ENGINEERING GRAPHICS LAB

(20A03101P)

LAB MANUAL

I – BTECH

Prepared by

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Department of Mechanical Engineering

VEMU INSTITUTE OF TECHNOLOGY

(Approved by AICTE, New Delhi and Affiliated to JNTUA, Ananthapuramu)
Accredited by NAAC, NBA (EEE , ECE & CSE) & ISO 9001-2015 Certified Institution
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Chittoor, Andhra Pradesh -517112
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R20 Regulations

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR
(Established by Govt. of A.P., ACT No.30 of 2008)
ANANTAPUR – 515 002 (A.P) INDIA

Course Code	**Engineering Graphics**	**L**	**T**	**P**	**C**
20A03101P		**0**	**0**	**2**	**1**

Course Objectives

- Instruct the utility of drafting & modeling packages in orthographic and isometric drawings.
- Train the usage of 2D and 3D modeling.
- Instruct graphical representation of machine components.

Course outcomes (CO) : After completion of the course, the student can able to

CO-1: Use computers as a drafting tool.

CO-2: Draw isometric and orthographic drawings using CAD packages.

LIST OF EXPERIMENTS

I.	Introduction to Computer Aided Drafting software packages.
II.	Practice on basic elements of a Computer Aided Drafting packages
III.	Practice on features of a Computer Aided Drafting package
IV	Drafting of Solids, Intersection of Solids
V	Drafting of Perspective views
VI	Drafting of Orthographic views of simple parts

References

1. Computer – Aided Engineering Drawing, S. Trymbaka Murthy. University Press.
2. Engineering Graphics for Degree, K.C. John. PHI Publications.

Engineering Graphics Lab (20A03101P)

2022-2023

VEMU INSTITUTE OF TECHNOLOGY

(Affiliated To JNTUA University)

NAME: ……

ROLL No. :……

Branch:

Course: B. Tech

Year/Semester : I/I

INSTRUCTIONS TO STUDENTS

1. Students are required to remove their footwear outside the center and keep it in the box provided for the same.
2. Students should leave their belongings outside the lab except their observation note book, the concerned books/manuals and calculators.
3. Students are requested not to place their legs on the wall or on the table.
4. Students should refrain from leaning on the table and sitting on it.
5. Before logging in to a particular terminal, if there is something wrong in the terminal, the student should report the same immediately to the concerned staff.
6. Students should not use any disks brought from outside without prior permission from the concerned staff.
7. Students can get the required manual or disks from the staff after signing in the appropriate register.
8. Students should collect their printouts before leaving the lab for that particular session.
9. Before leaving the Terminal, the students should logout properly and leave their chairs in position.
10. Students are not allowed to take any manual outside the lab.
11. Edibles are strictly prohibited in the lab.
12. No internet browsing allowed during the lab hours.

Engineering Graphics Lab (20A03101P)

LIST OF EXPERIMENTS:

I. Introduction to Computer Aided Drafting software packages.
II. Practice on basic elements of a Computer Aided Drafting **p**ackages
III. Practice on features of a Computer Aided Drafting **p**ackage
IV Drafting of Solids, Intersection of Solids
V Drafting of Perspective views
VI Drafting of Orthographic views of simple parts

Note: Any of the standard Software Packages like – AUTO CAD, Pro-E, Uni – Graphics, Catia …. Etc may be used

References:

1. Computer – Aided Engineering Drawing, S. Trymbaka Murthy. University Press.
2. Engineering Graphics for Degree, K.C. John. PHI Publications.

VISION OF THE DEPARTMENT

To become a Centre of excellence in the field of Mechanical Engineering by producing graduates with technical knowledge, research, consultancy and entrepreneurial skills along with leadership qualities, ethics and lifelong learning to cater the needs of the society.

MISSION OF THE DEPARTMENT

To impart quality education and training to nurture globally competitive mechanical engineers by effective teaching-learning practices and state-of-the art laboratories through eminent faculty.

To establish linkages with industries and research organizations to bring excellence in problem solving skills, research and consultancy services.

To empower the graduates with creative thinking, leadership qualities, lifelong learning skills, spirit of entrepreneurship, social and ethical values by offering value based education.

PROGRAM OUTCOMES

Engineering Graduates will be able to:

PO_1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO_2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO_3	**Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO_4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO_5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO_6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO_7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO_8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO_9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO_10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO_11	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO_12	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Program Specific Outcomes

A Graduate of the Mechanical Engineering Program will be able to:

S. No.	After graduation, the students will able to:
PSO_1	**Higher Education:** Apply the fundamental knowledge of mathematics, science, and Mechanical Engineering to pursue higher education in the areas of Design, Thermal and Manufacturing Engineering.
PSO_2	**Employment:** Get employed in core and allied industries through their proficiency in the program-specific domain knowledge, specialized computational package and programming or become an entrepreneur.

Programme Educational Objectives

PEO-1: To plan, design, construct, maintain and improve mechanical engineering systems that are technically sound, economically feasible and socially acceptable.

PEO-2: To apply modern computational, analytical, simulation tools and techniques to address the challenges faced in mechanical and allied engineering industries.

PEO-3: To communicate effectively by using innovative tools, demonstrate leadership qualities, research & entrepreneurial skills, exhibit professionalism, ethical attitude, team spirit along with lifelong learning to achieve career and organizational goals.

Course Outcomes:

At the end of the course the student will be able to

- Use computers as a drafting tool.
- Draw isometric and orthographic drawings using CAD packages.

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S.No	Date	Name of Experiment	Page	Initial
1.		Introduction to computer aided drafting Software packages		
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5.		Practice exercises 2d		
6.		3Dviewof V-block		
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INTRODUCTION TO COMPUTER AIDED DRAFTING SOFTWARE PACKAGES

Computer Aided Designing and Drafting(CADD)

CADD is an electronic tool that enables us to make quick and accurate drawings. CADD has number of advantages over drawings created on a drawing board. Electronic drawings can be modified quite be represented in a variety of formats.

CADD extends its power to yet another branch of engineering called computer aided manufacturing (CAM).CADD and manufacturing program are often integrated into one system called CAD-CAM. This system import CADD drawings into CAM program to automate the manufacturing process. When the design is finalized, the drawings are brought into a CAD-CAM system that uses numerical data from the CADD drawing for actual manufacturing.

There is separate category of programs called Computer Aided Engineering(CAE) that can use CADD drawing for engineering analysis. The CAE programs have a number of applications in Structural Design, CivilEngineering, Mechanical Engineering and Electrical Engineering. The Mechanical engineer can test a machine assembly and also a prototype electronic model and test it without building a physical model.

Expectations from CADD

We can do amazing things with CADD that we never thought possible while creating drawings with pen or pencil. The following are some of the important capabilities that make CADD a powerful tool.

- Presentations
- Flexibility in editing
- Unit and accuracy levels
- Storage and access for drawings
- Sharing CADD drawings

Presentations

There are a number of ready-made presentations symbols available in CADD that can be used to enhance the look of drawings. In addition to prepare impressive presentations on paper, we can use CADD to make an on – screen presentations. Advanced CADD programs ever allow us to create an animated image.

Flexibility in editing

CADD allows us to work with great accuracy. If we need to create highly accuracy geometric shapes, CADD is the answer. It can help avoid time-consuming mathematical calculations.

Unit and accuracy level

We can work with as high precession as $1/1000^{th}$ of an inch.

Storage and access of drawing

A computer electronic filing system has the following advantages over the traditional filing system.

- It is quick and convenient to organize CADD drawing in a computer.
- It enables us to create a highly organized environment.
- An electronic drawing never gets old and faded.

Sharing CADD Drawing

The electronic drawing can be shared by a number of users, allowing them to Co-ordinate projects and work as a team. This is accomplished by connecting different computer via a network.

About AutoCAD

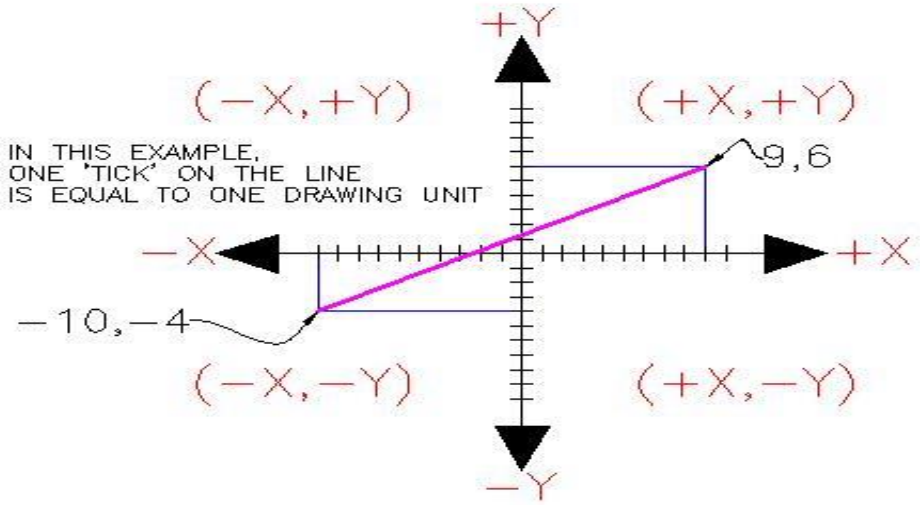
AutoCAD is a Computer Aided Design (CAD)program used by just about every Engineering and Design office in the world. Although there are alternative CAD packages, AutoCAD is by far the most widely used system. Autodesk's AutoCAD is the industry leader in CAD packages. Used by Civil Engineers, Architects, Mechanical and Electrical Engineers, Aeronautical Engineers plus many other disciplines.

There have been several versions of AutoCAD over the years, with each new version introducing new and more powerful features than its predecessor. The latest version of AutoCAD (at the time of writing) is

AutoCAD2011. Any courses, whether through community colleges or online universities, that are related to Engineering or Architecture should be considered incomplete if they do not introduce students to AutoCAD. Accurate, scale drawings can be created and published using AutoCAD powerfulfeatures.3D models can also be created giving the designer absolute control over the design from start to finish. The computerized model can be viewed through a 360ºangle,and even 'rendered' with a texture on screen to give an idea of the finished product.

The X, Y co-ordinate system

Everything that we draw in AutoCAD is exact. It will be more accurate than we will ever need it to be. We're taking14 decimal points accurate. All objects drawn on the screen are placed there based on a simple X,Y co-ordinate system. In AutoCAD this is known as the World Co-ordinate System(WCS).



AutoCAD uses points to determine where an object is located. There is an origin where it begins counting from. This point is (0,0). Every object is located in relation to the origin. If we were to draw a line straight out to the right from the origin, this would be considered the positive X-axis. If we were to draw a line straight up, this would be the positive Y-axis. The picture above shows a point located at(9,6).This means that the point is 9 units over in the X-axis and 6units up in the Y-axis. When we are working

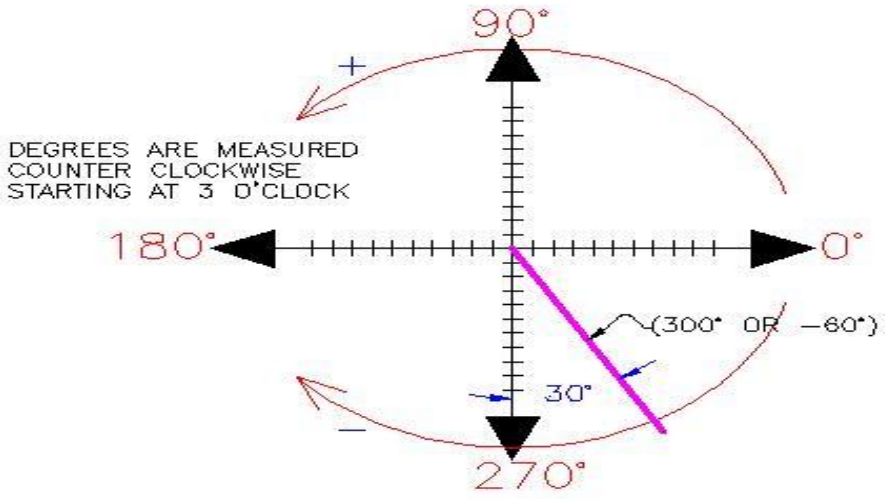
with points, X always comes first. The other point shown is (-10,-4).This means that the point is 10 units in the negative X-axis (left) and 4 units in the negative Y-axis (down).

A line has two points, a start point and an end point. AutoCAD works with the points to display the line on the screen. Most of the time we will not have an indication of where the origin. We may need todraw a line from the end point of an existing line. To do this we use relative points. These work the same way, but we have to add the @ symbol (shift+2) to tell AutoCAD that this next point is relative from the last point entered.

- ABSOLUTE POINTS are exact points on the drawing space.
- RELATIVE POINTS are relative to an OBJECT on the drawing space.

Angular Measurement

AutoCAD measures angles in a particular way also.



When drawing lines at an angle, we have to begin measuring the angle from 0 degrees, which is at the36o'clock position. If we drew a line at 90 degrees, it would go straight up. The example shows a line drawn at +300 degrees (270+30),or-60 degrees.

Entering Points in AutoCAD

We can enter points directly on the command line using three different systems. The one we use will depend on which is more applicable for the situation. The three systems are as follows:

Types of Co-Ordinates:

Absolute Co-Ordinates

Using this method, we enter the points as they relate to the origin of the

WCS. To enter a point just enters in the exact point as X,Y.

Relative Co-Ordinates

This allows us to enter points in relation to the first point we have entered. After we've entered one point, the next would be entered as @X,Y. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.

Polar Co-Ordinates

We would use this system if we know that we want to draw a line a certain distance at a particular angle. We would enter this as @D<A. In this case, D is the distance and A is the angle. Example:@10<90will draw a line 10units straight up from the first point.

The three ways of entering co-ordinates shown above are the ONLY way AutoCAD accepts input. First decide which style we need to use, and then enter as shown. Remember that X is always before Y(alphabetical). Don't forget the '@' symbol when we are entering relative points.

BASIC ELEMENTS OF A COMPUTER AIDED DRAFTING PACKAGES

FUNCTION KEYS

SHORTCUT	COMMAND	COMMENT
F1	HELP	Opens Autocad help
F2	COMMAND TEXT WINDOW	Switches between text screen and
F3	OSNAP	Switches osnap on/off
F5orCTRL+E	ISOPLANE	Cycles through iso planes
F6orCTRL+D	COORDS	Turns coordinate displayon/off
F7orCTRL+G	GRID	Turns grid on/off
F8orCTRL+L	ORTHO	Turns ortho on/off
F9orCTRL+B	SNAP	Turns snap on/off
F10orCTRL+U	POLAR	Turns polar on/off
F11orCTRL+W	OSNAPTRACK	Turns object snap tracking on/off
F12	DYNMODE	Turns dynamic input on/off

BASIC DRAWCOMMANDS:

Line(L): Draws straight lines between two points.  **Circle(C):** Draws a circle based on a center point and radius.  Circle with the following options:

- Center Radius
- Center Diameter
- Two Points
- Three Points
- Tangent, Tangent, radius (TTR)
- At the specify the center point for the circle or[3P/2P/TTR].

Arc(A): Draws an arc (any part of a circle or curve) through three known points.

Rectangle (REC): Draws a rectangle after you enter one corner and then the second. 

Display Commands:

Limits: Sets the size of the drawing paper.

Zoom(Z): Enlarges or reduces the display of a drawing. 

Pan (P): Panning allows you to quickly move around the drawing area at the same  magnification you currently have set. Left click and hold to move around your drawing.

Editing Commands:

Erase(E): Erases entities from the drawing. 

Extend(EXT): Lengthens a line to end precisely at a boundary edge.

Trim(TR): Trims objects to a selected cutting edge. 

Grips: We can edit selected objects by manipulating grips that appear at defining points on the object. Grips are not a command. To activate

grips simply pick the object. Small squares will appear at various entity-specific positions. By selecting an end grip we can stretch the entity to change its size. By selecting the center grip we can move the entity to a new location. To remove grips press CTL- C twice. We can perform the following using grips: Copy, Multiple Copy, Stretch, Move, Rotate, Scale, and Mirror.

Creating Layers:

Layer (LR): Starts the Layer and Line type property dialog box. 

Construction Commands:

Array(AR): Makes multiple copies of selected objects in a rectangular or circular pattern. 

Copy (CO): Copies object(s) once or multiple times. 

Mirror (MI): Creates a mirror image of an object or selection set. 

Move(M): Moves an object or objects. 

Offset(O): Constructs an entity parallel to another entity at a specified distance.  Offset can be used with lines, circles, arcs, and poly lines.

Fillet (FI): Creates a round corner between two lines. 

Chamfer (CHA): Creates an angled corner between two lines. 

Stretch (S): Stretches an object after you have selected a portion of it. 

Rotate(RO): Rotates objects to a certain angle. 

Scale(SC): Proportionately resizes (or scales) objects 

Boundary Hatch (H): Covers an area with a predefined pattern. 

Hatch Edit (HE): Edits an existing Hatch. 

Dimension (DIM): Dimensions previously drawn objects.

SKETCHER

Expt. No: 1 **Date:**

DRAFTING ORTHOGRAPHIC VIEW OF QUADRANT

AIM:

To draw the orthographic view of the QUADRANT using AutoCADSoftware and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

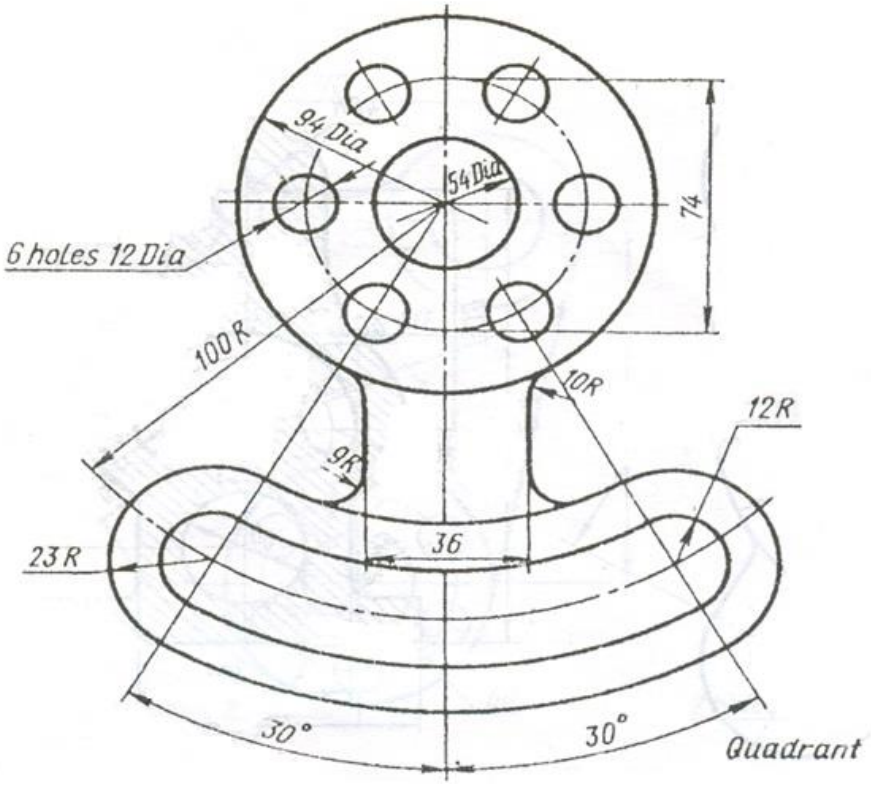
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file usingLIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, Polar Array, arc, fillet, offset) draw the given Quadrant as per the dimensions.
5. By using the Annotate tab, annotate the dimensions of the given Quadrant.
6. Save the finished drawing in the local disk using "quadrant.dwg" file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 2 **Date:**

2D SKETCH OF OBJECT MODEL-2

AIM:

To draw the orthographic view of the Object Model-2 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

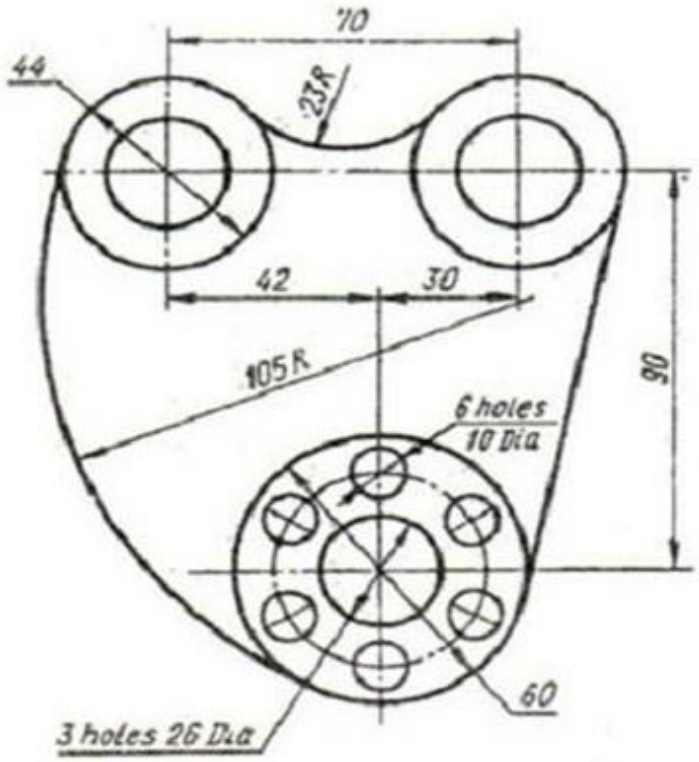
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, Polar Array, arc, fillet, offset) draw the given Object Model-2 as per thedimensions.
5. By using the Annotate tab, annotate the dimensions of the given Object Model-2.
6. Save the finished drawing in the local disk using “object model-2.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 3 **Date:**

2D SKETCH OF PRACTICE MODEL-1

AIM:

To draw the orthographic view of the Practice Model-1 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

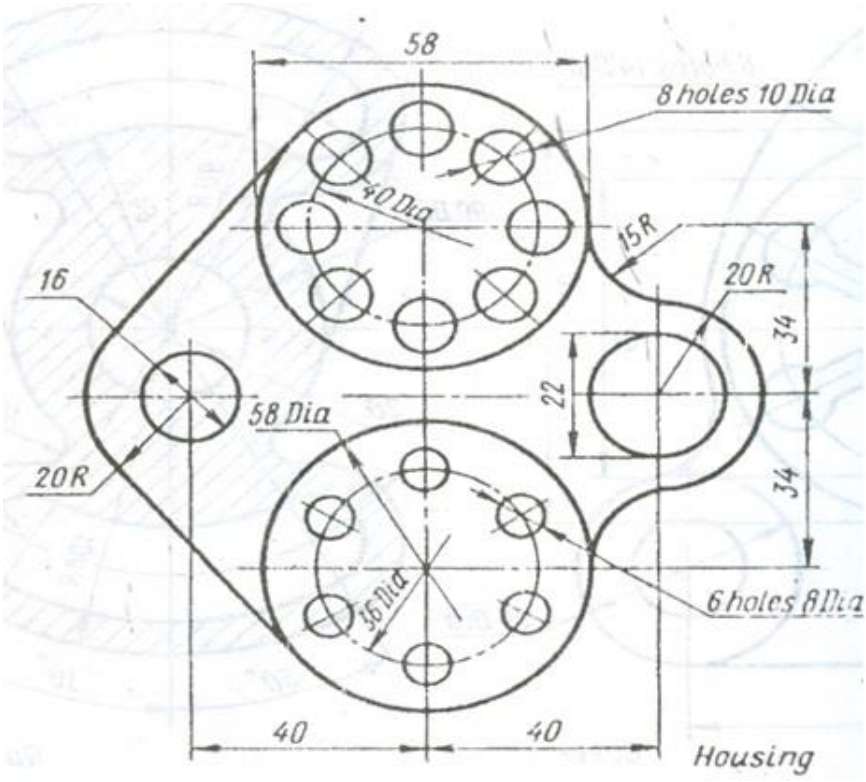
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, Polar Array, arc, fillet, offset) draw the given Practice Model-1as per thedimensions.
5. By using the Annotate tab, annotate the dimensions of the given Practice Model-1.
6. Save the finished drawing in the local disk using “practice model-1.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 4 **Date:**

2D SKETCH OF PRACTICE MODEL-2

AIM:

To draw the orthographic view of the Practice Model-2 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

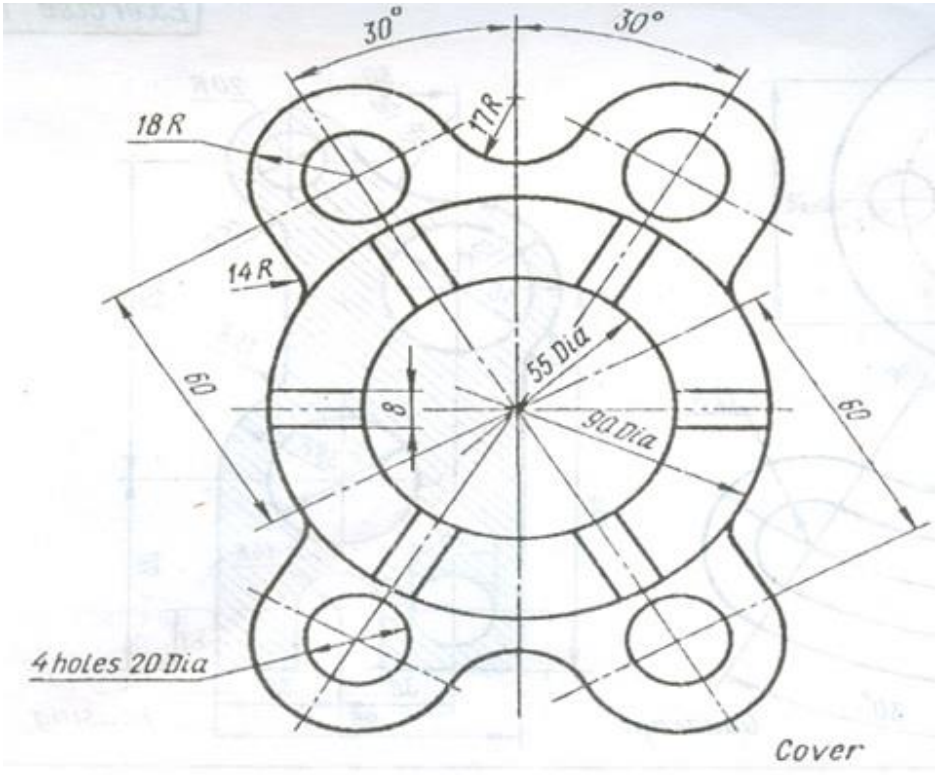
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, Polar Array, arc, fillet, offset) draw the given Practice Model-2 as per thedimensions.
5. By using the Annotate tab, annotate the dimensions of the given Practice Model-2.
6. Save the finished drawing in the local disk using “practice model-2.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 5 **Date:**

2D SKETCH OF PRACTICE MODEL-3

AIM:

To draw the orthographic view of the Practice Model-3 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

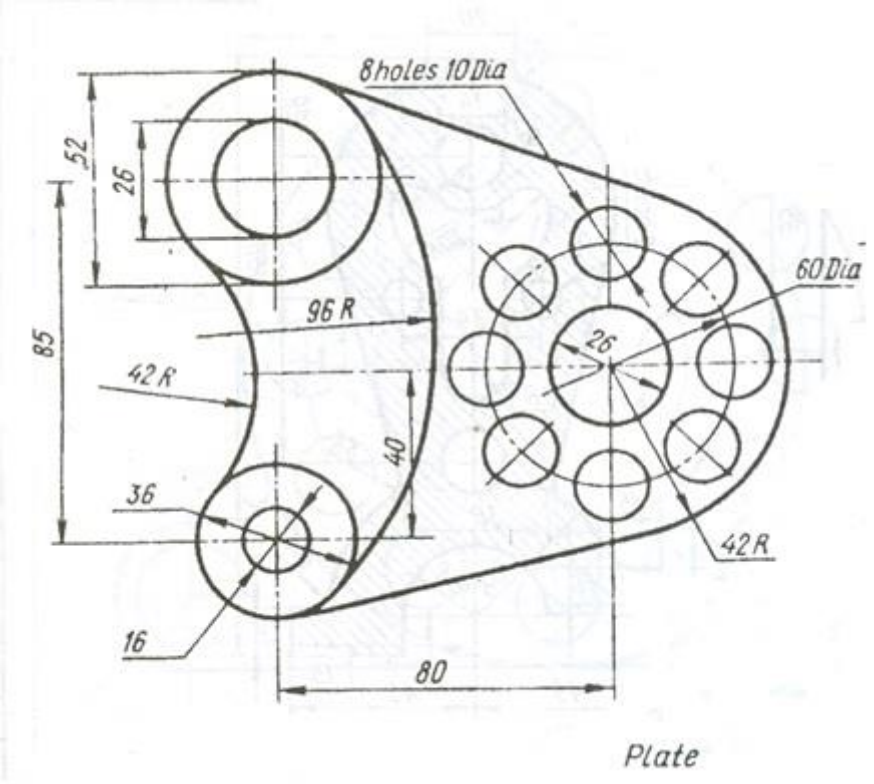
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, Polar Array, arc, fillet, offset) draw the given Practice Model-3 as per thedimensions.
5. By using the Annotate tab, annotate the dimensions of the given Practice Model-3.
6. Save the finished drawing in the local disk using “practice model-3.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 6 **Date:**

2D SKETCH OF PRACTICE MODEL-4

AIM:

To draw the orthographic view of the Practice Model-4 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

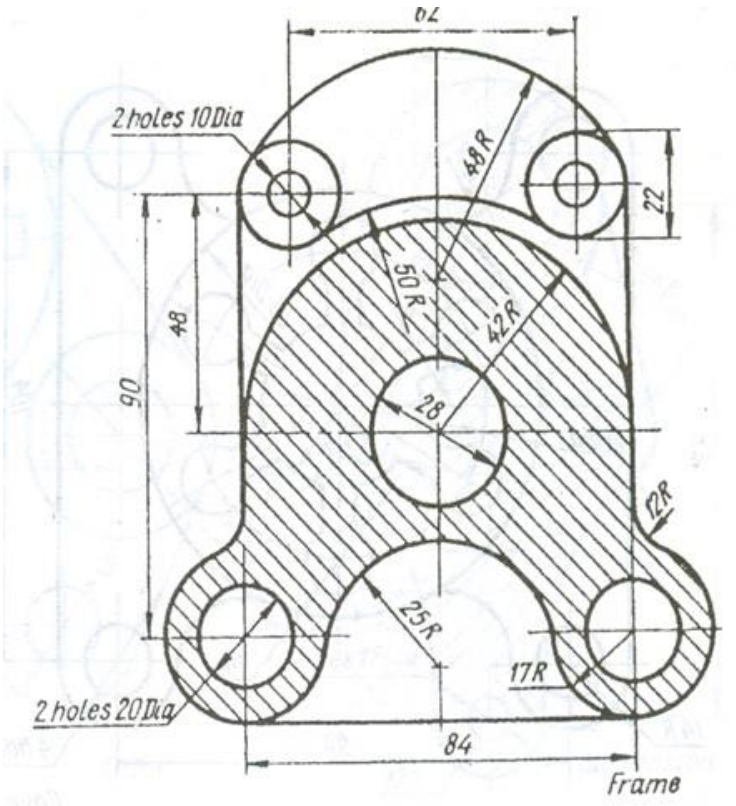
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Line,
6. Circle,
7. Arc,
8. Polar Array,
9. Offset,
10. Fillet,
11. Mirror,
12. Trim,
13. Copy,
14. Move,
15. Hatch,
16. Layer,
17. Dimensions
18. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, arc, fillet, offset) draw the given Practice Model-4 as per thedimensions.
5. By using the hatch command, hatch the sectioned portion as shown in the Model.
6. By using the Annotate tab, annotate the dimensions of the given Practice Model-4.
7. Save the finished drawing in the local disk using “practice model-4.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

Expt. No: 7 **Date:**

2D SKETCH OF PRACTICE MODEL-5

AIM:

To draw the orthographic view of the Practice Model-5 using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

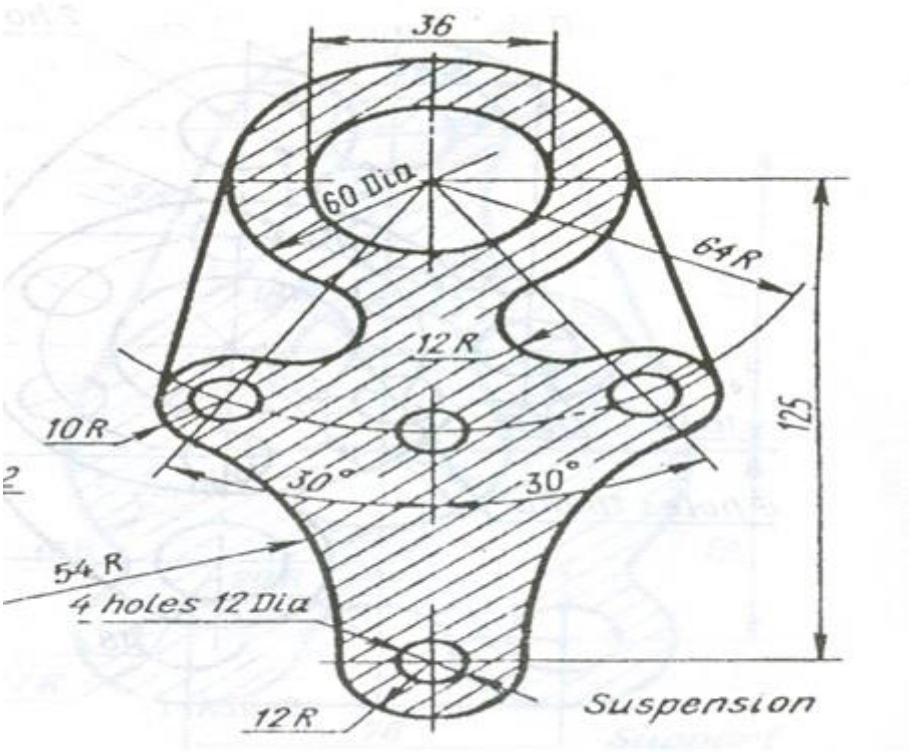
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

19. Limits,
1. Zoom,
2. Units,
3. Dim Style,
4. Line,
5. Circle,
6. Arc,
7. Polar Array,
8. Offset,
9. Fillet,
10. Mirror,
11. Trim,
12. Copy,
13. Move,
14. Hatch,
15. Layer,
16. Dimensions
17. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. By using draw tool bar commands and modify tool bar commands (line, circle, arc, fillet, offset) draw the given Practice Model-5 as per thedimensions.
5. By using the hatch command, hatch the sectioned portion as shown in the Model.
6. By using the Annotate tab, annotate the dimensions of the given Practice Model-5.
7. Save the finished drawing in the local disk using “practice model-5.dwg” file name and hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

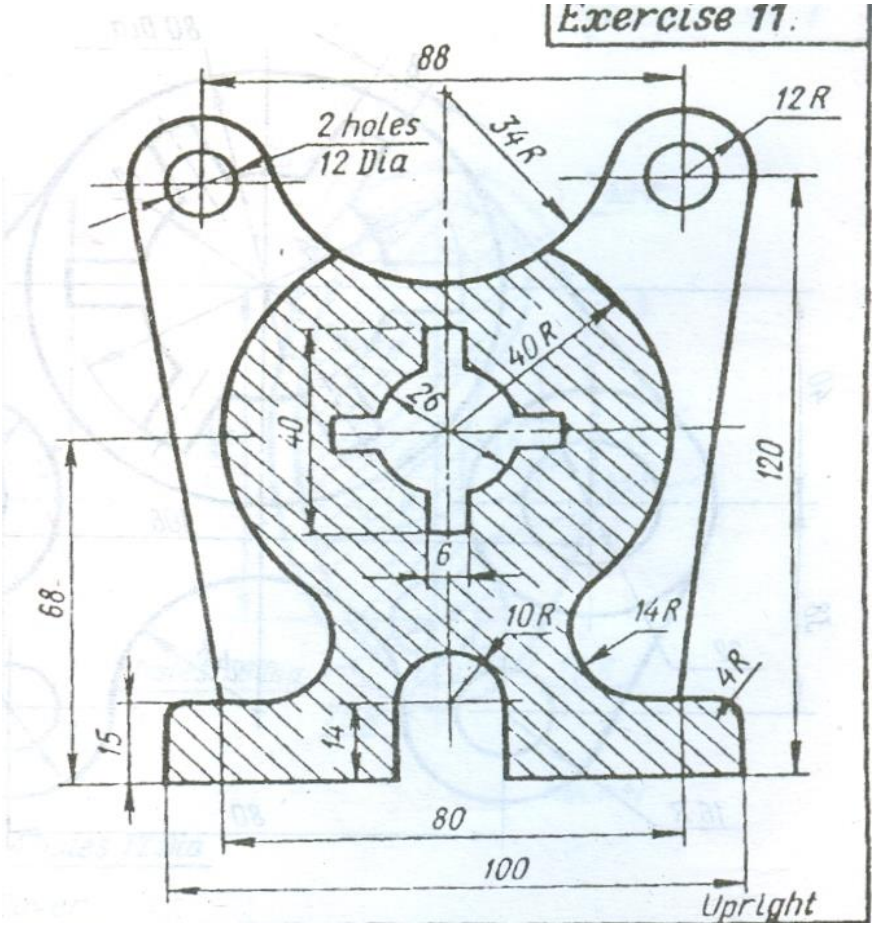
1. Set the ORTHO MODE to ON where ever necessary.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

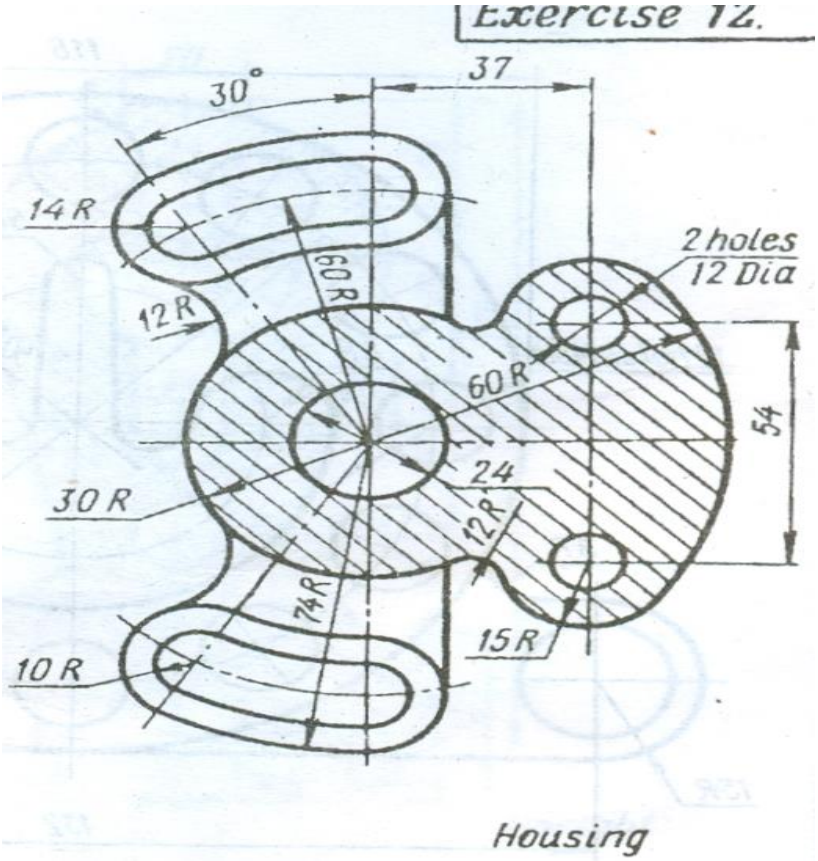
Thus the orthographic view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

PRACTICE MODELS:-

6.



7.



PART DRAWING

EXPNO: 1 **Date:**

3D VIEW OF V-BLOCK

AIM:

To draw the Isometric view of the V-Block using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

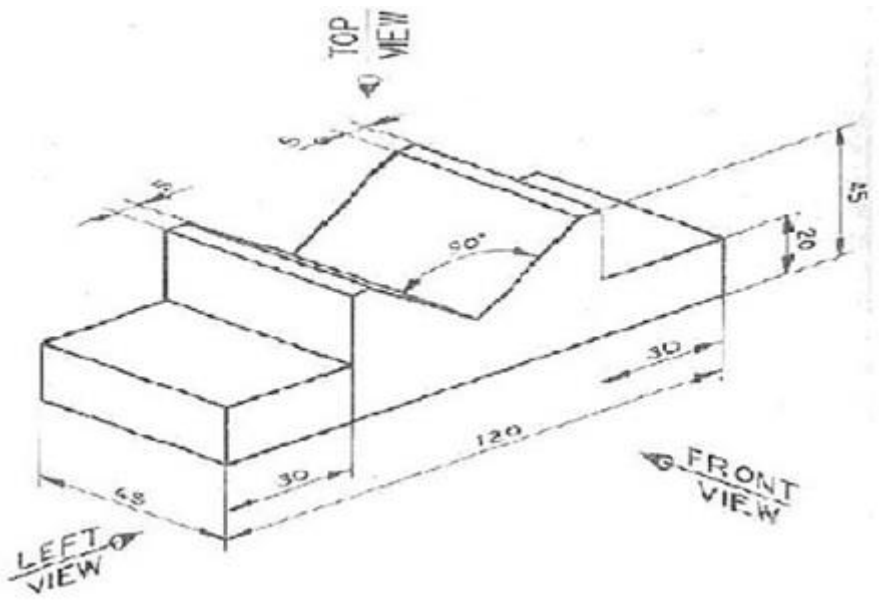
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using "v-block.dwg" file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 2 **Date:**

3D VIEW OF RECTANGULAR BLOCK

AIM:

To draw the Isometric view of the Rectangular Block using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

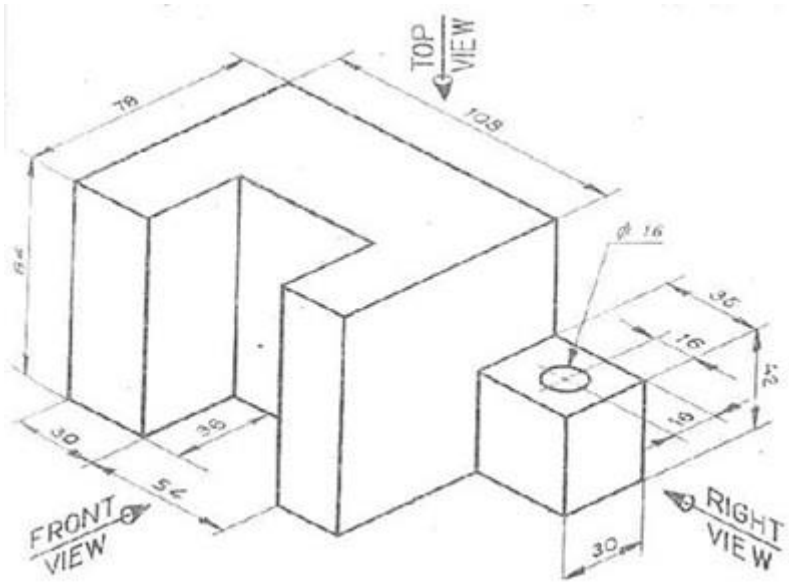
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using "rectangular block.dwg" file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 3 **Date:**

3D VIEW OF JOURNAL BLOCK

AIM:

To draw the Isometric view of the Journal Bearing using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

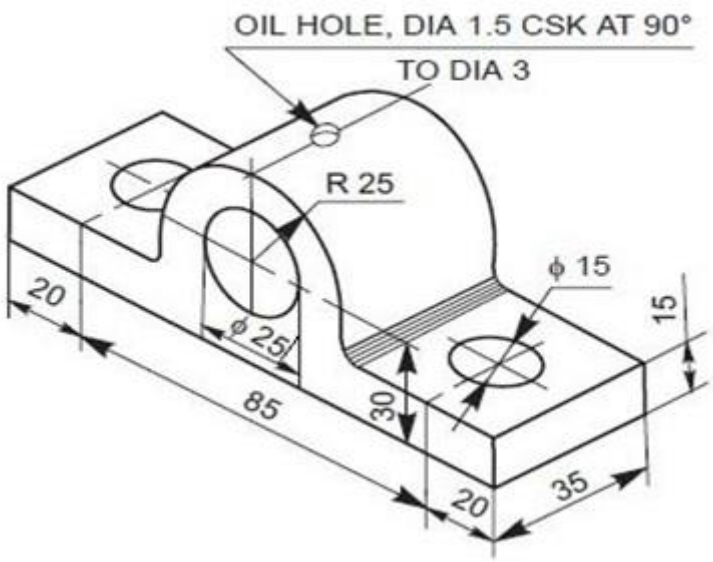
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using "journal bearing.dwg" file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 4 **Date:**

3D VIEW OF ANGULAR BLOCK

AIM:

To draw the Isometric view of the Angular Block using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

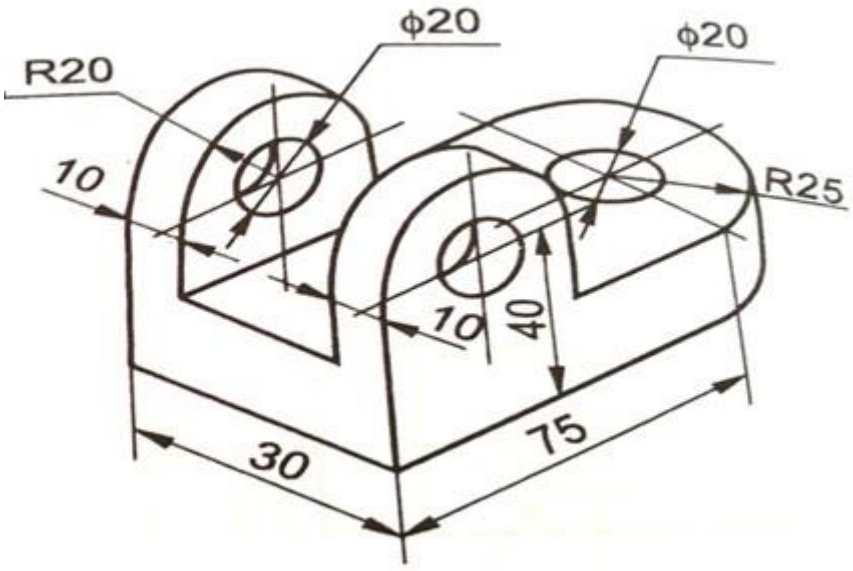
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using “angular block.dwg” file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 5 **Date:**

3D VIEW OF SUPPORT

AIM:

To draw the Isometric view of the Support using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

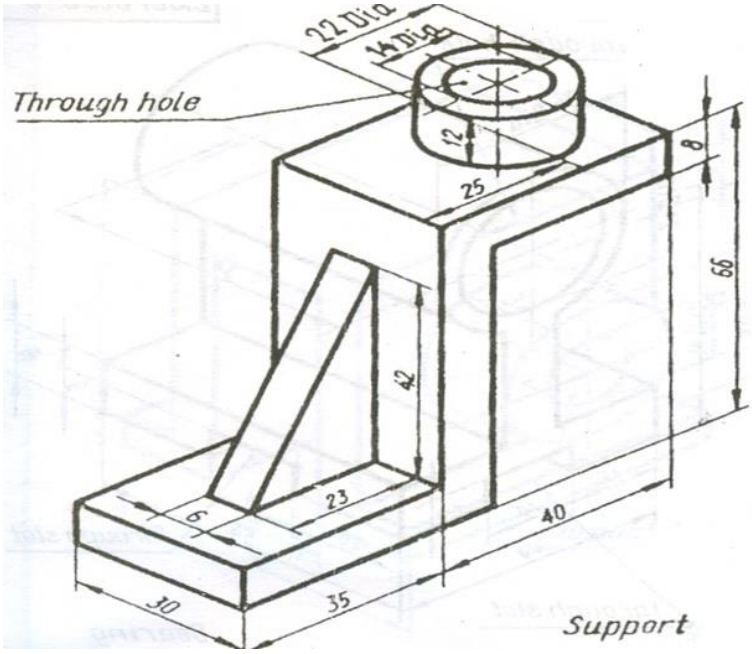
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using “support.dwg” file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 6 **Date:**

3D VIEW OF BODY

AIM:

To draw the Isometric view of the Body using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

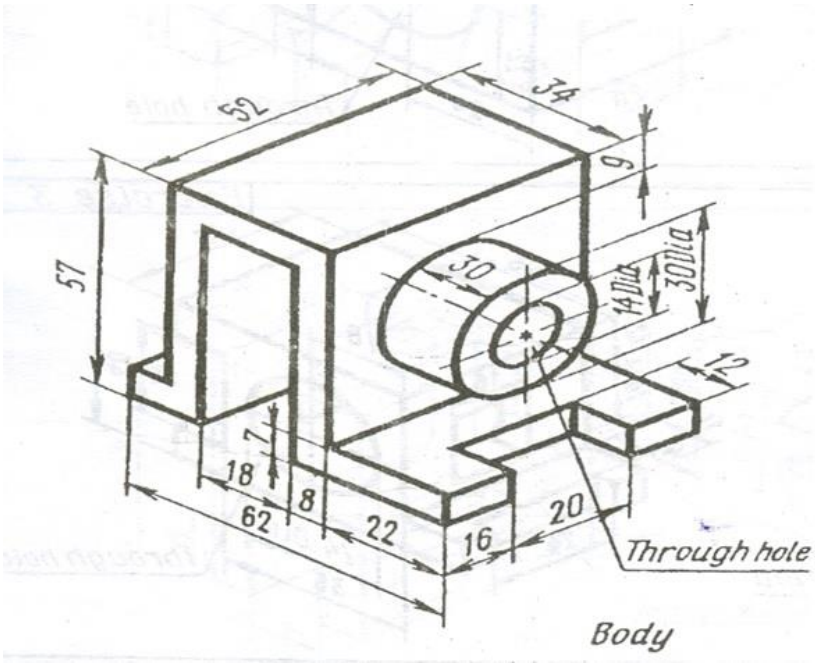
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using “body.dwg” file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

EXPNO: 7 **Date:**

3D VIEW OF SHOE

AIM:

To draw the Isometric view of the Shoe using AutoCAD software and to specify its dimensions.

SOFTWARE USED:

1. Operating system: Windows 7 Ultimate
2. AUTOCAD 2017

SYSTEM HARDWARE USED:

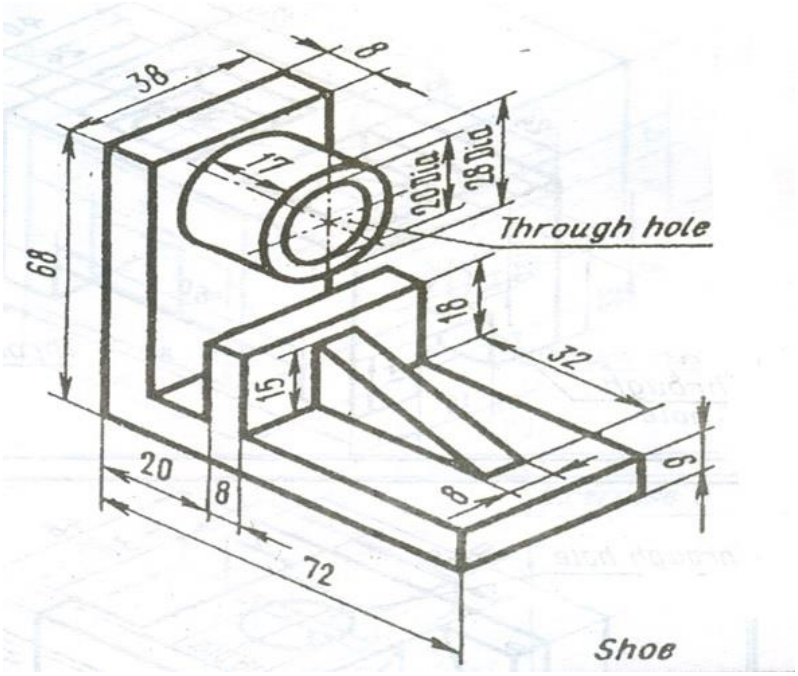
1. Intel ® Core I5 processor
2. 4GB RAM
3. 500GB Hard disk drive
4. Keyboard and Mouse
5. 15” Monitor and a Plotter

COMMANDS USED:

1. Limits,
2. Zoom,
3. Units,
4. Dim Style,
5. Polar Tracking (F10)
6. Iso Drafting (F5)
7. Line,
8. Ellipse – Iso circle,
9. Copy,
10. Trim,
11. Layer,
12. Dimensions,
13. Oblique,
14. Properties.

PROCEDURE:

1. Start the AutoCAD software and study the functional requirements of the component.
2. Set the appropriate limits to New AutoCAD file using LIMITS command.
3. Select the suitable UNITS for the entire drawing.
4. Set the Polar tracking (F10) and Isometric drafting (F5) to ON Mode.
5. By using draw tool bar commands and modify tool bar commands, the required views are drawn as per the dimensions by shifting the plane to iso-left, iso-right, and iso-top.
6. For drawing Iso circle, ellipse command is used.
7. By using the Annotate tab, annotate the dimensions of the given model to the oblique angle.
8. Save the finished model in the local disk using “shoe.dwg” file name and the hardcopy is printed as per the drawing sheet.



ALL DIMENSIONS ARE IN MM ONLY

PRECAUTIONS:

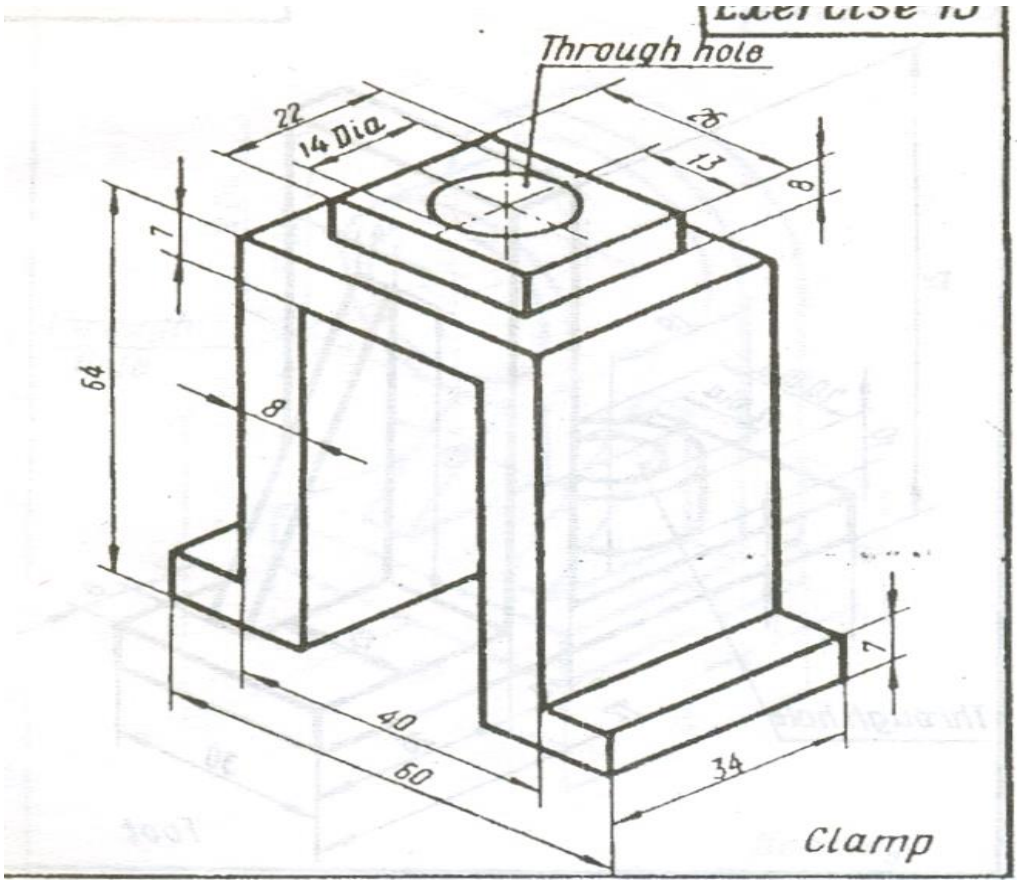
1. Set the Isometric Drafting (F10) & Polar Tracking (F5)to ON MODE.
2. Set the Units to Millimeters.
3. Use the modify tool bar commands where ever required.

RESULT:

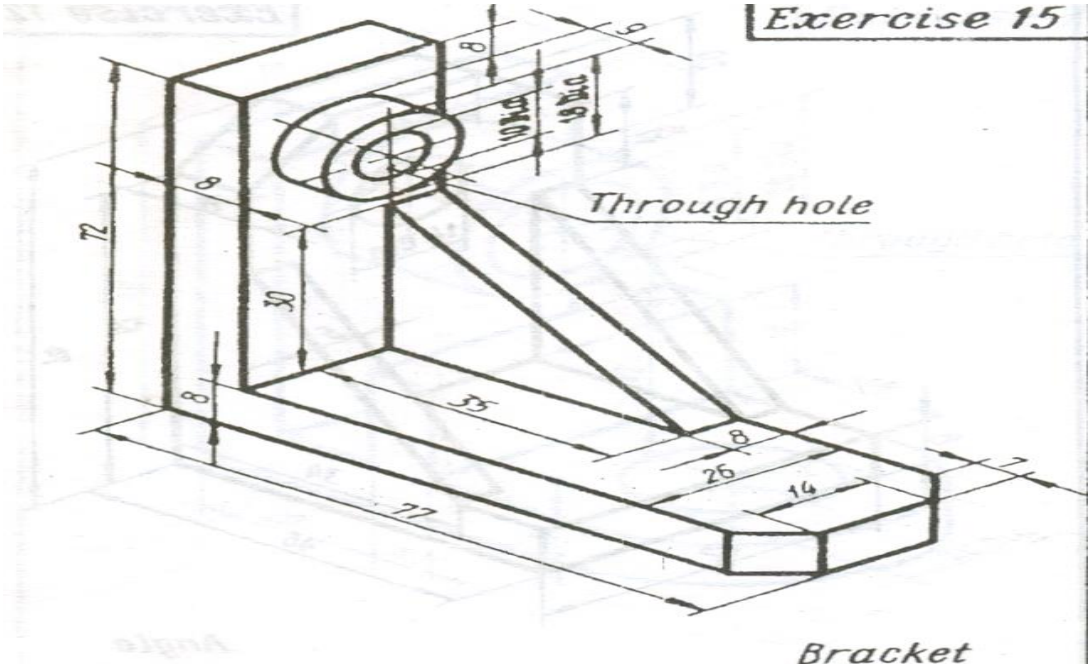
Thus the Isometric view of the given model is drawn using AUTOCAD 2017 Software as per the required dimensions successfully.

PRACTICE MODELS 3D DRAWING

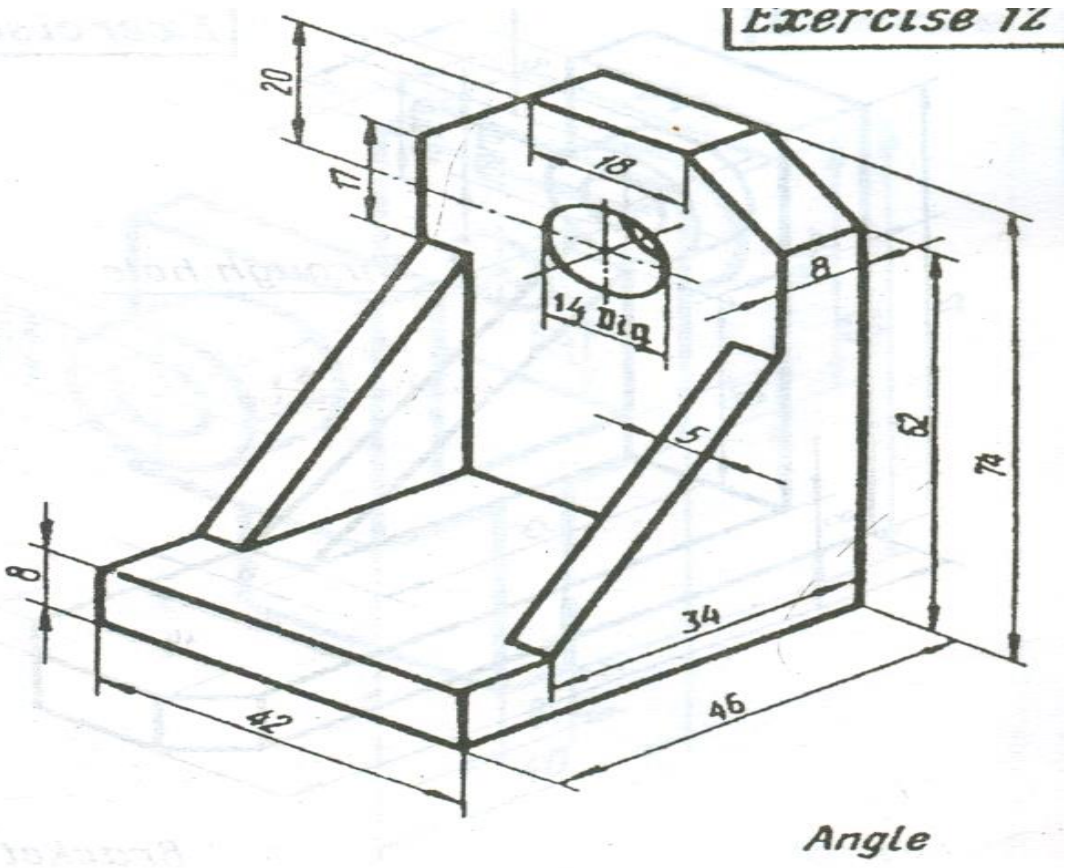
1.



2.

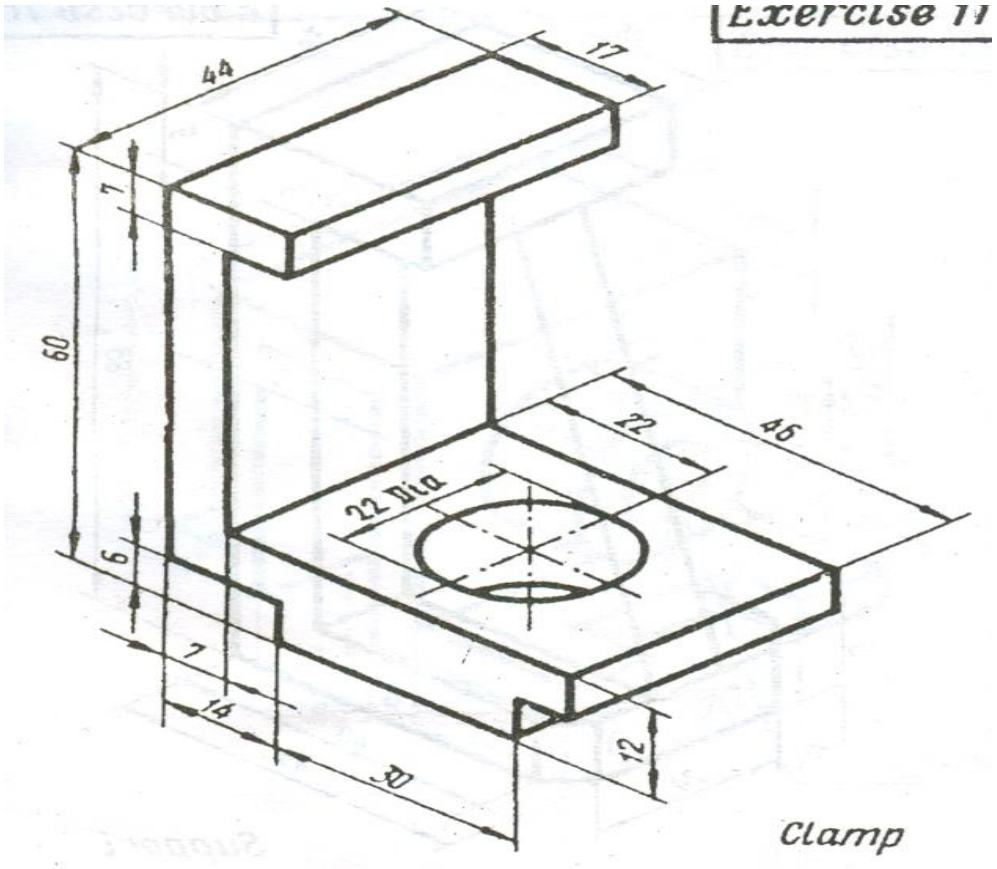


3.

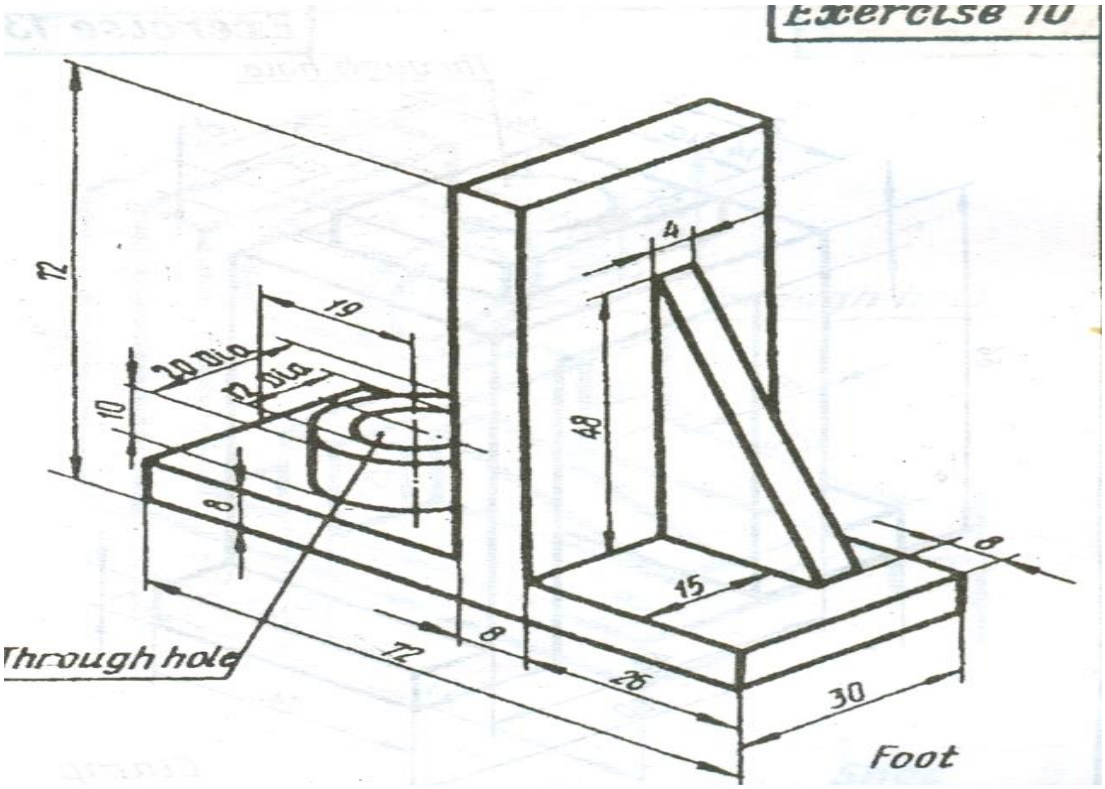

Exercise 12
20
17
18
14 Dia
8
5
62
74
8
34
42
46
Angle

4.

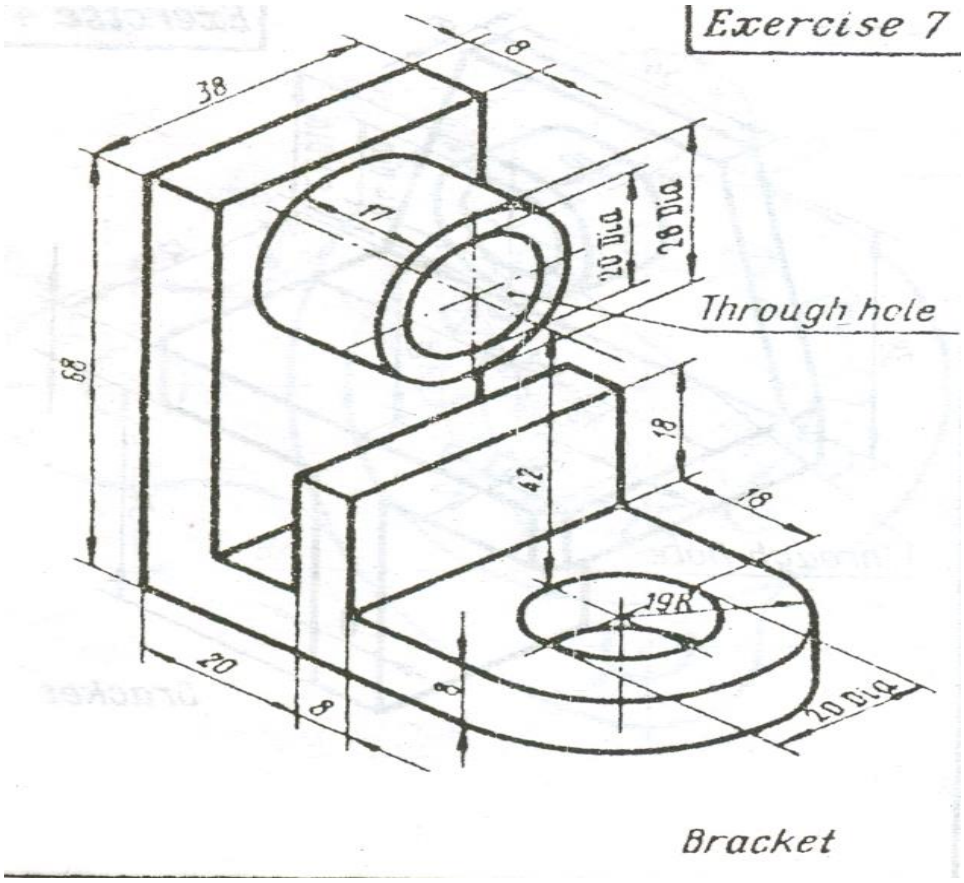
.


Exercise 11
44
17
7
60
22
46
22 Dia
6
7
14
30
12
Clamp

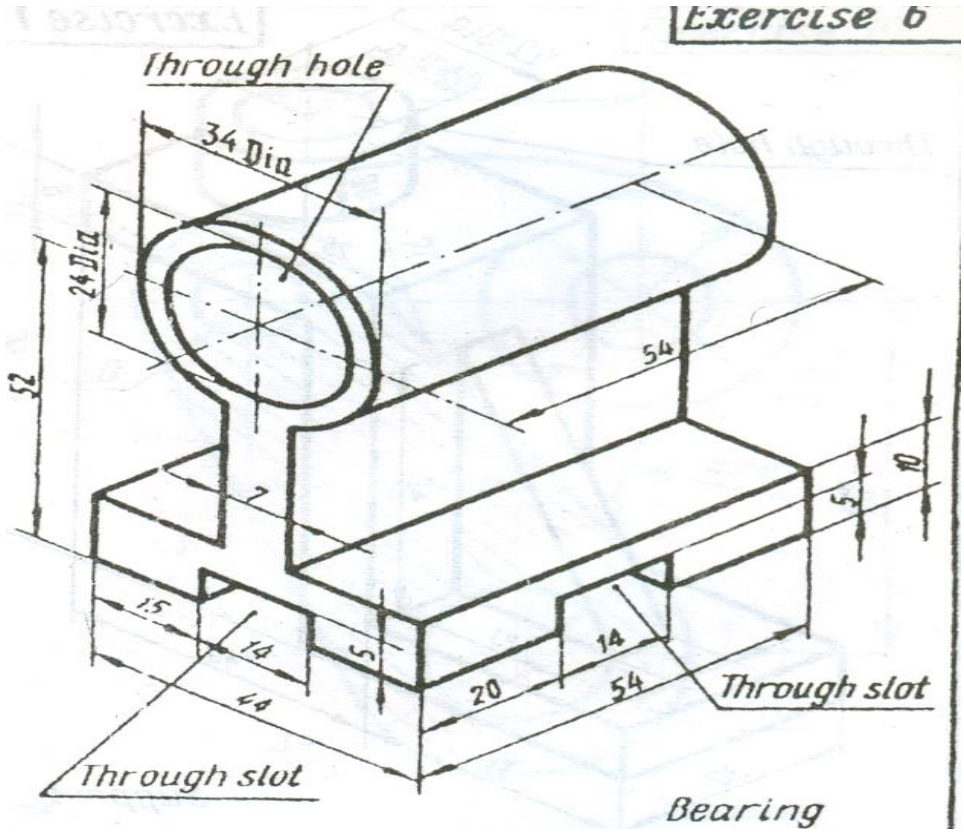
5.


Exercise 10
72
19
20 Dia
12 Dia
10
8
4
48
15
8
8
72
25
30
Through hole
Foot

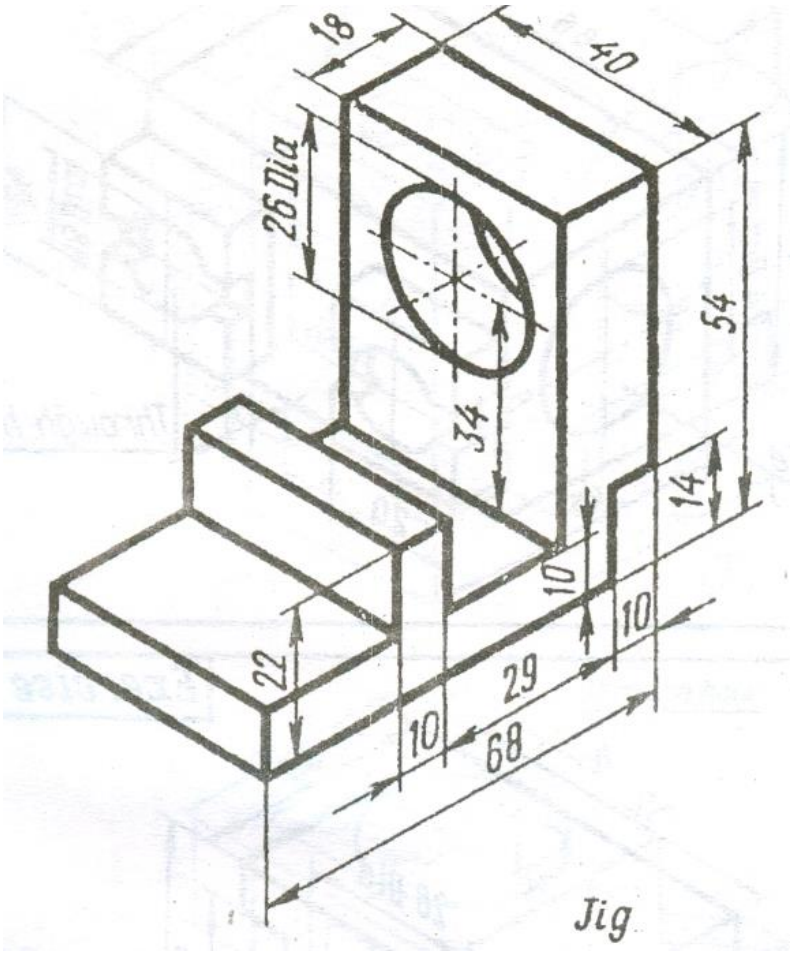
6.


Exercise 7
8
38
17
10 Dia
28 Dia
Through hole
68
18
42
18
19R
20
8
8
20 Dia
Bracket

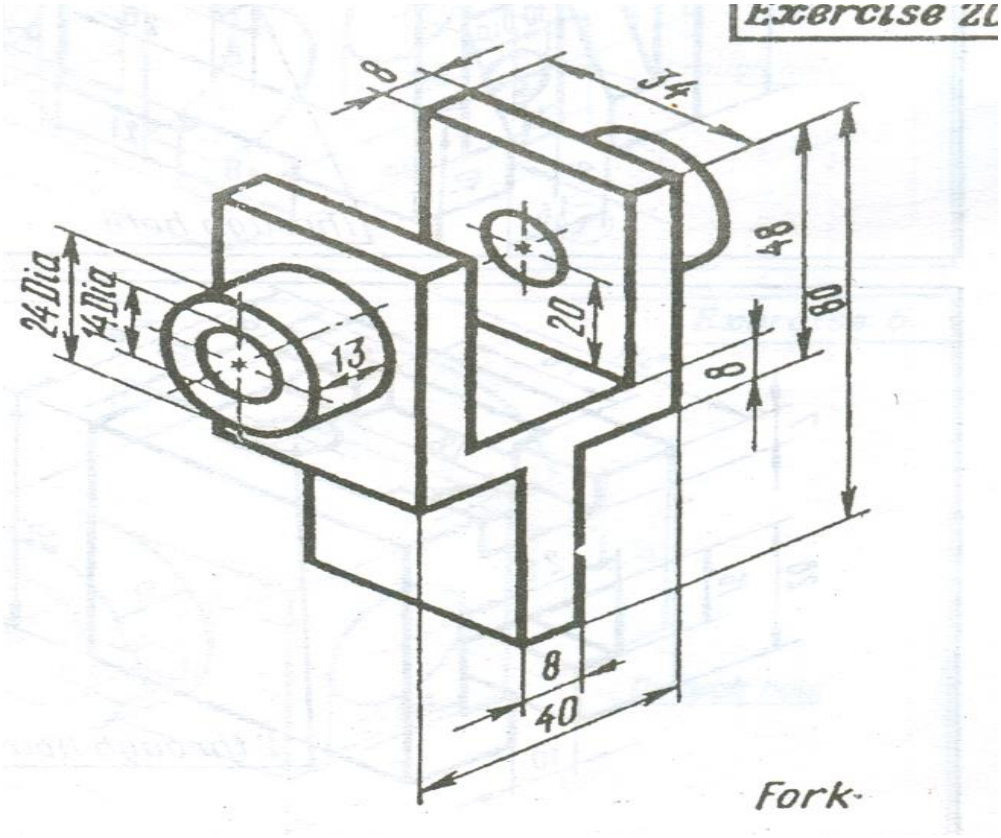
7.


Exercise 6
Through hole
34 Dia
24 Dia
52
54
7
5
10
15
14
5
14
20
54
44
Through slot
Through slot
Bearing

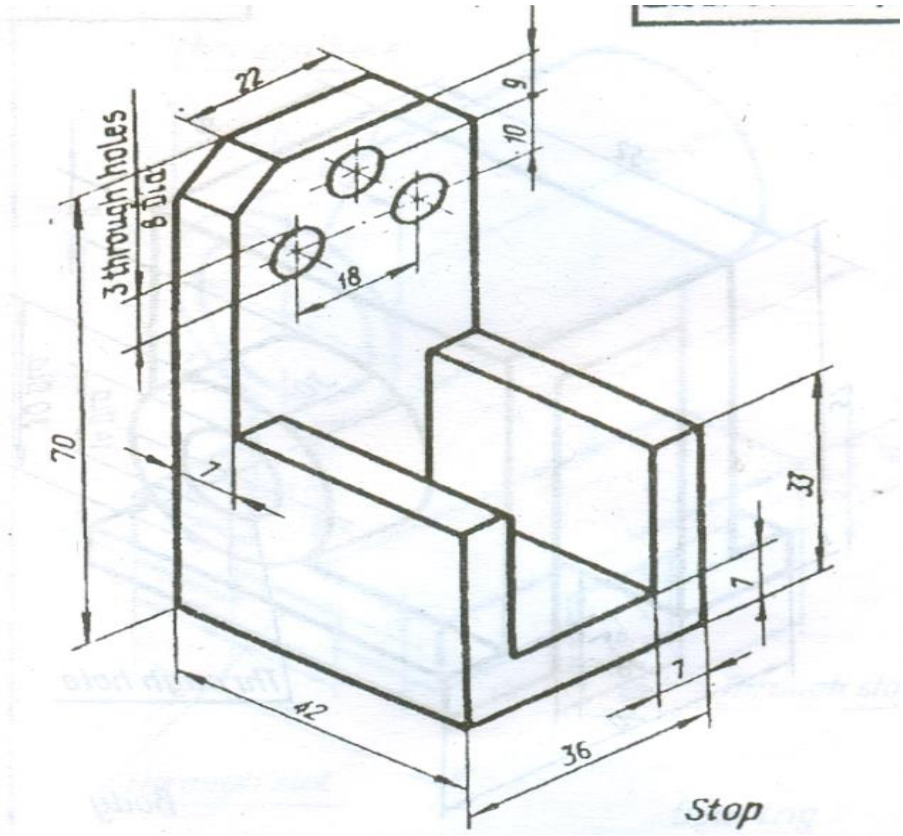
8.


18
40
26 Dia
54
34
14
10
10
22
29
10
68
Jig

9.


Exercise 2
8
34
24 Dia
14 Dia
13
20
48
80
8
8
40
Fork

10.


22
9
10
3 through holes
8 Dia
18
70
7
33
7
7
42
36
Stop

11.