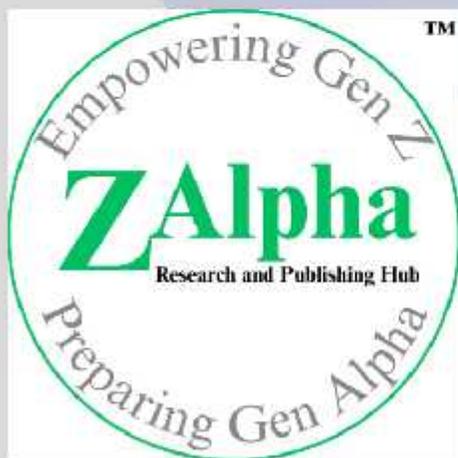# RE-ENGINEERING

FOR

INNOVATION

Dr. Manickavasaham G

Mr. L Chandra Sekaran

Mr. Dhushyanth Ajay S

Mr. Prakash Balan G

Re-Engineering for Innovation

First Edition
(For Undergraduate Engineering Students)

Dr. MANICKAVASAHAM G
Mr. L CHANDRA SEKARAN
Assistant Professor,
Department of Mechanical Engineering,
Mookambigai College of Engineering,
Pudukkottai District,
Tamil Nadu, India.

Mr. DHUSHYANTH AJAY S
Mr. PRAKASH BALAN G
Undergraduate Student
Department of Mechanical Engineering,
Mookambigai College of Engineering,
Pudukkottai District,
Tamil Nadu, India.

Re-Engineering for Innovation

Authors: Dr. MANICKAVASAHAM G, Mr. L CHANDRA SEKARAN, Mr. DHUSHYANTH AJAY S, & Mr. PRAKASH BALAN G

Preface

Engineering education today emphasizes not only theoretical knowledge but also the ability to observe, measure, analyze, and apply engineering concepts to real-world products. This book has been prepared with the objective of bridging the gap between theory and practical application, especially in the areas of product development, reverse engineering, re-engineering, basic measurements, freehand sketching, and CAD modeling.

The content of this book is structured to support Outcome-Based Education (OBE) and Revised Bloom's Taxonomy (RBT) levels, with special emphasis on Understand and Apply. Students are guided systematically from understanding the fundamentals of engineering design and reverse engineering to applying measurement techniques, sketching practices, and CAD modeling on simple, commonly used products and mechanical components.

This book covers topics such as:

- Engineering design and design processes
- Product development and product life cycle
- Reverse engineering and re-engineering concepts
- Basic and precision measuring instruments
- Hand sketching techniques
- CAD modeling of simple disassembled components
- Failure analysis, forensic engineering, and ethics in reverse engineering
- Practical laboratory exercises with real-life products

Special care has been taken to present the material in simple language, supported by step-by-step procedures, illustrations, tables, and practical exercises, making it easy for students to understand, remember, and apply. This book is intended to serve as a lab manual, reference text, and learning guide for undergraduate engineering students, particularly in mechanical and allied disciplines.

It is hoped that this book will help students develop practical skills, analytical thinking, and design confidence, which are essential for modern engineering practice.

Dr. MANICKAVASAHAM G
Mr. L CHANDRA SEKARAN
Mr. DHUSHYANTH AJAY S
Mr. PRAKASH BALAN G

Acknowledgement

The successful completion of this book would not have been possible without the guidance, encouragement, and support of many individuals.

First and foremost, I express my sincere gratitude to my teachers and mentors, whose knowledge, experience, and continuous motivation have been a constant source of inspiration. Their guidance has played a vital role in shaping the academic and practical orientation of this work.

I extend my heartfelt thanks to the students, whose curiosity, questions, and learning needs inspired the preparation of this book in a clear, student-friendly manner. Their participation in laboratory sessions and feedback helped in refining the exercises and explanations.

Finally, I acknowledge the encouragement and support of my family and well-wishers, whose patience and motivation made this work possible.

I sincerely hope this book will be useful to students and educators and contribute positively to engineering education.

This book is made available to students exclusively for educational purposes and is not intended for commercial sale.

<div align="right">

Dr. MANICKAVASAHAM G
Mr. L CHANDRA SEKARAN
Mr. DHUSHYANTH AJAY S
Mr. PRAKASH BALAN G

</div>

Contents

1. Introduction to Re-Engineering and Reverse Engineering

In engineering practice, existing products are often studied and improved instead of designing new products from scratch. Two important approaches used for this purpose are Reverse Engineering and Re-Engineering. Though they are closely related, their objectives are different.

1.1 Reverse Engineering

Reverse engineering is the process of understanding an existing product by carefully studying it. The product is disassembled to examine its components, materials, dimensions, and working principles. Engineers create sketches, drawings, or CAD models to represent the original design.

Primary objective: Understand how the product works

Activities

- ❖ Disassembly of product: Carefully taking apart a product to study its individual components and construction.
- ❖ Measurement and sketching: Measuring product parts and drawing neat sketches to record their size and shape.
- ❖ CAD modeling: Creating accurate digital 2D or 3D models of product components using CAD software.
- ❖ Functional analysis: Studying how each part works and how all parts together perform the product's function.

1.2 Re-Engineering

Re-engineering is the process of improving or modifying a product after it has been understood. Based on the knowledge gained from reverse engineering, changes are made to enhance performance, reduce cost, improve safety, or add new features.

Primary objective: Improve the product design

Activities

- ❖ Design modification: Changing the existing design to improve function, safety, or usability.
- ❖ Material or cost optimization: Selecting better or economical materials to reduce cost without affecting quality.
- ❖ Performance enhancement: Improving the efficiency, speed, strength, or reliability of the product.
- ❖ Prototype development: Creating a working model of the improved design for testing and validation.

1.3 Comparison: Reverse Engineering vs Re-Engineering

Aspect	Reverse Engineering	Re-Engineering
Purpose	Understanding	Improving
Design change	No	Yes
Focus	Study and analysis	Modification and enhancement
Outcome	Drawings and models	Improved product

1.4 Real-Life Examples

Example 1: Mobile Phone

- ❖ Reverse Engineering: Opening a mobile phone to study its battery, display, and internal layout.
- ❖ Re-Engineering: Redesigning the phone to increase battery life or make it thinner.

Example 2: Ceiling Fan

- ❖ Reverse Engineering: Studying the motor, blades, and speed regulator.
- ❖ Re-Engineering: Modifying blade design to reduce power consumption and noise.

Example 3: Agricultural Sprayer

- ❖ Reverse Engineering: Examining the pump, nozzle, and tank design.
- ❖ Re-Engineering: Improving nozzle design for better spray efficiency and less water usage.

Reverse engineering helps engineers learn from existing products, while re-engineering enables them to create better versions. Both play a vital role in modern product development and innovation.

2. Understand the Basics of Product Development, Product Life Cycle, and Reverse Engineering

2.1 Product Development

Product development is the systematic process of converting an idea into a usable product that meets customer needs. It involves planning, designing, developing, testing, and launching a product. The main aim of product development is to create products that are functional, reliable, economical, and user-friendly.

Stages of product development

- ❖ Idea generation: Identifying customer needs or problems
- ❖ Concept development: Creating possible solutions
- ❖ Design: Preparing sketches and CAD models
- ❖ Prototype development: Building a trial model
- ❖ Testing and validation: Checking performance and safety
- ❖ Production and launch: Manufacturing and releasing the product

2.2 Product Life Cycle (PLC)

The product life cycle describes the stages a product goes through from its introduction to its withdrawal from the market. Every product has a limited life span, and understanding the life cycle helps in planning improvements and innovations.

Stages of Product Life Cycle

- ❖ Introduction: Product is launched into the market
- ❖ Growth: Sales increase and product gains acceptance
- ❖ Maturity: Sales stabilize and competition increases
- ❖ Decline: Sales decrease due to new technologies or alternatives

2.3 Reverse Engineering

Refer section 1.1 and 1.4

2.4 Relationship between Product Development, PLC, and Reverse Engineering

In engineering, Product Development, Product Life Cycle, and Reverse Engineering work together to create, monitor, and improve products throughout their life.

2.4.1 Product Development → Creates

Product development focuses on creating a new product from an idea.

Function

- ❖ Converts ideas into real products
- ❖ Involves design, prototyping, testing, and production

Real-life example

A company designs and develops a new electric scooter with better battery and smart features.

2.4.2 Product Life Cycle (PLC) → Tracks

PLC tracks a product's journey in the market over time.

Function

- ❖ Shows how sales and demand change
- ❖ Helps decide when to improve or replace a product

Stages: Introduction → Growth → Maturity → Decline

Real-life example

A smartphone model

- ❖ Introduced with marketing
- ❖ Grows in sales
- ❖ Reaches maturity
- ❖ Declines when a newer model is released

2.4.3 Reverse Engineering → Understands

Reverse engineering focuses on understanding an existing product.

Function

- ❖ Studies how a product is designed and works
- ❖ Helps learn from existing designs

Real-life example

Opening and studying an old ceiling fan to understand its motor, blades, and wiring.

Product development brings new products into the market, the product life cycle tracks their success over time, and reverse engineering helps engineers understand and improve products when innovation is needed.

3. Engineering Design and Reverse Engineering

Engineering design is the process of planning, creating, and developing a product to meet a specific need. Reverse engineering supports engineering design by studying existing products to understand their structure, function, and performance. Together, they help engineers create better and improved products.

3.1 Engineering Design

Engineering design is a systematic approach used by engineers to convert ideas into functional products. It considers factors such as function, safety, cost, materials, and manufacturing methods. The goal of engineering design is to develop products that are efficient, reliable, and user-friendly.

3.2 Engineering Design Process

The engineering design process is a step-by-step method followed to develop a product.

Main steps include:

- ❖ Identifying the problem or need
- ❖ Collecting information and requirements
- ❖ Generating design ideas
- ❖ Selecting the best design
- ❖ Creating drawings and models
- ❖ Testing and improving the design

This process ensures that the final product meets the required performance and safety standards.

3.3 Types of Design

Original Design

Original design involves creating a completely new product or concept based on innovation and creativity, without directly copying existing designs.

Adaptive Design

Adaptive design modifies an existing design to suit a new application, environment, or requirement.

Variant Design

Variant design makes minor changes to an existing product, such as size, shape, or material, while keeping the basic design the same.

3.4 Motivations for Reverse Engineering

Cost Reduction

Reverse engineering helps in identifying cost-effective materials and manufacturing methods to reduce product cost.

Learning from Existing Products

Engineers study existing products to understand their design, working principle, and materials.

Competitive Analysis

Companies use reverse engineering to analyze competitor products and improve their own designs.

Obsolete Product Support

Reverse engineering helps in reproducing or repairing parts of products whose original design data is unavailable.

4. Uses, Benefits, and Risks of Reverse Engineering

Reverse engineering is widely used to study existing products, understand their design, and improve future designs. It plays an important role in learning, innovation, and product improvement.

4.1 Uses of Reverse Engineering

Reverse engineering is used to:

- ❖ Understand the working and structure of existing products
- ❖ Analyze competitor products
- ❖ Reproduce damaged or obsolete parts
- ❖ Improve product performance and design

4.2 Benefits of Reverse Engineering

Learning

Reverse engineering helps students and engineers learn design principles, materials, and manufacturing methods used in real products.

Innovation

By understanding existing designs, engineers can develop improved or innovative products.

Cost Saving

Reverse engineering helps in identifying low-cost materials and efficient manufacturing methods, reducing overall product cost.

4.3 Risks and Limitations

Legal Risks

Reverse engineering may violate patents or intellectual property laws if not done legally.

Ethical Risks

Copying designs without proper permission may be ethically unacceptable.

Incomplete Understanding

Some internal details or design intent may not be fully understood, leading to design errors.

5. Failure Analysis and Forensic Engineering

Failure analysis is an important engineering activity used to identify the causes of failure in mechanical components and systems. Forensic engineering applies failure analysis methods to investigate accidents and failures for legal and safety purposes.

5.1 Introduction to Failure Analysis

Failure analysis is the systematic study of why a component, material, or system has failed. The main objective is to prevent similar failures in the future by understanding the root cause.

5.2 Sources of Failures in Mechanical Systems

Failures in mechanical systems may occur due to:

- ❖ Poor design
- ❖ Manufacturing defects
- ❖ Improper material selection
- ❖ Overloading and misuse
- ❖ Wear, fatigue, and corrosion
- ❖ Inadequate maintenance

5.3 Mechanisms of Failure in Materials

Material failure mechanisms include:

- ❖ **Fracture:** Sudden breaking due to excessive stress
- ❖ **Fatigue:** Failure under repeated loading
- ❖ **Creep:** Deformation under constant load at high temperature
- ❖ **Corrosion:** Material damage due to chemical reactions
- ❖ **Wear:** Material loss due to friction

5.4 General Procedure for Failure Analysis

The general steps involved in failure analysis are:

- ❖ Visual inspection of the failed component
- ❖ Collection of background and service data
- ❖ Material and dimensional analysis
- ❖ Testing and examination
- ❖ Identification of root cause
- ❖ Preparation of a failure analysis report

5.5 Forensic Engineering

Forensic engineering involves the application of engineering principles to investigate failures related to accidents, property damage, or product malfunction. It is often used in legal investigations to determine responsibility.

6. Law and Ethics of Reverse Engineering

Law and ethics play an important role in guiding engineers to practice reverse engineering responsibly, legally, and professionally. While reverse engineering can support learning and innovation, it must be carried out within legal and ethical boundaries.

6.1 Importance of Ethics in Engineering

Ethics in engineering refers to the moral principles and professional values that guide engineers in their work. Ethical practices ensure public safety, honesty, fairness, and social responsibility in engineering activities.

6.2 Legal vs Ethical Issues

Legal issues are related to laws and regulations, while ethical issues are related to moral values and professional conduct. An action may be legal but still ethically questionable, and engineers must consider both aspects when performing reverse engineering.

6.3 Legality of Reverse Engineering

Reverse engineering is legal in certain cases such as education, research, interoperability, and repair. However, it may be illegal if it violates patents, copyrights, trade secrets, or license agreements.

6.4 Ethics of Reverse Engineering

Ethical reverse engineering focuses on learning, improvement, and innovation, not copying or plagiarism. Engineers should respect intellectual property rights and use reverse engineering only for legitimate and constructive purposes.

7. Basic Measurements

Measurement is a fundamental activity in engineering used to determine the accurate dimensions of components and products. It forms the basis for sketching, CAD modeling, manufacturing, and quality control. From simple instruments like steel scales to advanced systems such as coordinate measuring machines (CMM), measurement tools help engineers represent physical objects correctly and ensure proper fit, function, and reliability of engineering designs.

7.1 Purpose of Measurement in Engineering

Measurement is carried out to:

- ❖ Determine the actual size of a component
- ❖ Ensure dimensional accuracy
- ❖ Maintain interchangeability of parts
- ❖ Support sketching and CAD modeling
- ❖ Check manufacturing quality

Accurate measurement reduces errors and improves product reliability.

7.2 Classification of Measuring Instruments

Measuring instruments can be broadly classified into:

- ❖ Simple measuring instruments
- ❖ Precision measuring instruments
- ❖ Comparative measuring instruments
- ❖ Advanced measuring systems (CMM)

7.3 Simple Measuring Instruments

Steel Scale

A steel scale is the most basic measuring instrument used to measure length, width, and height. Figure 7.1 shows the steel rule.

Figure 7.1: Steel Rule

Applications:

- ❖ Measuring overall dimensions
- ❖ Quick inspection
- ❖ Preliminary measurements

Advantages:

- ❖ Easy to use
- ❖ Low cost
- ❖ Portable

Limitations:

- ❖ Low accuracy
- ❖ Parallax error

7.4 Precision Measuring Instruments

Precision measuring instruments are used when higher accuracy is required than what simple tools like steel scales can provide. They are essential in engineering to measure small dimensions accurately, ensuring proper fit, function, and quality of components. The most commonly used precision instruments are the Vernier Caliper and the Micrometer Screw Gauge.

7.4.1 Vernier Caliper

A vernier caliper is a precision measuring instrument used to measure external dimensions, internal dimensions, and depth with good accuracy. It consists of a main scale and a vernier scale, which together help in taking precise readings. Figure 7.2 shows the vernier caliper.
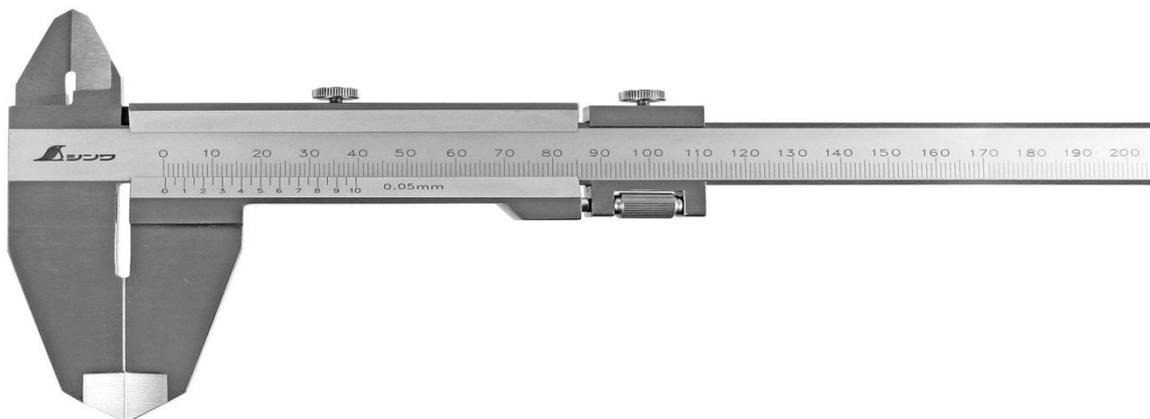
Figure 7.2: Vernier Caliper

Measurements Possible

- ❖ **External dimensions:** diameter of shafts, width of plates
- ❖ **Internal dimensions:** inner diameter of holes or pipes
- ❖ **Depth:** depth of holes, slots, or recesses

Least Count

❖ Commonly 0.02 mm or 0.01 mm, depending on the caliper
❖ Smaller least count means higher accuracy

Working Principle

The vernier scale slides over the main scale. The reading is obtained by adding the main scale reading and the vernier scale reading that coincides exactly with the main scale.

Applications

❖ Measuring shaft diameter
❖ Measuring hole diameter
❖ Measuring thickness of components
❖ Used widely in workshops, laboratories, and inspection departments

Advantages

❖ Measures three types of dimensions with one instrument
❖ Easy to use and portable
❖ More accurate than a steel scale

7.4.3 Micrometer Screw Gauge

A micrometer screw gauge is a highly precise measuring instrument used for very small dimensions. It works on the principle of a screw mechanism, where a small rotation of the screw produces a precise linear movement. Figure 7.3 shows micrometer screw gauge.
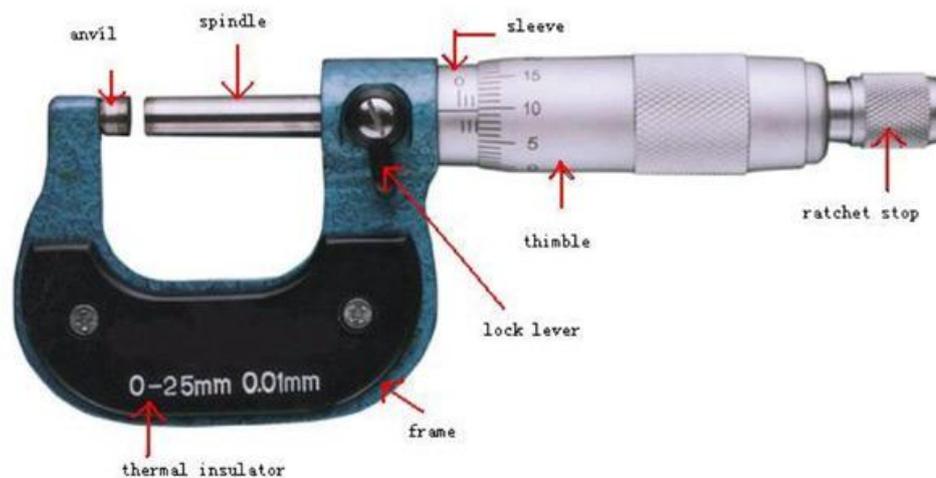
Figure 7.3: Micrometer Screw Gauge

Types of Micrometer

❖ **Outside Micrometer**

Used to measure external dimensions such as diameter or thickness.

❖ **Inside Micrometer**

Used to measure internal dimensions like the diameter of holes.

❖ **Depth Micrometer**

Used to measure the depth of holes, slots, and recesses.

Least Count

❖ Commonly 0.01 mm
❖ High-precision micrometers can measure up to 0.001 mm

Working Principle

When the thimble is rotated, the spindle moves linearly. The measurement is obtained by adding the sleeve (main scale) reading and the thimble scale reading.

Applications

❖ Measuring wire diameter
❖ Measuring thickness of thin sheets
❖ Inspecting precision components such as bearings and pins

Advantages

❖ Very high accuracy
❖ Suitable for precision engineering work
❖ Reliable and repeatable measurements

7.4.4 Comparison

Instrument	Accuracy	Typical Use
Vernier Caliper	Medium–High	General precision measurement
Micrometer Screw Gauge	Very High	Fine and precision measurement

7.5 Comparative Measuring Instruments

Comparative measuring instruments are used to compare the dimension of a component with a standard value rather than measuring the exact size directly. These instruments are highly useful for detecting small variations, misalignment, and surface errors. Two commonly used comparative instruments are the Dial Indicator and the Height Gauge.

7.5.1 Dial Indicator

A dial indicator is a precision instrument used to measure small linear displacements or deviations from a reference position. It does not give the absolute dimension but indicates how much a component deviates from the standard or desired position. Figure 7.4 shows Dial indicator with magnetic stand.

Figure 7.4: Dial Indicator with magnetic stand

Working Principle

The dial indicator works using a rack and pinion mechanism. When the contact tip touches the component surface, small movements are transmitted to the pointer on the dial, which shows the amount of deviation.

Applications

- ❖ **Checking runout:** Detects eccentricity in rotating shafts
- ❖ **Alignment of shafts:** Ensures proper alignment of machine parts
- ❖ **Flatness checking:** Identifies uneven surfaces
- ❖ **Parallelism checking:** Ensures two surfaces are parallel

Advantages

- ❖ High sensitivity, capable of detecting very small deviations
- ❖ Easy to read
- ❖ Widely used in inspection and machine setup

7.5.2 Height Gauge

A height gauge is a precision measuring instrument used in combination with a surface plate to measure vertical dimensions accurately. It is commonly used in layout and inspection work. Figure 7.5 shows Height gauge.

Figure 7.5: Height Gauge

Working Principle

The height gauge moves vertically along a rigid beam, and the measurement is taken from the surface plate reference. The reading is obtained using a vernier scale, digital display, or dial.

Applications

- ❖ **Height measurement:** Measures vertical distance of components
- ❖ **Scribing reference lines:** Used to mark layout lines accurately
- ❖ **Inspection work:** Checks step heights and feature locations

Advantages

- ❖ High accuracy for vertical measurements

❖ Provides a stable reference surface
❖ Useful for marking and inspection

7.5.3 Difference between Dial Indicator and Height Gauge

Instrument	Purpose	Type of Measurement
Dial Indicator	Detects deviation	Comparative
Height Gauge	Measures vertical dimensions	Direct / comparative

7.6 Profile Projector (Optical Comparator)

A profile projector, also known as an optical comparator, is an advanced measuring instrument used to inspect and measure the profile of mechanical components. It works by magnifying the outline (profile) of a component and projecting it onto a screen for comparison with standard templates or measurement scales. Figure 7.6 shows a Profile projector (optical comparator)
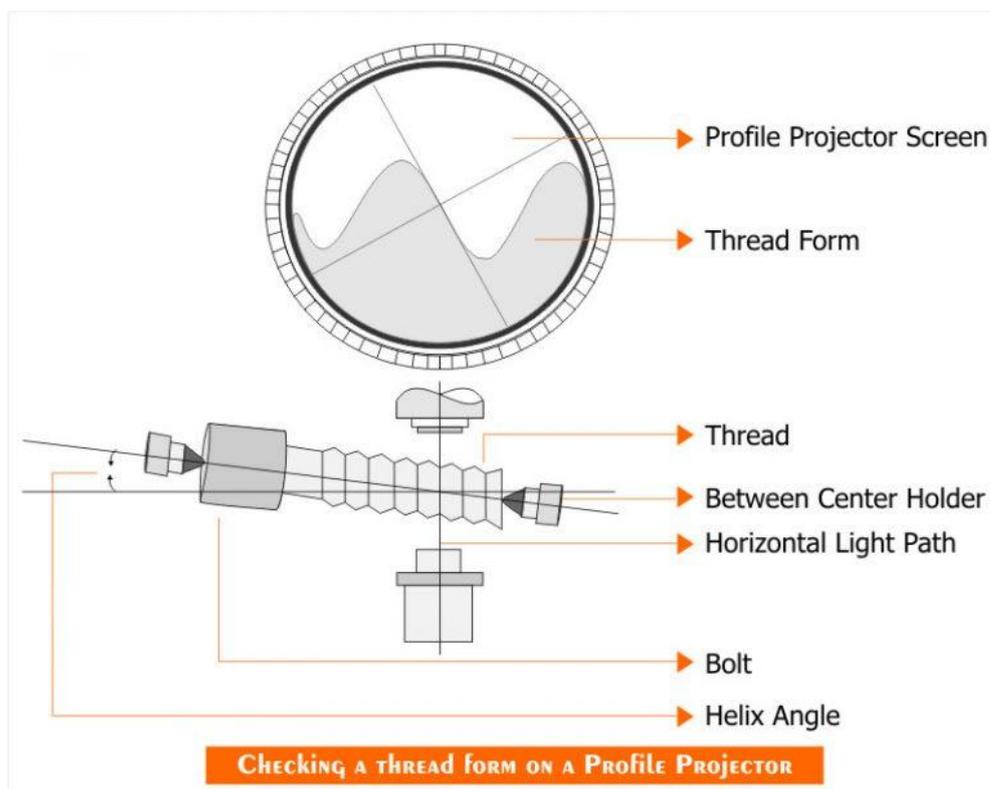

Figure 7.6: Profile Projector

Working Principle

The component to be measured is placed on the projector stage and illuminated by a strong light source. The light passes around the component, creating a magnified shadow image on the screen. This projected image can be measured directly using screen scales, overlays, or digital readouts.

Applications

- ❖ **Thread Measurement:**

Used to inspect thread angle, pitch, and profile accuracy.

- ❖ **Gear Profile Inspection:**

Checks the involute profile of gears to ensure smooth and accurate power transmission.

- ❖ **Complex Shape Measurement:**

Measures intricate shapes such as cams, slots, radii, and contours that are difficult to measure using contact instruments.

Advantages

- ❖ Non-contact Measurement:

No physical contact with the component, making it ideal for delicate or soft materials.

- ❖ High Accuracy:

Provides precise measurement due to magnification and optical clarity.

- ❖ Easy Visualization:

Enlarged image makes inspection simple and clear.

- ❖ Suitable for Small Components:

Ideal for very small or thin parts.

Limitations

- ❖ High initial cost
- ❖ Requires controlled lighting conditions
- ❖ Limited to profile and surface measurement

Importance in Engineering Practice

Profile projectors are widely used in quality control, inspection laboratories, and reverse engineering. They help engineers verify design accuracy and ensure that manufactured components meet required specifications.

7.7 Coordinate Measuring Machine (CMM)

A Coordinate Measuring Machine (CMM) is an advanced and highly accurate measuring system used to measure the geometric characteristics of components in three dimensions (X, Y, and Z). It is widely used in modern manufacturing and inspection environments to measure complex shapes and profiles that cannot be measured easily using conventional instruments. Figure 7.7 shows the Coordinate Measuring Machine.

Figure 7.7: Coordinate Measuring Machine (CMM)

Working Principle

The working of a CMM is based on the coordinate system. The component to be measured is placed on the CMM table, and a probe mounted on a movable arm touches the surface of the component at various points.

Each contact point generates precise X, Y, and Z coordinate values, which are recorded and processed by a computer. These coordinates are then used to calculate dimensions, form errors, and geometric tolerances.

CMMs can operate in:

- ❖ Manual mode (operator controlled)
- ❖ Semi-automatic mode
- ❖ Fully automatic (CNC) mode

Types of Probes

- ❖ **Touch trigger probe** – records a point when the probe touches the surface
- ❖ **Scanning probe** – continuously scans the surface
- ❖ **Optical or laser probe** – non-contact measurement

Applications

❖ **Inspection of Complex Parts:**

Used to measure intricate components such as turbine blades, engine blocks, molds, and castings.

❖ **Quality Control:**

Ensures components meet design specifications and tolerances.

❖ **Reverse Engineering:**

Captures the geometry of existing parts to recreate CAD models.

❖ **CAD Comparison:**

Compares measured data with CAD models to detect deviations and errors.

Advantages

❖ **Very High Accuracy:**

Capable of measuring with micrometer-level precision.

❖ **Measures Complex Shapes:**

Ideal for free-form surfaces and 3D geometries.

❖ **Computer-Based Data Output:**

Generates digital reports, graphs, and inspection results.

❖ **Repeatability and Reliability:**

Ensures consistent and accurate measurements.

Limitations

❖ **High Cost:**

Expensive to purchase, install, and maintain.

❖ **Requires Skilled Operator:**

Proper training is necessary for operation and data interpretation.

❖ **Environmental Sensitivity:**

Accuracy can be affected by temperature and vibration.

Importance in Modern Engineering

CMMs play a crucial role in precision manufacturing, aerospace, automotive, and medical industries. They support high-quality production, reduce errors, and enhance product reliability. In reverse engineering, CMMs help convert physical components into accurate digital models.

7.8 Measurement Procedure

A systematic measurement procedure helps reduce errors and improves reliability of results. The general steps involved in measurement are explained below.

1. Clean the Component

Before measurement, the component should be cleaned properly to remove dust, oil, grease, or burrs. Dirt on the surface can affect contact between the measuring instrument and the component, leading to incorrect readings.

Purpose:

- ❖ Ensures true contact
- ❖ Improves accuracy

2. Select the Appropriate Instrument

The measuring instrument should be chosen based on:

- ❖ Size of the component
- ❖ Required accuracy

Type of dimension (length, diameter, depth, etc.)

For example:

- ❖ Steel scale → rough measurement
- ❖ Vernier caliper → moderate precision
- ❖ Micrometer → high precision

Purpose:

- ❖ Avoids under- or over-precision
- ❖ Ensures correct measurement method

3. Check Zero Error

Before taking measurements, the instrument should be checked for zero error. This means ensuring that the instrument shows zero when no measurement is taken.

- ❖ If zero error is present, it must be noted and corrected.
- ❖ Zero error can be positive or negative.

Purpose:

- ❖ Prevents systematic error
- ❖ Improves measurement reliability

4. Measure Carefully

The component should be measured with proper alignment and steady handling. Excessive force should be avoided, especially with precision instruments like micrometers.

Guidelines:

- ❖ Hold the instrument perpendicular to the surface
- ❖ Avoid tilting or misalignment
- ❖ Apply uniform pressure

Purpose:

- ❖ Ensures true dimension measurement
- ❖ Reduces handling errors

5. Record Readings in a Table

All measured values should be recorded systematically in a measurement and observation table. Units must be clearly mentioned, and readings should be written immediately to avoid mistakes.

Purpose:

- ❖ Maintains proper documentation
- ❖ Helps in sketching and CAD modeling

6. Repeat for Accuracy

Measurements should be taken two or three times, and the average value should be considered as the final reading. This helps in identifying random errors.

Purpose:

- ❖ Improves accuracy
- ❖ Increases confidence in results

7.9 Measurement Errors

Measurement errors are deviations between the true value and the measured value. Understanding these errors helps in minimizing them.

1. Instrument Error

Instrument error occurs due to:

- ❖ Poor calibration
- ❖ Worn-out or damaged instruments
- ❖ Manufacturing defects

Example: Zero error in vernier caliper.

2. Parallax Error

Parallax error occurs when the reading is taken from an incorrect angle, especially in scale-based instruments.

Prevention:

- ❖ Keep eye directly above the scale
- ❖ Use digital instruments where possible

3. Human Error

Human error occurs due to:

- ❖ Improper handling
- ❖ Misreading scales
- ❖ Recording incorrect values

Prevention:

- ❖ Careful observation
- ❖ Proper training
- ❖ Rechecking readings

4. Environmental Effects

Environmental factors such as:

- ❖ Temperature variation
- ❖ Humidity
- ❖ Vibrations

Can affect measurement accuracy, especially in precision instruments.

Prevention:

- ❖ Controlled environment
- ❖ Allow components to reach room temperature

7.10 Reduction of Measurement Errors

Measurement errors can be reduced by:

- ❖ Proper handling of instruments
- ❖ Regular calibration
- ❖ Correct selection of measuring tools
- ❖ Careful reading and recording
- ❖ Following standard measurement procedures

8. Sketching

Sketching is the process of freehand drawing of engineering components to represent their shape, size, features, and relationships clearly. It is one of the most important skills for engineers, especially during measurement, reverse engineering, and CAD modeling. Sketching helps convert a physical object or idea into a visual form quickly and effectively without the need for software.

8.1 Purpose of Sketching in Engineering

Sketching is used to:

- ❖ Visualize the shape and geometry of components
- ❖ Communicate ideas clearly
- ❖ Plan CAD modeling steps
- ❖ Record measured dimensions
- ❖ Understand assembly and part relationships

8.2 Types of Sketches Used in Engineering

8.2.1. Orthographic Sketching

Orthographic sketching represents an object using multiple 2D views, usually:

- ❖ Front view
- ❖ Top view
- ❖ Side view

These views show the true size and shape of the object. Orthographic sketches are mainly used for dimensioning and manufacturing understanding. Figure 8.1 shows orthographic projection and isometric view of a simple solid.
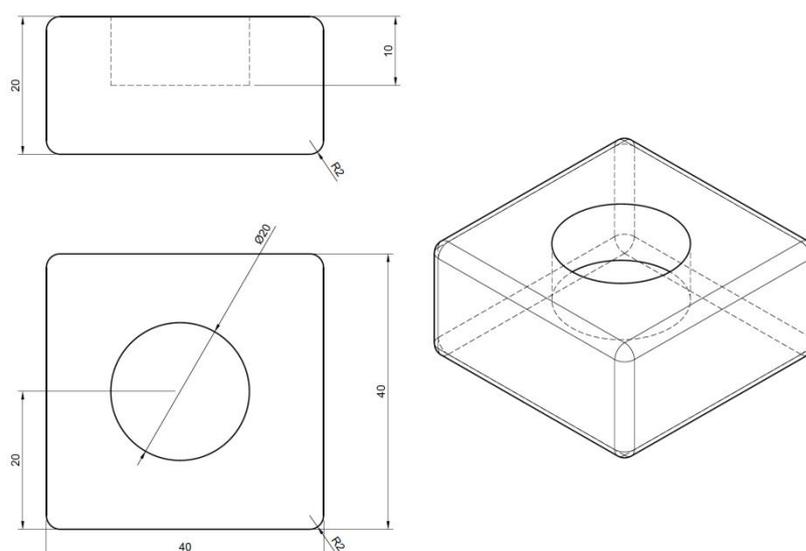

Figure 8.1: Isometric and Orthographic View of a Simple Solid

8.2.2. Isometric Sketching

Isometric sketching shows a three-dimensional view of the object on a two-dimensional plane. It helps in understanding the overall appearance of the component.

Characteristics:

- ❖ All three axes are drawn at 120°
- ❖ Length, width, and height are visible
- ❖ Isometric sketches are commonly used for visual explanation and assembly understanding.

8.2.3. Sectional Sketching

Sectional sketches are drawn by imagining a cut through the object to reveal internal features.

Used to show:

- ❖ Holes
- ❖ Cavities
- ❖ Slots
- ❖ Threads

Sectional sketches improve clarity when internal details are not visible from outside. Figure 8.2 shows the sectional view of a simple mechanical object.
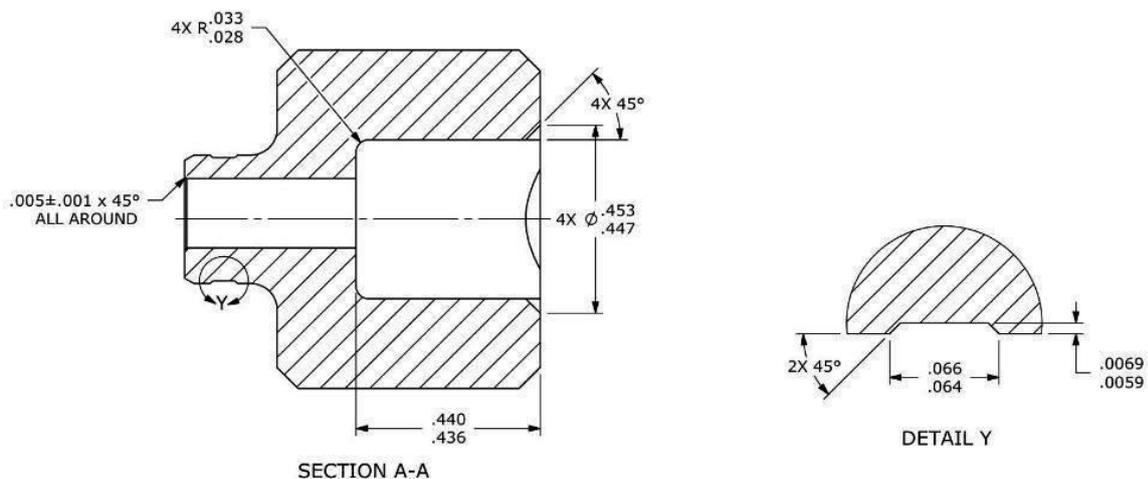

Figure 8.2: Sectional View of a Simple Solid

8.3 Basic Rules of Sketching

To create a clear and correct sketch, the following rules should be followed:

- ❖ Use light construction lines initially
- ❖ Maintain proper proportions
- ❖ Draw straight lines confidently
- ❖ Avoid excessive erasing

- ❖ Use dark final outlines
- ❖ Keep sketches neat and readable

Accuracy in proportion is more important than artistic perfection.

8.4 Dimensioning in Sketching

Dimensioning is the process of adding measured values to the sketch.

Guidelines:

- ❖ Place dimensions clearly
- ❖ Avoid overlapping dimension lines
- ❖ Use arrowheads properly
- ❖ Mention units (mm)

Correct dimensioning ensures the sketch can be accurately converted into a CAD model.

8.5 Sketching Procedure (Step-by-Step)

- ❖ Observe the Component

  Identify overall shape and features

- ❖ Select Appropriate Views

  Decide front, top, side, or sectional views

- ❖ Draw Light Outline

  Maintain proportion

- ❖ Add Details

  Holes, slots, fillets, threads

- ❖ Add Dimensions

  Based on measured values

- ❖ Label Important Features

8.6 Applications of Sketching

Sketching is used in:

- ❖ Reverse engineering
- ❖ Product development
- ❖ Failure analysis
- ❖ Maintenance and repair work
- ❖ Manufacturing planning

8.7 Advantages of Sketching

- ❖ Quick and flexible
- ❖ No software required
- ❖ Enhances visualization
- ❖ Improves engineering thinking

8.8 Limitations of Sketching

- ❖ Less accurate than CAD
- ❖ Depends on individual skill
- ❖ Not suitable for final production drawings

9. Computer Aided Design (CAD)

Computer Aided Design (CAD) refers to the use of computer systems and software to assist in the creation, modification, analysis, and optimization of engineering drawings and designs. CAD has become an essential tool in modern engineering, architecture, manufacturing, and design fields due to its accuracy, efficiency, and ability to handle complex geometries.

In traditional manual drafting, drawings are prepared using drawing boards, instruments, and sheets, which is time-consuming and prone to errors. CAD overcomes these limitations by allowing designers to create precise drawings digitally with greater speed and flexibility. Any modification in a CAD drawing can be done easily without redrawing the entire figure, which significantly improves productivity.

CAD software enables the creation of 2D drawings such as orthographic views, sections, and dimensioned layouts, as well as 3D models that represent real-world objects. These 3D models help engineers visualize the design, detect errors, and understand assembly relationships before actual manufacturing.

9.1 Features of CAD

- ❖ High accuracy and precision in drawings
- ❖ Easy editing, scaling, and modification
- ❖ Standardization of drawings using layers, templates, and blocks
- ❖ Efficient storage, retrieval, and sharing of drawings
- ❖ Integration with manufacturing and analysis tools

9.2 Other Drawing Software

Apart from AutoCAD, several other drawing and design software tools are widely used in engineering and design applications.

9.2.1. SolidWorks

SolidWorks is a popular 3D parametric modeling software used mainly for mechanical design. It allows users to create parts, assemblies, and detailed engineering drawings. SolidWorks is widely used in product design, simulation, and manufacturing industries.

9.2.2. CATIA

CATIA is an advanced CAD/CAM/CAE software extensively used in aerospace, automobile, and heavy engineering industries. It supports complex surface modeling and large-scale assemblies and is suitable for high-end industrial applications.

9.2.3. Creo (Pro/ENGINEER)

Creo is a powerful 3D CAD software used for product design and development. It supports parametric modeling, simulation, and manufacturing processes, making it suitable for mechanical engineering applications.

9.2.4. Fusion 360

Fusion 360 is cloud-based CAD software that integrates design, simulation, and manufacturing. It is widely used for product design, prototyping, and CNC machining, especially in startups and educational institutions.

9.2.5. SketchUp

SketchUp is an easy-to-use 3D modeling software mainly used for architectural drawings and interior design. It is suitable for beginners and provides quick visualization of structures and layouts.

9.2.6. FreeCAD

FreeCAD is an open-source parametric CAD software used for basic mechanical and architectural designs. It is free to use and suitable for students and educational purposes.

9.3 CAD Commands – 2D and 3D

A. Basic & General Commands

Command	Explanation
NEW	Creates a new drawing file
OPEN	Opens an existing drawing
SAVE	Saves the current drawing
SAVEAS	Saves the drawing with a new name
CLOSE	Closes the current drawing
EXIT / QUIT	Exits AutoCAD
UNDO	Reverses the last action
REDO	Restores the undone action
REGEN	Regenerates the drawing
SNAP	Controls cursor movement
GRID	Displays grid on the screen
OSNAP	Enables object snapping
ORTHO	Restricts cursor movement to horizontal/vertical
POLAR	Enables polar tracking

DSETTINGS   Drafting settings

B. Draw Commands (2D)

Command	Explanation
LINE	Draws straight line segments
POLYLINE (PL)	Draws connected line segments
CIRCLE	Draws a circle
ARC	Draws an arc
RECTANGLE	Draws a rectangle
POLYGON	Draws regular polygons
ELLIPSE	Draws an ellipse
SPLINE	Draws smooth curves
POINT	Creates a point
DONUT	Draws filled rings
XLINE	Draws infinite construction lines
RAY	Draws semi-infinite lines
MULTILINE (MLINE)	Draws multiple parallel lines

C. Modify Commands (2D)

Command	Explanation
MOVE	Moves objects
COPY	Copies objects
ROTATE	Rotates objects
SCALE	Scales objects
MIRROR	Creates mirror image
OFFSET	Creates parallel objects
TRIM	Trims objects
EXTEND	Extends objects

FILLET	Rounds corners
CHAMFER	Bevels corners
ARRAY	Creates multiple copies
STRETCH	Stretches objects
BREAK	Breaks objects
JOIN	Joins objects
EXPLODE	Breaks compound objects
ALIGN	Aligns objects
LENGTHEN	Changes length

D. Annotation Commands

Command	Explanation
TEXT	Single-line text
MTEXT	Multi-line text
DIMLINEAR	Linear dimension
DIMALIGNED	Aligned dimension
DIMRADIUS	Radius dimension
DIMDIAMETER	Diameter dimension
DIMANGULAR	Angular dimension
DIMSTYLE	Sets dimension style
LEADER	Leader annotation
MLEADER	Multi-leader
TOLERANCE	Geometric tolerances

E. Layer & Property Commands

Command	Explanation
LAYER (LA)	Layer management
PROPERTIES (PR)	Object properties

MATCHPROP	Matches properties
COLOR	Sets object color
LINETYPE	Sets line type
LINEWEIGHT	Sets line thickness

F. Block & Reference Commands

Command	Explanation
BLOCK	Creates a block
INSERT	Inserts a block
WBLOCK	Writes block to file
BEDIT	Block editor
XREF	External reference
REFEDIT	Edit reference

G. Selection & View Commands

Command	Explanation
ZOOM	Zooms view
PAN	Moves view
VIEW	Saves named views
REGENALL	Regenerates all viewports
UCS	User coordinate system
PLAN	Sets view orientation

H. Hatch & Region Commands

Command	Explanation
HATCH	Applies hatch patterns
BHATCH	Boundary hatch
GRADIENT	Gradient fill

| REGION | Converts closed shape to region |
| BOUNDARY | Creates boundary |

I. 3D Drawing Commands

Command	Explanation
BOX	Creates 3D box
CYLINDER	Creates cylinder
CONE	Creates cone
SPHERE	Creates sphere
TORUS	Creates torus
WEDGE	Creates wedge
PYRAMID	Creates pyramid
POLYSOLID	Creates wall-like solids

J. 3D Modify Commands

Command	Explanation
EXTRUDE	Converts 2D to 3D
PRESSPULL	Extrudes by selecting faces
REVOLVE	Revolves object
SWEEP	Sweeps profile
LOFT	Creates solid between profiles
UNION	Combines solids
SUBTRACT	Removes solid
INTERSECT	Common volume
FILLETEDGE	Rounds edges
CHAMFEREDGE	Chamfers edges
SHELL	Hollows solid

K. 3D Surface Commands

Command	Explanation
SURFACE	Creates surface
SURFEXTRUDE	Extruded surface
SURFREVOLVE	Revolved surface
SURFSWEEP	Swept surface
SURFLOFT	Lofted surface
THICKEN	Converts surface to solid

L. 3D View & Visual Commands

Command	Explanation
3DORBIT	Rotates view
VPOINT	Sets viewpoint
VISUALSTYLES	Shaded / wireframe
SHADEMODE	Shading mode
HIDE	Hides hidden lines
RENDER	Renders image
LIGHT	Adds lighting
MATERIALS	Applies materials

M. Solid Editing Commands

Command	Explanation
SOLIDEDIT	Edits solid faces
SLICE	Cuts solid
SECTION	Creates section
INTERFERE	Checks interference

N. Output & Plotting Commands

Command	Explanation
LAYOUT	Paper space
PAGESETUP	Page settings
PLOT	Prints drawing
EXPORT	Exports file
PDFEXPORT	Exports to PDF

O. Inquiry Commands

Command	Explanation
DIST	Measures distance
AREA	Calculates area
LIST	Lists object properties
ID	Displays coordinates
MASSPROP	Mass properties

10. Brainstorming in Engineering Design

Brainstorming is a creative problem-solving technique in which a group or individual generates a large number of ideas in a short time without immediate evaluation or criticism. The main objective is to explore all possible solutions to a design problem.

10.1 Objectives of Brainstorming

- ❖ To encourage creative and innovative thinking
- ❖ To generate multiple alternative design ideas
- ❖ To avoid fixation on a single solution
- ❖ To promote teamwork and idea sharing
- ❖ To improve problem understanding

10.2 Types of Brainstorming Techniques

1. Individual Brainstorming

Ideas are generated independently by an individual. This method is useful when quick idea generation is required or when working alone.

2. Group Brainstorming

A group of members collectively generate ideas. Each participant builds upon others' ideas, leading to diverse and improved design alternatives.

3. Mind Mapping

A visual brainstorming technique where ideas are connected to a central problem using branches. It helps in organizing thoughts and identifying relationships between ideas.

4. Reverse Brainstorming

Instead of finding solutions, the focus is on identifying possible problems or failures. These are then reversed to develop better design solutions.

10.3 Rules of Effective Brainstorming

- ❖ No criticism or judgment during idea generation
- ❖ Encourage free and unconventional ideas
- ❖ Quantity of ideas is more important than quality initially
- ❖ Combine and improve ideas after brainstorming

10.4 Application of Brainstorming in Design

- ❖ Identifying possible product concepts
- ❖ Exploring different shapes, mechanisms, and materials
- ❖ Developing multiple solutions for a given engineering problem
- ❖ Selecting optimal designs based on feasibility and requirements

10.5 Integration of Brainstorming and Sketching

Brainstorming and sketching are most effective when used together. Brainstorming generates multiple ideas, while sketching helps visualize and refine those ideas. Each brainstormed idea can be quickly converted into a sketch, allowing comparison and evaluation of alternative designs.

10.6 Steps to Create Alternative Design Ideas

❖ Identify the design problem clearly
❖ Conduct brainstorming to generate multiple ideas
❖ Select promising ideas
❖ Prepare rough sketches for each idea
❖ Compare alternatives based on functionality, cost, and feasibility
❖ Refine the best design concept

10.7 Educational and Industrial Relevance

The ability to apply brainstorming and sketching techniques is essential in product design, machine design, and manufacturing industries. Engineers use these techniques to develop innovative products, improve existing designs, and solve complex engineering problems efficiently.

11. Prototyping Skills for Manufacturing and Assembling a Functional Prototype

Prototyping is a critical phase in the engineering design and product development process. It involves transforming a conceptual design into a physical model that demonstrates the functionality, form, and performance of the product. Prototyping helps engineers validate design ideas, identify design flaws, and improve product quality before mass production.

This course outcome focuses on developing students' ability to apply practical prototyping skills to manufacture and assemble a functional prototype using conventional tools or advanced technologies such as 3D printing.

11.1 Objectives of Prototyping

- ❖ To verify design feasibility and functionality
- ❖ To identify errors and improve design quality
- ❖ To evaluate ergonomics and aesthetics
- ❖ To test assembly and fit of components
- ❖ To reduce development cost and time

11.2 Types of Prototypes

- ❖ **Proof-of-Concept Prototype**

  Used to validate basic working principles and functionality.

- ❖ **Functional Prototype**

  Demonstrates actual working conditions and performance of the design.

- ❖ **Visual Prototype**

  Focuses on appearance, shape, and aesthetics rather than functionality.

- ❖ **Rapid Prototype**

  Produced quickly using additive manufacturing techniques such as 3D printing.

11.3 Prototyping Methods

A. Conventional Tool-Based Prototyping

This method involves manufacturing components using hand tools and basic workshop machines.

Common Tools Used

- ❖ Cutting tools (hacksaw, cutter)
- ❖ Shaping tools (files, chisels)
- ❖ Drilling tools (hand drill, drilling machine)

- ❖ Fastening tools (spanners, screwdrivers)
- ❖ Measuring tools (vernier caliper, steel rule)

Process Steps

- ❖ Interpretation of design drawings
- ❖ Selection of suitable materials
- ❖ Cutting and shaping components
- ❖ Drilling and finishing operations
- ❖ Assembly of parts using fasteners or joints

B. 3D Printing-Based Prototyping

3D printing, also known as additive manufacturing, builds components layer by layer from digital 3D models.

Common 3D Printing Technologies

- ❖ Fused Deposition Modeling (FDM)
- ❖ Stereolithography (SLA)
- ❖ Selective Laser Sintering (SLS)

Materials Used

- ❖ PLA (Polylactic Acid)
- ❖ ABS (Acrylonitrile Butadiene Styrene)
- ❖ Nylon
- ❖ Resin

Process Steps

- ❖ Creation of 3D CAD model
- ❖ Conversion to STL file format
- ❖ Slicing and setting print parameters
- ❖ Printing the component
- ❖ Post-processing and finishing

11.4 Assembly of the Functional Prototype

Assembly involves joining individual components to form a complete working model.

Assembly Methods

- ❖ Mechanical fastening (nuts, bolts, screws)
- ❖ Snap-fit joints
- ❖ Adhesive bonding
- ❖ Press fitting

Assembly Considerations

- ❖ Proper alignment of parts
- ❖ Ease of assembly and disassembly
- ❖ Strength and stability of joints
- ❖ Functional movement of components

11.5 Testing and Validation of Prototype

After assembly, the prototype is tested to ensure functionality.

Testing Parameters

- ❖ Dimensional accuracy
- ❖ Mechanical strength
- ❖ Fit and tolerance
- ❖ Operational performance
- ❖ Safety and reliability

Feedback obtained from testing is used to modify and improve the prototype.

11.6 Skills Developed Through Prototyping

- ❖ Practical manufacturing skills
- ❖ Material selection knowledge
- ❖ Assembly and fitting techniques
- ❖ Problem-solving and troubleshooting
- ❖ Teamwork and project management

Expt. No.: 1

Date:

Study of Product Development, Product Life Cycle, and Reverse Engineering of a Ceiling Fan

Aim

To study a ceiling fan in order to understand product development, product life cycle, and reverse engineering concepts.

Outcome

- ❖ After completing this experiment, students will be able to:
- ❖ Understand the product development process of a ceiling fan
- ❖ Identify the product life cycle stage of a ceiling fan
- ❖ Understand the design and working of components through reverse engineering

Apparatus / Tools Required

- ❖ Ceiling fan
- ❖ Screwdriver set, Spanner set
- ❖ Measuring scale / Vernier caliper

Theory

A ceiling fan is a commonly used electro-mechanical household product designed to circulate air efficiently. It consists of major components such as an electric motor, blades, bearings, capacitor, and mounting arrangement. Studying this product helps in understanding how engineering products are developed, how they perform in the market over time, and how their design can be analyzed through reverse engineering. Figure Ex.1 shows the exploded view of ceiling fan showing major components.

Procedure

Part A: Product Development Study

1. Identify the need fulfilled by the ceiling fan

The primary need fulfilled by a ceiling fan is air circulation and thermal comfort. It helps circulate air in a closed space, reduces the feeling of heat, and improves comfort at low energy consumption. Ceiling fans are widely used because they are cost-effective, energy-efficient, and suitable for all climatic conditions.
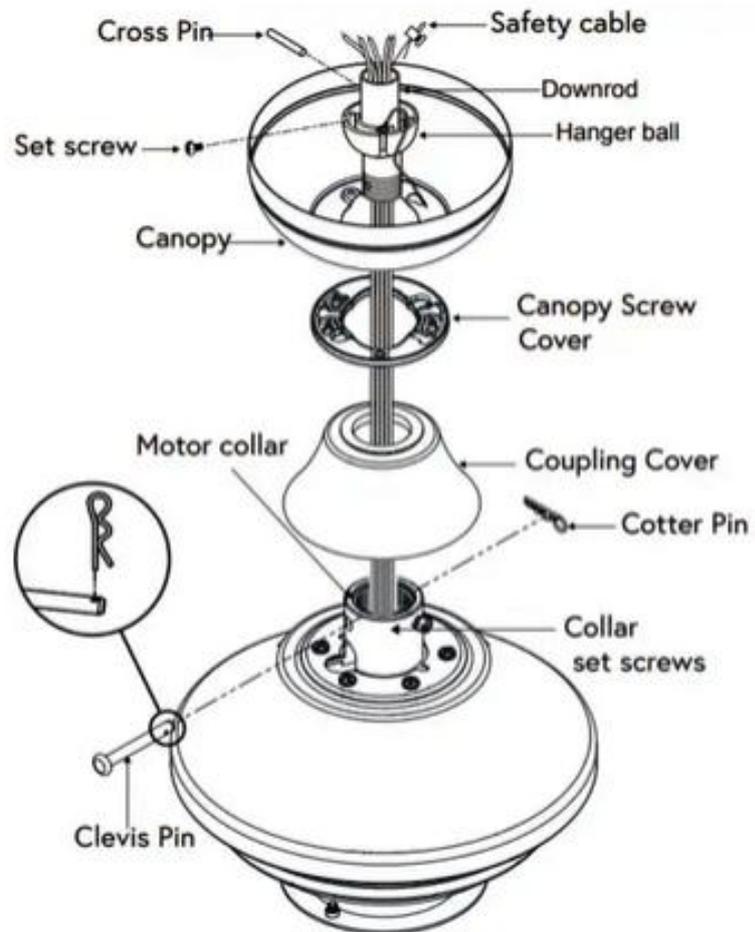
Figure Ex.1: Exploded view of ceiling fan showing major components

2. List the major components and their purpose (Ceiling Fan)

Electric Motor:

Converts electrical energy into mechanical energy to rotate the fan blades.

Fan Blades:

Push air downward to create airflow and provide cooling effect.

Bearings:

Support smooth and frictionless rotation of the motor shaft.

Capacitor:

Provides starting torque and helps in smooth operation of the single-phase motor.

Mounting System (Down rod and canopy):

Fixes the fan securely to the ceiling and supports its weight.

Each component is designed to work together to ensure safe, efficient, and continuous operation of the ceiling fan.

3. Design considerations in ceiling fan development

Safety:

Proper insulation of wires, strong mounting system, and balanced blades prevent accidents.

Efficiency:

Aerodynamic blade design and energy-efficient motors reduce power consumption.

Noise Reduction:

Use of high-quality bearings and balanced blades minimizes vibration and noise.

Durability:

Corrosion-resistant materials and robust motor construction increase product life.

Part B: Product Life Cycle (PLC) Study

1. Identify the PLC stage of the ceiling fan
The ceiling fan is currently in the maturity stage of the product life cycle.

2. Observe design changes over time

Over the years, ceiling fan designs have improved:
- ❖ Blade shapes have become more aerodynamic for better airflow.
- ❖ Traditional AC motors are being replaced by BLDC motors for energy savings.
- ❖ Improved materials reduce weight and noise.
- ❖ Smart features like remote control and timers are added.
- ❖ These changes help the product remain competitive in the market.

Observation

Sl. No.	Component Name	Dimension Measured	Measured Value (mm)	Tool Used
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				

From the measured values, it is observed that the dimensions of the fan blades, motor housing, shaft, and bearings etc. are carefully designed to ensure balanced rotation, smooth operation, and efficient air circulation. Proper sizing of components also contributes to noise reduction, safety, and durability of the ceiling fan.

3. Product Life Cycle Curve (Description)

The product life cycle consists of:

Introduction: Initial launch of the ceiling fan

Growth: Rapid increase in usage

Maturity: Stable demand and wide usage

Decline: Possible in future due to new cooling technologies

Inference

The ceiling fan is in the maturity stage of the product life cycle due to its widespread use, stable demand, and continuous design improvements.

Part C: Reverse Engineering Study

1. Study of exploded view

The exploded view of the ceiling fan (Figure Ex.1) shows how individual components are arranged and assembled. It helps in understanding the internal structure without physically disassembling the product.

2. Identification of components and their function

Motor housing: Protects internal motor components

Rotor and stator: Produce rotational motion

Shaft: Transfers motion from motor to blades

Blades: Move air

Bearings: Reduce friction

Capacitor: Assists motor starting

Each part has a specific function and contributes to the overall working.

3. Assembly sequence and working principle

The motor components are assembled inside the housing, followed by mounting of the shaft and bearings. The blades are fixed to the rotor assembly, and the entire unit is mounted to the ceiling. When power is supplied, the capacitor helps start the motor, the motor rotates the shaft, and the blades rotate to circulate air.

Result

The ceiling fan was successfully studied to understand its product development process, product life cycle stage, and reverse engineering aspects. The measurement and observation data provided practical insight into the dimensional design of major components, supporting the understanding of assembly, working principle, and performance. This experiment helped in relating theoretical concepts with a real-life household product.
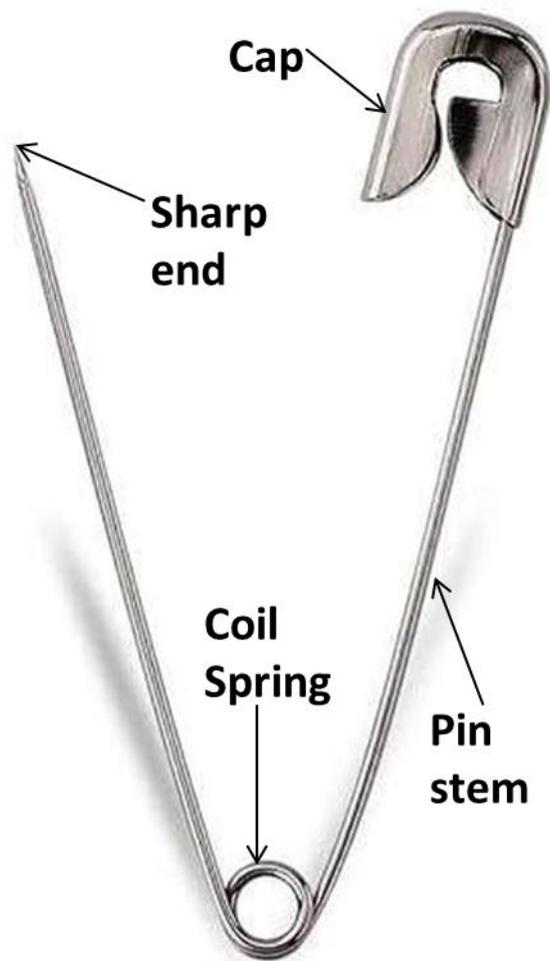
Figure Ex.2: Safety Pin

Expt. No.: 2

Date:

Reverse Engineering and Re-Engineering of a Safety Pin

Aim

To study a safety pin through reverse engineering to understand its design and working, and to propose re-engineering improvements for better performance and usability.

Outcome

After completing this experiment, the student will be able to:

- ❖ Identify the components of a safety pin
- ❖ Understand its working principle
- ❖ Measure and analyze its design
- ❖ Suggest simple improvements through re-engineering

Apparatus / Tools Required

- ❖ Safety pin
- ❖ Scale
- ❖ Vernier caliper
- ❖ Micrometer

Theory

Reverse Engineering

Reverse engineering is the process of studying an existing product to understand how it is designed and how it works without knowing the original design details.

Re-Engineering

Re-engineering involves modifying or improving an existing product to enhance its performance, safety, durability, or cost effectiveness.

In this experiment, a safety pin, a common daily-use product, is chosen because of its simple design and clear working mechanism. Figure Ex.2 shows the safety pin used for the experiment.

Observation

Sl. No.	Parameter	Measured Value (mm)	Tool Used
1	Overall length		
2	Wire diameter		
3	Length of spring coil		
4	Number of coils		

Procedure

Part A: Reverse Engineering of Safety Pin

1. Observe the safety pin carefully.
2. Identify the major parts
   ❖ Pin (sharp end)
   ❖ Spring / coil
   ❖ Clasp (locking head)
3. Study how the spring provides tension.
4. Understand the locking mechanism that prevents accidental opening.
5. Note the material used (usually steel or stainless steel).

Part B: Measurement and Observation

   ❖ Measure the overall length of the safety pin.
   ❖ Measure the wire diameter.
   ❖ Observe the shape and number of spring coils.
   ❖ Record the observations in the table.

Part C: Working Principle

   ❖ The safety pin works on the principle of spring force.
   ❖ When the pin is opened, the spring stores energy. When the pin is released, the stored spring force causes the pin to close and lock inside the clasp, ensuring safe and secure fastening.

Part D: Re-Engineering Study (Improvement Ideas)

Based on the reverse engineering study, suggest improvements such as:

   ❖ Using rust-free material for longer life
   ❖ Adding a plastic cover on the clasp for safety
   ❖ Improving the grip area for easy handling
   ❖ Designing child-safe or decorative versions

Inference

The safety pin has a simple but effective design that combines spring action and locking mechanism. Reverse engineering helps understand its working, while re-engineering allows scope for improving safety, durability, and user comfort.

Result

Thus, the safety pin was successfully studied using reverse engineering to understand its design and working, and re-engineering concepts were applied to suggest possible improvements.

Project (Group / Individual)

Title: Study of Product Development, Life Cycle, and Reverse Engineering of a Common Product

Objective:

To understand how a product is developed, how it performs in the market over time, and how its design can be studied through reverse engineering.

Outcome:

Students will clearly understand how products are created, tracked in the market, and studied for learning.

Project Description:

Students shall select a simple daily-use product and study it using product development stages, product life cycle analysis, and reverse engineering concepts.

Suggested Products:

Mechanical Products	Electrical Products	Electronics Products	Household Products
Water tap	Extension board	TV remote control	Water bottle
Bicycle	Mixer grinder	Mouse (wired)	Pressure cooker
Scissors	Water heater (geyser)	Power bank	Tiffin box lock system
Adjustable spanner	Mobile phone charger	Bluetooth speaker	Office chair (mechanism)
Table fan stand mechanism	Electric bell	Torch light	Umbrella
Screw jack	Electric iron	LED bulb	Gas stove
Door hinge	Electric kettle	Calculator	Foot pedal dustbin

Project Tasks:

1. Describe the product development stages of the selected product
2. Explain the product life cycle stages with reasons
3. Perform basic reverse engineering (visual study or simple disassembly)
4. Identify main components and their functions
5. Prepare sketches or simple diagrams
6. Write conclusions based on understanding

Assignment (Individual)

Title: Product Life Cycle Analysis of a Consumer Product

Task:

- ❖ Select one product
- ❖ Identify its PLC stage (Introduction / Growth / Maturity / Decline)
- ❖ Justify the stage with real-life reasons

Deliverable:

- ❖ 2–3 pages handwritten or typed
- ❖ Simple PLC diagram
- ❖ One real-life example

(Or)

Title: Reverse Engineering Study of a Simple Product

Task:

- ❖ Choose a product
- ❖ List main parts and functions
- ❖ Explain how the product works

Deliverable:

- ❖ Block diagram or sketch
- ❖ Short explanation (1–2 pages)
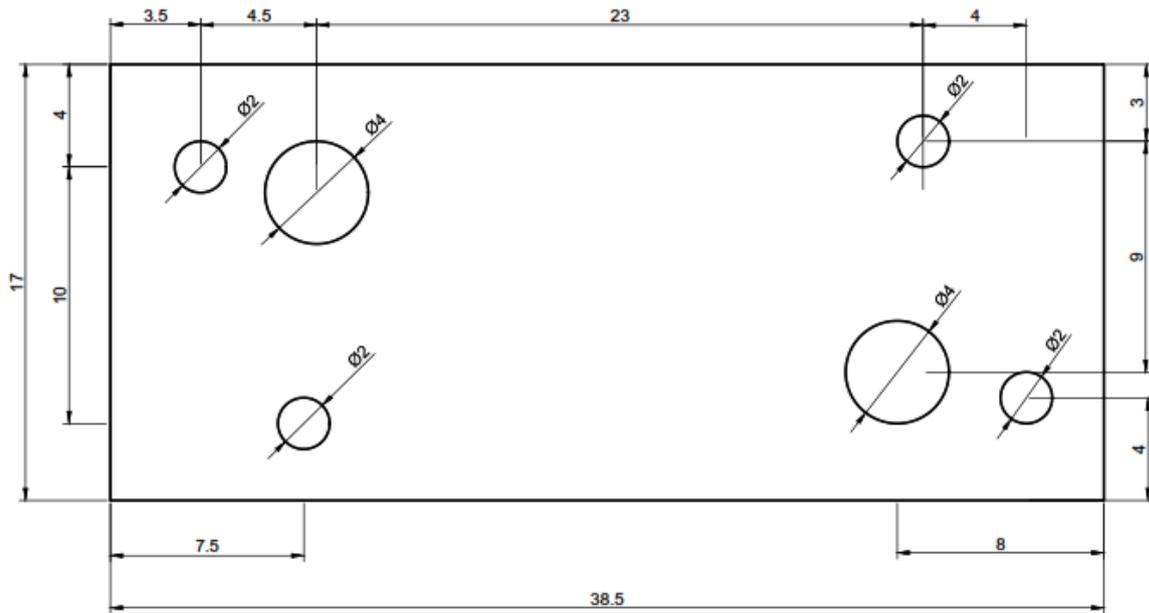
Figure Ex.3: Mechanical Component with Circles and Lines
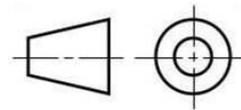
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 3

Date:

2D Drafting of a Mechanical Component with Circles and Lines

Aim

To prepare a 2D CAD drawing of a given mechanical component consisting of straight lines and circles using CAD software.

Outcome

Apply CAD tools to create accurate 2D engineering drawings using basic geometric entities.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Procedure

1. Open the CAD software and create a new drawing file.
2. Set drawing units to millimeters and select appropriate limits.
3. Draw the base geometry using LINE command.
4. Draw circles of required diameters using CIRCLE command.
5. Use OFFSET to create parallel features wherever required.
6. Apply TRIM and EXTEND to remove unwanted portions.
7. Add dimensions using standard dimensioning tools.
8. Save the drawing with proper file name.

Result

The given mechanical component with circles and straight lines was drawn accurately using CAD software.
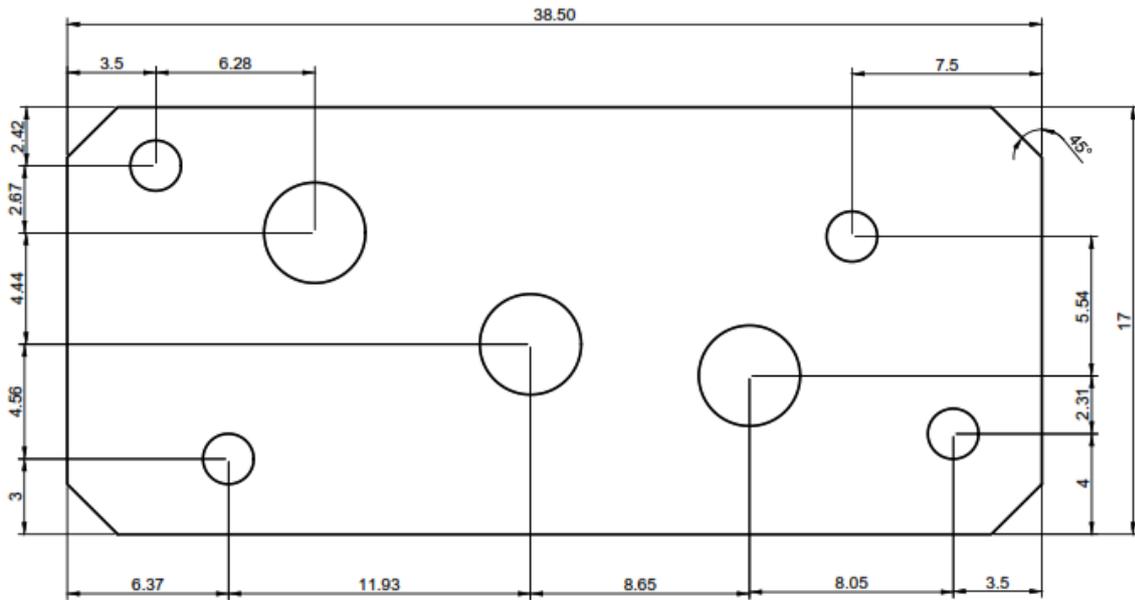
Figure Ex.4: Mechanical Component with Angular and Linear Dimensions
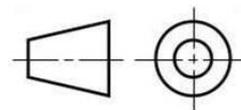
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 4

Date:

2D Drafting of a Component with Angular and Linear Dimensions

Aim

To create a 2D CAD drawing of a component containing angular and linear dimensions as per given specifications.

Outcome

Apply angular and linear dimensioning techniques using CAD tools.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Procedure

1. Create a new drawing and set units to millimeters.
2. Draw the outline of the component using LINE and ARC commands.
3. Use POLAR tracking to draw angular features accurately.
4. Apply ROTATE command where necessary.
5. Trim unwanted lines using TRIM command.
6. Add linear and angular dimensions using appropriate dimension commands.
7. Save the drawing.

Result

A precise 2D CAD drawing with angular and linear dimensions was successfully completed.
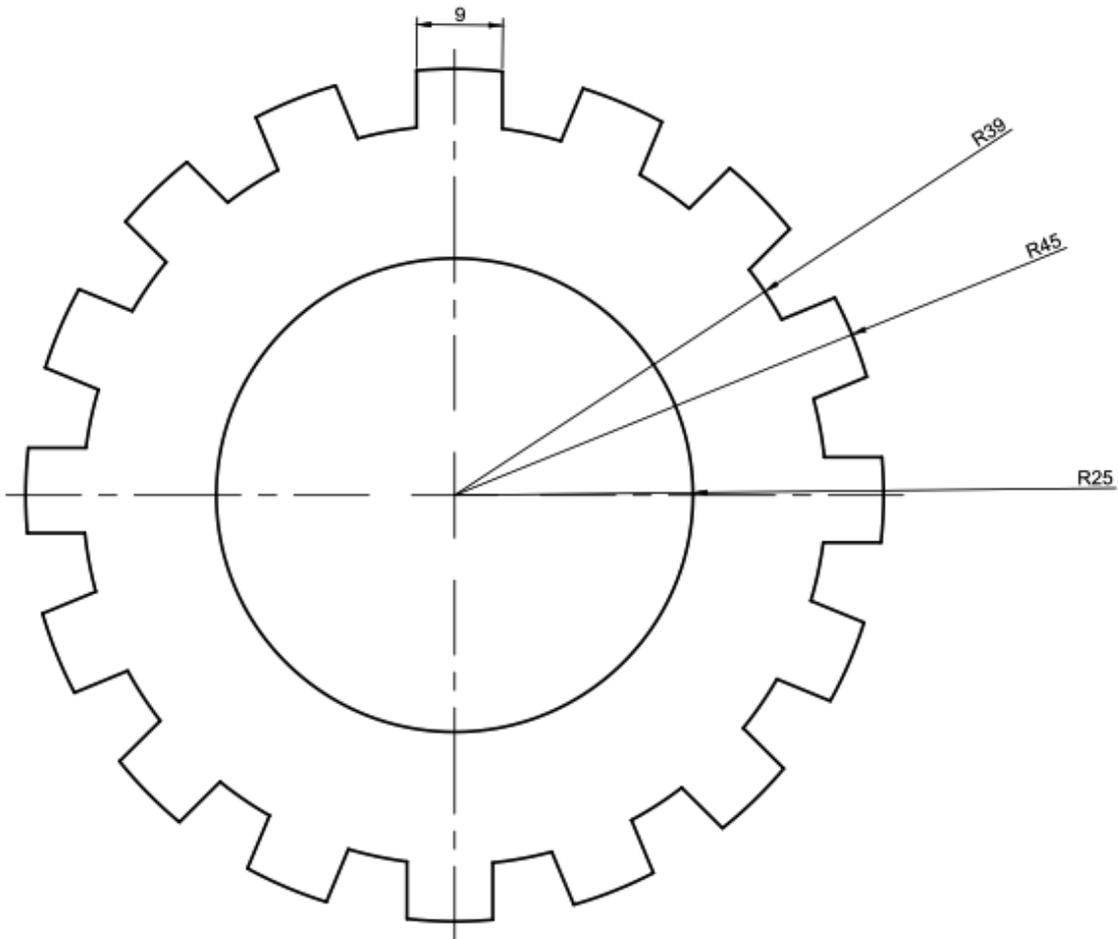
Figure Ex.5: Spur Gear
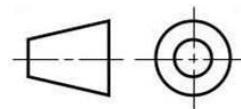
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 5

Date:

2D Drafting of a Spur Gear

Aim

To prepare a detailed 2D CAD drawing of a spur gear using standard proportions and given data.

Outcome

Apply CAD drafting techniques to draw standard mechanical elements using engineering design principles.

Software Used

❖ AutoCAD
❖ FreeCAD
❖ Autodesk Fusion 360

Theory

A spur gear is the simplest and most commonly used type of gear in mechanical systems. It consists of a cylindrical body with straight teeth cut parallel to the axis of rotation. Spur gears are primarily used to transmit motion and power between parallel shafts with constant velocity ratio.

The design of a spur gear is based on standard gear terminology such as:

❖ Pitch Circle – Imaginary circle that rolls without slipping with the mating gear
❖ Addendum – Radial height of the tooth above the pitch circle
❖ Dedendum – Radial depth of the tooth below the pitch circle
❖ Module – Ratio of pitch circle diameter to number of teeth
❖ Tooth Thickness – Width of the tooth measured along the pitch circle

In CAD drafting, spur gears are constructed using standard proportions to ensure proper meshing and smooth power transmission. Accurate drafting of gear profiles is essential in machine design to avoid interference, excessive wear, and noise.

Spur gears are widely used in gearboxes, machine tools, clocks, conveyors, and automotive transmissions due to their high efficiency and ease of manufacture.

Procedure

1. Open the CAD software and start a new drawing file.
2. Set drawing units to millimeters.
3. Calculate gear parameters such as:
4. Pitch circle diameter
5. Addendum circle diameter
6. Dedendum circle diameter
7. Draw the pitch circle, addendum circle, and dedendum circle using CIRCLE command.
8. Construct one gear tooth profile using LINE and ARC commands.
9. Trim unwanted lines to obtain a clean tooth shape.
10. Use ARRAY (Polar Array) to replicate the tooth around the pitch circle.
11. Ensure proper alignment and symmetry of teeth.
12. Add necessary dimensions and annotations.
13. Save the drawing file.

Precautions

❖ Maintain correct gear proportions.
❖ Ensure accurate angular spacing of teeth.
❖ Use polar array carefully to avoid overlapping.

Result

A complete and accurate 2D CAD drawing of a spur gear was successfully drawn using standard gear proportions.
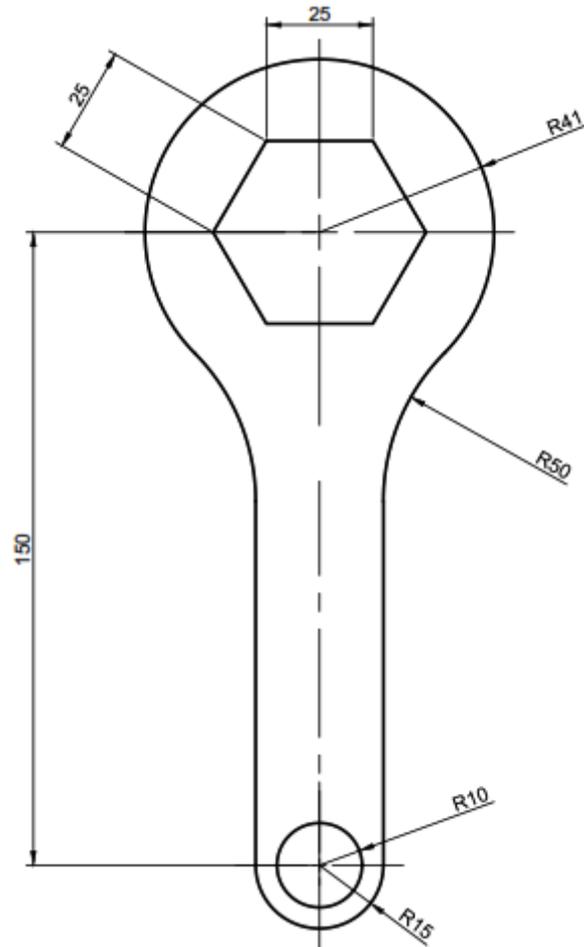
Figure Ex.6: Hexagonal Profile Ring Spanner
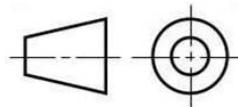
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 6

Date:

2D Drafting of a Ring Spanner

Aim

To prepare a 2D CAD drawing of a ring spanner using standard dimensions.

Outcome

Apply CAD drafting skills to draw hand tools used in mechanical applications.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A ring spanner is a commonly used hand tool in mechanical workshops for tightening or loosening nuts and bolts. Unlike open-ended spanners, ring spanners completely enclose the nut or bolt head, providing better grip and reducing the chances of slippage.

The ring end generally has a 12-point (bi-hexagonal) or 6-point hexagonal profile, which allows easy engagement at different angles. Ring spanners are designed to apply uniform force on the fastener, minimizing damage to edges and improving safety.

In engineering drawing and CAD drafting, ring spanners are drawn using standard proportions, including:

- ❖ Ring diameter
- ❖ Handle length
- ❖ Handle width and thickness
- ❖ Fillets and chamfers for smooth edges

CAD drafting of a ring spanner helps students understand tool geometry, symmetry, and the importance of ergonomic design. It also develops the ability to draw workshop tools accurately as per industrial standards.

Ring spanners are widely used in automotive servicing, machine assembly, maintenance work, and fabrication industries.

Procedure

1. Start a new drawing and set units to millimeters.
2. Draw the center line of the spanner.
3. Create the ring end using CIRCLE and POLYGON commands.
4. Draw the handle portion using LINE and OFFSET commands.
5. Apply FILLET to smoothen corners.
6. Trim unnecessary lines.
7. Add complete dimensioning.
8. Save the drawing.

Result

The 2D CAD drawing of a ring spanner was drawn accurately as per given dimensions.
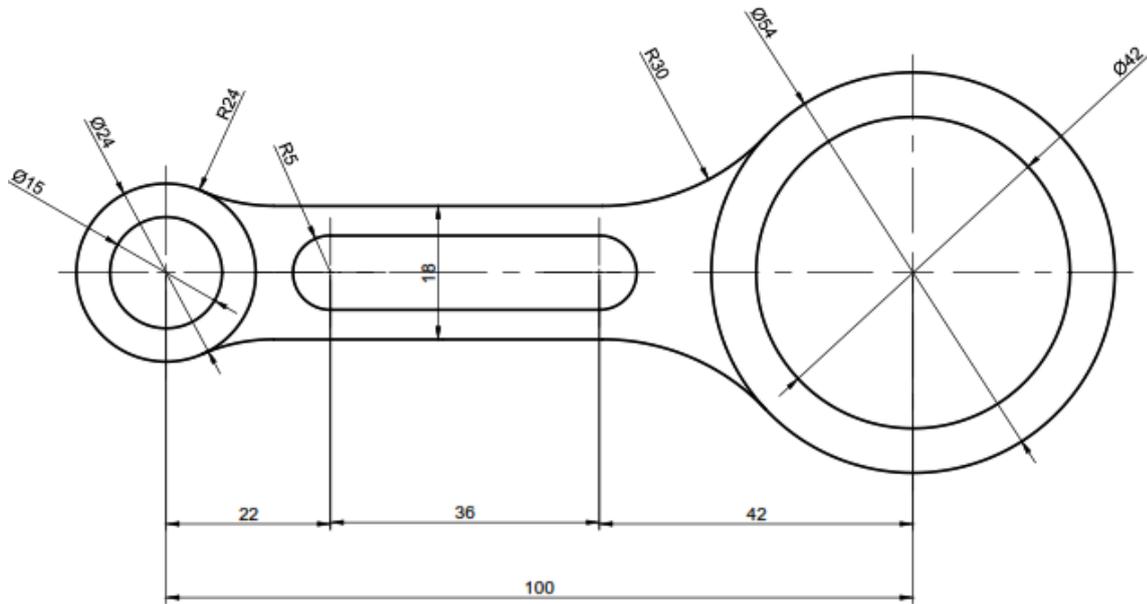
Figure Ex.7: Connecting Rod
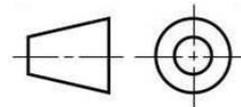
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 7

Date:

2D Drafting of a Connecting Rod

Aim

To draw a 2D CAD drawing of a connecting rod used in internal combustion engines.

Outcome

Apply CAD tools to draw mechanical components used in engines.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A connecting rod is a critical mechanical component used in internal combustion engines. It connects the piston to the crankshaft and converts the reciprocating motion of the piston into rotary motion of the crankshaft.

A typical connecting rod consists of:

- ❖ Small End – Connected to the piston pin
- ❖ Big End – Connected to the crankshaft journal
- ❖ Shank – The body connecting the two ends

Connecting rods are subjected to high tensile, compressive, and bending stresses during engine operation. Hence, they are designed with smooth curves, fillets, and uniform cross-sections to reduce stress concentration.

In CAD drafting, accurate representation of the connecting rod involves:

- ❖ Drawing concentric circles for big and small ends
- ❖ Maintaining correct center-to-center distance
- ❖ Providing smooth transitions using arcs and fillets

Understanding the geometry of a connecting rod through CAD drafting helps students gain insight into engine mechanics, component assembly, and functional design considerations.

Procedure

1. Set drawing units and limits.
2. Draw the big end and small end circles using CIRCLE command.
3. Connect both ends using LINE and ARC commands.
4. Apply OFFSET to obtain thickness.
5. Use FILLET for smooth transitions.
6. Trim unwanted portions.
7. Add dimensions and center lines.
8. Save the drawing.

Result

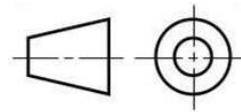The 2D CAD drawing of a connecting rod was completed successfully.
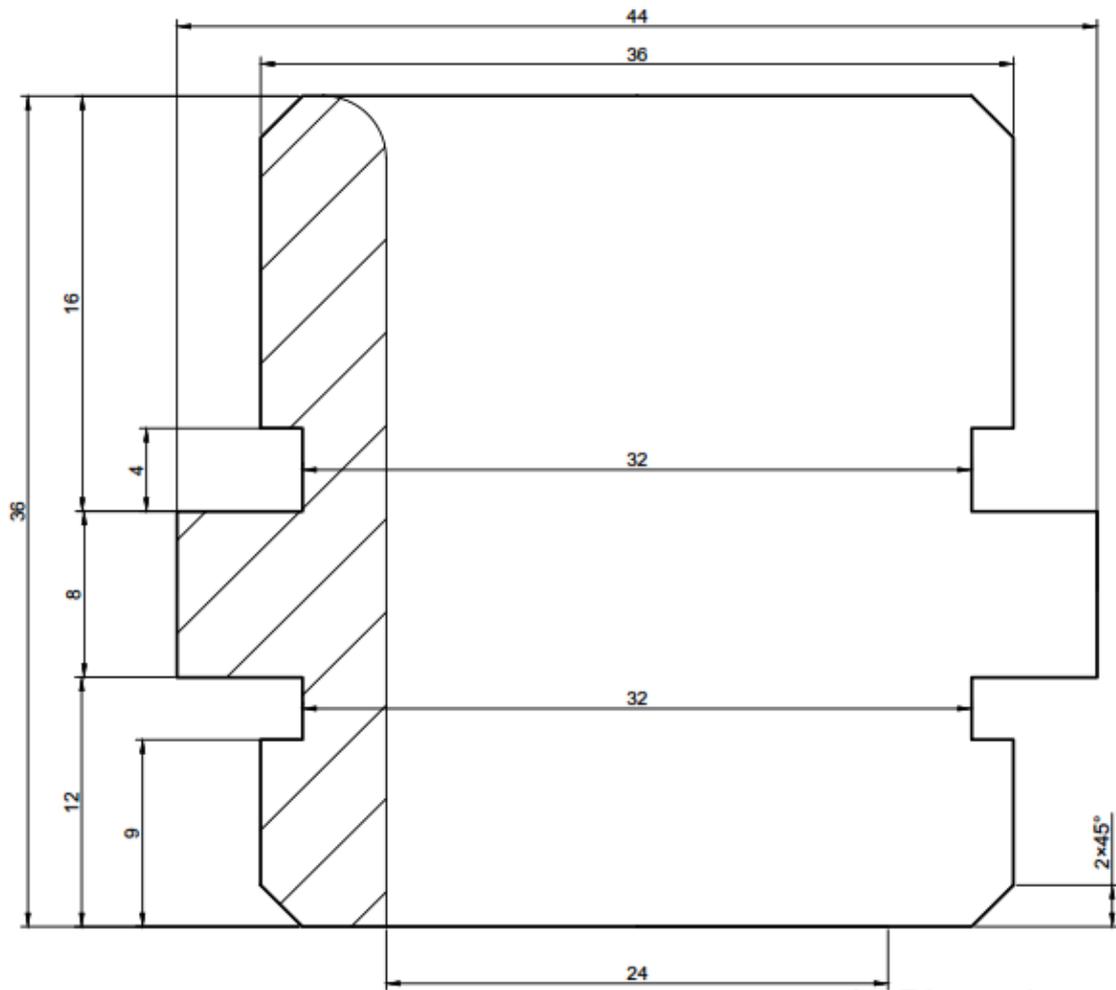
Figure Ex.8: Piston

SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 8

Date:

2D Drafting of a Piston

Aim

To prepare a 2D CAD drawing of a piston used in an internal combustion engine.

Outcome

Apply CAD drafting skills to draw engine components with dimensional accuracy.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A piston is a cylindrical component that moves reciprocally inside the cylinder of an internal combustion engine. It plays a vital role in transferring the force generated by combustion to the connecting rod.

The main parts of a piston include:

- ❖ *Piston Head (Crown)* – Exposed to combustion pressure
- ❖ *Piston Rings and Ring Grooves* – Provide sealing and prevent gas leakage
- ❖ *Piston Skirt* – Guides piston movement inside the cylinder
- ❖ *Piston Pin Holes* – Connect piston to the connecting rod

Pistons are usually made of aluminum alloys to reduce weight and improve heat dissipation. The design of a piston ensures strength, durability, and efficient sealing under high temperature and pressure conditions.

In CAD drafting, piston drawings require:

- ❖ Accurate representation of grooves and clearances
- ❖ Proper alignment of pin holes
- ❖ Symmetry about the vertical axis

Drafting a piston using CAD enhances understanding of engine components, dimensional accuracy, and the importance of precision in manufacturing.

Procedure

1. Open a new drawing and set units to millimeters.
2. Draw the piston profile using LINE and ARC commands.
3. Create piston rings grooves using OFFSET.
4. Draw piston pin holes using CIRCLE command.
5. Trim unwanted lines.
6. Apply dimensioning as per standards.
7. Save the drawing.

Result

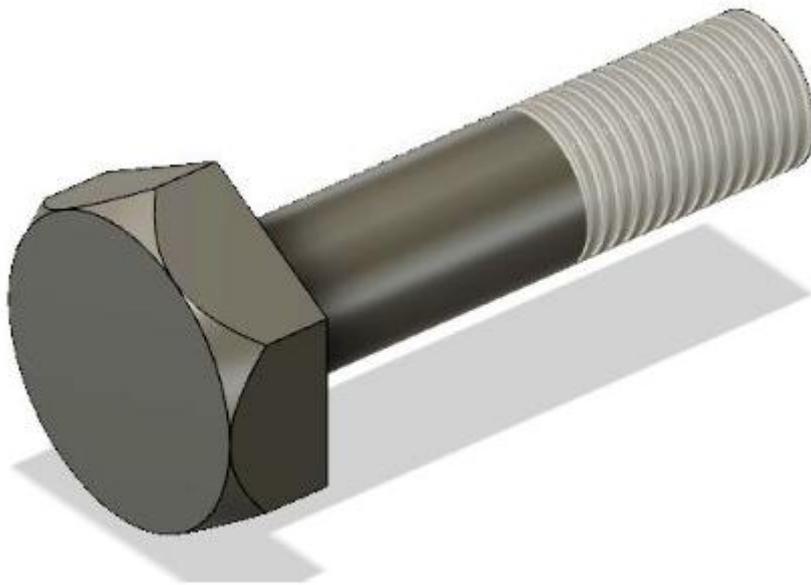The 2D CAD drawing of a piston was drawn accurately using CAD software.

(a)
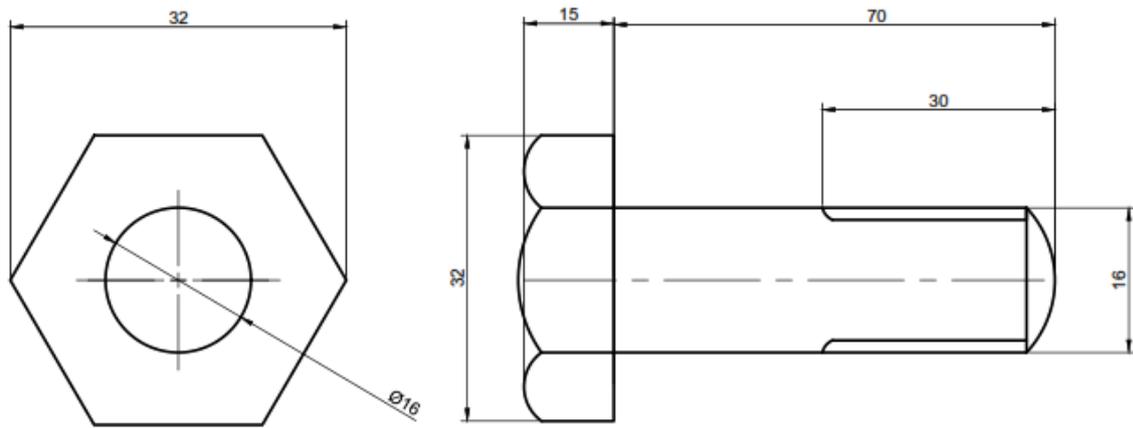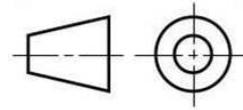
SCALE 1:1

ALL DIMENSIONS ARE IN mm

(b)

Figure Ex.9.1: (a) 2D View of Hexagonal Headed Bolt, (b) 3D Model of Hexagonal Headed Bolt

Expt. No.: 9

Date:

Design, Modeling, and Assembly of a Hexagonal Headed Bolt, Nut, and Washer Using CAD

Aim

To design the 2D drawings, develop 3D solid models, and assemble a hexagonal headed bolt, nut, and washer using CAD software as per given dimensions.

Outcome

Apply CAD tools to design mechanical components, generate 3D models, and assemble standard fasteners into a functional assembly.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

Bolts, nuts, and washers are commonly used mechanical fasteners that provide detachable joints in machine components. A hexagonal headed bolt consists of a cylindrical shank with external threads and a hexagonal head, which allows torque to be applied using a spanner or wrench. A nut is internally threaded and mates with the bolt, while a washer is placed between the nut and the surface to distribute load uniformly and prevent damage to the mating surface.

In engineering design, accurate 2D drafting ensures correct interpretation of dimensions and standards, while 3D modeling helps visualize the actual shape and size of components. Assembly modeling verifies proper fit, alignment, and functionality of individual parts. This experiment integrates design, modeling, and assembly to provide a complete understanding of fastener components used in mechanical systems.
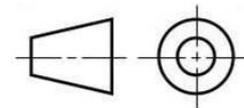
Procedure

PART A: 2D Design and Drafting
1. Open the CAD software and create a new drawing file.
2. Set the drawing units to millimeters.
3. Design the 2D views (front and top views) of:
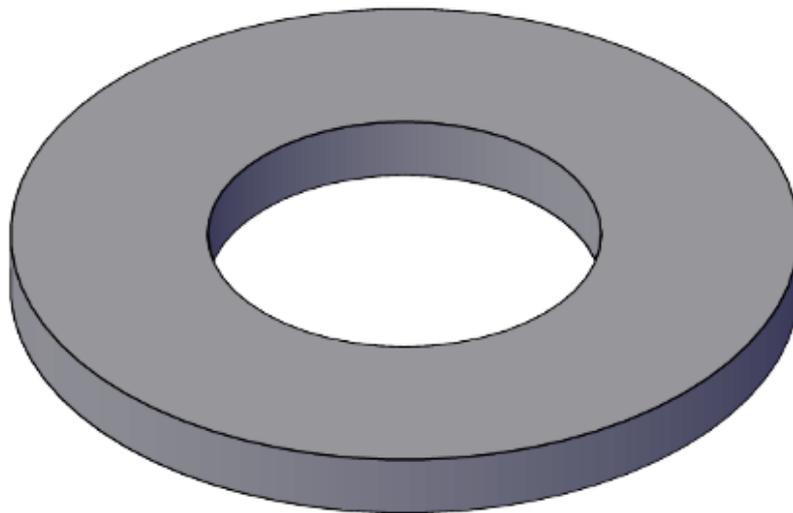   - ❖ Hexagonal headed bolt

(a)
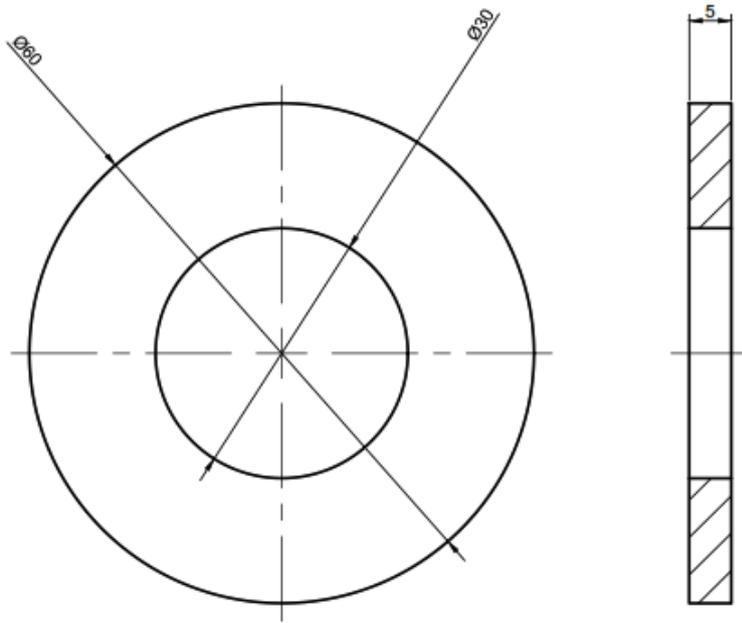
SCALE 1:1

ALL DIMENSIONS ARE IN mm

(b)

**Figure Ex.9.2: (a) 2D View of washer, (b) 3D Model of washer**

❖ Nut

❖ Washer

4. Use basic drawing commands such as LINE, CIRCLE, POLYGON, and ARC.

5. Apply OFFSET, TRIM, and EXTEND commands to obtain accurate geometry.

6. Add center lines and complete dimensioning as per the given specifications.

7. Save the 2D drawing file.

PART B: 3D Modeling

1. Switch to the 3D modeling workspace.

2. Create individual 3D models:

   ❖ *Bolt:* Create the shank using EXTRUDE and the hexagonal head using POLYGON followed by EXTRUDE.

   ❖ *Nut:* Create a hexagonal profile, extrude it, and subtract the inner cylindrical hole.

   ❖ *Washer:* Create two concentric circles and extrude to the required thickness.

3. Apply fillets or chamfers wherever necessary.

4. Save each component as a separate 3D part file.

PART C: Assembly

1. Open a new assembly file in the CAD software.

2. Insert the 3D models of the bolt, nut, and washer.

3. Align the washer concentrically with the bolt shank.

4. Position the nut on the threaded portion of the bolt.

5. Apply suitable assembly constraints such as concentric and coincident constraints.

6. Check the alignment and proper fit of all components.
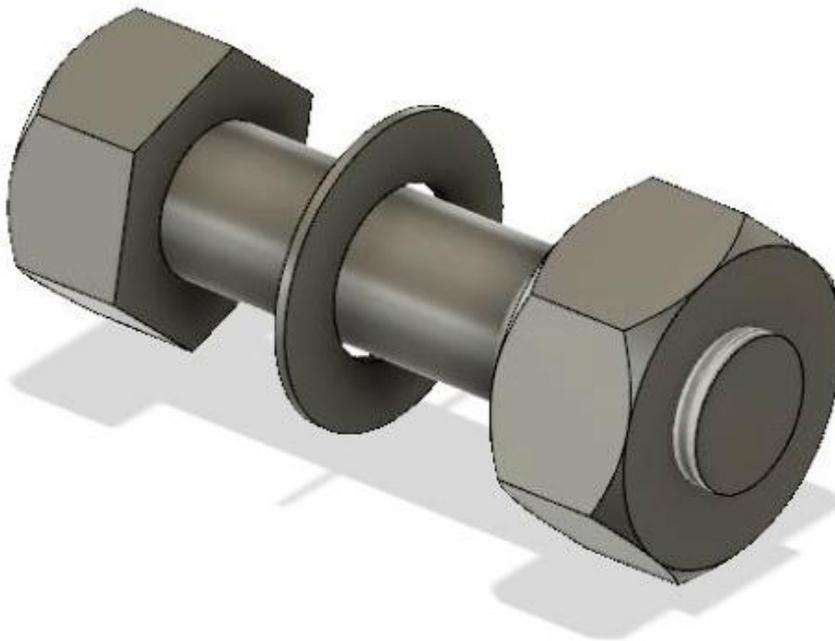
7. Save the assembled model.
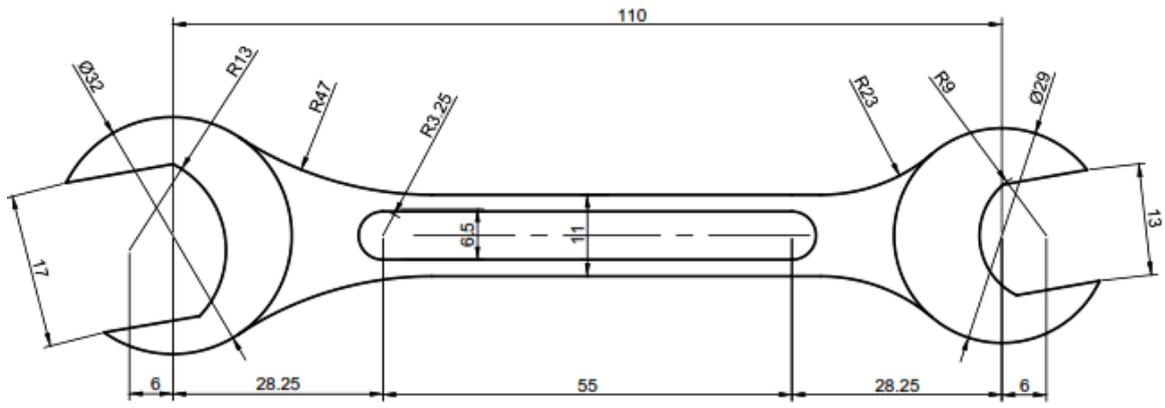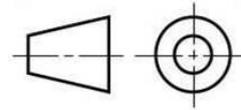
(a)
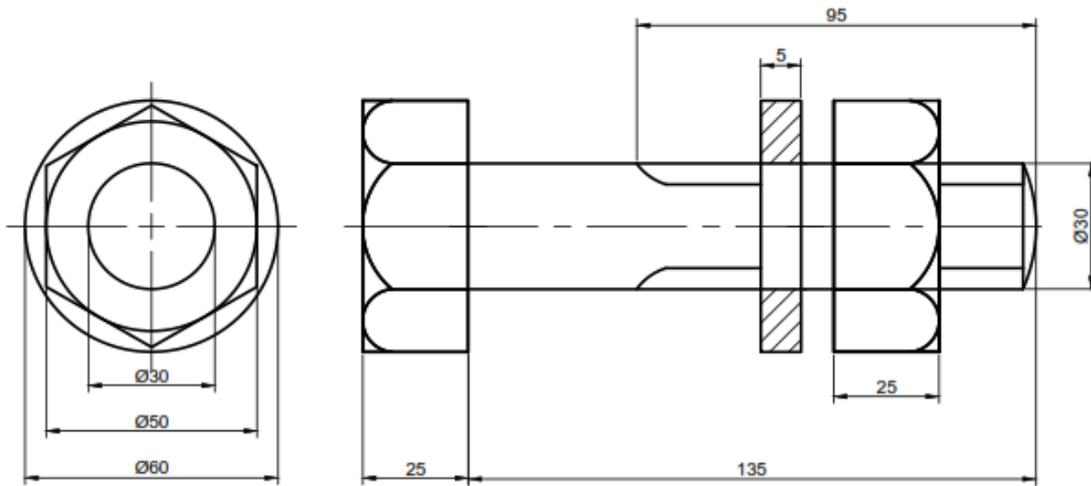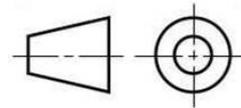
SCALE 1:1

ALL DIMENSIONS ARE IN mm

(b)

Figure Ex.9(c): (a) 2D Assembly of Hexagonal Headed Bolt, Nut, and Washer, (b) 3D Assembly of Hexagonal Headed Bolt, Nut, and Washer

Result

The design, 3D modeling, and assembly of a hexagonal headed bolt, nut, and washer were successfully completed using CAD software as per the given dimensions.
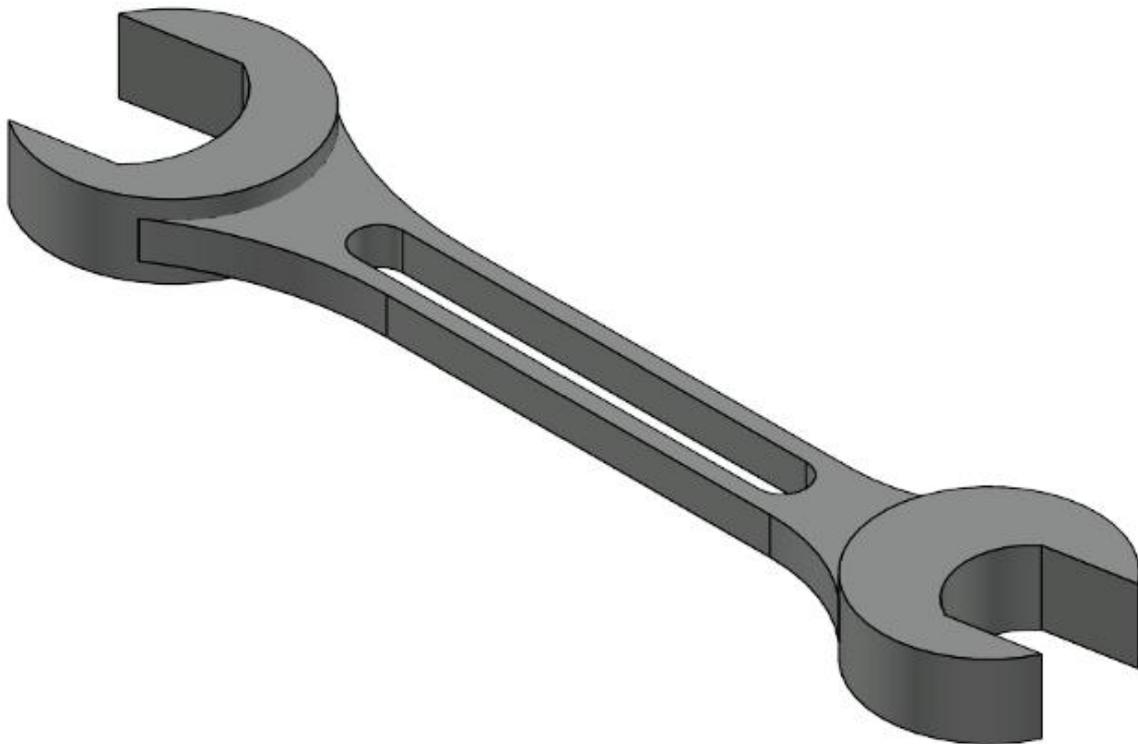
(a)

SCALE 1:1

ALL DIMENSIONS ARE IN mm

(b)

Figure Ex.10: (a) 2D View of Double End Spanner, (b) 3D Model Double End Spanner

Expt. No.: 10

Date:

2D Drafting and 3D Modeling of a Double End Spanner

Aim

To prepare the 2D drafting and 3D solid model of a double end spanner (17–13 size) using CAD software as per the given dimensions.

Outcome

Apply CAD tools to create accurate 2D drawings and develop 3D solid models of mechanical hand tools.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A double end spanner is a commonly used hand tool in mechanical workshops for tightening and loosening nuts and bolts of different sizes. It consists of two open-ended jaws of different dimensions provided at both ends of a single handle. The open ends are generally inclined at a slight angle to improve accessibility in confined spaces.

Double end spanners are manufactured with standardized jaw sizes and handle proportions to ensure strength, durability, and user comfort. The geometry of the spanner includes curved profiles, fillets, and chamfers to reduce stress concentration and improve ergonomics.

In CAD applications, 2D drafting helps in understanding the exact shape and dimensions of the spanner, while 3D modeling provides a realistic visualization of the tool. Modeling such tools enhances students' ability to interpret engineering drawings and apply geometric constraints effectively.

Procedure

PART A: 2D Drafting

1. Open the CAD software and start a new drawing file.
2. Set the drawing units to millimeters (mm).
3. Draw the centerline of the spanner to maintain symmetry.

4. Construct the open-end jaw profiles for 17 mm and 13 mm sizes using lines and arcs.
5. Draw the handle profile using arcs and straight lines as per given dimensions.
6. Apply FILLET, TRIM, and OFFSET commands to obtain smooth curves and accurate geometry.
7. Complete the drawing by adding all dimensions as per the given drawing.
8. Save the completed 2D drawing.
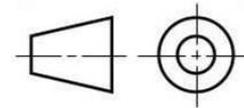
PART B: 3D Modeling

1. Switch to the 3D modeling workspace.
2. Convert the 2D profile of the spanner into a closed region.
3. Use the EXTRUDE command to create a 3D solid of required thickness.
4. Apply FILLET or CHAMFER to edges wherever required for realistic appearance.
5. Inspect the 3D model for accuracy and symmetry.
6. Save the 3D model file.

Result

The 2D drafting and 3D modeling of a double end spanner (17–13 size) were successfully completed using CAD software as per the given dimensions.
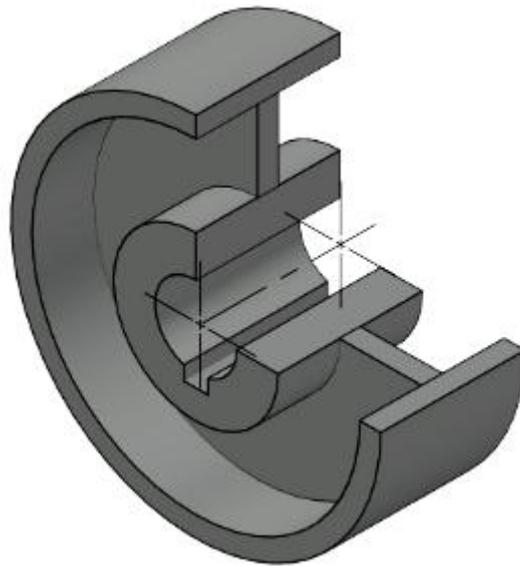
(a)
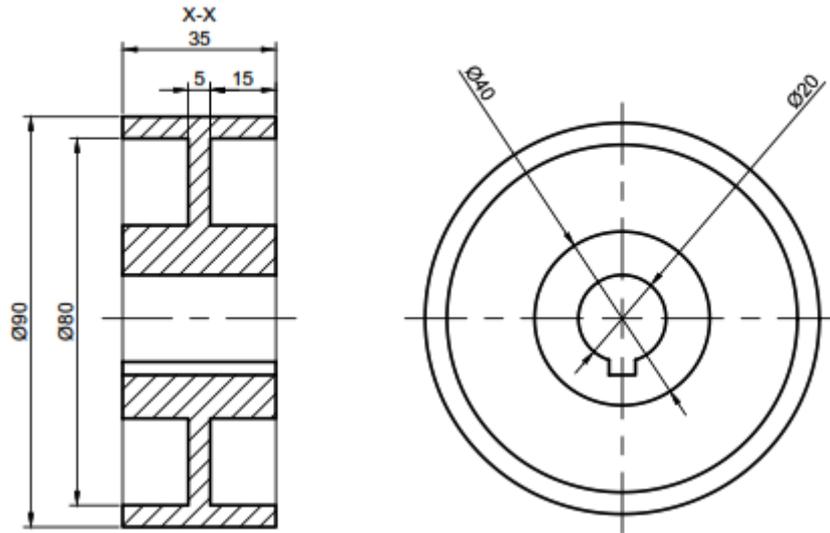
SCALE 1:1

ALL DIMENSIONS ARE IN mm

(b)

Figure Ex.11: 2D View of Pulley with Web, (b) 3D Model Pulley with Web

Expt. No.: 11

Date:

2D Drafting and 3D Modeling of a Pulley with Web Using CAD

Aim

To prepare the 2D drafting and 3D solid model of a pulley with web using CAD software as per the given dimensions.

Outcome

Apply CAD tools to create accurate 2D drawings and develop 3D solid models of mechanical power transmission components.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A pulley is a mechanical power transmission element used to transmit motion and power through belts or ropes. It consists of a grooved rim mounted on a shaft and is commonly used in machines such as conveyors, lifting equipment, and machine tools.

A pulley with web includes a central web connecting the hub and the rim. The web reduces the overall weight of the pulley while maintaining sufficient strength and rigidity. It also helps in uniform stress distribution during rotation.

In engineering practice, pulleys are designed with specific proportions for:

- ❖ Outer diameter and groove width
- ❖ Hub diameter and bore
- ❖ Web thickness

In CAD applications, 2D drafting provides a clear understanding of dimensions and sectional views, while 3D modeling offers realistic visualization and helps verify shape, thickness, and symmetry. This experiment helps students understand the geometry and design features of power transmission components.

Procedure

PART A: 2D Drafting

1. Open the CAD software and start a new drawing file.
2. Set the drawing units to millimeters (mm).
3. Draw the centerline of the pulley to maintain symmetry.
4. Construct the front view and sectional view (X–X) of the pulley using LINE, CIRCLE, and ARC commands.
5. Draw the rim, hub, bore, and web as per the given dimensions.
6. Use OFFSET, TRIM, and EXTEND commands to obtain accurate geometry.
7. Add center lines and complete dimensioning according to the given drawing.
8. Save the completed 2D drawing.

PART B: 3D Modeling

1. Switch to the 3D modeling workspace.
2. Convert the 2D sectional profile into a closed region.
3. Use the REVOLVE command to generate the 3D pulley model about the central axis.
4. Apply FILLET or CHAMFER to edges wherever required.
5. Check the model for symmetry and dimensional accuracy.
6. Save the 3D model file.

Result

The 2D drafting and 3D modeling of a pulley with web were successfully completed using CAD software as per the given dimensions.
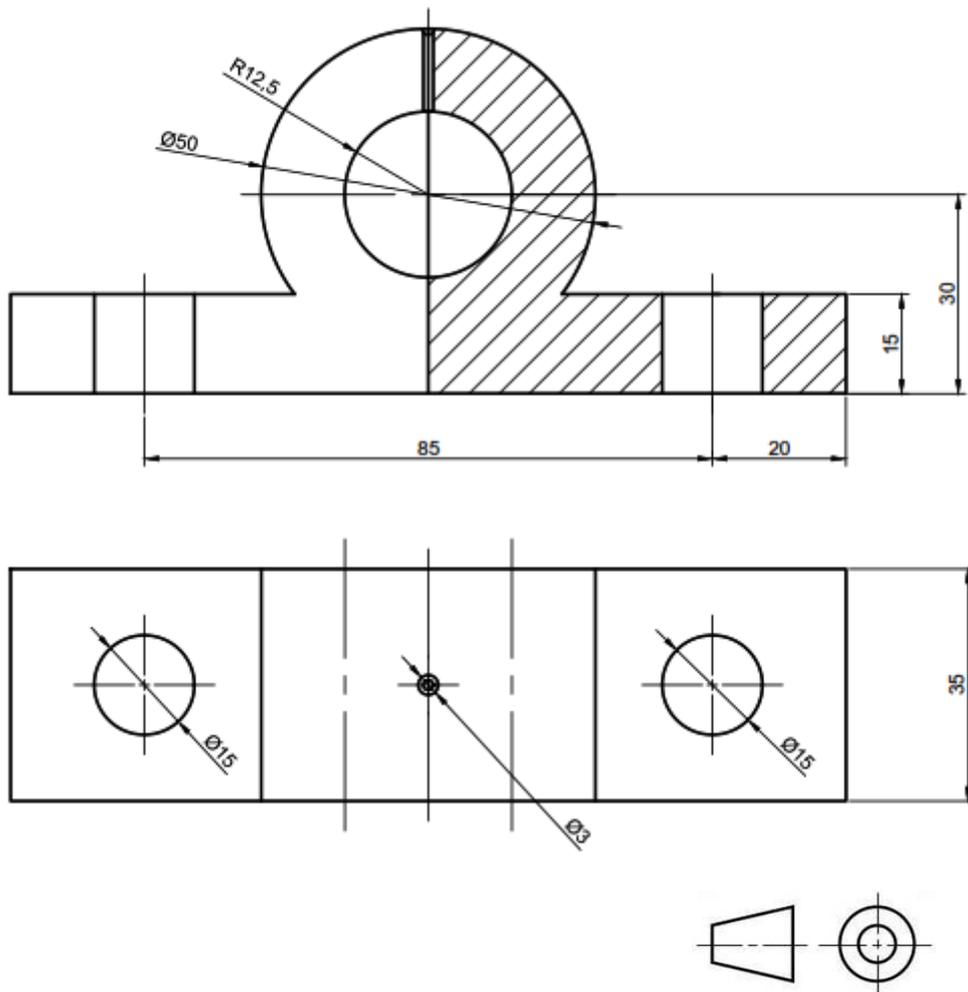
SCALE 1:1

ALL DIMENSIONS ARE IN mm

Figure Ex.12: (a) 2D View of Solid Journal Bearing

Expt. No.: 12

Date:

2D Drafting and 3D Modeling of a Solid Journal Bearing Using CAD

Aim

To prepare the 2D drafting and 3D solid model of a solid journal bearing using CAD software as per the given dimensions.

Outcome

Apply CAD tools to create accurate 2D drawings and develop 3D solid models of standard machine elements used in power transmission systems.

Software Used

- ❖ AutoCAD
- ❖ FreeCAD
- ❖ Autodesk Fusion 360

Theory

A journal bearing is a type of sliding contact bearing that supports a rotating shaft (journal) and allows relative motion between the shaft and the bearing surface. Unlike rolling contact bearings, journal bearings operate on the principle of sliding friction and are commonly used in machines where high load capacity and smooth operation are required.

A solid journal bearing is made as a single-piece cylindrical component with an internal bore to accommodate the shaft. It is usually manufactured from materials such as bronze, cast iron, or bearing alloys to reduce friction and wear. The bearing provides radial support to the shaft and maintains proper alignment during rotation.

Important features of a solid journal bearing include:

- ❖ Outer diameter for mounting into the housing
- ❖ Inner bore diameter for shaft support
- ❖ Length of bearing for load distribution
- ❖ Fillets and radii to reduce stress concentration

In CAD practice, 2D drafting helps in understanding sectional views and dimensional details, while 3D modeling provides clear visualization of the bearing geometry and assists in manufacturing and assembly planning.
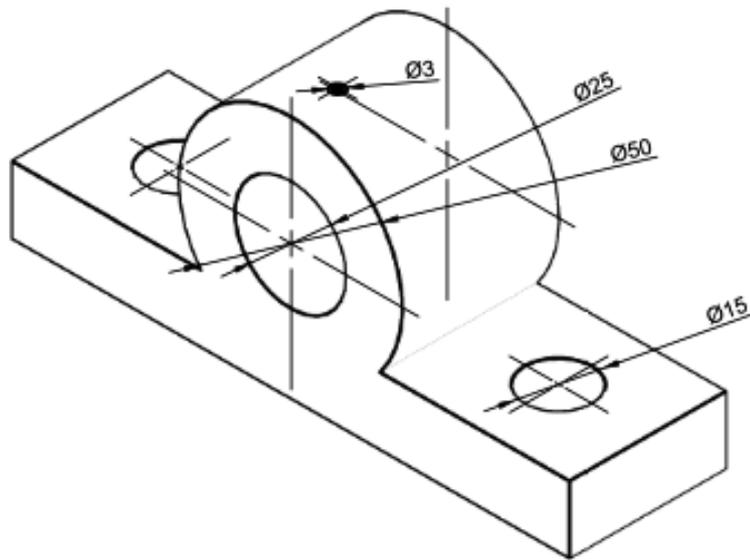
Figure Ex.12: (b) 3D Model Solid Journal Bearing

Procedure

PART A: 2D Drafting

1. Open the CAD software and start a new drawing file.
2. Set the drawing units to millimeters (mm).
3. Draw the centerline to maintain symmetry of the bearing.
4. Construct the front view and sectional view of the solid journal bearing using LINE, CIRCLE, and ARC commands.
5. Draw the inner bore, outer profile, and fillets as per the given dimensions.
6. Use OFFSET, TRIM, and EXTEND commands to obtain accurate geometry.
7. Add center lines and complete dimensioning according to the given drawing.
8. Save the completed 2D drawing file.

PART B: 3D Modeling

1. Switch to the 3D modeling workspace.
2. Convert the 2D sectional profile into a closed region.
3. Use the REVOLVE command about the central axis to generate the 3D solid model of the journal bearing.
4. Apply FILLET to edges wherever required to represent smooth transitions.
5. Verify the dimensions and symmetry of the model.
6. Save the 3D model file.

Result

The 2D drafting and 3D modeling of a solid journal bearing were successfully completed using CAD software as per the given dimensions.
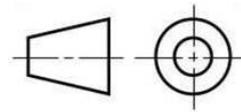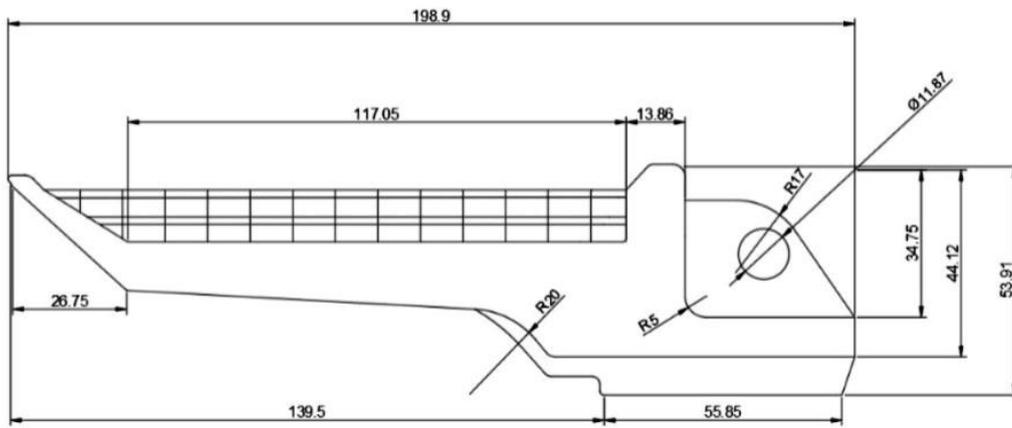
SCALE 1:1

ALL DIMENSIONS ARE IN mm
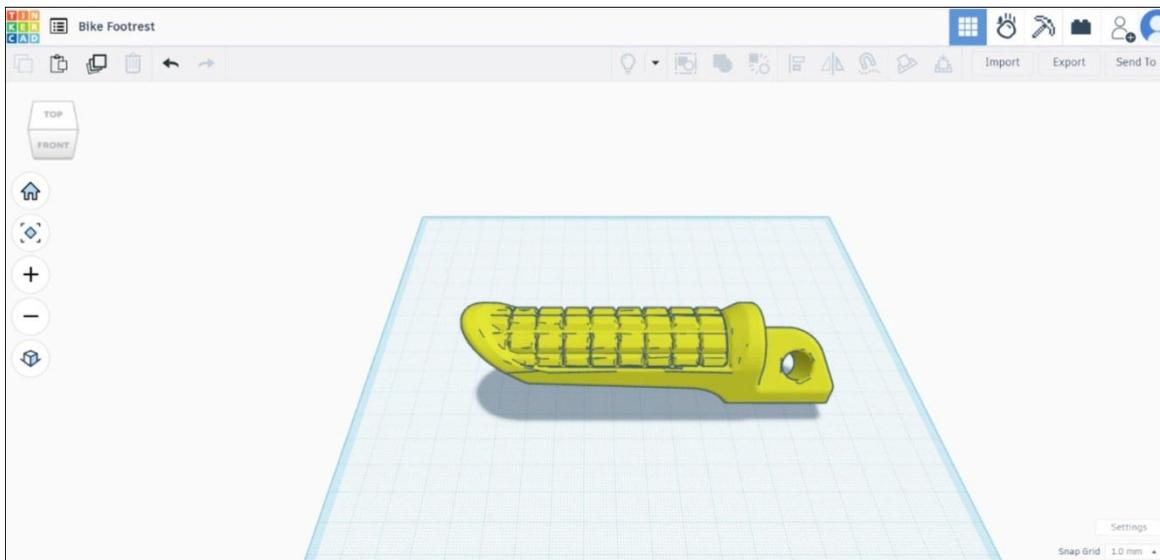
Figure Ex.13: (a) 2D View of Two-Wheeler Foot Rest

Figure Ex.13: (b) 3D Model of Two-Wheeler Foot Rest

Expt. No.: 13

Date:

Product Development of a Two-Wheeler Foot Rest Using CAD Modeling and 3D Printing

Aim

To design, model, and fabricate a two-wheeler foot rest by applying product development principles, CAD modeling techniques, and 3D printing, and to compare the CAD model dimensions with the fabricated prototype.

Outcome

After completing this experiment, students will be able to apply product development concepts to a real-life mechanical component by converting a 2D engineering drawing into an accurate 3D CAD model, effectively using CAD software for part design, fabricating a functional prototype using 3D printing technology, and comparing the CAD model with the manufactured part to analyze dimensional deviations and design accuracy.

Software Used

- ❖ Tinkercad / Fusion 360 / SolidWorks – 3D CAD modeling
- ❖ Cura / PrusaSlicer – Slicing software
- ❖ FDM 3D Printer – Fabrication

Apparatus / Tools Required

- ❖ Computer system
- ❖ 3D printer (FDM type)
- ❖ PLA filament
- ❖ Vernier caliper / steel scale
- ❖ Cleaning tools for post-processing

Theory

Product development involves transforming a concept into a functional product through design, modeling, prototyping, and evaluation. In modern engineering practice, CAD modeling enables accurate visualization and modification of designs, while 3D printing allows rapid fabrication of prototypes for testing and validation.
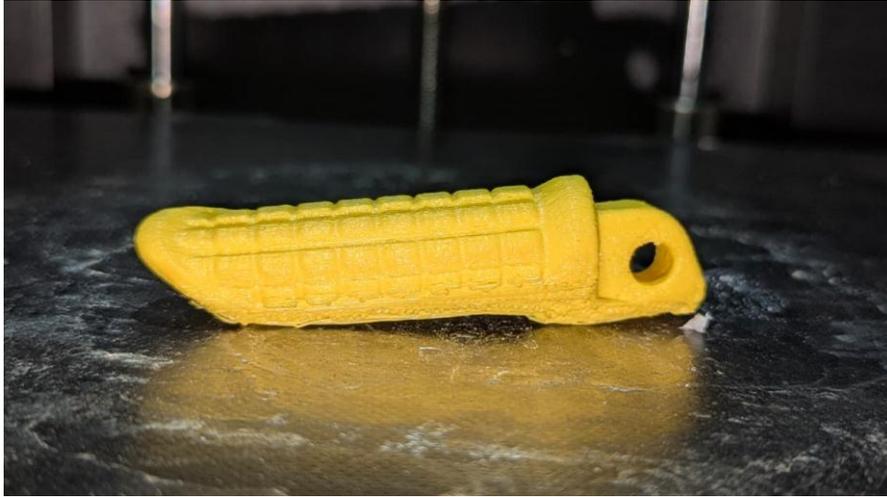
Figure Ex.13: (c) 3D Printed Two-Wheeler Foot Rest

Table 1. Comparison of CAD Model and 3D Printed Part

Sl. No.	Feature	Dimensions (mm)		Deviation (mm)	Remarks
		CAD Model	Printed Part		
1					
2					
3					
4					
5					
6					
7					
8					
9					
10					
11					
12					
13					

A two-wheeler foot rest is a functional and safety-critical component designed to support the rider's foot. It must possess adequate strength, ergonomics, grip, and durability. By using CAD and additive manufacturing, design iterations can be completed quickly and cost-effectively.

This experiment demonstrates the integration of engineering drawings, CAD modeling, and additive manufacturing in product development.

Procedure

Part A: 2D Drafting of Two-Wheeler Foot Rest (with Comments)

1. Study the given dimensions and shape of the two-wheeler foot rest.
2. Open the CAD software and select the 2D drafting environment.
3. Draw the base profile of the foot rest using line, arc, and circle commands.
4. Apply all required dimensions such as length, width, height, hole diameter, and fillet radius.
5. Ensure proper geometric constraints (horizontal, vertical, concentric, tangent).
6. Add center lines and annotations where required.

Comments:

1. 2D drafting helps in understanding the exact geometry of the component.
2. Proper dimensioning avoids modeling errors in later stages.
3. Accurate 2D drawings form the foundation for precise 3D CAD modeling.

Part B: 3D CAD Modeling

1. Convert the completed 2D sketch into a 3D model using extrusion.
2. Add necessary features such as:
    ❖ Mounting hole
    ❖ Fillets and chamfers
    ❖ Grip patterns on the foot rest surface
3. Maintain uniform wall thickness suitable for 3D printing.
4. Inspect the model for symmetry and dimensional accuracy.
5. Save the final model and export it in STL format.

Comments:

1. 3D modeling provides a realistic visualization of the product.
2. Fillets improve strength and reduce stress concentration.
3. Design optimization helps reduce material usage and printing time.

Part C: Slicing and 3D Printing

1. Import the STL file into slicing software.
2. Select the appropriate printing material (PLA).
3. Set printing parameters such as:
   - ❖ Layer height
   - ❖ Infill percentage
   - ❖ Print orientation
   - ❖ Printing speed
4. Generate the G-code file.
5. Load the filament and start the 3D printing process.

Comments:

1. Proper slicing ensures good surface finish and dimensional accuracy.
2. Print orientation affects strength and appearance of the component.
3. Correct parameter selection minimizes printing defects.

Part D: Fabrication and Inspection

1. Allow the printed foot rest to cool completely.
2. Remove the printed part from the build platform.
3. Clean the part and remove any supports.
4. Measure critical dimensions using a vernier caliper.
5. Compare the measured values with the CAD model dimensions.
6. Record observations and deviations.

Comments:

1. Inspection confirms the accuracy of the product development process.
2. Minor deviations may occur due to material shrinkage and layer deposition.
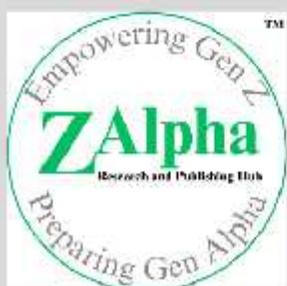3. Comparison helps in understanding manufacturing limitations.

Result

The two-wheeler foot rest was successfully designed using CAD software and fabricated using 3D printing. The comparison showed that the printed part closely matched the CAD model with acceptable dimensional variations.

RE-ENGINEERING FOR INNOVATION

This book bridges engineering theory and practical application through hands-on learning in product development, reverse engineering, measurements, sketching, and CAD modeling. Aligned with Outcome-Based Education and Revised Bloom's Taxonomy (Understanding–Applying), it guides students from conceptual understanding to practical implementation using real-life components. Written in simple language and supported by illustrations and lab exercises, it serves as a lab manual and reference for undergraduate engineering students.

This textbook is made available to students exclusively for educational purposes and is not intended for commercial sale.

Empowering Gen Z
ZAlpha
Research and Publishing Hub
Preparing Gen Alpha