

Command alias	Button	Classic menu	Ribbon / Application menu
DIMARODINATE		Dimensions => Ordinate	Annotate => Dimensions => Ordinate

Ordinate dimensions are not really dimensions at all in that they do not indicate a measurement. Rather they annotate known co-ordinate points. The DIMORDINATE command is used to indicate the X and Y ordinate values at any point. (Fig 8)

Fig 8



The following prompts are displayed.
Specify feature location:
Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]:

Angle

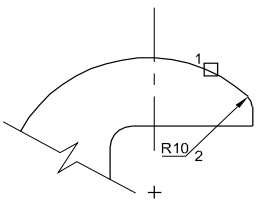
Changes the angle of the dimension text.

Dimension radius

Command alias	Button	Classic menu	Ribbon / Application menu
DIMRADIUS		Dimensions => Radius	Annotate => Dimensions => Linear => Radius

The radius dimensioning is used to dimension a circle or an arc. Radius and diameter dimensioning are similar; the only difference is that instead of dimension line, a radius line is drawn, which is measured from the center to any point on the circumference. (Fig 9)

Fig 9



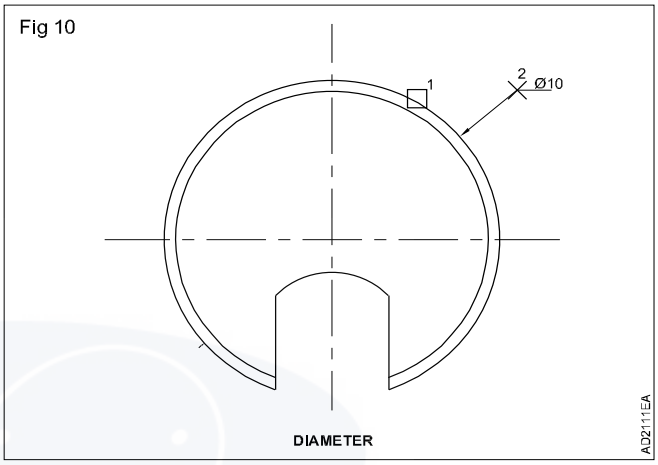
Dimension diameter

Command alias Button Classic menu Ribbon/Application menu

Command alias	Button	Classic menu	Ribbon / Application menu
DIMDIAMETER		Dimensions => Diameter	Annotate => Dimensions => Linear => Diameter

Diameter dimensioning is used to dimension circle or an arc. Here the measurement is done between two diametrically opposite points on the circumference of the circle or an arc. The dimension text is generated in AutoCAD commence with symbol to indicate diameter dimension. (Fig 10)

Fig 10

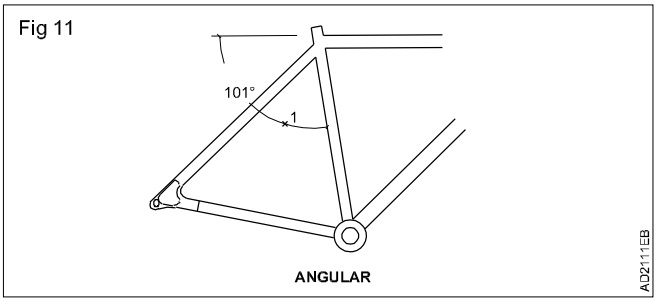


Dimension angular

Command alias	Button	Classic menu	Ribbon / Application menu
DIMANG		Dimensions => Angular	Annotate => Dimensions => Linear => Angular

The angular dimensioning is used when you want to dimension an angle. This command generates a dimension arc. The location of the dimension line and dimension arc is determined by the way you locate the dimension arc. We can place dimension text outside the quadrant in which you measure the angle by extending the dimension arc. Choose the quadrant option from the short cut menu. Next you will be prompted to specify the quadrant in which you want to measure the angle. Then specify the quadrant with the mouse. (Fig 11)

Fig 11



Dimensioning the angle of arc

Angular dimensioning can also be used to provide a dimension to the angle of an arc. In this case the center point of the arc is considered as the vertex and end point of arc is the extension line origin.

Dimension jogged

Command alias	Button	Classic menu	Ribbon / Application menu
DIMJOGGED		Dimensions => Jogged	Annotate => Dimensions =>Jogged

The necessity of the jogged dimension arises because of the space constraint. Also the jogged dimension is used when you want to avoid merging of the dimension line with other dimensions. Also there are instances when it is not possible to show the centre of circle in the sheet. In such situations, jogged dimensions are of immense use. Jogged dimensions are added using the jogged tool. Note that with this command you can add dimensions to only jogged radius dimensions.

Leader

Command alias	Button
LEADER	

The leader command can be used to annotate any point on a drawing. Unlike other dimension commands the leader and annotation text are drawn as separate objects. So, if you need to move or edit the text, you can do so without affecting the leader line.

The following prompts are displayed.

Specify leader start point:

Specify next point:

A leader line segment is drawn and prompts for points and options are displayed.

Specify next point of [Annotation/Format/Undo] <Annotation>:

Mleader

Command alias	Button	Classic menu	Ribbon / Application menu
MLEADER		Dimensions => Mleader	Annotate => Leaders =>Multi leader

The following prompts are displayed.

Specify leader arrowhead location or [leader landing first/ Content first/Options] <Options>:

Leader Arrowhead first

Specifies a location for the arrowhead of the multileader object.

Specify leader landing location

Sets placement of the leader landing for the new multileader object.

If you exit the command at this point, then no text is associated with the multileader object.

Leader landing first

Specifies a location for the landing line of the multileader object.

Specify leader arrowhead location

Sets placement of the arrowhead for the new multileader object.

If you exit the command at this point, then no text is associated with the multileader line.

Content first

Specifies a location for the text or block associated with the multileader object.

Point selection

Sets placement for the text box for the text label associated with the multileader object. When you finish entering the text, click ok or click outside the text box.

If you choose end at this point, then no landing line is associated with the multileader object.

Options

Specifies options for placing the multileader object.

Leader type

Specifies a straight, spline, or no leader line.

- Straight.
- Spline.
- None.

Leader landing

Changes the distance of the horizontal landing line

- Yes.
- No.

If you choose no at this point, then no landing line is associated with the multileader object.

Content type

Specifies the type of content that will be used for the multileader.

- Block
- Mtext
- None

Leader

Specify leader association point <next>: Specify an object snap location, or press enter to skip to the next dimension object, if any

Ordinate

Specify feature location <next>: Specify an object snap location, or press enter to skip to the next dimension object, if any

Radius

Select arc or circle <next>: Select an arc or a circle, or press enter to skip to the next dimension object, if any

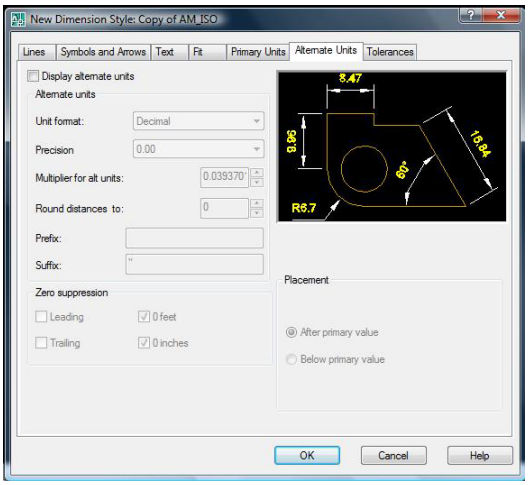
Dimension style

Command alias	Button	Classic menu	Ribbon / Application menu
DS		Dimensions => Dimension style	Home => Annotation =>Dimension style

The dimension style command can be used to change the appearance of dimensions. The best method is to create a new style before creating dimensions so that you can leave the STANDARD style as a default option. Having created a new style from STANDARD you can then apply any modifications you generally require to the parent style and then more specific modifications to the child styles in order to create a style family.

Dimension styles are created using the dimension styles dialogue box. As you can see from the dialogue box, a style is applied to a family of dimensions. By default, any style changes are made to the parent. Each style parent has six child styles. The child styles, linear, radial, angular, diameter, ordinate and leader can be used to modify the parent style when that particular dimension is used. For example, you may like to use a tick rather than an arrow head for your dimensions but this isn't really appropriate for a leader, so the leader child style can be changed so that leaders will always be drawn with an arrow head whilst all other dimensions of the same style family are drawn using ticks. (Fig 12)

Fig 12



The following options are displayed.

Current dimension style

Displays the name of the dimension style that is current. The default dimension style is STANDARD. The current style is applied to dimensions you create.

Styles

Lists dimension styles in the drawing. The current style is highlighted. Right-click in the list to display a shortcut menu with options to set the current style, rename styles, and delete styles. You cannot delete a style that is current or in use in the current drawing. An icon before the style name indicates that the style is annotative.

Unless you select the option, 'Don't list styles in Xrefs', dimension styles are displayed in externally referenced drawings using the syntax for externally referenced named objects. Although you cannot change, rename, or make current externally referenced dimension styles, you can create new styles based on them.

The item selected in list controls the dimension styles being displayed.

List

Controls the display of styles in the styles list. Select all styles if you want to see all dimension styles in a drawing. Select styles in use if you want to see only the dimension styles being currently used by the dimensions in the drawing.

Don't list styles in Xrefs

When selected, suppresses display of dimension styles in externally referenced drawings in the styles list.

Preview

Shows a graphic representation of the style selected in the styles list.

Description

Describes the style selected in the styles list relative to the current style. If the description is longer than the space provided, you can click in the pane and use arrow keys to scroll down.

Set current

Sets the style selected under styles to current. The current style is applied to dimensions you create.

New

Displays the create new dimension style dialog box, in which you can define a new dimension style.

Modify

Displays the modify dimension styles dialog box, in which you can modify dimension styles. Dialog box options are identical to those in the new dimension style dialog box.

Override

Displays the override current style dialog box, in which you can set temporary overrides to dimension styles. Dialog box options are identical to those in the new

dimension style dialog box. Overrides are displayed as unsaved changes under the dimension style in the styles list.

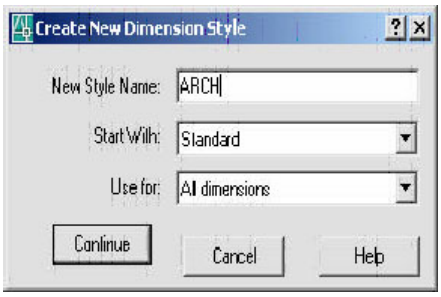
Compare

Displays the compare dimension styles dialog box, in which you can compare two dimension styles or list all the properties of one dimension style.

New

Names the new dimension style, sets the style on which to start the new one, and indicates the dimension types to which you want the new style to apply. (Fig 13)

Fig 13



The following options are displayed.

New style name

Specifies the new dimension style name.

Start with

Sets a style to use as a basis for the new one. For the new style, you need to change only the properties that differ from the properties you start with.

Annotative

Specifies that the dimension style is annotative.

Use for

Creates a dimension substyle that applies only to specific dimension types. For example, you could create a version of the STANDARD dimension style to be used only for diameter dimensions.

Continue

Displays the new dimension style dialog box, in which you define the new dimension style properties.

Lines tab

The following options are displayed. (Fig 14)

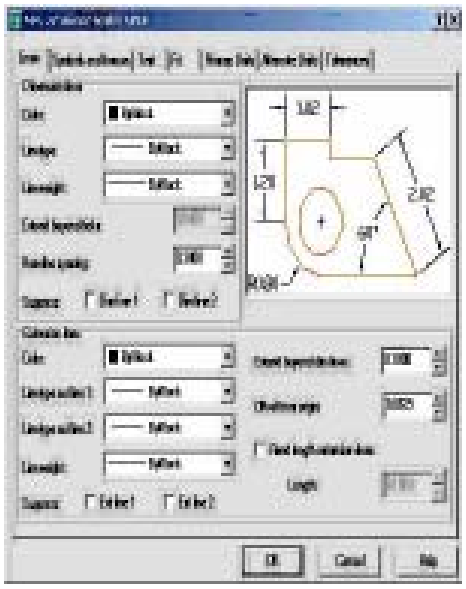
Dimension lines

Color

Displays and sets the color for the dimension line. If you click select color (at the bottom of the color list), the select color dialog box is displayed. You can also enter a color name or number. (DIMCLRD system variable)

You can select colors from the 255 AutoCAD color index (ACI) colors, true colors, and color book colors.

Fig 14



Line type

Sets the linetype of the dimension line. (DIMLTYPE system variable)

Lineweight

Sets the lineweight of the dimension line. (DIMLWD system variable)

Extend beyond ticks

Specifies a distance to extend the dimension line past the extension line when you use oblique, architectural, tick, integral, and no marks for arrowheads. (DIMDLE system variable)

Baseline spacing

Sets the spacing between the dimension lines of a baseline dimension. Enter a distance. (DIMDLI system variable)

Suppress

Suppress display of dimension lines. Dim line 1 suppresses the first dimension line; dim line 2 suppresses the second dimension line. (DIMSD1 and DIMSD2 system variable)

Extension lines

Controls the appearance of the extension lines.

Color

Sets the color for the extension line. If you click select color (at the bottom of the color list), the select color dialog box is displayed. You can also enter a color name or number. (DIMCLRE system variable)

You can select colors from the 255 AutoCAD color index (ACI) colors, true colors, and color book colors.

Linetype Ext line 1

Sets the linetype of the first extension line. (DIMLTEX1 system variable)

Linetype Ext line 2

Sets the linetype of the second extension line. (DIMLTEX2 system variable)

Line weight

Sets the lineweight of the extension line. (DIMLWE system variable)

Suppress

Suppress the display of extension lines. Ext line 1 suppresses the first extension line; ext line 2 suppresses the second extension line. (DIMSE1 and DIMSE2 system variables)

Extend beyond dim lines

Specifies a distance to extend the extension lines above the dimension line. (DIMEXE system variable)

Offset from origin

Sets the distance to offset the extension lines from the points on the drawing that define the dimensions. (DIMEXO system variable)

Fixed length extension lines

Enables fixed length extension lines. (DIMFXLON system variable)

Length

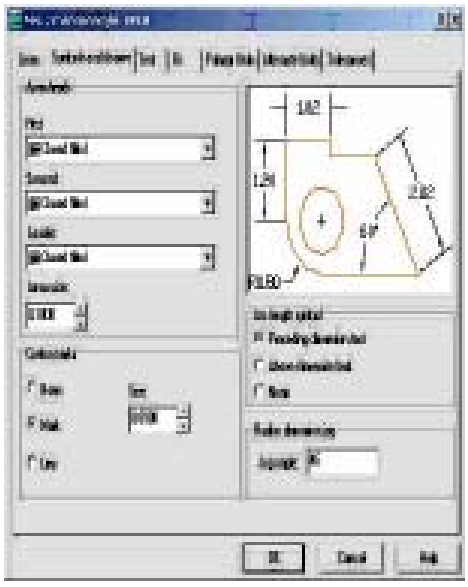
Sets the total length of the extension lines starting from the dimension line toward the dimension origin. (DIMFXL system variable)

Preview

Display sample dimension images that show the effects of changes you make to dimension style settings.

Symbols and arrows tab (Fig 15)

Fig 15



Arrowheads

First

Sets the arrowhead for the first dimension line. When you change the first arrowhead type, the second arrowhead automatically changes to match it. (DIMBLK1 system variable)

To specify a user-defined arrowhead block, select user arrow. The select custom arrow block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing)

Second

Sets the arrowhead for the second dimension line. (DIMBLK2 system variable)

To specify a user-defined arrowhead block, select user arrow. The select custom arrow block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing)

Leader

Sets the arrowhead for the leader line. (DIMLDRBLK system variable)

To specify a user-defined arrowhead block, select user arrow. The select custom arrow block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing)

Arrow size

Displays and sets the size of arrowheads. (DIMASZ system variable)

Note : Annotative blocks cannot be used as custom arrowheads for dimensions or leaders.

Center marks

Controls the appearance of center marks and centerlines for diameter and radial dimensions. The DIMCENTER, DIMDIAMETER, and DIMRADIUS commands use center marks and centerlines. For DIMDIAMETER, and DIMRADIUS, the center mark is drawn only if you place the dimension line outside the circle or arc.

Type

Sets the type of center mark or line to use.

None

Creates no center mark or centerline. The value is stored as 0 in the DIMCEN system variable.

Mark

Creates a center mark. The size of the center mark is stored as a positive value in the DIMCEN system variable.

Line

Creates a centerline. The size of the centerline is stored as a negative value in the DIMCEN system variable.

Size

Displays and sets the size of the center mark or centerline. (DIMCEN system variable)

Dimension break

Controls the gap width of dimension breaks.

Break size

Displays and sets the size of the gap used for dimension breaks.

Arc length symbol

Controls the display of the arc symbol in an arc length dimension. (DIMARCSYM system variable)

Preceding dimension text

Places arc length symbols above the dimension text. (DIMARCSYM system variable)

Above dimension text

Places arc length symbols above the dimension text. (DIMARCSYM system variable)

None

Suppresses the display of arc length symbols. (DIMARCSYM system variable)

Radius jog dimension

Controls the display of jogged (zigzag) radius dimensions.

Jogged radius dimensions are often created when the center point of a circle or arc is located off the page.

Jog angle

Determines the angle of the transverse segment of the dimension line in a jogged radius dimension. (DIMJOGANG system variable)

Linear jog dimension

Controls the display of the jog for linear dimensions.

Jog lines are often added to linear dimensions when the actual measurement is not accurately represented as the dimension. Typically the actual measurement is smaller than the desired value.

Jog height factor

Determines the height of the jog, which is determined by the distance between the two vertices of the angles that make up the jog.

Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

Text Tab (Fig 16)

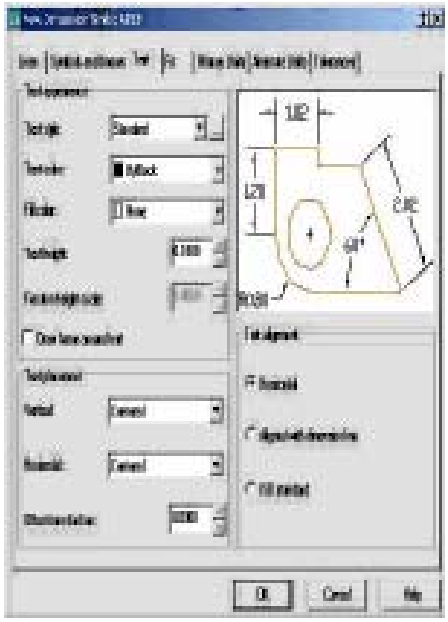
Text appearance

It controls the dimension text format and size.

Text style

Lists the available text styles.

Fig 16



Text style button

Displays the text style dialog box where you can create or modify text styles. (DIMTXSTY system variable)

Text color

Sets the color for the dimension text. If you click select color (at the bottom of the color list), the select color dialog box is displayed. You can also enter color name or number. (DIMCLRT system variable)

You can select colors from the 255 AutoCAD color Index (ACI) colors, true colors, and color book colors.

Fill color

Sets the color for the text background in dimensions. If you click select color (at the bottom of the color list), the select color dialog box is displayed. You can also enter color name or number. (DIMTFILL and DIMTFILLCLR system variables)

You can select colors from the 255 AutoCAD color index (ACI) colors, true colors, and color book colors.

Text height

Sets the height of the current dimension text style. Enter a value in the text box. If a fixed text height is set in the text style (that is, the text style height is greater than 0), that height overrides the text height set here. If you want to use the height set on the text tab, make sure the text height in the text style is set to 0. (DIMTXT system variable)

Fraction height scale

Sets the scale of fractions relative to dimension text. This option is available only when fractional is selected as the unit format on the primary units tab. The value entered here is multiplied by the text height to determine the height of dimension fractions relative to dimension text. (DIMTFAC system variable)

Draw frame around text

When selected, draws a frame around dimension text. Selecting this option changes the value stored in the DIMGAP system variable to a negative value.

Text placement

Controls the placement of dimension text.

Vertical

Controls the vertical placement of dimension text in relation to the dimension line. (DIMTAD system variable)

Vertical position options include the following

- **Centered :** Centers the dimension text between the two parts of the dimension line.
- **Above:** Places the dimension text above the dimension line. The distance from the dimension line to the baseline of the lowest line of text is the current text gap. See the offset from dim line option.
- **Outside :** Places the dimension text on the side of the dimension line farthest away from the first defining point.
- **JIS :** Places the dimension text to conform to a Japanese Industrial Standard (JIS) representation.
- **Below :** Places the dimension text under the dimension line. The distance from the dimension line to the baseline of the lowest line of text is the current text gap. See the offset from dim line option.

Horizontal

Controls the horizontal placement of dimension text along the dimension line, in relation to the extension lines. (DIMJUST system variable)

Horizontal position options include the following:

- **Centered :** Centers the dimension text along the dimension line between the extension lines.
- **At ext line 1 :** Left-justifies the text with the first extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value. See arrowheads and offset from dim line.
- **At ext line 2 :** Right-justifies the text with the second extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value. See arrowheads and offset from dim line.
- **Over Ext line 1 :** Positions the text over or along the first extension line.
- **Over Ext line 2 :** Positions the text over or along the Second extension line.

View direction

Controls the dimension text viewing direction. (DIMTXTDIRECTION system variable)

View direction includes the following options:

- Left-to-right: Places the text to enable reading from left to right.
- Right-to-left: Places the text to enable reading from right to left.

Offset from dim line

Sets the current text gap, which is the distance around the dimension text when the dimension line is broken to accommodate the dimension text.

This value is also used as the minimum length required for dimension line segments.

Text is positioned inside the extension lines only if the resulting segments are at least as long as the text gap. Text above or below the dimension line is placed inside only if the arrowheads, dimension text, and a margin leave enough room for the text gap. (DIMGAP system variable)

Text alignment

Controls the orientation (horizontal or aligned) of dimension text whether it is inside or outside the extension lines. (DIMTIH and DIMTOH system variables)

Horizontal

Places text in a horizontal position.

Aligned with dimension line

Aligns text with the dimension line.

ISO standard

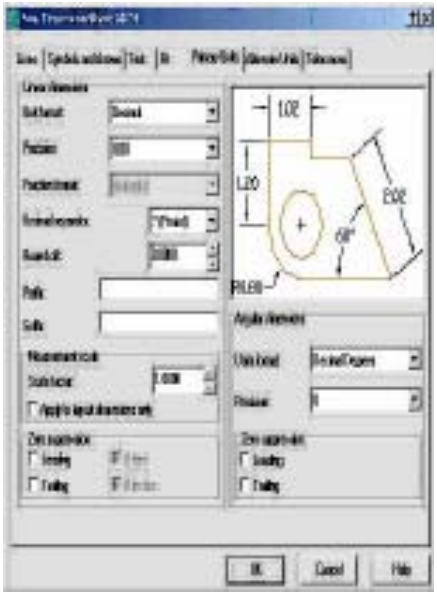
Aligns text with the dimension line when text is inside the extension lines, but aligns it horizontally when the text is outside the extension lines.

Preview

Displays sample dimension images that show the effects of changes you make to the dimension style settings.

Fit tab (Fig 17)

Fig 17



Fit options

Controls the placement of text and arrowheads based on the space available between the extension lines. When space is available, text and arrowheads are placed between the extension lines. Otherwise, text and arrowheads are placed according to the fit options. (DIMATFIT, DIMTIX, and DIMSOXD system variables)

Either text or arrows (Best fit)

Moves either the text or the arrowheads outside the extension lines based on the best fit (DIMATFIT system variable).

- When enough space is available for text and arrowheads, this option places both between the extension lines. Otherwise, either the text or the arrowheads are moved based on the best fit.
- When enough space is available for just the text alone, places text between the extension lines and places arrowheads outside the extension lines.
- When enough space is available for just arrowheads alone, the option places them between the extension lines and places text outside the extension lines.
- When space is available for neither the text nor the arrowheads, it places them both outside the extension lines.

Arrows

Moves the arrowheads outside the extension lines first, followed by the text (DIMATFIT system variable).

- When enough space is available for both the text and the arrowheads, it places both between the extension lines.
- When space is available for just the arrowheads alone, this option places them between the extension lines and places the text outside them.
- When enough space is not available for the arrowheads, the option places both text and arrowheads outside the extension lines.

Text

Moves text outside the extension lines first, followed by the arrowheads (DIMATFIT system variable).

- When space is available for both the text and the arrowheads, it places both between the extension lines.
- When space is available for just the text alone, the option places the text between the extension lines, whereas the arrowheads are placed outside them.
- When enough space is not available for the text, it places both the text as well as the arrowheads outside the extension lines.

Both text and arrows

When enough space is not available for both the text and the arrowheads, this option moves both outside the extension lines (DIMATFIT system variable).

Always keep text between ext lines

Always places the text between extension lines. (DIMTIX system variable)

Suppress arrows if they don't fit inside extension lines

Suppresses the arrowheads if not enough space is available inside the extension lines. (DIMSOXD system variable)

Text placement

Sets the placement of dimension text when it is moved from the default position, that is, the position defined by the dimension style. (DIMTMOVE system variable)

Beside the dimension line

If selected, moves the dimension line whenever dimension text is moved. (DIMTMOVE system variable)

Over the dimension line, with leader

If selected, the dimension lines are not moved while the text is being moved. If the text is moved away from the dimension line, a leader line is created connecting the text to the dimension line. The leader line is omitted when the text is too close to the dimension line. (DIMTMOVE system variable)

Over the dimension line, without leader

If selected, dimension lines are not moved while the text is being moved. The text moved away from the dimension line is not connected to the dimension line with a leader. (DIMTMOVE system variable)

Scale for dimension features

Sets the overall dimension scale value or the paper space scaling.

Annotative

Specifies that the dimension is annotative. Click the information icon to learn more about annotative objects.

Scale dimensions to layout

Determines a scale factor based on the scaling between the current model space viewport and paper space. (DIMSCALE system variable)

When you work in paper space, but not in a model space viewport, or when TILEMODE is set to 1, the default scale factor of 1.0 is used or the DIMSCALE system variable.

Use overall scale of

Sets a scale for all dimension style settings that specify size, distance, or spacing, including text and arrowhead sizes. This scale does not change dimension measurement values. (DIMSCALE system variable)

Fine tuning

Provides additional options for placing dimension text.

Place text manually

Ignores any horizontal justification settings and places the text at the position you specify upon the dimension line location prompt. (DIMUPT system variable)

Draw dim line between ext line

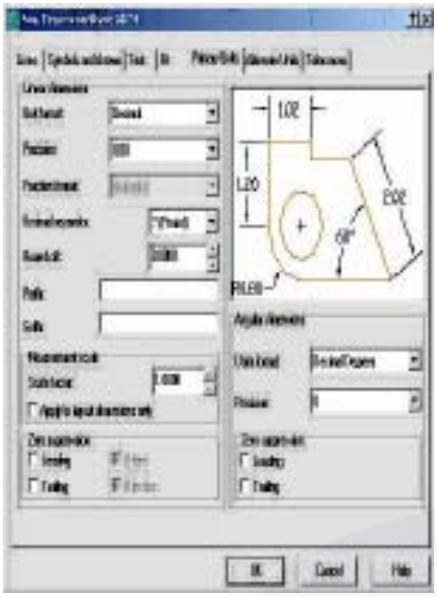
Draws dimension lines between the measured points even when the arrowheads are placed outside the measured points. (DIMTOFL system variable)

Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

Primary Units Tab (Fig 18)

Fig 18



Linear dimensions

Sets the format and precision for linear dimensions.

Unit format

Sets the current units format for all dimension types except angular. (DIMLUNIT system variable).

The relative sizes of numbers in stacked fractions are based on the DIMTFAC system variable (in the same way that tolerance values use this system variable).

Precision

Displays and sets the number of decimal places in the dimension text. (DIMDEC system variable)

Fraction format

Sets the format for fractions. (DIMFRAC system variable)

Decimal separator

Sets the separator for decimal formats. (DIMDSEP system variable)

Round off

Sets rounding rules for dimension measurements for all dimension types except angular. If you enter a value of 0.25, all distances are rounded to the nearest 0.25 unit. If you enter a value of 1.0, all dimension distances are rounded to the nearest integer. The number of digits displayed after the decimal point depends on the precision settings. (DIMRND system variable)

Prefix

Includes a prefix in the dimension text. You can either enter a text or use control codes to display special symbols. For example, entering the control code %%c displays the diameter symbol. When you enter a prefix, it overrides any default prefixes such as those used in diameter and radius dimensioning. (DIMPOST system variable)

If you specify tolerances, the prefix is added to the tolerances as well as to the main dimension.

Suffix

Includes a suffix in the dimension text. You can enter text or use control codes to display special symbols. For example, entering the text mm results in dimension text similar to that shown in the illustration. When you enter a suffix, it overrides any default suffixes. (DIMPOST system variable)

If you specify tolerances, the suffix is added to the tolerances as well as to the main dimension.

Measurement scale

Defines linear scale options. Applies primarily to legacy drawings.

Scale factor

Sets a scale factor for linear dimension measurements. It is recommended that you do not change this value from the default value of 1.00. For example, if you enter 2, the dimension for a 1-inch line is displayed as two inches. The value does not apply to angular dimensions and is not applied to rounding values or to the plus or minus tolerance values. (DIMLFAC system variable)

Apply to layout dimensions only

Applies the measurement scale factor only to dimensions created in layout viewport. Except while using for non-associative dimensions, this setting should remain unchecked. (DIMLFAC system variable)

Zero suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (DIMZIN system variable)

Leading

Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes. 5000. Select leading to enable display of dimension distances less than one unit in sub units.

Sub-units factor

Sets the number of sub units to a unit. It is used to calculate the dimension distance in a sub unit when the distance is less than one unit. For example, enter 100 if the suffix is m and the sub-unit suffix is to display in cm.

Sub-unit suffix

Includes a suffix to the dimension value sub unit. You can either enter a text or use control codes to display special

symbols. For example, enter cm for .96m to display as 96 cm.

Trailing

Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

0 feet

Suppresses the feet portion of a feet-and-inches dimension when the distance is less than one foot. For example, 0*-6 ½ " becomes 6 ½ ".

0 inches

Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1*-0" becomes 1*.

Angular dimensions

Displays and sets the current angle format for angular dimensions.

Units format

Sets the angular units format. (DIMAUNIT system variable)

Precision

Sets the number of decimal places for angular dimensions. (DIMADEC system variable)

Zero suppression

Controls the suppression of leading and trailing zeros. (DIMAZIN system variable)

Leading

Suppresses leading zeros in angular decimal dimensions. For example, 0.5000 becomes .5000.

You can also display dimension distances less than one unit in sub units.

Trailing

Suppresses trailing zeros in angular decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

Alternate units tab (Fig 19)

Display alternate units

Adds alternate measurements units to dimension text. Sets the DIMALT system variable to 1.

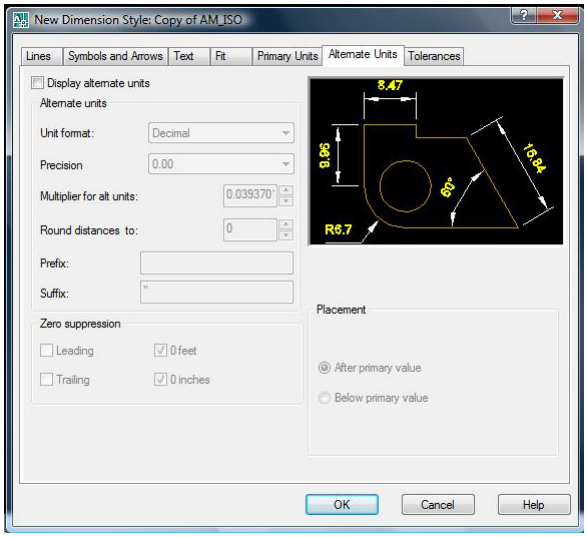
Alternate units

Displays and sets the current alternate units format for all dimension types except angular.

Unit format

Sets the unit format for alternate units. (DIMALTU system variable)

Fig 19



The relative sizes of numbers in stacked fractions are based on DIMTFAC (in the same way that tolerance values use this system variable).

Precision

Sets the number of decimal places for alternate units. (DIMALTD system variable)

Multiplier for alt units

Specifies the multiplier used as the conversion factor between primary and alternate units. For example, to convert inches to millimeters, enter 25.4. The value has no effect on angular dimensions, and it is not applied to the rounding value or the plus or minus tolerance values. (DIMALTF system variable)

Round Distance to

Sets rounding rules for alternate units for all dimension types except angular. If you enter a value of 0.25, all alternate measurements are rounded to the nearest 0.25 unit. If you enter a value of 1.0, all dimension measurements are rounded to the nearest integer. The number of digits displayed after the decimal point depends on the precision setting. (DIMALTRND system variable)

Prefix

Includes a prefix in the alternate dimension text. You can either enter text or use control codes to display special symbols. For example, entering the control code %%c displays the diameter symbol. (DIMAPOST system variable)

Suffix

Includes a suffix in the alternate dimension text. You can either enter text or use control codes to display special symbols. For example, entering the text cm results in dimension text similar to that shown in the illustration. When you enter a suffix, it overrides any default suffixes. (DIMAPOST system variable)

Zero Suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (DIMALTZ system variable)

Leading

Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes. 5000.

Sub-units factor

Sets the number of sub units to a unit. It is used to calculate the dimension distance in a sub unit when the distance is less than one unit. For example, enter 100 if the suffix is m and the sub-unit suffix is to display in cm.

Sub-unit suffix

Includes a suffix to the dimension value sub unit. You can enter the text or use control codes to display special symbols. For example, enter cm for .96m to display as 96 cm.

Trailing

Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

0 Feet

Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0*-6 1/2" becomes 6 ½ ".

0 Inches

Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1*-0" becomes 1*.

Placement

Controls the placement of alternate units in the dimension text.

After primary value

Places alternate units after the primary units in the dimension text.

Below primary value

Places alternate units below the primary units in the dimension text.

Preview

Displays sample dimension image that show the effects of changes you make to dimension style settings.

Tolerances tab (Fig 20)

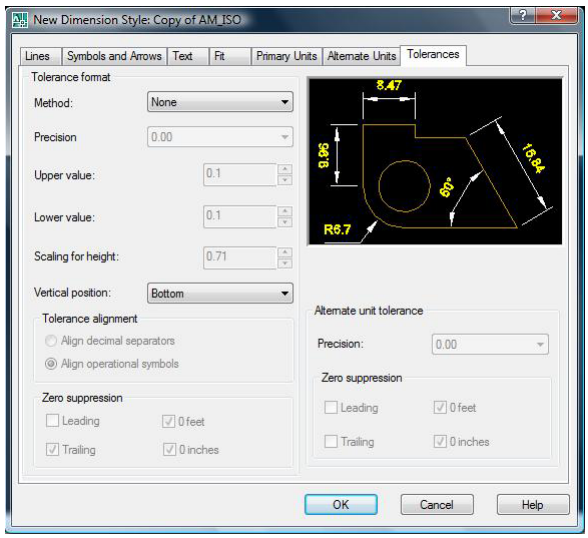
Tolerance format

Controls the tolerance format.

Method

Sets the method for calculating the tolerance. (DIMTOL system variable)

Fig 20



- **None :** this feature does not add a tolerance. The DIMTOL system variable is set to 0.
- **Symmetrical :** Adds a plus/minus expression of tolerance in which a single value of variation is applied to the dimension measurement. A plus-or-minus sign appears after the dimension. Enter the tolerance value in upper value. The DIMTOL system variable is set to 1. The DIMLIM system variable is set to 0.
- **Deviation :** Adds a plus/minus tolerance expression. Different plus and minus values of variation are applied to the dimension measurement. A plus sign (+) precedes the tolerance value entered in upper value, and a minus sign (-) precedes the tolerance value entered in lower value. The DIMTOL system variable is set to 1. The DIMLIM system variable is set to 0.
- **Limits :** Creates a limit dimension. A maximum and a minimum value are displayed, one over the other. The maximum value is the dimension value plus the value entered in upper value. The minimum value is the dimension value minus the value entered in lower value. The DIMTOL system variable is set to 0. The DIMLIM system variable is set to 1.
- **Basic:** Creates a basic dimension, which displays a box around the full extents of the dimension. The distance between the text and the box is stored as a negative value in the DIMGAP system variable.

Precision

Sets the number of decimal places. (DIMTDEC system variable)

Upper value

Sets the maximum or upper tolerance value. When you select symmetrical in method, this value is used for the tolerance. (DIMTP system variable)

Lower value

Sets the minimum or lower tolerance value. (DIMTM system variable)

Scaling for height

Sets the current height for the tolerance text. The ratio of the tolerance height to the main dimension text height is calculated and stored in the DIMTFAC system variable.

Vertical position

Controls text justification for symmetrical and deviation tolerances.

- **Top :** Aligns the tolerance text with the top of the main dimension text. When you select this option, the DIMTOLJ system variable is set to 2.
- **Middle :** Aligns the tolerance text with the middle of the main dimension text. When you select this option, the DIMTOLJ system variable is set to 1.
- **Bottom :** Aligns the tolerance text with the bottom of the main dimension text. When you select this option, the DIMTOLJ system variable is set to 0.

Tolerance alignment

Controls the alignment of upper and lower tolerance values when stacked.

Align decimal separators

Values are stacked by their decimal separators.

Align operational symbols

Values are stacked by their operational symbols.

Zero suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (DIMTZIN system variable)

Leading

Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

Trailing

Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

0 Feet

Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0*-6 ½ " becomes 6 1/2".

0 Inches

Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1*-0" becomes 1^.

Alternate unit tolerance

Formats alternate tolerance units.

Precision

Displays and sets the number of decimal places. (DIMALTTD system variable)

Zero suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (DIMALTTZ system variable)

Leading

Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

Trailing

Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

0 Feet

Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0**-6 ½ " becomes 6 1/2".

0 Inches

Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1*-0" becomes 1^.

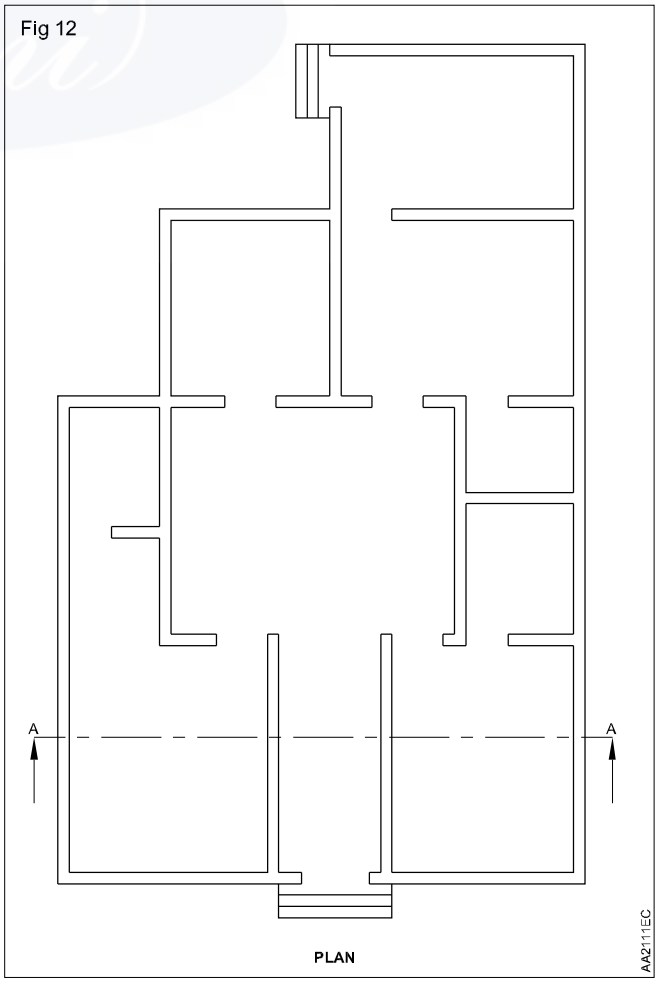
Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

Hads on -1

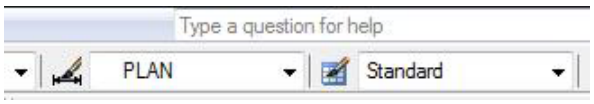
1 Open the given file plan.dwg. (Fig 21)

Fig 12



2 Set the dimension style as plan. (Fig 22)

Fig 22



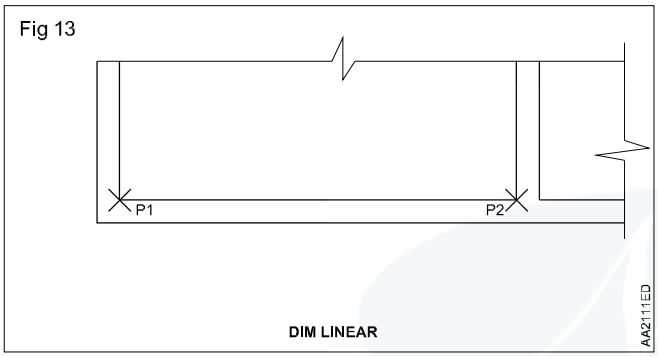
3 Set the layer DIMENSION as the current layer. (Fig 23)

Fig 22

4 Dimension the distance P1 to P2 with linear dimension.

Command: dimlinear (Fig 24)

Fig 13

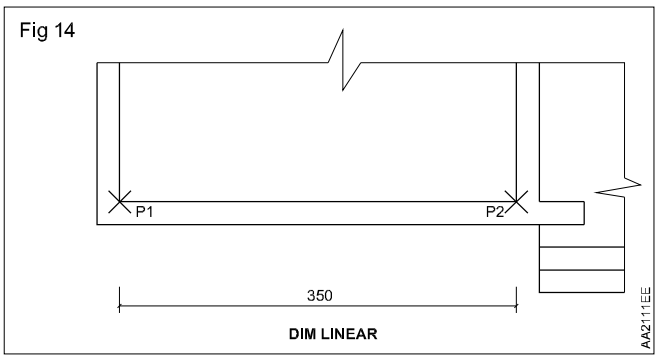


Specify first extension line origin or <select object>:<pick on P1>

Specify second extension line origin: <Pick on P2>

Specify dimension line location or [Mtext/Text/Angle/ Horizontal/Vertical/Rotated]: <Drag and click on a location> (Fig 25)

Fig 14

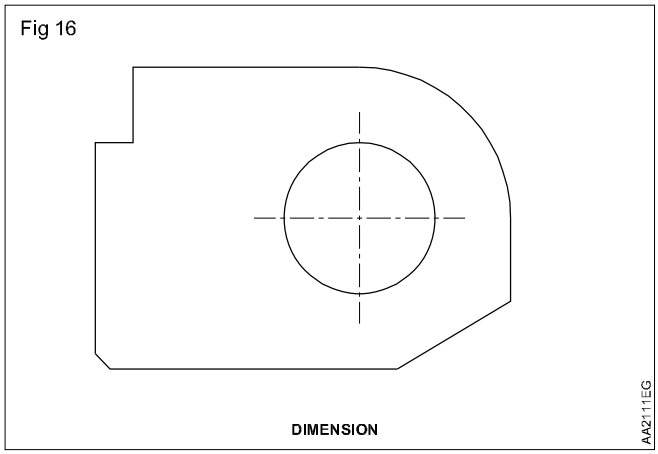


5 Repeat the steps to complete the dimensioning as shown in the following image and save the file. (Fig 26)

Hands on - 2

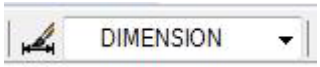
1 Open the file model1.dwg. (Fig 27)

Fig 16



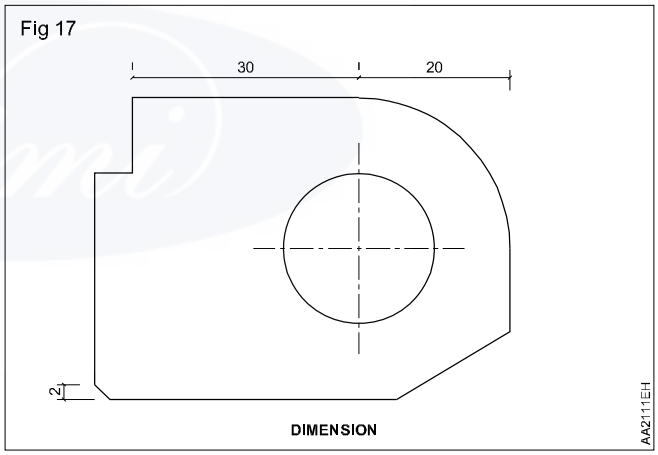
2 Set the dimension style as 'dimension'. (Fig 28)

Fig 28



3 Dimension model with linear dimension as shown in Fig 29.

Fig 17



4 Use dimension baseline to complete the dimensions as shown in Fig 30

Fig 18

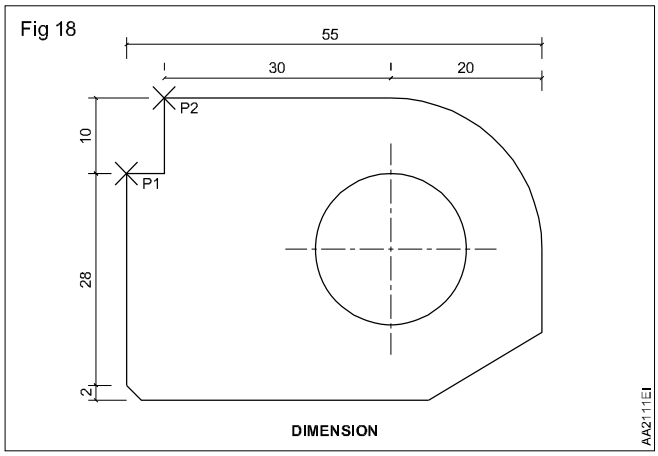
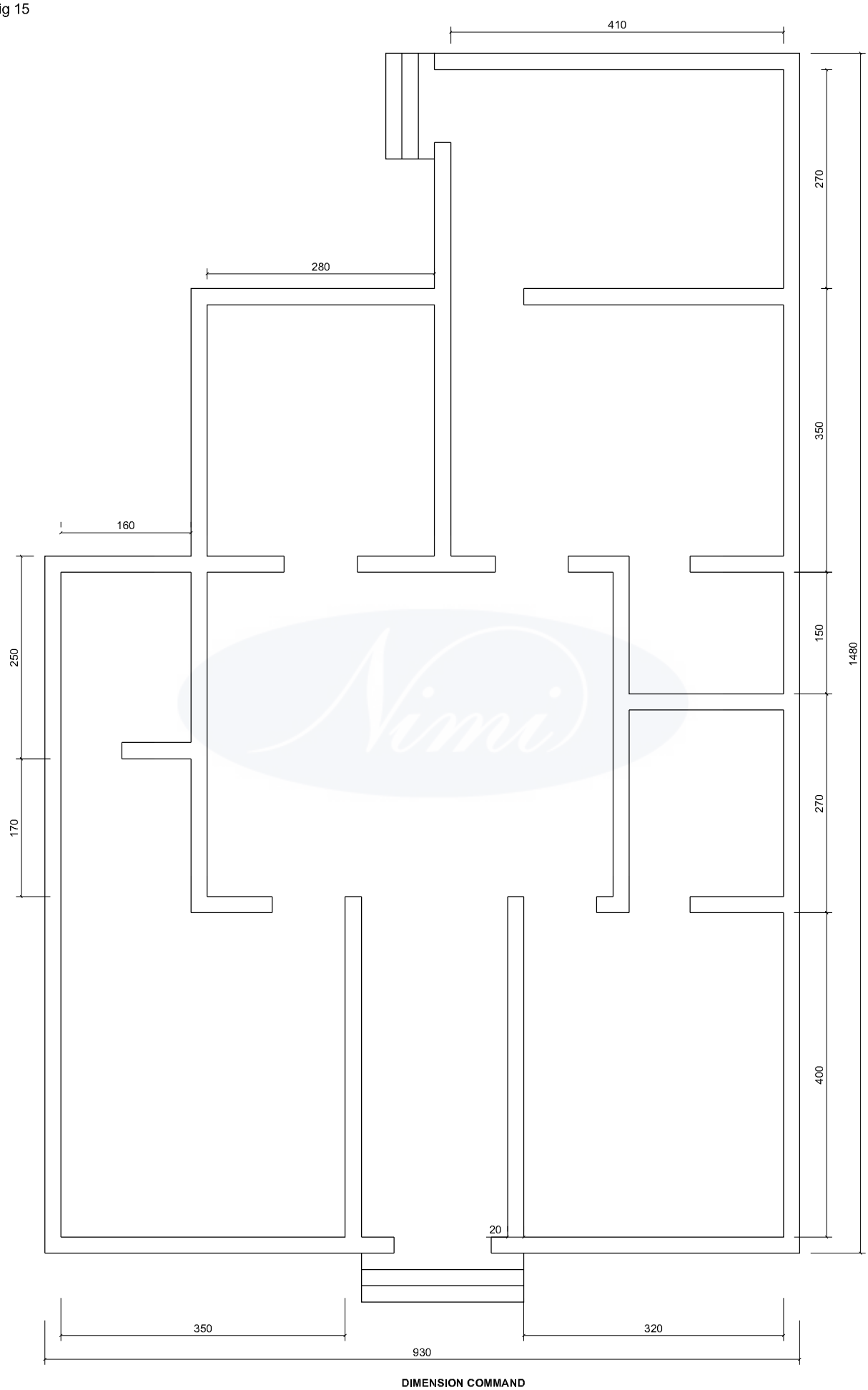


Fig 15


410
270
280
350
160
250
150
1480
170
270
400
20
350
320
930
DIMENSION COMMAND
AA2111EF

Command: dimcontinue

Select base dimension: <select the dimension with text 2>

Specify a second extension line origin or [Undo/Select] <Select>: <Click on P1>

Dimension text = 28

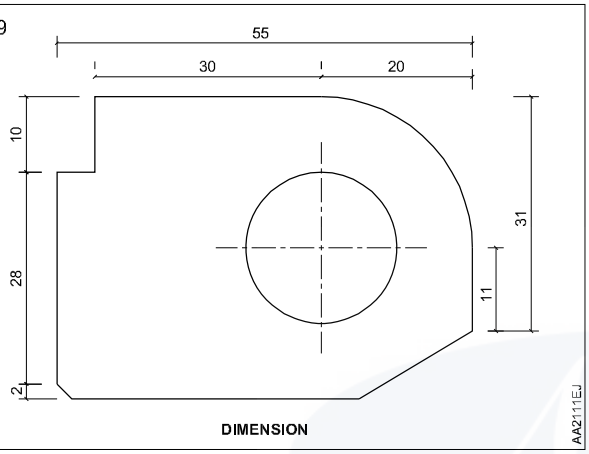
Specify a second extension line origin or [Undo/Select] <Select>:<Click on P2>

Dimension text = 10

Specify a second extension line origin or [Undo/Select] <Select>:<ESC>

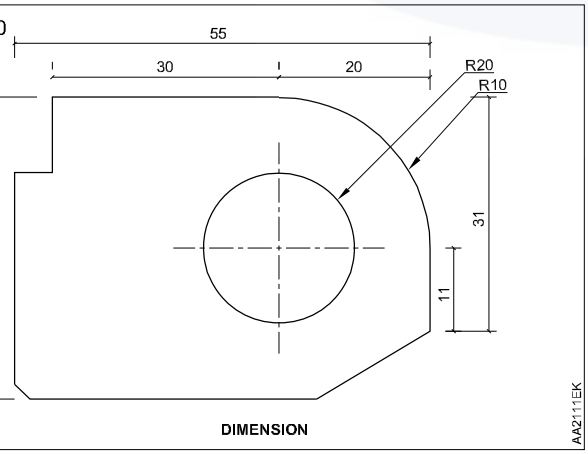
5 Use the linear dimension to complete dimension as shown in Fig 31.

Fig 19



6 Choose radius dimension and complete the dimensioning like this. (Fig 32)

Fig 20



Command: dimradius

Select arc or circle: <Click on R1 and drag to a point>

Dimension text = 20

Specify dimension line location or [Mtext/Text/Angle]

Choose leader command and make a leader like this.

Command: LEADER

Specify leader start point: <Click on C1>

Specify next point: <Click C2>

Specify next point or [Annotation/Format/Undo] <Annotation>: <Click C3>

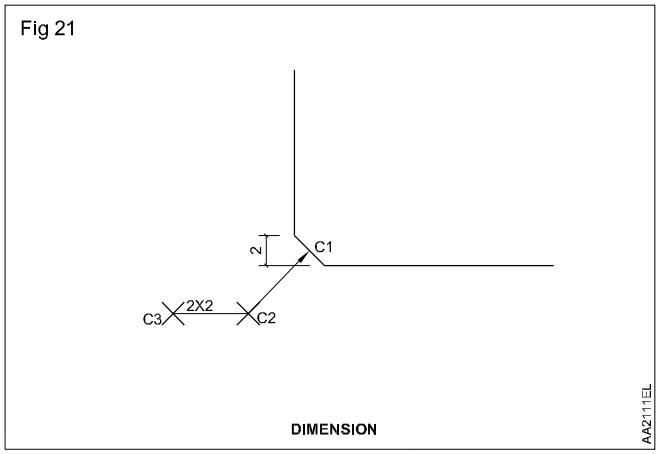
Specify next point or [Annotation/Format/Undo] <Annotation>: <Press enter>

Enter first line of annotation text or <options>: <In command line type as 2 x 2>

Enter next line of annotation text: <Press enter>

Command: (Fig 33)

Fig 21



8 Save and exit the drawing file.

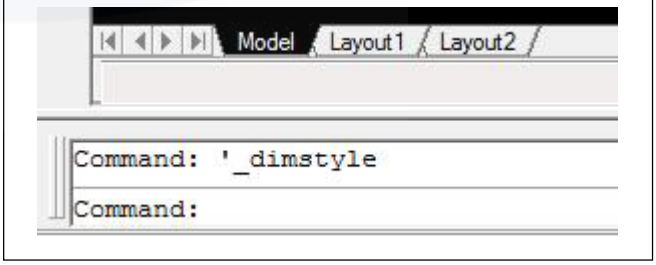
Transparency

Displays the transparency dialog box.

Layouts

- Choose the layout 1 TAB at the bottom of the screen. (Fig 34)

Fig 34



- Change the name of the layout using the layout wizard.
- Change the remaining layout options for page setup and plots. (Fig 35)
- Paper space layout with one view is shown in Fig 36.

Creating multiple layouts

- Choose the layout 2 TAB at the bottom of the screen.
- Right click on the layout tab, choose new layout. (Fig 37)
- Change the remaining layout options for page setup and plots.

Fig 35

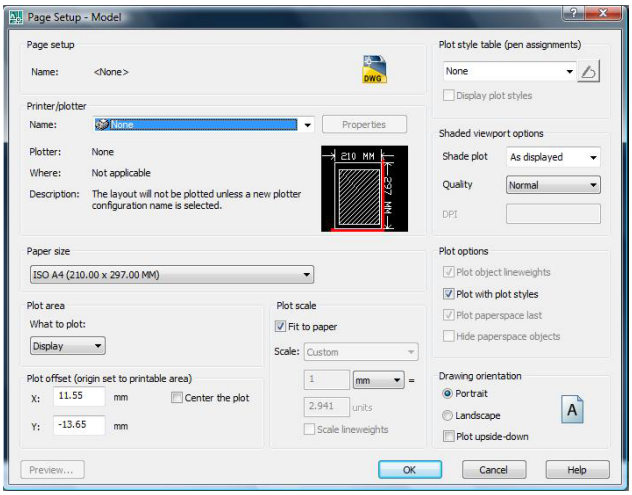


Fig 36

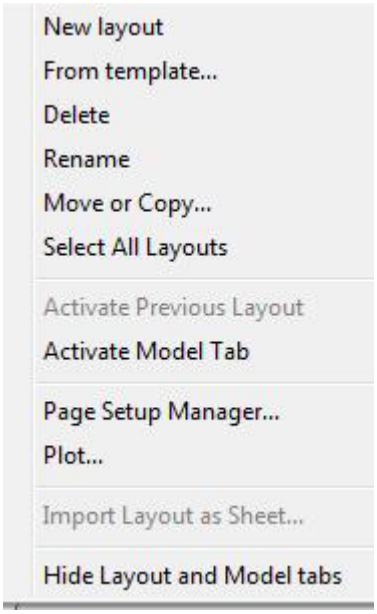
Model space view ports

In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others. Viewports created on the model tab completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.

Tiled viewports let you divide the drawing area into tiles and display different parts of your drawing in each tile. The major advantage with tiled viewports in that you can draw from one viewport to another without missing a best, expanding them for large drawings where you need to alternate between working in detail in a small area and working in a larger area. In fact, through the use of one viewport, you can have a view of the entire viewport, and others for zoomed in details.

You can set up a wide variety of configurations of tiled viewports, and set the view however you want in each one, using standard zooming and panning. But no matter how you set them, they always occupy the entire drawing area, unlike floating viewports.

Fig 37



Note : Floating viewports are useful for laying out a drawing in paper space for plotting. Whereas, tiled viewports are meant for drawing and editing.

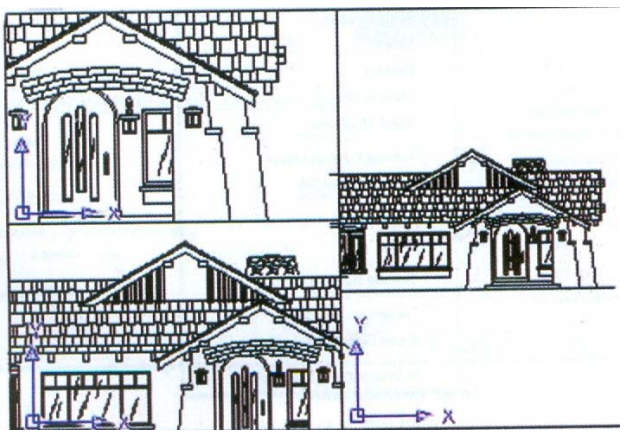
Only one tiled viewport can be active at a time, and the crosshairs appear only in the active viewport. To make a viewport active, all you have to do is to click inside the viewport you wish to open. This being done, you can draw or edit there, in the same way you usually do in a model space, when you only have one viewport, which is the default. (Fig 38)

You can divide a single drawing window into multiple tiled windows (called viewports) on the model tab. You can control the number of windows created and the arrangement of the windows. You can also save and restore named window configurations and display a list of the current and saved window configurations.

Create multiple views

- Choose view => Viewports.
- In viewports menu, choose 1, 2, 3, or 4 viewports.

Fig 38

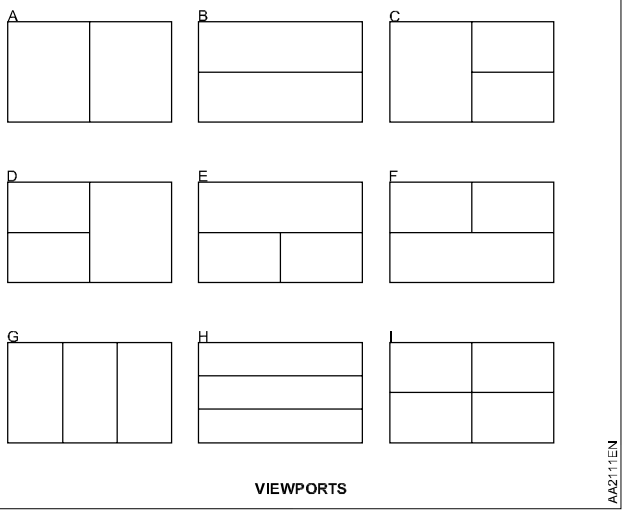


- Type h if you want the horizontal orientation or type v if you want the vertical orientation.

Command line = VPORTS

You can divide a drawing window into two windows arranged vertically (A) or horizontally (B); three windows arranged left (C), right (D), above (E), below (F), vertically (G), or horizontally (H); or four tiled windows (I). (Fig 39)

Fig 23



Join two views

- Choose view => Viewports => Join.
- Click anywhere inside the window you want to keep.
- Click anywhere inside the adjacent window you want to join to the first window.

Command line = VPORTS

Name and save a window configuration

- Choose view => Viewports => Save viewports
- Type a configuration name, and then press enter.

The name can be up to 31 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_).

Command line = VPORTS

Restore a named window configuration

- Type - vports and press ENTER.
- On the command line, type restore.
- Type the name of the window configuration you want to restore.

Commands reference

VPORTS : Creates multiple viewports

System variables reference

Maxactvp : Sets the maximum number of viewports that can be active at one time in a layout

Tilemode : Makes the model tab or the last layout tab current

Cvport : Sets the identification number of the current viewport

Viewctr : Stores the center of view in the current viewport

Viewsize : Stores the height of the view in the current viewport.

Mview

The Mview command controls the size and position of the mview viewports (from now on called mviews).

Mview is to tilemode = 0, as vports is to tilemode = 1. You can use mview when you would like to have a view of the model. Yet for this, pspace mode must be active to use mview. AutoCAD will automatically switch to pspace when you issue the mview command. The default mview option is "<<first point>>" to use this option, pick a point which represents one corner of the mview. At the "other corner" prompt, pick a point which represents the opposite corner of the mview.

- Choose view, Viewports, 1 viewport.

  or

  Type type Mview at the command prompt.

  Command: Mview or Mv

  ON/OFF/Hideplot/Fit/2/3/4/Restore/<<First point>>:

  P1

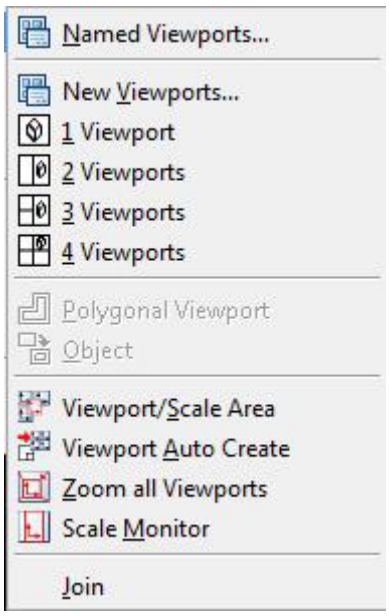
  Other corner: P2

TIP : Mviews should be created on their own layers in order to be turned ON/OFF.

Irregular shaped viewports

- Draw a shape in paper space (e.g. circle, polygon, ellipse)
- Choose view => viewports => object (Fig 40)
- Choose the object to make a viewport.

Fig 40



Model space and paper space

Mspace (model space) can be activated only when there is a minimum of one mview.

To enter model space mode use "Mspace".

- Type Mspace at the command prompt.

  Command: Mspace or Ms

  or

  Double - click the word "paper" on the status bar to toggle to model space.

- Notice the ucs icon will appear in each of the mviews when you enter modelspace.

Pspace mode should be entered to create a border, a title, mviews, and annotations only. This environment is used to lay out a 2 dimensional working drawing suitable for plotting. When you plot from pspace, you should plot 1 =1.

- Type Pspace at the command prompt.

  Command: Space or Ps

  or

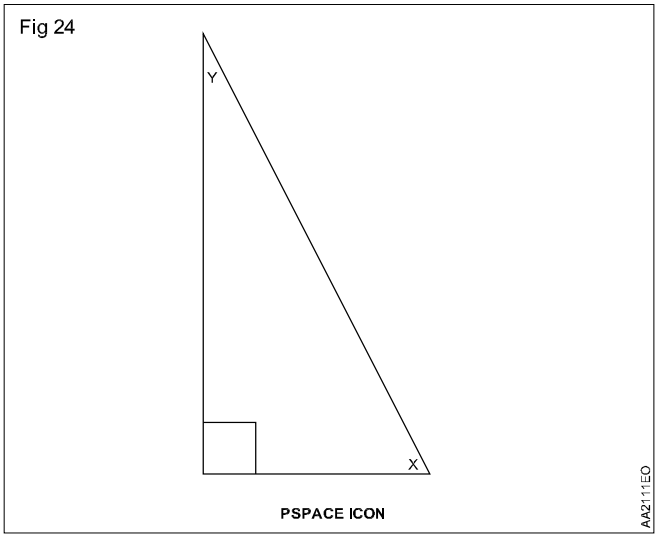
  Double - click the word "MODEL" on the status bar to toggle to paper space.

- Notice the pspace icon. (Fig 41)

Scales - zooming in model space

Use zoom "XP" to zoom the model on a certain factor of the paper. If you enter a value followed by xp, AutoCAD specifies the scale relative to paper space units. For example, entering .5xp displays model space at half the scale of paper space units. If you want to plot the model at ¼ " = 1', type zoom ¼ 8xp. If you want to plot a part at 3 times, type zoom 3xp. Views can also be shown in 3D by using the Vpoint command.

Fig 24



PSPACE ICON

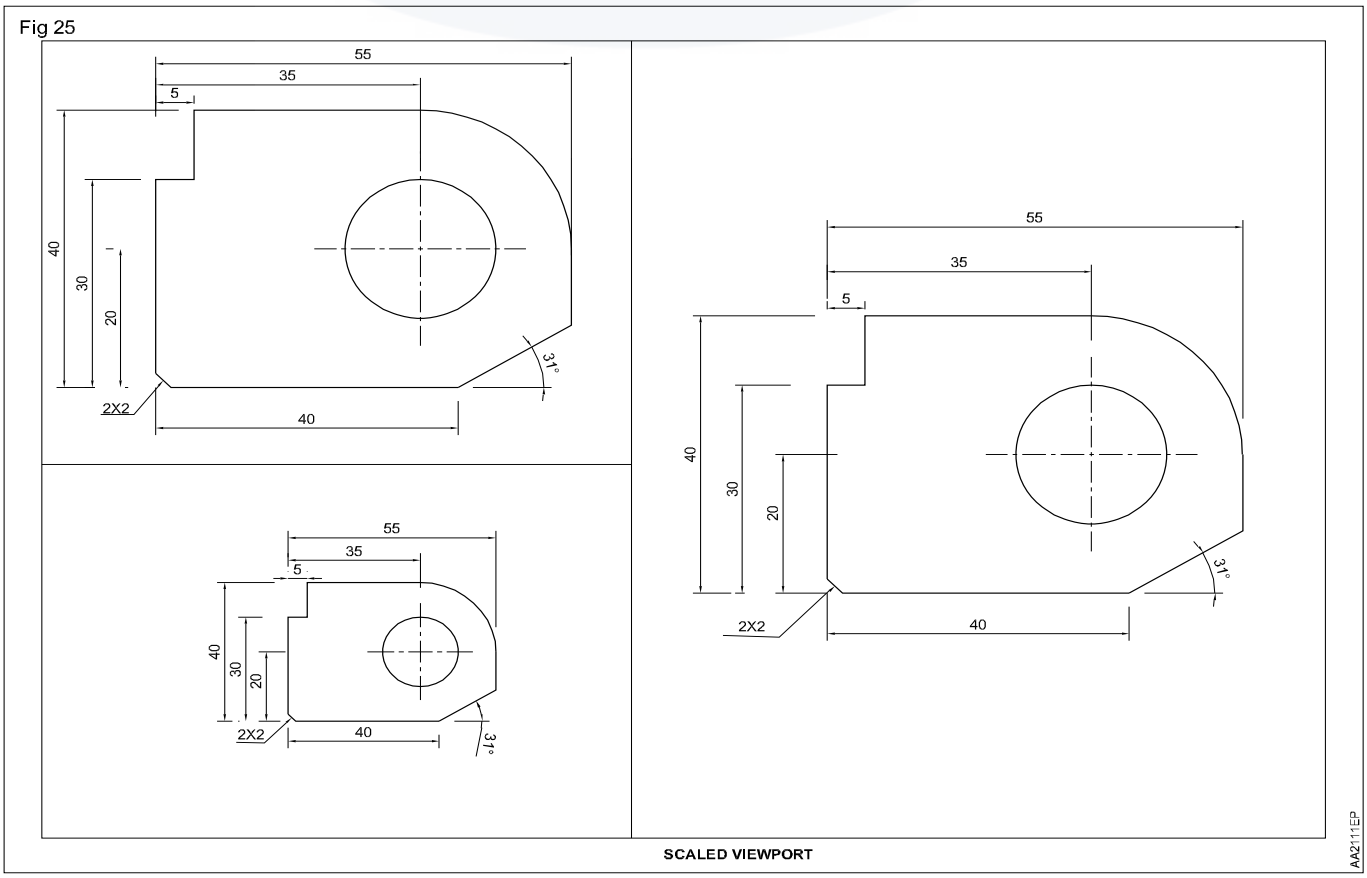
- Type MS at the command prompt to enter model space for each individual viewport.
- Type zoom at the command prompt. Command: Zoom

  All/Center/Dynamic/Extents/Previous/Scale (X/XP)/ Window/<Realtime>: 3XP

  You can also change the scale from the viewports toolbar.

  Scale1 = 1 scale2= 1Zoom ¼ XP (Fig 42)

Fig 25



SCALED VIEWPORT

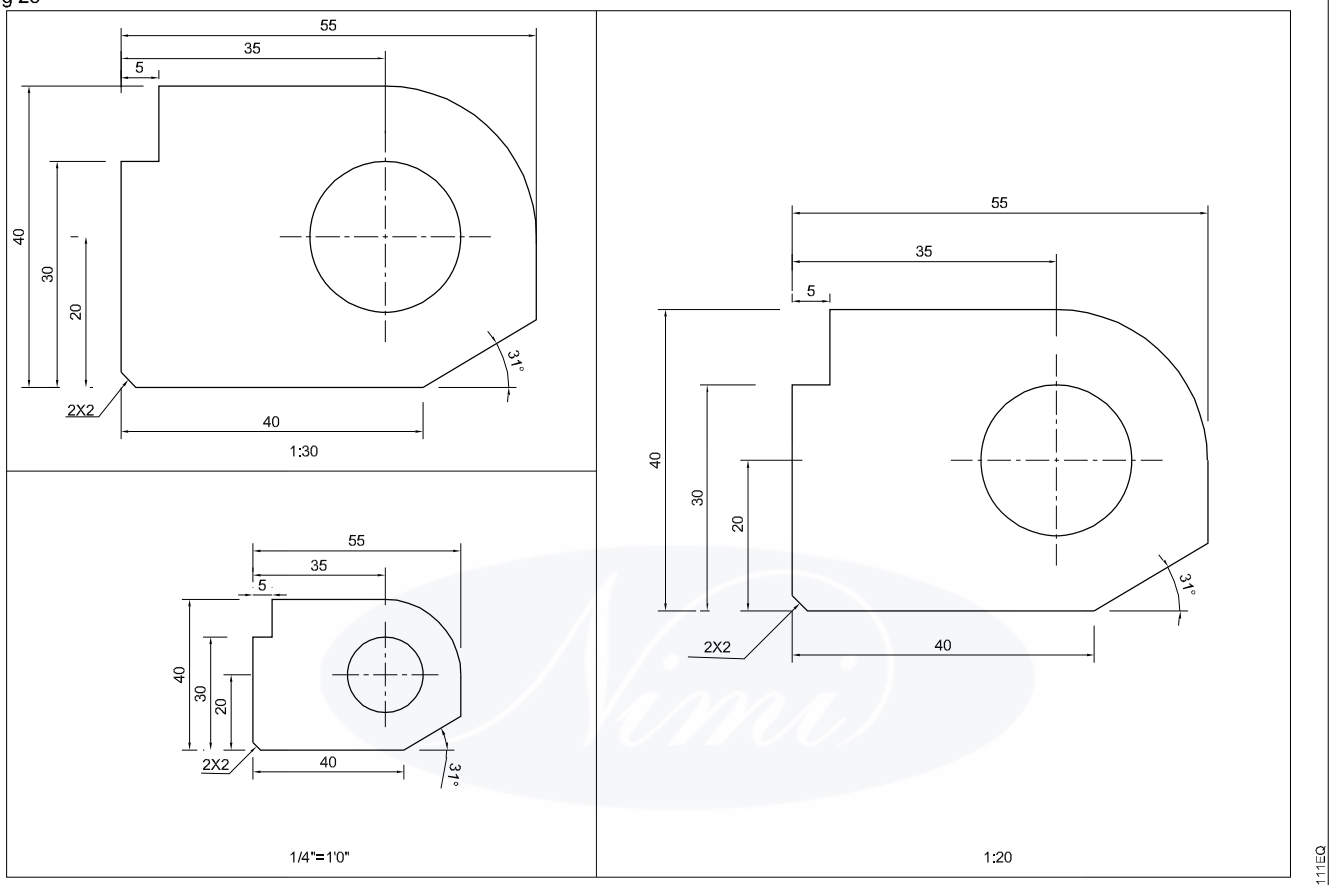
Adding text in paper space

Title block text and miscellaneous text can also be added in paper space. (Fig 43)

Plotting in paper space

Plotting of all mviews should be done from paper space, and not from modelspace. When you plot from pspace, you should plot1 =1. For hidden line removals, remember to use the hideplot option in the mview command. Once a zoom scale has been defined, do not zoom again before plotting. You can change the display with the pan command.

Fig 26



TEXT IN PAPER SPACE

Layout wizard

Layout wizard helps to create a new layout with the following assigned properties.

- Layout name
- Printer
- Paper size
- Paper orientation
- Title block
- Viewports

Viewport layers

Vplayer (Viewport layer) controls layers on and off, and freeze and thaw, for each mview. Layer controls the on and off, freeze and thaw, globally. Layers must be on and thawed globally before they can be effected per mview using vplayer.

- Click in the viewport to change layer status.
- Choose the layer dialog box.
- Highlight the layer to freeze or thaw in the current or new viewport.
- The layer dialog box also allows control of layers for each viewport. (Fig 43)

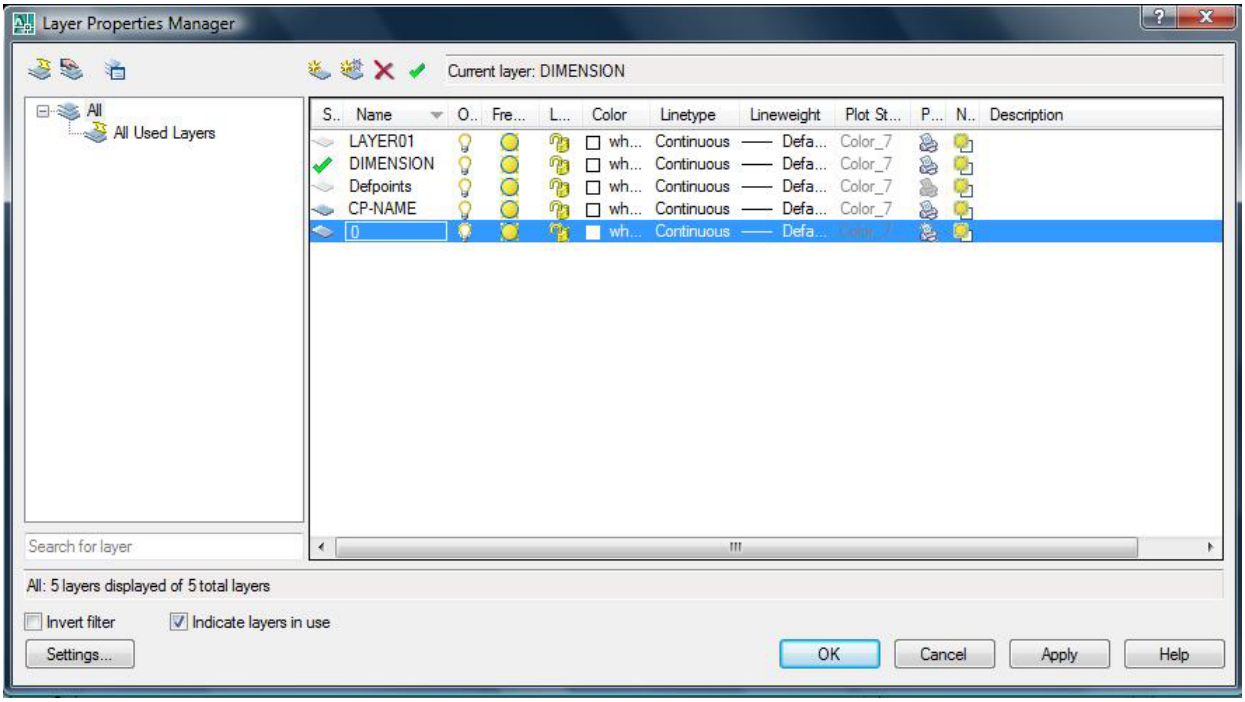
Page setup dialog box

Command alias	Button	Classic menu	Ribbon / Application menu
Page setup		Application menu => Print => Page Setup	Output tab =>Plot panel =>Page setup

Specifies page layout and plotting device settings. The page setup dialog box is displayed in the following cases.

- When you create a new page setup through the page setup manager

Fig 43

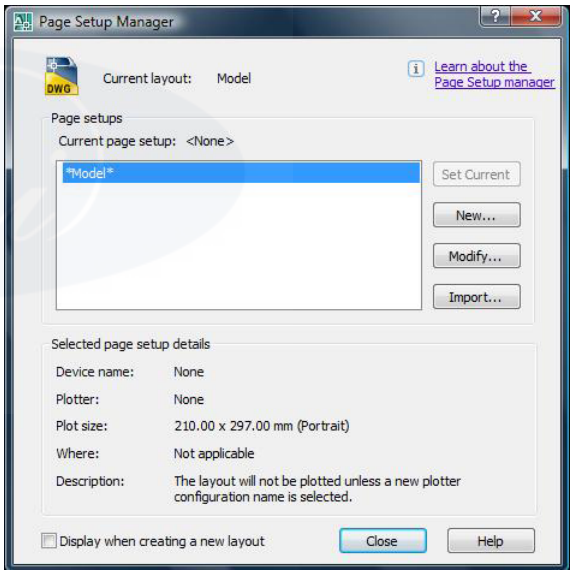


- When you modify an existing page setup through the page setup manager

The page setup settings that you specify are stored with the layout and can be applied to other layouts or imported into other drawings.

The title of the page setup dialog box also displays the name of the current layout or sheet set. (Fig 44)

Fig 44



Design bubble diagram (manual)

Objectives: At the end of this exercise you shall be able to
- **determine space requirements by gathering information**
- **develop alternative solutions for bubble diagram**
- **select the final solutions from the bubble diagram.**

Data :

Ground floor - Foyer
Verandah
1 bed room
Living
Stairs
Dining
Kitchen
Sitout
Utility

First floor - Family
Stairs
Terrace
2 bed room
2 attach toilet
1 dress

Note: There is no book of rules for design. The original idea behind the sketch is called the concept of design. The mental ideas helps to

1 Identify the purpose of the object (building)

2 Analyse the aesthetic and functional aspects

3 Think a good solution (design)

The trainer has to give a specific plot size for design or the trainee can select a plot size and indicate north point and road selection.

The trainees must workout 2 or 3 such options of bubble diagram for practice.

PROCEDURE

TASK 1 :

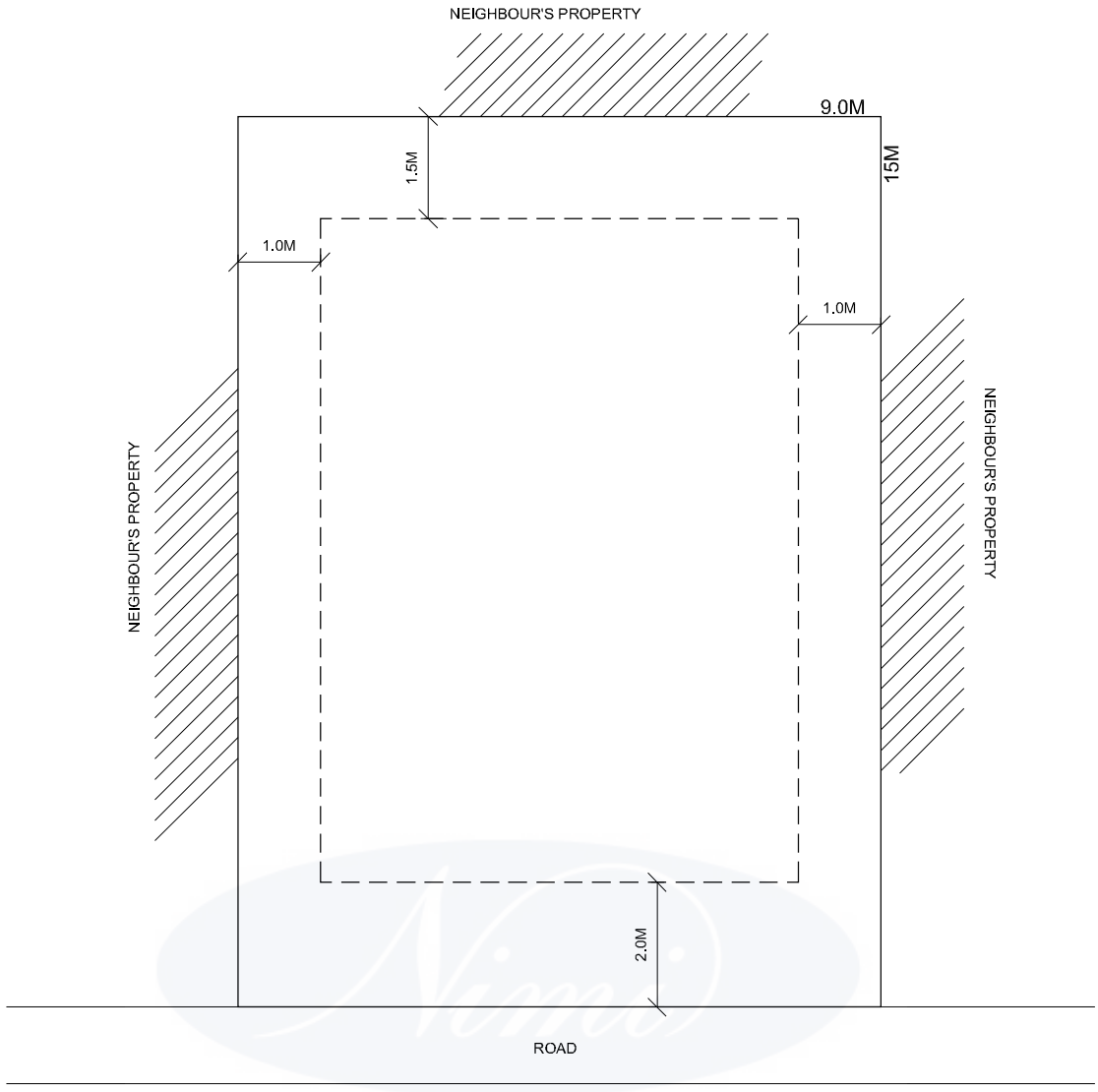
1 Firstly you have to determine the space requirements based on your needs and requirements.

2 The second step is to plot the boundary baded on the given plot size as shown in Fig 1 in 1:100 scale.

3 Mark the set backs based on the building byelaws.

4 Mark the Road direction and north point. (Fig 1)

Fig 1



TASK 2 :

1 Develop a spatial bubble flow diagram based on the information gathered, and within the space left after marking set backs.

2 Show the spatial relationship of the spaces between each other, their relative size and patterns connecting the spaces as shown in Fig 2 & 3

3 Several bubble diagram may be used to illustrate possible solutions.

4 You must make connections between rooms as per circulation movement.

5 These should define the linkages between the different spaces and you must differentiate between the more public and private areas.

6 Develop ground floor and first floor in relation to the spaces and position of staircase has to be same.

7 It is then best to convert the bubble diagram which best suits you into the concept design. (Fig 2 & Fig 3)

You can also colour the different spaces with different colours for better representation.

Fig 2

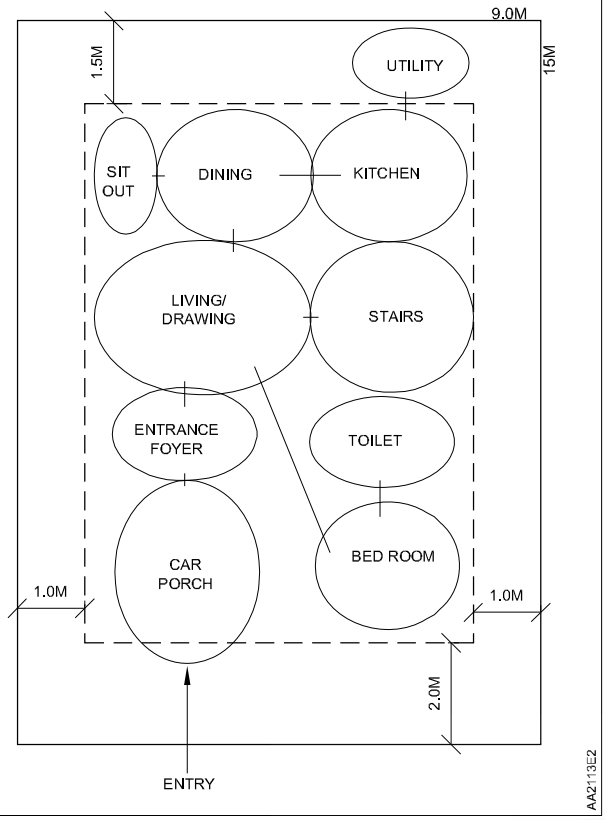
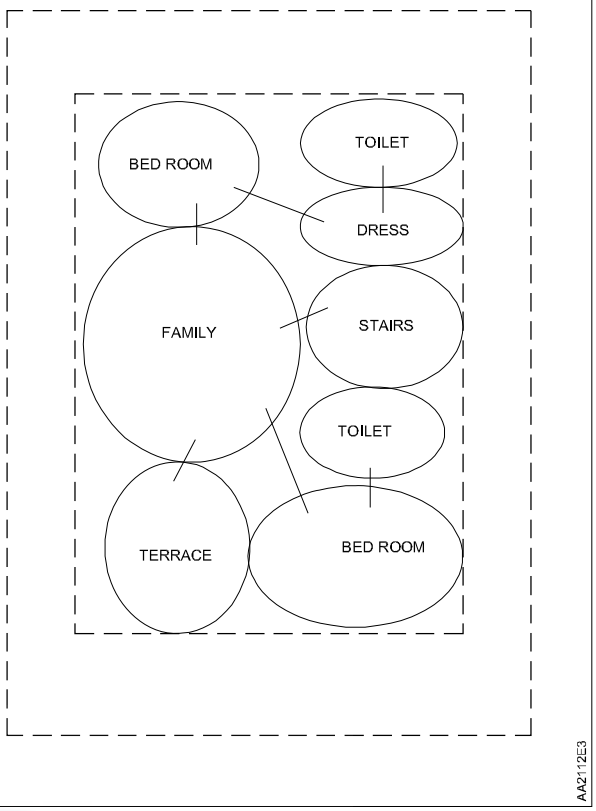

9.0M
1.5M
15M
UTILITY
SIT OUT
DINING
KITCHEN
LIVING/ DRAWING
STAIRS
ENTRANCE FOYER
TOILET
CAR PORCH
BED ROOM
1.0M
1.0M
2.0M
ENTRY
AA2113E2

Fig 3


BED ROOM
TOILET
DRESS
FAMILY
STAIRS
TOILET
TERRACE
BED ROOM
AA2112E3

Development of conceptual design (Manual)

Objectives: At the end of this exercise you shall be able to
- **orient the spaces according to the bubble diagram**
- **design the spaces considering the surrounding built and open spaces**
- **define the linkage between different spaces**
- **position the spaces according to the need for privacy easy access, wind movement, cross ventilation, pathway, etc.**

Data :

Ground floor -	Foyer	First floor -	Family
	Verandah		Stairs
	1 bed room		Terrace
	Living		2 bed room
	Stairs		2 attach toilet
	Dining		1 dress
	Kitchen		
	Sitout		
	Utility		

PROCEDURE

TASK 1 :

1 Conceptual design should be started with respect to the bubble diagram designed in the previous exercise.

2 To make the plan, put the established facts of the site its shape and existing features-down on paper.

3 Use a simple scale, say 1:100 and you could use a graph paper, which in fenths, or a grid of one me be might be easier.

4 Plot the boundary, take guide lines from an established point.

5 Mark the views you wish to keep, positon the spaces according to the need for Pricacy, easy access, wind movement, cross ventilation, circulation etc.

6 Now evolve a satisfactory pattern on a tracing paper put over this plan in an amalgamation of the two.

7 Complete the concept design as shown in Fig 1&2.

Fig 1

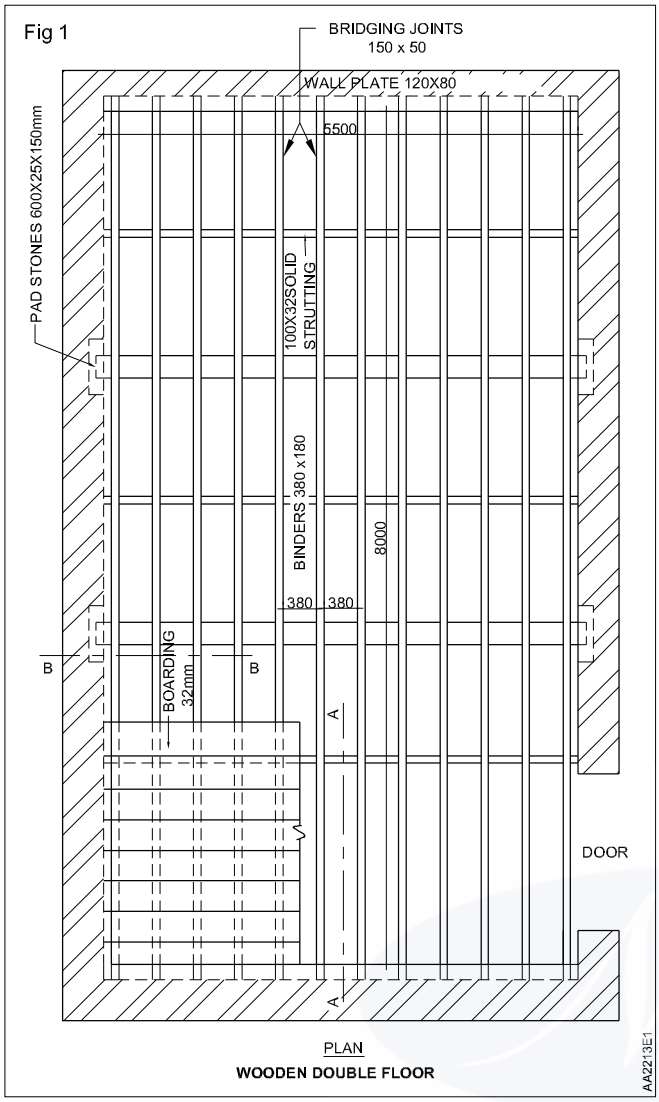


Fig 2

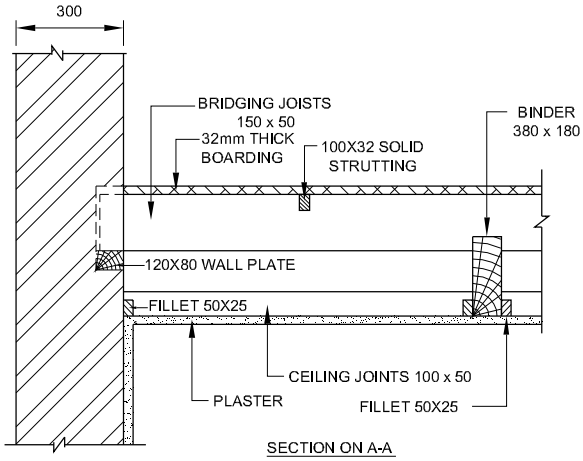
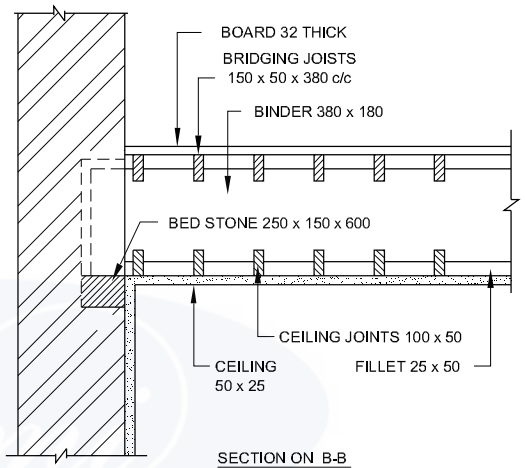


Fig 3



Design single line ground floor plan(Manual) & first floor plan

Objectives: At the end of this exercise you shall be able to
- **convert the conceptual design into a single line plan**
- **indicate the positions of doors and windows**
- **orient the spaces of individual rooms with proper dimensions**
- **check and modify the dimensions of units with reference to the consideration of roominess.**

PROCEDURE

TASK 1 : **Design single line ground floor plan (Fig 1)**

1 With reference to the conceptual plan, single line ground floor plan has to be developed.

2 Make the scheme over a 2 X 2 grid to define the plan and to work from the inside out.

3 Adjustments to the rooms and spces have to be made on the grid.

4 Natural ventilation to the rooms must be achieved by designing windows and ventilators opposite to each other.

5 The kitchen and dining room must be close to each other.

6 The staircase must be approachable from the maximum number of rooms.

7 The passage area must be minimum, well ventilated and sufficiently well it.

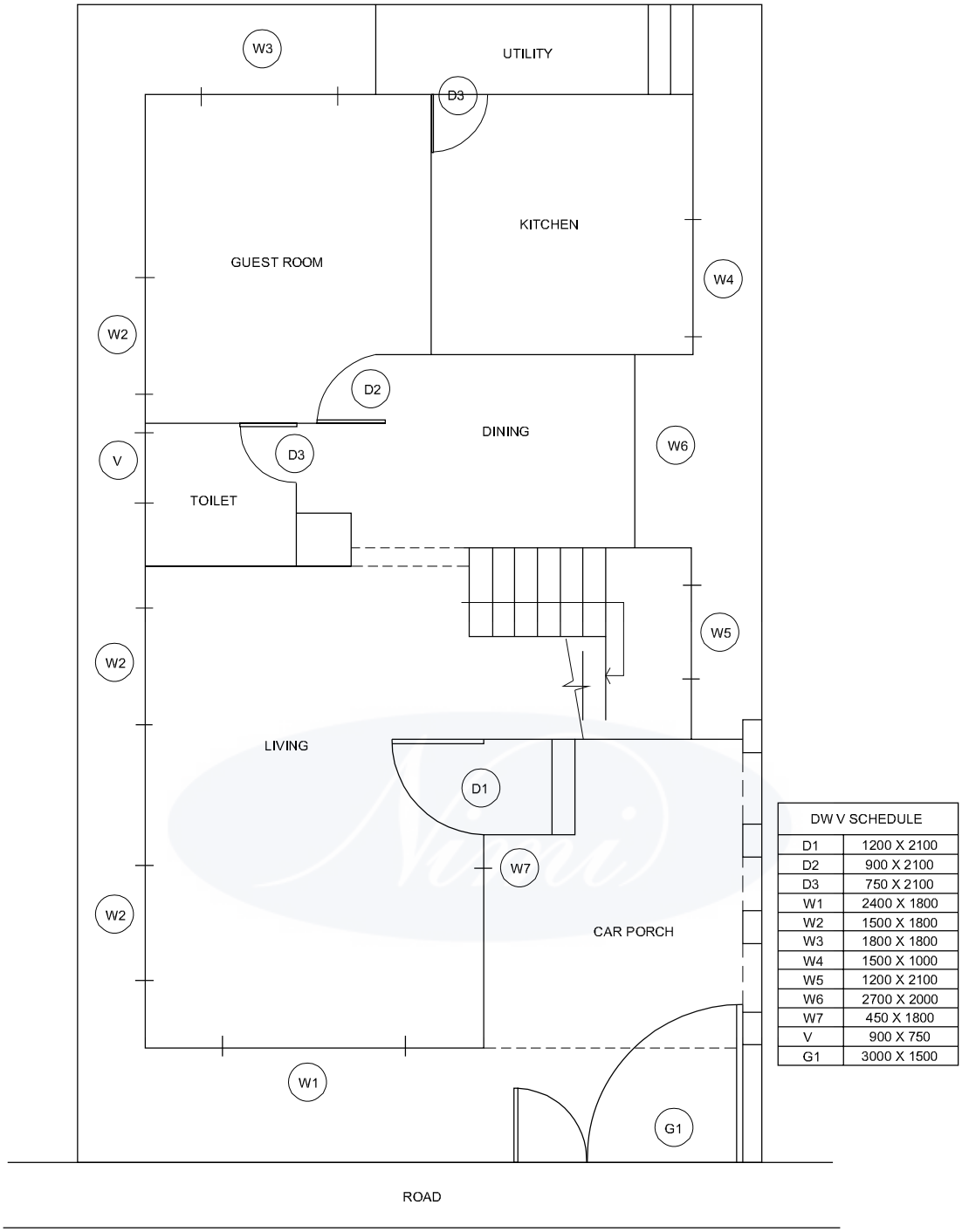
8 Location of the main door has to be easily accessible from the main road.

9 The visualisation of elevation should always be kept in mind while preparing a plan.

10 While drawing the line plan, the elevation of all the 4 sides as well as the perspective view from the road should be kept in mind.

11 An effect of spaciousness within comparatively limited dimension can be achieved by skilful treatment of form, colour natural and artifitial lighting and all the elements of furnishing.

Fig 1



GROUND FLOOR PLAN (SINGLE LINE DIAGRAM)

TASK 2 : **Design single line first floor plan**

1 While designing first floor plan take reference from ground floor for where ever walls can over lap.

2 All the areas and room spaces should be designed taking into consideration the purpose for which they will be used.

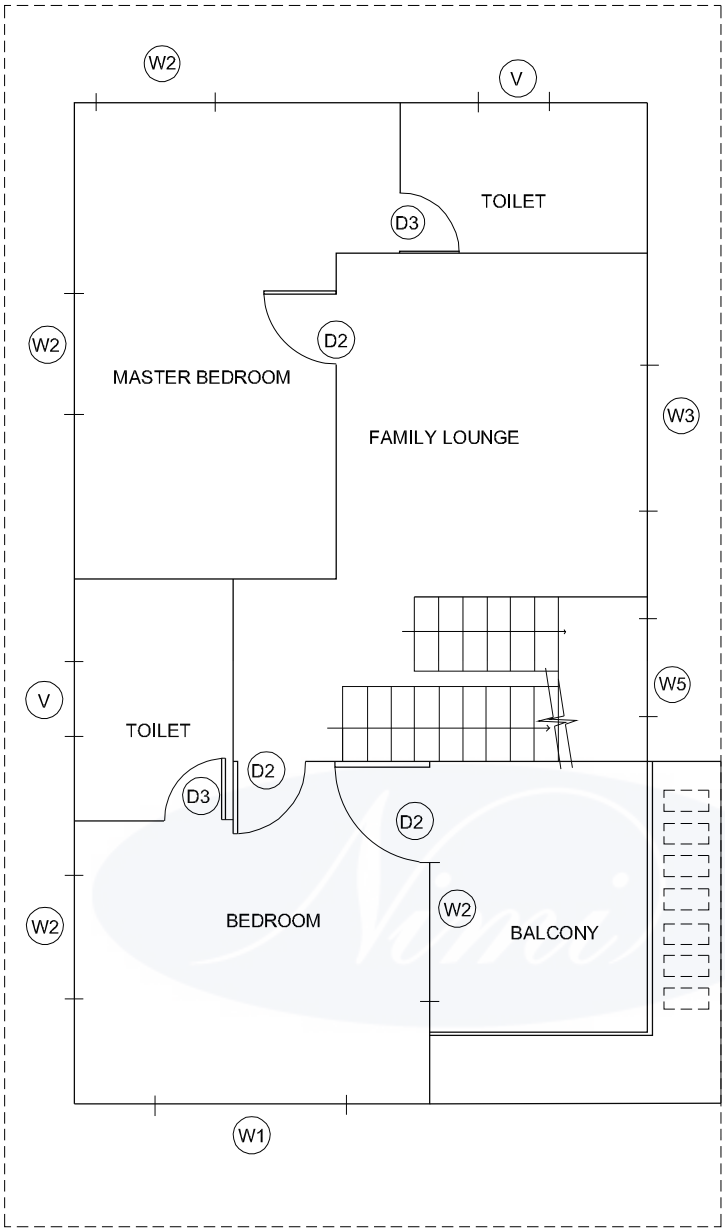
3 All the rooms should be well-ventilted.

4 Bed rooms may be with attach toilet and dress where ever possible.

5 The line plan with reference to the site plan, shape of the plot, north direction, wind and rain directions, main road and surrounding views etc should be kept in mind while designing bedrooms and terraces or balcony.

6 Complete the plan as shown in Fig 2.

Fig 2



DW V SCHEDULE	
D2	900 X 2100
D3	750 X 2100
W1	2400 X 1800
W2	1500 X 1800
W3	1800 X 1800
W5	1200 X 2100

FIRST FLOOR PLAN (SINGLE LINE DIAGRAM)

AA2114E2

Draw preliminary ground floor plan(CAD) and first floor plan

Objectives: At the end of this exercise you shall be able to
- **assign layers and its colour and appropriate line weight to different materials used**
- **convert the single line plan into a conventional plan**
- **complete the preliminary ground floor plan in CAD with all doors and windows.**

PROCEDURE

TASK 1 : **Assign layers and colours and line weight**

1 Start the autocad applications and start a new drawing.

2 Save the file as plans with some reference and in a new folder.

3 Set units in metric and make the plan in millimeters.

4 Before starting the drawing set all layers to the materials used for the drawing.

5 For architectural drawing set layer as follows.

Layer	Colour	Line weight	Line type
A-Wall	Red	0.35	Continous
A-DWV	Green	0.15	Continous
A-Text	Black	0.15	Continous
A - Above	08	0.15	Acad------
A-boundary	Magenta	0.35	Continous
A-ele-1	Blue	0.20	Continous
A-Layout	Black	0.20	Continous

— — — — — — — —

TASK 2 : **Draw the preliminary ground floor plan**

1 Mark the plot as per the given site size.

2 Mark the offset as per building byelaws.

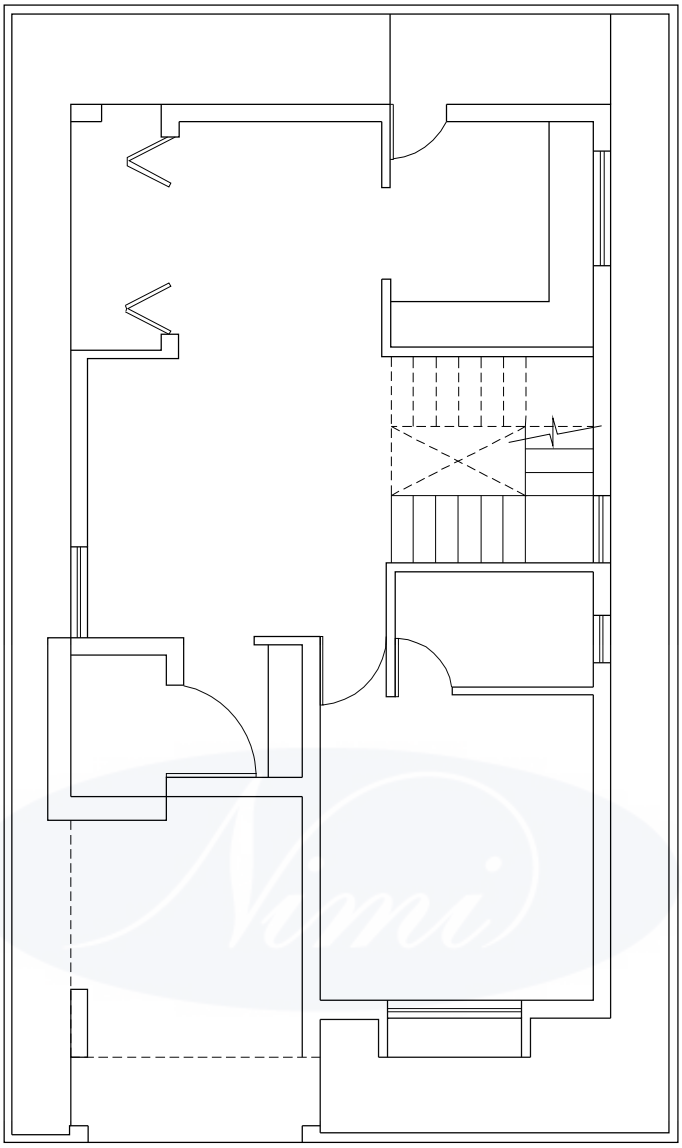
3 With reference to the single line diagram convert the plan into wall diagram with 230mm as exterior wall thickness and 115 as internal wall thickness. (Fig 1)

4 Draw the doors and windows in the marked positions.

5 Use appropriate layers for walls, DWV, Text etc.

6 Complete the ground floor plan as shown in Fig 2.

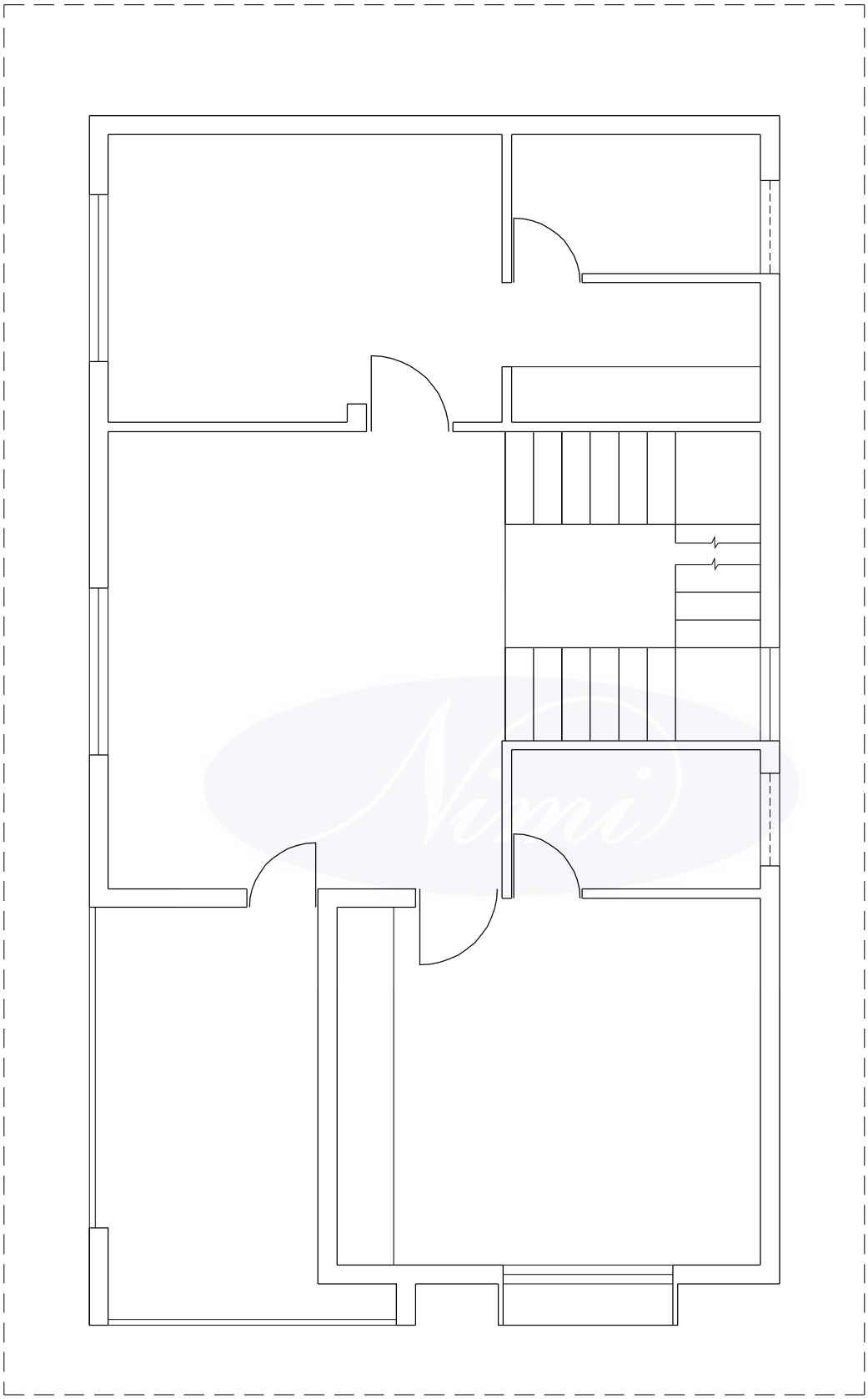
Fig 1



GROUND FLOOR PLAN

AA2115E1

Fig 2



FIRST FLOOR PLAN

AA2115E2

TASK 3 : **Draw the preliminary first floor plan**

1 The first floor plan should be drawn with reference to the walls given in the ground floor plan.

2 Draw the arrangement of rooms as designed in the single line diagram. (Fig 3)

3 Wherever necessary the ground floor walls and first floor walls need to the matched.

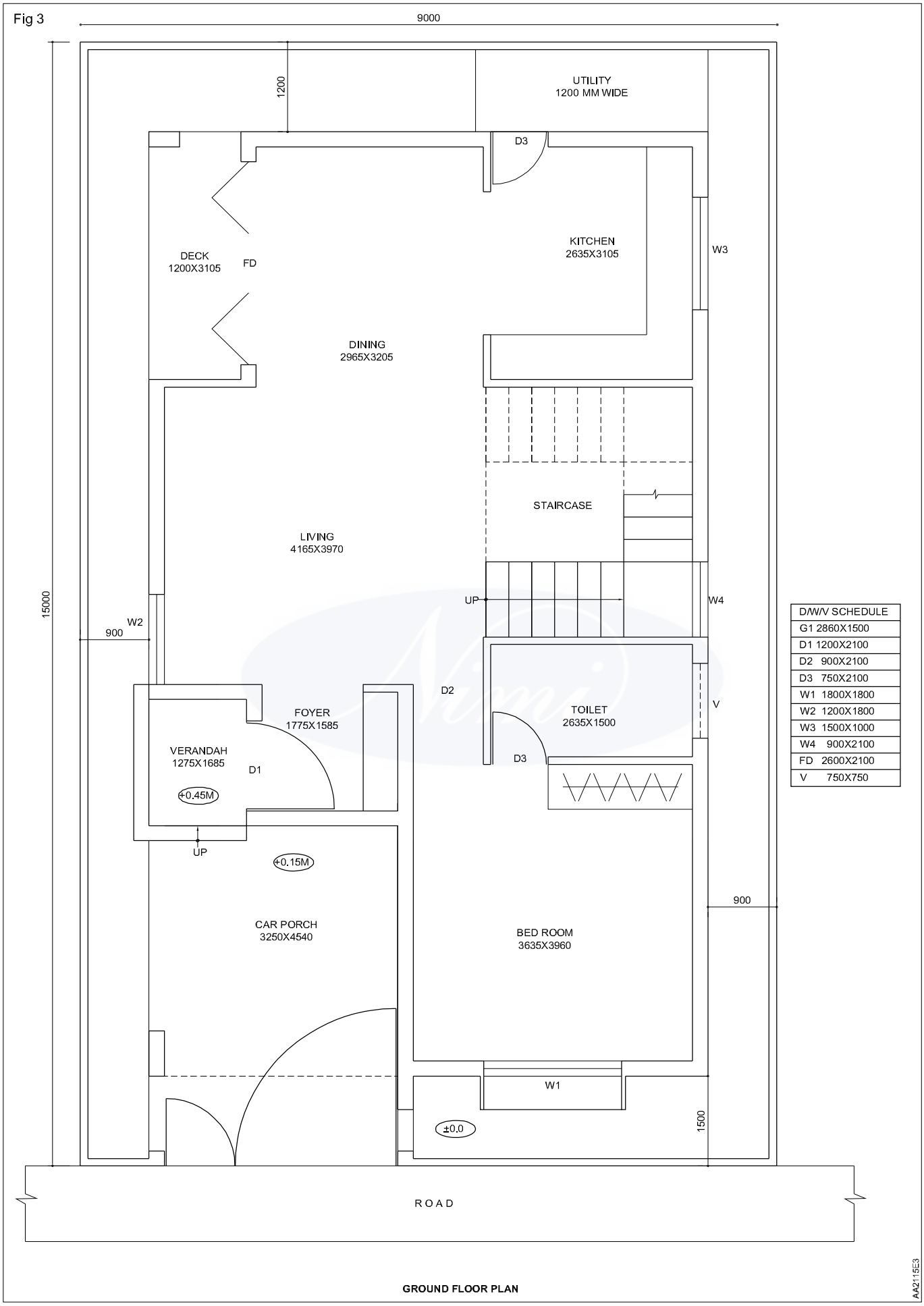
4 Mark all the doors and windows in positions.

5 The drawing in CAD has to be done in 1:1 scale.

6 Indicate the staircase up and down with proper break lines.

7 Complete the first floor plan as shown in Fig 4.

Fig 3

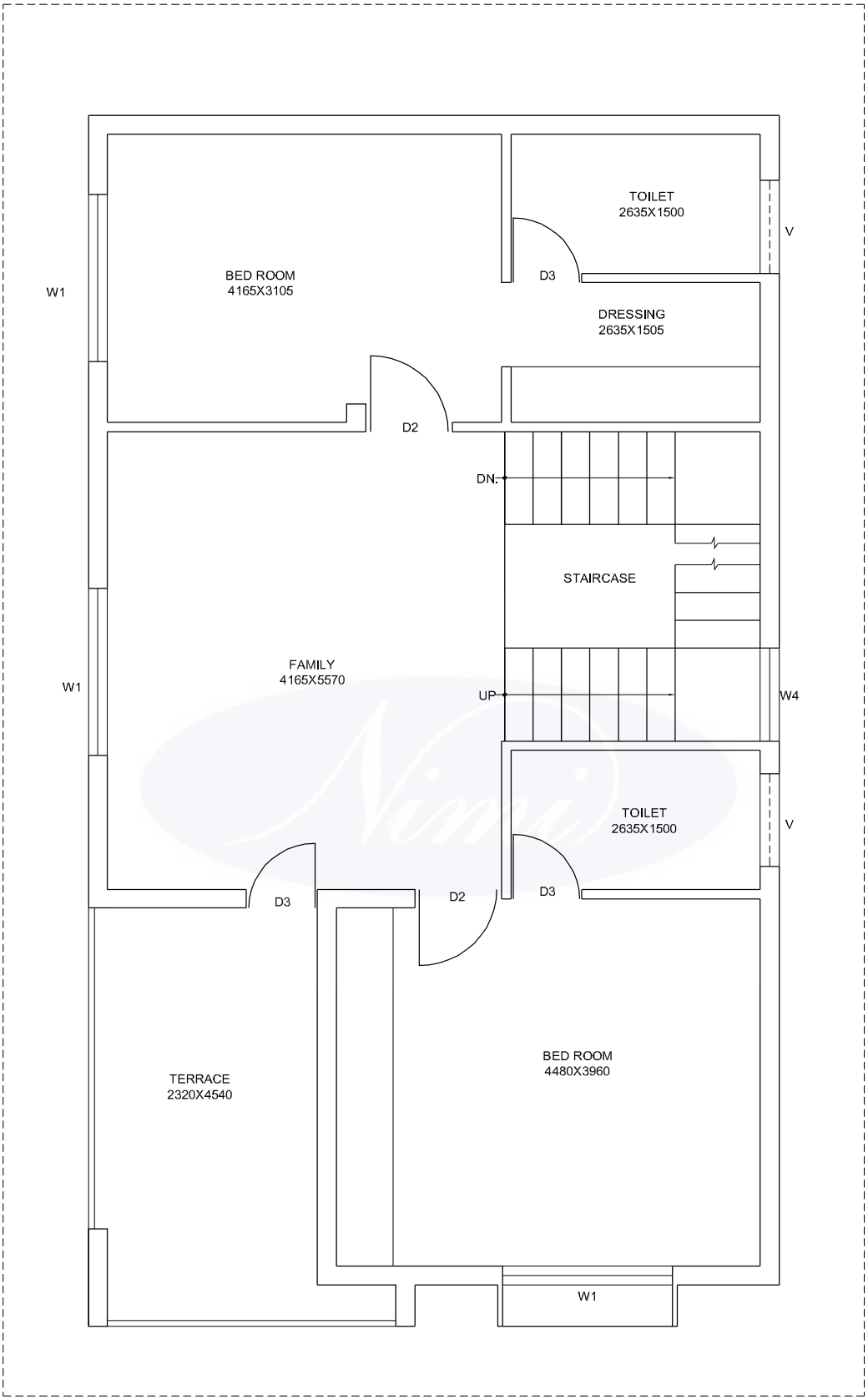


D/W/V SCHEDULE
G1 2860X1500
D1 1200X2100
D2 900X2100
D3 750X2100
W1 1800X1800
W2 1200X1800
W3 1500X1000
W4 900X2100
FD 2600X2100
V 750X750

GROUND FLOOR PLAN

AA2115E3

Fig 4



D/W/V SCHEDULE	
D2	900X2100
D3	750X2100
W1	1800X1800
W4	900X2100
V	750X750

AA2115E4

FIRST FLOOR PLAN

Draw final ground floor and first floor plans with furniture layout rendered(CAD)

Objectives: At the end of this exercise you shall be able to
- **design the interior layout for the residence plan**
- **render the plan and all furnitures with pleasing colours**
- **dimension and name the plan completely.**

PROCEDURE

TASK 1 : **Organization of interior space (manual)**

1 Print the previously drawn ground floor plan & first floor plan.

2 Economic utilisation of space is very important in designing interior.

3 Where should be minimum circulation area and best use of avilable space for the activity to be performed.

4 Minimum number of doors and partitions should be considered for free flow of space and spacious look.

5 To get a spacious look select limited variaty of textures, patterns and colours.

6 The 'Interior activity diagram' should be drawn showing the division of interior spaces with proper circulation.

7 Where should be acoustical and visual barries between entry and private area.

8 While designing seating for living room L-shaped or U-shaped seating makes the most economical use of space because it can accomodate a large group of people, leaving ample circulation space.

9 Do not clutter a small space with more items than needed.

10 Low seating helps in making the room look larger.

11 The dining table could be circular or rectangular.

12 Work Triangles consists of the three essential elements in the kitchen; the clean up or sink area, the cooking centre, the mixing centre and the refrigeration centre.

13 Width of the kitchen plat form should be 600mm.

14 The position of the bed depends on a number of factors. The window should be parallel to the bed position.

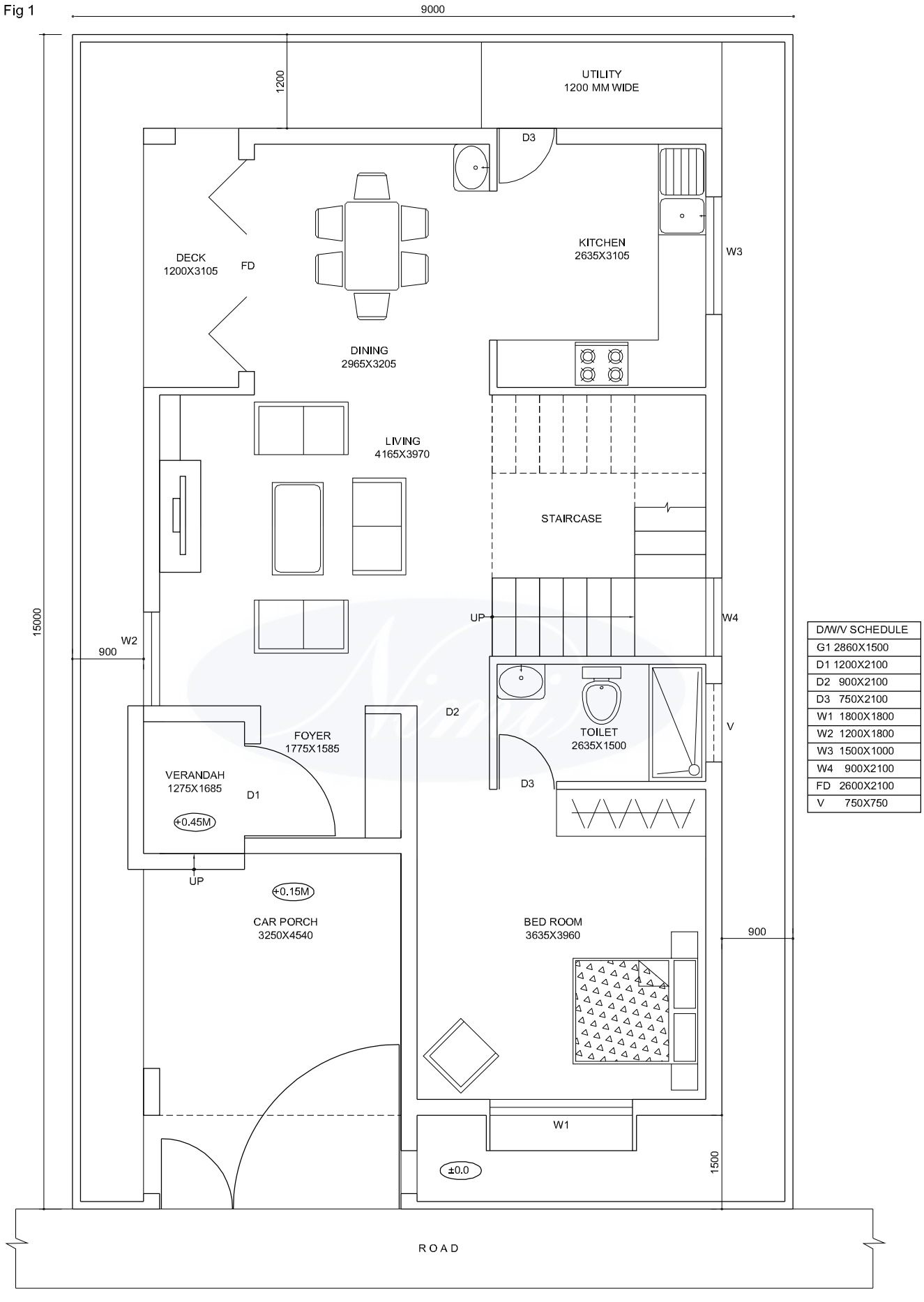
15 The window should not be over the head.

16 The toilet can be divided into two types of areas-the dry area including the sink and the toilet and a wet area, which includes the bathing area. In well-planned toilets, you must separate these two areas.

17 Some cabinets or open shelves must be placed in the bathroom for storage.

18 With all consideration of furnitures and fixtures complete the sketch of organisation of space planning as Fig 1 & Fig 2.

Fig 1

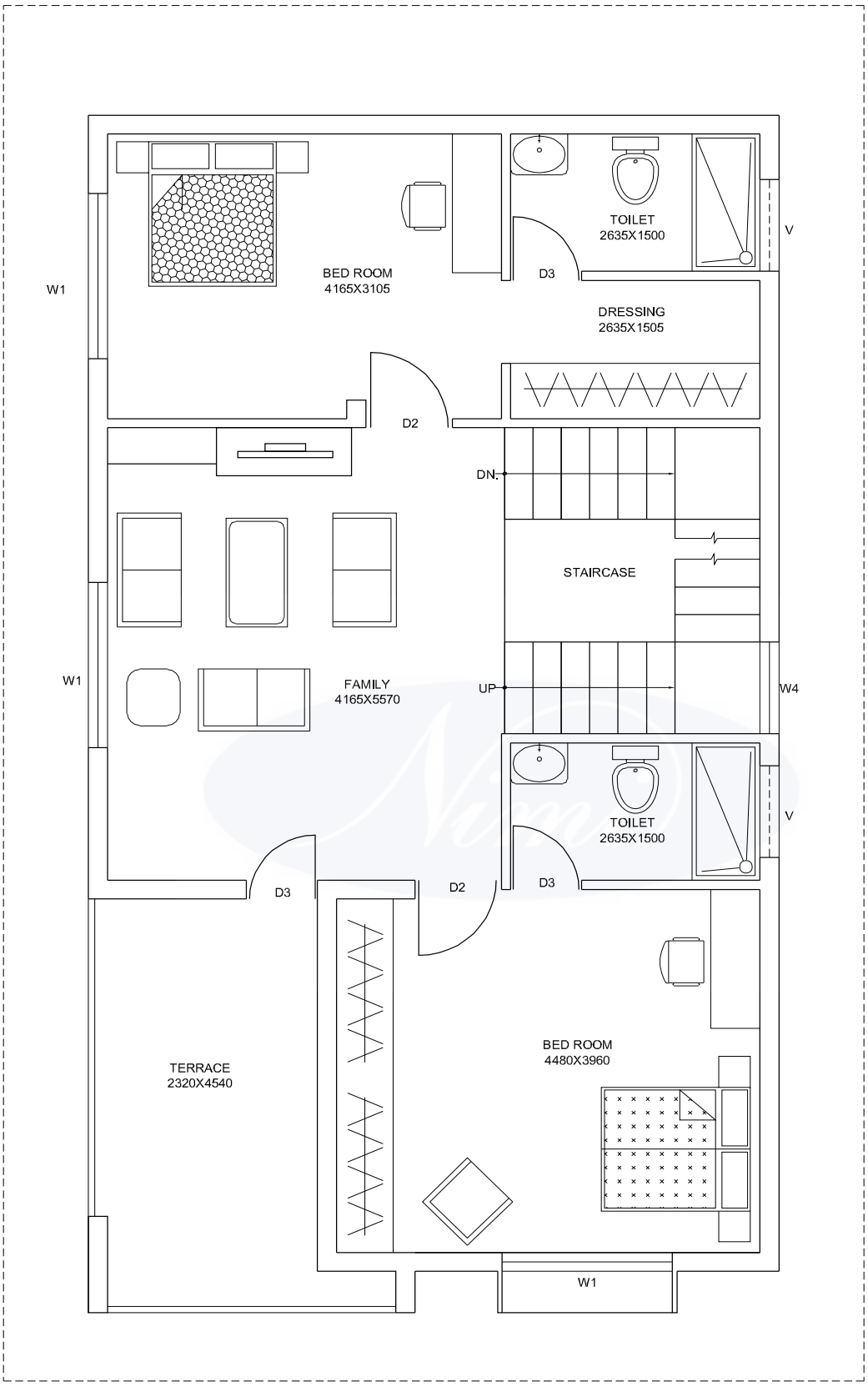


D/W/V SCHEDULE
G1 2860X1500
D1 1200X2100
D2 900X2100
D3 750X2100
W1 1800X1800
W2 1200X1800
W3 1500X1000
W4 900X2100
FD 2600X2100
V 750X750

GROUND FLOOR PLAN

AA2116E1

Fig 2



D/W/V SCHEDULE	
D2	900X2100
D3	750X2100
W1	1800X1800
W4	900X2100
V	750X750

AA2116E2

FIRST FLOOR PLAN

TASK 2 : **Rendering ground floor and first floor plan with furniture layout**

1 With reference to the organisation plan, the detailed furniture layout should be designed.

2 Choosing colour scheme should be considered carefully while rendering the plans and furnitures.

3 For the living room a single solid colour on the floor, or unpatterned floors tend to sketch the space. These include stone, tiles, wood etc.

4 Lighter the shade and the moe reflective the surface, the more effective it will be in lightening up the room.

5 For dining avoid using carpet below the table.

6 Use the same flooring material as in the living.

7 For kitchen you can choose from a variety of tiles and stones.

8 Any type of flooring can be used in the bedroom-from carpet to wood flooring to tiles or ferazzo.

9 You must combine the shape, colour, texture for furniture to provide a pleasing environment, which will truly rejunvenate and refresh you.

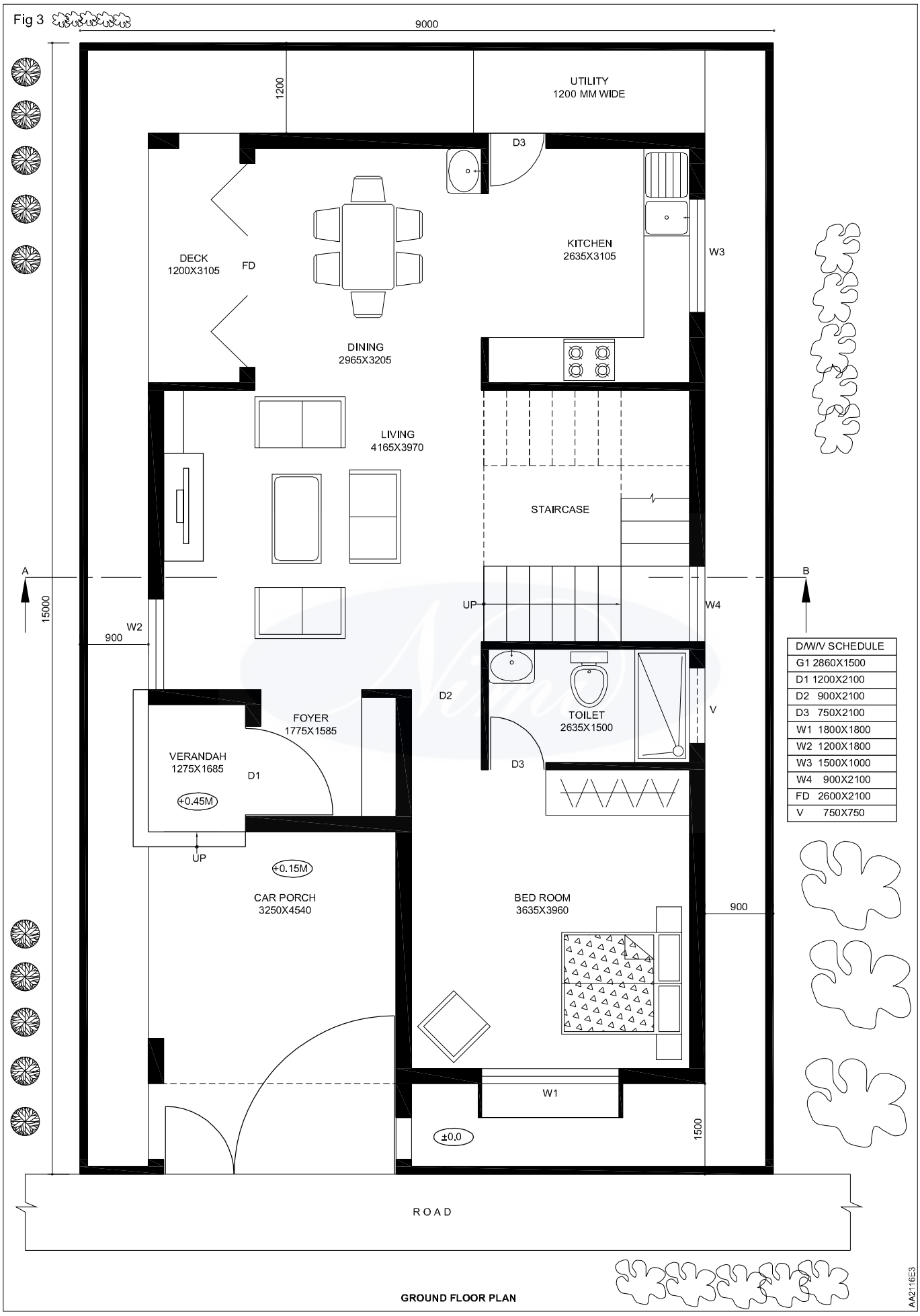
10 Concrete, brick paving, slate, unpolished kota, sandstone are effective floor materials for the porch, out door sit out, inner courtyard, terraces and balconies.

11 Plan the garden simultaneously with the house so that the levels of drive way, terraces are co-ordinated.

12 The amount of soft surfacing (Lawn or ground cover) should be considered in designing the landscape.

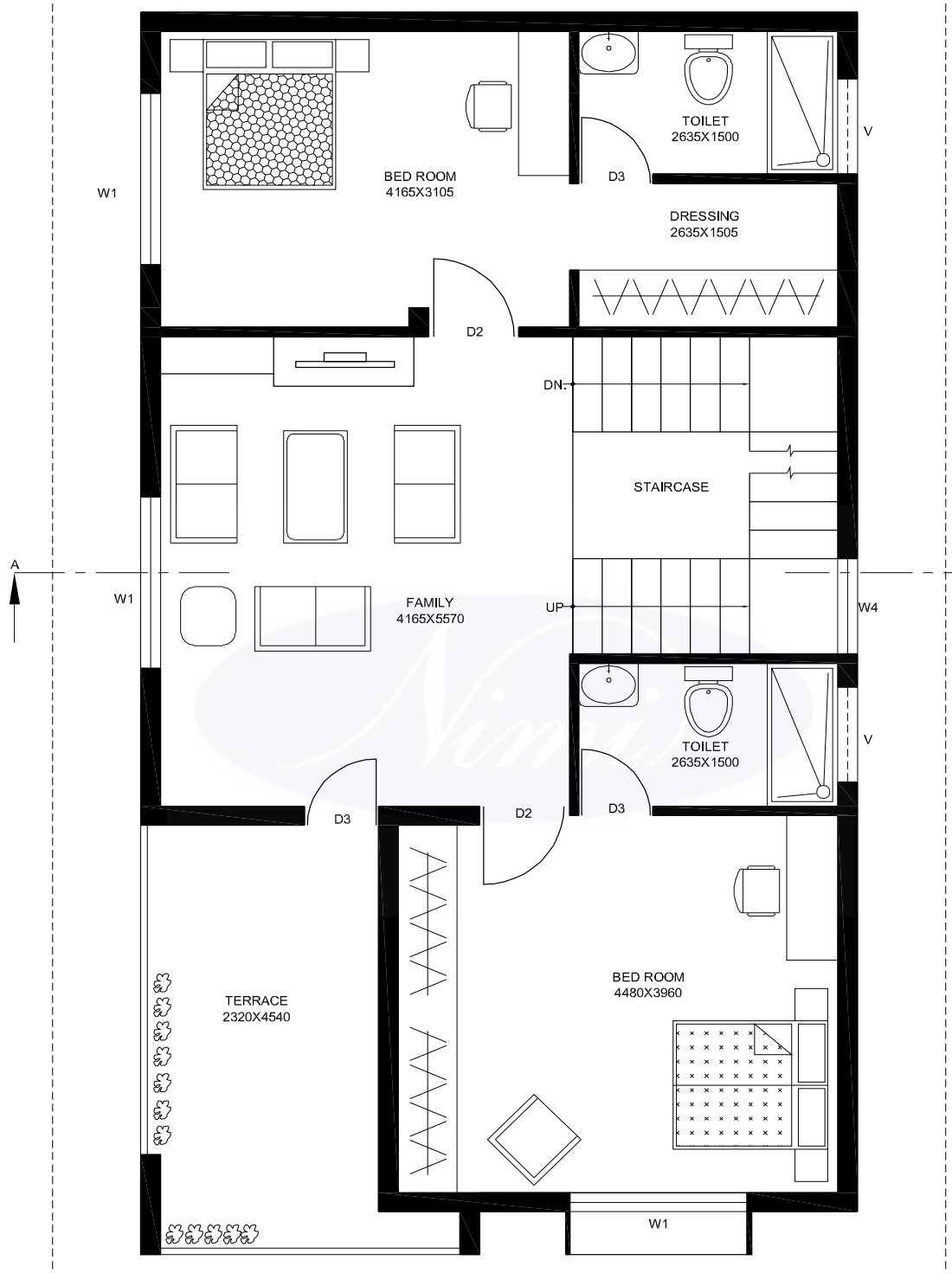
13 Complete the drawing in colour as per Fig 3 & 4.

Fig 3



GROUND FLOOR PLAN

Fig 4



D/W/V SCHEDULE	
D2	900X2100
D3	750X2100
W1	1800X1800
W4	900X2100
V	750X750

AA2116E4

FIRST FLOOR PLAN

Design development of front elevation and one side elevation(Manual)

Objectives: At the end of this exercise you shall be able to
- **design the front elevation based on the designed plan**
- **design the side elevation based on the designed plan.**

Data :

Floor to floor height = 3000 mm

Plinth height = 450 mm

Parapet height = 1000 mm

Staircase headroom height = 2400 mm

PROCEDURE

TASK 1 : **Design front elevation and side elevation(manual)**

1 The first step in elevation design is to choose an architectural style.

2 Sketch the outline of an exterior wall showing the roof shape and the position of doors, windows and other key features.

3 Create a series of progressive sketches to develope an elevation design.

4 Experiment with different roof styles, doors and window designs, siding materials for the exterior walls, overhangs, roof materials, etc.

5 Sketches can also show various architectural styles derived from the same floor.

6 Sufficient roof over hang should be provided to afford protection from the sun, rain and snow.

7 The portion facing the road side is considered as front elevation and any left or right side is considered as side elevation.

8 Plan window height to allow for furniture and built in components that are placed near windows.

9 Windows can be detailed to represent the exact type of window called for in this design.

10 Door, windows or other openings should be uniform in design and located to present a symmetrical appearance to the elevation except where the variations are an integral and necessary part of the exterior design.

11 Roof and roof lines should be continuous in design except where there is a major change in an element of a building elevation. Such elements include wind walls, fanwalls and interior building corners.

12 Roof line elements including parapet walls should be developed along all elevations, regardless of orientation away from sheets or towards a neighbouring structure.

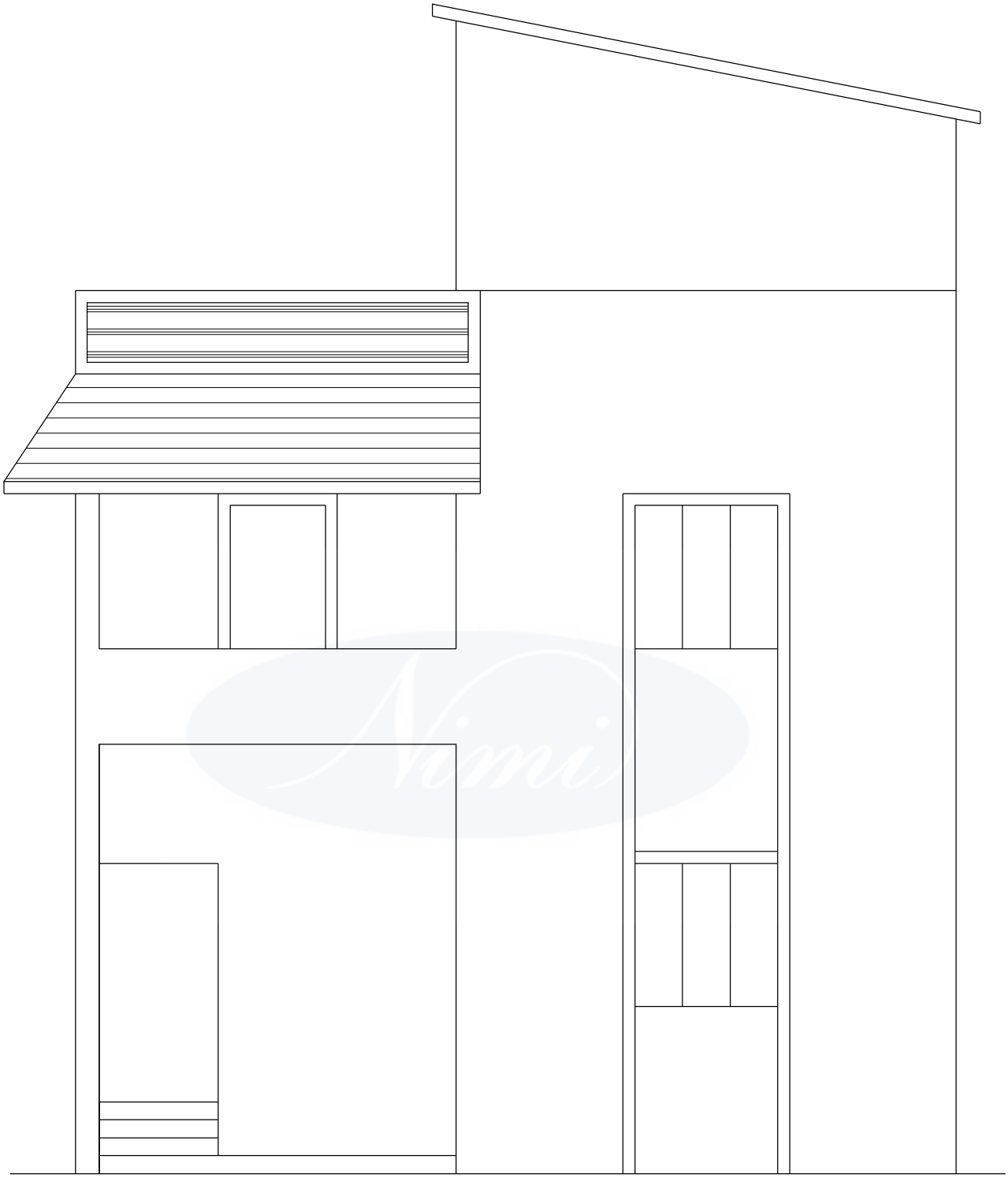
13 No mechanical equipment, dueting, meters or other elements should be left exposed at the ground level or on roofs.

14 Attachments in the form of pergolas, verandahs, screens improve the visual look of a house through layering and increasing perceived depth of facades in their shadowing effects.

15 The general form and style of attachments should enhance the visual look of the house and not detract from it.

16 With consideration of all the element in to design complete the sketches an per Fig 1 & Fig 2.

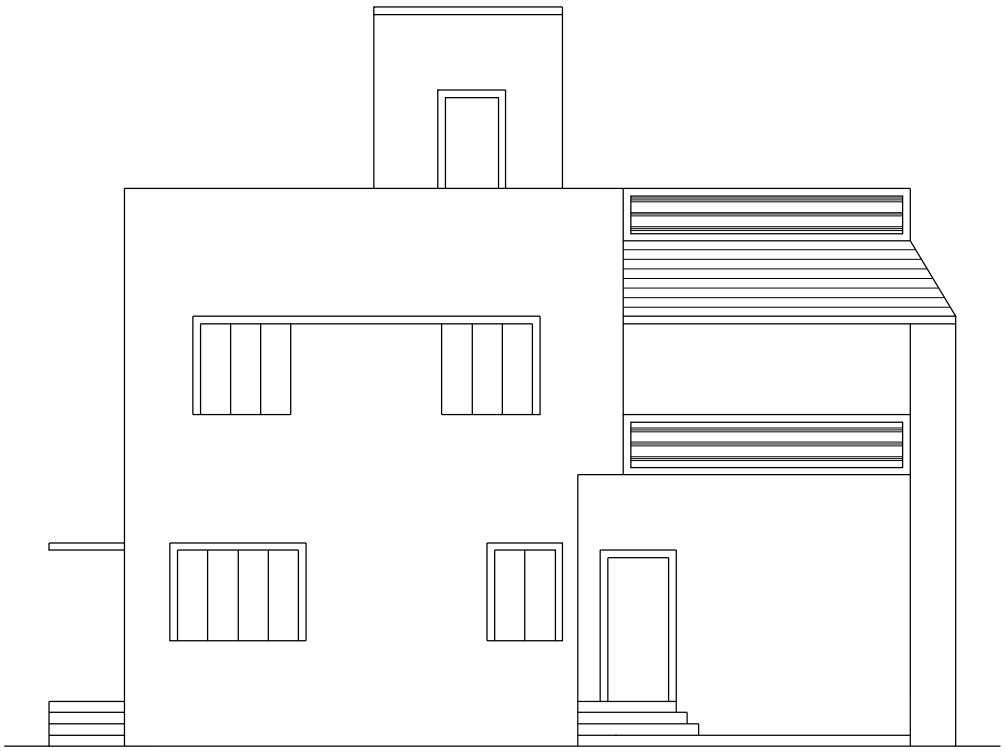
Fig 1



FRONT ELEVATION

AA2117E1

Fig 2



SIDE ELEVATION

AA2117E2

Draft front elevation and side elevation (CAD)

Objectives: At the end of this exercise you shall be able to
- **draft the front elevation in CAD**
- **draft the side elevation in CAD.**

PROCEDURE

TASK 1 : **Draft the front elevation (CAD)**

1 Using the floor plan, project the vertical lines that represent the main lines of the building.

2 These lines show the overall length or width of the building.

3 When projecting an elevation on a cad system, use the grid pick junction to project the major lines from the floor plan to the elevation plan.

4 Measure and project horizontal lines that represent the height of the ground line, doors, top & bottom of windows and other key features.

5 Add details and symbols, such as indicating door and window bim, mullions, roofing materials, etc.

6 Represent textures or features as designed in front elevation.

7 Complete the front elevation with all the details as shown in Fig 1.

— — — — — — — —

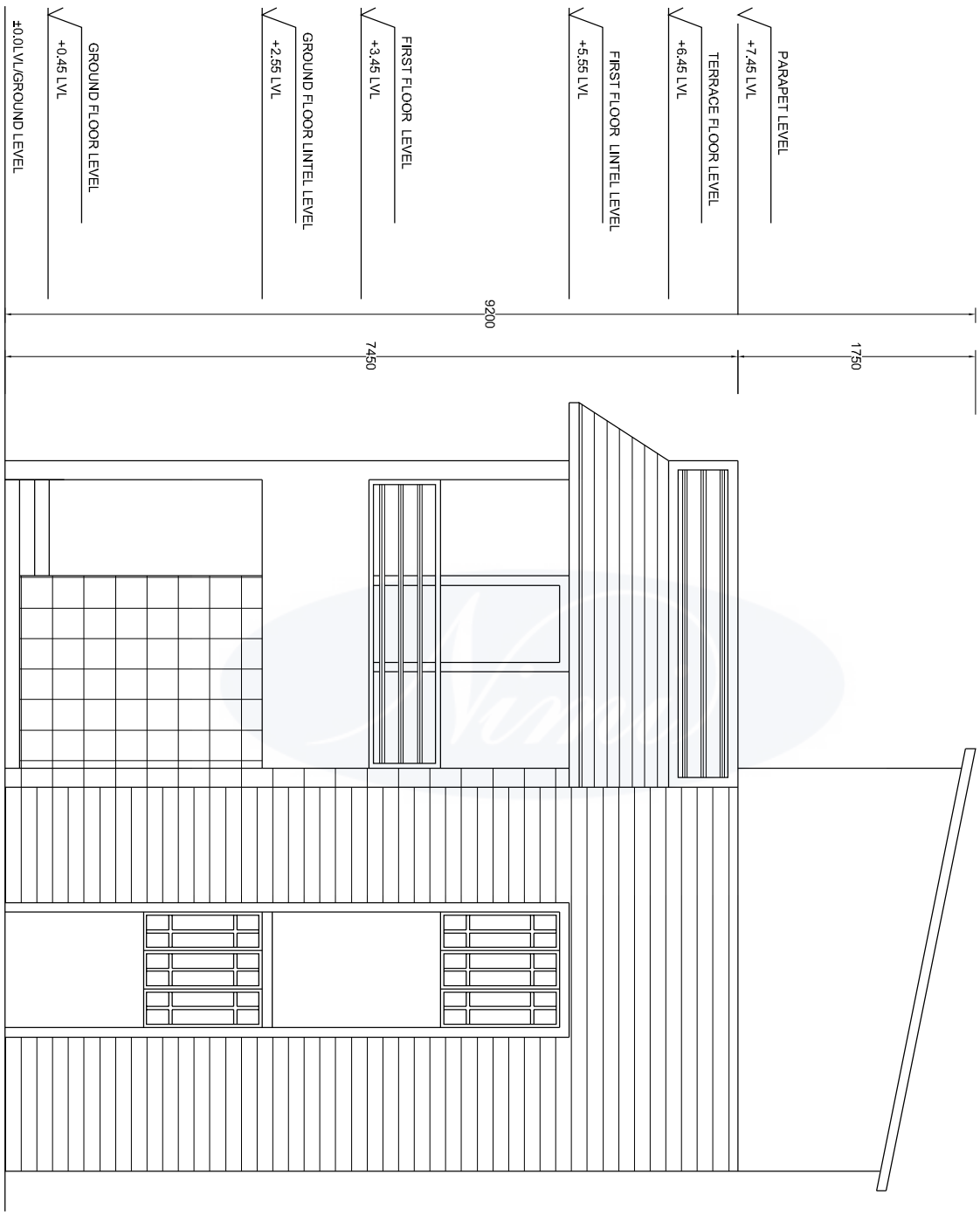
TASK 2 : **Side elevation - CAD**

1 During the drawing process floor plans can be rotated 90^0 to position for side elevation.

2 Follow the same procedure as TASK 1 and complete the side elevation as shown in Fig 2.

— — — — — — — —

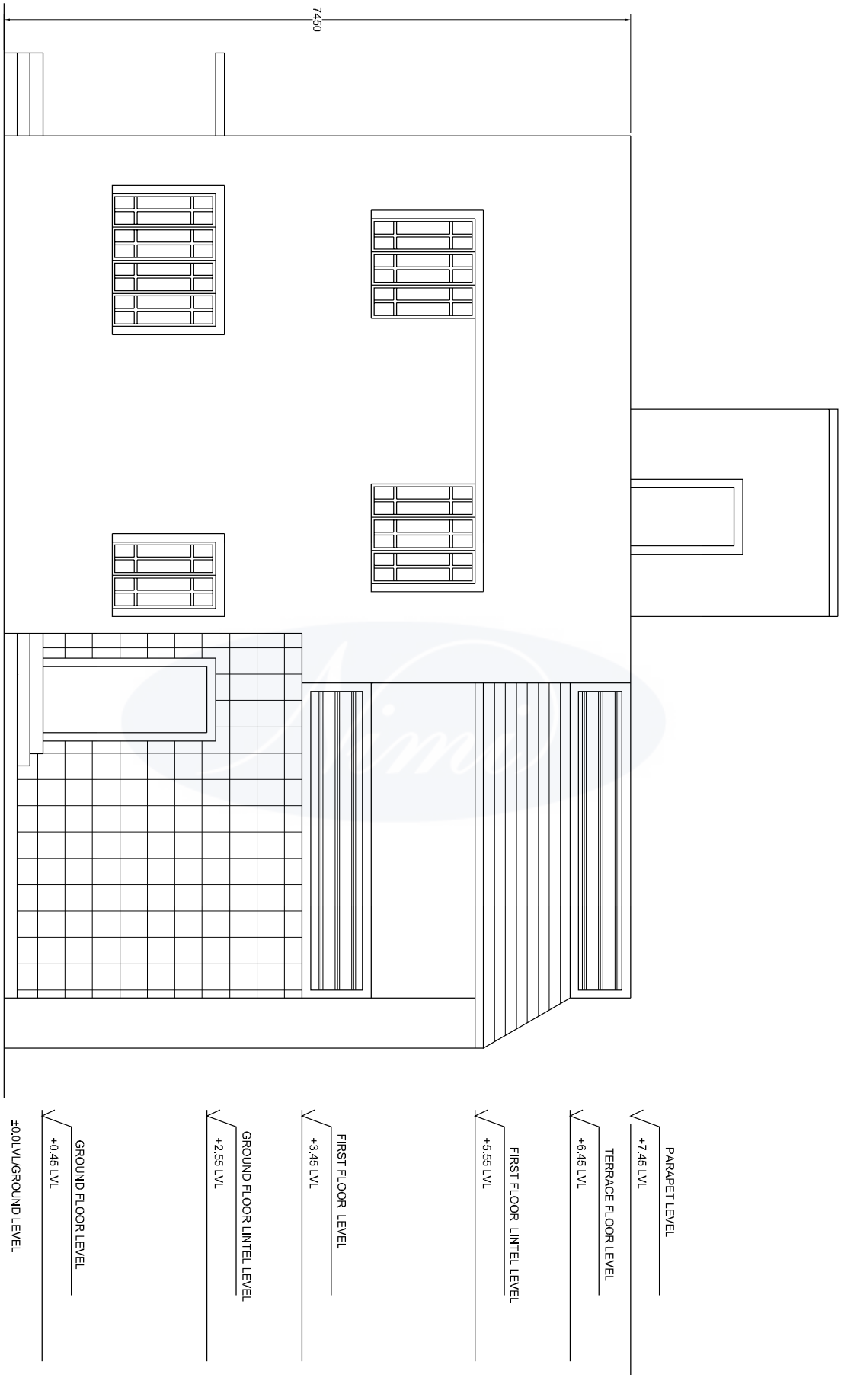
Fig 1



FRONT ELEVATION

AA2118E1

Fig 2



SIDE ELEVATION

AA2118E2

Draw final front elevation and one side elevation with rendering (CAD)

Objectives: At the end of this exercise you shall be able to
- **render the front elevation (CAD)**
- **render side elevation (CAD).**

PROCEDURE

TASK 1 : **Render fornt elevation and side elevation**

1 A rendered elevation is often used as a presentation drawing to help show the shape of the structure.

2 Using CAD you can add patterns representing building materials or shading to your drawing.

3 For the main building light colours should be used since dark colours make the building look dull and condensed.

4 Feature elements should be identified and different materials or colours may be choosen.

5 For doors and windows use shades of brown and for glass use gradient of light blue.

6 The elevation can be enhanced by adding bees, shrubs, people, sky, cars, etc.

7 The outline of the building needs to be dark and all the hatches and colours need to light.

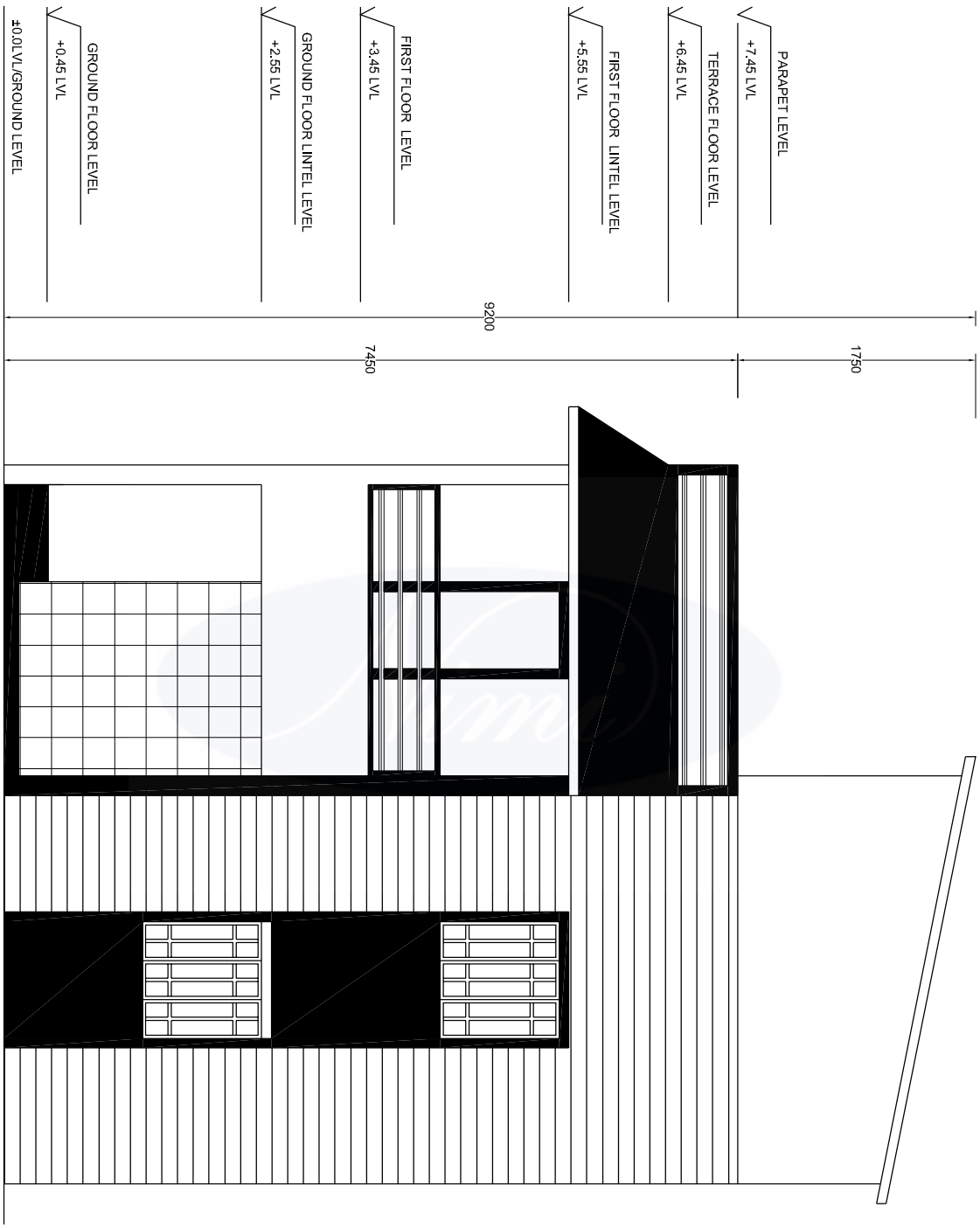
8 The use of texture in design must be carefully considered in combination with the use of colour.

9 Different colours for hatches can also be used.

10 Prepare a layout individually for front elevation and side elevation.

11 Complete the rendering of front elevation and side elevation as shown in Fig 1 & Fig 2.

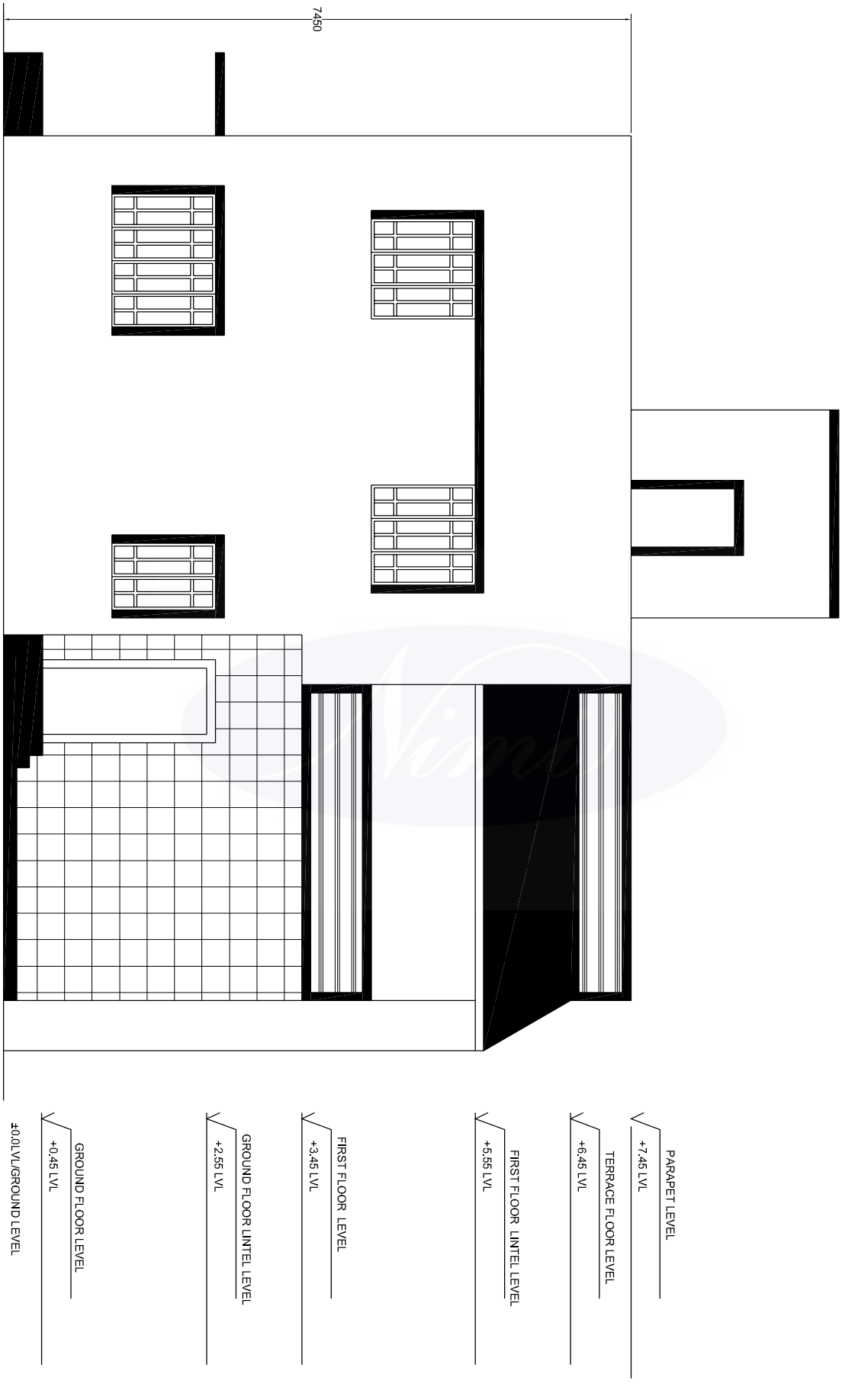
Fig 1



FRONT ELEVATION

AA2119E1

Fig 2



SIDE ELEVATION

AA2119E2

Develop section of the building (CAD)

Objectives: At the end of this exercise you shall be able to
- **develop the section of the building in CAD**
- **place the section in a layout.**

PROCEDURE

TASK 1 : **Section of the building**

1 Mark the section plane in plan in such a way that max details can be viewed through.

2 Once the section line is marked, project lines perpendicular from the floor plan outline at each corner.

3 Add ceiling lines and floor lines to give each wall its specified height.

4 Once the features of the walls are projected to the section from the floor plan, dimensions, instructional notes and additional features can be added to the drawing.

5 Interior elevations provide a great amount of detail the height of all cabinets, shelving, ledges, railing, wall lamps, fixtures, mirror, furnitures, etc.

6 The vertical dimensions are as important as horizontal dimensions are on floor plan.

7 Many dimensions on a section drawings show the vertical distance from a datum line.

8 Vertical dimensions should be read from the right side of the drawing.

9 Levels to be dimensioned should be labeled with a note or term.

10 Room heights are shown by dimensioning from the floor line to the ceiling line.

11 The depth of the footing is shown as break in midway.

12 Heights of windows and doors are dimensioned from the floor line to the top of the window or door.

13 Windows and doors may be indexed by a code or a symbol to a door or window schedule.

14 Dimensions for small or obscure areas should be indexed to a separate detail.

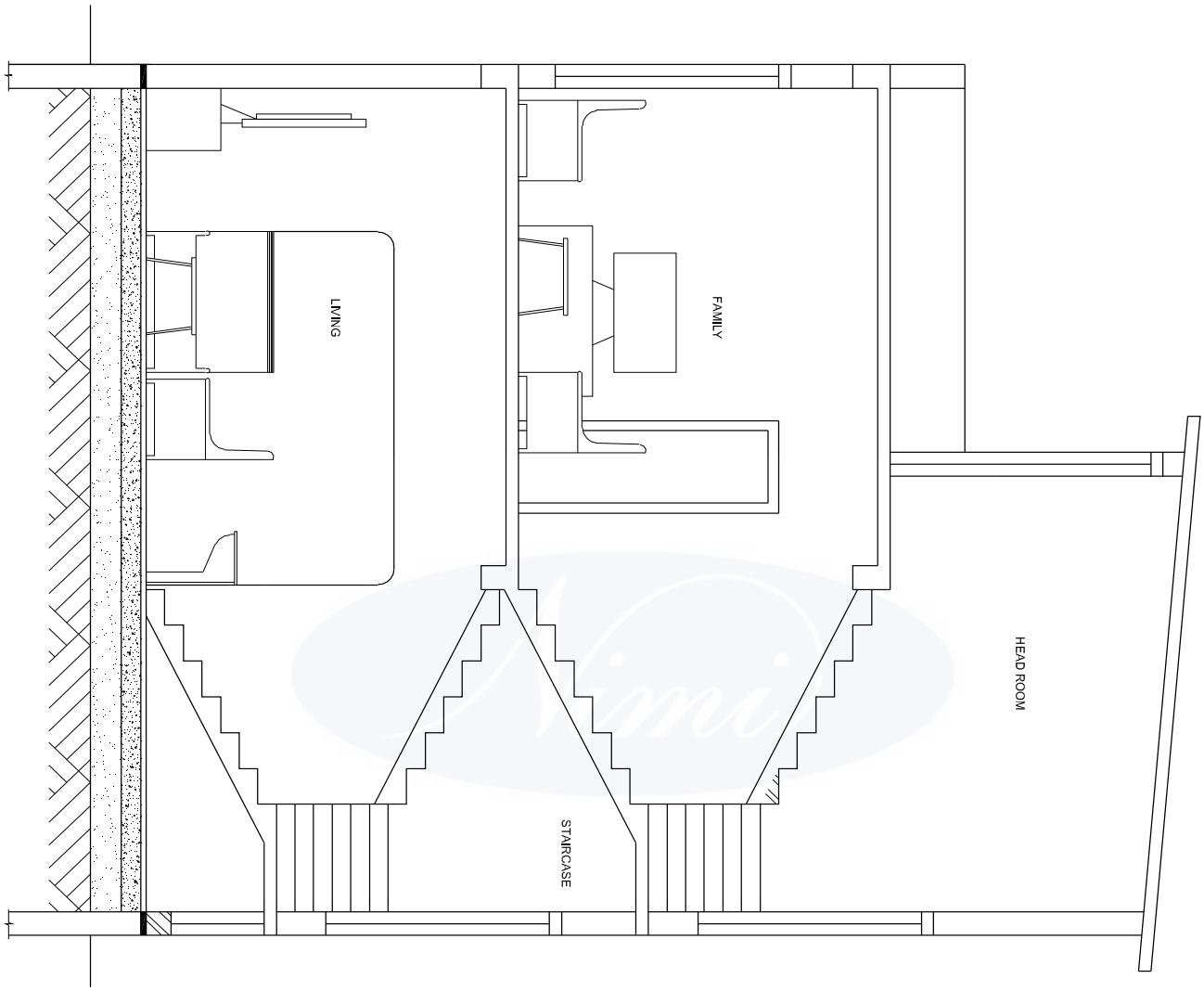
15 Floor and ceiling lines are shown with hidden lines.

16 Thickness of slabs are dimensionel.

17 Overall height are placed on the outside of sub dimensions.

18 Complete the drawing as shown in Fig 1.

Fig 1



SECTION ON AB

AA2120E1

Draw final section of the building with rendering (CAD)

Objectives: At the end of this exercise you shall be able to
- **render the section in CAD**
- **place the section in a layout.**

PROCEDURE

TASK 1 : **Rendered section of the building**

1 Using CAD you can add patterns representing various building materials or shading to your drawing.

2 Slabs can be of solid hatch and black in colour.

3 Walls can be solid hatch of red colour or brick symbol.

4 The flooring concrete can be indicated by hatching concrete symbol.

5 The colours for walls should be choosen with proper considerations mostly light shades are preferred.

6 The furnitures should be hatched with patterns or colours.

7 Elements like shrubs, lamp post, fan, vase etc. can also be added to enhance the section.

8 For doors and windows use shades of brown and for glass use gradient of light blue.

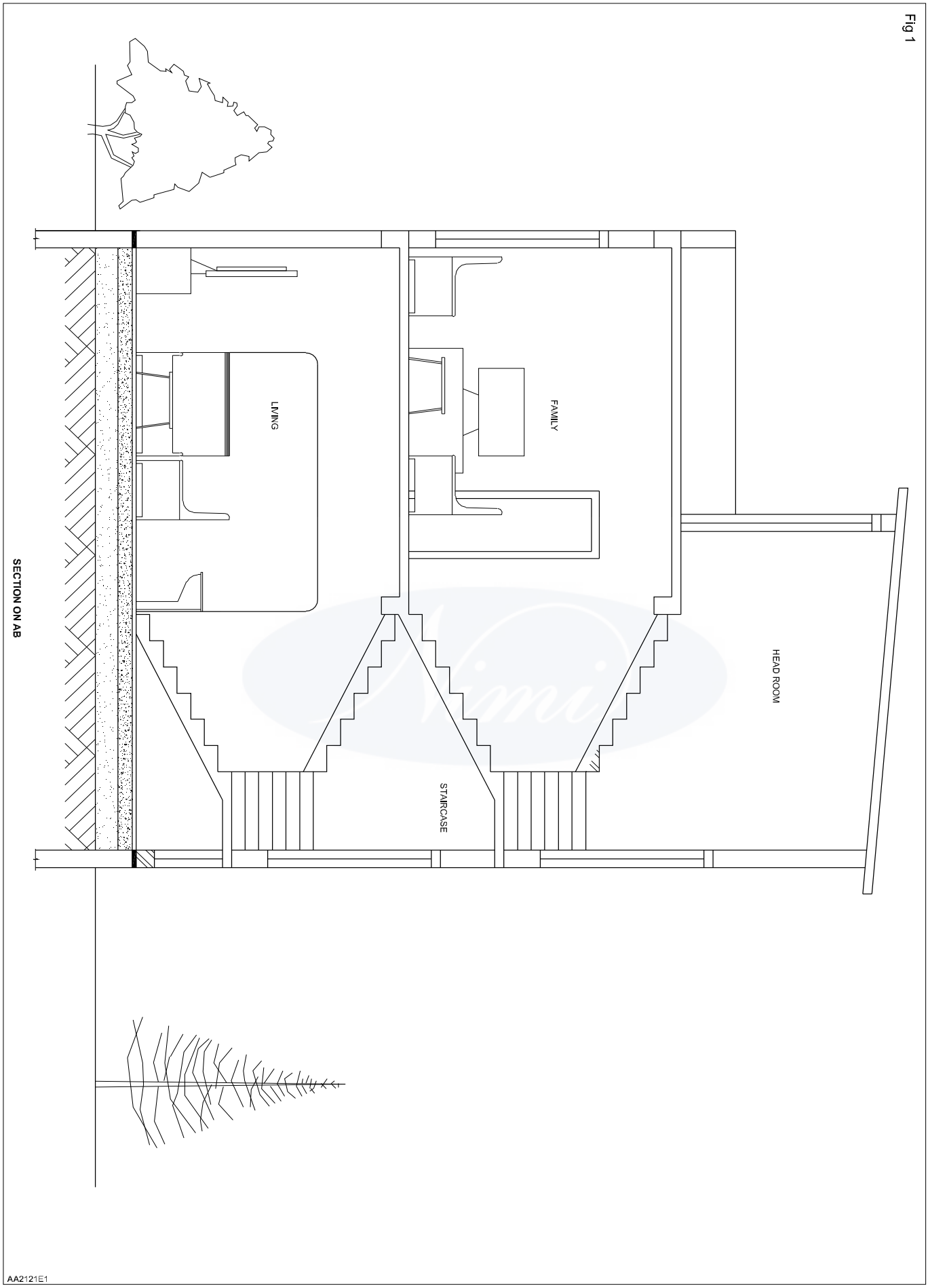
9 To enhance the section trees, shrubs, people, sky, etc. can be added.

10 The use of texture in design should be considered in combination with the use of colour.

11 Prepare a layout individually and place the drawing.

12 Complete the rendering of the section as shown in Fig.

Fig 1



Drawing of different types of arches

Objectives: At the end of this exercise you shall be able to

- **Draw the elevation of**
  - **flat Arch**
  - **semi circular Arch**
  - **semi Elliptical Arch (with five centres) and**
  - **venetian Arch.**

PROCEDURE

Exercise No 2.1 to 2.4

Draw the elevations of Flat arch semi circular arch, semi elliptical arch and venetian arch.

Exercise No: 2.1 (Flat arch)

Draw the elevation of Flat arch (gauged type)

Width of opening = 1200 mm

The depth or height of arch = 300 mm

Angle of the skewback = 60^0

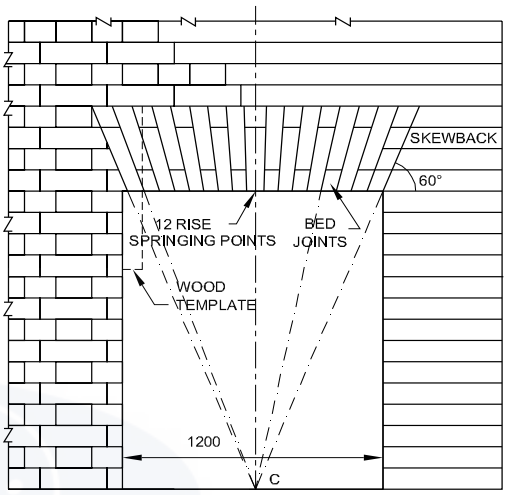
Rise of the arch at the middle = 12 mm

Width of each voussoir at extrados = 100 mm

Thickness of mortar joints = 10 mm

- Draw the opening to a width 1200 mm and to a convenient height by indicating the break line.
- Draw the skewback at an angle of 60^0 on both sides of opening.
- Draw a line parallel to the top of opening at a distance of 300mm for fixing the height of the arch.
- Project the skewback lines backwards to findout the centre 'c' of the arch.
- Mark the width of voussoirs on top of extrados. Join these points with the arch centre to draw voussoirs.
- Draw the brick layers in elevation on both sides of the opening and top of the arch.
- Complete the elevation of flat gauged arch as shown in Fig 1.

Fig 1



FLAT ARCH

Exercise No:2.2 (Semi circular arch):

Draw the elevation of semi circular arch

Width of opening = 1200 mm

The depth or height of arch = 2 rings. = 200 mm

Angle of skewback = 0^0 (i.e horizontal)

Rise of the arch at the middle = 1/2 span of opening

= 600 mm

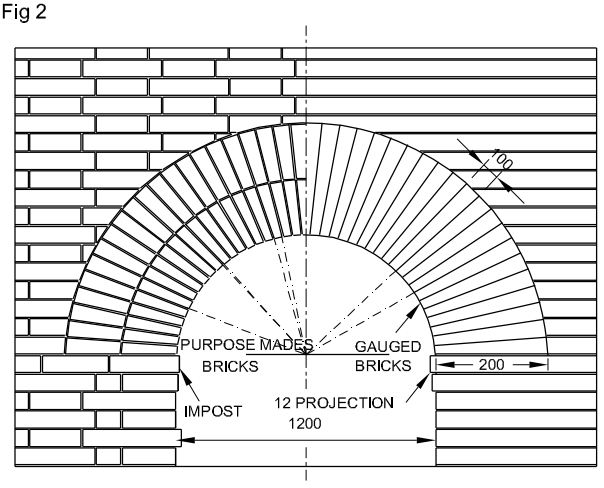
Width of each voussoir at extrados = 100mm

Thickness of mortar joints = 10mm

Procedure

- Draw the opening to a width of 1200 mm
- Draw the springing line and mark the centre of arch
- Draw the intrados with a radius of 600 mm and extrados with a radius of 800 mm from the arch centre.
- Draw the voussoirs as detailed in Fig 2 (Left side half elevation is made with purpose made bricks with 2 rings and the right half elevation is made with gauged bricks).The arch may be constructed with two rings of purpose made bricks of single ring of structure.
- Draw the elevation of remaining wall and complete the drawing as shown in Fig 2

Fig 2



SEMI CIRCULAR ARCH

Exercise No:2.3 (Semi Elliptical arch with five centres):

Draw the elevation of semi Elliptical arch with five centres

Width of opening = 1800 mm

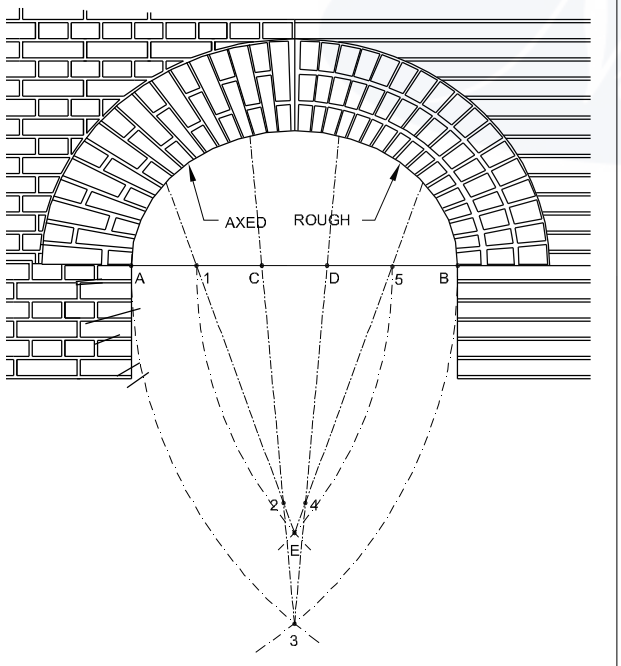
The depth or height of arch = 300 mm

Angle of the skew back = 0^0 (ie horizontal)

- Draw the opening to a width of 1800 mm
- Draw the springing line AB and divide it into 5 equal parts A1 = 1 C = CD = D5 = 5B

  From the Centre A and B draw arc of radius equal to the span, to intersect at 3. Join the point 3 with C and D.
- From the centres 1 and 5, draw arcs of radius equal to 3/5th of span, meeting each other at point E.
- Join points 1 and E. The point of intersection of lines 3C and E1 is 2. Similarly join 5 and E. The point of intersection of lines 3D and E5 is 4
- The points 1,2,3,4 and 5 represent five centres of the arch.
- Draw the left half elevation as axed arch and the right half elevation as rough arch.
- Draw the arrangement of voussoirs and the remaining elevation of wall as shown in Fig 3.

Fig 3



SEMI ELLIPTICAL ARCH

Exercise No. 2.4 (Venetian arch)

Draw the elevation of venetian arch

Width of opening = 1200 mm

Rise = 900 mm

The depth or height or arch at skewback = 200 mm

- Draw the springing line AB to 1200mm length.
- From the midpoint of AB, mark the rise at C.
- Join AC and BC to complete the triangle ABC
- Draw the skewback AD and BE on eitherside to a length 200 mm
- Draw DF parallel to AC and EF parallel to BC to Complete the triangle DEF.
- Draw perpendicular bisectors for AC and BC. They meet the springing line at r1 and r2 which are centre points to draw intrados.
- Similarly draw perpendicular bisectors for DF and EF. Let them meet the springing line at R1 and R2 which are centre points to draw extrados.
- Draw intrados with centres r1 and r2 and extrados with centres R1 and R2 as shown in the figure 4.
- Draw the voussoirs

Complete the remaining elevation as shown in Fig 5.

Fig 4

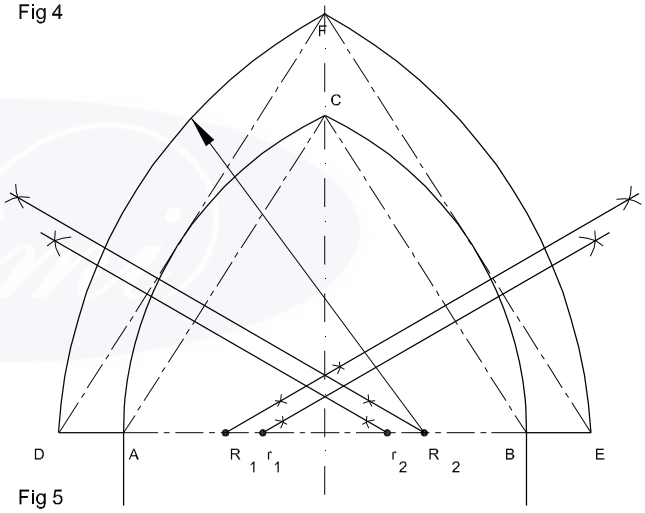
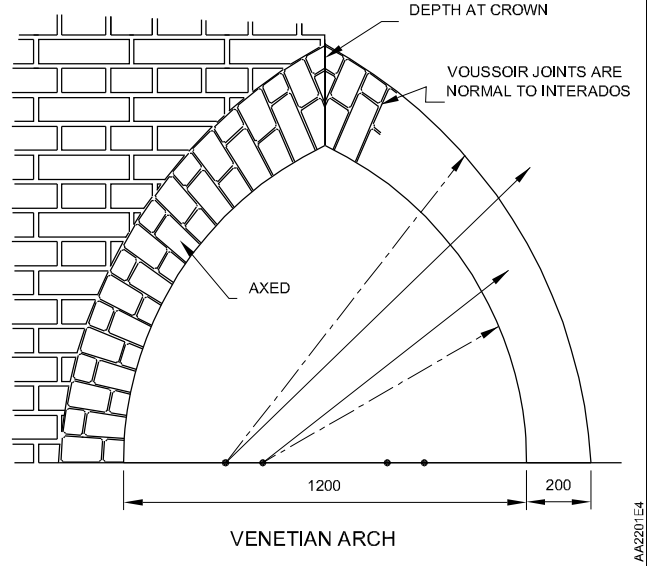


Fig 5



VENETIAN ARCH

Drawing of different types of lintels

Objectives: At the end of this exercise you shall be able to
- **draw the elevation and section of wooden lintel**
- **draw the elevation of brick lintel**
- **draw the elevation and section of stone lintel**
- **draw the section of R.C.C lintel and details of bars.**

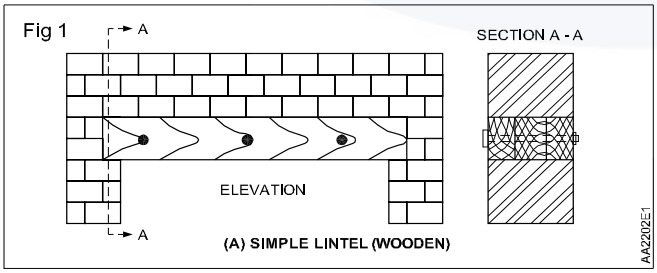
PROCEDURE

Exercise No 2.1 to 2.4

Draw various views of wooden, stone, brick, steel and R.C.C. lintel

- Draw a span of 1000mm
- Draw 100 mm thick wooden Lintel
- Draw bearing of Lintel 100 mm
- Draw three bolt and nuts in elevation
- Draw three courses of Brick masonry on the top of the wooden Lintel
- Draw a section at AA
- Draw a section of wall 300 mm
- Draw and develop the section at AA
- Draw convensional symbols
- Complete the drawing as shown in Fig 1.

Fig 1



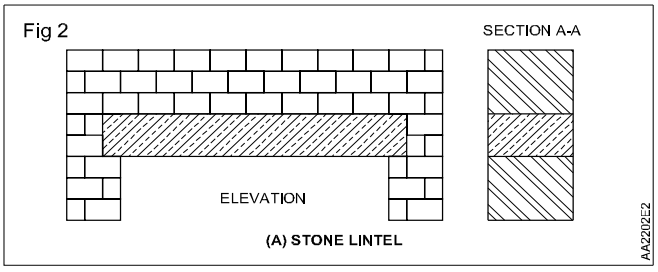
Exercise No: 2.2 Stone Lintel

The size of stone Lintel 220 mm

Draw the elevation of stone Lintel similar to wooden Lintel

Draw bearing of Lintel 250 mm (Fig 2)

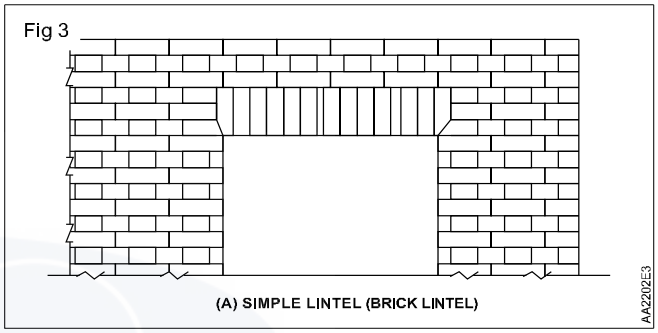
Fig 2



Exercise No.2.3 (Brick lintel)

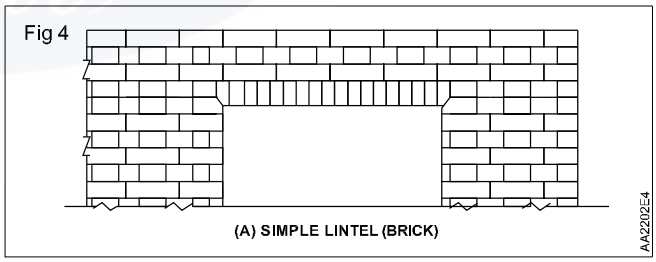
- Draw the elevation of brick Lintel
- Draw a span of 1000 mm
- Draw the elevation of wall
- Draw brick Lintel width of brick 100 mm and depth of brick 300 mm
- Draw and develop the brick Lintel as shown in Fig 3.

Fig 3



- Similarly Draw another form of brick Lintel as shown in Fig 4

Fig 4



- width of brick 100 mm and
- depth of brick 150 mm.

Exercise No. 2.4

Steel Lintel :(Fig .5,6,7)

Size of Rolled steel Joist

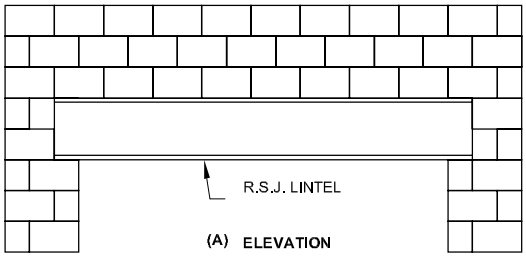
Indian standard medium Weight beam (ISMB) 150 x 80 - 14.9 Kg - 1 No.

Indian standard Junior beam (ISJB)

175 x 50 - 8.1Kg.

- Draw a span of 2500 mm
- Draw ISMB 150 x 80 - 1No. at the middle of the wall.
- Draw Bearing 150 mm on both sides.
- Draw three courses of brick masonry on the top of the lintel (Fig 5)

Fig 5

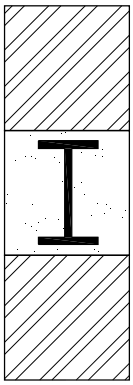


(A) ELEVATION

AA2202E5

- Draw a section view of steel beam. (Fig 6)

Fig 6

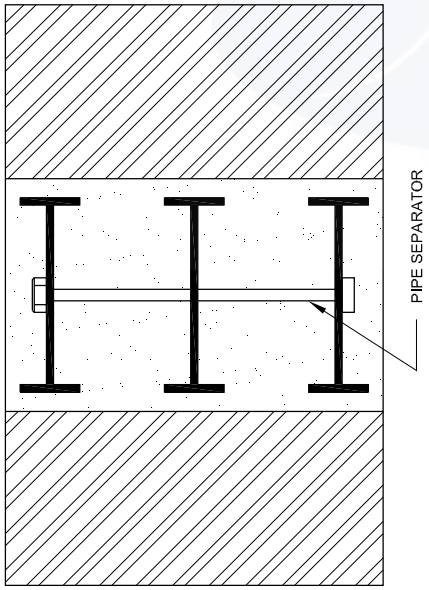


(b) CONCRETE EMBEDMENT

AA2202E6

- Draw and develop multiple units of I - section connected with pipe separator. (Fig 7)

Fig 7



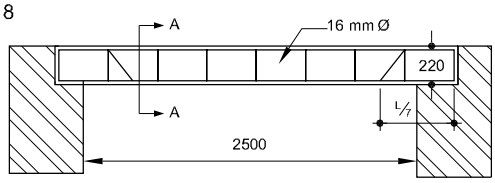
(C) MULTIPLE UNITS

AA2202E7

Exercise No: 2.5 R.C.C Lintel

- Draw a span of 2500 mm
- Draw a depth of beam 220 mm
- Draw a bearing of Lintel 200 mm
- Draw a bent up bar 300mm from L/7 both supports - 1 No.
- Draw straight bar at bottom of Lintel 16mm f - 2 Nos.
- Draw anchor bar at Top of Lintel 16mm f - 2 Nos.
- Draw stirrup 6 mm f uniformly spaced 360 mm c/c Fig 8

Fig 8

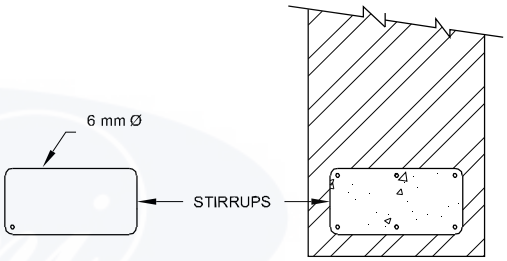


(A) ELEVATION

AA2202E8

- Draw a section at AA (Fig 9)

Fig 9

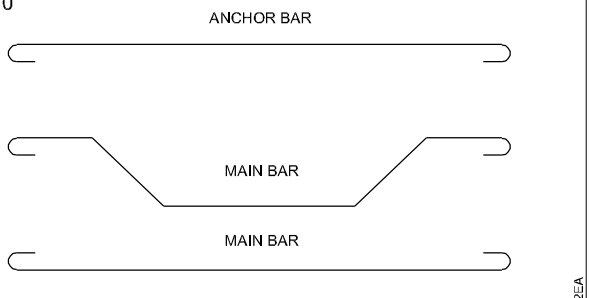


DETAILS OF A REINFORCED CONCRETE LINTEL

AA2202E9

- Draw the details bar bending. (Fig 10)
- Complete the drawing.

Fig 10



(A) ELEVATION

AA2202EA

Draw plan and elevation of straight stairs

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of a straight stair.**

PROCEDURE

TASK 1 : **Draw the plan of a straight stair**

Draw the plan and elevation of a straight stair from floor to floor

Data

Height betweeen the floors : 3000

The rise and going : 150 and 300 respectively

Total no of steps in the flight : as per the rule

Width of the flight : 1000

Diameter of GI pipe handrail : 50

Diameter of newel post GI pipe : 25

Height of the baluster : 900

Width of the handrail : 50

Size of the window : 1000 x 1200

Note : No of risers -3 (min) & 12 maximum (all dimensions are in mm).

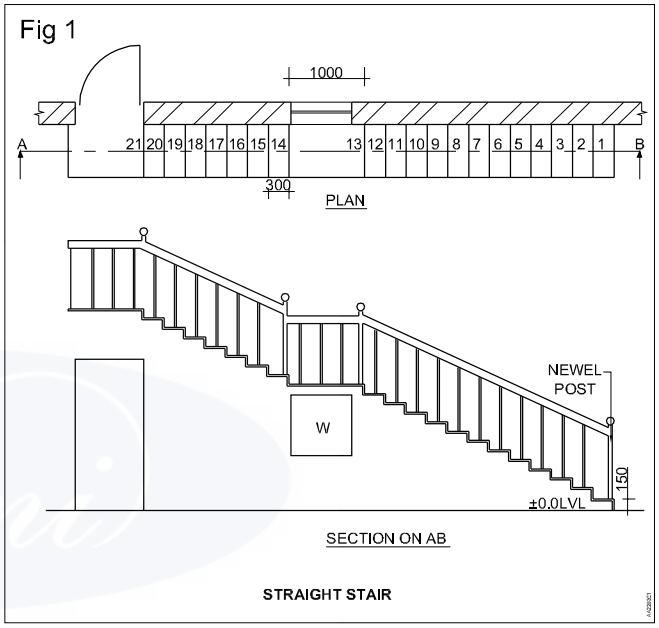
1 Draw the plan of the with proper number of treads.

2 Draw the landing after twelve risers.

3 Draw the treads (8nos) after the landing.

4 Dimensions the drawing properly. (Fig 1)

Fig 1



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TASK 2 : **Draw the elevation of the straight stair**

1 Draw the projected vertical lines to mark the risers.

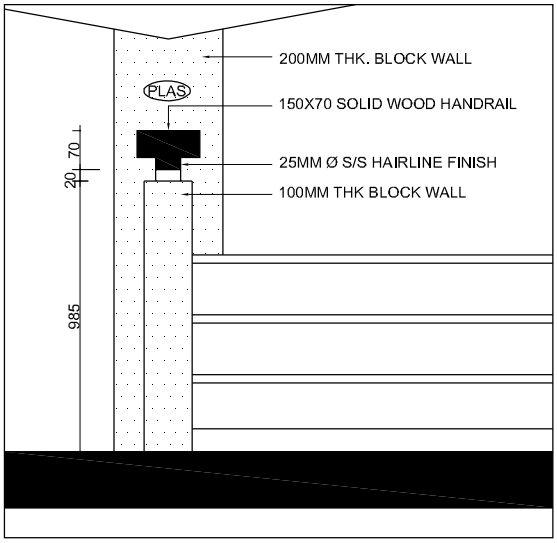
2 Draw the vertical lines to show the baluster.

3 Draw the handrail as per the drawing.

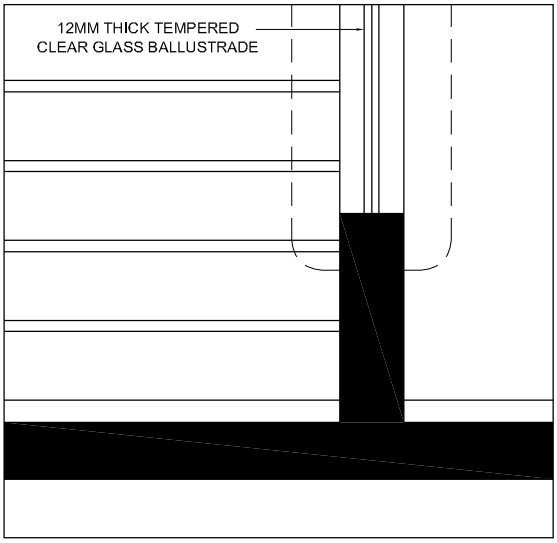
4 Draw the elevation of the window.

5 Dimension the drawing properly. (Fig 2)

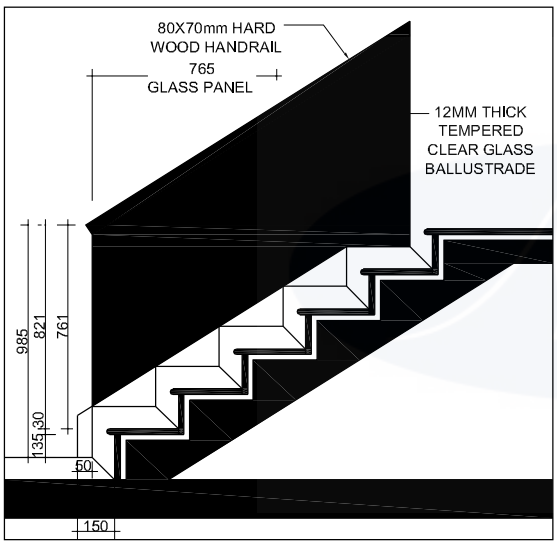
Fig 2



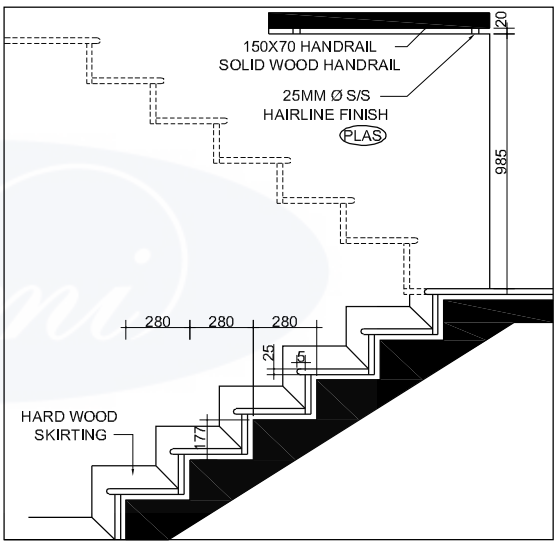
DETAIL - 4



DETAIL - 3



DETAIL - 1 (GF TO FF FLOOR STEP DETAIL)



DETAIL - 2 (FF TO TERRACE FLOOR)

AA2202E2

DETAILS

Draw plan and elevation of open well stairs

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of a open well stairs.**

PROCEDURE

TASK 1 : **Draw the plan of on open well stair**

Data

Height between the floors : 3000

Rise and going 150 and 300 respectively

Total no of steps in the flight as per the rule

Width of the flight : 1000

Width of the wall : 300

Diameter of GI pipe handrail : 50

Height of the baluster : 900

Width of the handrail : 50

Size of teh window : 1900 x 1200

Size of the well : 2400 x 500

Note : No of risers - 3 (min) & 12 maximum (all dimensions are in mm)

1 Draw the plan of the stair room with proper numbner of treads.

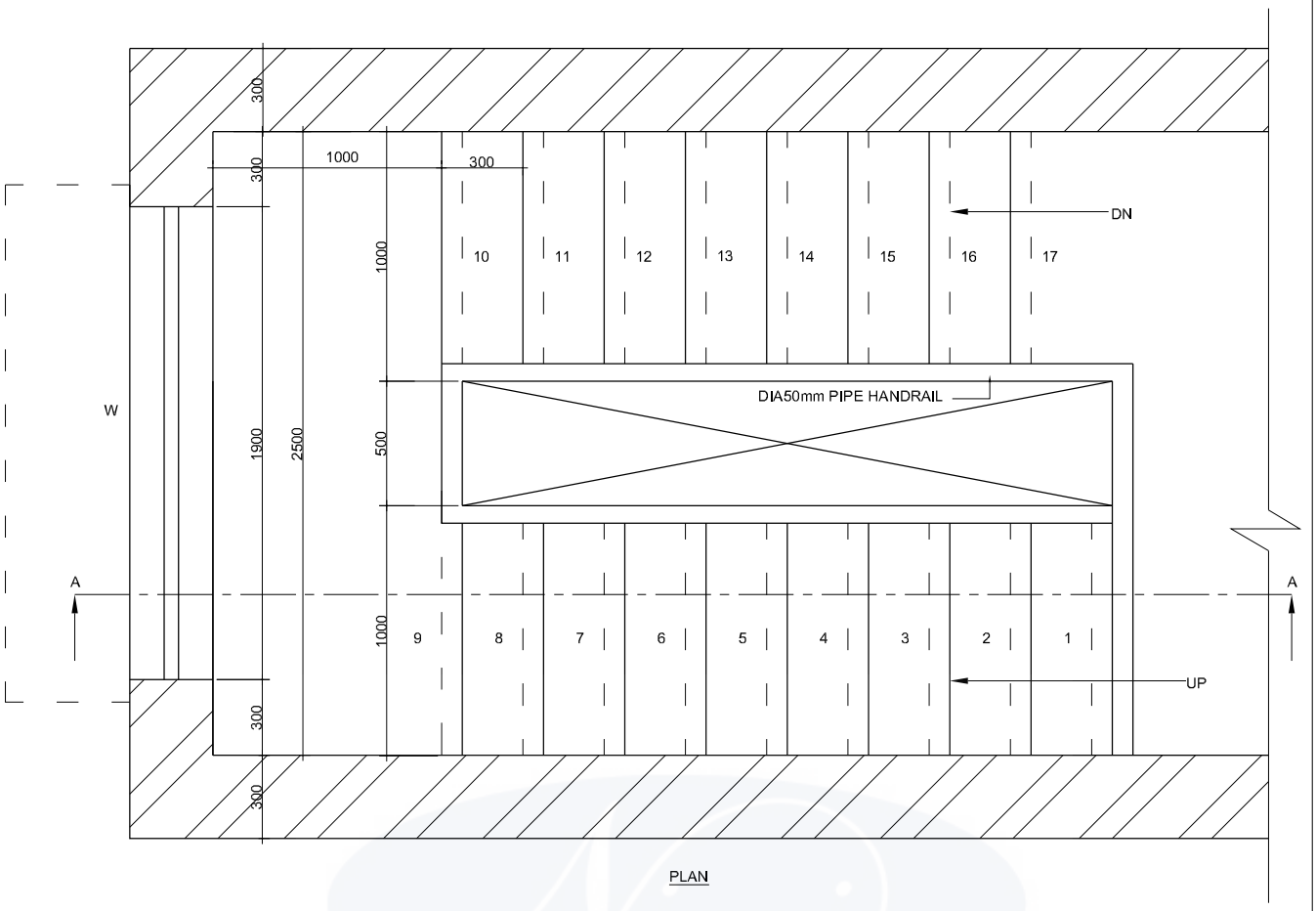
2 Draw the landing.

3 Draw the treads in the reverse direction with open well.

4 Draw the window opening.

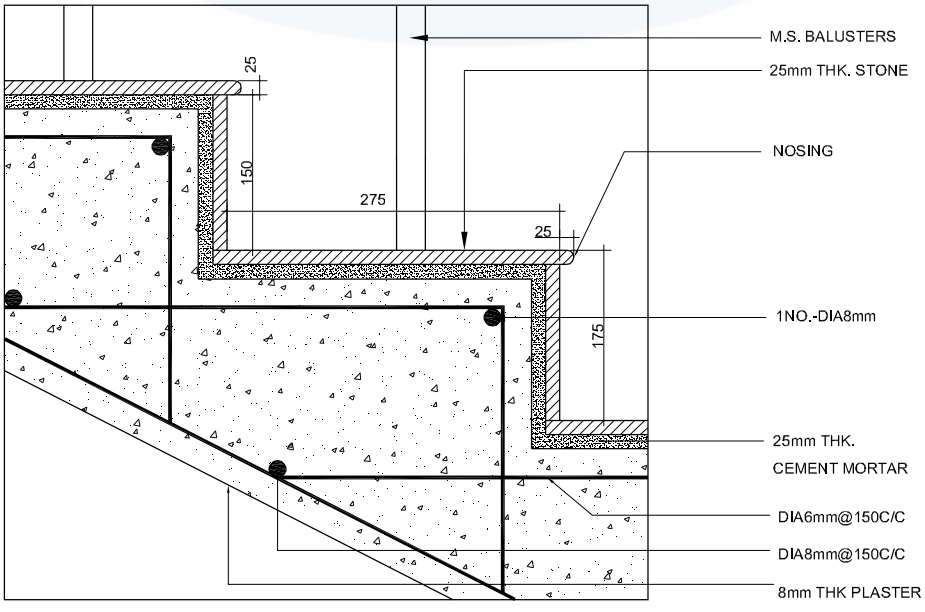
5 Dimensions and the drawing properly.

Fig 1



PLAN

Fig 1a



DETAIL A

OPEN WELL STAIR

AA2204E1

TASK 2 : **Draw the elevation of the open well stair**

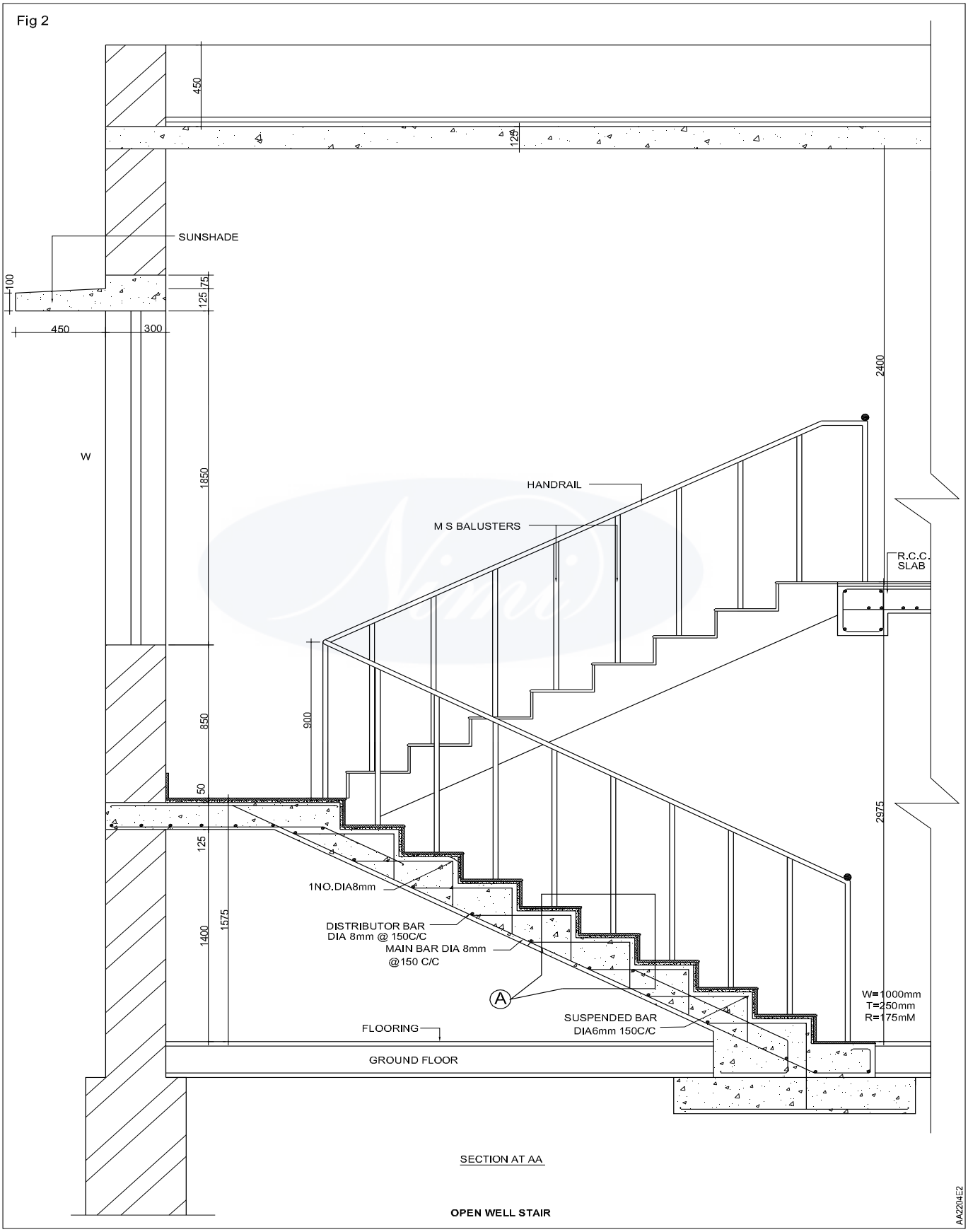
1 Draw the projected vertical lines to mark the sectional elevatin of the first flight.

2 Draw the vertical lines to show the elevation.

3 Draw the handrail, nosing, window and reinforcement details as per the drawing.

4 Complete the drawing with proper dimensioning.

Fig 2



Draw plan and elevation of bifurcated stairs

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of bifurcated stair.**

PROCEDURE

TASK 1 : **Draw the plan and elevation of a bifurcated stair from floor to floor**

Data

Height between the floors : 3000

Rise and going 150 and 300 respectively

Total no of steps in the flight as per the rule

Width of the flight : 1000

Diameter of GI pipe handrail : 50

Height of the baluster : 900

Note : No of risers - 3 (min) & 12 maximum (all dimensions are in mm)

1 Draw the plan of the stair with proper number of treads.

2 Draw the landing after risers.

3 Drawt the treads (8nos) after the landing in both right and left sides.

4 Draw the handrail in plan.

5 Dimensions and the drawing properly. (Fig 1)

Draw the elevation of the bifurcated stair

1 Draw the projected vertical lines to mark the risers.

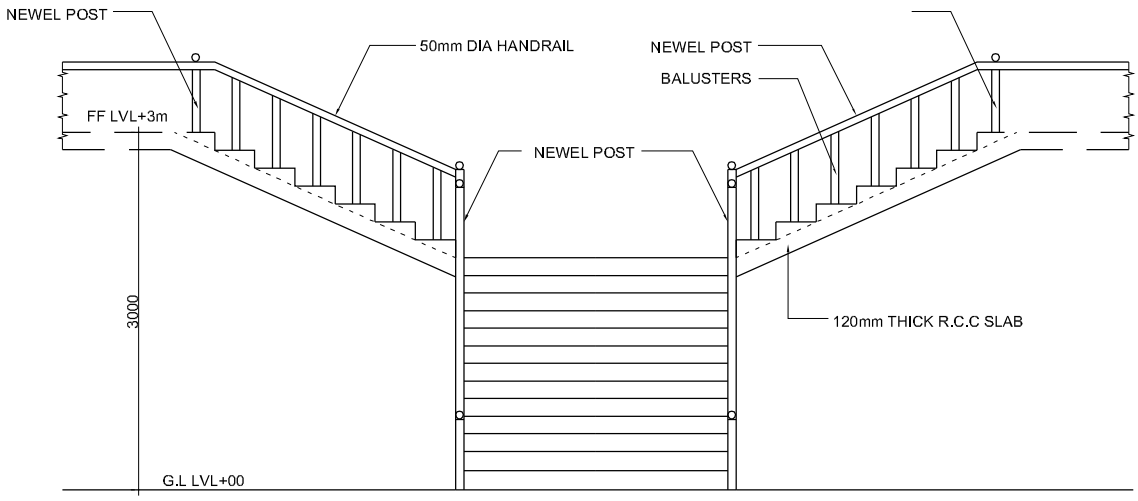
2 Draw the vertical lines to mark the baluster.

3 Draw the handrail details as per the drawing.

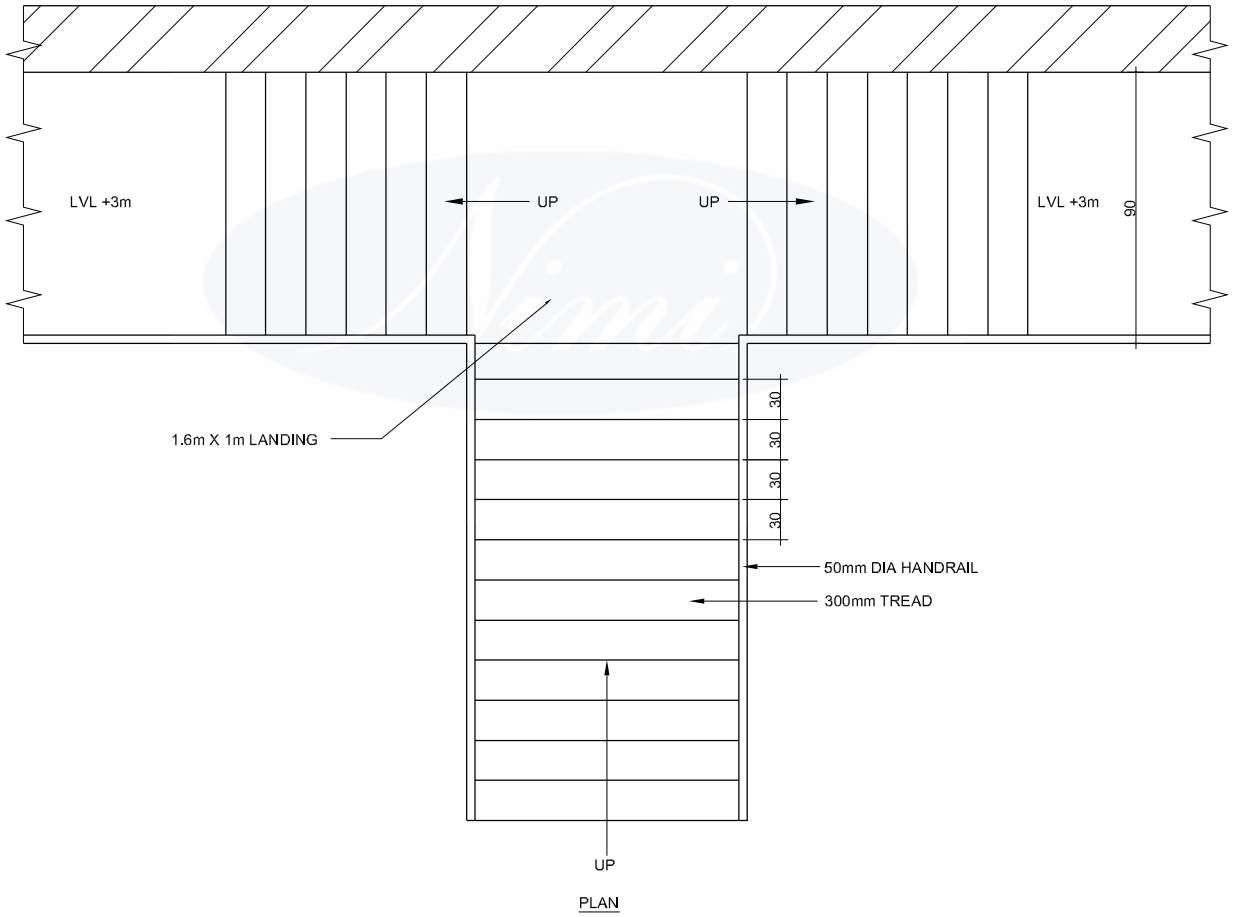
4 Draw the elevation of the handrail in both directions.

5 Dimension the drawing properly. (Fig 1)

Fig 1



ELEVATION



PLAN

BIFURCATED STAIR

AA2205E1

Draw plan and elevation of geometrical stair

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of a geometrical stair.**

PROCEDURE

TASK 1 : **Draw the plan of geometrical stair**

Data

Height between the floors : 3000

Rise and going 150 and 300 respectively

Total no of steps in the flight as per the rule

Width of the flight : 900

Diameter of GI pipe handrail : 50

Height of the baluster : 900

Width of the handrail : 50

Size of the window : 1000 x 1200

Note: No of risers - 3 (min) & 12 maximum (all dimensions are in mm)

1 Draw the plan of the stair and treads.

2 Draw the radiating treads from the centre.

3 Draw the handrail and window in plan.

4 Complete the drawing with necessary dimensioning.

Draw the elevation of the geometrical stair

1 Draw the projected vertical lines to show the risers.

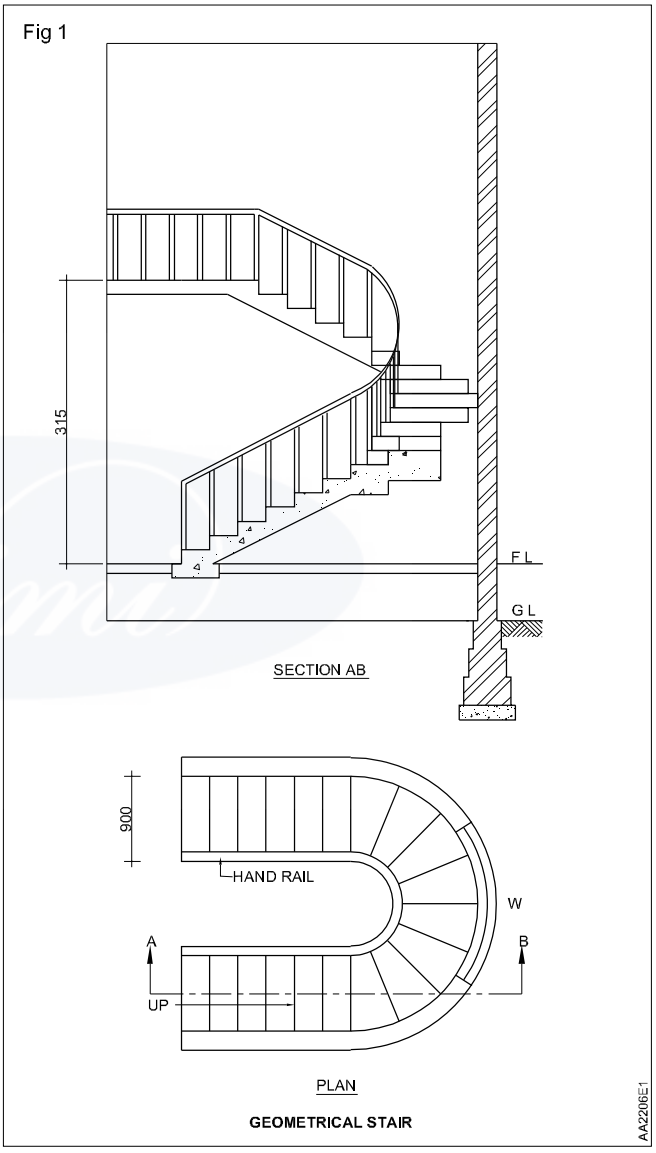
2 Draw the vertical lines to show the baluster.

3 Draw the handrail, window as per the drawing.

4 Draw the elevation of the window.

5 Dimension the drawing properly. (Fig 1)

Fig 1



Draw plan and elevation of circular stair

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of a circular stair.**

PROCEDURE

TASK 1 : **Draw the plan of circular stair**

Data

Height between the floors : 3000

Width of the flight : 900

Diameter of GI pipe handrail : 50

Height of the baluster : 900

Width of the flight : 50

Rise and going 173.6 and 300 respectively

Total no of steps in the flight as per the rule.

Note : No of risers - 3 (min) & 12 maximum (all dimensions are in mm)

1 Draw the plan of the stair with the given diameter.

2 Draw the radiating treads from the centre.

3 Draw the handrail in plan.

4 Complete the drawing with necessary dimensioning. (Fig 1)

Draw the elevation of the circular stair

1 Draw the projected vertical lines to show the risers.

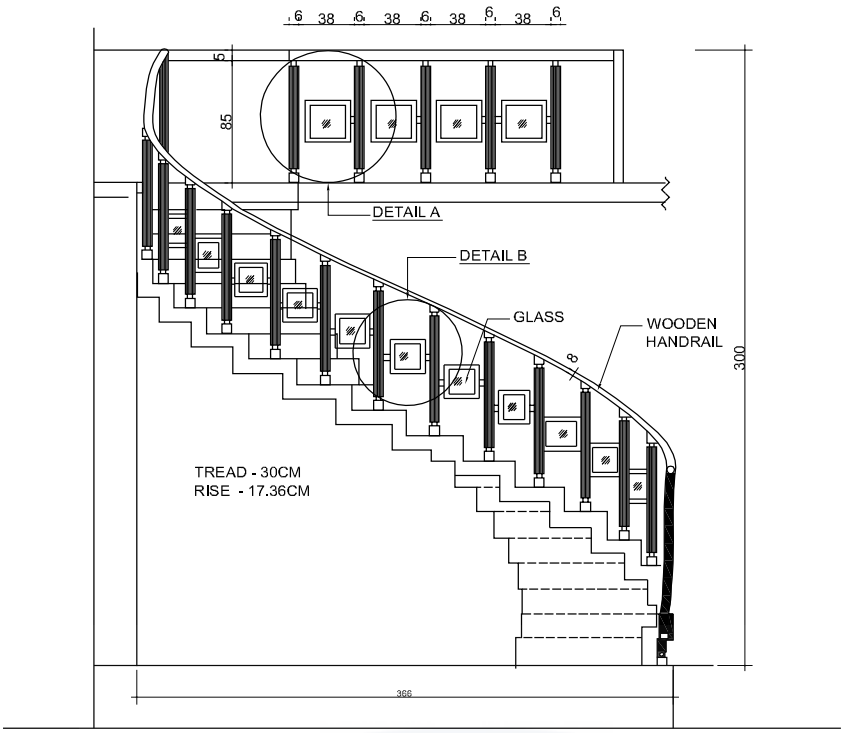
2 Draw the vertical lines to show the baluster.

3 Draw the handrail, and as per the details in drawing.

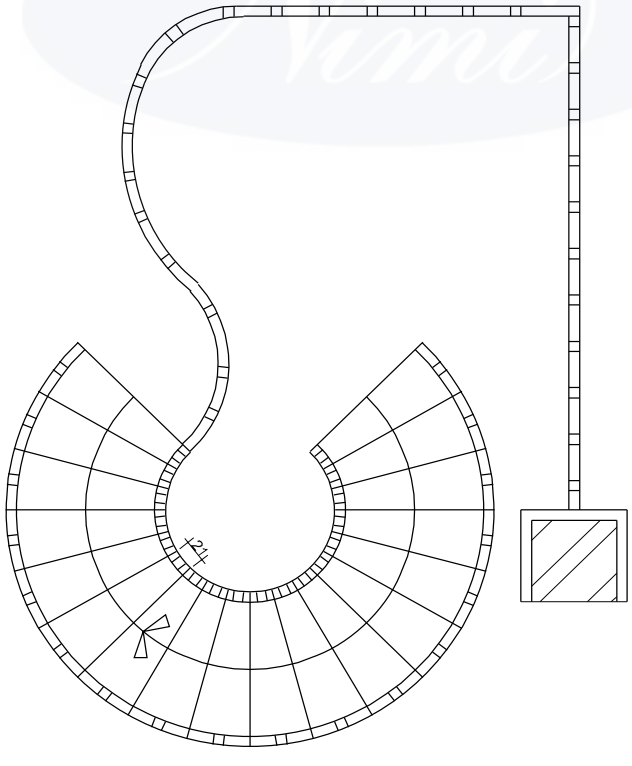
4 Draw the details of handrail and balusters in enlarged scale.

5 Dimension the drawing properly. (Fig 2)

Fig 1



ELEVATION

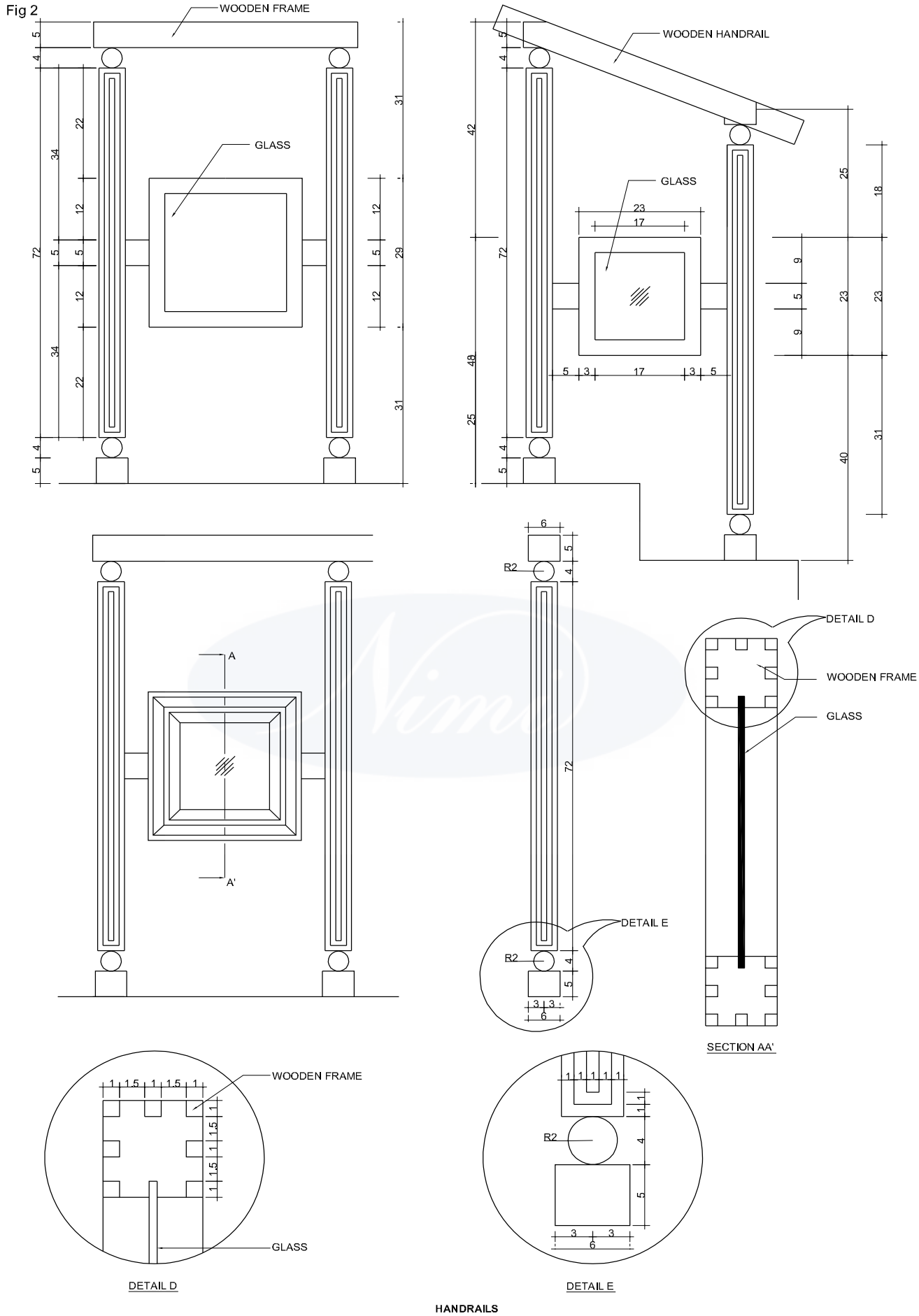


PLAN

CIRCULAR STAIR

AA2207E1

Fig 2


WOODEN FRAME
GLASS
WOODEN HANDRAIL
GLASS
A
A'
DETAIL D
WOODEN FRAME
GLASS
SECTION AA'
DETAIL E
WOODEN FRAME
GLASS
DETAIL D
DETAIL E
HANDRAILS
AA2207E2

Draw the construction details of RCC dog legged stairs

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of a RCC doglegged stair.**

PROCEDURE

TASK 1 : **Draw the plan of doglegged stair**

Data

Height between the floors : 3000

Rise and going 150 and 300 respectively

Total no of steps in the flight as per the rule

Width of the flight : 1000

Diameter of GI pipe handrail : 50

Height of the baluster : 900

Width of the handrail : 50

Size of the window : 2000 x 1200

Note: No of risers - 3 (min) & 12 maximum (all dimensions are in mm)

1 Draw the plan of the stair room with proper number of treads.

2 Draw the landing after twelve risers.

3 Draw the treads in the reverse direction after the landing.

4 Draw the window in plan.

5 Dimension the drawing properly. (Fig 1)

Draw the elevation of the doglegged stair

1 Draw the sectional elevation of the stair as per the drawing.

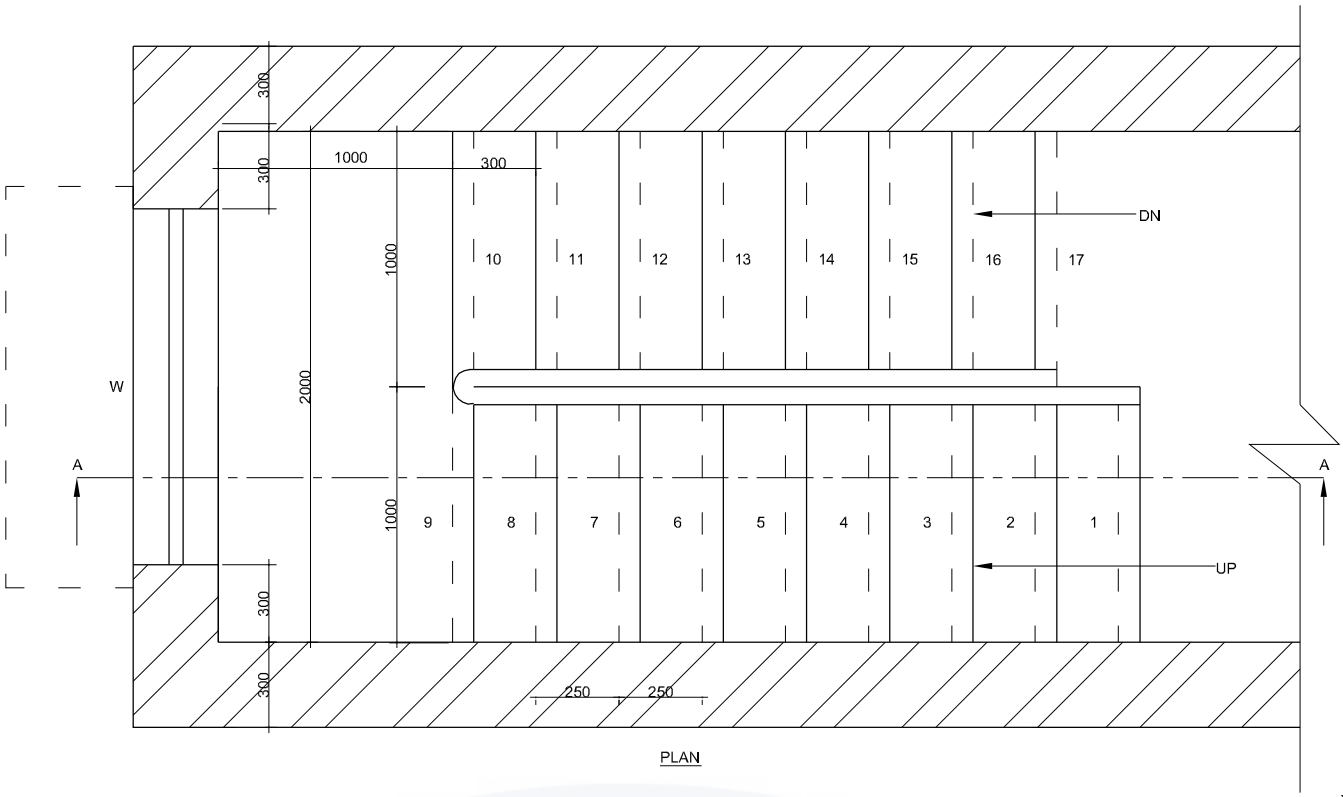
2 Draw the vertical lines showing the baluster.

3 Draw the handrail and sectional details as per the drawing.

4 Draw the elevation of the window.

5 Dimension the drawing properly. (Fig 2 & Fig 3)

Fig 1

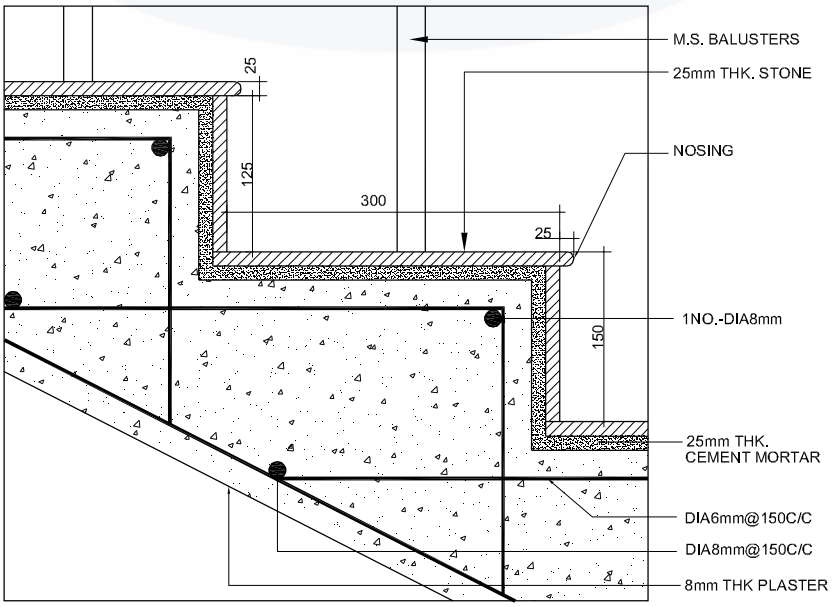


PLAN

DOG LEGGED STAIR

AA2208E1

Fig 3

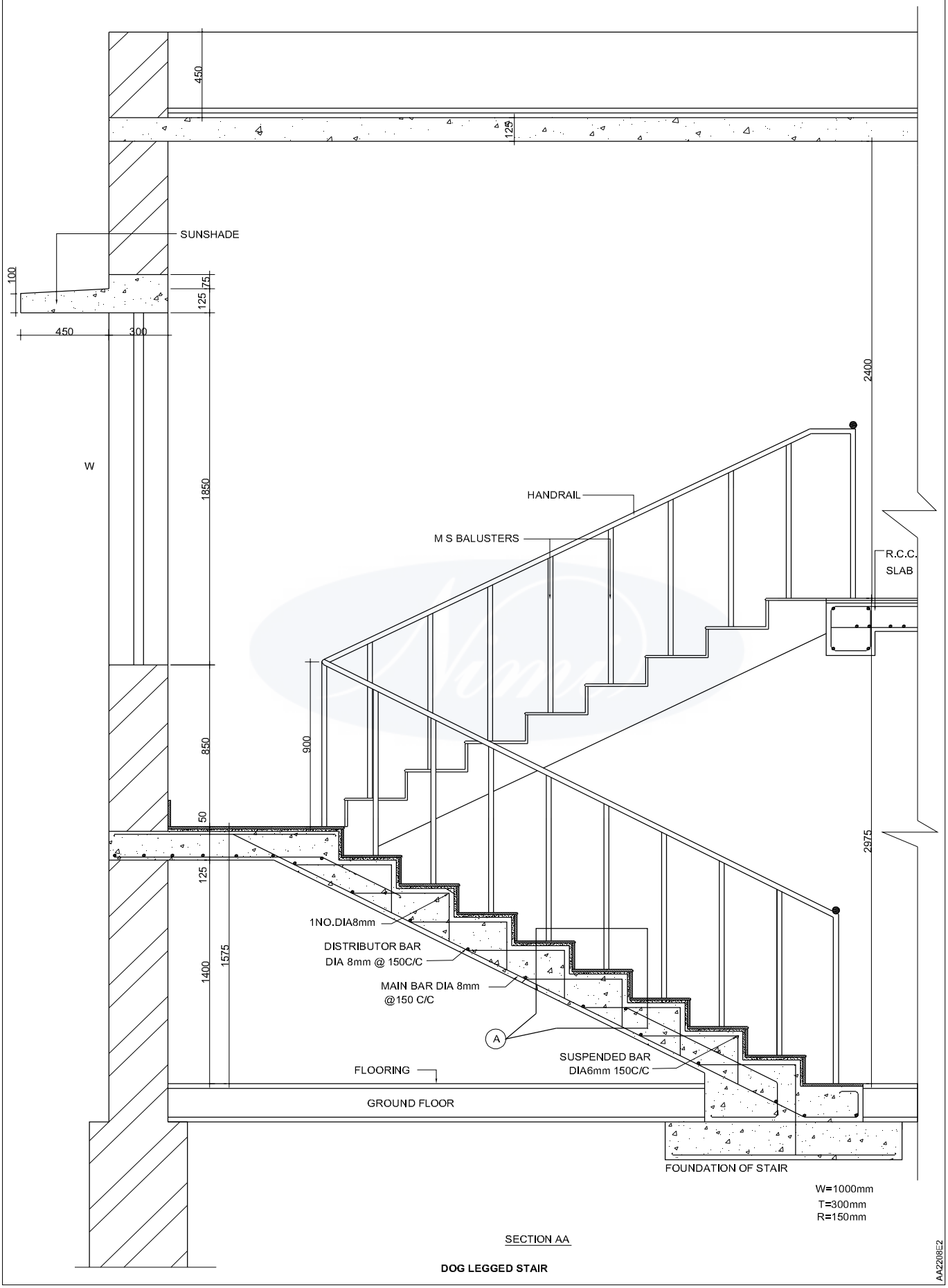


DETAIL A

DOG LEGGED STAIR

AA2208E3

Fig 2


450
125
SUNSHADE
100
75
125
450
300
2400
W
1850
HANDRAIL
M S BALUSTERS
R.C.C.
SLAB
900
850
50
125
2975
1400
1575
1NO.DIA8mm
DISTRIBUTOR BAR
DIA 8mm @ 150C/C
MAIN BAR DIA 8mm
@150 C/C
A
SUSPENDED BAR
DIA6mm 150C/C
FLOORING
GROUND FLOOR
FOUNDATION OF STAIR
W=1000mm
T=300mm
R=150mm
SECTION AA
DOG LEGGED STAIR
AA2208E2

Draw the construction details of MS spiral stairs

Objectives: At the end of this exercise you shall be able to
- **draw the plan and elevation of MS spiral stairs.**

PROCEDURE

TASK 1 : **Draw the plan of a MS spiral stair**

Data

Rise and going 150 and 250 respectively

Total no of steps in the flight 15

Diameter of the stair : 900

Diameter of the central post : 250

Height of the baluster : 900

Width of the handrail : 50

1 Draw the plan of the column of specified radius.

2 Draw the outer circle.

3 Draw the winders radiating from the centre.

4 Draw the handrail in plan.

5 Comlete the draing with necessary dimensioning.

Draw the elevation of the spiral stair

1 Draw the projected vertical lines to show the risers around the central column.

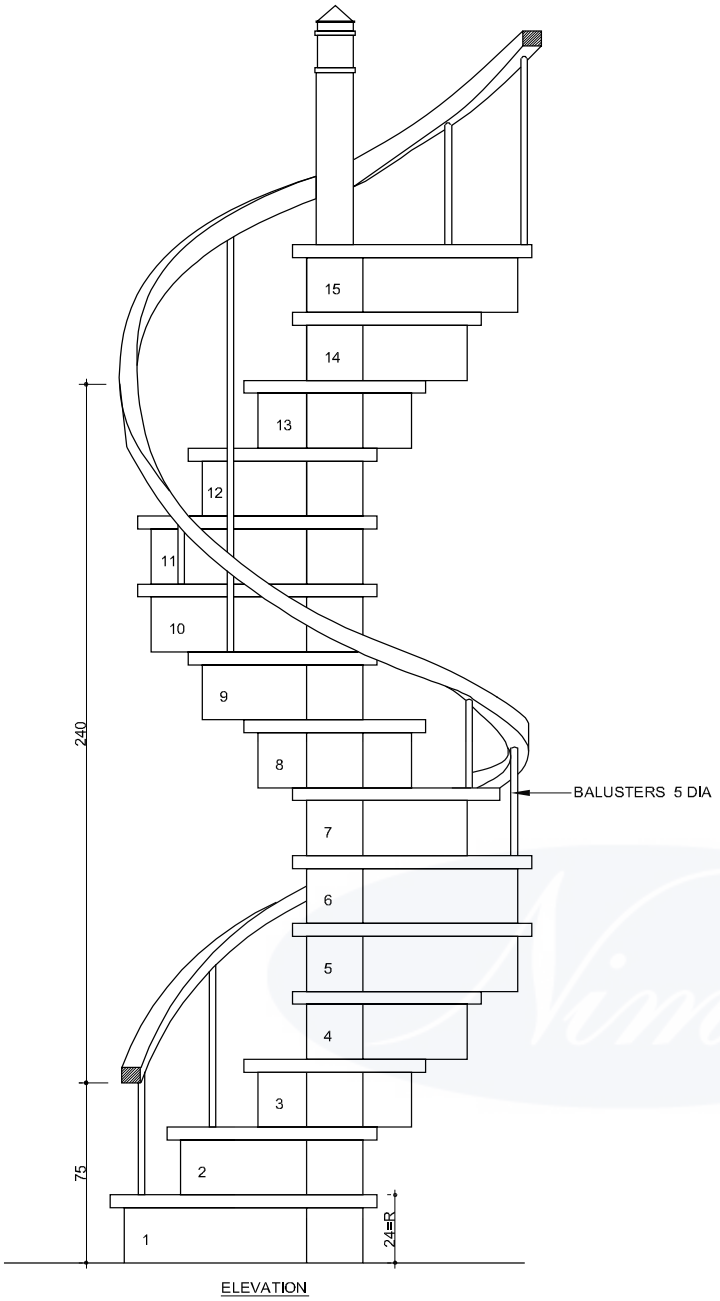
2 Draw the vertical lines showing the baluster.

3 Draw the vertical lines to show the baluster, handrail as per the drawing.

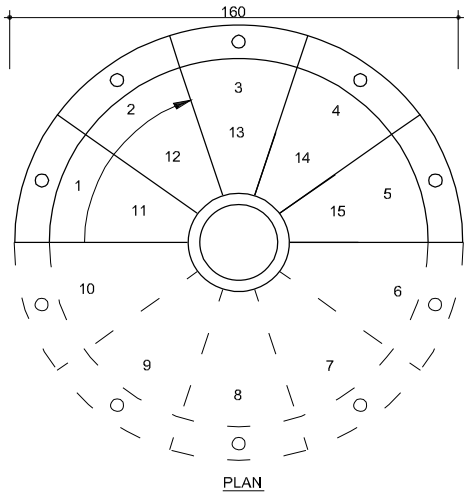
4 Complete the elevation of the spiral stair.

5 Complete the drawing with necessary dimensioning.

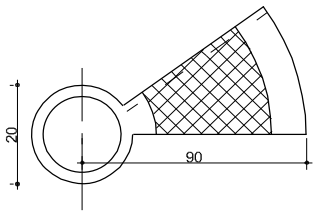
Fig 1



ELEVATION



PLAN



STEP DETAILS

STEEL SPIRAL STAIR

AA2209E1

Draw details of wooden stairs

Objectives: At the end of this exercise you shall be able to
- **draw plan and sectional elevation of wooden straight flight stair**
- **draw the details of steps, landing and hand rails**
- **draw plan and sectional elevation of wooden dog-legged stair**
- **draw the details of landing and steps.**

PROCEDURE

TASK 1 : **Draw the plan and sectional elevation of wooden straight flight stair and also draw the details.**

- Draw the length of stair case 2640mm
- Draw the width of stair 900mm
- Draw length of going 220 mm (1 to 13 Nos) (Fig 1,2)
- Draw projection of Nosing 16mm
- Draw one brick thick wall on both sides.
- Complete the drawing of plan (300 x 32mm)
- Draw strings on both sides of wall
- Draw section Elevation at KK (Fig 1)
- Draw tread and riser (220 to 200mm)
- Draw 1 to 13 steps (upto landing)
- Draw the ceiling height 2600mm
- Draw the hand rails parallel to strings (height 840mm)
- Draw stud partition under the 5th step
- Draw R.C.C lintel 15cm thick below 13th step
- Draw a wooden joist (225 x 75mm) above R.C.C lintel.
- Complete the drawing of straight flight wooden stair (Fig 1,2)
- Draw the details at 'M' (Fig 3)
- Draw the details at 'L -L' (Fig 4)

Fig 2

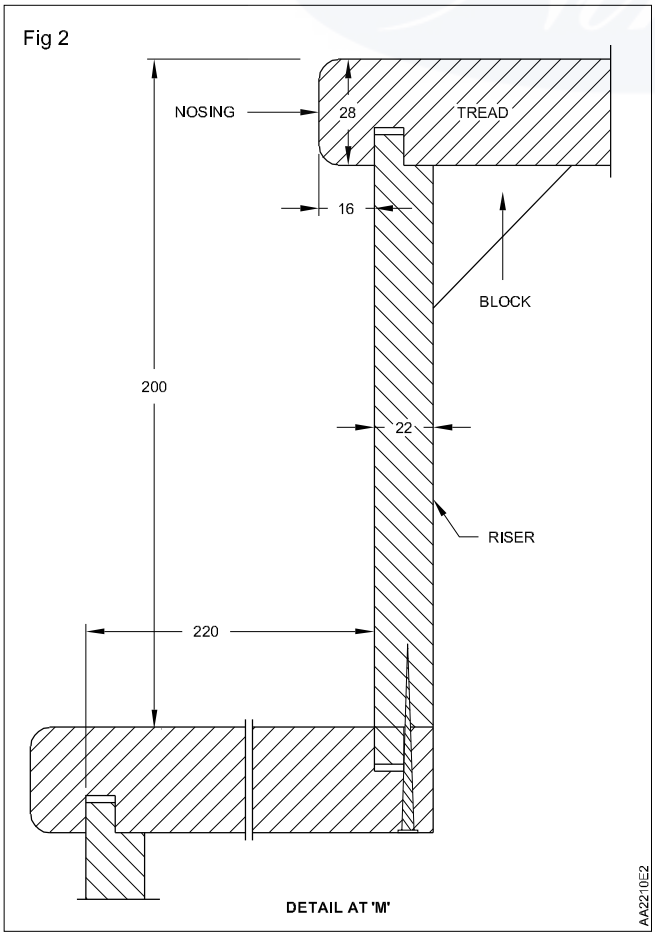
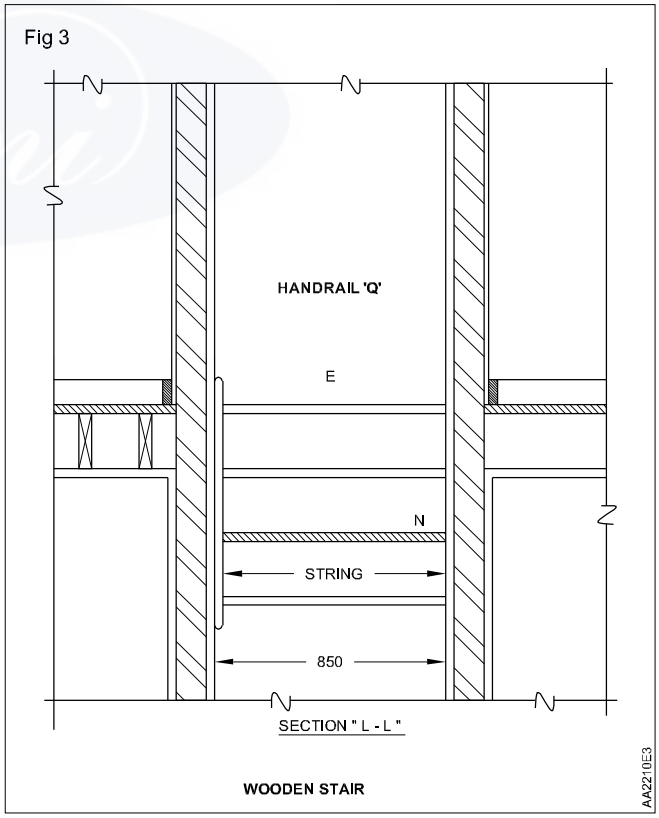
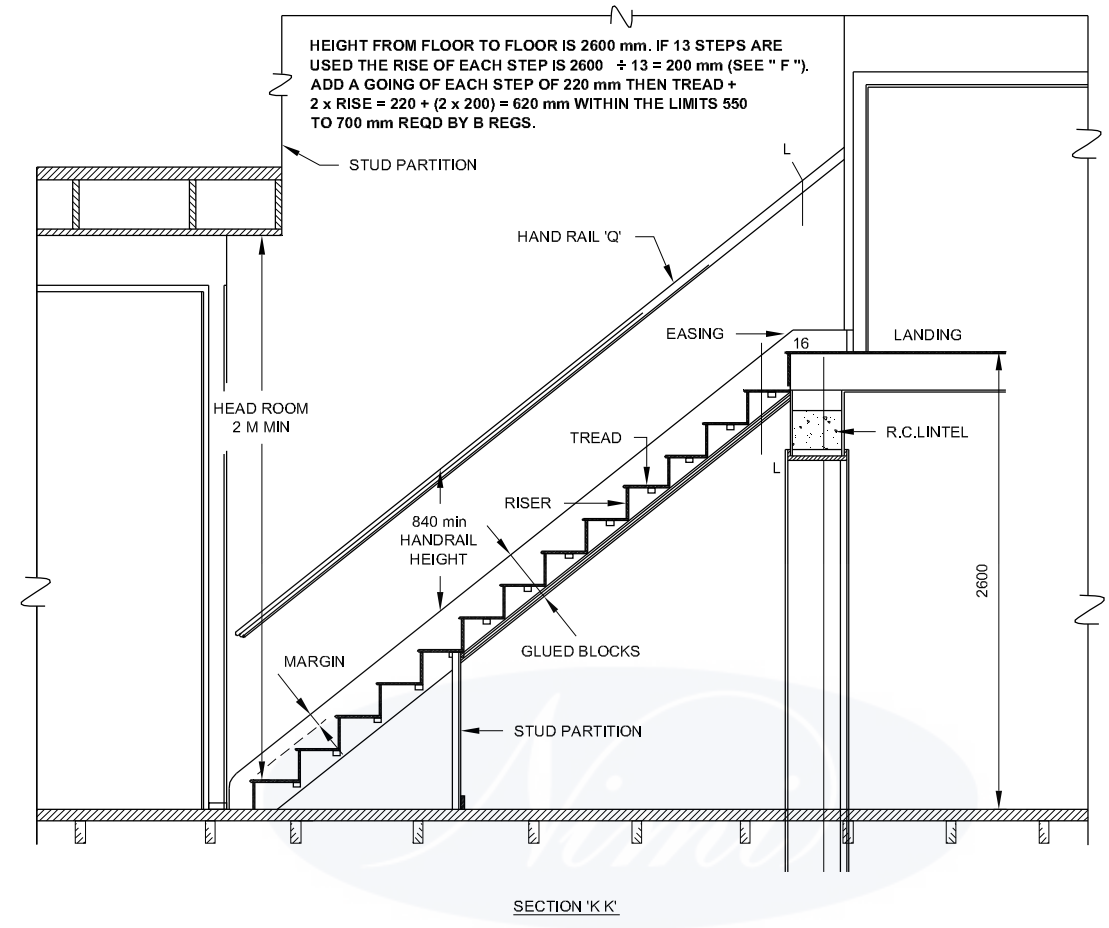


Fig 3



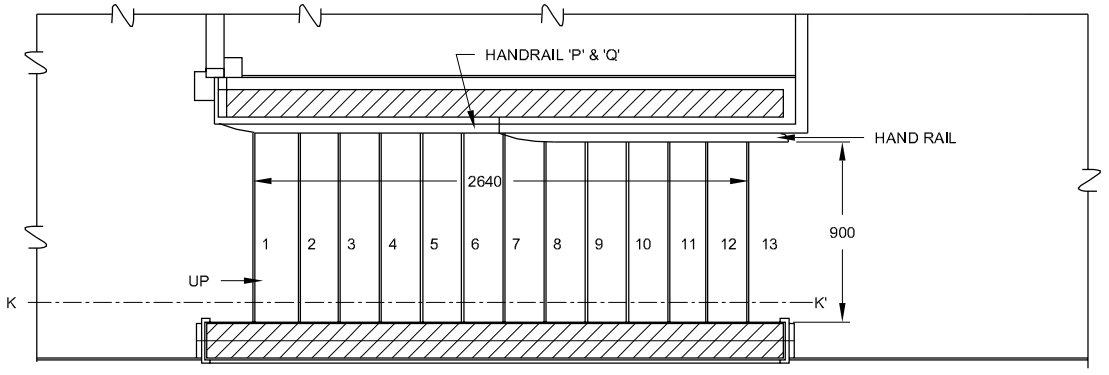
- Draw the details of Handrail (Fig 5,6)

Fig 1



SECTION 'K K'

Fig 1a



PLAN

WOODEN STAIR

AA2210E1

Fig 4

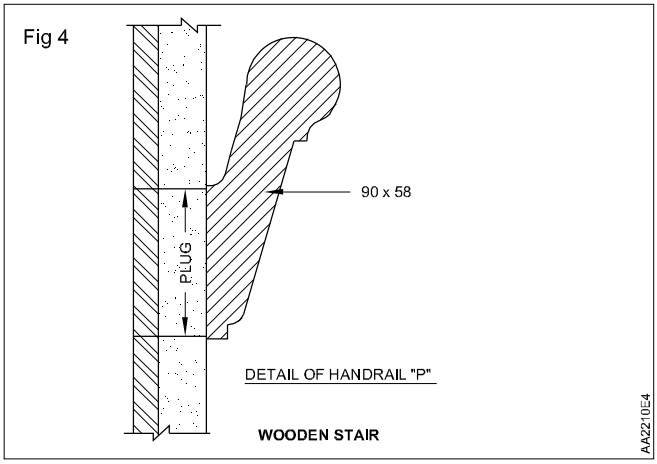
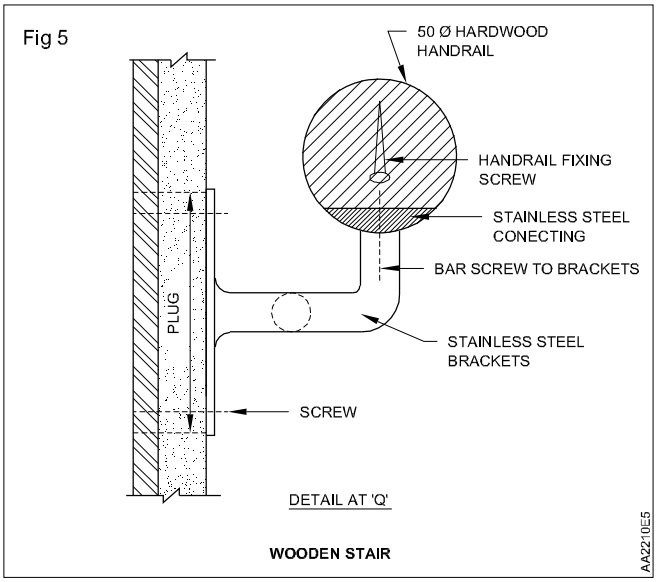


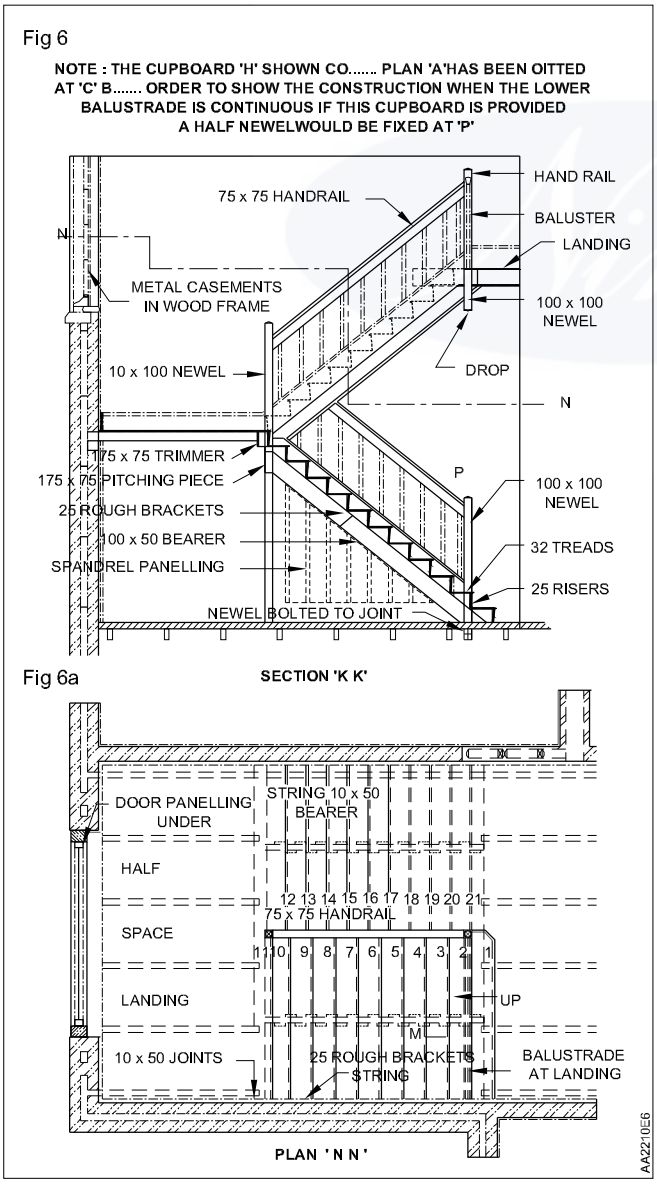
Fig 5



TASK 2 : **Draw the plan and sectional elevation of Dog-logged wooden stair case**

- Plan of Dog-logged stair case (Fig 7)

Fig 6



- Draw the width of stair 1830 mm
- Draw the length of first flight 1575mm
- Draw 8 Nos going to size of 225mm
- Draw Two Newel post 100 x 100 mm
- one at starting place another at landing
- Draw the width of Half space landing of size 1140mm
- Draw the another flight in opposite direction.
- Draw Half space landing 1090 mm
- Draw 7 steps in second flight of length 1350mm
- Draw 6 Nos Joist in half space landing (100 x 50mm)
- Draw 6 Nos Floor joist (225 x 50)
- Draw a trimmer joist below the floor joist (225 x 75)
- Draw wall thickness 300mm thick. (Fig 8)
- Draw and develop sectional elevation at KK (Fig 7)
- Draw and develop section at 'LL' (Fig 4,9)
- Draw and develop section Tread and Rise. (Fig 3,8)

Fig 7

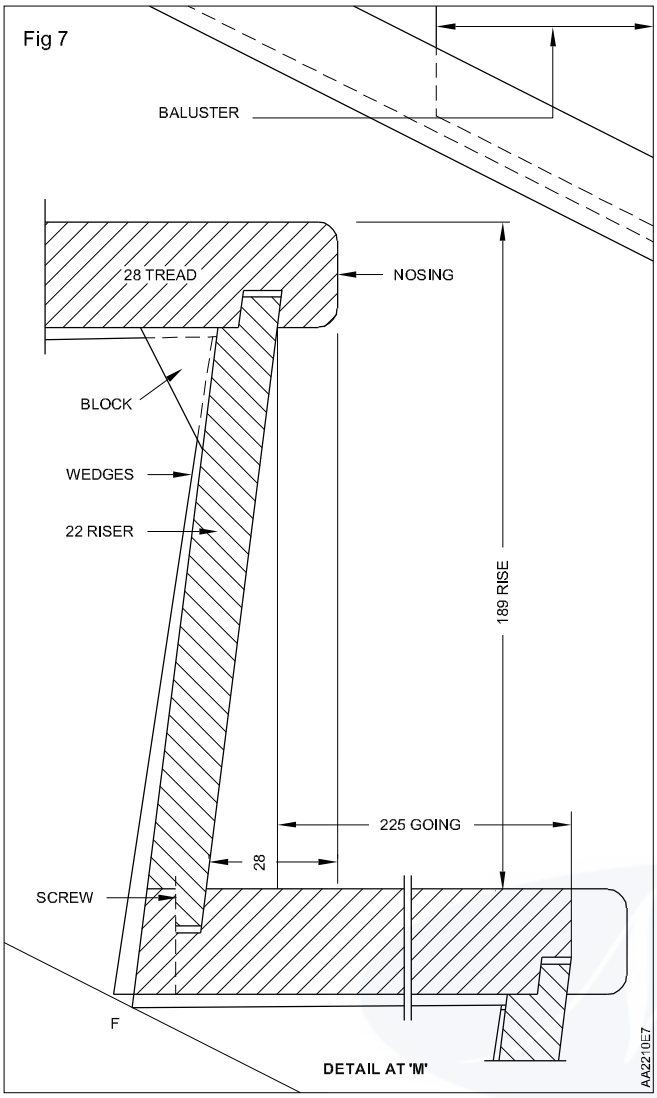
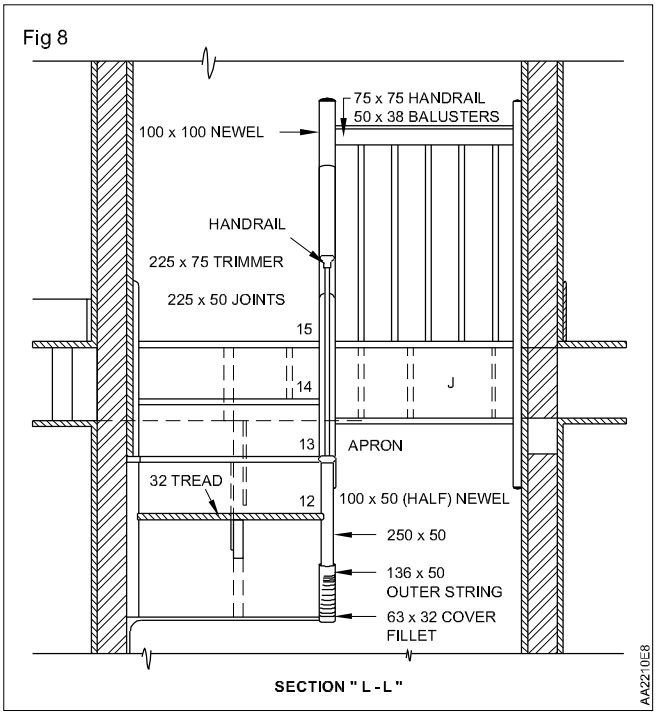

BALUSTER
28 TREAD
NOSING
BLOCK
WEDGES
22 RISER
189 RISE
225 GOING
28
SCREW
F
DETAIL AT 'M'
AA2210E7

Fig 8


75 x 75 HANDRAIL
50 x 38 BALUSTERS
100 x 100 NEWEL
HANDRAIL
225 x 75 TRIMMER
225 x 50 JOINTS
15
14
J
13
APRON
32 TREAD
12
100 x 50 (HALF) NEWEL
250 x 50
136 x 50
OUTER STRING
63 x 32 COVER
FILLET
SECTION " L - L "
AA2210E8

Draw the different components of floor

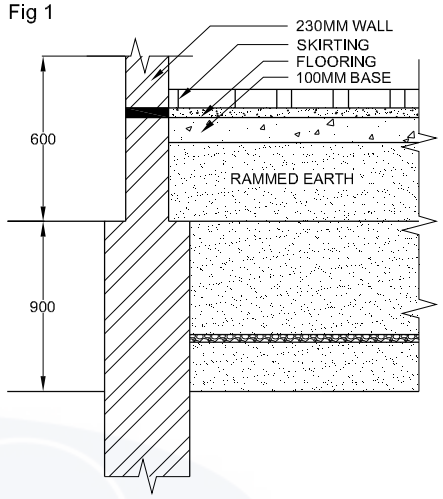
Objectives: At the end of this exercise you shall be able to
- **draw the detail of floor, components.**

PROCEDURE

TASK 1 : **Components of ground floor**

- Draw 230mm wall with 2.5cm DPC at plinth level along with stone foundation 450mm thick below.
- Draw sub floor total depth of 925 mm.
- Bottom soling is 40mm aggregate/random rubble of 150mm.
- Rammed earth filling 600mm thick.
- Flat brick soling 75mm thick (when black cotton soil is present).
- Base of 100mm thick cement concrete in 1:4:8 and smeared with 1:2: 4 concrete.
- Draw floor covering or flooring of 20mm thick of clossed tile granite / marble.
- Draw skirting of same flooring. (Fig 1)

Fig 1



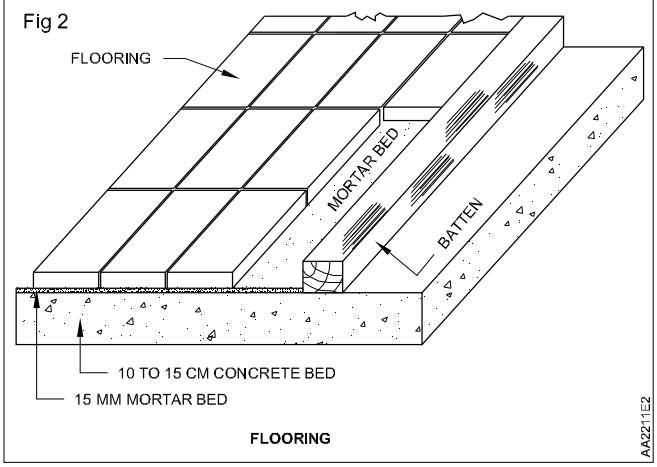
COMPONENTS OF GROUND FLOOR

AA2211E1

TASK 2 : **Draw the isometric view of flooring (Fig 2)**

- Data : Stone size - 300 x 50 x 20mm bedjoint - 15mm thick.
- Draw the isometric view of flooring.

Fig 2



FLOORING

AA2211E2

Draw the details of cement concrete floor

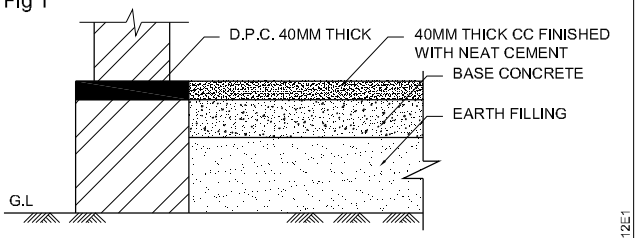
Objectives: At the end of this exercise you shall be able to
- **draw the detail of cement concrete floor**
- **draw section of supported type timber floor along with isometric view.**

PROCEDURE

TASK 1 : **Details of cement concrete floor**

- Draw the section of wall at plinth level
- Draw the base of flooring with
- Cement concrete bed of 1: 4: 8 100mm thick
- Draw 40mm thick finished floor covering of 1:2:4 cement concrete. (Fig 1)

Fig 1

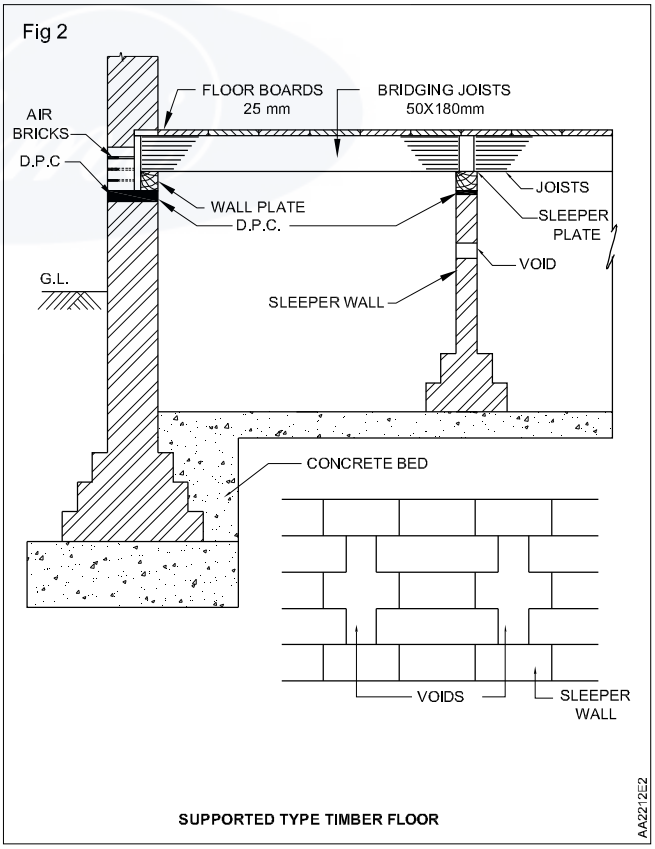


C.C. FLOORING

TASK 2 : **Detail of wooden suspended floor (ground floor)**

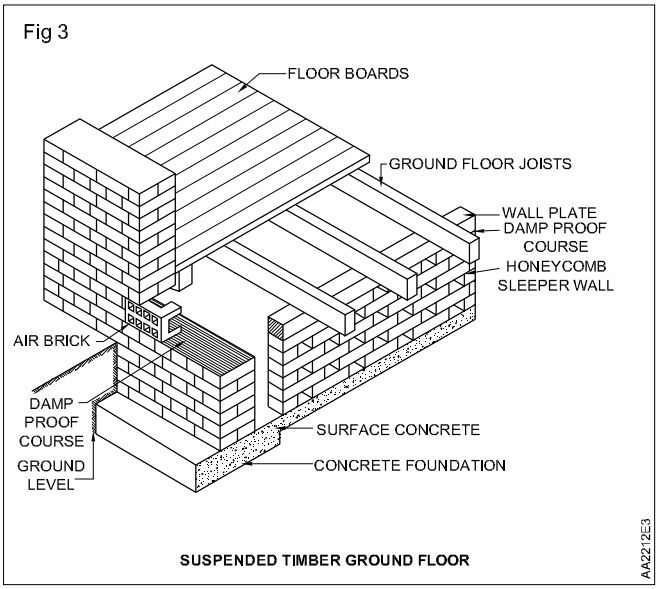
- Draw 230mm wall with foundation detail. (Fig 2 & Fig 3)
- Draw base concrete bed 1:2:4 15cm to 20cm thick which laid below 1m level from ground
- Draw a layer of mastic asphalt through out the width of wall immediately below wall plate and also on sleeper wall
- Draw sleeper walls of one brick thickness constructed at centre to centre distance of 1:2m to 1.5m.
- Draw the bridging joists should rest on wall plates 125 x 75mm.

Fig 2



SUPPORTED TYPE TIMBER FLOOR

Fig 3



SUSPENDED TIMBER GROUND FLOOR

- Carity space 25mm for expansion of material
- Draw floor boarding 18mm thick as covering with widening joints
- Air bricks and voids in sleeper walls provide ventilation.

Draw the details of wooden floor (Upper floor)

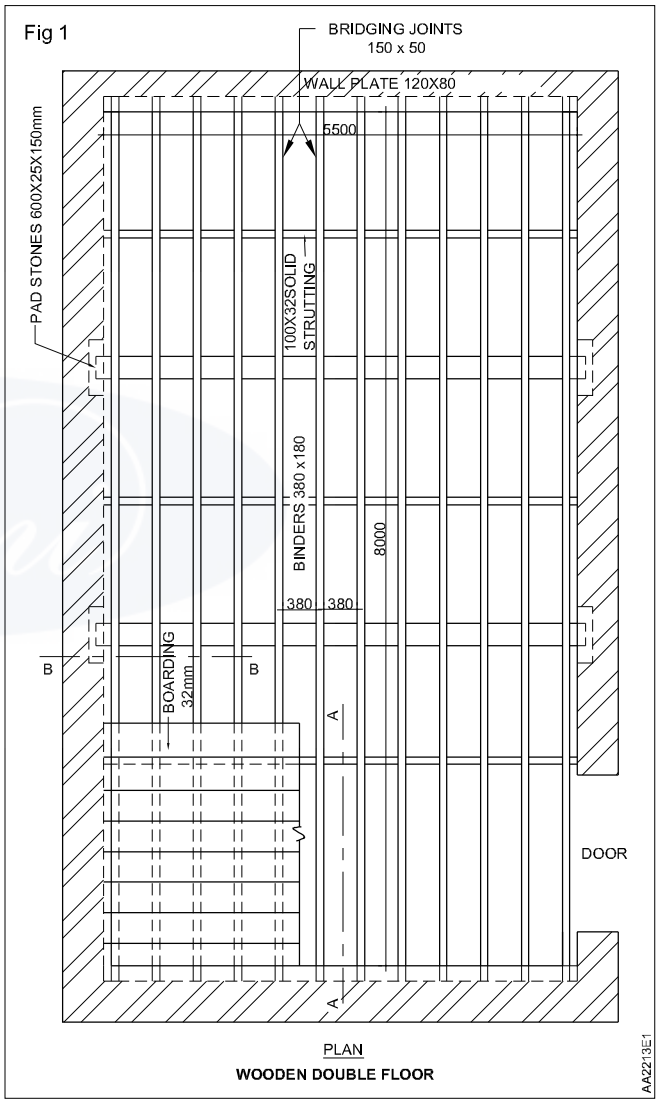
Objectives: At the end of this exercise you shall be able to
- **draw the plan showing arrangements of supporting boarding in a wooden double floor**
- **details showing different methods of supporting bridging joists**
- **section through a framed floor.**

PROCEDURE

TASK 1 : **Plan showing arrangements of supporting boarding in a wooden double floor**

- Draw the plan of 4.0 meter width room.
- Consider 230mm wall thickness and hatch the wall.
- Draw the binders of size 75 x 150mm with a gap of 900mm.
- Draw the bridging joist of 60 x 100mm at 300 mm c/c distance.
- Indicate the boarding as shown in Fig 1 and complete the drawing.

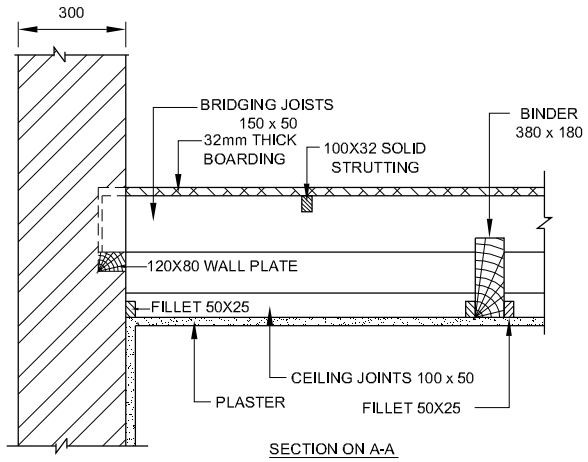
Fig 1



TASK 2 : **Sections at AA & BB**

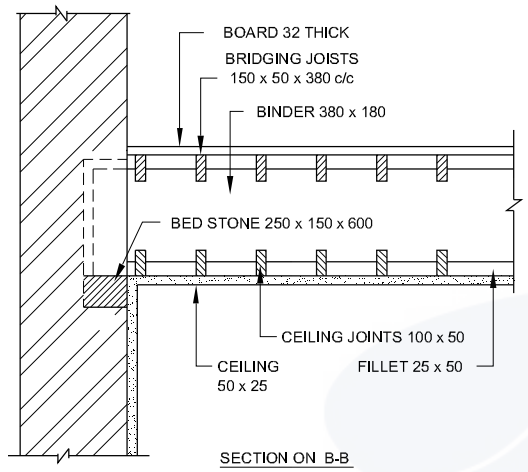
- Mark the section AA and BB in plan
- Draw the section at AA showing wall plate, binders, fillets next to binders etc
- Draw the ceiling joist and bridging joists
- Draw the boarding on top of the bridging joist as shown in Fig 2a and complete the drawing
- Draw the section at BB showing the bridging joist in section
- Draw the stone bed plate fixed in the wall for supporting the binders
- Draw the ceiling joist, fillets, boardings etc as shown in Fig 2b and complete the drawing.

Fig 2



SECTION ON A-A

Fig 3



SECTION ON B-B

WOODEN DOUBLE FLOOR

AA2213E2

Draw lean to roof details

Objectives: At the end of this exercise you shall be able to
- **draw the elevations of lean to roof.**

PROCEDURE

TASK 1 : **Draw the section of lean to roof**

Draw the section of lean- to roof to a scale of 20

Data required

Span of the roof : 2400 mm

Thickness of the main wall : 230mm

Cross section size of wall plate : 150 x 100 mm (verandah wall)

Cross section size of rafter : 125 x 75 mm

Cross section size of battens : 50 x 30 mm at 350 mm C/ C

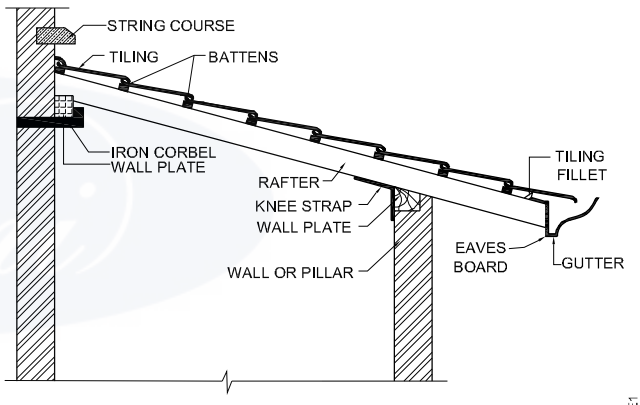
Cross section size of eave boards : 25 x 125mm

Eave projection : 300 mm

Pitch of the roof : 30° or 1/3rd of the span

- Draw the main wall and verandah wall with the given span.
- Draw the wall plate on the top of the verandah wall.
- Draw the rafter at angle 30° to the horizontal.
- Draw the corbel in themain wall at the meeting point of the rafter and the wall.
- Draw the battens above the rafter at angle 30° to the horizontal.
- Draw the corbel in the main wall at the meeting point of the rafter and the wall.
- Draw the battens above the rafter.
- Draw the roof tiles above the battens.
- Draw the eave board and gutter at the end of the rafter.
- Complete the drawing with proper dimensioning. (Fig 1)

Fig 1



LEAN TO ROOF

Draw flat roof details

Objectives: At the end of this exercise you shall be able to
- **draw the section of flat roof with brick tile flooring**
- **draw the section of flat roof with file flooring & mud phuska.**

PROCEDURE

TASK 1 : **Draw the section of flat roof with brick tile flooring**

1 Painting the roof slab uniformly with a layer of hot bitumen spread at the rate of 1.70 kg of bitumen per sq m of roof surface.

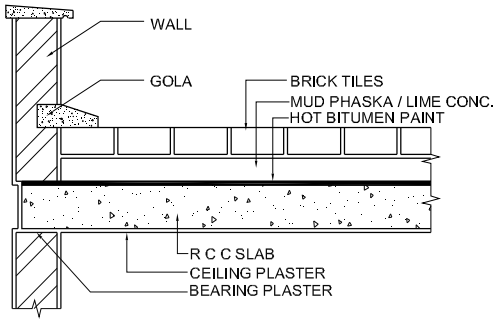
2 Spreading immediately coarse sand at the rate of 0.6 cu.m. of sand per 100 sq m of roof surrface, when the bitumen is still hot.

3 Laying 10cm thick (average) mud phuska the slope for the proper drainage of roof being given to this layer. The slope should not be less than I in 40.

4 Plastering the consolidated layer of mud phuska with a 13 mm thick coat of mug gobri mortar 3:1 (3 mud : 1 cowdung). (Fig 1)

5 Laying tile bricks flat on the plastered surface and grouting the joints with cement mortar 1:3.

Fig 1



BRICK TILE FLOORING

— — — — — — — —

TASK 2 : **Draw the section of flat roof with brick tile flooring and mud phuska**

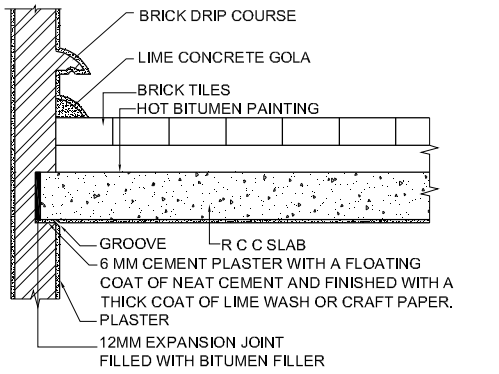
1 Painting the top of roof uniformly with layer of hot bitumark

2 Spreading immediately, coarse sand at the rate of 0.6 cu.m of sand per 100 sq m of roof surface when the bitumen is still hot.

3 Laying lime concrete in an average thickness of 10cm. The slope for proper drainage of roof being given to the lime concrete layer

4 Laying two courses of tiles (with breaking joints) over the compacted later of lime concrete in cement mortar 1:3 Thickness of each course of tile varies from 13 to 20mm. Instead of the flat tiles, precast concrete tiles or 25mm thick Shahabad stone can also be used depending upon availability and cost.

Fig 2



WATER PROOFING TREATMENT

— — — — — — — —

Draw the fixing details of AC sheets and corrugated sheets

Objectives: At the end of this exercise you shall be able to
- **draw the planning fixing details of asbetos sheet and corrugated sheets**
- **draw the section of fixing of asbestos sheet**
- **draw the fixing of screws and bolts.**

PROCEDURE

TASK 1 :

Data

Spacing between the purlins 1:65m maximum

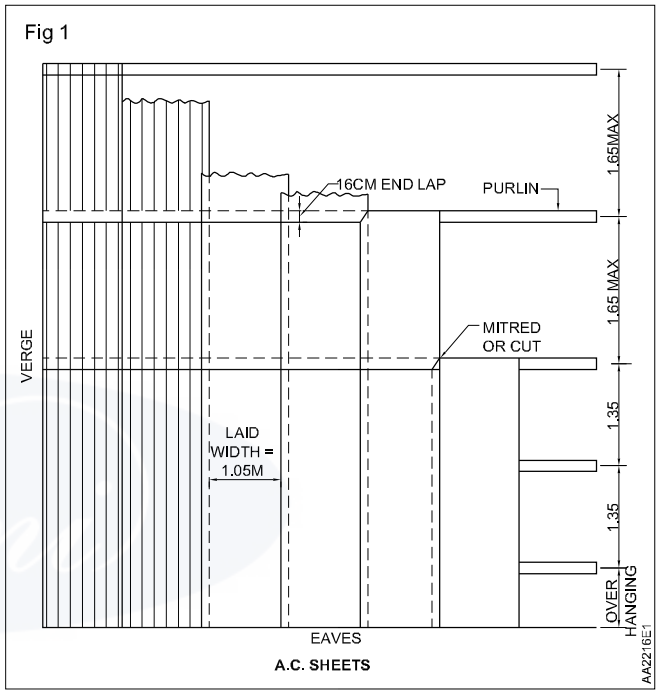
Minimum endlap : 16 cm

Overhang at the ridges : 75

Side lap : 1/2 corrugation minimum (All dimensions in mm)

- Draw the purlins at the given spacing.
- Draw the corrugated sheets as per the drawing.
- Complete the drawing with proper dimensioning as shown in Fig 1.

Fig 1



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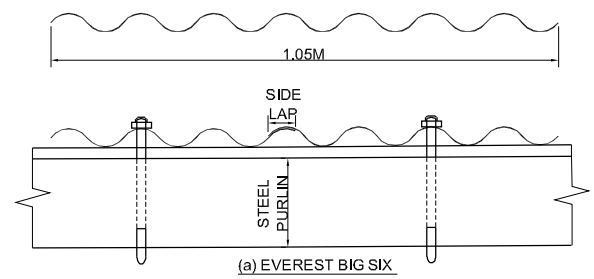
TASK 2 : **Draw the section of fixing of Asbestos sheet**

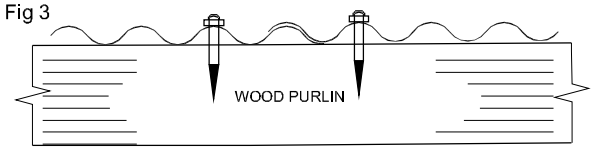
- Draw the section of purlins.
- Draw the asbestos and corrugated sheets as shown in Fig 2.
- Complete the drawing with proper dimensioning.

Fig 2



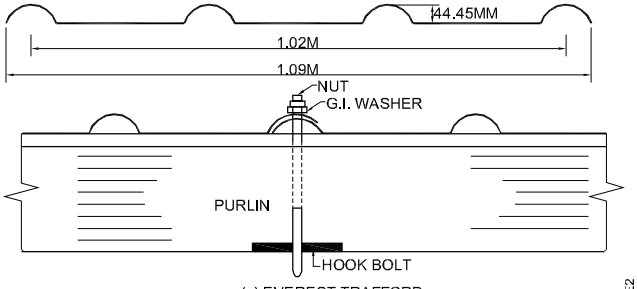
(a) EVEREST BIG SIX

Fig 3



(b) EVEREST STANDARD

Fig 4



(c) EVEREST TRAFFORD

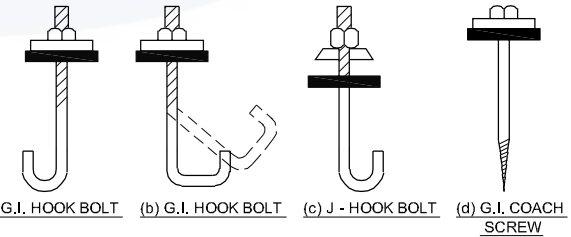
A.C. SHEETS SECTION

AA2216E2

TASK 3 : **Draw the section of fixing of Asbestos sheet**

- Draw the 6.1 hook bolt with bitumen washers
- Draw the J-hook bolt with Asbestos washers and lead washers
- Draw the 6.1 croach screw
- Complete the drawing as shown in Fig 3.

Fig 5



(a) G.I. HOOK BOLT (b) G.I. HOOK BOLT (c) J - HOOK BOLT (d) G.I. COACH SCREW

FIXING BOLTS AND SCREWS

AA2216E5

Draw details of king post truss

Objectives: At the end of this exercise you shall be able to
- **draw line diagram of king post truss**
- **draw king post truss 8m span length**
- **draw detail of each joint in king post truss.**

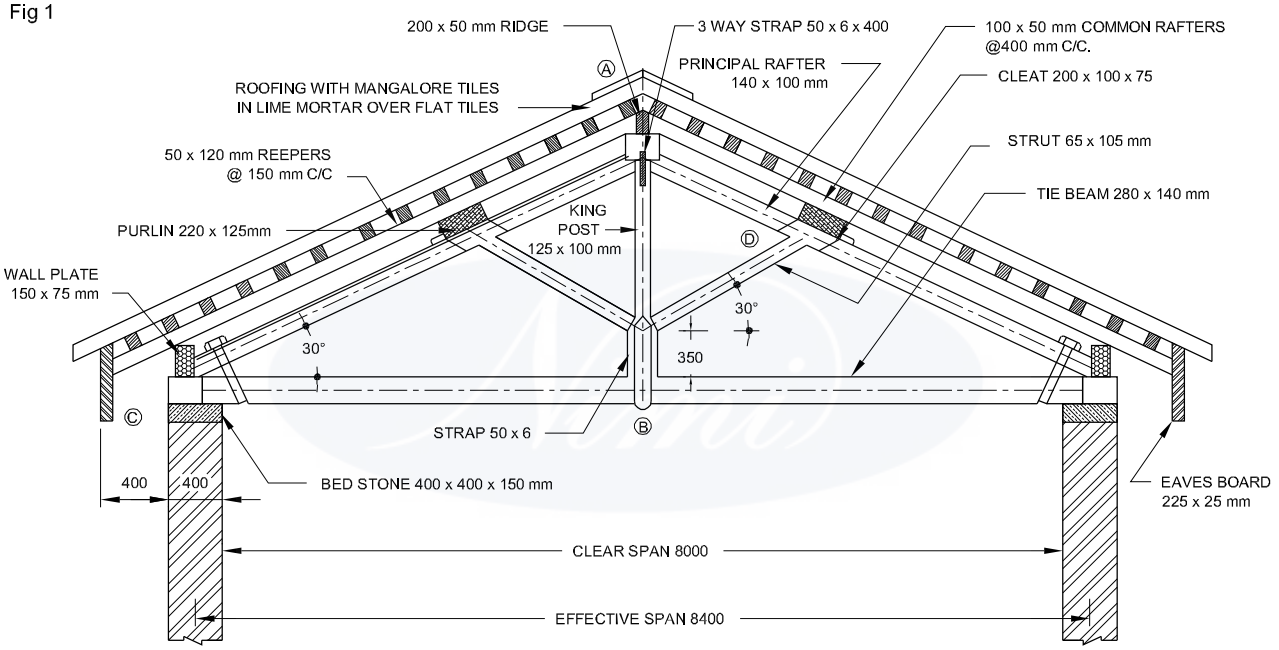
PROCEDURE

- Draw the view of king post Truss Fig 1
- Draw given span length Details of king post truss
- Draw Details of Joint in King post Fig 1
- Draw sectional view of king post truss Fig 1.

Draw the view of King post truss (Fig 1)

- Draw a horizontal line take some else and its ends of support a draw vertical line to below the horizontal line.

Fig 1

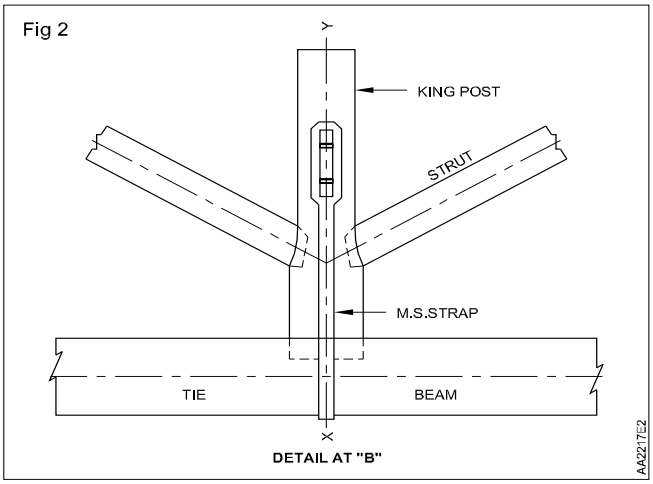


- Draw a pitch of 30’ inclined at both ends horizontal line.
- Draw the inclined line to parallel certain interval.
- Draw a vertical line to middle of horizontal line.
- Draw a inclined line to vertical line pitch of 30’ at both side.
- Draw a ridge pices at end of top (or) crown To show Roof covering, cleat, sectional view of purlin. Principal rafter and common rafter as an shown in Fig.

Draw details at ‘B’ of king post

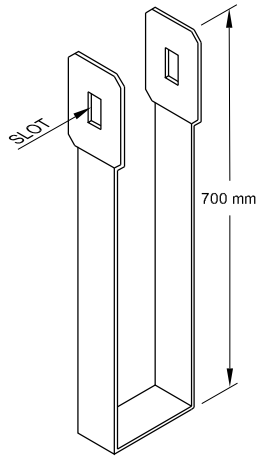
- Draw the exercise as same.
- Draw Dimentional sketch of each part of king post truss strut and tie beam (Fig 2)

Fig 2



- Draw the detail of stirrup strap (Fig 3)

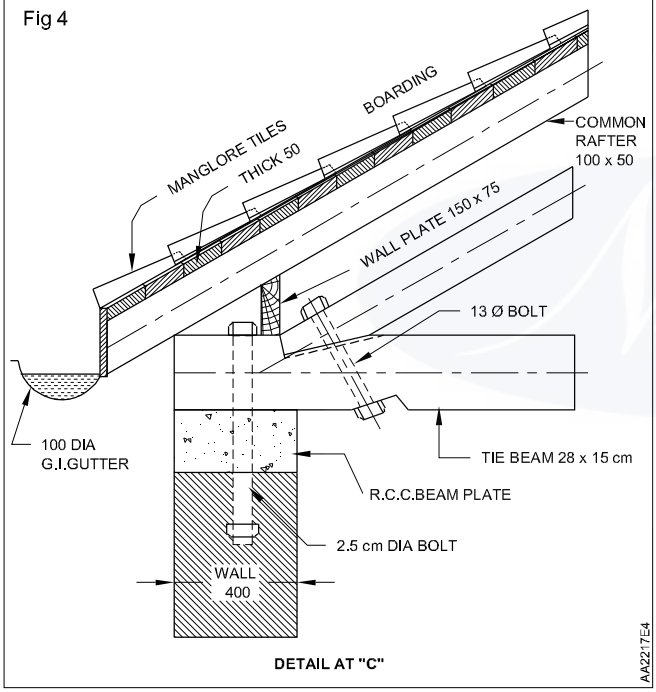
Fig 3



STIRRUP STRAP

- Complete the drawing.
- Draw the detail at ‘C’ of King post Truss (Fig 4)

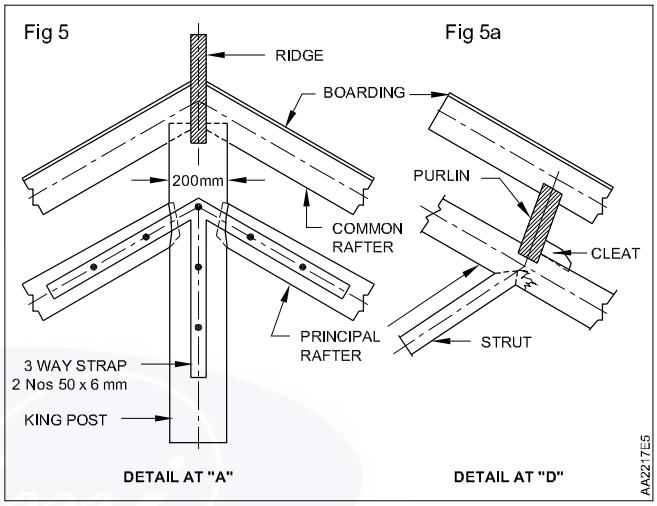
Fig 4



DETAIL AT "C"

- Draw wall thickness
- Draw Tie beam and principle rafter
- Draw common rafter and setaing of mangalore tiles
- Draw connection of Bolt
- Complete the drawing as shown in Fig 3.
- Draw the details at ‘A’
- Draw King post and principal rafter
- Draw common rafter and ridges
- Complete the drawing as shown in Fig 5.
- Draw the details at ‘D’
- Draw the drawing as shown in Fig (5a)

Fig 5 Fig 5a



DETAIL AT "A" DETAIL AT "D"

Draw details of queen post truss

Objectives: At the end of this exercise you shall be able to
- **draw queen post truss elevation**
- **draw queen post truss detail.**

PROCEDURE

Diagram of span length 8000 to 12000 mm

- Draw the elevation of Queen post truss of span length 8000 to 12000 mm
- Draw the details drawing of Queen post truss. Span up to length 11000 mm

Data:

Span 11,000 mm

Wall thickness 500 mm

Bed plate 400 x 300 x 150

Parapet 300 mm

Rain water pipe f 20 mm

Tie beam 150 x 250 mm

Queen Post 180 x 150 mm

Straining sill 50 x 150 mm

Straining beam 180 x 150 mm

Strut 100 x 150 mm

Cleat 180 x 80 mm

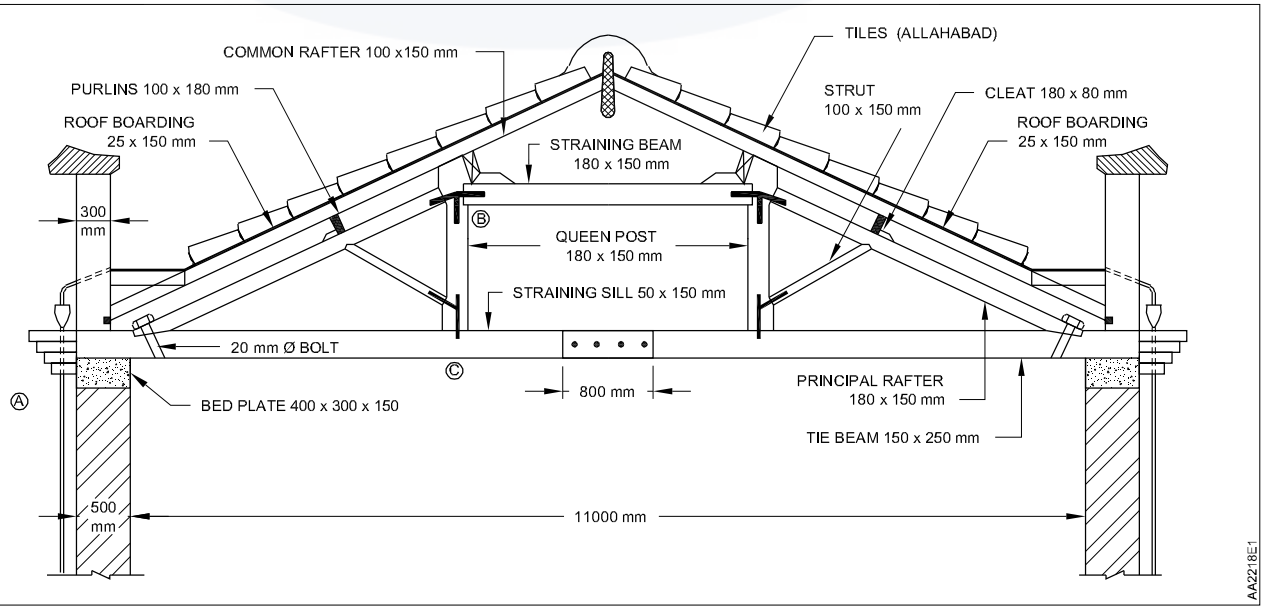
Roof Boarding 25 x 150 mm

Purlin 100 x 180 mm

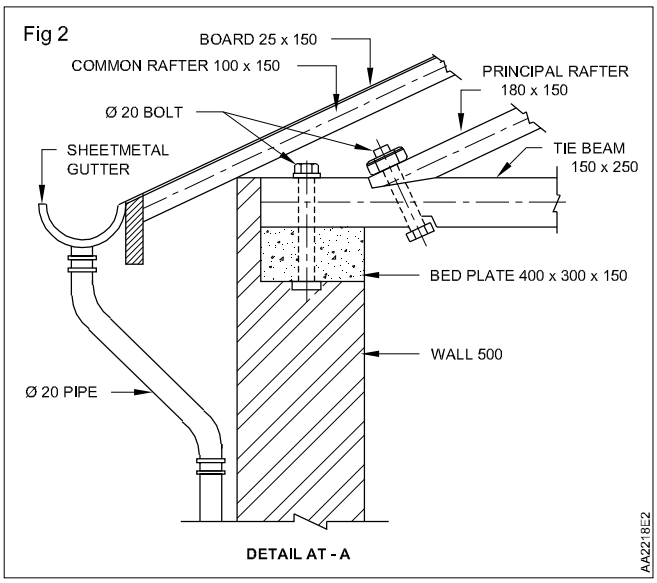
Dia of Bolt 20 mm

- Draw Horizontal line 11000 mm to draw two wall thickness.
- Draw 30^0 inclined to pitch of horizontal line to both ends. it will form a triangle shape
- Horizontal line to divide three equal points 1,2 put mark it. Draw vertical line above it. The vertical line cut in the inclined line in both side, That it cutting point connect a horizontal line. This line is called springer beam and below it called tie beam. It will shown in Fig 26.1
- Complete the drawing as shown in Fig 1.

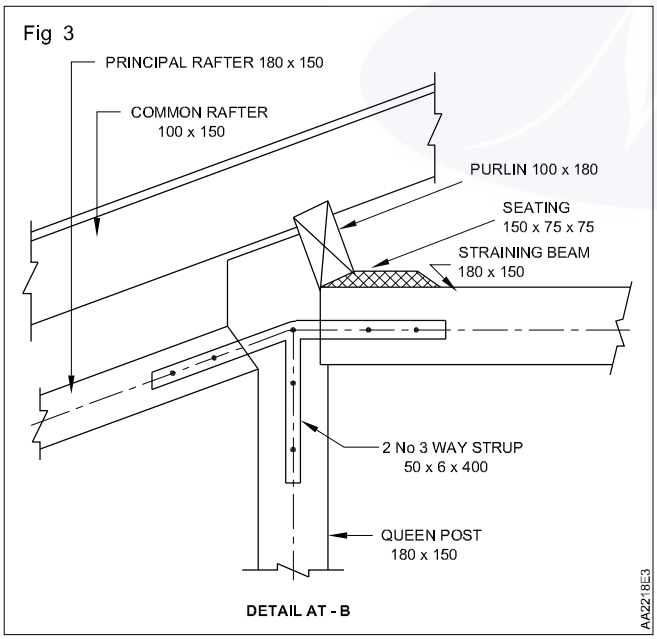
Fig 1



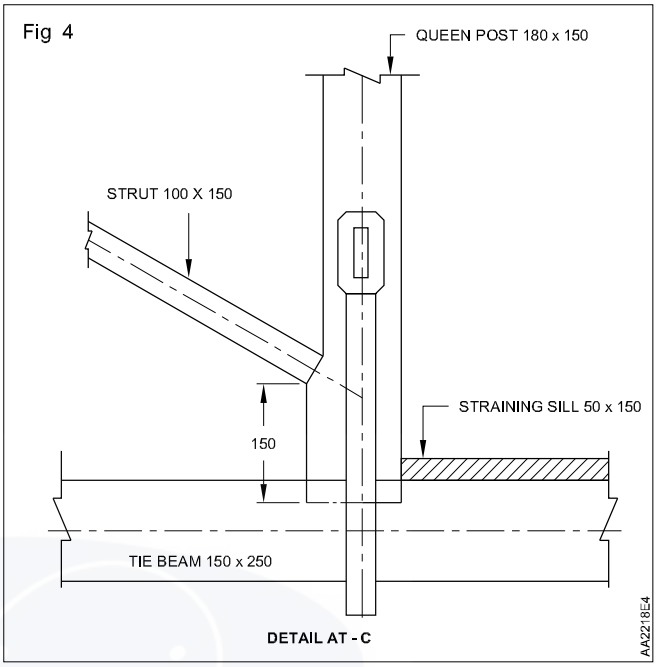
- Draw Gutter and rain water pipe as shown in Fig 2.



- Details of Queen post truss at ‘B’.
- Draw the length of straining beam
- Draw inclined member of common rafter and principal rafter
- Draw connection of stirrup and purlin
- Complete the drawing as shown in Fig 3.



- Draw the details at ‘C’ of Queen post truss
- Draw the length of tie beam
- Draw vertical Queen post
- Draw straining sill
- Draw incline member of strut
- Draw the connection of strap
- Complete the drawing as shown in Fig 4.



Draw details at ‘A’ of Queen post truss

- Draw a Horizontal member of tie beam and inclined member of principal rafter and common rafter.
- Draw wall thickness and bed plate.

Draw details of steel roof truss

Objectives: At the end of this exercise you shall be able to
- **draw the elevation of steel roof truss**
- **draw the details of steel roof truss**
- **draw the elevation of north light roof truss**
- **draw the elevation tabular steel roof truss**
- **draw the details of tubular steel roof truss.**

PROCEDURE

Draw Elevation of steel Roof Truss of span 6000 mm

Data

Tie beam 60 x 60 x 6mm

Principle Rafter 75 x 75 x 6mm

Purlin 100 x 75 x 10mm

Gusset plate 10mm thick

Bearing plate 20mm thick

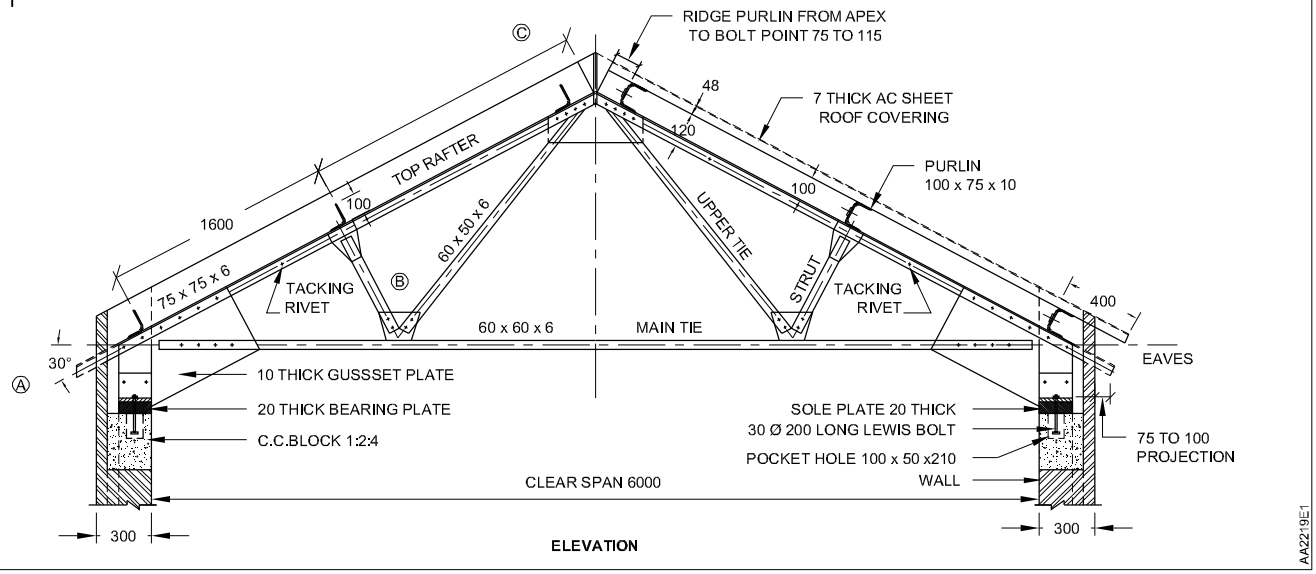
Lewis Bolt 30mm f, 200mm long

Pitch of Roof 30^0

Wall thickness 300mm

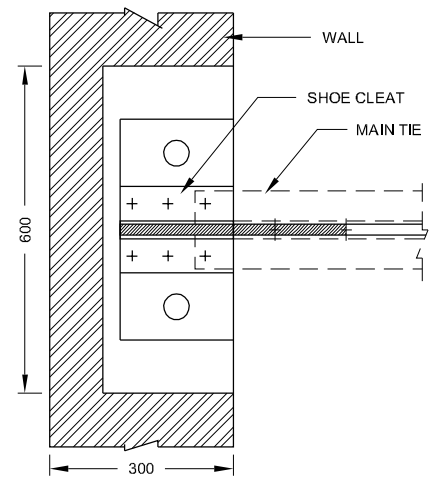
- Draw a horizontal centre line 6300 mm
- Draw on both side 30^0 pitch to form a traingle
- Draw a 60^0 line from ridge towards Tie beam
- Draw a 90^0 line from principle rafter on both sides
- Complete the shape of central line
- Draw the thickness of each member
- Draw the Riveted joints with gusset plate
- Draw purlin on each side at on the top of the principle rafter at ridge, Junction of strut and at bottom of principle rafter.
- Draw the support of wall thickness
- Draw c.c Block, Bearing plate, gusset plate
- Draw a line roof covering sheet on both side.
- Complete the Drawing of steel Roof Truss (Fig 1)

Fig 1



- Draw and develop the following Details of joints at fixed end (Fig 2, 3, 4)
- Draw the elevation of North Light Roof Truss (Fig 27.5)

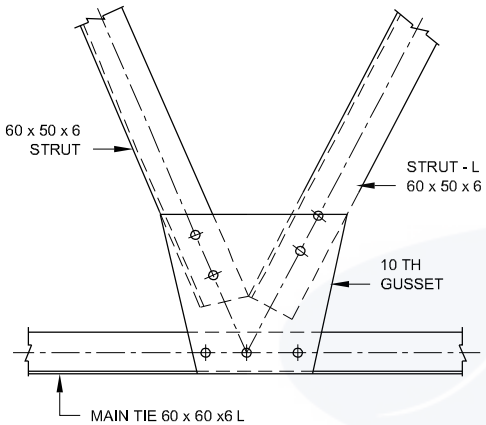
Fig 2



DETAIL AT " SUPPORT " Ⓐ
FIXED END (PLAN)

AA2219E2

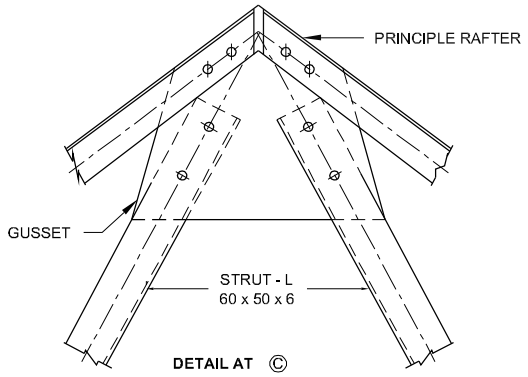
Fig 3



DETAIL AT Ⓑ
TIE BEAM

AA2219E3

Fig 4



DETAIL AT Ⓒ
RIDGE

AA2219E4

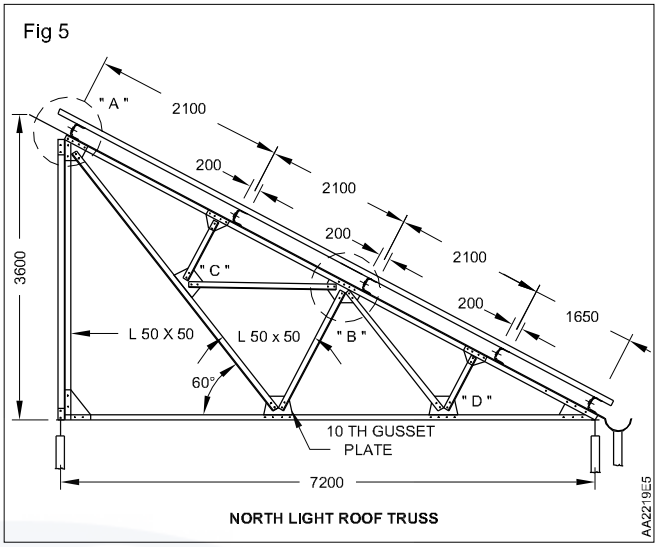
Data

All L angle 50 x 50 x 6mm

Gusset plates 10mm thick

- Draw a span of 7200 mm as Centre line
- Draw Rise of Roof 3600 mm as centre line
- Draw and complete the Triangle
- Draw inclined line from ridge to Tie beam at an angle of 60^0
- Draw inclined line from Tie beam to principle rafter at angle of 60^0
- Draw parallel line to Tie beam at point B
- Draw perpendicular line from principle raftes to the point C and D
- Complete the drawing of North light Roof truss (Fig 5)

Fig 5



NORTH LIGHT ROOF TRUSS

AA2219E5

- Draw enlarged Details at B,C and D
- Draw the elevation of Tubular Roof Truss

Data

Dia of principal rafter 150mm

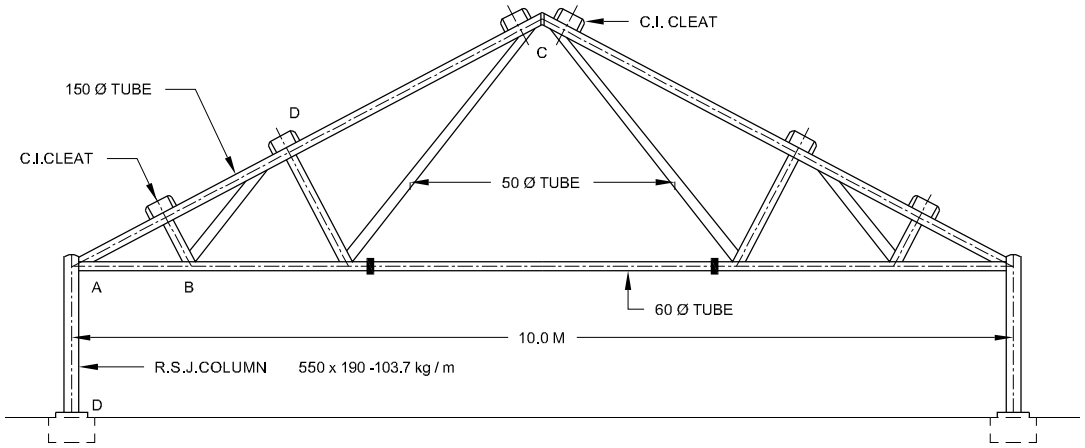
Dia of Tube 50mm (All strut)

Dia. of Tube 60mm (Tie beam) plate 6mm thick

Size of (R.S.J) ISMB 550 x 190 - 103.7Kg/m

- Draw a span of 10000mm length
- Rise of Roof 1/4 of span.
- Complete the Drawing of tubular steel Roof Truss
- Draw the symbol of welded Joints (Fig 6)
- Draw the enlarge details of at A,B,C and at E (Fig 7, 8, 9 and 10)

Fig 6

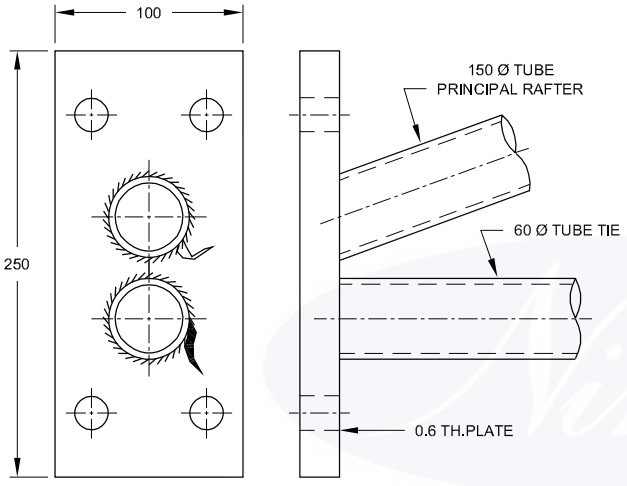


TUBULAR STEEL ROOF TRUSS

ELEVATION

AA2219E6

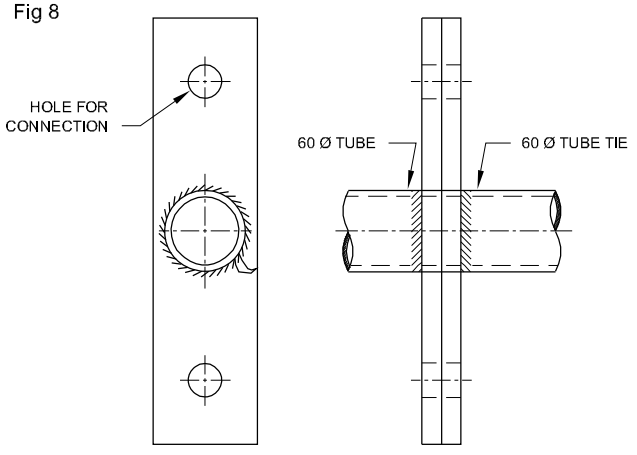
Fig 7



DETAIL AT 'A' (SUPPORT)

AA2219E7

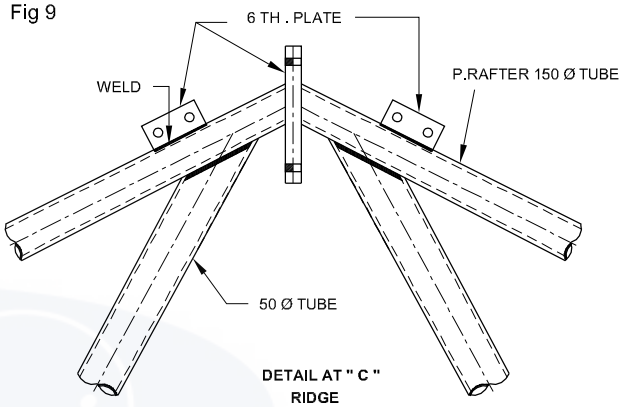
Fig 8



DETAIL AT 'B' (TIE AND STRUT)

AA2219E8

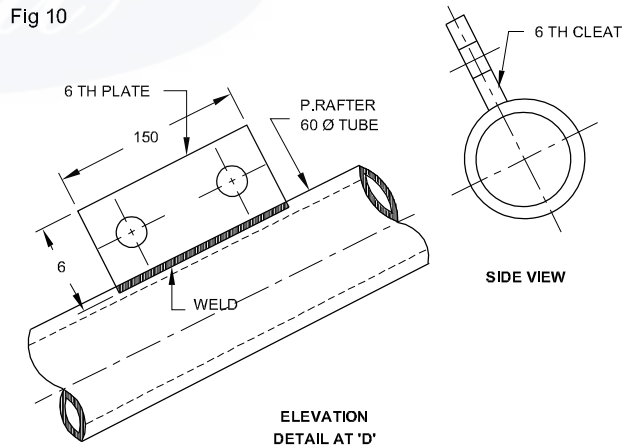
Fig 9



DETAIL AT " C "
RIDGE

AA2219E9

Fig 10



ELEVATION
DETAIL AT 'D'

AA2219EA

Water supply for toilet and kitchen

Objectives: At the end of this exercise you shall be able to
- **draft the details of underground as well as overhead water tank**
- **draft the plan showing the pipes connecting the tank and distributing water to respective floors.**

PROCEDURE

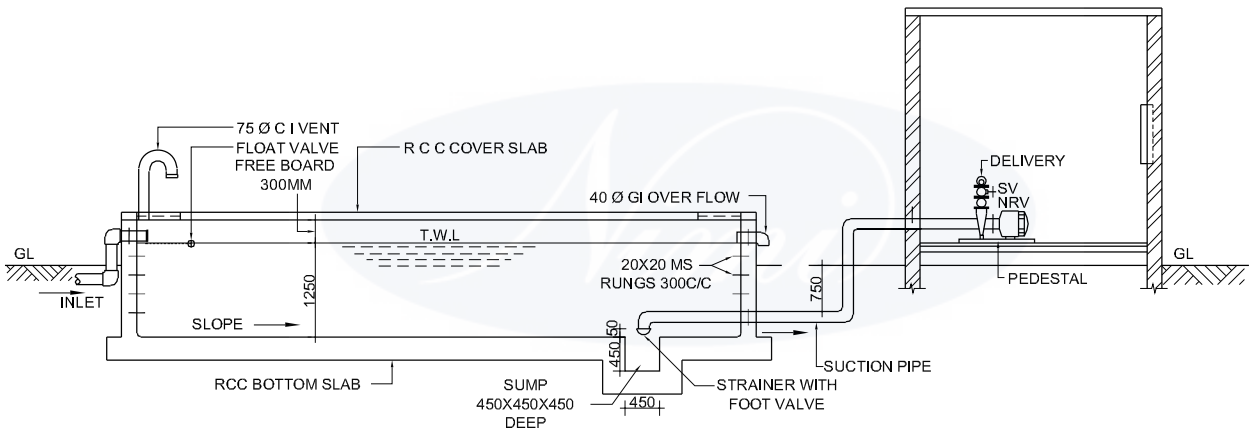
Draw an overhead tank on the terrace of size 1500x200 (size varies according to the number of occupancy of residence/building). This water over head water tank meets demand of water of domestic consumption as well as fr fire fighting requirments Fig 3.

Water supply from the municipal is first collected in under ground tank 400 x 800 x 1250 of capacity 4000 litres then the water is pumped to overhead tank for distribution to various floors by gravity (Fig 1)

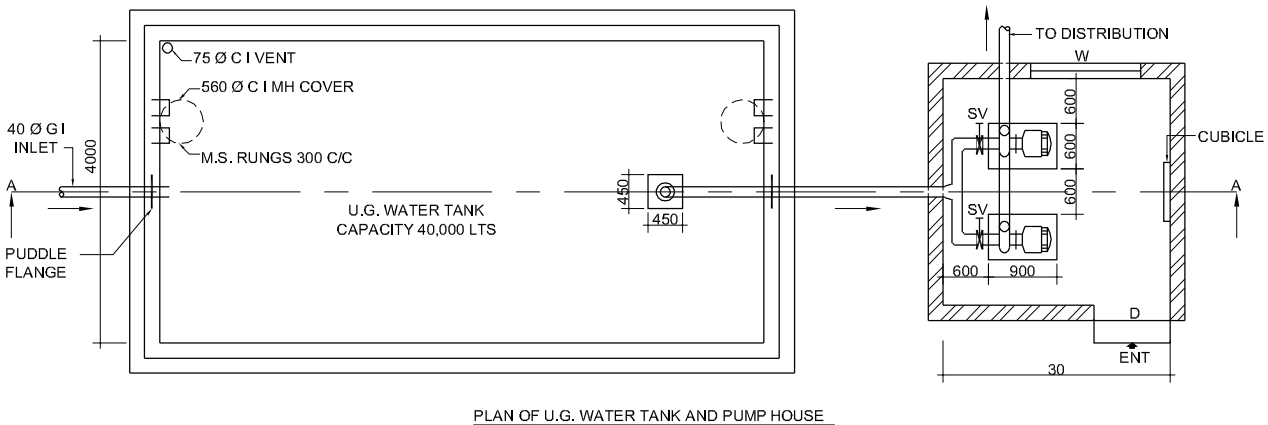
Draw a pump house of size 3000 x 300 from where water is pumped up to the overhead tank for distribution to different floors (Fig 2)

Draw GI pipe (down take pipe) from overhead tank to various floors connecting to every floor to supply water is toilets and kitchen.

Fig 1



SECTION AT A-A

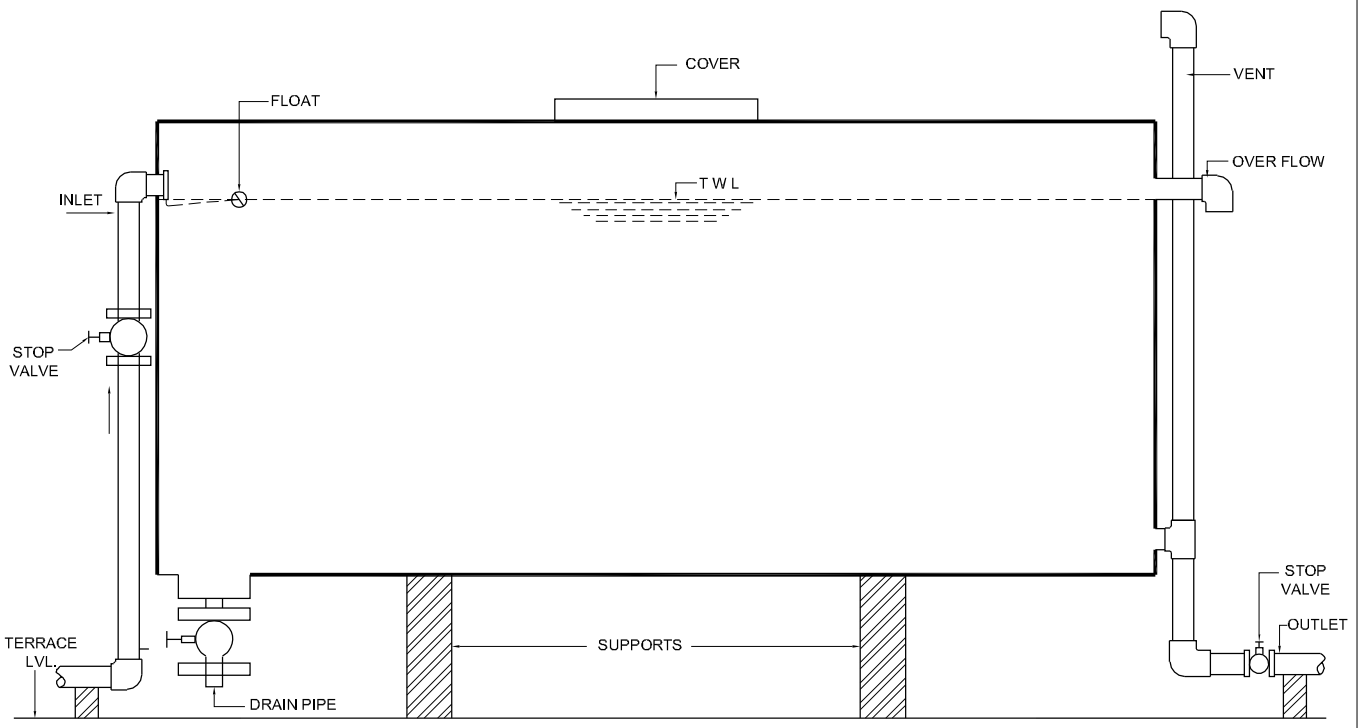


PLAN OF U.G. WATER TANK AND PUMP HOUSE

AA2301E1

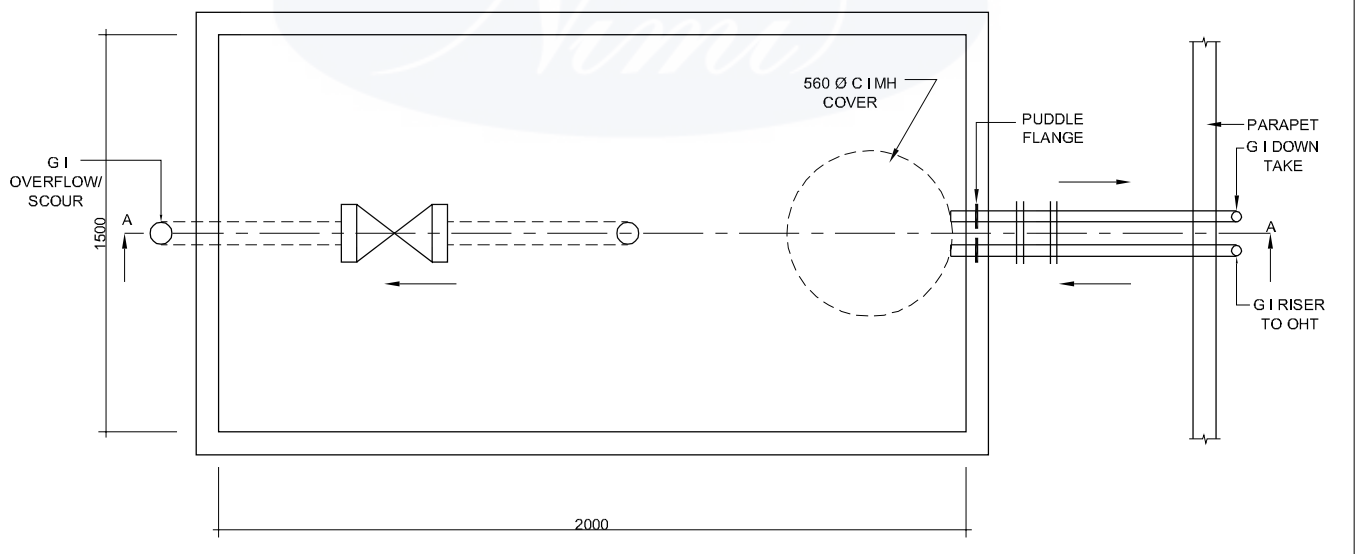
WATER TANK AND PUMP HOUSE

Fig 2



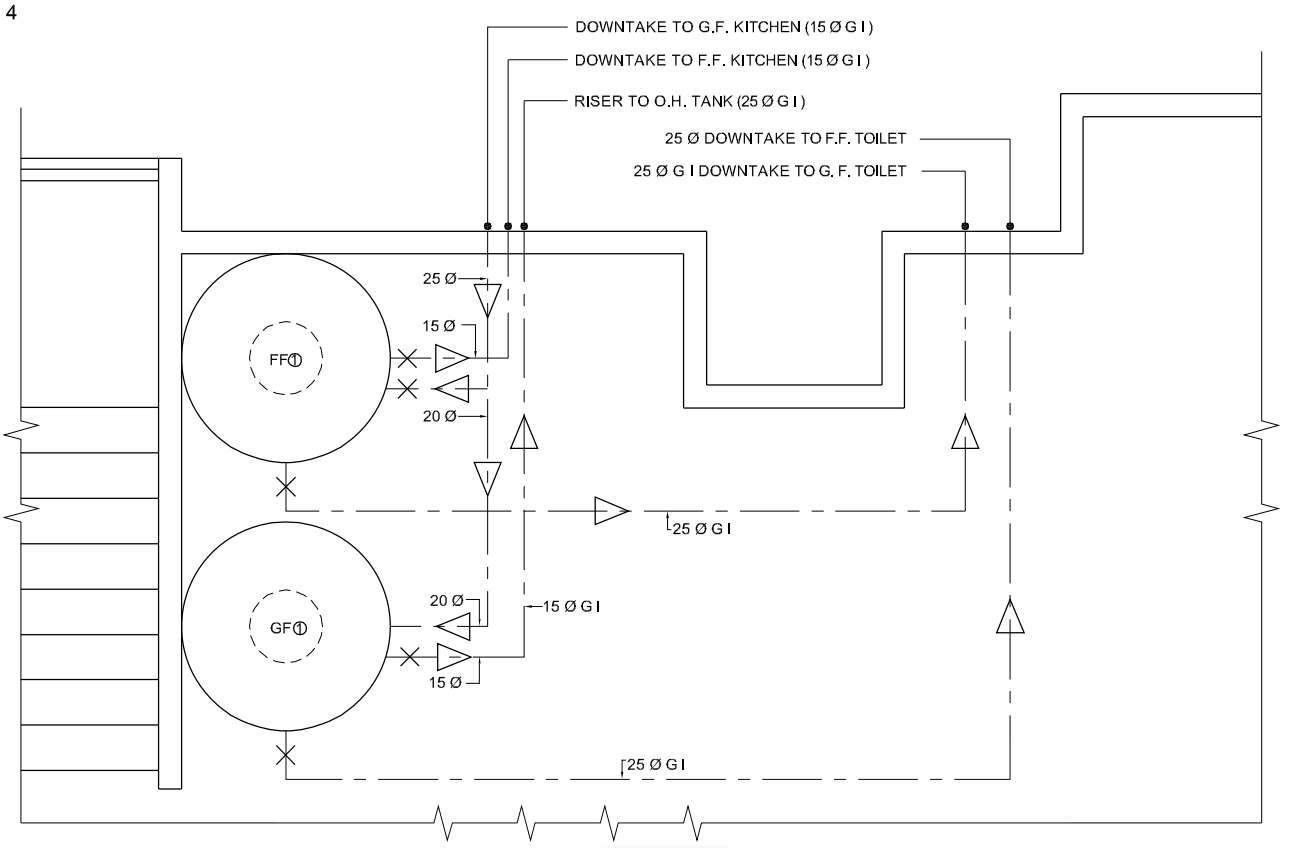
SECTION OF OVERHEAD TANK

Fig 3



DETAIL SHOWING PIPE CONNECTIONS TO OVERHEAD TANK ON TERRACE

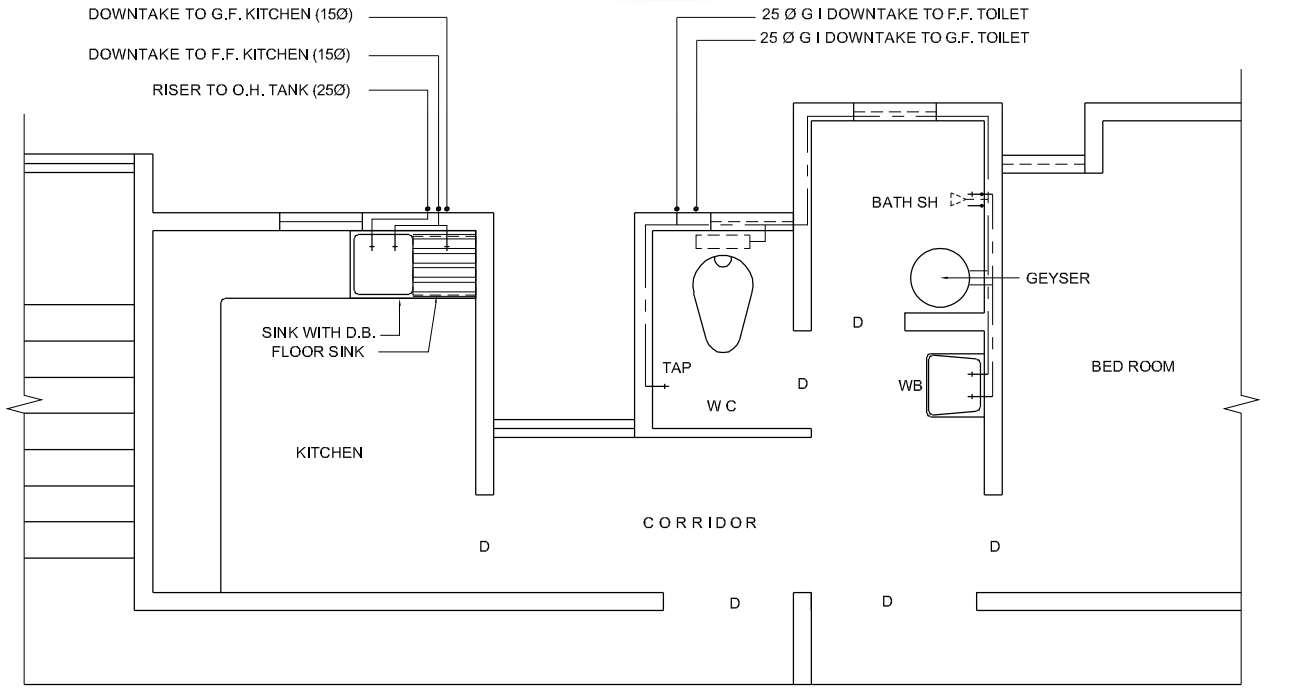
Fig 4



DOWNTAKE PIPES FROM OVERHEAD TANKS

AA2301E4

Fig 5



WATER SUPPLY SCHEME FOR A HOUSE

AA2301E5

Water supply for toilet and kitchen

Objectives: At the end of this exercise you shall be able to
- **connection of pipes (soil and waste pipes)**
- **location of floor traps**
- **connectivity of all pipes to main man hole.**

PROCEDURE

Draw the floor trap under the wash basin to collect waste water

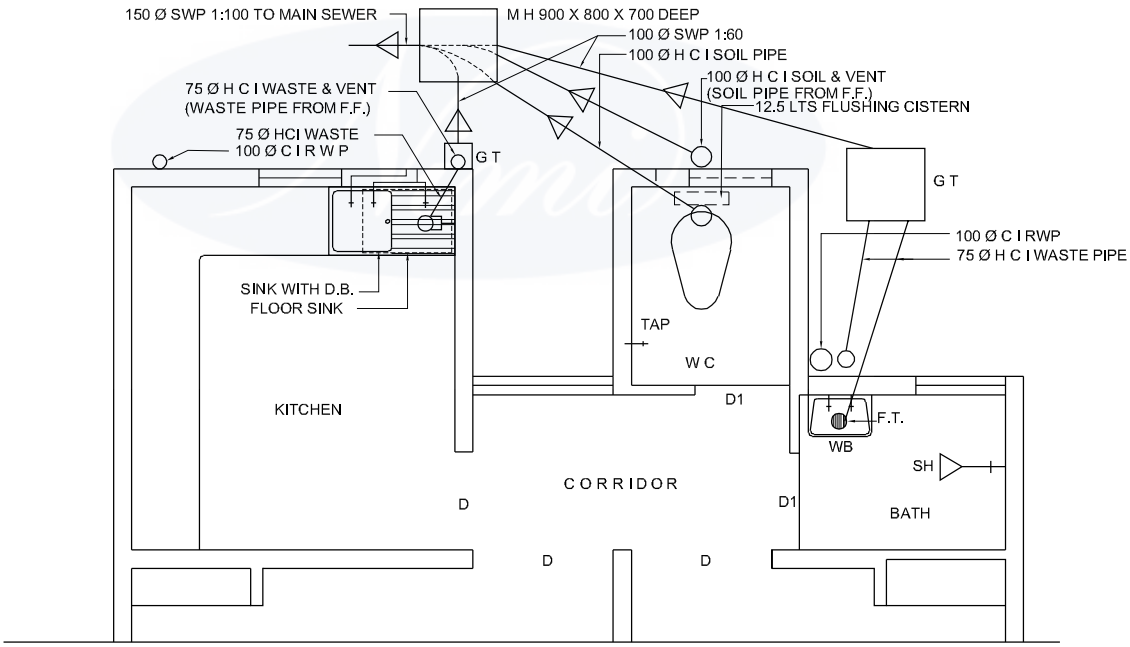
Draw the dia pipe connecting floor trap to gully trap

Draw 100 dia CI pipe connecting W.C to the man hole

Draw 75 dia pipe (rain water pipe from terrace)to the gully trap which the gets connected to man hole.

Draw a man hole of size 900 x 800 x 700 deep where all soil and waste water gets tollected which the connects to main man hole and then to the main sewer line (refer Fig 1 & Fig 2)

Fig 1



DRAINAGE LAYOUT

Draw gully trap, detail

Objectives: At the end of this exercise you shall be able to
- **draft the gully trap detail with referance to waste water pipe brick work slope maintained in the pipes also PCC structure.**

Requirements

Tools/Instruments

- Drawing board
- T square
- Set squares
- Drawing sheet
- Pencil
- Eraser
- Metric scale

PROCEDURE

TASK 1 :

Draw 300 x 300mm size of gully trap with brick wall of 115 thick. The trap again connects to IC (Inspection chamber or man hole)

— — — — — — — —

TASK 2 :

Draw section at “AA” of gully trap the depth of the trap is 675 mm which has the fixtures of 75mm HCI or PVC) waste and vent pipe, 300 x 300mm CI cover), cast iron (CI) covering.

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Fig 1

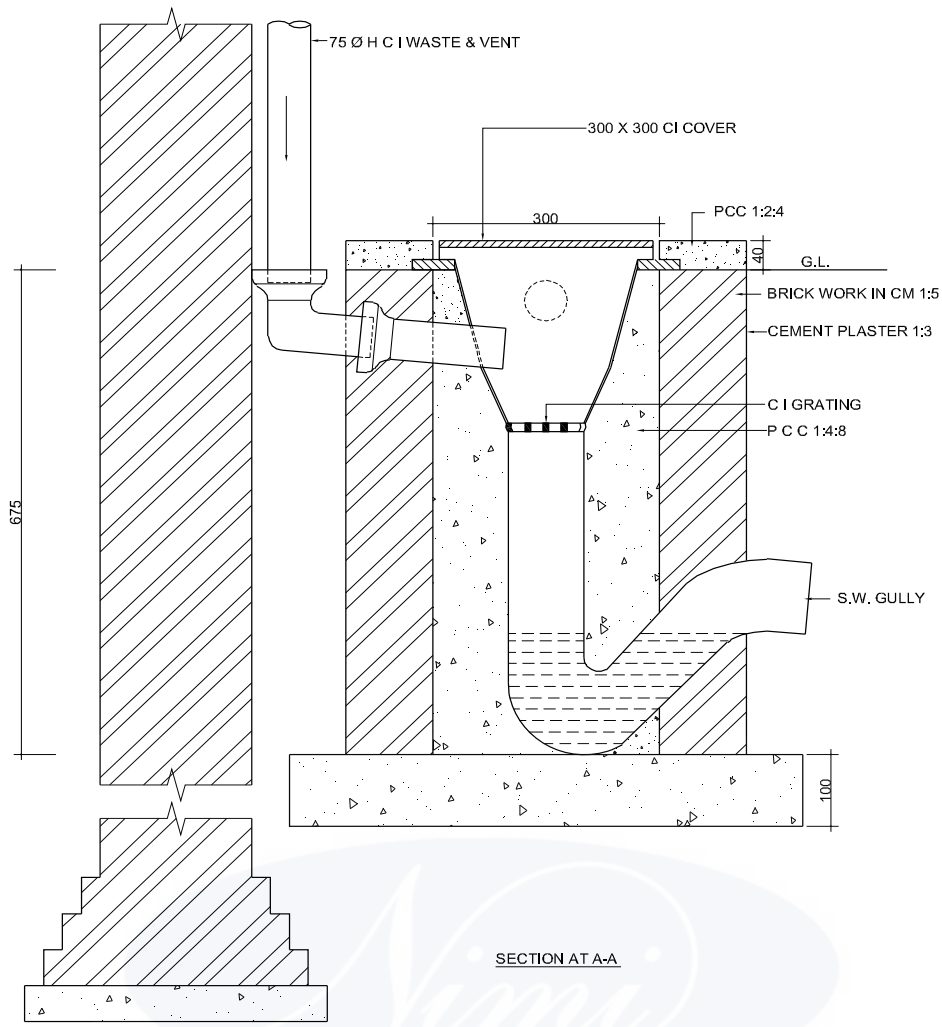
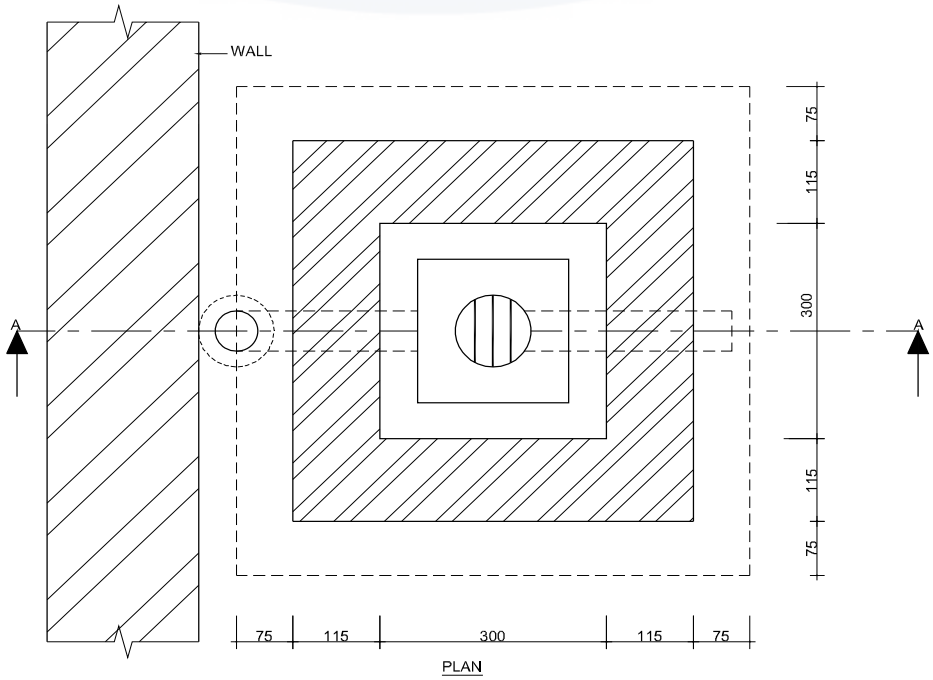


Fig 2



DETAIL OF GULLY TRAP

AA2303E1

Draw manhole detail

Objectives: At the end of this exercise you shall be able to
- **draft the manhole details with main sewers.**

Requirements

Tools/Instruments

- Drawing board
- T square
- Set squares
- Drawing sheet
- Pencil
- Eraser
- Metric scale

PROCEDURE

TASK 1 : **Metric Scale**

Draw the sectional plan of chamber of size 180 x 180cm and depth of the 150 cm depth. The portion of the chamber may be made smaller by controlling the brick mansory by laping 5cm (1/4th)(20 x 10 x 10) Brick mason of walls a shown in the drawing.

The manhole cover is put on the RCC slab covering the hole so reduced in size of 50cms thickness of the brick masonry all round the manhole is of (20 x 10 x10) brick.

Alternatively this may be done by concreting or using a pre fabricated concrete elements of this square shape or circular type also. (Fig 1)

— — — — — — — —

TASK 2 : **Draw the sectional elevation with the depth of 150cm.**

Draw the section of refilled pit for recharging borewell of different layers.

a) 40 mm aggregate : 1.5m depth

b) 20 mm aggregate : 0.60 m

c) Course sand : 0.60 m

d) Fine sand : 0.30 m

Draw the casing pipe 0.15m should be left to ground.

Draw isometric view of percolation pit (Fig 2)

Man hole of depth 1.5m

Full shallow manholes a chamber with vertical walls and covered with a slab, shall suffice. When the depth increases to 1.5m, the entry portion of the chamber may be made smalled by corbeling the brick masnory of the walls, as shown in this drawing. The method of corbelling involves projecting the bricks of successive courses by a distance not exceeding 1/4 of its length and reducing the opening as shown. The manhole cover is put on the RCC slab covering the hole, so reduced in size.

The sectional plan shows the chamber of size 120cm x 50 cm opening for the manhole. The corbelling is done on three sides. The fourth side is the verical wall, on which the rungs are fixed. The two sections of the manhole give details of the construction, arranged of rungs and the position of drains with benching. The corbelling of brick is the easiest method of reducing the size of the opening. Alternatively this may be done by concreting or using a prefabricated concrete element of this shape. (Fig 3)

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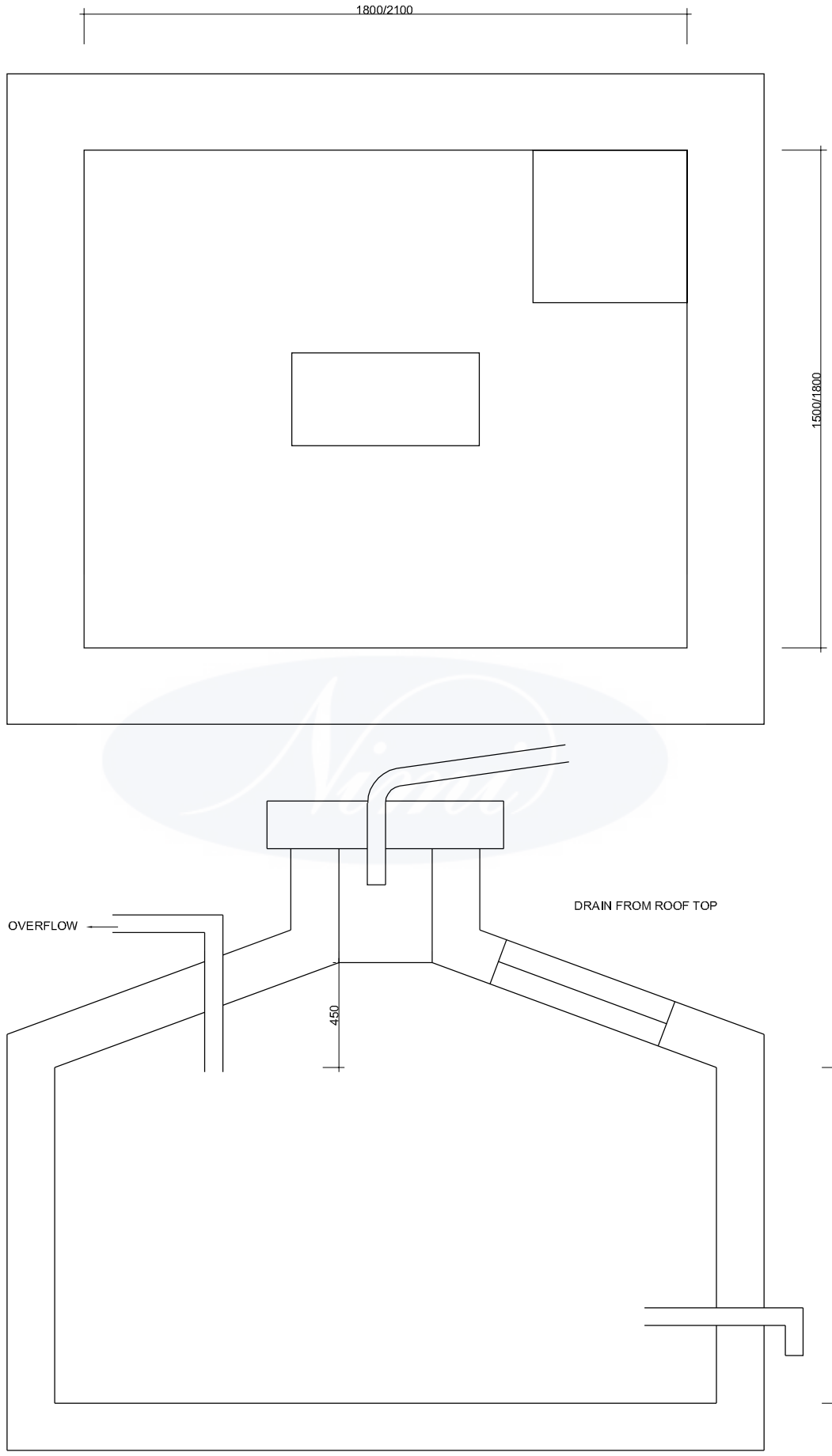
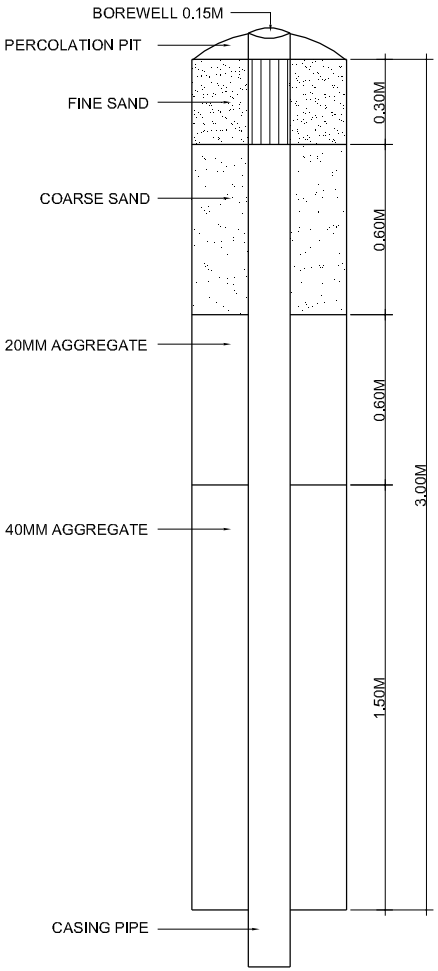
Fig 1



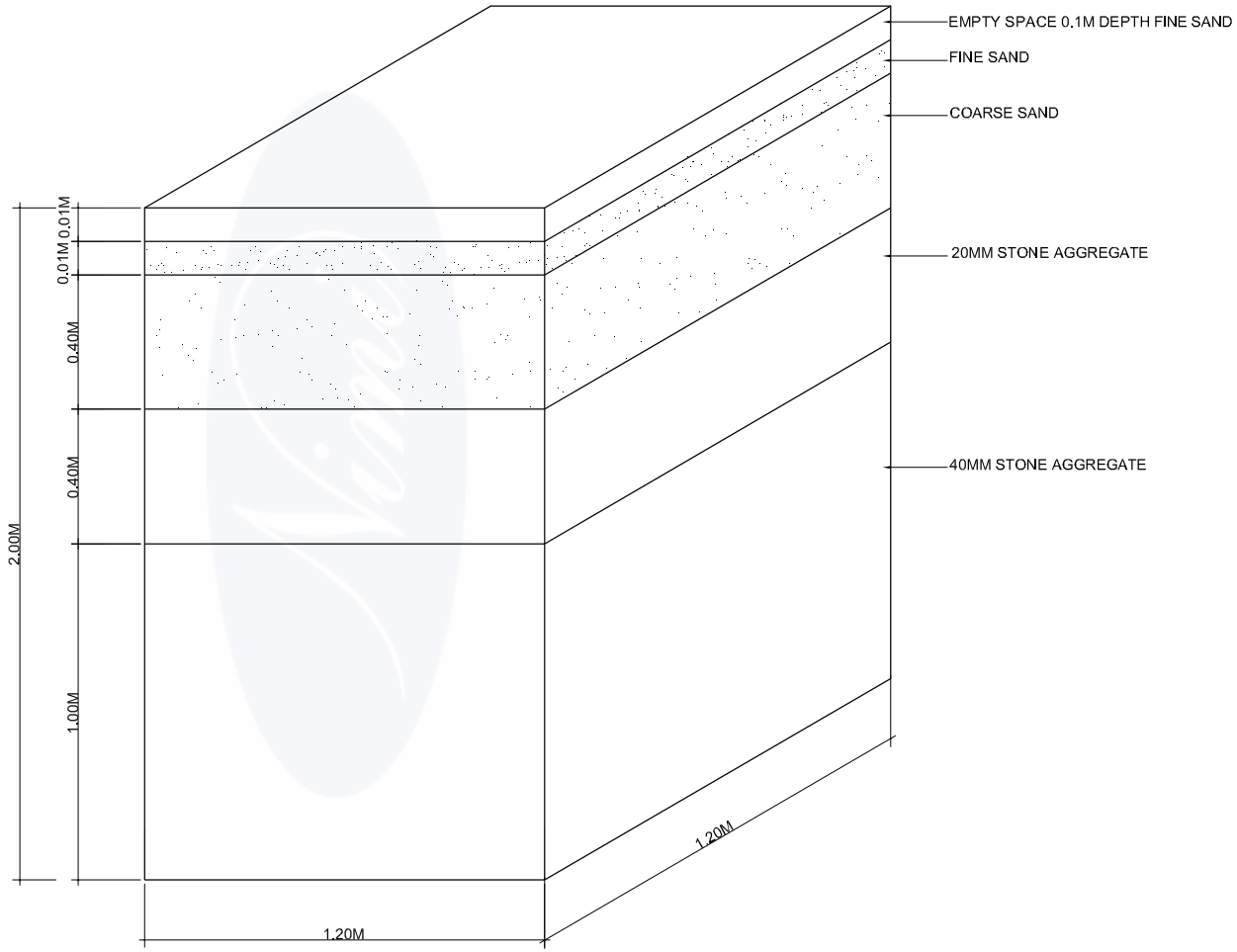
RAIN WATER HARVESTING

AA2304E1

Fig 2



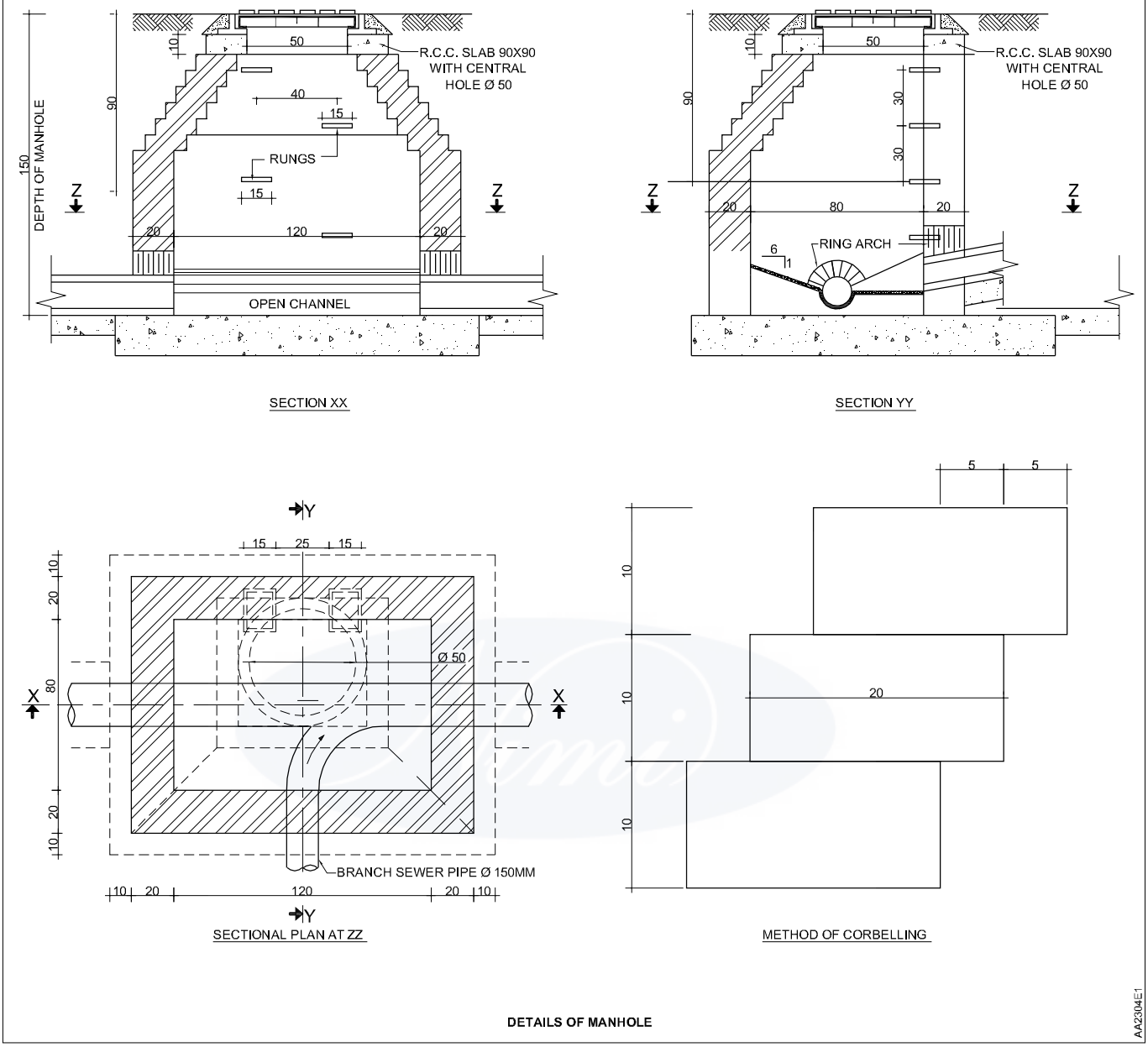
SECTION OF REFILLED PIT FOR RECHARGING BOREWELL



SECTION OF PERCOLATION PIT

RAIN WATER HARVESTING STRUCTURES

Fig 3



DETAILS OF MANHOLE

AA2304E1

Draw site plan showing building services

Objectives: At the end of this exercise you shall be able to
- **draw siteplan showing manhole, I.C. and building service.**

Requirements

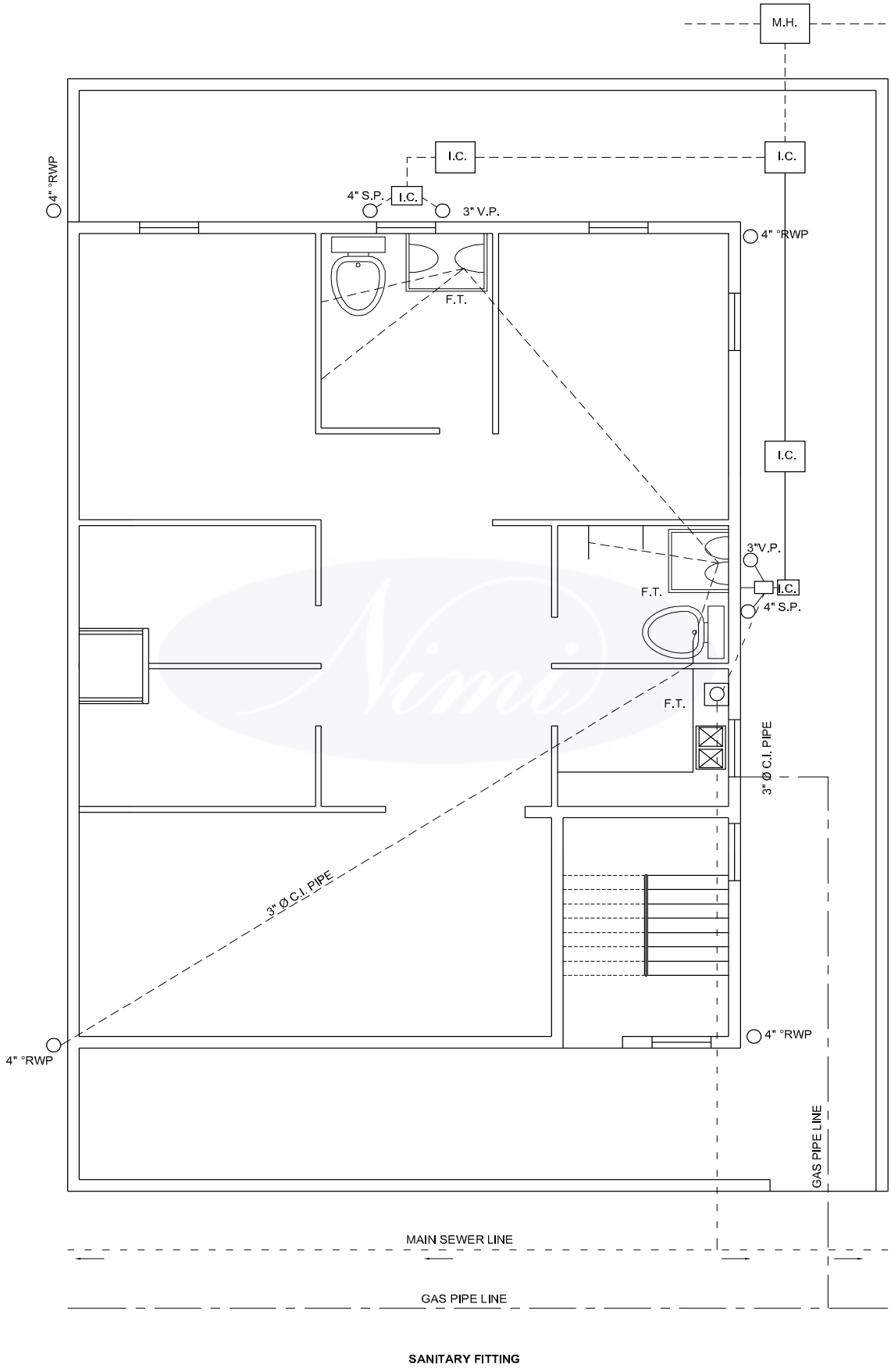
Tools/Instruments

- Drawing board
- T square
- Pencil

PROCEDURE

- Draw siteplan with toilet, kitchen firtures.
- Draw plumbing service lines of soild pipe vent pipe which laid in slope.
- Draw gully traps along with Inspection chambers (manhole) every junction.
- Draw the connecting line to main sewer line.
- Draw gas pipe line which supply from out source. (Fig 1)

Fig 1


M.H.
I.C.
I.C.
4" S.P.
I.C.
3" V.P.
4" °RWP
4" °RWP
F.T.
I.C.
3"V.P.
I.C.
4" S.P.
F.T.
F.T.
3" Ø C.I. PIPE
3" Ø C.I. PIPE
4" °RWP
4" °RWP
GAS PIPE LINE
MAIN SEWER LINE
GAS PIPE LINE
SANITARY FITTING
AA2305E1

Draw rainwater harvesting details

Objectives: At the end of this exercise you shall be able to
- **detail rainwater harvest pit and recycling the water.**

Requirements

Tools/Instruments

- Drawing board A3 size
- T square
- Pencil
- Eraser

PROCEDURE

TASK 1 :

- Draw rain water storage tank size of 2100 x 1800 of brick work with water proof.
- Draw sectional detail of tank with filter and out let.

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TASK 2 : **Draw rainwater harvest, structure recharging borewell.**

- Draw the section of refilled pit for recharging borewell of different layers

  40mm aggregate : 1.5 m depth

  20mm aggregate : 0.60.m depth

  course sand : 0.60 m

  Fine sand : 0.30m
- Draw the casing pipe 0.15 m should be left to ground
- Draw Isometric view of per location pit.

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Fig 1

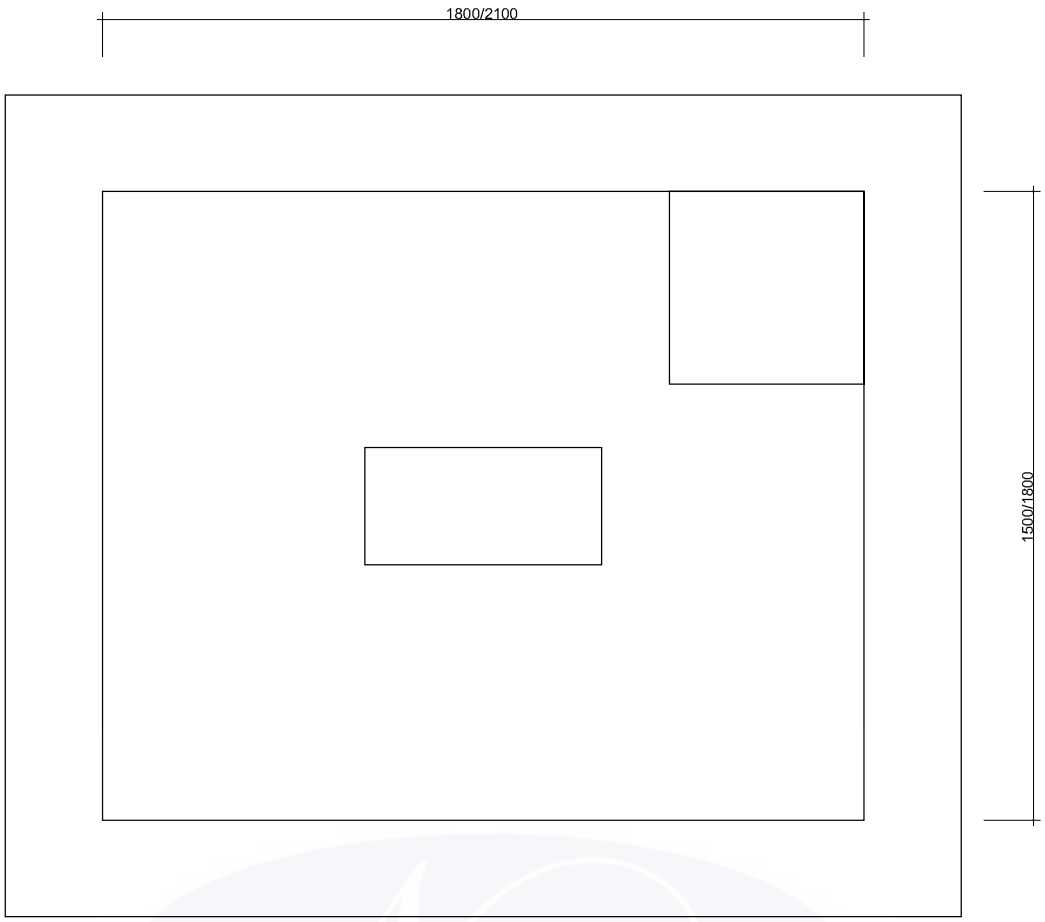
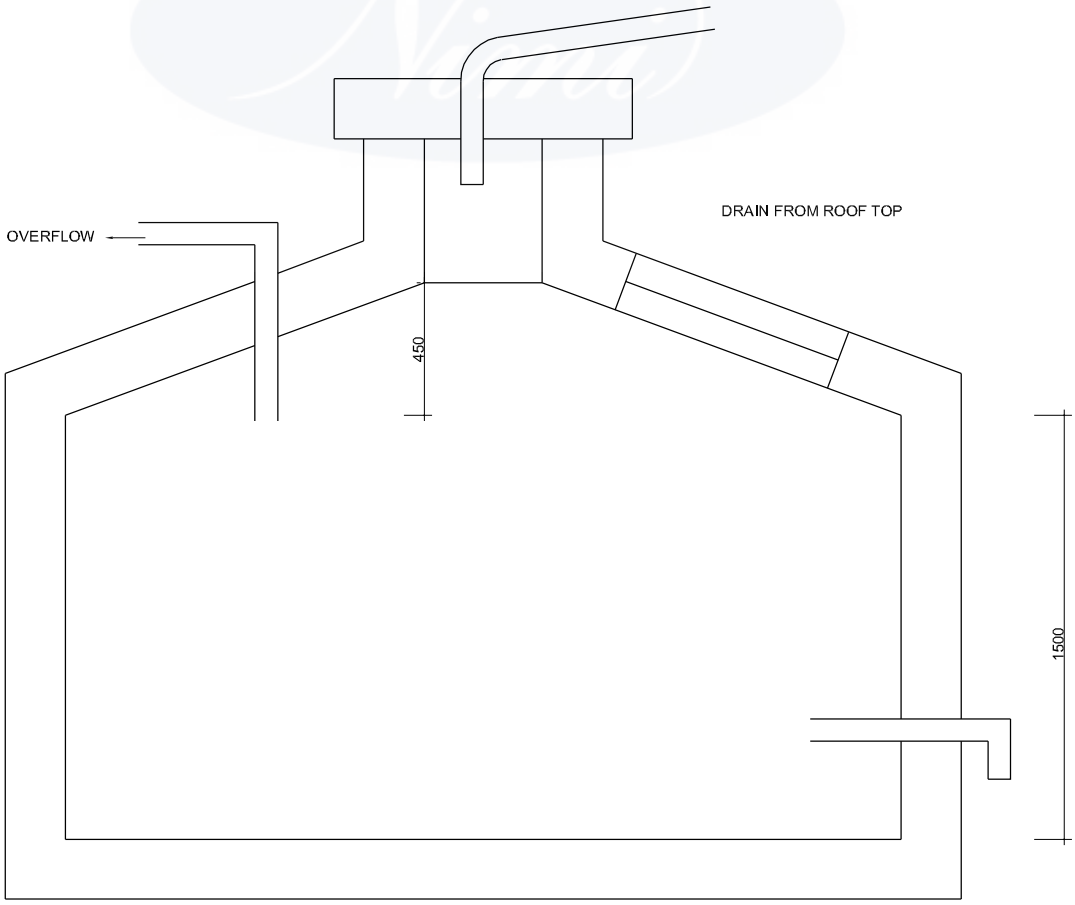


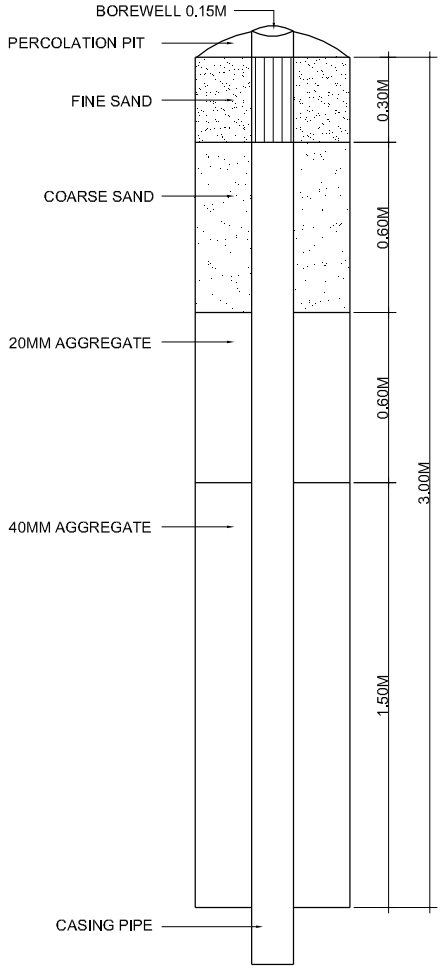
Fig 2



RAIN WATER HARVESTING

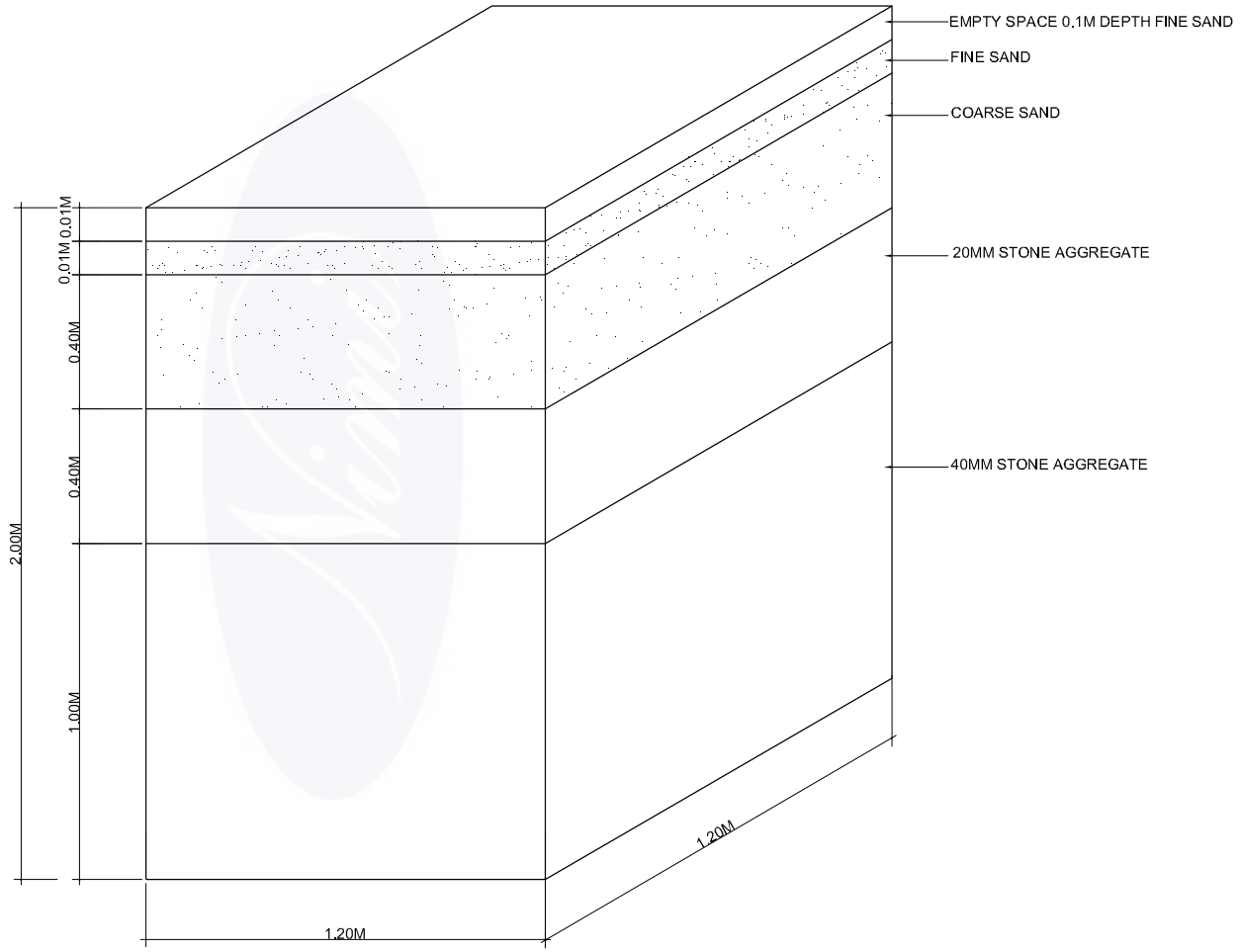
AD2306E1

Fig 3



SECTION OF REFILLED PIT FOR RECHARGING BOREWELL

Fig 4



SECTION OF PERCOLATION PIT

RAIN WATER HARVESTING STRUCTURES

AD2306E3

Nimi

Nimi