DRAUGHTSMAN MECHANICAL

NSQF LEVEL - 4

2nd Year

TRADE PRACTICAL

SECTOR: CAPITAL GOODS & MANUFACTURING

(As per revised syllabus July 2022 - 1200Hrs)

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 : **Capital Goods & Manufacturing**

Duration : **2 Year**

Trade : **Draughtsman Mechanical - 2nd year Trade Practical - NSQF Level - 4 (Revised 2022)**

Developed & Printed by

National Instructional Media Institute
Post Box No.3142
Guindy, Chennai - 600032
INDIA
Email: chennai-nimi@nic.in
Website: www.nimi.gov.in

First Edition : December 2023 Copies : 1000

Rs.190 /-

FOREWORD

The Government of India has set an ambitious target of imparting skills to 30 crores people, one out of every four Indians, to help them secure jobs as part of the National Skills Development Policy. Industrial Training Institutes (ITIs) play a vital role in this process especially in terms of providing skilled manpower. Keeping this in mind, and for providing the current industry relevant skill training to Trainees, ITI syllabus has been recently updated with the help of Media Development Committee members of various stakeholders viz. Industries, Entrepreneurs, Academicians and representatives from ITIs.

The National Instructional Media Institute (NIMI), Chennai, has now come up with instructional material to suit the revised curriculum for **Draughtsman Mechanical 2nd Year Trade Practical** in **CG & M Sector** under **Yearly Pattern.** The NSQF Level - 4 (Revised 2022) Trade Practical will help the trainees to get an international equivalency standard where their skill proficiency and competency will be duly recognized across the globe and this will also increase the scope of recognition of prior learning. NSQF Level - 4 (Revised 2022) trainees will also get the opportunities to promote life long learning and skill development. I have no doubt that with NSQF Level - 4 (Revised 2022) the trainers and trainees of ITIs, and all stakeholders will derive maximum benefits from these Instructional Media Packages IMPs and that NIMI's effort will go a long way in improving the quality of Vocational training in the country.

The Director General, Executive Director & Staff of NIMI and members of Media Development Committee deserve appreciation for their contribution in bringing out this publication.

Jai Hind

ATUL KUMAR TIWARI, ***I.A.S***
Secretary
Ministry of Skill Development & Entrepreneurship,
Government of India.

December 2023
New Delhi - 110 001

PREFACE

The National Instructional Media Institute (NIMI) was established in 1986 at Chennai by then Directorate General of Employment and Training (D.G.E & T), Ministry of Labour and Employment, (now under Ministry of Skill Development and Entrepreneurship) 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 provide instructional materials for various trades as per the prescribed syllabus under the Craftsman and Apprenticeship Training Schemes.

The instructional materials are created keeping in mind, the main objective of Vocational Training under NCVT/NAC in India, which is to help an individual to master skills to do a job. The instructional materials are generated in the form of Instructional Media Packages (IMPs). An IMP consists of Theory book, Practical book, Test and Assignment book, Instructor Guide, Audio Visual Aid (Wall charts and Transparencies) and other support materials.

The trade practical book consists of series of exercises to be completed by the trainees in the workshop. These exercises are designed to ensure that all the skills in the prescribed syllabus are covered. The trade theory book provides related theoretical knowledge required to enable the trainee to do a job. The test and assignments will enable the instructor to give assignments for the evaluation of the performance of a trainee. The wall charts and transparencies are unique, as they not only help the instructor to effectively present a topic but also help him to assess the trainee's understanding. The instructor guide enables the instructor to plan his schedule of instruction, plan the raw material requirements, day to day lessons and demonstrations.

In order to perform the skills in a productive manner instructional videos are embedded in QR code of the exercise in this instructional material so as to integrate the skill learning with the procedural practical steps given in the exercise. The instructional videos will improve the quality of standard on practical training and will motivate the trainees to focus and perform the skill seamlessly.

IMPs also deals with the complex skills required to be developed for effective team work. Necessary care has also been taken to include important skill areas of allied trades as prescribed in the syllabus.

The availability of a complete Instructional Media Package in an institute helps both the trainer and management to impart effective training.

The IMPs are the outcome of collective efforts of the staff members of NIMI and the members of the Media Development Committees specially drawn from Public and Private sector industries, various training institutes under the Directorate General of Training (DGT), Government and Private ITIs.

NIMI would like to take this opportunity to convey sincere thanks to the Directors of Employment & Training of various State Governments, Training Departments of Industries both in the Public and Private sectors, Officers of DGT and DGT field institutes, proof readers, individual media developers and coordinators, but for whose active support NIMI would not have been able to bring out this materials.

Chennai - 600 032 **EXECUTIVE DIRECTOR**

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 organisations to bring out this Instructional Material **(Trade Practical)** for the trade of **Draughtsman Mechanical 2nd Year NSQF Level - 4 (Revised 2022)** under **Capital Goods & Manufacturing** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. V. Dhivakar	-	ATO, Govt ITI , Guindy, Tamil nadu.
Shri. C.S. Kumar	-	ATO, Govt ITI, North Chennai.
Shri. T. Chithra	-	ATO, Govt ITI, Cuddalore.
Shri. C.S. Nagarathinam	-	ATO, Govt ITI, Ulundurpet.
Shri. M.A. Mangaleshwaran	-	JTO, Govt ITI, Ramanadhapuram.
Shri. V. Dhanasekaran	-	ADT,(Retd) Mdc member, Nimi Chennai.
Shri. V. Janarthanan	-	Asst prof.(Retd) Mdc member, Nimi Chennai.

NIMI CO-ORDINATORS

Shri.Nirmalya Nath	-	Deputy Director of Training, NIMI- Chennai - 32.
Shri. V. Gopalakrishnan	-	Manager, NMI, Chennai - 32

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 intended to be used in practical workshop. It consists of a series of practical exercises to be completed by the trainees during the course. These exercises are designed to ensure that all the skills in compliance with NSQF Level - 4 (Revised 2022) syllabus are covered.

The manual is divided into Nine modules

Module 1	**Computer aided drafting practice**
Module 2	**Types of Pulleys**
Module 3	**Pipe joints**
Module 4	**Gears and cams**
Module 5	**IC Engine parts and Assembly**
Module 6	**3D Solid objects**
Module 7	**Detailed and assemble drawing**
Module 8	**Solid works**
Module 9	**Production drawing**

The skill training in the shop floor is planned through a series of practical exercises centered around some practical project. However, there are few instances 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 trainee. However the development team accept that there is a scope for further improvement. 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 Course of the **Draughtsman Mechanical 2nd Year NSQF Level - 4 (Revised 2022)** in **CG & M**. The contents are sequenced according to the practical exercise contained in NSQF Level - 4 (Revised 2022) syllabus on Trade Theory attempt has been made to relate the theoretical aspects with the skill covered in each exercise to the extent possible. This correlation is maintained to help the trainees to develop 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 indications about the corresponding practical exercises are given in every sheet of this manual.

It will be preferable to teach/learn trade theory connected to each exercise at least 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 for the purpose of self-learning and should be considered as supplementary to class room instruction.

CONTENTS

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

S.No	Learning Outcome	Ref. Ex.No.
1	Construct projection views of geometrical figures with dimension and annotation on CAD in model space and viewport in layout space. (CSC/NO402)	2.1.94 - 2.1.101
2	Draw in CAD detail and assembly drawing of machine parts viz., Pulleys, Pipe fittings, Gears and Cams applying range of cognitive and practical skills. (CSC/NO402)	2.2.102 - 2.2.112
3	Construct drawing of engine parts with detailed and assembly in template layout applying quality concept in CAD. (CSC/NO402)	2.3.113 - 2.3.115
4	Create 3D solid by switching to 3D modeling workspace in CAD, generate views, Print Preview and Plotting. (CSC/NO402)	2.4.116
5	Construct detailed and assembled drawing applying conventional sign & symbolsusing CAD. (CSC/NO402)	2.5.117 - 2.4.132
6	Prepare drawing of machinepart by measuring with gauges and measuring instruments. (CSC/NO402)	2.6.133
7	Draw a machine shop layout considering process path and ergonomics (human factor). (CSC/NO402)	2.7.134
8	Create and plot assembly and detail views of machine part with Dimensions, Annotations, Title Block and Bill of materials in SolidWorks/AutoCAD Inventor/ 3D Modeling. (CSC/NO402)	2.8.135 - 2.8.143
9	Create production drawing of machine part. (CSC/NO402)	2.9.144 - 2.9.146

SYLLABUS

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 110 Hrs; Professional Knowledge 34 Hrs	Construct projection views of geometrical figures with dimension and annotation on CAD in model space and viewport in layout space. (Mapped NOS: CSC/NO402)	94. CAD: draw 2D object using line, polyline, ray, polygon, circle, rectangle, arc, ellipse commands. (20 hrs)	Drawing of Line, polyline, ray, polygon, circle, rectangle, arc, ellipse using different options. (07 hrs.)
		95. CAD: modify 2D objects using Break, Erase, Trim, Offset, Fillet, Chamfer Commands. (10 hrs) 96. CAD: manage 2D objects using Move, Copy, Array, Insert Block, Make Block, Scale, Rotate, Hatch Commands. (12 hrs)	Trim, Offset, Fillet, Chamfer, Arc and Circle under modify commands. Move, Copy, Array, Insert Block, Make Block, Scale, Rotate, Hatch Commands. (07 hrs.)
		97. CAD: Create templates, Insert drawings. Create objects in different Layers and Modify Layer properties. (20 hrs)	Creating templates, Inserting drawings, Layers, Modify Layers. (07 hrs.)
		98. CAD: Provide dimension on object. Create dimension by customizing dimension styles (lines, arrows, text, unit and alignment) Put dimension with scale factor. (20 hrs)	Format dimension style, creating new dimension style, Modifying styles in dimensioning. Writing text on dimension line and on leader. Edit text dimension. (07 hrs.)
		99. CAD: Construct orthographic sectional view of a steel bracket with dimension using shortcut keyboard command.(10 hrs) 100. Construct isometric view of machine blocks. (10 hrs) 101. Create viewports in layout space and place views for model space in different scale. (08 hrs)	Knowledge of shortcut keyboard command. Customization of keyboard command. Customization of drafting settings, changing orthographic snap to isometric snap. Procedure to create viewport in layout space in zooming scale. (06 hrs.)
Professional Skill 140Hrs; Professional Knowledge 50 Hrs	Draw in CAD detail and assembly Drawing of machine parts viz., Pulleys, Pipe fittings, Gears and Cams applying range of cognitive and practical skills. (Mapped NOS: CSC/NO402)	102. Construct Pulleys: solid, stepped and built up pulleys. (10 hrs) 103. Construct pulley with different types of arms. (10 hrs) 104. Draw rope pulley and v-belt pulley using CAD. (10 hrs)	Belt-drive. Materials of belts, slip and creep, Velocity of belt. Arc of contact. Simple exercise in calculation of belt speeds, nos. of belts needed in V-belt drive, velocity, pulley ratio etc. Standard pulleys width of pulley face, velocity ratio chain drive. (07 hrs.)
		105. Draw pipe fittings: tee, elbow (90° & 45°), flange, union and valve. (10 hrs) 106. Draw conventional symbols of different types of valves and joints used in pipe line diagram. (10 hrs) 107. Draw a piping layout systems from a sump to an overhead tank through a pump with possible fittings and valves. (10 hrs)	Knowledge of different pipe materials and specifications of Steel, W.I. & PVC pipes. Brief description of different types of pipe joints. Pipe threads. Pipe fittings (threaded, welded and pressed). Specifications of pipe fittings.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		108. Draw sectional views of different types of pipe joints using CAD. (10 hrs)	Different types of valves. (14 hrs.)
		109. Draw: i) spur gear, (08 hrs) ii) helical gear, (08 hrs) iii) bevel gear, (08 hrs) iv) worm and worm wheel. (08 hrs) 110. Construct involute tooth profile of a gear (using CAD). (08 hrs)	Gear drive- Different types of gears. Cast gears and machined gears. Knowledge of profile of gears etc. (14 hrs.)
		111. Draw a symmetrical cam profile. (15 hrs) 112. Draw different types of follower (using CAD).(15 hrs)	Use of Cams in industry. Types of cam, kinds of motion in cam, displacement diagrams. Terms used in cam. Types of follower. (15 hrs.)
Professional Skill 110Hrs; Professional Knowledge 35 Hrs	Construct drawing of engine parts with detailed and assembly in template layout applying quality concept in CAD. (Mapped NOS: CSC/NO402)	113. Construct detailed and assembly drawing (using CAD) of i) Eccentrics (10 hrs), ii) Stuffing box (12 hrs) iii) Piston assembly of a petrol engine (20 hrs), iv) IC engine connecting rod. (20 hrs)	Knowledge of engine mechanism. Transmission of motion from reciprocating to circular through eccentric, crank and connecting rod. (21 hrs.)
		114. Construct detailed drawing of an air valve. (28 hrs) 115. Construct detailed drawing of a fuel injector of a diesel engine. (20 hrs) (using CAD)	Knowledge of fuel injection system in petrol and diesel engine. (14 hrs.)
Professional Skill 46Hrs; Professional Knowledge 12Hrs	Create 3D solid by switching to 3D modeling workspace in CAD, generate views, Print Preview and Plotting. (Mapped NOS: CSC/NO402)	116. 3D Modeling: i) Create 3D solid objects using command from 3D primitive (viz. box, sphere, cylinder and poly-solids) , from solid (extrude, revolve, sweep and loft), from Boolean (union, subtract and intersect) (20 hrs) ii) Create 3D drawing using User co-ordinate systems. (13 hrs) iii) Annotate and dimension of the 3D model. (05 hrs) iv) Generate views from model space to layout space. (05 hrs) v) Generate Print preview and Plotting. (03 hrs)	Introduction to 3D modeling, 3D primitives (viz. box, sphere, cylinder, mesh and poly-solids), solid figure by extrude, revolve, sweep and loft command, solid editing: fillet, offset, taper, shell and slice command. Setting of User co-ordinate Systems, Rotating, Print preview and Plotting. (12 hrs)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 260 Hrs; Professional Knowledge 90 Hrs	Construct detailed and assembled drawing applying conventional sign & symbols using CAD. (Mapped NOS: CSC/NO402)	117. Construct detailed drawing of a lever safety valve.(20 hrs) 118. Construct detailed drawing of a gate valve.(20 hrs)(using CAD)	Working principle of valves and their description. (13 hrs.)
		119. Construct detailed drawing of a steam stop valve and blow off cock. (20 hrs) (using CAD)	Knowledge of simple stationary fire tube boiler, boiler mountings. Function and purpose of blow off cock. (07 hrs.)
		120.Create library folder containingblocks of hydraulic and pneumatic conventional signs and symbols. (10 hrs) 121. Draw a sectional view of a hydraulic jack and a pneumatic valve actuator. (10 hrs)(using CAD)	Brief description of a typical hydraulic system, components, working principle and function of hydraulic jack. Different types of hydraulic actuator. Symbol and working of hydraulic DC valve, non-return valve and throttle valve. Knowledge of typical pneumatic system, FRL or air service unit and pneumatic actuator. (07 hrs.)
		122. Draw detail and full sectional view of a volute casing centrifugal pump(using CAD). (20 hrs)	Different types of pump systems.Characteristics of a pump system: pressure, friction and flow.Energy and head in pump systems. (07 hrs.)
		123. Draw assembly and detailed drawing of tool post of a lathe. (using CAD) (20 hrs)	Different clamping devices on lathe. (07 hrs.)
		124. Construct detailed &assembly drawing of tail stock and revolving centre. (using CAD) (20 hrs)	Description of different job holding devices in lathe operation. (07 hrs.)
		125. Construct detailed drawing of a milling fixture. (using CAD) (20 hrs)	Different clamping devices on milling operation. (07 hrs.)
		126. Construct detailed & assembly drawing of shaper tool head slide. (using CAD) (20 hrs)	Different clamping devices on shaping operation. (07 hrs.)
		127. Draw a simple drilling jig for drilling holes in a given component. (using CAD) (20 hrs)	Knowledge of accuracy and interchangeabilityinthe manufacturing of products. (07 hrs.)
		128. Draw a Press Tool giving nomenclature of each part. (08 hrs) 129. Draw dies & punches for the production of simple work pieces. (using CAD) (06 hrs) 130. Develop isometric drawing for manufacturing 2 cavity injection moulds with side cavities. (using CAD)(06 hrs)	Knowledge of various parts of press tools and their function. Knowledge of different moulding processes. Introduction to Die casting, gating system design, force calculation, defects and remedies and estimation. (07 hrs.)
		131. Construct detailed drawing of a simple carburetor.(using CAD) (20 hrs)	Description of different parts of petrol engine. (07 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		132. Construct detailed and assembly drawing of a simple pressure vessel. (using CAD) (20 hrs)	Knowledge of design, manufacture, and operation of pressure vessels. (07 hrs.)
Professional Skill 20Hrs; Professional Knowledge 08Hrs	Prepare drawing of machineparts by measuring with gauges and measuring instruments. (Mapped NOS: CSC/NO402)	133. Prepare detailed drawing of a C-clamp and a machine vice by taking measurement using gauges and measuring instrument. (using CAD) (20 hrs)	Proper measurement practice in workshop. Principles of good measurement result: right measurement, right tools, right sketching, review and right procedures.(08 hrs.)
Professional Skill 20Hrs; Professional Knowledge 06Hrs	Draw a machine shop layout considering process path and ergonomics (human factor). (Mapped NOS: CSC/NO402)	134. Draw a machine shop layout of small production industry showing material inflow to finished product stock. (using CAD) (20 hrs)	Lay out of Machine foundations. Brief treatment of the principle Involved and the precautions to be observed. Lay out of machine Foundation. Consideration of ergonomics (human factor) for shop layout. (06 hrs.)
Professional Skill 110 Hrs; Professional Knowledge 35 Hrs	Create and plot assembly and detail views of machine part with Dimensions, Annotations, Title Block and Bill of materials in SolidWorks/AutoCAD Inventor/ 3D Modeling. (Mapped NOS: CSC/NO402)	**SolidWorks/AutoCAD Inventor/ 3D Modeling:** 135. Draw 3D solid figures by Sketching features & applied features. (08 hrs) 136. Sketch an angle plate and a block – Create/ Modify constraints. (06 hrs) 137. Create a sketch of a new part. (08 hrs)	Introduction to SolidWorks/ AutoCAD Inventor/ 3D Modeling User interface - Menu Bar – Command manager – Feature manager – Design Tree – settings on the Default options – suggested settings – key board short cuts. Create the best profile – create a sketch – create a new part. (07 hrs.)
		138. Create 3D solid and edit using: i) Copy & Paste, (03 hrs) ii) Filleting, (03 hrs) iii) Chamfering, (03 hrs) iv) Editing a feature definition. (03 hrs) v) Create ribs, mirror pattern, the Hole wizard, (03 hrs) vi) Create part configurations, Part design tables, (03 hrs) vii) Inset Design Table, Inset new design table. (03 hrs)	Extrude bosses and cuts, add fillets, and chamfer changing dimensions. Revolved features using axes, circular patterning changes and Rebuild problems. (07 hrs.)
		139. Create New assembly part: i) Create a new assembly (06 hrs) ii) Insert components into an assembly, (03 hrs) iii) Add mates (degree of freedom). (03 hrs) iv) Perform components configuration in an assembly, (03 hrs)	Bottom up assembly modeling Components configuration in an assembly, Insert subassemblies, Interference detection. (07 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		v) Insert subassemblies, (03 hrs) vi) Perform Interference detection. (03 hrs)	
		140. Create a 3D model putting: i) Driving dimensions, (02 hrs) ii) Bill of materials, (02 hrs) iii) Driven (Reference) Dimensions, (02 hrs) iv) Annotations, (02 hrs) v) Alternate position view. (02 hrs) 141. Prepare drawings & detailing: i) Create drawing sheets, (02 hrs) ii) Add drawing items, (02 hrs) iii) Named views, standard 3 views, auxiliary views, section views, detail views. (02 hrs) iv) Reattach and replace dimensions, (02 hrs) v) Edit sketch, (02 hrs) vi) Edit sketch plane, (02 hrs) vii) Edit definition. (02 hrs)	Drawings & Detailing, create drawing sheets, Add drawing items, Named views, std. 3 views, auxiliary views, section views, detail views. Drawings & Detailing, create drawing sheets, Add drawing items, Named views, standard 3 views, auxiliary views, section views, detail views. (07 hrs.)
		142. Create a 3D transition figure • using loft feature. (03 hrs) • using sweep feature. (03 hrs) • using library features.(03 hrs) i) Create 3D model by annotating Holes and Threads, (03 hrs) ii) Create Centerlines, symbols and leaders, (03 hrs) iii) Create Simulation. (03 hrs) iv) Plot the model. (01 hr) 143. Convert or save as Solid Works and Inventor file into .dwg format. (03 hrs)	Difference between sweep and loft. Exploded views – Configuration manager, Animation controller. Annotating Holes and Threads, Creating Centerlines, symbols and leaders, Simulation. Introduction to plot & Different ways of plotting. (07 hrs.)
Professional Skill 24 Hrs; Professional Knowledge 06 Hrs	Create production drawing of machine part. (Mapped NOS: CSC/NO402)	144. Create production drawing of a simple Drill jig – Part model – assembly-detailing (using CAD). (12 hrs) 145. Create production drawing of a Screw jack – Part model – assembly-detailing. (10 hrs) (using CAD) 146. Create a check list by self-assessment and provide Revision mark by noting in the Revision table. (02 hrs)	Knowledgeof production drawing, name plate and bill of materials, etc. Study of production drawing. Procedure of preparing Revision Drawing: putting revision mark, writing remarks in the table as per check list. (06 hrs.)

Drawing 2D object using line, polyline, regular polygon, Circle rectangle, arc ellipse commands

Objectives: At the end of this exercise you shall be able to
- **draw different objects using Auto CAD commands like, line, poly line, rectangle, arc, circle, Ellipse polygons.**

Requirements

Tools/Equipment/Machines

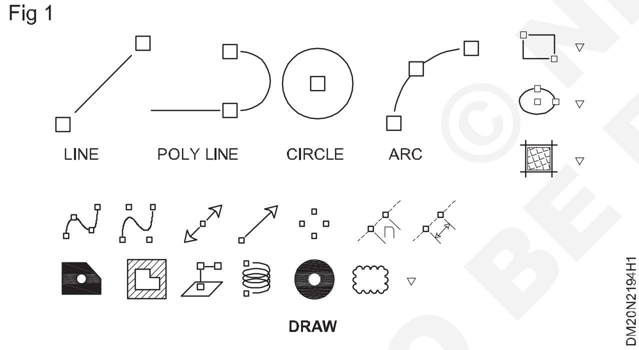
- Auto CAD 2018 or Higher version.

PROCEDURE

1 Draw tool bar:

The Draw commands can be used to create new objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components. A good understanding of the Draw commands is fundamental to the efficient use of Auto CAD.(Fig 1)

Fig 1



2 Drawing Entity-LINE:

Lines can be drawn by anyone of the following three methods using LINE command.

a Using Absolute Co-ordinates:

Example:

Draw a line from point (5, 5) to point (10, 10).

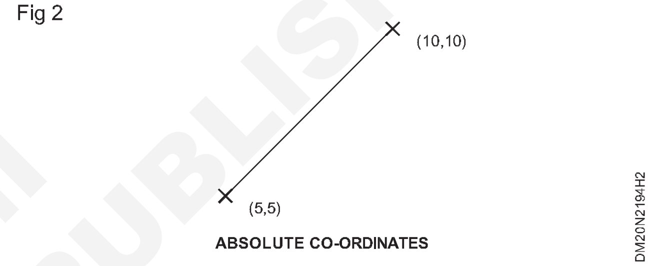
Command: LINE

From point: 5, 5 (Select the point by mouse or enter the Co-ordinates by keyboard)

To point: 10, 10

To point: (Press ENTER) (Fig 2)

Fig 2



b Using relative Co-ordinates:

Example:

Draw a line from point (2, 2) to point 5 units in X-axis and 8 units first co-ordinate.

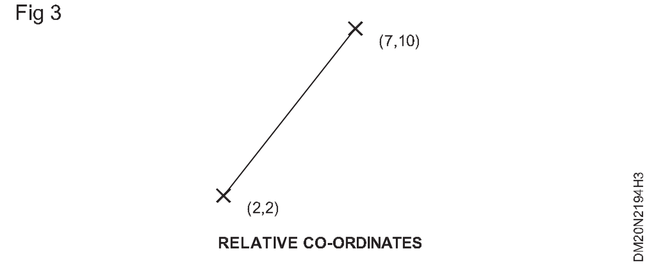
Command: LINE

From point: 2,2

To point: @5, 8

To point: (press ENTER) (Fig 3)

Fig 3



c Using Polar Co-ordinates:

Example:

Draw a line from point (1, 2) to a length of 6 units at 90 degree.

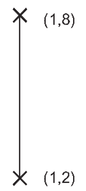
Command: LINE

From point: 1,2

To point:6@<90

To point:(Press ENTER) (Fig 4)

Fig 4



POLAR CO-ORDINATES

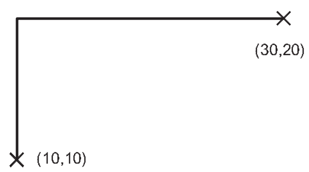
3 Drawing Entity -PLINE:

A polyline is a connected sequence of line and arc segments. The command is PLINE.

Example:

Draw a thick line of width 2 units from point to point using PLINE command. (Fig 5)

Fig 5



PLINE

4 Drawing Entity - RECTANGLE:

A rectangle is a polyline based on two opposite corner points called diagonal points. (A polyline is a connected sequence of line/ arc segments)

Example:

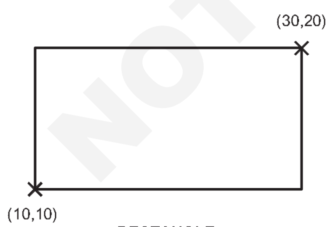
Draw a rectangle defined by diagonal points (10, 10) and (30, 20).

Command: RECTANGLE

First corner: 10, 10

Second Corner: 30, 20 (Fig 6)

Fig 6



RECTANGLE

5 Drawing Entity - CIRCLE

Circle can be drawn by any one of following five methods using CIRCLE command

a Using Centre and Radius:

Example:

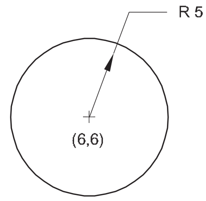
Draw a circle with centre (6, 6) and radius 5 units.

Command: CIRCLE

3P/2P/TTR/ <Centre point>: 6,6

Diameter/<Radius>:5 (Fig 7)

Fig 7



CIRCLE

b Using Centre and Diameter:

Example:

Draw a circle with centre (6, 17) and diameter 10 units.

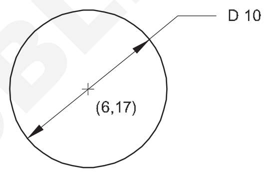
Command: CIRCLE

3P/2P/TTR <Centre point>: 6, 17

Diameter/<Radius>: D

Diameter: 10 (Fig 8)

Fig 8



CENTRE & DIAMETER

c Using Three given Points: (3P)

Example:

Draw a circle using the given 3points: (5,30), (4,26), (10,25). By entering 3 given points to be on the circumference of the circle:

Command: CIRCLE

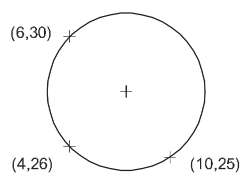
3p/2P/TTR/ <Centre point: 3P

First point: 5, 30

Second point: 4,26

Third point: 10.25 (Fig 9)

Fig 9



THREE POINT

d Using Two given Points: (2 P)

Example:

Draw a circle using the given 2 points: (7, 35), (7,47).

By entering 2 end points of the circle diameter:

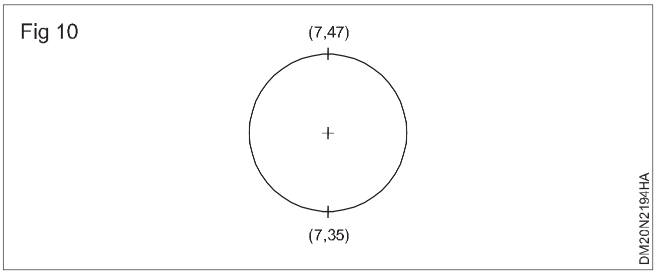
Command: CIRCLE

3P/2P/TTR/<Centre point>:2P

First point on diameter: 7, 35

Second point on diameter: 7, 47 (Fig 10)

Fig 10



e Using Tangent, Tangent and Radius:(TTR)

We can draw a circle by specifying two lines or two circles or a combination of line and circle, and also radius of circle. These two lines or two circles act as tangents to the circle.

6 Drawing Entity - ELLIPSE

Ellipse can be drawn by any one of following four methods using ELLIPSE command.

a Using First axis end points and other axis distance:

Example:

Draw an ellipse using major axis

endpoints (10, 20), (60, 20) and minor

axis end point (35, 35).

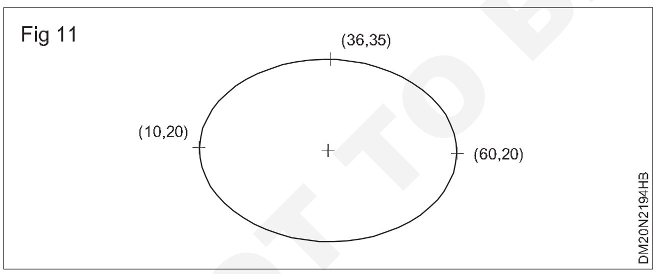
Command: ELLIPSE

<Axis endpointl >/Center: 10, 20

Axis endpoint2: 60, 20

<Other axis distance> /Rotation: 35, 35 (Fig 11)

Fig 11



b Using Centre of ellipse axis, End point and other axis distance:

Example:

Draw an ellipse with centre (100, 20), major axis endpoint (125, 20) and minor axis end point (100.35).

Command: ELLIPSE

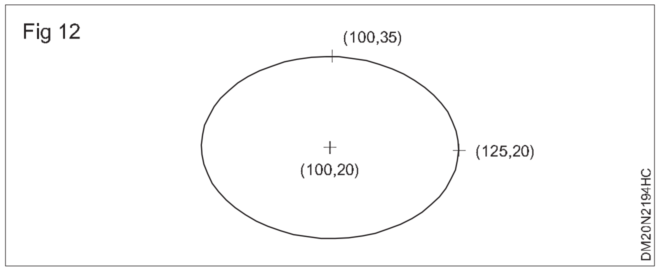
<Axis endpoint] >/ Centre: C

Centre of Ellipse: 100, 20

Axis endpoint2: 125.20

<Other axis distance> /Rotation: 100, 35 (Fig 12)

Fig 12



c) Using Centre, End point and Rotation angle of circle around the axis:

Example:

Draw an ellipse with centre (35, 48), major axis endpoint (60, 48) and 65 degree rotation around the major axis.

Command: ELLIPSE

<Axis endpoint!>/Centre: C

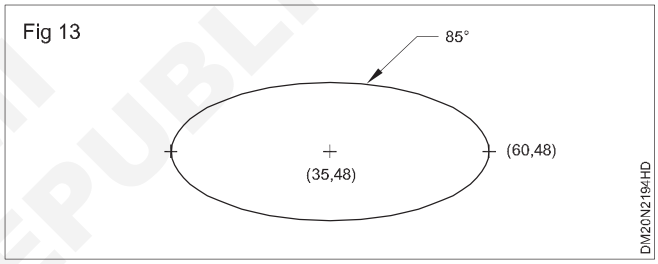
Centre of Ellipse: 35, 48

Axis endpoint2: (60, 48)

<Other axis distance> /Rotation: R

Rotation around major axis: 65 (Fig 13)

Fig 13



7 Drawing Entity - ARC

Arcs are partial circles and can be drawn in eight different methods using ARC command.

a Using Three given Points:

Example:

Draw an arc using the given three points: (75, 50), (55, 90), (105,110).

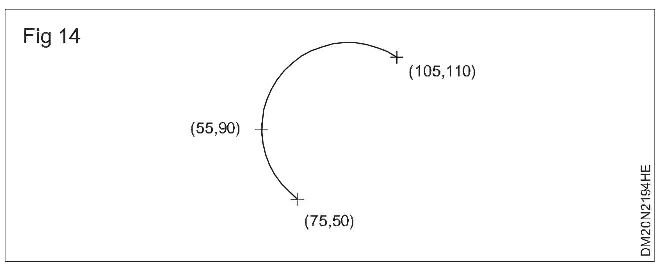
Command: ARC

Center /<Start point>:75, 50

Center/End/<Second point>: 55, 90

Endpoint: 105,110 (Fig 14)

Fig 14



b Using Start point, Centre and Endpoint: (S, C, E)

Example:

Draw an arc using start point (240, 20), centre point (250, 60) and endpoint (250, 100).

Command: ARC

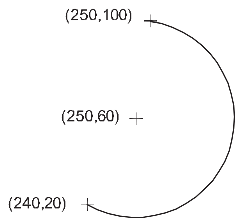
Center/<Start point> :240,20

Center/End/<Second point>: C

Center point: 250, 60

Angle/Length of chord! <Endpoint>:250,100 (Fig 15)

Fig 15



c Using Start point, Centre and Included Angle: (S, C, A)

Example:

Draw an arc using Start point (100,190),

centre point (40,190) and included angle 90 degrees.

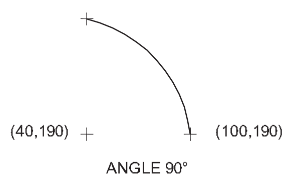
Command: ARC

Center/<Start point>: 100, 190

Center/End/ <Second point>: C

Center: 40, 190 (Fig 16)

Fig 16



> **NOTE: Positive "included angle" draws are in clockwise and negative in anti clockwise direction.**

d Using Start point, Centre and Length of chord : (S, C, L)

Example:

Draw an arc using start point (140.10),

centre point (100, 10) and chord length 45 units.

Command: ARC

Center/<Start point>: 140, 10

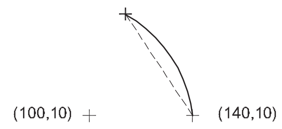
Center/End/< Second point>: C

Center: 100, 10

Angle/Length of chord/ <Endpoint>: L

Length of chord: 45 (Fig 17)

Fig 17



e Using Start point, End point and Radius : (S, E, R)

Example:

Draw an are using start point (230, 80), endpoint (190, 80) and radius 22 units.

Command: ARC

Center/<Start point>:230, 80

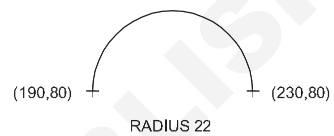
Center/End/<Second point E

Endpoint: 190, 80

Angle/Direction/Radius/ <Centre point>: R

Radius: 22 (Fig 18)

Fig 18



f Using Start point, End point and Included Angle: (S, E, A)

Example:

Draw an arc using start point (300, 60), end point (340,120) and included angle 90 degrees.

Command: ARC

Center/<Start point 300, 60

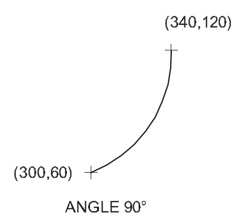
Center/End/ <Second point>: E

End point: 340, 120

Angle/Direction/Radius/ <Centre point>: A

Included Angle: 90 (Fig 19)

Fig 19



g Using Start point, End point and Starting Direction: (S, E, D)

Example:

Draw an arc with start point (40, 170), endpoint (70,230) and direction from start point 120 degrees.

Command: ARC

Center/<Start point>:40.170

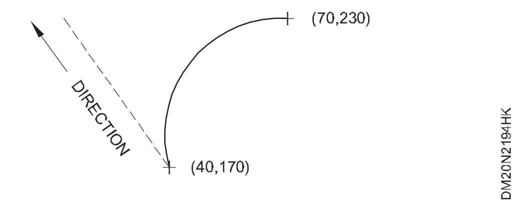
Center/End/ <Second point>: E

End point: 70, 230

Angle/Direction/Radius/ <Center point>:D

Direction from Start point: 120 (Fig 20)

Fig 20



h Using Line/Arc Continuation:

Example: Draw an arc with end point (200, 150) and tangential to the existing line.

Take Existing line from point (150, 200) to point (200,200).

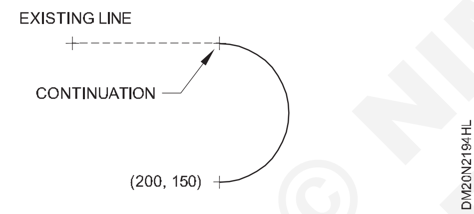
Command: ARC

Center/< Start point>: (Press ENTER)

End point: 200, 150 (Fig 21)

NOTE: This command is also used to draw a continuous are after an arc is drawn.

Fig 21



8 Drawing Entity -POLYGON

The Polygon command draws regular 2D polygons with 3 to 1024 sides. Any polygon can be drawn by the following three methods using POLYGON command.

a Using Radius of given Circle in which polygon is inscribed:

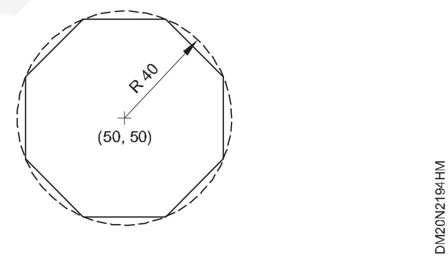
Example:

Draw a polygon of eight sides with centre (50, 50) inscribed in a circle of radius 40 units.

Command: POLYGON

Number of sides: 8 (Fig 22)

Fig 22



Edge/ <Center of polygon>: 50, 50

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

Radius of circle: 40

b Using Radius of given Circle on which polygon is circumscribed:

Example:

Draw an octagon with centre (140, 50) circumscribed on a circle of radius 40 units.

Command: POLYGON

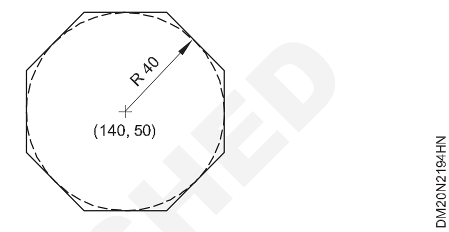
Number of sides: 8

Edge/Center of polygon 140, 50

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

Radius of circle: 40 (Fig 23)

Fig 23



c Using Edge Method:

Example:

Draw a polygon of ten sides using “edge method.” The first end point of the edge is (90,100) and second end point of the edge is (120,100).

Command: POLYGON

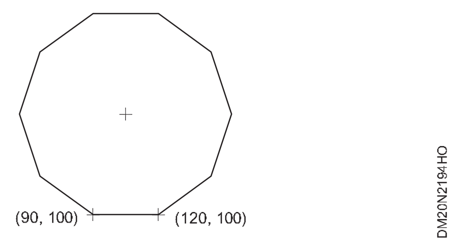
Number of sides: 10

Edge/ <Center of polygon>: E

First end point of edge: 90, 100

Second end point of edge: 120, 100 (Fig 24)

Fig 24



9 Drawing Entity-RAY

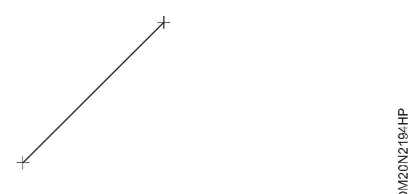
The RAY command can create a line starts at a point and continues to infinity in any specified direction.

Command: RAY

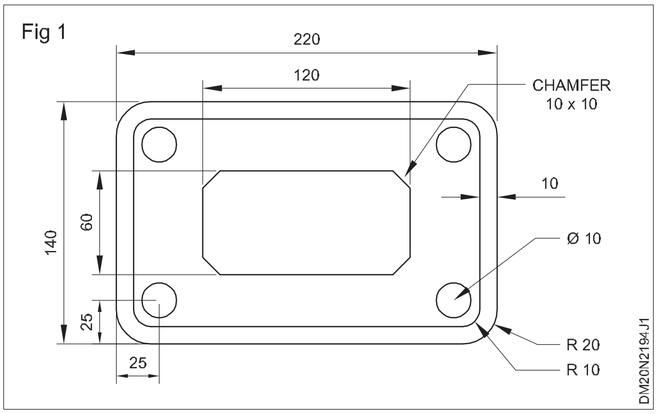
Specify starting point: XI, YI

Specify through point: X2,Y2 (Fig 25)

Fig 25



TASK 1: **Draw the following figure start from original point (0,0) (Fig 1)**



Absolute Coordinate System

Command: Rectangle

Command: fillet

Specify fillet radius for rectangles:20

Specify first corner point:0,0

Specify other corner point: 220,140

Command: Rectangle

Command: fillet

Specify fillet radius for rectangles:10

Specify first corner point:10,10

Specify other corner point: 210,130

Command: Rectangle

Command: chamfer

Specify first chamfer distance for rectangles:10

Specify second chamfer distance for rectangles:10

Specify first corner point:50,40

Specify other corner Point: 170,120

Command: Circle

Specify center point for circle: 25,25

Specify radius of circle:5

Command: Circle

Specify center point for circle:25,115

Specify radius of circle:5

Command: Circle

Specify center point for circle: 195,25

Specify radius of circle:5

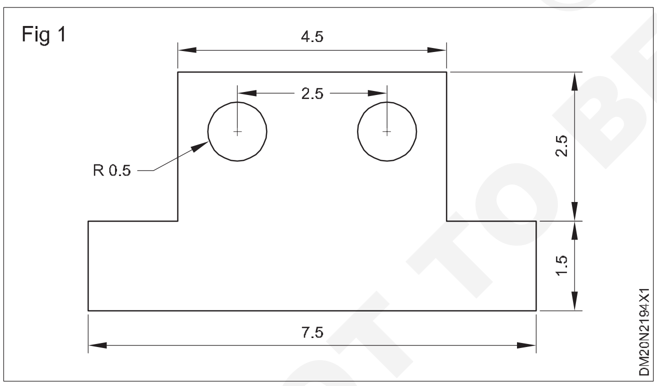
Command: Circle

Specify center point for circle: 195,115

Specify radius of circle:5

— — — — —

TASK 2: **Draw the following figures using Auto CAD commands (Fig 1)**



Specify included angle:90

Command: a

Specify first point:10,0

Specify center point:5,0

Specify included angle:180

Command: a

Specify first point:0,10

Specify center point:5,10

Specify included angle: 180

— — — — —

TASK 3: **Draw the following figure start from original point (0.0) (Fig 1)**

Absolute Coordinate System

Command: L

Specify first point:0,0

Specify next point:0,3

Command: L

Specify first point:2,0

Specify next point:2,3

Command: L

Specify first point:4,0

Specify next point:4,3

Command: L

Specify first point:6,0

Specify next point:6,3

Command: a

Specify first point:0,0

Specify second point:1,-1

Specify end point:2,0

Command: a

Specify first point:4,0

Specify second point:5,-1

Specify end point:6,0

Command: a

Specify first point:2,3

Specify second point:3,4

Specify end point:4,3

Command: a

Specify first point:0,3

Specify second point:3,6

Specify end point:6,3

Command: C

Specify center point for circle: 1,0

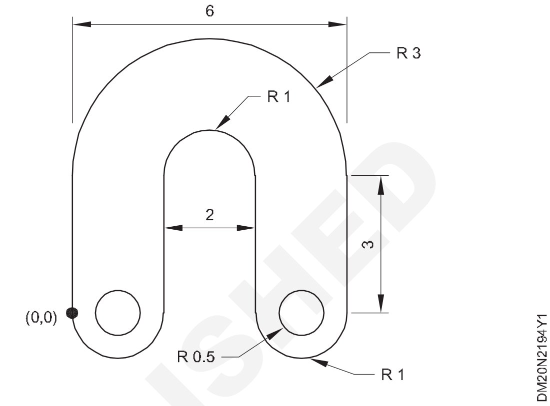
Specify radius of circle:0.5

Command: C

Specify center point for circle: 5,0

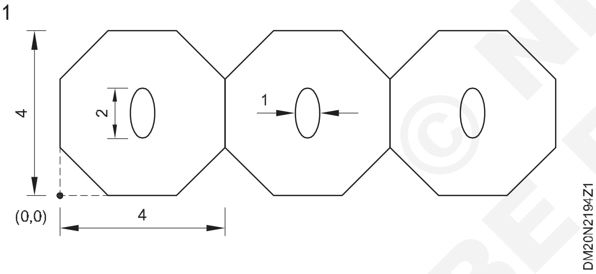
Specify radius of circle:0.5

Fig 1



TASK 4: **Draw the flowing figure start from original point (0,0) (Fig 1)**

Fig 1



Absolute Coordinate

Command: Polygon

Enter number of side: 8

Specify center point :2,2

Specify radius of circle:2

Command: Polygon

Enter number of side: 8

Specify center point :6,2

Specify radius of circle:2

Command: Polygon

Enter number of side: 8

Specify center point :10,2

Specify radius of circle:2

Command: ellipse

Specify axis end point of ellipse:2,1

Specify other end point od axis:2,3

Specify distance to other axis:0.5

Command: ellipse

Specify axis end point of ellipse 6,1

Specify other end point od axis:6,3

Specify distance to other axis:0.5

Command: ellipse

Specify axis end point of ellipse:10,1

Specify other end point od axis:10,3

Specify distance to other axis:0.5

Modify 2D objects using break, erase, trim, offset, fillet, chamber, move, copy, array, rotate, Hatch command

Objectives: At the end of this exercise you shall be able to
- **practice the commands used to modify different objects using AutoCAD like move, copy, rotate, trim, Array, Break, Erase, Offset, Fillet, Chamfer, scale, Hatch, Insert Block.**

Requirements

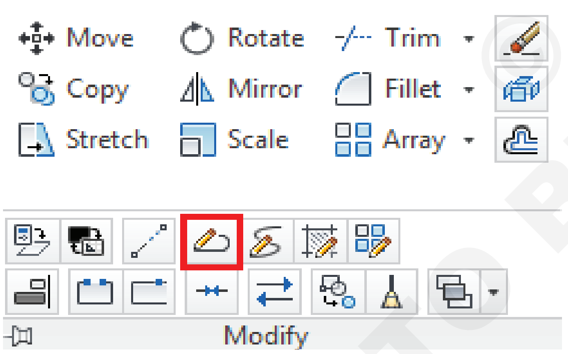
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

Modify:

The MODIFY command is used to modify the existing drawings or complete the drawing easily. AutoCAD drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to MODIFY these basic drawing objects in some way in order to create the image you need. AutoCAD provides a whole range of modify tools such as Move, Copy, Rotate and Mirror. A good understanding of the Draw commands is fundamental to the efficient use of Auto CAD. (Fig 1)

Fig 1

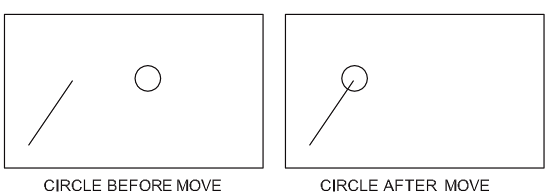


Modifying Entity-MOVE

The modify entity MOVE command is used to move an object from one place to another place in the work plane. (Fig 2)

Command: MOVE

Fig 2



MOVE COMMAND

DM20N2195H2

Select object to move:

Select through point:

Specify through distance.

Modifying Entity -COPY

The modify entity COPY command copies the selected objects and creates any number of copies on the workspace at specified distance with specified rotation. (Fig 3)

Command: COPY

Select an object to copy:

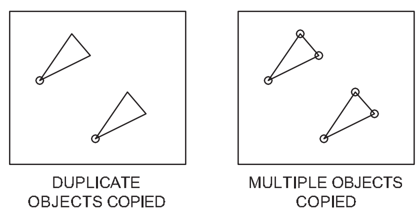
Select a through point:

Select through distance:

Esc to close command.

Esc - Escape key

Fig 3

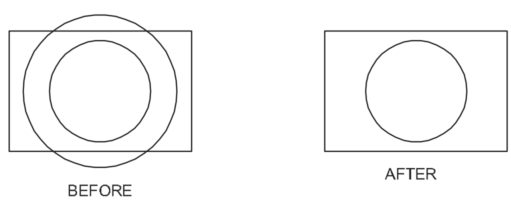


COPY COMMAND

DM20N2195H3

Modifying Entity-ERASE

Fig 4



ERASE COMMAND

DM20N2195H4

The modify entity ERASE command is used to delete an object or some part of the object from the drawing permanently. It can be also done by selecting an object and pressing delete button on the keyboard. (Fig 4)

Command: ERASE

Select an object to erase:

Close the command.

Modifying Entity mirror:

The Modify command is used to create a mirror image of the subjected object. The symmetric objects can be easily created using this command. (Fig 5)

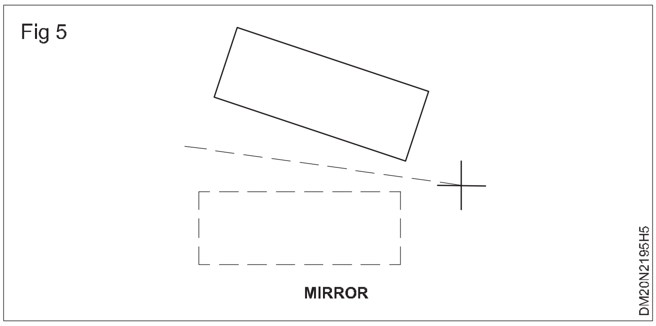
Select an object:

Specify first point of mirror line:

Specify second point of mirror line:

Close the command.

Fig 5



Modifying Entity-TRIM

The modify entity TRIM command is remove the unnecessary parts of the drawing corresponding to the other drawing entities. It is also used to meet the edges of one object to the other in the drawings. (Fig 6)

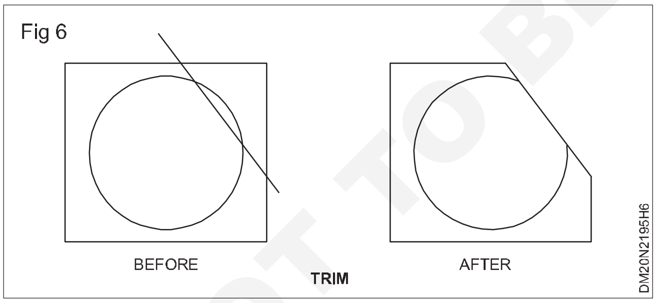
Command: TRIM

Select the objects to trim:

Trim unnecessary parts:

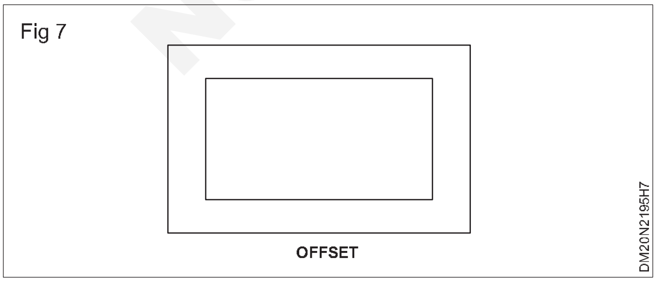
Close the command.

Fig 6



Modifying Entity-OFFSET

Fig 7



The modify entity OFFSET command is used to OFFSET the objects through specified distances. This command duplicates the same object around itself with some distance. (Fig 7)

Command: OFFSET

Select the objects to offset:

Specify offset distance:

Specify direction to offset:

Close the command.

Modifying Entity- ARRAY

The modify entity ARRAY command creates the selected objects into a number of objects arranged in a specified order. ARRAY command can be used in two different ways.

a Rectangular array:

The rectangular array is used to create the selected objects into number of objects and arrange them in a rectangular order. (Fig 8)

Command: RA

Select the object:

Select number of rows:

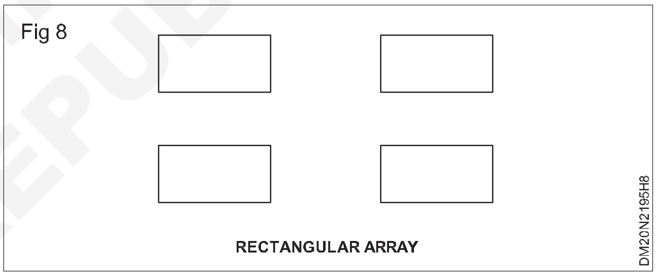
Specify distance between two consecutive rows:

Select number of Columns:

Specify distance between two consecutive columns:

Close the command:

Fig 8



b Polar array:

The polar array is used to create the selected objects into number of objects and arrange them in a circular order. (Fig 9)

Command: PA

Select the object:

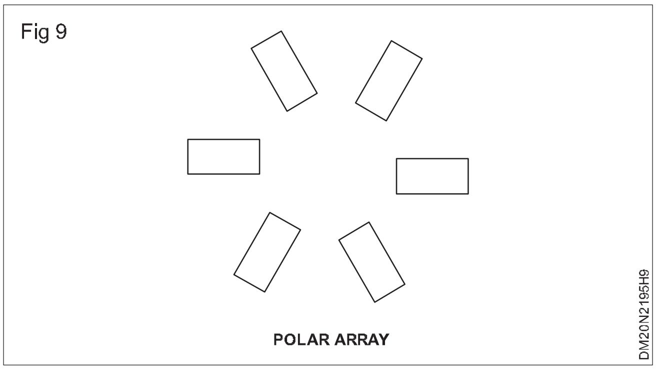
Select number of rows:

Specify distance between two consecutive rows:

Select angle of rotation:

Close the command.

Fig 9



Modifying Entity-ROTATE

The modify entity ROTATE command is used to rearrange the selected object in the required order or at the specified angle. (Fig 10)

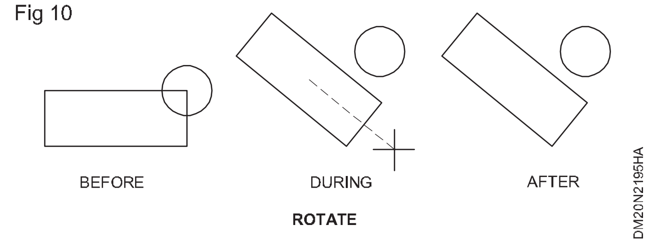
Command: ROTATE

Select the object to rotate:

Specify the through point to rotate:

Close the command.

Fig 10



Modifying Entity-SCALE

The modify entity SCALE command is used to resize the selected objects to specified scaling ratio. This command is used to view the large objects into a considered size. (Fig 11)

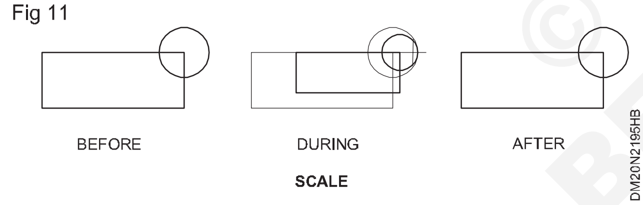
Command: SCALE

Select an object to be scaled:

Specify scaling ratio:

Close the command.

Fig 11



Modifying Entity-STRETCH

The modify entity STRETCH command is used to strech the required object to the required length. This command is also used to make join one entity with the other. (Fig 12)

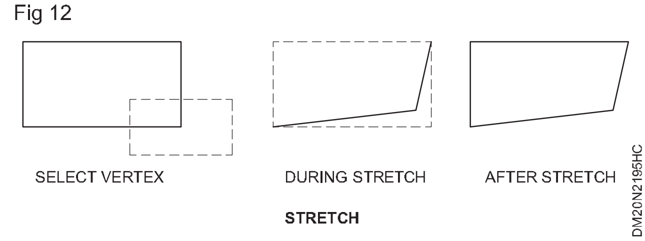
Command: STRETCH

Select an object to stretch:

Specify the distance or end object:

Close the command.

Fig 12



Modifying Entity-LENGTHEN

The modify entity LENGTHEN command is used to increase the length of the line command. (Fig 13)

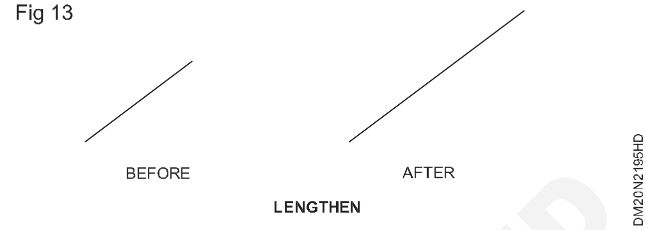
Command: LENGTHEN

Select an object to lengthen:

Specify the distance or end object.

Close the command.

Fig 13



Modifying Entity-EXTEND

The modify entity EXTEND command is used to extend the required object to the required length. This command is also used to make join one entity with the other. (Fig 14)

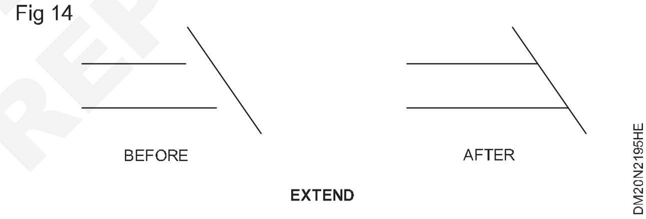
Command: EXTEND

Select an object to stretch:

Specify the distance or end object:

Close the command.

Fig 14



Modifying Entity - BREAK

The modify entity BREAK command is used to create breaks in the objects. Any entity can be breaked at any place using this break command. (Fig 15)

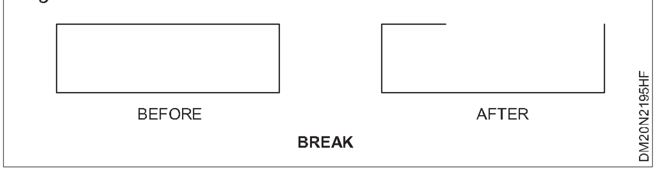
Command: BREAK

Select an object to break:

Specify break position:

Close the command.

Fig 15



Modifying Entity-CHAMFER

The modify entity CHAMFER command is used to round off the sharp edges of the object. (Fig 16)

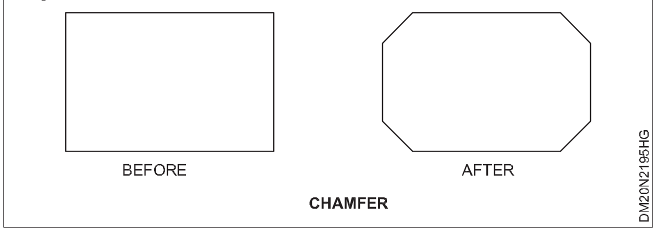
Command: CHAMFER

Select an end to chamfer:

Select the chamfer radius:

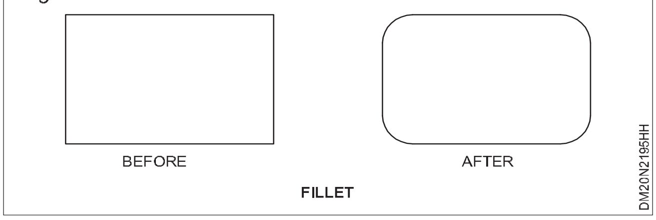
Close the command.

Fig 16



Modifying Entity-FILLET

Fig 17



The modify entity FILLET command is used to remove the sharp edges of the object. (Fig 17)

Command: FILLET

Select an end to chamfer:

Select the chamfer radius:

Close the command.

Modifying Entity-HATCH

The modify entity HATCH command is used to define hatching to an object. Hatch is used to clearly identify the different parts of an object and different sectional views of object. Different types of hatches can be used by changing properties of hatch. (Fig 18)

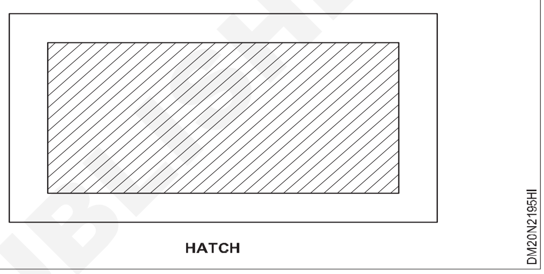
Command: HATCH

Select the area to hatch:

Edit hatch properties:

Close the command.

Fig 18



CAD: Create templates, insert drawing-create objects in different layers and modify layer properties

Objectives: At the end of this exercise you shall be able to
- **create a new template**
- **Interest a drawing**
- **creation of objects in different layers.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

Layers Definition (Fig 1)

The way to gain complete layer control is through the Layer Properties Manager. . The Layer Properties Manager is invoked by:

1 Command line Window: Layer or LA

2 Tool Bar: format ↔Layer

3 layer properties manager:

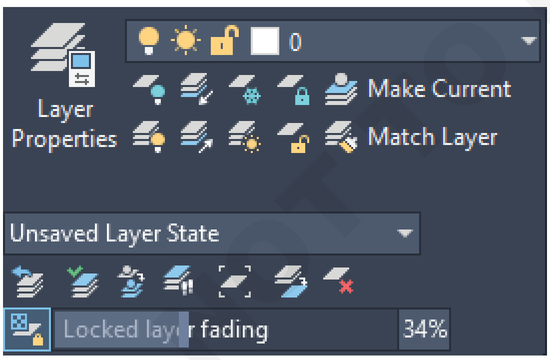
Command:-Layer

Current Layer."0"

Enter an option

[?/Make/Set/New/On/Off/Color/Ltype/Lweight/Plot/Pstyle/Freeze/Thaw/Lock/Unlock/state]:

Fig 1



Layer Properties manager Layer List

The right central area of the layer properties manager is the layer list that displays the list of layers in the drawing. Here you can control the current visibility settings and the properties assigned to each layer. This list can display all the layers in the drawing or a subset of layers if a layer filter is applied. Layer filter are created and applied in the filter tree view on the left side of the layer properties manager. For new drawings such as those created from the AutoCAD, dwg template. only one layer may exist in the drawing layer "0". New layers can be created by selecting the new layer button or the new layer option of the right click menu.

Steps of Create Layers (Fig 2)

1 Click on the left of layer properties bar. The following dialog box appear.

2 Click (New), new raw adding to existing layers, we saw always there was first layer it is zero layer which created by AutoCAD Automatically when open new drawing. Appear in new raw name of new layer we can change it to objective name to easily access to layer we need, such as wall, column. steel, ground.......

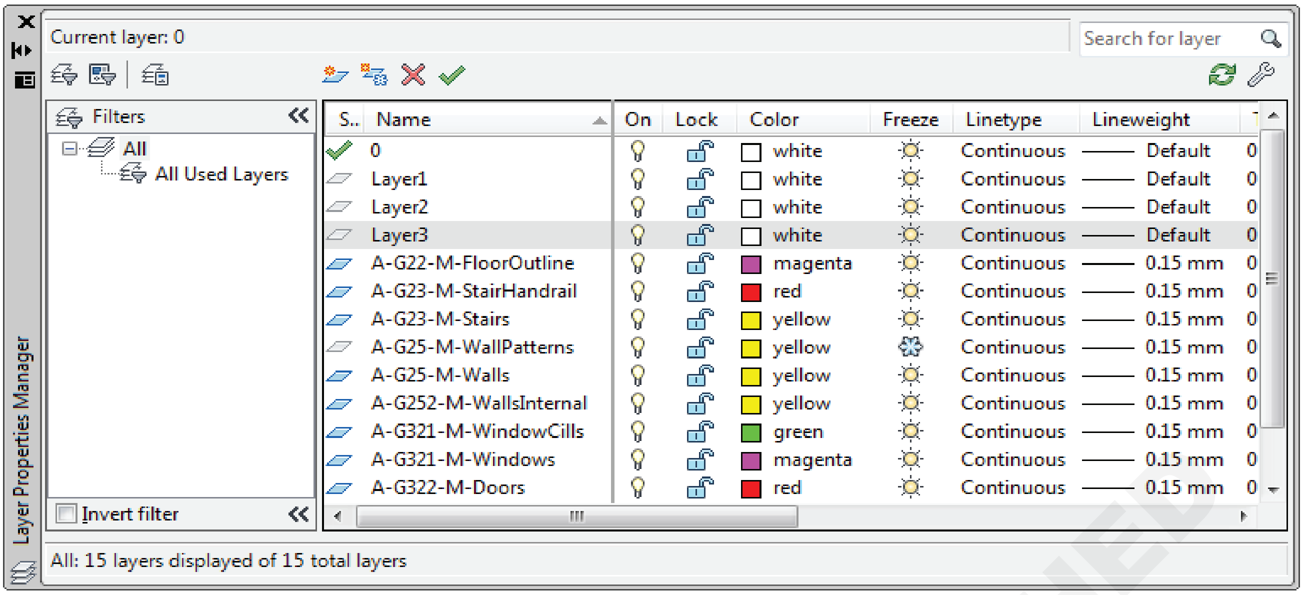
3 To determine layer color click on black square under address (color) in new raw color window appear select from it appropriate color.

4 To determine line style click on (continuous) in new raw under address (line type). window appear, select from it appropriate line type click on (load) to appear more line types.

5 To determine line weight used for drawing objects click (default) under address (line weight) then select weight we need from the window that appear.

6 Repeat the operation for other layers then click (ok).

Fig 2



Layers Managements

***Select drawing layer to be activate**

1 Click on manager Layer List.

2 Select layer from it.

3 Click on drawing board the selected layer be the active layer note that object color and line type and line weight all must be assistant to(By layer).

***move drawing object from layer to another.**

1 Select drawing object.

2 Click on manager Layer List to open it.

3 Select layer want to move the object to it.

4 Click on ESC twice to undo select object, we saw change the layer color and line to color of selected layer.

***hid objects belong to determine drawing layer.**

1 Click on manager Layer List to open it.

2 Click on yellow light for layer wants to hid it by put it to off then click on enter we obtain the drawing without the selected layer.

***lock objects belong to determine layer.**

1 Click on manager Layer List to open it.

2 Click on lock button for layer wants to lock it then click enter, the layer is locked. The locked layer it is the layer that can't change or modified to it unless we open it by click the lock button again.

***change color of drawing layer.**

1 Click on properties bar, the dialog box appear for control layers and lines types.

2 Click on color, color control box appear

3 Select appropriate color.

4 Click ok in two open boxes to close them we saw all the objects belong to the layer their color are change to new color.

— — — —

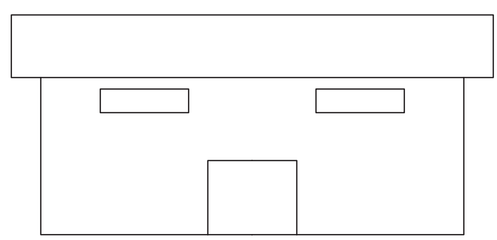
TASK 1: **Setup the layers shown in the following table, then create the house shown in figure (Fig 1)**

Then apply the following:

- Hide layer windows.
- Lock layer door.
- Change layer walls color from yellow to pink.

Layer Name	Color	Line Type	Line Weight
Walls	Yellow	Continuous	0.40mm
Windows	blue	Center2	0.20mm
Door	red	Continuous	Default

Fig 1



DM20N2197H1

— — — —

Dimensioning by Auto CAD

Objectives: At the end of this exercise you shall be able to
- **draw the techniques of the dimensioning in Auto CAD.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

Dimensioning:

The correct use of AutoCAD's dimension tools is the key to producing clear and concise measured drawings. Any drawing is incomplete without specifying its dimensions. Dimensions are necessary to understand the drawings and also to manufacture is the parts in the drawings. In this experiment, we are going to learn how to use different types of dimensioning commands.

1 Linear Dimension command:

The LINENEAR DIMENSION command is used to generate the horizontal and vertical dimensions in the drawing. The dimensions of vertical and horizontal lines are represented in the drawing using this command. (Fig 1)

Command: DIMLINEAR

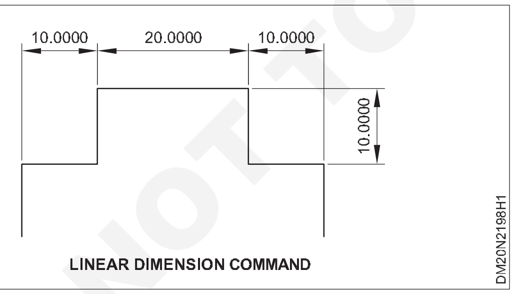
Select a line or first point:

Select end point:

Select dimension direction:

Close the command.

Fig 1



2 Baseline Dimension command:

The BASELINE DIMENSION command is also used to generate the horizontal and vertical dimensions in the drawings. But the only difference is that every dimension is started from a fixed basic line unlike to linear dimension. (Fig 2)

Command: DIMBASELINE

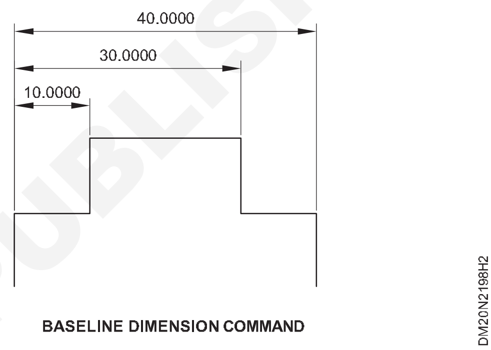
Select the base line:

Select second point for dimension:

Select dimension direction:

Close the command

Fig 2



3 Aligned Dimension command:

The ALIGNED DIMENSION command is used to generate the dimensions of the aligned lines and aligned portions of the drawings. By using this command, we can generate the dimensions of the aligned lines parallel to them. (Fig 3)

Command: DIMALIGNED

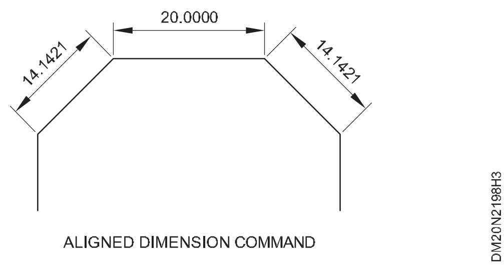
Select a line or first point:

Select end point:

Select dimension direction:

Close the command.

Fig 3



4 Diameter Dimension command:

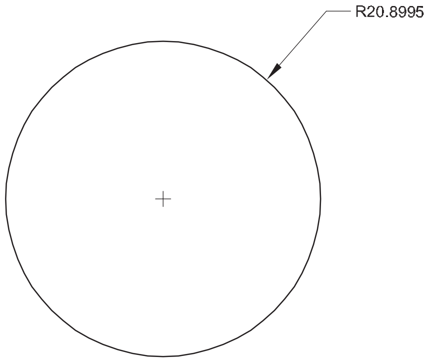
The DIAMETER DIMENSION command is used to specify the diameter of the circles or ares that are present in the drawings. (Fig 4)

Command: DIMDIAMETER

Select the circle to annotate:

Close the command.

Fig 4



DIAMETER DIMENSION COMMAND

DM20N2198H4

5 Radius Dimension command:

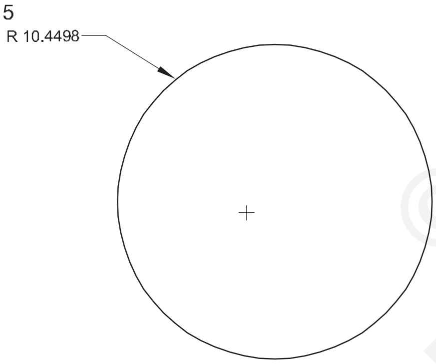
The DIAMETER DIMENSION command is used to specify the diameter of the circles or ares that are present in the drawings. (Fig 5)

Command: DIMDIAMETER

Select the circle to annotate;

Close the command.

Fig 5



RADIUS DIMENSION COMMAND

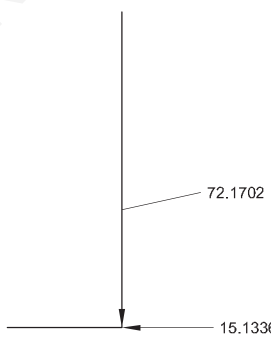
DM20N2198H5

6 Ordinate Dimension command:

The ORDINATE DIMENSION command is used to annotate co-ordinate points with X or Y values of any point in the drawing. This may be useful for setting-out on site plans. (Fig 6)

Command: DIMORDINATE

Fig 6



ORDINATE DIMENSION COMMAND

DM20N2198H6

Select the point to dimension ordinates:

Close the command.

7 Angular Dimension command:

The ANGULAR DIMENSION command is used to represent the angles of angular lines with respect to a base line in the drawings. (Fig 7)

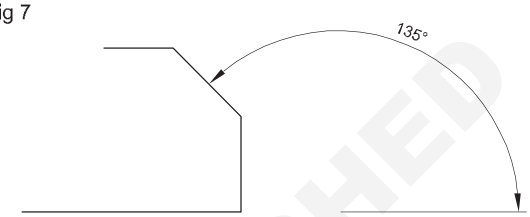
Command: DIMANGULAR

Select base line:

Select the line to represent angle:

Close the command.

Fig 7



ANGULAR DIMANSION COMMAND

DM20N2198H7

8 Dimension Text Edit command:

DIMENSION TEXT EDIT command is used to edit the text in the dimension. This command option is used to edit the dimension to represent the repetitive dimensions.

Command: DIMTEDIT

Select the dimension to edit:

Edit dimension:-

Close the command.

9 Dimension Edit command:

DIMENSOIN EDIT command is used to edit the properties of the dimensions such as dimension line type, line thickness, arrow size, text height etc. (Fig 8)

Command DIMEDIT

Select the dimensions to edit properties:

Edit required properties:

Close the command.

AutoCAD 2D

Design Center Blocks 20.2

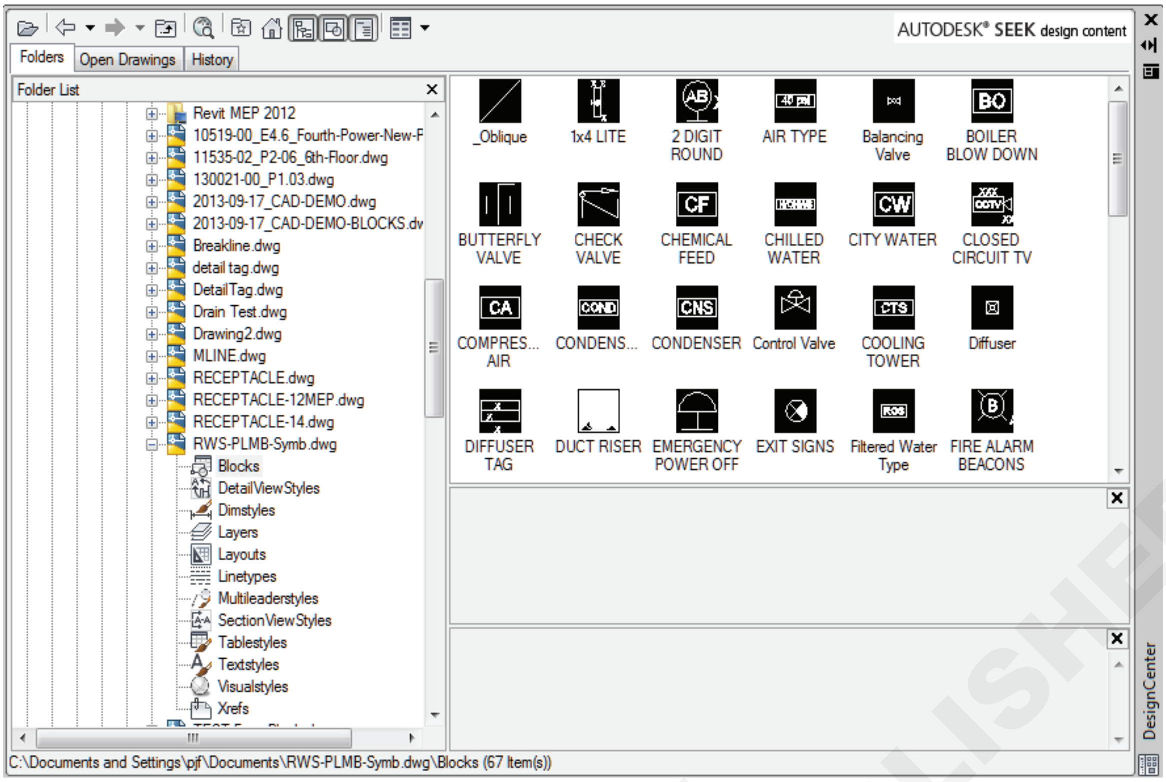
1 Choose: Blocks from one of the Design Center menus.

2 Drag: and drop a block from the Design Center into a drawing.

TIP:

- Blocks with attributes will be prompted as they are inserted into the drawing.

Fig 8


AUTODESK® SEEK design content
Folders
Open Drawings
History
Folder List
Revit MEP 2012
10519-00_E4.6_Fourth-Power-New-F
11535-02_P2-06_6th-Floor.dwg
130021-00_P1.03.dwg
2013-09-17_CAD-DEMO.dwg
2013-09-17_CAD-DEMO-BLOCKS.dv
Breakline.dwg
detail tag.dwg
DetailTag.dwg
Drain Test.dwg
Drawing2.dwg
MLINE.dwg
RECEPTACLE.dwg
RECEPTACLE-12MEP.dwg
RECEPTACLE-14.dwg
RWS-PLMB-Symb.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
_Oblique
1x4 LITE
2 DIGIT ROUND
AIR TYPE
Balancing Valve
BOILER BLOW DOWN
BUTTERFLY VALVE
CHECK VALVE
CHEMICAL FEED
CHILLED WATER
CITY WATER
CLOSED CIRCUIT TV
COMPRES... AIR
CONDENS...
CONDENSER
Control Valve
COOLING TOWER
Diffuser
DIFFUSER TAG
DUCT RISER
EMERGENCY POWER OFF
EXIT SIGNS
Filtered Water Type
FIRE ALARM BEACONS
BO
CF
CW
CA
COND
CNS
CTS
DesignCenter
C:\Documents and Settings\pjf\Documents\RWS-PLMB-Symb.dwg\Blocks (67 Item(s))

CAD: Construct or thographic sectional view of a steel bracket with dimension

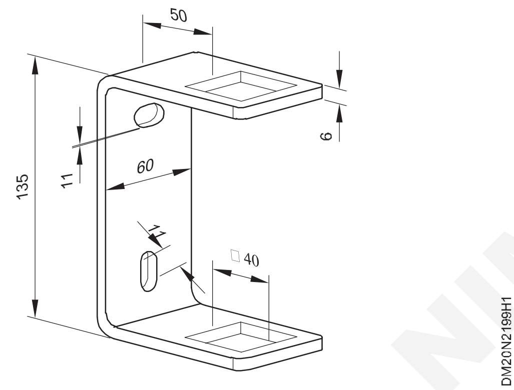
Objectives: At the end of this exercise you shall be able to
- **draw the sectional view, plan, Elevation using CAD.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the sectional view, plan, Elevation using CAD (Fig 1)**

Fig 1



Construct isometric view of machine blocks

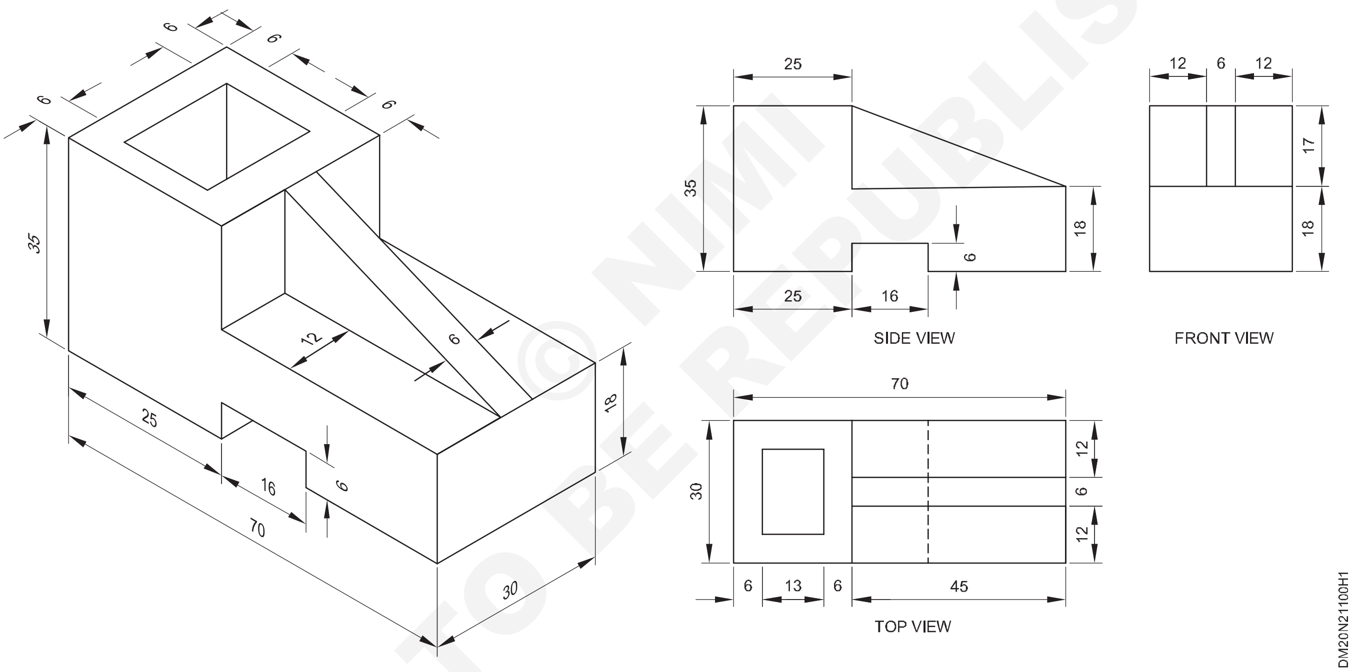
Objectives: At the end of this exercise you shall be able to
- **draw isometric view using orthographic view.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw isometric view using orthographic view using CAD (Fig 1)**

Fig 1



— — — — —

Create view ports in layout Space

Objectives: At the end of this exercise you shall be able to
- **create viewports in layout space**
- **place views for model space.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

Typing the View Command (Fig 1)

1 **Type** View at the command prompt
Command: -VIEW

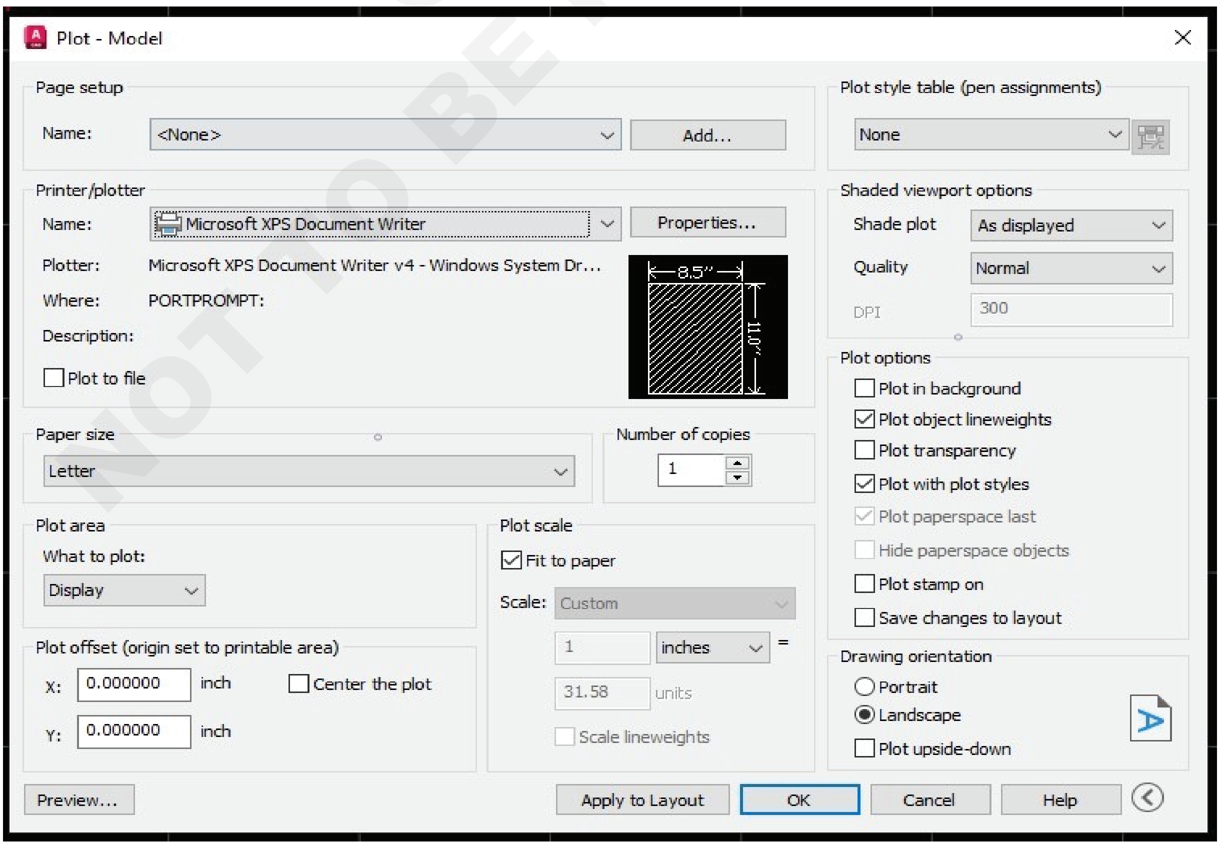
2 **Type** One of the following view options:
?/Delete/Restore/Save/Window:

View options:

?	Lists the named views for this drawing
Delete	Deletes the named view
Restore	Displays the specified view
Save	Attaches a name to the current view of the drawing
Window	Attaches a name to specified window

Plotting Named Views

Fig 1



Viewports

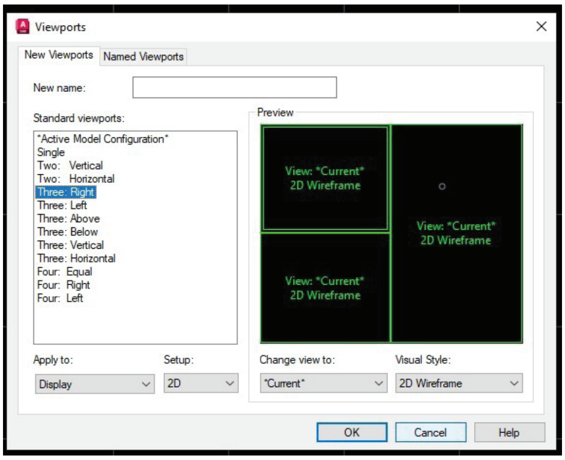
Vports Command (Figs 2 - 5)

1 **Choose** View, Viewports, New Viewports...

2 **Choose** one of the viewports configurations

3 **Click** OK.

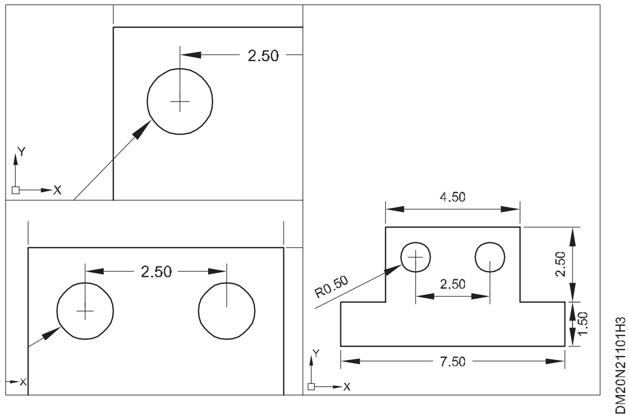
Fig 2



4 **Click** once in each vport to make it active.

5 **Type** a ZOOM option in each viewport.

Fig 3



Creating a Layout

1 **Choose** the Layout1 TAB at the bottom of the screen.

Fig 4

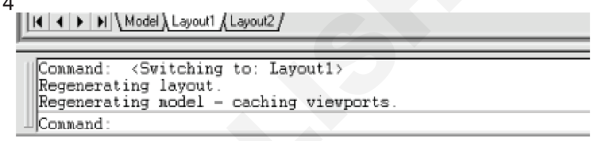
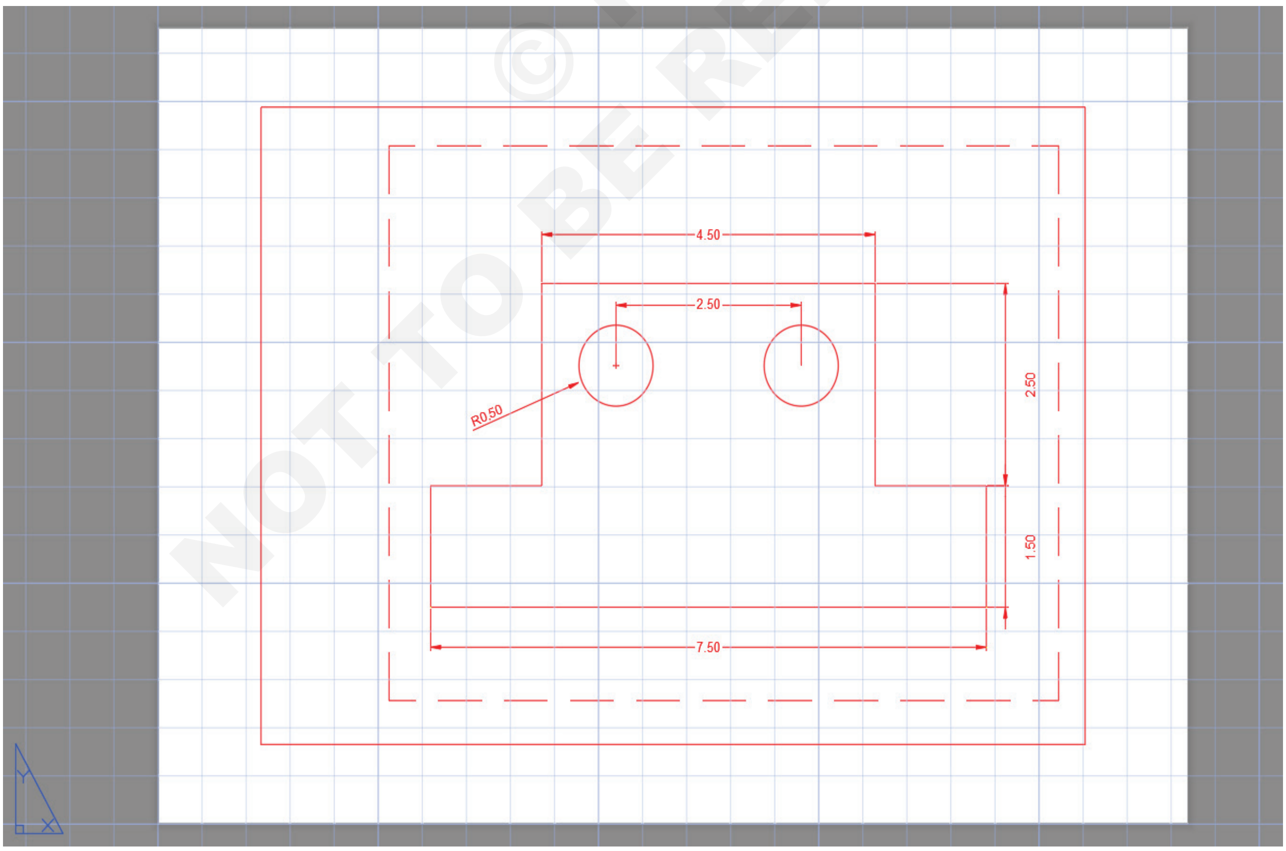
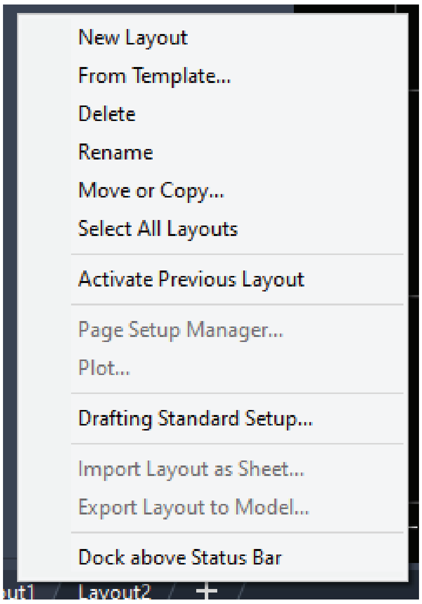


Fig 5



2 Right-Click Layout 1 to change the name and other properties of layout (Fig 6)

Fig 6



Model Space

MSPACE (model space) can only be activated if there is at least one review. To enter model space mode use “MSPACE”.

1 Type MSPACE at the command prompt. Command: **MSPACE or MS**

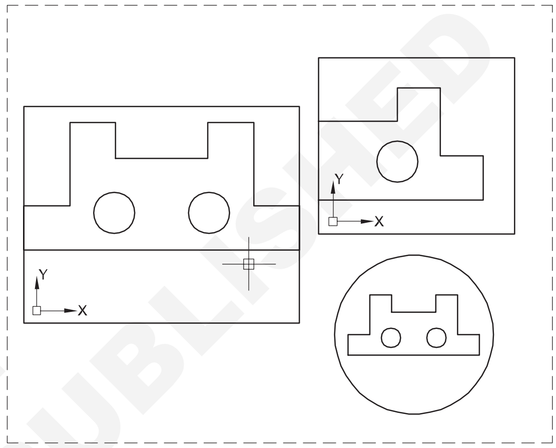
or

2 Double-Click the word “PAPER” on the Status Bar to toggle to model space. (Fig 7)

Fig 7

Notice the ucsicon will appear in each of the m views when you enter model space. (Fig 8)

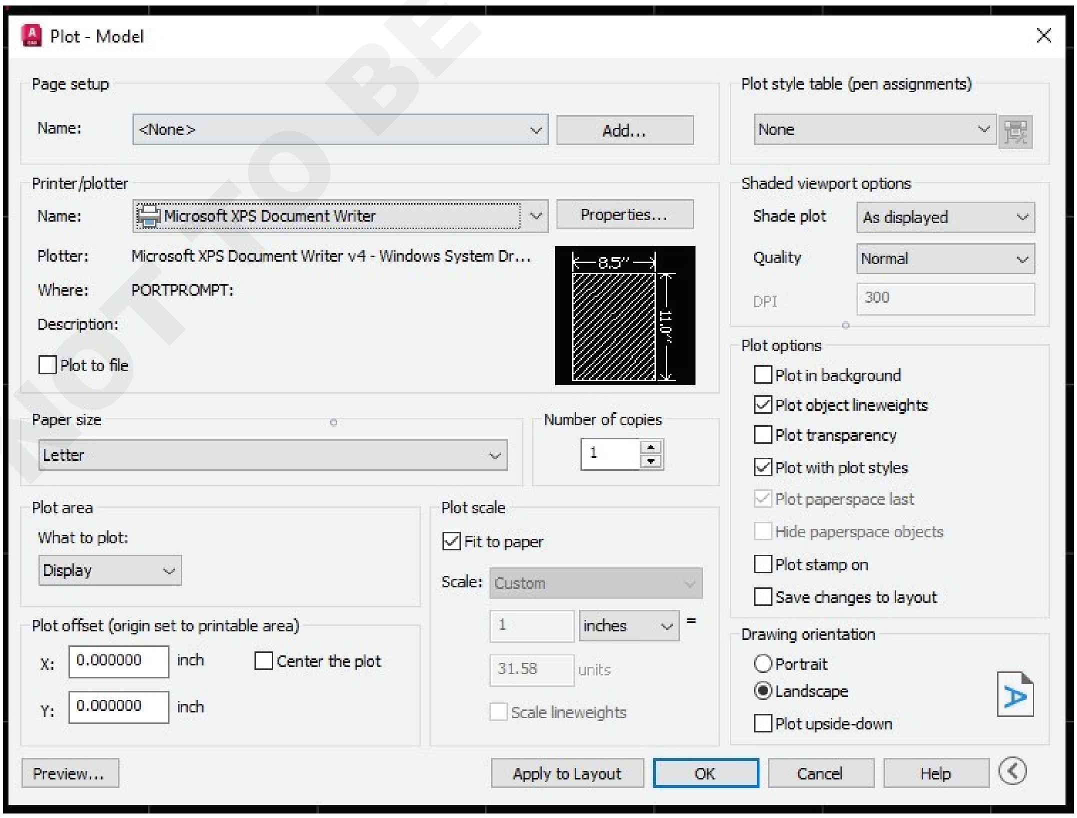
Fig 8



DM20N21101H8

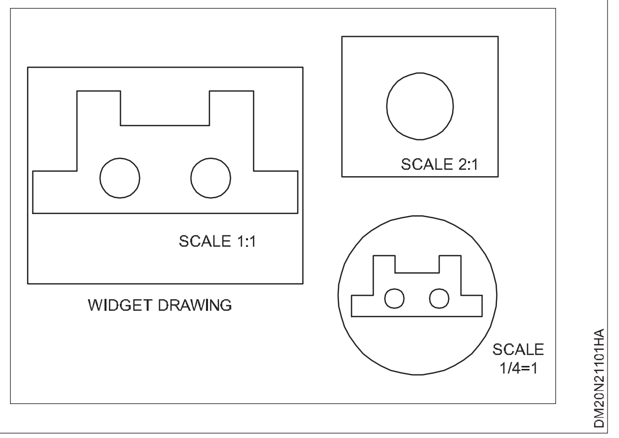
Plotting in Paper Space (Fig 9)

Fig 9



- Plotting all M VIEWS should be done from Paper Space not from Model Space.
- When you plot from P space, 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 10)

Fig 10



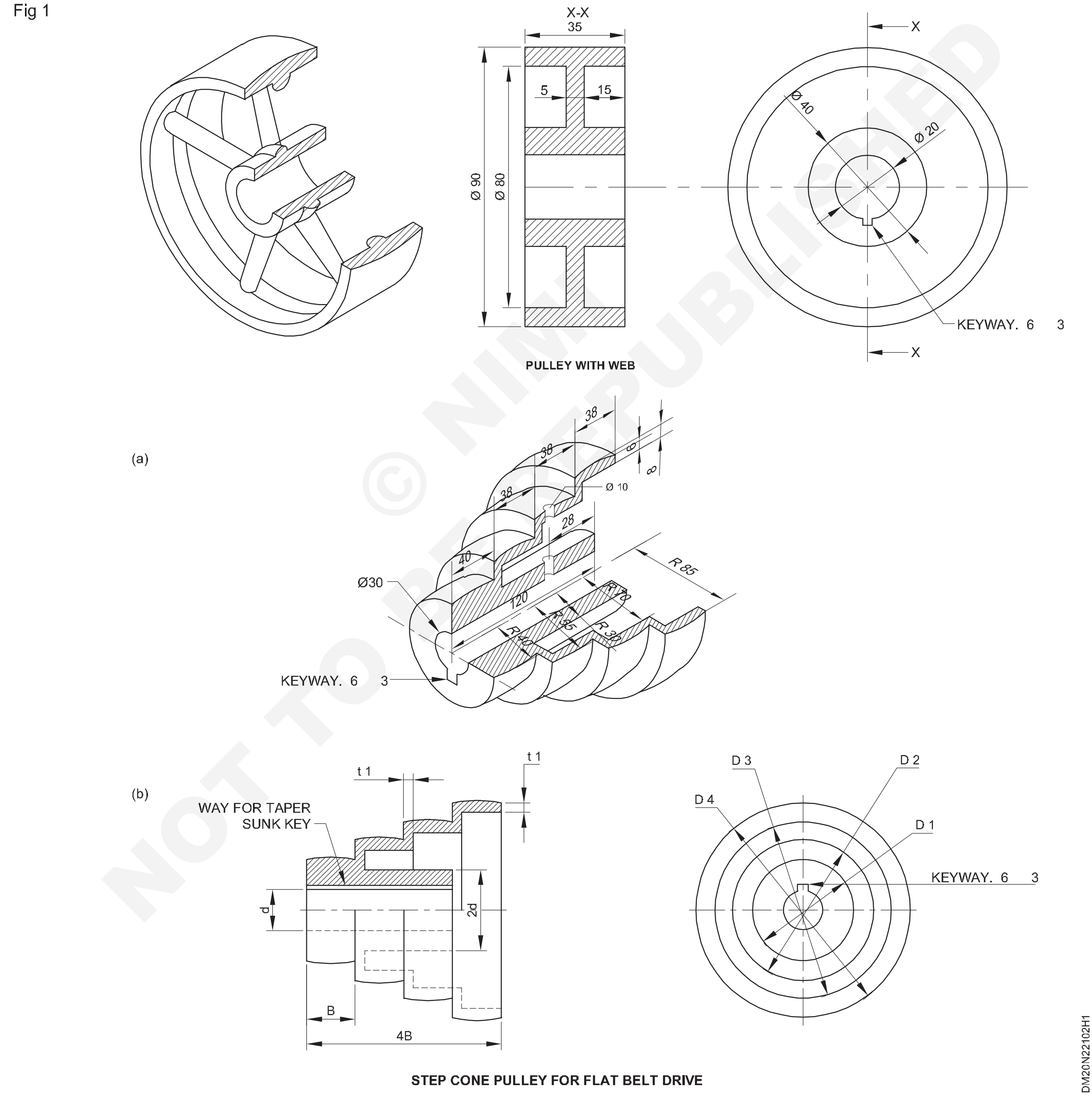
Construct pulleys using CAD

Objectives: At the end of this exercise you shall be able to
- **draw a solid pulley**
- **draw stepped pulley.**

PROCEDURE

TASK 1: **Draw the solid pulley, stepped pulley using CAD (Fig 1)**

Fig 1



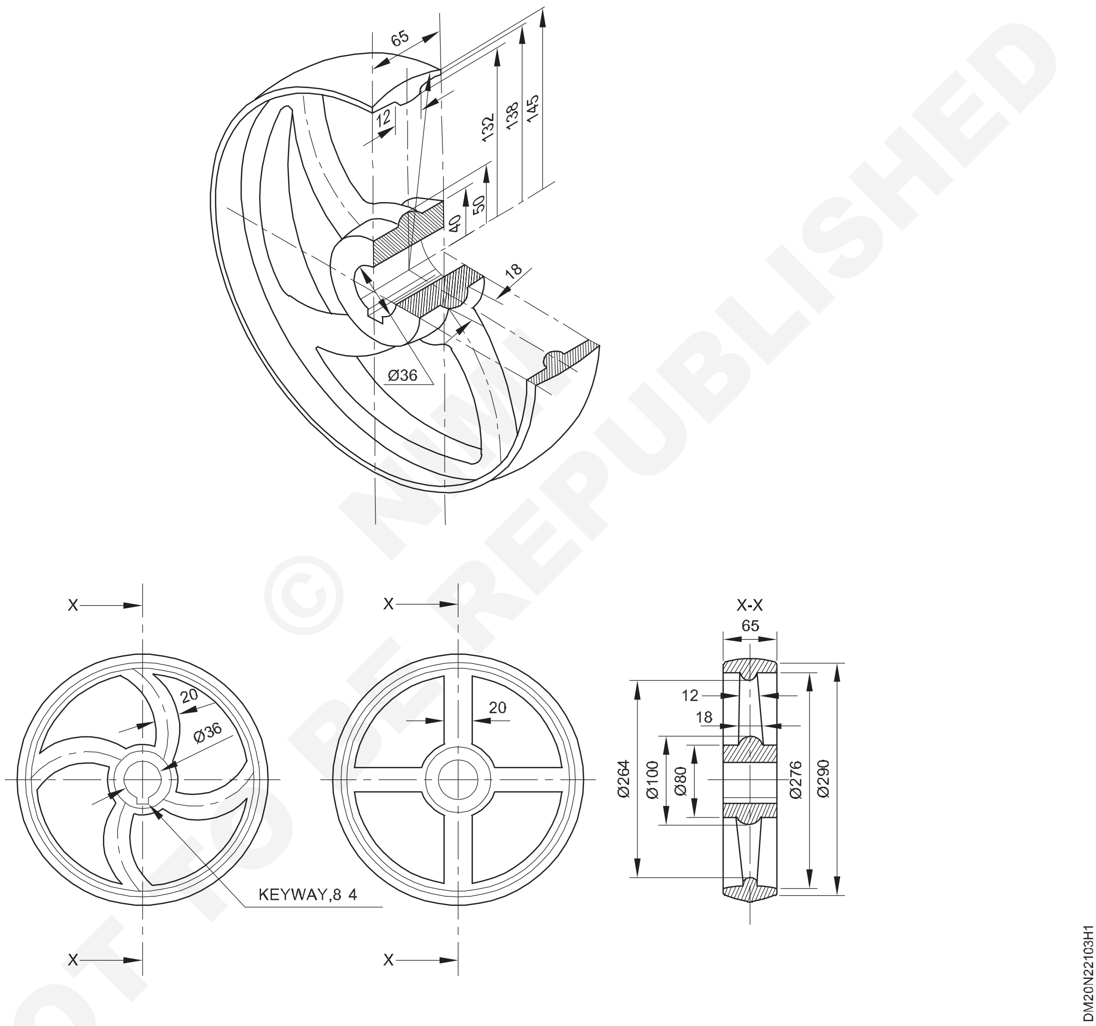
Construct pulley with different types of arms

Objectives: At the end of this exercise you shall be able to
- **construct pulley with arms.**

PROCEDURE

TASK 1: **Construct pulley with arms using CAD (Fig 1)**

Fig 1



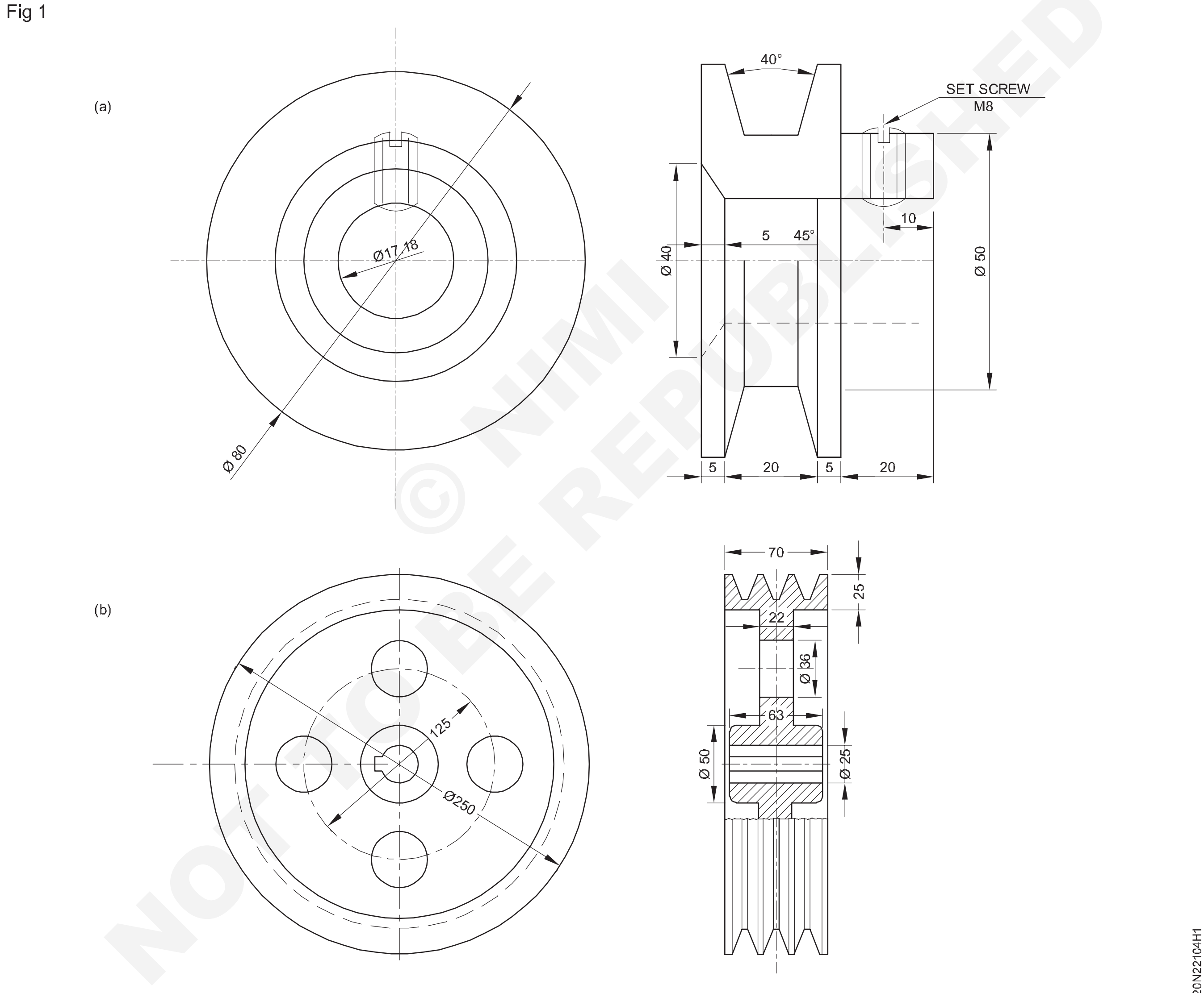
Draw rope pulley and vee belt pulley using CAD

Objectives: At the end of this exercise you shall be able to
- **draw rope pulley**
- **draw V belt pulley.**

PROCEDURE

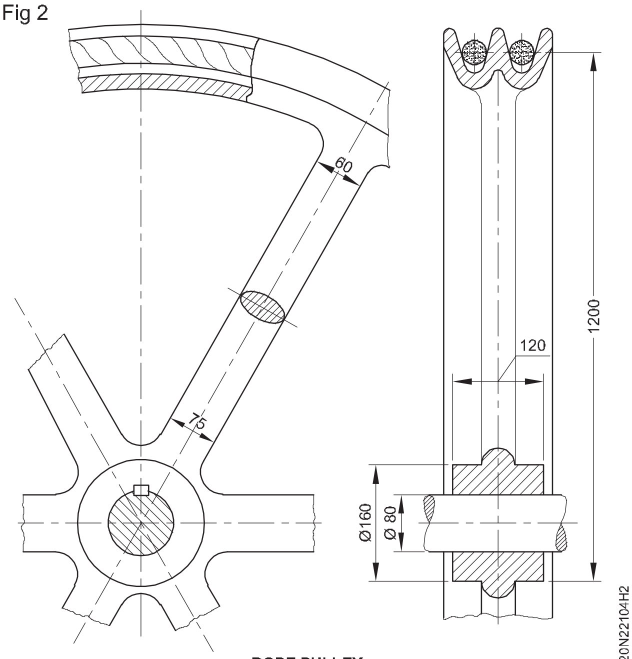
TASK 1: **Draw the rope pulley, V belt pulley using CAD (Figs 1&2)**

Fig 1



V-BELT PULLEY

Fig 2



ROPE PULLEY

— — — — —

Draw pipe fittings

Objectives: At the end of this exercise you shall be able to
- **draw pipe fittings.**

Requirements

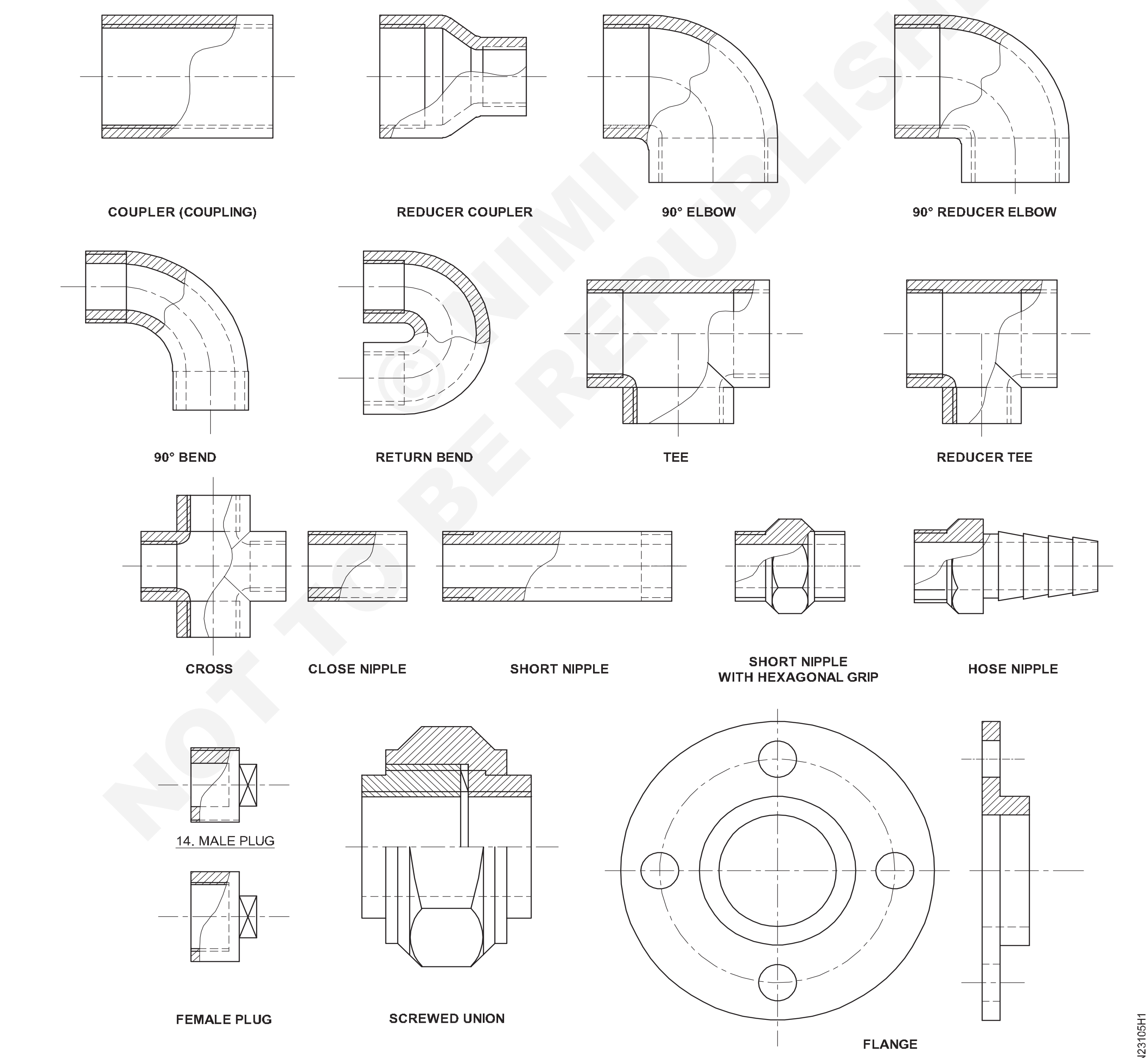
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw pipe fittings (Fig 1)**

Fig 1



PIPE FITTINGS (SPECIALS)

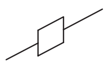
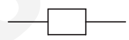
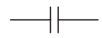
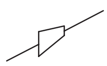
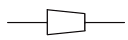
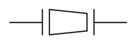
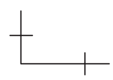
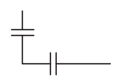
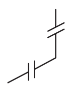
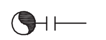
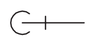
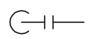
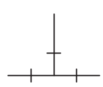
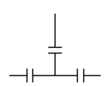
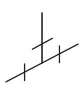
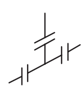
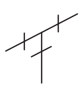
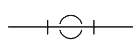
Draw conventional symbols of different types of valves and joints used on pipe line diagram

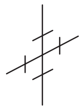
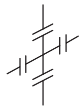
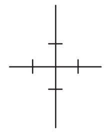
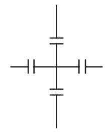
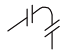
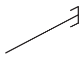
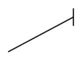
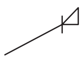
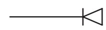
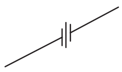
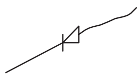
Objectives: At the end of this exercise you shall be able to
- **draw symbols for pipe fittings**
- **draw the symbols for valves**
- **draw pipes layout systems.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

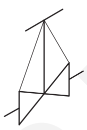
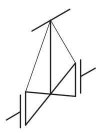
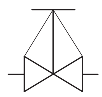
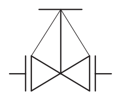
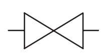
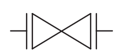
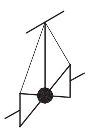
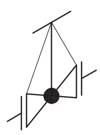
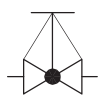
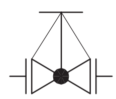
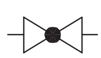
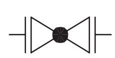
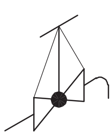
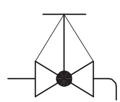
PROCEDURE

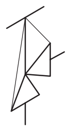
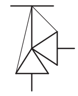
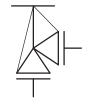
TASK 1: **Draw the symbols shown in the table below**

SL. No	Description	Isometric symbol (right face)		Orthographic symbol	
		Screwed	Flanged	Screwed	Flanged
1	Joint coupling				
2	Reducer				
3	90° elbow	— — —	— — —		
	i.Turned up				
	ii.Turned down				
4	Tee	— — —	— — —		
	i.Turned up				
	ii. Turned down				

SL. No	Description	Isometric symbol (right face)		Orthographic symbol	
		Screwed	Flanged	Screwed	Flanged
5	Cross				
6	Bend			— — —	— — —
7	Cap (female) / (dead end)				
8	Plug (male)		— — —		— — —
9	Union		— — —		— — —
10	Hose nipple		— — —		— — —

TASK 2: **Draw the symbols shown in the table below**

Sl.	Description	Isometric symbol (right face)		Orthographic symbol	
		Screwed	Flanged	Screwed	Flanged
1	Gate valve (Ele.)				
	Plan	— — —	— — —		
2	Globe valve (Ele)				
	Plan	— — —	— — —		
3	Water tap		— — —		— — —

Sl.	Description	Isometric symbol (right face)		Orthographic symbol	
		Screwed	Flanged	Screwed	Flanged
4	Angle valve (Ele)				
	Plan	— — —	— — —		

— — — — —

Draw the piping layout system

Objectives: At the end of this exercise you shall be able to
- **draw piping layout system from sump to a over head tank.**

Requirements

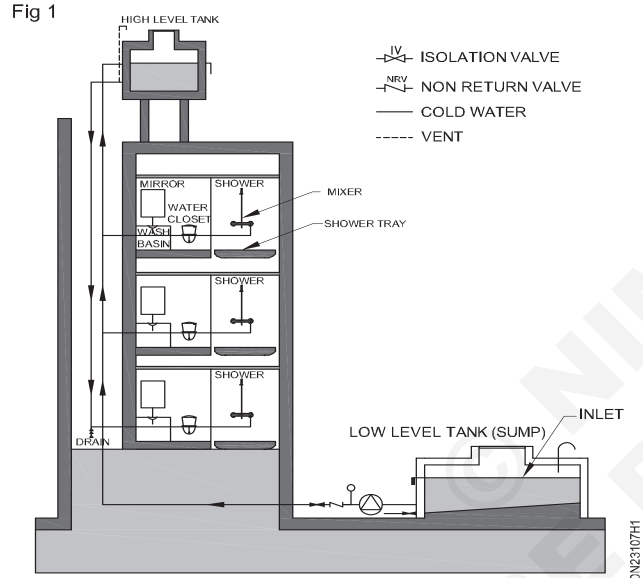
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the water supply gravity system shown in figure with suitable dimensions (Fig 1)**

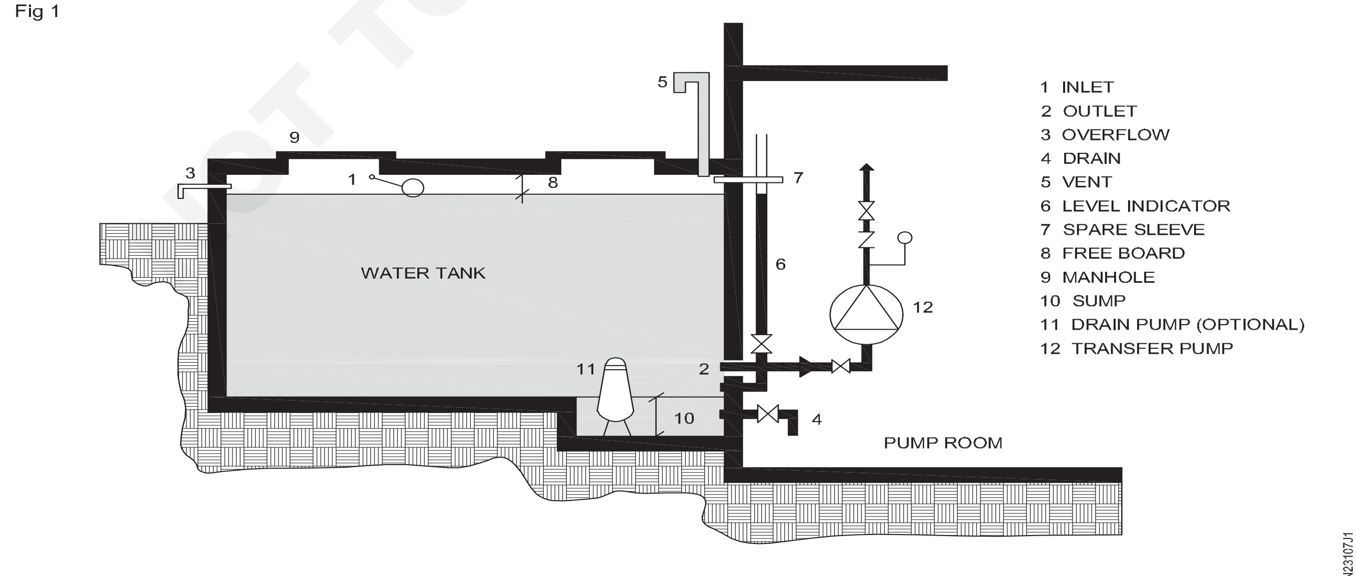
Fig 1



WATER SUPPLY-GRAVITY SYSTEM

TASK 2: **Draw the assemble view of the water tank with suitable dimensions shown in (Fig 1)**

Fig 1



WATER TANK DETAILS

Draw the sectional view of different types of pipe joints

Objectives: At the end of this exercise you shall be able to
- **draw the different types of pipe joints.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

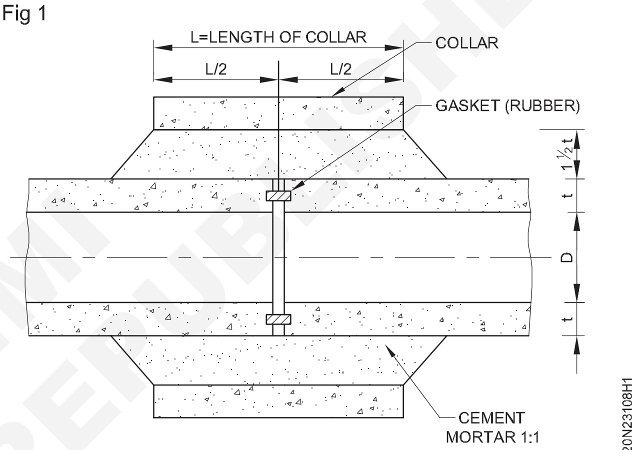
TASK 1: **Draw collar joint**

Collar joint (Fig 1)

D (Diameter of pipe) = 50 cm

t = (Thickness of pipe) = 5 cm

Fig 1



COLLAR JOINT

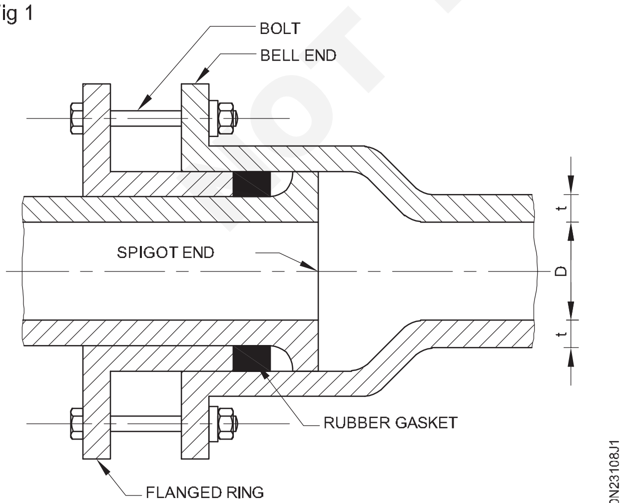
TASK 2: **Draw the typical pipe joints**

Draw the pipe joints as per given drawing

D = Diameter of pipe 10 cm

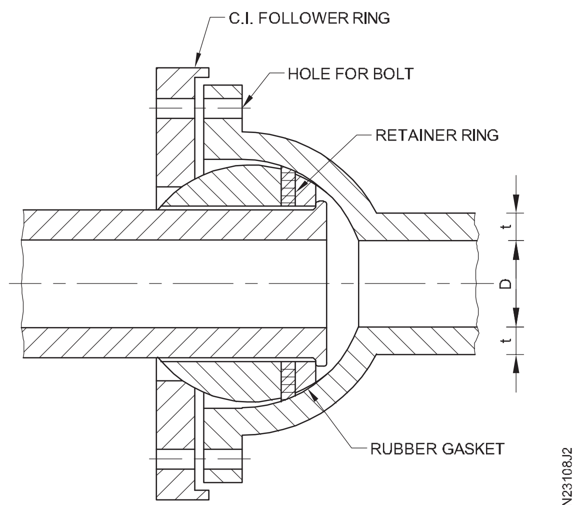
t = Thickness of pipe 1 cm (Figs 1 to 6)

Fig 1



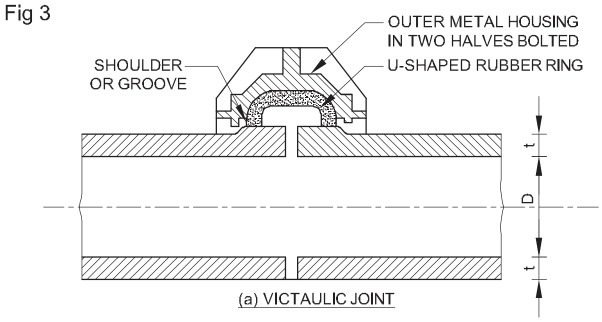
EXPANSION JOINT

Fig 2



FLEXIBLE JOINT (UNIVERSAL JOINT)

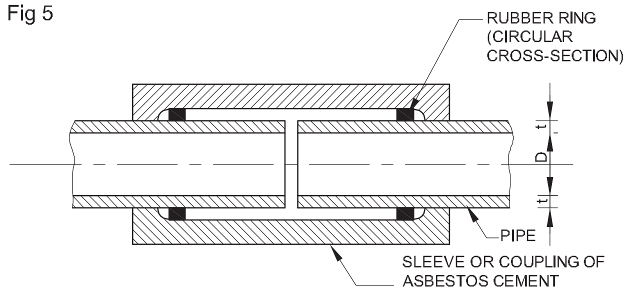
Fig 3



(a) VICTAULIC JOINT

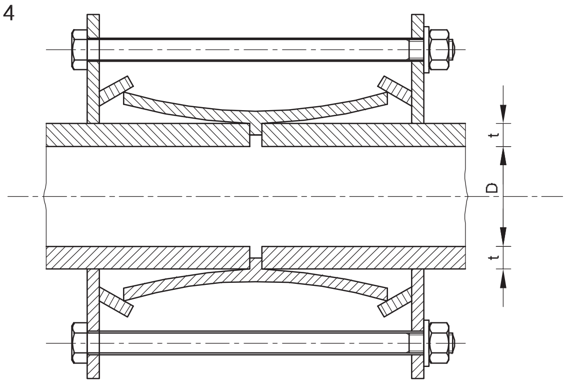
MECHANICAL JOINT

Fig 5



SIMPLEX JOINT

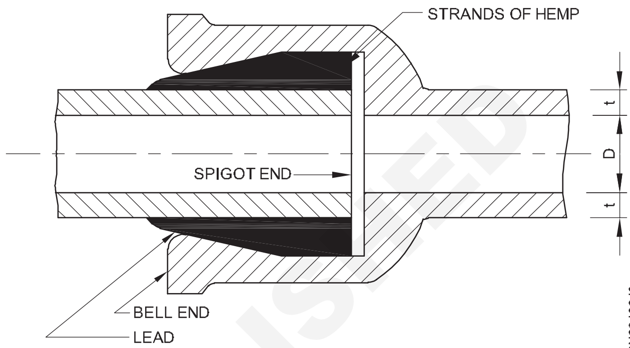
Fig 4



(b) DRESSER COUPLING JOINT

MECHANICAL JOINT

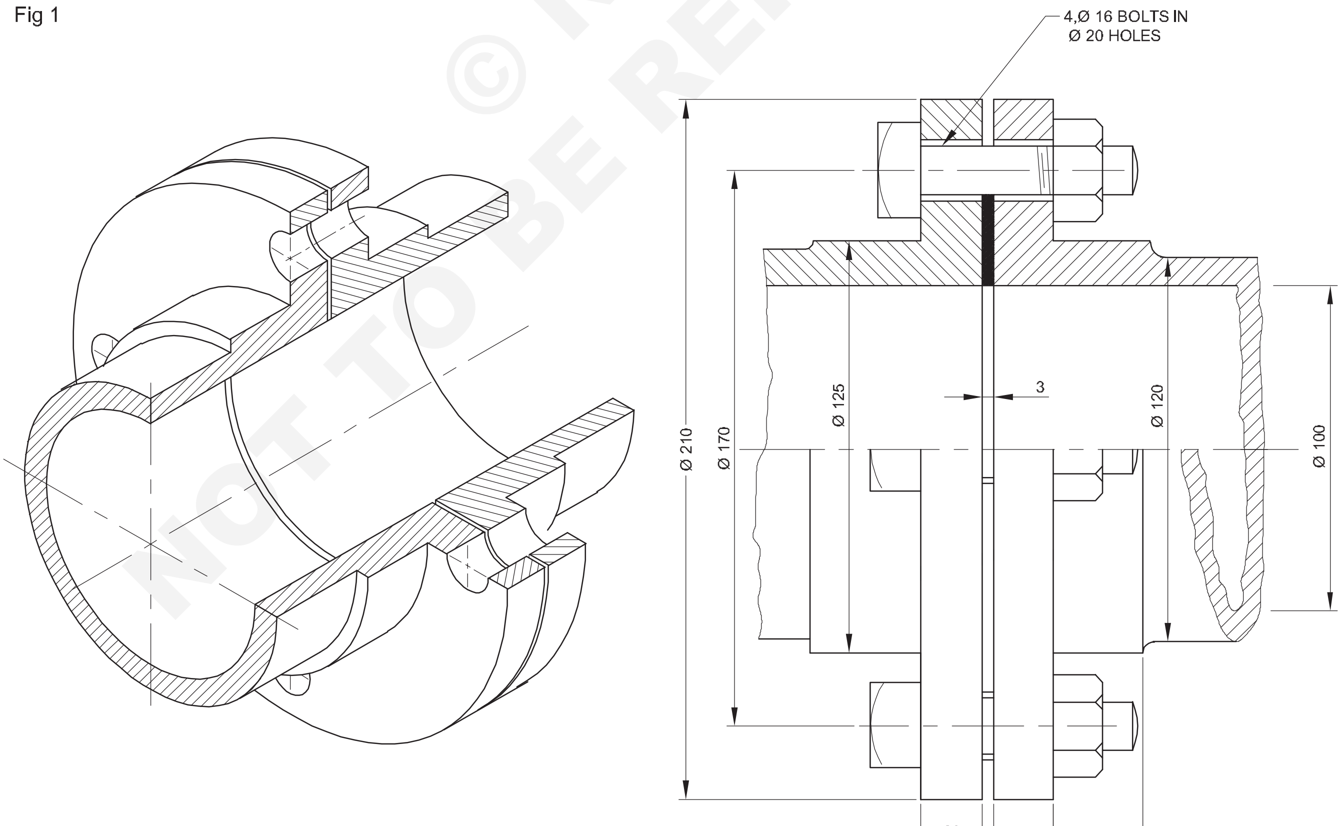
Fig 6



SPIGOT AND SOCKET JOINT

TASK 3: **Draw the cast iron for two 100 mm diameter (Fig 1)**

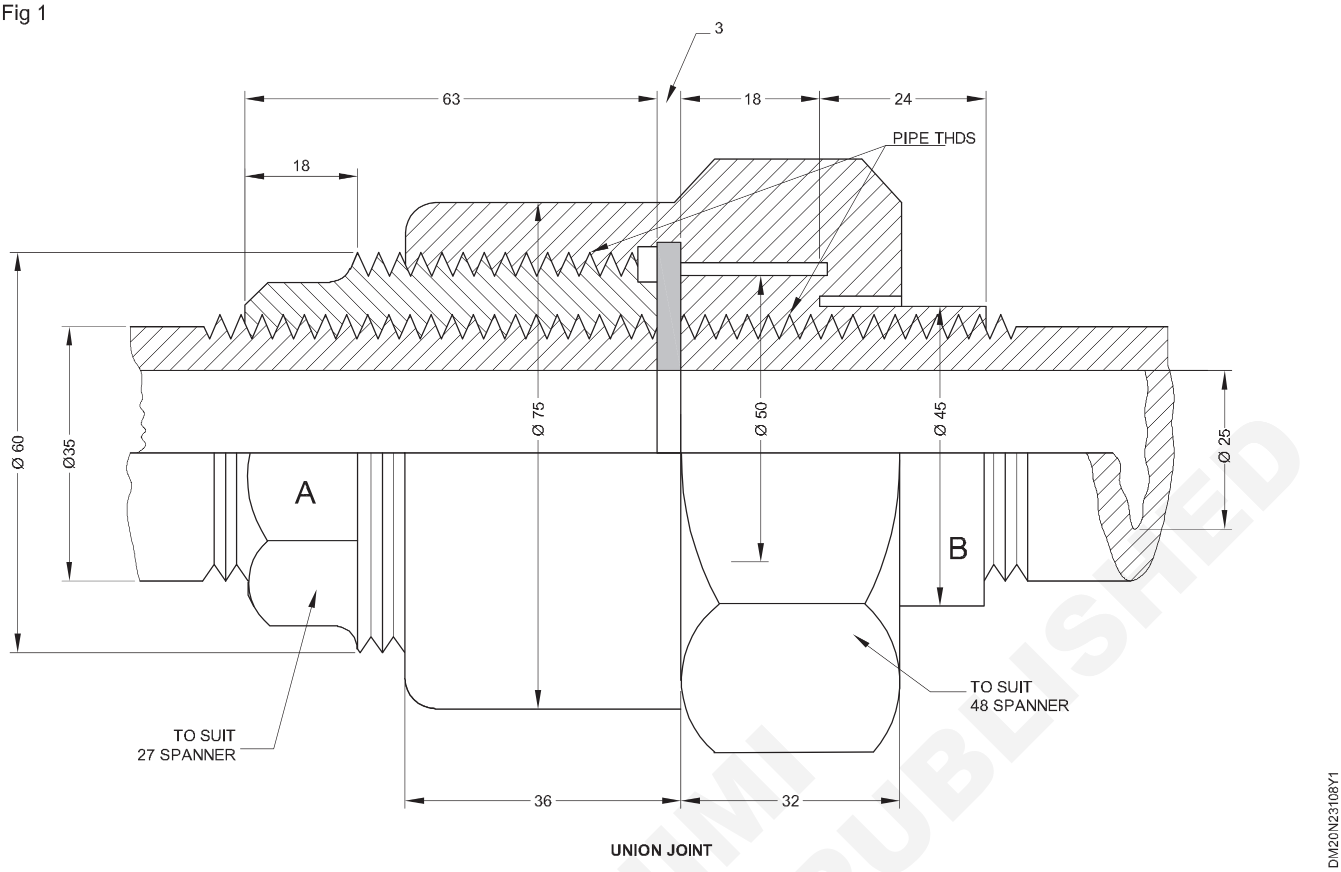
Fig 1



CAST-IRON FLANGED JOINT

TASK 4: **Draw full size two views of an union joint for 25 mm diameter pipes (Fig 1)**

Fig 1



UNION JOINT

Gears

Objectives: At the end of this exercise you shall be able to
- **gear calculation**
- **construction of base circles**
- **construct tooth profile of a spur gear above 30 teeth**
- **draw two spur gear in mesh**

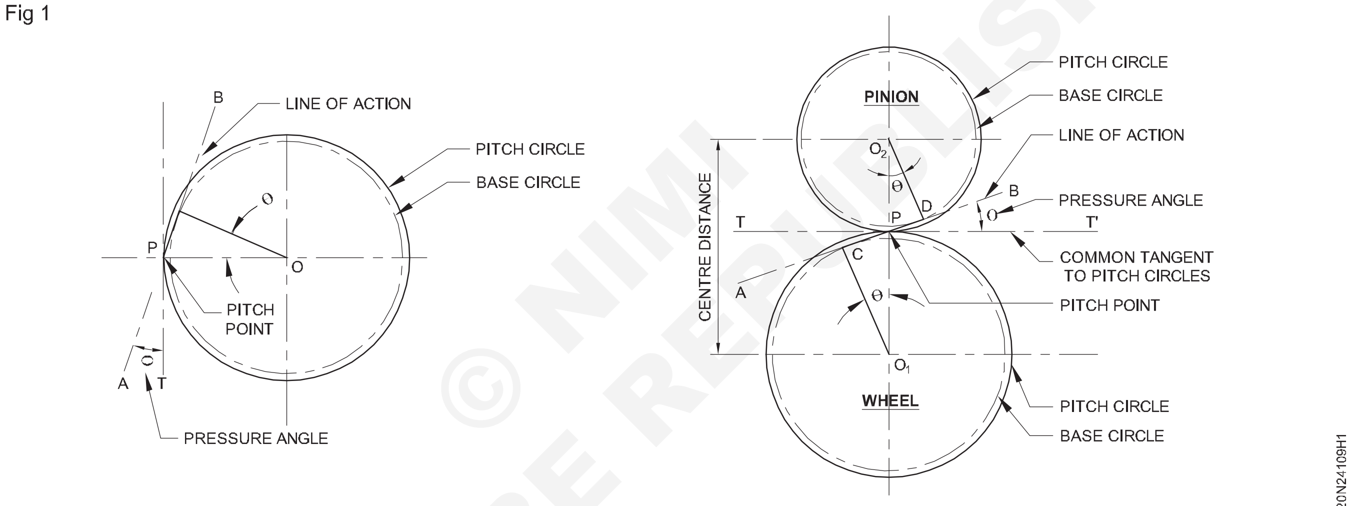
Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

Fig 1



METHOD OF DRAWING BASE CIRCLES

Gear calculation (Fig 1)

$$\text{Circulr pitch} = \frac{\text{Circumference of P.C.D}}{\text{No of teeth}}$$

$$= \frac{\text{Pitch circle diameter x } \pi}{\text{No of teeth}}$$

$$C.P = \frac{P.C.D \times \pi}{N}$$

Pitch diameter and number of teeth can also be found out from this relation.

$$\text{Diametral Pitch} = \frac{\text{No.of teeth}}{\text{Pitch circle diameter}}$$

$$D.P = \frac{N}{P.C.D}$$

Number of teeth or pitch circle diameter can also found from this relation

N = P.C.D x D.P

and $P.C.D = \frac{N}{D.P}$

$$C.P \times D.P = \frac{P.C.D \times \Pi}{N} \times \frac{N}{P.C.D} = \Pi$$

$$C.P = \frac{\pi}{D.P}$$

and $D.P = \frac{\pi}{C.P}$

Module pitch or metric module is generally used in metric system.

Metric module,

$$m = \frac{\text{pitch circle diameter}}{\text{Number of teeth}}$$

$$m = \frac{P.C.D}{N} = \frac{1}{D.P}$$

$$C.P = \frac{\pi}{D.P} = \pi \times m$$

Outside Diameter = Pitch diameter + 2 x addendum.

Root Diameter = Pitch Diameter - 2 x Dedendum

Clearance = Dedendum - Addendum

$$\text{Tooth Thickness} = \frac{C.P}{2}$$

$$\text{Tooth Thickness} = \frac{C.P}{2}$$

$$\text{Width of space} = \frac{C.P}{2}$$

$$\text{Addendum} = \frac{1}{D.P} = \frac{C.P}{\pi} = m$$

$$\text{Clearance} = \frac{C.P}{20} = \frac{\pi}{D.P} x \frac{1}{20} = 0.157 \times m$$

Exercise :1 Construct tooth profile of a spur gear above 30 teeth (Fig 2)

1 Construction of gear teeth suitable for gears of 30 teeth and above pressure angle 14.5°

Gear data - Metric module (m) 6; Number of teeth 30

Calculation data

Pitch circle diameter (P.C.D) = N x m = 30 x 6 = 180 mm

Addendum = m = 6 mm

Circular pitch (C.P) - π .m = 3.142 x 6 = 18.852 mm

$$\text{Clearance} = \frac{C.P}{20} = \frac{18.852}{20} = .9426$$

Dedendum = Addendum + Clearance

=6 + .9426 = 6.9426 mm

Tooth thickness = C.P/2 = 9.426 mm

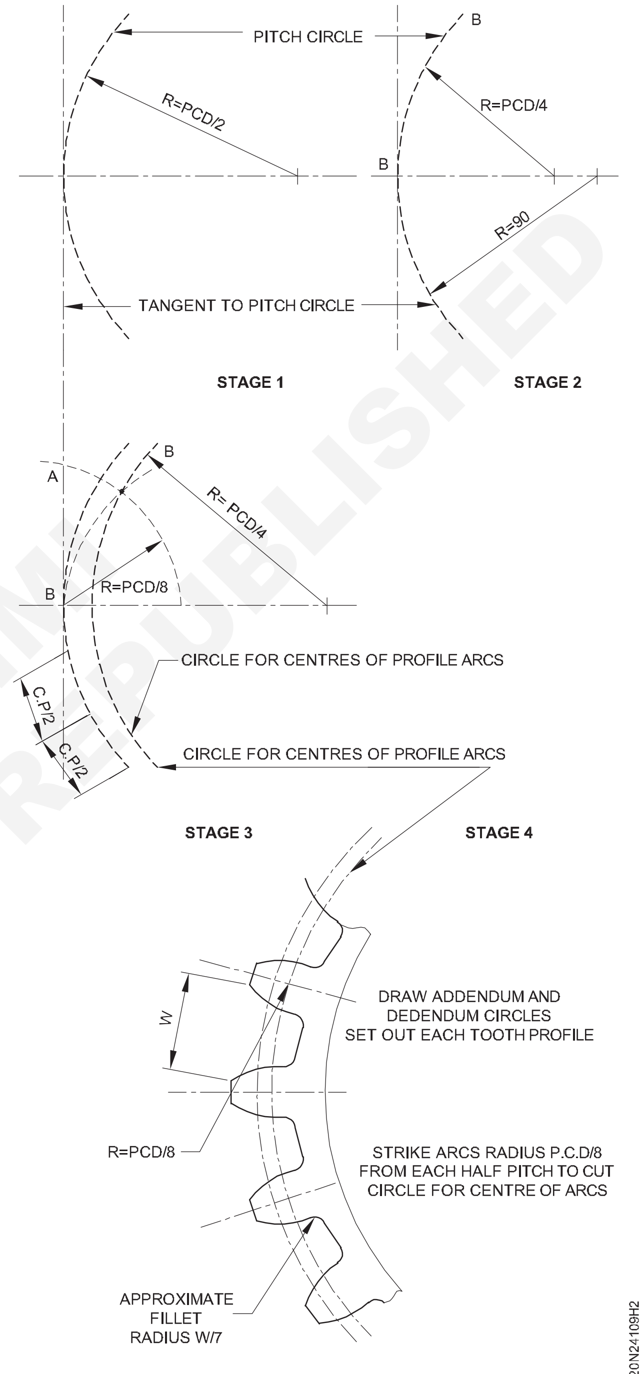
Exercise 2: Layout a pair of spur standard gears, given the following data:

Number of teeth on large gear = 24

Number of teeth on small gear = 16

Metric module m = 8

Fig 2



APPROXIMATE CONSTRUCTION METHOD (FOR 30 TEETH AND OVER)

Pressure angle ϕ = 20°

Calculations

Large gear (Fig 3)

P.C.D = m x N = 8 x 24 = 192 mm

C.P = π x m = π x 8 = 25.133 mm

Add = $\frac{C.P}{\pi} = 8mm$

Add. Circle dia = P.C.D + 2 x 8

= 192 + 16 = 208 mm

Clearance = $\frac{C.P}{20} = 1.257$ mm

Dedendum = Add. + Clearance

= 8 + 1.257 = 9.257 mm

Ded. Circle Dia. = P.C.D - 2 x Ded.

= 192 - 2 x 9.257

= 173.49 mm

Tooth thickness = $\frac{C.P}{2} = 12.566$ mm

Small Gear

P.C.D. = 8 x 16 = 128 mm

C.P = π x 8 = 25.133 mm

Addendum = 8 mm

Add. Circle Dia. = P.C.D + 2 x Add.

= 128 + 16 = 144 mm

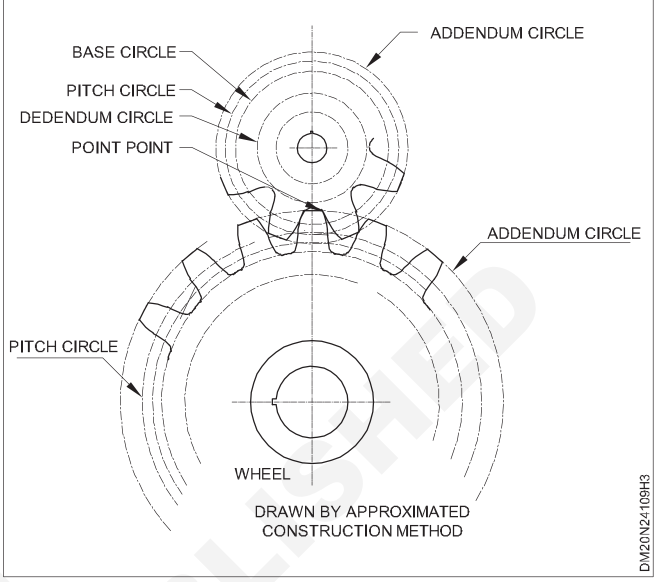
Clearance = 1.257 mm

Dedendum = 8 + 1.257 = 9.257 mm

Ded. Circle Dia. = P.C.D. - 2 x Ded.

= 109.486 mm

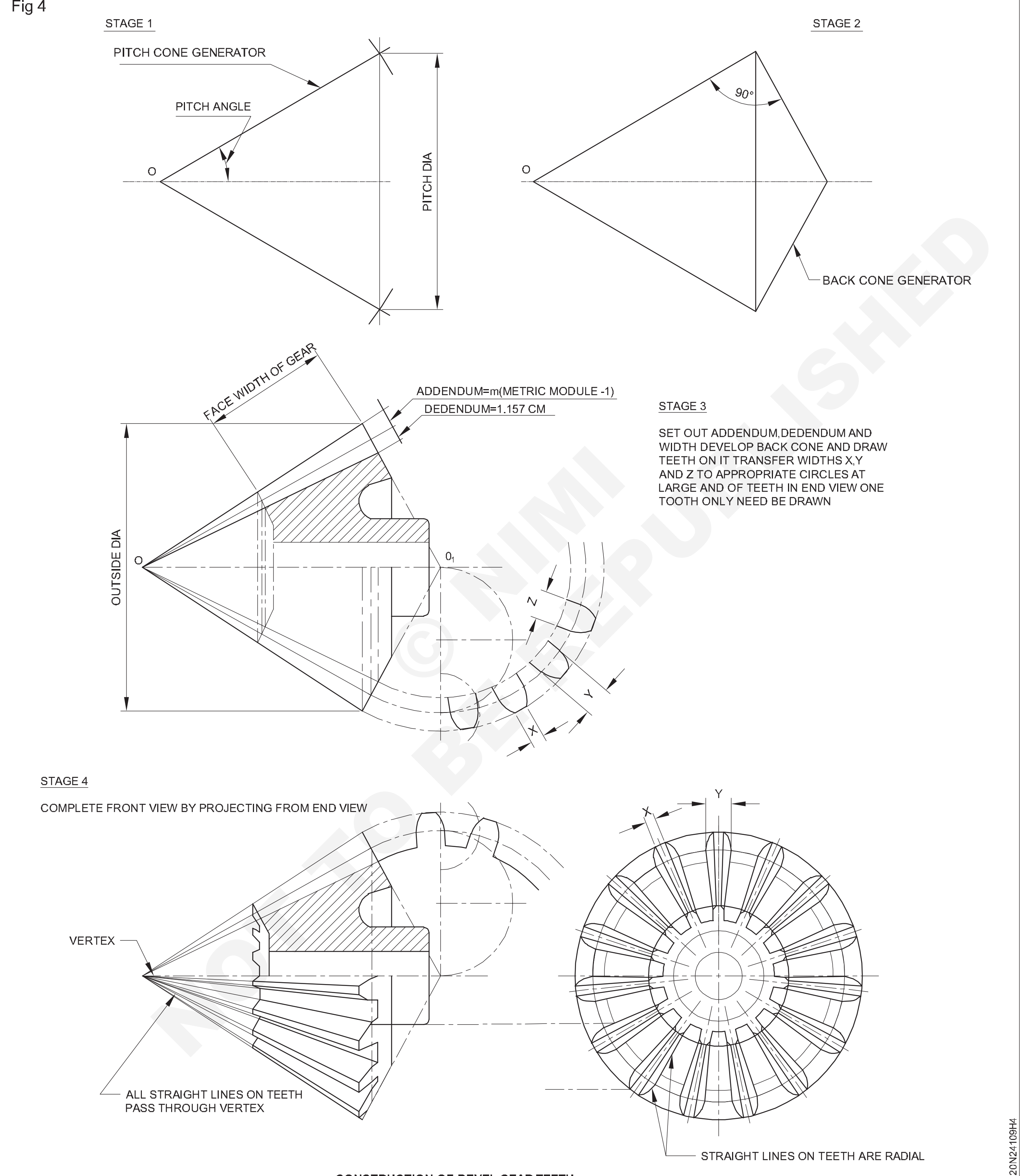
Fig 3



Exercise :3 Construct of bevel gear teeth to the given date gear data:

Number of teeth 15 pitch cone angle 15°pitch circle dia 105mm pressure angle 14.5°illustration drawing given below. (Fig 4)

Fig 4



CONSTRUCTION OF BEVEL GEAR TEETH

Exercise :4 Draw a Bevel gear

Draw front and top view of a bevel gear having the following the specification (Fig 5)

No. of teeth = 25

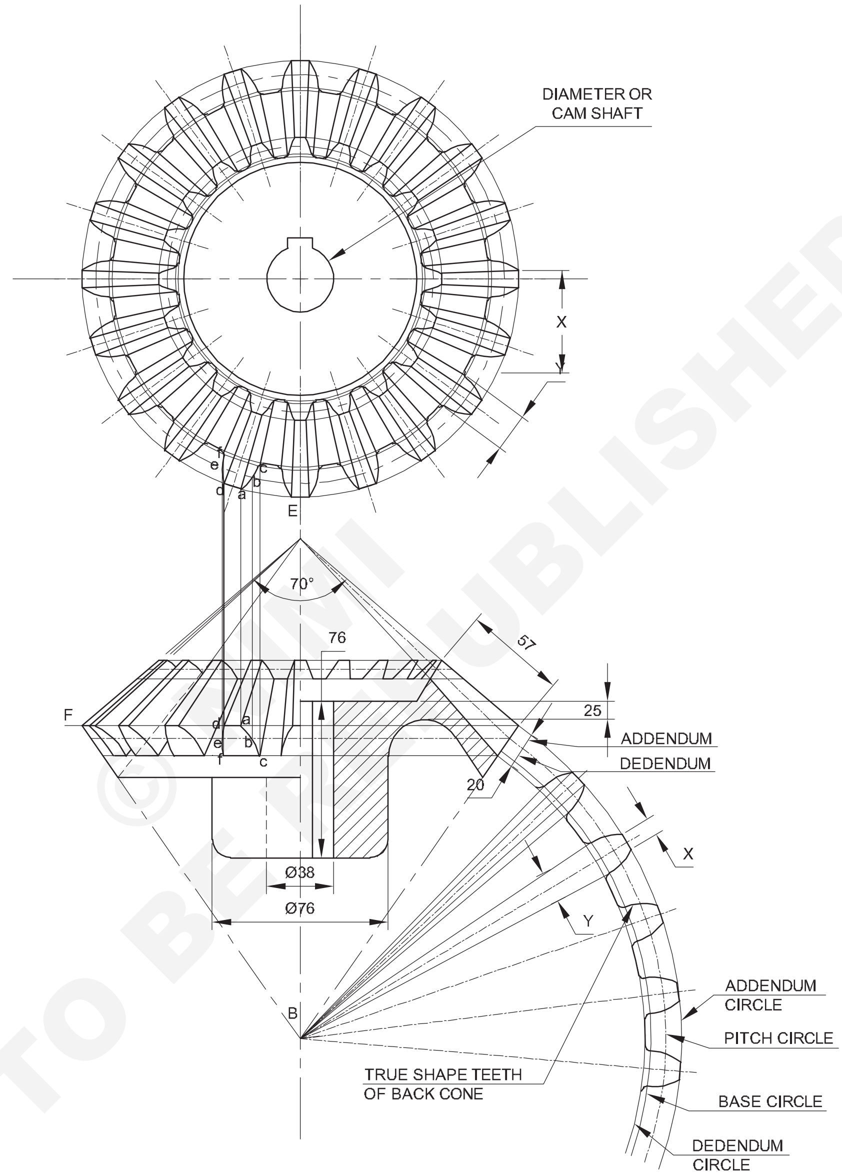
C.P = 25 mm

Face width = 57 mm

Pitch cone angle = 70°

$$\text{Diameter of pitch cone} = \frac{\text{No. of teeth x C.P}}{\pi}$$

Fig 5



BEVEL GEAR

Construct involute tooth profile of a gear

Objectives: At the end of this exercise you shall be able to
- **draw involute profile of gear using CAD.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

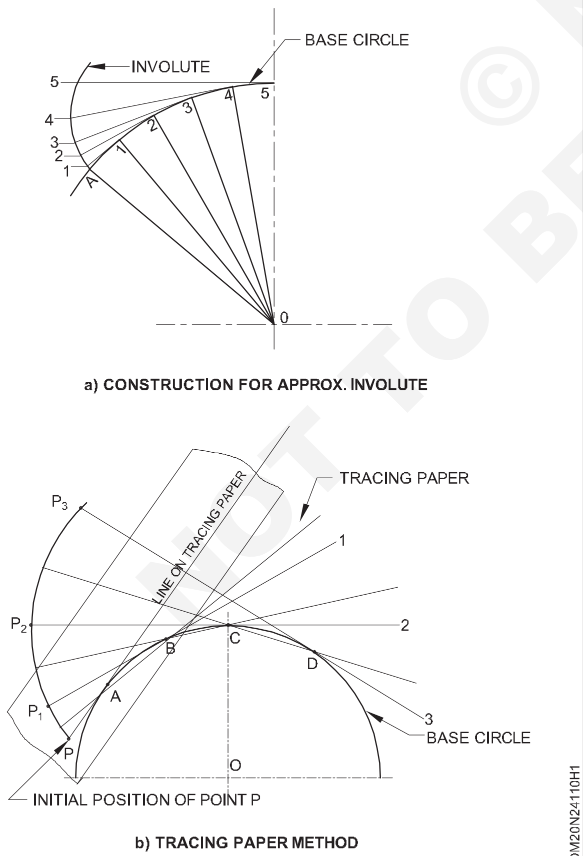
PROCEDURE

TASK 1: **Draw involute profile of gear using CAD**

1 Construction for approximate involute

Describe the base circle. Cut-off any number of equal spaces along the base circle, i.e, A-1, 1-2, 2-3 etc. Join these points with the centre of the base circle. Draw tangents to the base circle from these points. Now with centre 1 and radius equal to 1A draw an arc to cut the first tangent at 1'. Again with 2 as centre and radius equal to 2 A draw an arc to cut the second tangent at 2' and so on. The Curve joining 1', 2', 3' etc. Will be the involute curve (Fig 1).

Fig 1



a) CONSTRUCTION FOR APPROX. INVOLUTE

b) TRACING PAPER METHOD

2 Approximate constructions for the involute tooth profile

Since accurate drawing of actual involute curves on the gear teeth is a lengthy procedure, it is usual to represent those involute curves approximately by circular arcs.

Four such approximate methods are shown here in Fig1,Fig 2, Fig 3. The first two methods differ only at one point, depending on the number of teeth on the gear. First draw the pitch circle of diameter D and a tangent to it to fix the pitch point. On the pitch circle radius draw a semi circle of radius D/4 to cut a P an arc of radius D/8 drawn with the pitch point as centre. On this circle lie the centres of the profile arcs of the teeth Fig 1, Then draw the addendum and dedendum circles and from the pitch point mark off tooth widths C.P/2 around the pitch circle. From these points draw arcs of radius D/8 to the circle for the centres of the profile areas. Then:

1 For a gear of 30 teeth and over (Fig 1) draw the profile arcs radius D/8 with these points as centres from the addendum circle to the Dedendum circle and complete the teeth with fillet radii equal to one-seventh of the widest tooth space.

2 For a gear with less than 30 teeth draw the profile arcs as in 1 above and draw radial lines tangential to them before adding the fillet radii. This method avoids having teeth, which appear to be excessively undercut.

3 The third method can be used for a gear having any number of teeth and makes use of the base circle. This construction is more suitable for teeth 20° pressure angle than the methods described in 1 and 2 as it produces teeth which have wider roots Fig 3.

- Draw the pitch, addendum and dedendum circles, and the base circle tangential to the line of action.
- On a convenient radial line, fix a point A on the addendum circle and a point E on the base circle.
- Divide AE into three equal parts and through B, the division nearest A, draw a tangent to the loose circle at D.

- Divide BD into four equal parts and through F, the division nearest to F draw circle. The centre for F is the centre for the profile arc passing through B.
- From C, the intersection of this profile arc and the pitch circle, mark off teeth thickness round the circle.

Construction of gear teeth suitable for gears of 30 teeth and above pressure angle 14.5°

Gear data - Metric module (m)6; Number of teeth 30

Calculation Data

Pitch circle diameter (P.C.D) = N x m = 30 x 6 = 180 mm

Addendum = m = 6 mm

Circular pitch (C.P) = m = π x 6 = 18.852

$$\text{Clearance} = \frac{C.P}{20} = \frac{18.852}{20} = 0.9426$$

Dedendum = Addendum + Clearance

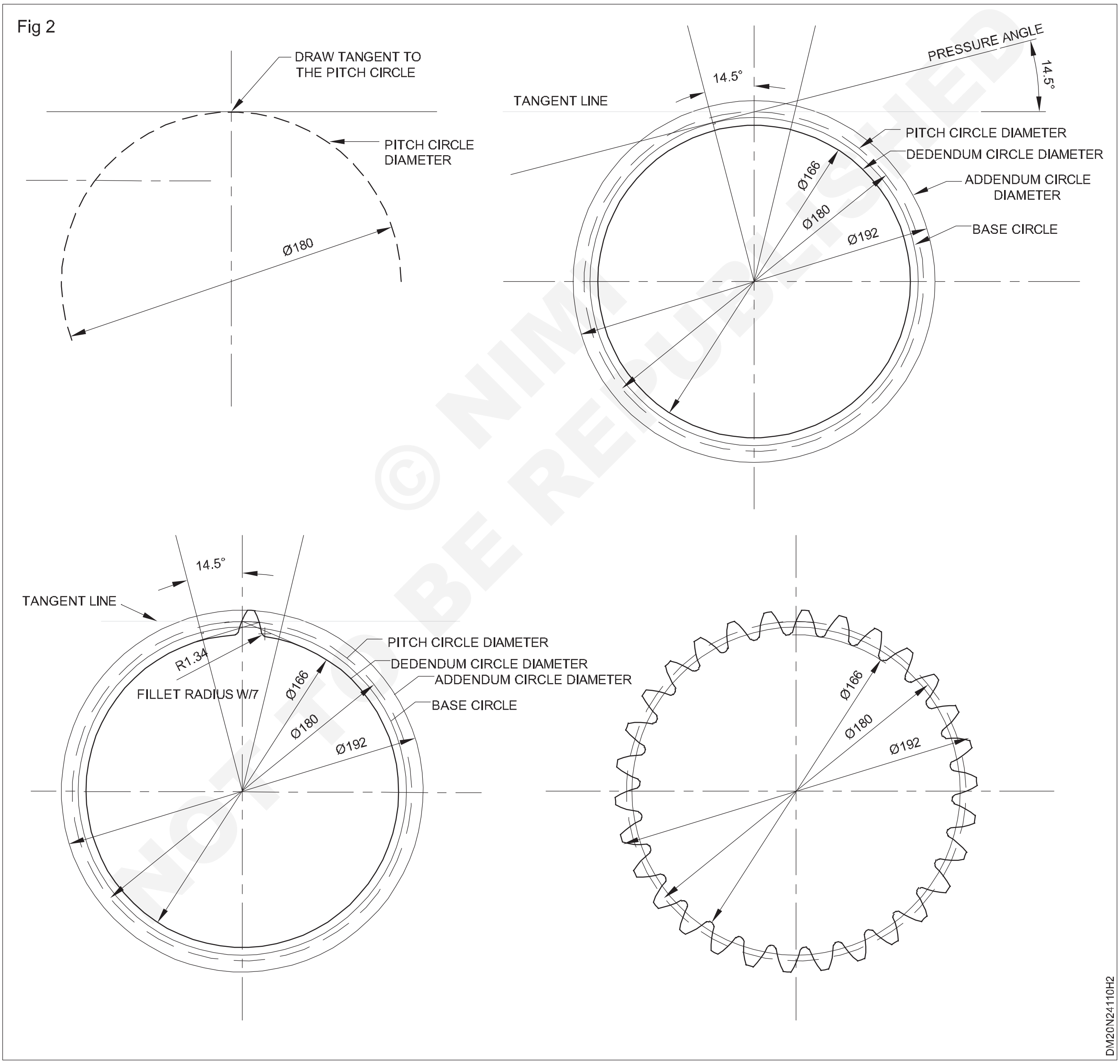
= 6 + 0.9426 = 6.9426

Tooth thickness = C.P/2 = 9.426 mm

Addedum circle dia = PCD + 2 $\times$ Addendum

Dedendum circle dia = PCD - 2 × Dedendum

Fig 2



3 Construction of gear teeth using base circle as a basis of construction

Gear data - Pressure angle 20°, Metric module (m)10, number of teeth 25.

Calculation data

Pitch circle Diameter = D = m.N. = 10 x 25 = 250 mm

Addendum = m = 10mm

$$\text{Circular pitch} = C.P = \frac{pD}{N} = \pi.m = 3.142 \times 10 = 31.42\ \text{mm}$$

Clearance = C.P/20 = 31.42/20 = 1.571 mm

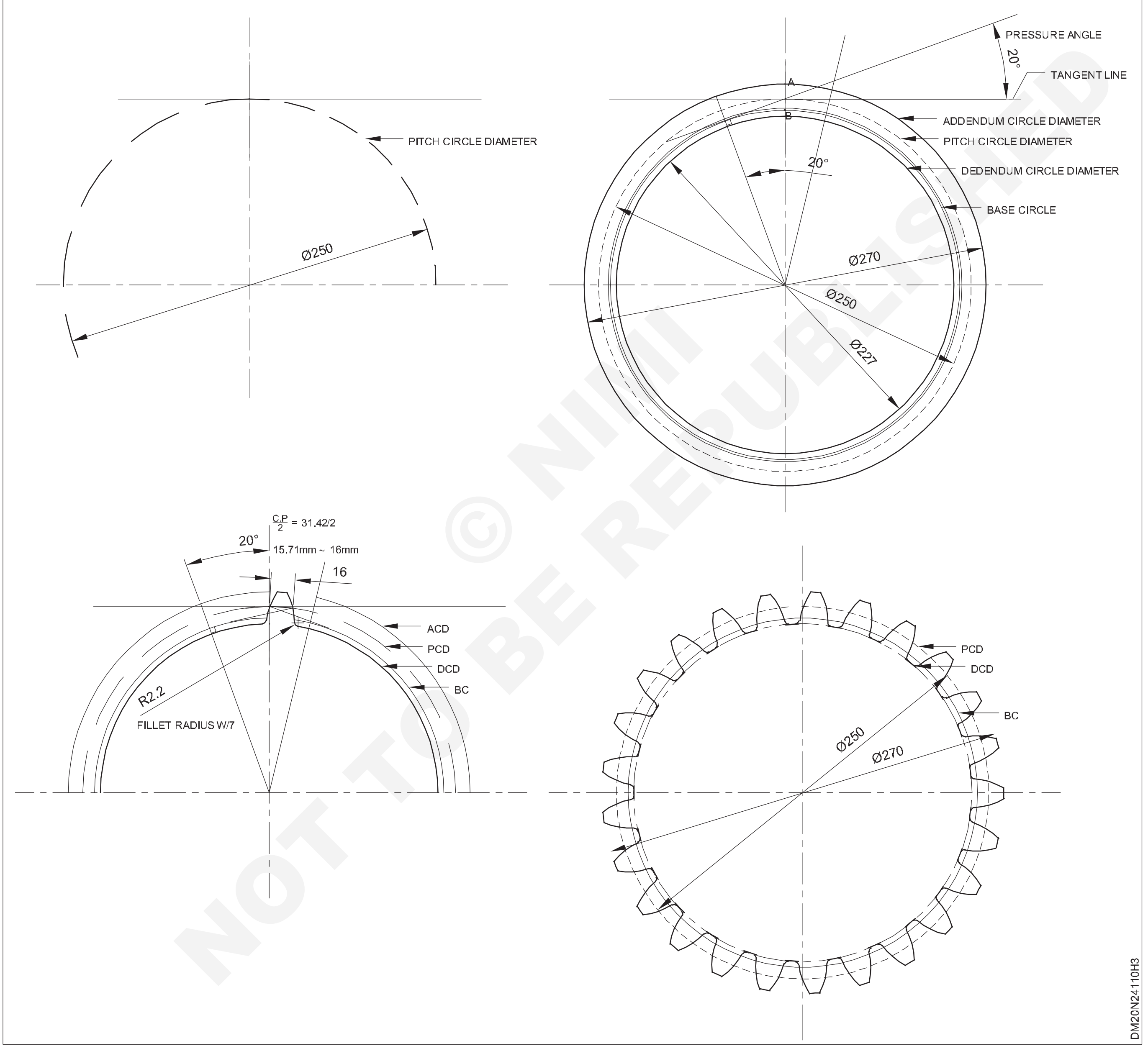
Dedendum = Addendum + Clearance = 10 + 1.571 = 11.571 mm

Tooth thickness = C.P/2 = 31.42/2 = 15.71 mm

Addedum circle dia = PCD + 2 × Addendum

Dedendum circle dia = PCD - 2 × Dedendum

Fig 3



Draw a symmetrical cam profile

Objectives: At the end of this exercise you shall be able to
- **draw cam profile.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw cam profile (Fig 1)**

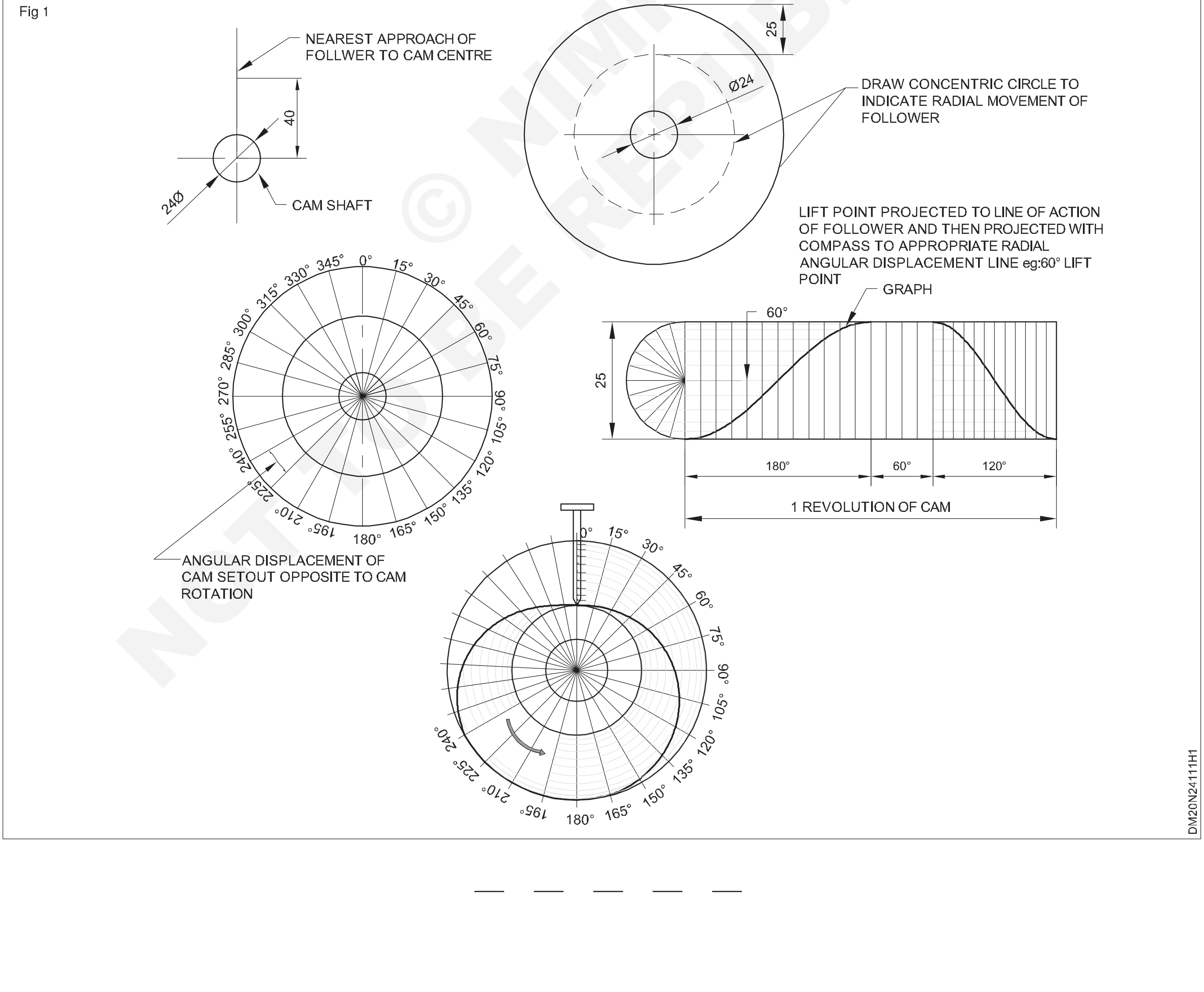
1 Construct a plate cam nith knife edge follower to the given data

Cam data: 0° - 180° Lift 25 mm with S.H.M,

240° - 360° Fall 25 mm with acceleration and retardation at uniform rate. Rotation of CAM Anti - Clockwise

Least Thickness of metal around CAM centre 41 mm. Dia of camshaft 24 mm.

Fig 1



TASK 2: **Construct a cam profile with inline roller follower to the given dat**a

A cam, with a minimum radius of 25 mm, rotating clockwise at a uniform speed is to be designed to give a roller follower, at the end of a valve rod, motion described below:

1 To raise the valve through 50 mm during 120° rotation of the cam;

2 To keep the valve fully raised through next 30°;

3 To lower the valve during next 60°; and

4 To keep the valve closed during rest of the revolution i.e.150°;

The diameter of the roller is 20 mm and the diameter of the cam shaft is 25 mm.

Draw the profile of the cam when (a) the line of stroke of the valve rod passes through the axis of the cam shaft, and (b) the line of the stroke is offset 15 mm from the axis of the cam shaft.

The displacement of the valve, while being raised and lowered, is to take place with simple harmonic motion. Determine the maximum acceleration of the valve rod when the cam shaft rotates at 100 r.p.m.

Draw the displacement, the velocity and the acceleration diagrams for one complete revolution of the cam.

Solution: Given: S = 50 mm = 0.05 mm; θ_O = 120° = π/3 rad = 2.1 rad; θ_R = 60° = π/3 rad = 1.047; N = 100 r.p.m

Since the valve is being raised and lowered with simple harmonic motion, therefore the displacement diagram, as shown in Fig 1, is drawn in the similar manner as discussed in the previous example.

a Profile of the cam when the line of stroke of the valve rod passes through the axis of the cam shaft

The profile of the cam, as shown in Fig 1 is drawn as discussed in the following steps:

1 Draw a base circle with centre O and radius equal to the minimum radius of the cam (i.e. 25 mm).

2 Draw a prime circle with centre O and radius,

$$OA = \text{Min. radois of cam} + \frac{1}{2}\text{Dia. of roller} = 25 + \frac{1}{2} \times 20 = 35 \text{ mm}$$

3 Draw angle AOS = 120° to represent raising or out stroke of the valve, angle SOT = 30° to represent dwell and angle TOP = 60° to represent lowering or return stroke of the valve.

4 Divide the angular displacements of the cam during raising and lowering of the valve (i.e. angle AOS and TOP) into the same number of equal even parts as in displacement diagram.

5 Join the points 1,2,3 etc. with the centre O and produce the lines beyond prime circle as shown in Fig 1.

6 Set off 1B, 2C,3D etc. equal to the displacements from displacement diagram.

7 Join the points A, B,C ...N,P,A. The curve drawn through these points is known as pitch curve.

8 From the points A, B, C....N,P, draw cricles of radius equal to the radius of the roller.

9 Joint the bottoms of the circles with a smooth curve as shown in Fig 1. This is required profile of the cam.

b Profile of the cam when the line of stroke is offset 15 mm from the axis of the cam shaft

The profile of the cam when the line of stroke is offset from the axis of the cam shaft, as shown in Fig 1, may be drawn as discussed in the following steps:

1 Draw a base circle with centre O and radius equal to 25 mm.

2 Draw a prime circle with centre O and radius OA = 35 mm.

3 Draw an off-set circle with centre O and radius equal to 15 mm.

4 Join OA . From OA draw the angular displacement of cam i.e. draw angle AOS = 120° angle SOT = 30° and angle TOP = 60°.

5 Divide the angular displacements of the cam during raising and lowering of the valve into the same number of equal even parts(i.e. six parts) as in displacement diagram.

6 From points 1,2,3 .... etc and 0', 1', 3', ... etc. on the prime circle, draw tangents to the offset circle.

7 Set off 1B, 2C, 3D..., etc. equal to displacements as measured from displacement diagram.

8 By joining the points A, B, C .... M, N,P, with a smooth curve, we get a pitch curve.

9 Now A, B,C ... etc. as centre, draw circles with radius equal to the radius of roller.

10 Join the bottoms of the circles with a smooth curve as shown in Fig 1. This is the required profile of the cam.

Maximum acceleration of the valve rod

We know that angular velocity of the cam shaft,

$$\omega = \frac{2\pi N}{60} = \frac{2\pi \times 100}{60} = 10.47 \text{ rad/s}$$

We also know that maximum velocity of the valve rod to raise valve,

$$\nu O = \frac{\pi\omega S}{2\theta_O} = \frac{\pi \times 10.47 \times 0.05}{2 \times 2.1} = 0.39 \text{ m/s}$$

and maximum velocity of the valve rod to lower the valve,

$$\nu R = \frac{\pi\omega S}{2\theta_R} = \frac{\pi \times 10.47 \times 0.05}{2 \times 1.047} = 0.785 \text{ m/s}$$

The velocity diagram for one complete revolution of the cam is shown in Fig 1. We know that the maximum acceleration of the valve rod to raise the valve,

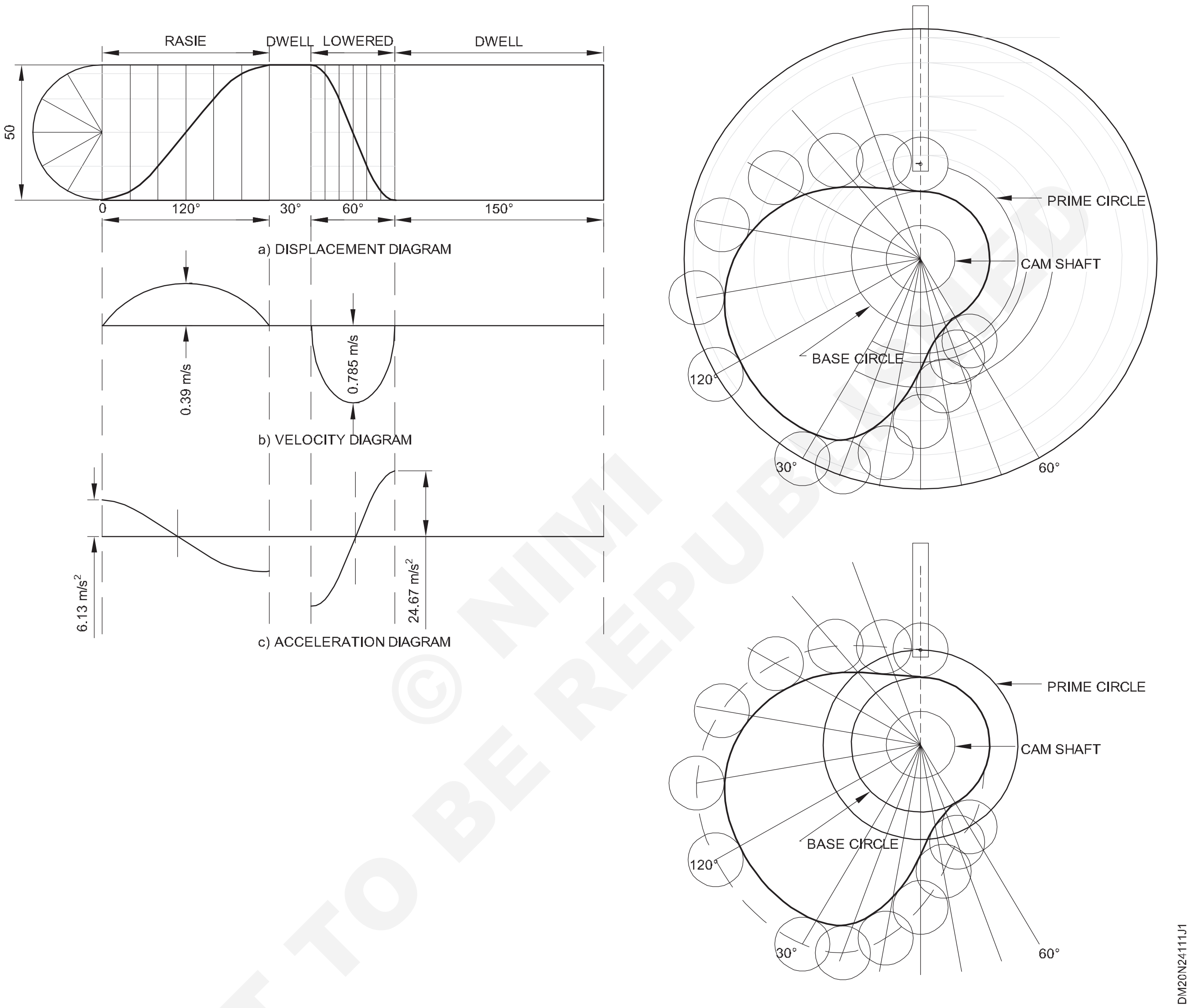
$$a_O = \frac{\pi^2 \omega^2 .S}{2(\theta_0)^2} = \frac{\pi^2 (10.47)^2\ 0.05}{2\ (2.1)^2} = 6.13\ m/s^2 \text{Ans.}$$

and maximum acceleration of the valve rod to lower the valve,

$$a_R = \frac{\pi^2 \omega^2 .S}{2(\theta_R)^2} = \frac{\pi^2 (10.47)^2\ 0.05}{2\ (1.047)^2} = 24.67\ m/s^2 \text{Ans.}$$

The acceleration diagram for one complete revolution of the cam is shown in Fig 1.

Fig 1



— — — — —

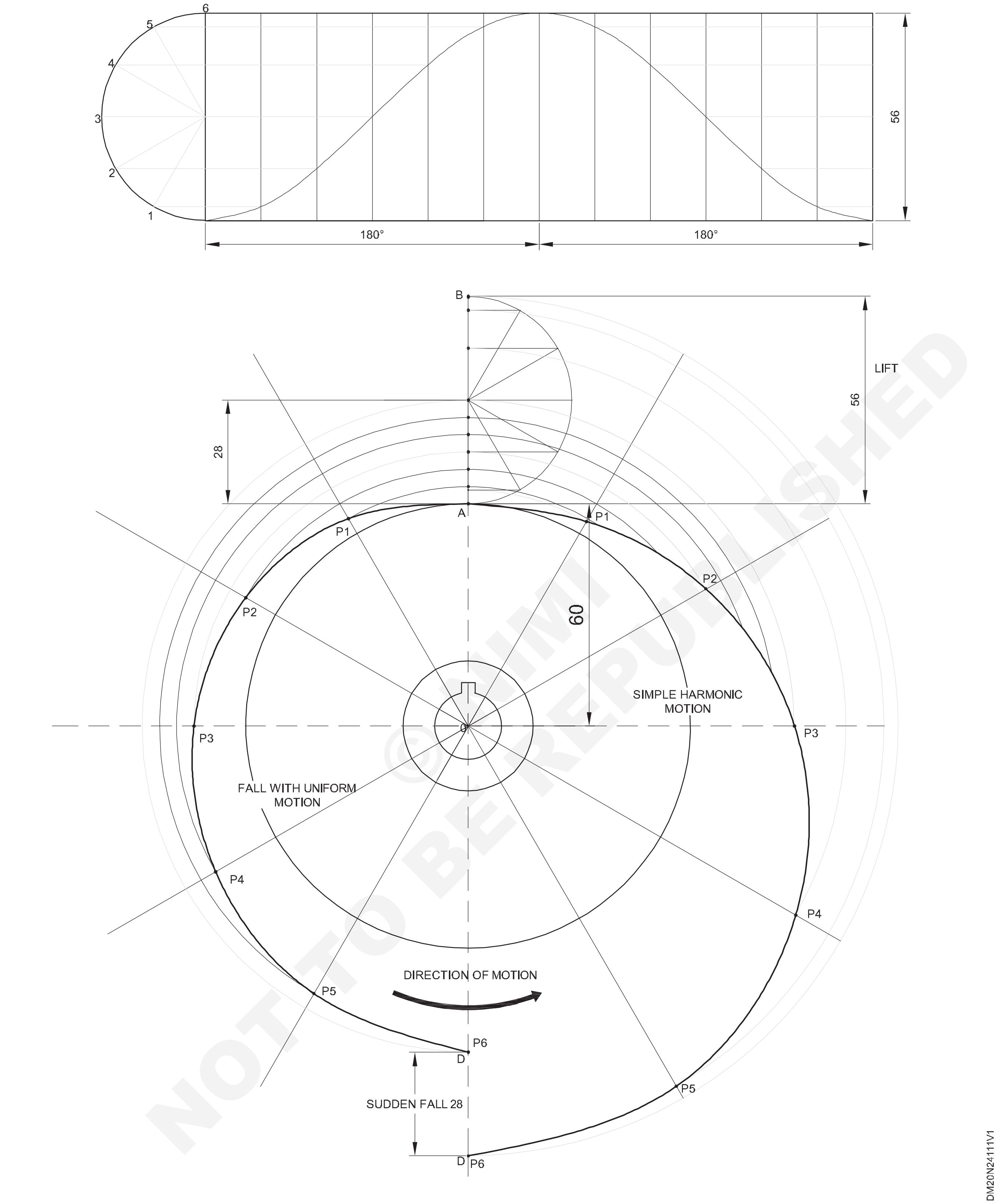
TASK 3: **Design an edge cam to the following particulars (Fig 1)**

- Minimum distance of centre of cam from edge of follower 60 mm, lift 56 mm. The cam is to lift the follower with simple harmonic motion during one half of revolution, and then allows it suddenly to drop halfway, then to fall with uniform motion during the remaining half of the revolution.

- **Construction:** Through O draw intersecting centrelines. Mark off OA = 60 mm, AB = 56 mm. On AB describe a semicircle and divide its circumference into six equal/sets. From each of these drop a perpendicular to AB. We thus, divide the circumference into six equal parts from each of the drop a perpendicular to AB. Thus we divide rise •into parts proportional to harmonic motion. This harmonic motion applies to 180° therefore divide the first 180° in the same number of angular parts (six in this case) as was chosen when dividing the lift. Let these radii intersect the arcs from points 1 to 6 (on the lift) in points P_1 and P_6. Draw smooth curve through these points.

- Mark off CD equal to half AB. The curve form D to A is part of an Archimedean spiral. For uniform motion the follower must rise or fall through equal distance for equal angles of rotation by the cam.

Fig 1



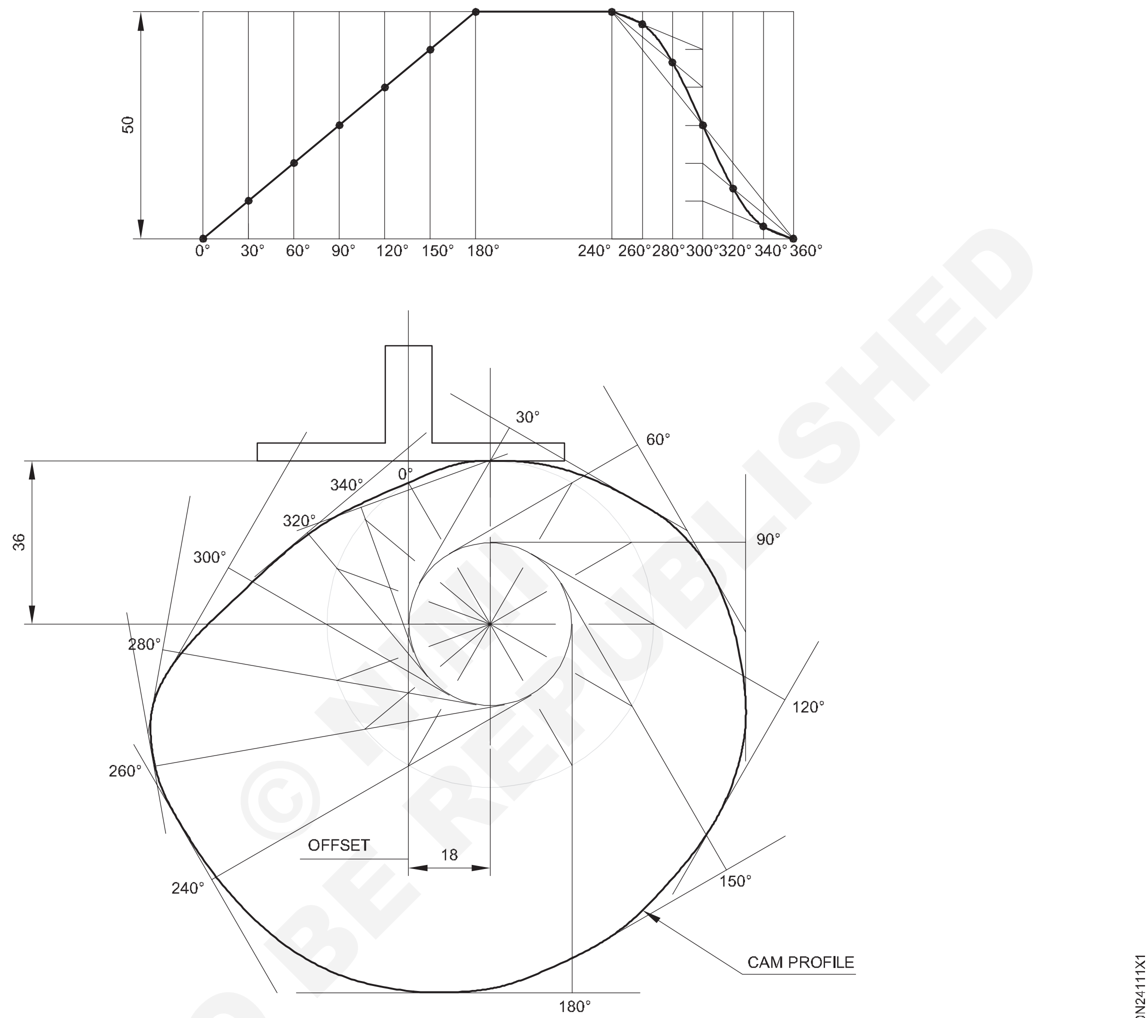
6
5
4
3
2
1
56
180°
180°
B
LIFT
56
28
A
P1
P1
P2
P2
60
SIMPLE HARMONIC
MOTION
P3
P3
0
FALL WITH UNIFORM
MOTION
P4
P4
DIRECTION OF MOTION
P5
P5
P6
D
SUDDEN FALL 28
D
P6
DM20N24111V1

— — — — —

TASK 4: **Layout, full size, the profile of a plate cam to give an offset flat follower the specified motion (Fig 1)**

- 0° - 180° lift of 50mm with uniform velocity.
- 180° - 240° dwell.
- 240° - 360° fall of 50 mm with uniform acceleration and retardation.
- The follower line of action is offset by 18 mm to the left of the cam shaft centre. The neatest approach of the follower face to the camshaft centre is 36 mm and the cam rotates anticlockwise.

Fig 1



TASK 5: **Construct a plate cam profile with an offset roller follower to the given data (Fig 1)**

CAM data: Rotation of cam clockwise

- Dia of camshaft 26 mm. Nearest approach of roller centre to cam centre 40 mm.
- Line of action of follower offset 20 mm to right of cam C/L Roller follower ϕ 20.
- 0° - 105° Lift 25 mm with uniform velocity. 105° - 150° Dwell period
- 150° - 240° Lift 13 mm S.H.M.,
- 240° - 360° Return to start position with uniform
- Acceleration and retardation

Fig 1

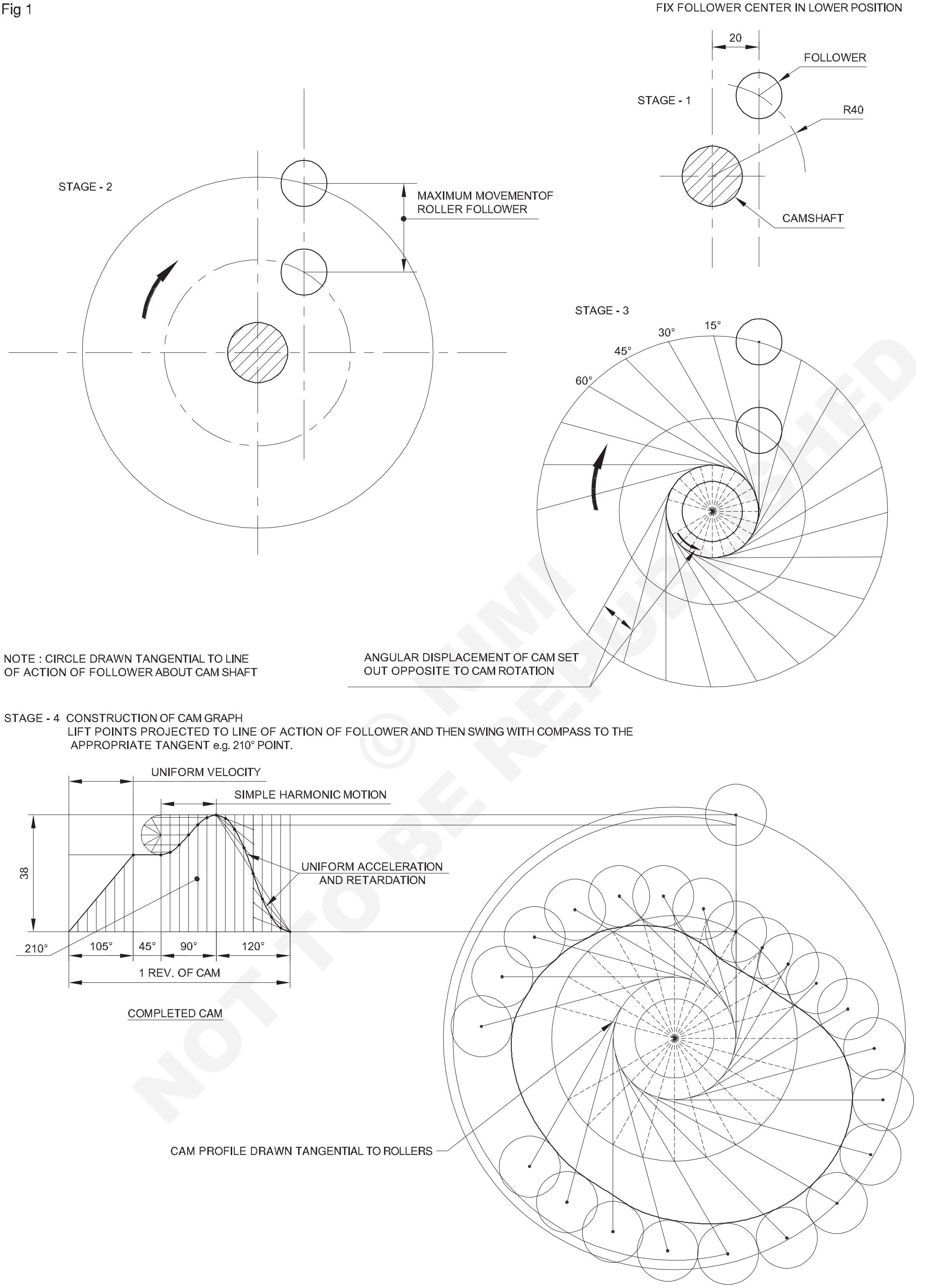


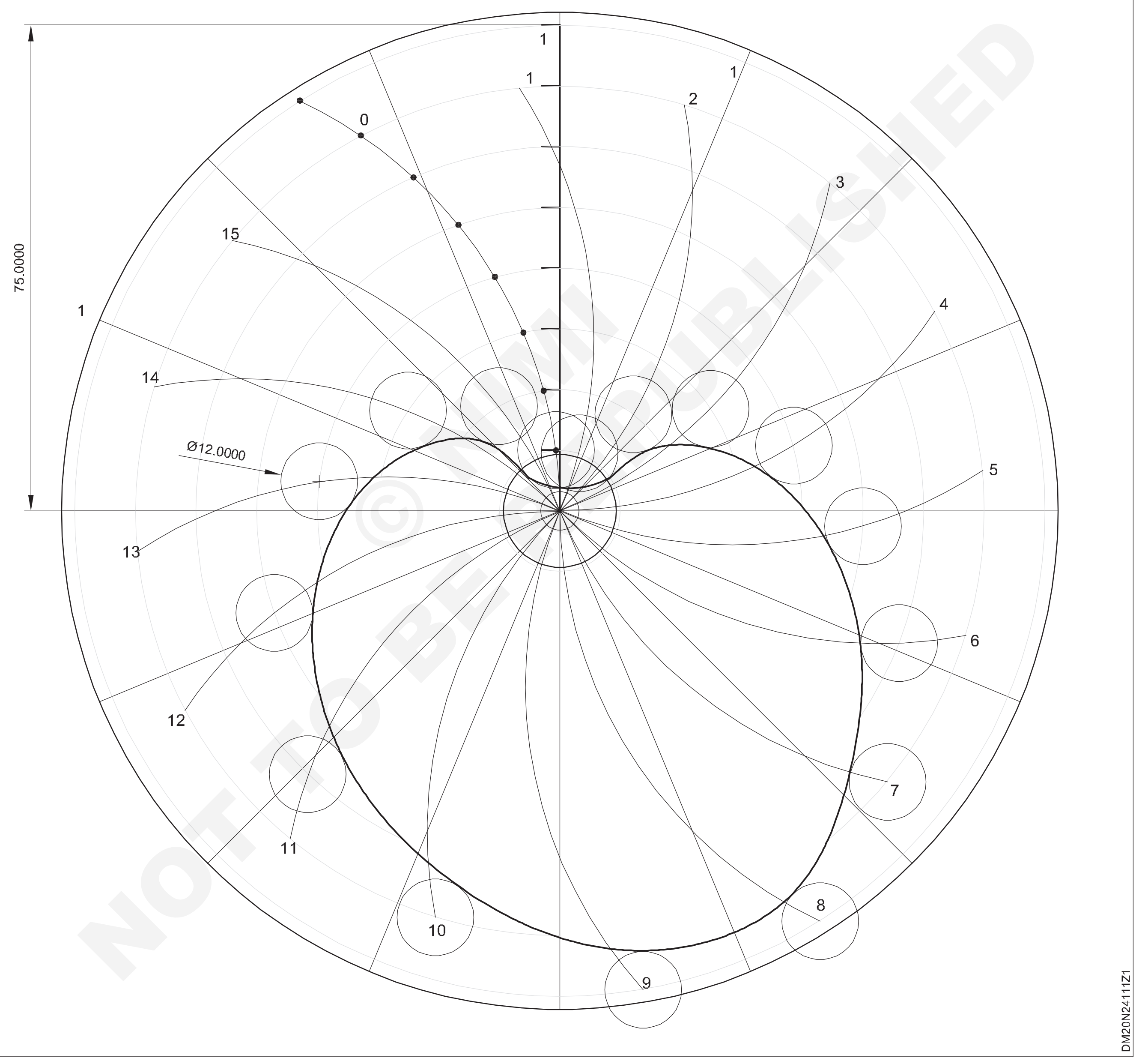
PLATE CAM PROFILE WITH OFF-SET ROLLER FOLLOWER

DM20N24111Y1

TASK 6: **Construct profile of a cam with follower**

- Determine the profile of a cam, which allow the follower to oscillate with uniform angular velocity about a fixed centre. Every revolution of the cam completes one oscillation of the follower. Distance between the centre of cam and the roller is 35 mm, Lift = 75 mm, roller diameter = 12 mm, lever FC - 154 mm.
- **Construction:** Fig 1 with centre O and radius OF = 140 mm describe a circle. Starting from F divide its circumference into 16 equal parts. From each of these parts in turn, describe arcs passing through O. Locate point C, the centre of roller. Set-off the lift and divide it into eight equal parts. Describe concentric circles through each of the points. The successive points of intersection between these circles and the arcs previously drawn are on the locus of the roller centre. Draw the arcs, from the points of intersection of circles and arcs, with radius equal to radius of the roller. A smooth curve touching all these curved arcs gives the profile of the cam

Fig 1



Construct detailed and Assembly drawing

Objectives: At the end of this exercise you shall be able to
- **draw eccentrics**
- **draw stuffing box**
- **draw piston assembly**
- **draw Ic engine connecting rod.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

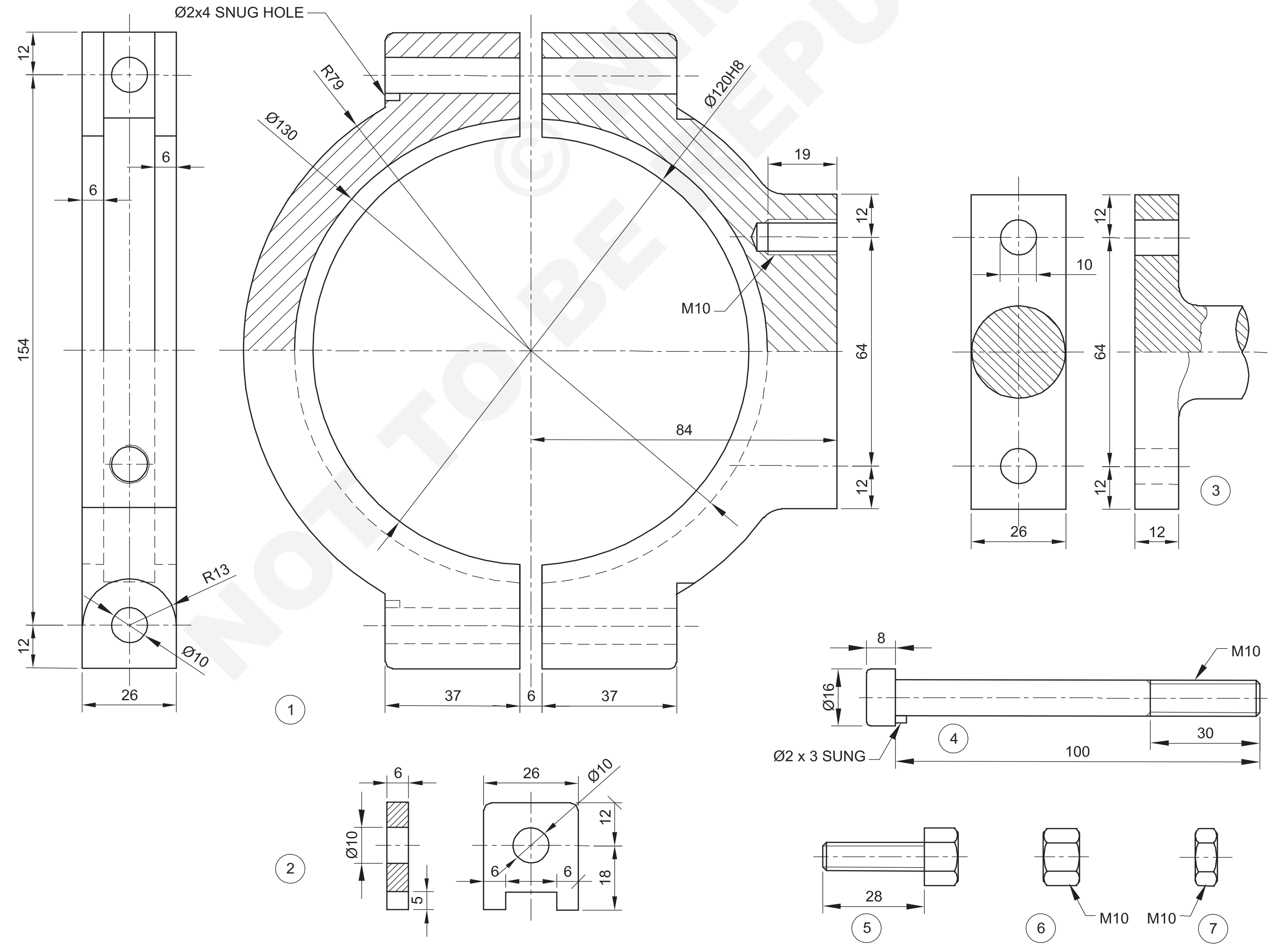
TASK 1: **Construct detail and assembly drawings of an Eccentrics using CAD (Figs 1 -3)**

In assemble drawing draw the following views

i Top half - sectional elevation

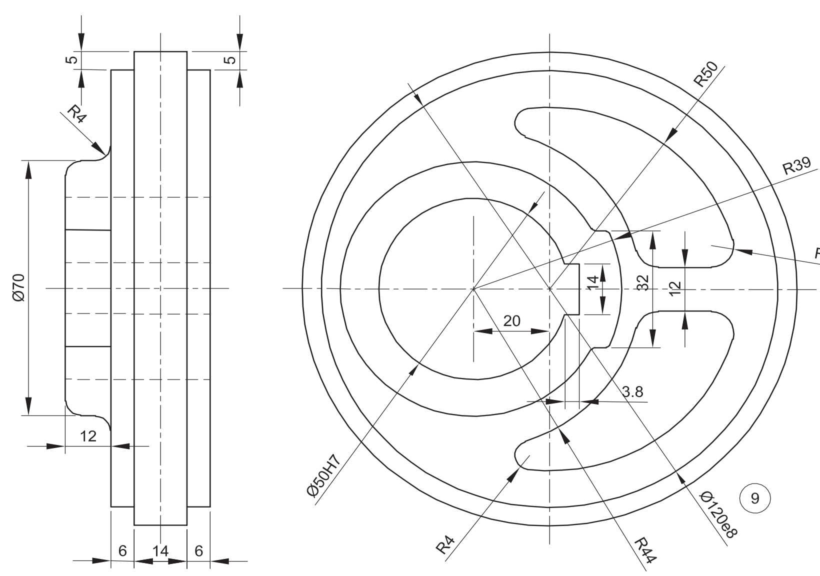
ii End view

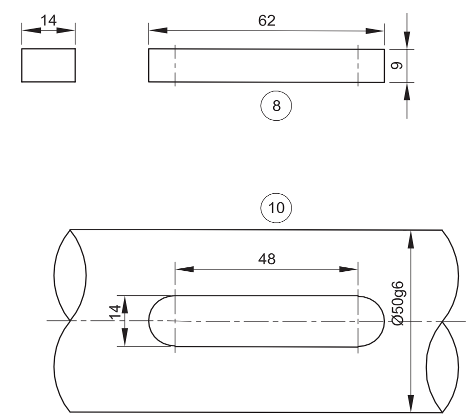
Fig 1



ECCENTRIC (PARTS DETAIL DRAWING)

Fig 2



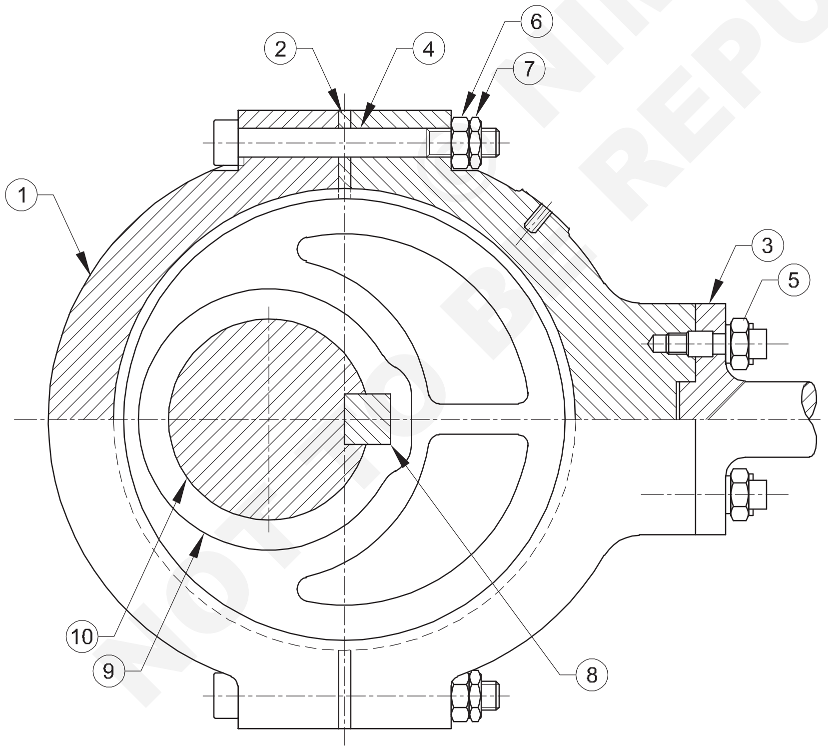


PART NO.	NAME	MATL.	QTY.
1	STRAPS	C.I.	1 SET
2	SHIM	G.M.	2
3	ROD	STEEL	1
4	BOLT (M10x100)	M.S.	2
5	BOLT (M10x28)	M.S.	2
6	NUT	M.S.	2
7	LOCK NUT	M.S.	2
8	PARALLEL KEY	M.S.	1
9	SHEAVE	C.I.	1
10	SHAFT	STEEL	1

ECCENTRIC (PARTS DETAIL DRAWING)

DM20N25113H1

Fig 3



PART NO.	NAME	MATL.	QTY.
1	STRAPS (SET)	C.I.	1
2	SHIM	G.M.	2
3	END ROD	STEEL	1
4	BOLT (M10 x 100)	M.S.	2
5	BOLT (M10 x 28)	M.S.	2
6	NUT	M.S.	2
7	LOCK NUT	M.S.	2
8	PARALLEL KEY	M.S.	1
9	SHEAVE	C.I.	1
10	SHAFT	STEEL	1

ELEVATION, TOP HALF IN SECTION ECCENTRIC (ASSEMBLY)

DM20N25113H3

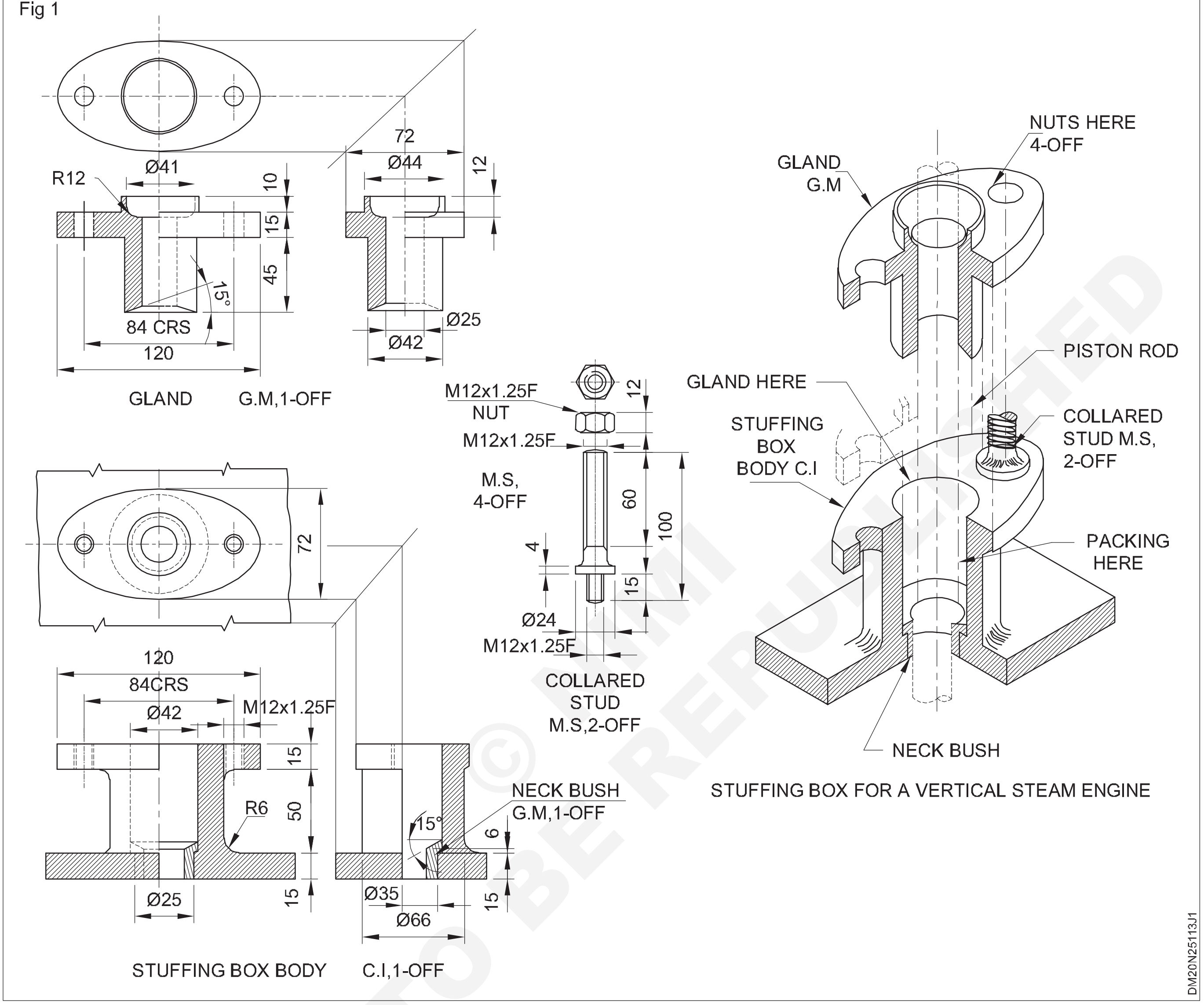
TASK 2: **Construct detail and assembly drawings of an stuffing box using CAD (Fig 1)**

In assemble drawing draw the following views

i Left half - sectional elevation

ii Top view

Fig 1



— — — — —

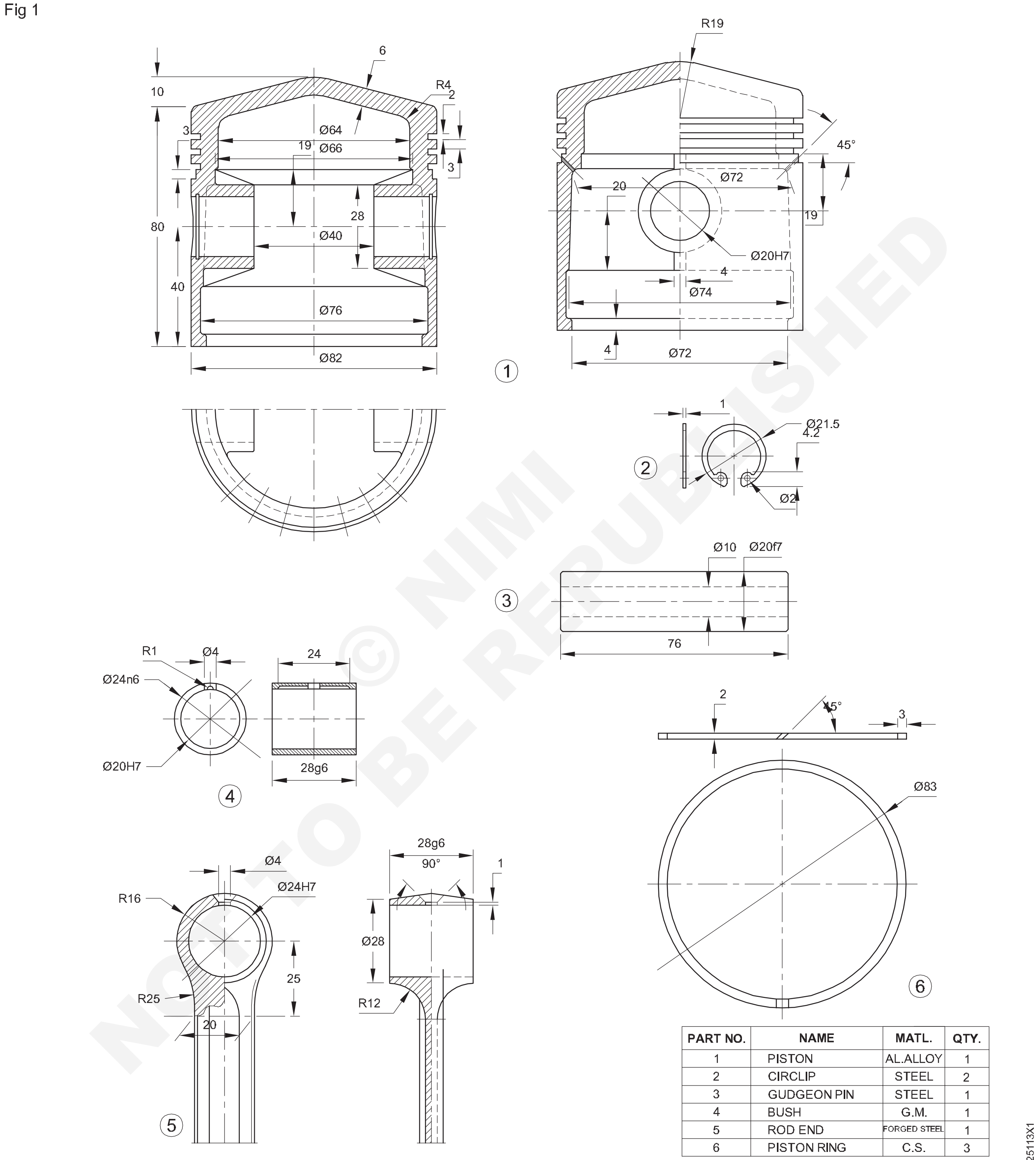
TASK 3: **Construct detail and assemble drawings of piston assembly using CAD (Figs 1&2)**

In assemble drawing draw the following views

i Full sectional elevation

ii Top view

Fig 1

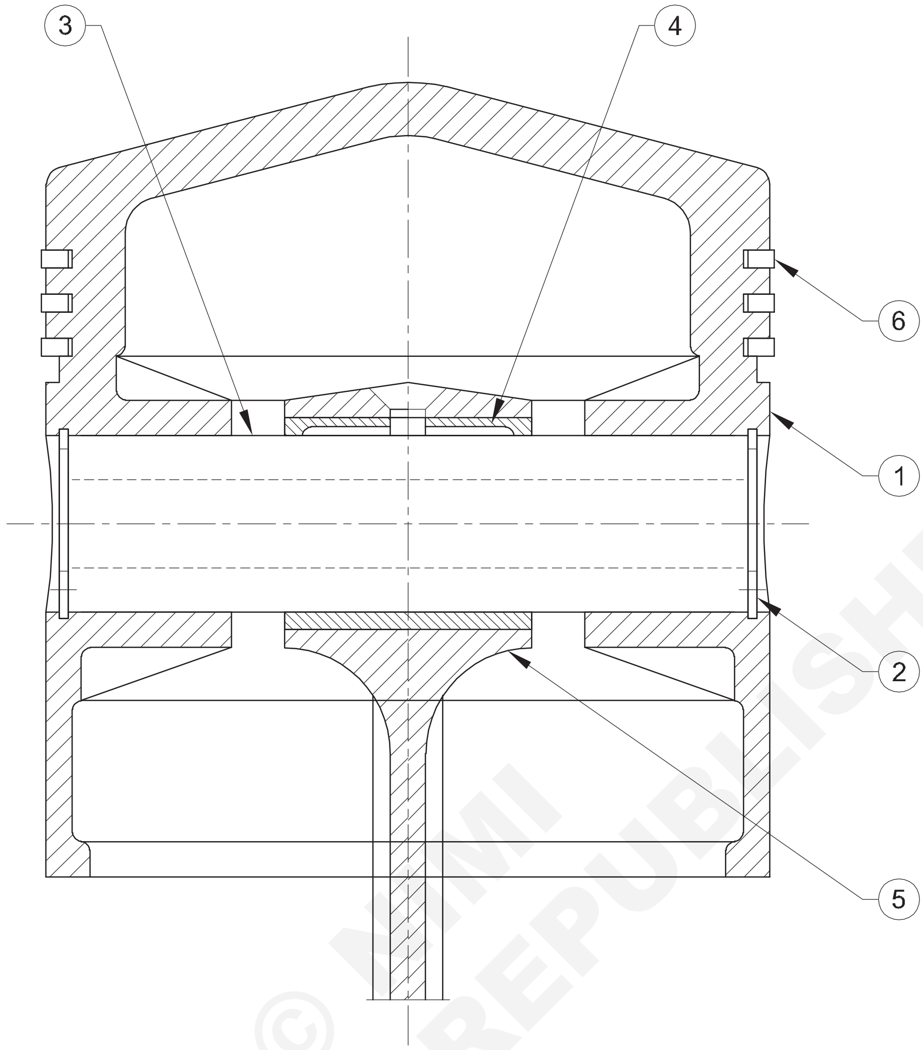


PART NO.	NAME	MATL.	QTY.
1	PISTON	AL.ALLOY	1
2	CIRCLIP	STEEL	2
3	GUDGEON PIN	STEEL	1
4	BUSH	G.M.	1
5	ROD END	FORGED STEEL	1
6	PISTON RING	C.S.	3

PISTON AND ROD END (PETROL - FOUR STROKE) PARTS

DM20N25113X1

Fig 2



PART NO.	NAME	MATL.	QTY.
1	PISTON	AL.ALLOY	1
2	CIRCLIP	STEEL	2
3	GUDGEON PIN	STEEL	1
4	BUSH	G.M.	1
5	ROD END	STEEL	1
6	PISTON RING	C.S.	3

SECTIONAL ELEVATION
PISTON AND ROD END (PETROL - FOUR STROKE) ASSEMBLY

DM20N25113X2

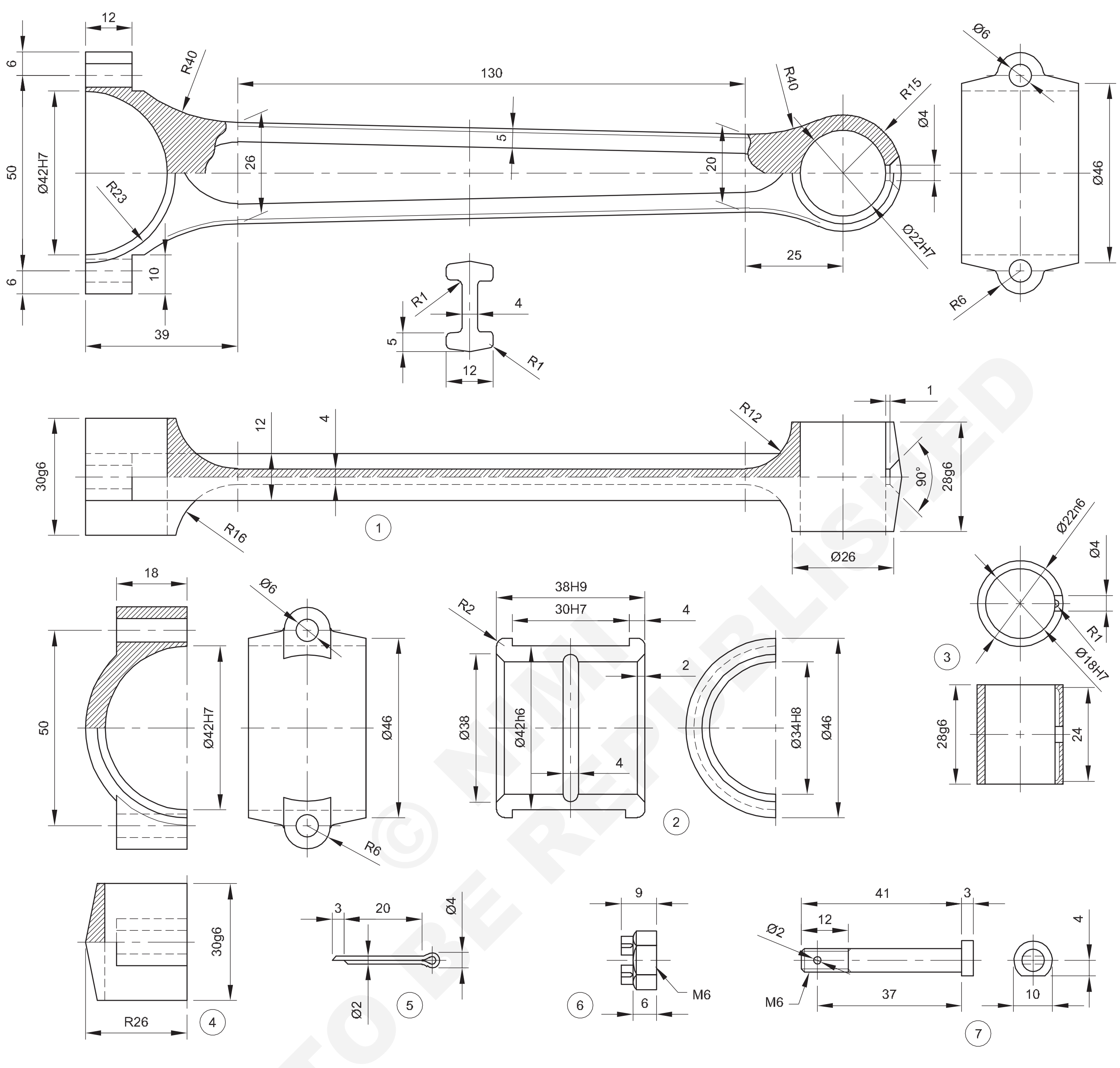
TASK 4: **Construct detail and assemble drawings of IC Engine connecting rod using CAD (Fig 1)**

In assemble drawing draw the following views

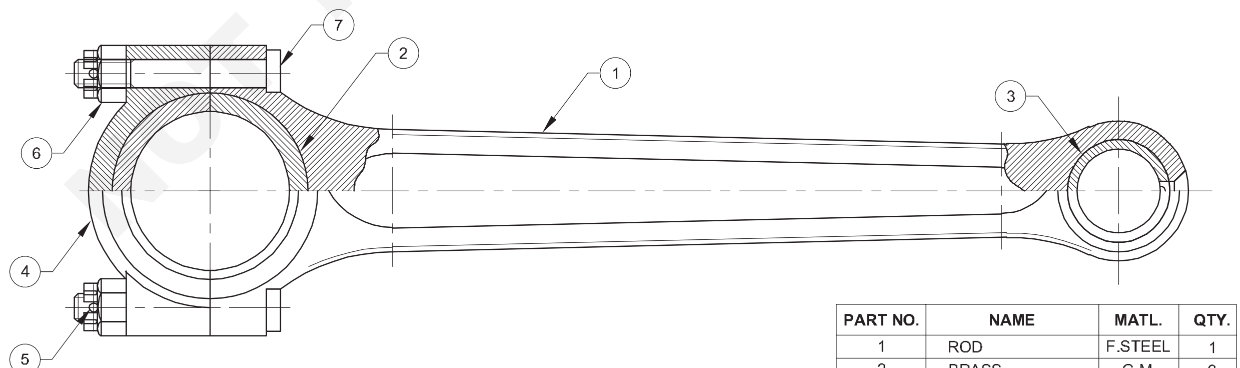
i Sectional elevation

ii Top view

Fig 1



IC ENGINE CONNECTING ROD (PARTS)



PART NO.	NAME	MATL.	QTY.
1	ROD	F.STEEL	1
2	BRASS	G.M.	2
3	BUSH	G.M.	1
4	CAP	F.STEEL	1
5	SPLIT PIN	M.S.	2
6	CASTLE NUT	STEEL	2
7	BOLT	STEEL	2

IC ENGINE CONNECTING ROD (ASSEMBLY)

DM20N25113Y1

Construct detailed drawing of an Air valve

Objectives: At the end of this exercise you shall be able to
- **draw the detailed drawing of the air valve.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

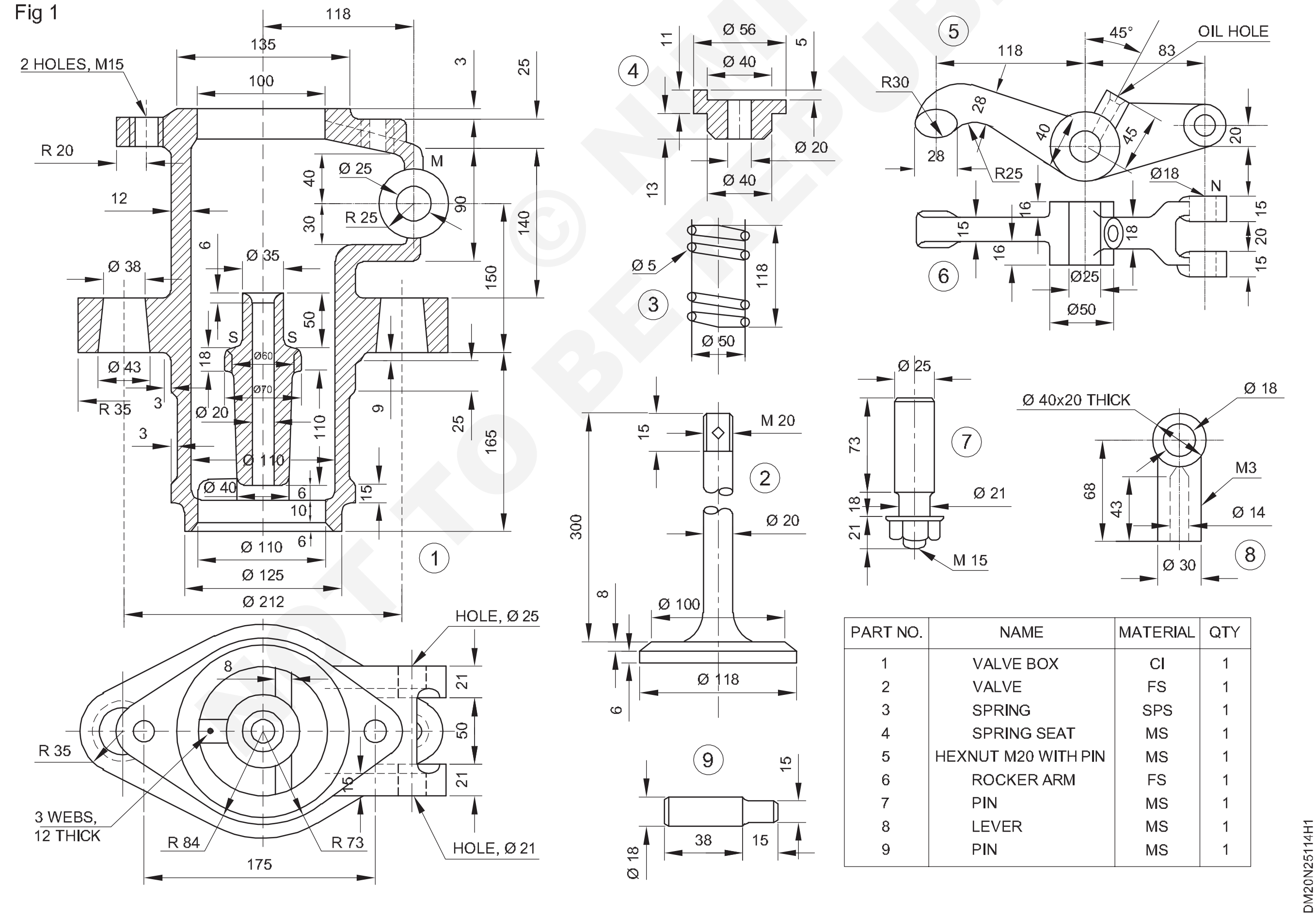
TASK 1: **Construct detail and assemble drawings of Air valve using CAD (Figs 1&2)**

In assemble drawing draw the following views

i Full sectional elevation

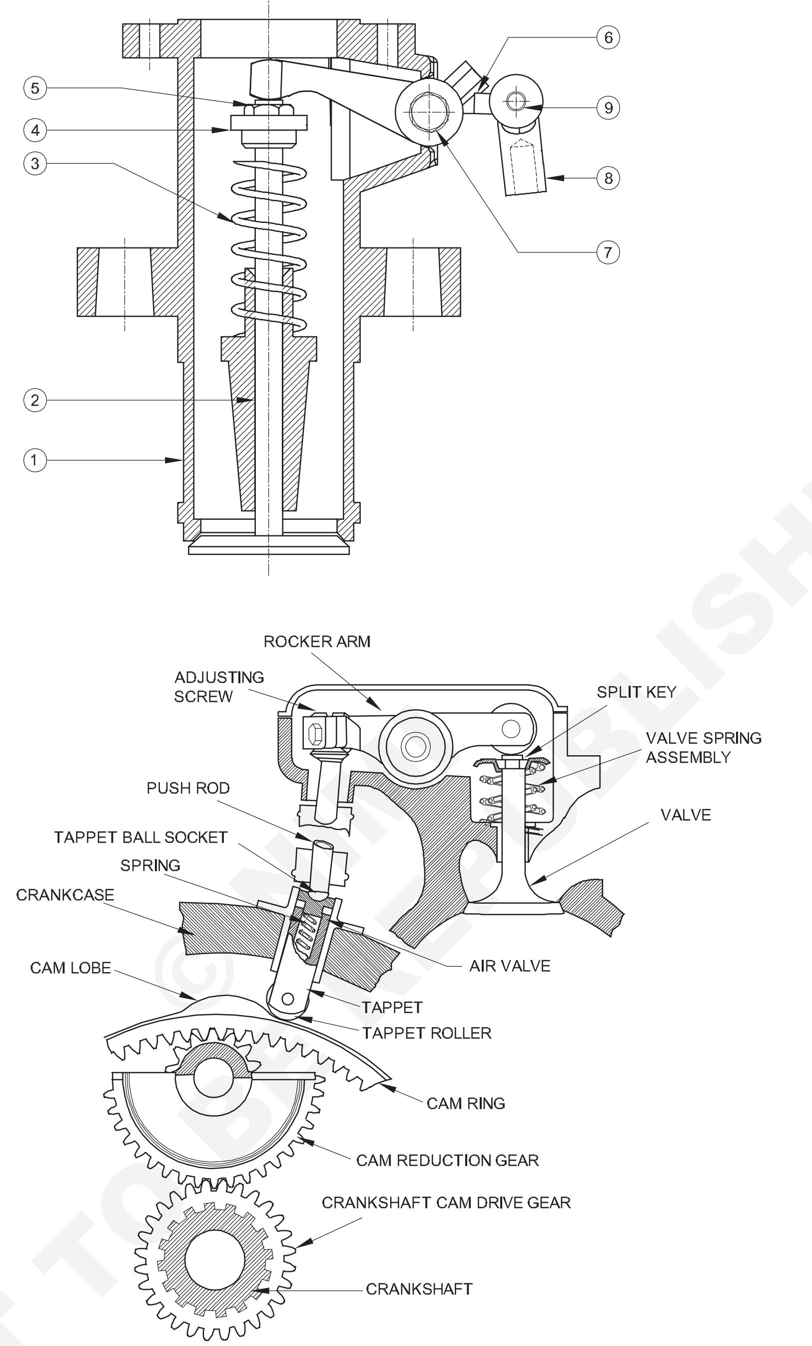
ii Top view

Fig 1



PART NO.	NAME	MATERIAL	QTY
1	VALVE BOX	CI	1
2	VALVE	FS	1
3	SPRING	SPS	1
4	SPRING SEAT	MS	1
5	HEXNUT M20 WITH PIN	MS	1
6	ROCKER ARM	FS	1
7	PIN	MS	1
8	LEVER	MS	1
9	PIN	MS	1

Fig 2


1
2
3
4
5
6
7
8
9
ROCKER ARM
ADJUSTING SCREW
SPLIT KEY
VALVE SPRING ASSEMBLY
PUSH ROD
VALVE
TAPPET BALL SOCKET
SPRING
CRANKCASE
AIR VALVE
CAM LOBE
TAPPET
TAPPET ROLLER
CAM RING
CAM REDUCTION GEAR
CRANKSHAFT CAM DRIVE GEAR
CRANKSHAFT
DM20N25114H2

Fuel injector of diesel Engine

Objectives: At the end of this exercise you shall be able to
- **draw detailed drawing of the fuel injector.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

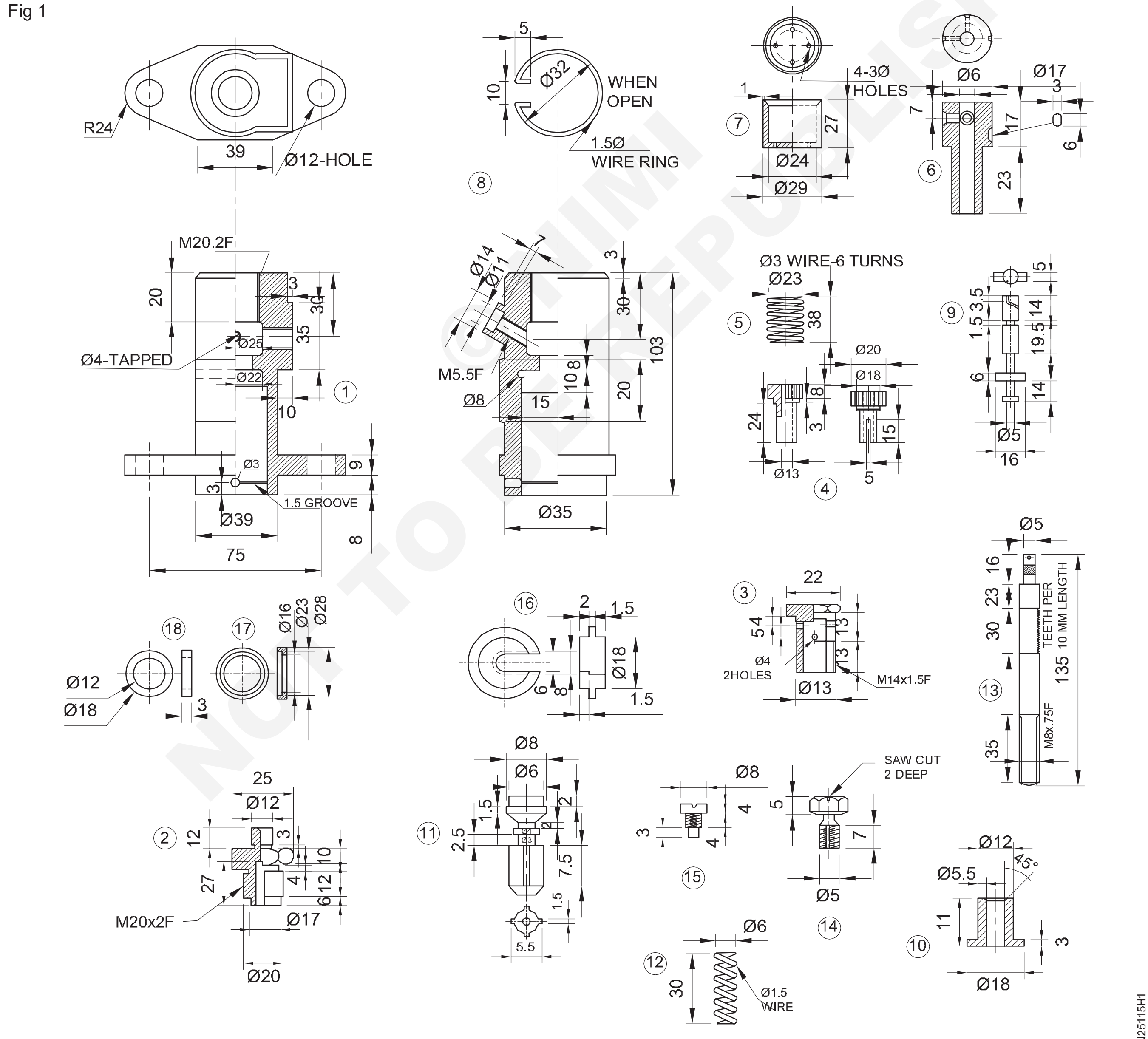
PROCEDURE

TASK 1: **Construct detail and assemble drawings of fuel injector using CAD (Figs 1&2)**

In assemble drawing draw the following views

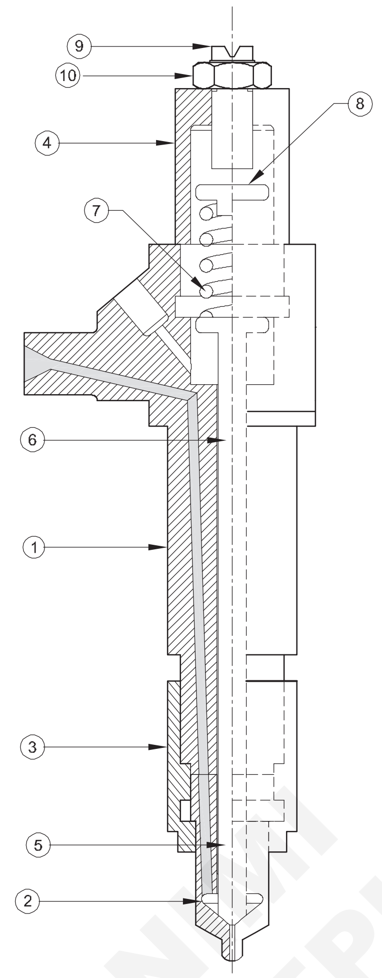
i Half sectional elevation.

Fig 1



DETAILS OF A FUEL INJECTOR

Fig 2



FUEL INJECTOR ASSEMBLY

3D Modelling

Objectives: At the end of this exercise you shall be able to
- **draw 3D primitives**
- **create co ordinate system**
- **annotate and dimension of the 3D model**
- **generate views from model space to layout**
- **generate print preview.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Creating solids using basic shapes**

AutoCAD provides seven basic solid shapes. Basic shapes will be our first step to create more complex shapes. To find all of these commands go to the **Home** tab, locate the **Modeling** panel, and select one of the following:

Here is a discussion of each command.

Box Command

This command will allow you to create a box or cube. Choose the Box command; the following prompts will be displayed:

Specify first corner or [Center]:

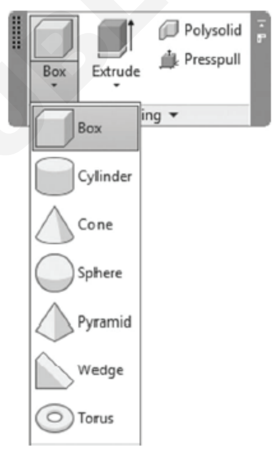
Specify other corner or [Cube/Length]:

Specify height or [2Point]:

To draw a box in AutoCAD, you have to define the base first, then the height. The base will be drawn on the current XY plane. Some variations include:

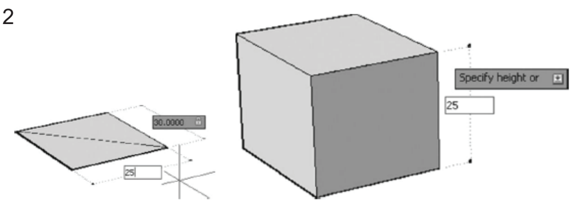
- Specify the base by typing the coordinates of the first corner, and the coordinates of the opposite corner.
- Specify the base by typing the coordinates of the first corner. To specify the opposite corner, specify the length and height using typing and [Tab] key.
- Specify the base by specifying the first corner; then specify Length (in X axis direction) and Width (in Y axis direction).
- Specify the base by specifying the first corner; then specify Cube option. Auto- CAD will ask you to input one dimension for all the sides (length, width, and height).
- Specify the base by specifying Center point and one of the corners of the base. After specifying the base, specify the height, either by typing, by using the mouse, or by specifying two points by clicking (this is a good method if you have an existing object). (Figs 1&2)

Fig 1



Note:From now on, you can use dynamic input to input all or any of the distances needed to complete any command.

Fig 2



Cylinder Command (Fig 3)

This command will allow you to create a cylinder. Choose the Cylinder command and the following prompts will appear:

Specify center point of base or [3P/2P/Ttr/Elliptical]:

Specify base radius or [Diameter]:

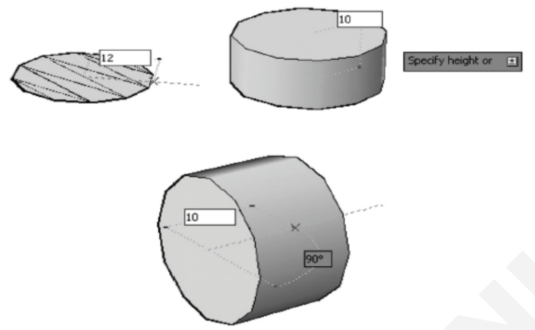
Specify height or [2Point/Axis endpoint]:

To draw a cylinder in AutoCAD, you have to define the base first, then the height. Some variations include:

- The base could be a circle or ellipse. The options to draw either a circle or ellipse are identical to the 2D prompts. The base will be drawn on the current XY plane.
- After defining the base, specify the height, either by typing or by using the mouse or by using 2 Points option (using the height of an existing object).

Axis endpoint option will allow you to rotate the cylinder by 90 degrees to make the height in the XY plane and not perpendicular to it. Using Polar option you can direct the cylinder as you wish.

Fig 3



Cone Command (Fig 4)

This command will allow you to create a cone. Choose the Cone command and the following prompts will be displayed:

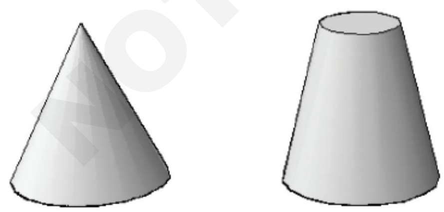
Specify center point of base or [3P/2P/Ttr/Elliptical]:

Specify base radius or [Diameter):

Specify height or [2Point/Axis endpoint/Top radius):

Cone and Cylinder have identical prompts; hence, we will not discuss them again, except for one thing. The Cone command will allow you to draw a cone with Top radius.

Fig 4



Sphere Command (Fig 5)

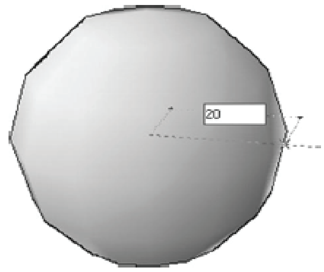
This command will allow you to create a sphere. Choose the **Sphere** command and the following prompts will appear:

Specify center point or [3P/2P/Ttr]:

Specify radius or [Diameter]:

These prompts are identical to Circle command prompts.

Fig 5



Pyramid Command (Fig 6)

This command will allow you to create a pyramid. Choose the **Pyramid** command and the following prompts will appear:

4 sides Circumscribed

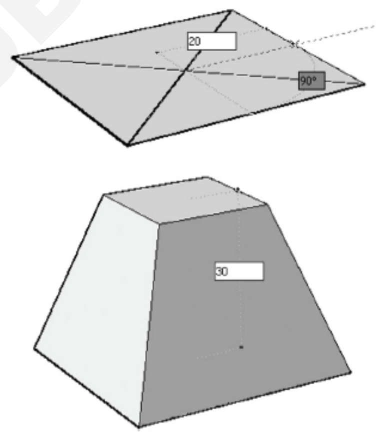
Specify center point of base or [Edge/Sides]:

Specify base radius or [Inscribed]:

Specify height or [2Point/Axis endpoint/Top radius]:

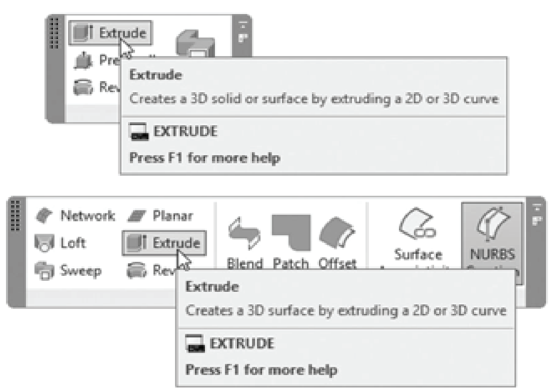
The default is 4-sides base. You can change the number of sides by selecting the Sides option. To draw a base, use the same methods to draw a 2D polygon. Height options were discussed in the previous commands. The example below is for a 4-sides base with the top smaller polygon:

Fig 6



To issue this command, go to the Solid tab, locate the Solid panel, and select the Extrude button, or go to the Surface tab, locate the Create panel, and select the Extrude button: (Fig 7)

Fig 7



The following prompts will be shown:

Select objects to extrude or (Mode]:

Specify height of extrusion or [Direction/Path/Taper angle]

The first line is asking you to select object(s) to extrude. The second line says:

Specify height of extrusion or [Direction/Path/Taper angle]

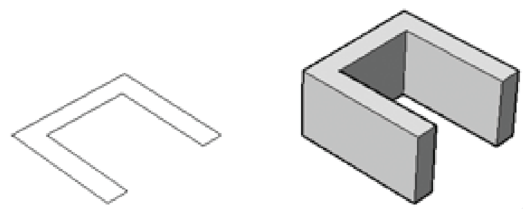
This line will tell you there are four ways to create an extrusion in AutoCAD:

- Specify a height.
- Specify a direction.
- Specify a path.
- Specify a taper angle.

Create an Extrusion Using Height (Fig 8)

This is the default option; it will create an extrusion perpendicular to the 2D profile using height. Whenever you select the 2D object, edge, or face, you will see the extrusion in the screen. You can specify the height graphically, or you can type the height value. See the following:

Fig 8



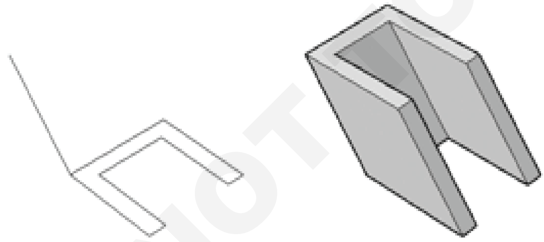
Create an Extrusion Using Direction (Fig 9)

If option height is to create an extrusion perpendicular on the profile, this option will create an extrusion with any angle desired. The following prompts will be shown:

Specify start point of direction: Specify end point of direction:

The following picture clarifies the concept:

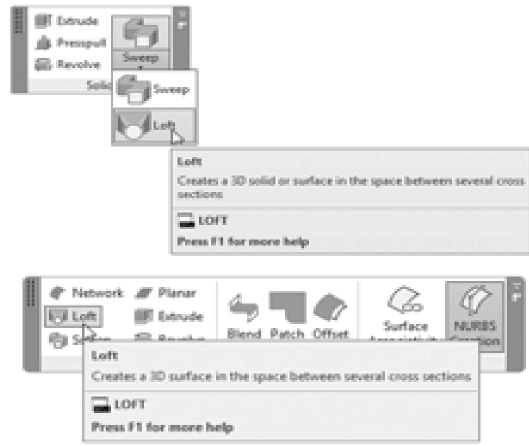
Fig 9



Using loft command (Figs 10&11)

This command will create a solid or surface by using multiple cross sections in different UCSs whether opened or closed. The condition here is to have all profiles opened or all of them closed; you cannot use a mix. You can use different options like guide, path, and cross sections. To issue this command, go to the Solid tab, locate the Solid panel, and select the Loft button, or go to the Surface tab, locate the Create panel, and select the Loft button:

Fig 10

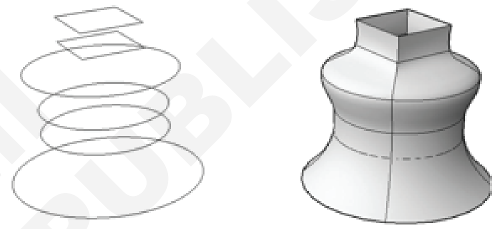


The following prompts will be displayed:

Select cross sections in lofting order or [Point/Join multiple edges/Mode]:

The first line will ask you to select the cross sections in lofting order. Refer to the following example:

Fig 11

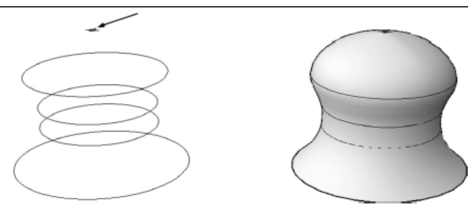


This prompt will be repeated to select more and more closed or opened objects. While you are selecting, you can select one of three more options. They are the following:

- Point, which will allow you to select the loft end point and end the command at this point. The cross section selected should be always closed, and you should specify this point either by typing a real 3D point or by selecting a point object. You will see the following prompt:

Specify loft end point: (Fig 12)

Fig 12



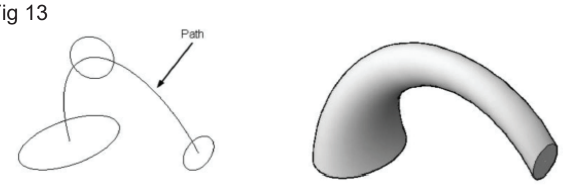
Lofting Using Path (Fig 13)

You have the ability to select cross section alone, but with this option, users will select both cross sections and path to dictate the final shape. The following prompt will be shown:

Select path profile:

You should receive the following:

Fig 13

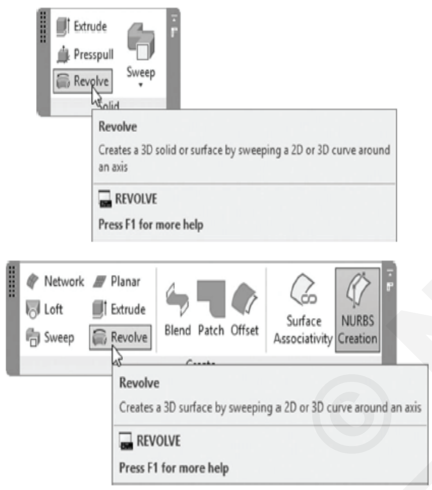


Using revolve command (Figs 14 & 15)

This command will create a solid or surface by revolving 2D objects closed or opened, or using the face or edge of solids and surfaces. To issue this command, go to the Solid tab, locate the Solid panel, and select the Revolve button, or go to the Surface tab, locate the Create panel, and select the Revolve button:

The following prompt will be shown: Select objects to revolve or [Mode]:

Fig 14



AutoCAD is asking you to select the desired object(s) to revolve When done press [Enter]; the following prompts will be shown:

Specify axis start point or define axis by

[Object/X/Y/Z] <Object>:

Specify angle of revolution or

[STart angle/Reverse/Expression] <360>:

AutoCAD is offering three different methods to specify the axis of revolution. They are the following:

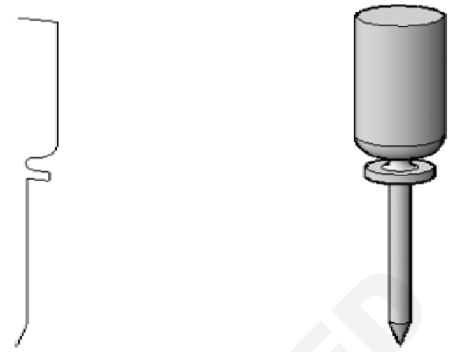
- Specify an axis by inputting two points
- Specify an axis by selecting a drawn object.
- Specify an axis by specifying one of the three axes.

Finally you have to specify the angle of revolution, there are multiple options here.

- Input the angle either by typing or by using the mouse. CCW is always positive. If you define the axis of revolution by two points, then the positive angle will be the curling of your right hand, with your thumb pointing to the direction of the nearest point to the farthest point.
- Specify the start angle if you want the revolving process to start not from the cross section.
- If you want to reverse the direction of the angle, use Reverse option.

The following illustration will explain the concept:

Fig 15

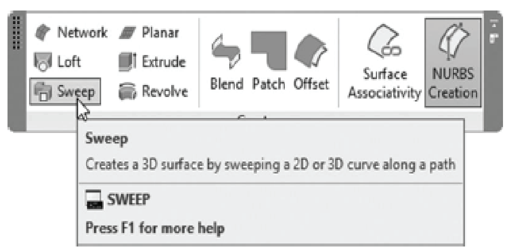


Using sweep command (Figs 16 - 18)

This command will create complex a solid or surface by sweeping a 2D open or closed shape, or a solid or surface face or edge along a path. To issue this command, go to the Solid tab, locate the Solid panel, and select the Sweep button, or go to the Surface tab, locate the Create panel, and select the Sweep button

Fig 16

Fig 17



The following prompts will be shown:

Select objects to sweep or [Mode]:

Select objects to sweep or [Mode]:

Select sweep path or [Alignment/Base point/Scale /Twist]:

The default option is to select object(s) to sweep, then select the sweep path if you follow this path. The following is a result of what you will receive:

Or you can use the other options which are the following:

- Alignment
- Base point
- Scale
- Twist

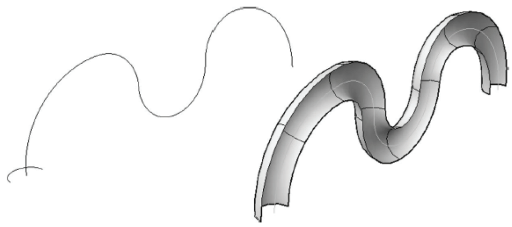


Fig 18

— — — — —

TASK 2: **Creating a new UCS using UCS command**

This command will create a new UCS based on information supplied by you, and it does not ask for objects to be drawn before hand. This command offers so many ways to create a new UCS, which makes it a very important command for any AutoCAD user who wants to master 3D modeling in AutoCAD. We will discuss UCS options not according to their alphabetical order, but according to their importance. You can find all UCS options in the Home tab and Coordinates panel. (Fig 1)

3-Point Option

To create any plane in XYZ space, you need three points. This fact makes this option the most practical when creating a new UCS. As the name of the option suggests, you will specify three points:

1 New origin point

2 Point on the positive X axis

3 Point on the positive Y axis

To issue this command, select the 3-Point button:

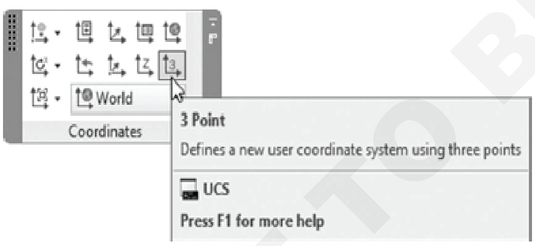


Fig 1

Origin Option (Fig 2)

This option will create a new UCS by moving the origin point from one point to another without modifying the orientation of the XYZ axes. To issue this command, select the Origin button:

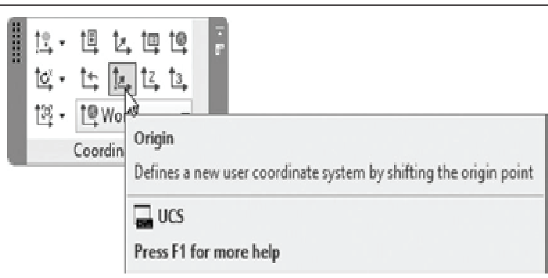


Fig 2

X/Y/Z Options (Fig 3)

These are three options but they are similar. So, we put them together. The idea is very simple; you will fix one of the three axes, and then rotate the other two to get a new UCS. But how will you know whether the angle you have to input should be positive or negative? We will use the second right-hand-rule, which states, "Point the thumb of your right hand toward the positive portion of the fixed axis; the curling of your four fingers will represent the positive direction." In the below picture we took Y as an example. To issue the Y command, select the Y button.

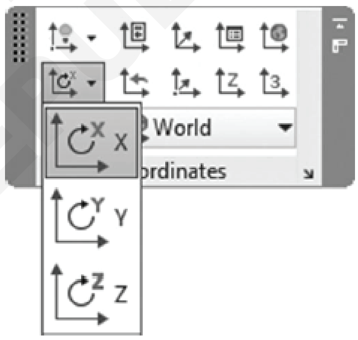


Fig 3

Named Option (Figs 4 -7)

If you use a certain UCS a lot, AutoCAD gives you the ability to name/save it. This will help you retrieve it whenever you need it. After creating the UCS, go to the View tab, locate the Coordinates panel, select the Named button:

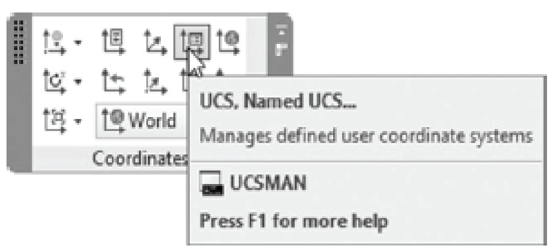


Fig 4

You will see the following dialog box:

Under the Named UCSs tab, you will see a list of the saved UCSs, plus, Unnamed, World, and Previous. Unnamed is your current UCS, which you created before issuing this command. Click it to rename it; then type the new name. Then press [Enter]. You will see the following:

Fig 5

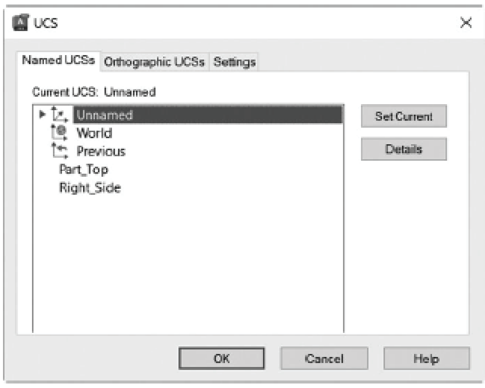
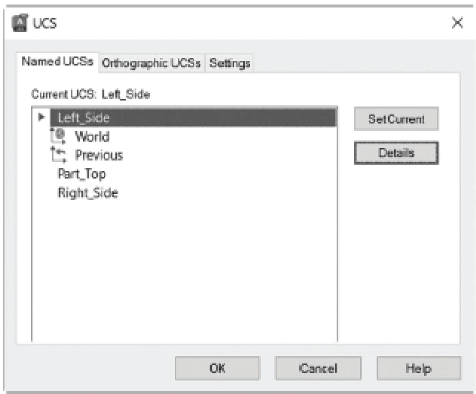


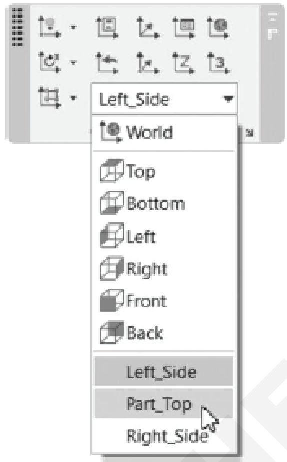
Fig 6



To end the command, click OK.

In the Coordinates panel, there is a list of the preset UCS. You will see your saved UCS among them, as shown in the following.

Fig 7



— — — — —

TASK 3: **Annotate or dimension a 3D object in auto CAD**

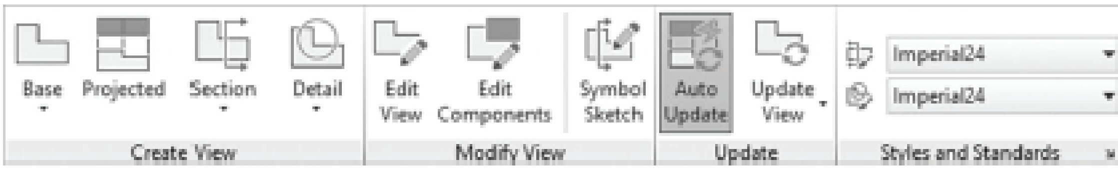
- Dimension using 3D Snaps
- Replace the Z value with the current elevation in the layout:
  1 Open the OPTIONS dialog.
  2 Switch to the Drafting Tab.
  3 At the bottom of the window, check Replace Z value with current elevation.
  4 Press OK.
- Place dimension using UCS:
  1 Go to Home Tab
  2 Hover to mouse to Coordinates panel
  3 Click on Origin UCS
  4 Place the UCS on the object face where the dimension needs to be added
  5 Go to Annotate Tab
  6 Click on Dimension
  7 Place the dimension where are needed

— — — — —

TASK 4: **Model documentation - Introduction**

Model Documentation will generate intelligent documentation for AutoCAD 3D models. Model Documentation will generate views, sections, and details that will be instantly updated when the model changes. In order to start working with this tool, you should go to any layout; then you will find all the desired panels in the Layout tab (which will appear only if you are in Layout and not in the Model space). They are: Create View, Modify View, Update, and Style and Standards. (Fig 1)

Fig 1



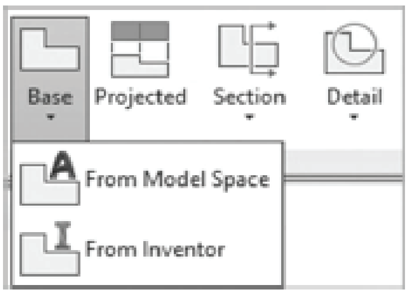
The procedure is very simple:

- Create Base view.
- Create Projected views.
- Create Sections and Details.
- Edit and update the views.
- Control Styles and Standards.

Model documentation - Create base view (Fig 2)

This should be always your first step. If you have more than one 3D model, AutoCAD will allow you to select the desired objects to be included in the creation process. You can start your step from the Model space or you can go to the desired layout. To start this command, go to the Layout tab, locate the Create View panel, and select the Base button:

Fig 2



You will have two choices to choose from:

- The current From Model Space.
- From Inventor. This option will allow you to bring in Inventor file (*.iam, *.ipt, or *.ipn).

Let's assume that you want to bring 3D objects from AutoCAD created in the Model Space. In this case, we have two choices:

- Start the command in Model Space. AutoCAD will ask you either to select the desired objects to include in the view/section/detail, or select the entire model.

Because you are in the Model Space, AutoCAD will ask to specify the name of the layout to be used. You will see the following prompts:

Select objects or [Entire model] <Entire model>: Enter new or existing layout name to make current or [?] <Lay-out1>: layout1

Start the command in one of the layouts. AutoCAD will ask you to specify the location of the base view. The most important thing at this stage is Select option. By default AutoCAD will select all objects in the Model Space; once you start the Select option, you will be taken to Model space to remove the undesired objects. When done, AutoCAD will take you back to the layout you were in. You will see the following prompts:

Type Base only Hidden Lines Visible and hidden lines

Scale = 1/8" = 1'-0"

Specify location of base view or [Type/s Elect/ Orienta-tion/ Hidden lines/Scale/Visibility] <Type>:

Meanwhile, a new context tab named Drawing View Creation will appear. It appears as the following:

At the left you can see the Model Space Selection button, which is identical to the Select option we discussed above.

The Default view will appear. You can change the default view to your desired view, which will act as your Base view using two methods. The first is Using the Orientation panel in the context tab, which will allow you to select the desired view. The second is to use the Orientation option in the prompts above. (Fig 3)

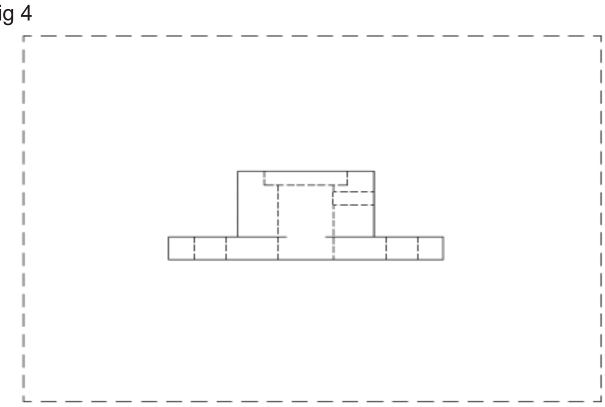
Fig 3

By default, using the Base command will insert the base view and the Projected views, as well. But using the Type option, users can select whether to insert Base or Base and Projected.

Using the Appearance panel in the context tab will allow you to pick the Hidden Lines settings. You will select one of four: Visible lines, Visible and hidden lines, Shaded with visible lines, and Shaded with visible and hidden lines. Also, using the same panel, you can set the scale and the edge visibility. (Fig 4)

Assuming the inserted only the base view, you will see the following:

Fig 4



TASK 5: **Named views and 3D**

Users normally use all 3D viewing commands to look at the desired model from different angles. While using this command, you can save and name the views to look at them later or to use them in layouts for printing purposes. To issue this command, go to the Visualize tab, locate the Named Views panel, and select the New View button:

The following dialog box will be shown: (Figs 1&2)

Fig 1

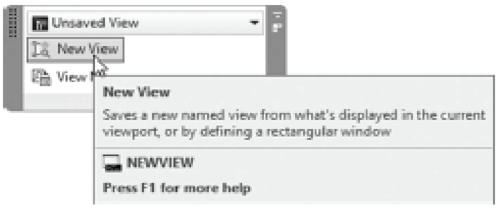
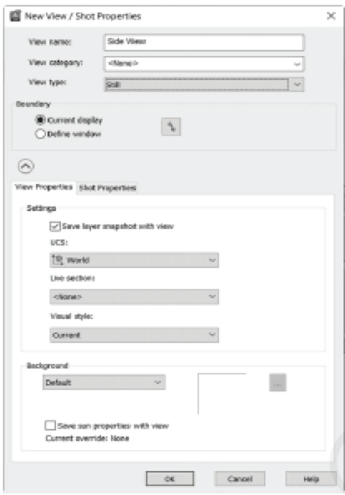


Fig 2



First type in the View's name. Users can save with the view several things, they are:

- The current layer status
- The current UCS
- The current Live section
- The current visual style

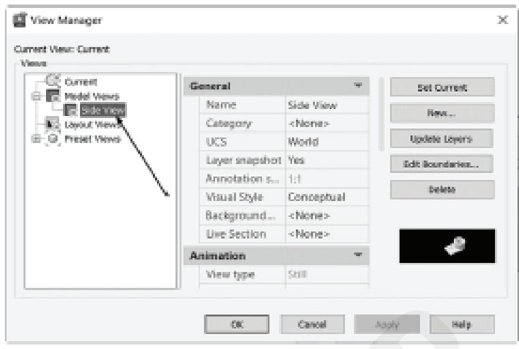
Users can add a background to the saved view; select one out of the five different choices: (Fig 3)

Fig 3

A dialog box will appear for you to manipulate the relative parameters of each one of the five selections. Once you have finished, click OK to end the command.

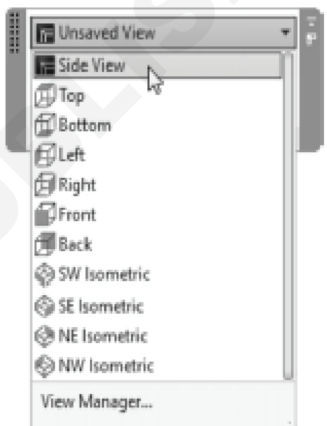
To control your views, go to Visualize tab, locate Named Views panel, and click View Manager button, you will see the following dialog box. which contains a list of saved views: (Fig 4)

Fig 4



Another place you will see a list of the saved views is the Named Views panel; see the following image (Fig 5)

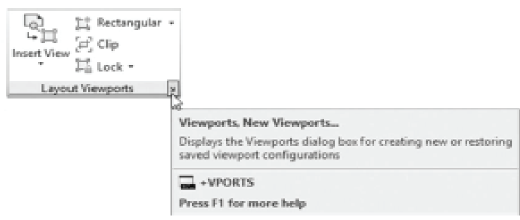
Fig 5



3D and viewport creation (Fig 6)

In this part, we will discuss the special features of 3D in viewport creation. To issue this command, you should be in one of the layouts, go to the Layout tab, locate Layout Viewports; then click the New Viewports button:

Fig 6



You will see the following dialog box:

Change the setup to 3D, set a named viewport, and then set the visual style for each viewport. When done click OK, and specify two opposite corners for the viewports, you will receive the following:

User can assign a shade mode to one or all viewports. Select the desired viewport; then right click, select shade plot, and select one of the modes as shown below:

Notice the last group of the shade plots. They are rendering options; hence you can plot a rendered image if you want, but you can only see the effects of this action using the print preview command. (Figs 7 - 9)

Fig 7

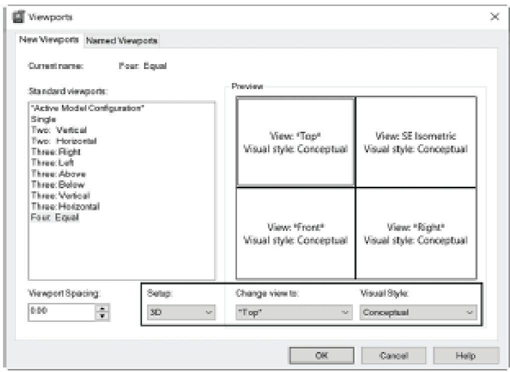


Fig 8

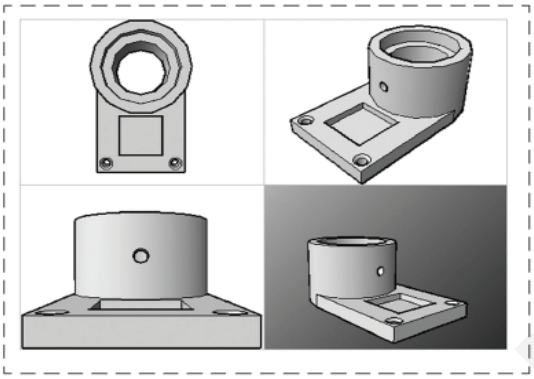
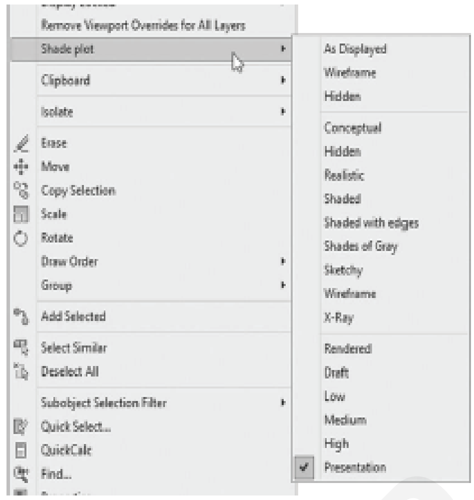


Fig 9



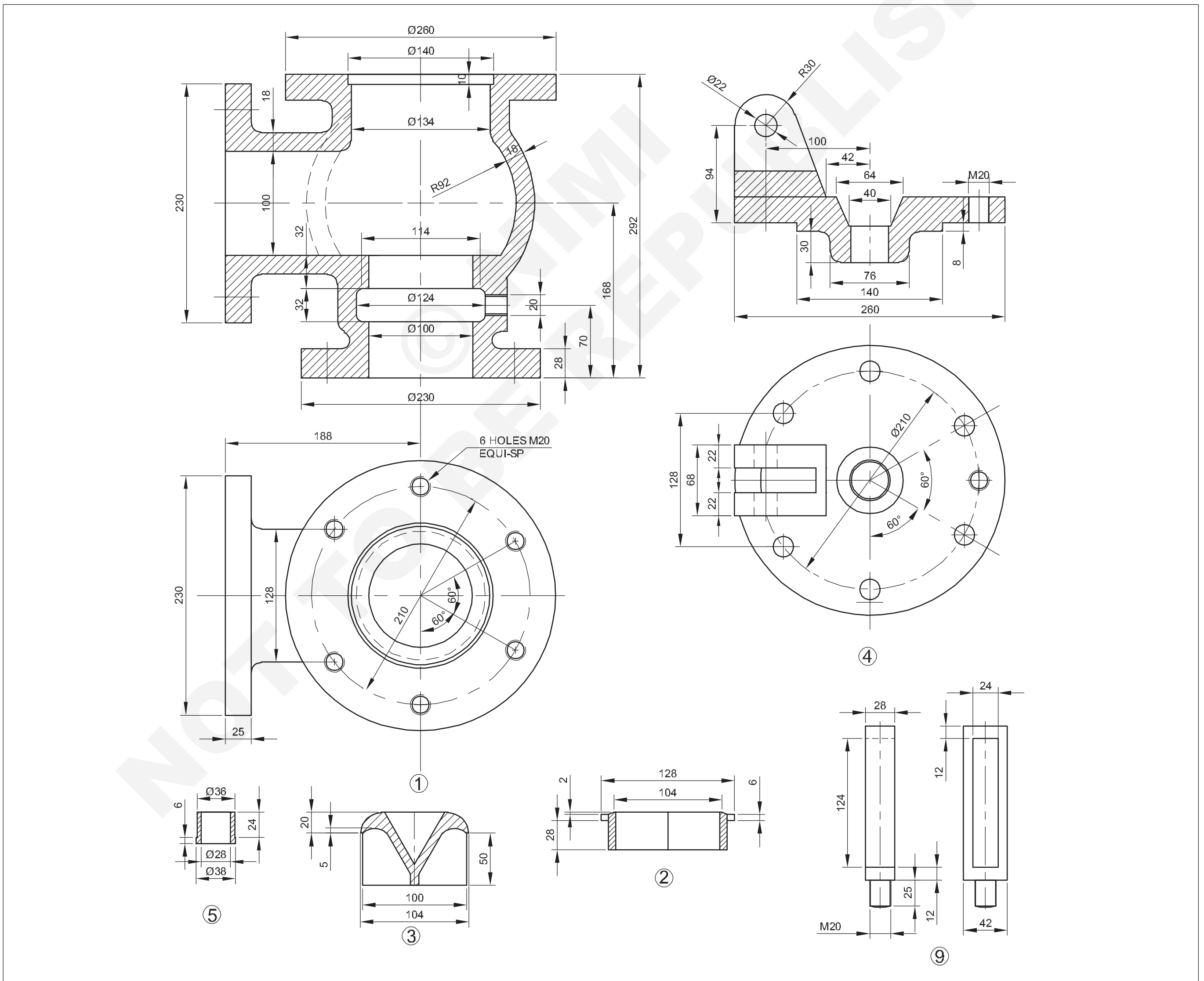
Construct detailed drawing of a lever safety valve

Objectives: At the end of this exercise you shall be able to
- **draw the details of the lever safety valve**
- **draw hte full sectional front elevation**
- **draw the plan.**

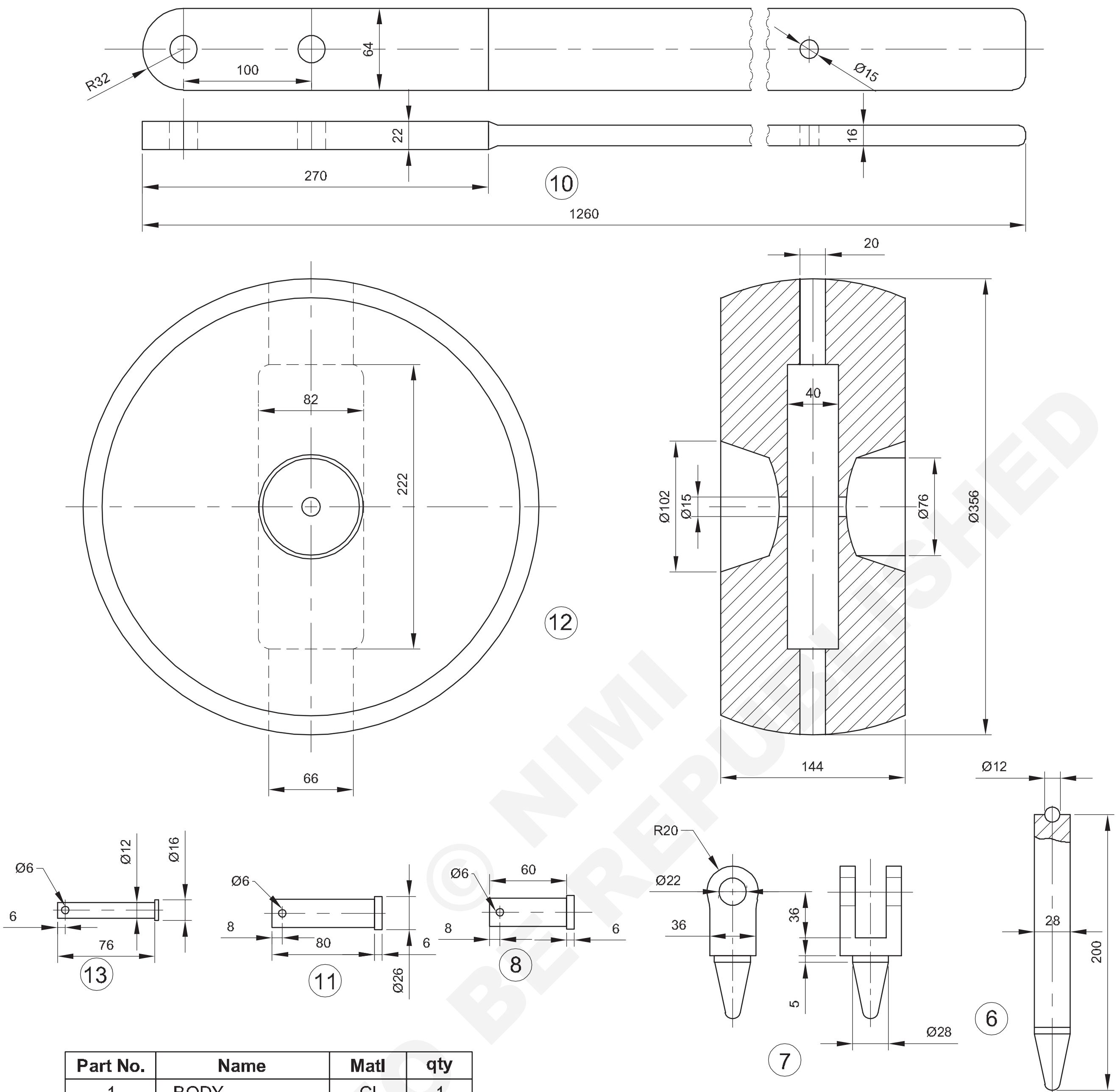
Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the given details of the lever safety valve with required elevations and plan by using CAD (Fig 1)**



SCALE 1:1	LEVER SAFETY VALVE (DETAILS)	EX NO : 2.7.117
		TIME : 28hrs
PROJECTION		CODE : DM20N27117E1

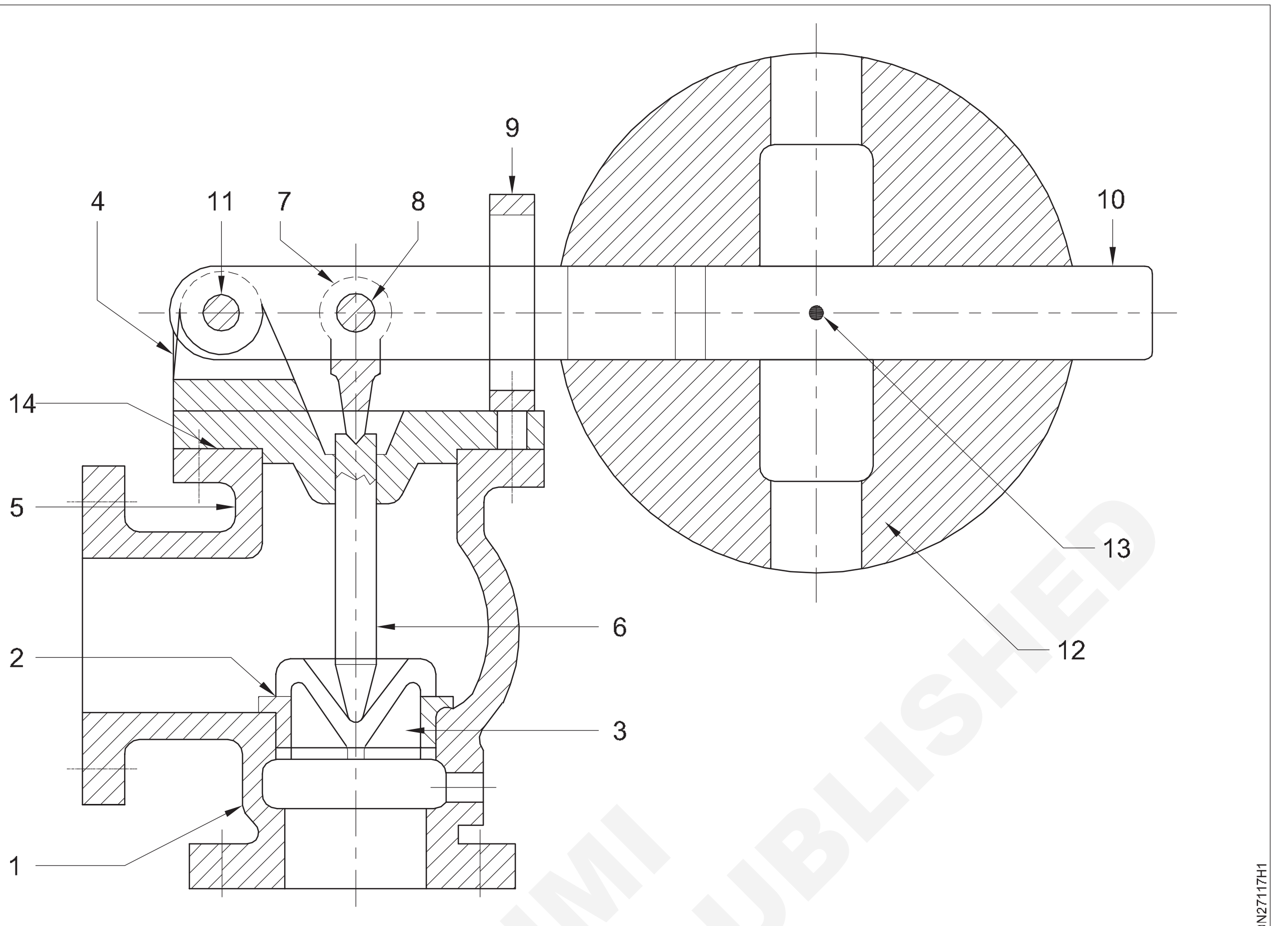


Part No.	Name	Matl	qty
1	BODY	CI	1
2	VALVE SEAT	GM	1
3	VALVE	GM	1
4	COVER	CI	1
5	COVER BUSH	BRASS	1
6	SPINDLE	MS	1
7	TOGGLE	MS	1
8	TOGGLE-PIN	MS	1

Part No.	Name	Matl	qty
9	LEVER GUIDE	MS	1
10	LEVER	MATL	1
11	FULCRUM PIN	MS	1
12	WEIGHT	MATL	1
13	LEVER PIN	MS	1
14	STUD WITH NUT M20	-	1

SCALE 1:1	**LEVER SAFETY VALVE (DETAILS)**	EX NO : 4.1.115
		TIME : 28hrs
PROJECTION		CODE : DM20N27117E2

Fig 1



LEVER SAFETY VALVE ASSEMBLY

Construct detailed drawing of a gate valve

Objectives: At the end of this exercise you shall be able to
- **construct detailed drawing of a gate valve.**

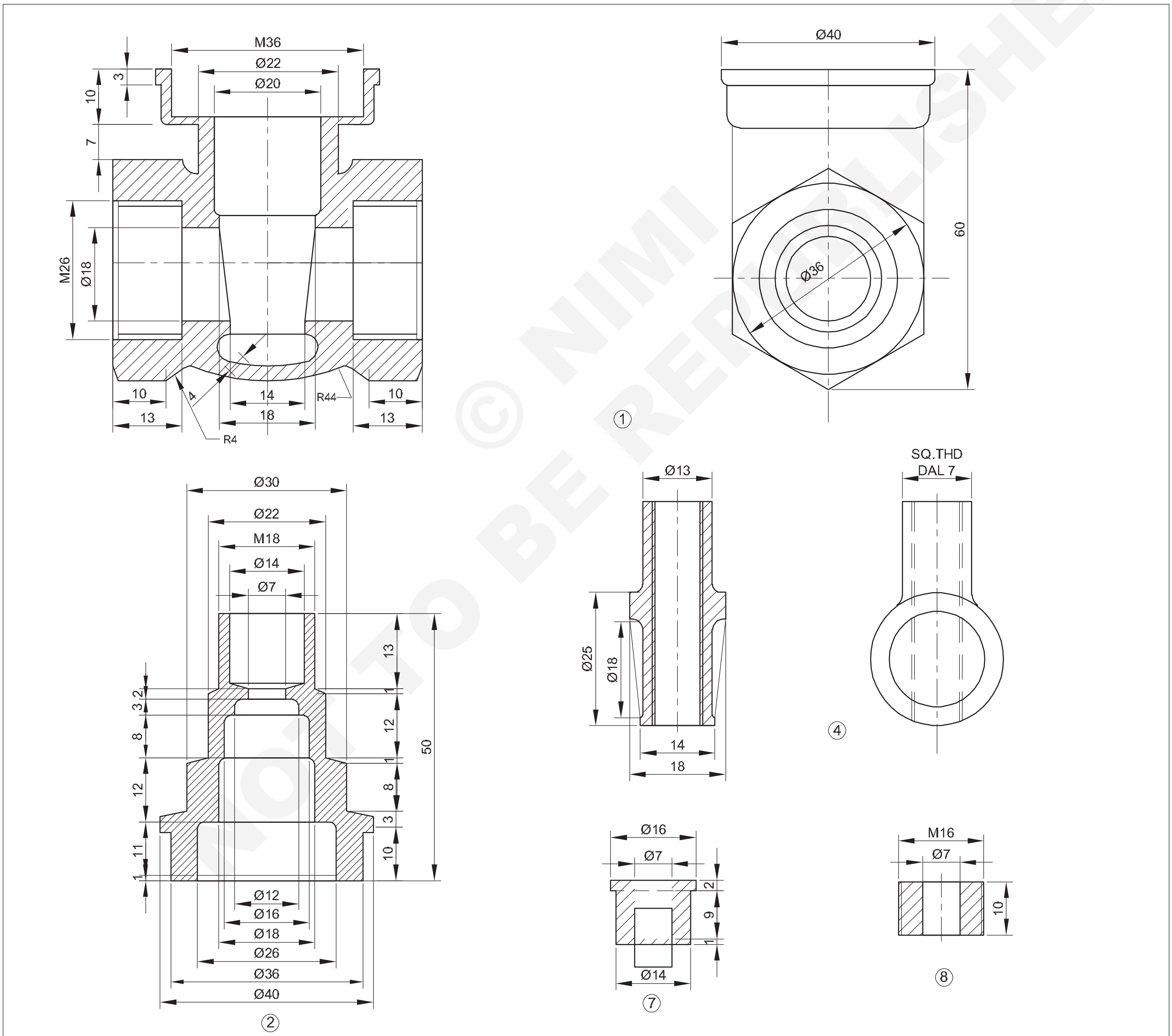
Requirements

Tools/Equipment/Machines

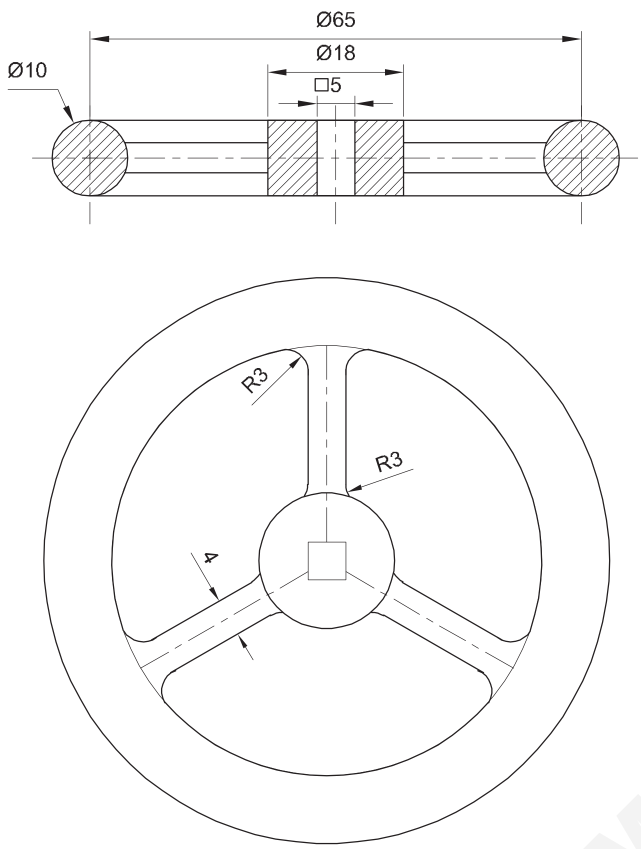
- Auto CAD 2018 or Higher version.

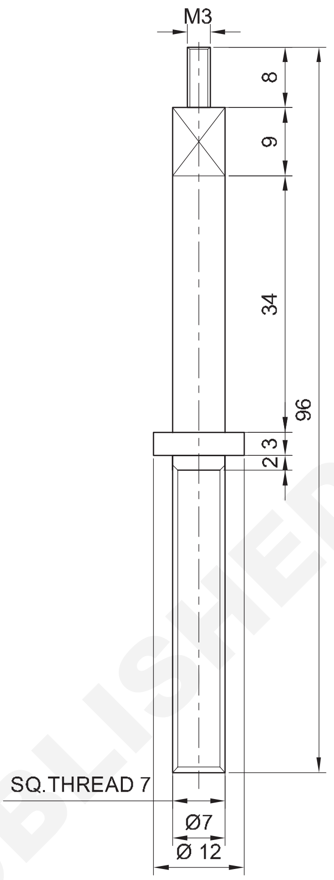
PROCEDURE

TASK 1: **Construct detailed drawing of gate valve**



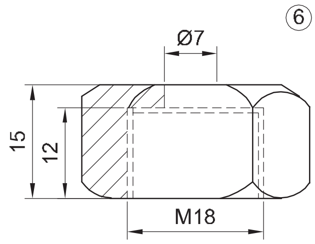
SCALE 1:1	GATE VALVE (DETAILS)	EX NO : 4.1.116
		TIME : 28hrs
PROJECTION		CODE : DM20N27118E1



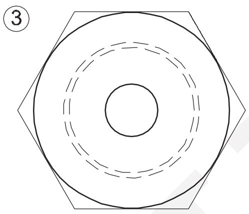


5

6



3



PART NO.	NAME	MATERIALS	QTY
1	VALVE BODY	BRASS	1
2	UNION	BRASS	1
3	UNION RING	BRASS	1
4	WEDGE VALVE	BRASS	1
5	STEM	BRASS	1
6	HAND WHEEL	CI	1
7	GLAND	BRASS	1
8	INSIDEUNION	BRASS	1

SCALE 1:1	**GATE VALVE (DETAILS)**	EX NO : 2.7.118
		TIME : 28hrs
PROJECTION		CODE : DM20N27118E2

— — — — —

Construct detailed drawing of a steam stop valve and blow of cock

Objectives: At the end of this exercise you shall be able to
- **draw the steam stop valve detailed drawing by using CAD.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

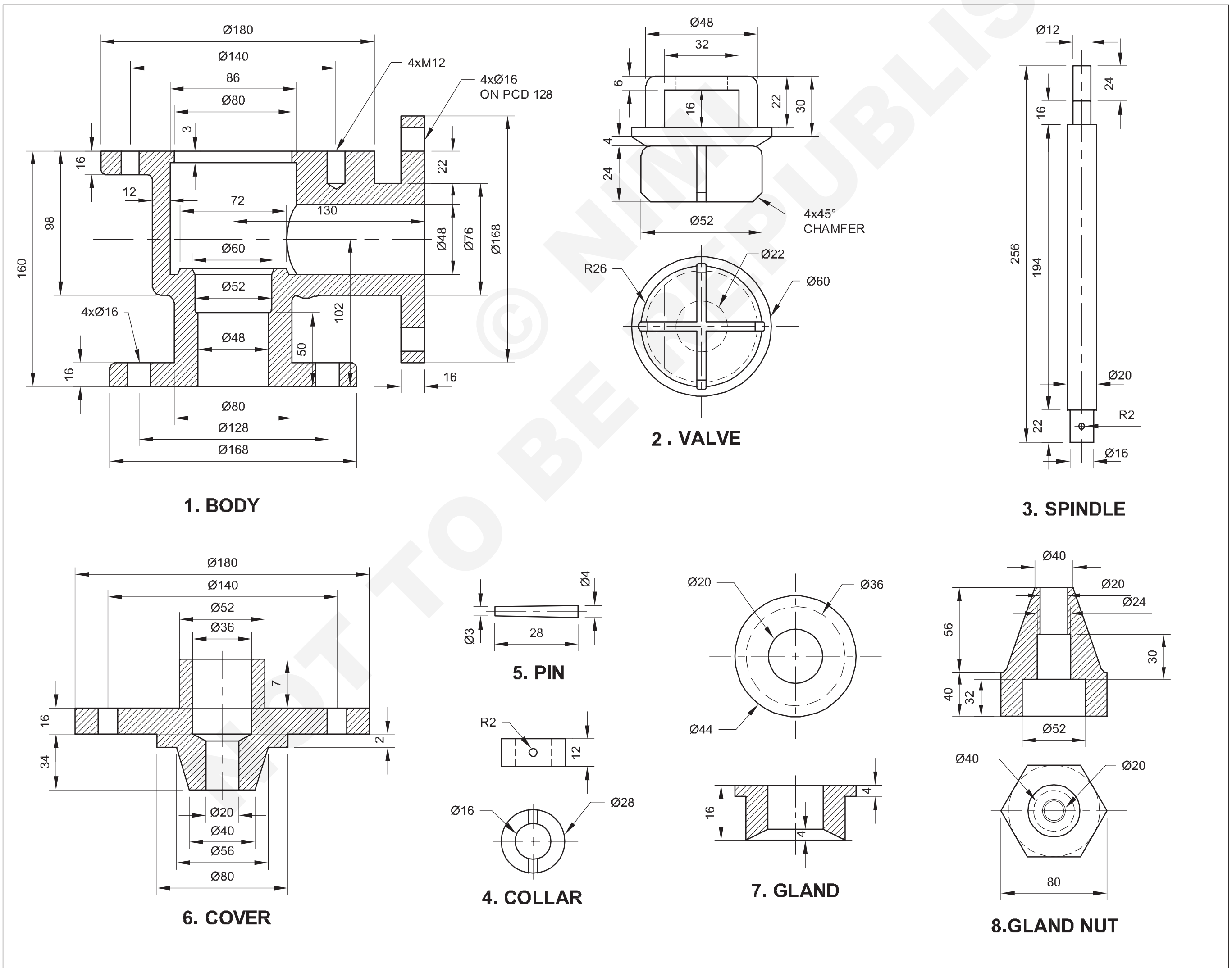
PROCEDURE

TASK 1: **Construct detail drawing of steam stop valve using cad**

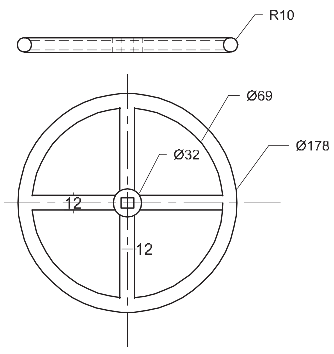
i Draw the full plan view using CAD.

ii Prepare a bill of material.

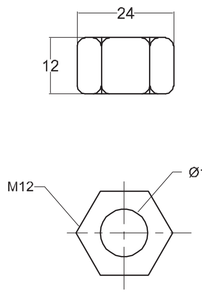
Note: For assembling the steam stop valve refer the detail drawing indicate the parts as per the bill of material list in the views.



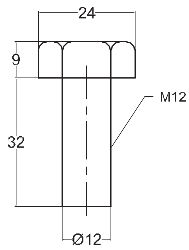
SCALE 1:1	DETAILS OF STEAM STOP VALVE	EX NO : 2.7.119
		TIME : 28hrs
PROJECTION		CODE : DM20N27119E1



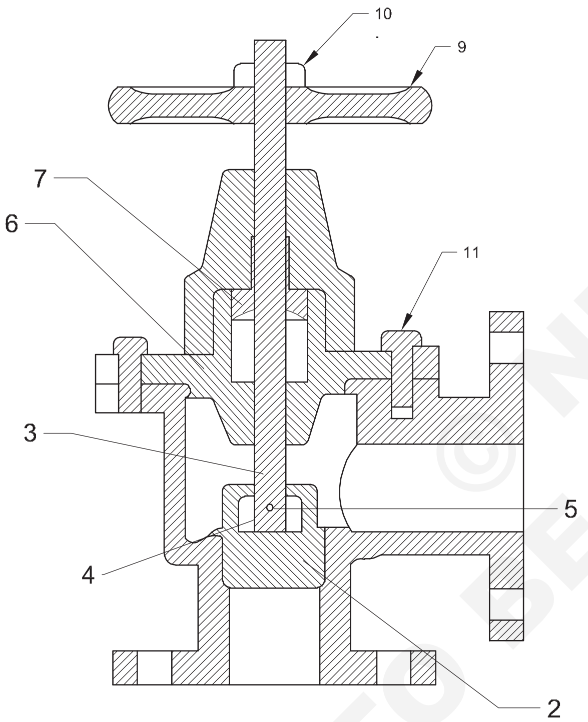
9.HANDWHEEL



10. NUT



11. BOLT



STEAM STOP VALVE ASSEMBLY

PARTS LISTS				
ITEM	QTY	PART NAME	DESCRIPTION	MATERIAL
1	1	BODY	PART	IRON,CAST
2	1	VALVE	PART	STAINLESS STEEL
3	1	SPINDLE	PART	STAINLESS STEEL
4	1	COLLAR	PART	IRON,GRAY
5	1	PIN	PART	STEEL
6	1	COVER	PART	IRON,CAST
7	1	GLAND	PART	STEEL,CAST
8	1	GLAND NUT	PART	STEEL
9	1	HANDWHEEL	PART	STAINLESS STEEL
10	1	NUT	PART	STEEL,CAST
11	4	BOLT	PART	STEEL,CAST

SCALE 1:1	**DETAILS OF STEAM STOP VALVE**	EX NO : 2.7.119
		TIME : 28hrs
PROJECTION		CODE : DM20N27119E2

Hydraulics & pneumatics conventional signs and symbols

Objectives: At the end of this exercise you shall be able to
- **draw the hydraulic symbols used in mechanic drawing**
- **draw the pneumatic symbols used in mechanic drawing.**

Requirements

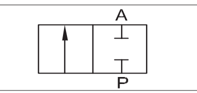
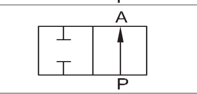
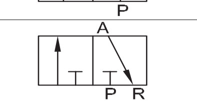
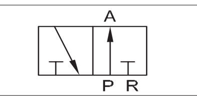
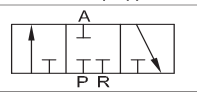
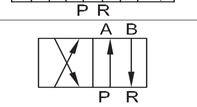
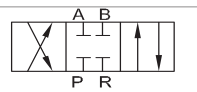
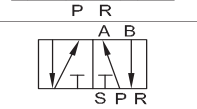
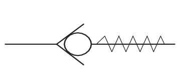
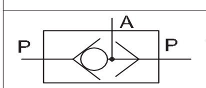
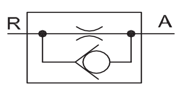
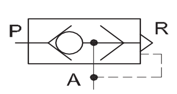
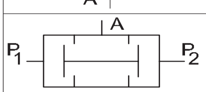
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

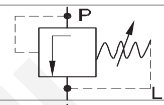
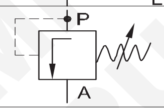
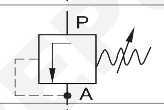
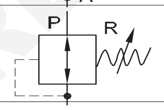
PROCEDURE

TASK 1: **Hydraulics connections and symbols (IS1219,1:1991)**

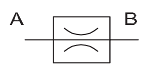
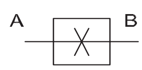
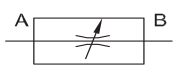
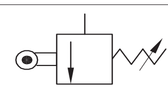
i Draw the control valve symbols and its description.

CONTROL VALVE	DIRECTIONAL CONTROL VALVE
	2/2 DIRECTIONAL CONTROL VALVE - NORMALLY CLOSED
	2/2 DIRECTIONAL CONTROL VALVE - NORMALLY OPEN
	3/2 D.C. VALVE NORMALLY CLOSED
	3/2 D.C. VALVE NORMALLY OPEN
	3/3 D.C.VALVE - ZERO POSITION ALL PORTS CLOSED
	4/2 D.C.VALVE
	4/3 D.C.VALVE - ZERO POSITION ALL PORTS CLOSED
	5/2 D.C. VALVE
	NON RETURN VALVE
	SHUTTLE VALVE
	FLOW CONTROL VALVE
	QUICK EXHAUST VALVE
	TWIN PRESSURE SEQUENCE VALVE

ii Draw the Pressure control valve symbols and it's descriptions

PRESSURE CONTROL VALVE	
	PRESSURE RELIFE VALVE
	SEQUENCE VALVE
	PRESSURE REGULATOR
	PRESSURE REGULATOR WITH SELF RELIEVING

III Draw the Flow valve symbols and it's description.

FLOW VALVE	
	FLOW CONTROL GEN.SYMBOL
	FLOW CONTROL INSIGNIFICANT INFLUENCE OF VISCOSITY
	ADJUSTABLE FLOW CONTROL
	FLOW CONTROL VALVE CONTROLLED MECHANICALLY WITH SPRING RETURN

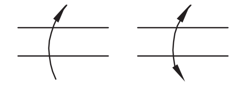
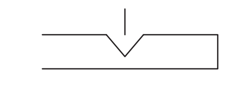
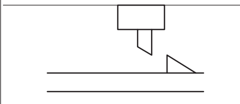
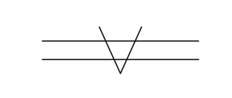
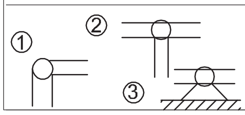
IV Draw the Shut – off valve and it's description

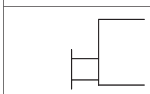
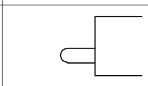
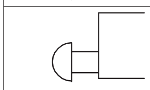
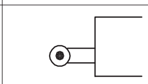
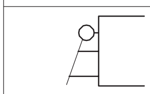
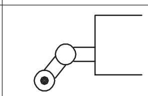
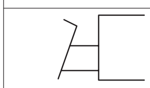
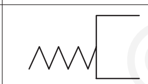
SHUT OFF VALVE	
	SHUT - OFF VALVE

V Draw the Actuation's mechanical components symbols and its descriptions.

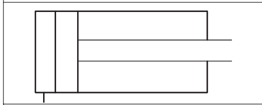
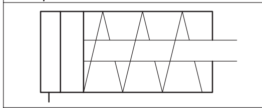
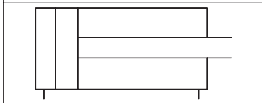
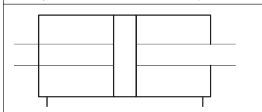
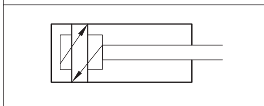
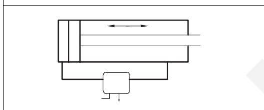
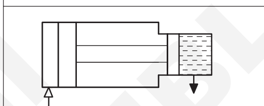
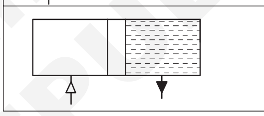
Vi Draw the control methods symbols and its description.

Hydraulic symbols

ACTUATIONS - MECHANICAL COMPONENTS	
	ONE DIRECTION OF ROTATION OR TWO RESPECTIVELY OF SHAFT.
	DETENT
	LOCKING DEVICE
	OVER - CENTRE DEVICE
	LINKAGES 1) SIMPLE. 2) WITH LINKAGE 3) WITH FIXED FULCRUM.

CONTROL METHODS			
MANUAL CONTROAL		MECHANICAL CONTROAL	
	GENERAL SYMBOL		PLUNGER
	PUSH BUTTON		ROLLER
	LEVER		ROLLER TRIP
	PEDAL		SPRING

vii Draw th cylinder symbols and its description.

CYLINDER	
	SINGLE ACTING CYLINDER
	SINGLE ACTING CYLINDER RETURN BY SPRING
	DOUBLE ACTING CYLINDER
	DOUBLE ACTING CYLINDER WITH THROUGH ROD
	D.A. CYLINDER WITH ADJUSTABLE CUSHIANING OF BOTH END
	CYLINDER WITH BUILT ON CONTROL VALVE
	PRESSURE INTENSIFIER
	PRESSURE MEDIUM CHANGER

Task 2: **Pneumatics connections and symbols (IS,1219:1991)**

i Draw the pneumatic circuits (IS,1219:1991) symbols and it's description

ii Draw the Exhaust (PNEU) symbols and their descriptions.

iii Draw the power take off symbols and their descriptions

iv Draw the quick release coupling symbols and their description.

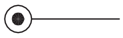
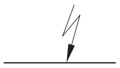
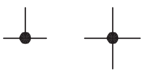
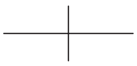
v Draw the servicing (PNEU) symbols and their descriptions.

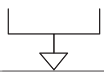
vi Draw the air motor conventional symbols and their description.

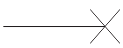
Draw the conventional symbols to a suitable scale and print their descriptions.

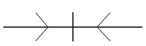
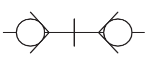
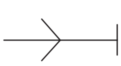
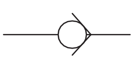
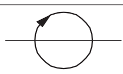
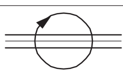
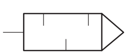
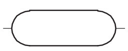
Pneumatic symbols

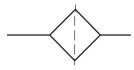
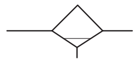
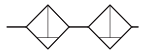
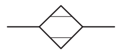
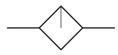
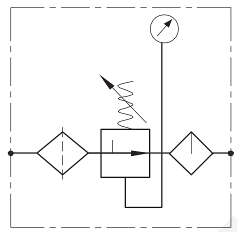
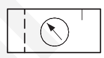
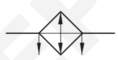
SYMBOLS USED IN PNEUMATIC AND HYDRAULIC
(IS 1219 - 1 : 1991)

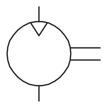
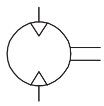
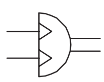
PNEUMATIC CIRCUITS	
	MAIN SUPPLY PRESSURE
	WORKING RETURN & FEED LINE
	CONTROL LINE
	DRAIN LINE
	ASSEMBLY LINE
	ELECTRICAL LINE
	FLIXIBLE LINE
	LINE JUCTION
	LINE CROSSING

EXHAUST (PNEU)	
	UNTHREADED AIR EXHAUST PORT
	THREADED AIR EXHAUST PORT

POWER TAKE OFF	
	PLUGGING
	PLUGGING WITH TAKE OFF LINE

QUICK RELEASE COUPLING	
	COUPLED WITHOUT CHECK VALVE
	COUPLE WITH CHECK VALVE
	HALF OF QUICK RELEASE COUPLING WITHOUT VALVE
	HALF OF QUICK RELEASE COUPLING WITH CLOSED VALVE
	ONE LINE OF FLOW
	THREE LINES OF FLOW
	SILENCER (PNEU)
	ACCUMULATOR

SERVICING (PNEU)	
	FILTER
	WATER TRAP MANUAL DRAIN
	WATER TRAP - AUTOMATIC DRAINING
	FILTER WITH WATER TRAP & DRAIN
	DESICATOR AIR DRY BY CHEMICALS
	LUBRICATOR
	SERVICE UNIT (DETAILED SYMBOL)
	SERVICE UNIT (SIMPLE SYMBOL)
	COOLER WITHOUT FLOW LINES
	COOLER WITH FLOW LINES

AIR MOTOR	
	PNEUMATIC MOTOR UNI - DIRECTIONAL
	PNEUMATIC MOTOR TWO DIRECTIONAL
	OSCILLATING MOTOR

Draw sectional view of hydraulic jack and pneumatic valve actuator

Objectives: At the end of this exercise you shall be able to
- **draw hydraulic and pnematic valve actuator.**

Requirements

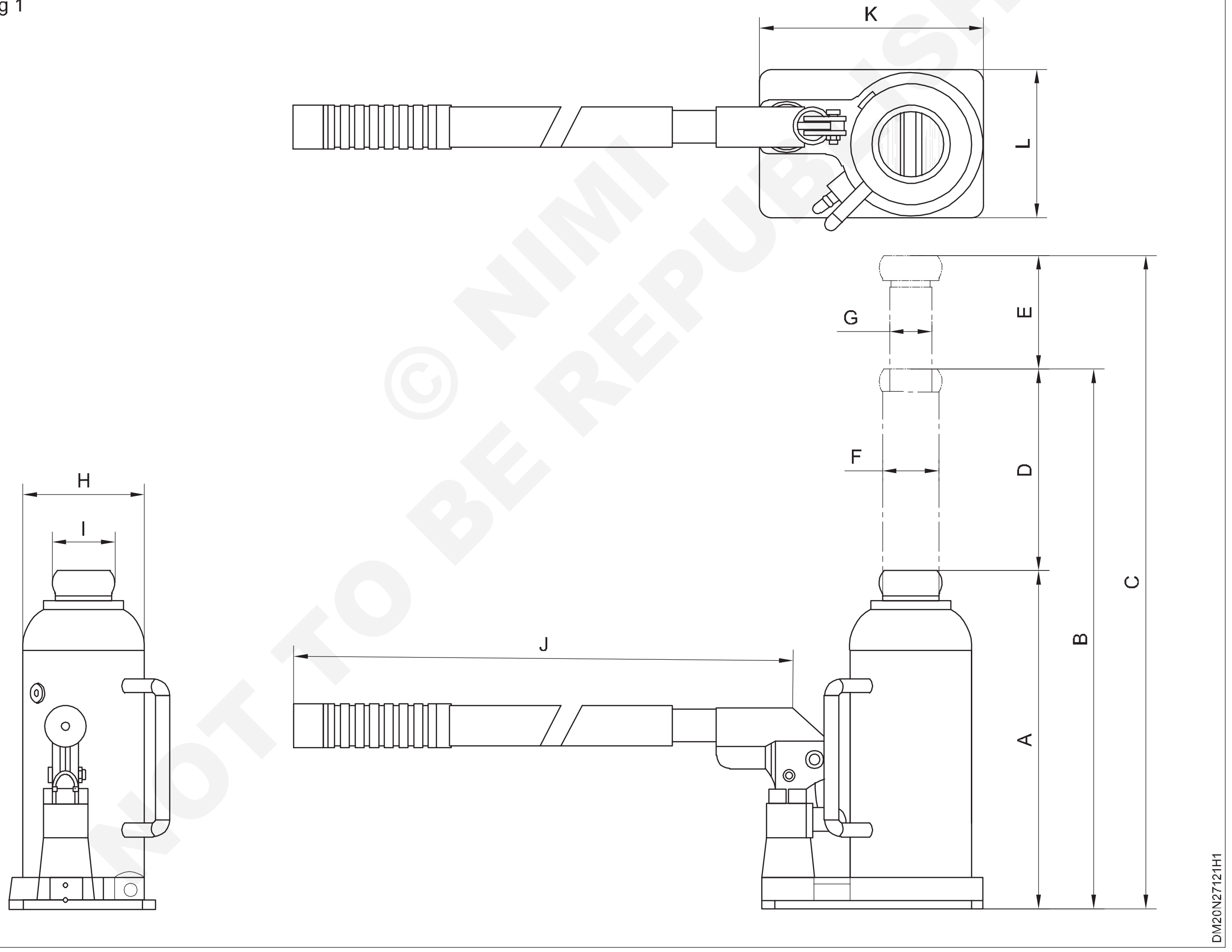
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw orthographic view of hydraulic and pnematic valve actuator (Fig 1)**

Fig 1



Draw sectional view of a volute casing centrifugal pump

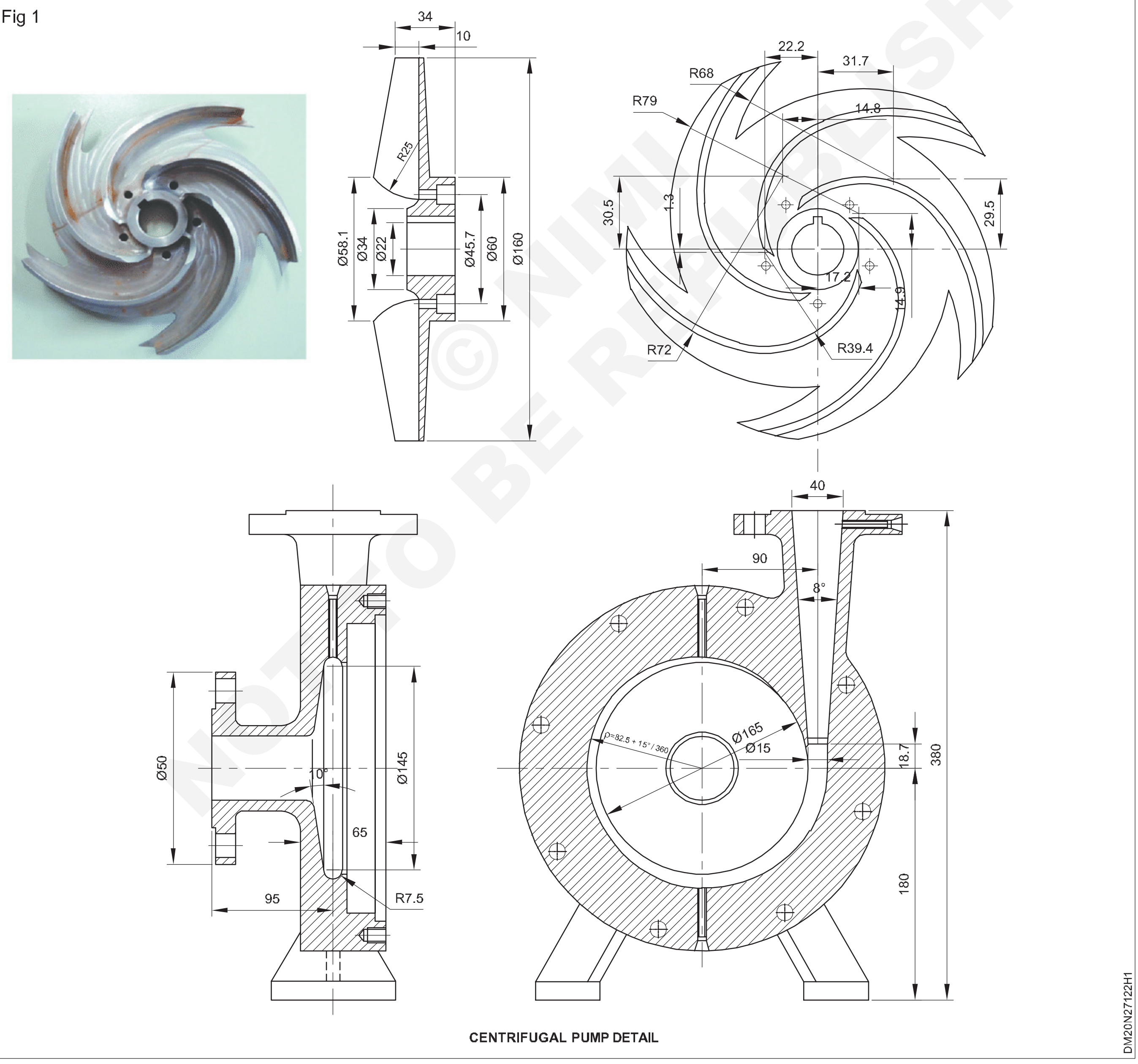
Objectives: At the end of this exercise you shall be able to
- **draw the sectional view of centrifugal pump.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the sectional view of centrifugal pump (Fig 1)**

Fig 1



CENTRIFUGAL PUMP DETAIL

Draw the detailed drawing of a lathe tool post

Objectives: At the end of this exercise you shall be able to
- **draw the detailed drawing of length tool post using AutoCAD.**

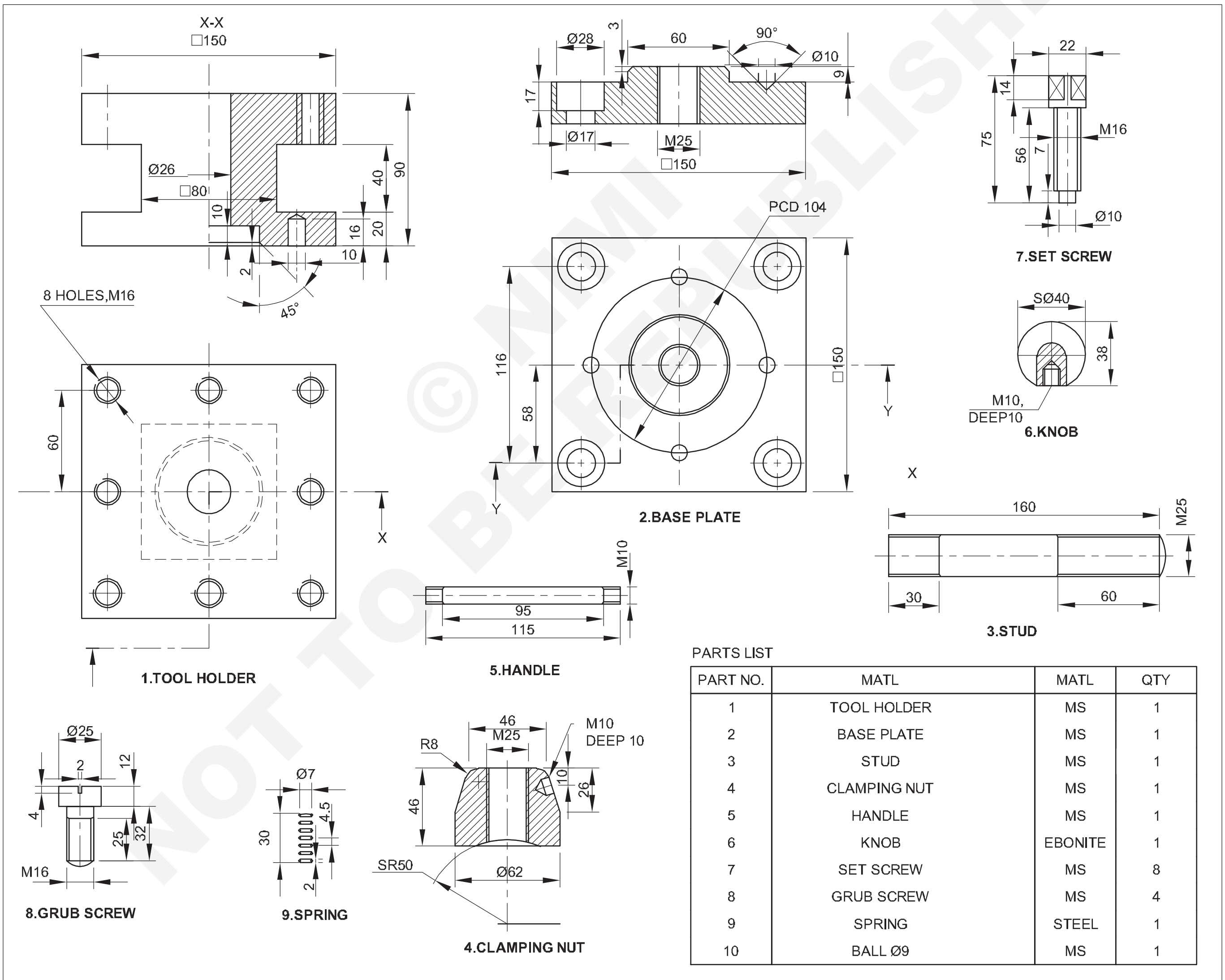
Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

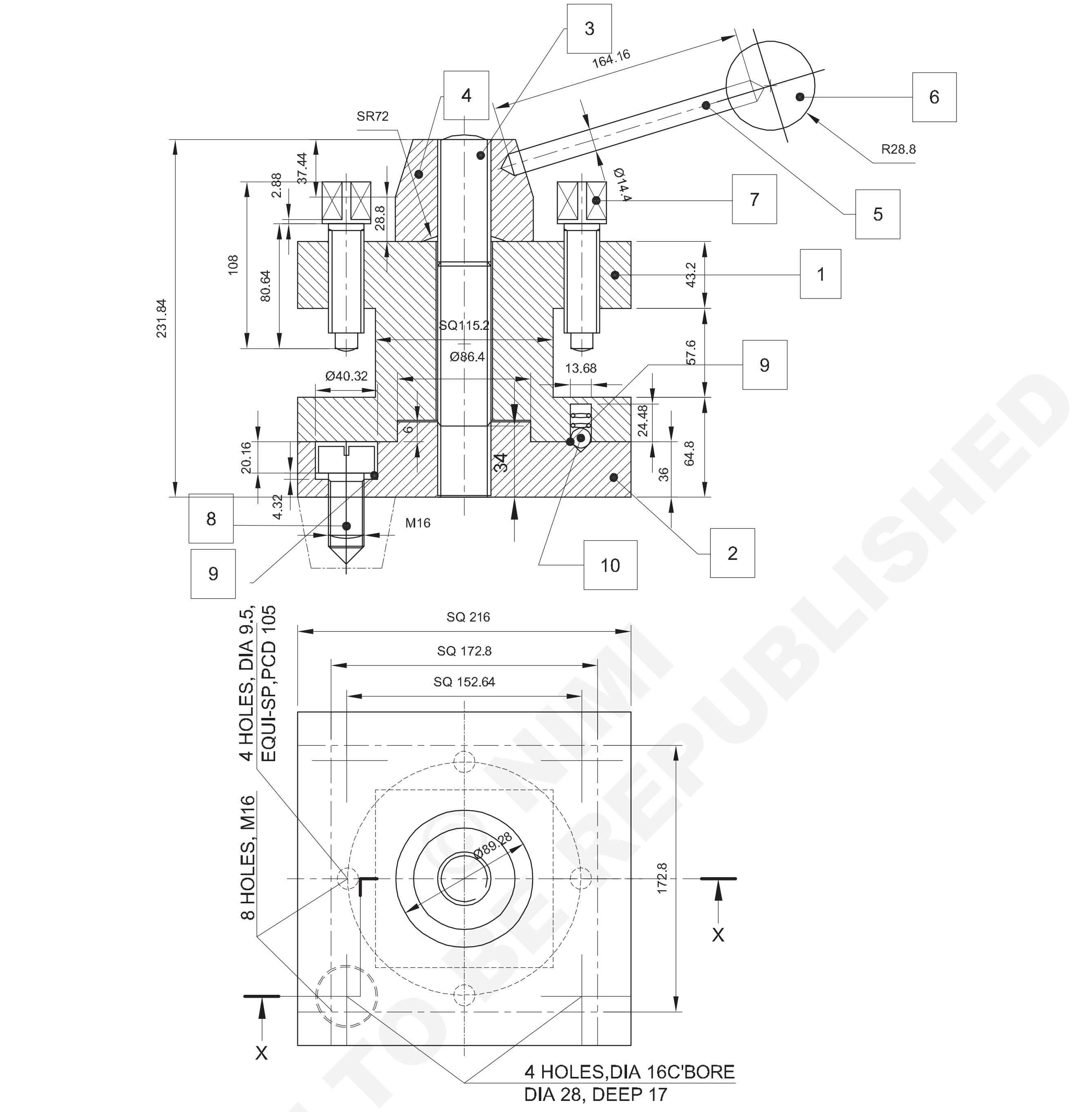
TASK 1: **Draw the detailed drawing of lathe tool post using AutoCAD**



PARTS LIST

PART NO.	MATL	MATL	QTY
1	TOOL HOLDER	MS	1
2	BASE PLATE	MS	1
3	STUD	MS	1
4	CLAMPING NUT	MS	1
5	HANDLE	MS	1
6	KNOB	EBONITE	1
7	SET SCREW	MS	8
8	GRUB SCREW	MS	4
9	SPRING	STEEL	1
10	BALL Ø9	MS	1

SCALE 1:1	LATHE TOOL POST (DETAILS)	EX NO : 2.7.123
		TIME : 28hrs
PROJECTION		CODE : DM20N27123E1



SCALE 1:1	**SQUARE TOOL POST ASSEMBLY**	EX NO : 2.7.123
		TIME : 28hrs
PROJECTION		CODE : DM20N27123E2

Construct detailed drawing of tail stock

Objectives: At the end of this exercise you shall be able to
- **draw the detailed drawing of rain stock.**

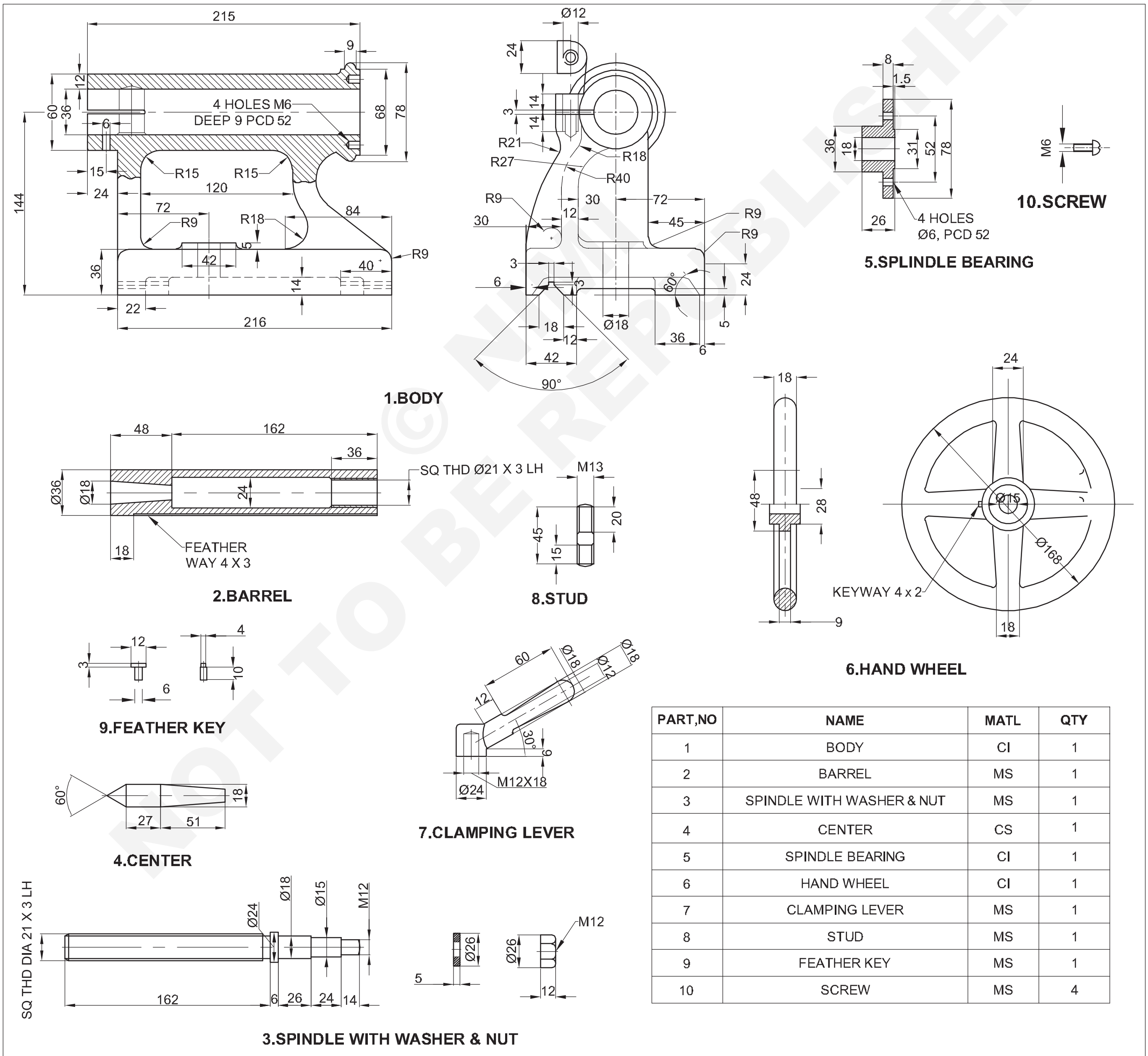
Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

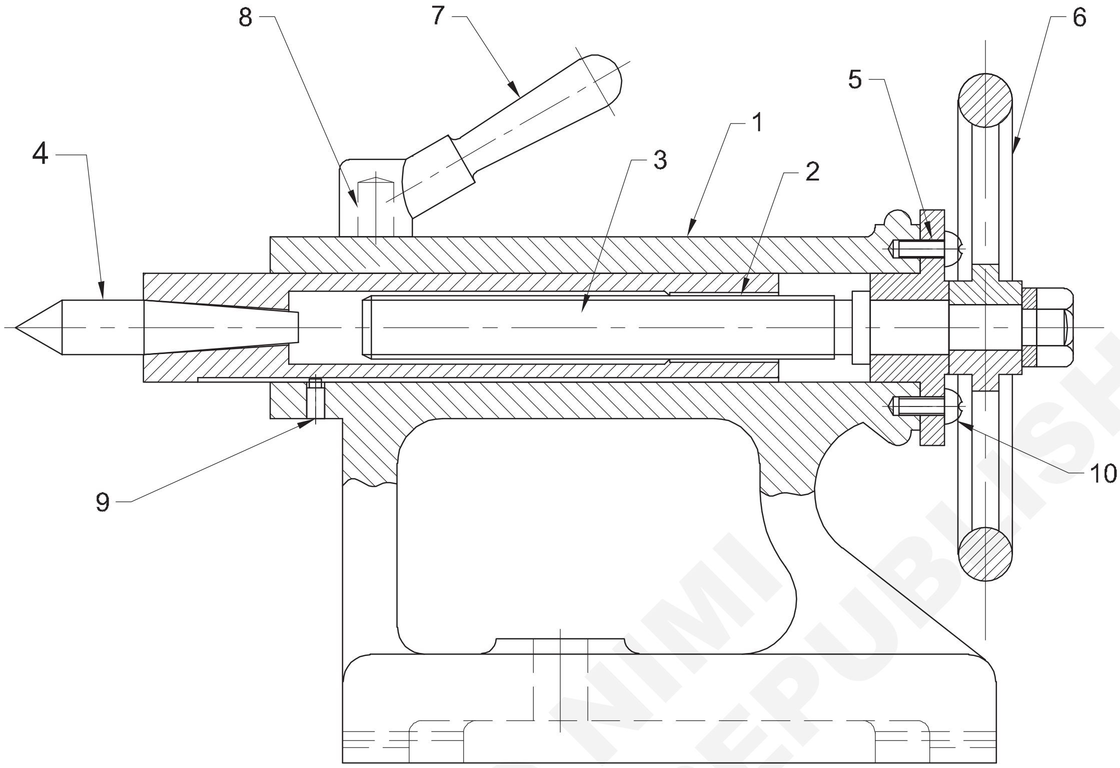
PROCEDURE

TASK 1: **Draw the detailed drawing of tail stock**

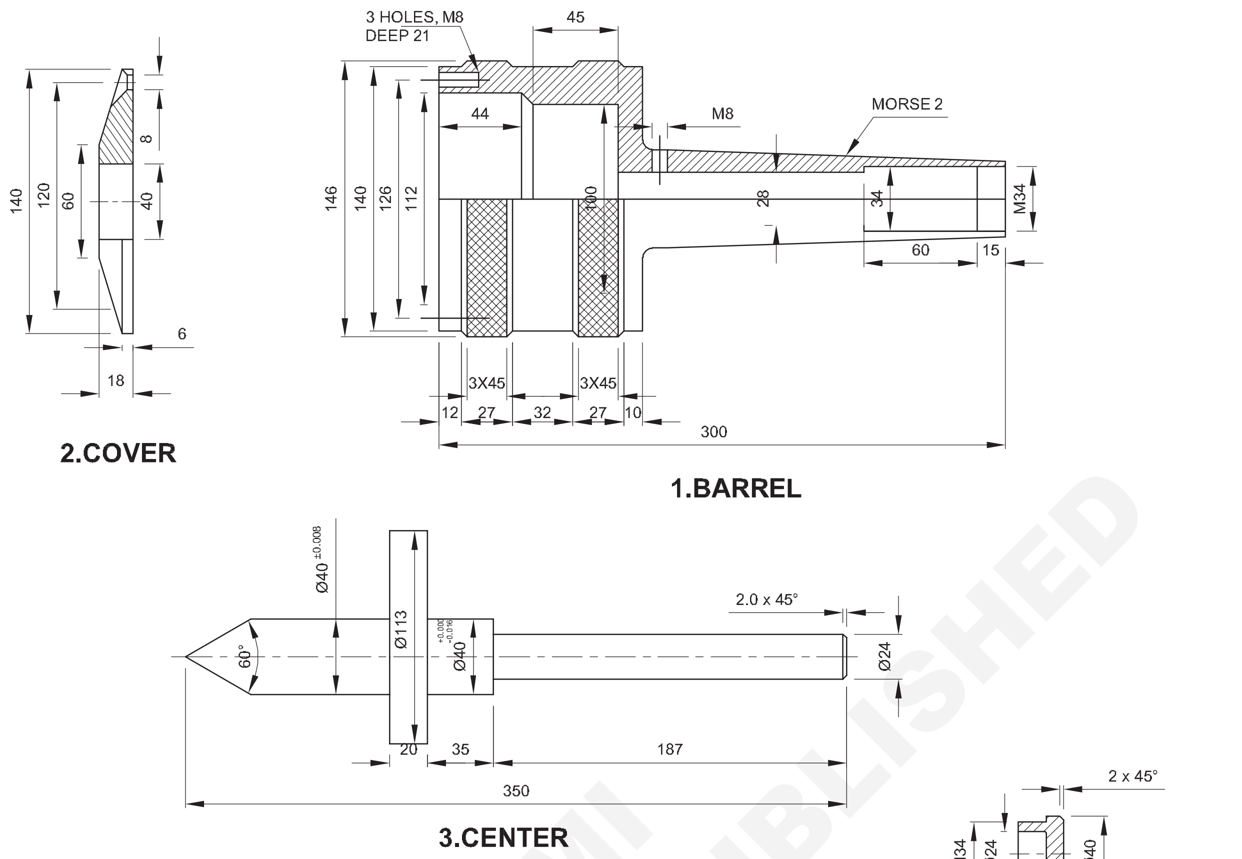


PART,NO	NAME	MATL	QTY
1	BODY	CI	1
2	BARREL	MS	1
3	SPINDLE WITH WASHER & NUT	MS	1
4	CENTER	CS	1
5	SPINDLE BEARING	CI	1
6	HAND WHEEL	CI	1
7	CLAMPING LEVER	MS	1
8	STUD	MS	1
9	FEATHER KEY	MS	1
10	SCREW	MS	4

SCALE 1:1	LATHE PART TAIL STOCK (DETAILS)	EX NO : 2.7.124
		TIME : 28hrs
PROJECTION		CODE : DM20N27124E1

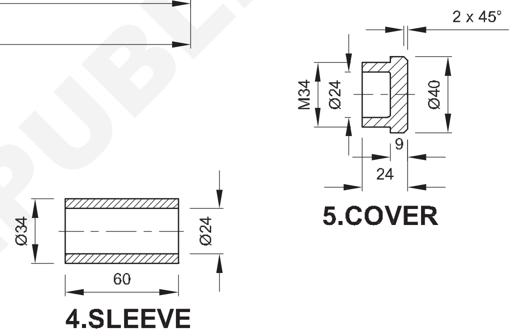


SCALE 1:1	**LATHE PART TAIL - STOCK ASSEMBLY**	EX NO : 2.7.124
		TIME : 28hrs
PROJECTION		CODE : DM20N27124E2



PARTS LIST

PART,NO	NAME	MATL	QTY
1	BARREL	MS	1
2	COVER	MS	1
3	CENTER	ALLOY STEEL	1
4	SLEEVE	MS	1
5	COVER	MS	1



SCALE 1:1	TAIL STOCK- REVOLVING CENTRE (DETAILS)	EX NO : 2.7.124
		TIME : 28hrs
PROJECTION		CODE : DM20N27124E2

Construct detailed drawing of a milling fixture

Objectives: At the end of this exercise you shall be able to
- **draw a detailed drawing of milling fixture.**

Requirements

Tools/Equipment/Machines

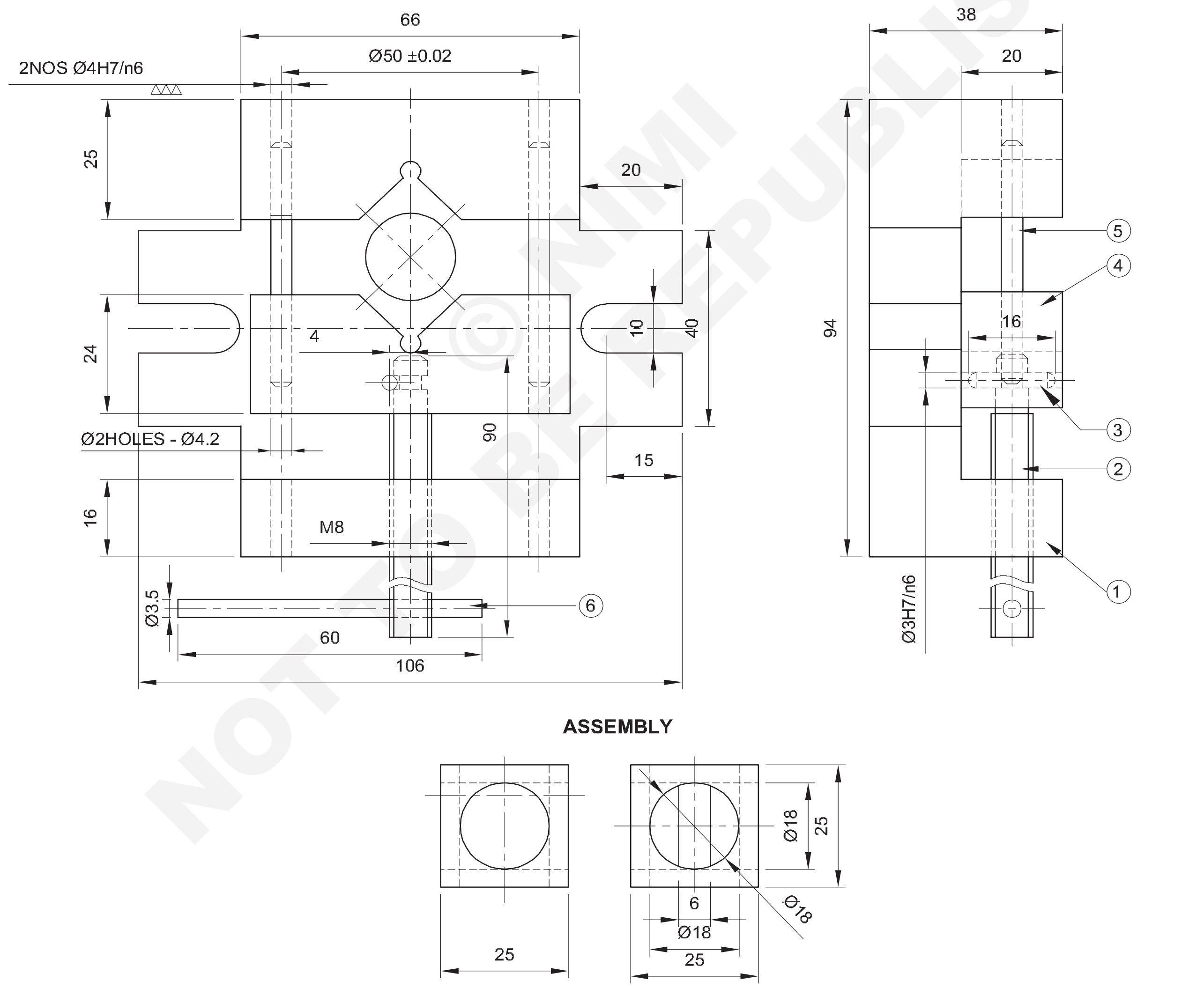
- Auto CAD 2018 or Higher version.

PROCEDURE

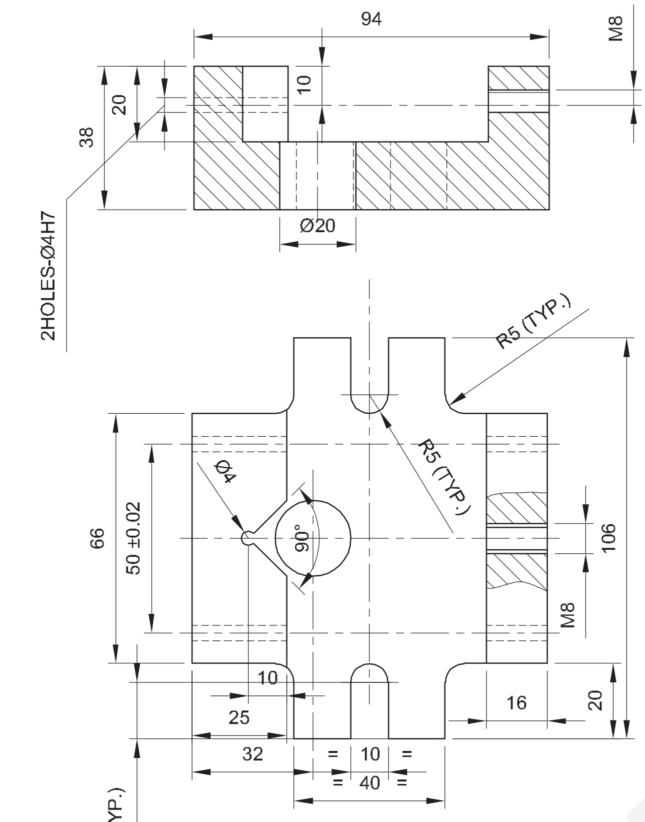
TASK 1: **Draw a detailed drawing of milling fixture**

Milling fixture

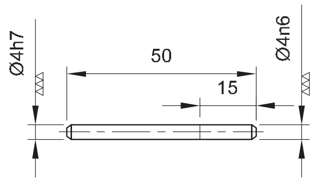
The three orthographic views of the Assembly of a Milling fixture production drawing are shown in Fig. The fixture is designed to cut a Slot in a round rod of 15 mm diameter at its end as shown in the drawing. The slot is cut on a vertical milling Machine with a slitting saw cutter.



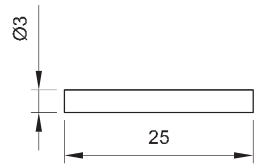
SCALE 1:1	COMPONENT MILLING FIXTURE ASSEMBLY	EX NO : 2.7.125
		TIME : 28hrs
PROJECTION		CODE : DM20N27125E1



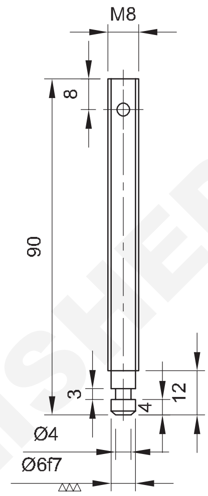
1. BASE PLATE



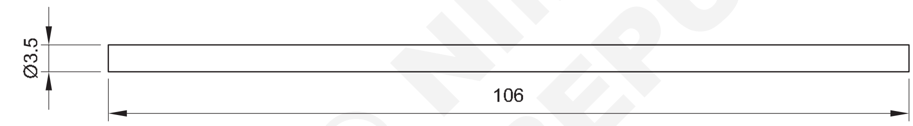
DETAILS OF ITEM - 5
GUIDE PIN



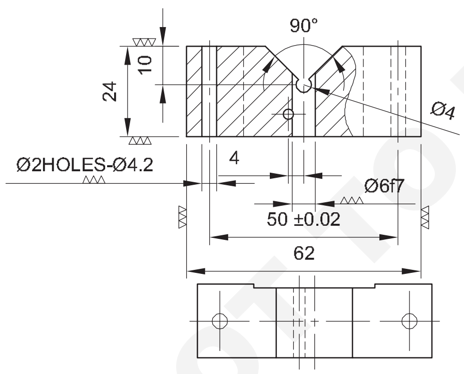
3.DOWEL PIN

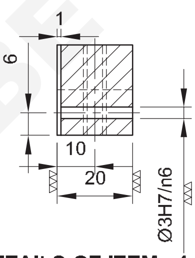


DETAILS OF ITEM - 2
SPL.SCREW



6.HANDLE





DETAILS OF ITEM - 4
MOVABLE JAW

PART NO	NAME	MATERIAL	QTY
1	BASE PLATE	MS	1
2	SPL.SCREW	MS	1
3	DOWEL	MS	2
4	MOVABLE JAW	CARBON TOOL STEEL	1
5	GUIDING PIN	MS	2
6	HANDLE	CARBON TOOL STEEL	1

SCALE 1:1	MILLING FIXTURE (DETAILS)	EX NO : 2.7.125
		TIME : 28hrs
PROJECTION		CODE : DM20N27125E2

Construct detailed assembly drawing of shaper tool head slide

Objectives: At the end of this exercise you shall be able to
- **draw detailed drawing of shaper tool head.**

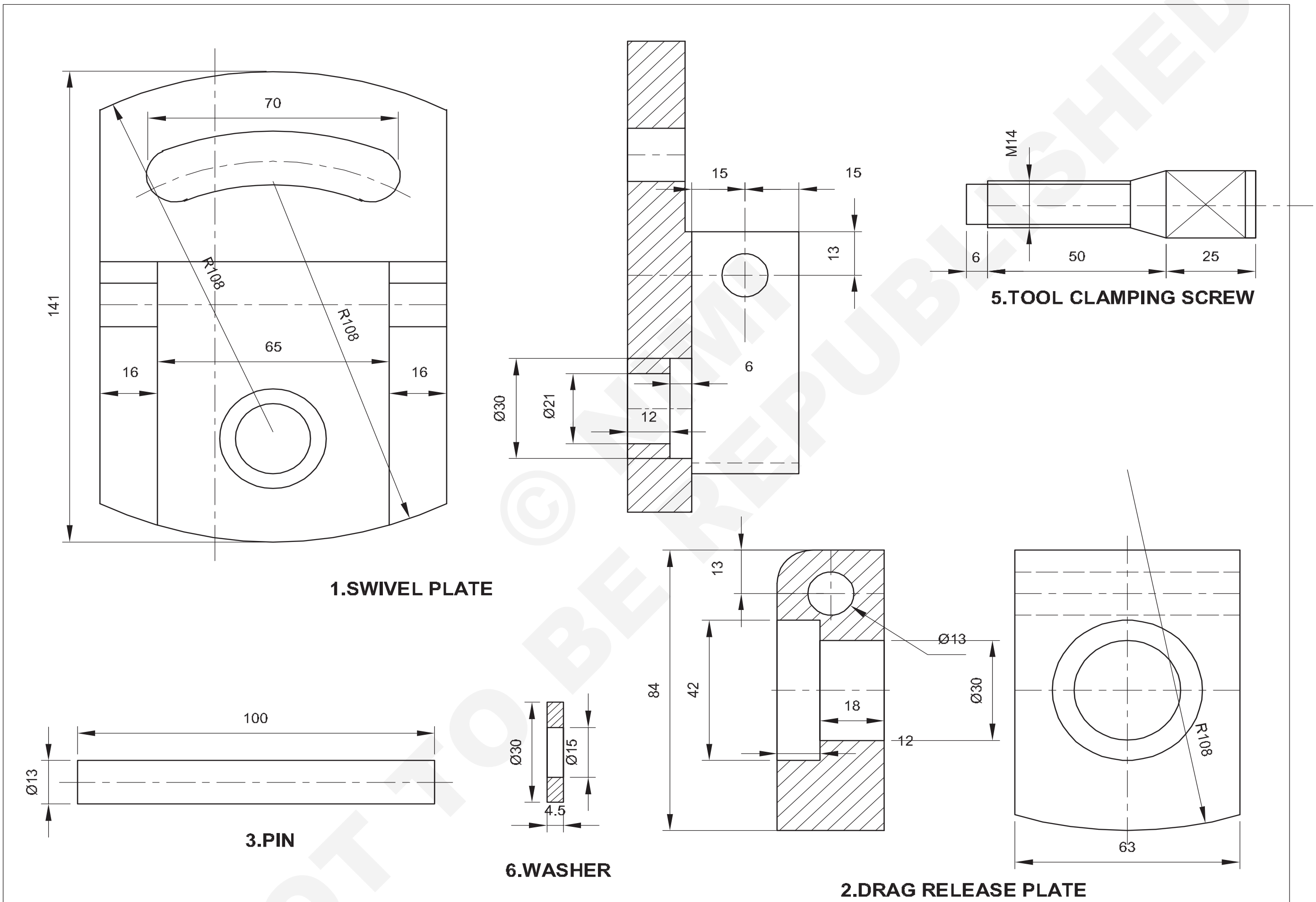
Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw detailed drawing of shaper tool head**



PART NO	NAME	MATL	QTY
1	SWIVEL PLATE	CI	1
2	DRAG RELEASE PLATE	CI	1
3	PIN	MS	1
4	TOOL HOLDER	MCS	1
5	TOOL CLAMPING SCREW	MS	1
6	WASHER	MS	1

SCALE 1:1	SHAPER MACHINE TOOL POST (DETAILS)	EX NO : 2.7.126
		TIME : 28hrs
PROJECTION		CODE : DM20N27126E1

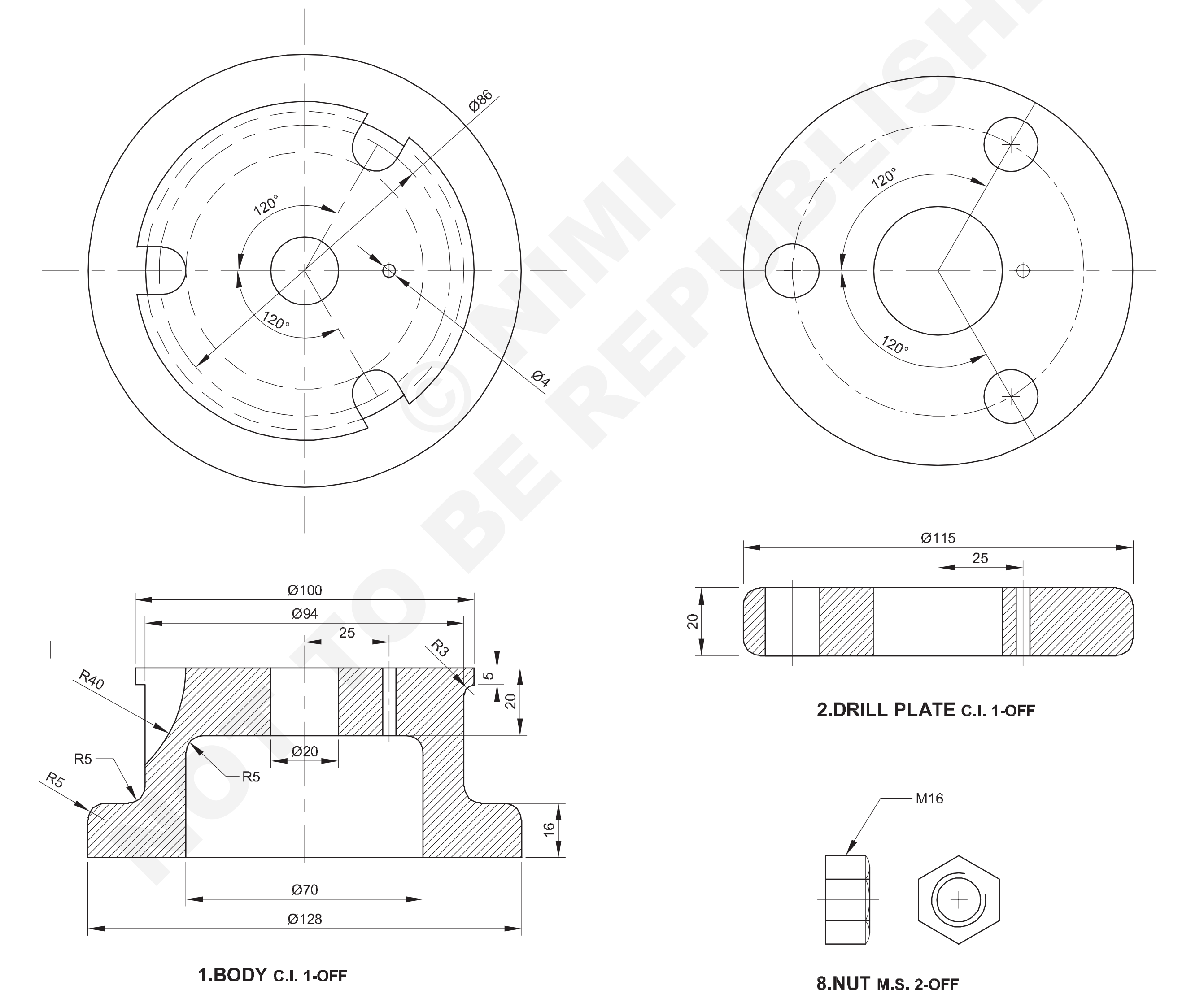
Draw a simple drilling Jig for drilling holes

Objectives: At the end of this exercise you shall be able to
- **draw detailed drawing of dril jig.**

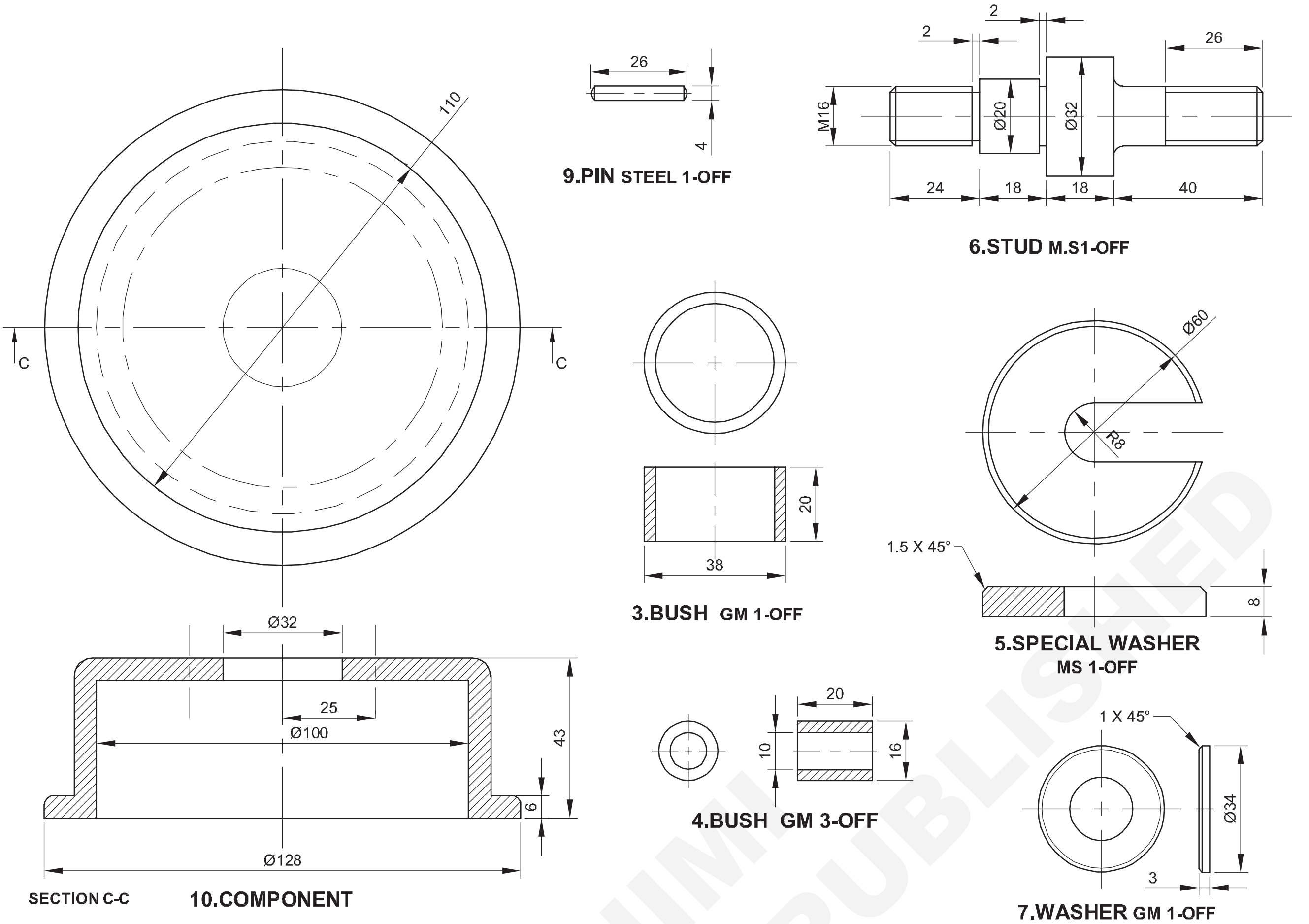
Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw detailed drawing of drill jig**



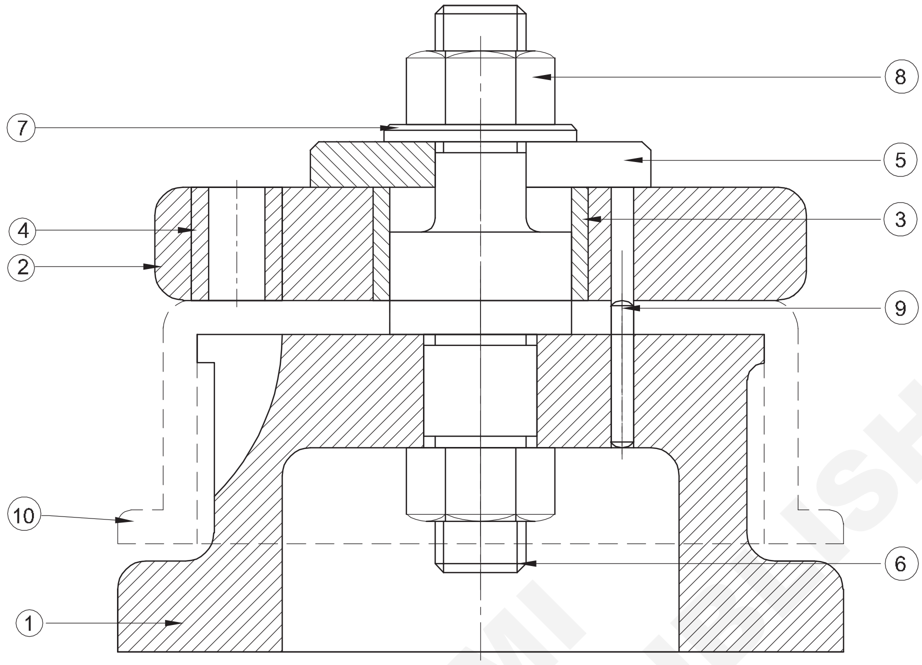
SCALE 1:1	DRILLING JIG (DETAILS)	EX NO : 2.7.127
		TIME : 28hrs
PROJECTION		CODE : DM20N27127E1



PRT NO.	NAME OF THE PART	MATERIAL	QTY
1	BODY	CAST IRON	1
2	DRILL PLATE	CAST IRON	1
3	BUSH	GUN METAL	1
4	BUSH	GUN METAL	3
5	SPECIAL WASHER	MILD STEEL	1

PRT NO.	NAME OF THE PART	MATERIAL	QTY
6	STUD	MILD STEEL	1
7	WASHER	MILD STEEL	1
8	NUT	MILD STEEL	2
9	PIN	STEEL	1
10	COMPONENT	CAST STEEL	1

SCALE 1:1	**DRILLING JIG (DETAILS)**	EX NO : 2.7.127
		TIME : 28hrs
PROJECTION		CODE : DM20N27127E2



SCALE 1:1	DRILLING JIG ASSEMBLY	EX NO : 4.1.125
		TIME : 28hrs
PROJECTION		CODE : DMN41125E3

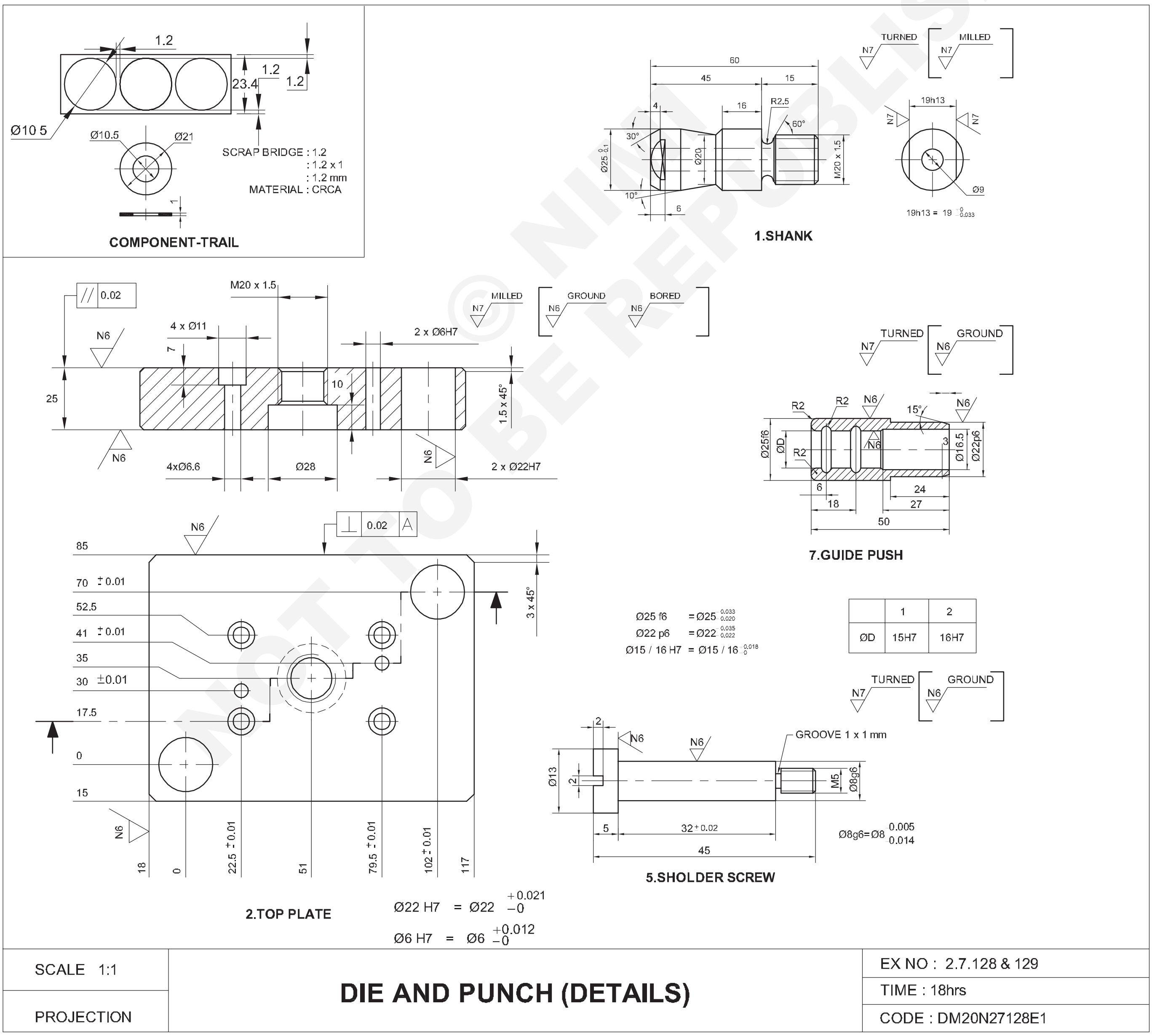
Draw the detailed drawing of a Press tools - dies - punches

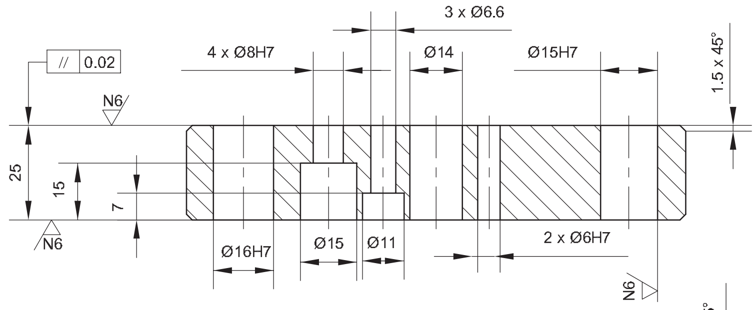
Objectives: At the end of this exercise you shall be able to
- **draw the details of punch and die by using CAD**
- **draw the Assembly view of the punch and dye by using CAD**
- **draw the exploded view by using CAD**
- **prepare the bill of material.**

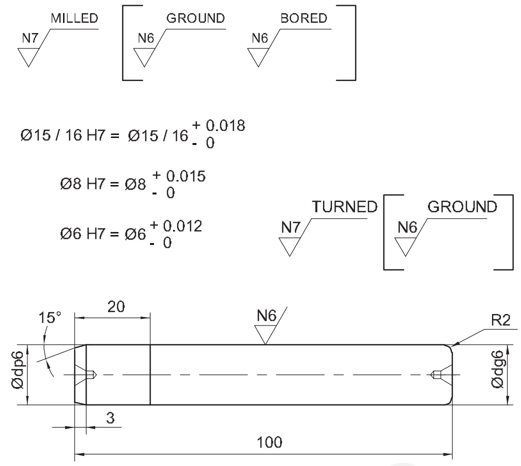
Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

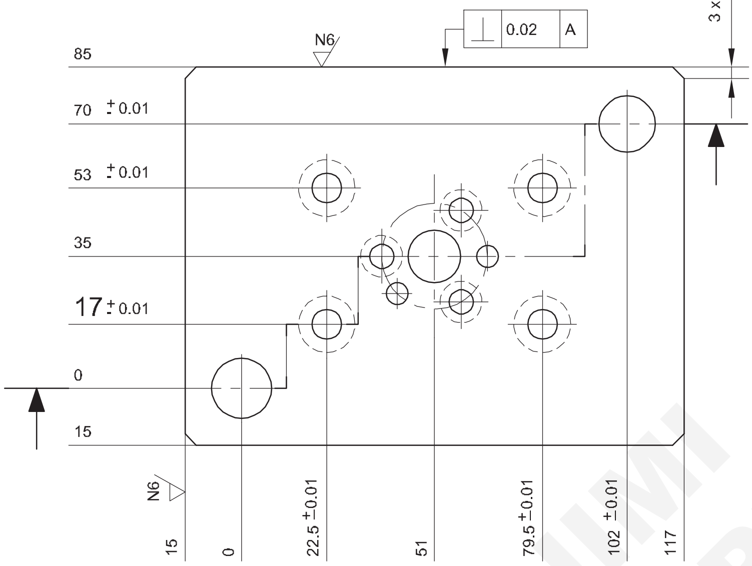
TASK 1: **Draw the details of punch and dye by using CAD (Fig 1)**

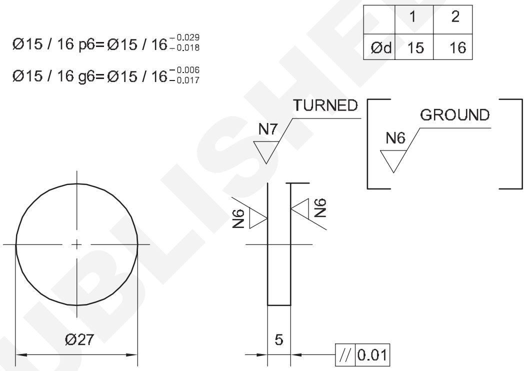






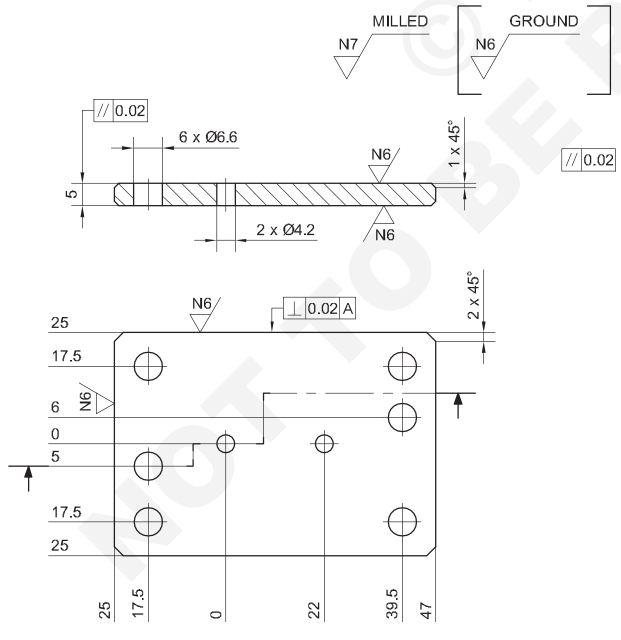
6.GUIDE PILLAR

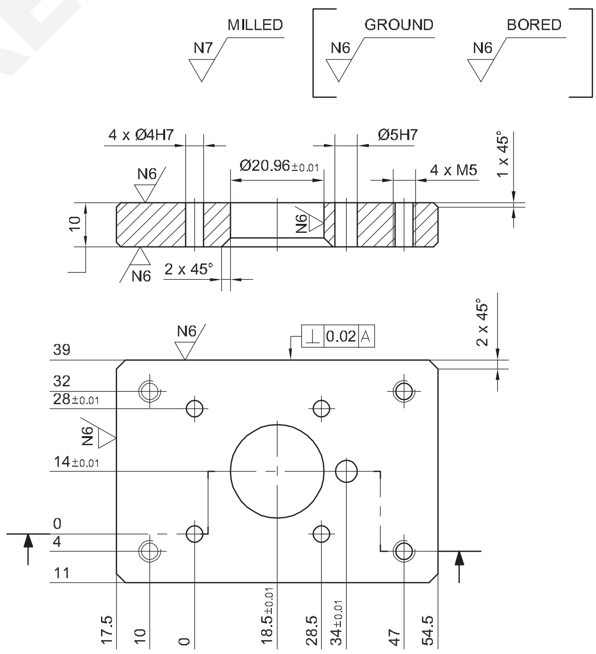




8.TRANSFER PLATE

3.BOTTOM PLATE

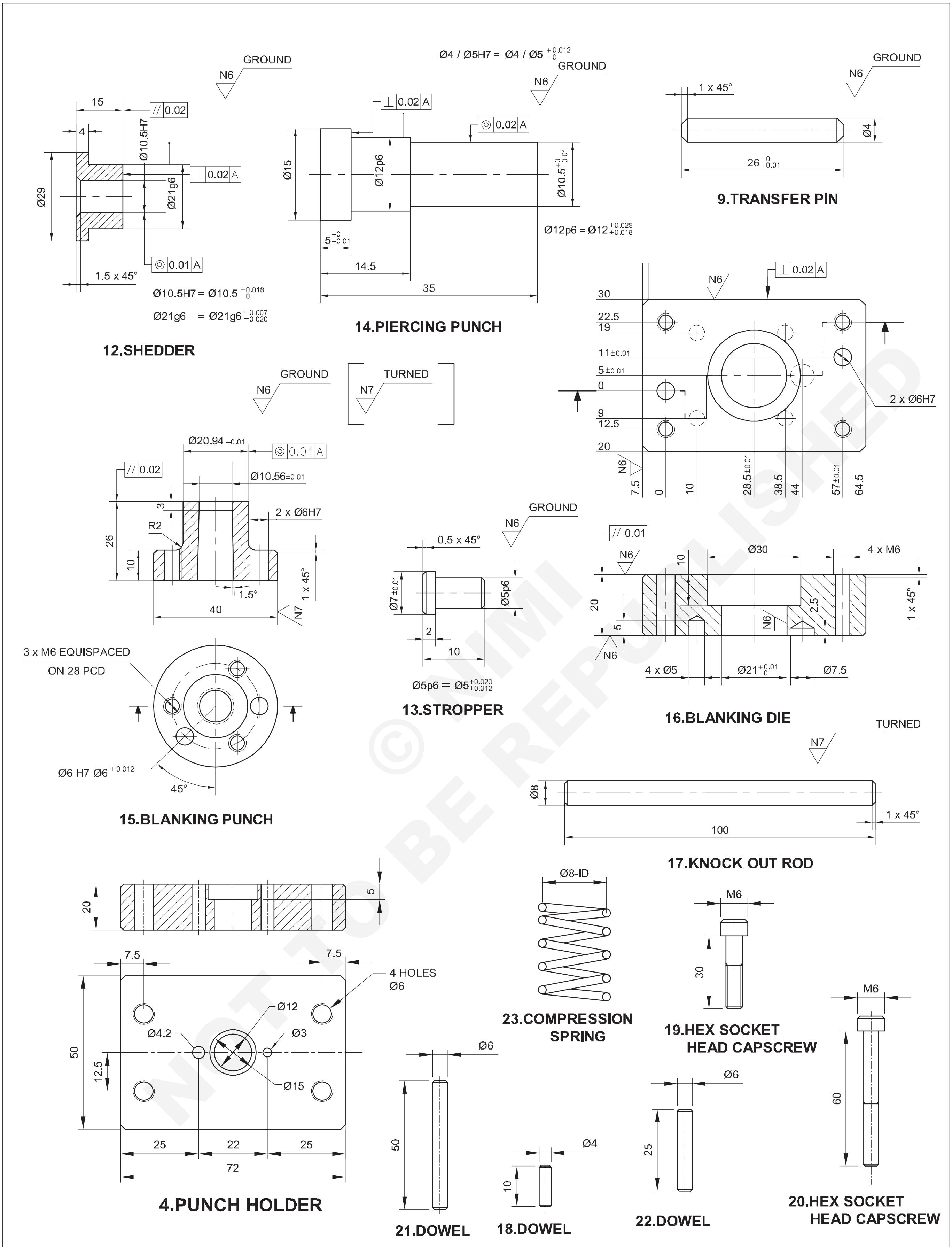




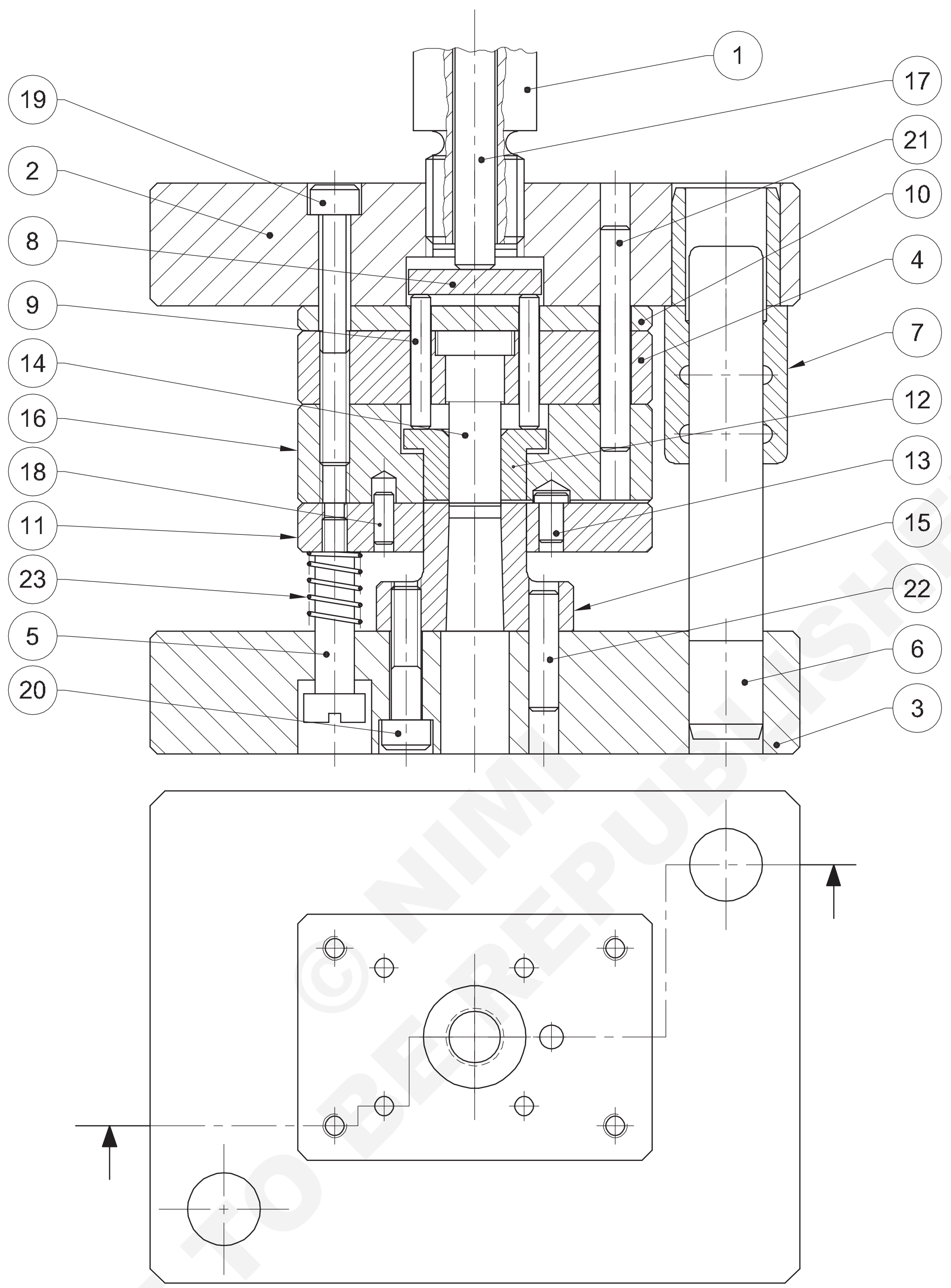
10.BACK PLATE

11.STRIPPER

SCALE 1:1	DIE AND PUNCH (DETAILS)	EX NO : 2.7.128 & 129
		TIME : 18hrs
PROJECTION		CODE : DM20N27128E2

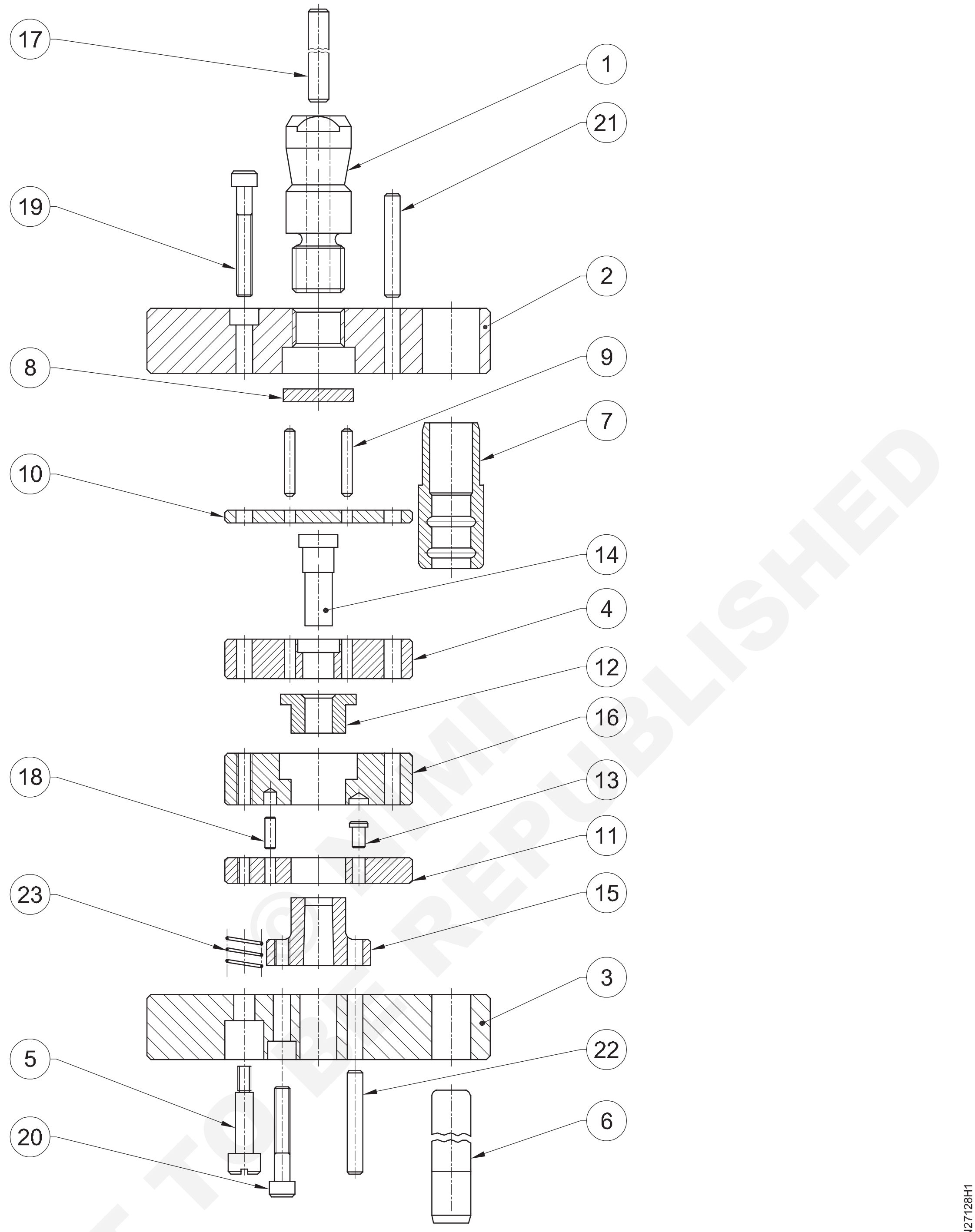


SCALE 1:1	PRESS TOOL(DETAILS)	EX NO : 2.7.128 &129
		TIME : 28hrs
PROJECTION		CODE : DM20N27128E3



SCALE 1:1	**COMPOUND DIE ASSEMBLY**	EX NO : 2.7.128 & 129
		TIME : 18hrs
PROJECTION		CODE : DM20N27128E4

Fig 1



COMPOUND DIE ASSEMBLY - EXPLODED VIEW

BILL OF MATERIAL

4	COMPRESSION SPRING	I.D=15,WØ=1.2,P=4,L=25	SPRING STEEL		23
2	DOWEL	Ø6 x 25	Fe310	STD	22
2	DOWEL	Ø6 x 50	Fe310		21
3	HEX. SOCKET HEAD CAP SCREW	M6 x 30 IS 2269-12.9	Fe310		20
4	HEX. SOCKET HEAD CAP SCREW	M6 x 60 IS 2269-12.9	Fe310		19
4	DOWEL	Ø4 x 10	Fe310	STD	18
1	KNOCKOUT ROD	ISRO 10-110	Fe310		17
1	BLANKING DIE	75 ISF 25-55	Fe310		16
1	BLANKING PUNCH	ISRO 45-40	Fe310		15
1	PIERCING PUNCH	ISRO 18-45	Fe310		14
1	STOPPER	ISRO 10-20	Fe310		13
1	SHEDDER	ISRO 36-25	Fe310		12
1	STRIPPER	60 ISF 15-77	Fe310		11
1	BACK PLATE	60 ISF 10-77	Fe310		10
2	TRANSFER PIN	ISRO 6-40	Fe310		09
1	TRANSFER PLATE	ISRO 32-20	Fe310		08
2	GUIDE BUSH	ISRO 28-55	Fe310		07
2	GUIDE PILLAR	ISRO 20-115	Fe310		06
4	SHOULDER SCREW	ISRO 16-50	Fe310		05
1	PUNCH HOLDER	60 ISF 20-77	Fe310		04
1	BOTTOM PLATE	140 ISF 30-105	Fe310		03
1	TOP PLATE	140 ISF 30-105	Fe310		02
1	SHANK	ISRO 28-65	Fe310		01
NO.OFF	DESCRIPTION	STOCK SIZE	MATERIAL	REMARKS	PART NO.

SCALE 1:1	**DIE AND PUNCH (DETAILS)**	EX NO : 2.7.128 & 129
		TIME : 18hrs
PROJECTION		CODE : DM20N27128E5

- Draw all the details of the die and punch as given in the drawing
- In corporate all the dimensions in the detail drawing
- Indicate all the machinery symbols
- Add the part list

Develop isometric drawing for injection mould with side cavities

Objectives: At the end of this exercise you shall be able to
- **draw the isometric view of the moulding ejected from injection mould**
- **draw the cross-section view of the two plate mould.**

Requirements

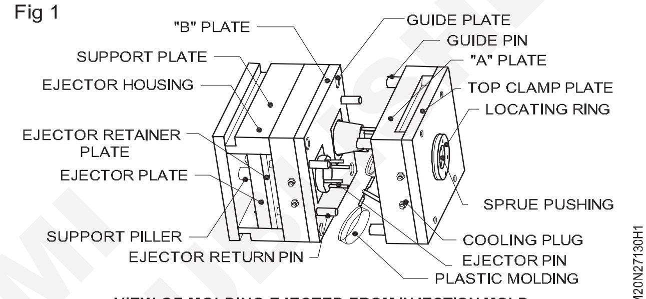
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the pictorial view of the moulding ejected from injection mould (Fig 1)**
- Select suitable dimensions

Fig 1

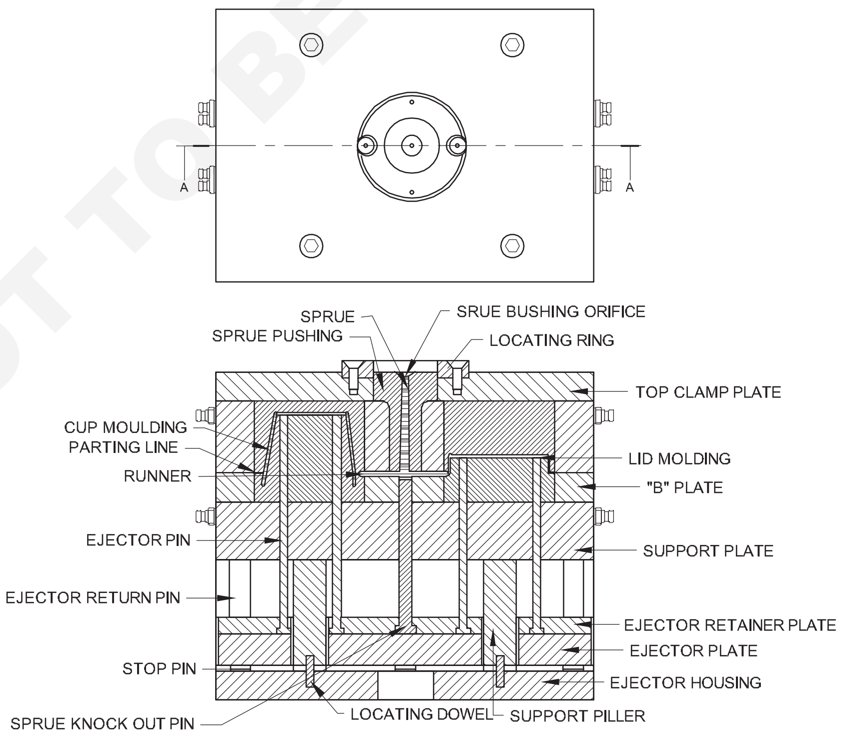


VIEW OF MOLDING EJECTED FROM INJECTION MOLD

— — — — —

TASK 2: **Draw the sectional view and plan of the two plate mould (Fig 1)**
- Select and draw with suitable dimensions.
- Replace the names in the views by numerical number
- Prepare a part list.

Fig 1



TOP AND CROSS-SECTION VIEWS OF A TWO-PLATE MOLD

— — — — —

Construct a detailed drawing of a simple carburettor

Objectives: At the end of this exercise you shall be able to
- **draw the given sectional front elevation view of the carburettor by using CAD**
- **draw the side looking front right of the front elevation**
- **draw the plan view by using CAD.**

Requirements

Tools/Equipment/Machines

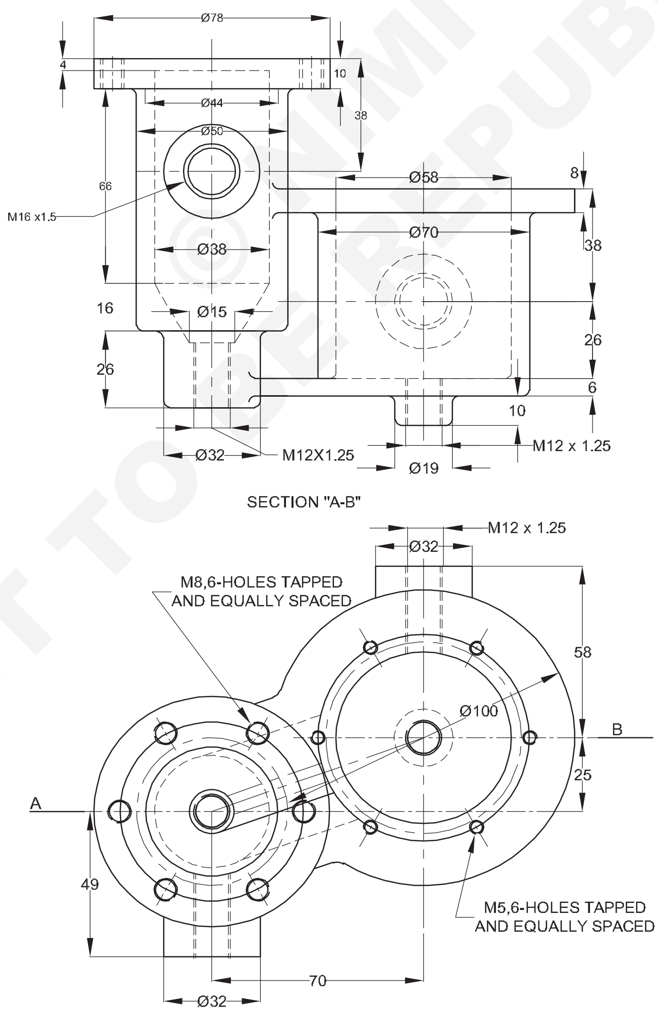
- Auto CAD 2018 or Higher version.

PROCEDURE

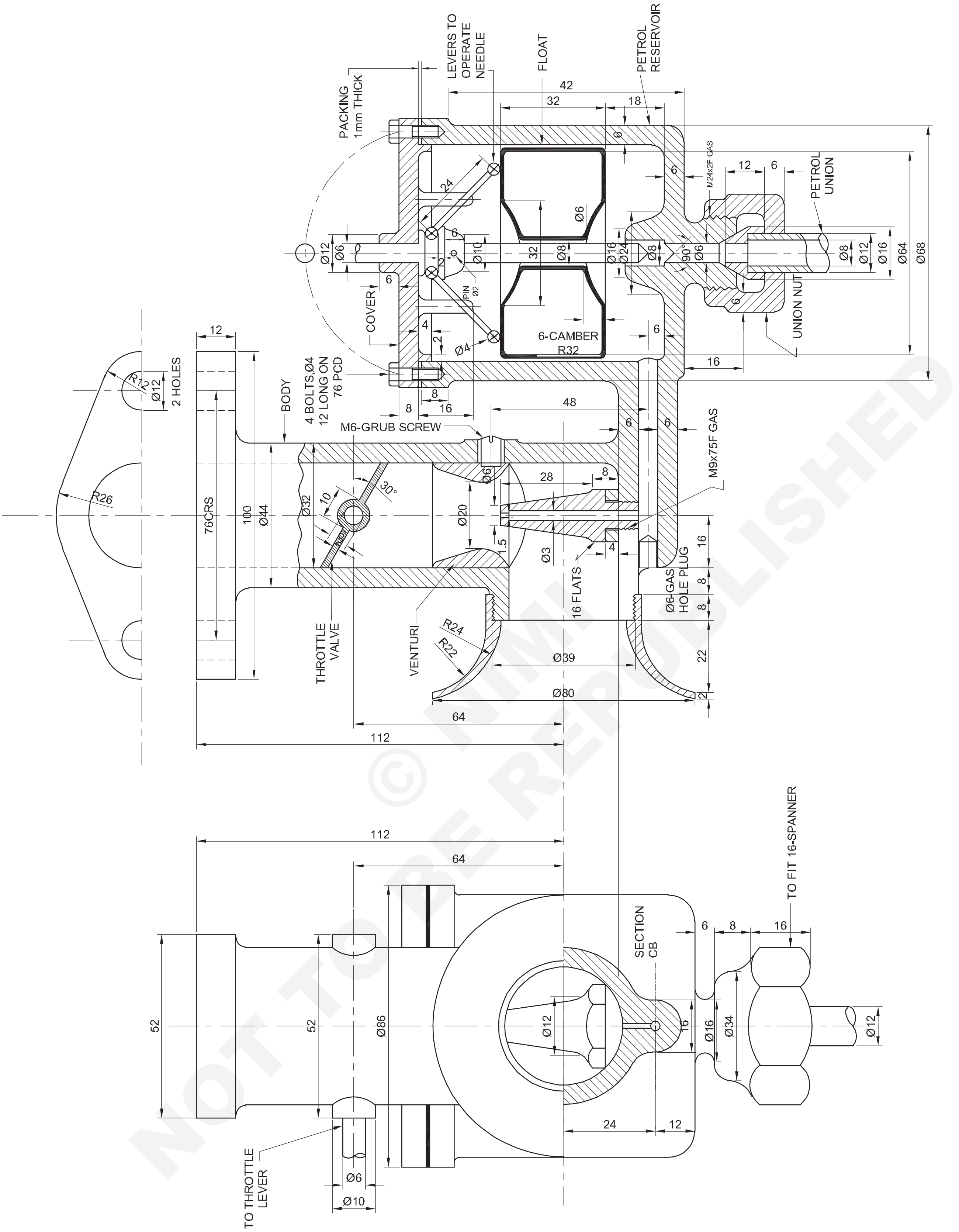
TASK 1: **Draw the full sectional elevation of the carburettor by using CAD**

- Draw the side elevation looking from right of the front elevation by using CAD.
- Draw the full plan view by using CAD.

Note: Provide all the necessary dimensions and prepare title block.



SCALE 1:1	SIMPLE CARBURETER	EX NO : 2.7.131
		TIME : 28hrs
PROJECTION		CODE : DM20N27131E1



SCALE 1:1	**SIMPLE JET CARBURETTOR**	EX NO : 2.7.131
		TIME : 28hrs
PROJECTION		CODE : DM20N27131E2

Construct detailed drawing of a Pressure vessel

Objectives: At the end of this exercise you shall be able to
- **draw the detailed views of pressure vessel by using CAD**
- **draw the assembled views of pressure vessel by using CAD**
- **prepare part list.**

Requirements

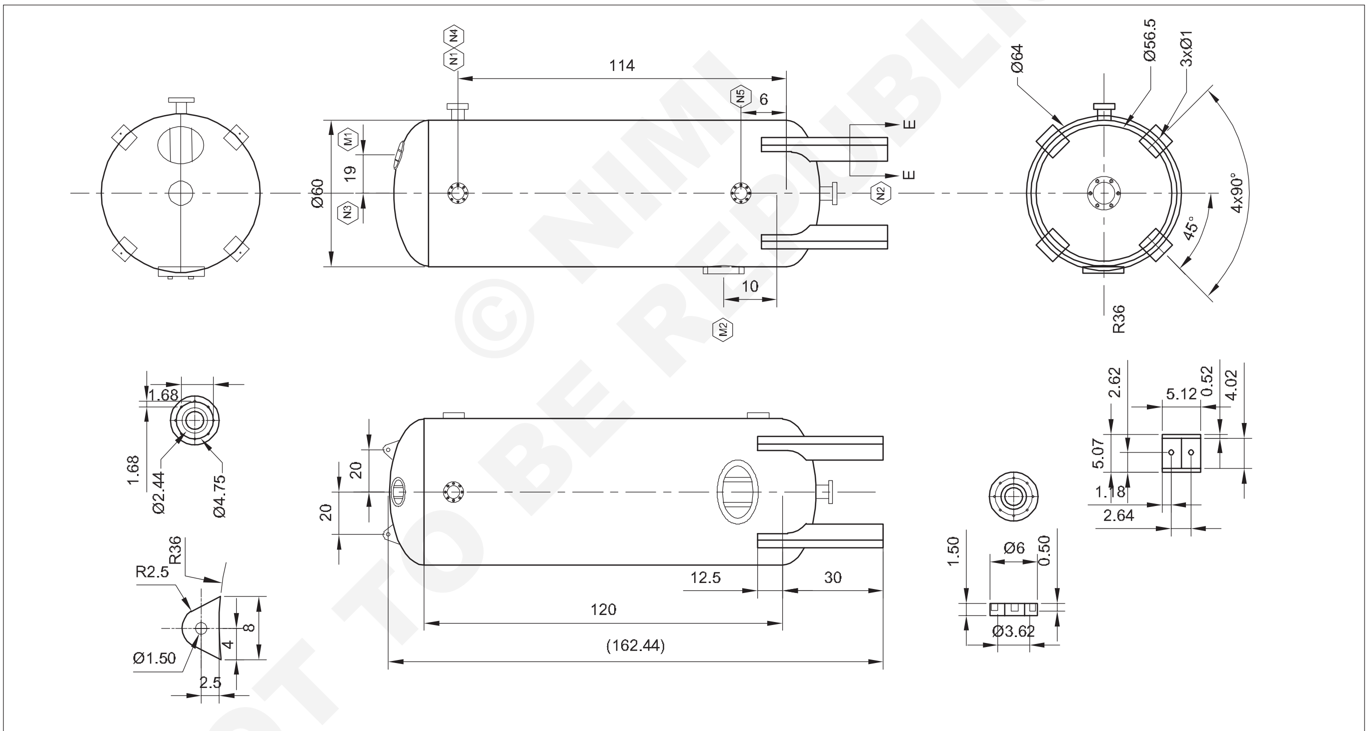
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw the detailed views of the compress simple pressure vessel by using CAD**

- Provide all dimensions and each detail.
- Provide part number for each detail as in assembled views.



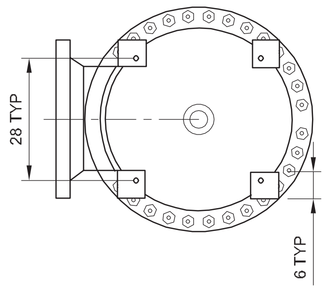
SCALE 1:1	**PRESSURE VESSEL**	EX NO : 2.7.132
		TIME : 28hrs
PROJECTION		CODE : DM20N27132E1

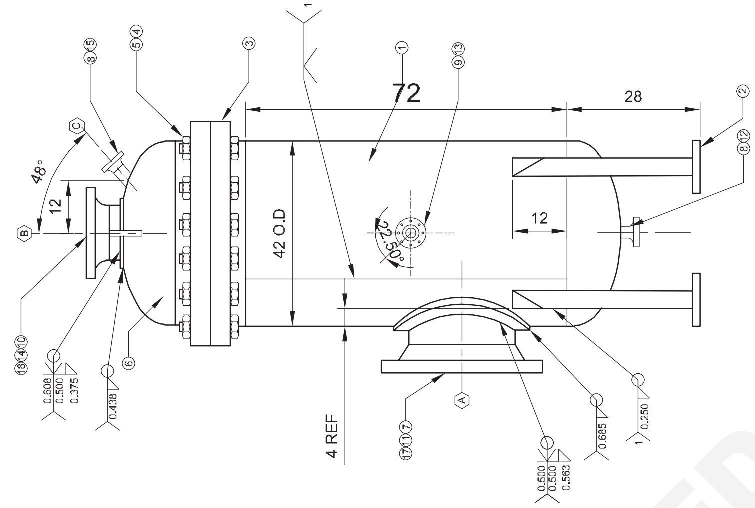
— — — — —

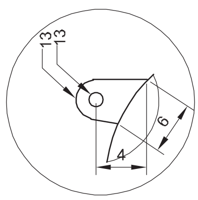
TASK 2: **Draw the following assembled views by using CAD**

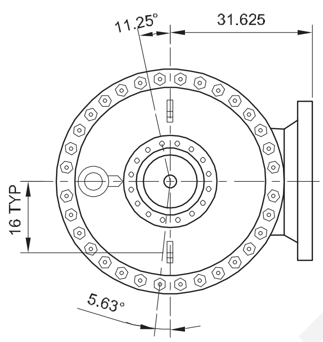
- Full sectional front elevation
- Side view looking from right of the sectional elevation
- Full plan
- Prepare bill of materials

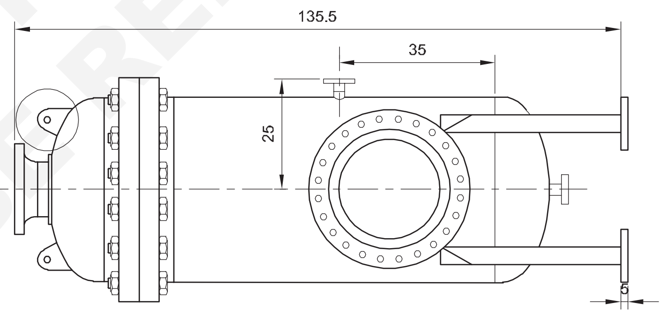
Note: for assembly and side elevation refer the Assembly view.

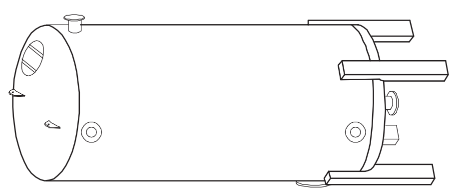












SCALE 1:1	PRESSURE VESSEL & ASSEMBLY	EX NO : 2.7.132
		TIME : 28hrs
PROJECTION		CODE : DM20N27132E2

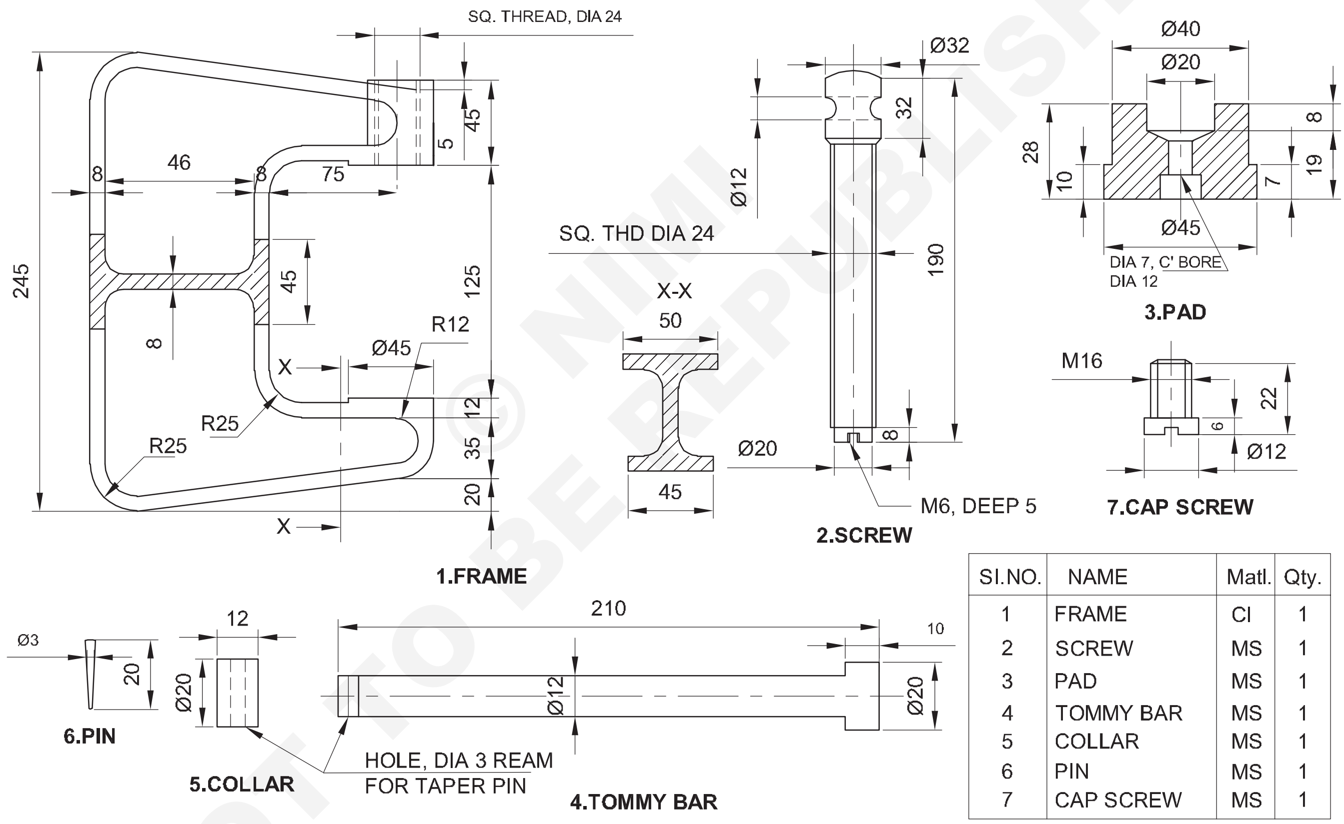
Prepare detailed drawing of a ‘c’ clamp

Objectives: At the end of this exercise you shall be able to
- **draw detailed drawing of a ‘c’ clamp.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw detailed drawing of a ‘c’ clamp**



SI.NO.	NAME	Matl.	Qty.
1	FRAME	CI	1
2	SCREW	MS	1
3	PAD	MS	1
4	TOMMY BAR	MS	1
5	COLLAR	MS	1
6	PIN	MS	1
7	CAP SCREW	MS	1

SCALE 1:1	'C'-CLAMP (DETAILS)	EX NO : 2.7.133
		TIME : 28hrs
PROJECTION		CODE : DM20N27133E1

Draw a simple machine shop layout of small industry

Objectives: At the end of this exercise you shall be able to
- **prepare a shop layout of a production.**
- **design and placement of machines.**

Requirements

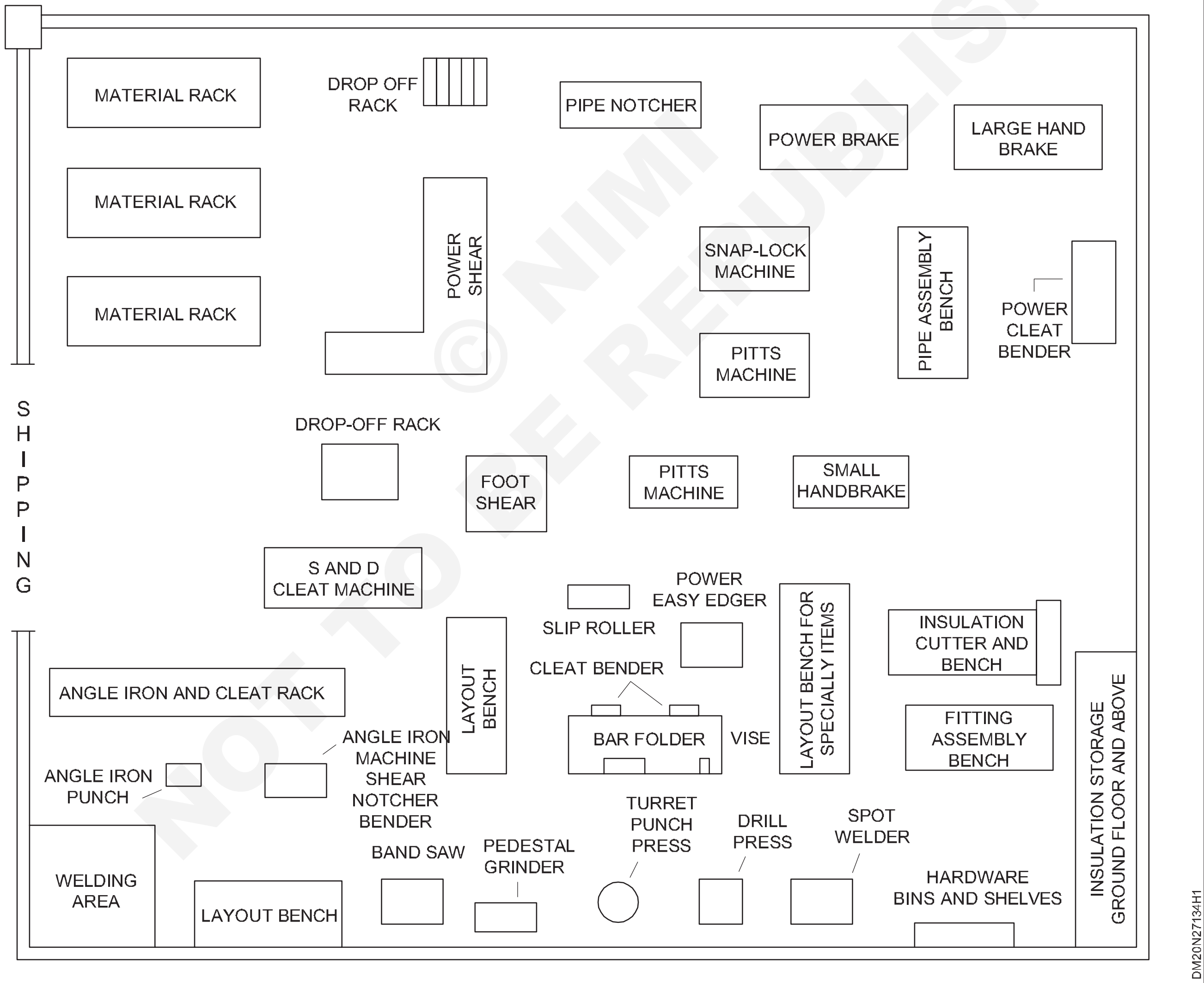
Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

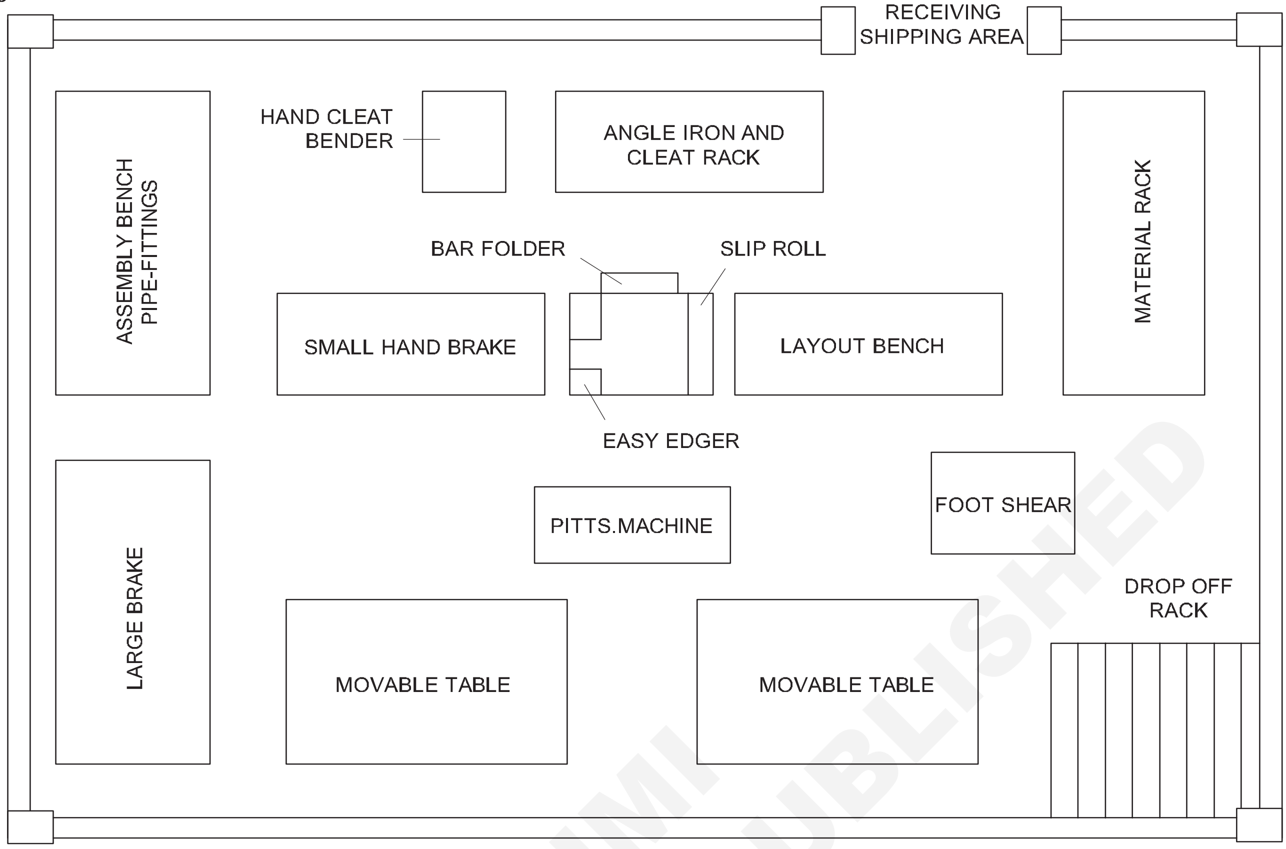
TASK 1: **Draw the shop layout with suitable dimension by using CAD (Fig 1)**

Fig 1



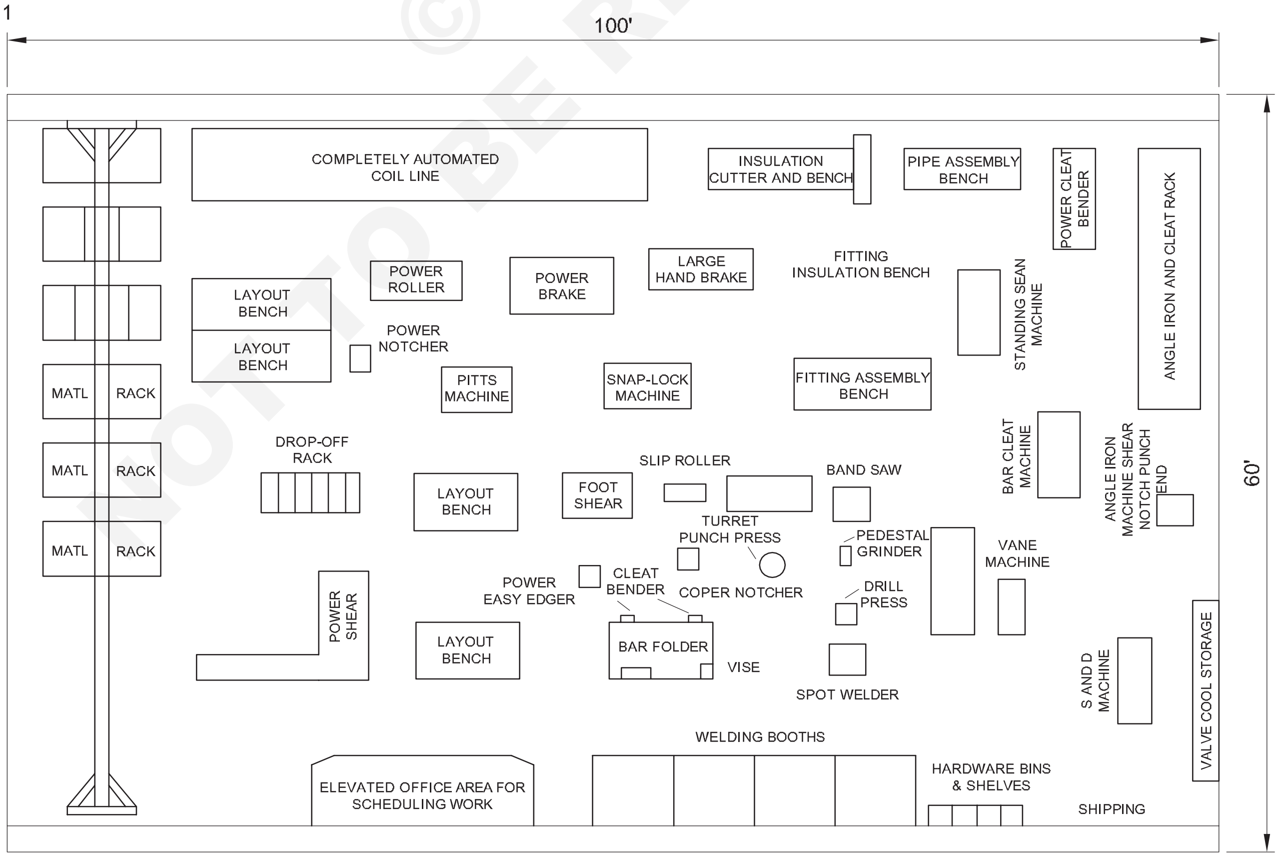
TASK 2: **Draw the shop layout with suitable dimension by using CAD. (Fig 1)**

Fig 1



TASK 3: **Draw the shop layout with suitable dimension using CAD (Fig 1)**

Fig 1



Draw 3D solid figure by sketching features and applied feature

Objectives: At the end of this exercise you shall be able to
- **draw 3D solid figure by sketching feature.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Draw 3D solid figure by sketching feature (Figs 1&2)**

Fig 1

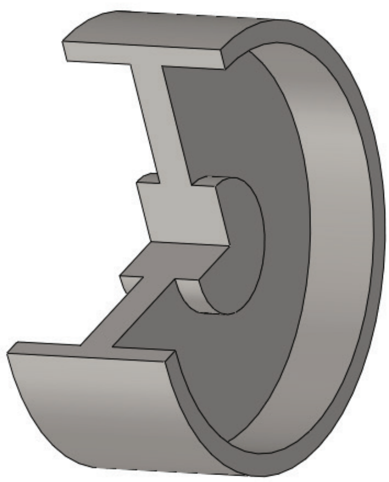
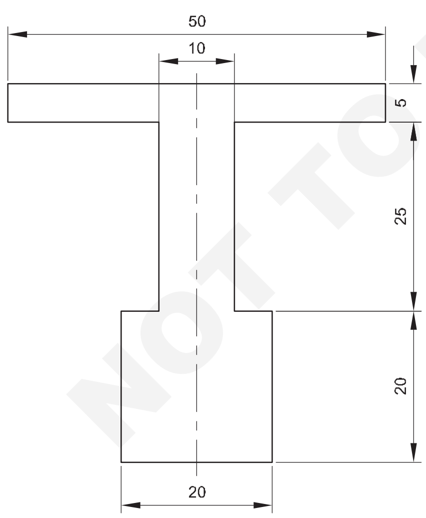


Fig 2



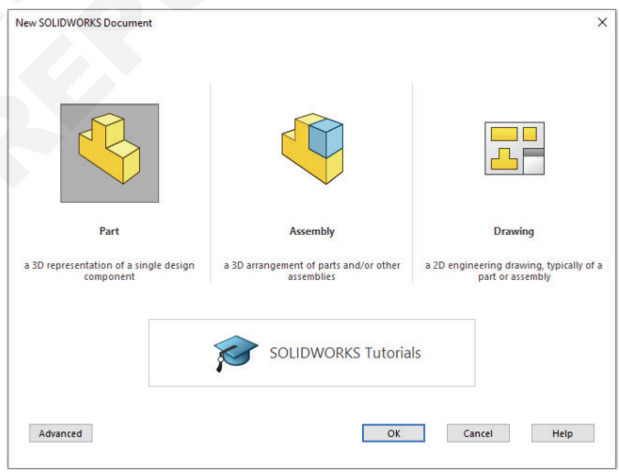
SKETCH OF THE REVOLVED SOLID MODEL

Opening a new part Document

1 Start SOLIDWORKS By double-clicking on the shortcut icon of SOLIDWORKS available on the desktop of your computer; the SOLIDWORKS 2017 window along with the SOLIDWORKS Resources task pane on its right is displayed. The Getting Started, SOLIDWORKS Tools, Community, 3D Interconnect, Online Resources, Subscription Services and Tip of the Day options are displayed in this task pane.

2 Choose the New Document option from the Getting Started roll out of the SOLIDWORKS Resources task pane; the New SOLIDWORKS Document dialogue box is displayed, as shown in (Fig 3).

Fig 3



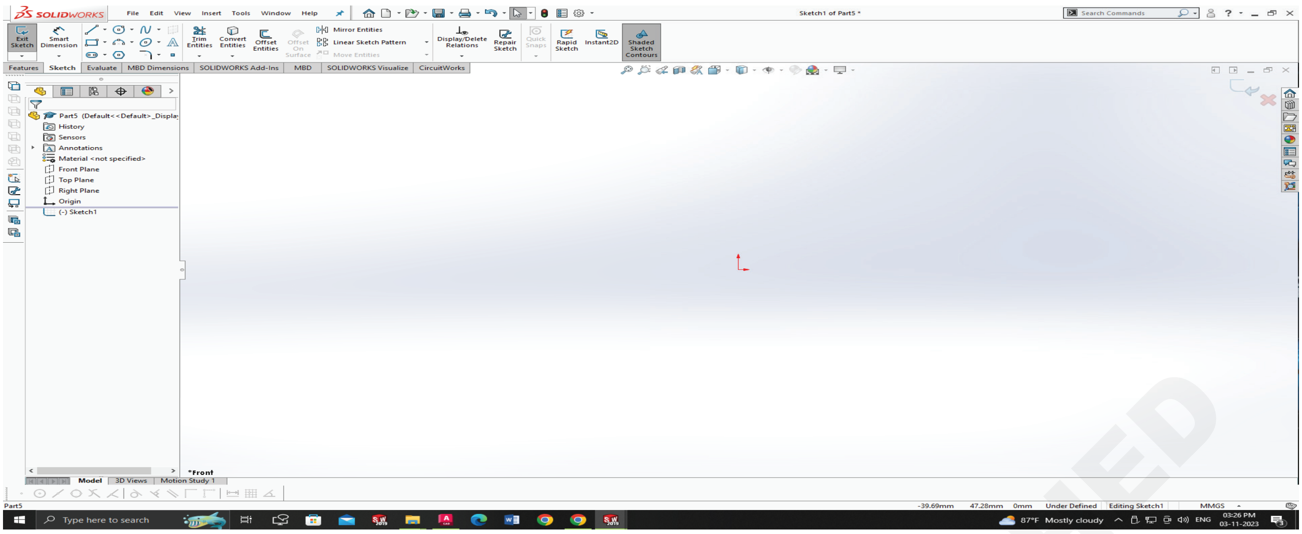
3 In the New SOLIDWORKS Document dialogue box, the Part button is chosen by default. Therefore, choose the OK button; a new SOLIDWORKS part document starts.

4 Choose the Sketch tab from the Command Manager and then the Sketch button in the Sketch Command Manager; the Edit Sketch Property Manager is displayed and you are promoted to select a plane on which you want to draw the sketch.

5 Select the Front Plane from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that red coloured arrows are displayed at the centre of the screen, indicating that you are in the sketching environment. Also, the confirmation corner with

the Exist Sketch and Cancel options is displayed on the upper right corner in the graphics area. The screen display in the sketching environment of SOLIDWORKS 2017 is shown in (Fig 4).

Fig 4



Screen display in the sketching environment.

1 Choose the Options button from the Menu Bar: the System Options - General dialogue box is displayed.

2 Choose the Document Properties tab; the name of the dialogue box changes to the Document Properties - Drafting Standard.

3 Select the Units option from the area on the left to display the options related to the linear and angular units.

4 Select the MMGS (millimetre, gram, second) radio button in the Unit System area. Also, select the Degree option in the Units column as unit of angle, if it is not selected by default. You can also change the unit system using the Unit System button, which is located at the right side of the status bar.

5 Select the Grid/ Snap option from the area on the left to display grid options set the value in the Major grid spacing spinner to 50 and the value in the Minor lines per major spinner to 10.

6 Select the Display grid check box if it is cleared. Next choose the Go To System Snaps button the system options related to relations and snaps are displayed.

7 Select the GRID check box from the Sketch Snaps area and clear the Snap only when grid is displayed check box. Choose OK to exit the dialogue box.

Drawing the Sketch

It is evident from figure that the sketch will be drawn using the Line tool. Therefore you need to start drawing the sketch from the lower left corner of the sketch.

1 Choose the Line tool from the Sketch Command Manager the arrow cursor is replaced by the line cursor.

2 Move the Line cursor to the origin.

3 Left click at this point and move the cursor Horizontally toward the right. You will notice that the symbol of the Horizontal relation is displayed below the line cursor and the length and angle of the line are displayed above the line cursor.

4 Left click again when the length of the line above the line cursor shows 20.

The first horizontal line is drawn. As your drawing continuous lines the end point of the line drawn is automatically selected as the the start point of the next line.

5 Move the line cursor vertically upward. The symbol of the Vertical relation is displayed on the right of the line cursor and the length of the line is displayed above the line cursor. Click when the length of the line on the line curs or is displayed as 20.

6 Move the cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 5.

7 Move the Line cursor vertically upward and press the left mouse button when the length of the line on the line cursor is displayed as 25.

8 Move the line cursor Horizontally towards the right and click when the length of the line on the line cursor is displayed as 20.

9 Move the line cursor vertically upward and click when the length of the line on the line cursor is displayed as 5.

10 press F on the keyboard to fit the sketch on the screen.

11 Move the line cursor Horizontally toward the left and click when the length of the line on the line cursor is displayed as 50.

12 Move the line cursor ventrically downward and click when the length of the line on the line cursor is displayed as 5.

13 Move the line cursor Horizontally toward the right and press the left mouse button when the length of the line on the line cursor is displayed as 20.

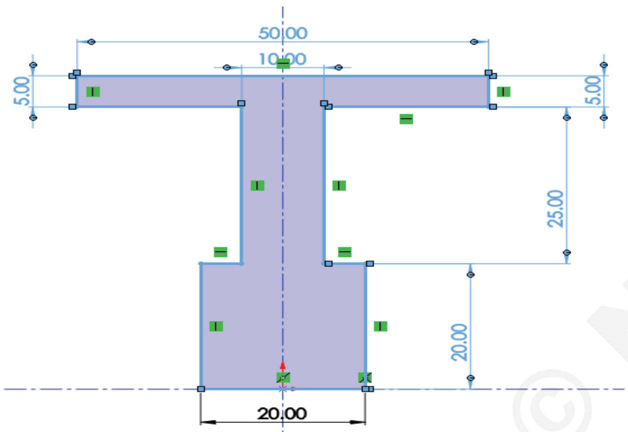
14 Move the Line cursor vertically downward and click when the length of the line on the line cursor is displayed as 25.

15 Move the line cursor Horizontally toward the left and click when the length of the line on the line cursor is displayed as 5.

16 Move the line cursor vertically downward to the start point of the first line. Click when an orange circle is displayed. The final sketch for Tutorial 1 is created as shown in figure. In this figure grid display and Shaded Sketch Contours settings or turned off for clarity.

17 Right click and then choose the Select option from the shortcut menu to exit the Line tool. (Fig 5)

Fig 5



Final sketch for Tutorial 1

1 Choose the Save button from the Menu Bar to invoke the Save As dialogue box. Create the SOLIDWORKS folder inside the Documents folder and then create the c02 folder inside the SOLIDWORKS folder.

2 Enter c02_tut01 as the name of the document in the File name edit box and choose the Save button. The document is saved at the location \Documents \ SOLID WORKS\c02

3 Close the document by choosing File > Close from the SOLIDWORKS menus.

Sketch an angle plate and a block modify constraints:

Create an Angle plate:

1 Open Solid Works: Launch Solid Works and create a new part document.

2 Sketching the angle plate.

- Click on the "Sketch" tab in the Command manager.
- Choose a plane (e.g. Front Plane) to sketch on.
- Use the Line, Rectangle and Dimensions tools to create the outline of the angle plate

3 Adding Dimensions:

- Add dimensions to define the size of the angle plate.
- Use Smart Dimension to specify the lengths and the angles of the lines.

4 Applying constraints

- Use geometric constraints like coincident and parallel to align lines and corners as needed.
- Apply dimensions to fully define the shape.

5 Extruding the sketch:

- Exit the sketch mode and go to the "Features" tab in the Command Manager.
- Click on "Extruded Boss/Base" and select the sketch profile.
- Specify the extrusion distance to create the 3D angle plate.

Creating a Block:

1 Sketching the block

- Follow similar step us above to create a new sketch on a plane (e.g. Top plane)
- Use the rectangle tool to draw the outline of the block.

2 Adding dimensions:

- Add dimensions to define the length width and height of the block.

3 Applying constraints:

- Use constraints like horizontal and vertical to align edges of the rectangle.
- Apply dimensions to fully define the block is size.

4 Extruding the sketch :

- Exit the sketch mode and go to the "Features" tab.
- Click on "Extruded Boss/Base" and select the sketch profile.
- Specify the extrusion death to create a 3D block

Modifying constraints:

After you have created the angle plate and block, you can modify constraints to change their positions.

1 Editing the Sketch

- Double click on the sketch in the Future Manager Design Tree to edit it.

2 Modifying Dimensions and Constraints.

- Change the values of dimensions to adjust the size and angles of the sketch elements.
- You can also add to remove geometric constraints to achieve the desired changes.

3 Updating the model:

- After editing the sketch, exiting the sketch mode.

Sketch an angle plate and a block

Objectives: At the end of this exercise you shall be able to
- **sketch an angle block.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Sketch an angle block (Figs 1 - 9)**

Fig 1

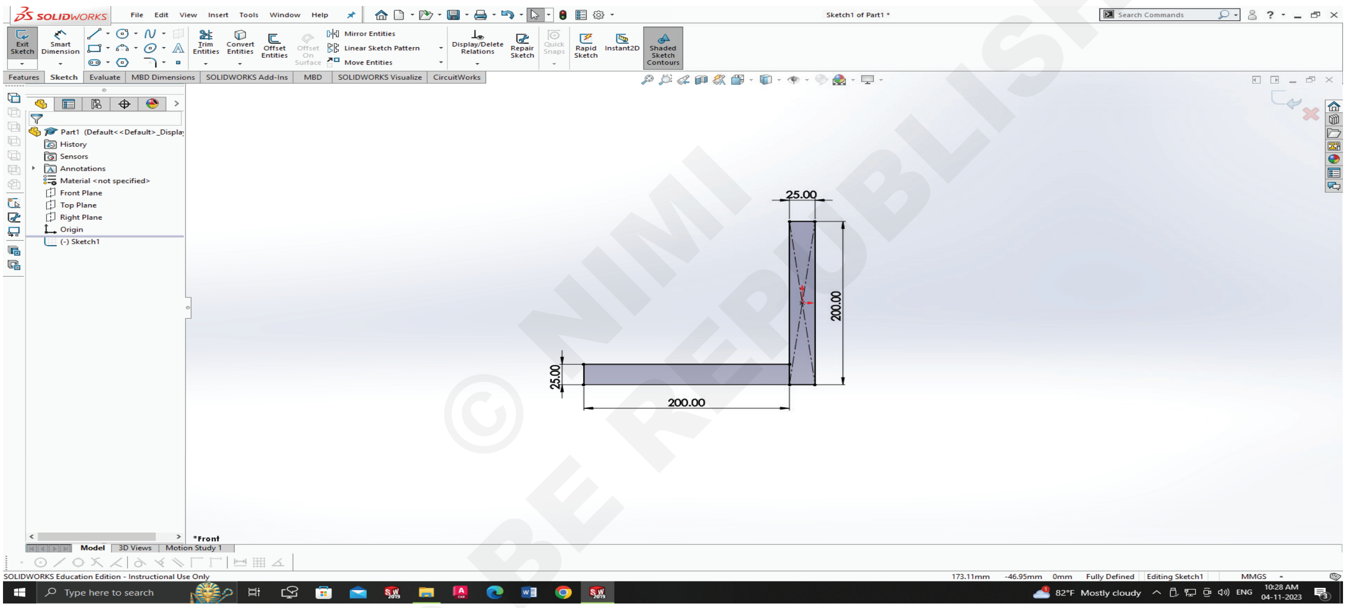


Fig 2

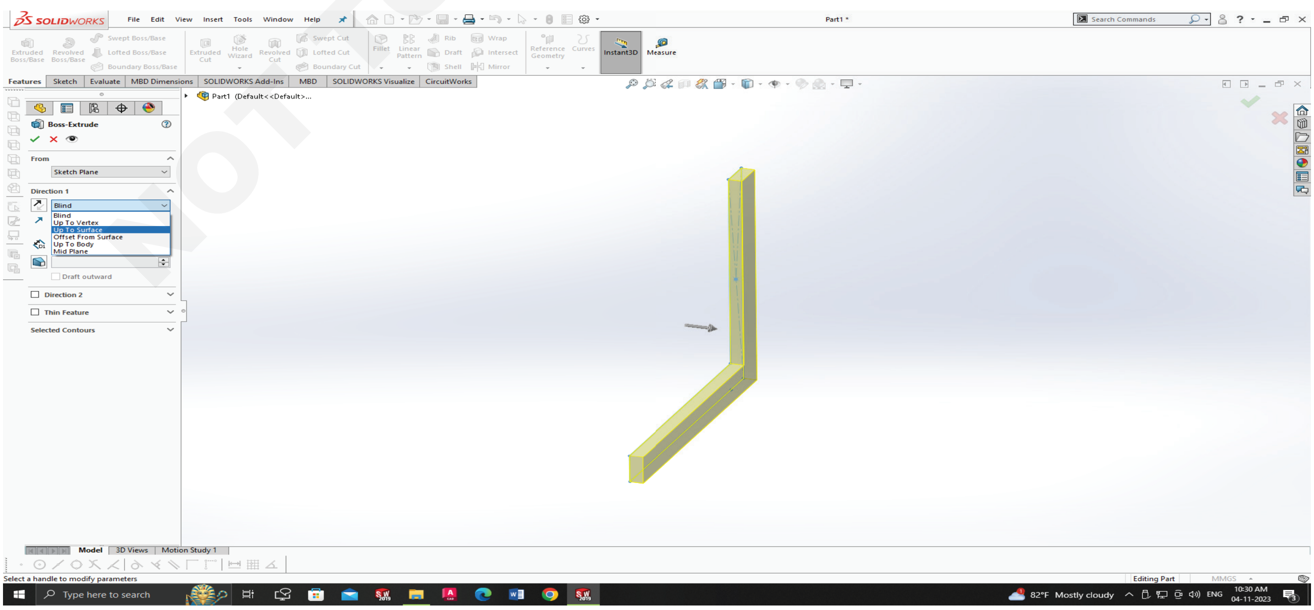


Fig 3

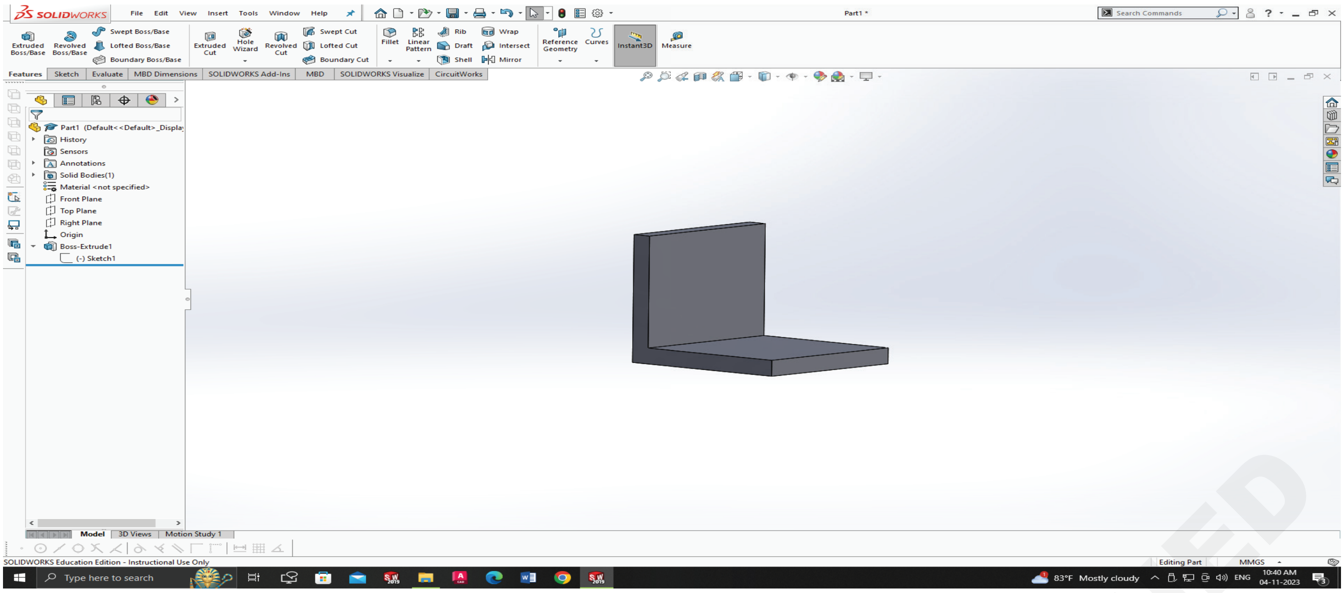


Fig 4

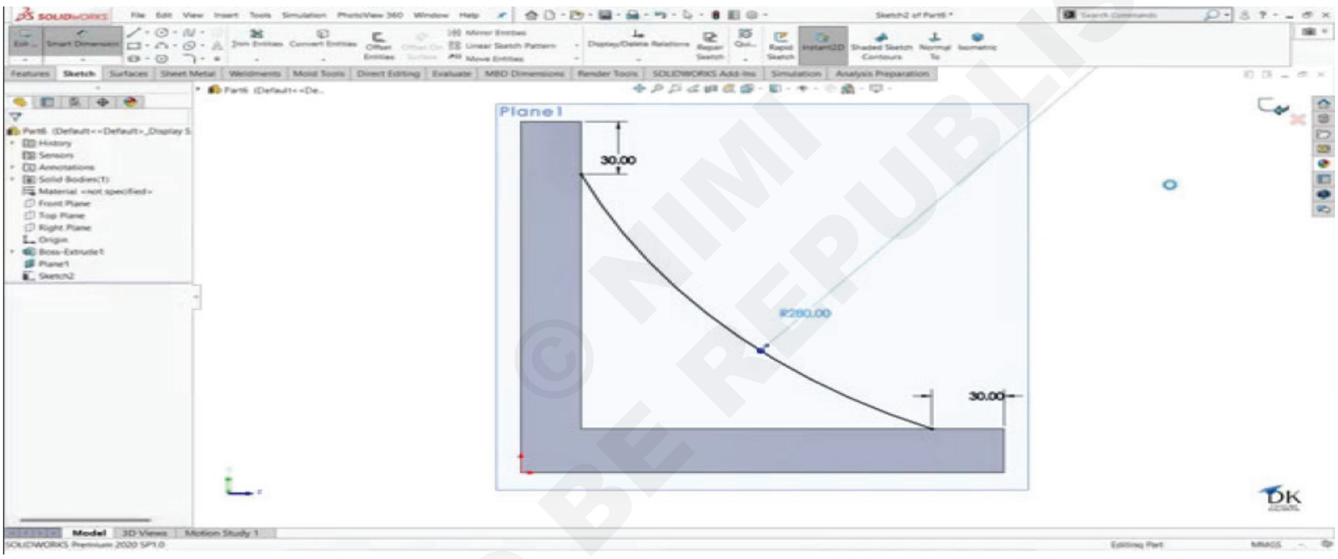


Fig 5

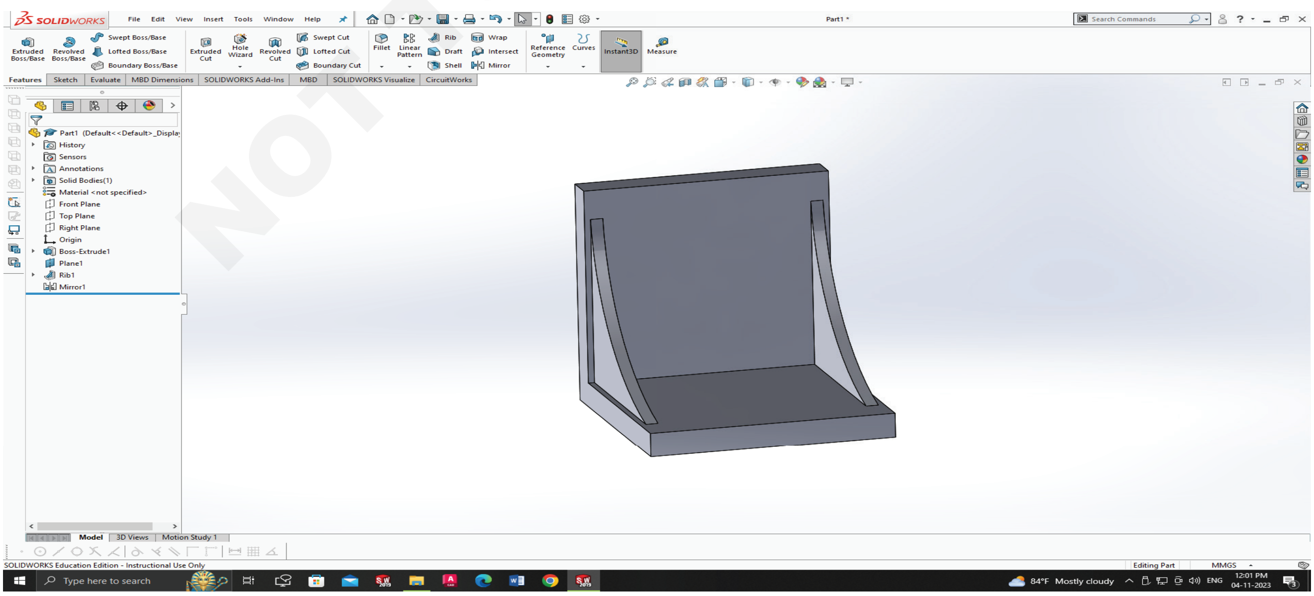


Fig 6

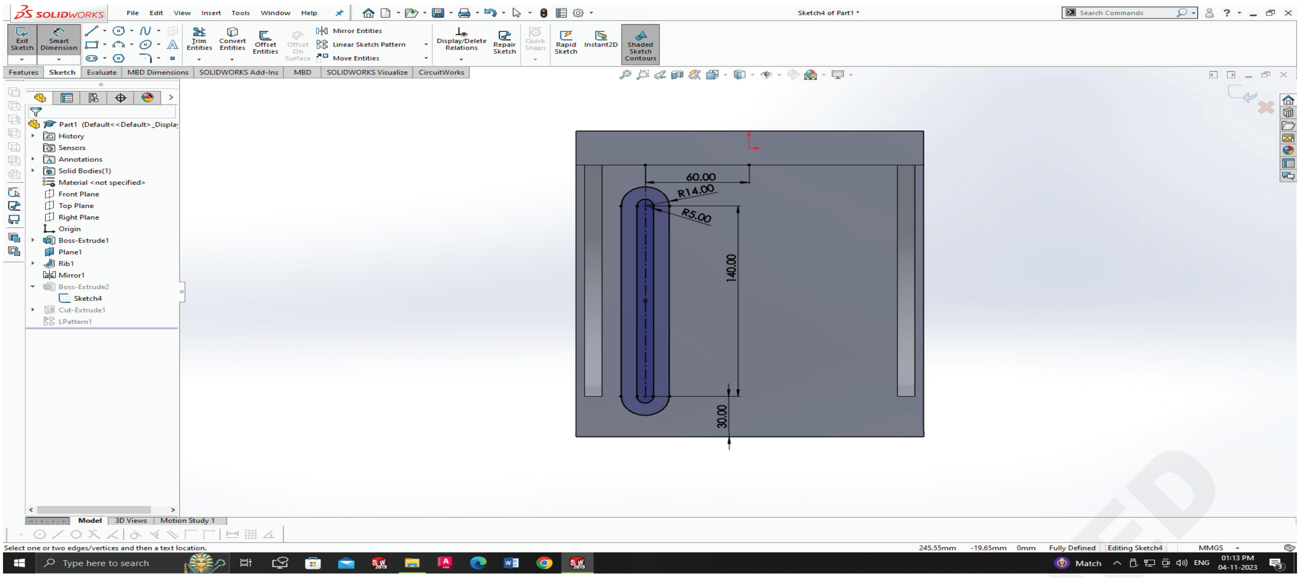


Fig 7

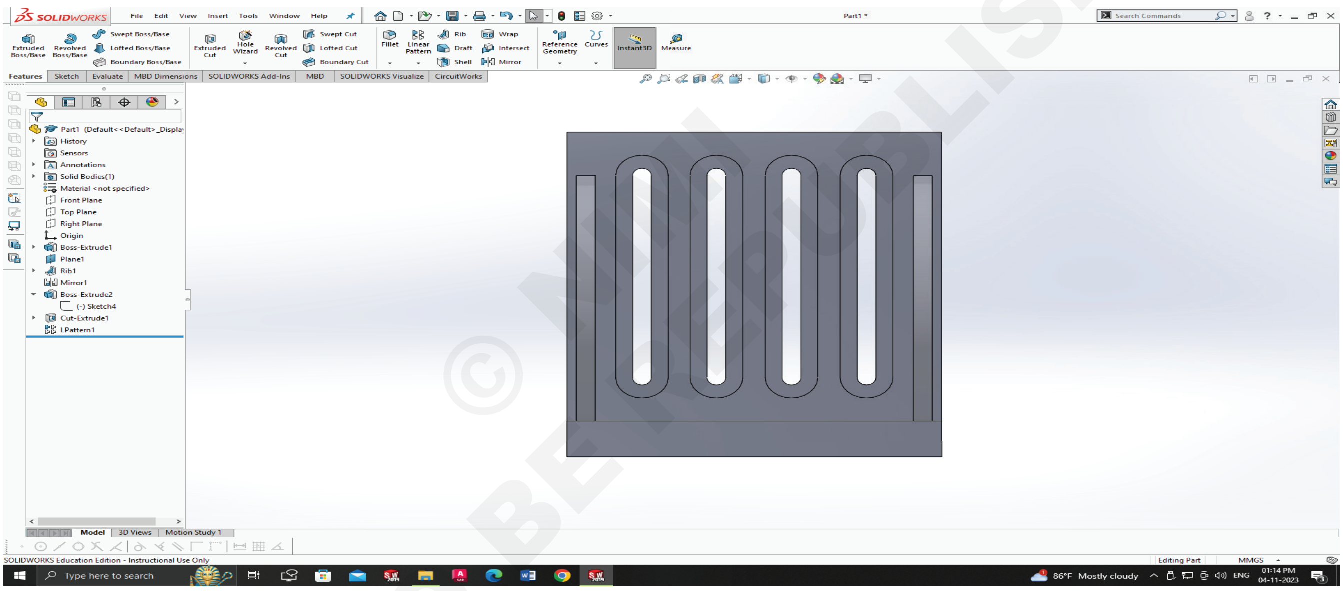


Fig 8

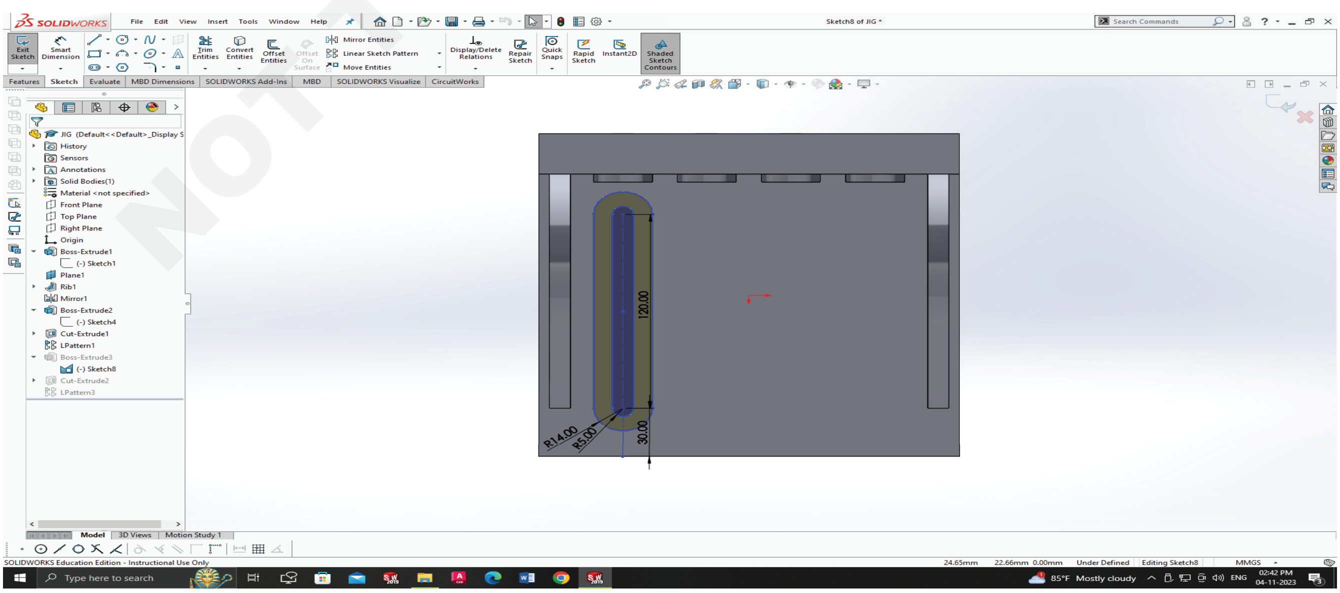
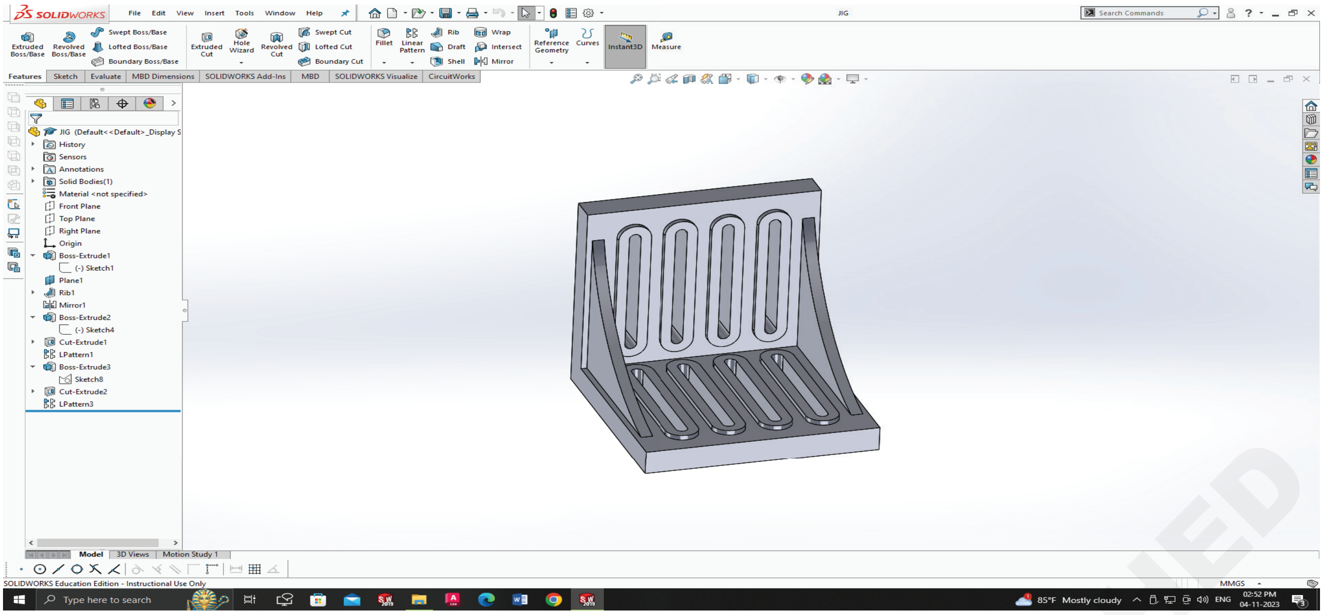


Fig 9


SOLIDWORKS
File
Edit
View
Insert
Tools
Window
Help
Features
Sketch
Evaluate
MBD Dimensions
SOLIDWORKS Add-Ins
MBD
SOLIDWORKS Visualize
CircuitWorks
History
Sensors
Annotations
Solid Bodies(1)
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1
Plane1
Rib1
Mirror1
Boss-Extrude2
Cut-Extrude1
LPattern1
Boss-Extrude3
Cut-Extrude2
LPattern3
Model
3D Views
Motion Study 1
SOLIDWORKS Education Edition - Instructional Use Only

Create a Sketch of a new part

Objectives: At the end of this exercise you shall be able to
- **create a Sketch of a part.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Create a Sketch of a part**

The following steps are required to complete this tutorial:

a Start a new part document.

b Invoke the sketching environment.

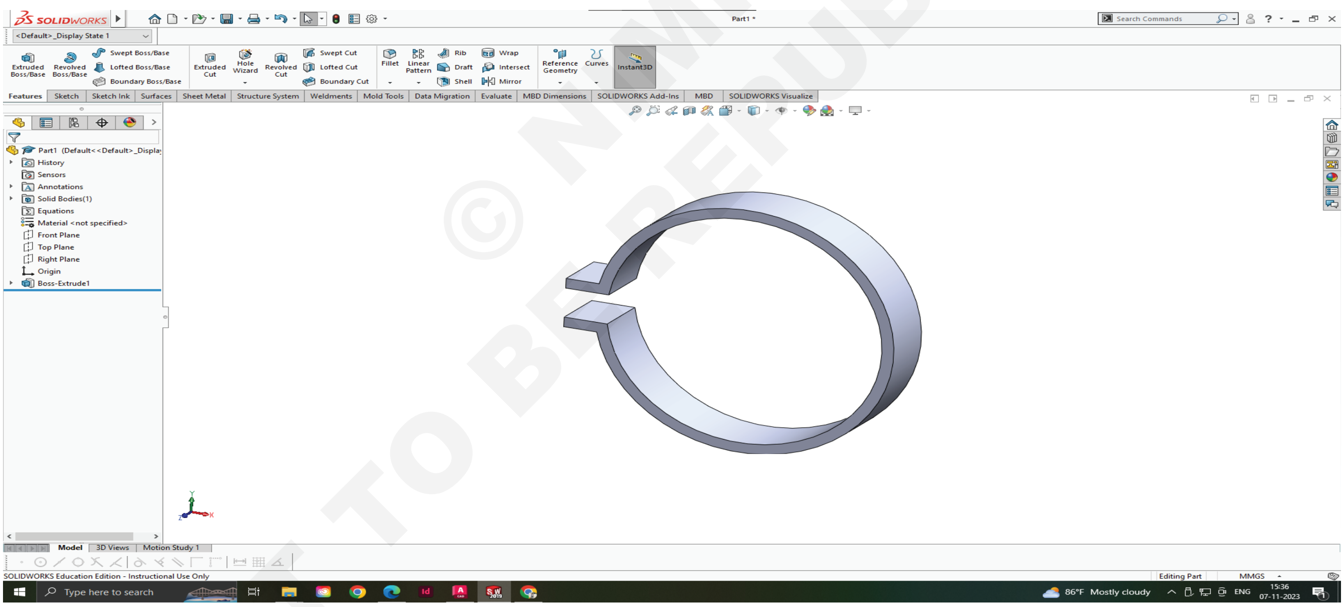
c Create a centre line.

d Draw and edit the sketch using the Mirror Entities and Trim Entities tools.

e Offset the entire sketch.

f Complete the final editing of the sketch using the Extend Entities and Trim Entities tools. (Fig 1)

Fig 1



Starting a New Document

1 Choose the New button from the Menu Bar to invoke the New SOLIDWORKS Document dialog box.

2 The Part button is chosen by default in the New SOLIDWORKS Document dialog box. Choose the OK button.

3 Choose the Sketch button from the Sketch Command Manager and select Front Plane to invoke the sketching environment.

4 Invoke the Document Properties – Units dialog box and change the units to MMGS (millimetre, gram, second) if it is not selected by default.

5 Invoke the Document Properties – Grid/Snap dialog box. Now, set 100 in the Major grid spacing spinner and 10 in the Minor-lines per major spinner. Choose the OK button.

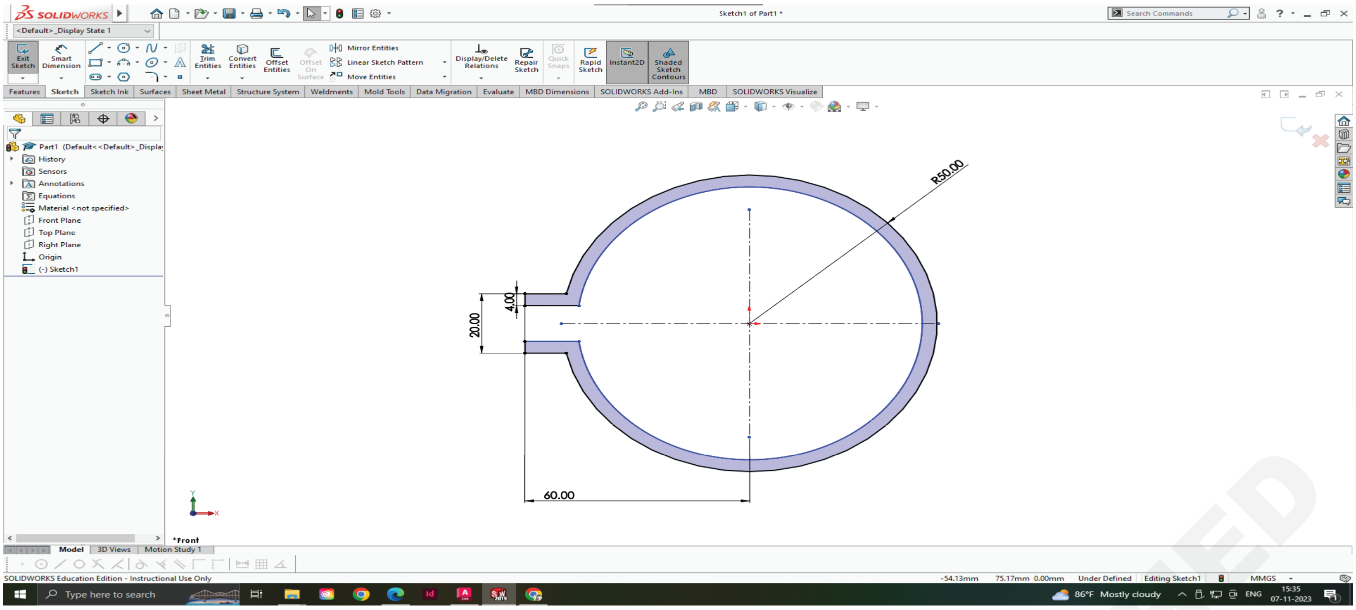
Drawing the Centre line

Before drawing the sketch, you need to draw a centre line that will act as reference for other sketched entities. This centreline will also be used for mirroring.

1 Choose the Centre line button from the Line flyout in the Sketch Command Manager.

2 Move the cursor to a location whose coordinates are – 70 mm, 0 mm, and 0 mm.

Fig 2



3 Specify this point as the start point and move the cursor horizontally toward the right.

4 Specify the end point of the centreline by clicking at a location when the value of the length of the centreline above the line cursor is 140.

5 Double-click anywhere in the drawing area to end the line creation and exit the Centre line tool.

Drawing the Outer Loop of the Sketch

Next, you need to draw the outer loop of the sketch using the sketch tools. As it is evident from (Fig 2), the sketch needs to be drawn using the Circle and Line tools.

1 Press and hold the right-mouse button and drag the cursor to the right, the mouse gesture is displayed with sketching tools. Move the cursor over the Circle tool, the Circle tool is invoked and the Circle Property Manger is displayed.

2 Make sure that the Circle button is chosen in the Circle Type roll out in the Circle Property Manger. Move the circle cursor to the origin and click when an orange circle is displayed to specify the centre point of the circle.

3 Move the cursor horizontally toward the right and draw a circle of 100 mm diameter.

4 Choose the Zoom to fit button from the View (Heads-Up) tool bar to increase the display of the sketch.

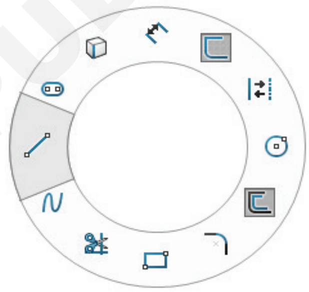
5 Choose the Line button from the Sketch Command Manger and move the cursor to a location whose coordinates are 60 mm, 10 mm, and 0 mm. Specify the start point of the line at this location.

6 Move the cursor horizontally toward the right and click to specify the end point of the line when the line cursor snaps the circle. Exit the Line tool.

7 Press the hold the SHIFT key and then using the left mouse button, select the centre line, and the horizontal line created in the last step.

8 Choose the Mirror Entities button from the Sketch Command Manager, the mirror image of the horizontal line is created on the other side of the centre line.

Fig 3



9 Choose the Line button using the mouse gesture, refer to (Fig 3) and then move the cursor to the left end point of the upper horizontal line. Click to specify the start point of the line when the orange circle is displayed.

10 Move the cursor vertically downward. Click in the drawing area when the cursor snaps to the left end point of the lower horizontal line. Exit the Line tool.

11 Choose the Trim Entities button from the Sketch Command Manger, the Trim Property Manager is displayed.

12 Choose the Power trim button from the Property Manger. Press and hold the left mouse button and drag the cursor over the portion to be removed. Choose the Close Dialog button from the Property Manager. The sketch after removing the unwanted portion is shown in Figure.

Offsetting the Entities

After drawing the outer loop of the sketch, you need to draw to the inner loop. The first step for drawing the inner loop of the sketch is offsetting the entire sketch inward.

1 Choose the Offset Entities button from the Sketch Command Manager, the Offset Entities Property Manager is displayed on the left in the drawing area.

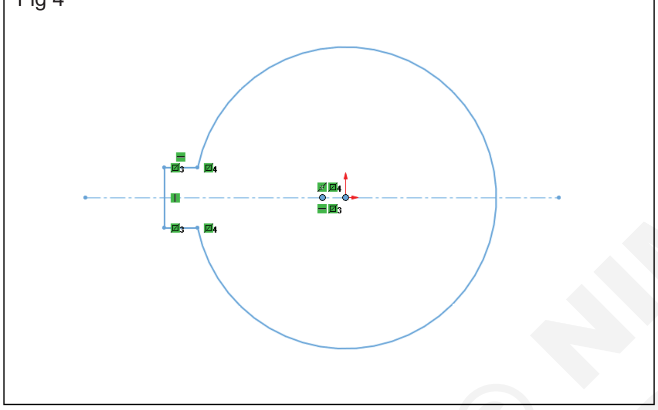
2 Set the value of the Offset Distance spinner to 4. Select the Add dimensions and Select chain check boxes, if they are not selected. Select any one entity of the sketch; the entire sketch is selected.

When you select the sketch, the preview of the offset sketch is displayed in the drawing area. The direction of the offset is outward. However, the direction of the offset should be inward of the sketch. Therefore, you need to flip the direction.

3 Move the cursor inside the sketch and press the left mouse button to offset the sketch inside the original sketch; a dimension with the value 4 is displayed with the sketch.

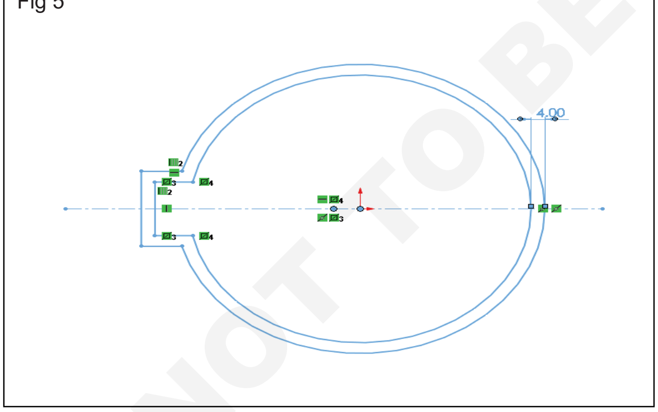
The sketch after offsetting the outer loop is shown in Figure.

Fig 4



(Fig 4) The sketch after removing the unwanted portion

Fig 5



(Fig 5) Sketch after offsetting the outer loop

Extending and Trimming the Entities

1 Choose the Extend Entities button from the Trim Entities flyout in the Sketch Command Manager, the select cursor is replaced by the extend cursor.

2 Move the cursor close to the left end of the lower horizontal line of the inner loop. You can preview the extended line appearing in orange.

You need to move the cursor a little toward the left if the preview of the extended line appears on the right.

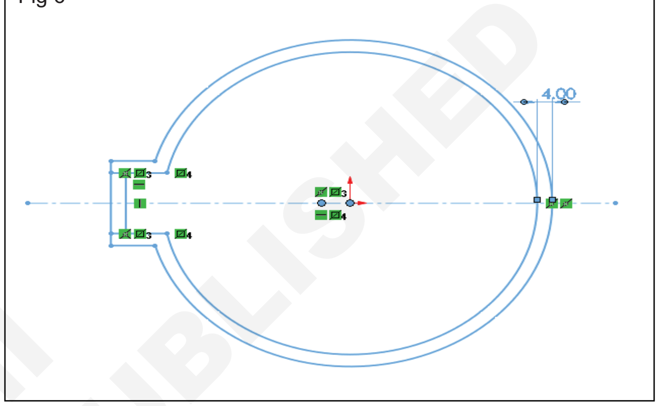
3 Press the left mouse button to extend the line.

4 Similarly, extend the upper horizontal line of the inner sketch. The sketch after extending the lines is shown in (Fig 6).

5 Right-click and choose the Trim Entities option from the shortcut menu; the extend cursor is replaced by t he trim cursor.

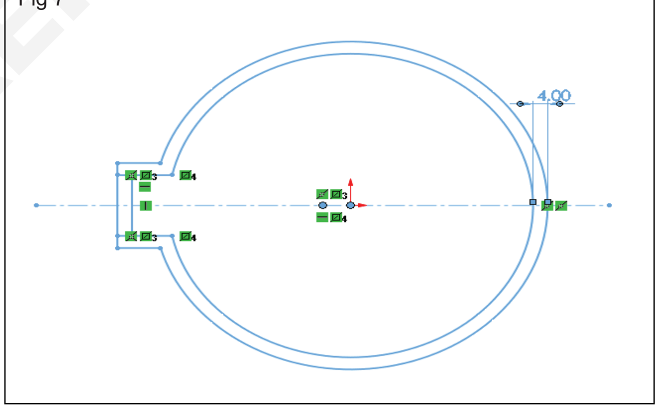
6 Trim the unwanted entities as discussed earlier and exit the t rim command. The final sketch is shown in (Fig 7).

Fig 6



Sketch after extending the lines

Fig 7



(Fig 7) Final sketch for Tutorial 2

Saving the Sketch

1 Choose t he Save button from the Menu Bar to invoke the Save As dialog box.

2 Enter c03_tut02 as the name of the document in the Film name edit box and then choose the Save button.

3 Close the file by choosing File>Close from the SOLIDWORKS menus.

Create 3D Solid and Edit using copy Loop, File time, C hampering, Editing, Create rib, mirror, Hole wizard, create part confirmation

Objectives: At the end of this exercise you shall be able to
- **draw 3D solid and edit using modify features.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Draw 3D solid and edit using modify features (Figs 1 - 3)**

Fig 1

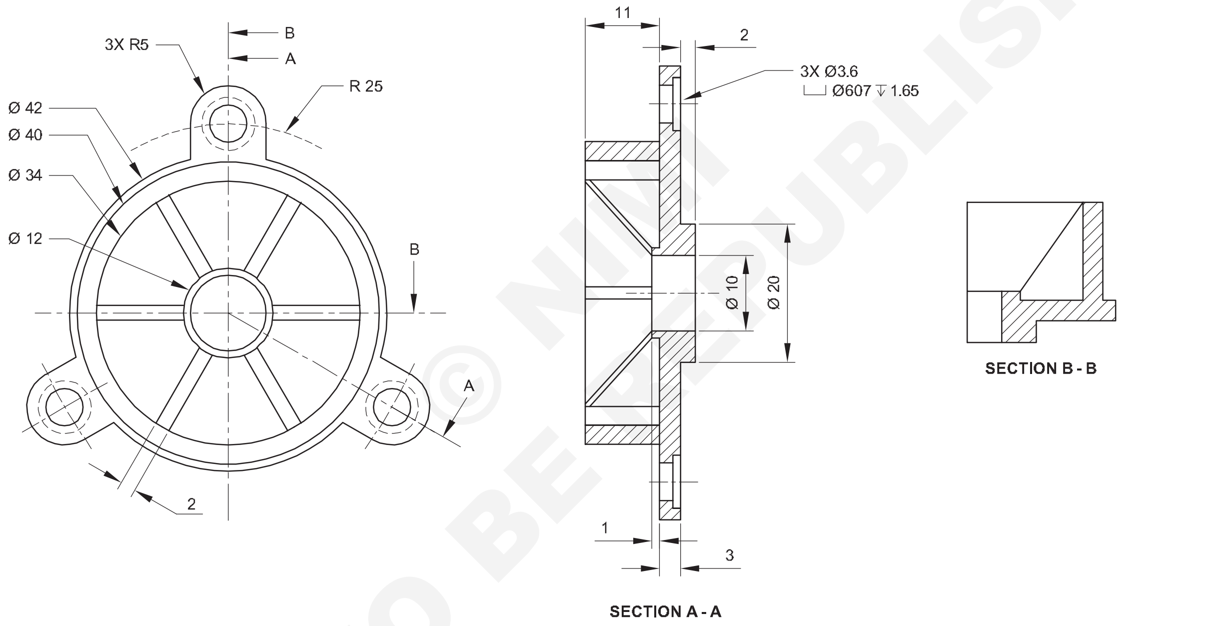


Fig 2

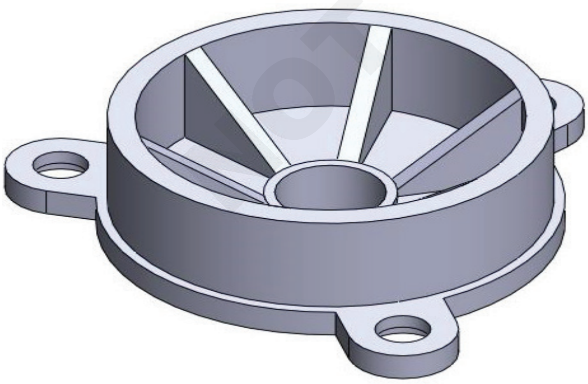
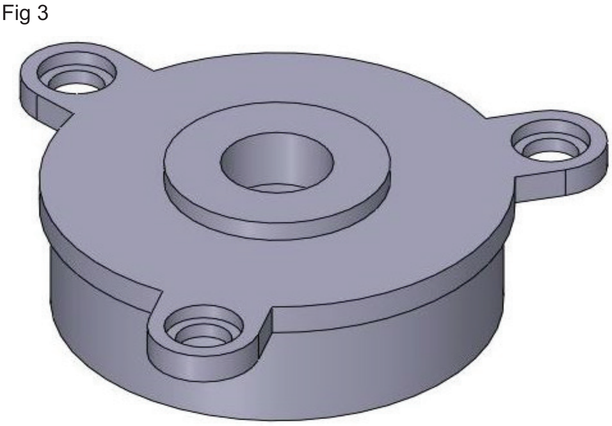


Fig 3



The following steps are required to complete this tutorial:

a Create the base feature of the model by revolving the sketch along the centre line.

b Create the second feature by extruding the sketch from the sketch plane to the selected surface.

c Place a counter bore hole feature on the bottom face of the second feature using the Hole Wizard tool.

d Pattern the second and third features along the temporary axis using the Circular Pattern tool.

e Create the rib feature.

f Pattern the rib feature along a temporary axis using the Circular Pattern tool.

Creating the Base Feature

Start a new SOLIDWORKS part document. First, you need to create the base feature of the model by revolving the sketch along the axis of revolution. The axis of revolution will be a centre line and the sketch for the base feature will be drawn on the Right Plane.

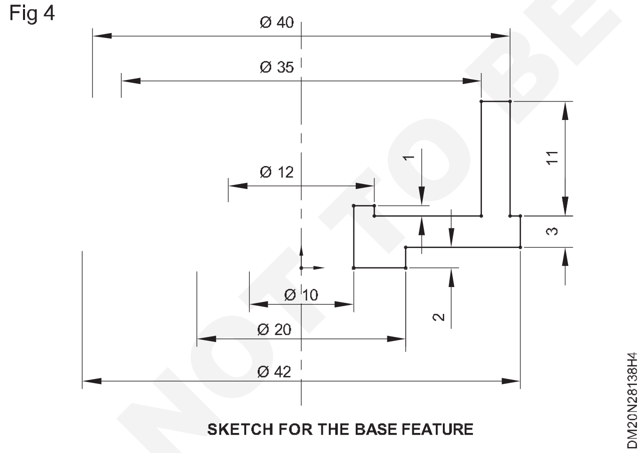
1 Invoke the Revolved Boss/Base tool and select the Right Plane as the sketching plane.

2 Create the Sketch for the base feature and add the required relations and dimensions to the sketch, as shown in (Fig 4)

3 Exit the sketching environment and set the value in the Direction 1 Angle spinner as 360

4 Choose the OK button from the Revolve Property Manager, the base feature is created, as shown in (Fig 5).

Fig 4



Creating the Second Feature

The second feature needs to be created by extruding the sketch up to the selected surface.

1 Invoke the Extruded Boss/Base tool and select the face as the sketching plane, as shown in (Fig 6).

2 Create the sketch of the second feature and add the required relations and dimensions to it, as shown in (Fig 7)

Fig 5

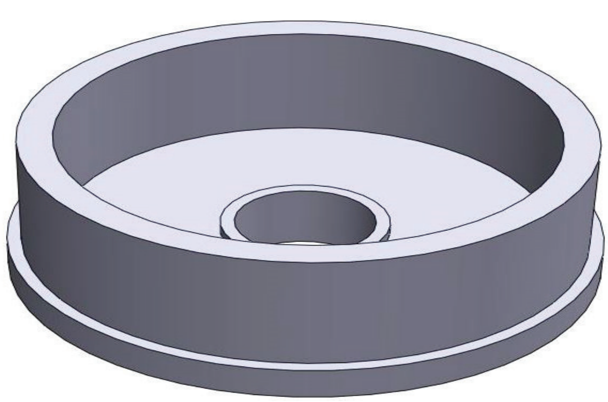


Fig 6

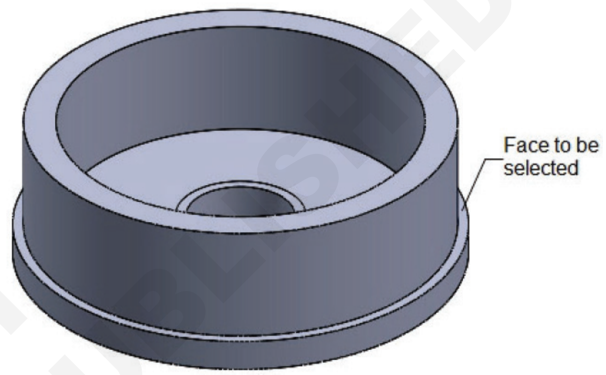
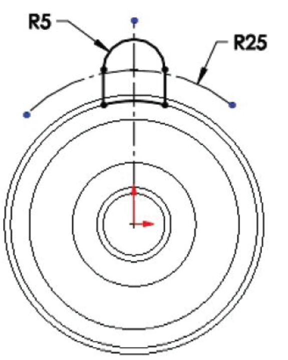


Fig 7



3 Exit the sketching environment. Use the Up To Surface option to extrude the sketch. The surface to be selected is shown in (Fig 8).

4 Choose the OK button from the Boss-Extrude Property Manager, the second feature is created, as shown in (Fig 9).

Creating the Hole Feature

It is evident from (Fig 1) that a counter bore hole needs to be added to the model. It can be added by using the Hole Wizard tool.

1 Select the bottom face of the second feature as the placement plane for the hole feature and press the S key; the Shortcut toolbar is displayed.

2 Choose Hole Wizard from the Shortcut toolbar to invoke the Hole Specification Property Manager.

Fig 8

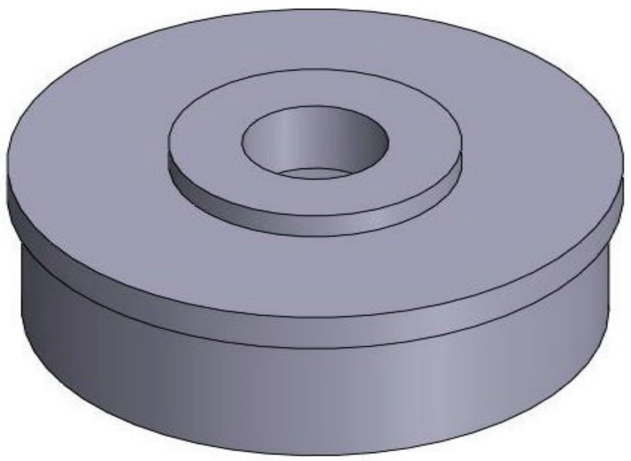
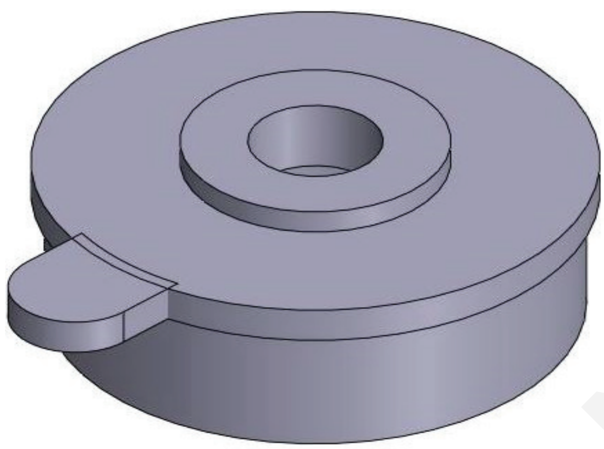


Fig 9



3 Choose the Counter bore button from the Hole Type roll out, if it is not chosen by default, and select the ANSI Metric option from the Standard drop-down list.

4 Select the Socket Button Head Cap Screw – ANSI B18.3.4M option from the Type drop-down list. Select the M3 option from the Size drop-down list. Select Through All in the End Condition drop-down list in the End Condition roll out.

5 Select the Show custom sizing check box to customize the size of the counter bore hole and then enter the following parameters:

Through Hole Diameter: 3.6 mm

counter bore Diameter: 6.70 mm

counter bore Depth: 1.65 mm

6 Choose the Positions tab from the Hole Specification Property Manager, you are prompted to use dimensions and other sketching tools to position the centre of the holes.

7 Specify the position of the hole and then choose the Add Relation button from the Sketch Command Manager, the Add Relations Property Manager will be invoked. Now apply the concentric relation between the centre point of the hole feature and the circular edge of the second feature. Choose the OK button to exit the Add Relations Property Manager.

8 Now, choose the OK button from the Hole Position Property Manager to end the feature creation.

Patterning the Features

After creating the second and third features, you need to pattern them using the Circular Pattern tool.

1 Choose the Circular Pattern button from the Linear Pattern flyout; the Cir Pattern Property Manager is invoked.

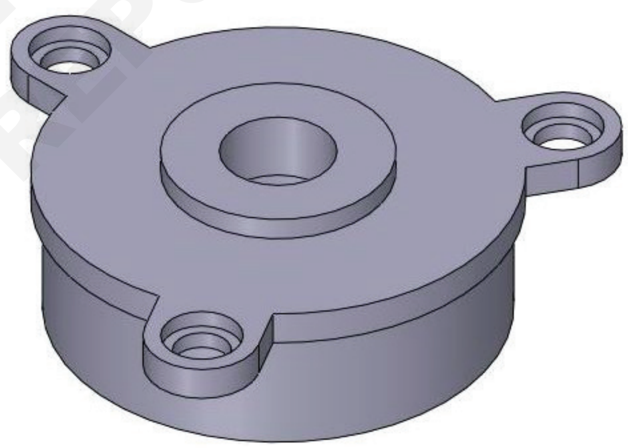
2 Select the second and third features from the drawing area or from the Feature Manager Design Tree that is displayed in the drawing area.

3 Click on the Pattern Axis selection box in the CirPattern Property Manager and select the circular edge of the base feature; preview of the circular pattern of the hole feature is displayed.

4 Set the value in the Number of Instances spinner to 3 and make sure the Equal spacing check box is selected. Clear the Geometry pattern check box from the Options roll out, if it is selected.

5 Choose the OK button from the Cir Pattern Property Manager, the features are patterned, as shown in (Fig 10).

Fig 10



Creating the Rib Feature

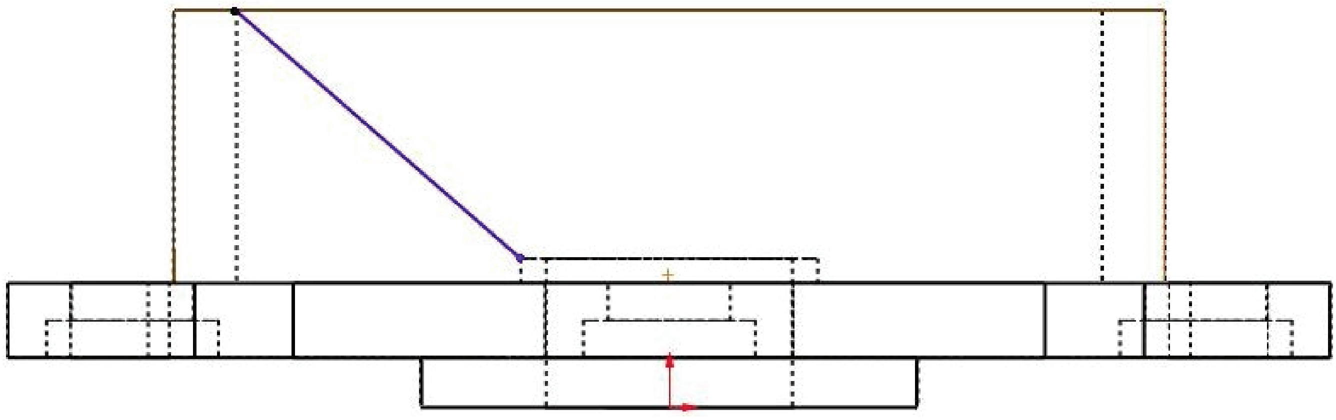
The next feature is a rib feature. The sketch for the rib feature will be created on the Front Plane.

1 Choose the Rib button from the Features Command Manager and select the Front Plane from the Feature Manager Design Tree; the sketch environment will be invoked.

2 Set the display mode to wire frame and then create the sketch for the rib feature and add the required relations, as shown in (Fig 11).

3 Exit the sketching environment and set the value of the Rib Thickness spinner to 2. Reverse the direction of material, if required, using the Flip material side

Fig 11



check box. Use the default values for other options and choose the OK button from the Rib Property Manager.

4 Change the display mode of the model to shaded with edges.

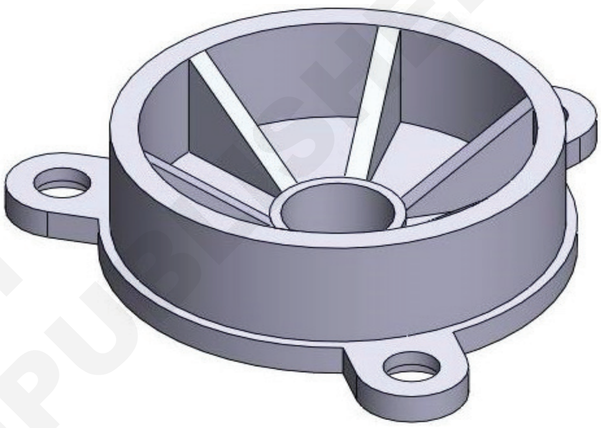
5 Use the Circular Pattern tool to create six instances of the rib feature. The final model after creating all the features is shown in (Fig 12).

Saving the Model

1 Save the model with name c08_tut02 at the location given below: \Documents\SOLIDWORKS\c08

2 Choose File > Close from the SOLIDWORKS menus to close the document.

Fig 12



Create New Assembly Part

Objectives: At the end of this exercise you shall be able to
- **create a new assembly.**

Requirements

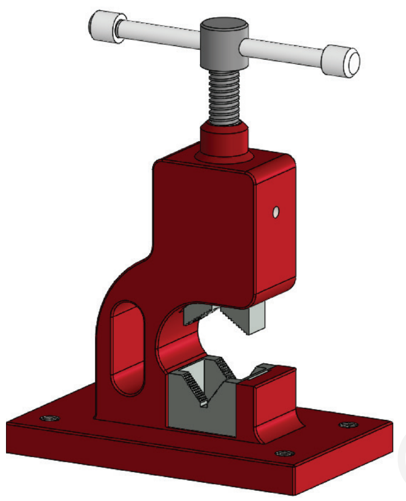
Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Create a new assembly**

Fig 1



You will create all components of the Pipe Vice assembly as separate part documents. After creating the parts, you will assemble them in the assembly document. In this tutorial, you need to use the bottom-up approach for creating the assembly. (Figs 1&2)

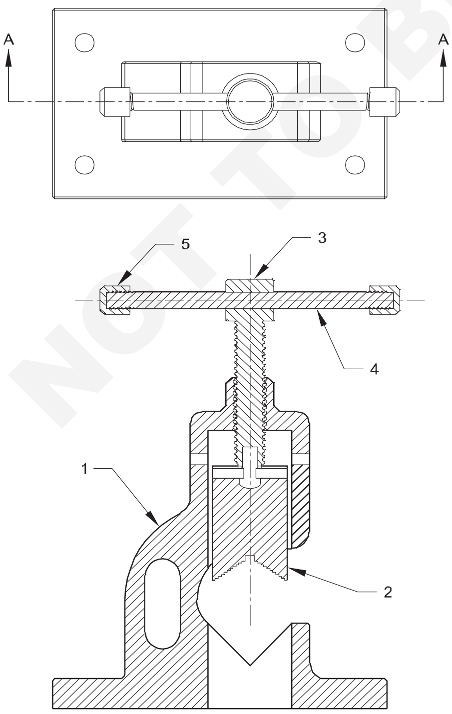
The following steps are required to complete this tutorial:

a Create all the components as separate part documents and then save them. The part documents will be saved at \Documents\SOLIDWORKS\c12\Pipe Vice.

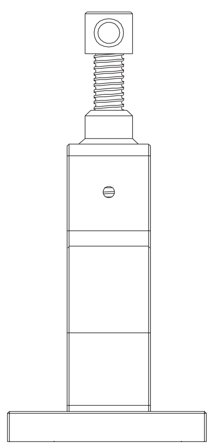
b Place the Base at the origin of the assembly.

c Place the Moveable Jaw and the Screw in the assembly. Apply mates between the Moveable Jaw and the Screw.

Fig 2



ITEM NO.	PART NO.	DESCRIPTION	QTY.
1	BASE		1
2	MOVEABLE JAW		1
3	SCREW		1
4	HANDLE		1
5	HANDLE STOP		2



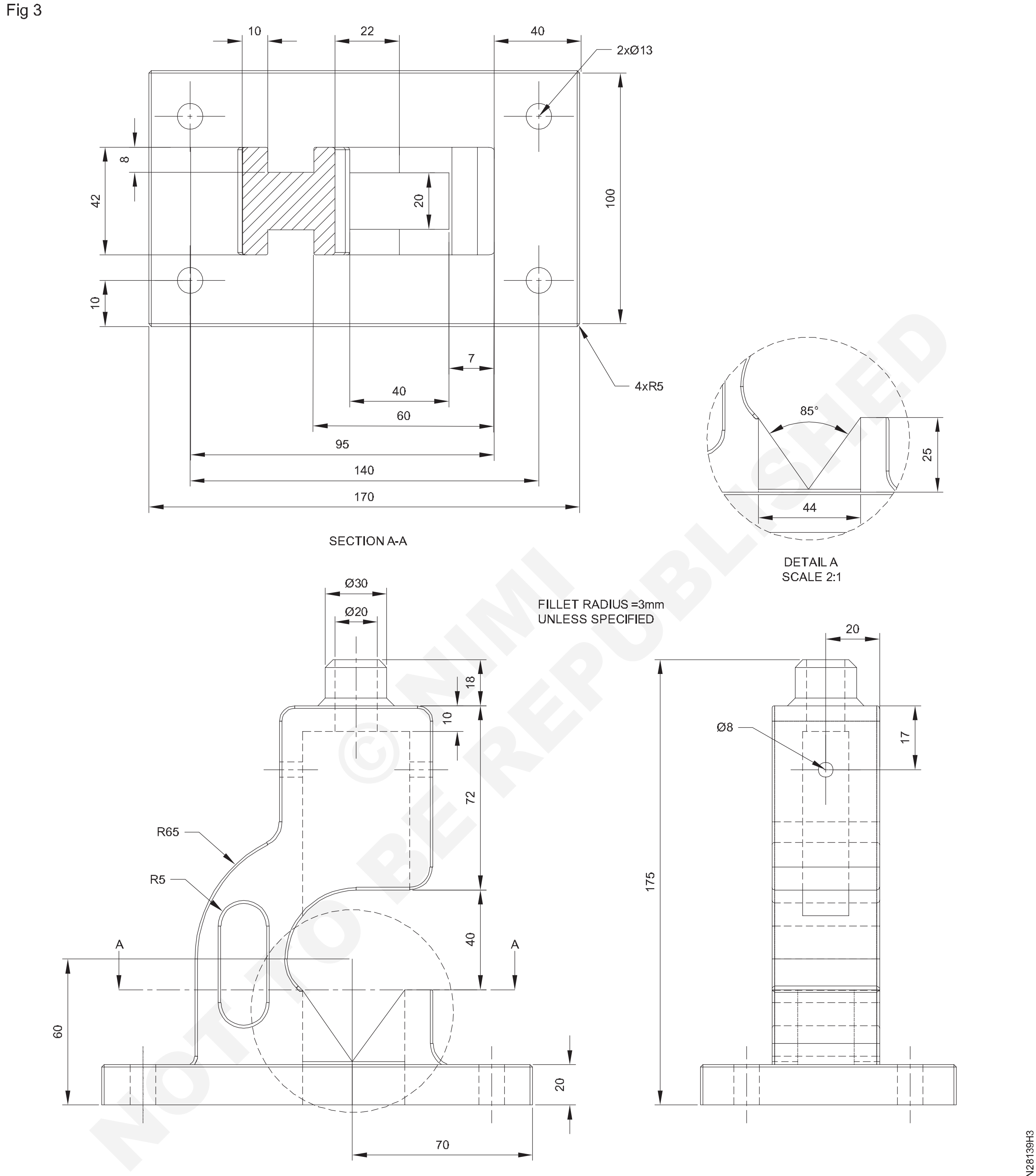
PIPE VICE ASSEMBLY

DM20N28139H2

d Assemble the Screw with the Base.

e Place other components in the Assembly and apply mates to the assembly

Fig 3



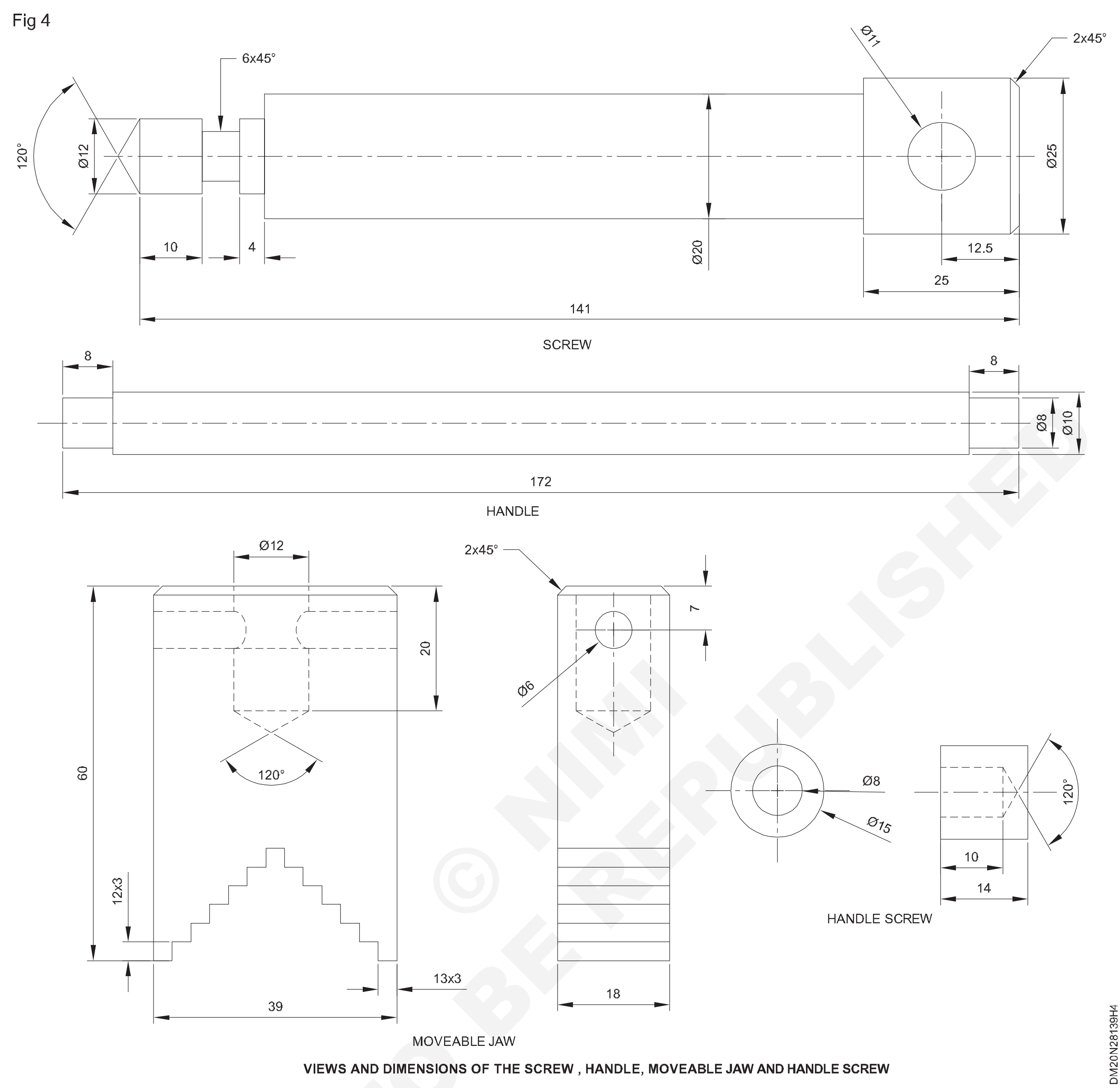
VIEWS AND DIMENSIONS OF THE BASE

Creating Components

1 Create all the components of the Pipe Vice assembly as separate part documents. Specify the names of the files, as shown in (Fig 3). Save the documents at the location \Documents\SOLIDWORKS\c12\Pipe Vice.

Inserting the First Component into the Assembly

After creating all the components of the Pipe Vice assembly, you need to start a new SOLIDWORKS assembly document.



VIEWS AND DIMENSIONS OF THE SCREW , HANDLE, MOVEABLE JAW AND HANDLE SCREW

1 Start a new SOLIDWORKS assembly document; the Begin Assembly Property Manager is displayed by default. (Fig 4)

2 Choose the Browse button from the Part/Assembly to Insert roll out to display the Open dialog box. Open the Pipe Vice folder and double-click on the Base.

3 Choose the OK button from the Begin Assembly Property Manager to place the Base part origin coincident at the origin of the assembly document.

4 Change the view orientation to isometric.

Inserting and Assembling the Moveable Jaw and the Screw

After placing the first component in the assembly document, you need to place the Moveable Jaw and the Screw in the assembly document. After placing these components, you need to apply the required mates.

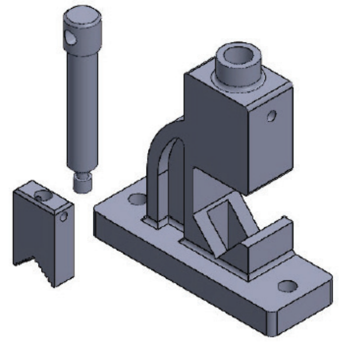
1 Choose the Insert Components button from the Assembly Command Manager. Then, choose the Keep Visible button from the Property Manager to keep it visible. Next, invoke the Open dialog box by choosing the Browse button from the Part/Assembly to Insert rollout.

2 Double-click on the Movable Jaw. Place the component anywhere in the assembly document such that it does not interface with the existing component.

3 Similarly, place the Screw in the assembly document and choose the OK button from the Insert Component Property Manager. (Fig 5) shows the Moveable Jaw, Screw, and Base placed in the assembly document.

First you need to assemble the Screw with the Moveable Jaw. Therefore, you need to fix the Moveable Jaw.

Fig 5



4 Select the Moveable Jaw from the drawing area or from the Feature Manager Design Tree. Right-click to invoke the shortcut menu.

5 Choose the Fix option from the shortcut menu; the Moveable Jaw becomes fixed and you cannot move or rotate it.

6 Invoke the Move Component Property Manager and choose the Smart Mates button from the Move roll out. Double-click on the lower most cylindrical face of the Screw; the Screw appears transparent.

7 Drag the cursor to the hole located at the top of the Moveable Jaw. Release the left mouse button as soon as the concentric symbol is displayed below the cursor. Next, choose the Add/Finish Mate button from the Mate pop-up toolbar.

8 Select the Screw and move it up so that it is not inside the Moveable Jaw.

9 Right-click in the drawing area and then choose Clear Selections from the shortcut menu to clear the current selection.

10 Rotate at the assembly and double-click on the lower flat face of the Screw; the Screw appears transparent.

11 Rotate the model again and select the top planar face of the Movable Jaw. The coincident mate is applied between the two selected faces. Choose the Add/ Finish Mate button from the Mate pop-up toolbar.

12 Choose the OK button from the Smart Mates Property Manager. (Fig 6) shows the Screw after applying mates.

Next, you need to assemble the Screw and the Moveable Jaw with the Base.

13 Select the Moveable Jaw and right click on it; a shortcut menu will be displayed. Choose the Float option from this shortcut menu. Now, you can move the Moveable Jaw and the Screw assembled to it.

14 Press the ALT key, select the cylindrical face of the screw and move the screw toward the hole created on the top face of the Base to add the concentric mate.

15 Invoke the Mate Property Manager and then select the front planar face of the Moveable Jaw and the front planar face of the Base.

16 Choose the Parallel button from the Mate pop-up toolbar and then choose the Add/Finish Mate button to add the Parallel mate between the selected faces.

17 Next, select the faces, as shown in (Fig 7).

Fig 6

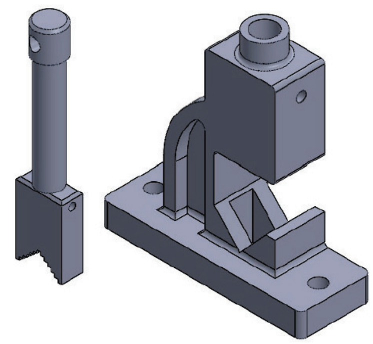
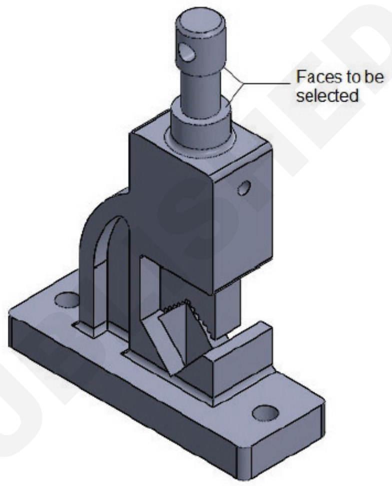


Fig 7



18 Expand the Advanced Mates roll out and then choose the Distance button from it; the Maximum Value and Minimum Value edit boxes become available.

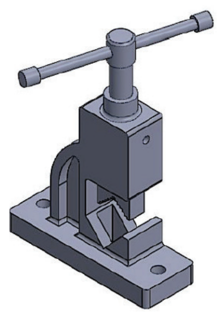
19 Enter 65 and 5 in the Maximum Value and Minimum Value edit boxes respectively. Next, choose OK from the Mate Property Manager.

You will learn more about the advanced mates in the next chapter.

20 Similarly, assemble the other components of the Pipe Vice assembly. (Fig 8) shows the final Pipe Vice assembly.

21 Choose the Save button to save the assembly document at the location \Documents\SOLIDWORKS\c12\Pipe Vice.

Fig 8



— — — — —

Create a 3D Model

Objectives: At the end of this exercise you shall be able to
- **create a 3D Model 2 Bill of material.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Create a 3D Model 2 Bill of material (Fig 1)**

a Copy the Bench Vice folder, which contains parts, assembly, and the drawing document, from Chapter 14 to the folder of the current chapter.

b Delete the views that are not required in the drawing sheet.

c Move the views and arrange them in the drawing sheet.

d Set the anchor on the drawing sheet where the BOM will be attached.

e Generate the BOM.

f Add balloons to the isometric view.

Copying the Bench Vice Assembly Folder to the Current Folder

Before proceeding, you need to copy the model and the drawing document in the folder of the current chapter.

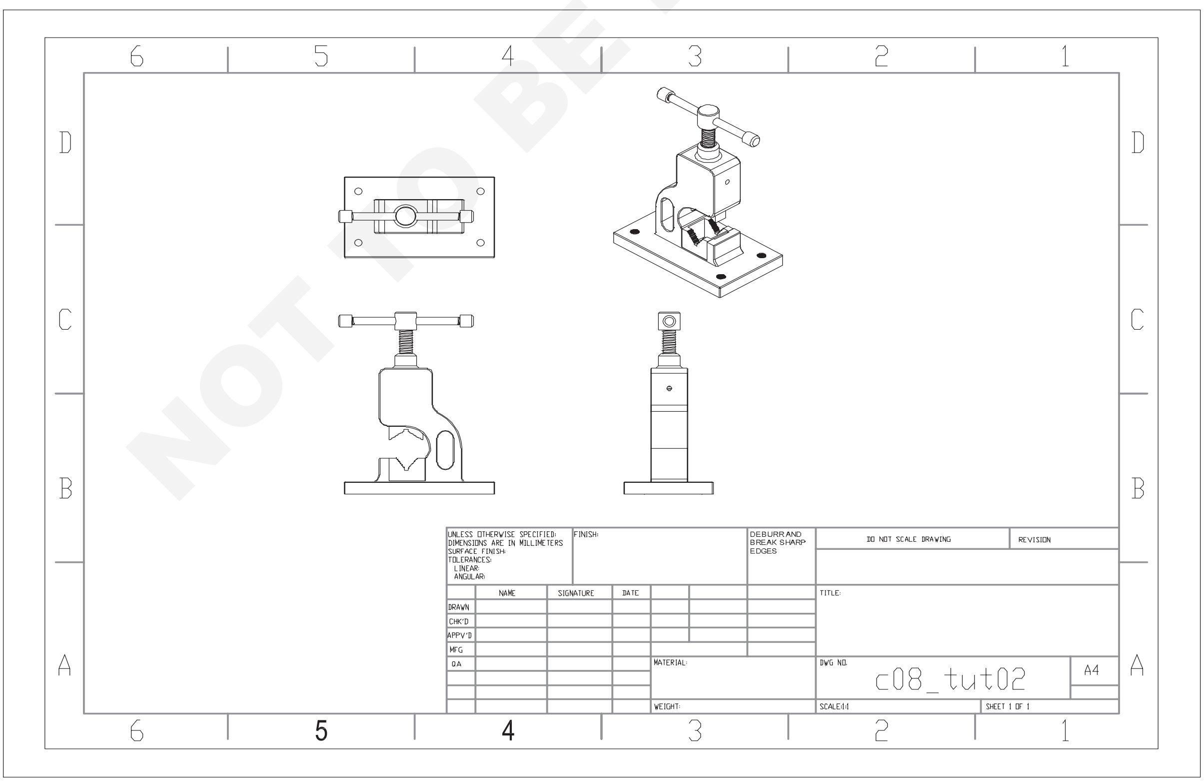
1 Copy the Bench Vice folder from the \Documents\SOLIDWORKS\c14 folder to the folder of the current chapter.

Opening the Drawing Document

After copying the folder, you need to open the drawing document in the SOLIDWORKS window.

The drawing document in which you need to generate the BOM and balloons is displayed in Figure.

Fig 1



DRAWING VIEWS

DM20N28140H1

Deleting the Unwanted View

You need to delete the right view because it is not required in this tutorial.

1 Select the right-side view and press the DELETE key; the Confirm Delete dialog box is displayed.

2 Choose the Yes button from this dialog box; the view is deleted from the current drawing sheet.

Moving the Isometric View

You need to move the exploded isometric view because the BOM will be generated and placed at the top right corner of the drawing sheet.

1 Select the isometric view; the border of the view is highlighted.

2 Move the cursor to the border; the cursor is replaced by the move cursor.

3 Select the border and drag the cursor to move the drawing view and then place it at the required location.

Setting the Anchor for the BOM

Before generating the BOM, you need to set its anchor. The anchor is a point on the drawing sheet to which one of the corners of the BOM coincides. By default, the anchor is defined at the top left corner of the drawing sheet. But in this tutorial, you need to add the BOM at the top right corner of the drawing sheet. Therefore, you need to set the anchor before generating the BOM.

1 Expand Sheet 1 from the Feature Manager Design Tree and then expand Sheet Format 1.

2 Select the Bill of Materials Anchor 1 option, right-click on it to invoke a shortcut menu and then choose the Set Anchor option from it; the drawing views disappear from the sheet.

3 Specify the anchor point on the inner top right corner of the drawing sheet; a point is placed at the selected location.

After you specify the anchor point, the drawing views are displayed automatically in the sheet because the sheet editing environment is invoked automatically.

Generating the BOM

Next, you need to generate the BOM. As discussed earlier, the BOM generated in SOLIDWORKS is parametric. If a component is deleted or added in the assembly, the change is reflected automatically in the BOM. But before generating the BOM, you need to set its text parameters.

1 Invoke the System Options – General dialog box by choosing the Options button from the Menu Bar.

2 Next, choose the Document Properties tab in the System Options – General dialog box to invoke the Document Properties – Drafting Standard dialog box and then choose Annotations > Notes from the area on the left.

3 Choose the Font button from the Test area of the dialog box; the Choose Font dialog box is invoked. Select the Points radio button from the Height area and set the value of the font size to 9 from the list box.

4 Choose the OK button from the Choose Font dialog box. Similarly, change the text height of balloons to 14 and close the dialog box.

5 Select the isometric view and choose Tables > Bill of Materials from the Annotation Command Manger, the Bill of Materials Property Manager is displayed.

6 Select the Attach to anchor point check box in the Table Position roll out.

7 Choose the OK button from the Bill of Materials Property Manager, the BOM is generated. If the BOM is displayed outside the drawing sheet, move the cursor over the BOM; an anchor symbol is displayed. Click on the symbol; the Bill of Materials Property Manager is displayed. Select an appropriate position for placing the BOM from the Table Position roll out and then choose the Close Dialog button; you will notice that the Description column is also displayed in the BOM. But this column is not required, so you need to delete it.

8 Move the cursor over the Description heading and right-click to display a shortcut menu. Choose Delete > Column from the shortcut menu; the column is deleted. The drawing sheet after generating the BOM and deleting the Description column is displayed.

Adding Balloons to the Components

After generating the BOM, you need to add balloons to the components. Before proceeding, make sure that you have changed the font height of balloons to 14, as discussed in the previous section.

1 Select the isometric view and choose the Auto Balloon button from the Annotation Command Manager, the balloons are automatically added to all the components in the isometric view and the Auto Balloon Property Manager is also displayed.

The multiple instances of any component are ignored because the Ignore multiple instances check box is already selected in the Balloon Layout roll out. Also, make sure that the Insert magnetic line(s) check box is cleared.

2 Select 1 Character from the Size drop-down list in the Balloon Settings roll out. Next, choose OK to close this Property Manager.

The balloons are added to all the components. You will notice that the balloons are not properly arranged on the sheet and are placed arbitrarily. Therefore, you need to drag each balloon manually to place it properly.

3 Move the cursor over any one of the balloons and when it is highlighted, drag it to place it at another location.

4 Similarly, drag and place the remaining balloons at proper locations. The final drawing sheet after adding and rearranging balloons.

— — — — —

Prepare drawing & detailing

Objectives: At the end of this exercise you shall be able to
- **create drawing sheets.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Create drawing Sheets**

The following steps are required to complete this tutorial:

a Copy the part document of Tutorial 2 of Chapter 8 in the folder of the current chapter.

b Open the copied part document and start a new drawing document from within the part document.

c Select the standard A4 landscape sheet format and generate the parent view.

d Generate projected views using the Projected View tool.

e Generate the aligned section view using the Section View tool.

f Generate the detail view.

g Save and close the drawing document.

Copying and Opening the Part Document

1 Create a folder with the name c14 in the SOLIDWORKS directory and copy c08_tut02.sldprt from the location\Documents\SOLIDWORKS\c08.

2 Start SOLIDWORKS and open the part document that you copied in the c14 folder.

Starting a New Drawing Document

As mentioned earlier in this chapter, you can start a new drawing document from the part document. This way, the model in the part document is automatically selected and you can generate its drawing views.

1 Choose New > Make Drawing from Part/Assembly from the Menu Bar; the Sheet Format/Size dialog box is displayed.

2 Clear the only show standard formats check box and select the A4 (ANSI) landscape sheet from the list box in this dialog box and choose the ok button, a new drawing document is started with the standard A4 sheet and the view palette task pane is displayed automatically.

Generating the Parent View and the Projected Views

Before you proceed to generate the drawing views, you need to confirm, whether the projection type for the current sheet is set to the third angle.

1 Click anywhere on the sheet to close the View Paletee task pane. Select Sheet 1 from the Feature Manager Design Tree and then right-click to display shortcut menu. Choose the Properties option from the shortcut menu; the Sheet Properties dialog box is displayed.

2 Select the Third angle radio button from the Type of projection area if not selected by default and then choose the Apply Changes button.

3 Choose the View Palette tab to view the View Palette task pane and make sure the Auto-Start Projected View check box is selected.

4 Selected the Front view from the View Palette task pane and drag it to the middle left of the drawing sheet above the title block. Drop the view at this location to place the front view, refer to (Fig 1). The Projected View Property Manager is invoked automatically and preview of the projected view is attached to the cursor.

5 Next, move the cursor vertically upward from the front view; a preview of the top view is displayed. Specify a point to place the top view, refer to (Fig 1). Preview of another projected view with the front view as the parent view is attached to the cursor.

6 Next, move the cursor horizontally toward right from the front view, a preview of the side view is displayed. Specify a point to place the side view, refer to (Fig 1).

7 Move the cursor horizontally toward the right and then move it upward; a preview of the isometric view is displayed. Specify a point to place the isometric view. Next, choose the OK button in the Property Manager.

The current location of the isometric view of the model is such that it will interfere with the aligned section view that you need to place next. Therefore, you need to move the isometric view close to the top right corner of the drawing sheet.

8 Move the cursor over the isometric view; the bounding box of the view is displayed in orange.

9 Click to select the view; the border of the view is highlighted.

10 Move the cursor on one of the border lines of the view; the cursor changes into a move cursor.

11 Press and hold the left mouse button and drag the view close to the upper right corner of the drawing sheet. The drawing sheet after generating and moving the isometric view is shown in (Fig 1).

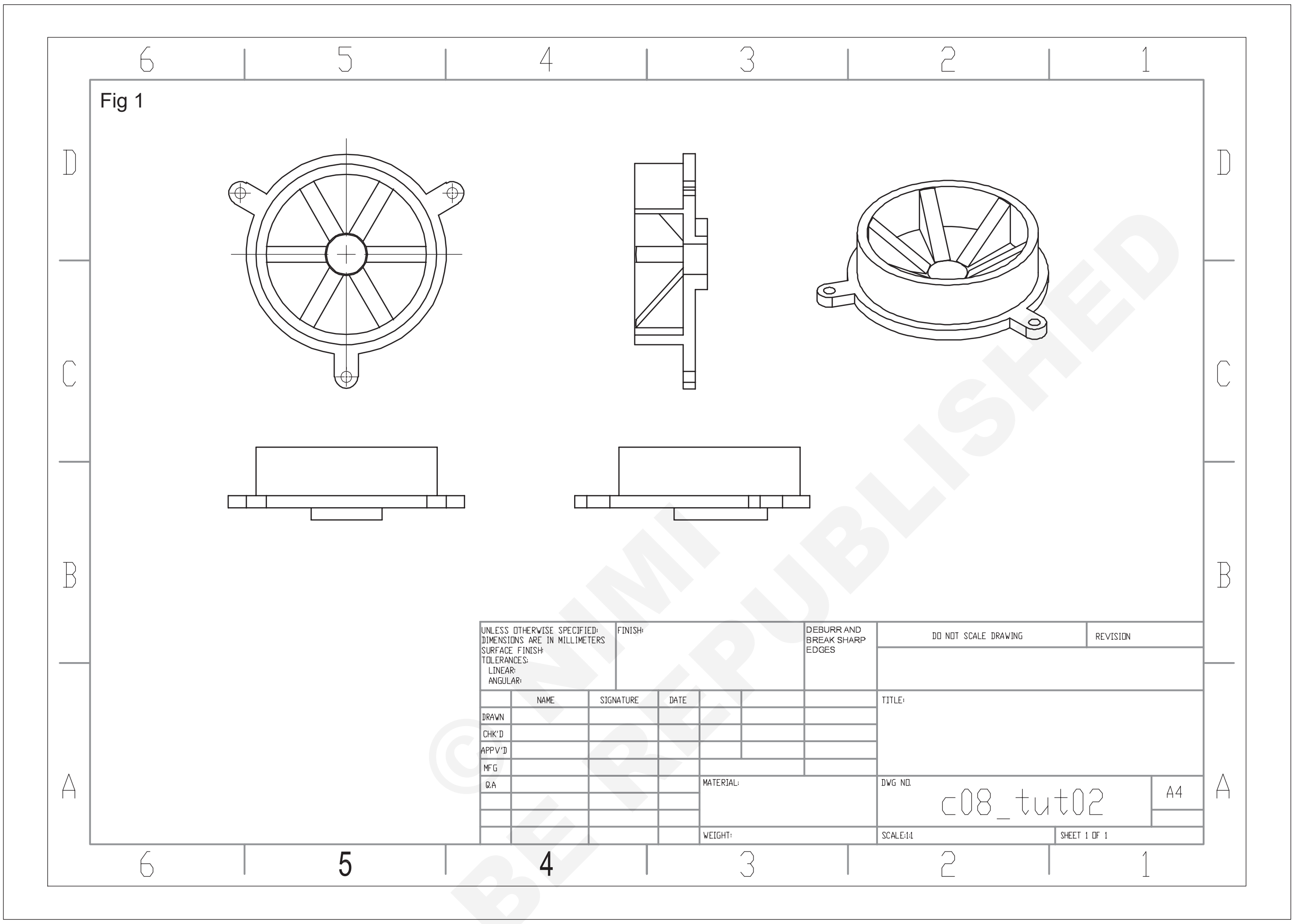
The center marks are automatically created in the drawing views of the circular features in a model.

Generating the Aligned Section View

1 Click on the top view to activate it.

2 Invoke the Design Command Manager and choose the Section View button; the Section View Assist Property Manager is displayed.

Fig 1



3 Choose the Aligned button in the Cutting Line rollout and create the cutting plane line, as shown in the Section View pop-up toolbar is displayed. Choose the OK button from the pop-up toolbar; the SOLIDWORKS information box is displayed with the message that the selected model has a rib feature that will vary the hatching of the section view. It also asks you whether to make an aligned section view or a foreshortened section view.

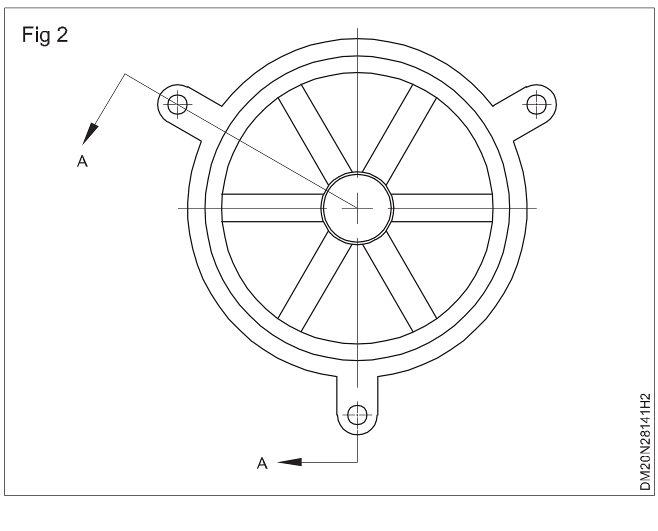
4 Choose the Make aligned section view option from the SOLIDWORKS information box; the aligned section view is attached to the cursor.

The view generated is normal to the vertical line. If the direction of viewing the aligned section view is in opposite direction, you need to flip it after placing the view.

5 Move the cursor to the right of the top view and place the aligned section view; the Section View Property Manger is displayed. If the direction of viewing is not the required one, select the Flip Direction button. Click anywhere on the sheet to exit the Property Manager.

The sheet after generating the aligned section view is shown in (Fig 2).

Fig 2



— — — — —

Create a 3D transition figure: Create 3D model by annotating holes and threads, create centre lines, symbols and leaders, create simulation, plot the model

Objectives: At the end of this exercise you shall be able to
- **create a 3D transition figure.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Create a 3D transition figure**

1 Create a 3D Model with Annotations, Holes and Threads:

- Create a new part documents.
- Use sketch tools to create the base profile for your transition figure.
- Add holes and threads using the "Hole Wizard" and "Thread" features from the "Features" tab.
- For annotations, use the "Annotations" tab to add dimensions to define the geometry.

2 Create Centre Lines, Symbols, and Leaders:

- Use the "Centre line" tool in the "Annotations" tab to create centre lines that represent symmetry or reference axes.
- Insert symbols (e.g. surface finish symbols) using the "Symbols" tool from the Annotation" tab.
- Add leaders to annotations using the "Leader" tool to connect notes or symbols to specific locations.

3 Create Simulation:

- Go to the "Simulation" tab (if available in your SOLIDWORKS version) or use SOLIDWORKS Simulation.
- Define the material properties, loads, restraints and other simulation parameters.
- Run the simulation to analyze stress, deformation, and other mechanical behaviour.

4 Plot the Model:

- Create a new drawing document.
- Insert the 3D model view by using the "Model View" tool and selecting the 3D model from the part.
- Use the "Annotations" tab to add dimensions, centre lines, symbols and leaders in the drawing view.
- You can also add notes and other information to explain the design or simulation results.
- Set up the drawing sheet with title blocks, and any required information.
- If you're plotting to paper or a PDF, use the "File" menu to print or save the drawing as a PDF.

— — — — —

Convert or save as solid works file into drawing format

Objectives: At the end of this exercise you shall be able to
- **convert solid work into drawing format.**

Requirements

Tools/Equipment/Machines

- Solid works 2017 or Higher version.

PROCEDURE

TASK 1: **Convert Solid work into drawing format**

1 Open or Create the Drawing:

- Open the part or assembly file you want to create a drawing for.
- Go to “File” > “New” > “Drawing” to create a new drawing file.

2 Add Model Views:

- In the drawing, use the “Model View” tool to insert the 3D Model.
- Choose the desired orientation and scale for the model view.

3 Add Annotations and Details:

- Use the “Annotations” tab to add dimensions, centre lines, symbols and other annotations to the model view.
- Add title blocks, borders, and other necessary information to the drawing sheet.

4 Save the Drawing:

- Once you’ve added all the necessary views and annotations, go to “File” > “Save As”.
- Choose a location to save the drawing and give it a name.
- Select the appropriate file type for the drawing. The most common format is “SOLIDWORKS” Drawing (*.slddrw).”

5 Optional: Export to Other Formats:

- If you need to share the drawing with others who don’t have SOLIDWORKS, you can also export it to different file formats such as PDF, DXF, or DWG.
- Go to “File” > “Save As” and choose the desired file format from the drop/down list.

— — — — —

Construct detailed and Assembly drawing

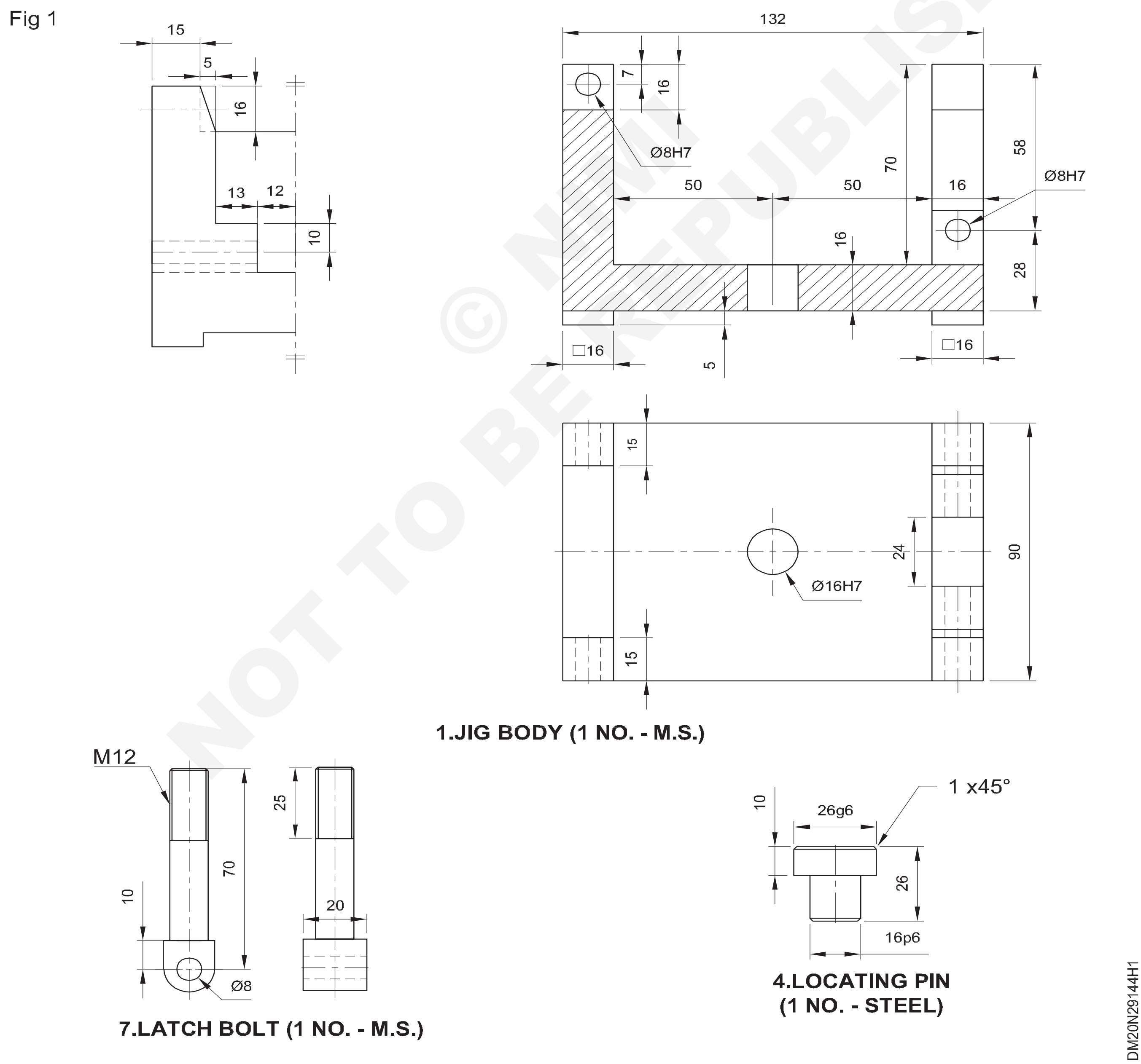
Objectives: At the end of this exercise you shall be able to
- **draw the details of the drilling jigs**
- **draw the full plan**
- **draw the full sectional elevation of the assembled view.**

Requirements
Tools/Equipment/Machines
• Auto CAD 2018 or Higher version.

PROCEDURE

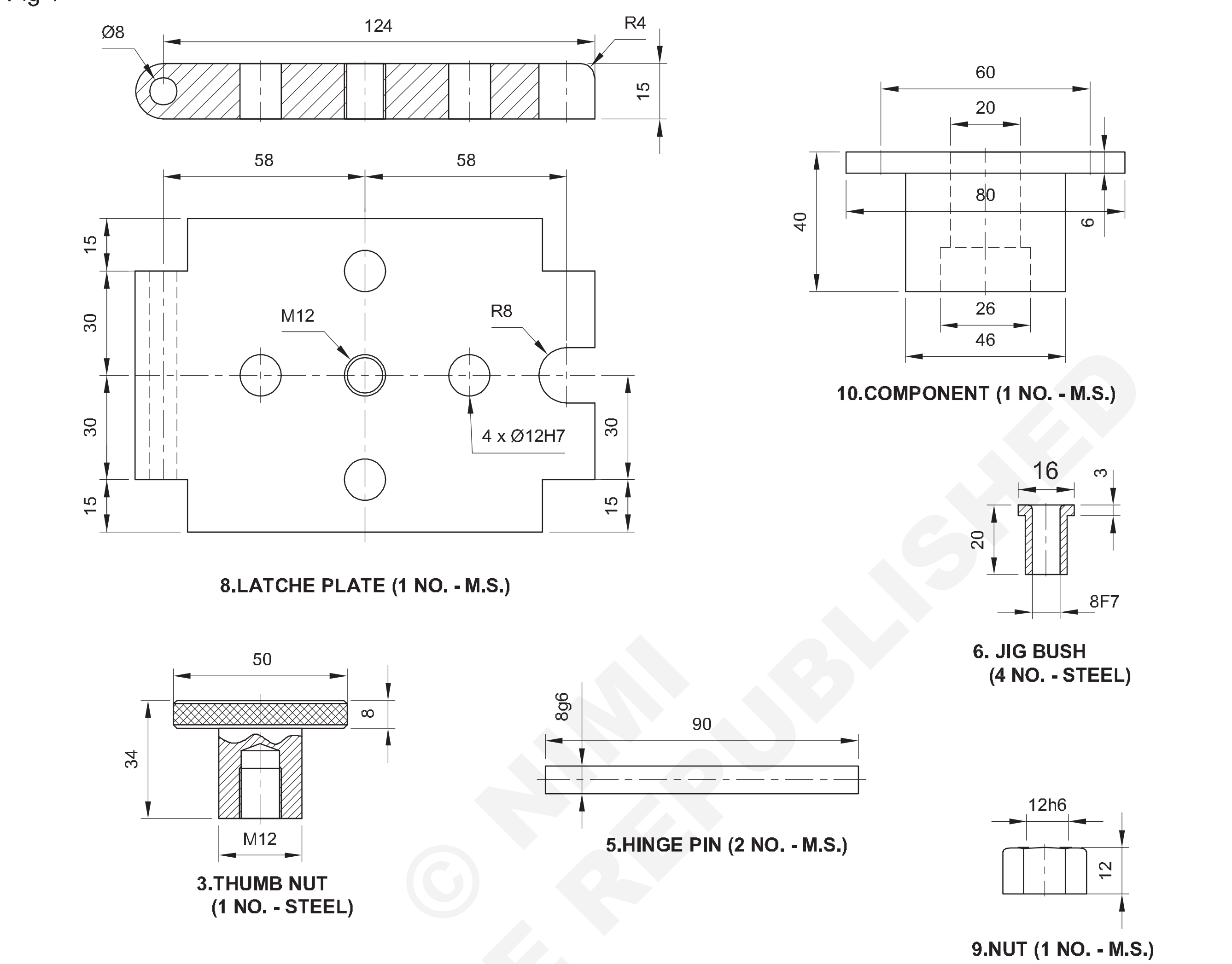
TASK 1: **Draw the assembly drawing of the drilling using the details by using CAD (Fig 1)**

Fig 1



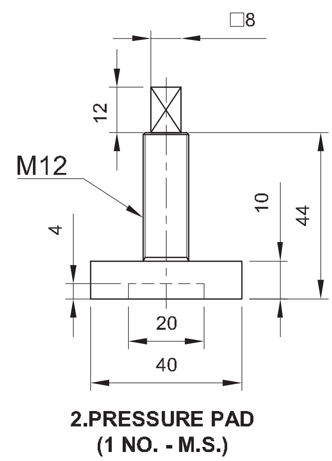
TASK 2: **Prepare the assembly during the drilling and also the part list**

Fig 1



ITEM LIST

ITEM	DESCRIPTION	QTY.	MATERIAL
1	JIG BODY	1	M.S.
2	PRESSURE PAD	1	M.S.
3	THUMB NUT	1	M.S.
4	LOCATING PIN	1	STEEL
5	HINGE PIN	2	M.S.
6	JIG BUSH	4	STEEL
7	LATCH BOLT	1	STEEL
8	LATCH PLATE	1	M.S.
9	NUT	1	M.S.
10	COMPONENT	1	M.S.



DM20N29144J1

Draw the following views

i Full sectional front elevation as given in (Fig 1).

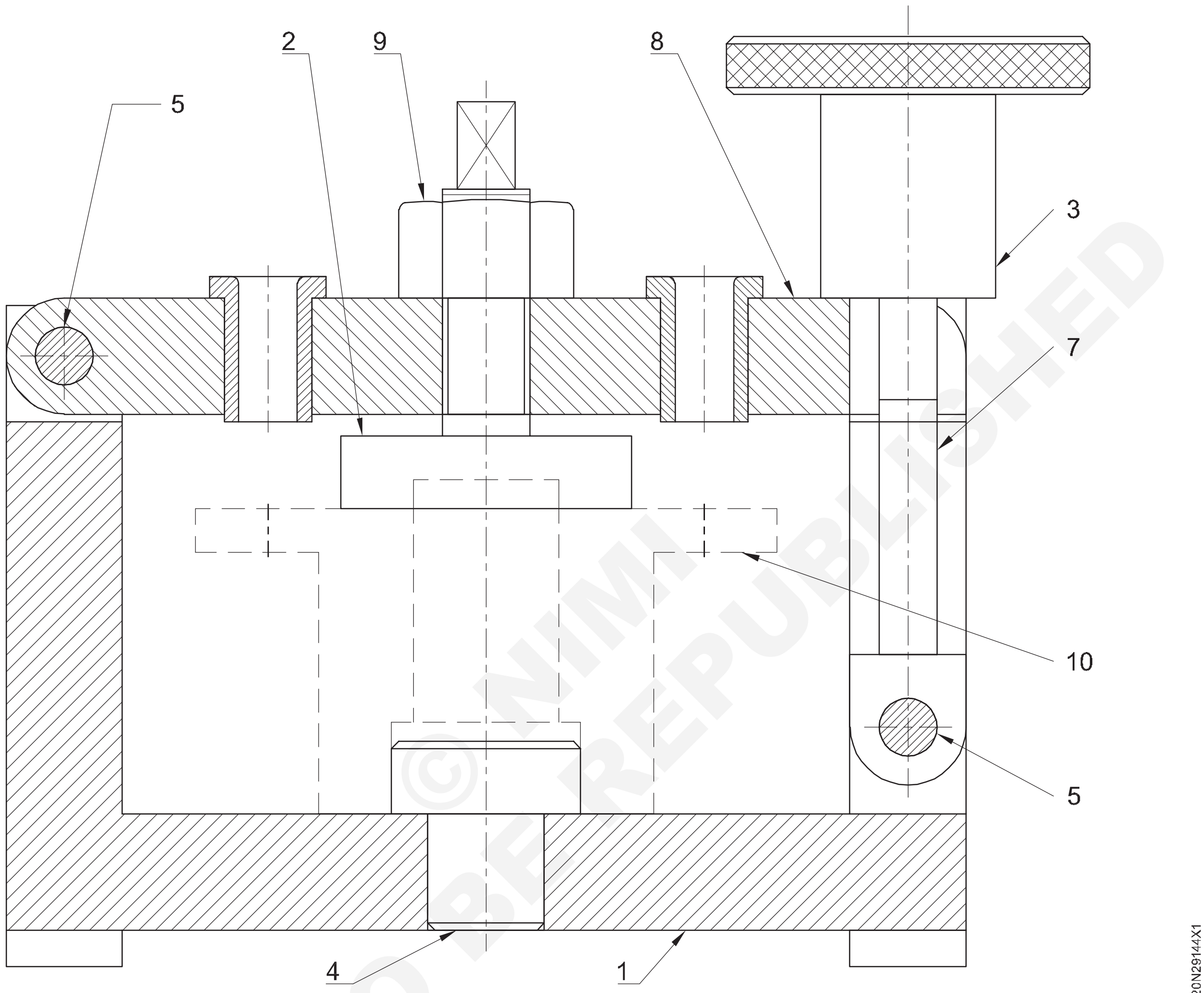
ii Full plan.

— — — — —

TASK 3: **Draw the followings views of the Drill Jig (Fig 1)**

- Draw the working drawing details of drill jig with required views.
- Draw the full sectional assemble elevation of the drill jig.
- Draw the plan.

Fig 1



FULL SECTIONAL ELEVATION OF DRILLING JIG

Create production drawing of a screw jack

Objectives: At the end of this exercise you shall be able to
- **draw use detailed drawing of a screw pack.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Draw use detailed drawing of a screw pack (Figs 1&2)**

Fig 1

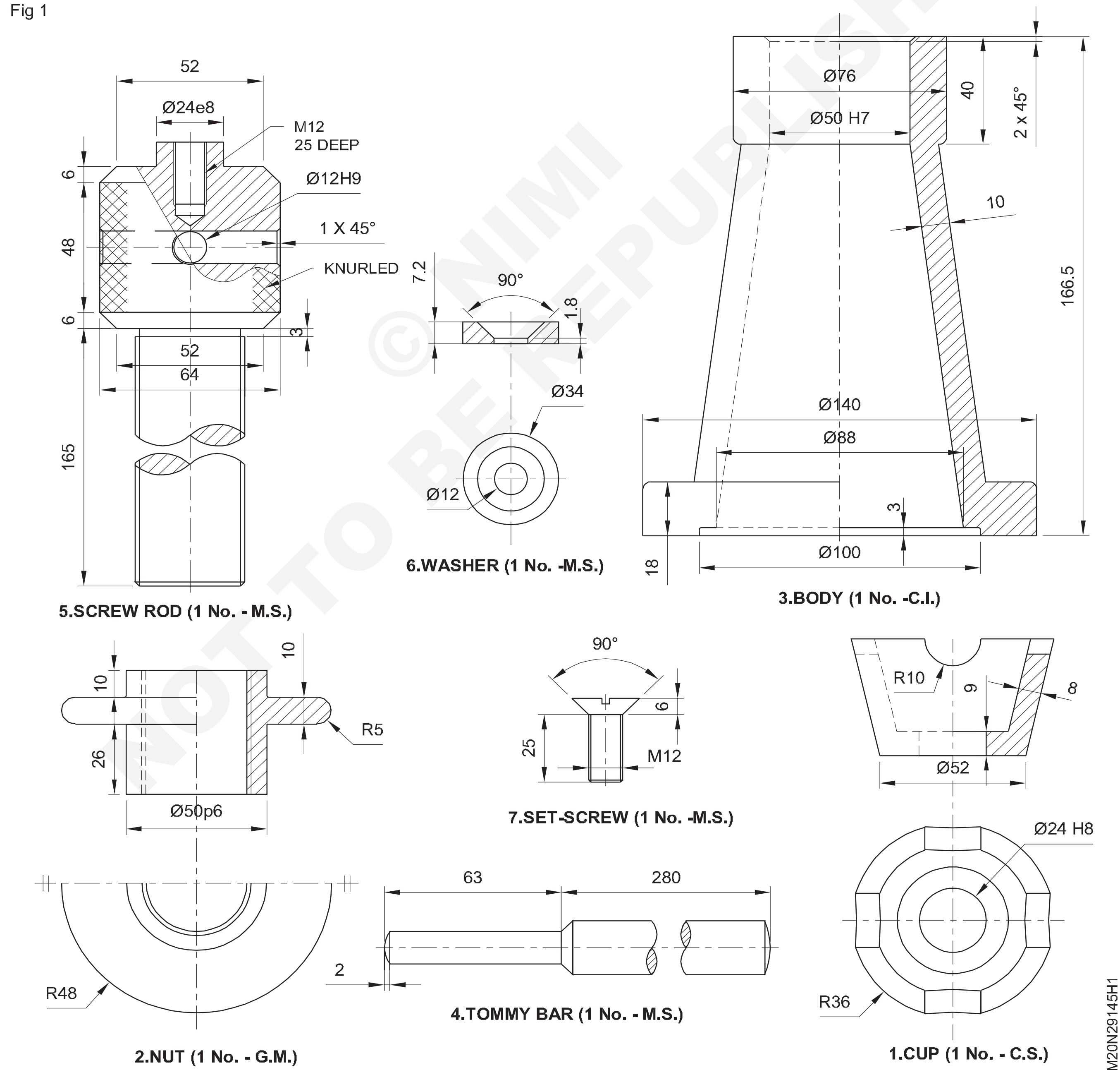
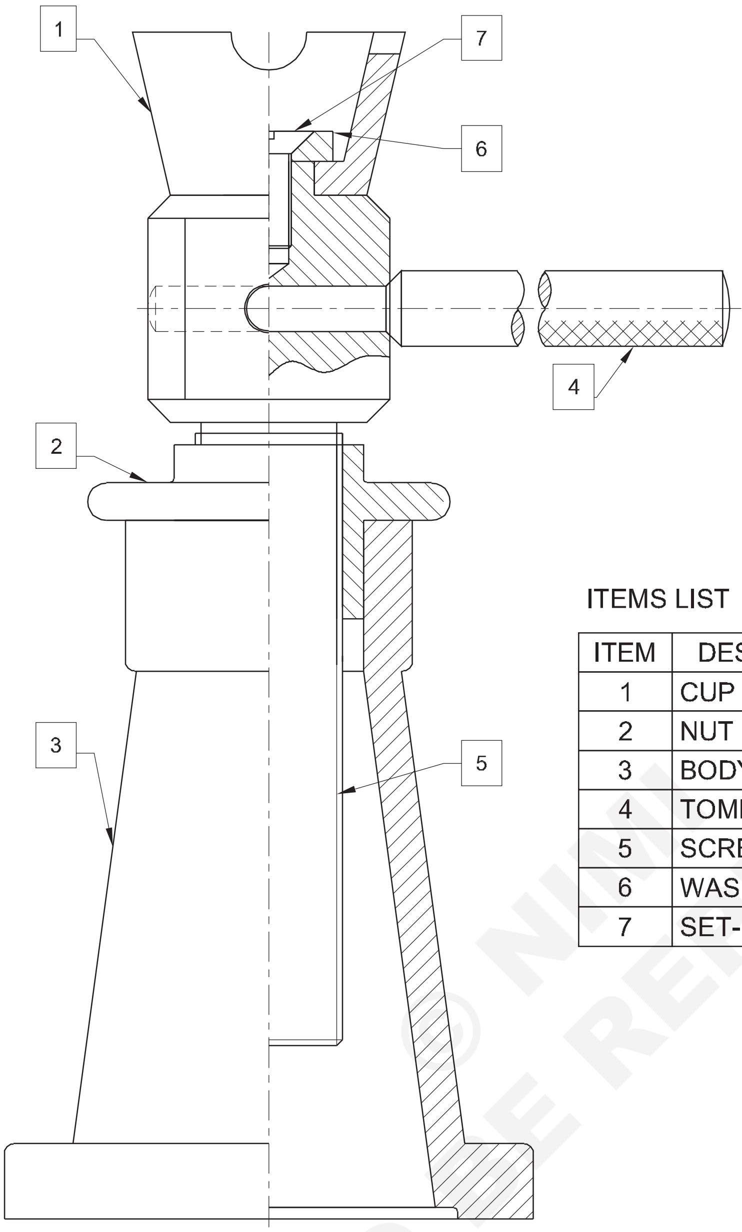


Fig 2



RIGHT HALF SECTIONAL ELEVATION

ITEMS LIST

ITEM	DESCRIPTION	QTY.	MATERIALS
1	CUP	1	C.S.
2	NUT	1	G.M.
3	BODY	1	C.I.
4	TOMMY BAR	1	M.S.
5	SCREW ROD	1	M.S.
6	WASHER	1	M.S.
7	SET-SCREW	1	M.S.

DM20N29145H2

Create the check list for self assessment revision table

Objectives: At the end of this exercise you shall be able to
- **prepare/create a check list for production drawing of a component**
- **prepare a revision mark by noting revision table.**

Requirements

Tools/Equipment/Machines

- Auto CAD 2018 or Higher version.

PROCEDURE

TASK 1: **Prepare/create a check list for production drawing of a component & revision marks**

Check list for the production drawing & Revision marks

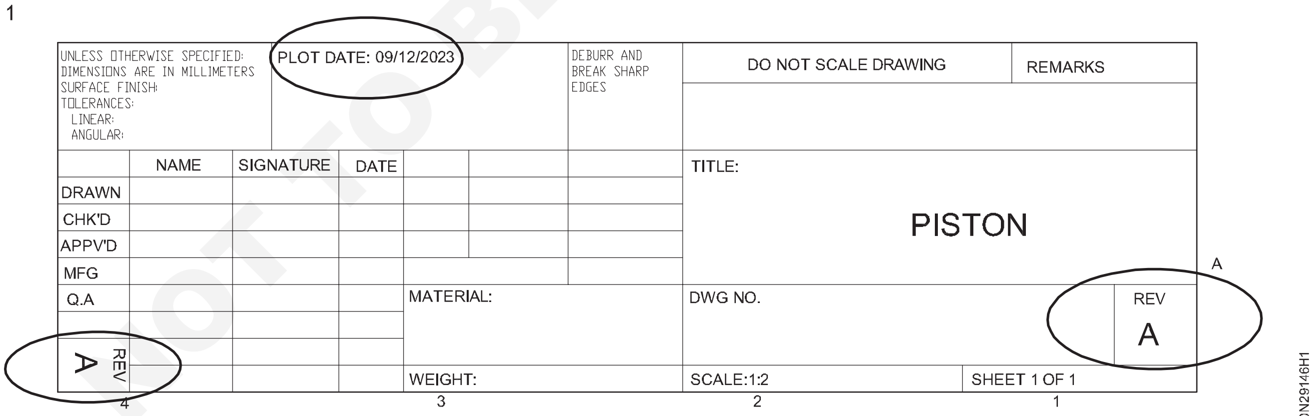
- To Ensure that all the part drawings are prepared with reference to assembly drawing.
- To Ensure that the proper projection method used in the drawing
- To Ensure that all the parts are dimensional as per BIS.
- To Ensure that Geometrical tolerance symbols and machining symbols are provided as per BIS.
- To Ensure that proper fit. Provided for the mating parts.
- To Ensure that the part no, material, qty & notes for heat treatment other important notes are provided.
- To Ensure that the welded parts have welding symbols.
- To Ensure that the bought - out items like - fasteners etc.., should be mentioned in the bill of material.
- To Ensure that the scale used and the drawing number are mentioned in the title block.
- To Ensure that if any changes are made in the drawing it should be mentioned in the revision table.

— — — — —

TASK 2: **Prepare a revision mark by noting revision table (Figs 1&2)**

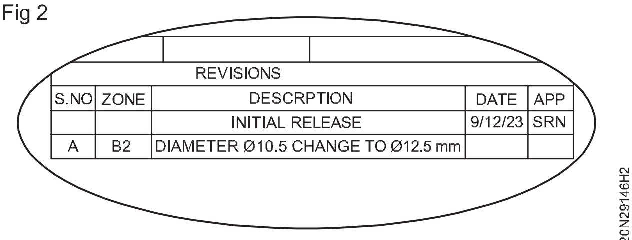
Revision mark by noting revision table

Fig 1



REVISED TITLE BLOCK

Fig 2



REVISIN HISTORY UPDATED

— — — — —