LAB MANUAL

Course Code: IPE 2110

Course Name: Computer-Aided Design (CAD) Sessional

Program: Industrial and Production Engineering (IPE)

AHSANULLAH UNIVERSITY OF SCIENCE AND TECHNOLOGY (AUST)

Introduction

Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM) are fundamental to modern engineering and industrial practices. CAD involves using computer software to create, modify, and optimize 2D and 3D designs with high precision, while CAM focuses on translating these designs into manufacturing instructions for processes such as CNC machining, cutting, or additive manufacturing. Together, CAD and CAM ensure seamless integration between product design and production, thereby reducing errors, costs, and lead times.

In this course, two industry-standard CAD platforms—**SolidWorks** and **CATIA**—will be used.
- SolidWorks is widely adopted in small to medium-scale industries for product design, machine parts, fixtures, and assembly drawings due to its user-friendly interface and parametric modelling capabilities.
- CATIA, developed by Dassault Systèmes, is a high-end CAD software used extensively in the aerospace, automotive, and advanced manufacturing sectors. It excels in handling complex surfaces, large assemblies, and design integration with engineering analysis and manufacturing simulations.

By learning both SolidWorks and CATIA, students will be exposed to tools ranging from beginner-friendly 2D and 3D modelling environments to advanced design platforms used in global industries. This combination ensures versatility and prepares students to adapt to diverse professional requirements.

Relevance to Industrial and Production Engineering (IPE):
1. Product and Component Design (e.g., gearbox housing, machine fixture in SolidWorks).
2. Design for Manufacturing and Assembly (e.g., automotive dashboard assembly in CATIA).
3. Process Planning and CAM Integration (e.g., CNC toolpaths for a die/mold).
4. Simulation and Virtual Validation (e.g., robotic arm simulation in CATIA).
5. Industrial Applications: SolidWorks for SMEs, CATIA for aerospace, automotive, and shipbuilding industries.

By mastering SolidWorks and CATIA, IPE students gain the dual advantage of practical design competency and exposure to industry-standard advanced platforms.

Scope of the Lab

This lab is designed to provide students with practical exposure to CAD modelling using SolidWorks and CATIA, focusing on the fundamental steps of computer-aided design. The lab sessions are structured to progressively build the students’ skills, starting from basic sketching in 2D, advancing to complex 2D profiles, and finally transitioning to 3D solid modelling. The scope of this lab extends beyond just drawing practice. It equips students with a systematic workflow—from interpreting engineering drawings to generating precise CAD models—that is essential for industrial applications.

Expected Learning Outcomes

- Identify and navigate CAD interfaces (SolidWorks and CATIA) and understand their application contexts.

- Create fully defined 2D sketches using geometric constraints and dimensional relations.

- Apply advanced sketching techniques (mirror, pattern, fillet, offset) to generate complex 2D profiles efficiently.

- Develop basic 3D models using sketch-based features such as Extrude, Revolve, and Cut, and visualize them in multiple orientations.

- Interpret and replicate technical drawings into digital CAD models with accuracy.

- Establish a foundation for CAM integration by understanding how digital models transition into toolpaths and manufacturing processes.

- Relate CAD skills to Industrial and Production Engineering practices, such as product design, process planning, fixture development, and system optimization.

Laboratory Rules and Regulations for Students

1. **Punctuality and Attendance**
   - Students must enter the lab on time and remain present for the full duration of the session.
   - Late entry beyond 10 minutes may result in loss of attendance for that class.
2. **Preparation Before Class**
   - Review the lab manual and assigned topics before each experiment.
   - Bring necessary materials (pens, notebooks, storage device).
3. **Conduct Inside the Lab**
   - Maintain silence and avoid unnecessary movement inside the lab.
   - Mobile phones must be kept on silent mode and should not be used for non-academic purposes.
4. **Computer and Software Usage**
   - Use only the assigned computer. Do not change settings or install unauthorised software.
   - Save your work in designated folders or storage devices before leaving.

    - Report immediately if a computer or software malfunctions.
5. **Handling of Lab Resources**
    - Do not eat, drink, or bring food inside the lab.
    - Handle all equipment with care. Misuse or negligence may lead to disciplinary action.
6. **Classwork and Assignments**
    - Complete the given classwork during the session and submit it as instructed.
    - Ensure your drawings are neat, accurate, and properly saved under your student ID.
7. **Academic Integrity**
    - Plagiarism or copying of assignments from others will not be tolerated.
    - Each student must complete their own work, even if collaboration is encouraged.
8. **Q&A and Interaction**
    - Actively participate in Q&A sessions at the end of each experiment.
    - Clarify doubts with the instructor during the session rather than outside.
9. **Exit Procedure**
    - Close all software and shut down the computer properly.
    - Ensure your workstation is clean before leaving the lab.

Percentages of Assessment Methods

Method	Percentage
Class Performance / Class Drawing	10
Project Submission SolidWorks	15
Project Submission Catia	15
Final Quiz (SolidWorks + Catia)	50
One Project Submission on 3D Printing from CIM Lab (Either SolidWorks or Catia)	10

Experiment 1: Introduction to SolidWorks Interface and Basic 2D Drawing

Objective

- To become familiar with the SolidWorks interface and sketching environment.
- To create simple 2D sketches using lines, circles, rectangles, and dimensions.
- To understand the concept of fully defined sketches.

Software Tools Used

- SolidWorks Software. See the software requirements here: https://www.solidworks.com/support/system-requirements
- Sketcher (Line, Circle, Rectangle, Trim, Mirror, Smart Dimension)

Theory

SolidWorks provides a parametric modelling environment where 2D sketches form the basis of 3D models. A 2D sketch must be fully defined using geometric relations (horizontal, vertical, perpendicular) and dimensions to ensure design accuracy. This experiment focuses on building simple 2D shapes and learning how to control them with constraints and dimensions.

Procedure

1. Open SolidWorks → Start a new Part file.
2. Select the **Front Plane** → enter **Sketch mode**.
3. Draw basic shapes such as rectangle, circle, and polygon as per provided drawing sheet.
4. Apply **Smart Dimension** to assign correct sizes.
5. Use **Geometric Relations** (horizontal, vertical, tangent) to fully define the sketch.
6. Use **Mirror Entities** to replicate symmetric features.
7. Save the sketch as ID_Exp1_Sketch.SLDPRT.

Task

Recreate the provided 2D sketches on the drawing sheet using SolidWorks tools and ensure all sketches are fully defined.

Expected Output

A set of fully defined 2D sketches created in SolidWorks, matching the provided dimensions.

Viva Questions

1. What does it mean for a sketch to be fully defined in SolidWorks?
2. Why are geometric relations important in 2D sketching?
3. What is the difference between the **Trim** and **Mirror Entities** tools?
4. Can a 2D sketch be left under-defined? What are the possible consequences?

2D Basic Drawings for Practice

Figure: 1

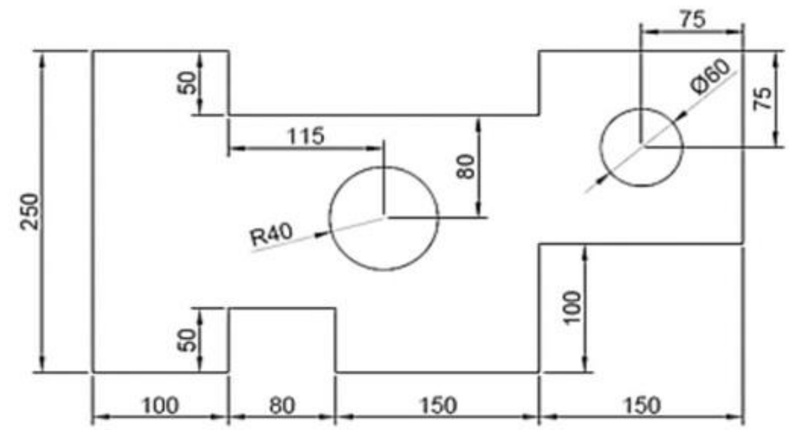


Figure:1.1

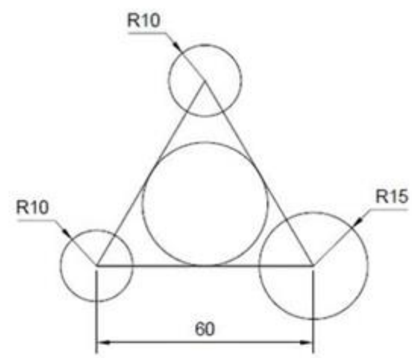
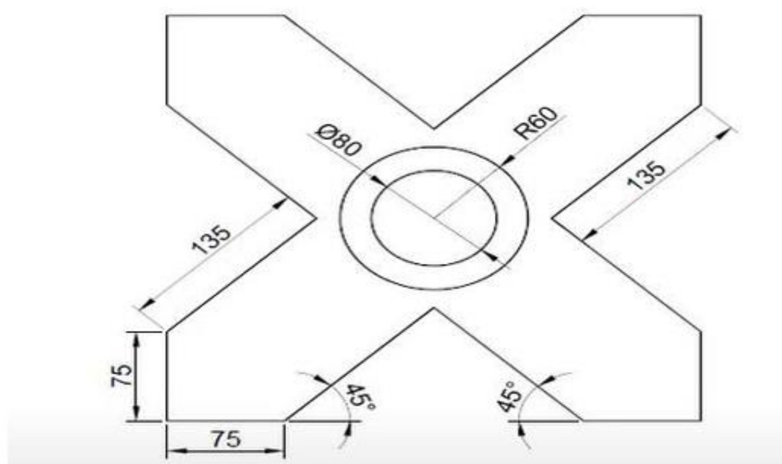


Figure: 1.2

Figure: 2



Experiment 2: Moderate to Advanced 2D Drawing in SolidWorks

Objective

- To create moderately complex 2D sketches with multiple features.
- To use advanced sketching tools like mirror, offset, fillet, and pattern.
- To practice geometric relations such as tangent, concentric, equal, and perpendicular.

Software Tools Used

- Sketcher (Line, Circle, Arc, Polygon, Fillet, Mirror, Offset, Smart Dimension)

Theory

Advanced 2D sketches in SolidWorks combine multiple geometric entities and constraints. Proper use of geometric relations and dimensions ensures the sketch is fully defined and ready for 3D modelling. Mirror and pattern tools increase efficiency by replicating features without redrawing.

Procedure

1. Open SolidWorks → Start a new Part file.
2. Select the **Front Plane** → enter **Sketch mode**.
3. Draw the complex shape as shown on the instructor-provided drawing sheet.
   Apply **Fillets** to rounded corners as required.
4. Use **Mirror Entities** to replicate symmetric parts across a centerline.
5. Apply **Offset** and **Pattern** tools to repeat shapes efficiently.
6. Assign dimensions using **Smart Dimension**.
7. Verify that all sketches are **fully defined**.
8. Save the sketch as ID_Exp2_.SLDPRT.

Task

Recreate the advanced 2D sketch from the drawing sheet, ensuring all dimensions and constraints match exactly.

Expected Output

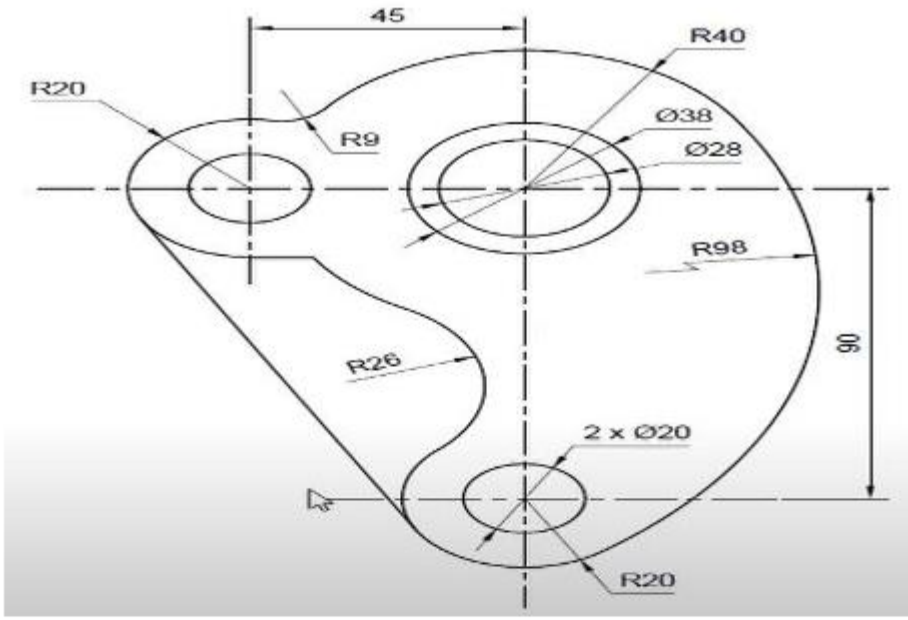
- A fully defined complex 2D sketch that matches the technical drawing.

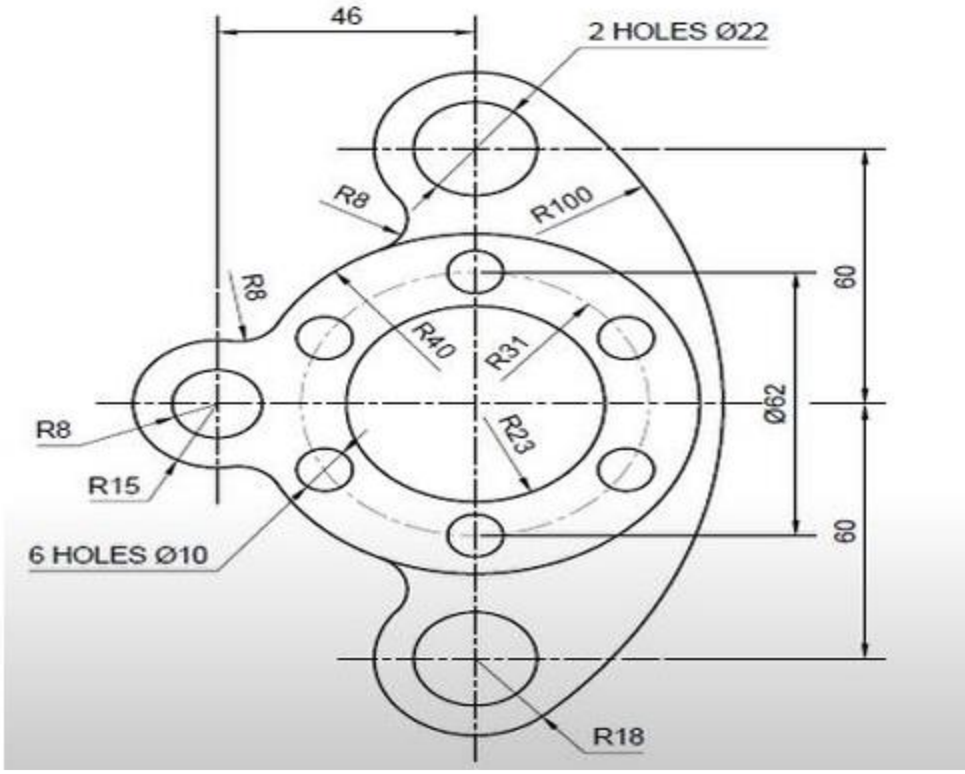
Viva Questions

1. What is the purpose of the mirror tool in sketching?

2. How do tangent and concentric relations improve sketch accuracy?
3. Why is it important to fully define a complex 2D sketch?
4. How do offset and pattern tools help save time in sketching?

2D Moderate to Advanced Drawings for Practice




46
2 HOLES Ø22
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R100
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R40
R31
Ø62
R8
R23
R15
6 HOLES Ø10
60
R18

Experiment 3: Introduction to 3D Modeling and Basic 3D Drawing

Objective

- To convert 2D sketches into 3D solid models.
- To practice basic 3D features: Extrude, Revolve, Cut-Extrude, and Fillet.
- To learn navigation in 3D space (rotate, zoom, standard views).

Software Tools Used

- Part Design (Extruded Boss/Base, Revolved Boss/Base, Cut-Extrude, Fillet)

Theory

3D modelling in SolidWorks starts from a 2D sketch and extends it into three dimensions. Features like **Pad/Extrude** and **Revolve** allow designers to create solid parts, while **Cut-Extrude** removes material. Edge fillets smooth transitions and improve manufacturability. Parametric modelling ensures that changes to the sketch automatically update the 3D model.

Procedure

1. Open SolidWorks → Start a new Part file.
2. Create a **base sketch** on the Front Plane.
3. Use **Extruded Boss/Base** to create a solid block from the sketch.
4. Sketch additional profiles on selected faces and use **Pad** or **Cut-Extrude** to add or remove features.
5. Use **Revolved Boss/Base** for cylindrical or rotational features if required.
6. Apply **Fillet** to smooth edges or corners.
7. Navigate using isometric, front, and top views to verify the model.
8. Save the model as ID_Exp3.SLDPRT.

Task

Create a simple 3D model based on the 2D drawing provided. Apply all required pads, cuts, and fillets.

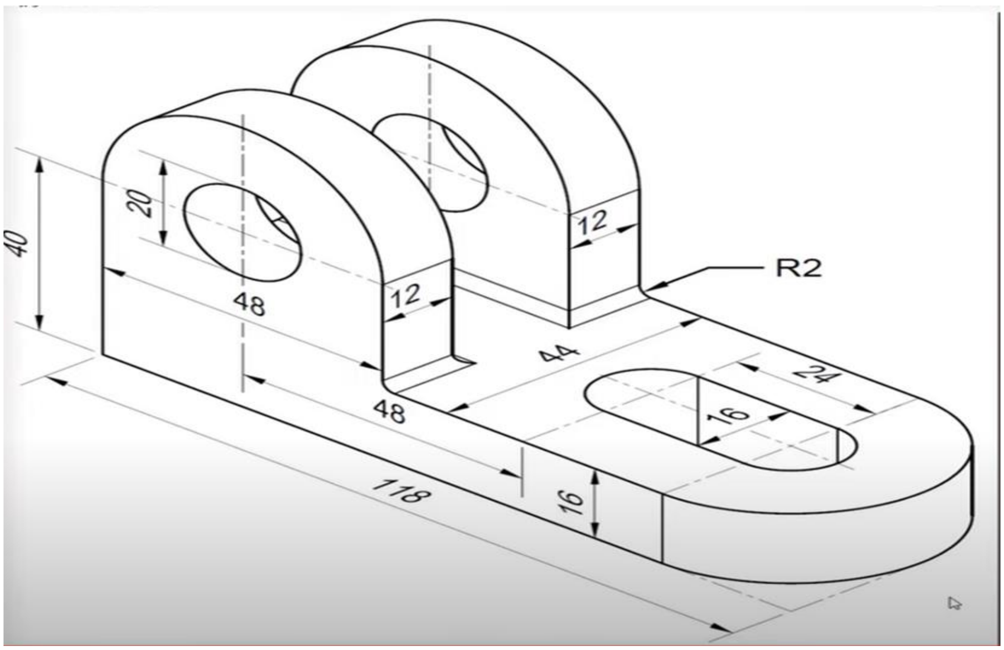
Expected Output

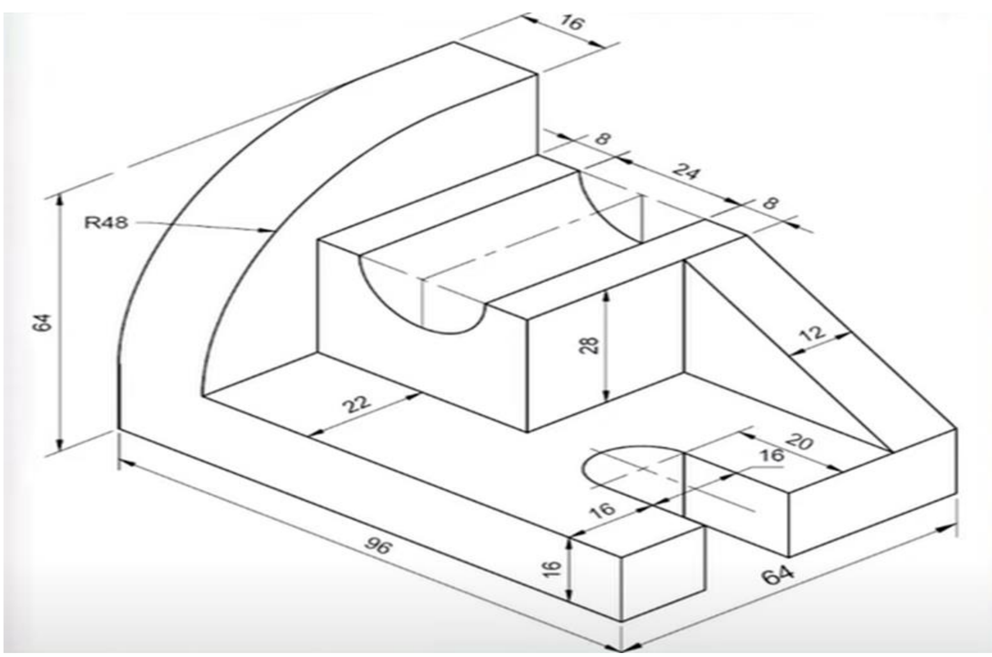
- A 3D solid model with accurate dimensions, features, and smooth edges ready for visualization or further processing.

Viva Questions

1. What is the difference between the Extrude and Revolve features?
2. How do reference planes help in 3D modelling?
3. Why is an isometric view important when inspecting a model?
4. How does parametric modelling in SolidWorks facilitate design changes?

3D Basic Drawings for Practice





Experiment 4: Moderate to Advanced 3D Drawing in SolidWorks

Objective

- To practice intermediate and advanced 3D modelling techniques in SolidWorks.
- To learn and apply advanced features such as **Loft, Sweep, Revolve, Revolve Cut, Shell, Rib, Draft, Chamfer, Fillet, and Pattern**.
- To explore feature editing and design intent using parametric relationships.

Software Tools Used

- **Part Design:** Lofted Boss/Base, Swept Boss/Base, Revolved Boss/Base, Revolved Cut, Shell, Rib, Draft, Chamfer, Fillet, Linear/Circular Pattern.

- **View Tools:** Section Views, Isometric Views, Zoom/Pan/Rotate.

Theory

Advanced 3D modelling in SolidWorks extends beyond simple extrusions.

- **Revolved Boss/Base** creates solid features by revolving a sketch profile around an axis.
- **Revolved Cut** removes material by revolving a closed or open profile around an axis.
- **Loft and Sweep** are used for complex shapes between multiple profiles or along paths.
- **Shell** hollows out solid bodies with a defined wall thickness.
- **Rib** adds thin supporting walls to strengthen the structure.
- **Draft** applies taper angles to faces, improving manufacturability.
- **Fillet and Chamfer** smooth sharp corners and edges.
- **Patterns** replicate features across the model efficiently. Parametric modelling ensures design intent is preserved during modifications.

Procedure

1. Start a **new Part file** in SolidWorks.
2. Create the base sketch and generate a **Revolved Boss/Base** around a central axis.
3. Create additional features using **Revolved Cut** to remove material.
4. Use **Lofted Boss/Base** between different profiles for smooth transitions.
5. Create a path curve and apply **Swept Boss/Base** for complex profiles along a path.
6. Apply **Shell** to hollow out the model.
7. Add **Rib** features where reinforcement is needed.
8. Use **Draft** on faces that require tapering.
9. Apply **Fillet and Chamfer** to refine sharp edges.
10. Use **Linear and Circular Patterns** to replicate holes, ribs, or slots.
11. Verify the model with **Isometric and Section Views**.
12. Save the model as **ID_Exp4.SLDPRT**.

Task

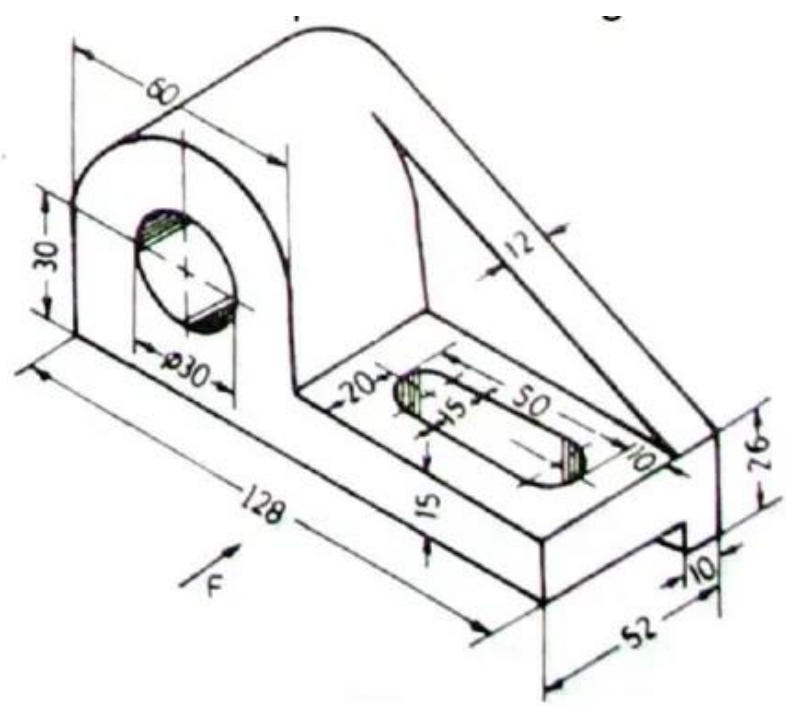
- Create a moderately complex 3D component (e.g., a mechanical bracket, bottle shape, or pulley) using **Revolve, Revolved Cut, Loft, Sweep, Shell, Rib, Fillet, Chamfer, and Pattern**.
- Check the design using **different views and section cuts**.

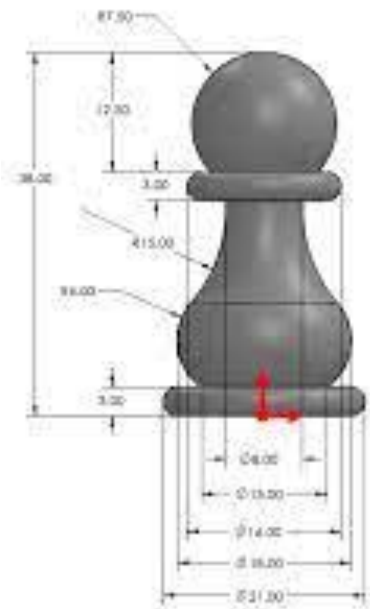
Expected Output

- A **3D solid model** incorporating **Revolved Boss/Base and Revolved Cut** along with advanced features like Loft, Sweep, Shell, Rib, Draft, Fillet, Chamfer, and Patterns.
- A parametric and fully editable model with accurate dimensions.

Viva Questions

1. Differentiate between Revolved Boss/Base and Revolved Cut.
2. What kind of shapes are best modelled using the Revolve feature?
3. Why is Shell preferred over multiple Cut-Extrudes when creating hollow parts?
4. How do Fillet and Chamfer improve manufacturability and safety?
5. Compare Loft and Sweep in terms of application.
6. How do Patterns help in reducing modelling time?

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Experiment 5: Standard and Advanced Assembly Features in SolidWorks

Objective

- To assemble multiple parts using **Standard and Advanced Mates**.
- To visualize assemblies using **Exploded Views** and **Section Views**.
- To create a realistic presentation using **Rendering tools**.
- To study basic motion between assembled components using **Motion Study**.

Software Tools Used

- **Assembly Design:** Insert Components, Standard Mates (Coincident, Concentric, Parallel, Distance, Angle), Advanced Mates (Limit, Symmetry, Path).
- **Visualization:** Exploded Views, Section Views, Rendering (PhotoView 360 / SolidWorks Visualize).
- **Simulation:** Motion Study (Basic Animation of mates and movements).

Theory

Assemblies in SolidWorks are created by combining individual parts with defined relationships:

- **Standard Mates**: Basic constraints like coincident, concentric, parallel, distance, and angle that fix part positions.
- **Advanced Mates**: More complex constraints such as limit (restricts movement), symmetry (aligns components symmetrically), and path (constrains movement along a curve or edge).
- **Exploded Views**: Show the separation of parts for better visualization and documentation.
- **Section Views**: Provide internal inspection of assemblies by cutting through them.
- **Rendering**: Adds realistic material, light, and background effects for presentations.
- **Motion Study**: Simulates the movement of parts in the assembly based on applied mates and motion constraints.

Procedure

1. Open SolidWorks → Start a **new Assembly file**.
2. Insert the **base component** and fix it in position.
3. Insert additional parts and apply **Standard Mates** (Coincident, Concentric, Parallel, Distance, Angle) to position them.
4. Apply **Advanced Mates** (Limit, Symmetry, Path) where needed to simulate realistic motion.
5. Create an **Exploded View** to visualize how parts fit together.

6. Use a **Section View** to check internal alignment and clearances.
7. Apply **materials and appearances**, then generate a **rendered view** using PhotoView 360 or SolidWorks Visualize.
8. Perform a **Motion Study** to demonstrate relative motion (e.g., sliding, rotation).
9. Save the assembly as **ID_Exp5.SLDASM**.

Task

- Assemble at least **three or more components** (e.g., bolt–nut–plate, piston–cylinder–connecting rod).
- Apply a combination of **Standard and Advanced Mates**.
- Generate an **Exploded View** and a **Section View**.
- Apply **rendering** to produce a realistic image of the assembly.
- Create a short **Motion Study** (e.g., piston movement, bolt tightening).

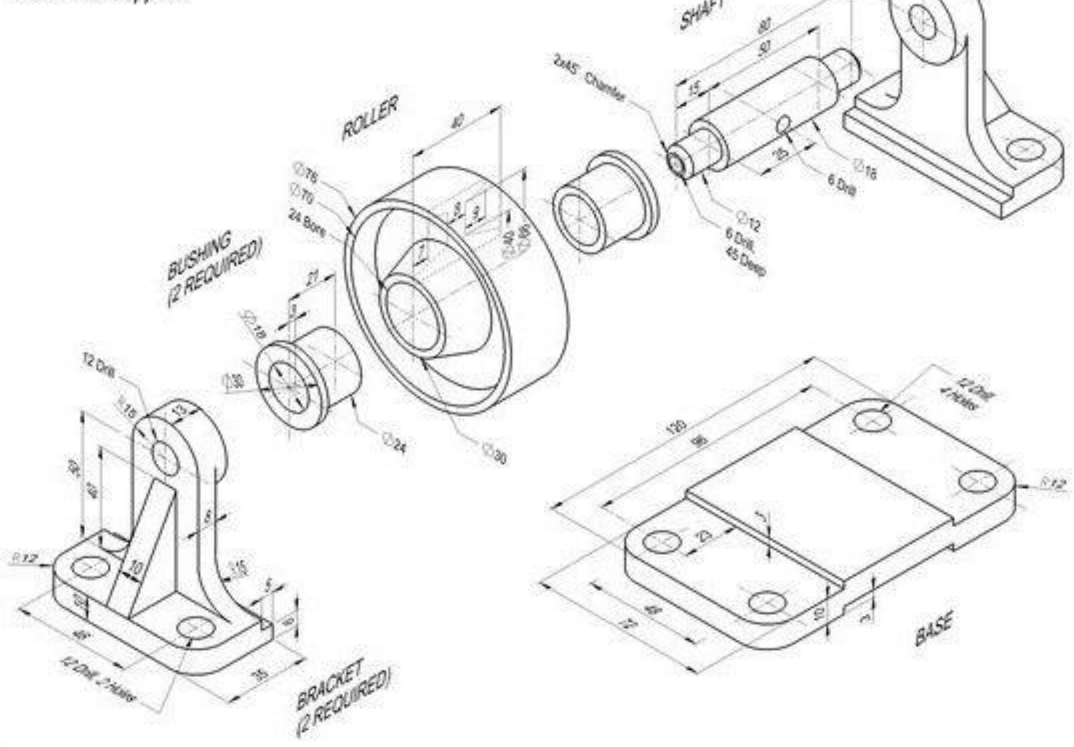
Expected Output

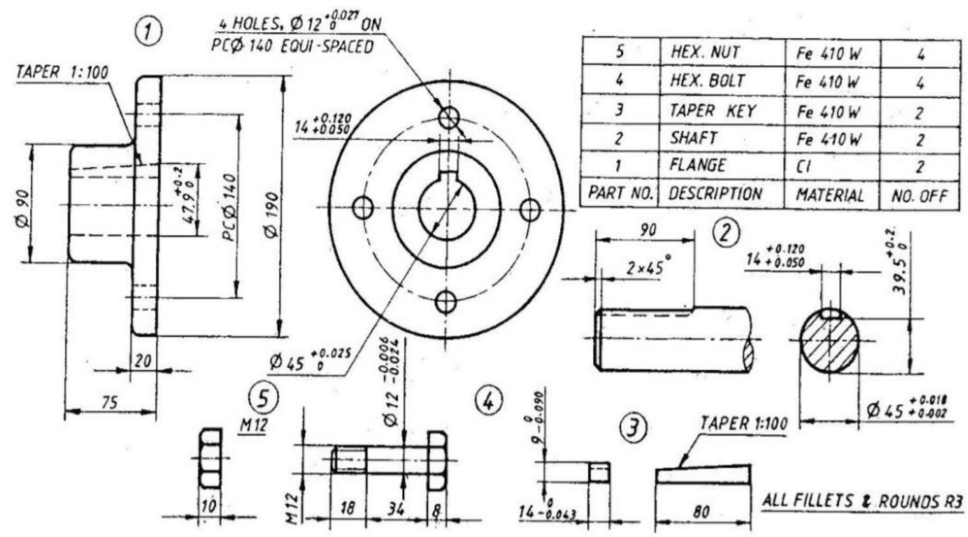
- A **fully constrained assembly** using both standard and advanced mates.
- Clear visualization with **Exploded View** and **Section Views**.
- A **rendered image** of the assembly.
- A **basic motion simulation** showing relative movement of parts.

Viva Questions

1. Differentiate between Standard Mates and Advanced Mates in assemblies.
2. What is the purpose of a Coincident Mate and a Concentric Mate?
3. How does a Limit Mate differ from a Distance Mate?
4. Explain the use of Symmetry and Path mates in assemblies?
5. How does a Motion Study enhance understanding of assemblies?

Belt Roller Support





PART NO.	DESCRIPTION	MATERIAL	NO. OFF
5	HEX. NUT	Fe 410 W	4
4	HEX. BOLT	Fe 410 W	4
3	TAPER KEY	Fe 410 W	2
2	SHAFT	Fe 410 W	2
1	FLANGE	CI	2

All Dimensions in mm
Details of Flanged Coupling — Unprotected Type

Experiment 6: Stress–Strain Analysis in SolidWorks

Objective

- To understand how to perform stress–strain analysis using SolidWorks Simulation.
- To apply loads and boundary conditions to a 3D model.
- To evaluate stress distribution, strain, and deformation under given loading conditions.
- To interpret results using plots, factor of safety, and reports.

Software Tools Used

- **SolidWorks SimulationXpress** or **SolidWorks Simulation** add-in.
- Tools: Fixtures (constraints), Loads (forces/pressures), Mesh generation, Stress/Strain plots, Displacement plots, Factor of Safety analysis.

Theory

Stress–strain analysis helps engineers predict the behavior of a component under loading.

- **Stress** is the internal resistance force per unit area ($\sigma = F/A$).
- **Strain** is the deformation per unit length ($\varepsilon = \Delta L/L$).
- In SolidWorks Simulation:
    - **Fixtures** represent boundary conditions (fixed, roller, hinge).
    - **Loads** simulate forces, pressures, or torques applied to the model.
    - **Meshing** divides the model into finite elements for numerical calculation.
    - **Stress and strain plots** show critical regions in the design.
    - **Factor of Safety** helps determine whether the design can withstand applied loads.

Finite Element Analysis (FEA) in SolidWorks solves governing equations at each mesh element to provide accurate stress–strain results.

Procedure

1. Open SolidWorks → Load the part or create a simple 3D component (e.g., a bracket or beam).
2. Go to **Tools → Add-ins → SolidWorks Simulation** and enable it.
   Start a **New Study → Static Analysis**.
3. Assign **material properties** (e.g., Steel, Aluminum).
4. Apply **Fixtures** (e.g., fix one face or edge of the model).
5. Apply **Loads** (force, pressure, or torque) to the required faces.
6. Generate a **Mesh** (automatic or manual).
7. Run the **Simulation**.
8. Observe results:
    - **Stress Plot** (von Mises stress distribution).
    - **Strain Plot**.
    - **Displacement Plot**.

- **Factor of Safety**.

9. Create a **Report** (optional: save as Word or PDF).
10. Save the study with filename **ID_Exp6_StressAnalysis.SLDPRT** or **.SLDASM** (if assembly).

Task

- Perform stress–strain analysis on a **bracket** or **cantilever beam**.
- Apply a fixed support at one end and a force/pressure at the other.
- Generate and study the **Stress, Strain, and Displacement plots**.
- Report maximum values and identify critical regions.

Expected Output

- Stress (von Mises) distribution across the model.
- Strain and displacement plots.
- Factor of Safety for the design.
- Clear identification of maximum stress and deformation regions.

Viva Questions

1. What is the difference between stress and strain?
2. Why do we use von Mises stress in SolidWorks Simulation?
3. How do boundary conditions affect simulation results?
4. What is the significance of Factor of Safety in design?
5. Why is material property assignment important in simulation?
6. How can stress analysis results guide design modifications?

<u>SolidWorks Group Project</u>

Objective

- To develop teamwork skills by collaborating on a SolidWorks design project.
- To integrate part modelling, assembly, analysis, and visualization in a single project.
- To present a complete design workflow: concept → CAD → analysis → rendering → documentation.

General Guidelines

1. **Group Formation:**
    - Work in groups of 3–5 students.
2. **Project Selection:**
    - A real-world mechanical product/system (e.g., gear box, robotic arm, chair design, bottle with cap, drone frame) will be assigned to each group.
    - Instructor approval required before starting.
3. **Software Tools Allowed:**
    - Part Modelling (Extrude, Revolve, Loft, Sweep, Patterns, Fillet, Chamfer).
    - Assembly Modelling (Standard Mates, Advanced Mates, Exploded View, Motion Study).
    - Simulation (Motion Analysis).
    - Rendering (PhotoView 360 / SolidWorks Visualize).

Instructions

1. **Part Modelling:**
    - Each member designs assigned parts in **.SLDPRT** format.
    - Ensure proper dimensions.
2. **Assembly:**
    - Combine all parts in **.SLDASM**.
    - Apply mates correctly to achieve realistic movement or constraints.
3. **Rendering:**
    - Apply appearances, materials, and backgrounds.
    - Generate rendered images for presentation.
4. **Documentation:**
    - Prepare **drawings** with dimensions and exploded views in **.SLDDRW**.
    - Compile a **final report** including objectives, methodology, results, and conclusions.

Deliverables

- All **part files (.SLDPRT)**
- Final **assembly file (.SLDASM)**
- **Rendered images** and exploded views

Evaluation Criteria

- **Design Accuracy & Complexity**
- **Assembly & Constraints**
- **Rendering & Visualization**

For Reference:
1. https://my.solidworks.com/solidworks/guide/SOLIDWORKS_Introduction_EN.pdf
2.https://www.solidworks.com/sites/default/files/2023-04/Fundamentals3DDesign_SIM_ENG_SV.pdf
3. https://www.solidworks.com/sw/docs/student_wb_2011_eng.pdf

Experiment 7: Introduction to CATIA Interface and 2D Sketching

Objective:

- To familiarize students with CATIA's interface, menus, and workbenches.
- To learn the use of basic 2D sketching tools in the Sketcher workbench.
- To replicate a given 2D profile with proper constraints and relations.

Software Used: CATIA V5

- **Sketcher Workbench:** Line, Circle, Trim, Constraint, Dimension, Grid Snap, Constraint Toolbar.

Theory:
CATIA (Computer Aided Three-dimensional Interactive Application) is a powerful CAD software used for designing mechanical parts and assemblies. It uses a **feature-based** and **parametric** approach to design.

The **Sketcher workbench** is the foundation for part creation in CATIA. All solid models start from a 2D sketch, which is then converted into 3D features. In Sketcher, geometric elements such as lines, circles, and arcs are created and controlled by **constraints**:

- **Geometrical Constraints** define relationships between elements (e.g., parallel, perpendicular, concentric, coincident).
- **Dimensional Constraints** specify exact measurements such as length, radius, or angle.

A **fully constrained sketch** ensures that the geometry is fixed and will behave predictably during modification. Without proper constraints, sketches may change unintentionally when edited.

Procedure:

1. Open CATIA and start a new **Part Design** file.
2. Select a plane and enter the **Sketcher** workbench.
3. Use **Line** and **Circle** tools to create the profile.
4. Apply **Geometrical Constraints** to control shape relationships.
5. Apply **Dimensional Constraints** to define size.
6. Use **Trim** tool to remove unnecessary elements.
7. Ensure the sketch is **fully constrained**.
8. Save the file.

Task:

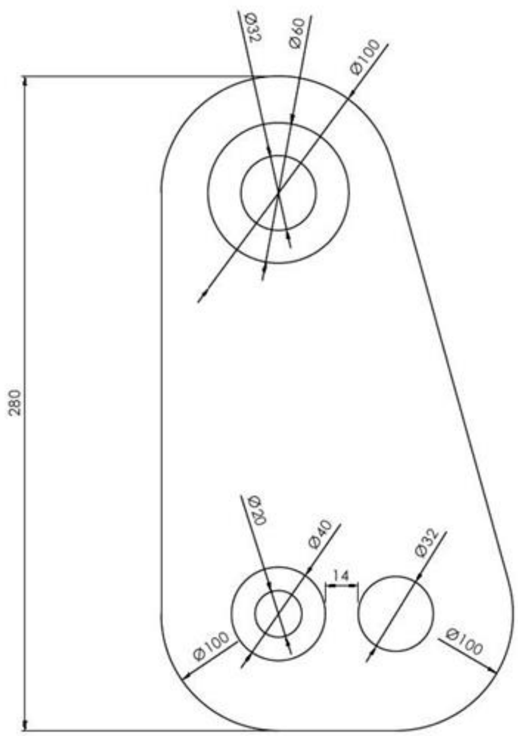
- Create the 2D profile in CATIA using the given drawing as reference, ensuring it is fully constrained.

Expected Output:

- A fully constrained 2D sketch in CATIA matching the provided reference drawing.

Viva Questions:

1. What is the difference between **Geometrical** and **Dimensional** constraints?
2. Why is it important to fully constrain a sketch before proceeding?
3. Which tool is used to remove unwanted portions of a profile?
4. What is the role of the **Sketcher workbench** in CATIA?



Experiment 8: Advanced 2D Sketching in CATIA

Objective:

- To develop skills in creating complex 2D profiles using CATIA's advanced Sketcher tools.

- To practice the use of constraints, symmetry, construction geometry, and pattern features in sketching.

Software Tools Used:

- **Sketcher Workbench:** Line, Circle, Arc, Trim, Fillet, Chamfer, Offset, Construction Geometry, Symmetry, Mirror, Pattern.

Theory:
While basic sketches are created using simple lines and circles, more complex profiles require advanced tools to save time and improve accuracy.
CATIA provides several **advanced sketching features**:

1. **Construction Geometry** – Reference geometry that doesn't become part of the final profile but is used for positioning elements.
2. **Symmetry Tool** – Allows mirrored duplication of geometry across an axis.
3. **Circular and Rectangular Patterns** – Repeats selected elements in a defined arrangement.
4. **Fillet/Chamfer** – Rounds or bevels sharp intersections within a sketch.
5. **Offset Tool –** Creates parallel curves or shapes at a set distance.

In this experiment, students will use these tools to create a complex symmetrical profile with multiple holes, arcs, and circular patterns, based on the provided technical drawing.

Procedure:

1. Open CATIA and start a new **Part Design** file.
2. Select a plane and enter the **Sketcher** workbench.
3. Create construction lines and center points to serve as reference geometry.
4. Draw main circles and arcs for the profile using **Circle** and **Arc** tools.
5. Use **Fillet** to create smooth transitions between arcs and lines.
6. Create equally spaced holes using **Circular Pattern** in Sketcher.
7. Apply **Symmetry** where applicable to duplicate features.

8. Apply **Geometrical** and **Dimensional Constraints** to fully define the sketch.

9. Save the file.

Task:

- Replicate the provided complex profile in CATIA using advanced sketching tools and ensure the sketch is fully constrained.

Expected Output:

- A fully constrained 2D profile in CATIA matching the given drawing, with correct use of construction geometry, symmetry, and patterns.

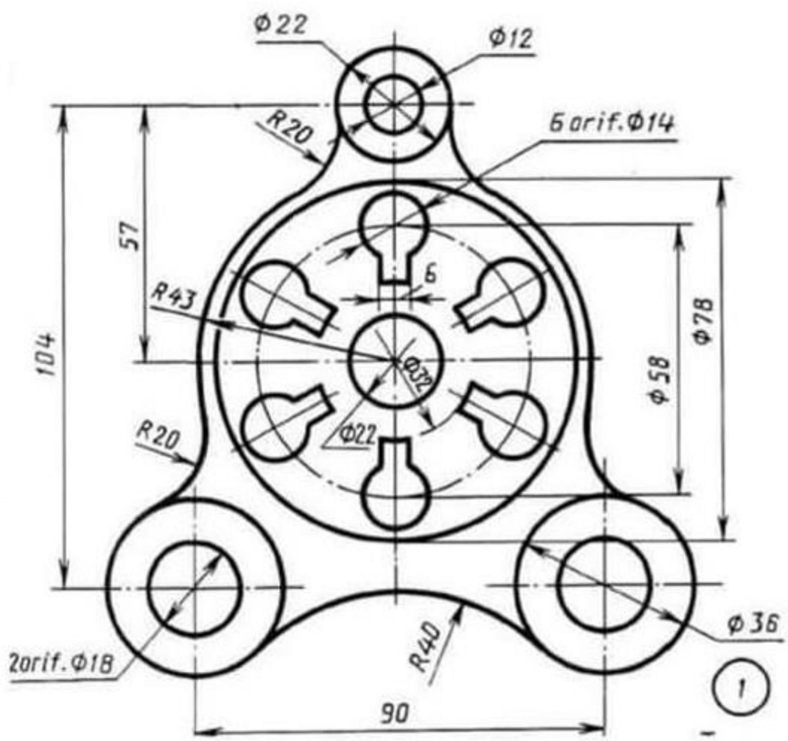
Viva Questions:

1. What is the purpose of **construction geometry** in CATIA?

2. How does the **Circular Pattern** tool in Sketcher work?

3. What is the difference between **Fillet** in Sketcher and Fillet in Part Design?

4. Why is symmetry useful in complex sketches



Experiment 9: Introduction to 3D Modeling in CATIA

Objective:

- To introduce students to 3D part creation from a 2D sketch in CATIA.
- To practice basic Part Design features such as **Pad**, **Pocket**, and **Hole**.

Software Tools Used:

- **Part Design Workbench:** Pad, Pocket, Hole, Edge Fillet.
- **Sketcher Workbench:** Line, Circle, Rectangle, Constraints.

Theory:
3D modeling in CATIA starts with a **2D sketch** drawn in the Sketcher workbench, which is then transformed into a 3D feature using tools like **Pad** (extrusion), **Pocket** (cut), and **Hole**.
Key tools introduced in this experiment:

1. **Pad –** Converts a closed 2D profile into a 3D solid by extruding it to a specified length.
2. **Pocket** – Removes material from a solid by cutting a sketch into it.
3. **Hole** – Creates cylindrical holes with defined dimensions and positions.
4. **Edge Fillet –** Rounds edges to improve aesthetics and reduce stress concentration.

Procedure:

1. Open CATIA and start a new **Part Design** file.
2. Select a plane and draw the base profile in **Sketcher**.
3. Use **Pad** to extrude the base to the required height.
4. Create the vertical semi-circular feature by sketching on a vertical face and **Pad** it.
5. Create holes using the **Hole** tool or **Pocket** from a circular sketch.
6. Create the square cutout using **Pocket** with a square sketch.
7. Apply **Edge Fillet** where necessary.
8. Save the file.

Task:

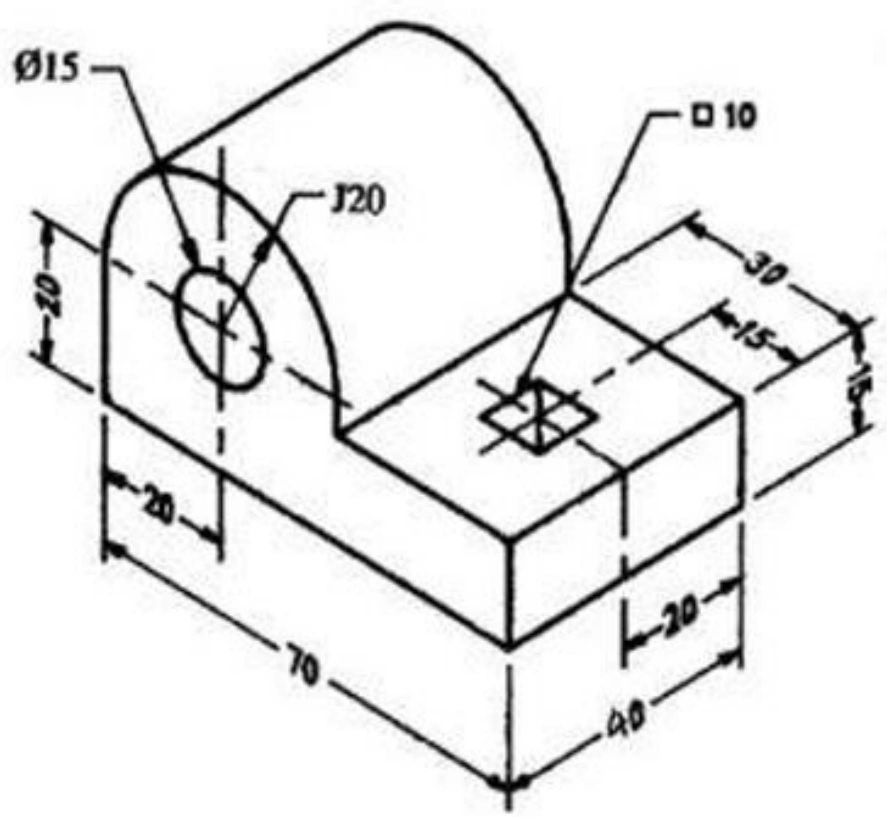
- Create the given part in CATIA, ensuring all features are correctly positioned and dimensioned.

Expected Output:

- A complete 3D part matching the given drawing, with correct holes, cutouts, and profiles.

Viva Questions:

1. What is the difference between **Pad** and **Pocket**?
2. How is the **Hole** feature different from creating a circle and using Pocket?
3. Why is it important to fully constrain a sketch before creating a 3D feature?
4. What is the purpose of **Edge Fillet** in a mechanical part?



Experiment 10: 3D Modeling of a Support Bracket

Objective:

- To create a 3D solid model of a mechanical part (support bracket) using CATIA.
- To practice **Pad, Pocket, and Fillet** features.

Software Tools Used:

- Sketcher
- Part Design (Pad, Pocket, Fillet)

Theory:
CATIA allows complex 3D modeling by combining basic sketch-based features. This part requires multiple sketches on different planes to create pads, apply pockets, and then fillet the edges.

Procedure:

1. Open CATIA → Start a new **Part Design** file.
2. Select the XY plane → enter **Sketcher**.
3. Draw the base rectangle **66 × 42 mm** with **12 mm offset borders** (as shown in drawing).
4. Exit Sketcher → use **Pad** (thickness as per design).
5. On the vertical face, sketch the triangular support (42 mm height, 27 mm slant) → **Pad**.
6. On the top arc, create a **21 mm radius arc** and complete the profile. Use **Pad** to generate the curved portion.
7. Apply **Pocket** where required (inside cut as per drawing).
8. Use **Edge Fillet** to smoothen the R21 arc.
9. Save as *Exp10_Bracket.CATPart*.

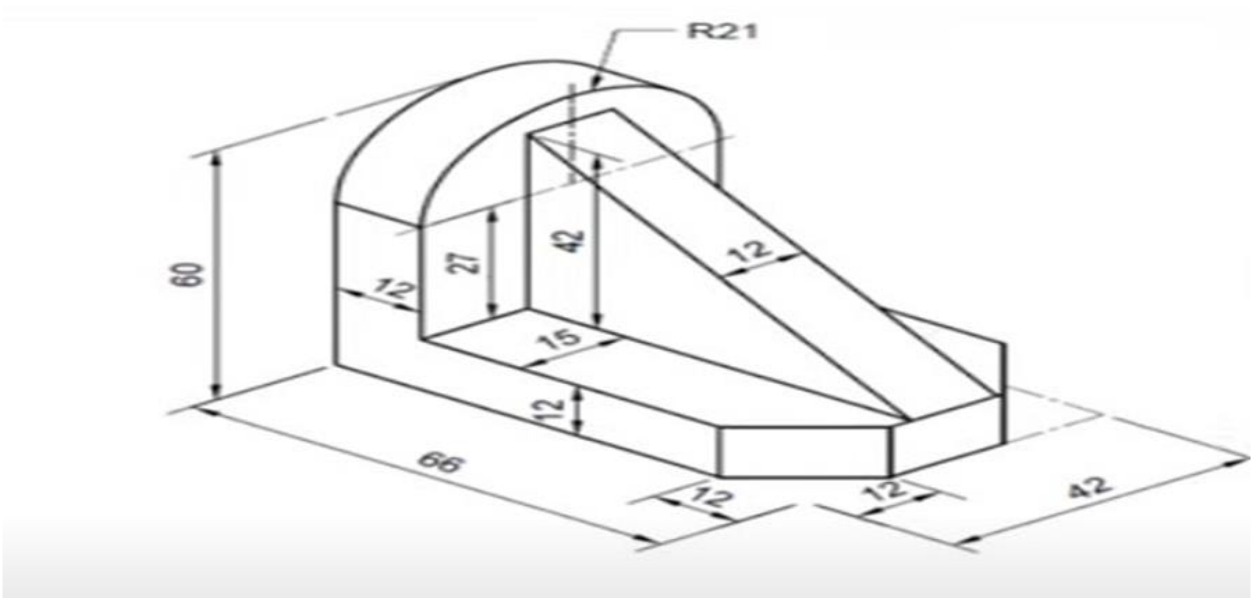
Task:
Recreate the 3D model of the support bracket as per given dimensions (see drawing provided).

Expected Output:

- A 3D bracket with triangular support, curved top, and pockets matching the technical drawing.

Viva Questions:

1. Which CATIA tool is used to create circular arcs like R21?
2. How do you constrain dimensions in a sketch?
3. Why is it important to follow exact drawing dimensions while modeling?


R21
60
42
27
12
12
15
12
66
12
12
42

Experiment 11: 3D Modeling of a Mechanical Part in CATIA

Objective:

- To create a 3D model of a mechanical part using CATIA.
- To practice **Pad**, **Pocket**, **Hole**, and **Circular Pattern** features.

Software Tools Used:

- **Sketcher**
- **Part Design**: Pad, Pocket, Hole, Circular Pattern

Theory:

CATIA enables solid modeling by converting 2D sketches into 3D features. This part includes operations like creating pads for solid volumes, pockets for cuts, and hole patterns on circular faces using PCD (Pitch Circle Diameter).

Procedure:

1. **Open CATIA → New Part Design.**
2. **Base:**
   - Sketch a rectangle: 300 × 100 mm → Pad: 25 mm.
3. **Cylindrical Boss:**
   - On top face, draw Ø150 mm circle at (250, 50) → Pad: 50 mm.
4. **Center Hole:**
   - On top of boss → Sketch Ø70 mm → Pocket: Through All.
5. **8 Holes on PCD:**
   - Sketch 1 hole Ø15 mm on radius 53 mm → Circular Pattern: 8 instances around center.
6. **4 Corner Holes:**
   - Sketch Ø20 mm holes at 4 corners (25 mm offset) → Pocket: Through All.
7. **Cutouts and Notch:**
   - Create rectangle cutout (25×25×10 mm) at bottom left → Pocket.
   - Create small block (50×25×25 mm) on top right as shown → Pad.
8. **Save the file as**: Exp11_Model_Part.CATPart.

Task:

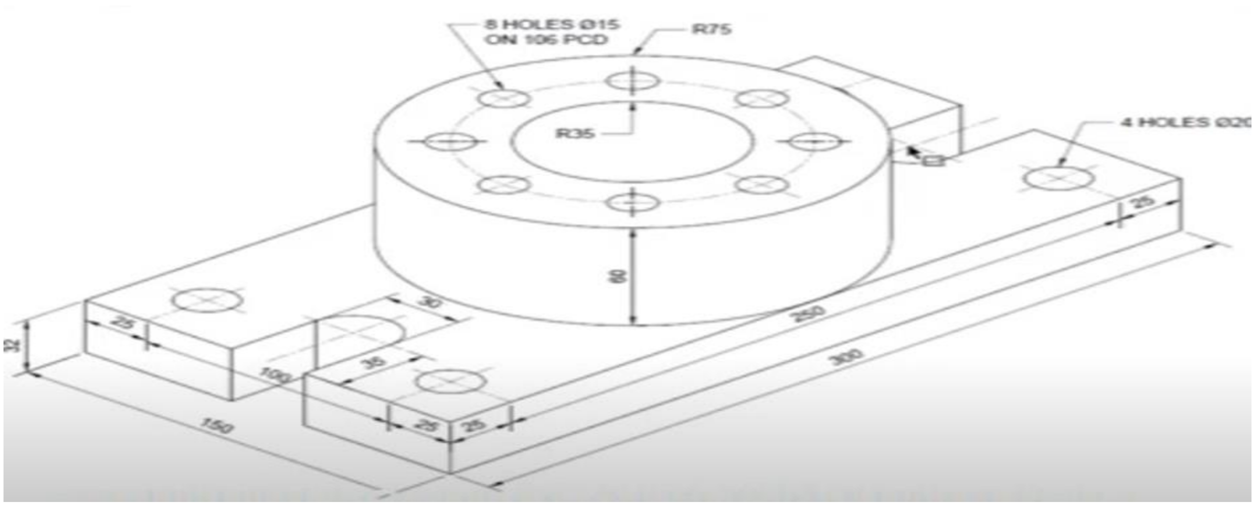
Recreate the full 3D model with correct dimensions and features as per the given drawing.

Expected Output:

- A 3D model with a circular boss, PCD holes, rectangular base, corner holes, and cutouts matching the drawing.

Viva Questions:

1. What is a **PCD**, and how is it used in modeling?
2. How do you create **multiple holes in a circular pattern**?
3. What is the function of the **Pocket** tool?
4. Why are **constraints** important in sketches?



Experiment 12: Assembly of a Wooden Bench using CATIA V5

Objective

- To create a 3D assembly of a bench using CATIA.
- To practice Assembly Design with constraints such as Coincidence, Contact, and Offset.

Software Tools Used

- **Part Design Workbench**: Pad, Pocket, Constraints
- **Assembly Design Workbench**: Coincidence Constraint, Contact Constraint, Offset Constraint

Theory

CATIA provides tools for both part modeling and assembly design. Individual parts are created in the Part Design workbench and then combined in the Assembly Design workbench. Constraints are applied to correctly position and fix the parts relative to one another. This bench model demonstrates the bottom-up assembly approach where separate parts (Top, Steps, Legs) are modeled individually and then assembled.

Procedure

1. **Open CATIA → New Part Design.**
2. **Create Parts:**
    - **Top**: Create rectangular pad for bench top.
    - **Leg**: Create trapezoidal pad with slots for steps.
    - **Step**: Create cylindrical or rectangular rods for bench steps.
    - Save all parts individually.
3. **Open Assembly Design.**
    - Insert all parts: Top, 2 Legs, 2 Steps.
    - Apply constraints:
        - Coincidence Constraint → Align Steps with Leg slots.
        - Contact Constraint → Seat Top correctly on Steps.
        - Offset Constraint → Maintain distance between Legs.
    - Ensure the assembly is fully constrained.
4. **Verification:**
    - Use Clash Detection to check for interference.
    - Save assembly file as **Bench_Assembly.CATProduct**.

Task

Recreate the full bench assembly with the given parts and apply correct constraints to achieve the expected final model.

Expected Output

- A fully assembled bench model with proper alignment of Top, Steps, and Legs.
- Correctly constrained assembly with no remaining degrees of freedom.

Viva Questions

1. What is the difference between Bottom-Up and Top-Down assembly in CATIA?
2. Explain the use of Coincidence and Contact constraints in assemblies.
3. How do you check if an assembly is fully constrained?
4. What is the purpose of Clash Detection in CATIA?
5. Can you create an exploded view of an assembly in CATIA? If yes, how?