Savitribai Phule Shikshan Prasarak Mandal's

SKN SINHGAD COLLEGE OF ENGINEERING, PANDHARPUR

(Approved by AICTE,New Delhi,Recognized by D.T.E .(M.S)& Affiliated to the Solapur University, Solapur)

Gat.No.6643,Korti, Tal.Pandharpur, Dist.Solapur-413304Office Ph.No.02186-250146

Accredited by NAAC with A+ Grade

Innovation in Teaching & Learning

Subject:CAD CAM CAE

Course Design, Module/Syllabus

Syllabus

A Course on CATIA Software

CATIA (Computer-Aided Three-dimensional Interactive Application) is a robust software suite widely used in various industries for product design and engineering. Its applications span from automotive and aerospace to consumer goods and industrial machinery. CATIA enables engineers and designers to create complex 3D models with high precision, facilitating everything from initial sketches to detailed part designs and comprehensive assemblies. It supports advanced surface modeling, which is crucial for designing aerodynamically efficient automotive bodies and aircraft components. CATIA also excels in simulating kinematics and dynamics, allowing for the analysis of mechanisms and motion within assemblies. Additionally, its capabilities in generative and interactive drafting ensure that accurate and detailed 2D drawings are produced for manufacturing purposes. In the sheet metal design domain, CATIA provides tools to create, modify, and optimize sheet metal parts, ensuring they meet manufacturing constraints and standards. Overall, CATIA's extensive range of tools and workbenches makes it indispensable for designing, validating, and documenting high-quality products efficiently.

Objective 1: Mastery of 2D and 3D Modeling

- To provide students with comprehensive knowledge and skills in creating, editing, and managing both 2D sketches and 3D models in CATIA. This includes understanding the tools and techniques for sketching, part design, and wireframe and surface design.

Objective 2: Proficiency in Assembly and Kinematic Simulation

- To enable students to proficiently create assemblies of multiple parts and simulate their interactions. This involves learning how to define relationships between parts, apply constraints, and perform kinematic analysis to understand the motion and behavior of assemblies.

Objective 3: Competence in Drafting and Sheet Metal Design

- To develop students' abilities in generating detailed 2D drawings for manufacturing and designing sheet metal components. This includes understanding drafting standards, creating accurate documentation, and applying specific sheet metal design tools and techniques to create manufactural parts.

Sr.No	Name of Course	Course Contents	Course Duration
1	CATIA	a) INTRODUCTION TO CATIA b) SKETCHER WORKBENCH c) PART DESIGN WORKBENCH d) WIRE FRAME AND SURFACE DESIGN WORKBENCH e) ASSEMBLY DESIGN WORKBENCH f) GENERATIVE AND INTERACTIVE DRAFTING WORKBENCH g) DMU KNEMATICS h) GENERATIVE SHEET METAL DESIGN	30 Hrs

Content Details

a) **Introduction to CATIA:**

- Overview of CATIA and its capabilities.
- Understanding the user interface and customization.
- Basic navigation and toolbar usage.
- Introduction to different workbenches.

b) **Sketcher Workbench:**

- Basics of 2D sketching.
- Creating and editing sketches.
- Applying geometric and dimensional constraints.
- Using tools like lines, circles, arcs, and rectangles.

c) **Part Design Workbench:**

- Transforming 2D sketches into 3D models.
- Basic features: pad, pocket, hole, and shaft.
- Advanced features: fillet, chamfer, patterns, and mirrors.
- Editing and modifying parts.

d) **Wireframe and Surface Design Workbench:**

- Creating and manipulating wireframe geometry.
- Basic surface creation: extrude, revolve, sweep, and loft.
- Operations on surfaces: trimming, splitting, and joining.

e) **Assembly Design Workbench:**

- Introduction to assembly creation.
- Importing and positioning parts.
- Applying assembly constraints: coincident, concentric, and distance.
- Creating exploded views and checking for interferences.

f) **Drafting Workbench:**

- Generating 2D drawings from 3D models.
- Creating different views: projection, section, and detail.
- Adding dimensions, annotations, and tolerances.
- Managing drawing sheets and templates.

g) **Basic DMU Kinematics:**

- Introduction to digital mock-up (DMU) and kinematic simulations.

- Defining and simulating kinematic joints.
- Analyzing motion and creating animations.

h) Generative Sheet Metal Design:

- Basics of sheet metal design.
- Creating sheet metal features: walls, bends, and flanges.
- Modifying sheet metal parts: cuts, holes, and patterns.
- Generating flat patterns for manufacturing.

Linking with Subject Syllabus

Section I

Unit 2: Computer Graphics and Geometric Modeling

- **Connection with CATIA:**
  - CATIA's Sketcher and Part Design workbenches are essential for geometric modeling, allowing students to create parametric models, apply geometric transformations, and work with advanced curves like Bezier, B-Spline, and NURBS.
  - **CATIA Content:**
    - **Sketcher Workbench:** Basics of 2D sketching, geometric and dimensional constraints.
    - **Part Design Workbench:** 3D modeling from sketches, advanced features like fillet and chamfer.

Unit 3: Finite Element Method and Automation

- **Connection with CATIA:**
  - CATIA integrates Finite Element Analysis (FEA) tools, enabling structural and thermal analysis directly within the software. This aligns with the syllabus content on FEM and automation, where students learn to apply FEA concepts using CATIA's robust simulation tools.
  - **CATIA Content:**
    - **FEA in CATIA:** Structural and thermal analysis of mechanical components.
    - **Automation in CATIA:** Utilizing CATIA's tools for automated design and manufacturing processes.

Industry Suggestion

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Industry Suggestions for Basic CATIA Learning Syllabus

Date: 21/8/2021

1. **Introduction to CATIA:**
   - **Application:** Familiarity with CATIA's interface and navigation is essential for any design or engineering role. It lays the groundwork for efficient use of the software.
   - **Suggestion:** Include a brief overview of how CATIA is used in different industries (e.g., automotive, aerospace) to highlight its relevance and importance.
2. **Sketcher Workbench:**
   - **Application:** 2D sketches are the foundation of 3D models. Accurate sketches are crucial for designing parts that meet industry standards.
   - **Suggestion:** Incorporate exercises that mimic real-world scenarios, such as creating sketches for automotive components or consumer product designs.
3. **Part Design Workbench:**
   - **Application:** Transforming sketches into 3D models is a core task in product development, from designing engine parts to household appliances.
   - **Suggestion:** Include projects that require students to model parts commonly found in their target industry, such as gears, brackets, or casings.
4. **Wireframe and Surface Design Workbench:**
   - **Application:** Complex surfaces are often required in industries like aerospace for aerodynamic components or in consumer products for aesthetic surfaces.

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8. **Generative Sheet Metal Design:**
   - **Application:** Designing sheet metal parts is crucial in industries such as automotive, aerospace, and consumer electronics, where lightweight and durable components are needed.
   - **Suggestion:** Provide projects that involve creating sheet metal enclosures, brackets, or panels, emphasizing manufacturability and cost-effectiveness.

General Suggestions:

- **Industry-Specific Projects:** Tailor projects and assignments to reflect real-world industry challenges, ensuring that students gain relevant and practical experience.
- **Guest Lectures:** Invite industry professionals to give talks or workshops, providing students with insights into current trends and applications of CATIA in their fields.
- **Certifications:** Encourage students to pursue certifications from Dassault Systems, which can enhance their credibility and job prospects.
- **Collaboration Tools:** Introduce collaborative features and cloud-based platforms like 3DEXPERIENCE to prepare students for modern, team-based engineering environments.
- **Continuous Updates:** Regularly update the syllabus to include the latest tools and features in CATIA, ensuring that students are learning the most up-to-date practices and technologies.

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Use of ICT

Use of ICT Tools in Teaching CATIA Software

1. **Video Lectures and Demonstrations:**
   - **Application:** Create or use existing video lectures to explain complex concepts and demonstrate software usage. These can be paused, replayed, and referenced as needed.
   - **Example:** Record detailed demonstrations of modeling processes, from sketch creation to assembly design.

Online tutorials created by faculty

a. https://www.youtube.com/watch?v=7PKT--byR4Q
b. https://youtu.be/dSZ1TR_Qqn8?si=_aPJGzhoTR-NKjr_
c. https://www.youtube.com/watch?v=uGedrS8_HRE&t=165s
d. https://youtu.be/SuxzTlHNgHY?si=jPvfEgEKcY6bdi9K
e. https://youtu.be/0O3Lj07R_pk?si=RJzvzsAy3-Nv0wcb
f. https://www.youtube.com/watch?v=8zuN3oT7fKQ
g. https://www.youtube.com/watch?v=rNkoohS9-TI
h. https://www.youtube.com/watch?v=9gpWFq9yRx8
i. https://www.youtube.com/watch?v=tFKeCxoUNEk&t=269s
j. https://www.youtube.com/watch?v=7jpsy1zpYg8&t=65s

2. **Learning Management Systems (LMS):**
   - **Application:** Use LMS platforms like Moodle, Blackboard, or Canvas to organize course materials, track progress, and manage assignments and quizzes.
   - **Example:** Create modules for each CATIA workbench with associated reading materials, videos, and exercises.

Moodle Server

Course Link http://103.99.146.35/course/view.php?id=494

By incorporating these ICT tools into CATIA software teaching, educators can enhance the learning experience, making it more interactive, accessible, and aligned with modern industry practices.

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COMPUTER AIDED DESIGN COURSE ON "CATIA" SOFTWARE

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Your progress

Announcements

CATIA Syllabus

a. **Introduction To CATIA**

b. **Sketcher Workbench**

c. **Part Design Workbench**

d. **Wire Frame And Surface Design**

e. **Assembly Design Workbench**

f. **Generative And Interactive Drafting**

g. **Dmu Kinematics**

Video How to install CATIA Software

Hello students above video contains download and Installation of CATIA V5 R20. CATIA was initially developed for use in designing the Dassault Mirage fighter jet. Catia is widely used for Designing purposes especially in the Aerospace sector and few other segments of industries

Download SetUp CATIA V5

Introduction To CATIA

This Course is to help you master your drawing skills utilizing the Profile Icon in Catia V5 software. Catia One of the leading design tools used by professionals working in the field of mechanical engineeringThe profile icon is the most efficient way to draw using a sketch. You can continuously draw lines and arcs/radii without interruption. Remember time is money. If you were working in industry your company would want you to work as proficiently as possible. There are short quiz's throughout this course. Click on arrows in the lower right and left hand corners to navigate your way around.

Overview of CATIA V5

2. **CATIA- One of the leading design tools used by professionals working in the field of mechanical engineering**
3. **What CATIA V5 do? CATIA V5 is an integrated Computer Aided Engineering tool that Incorporates CAD, CAM, CAE, and other applications Is a native Windows application Is based on Graphical User Interface based on Variation / Parametric technology and Encourages design flexibility and reuse • Supports knowledge based design**
4. **CATIA V5 Philosophy • Flexible modeling environment • Ability to easily modify models and implement design changes • Support for data sharing and data reuse • Knowledge enabled • Capturing design constraints and design intent as well as final model geometry • Managing information of non-geometric as well as geometric design • 3D Part is the master model • Drawings, Assemblies and Analyses are associative to the 3D parts. If the part design changes; the downstream models change too.**

VIDEO LECTURE 1

VIDEO LECTURE 2

VIDEO LECTURE 3

Sketcher Workbench

Definition of CAD/CAM:

Computer aided design (CAD) can be defined as the use of computer systems to assist in the creation, modification, analysis, or optimization of a design.

Computer aided manufacturing (CAM) can be defined as the use of computer systems to plan, manage, and control the operations of manufacturing plant through either direct or indirect computer interface with the plant's production resources.

- VIDEO LECTURE 1
- VIDEO LECTURE 2
- VIDEO LECTURE 3
- Practice Exercise no.1
- Practice Exercise no.2
- Practice Exercise no.3

Part Design Workbench

Solid modeling:

A solid model of an object is more complete representation than its surface model. It is unique from the surface. Defining an object with the solid model is the easiest of the available three modeling techniques. Solid model model in topological information it stores which potentially permits functional automation and integration can be quickly created without having to define individual locations as with wire frames. The completeness and unambiguity of solid models are attributed to the information that is related database of these models stores. **(Topology-- It determines the relational information between objects.)**

To model an object completely we need both geometry & topological information. Geometry is visible, whereas topological information are stored in solid model database are not visible to user. Two or more primitives can be combined to form the desire solid. Primitives are combined by Boolean Operations.

Practice Exercise no.4

Practice Exercise no.5

Wire Frame And Surface Design

Practice Exercise no.6

Practical Exercise no.7

Assembly Design Workbench

Assembly modeling

Assembly modeling is the process of creating designs that consist of two or more components assembled together at their respective work positions. The components are brought together and assembled in **Assembly Design** workbench by applying suitable parametric assembly constraints to them. The assembly constraints allow you to restrict the degrees of freedom of components on their respective work positions. The assembly files in CATIA are called Product files. There are two methods to invoke the **Assembly Design** workbench of CATIA. The primary method to start a new product file is by selecting **File** > **New** from the menu bar to open the **New** dialog box From this dialog box select **Product**, as shown in Figure. The other method of invoking the **Assembly Design** workbench is by choosing **Start** > **Mechanical Design** > **Assembly Design** from the menu bar.

Types of Assembly Design Approach

In CATIA you can create assembly models by adopting two types of approaches. The first design approach is the bottom-up approach, and the second one is the top-down approach. Both these design approaches are discussed below.

Bottom-up Assembly: The bottom-up assembly is the most preferred approach for creating assembly models. In this of approach, the components are created in the **Part Design** workbench as (**.CATPart*) file. Then the product (**.CATProduct*) file is started and all the previously created components are inserted and placed in it using the tools provided in the **Assembly Design** workbench. After inserting each component, constraints are applied to position them properly in the 3D space with respect to other components. Adopting the bottom-up approach gives the user the opportunity to pay more attention to the details of the components as they are designed individually. Because the other components are not present in the same window, it becomes much easier to maintain a relationship between the features of the current component. This approach is preferred for large assemblies, especially those having intricate individual components.

Top-down Assembly: In the top-down assembly design approach, components are created inside the Assembly Design workbench. Therefore, there is no need to create separate part files of the components. This design approach is completely different from the bottom-up design approach. Here you have to start the product file first and then, one by one, create all components. Note that even though the components are created inside the product file, they are saved as individual part files and can be opened separately later. Adopting the top-down design approach gives the user the distinctive advantage of using the geometry of one component to define the geometry of the other. Here the construction and assembly of the components takes place simultaneously. As a result of this, the user can view the development of the product in real time. This design approach is highly preferred, while working on a conceptual design or a tool design where the reference of previously created parts is required to develop a new part.

Practical Exercise no.8

Lectures by Prof.P.P.Kulkarni

Introduction to 2D sketch

- 2D Skecth and Introduction to 3D Sketch
- Sketch Based Feature in CATIA
- Basics of Assembly Design
- Rachet Modeling
- Modelling of Bicycle tyre
- Practice Models

Assembly Exercise

- Mini Project:Draw Assembly of Universal Coupling
- Assembly and animation of universal coupling -SKN SINHGAD COLLEGE OF ENGINEERING
- CATIA FILES Assembly universal :Prof.P.P.Kulkarni coupling

Practice exercises

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Instructional Delivery and Methods for each Module Unit

Instructional Delivery and Methods for CATIA Basic Course Modules

Module	Instructional Delivery	Instructional Method
1. Introduction to CATIA	Lecture, Demonstration	- Use presentations and live demonstrations to explain CATIA's interface and basic features.- Provide hands-on walkthroughs to familiarize students with navigation and tools.
2. Sketcher Workbench	Interactive Tutorial, Hands-On Practice	- Conduct guided exercises using interactive tutorials to teach 2D sketching.- Provide practice exercises where students create and edit sketches with real-time feedback.
3. Part Design Workbench	Lecture, Demonstration, Hands-On Practice	- Demonstrate basic and advanced part design features through video lectures.- Facilitate lab sessions for students to practice creating and modifying 3D models.
4. Wireframe and Surface Design Workbench	Demonstration, Hands-On Practice	- Use screen-sharing and live demonstrations to show wireframe and surface design techniques.- Assign hands-on projects for students to create and manipulate wireframes and surfaces.
5. Assembly Design Workbench	Lecture, Interactive Exercises, Collaborative Projects	- Present assembly design principles and constraints through lectures.- Conduct interactive exercises to practice assembling parts.- Facilitate group projects to create and manage complex assemblies.
6. Drafting Workbench	Demonstration, Hands-On Practice, Review Sessions	- Demonstrate the creation of 2D drawings from 3D models using live sessions.- Provide hands-on practice for generating and annotating drawings.- Review and critique student drafts to ensure accuracy and adherence to standards.
7. Basic DMU Kinematics	Lecture, Simulation, Hands-On Practice	- Lecture on basic kinematics and simulation principles.- Use simulation tools to demonstrate motion analysis.- Allow students to perform kinematic simulations on their models.
8. Generative Sheet Metal Design	Demonstration, Hands-On Practice	- Demonstrate sheet metal design techniques through live demonstrations.- Provide practice exercises for creating and modifying sheet metal parts.- Conduct reviews to check for proper use of sheet metal tools.

Key Points for Each Instructional Method

- **Lecture:** Provides foundational knowledge and theoretical background.
- **Demonstration:** Shows how to use specific tools and features in CATIA, typically through live or recorded videos.
- **Hands-On Practice:** Allows students to apply what they've learned through guided exercises and projects.
- **Interactive Tutorial:** Engages students with interactive elements to reinforce learning.
- **Collaborative Projects:** Encourages teamwork and real-world application of skills in group settings.
- **Review Sessions:** Offers feedback and critiques to improve accuracy and adherence to standards.

This table ensures that each module is delivered effectively, combining theoretical knowledge with practical application to provide a comprehensive learning experience.

Assessment /Project Work

Course Exercises Assessment Rubrics

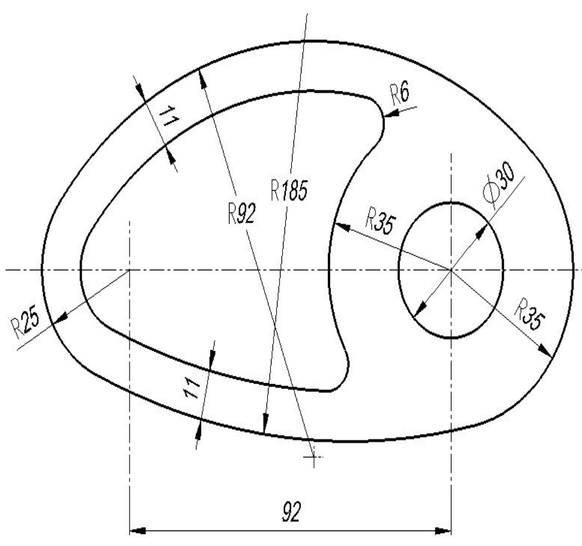
Category	Criteria	Exemplary (25-30 points)	Proficient (19-24 points)	Basic (13-18 points)	Needs Improvement (0-12 points)
Sketcher Workbench	**Accuracy of Sketches**	Sketches are precise and accurate with all geometric and dimensional constraints correctly applied.	Sketches are mostly accurate with minor errors in constraints.	Sketches have several inaccuracies, and constraints are inconsistently applied.	Sketches are inaccurate and lack proper constraints.
	Complexity of Sketches	Demonstrates high complexity using multiple tools and constraints effectively.	Shows moderate complexity with a good variety of tools and constraints used.	Simple sketches with limited use of tools and constraints.	Very basic sketches with minimal use of tools and constraints.
Part Design Workbench	**Transformation of Sketches to 3D Models**	3D models accurately reflect sketches with all features (extrude, revolve, fillet, chamfer) correctly applied.	3D models are mostly accurate with minor errors in feature application.	Several inaccuracies in transforming sketches to 3D models.	3D models do not accurately reflect sketches, and feature applications are incorrect.
	Use of Advanced Features	Effectively uses advanced features such as patterns, mirrors, and complex transformations.	Uses some advanced features with moderate effectiveness.	Limited use of advanced features.	Does not use advanced features effectively.

Assembly Design Workbench	**Correct Assembly of Parts**	Parts are assembled correctly with appropriate constraints, ensuring functional assemblies.	Most parts are assembled correctly with minor issues in constraints.	Several parts are incorrectly assembled or missing constraints.	Assemblies are incorrect and lack proper constraints.
	Complexity of Assembly	Demonstrates high complexity with multiple parts and intricate relationships.	Shows moderate complexity with a good number of parts and relationships.	Simple assemblies with limited parts and relationships.	Very basic assemblies with minimal parts and relationships.
Drafting Workbench	**Accuracy of 2D Drawings**	2D drawings are accurate with all views, dimensions, and annotations correctly applied.	Drawings are mostly accurate with minor errors in dimensions or annotations.	Several inaccuracies in drawings, dimensions, or annotations.	Drawings are inaccurate and lack proper dimensions and annotations.
	Adherence to Standards	Drawings adhere to industry standards (e.g., ISO, ASME) with all necessary details included.	Drawings mostly adhere to standards with minor deviations.	Limited adherence to standards with several deviations.	Drawings do not adhere to industry standards.
Wireframe and Surface Design Workbench	**Creation and Manipulation of Wireframe Geometry**	Wireframe geometry is accurately created and manipulated with all necessary elements.	Mostly accurate wireframe geometry with minor errors.	Several inaccuracies in wireframe creation and manipulation.	Wireframe geometry is inaccurate and poorly manipulated.

	Complexity of Surfaces	Demonstrates high complexity in surface creation using various tools effectively.	Shows moderate complexity with a good variety of tools used.	Simple surfaces with limited use of tools.	Very basic surfaces with minimal use of tools.

This table provides a structured and systematic approach to assessing CATIA course exercises, ensuring clarity and consistency in evaluation.

SKETCHER EXERCISES



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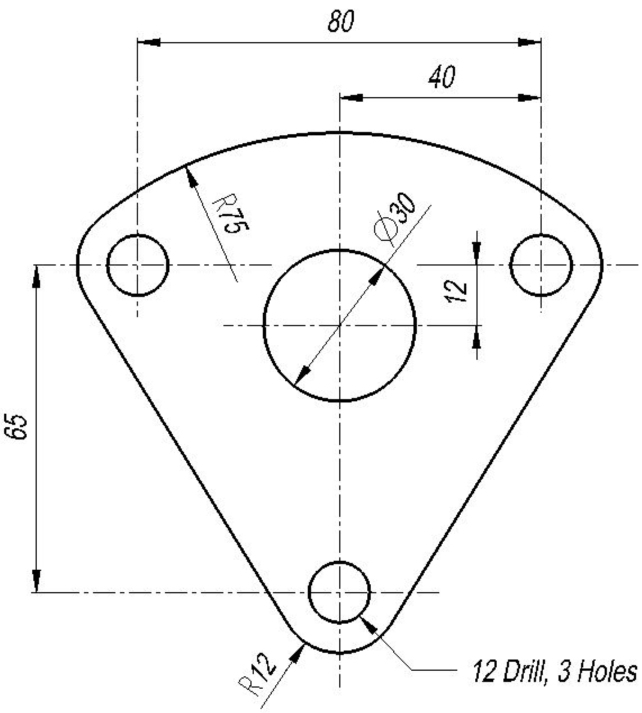

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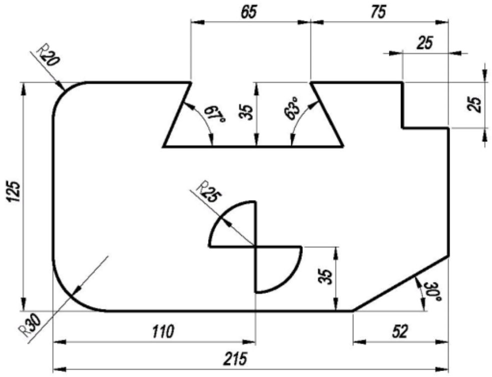

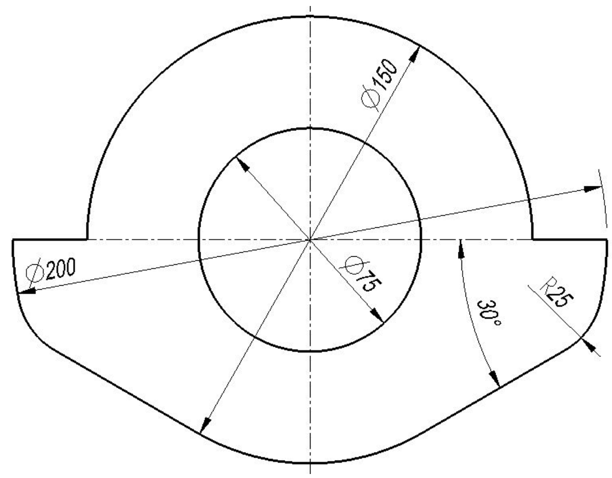

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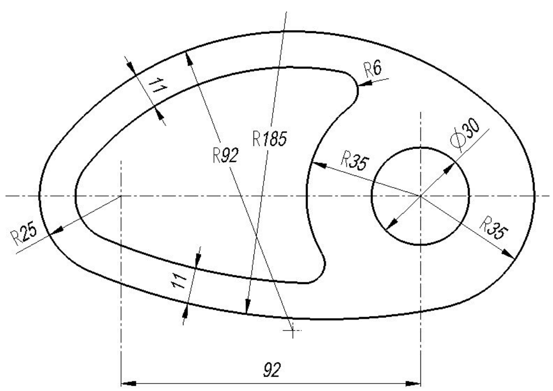

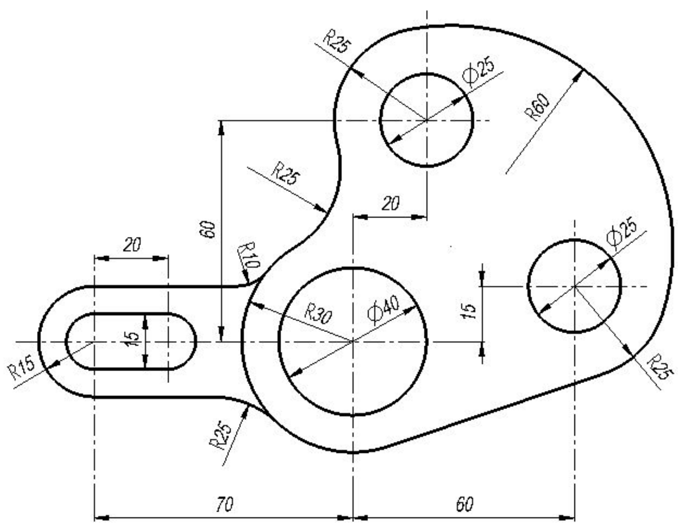

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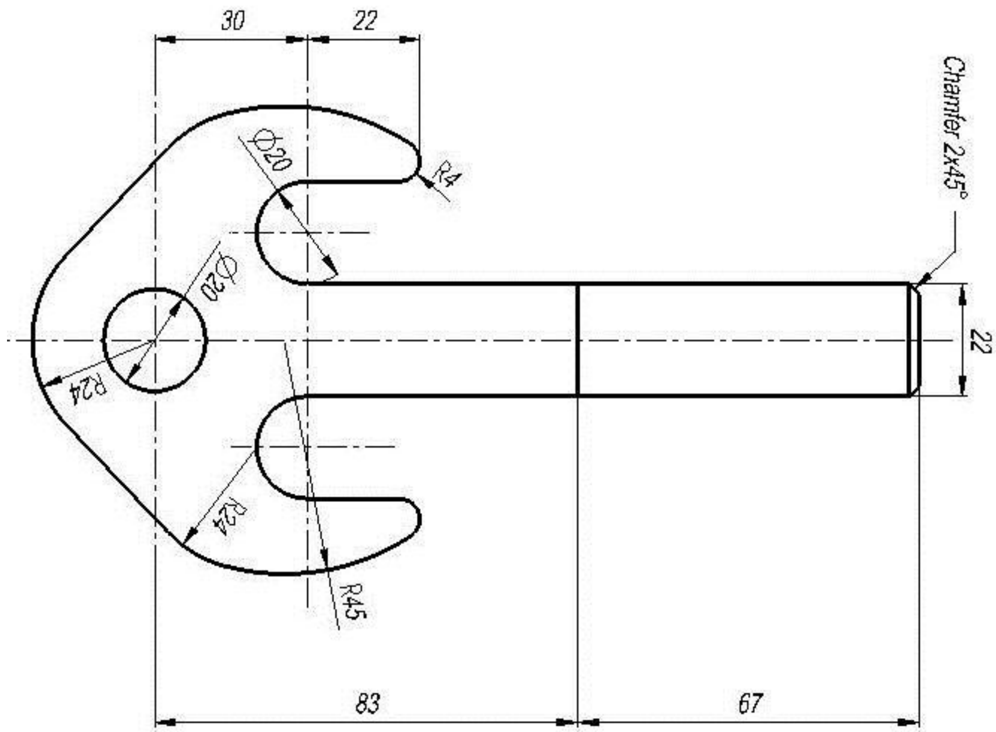

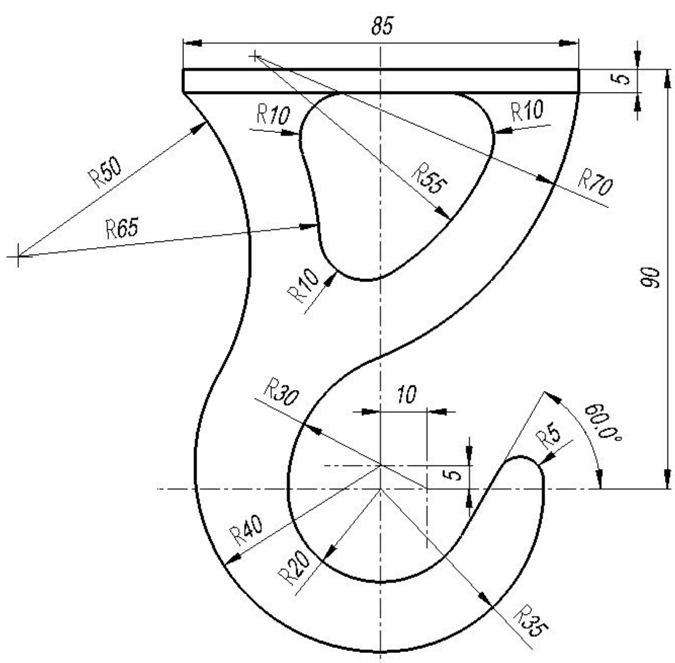

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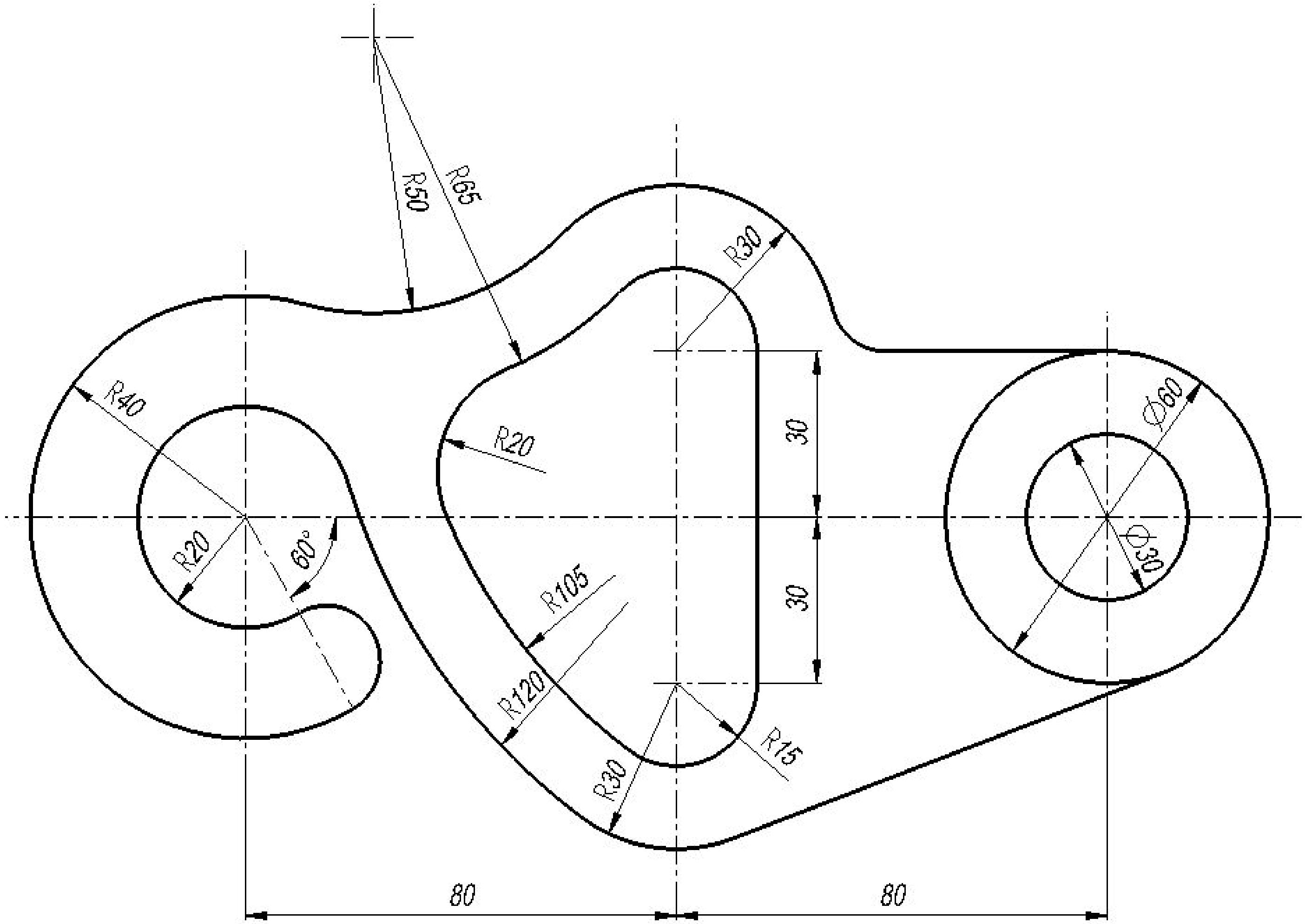

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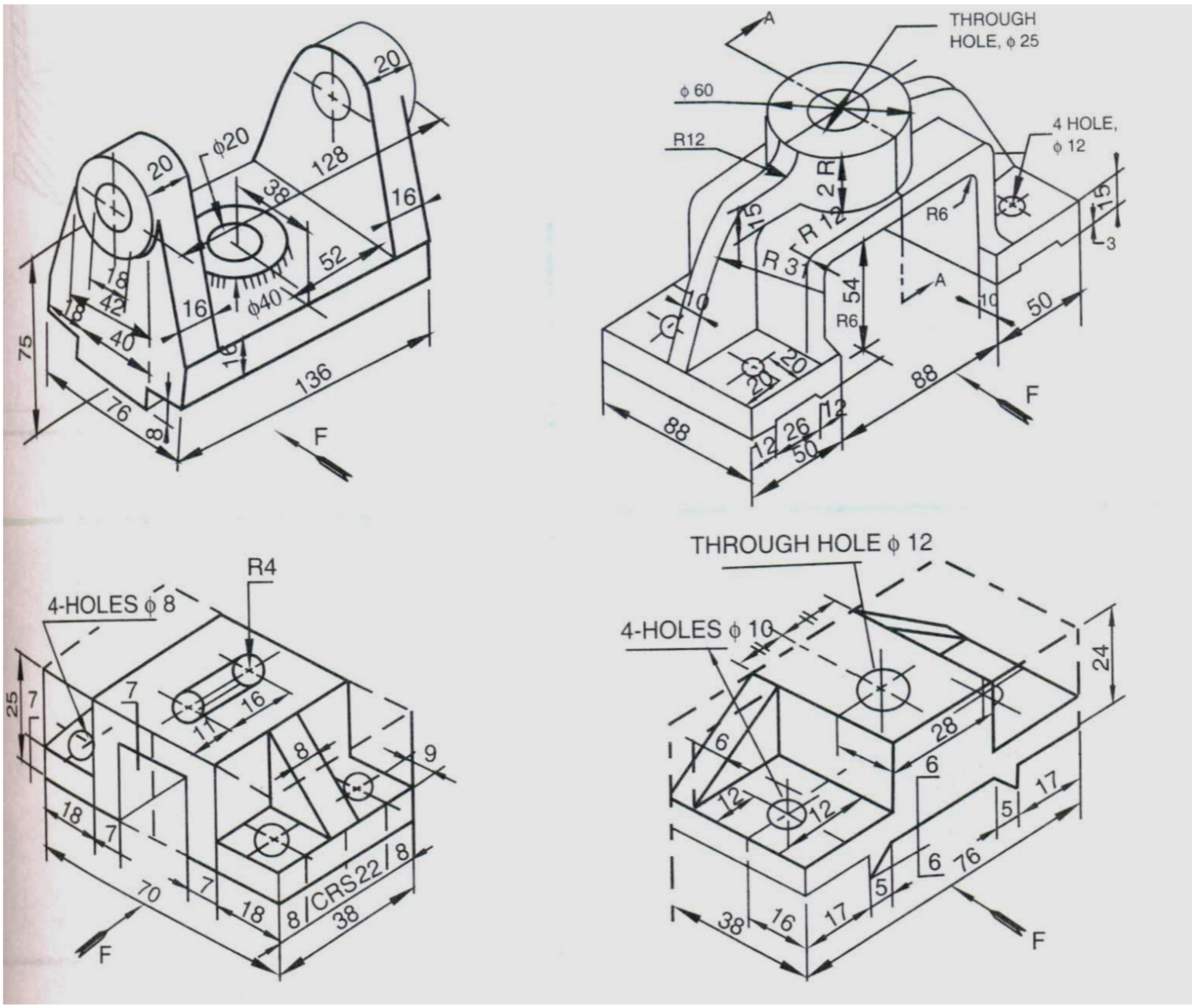
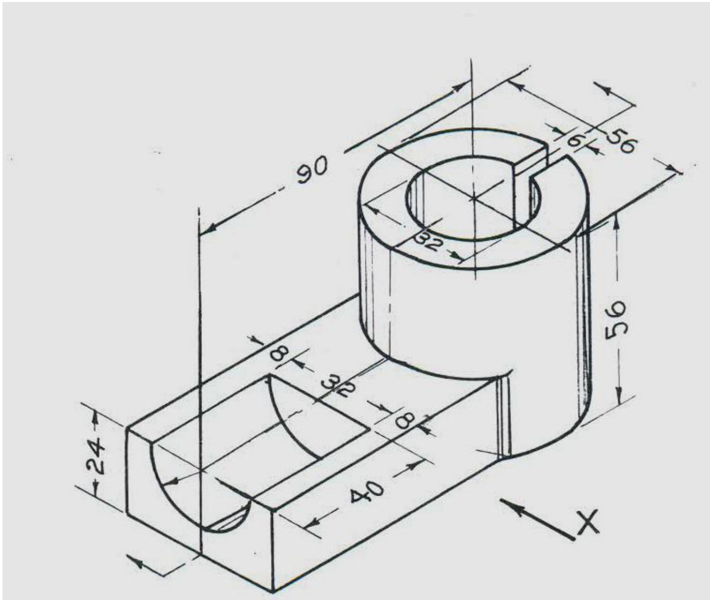
SOLID MODELLING EXERCISES

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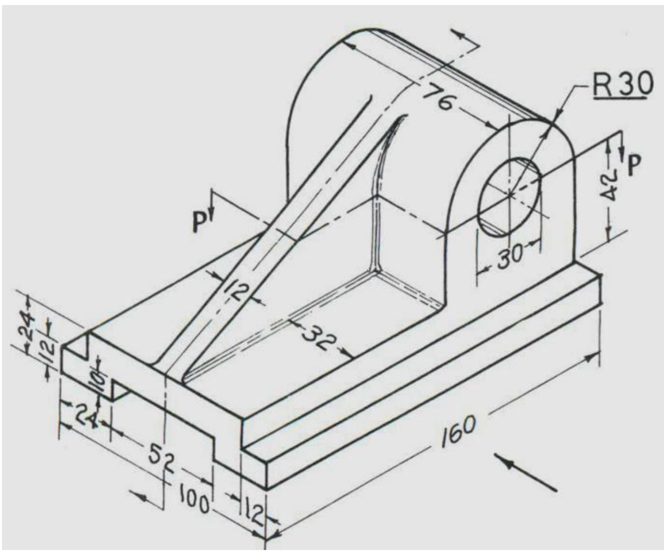
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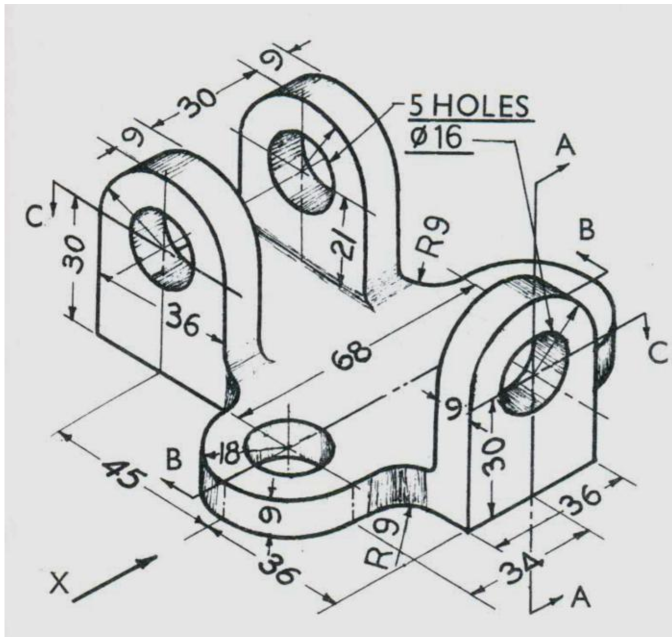
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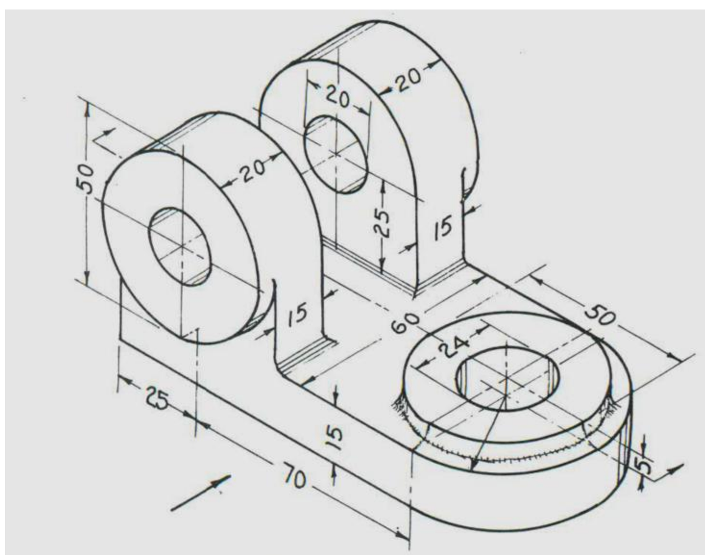
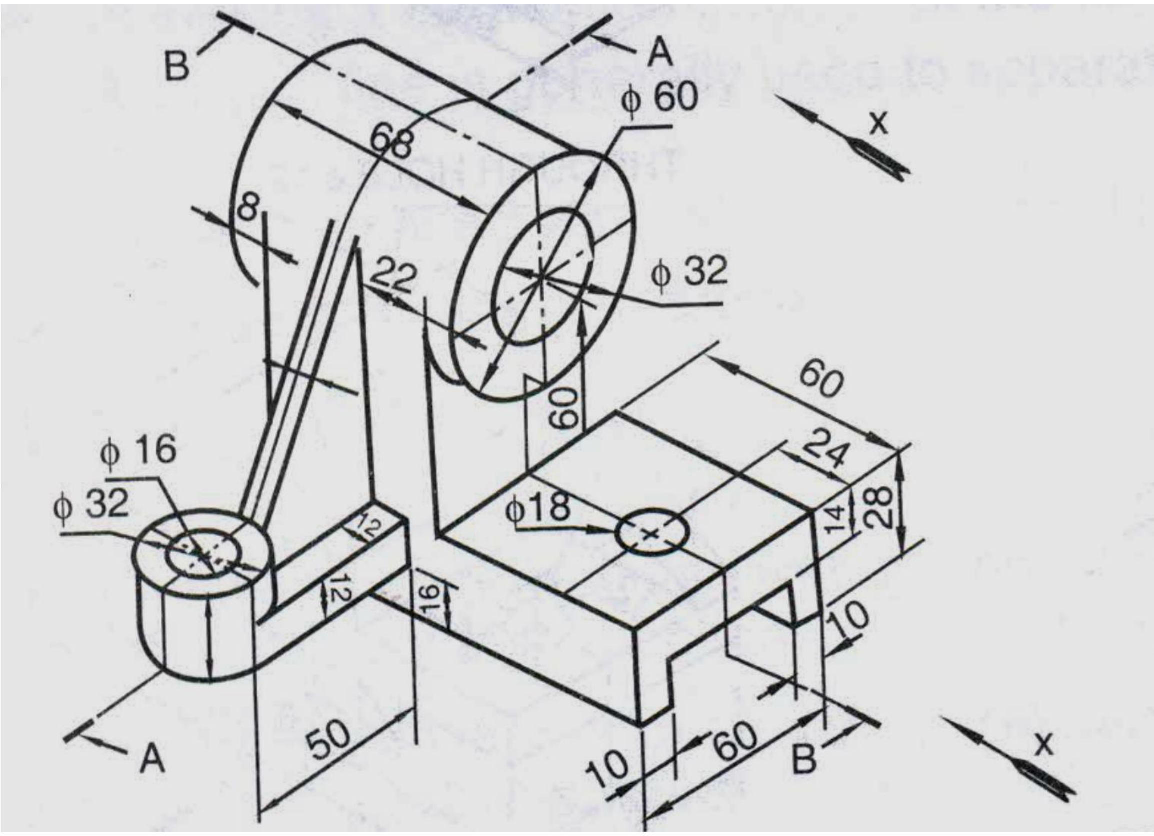
9
36
R9
34
36
A
X


20
20
20
50
25
15
15
60
50
24
25
15
70
5


B
A
ϕ 60
X
68
8
22
ϕ 32
60
60
24
ϕ 16
ϕ 32
ϕ18
14
28
12
12
16
10
A
50
10
60
B
X

Savitribai Phule Shikshan Prasarak Mandal's

SKN SINHGAD COLLEGE OF ENGINEERING, PANDHARPUR

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Gat.No.6643,Korti, Tal.Pandharpur, Dist.Solapur-413304Office Ph.No.02186-250146

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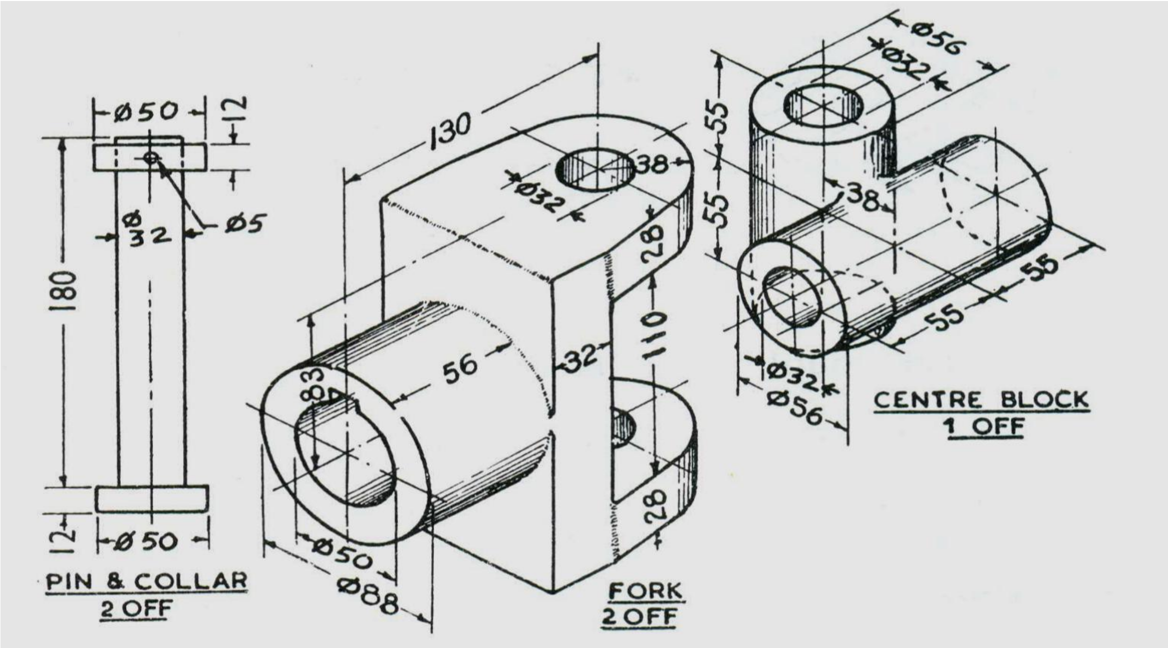
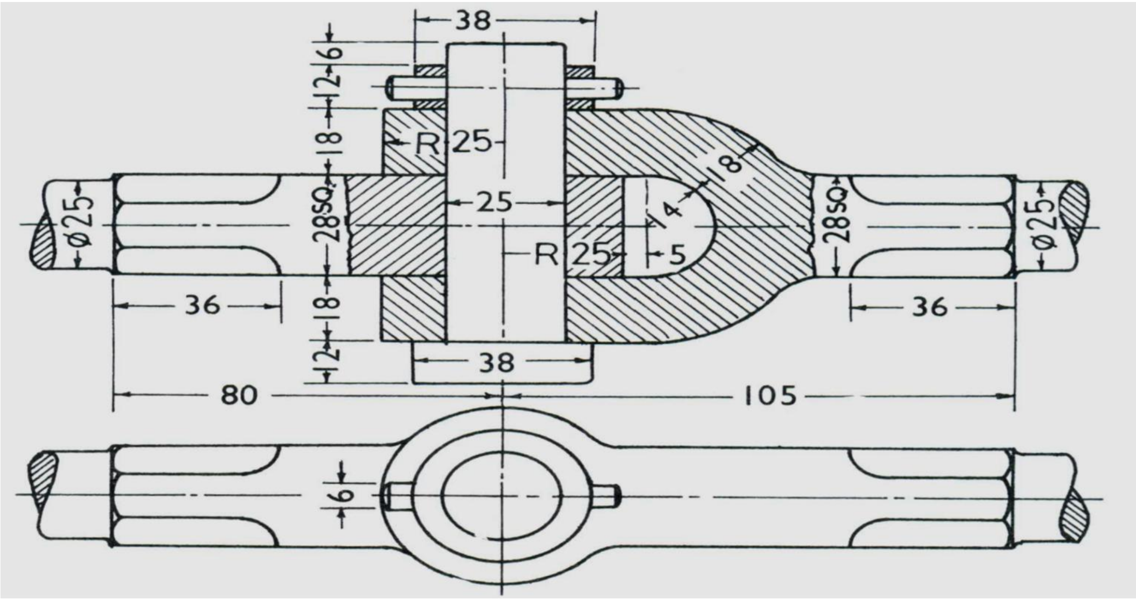
Assembly Exercise





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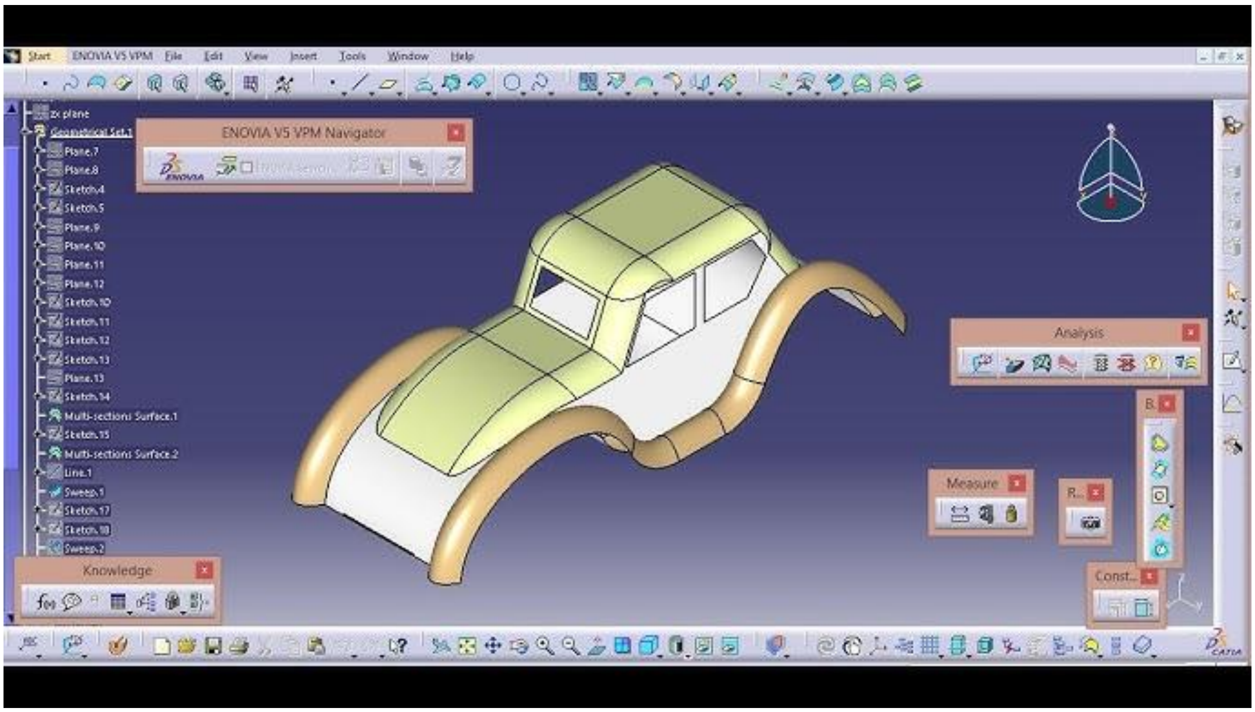
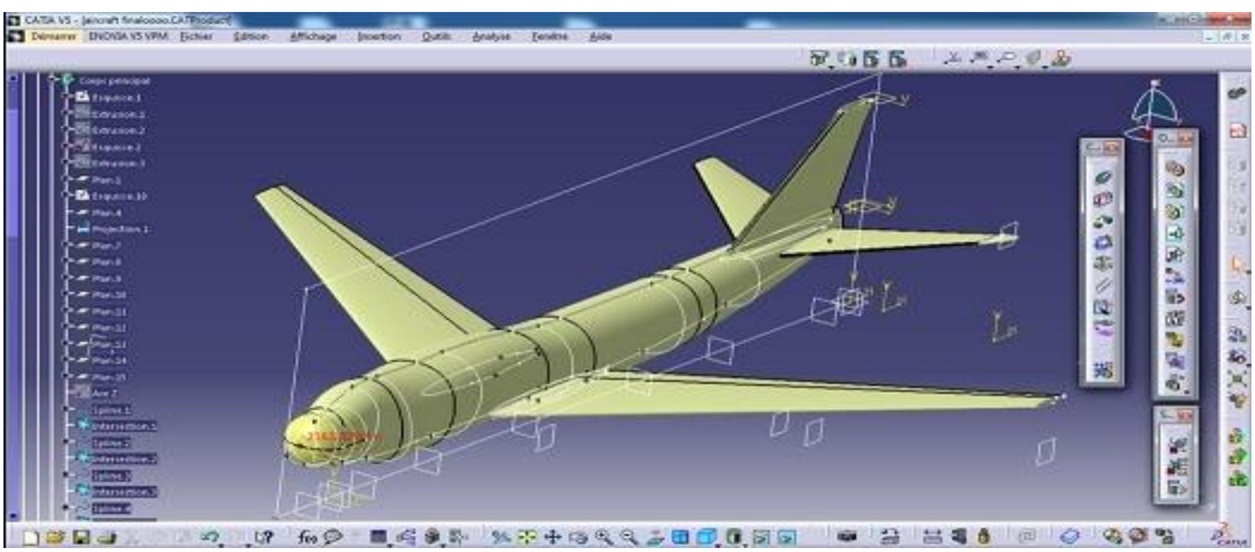
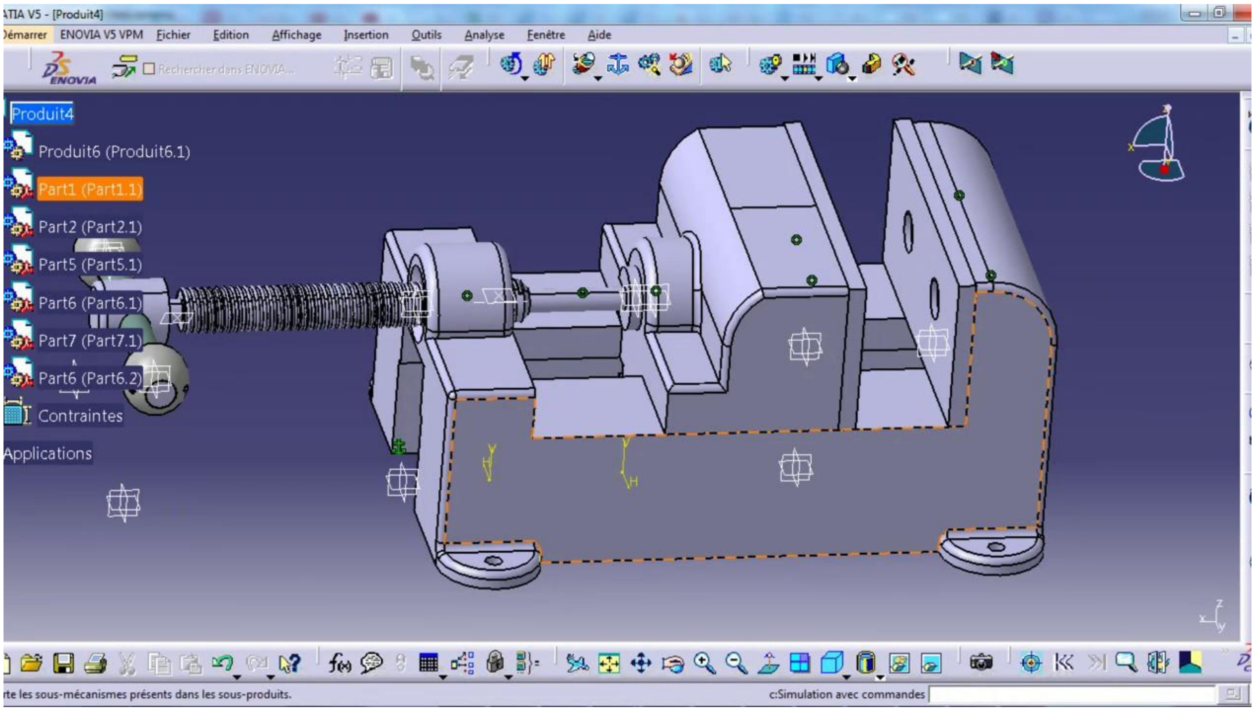
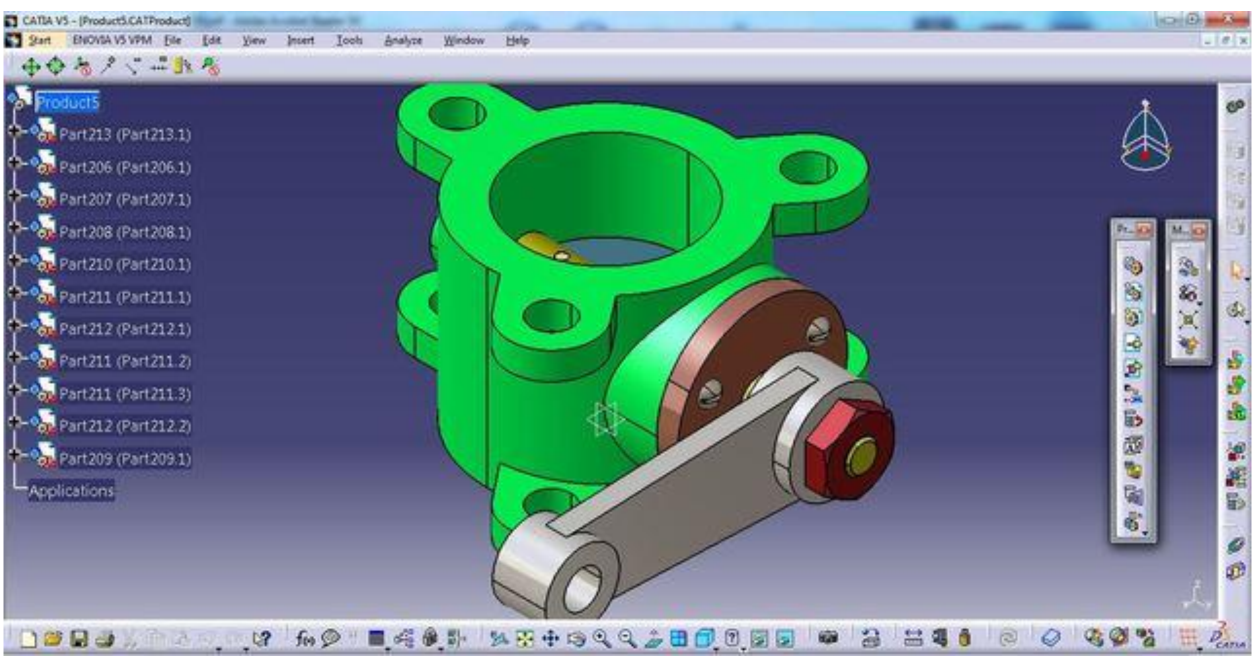
Project Executed by Student






ENOVIA V5 VPM
Fichier
Edition
Affichage
Insertion
Outils
Analyse
Fenêtre
Aide
Produit4
Produit6 (Produit6.1)
Part1 (Part1.1)
Part2 (Part2.1)
Part5 (Part5.1)
Part6 (Part6.1)
Part7 (Part7.1)
Part6 (Part6.2)
Contraintes
Applications
c:Simulation avec commandes


Start
ENOVIA V5 VPM
File
Edit
View
Insert
Tools
Analyze
Window
Help
Product5
Part213 (Part213.1)
Part206 (Part206.1)
Part207 (Part207.1)
Part208 (Part208.1)
Part210 (Part210.1)
Part211 (Part211.1)
Part212 (Part212.1)
Part211 (Part211.2)
Part211 (Part211.3)
Part212 (Part212.2)
Part209 (Part209.1)
Applications

Evaluation Report (Out of 100 Marks)

Category	Criteria	Rubrics	Prathamesh Bhoyate (Points)	Nikhil Chavan (Points)	Aadinath Salunkhe (Points)
Sketcher Workbench	**Accuracy of Sketches**	**Exemplary (8-10 points):** Sketches are precise and accurate with all geometric and dimensional constraints correctly applied. **Proficient (6-7 points):** Sketches are mostly accurate with minor errors in constraints. **Basic (4-5 points):** Sketches have several inaccuracies, and constraints are inconsistently applied. **Needs Improvement (0-3 points):** Sketches are inaccurate and lack proper constraints.	Exemplary (9.3/10)	Proficient (8/10)	Basic (5.3/10)
	Complexity of Sketches	**Exemplary (8-10 points):** Demonstrates high complexity using multiple tools and constraints effectively. **Proficient (6-7 points):** Shows moderate complexity with a good variety of tools and constraints used. **Basic (4-5 points):** Simple sketches with limited use of tools and constraints. **Needs Improvement (0-3 points):** Very basic sketches with minimal use of tools and constraints.	Proficient (7.3/10)	Basic (6/10)	Proficient (7/10)
Part Design Workbench	**Transformation of Sketches to 3D Models**	**Exemplary (8-10 points):** 3D models accurately reflect sketches with all features (extrude, revolve, fillet, chamfer) correctly applied. **Proficient (6-7 points):** 3D models are mostly accurate with minor errors in feature application. **Basic (4-5 points):** Several	Exemplary (9.2/10)	Basic (5.5/10)	Proficient (7/10)

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Prof.P.P.Kulkarni
Evaluator

Date:31/03/2022

		inaccuracies in transforming sketches to 3D models. **Needs Improvement (0-3 points):** 3D models do not accurately reflect sketches, and feature applications are incorrect.			
	Use of Advanced Features	**Exemplary (8-10 points):** Effectively uses advanced features such as patterns, mirrors, and complex transformations. **Proficient (6-7 points):** Uses some advanced features with moderate effectiveness. **Basic (4-5 points):** Limited use of advanced features. **Needs Improvement (0-3 points):** Does not use advanced features effectively.	Proficient (7/10)	Proficient (7/10)	Basic (5/10)
Assembly Design Workbench	**Correct Assembly of Parts**	**Exemplary (8-10 points):** Parts are assembled correctly with appropriate constraints, ensuring functional assemblies. **Proficient (6-7 points):** Most parts are assembled correctly with minor issues in constraints. **Basic (4-5 points):** Several parts are incorrectly assembled or missing constraints. **Needs Improvement (0-3 points):** Assemblies are incorrect and lack proper constraints.	Exemplary (9/10)	Basic (5.8/10)	Proficient (7/10)
	Complexity of Assembly	**Exemplary (8-10 points):** Demonstrates high complexity with multiple parts and intricate relationships. **Proficient (6-7 points):** Shows moderate complexity with a good number of parts and relationships. **Basic (4-5 points):** Simple assemblies with limited parts and	Exemplary (9/10)	Proficient (6.8/10)	Basic (5.7/10)

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		relationships. **Needs Improvement (0-3 points):** Very basic assemblies with minimal parts and relationships.			
Drafting Workbench	**Accuracy of 2D Drawings**	**Exemplary (8-10 points):** 2D drawings are accurate with all views, dimensions, and annotations correctly applied. **Proficient (6-7 points):** Drawings are mostly accurate with minor errors in dimensions or annotations. **Basic (4-5 points):** Several inaccuracies in drawings, dimensions, or annotations. **Needs Improvement (0-3 points):** Drawings are inaccurate and lack proper dimensions and annotations.	Proficient (8/10)	Basic (6/10)	Exemplary (9.2/10)
	Adherence to Standards	**Exemplary (8-10 points):** Drawings adhere to industry standards (e.g., ISO, ASME) with all necessary details included. **Proficient (6-7 points):** Drawings mostly adhere to standards with minor deviations. **Basic (4-5 points):** Limited adherence to standards with several deviations. **Needs Improvement (0-3 points):** Drawings do not adhere to industry standards.	Proficient (7.5/10)	Exemplary (8.8/10)	Basic (5.5/10)
Wireframe and Surface Design Workbench	**Creation and Manipulation of Wireframe Geometry**	**Exemplary (8-10 points):** Wireframe geometry is accurately created and manipulated with all necessary elements. **Proficient (6-7 points):** Mostly accurate wireframe geometry with minor errors. **Basic (4-5 points):** Several inaccuracies in wireframe creation and manipulation.	Proficient (7.7/10)	Exemplary (8.5/10)	Basic (5.5/10)

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		Needs Improvement (0-3 points): Wireframe geometry is inaccurate and poorly manipulated.			
	Complexity of Surfaces	**Exemplary (8-10 points):** Demonstrates high complexity in surface creation using various tools effectively. **Proficient (6-7 points):** Shows moderate complexity with a good variety of tools used. **Basic (4-5 points):** Simple surfaces with limited use of tools. **Needs Improvement (0-3 points):** Very basic surfaces with minimal use of tools.	Exemplary (9.3/10)	Proficient (7.8/10)	Basic (6/10)
Total Marks			**85.3/100**	**70.4/100**	**66.2/100**

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Gat.No.664,Korti,Tal.Pandharpur,Dist.Solapur-413304
Office Ph.No.02186-250146, Website:www.sknscoe.ac.in, Email-Id:principal @sknscoe.ac.in

Accredited by NAAC with 'A' Grade

Mechanical Engineering Department

Dept. of Mechanical Engineering SKNSCOE Korti Pandharpur 413304

CERTIFICATE

This is to certify that

Shendge Niyantaa Vinod

has successfully completed a short term course in

CATIA V6 R20

during the period from 17th Jan 2022 to 18th March 2022.

Course Contents	Duration
1) Introduction To Catia 2) Sketcher Workbench 3) Part Design Workbench 4) Wire Frame And Surface Design 5) Assembly Design Workbench 6) Generative And Interactive Drafting 7) Dmu Kinematics 8) Mini Project of Modelling and Animation of Universal Coupling	40HRS

Designed and Conducted by SKN SINHGAD COLLEGE OF ENGINEERING, PANDHARPUR.

Prof. P.P Kulkarni
Course Trainer

Prof.U.S.Gholap
Course Trainer

Dr.S.S.Kulkarni
Head of Department

Outcome

Placement of Students

Sr.No.	Name of Student	Name of Company	Ref.No.of Appointment
1	Aadinath Laxman Salunkhe	TE Connectivity India Pvt. Ltd.	31/Jan/24
2	Nikhil Nandkumar Chavan	Imco Alloy Pvt. Ltd. Mumbai	22/Mar/24
3	Madhavi Uttam Shende	Intence Engineering Pvt. Ltd.	IEPL2023/06/20/0270,dated 20-06-23