

Department of Mechanical Engineering

P.E.S COLLEGE OF ENGINEERING

MANDYA – 571401

(An Autonomous Institute Affiliated to VTU, Belagavi)



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An Internship Report on

“DESIGN AND DEVELOPMENT OF KNUCKLE JOINT BY USING CATIA V5”

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CERTIFICATE

This is to certify that **JAVAD BEIG [4PS24ME423]** has successfully completed the **Internship-I [P22INT408]** "**DESIGN AND DEVELOPMENT OF KNUCKLE JOINT BY USING CATIA V5**" in partial fulfillment for the award of Bachelor of Engineering in **Mechanical Engineering**, P.E.S College of Engineering, Mandya affiliated to **VTU**, Belagavi during the year 2024-2025.

The Internship has been approved as it satisfies the academic requirements in respect of Internship prescribed for the degree in Bachelor of Engineering.

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ABSTRACT

The internship program offers a comprehensive exploration of advanced CAD techniques through CATIA V5, equipping participants with valuable skills and practical experience in automotive design. Beginning with an introduction to surface modeling, interns will delve into the intricacies of creating complex surfaces, followed by an in-depth exploration of Generative Shape Design for generating organic shapes and forms. The training continues with hands-on projects, including base bracket modeling and B-pillar modeling, providing a solid foundation in component design. Participants will then transition to plastic modeling, mastering techniques essential for designing plastic components, before progressing to more complex projects such as car bumper modeling and car hood design. Additionally, interns will learn about attachment feature creation, facilitating the assembly of components, and remastering surface and solid designs, ensuring adaptability to evolving design requirements. Throughout the internship, emphasis will be placed on practical application, problem-solving, and attention to detail, preparing interns for future roles in the industry.

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CHAPTER – 01**CATIA V5****1.1 Introduction**

In the realm of modern engineering and design, Computer-Aided Design (CAD) software plays a pivotal role in shaping the way we conceptualize, model, and bring ideas to life. Among the myriad of CAD tools available, CATIA (Computer-Aided Three-dimensional Interactive Application) stands out as a cornerstone in the industry, renowned for its robust capabilities and versatility across a wide spectrum of domains.

Our college recently embarked on a journey to empower aspiring engineers with the expertise needed to harness the full potential of CATIA through a dedicated training program. This introduction sets the stage to illuminate the objectives, methodologies, and significance of the CATIA training initiative, shedding light on its pivotal role in nurturing the next generation of proficient CAD users and design innovators.

The CATIA training program was meticulously crafted to provide participants with a comprehensive understanding of the software's functionalities, ranging from basic modelling techniques to advanced simulation and analysis tools. Through a combination of theoretical instruction, hands-on exercises, and real-world applications, participants were immersed in a dynamic learning environment tailored to cultivate proficiency and fluency in CATIA.

This report endeavours to offer a holistic overview of the CATIA training program, encapsulating its curriculum, instructional approach, participant feedback, and the broader impact on skill development and career readiness. By documenting the journey and outcomes of the program, this report seeks to underscore the significance of CATIA proficiency in the contemporary engineering landscape and the transformative potential it holds for aspiring engineers striving to make their mark in the field.

Initially CATIA name is an abbreviation for Computer Aided Three-dimensional Interactive Application the French Dassault Systems is the parent company and IBM participates in the software and marketing, and Catia is invades broad industrial sectors, and has been explained in the previous post position of CATIA between 3d modeling software programs. CATIA was started in 1977 by French Aircraft Manufacturer Avions Marcel Dassault System.

CATIA is a comprehensive software solution that empowers organizations to innovate, collaborate, and streamline their product development processes. Its extensive capabilities, industry-specific modules, and integration with PLM systems make it a preferred choice for companies striving to achieve excellence in design, engineering, and manufacturing.

Version that most of the people works on it now is CATIA V5 or fifth version, which is a rewriting and revision the code of the fourth edition. For the fifth version, there are versions from 1 to 20, for example, CATIA V5 R17, it Means CATIA fifth edition version seventeenth, while years system was adoption in the sixth edition, for example. CATIA V6 2011 means CATIA sixth edition version of Year 2011

Advanced 3D modelling and surface design

- Integrated simulation and analysis tools
- Collaborative design environment
- Realistic behaviour modelling of products
- Cross-discipline development (mechanical, electrical, fluid systems)

CATIA originated in the late 1970s as an in-house tool for aircraft design by Avions Marcel Dassault. It was officially launched in 1982 and has since evolved into one of the most comprehensive design platforms available.

Let me know if you'd like a comparison with other tools like SolidWorks or AutoCAD, or a deeper dive into its modules.

1.2 The History of Catia

The history of CATIA spans over several decades, marked by significant advancements in computer-aided design (CAD) technology and its application across various industries. Here's a chronological overview of the key milestones in the evolution of CATIA:

- **1977:** CATIA (Computer-Aided Three-Dimensional Interactive Application) was initially developed by Avions Marcel Dassault, a French aircraft manufacturer, to support the design of their Mirage fighter jet series. This early version of CATIA focused primarily on aerodynamic design and analysis.
- **1981:** Dassault Systems, a subsidiary of Avions Marcel Dassault, was established to further develop and market CATIA as a standalone software solution. This marked the beginning of CATIA's transition into a comprehensive CAD/CAM/CAE platform.
- **1984:** CATIA Version 3 (V3) was released, introducing significant enhancements in 3D modeling capabilities and expanding its applicability beyond aerospace to industries such as automotive and consumer goods.
- **1988:** CATIA Version 4 (V4) represented a major technological advancement, featuring a completely rewritten codebase and a modular architecture. V4 significantly improved performance, scalability, and compatibility with various hardware platforms.
- **1995:** CATIA Version 5 (V5) was introduced, leveraging emerging technologies such as object-oriented programming and graphical user interfaces. V5 marked a paradigm shift in CAD software design, offering a more intuitive user experience, enhanced interoperability, and advanced surface modeling capabilities.
- **2000:** Dassault Systems introduced the ENOVIA brand, integrating CATIA with product lifecycle management (PLM) capabilities for improved collaboration, data management, and workflow automation. This integration strengthened CATIA's position as a comprehensive product development solution.
- **Present:** CATIA remains at the forefront of CAD/CAM/CAE technology, serving a diverse range of industries, including aerospace, automotive, industrial equipment, and consumer goods. Its continuous evolution and integration with the 3DEXPERIENCE platform position CATIA as a key enabler of digital transformation and sustainable innovation in product development.

1.3 CATIA Modules

1.3.1 Sketcher: The Sketcher module in CATIA is where users create 2D profiles that serve as the basis for creating 3D geometry. It provides tools for drawing lines, arcs, circles, and other geometric shapes. Users can apply constraints (such as dimensions, distances, angles) to ensure the sketch remains fully defined and editable.

1.3.2 Part Design: Part Design module is used for creating 3D solid parts from sketches or by directly modeling features. It allows users to apply operations like extrusion, revolution, fillet, chamfer, hole creation, etc., to shape the part. Features in Part Design are typically solid and have physical volume properties.

1.3.3. Wireframe and Surface Design: This module focuses on creating and manipulating wireframe geometry and surface models. Users can create complex surface shapes using tools like sweep, loft, blend, and boundary. Wireframe and Surface Design is often used for creating aesthetically pleasing or aerodynamic shapes that may be difficult to achieve with traditional solid modeling techniques.

1.3.4. Generative Sheet Metal: Generative Sheet Metal module is specialized for designing sheet metal components and structures. It provides tools for creating features specific to sheet metal design, such as bends, flanges, reliefs, and stamps. Users can unfold and flatten sheet metal parts to generate manufacturing drawings and calculate material usage.

1.3.5. Assembly Design: Assembly Design module is used for creating and managing assemblies of parts. It allows users to import parts, position them within the assembly, and define assembly constraints (such as mates, hinges, cams) to simulate real-world connections between components. Users can analyze assembly motion, detect interferences, and perform kinematic simulations to validate the design.

1.3.6. Drafting: Drafting module is used for creating 2D technical drawings and documentation. Users can generate views (such as front, top, side), add dimensions, annotations, symbols, and tables to fully document the design. Drafting ensures that design information is communicated accurately to manufacturing teams, suppliers, and stakeholders. It supports industry standards for drawing formats, scales, and tolerances, ensuring compliance with regulatory requirements and best practices in engineering documentation.

CHAPTER – 02

SKETCHER

2.1 Introduction

CATIA Sketcher is the foundational tool within the CATIA software suite, providing users with the essential means to initiate their design projects. As the initial step in the design process, Sketcher allows users to create precise 2D sketches that serve as the building blocks for complex 3D models. With its user-friendly interface and comprehensive set of drawing tools, Sketcher empowers designers and engineers to effortlessly craft geometric shapes, lines, arcs, and curves. Moreover, Sketcher offers robust constraint features, enabling users to maintain geometric relationships and dimensions, ensuring accuracy throughout the design process. One of the key strengths of CATIA Sketcher lies in its parametric modelling capabilities. This means that sketches created in Sketcher can be easily modified and adapted to meet changing design requirements, facilitating iterative design iterations and exploration of alternative concepts. Seamlessly integrated with other CATIA modules, Sketcher serves as the bridge between 2D conceptualization and 3D realization. Sketches created in Sketcher can be further developed in Part Design, Wireframe and Surface Design, or other specialized environments within CATIA, allowing users to transform their initial concepts into fully realized, three-dimensional models. In essence, CATIA Sketcher is not merely a drawing tool; it is the cornerstone of the CATIA design process, empowering users to unleash their creativity, refine their ideas, and ultimately, bring their designs to life with precision and efficiency.

2.2 Toolbars in sketcher: Toolbars are organized to help you create and constrain 2D geometry efficiently.

2.2.1 Profile Toolbar: The Profile toolbar contains 2D geometry commands. These geometries range from the very simple (point, rectangle, etc...) to the very complex (splines, conics, etc...). The Profile toolbar contains many sub-toolbars. Most of these sub-toolbars contain different options for creating the same geometry. For example, you can create a simple line, a line defined by two tangent points, or a line that is perpendicular to a surface. Profile toolbar Reading from left to right, the Profile toolbar contain the following commands

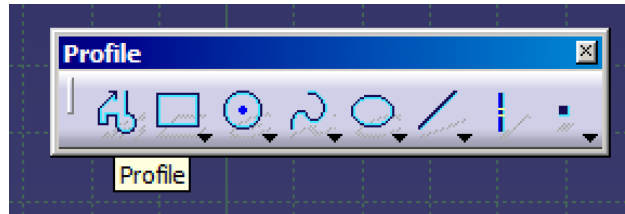


Fig 2.1 Profile toolbar

2.2.1.1 Profile: This command allows you to create a continuous set of lines and arcs connected together.

2.2.1.2 Rectangle/Predefined Profile toolbar: The default top command is rectangle. Stacked underneath are several different commands used to create predefined geometries.

- **Rectangle:** The rectangle is defined by two corner points. The sides of the rectangle are always horizontal and vertical
- **Oriented Rectangle:** The oriented rectangle is defined by three corner points. This allows you to create a rectangle whose sides are at an angle to the horizontal
- **Parallelogram:** The parallelogram is defined by three corner points.
- **Elongated Hole:** The elongated hole or slot is defined by two points and a radius
- **Cylindrical Elongated Hole:** The cylindrical elongated hole is defined by a cylindrical radius, two points and a radius
- **Keyhole Profile:** The keyhole profile is defined by two center points and two radii
- **Hexagon:** The hexagon is defined by a center point and the radius of an inscribed circle.
- **Centered Rectangle:** The centered rectangle is defined by a center point and a corner point.
- **Centered Parallelogram:** The centered parallelogram is defined by a center point (defined by two intersecting lines) and a corner point. Creates text that may be used to create a solid.

2.2.1.3 Circle/Circle toolbar: The default top command is circle. Stacked underneath are several different options for creating circles and arcs.

- **Circle:** A circle is defined by a center point and a radius.
- **Three Point Circle:** The three-point circle command allows you to create a circle using three circumferential ports.
- **Circle Using Coordinates:** The circle using coordinates command allows you to create a circle by entering the coordinates for the center point and radius in a Circle Definition window.
- **Tri-Tangent Circle:** The tri-tangent circle command allows you to create a circle whose circumference is tangent to three chosen lines.
- **Three Point Arc:** The three points arc command allows you to create an arc defined by three circumferential points.
- **Three Point Arc Starting with Limits:** The three points arc starting with limits allows you to create an arc using a start, end, and midpoint.
- **Arc:** The Arc command allows you to create an arc defined by a center point, and a circumferential start and end point.

2.2.1.4 Spline/Spline toolbars: The default top-command is spline which is a curved line created by connecting a series of point

- **Spline:** A spline is a curved profile defined by three or more points. The tangency and curvature radius at each point may be specified.
- **Connect:** The connect command connects two points or profiles with a spline.
- **Conic toolbar:** Reading from left to right, the Conic toolbar contains the following command.

2.2.1.5 Ellipse/Conic toolbar: The default top command is ellipse. Stacked underneath are commands to create different conic shapes such as a hyperbola.

- **Ellipse:** The ellipse is defined by a center point and major and minor axis points.

- **Parabola by Focus:** The parabola is defined by a focus, apex and start and end points.
- **Hyperbola by Focus:** The hyperbola is defined by a focus, center point, apex and start and end points.
- **Conic:** There are several different methods that can be used to create conic curves. These methods give you a lot of flexibility when creating the above three types of curves.

2.2.1.6 Line/Line toolbar: The default top command is line. Stacked underneath are several different options for creating lines

- **Line:** A line is defined by two points.
- **Infinite Line:** Creates infinite lines that are horizontal, vertical or defined by two points.
- **Bisecting Line:** Creates an infinite line that bisects the angle created by two other lines.
- **Line Normal to Curve:** This command allows you to create a line that starts anywhere and ends normal or perpendicular to another element.

2.2.1.7 Axis: An axis is used in conjunction with commands like mirror and shaft (revolve) It defines symmetry. It is a construction element so it does not become a physical part of your future.

2.2.1.8 Point/Point toolbar: The default top command is point. Stacked underneath are several different options for creating point

- **Point by Clicking:** Creates a point by clicking the left mouse button.
- **Point by using coordinates:** Creates a point at a specified coordinate point.
- **Equidistant Points:** Creates equidistant points along a predefined path curve.
- **Intersection Point:** Creates a point at the intersection of two different elements.
Projection Point: Projects a point of one element onto another.

2.2.2 Constraint toolbar

Constraints can either be dimensional or geometrical. Dimensional constraints are used to constrain the length of an element, the radius or diameter of an arc or circle, and the distance or angle between elements. Geometrical constraints are used to constrain the orientation of one element relative to another. For example, two elements may be constrained to be perpendicular to each other. Other common geometrical constraints include parallel, tangent, coincident, concentric, etc... Reading from left to right:

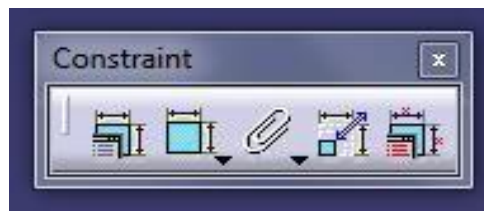


Fig 2.2 Constraint Toolbar

2.2.2.1 Constraints Defined in Dialoged Box: Creates geometrical and dimensional constraints between two elements.

2.2.2.2 Constraint: Creates dimensional constraints. **O Contact Constraint:** Creates a contact constraint between two elements.

2.2.2.3 Fix Together: The fix together command groups individual entities together. **O Auto Constraint:** Automatically creates dimensional constraints.

2.2.2.4 Animate Constraint: Animates a dimensional constraint between to limits.

2.2.2.5 Edit Multi-Constraint: This command allows you to edit all your sketch constraints in a single window.

2.3 Operation Toolbar

The operation toolbar in CATIA provides a set of essential tools for manipulating and modifying existing geometry in your sketches and 3D models. Here's a breakdown of some common functions found on this toolbar:

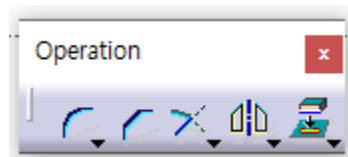


Fig 2.3 Operation Toolbar

Trim: This tool allows you to cut away unwanted portions of existing geometry. You can select entities like lines, curves, or surfaces and then define the cutting edges to remove specific sections.

Mirror: This tool creates a mirrored copy of selected geometry. Simply choose the entities you want to mirror and then define the mirror plane or axis for the reflection.

Chamfer: This tool creates a chamfer, which is a beveled edge on a solid model. You can define the chamfer size (distance) and angle to create a smooth transition between two intersecting faces.

Fillet: This tool creates a fillet, which is a rounded edge on a solid model. Similar to chamfer, you can specify the fillet radius to define the curvature of the rounded edge.

Offset: This tool allows you to create a parallel copy of existing geometry at a specified distance. Select the entities and define the offset distance to generate a new set of parallel curves or surfaces.

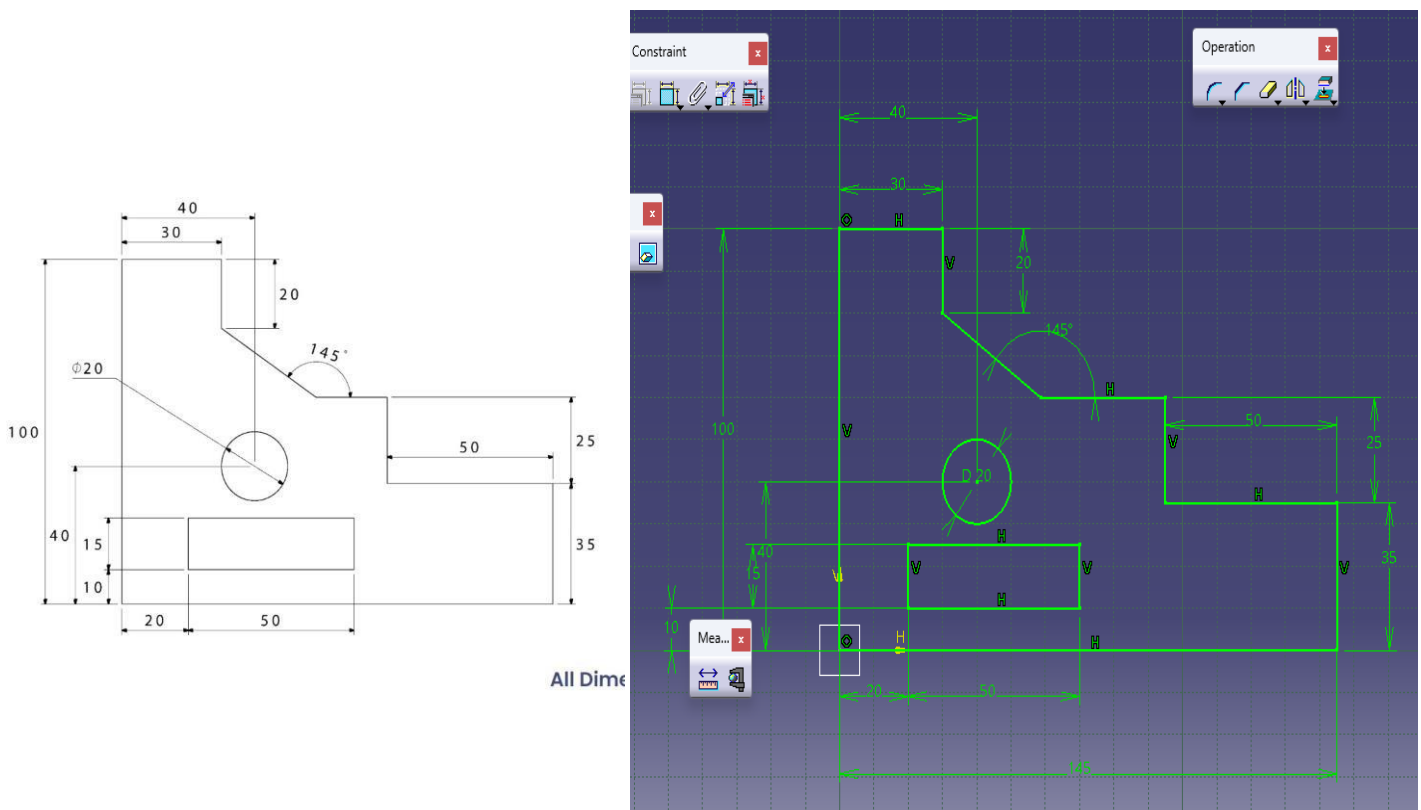


Fig 2.4 sketch drew using sketch and constraints

CHAPTER – 3

PART DESIGN

3.1 Introduction

CATIA Part Design module is the cornerstone of 3D solid modeling within the CATIA software suite. It offers a comprehensive set of tools and features tailored for creating precise and intricate solid parts. With intuitive controls and parametric capabilities, Part Design enables users to seamlessly translate conceptual designs into detailed 3D models. Whether crafting simple geometries or complex assemblies, this module empowers designers and engineers to iterate efficiently, ensuring design integrity and manufacturability. As an essential component of the CATIA ecosystem, Part Design plays a pivotal role in transforming ideas into tangible products with precision and versatility.

3.2 Toolbars in Part Design: Toolbars are organized to help you create and constrain 2D geometry efficiently.

3.2.1 Sketch-Based Features

This toolbar is mainly used to create a solid feature from a 2D sketch/profile

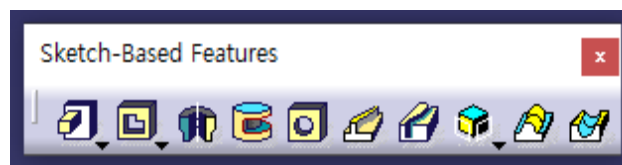


Fig 3.1 sketch based feature

3.2.1 Pad: This command is used to add material by extruding a sketch.

3.2.2 Pocket: This command is used to remove material by extruding a sketch.

3.2.1.3 Shaft: This command is used to add material by rotating a sketch.

3.2.1.4 Groove: This command is used to remove material by rotating a sketch.

3.2.1.5 Rib: This command is used to add material by sweeping a profile along a center curve.

3.2.1.6 Slot: This command is used to remove material by sweeping profile along a center curve.

3.2.1.7 Multi-sections Solid: This command is used to add material by sweeping one or more planar section curves along one or more guide curves.

3.2.1.8 Removed Multi-sections Solid: This command is used to remove material by sweeping one or more planar section curves along one or more guide curves.

3.2.1.9 Hole: This command is used for the circular material removal from the existing solid. Several types of holes are available. Such as: Simple, Tapered, Counter bored, counter drilled.

3.2.2 Dress-Up Features:

This toolbar is used to add fillets/chamfers on the solid edge, add a draft onto the solid faces. Hollow the solid, offset faces and many more.

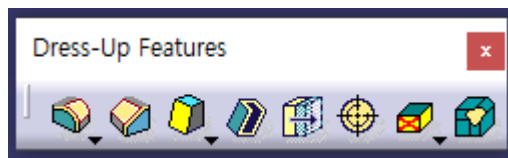


Fig 3.2 Dress-up Features

3.2.2.1 Fillet: This command is used for creating a curved face of a constant or variable radius that is tangent to, and that joins, two surfaces.

3.2.2.2 Chamfer: This command is used for removing & adding a flat section from a selected edge to create a beveled surface between the two original faces common to that edge.

3.2.2.3 Draft: This command is used for adding or removing material depending on the draft angle and the pulling direction.

3.2.2.4 Shell: This command is used to empty a solid while keeping a given thickness on its sides.

3.2.3 Transformation Features:

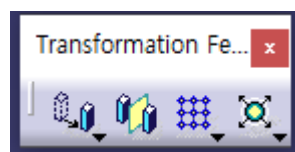


Fig 3.3 Transformation Feature

3.2.3.1 Translation: This command is used for translating a solid entity along a direction.

3.2.3.2 Rotation: This command is used for rotating a solid entity about an axis by a certain angle.

3.2.3.3 Symmetry: This command is used for translating a solid entity to the other side of the mirror plane.

3.2.3.4 Mirror: This command is used for duplicating a solid entity on the other side of the mirror plane.

3.2.3.5 Patterns: Generally, there are 3 types of patterns, which are:

- Rectangular Pattern
- Circular Pattern
- User defined Pattern

3.2.4 Boolean operation

CATIA's Part Design workbench offers specific tools for performing Boolean operations on solid models. These tools allow you to combine, subtract, or intersect existing solids to create new shapes.

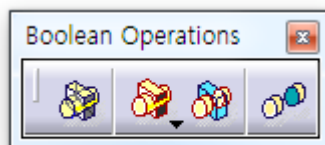


Fig 3.4 Boolean Operation

3.2.4.1 Assemble: This operation, though named "Assemble," actually performs a subtraction. It takes two solids and removes a portion of the first solid based on the second solid.

3.2.4.2 Add: This operation creates a single solid by combining two existing solids. Unlike Assemble, it doesn't remove any material. It simply merges the two solids into one, treating them as a single piece. The polarity of the second solid doesn't play a role here.

3.2.4.3 Remove: This operation is straightforward. It removes the entire second solid from the first solid. Think of it like cutting out a cookie with a cookie cutter, but here the entire cookie cutter shape is removed, not just its outline. Polarity doesn't affect this operation.

3.2.4.4 Intersect: This operation creates a new solid that represents the overlapping region where the two original solids intersect. Imagine placing two cylinders partially on top of each other. The resulting solid would be the cylindrical shape formed by the overlap, not the complete original cylinders. Polarity doesn't influence this operation either.

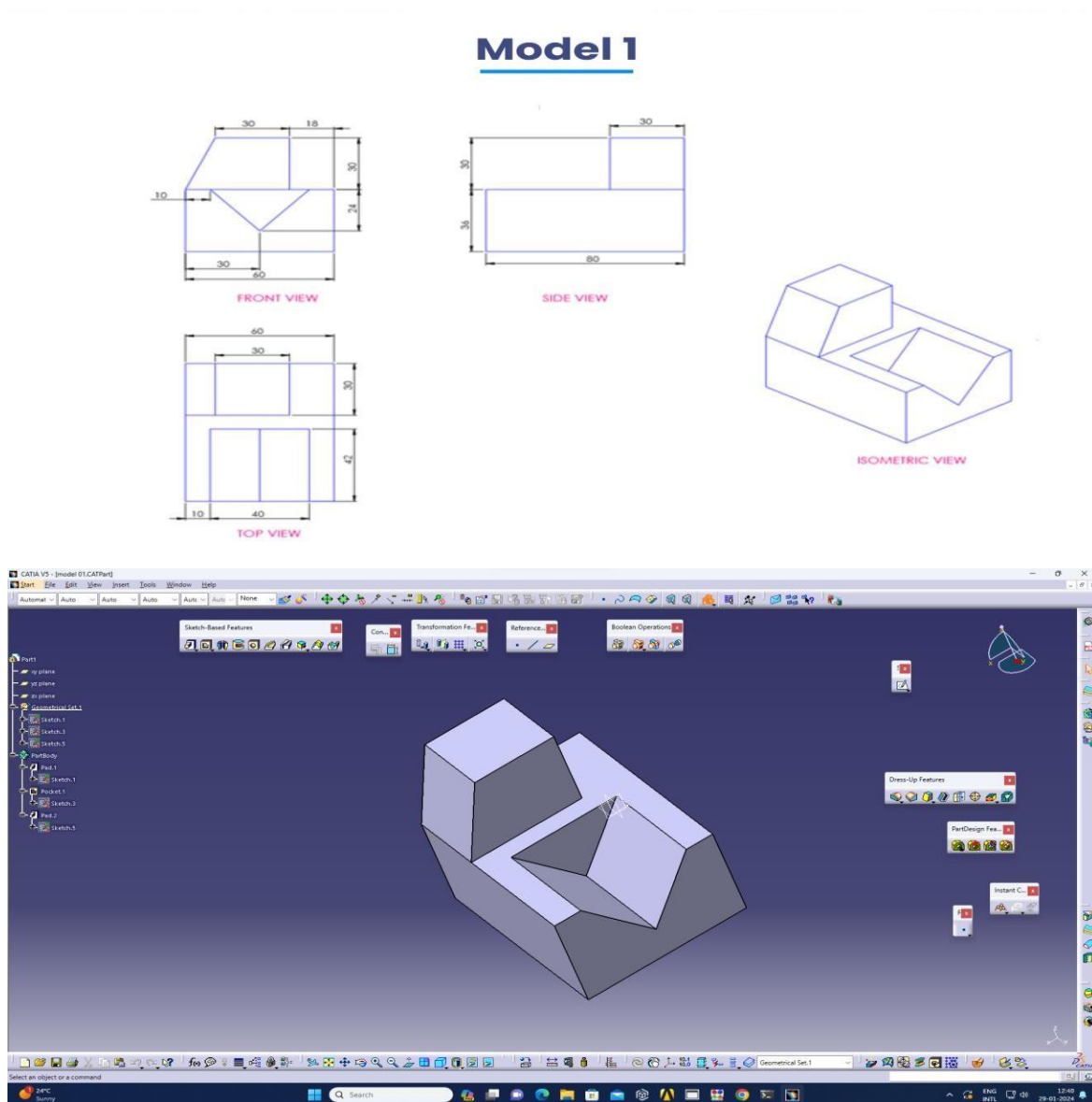


Fig 3.5 Part Designed Using Part Design Toolbar

CHAPTER – 4

WIREFRAME AND SURFACE DESIGN

4.1 Introduction

CATIA Wireframe and Surface Design module offer advanced capabilities for creating complex surface shapes and wireframe geometry within the CATIA software suite. It serves as a specialized environment for designers and engineers to sculpt organic forms, intricate surfaces, and aesthetically pleasing shapes that may be challenging to achieve with traditional solid modeling techniques.

With a rich set of tools for sweep, loft, blend, and boundary operations, Wireframe and Surface Design empower users to unleash their creativity and explore innovative design concepts. Whether designing automotive body panels, consumer products, or aerospace components, this module provides the flexibility and precision needed to bring even the most ambitious ideas to life.

Furthermore, Wireframe and Surface Design seamlessly integrate with other CATIA modules, allowing for smooth collaboration and transition between different stages of the design process. From conceptual sketches to final surface models, this module enables designers to iterate, refine, and optimize their designs with confidence, ensuring both aesthetic appeal and engineering excellence in the final product.

4.2 Surfaces Toolbar

The tools provided in Wireframe and Surface Design workbench to create simple and complex surfaces are discussed in the following section

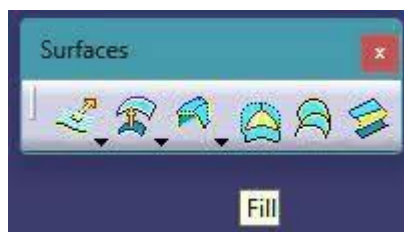


Fig 4.1 Surface Toolbar

4.2.1 Creating Extruded Surfaces: The extruded surfaces are created by extruding a profile and specifying the extrusion depth and direction vector. The basic parameters that are required to.

Menu: Insert > Wireframe > Helix Toolbar: Curve > Helix

Menu: Insert > Surfaces > Extrude Toolbar: Surfaces > Extrude

Create an extruded surface are profile, direction for extrusion, and extrusion limits. To create an extruded surface, you first need to draw the profile to be extruded using the Sketcher workbench or by using the tools available in the Wireframe toolbar. Once you have drawn the profile, choose the Extrude button from the Surfaces toolbar.

4.2.2 Creating Offset Surfaces: The Offset tool is used to create a surface that is at an offset distance from a reference surface. To do so, choose the Offset tool from the Surfaces toolbar. The Offset Surface Definition dialog box is displayed, and you are prompted to select a reference surface. Select the reference surface from the geometry area and specify the offset value in the Offset spinner. Choose the Reverse Direction button available in the dialog box to reverse the offset direction. The Both sides check box is selected to create the offset surface on both sides of the reference surface. The Repeat object after OK check box is used to create multiple offset surfaces. Select the Repeat object after OK check box and exit the Offset Surface Definition dialog box. The Object Repetition dialog box is displayed

Menu: Insert Surfaces > Cylinder Toolbar: Surfaces > Cylinder

Menu: Insert > Surfaces > Offset Toolbar: Surfaces > Offset

In this dialog box specify the required number of instance(s). Choose the OK button to create the offset surfaces. A reference surface and an offset surface. Note sometime for complex reference surfaces, the offset surface may not be created. In such cases, you need to reduce the offset value or modify the initial geometry.

4.2.3 Creating Swept surfaces: The sweep tool is provided to create surfaces by sweeping a profile along a guide curve in the Wireframe and Surfaces Design workbench of CATIA V5. To create a swept surface, you first need to draw a profile and a guide curve as two separate sketches. Next, choose the Sweep button from the Surfaces toolbar. The Swept Surface Definition dialog box is displayed, and you are prompted to select a profile. Select the profile

from the geometry area; you are prompted to select a guide curve. Select the guide curve from the geometry area. Now, choose the OK button from the Swept Surface Definition

dialog box. Profile and a guide curve and the resulting swept surface. Menu: Insert > Surface > Sweep Toolbar: Surface > Sweep. Swept Surface with two Guide Curves You can also create a swept surface using more than one various other tools to create swept surface are discussed following section. Guide curve. First draw a profile and two guide curves as separate sketches.

Now, select the Sweep button from the Surfaces toolbar, the Swept Surface Definition dialog box is displayed. Select With two guide curve option from the Subtypes drop- down list; you are prompted to select a profile. After you select the profile, you are prompted to select a guide curve. Select the first and second guide curves. Now, select the anchor point for the respective guide curves. Choose the Preview button from the Swept Surface Definition dialog box to preview the surface created. Choose the OK button from the Swept Surface Definition dialog box. Profile and guide curves. The swept surface created using the two-guide curve.

Swept Surface with Two Limits in CATIA V5, you can create a swept surface by defining the two limit curves. The limit curves can be in the same or different planes. To create swept surface with two limits, you need to draw two limit curves. Once you have drawn the curves, choose the Sweep button from the Surfaces toolbar; the Swept Surface Definition dialog box will be displayed. Select the Line button from the Profile Type area in the dialog box.

Swept Surface with Three Curve You can also create a circular swept surface using three guide curves. To create a surface using this tool, you first need to draw three guide curves, which should lie in different planes. After drawing the curves, invoke the Swept surface Definition dialog box. Choose the Circle button from the Profile type area in the dialog box. The parameters in the Swept surface

Definition dialog box will change. You are prompted to select a first guide curve that define the first extremity of the circular arc. Select the first guide curve. Next you are prompted to select the second guide curve. After selecting the second guide curve, you are prompted to select a guide curve that defines the second extremity of the circular arc. Select the guide curve and choose the OK button from the dialog box to complete the swept surface.

4.2.4 Creating Fill Surfaces: The Fill tool allows you to create fill surfaces between a numbers of boundary segments. These may be planar or non-planar, but there should not be a large gap between the consecutive boundary segments. Before creating a fill surface, draw the boundary curves. Next, choose Fill button from the Surface toolbar.

Next, you are required to select the boundary segments. You need to make sure that while selecting the boundary segments, the sequence of selection should be such that a

closed loop is formed. Once you have selected the boundary curves, choose the OK button. The curves drawn to create the fill surface and the resulting fill surface. Support surfaces may be selected with the respective curve to ensure the continuity between the fill surface and the support surface. Similarly, if you select a passing point, the fill surface will be created, such that it passes through the selected point.

4.2.5 Creating Lofted Multisection Surfaces:

This tool allows you to create lofted multisection surfaces. The surface is created between the sections along the computed or user-defined spine. To create a multisection surface, you first need to create sections and guide curves. Next, choose the Multisections surface button from the Surface toolbar, the Multi-sections Surface Definition dialog box is displayed.

You are prompted to select a curve. Select the first section curve; you are prompted to select new curve or select a tangent surface. Select the second section curve from the geometry area and click on the Guide selection area to activate it. You are prompted to select a curve. One by one, select the guide curves that were drawn earlier. Choose the OK button to exit the Multi-sections Surface Definition dialog, box and complete the multi section surface. Sections and guide curves to create the multisection surface.

Note while selecting the section curve, make sure the arrow associated with each section curve is pointing in the same direction. Else, the surface will result in a cusp and will not be created. In some cases, with arrows pointing in opposite direction, a twisted surface may be formed.

Select the first section curve; you are prompted to select new curve or select a tangent surface. Select the second section curve from the geometry area and click on the Guide selection area to activate it.

Menu: Insert > Surface > Multisections surface Toolbar: Surface Multisections surface

4.2.6 Creating Blended Surfaces: This tool allows you to create a surface by blending two curves. These curves can be sketched curves, wireframe geometries, or edge of existing surfaces. If you select support surfaces with curves, the resulting blend surface will be tangent to the support surfaces. To create a blend surface, draw some curves and create support surfaces. Choose the Blend button from the Surfaces toolbar.

You are now prompted to select the first curve and first support. Select the curve and support. Next, you are prompted to select the second curve and second support. Select them choose the OK button from the Blend Definition dialog box. The curves and support surfaces to create the blend surface.

4.3 Operation on Shape Geometry:

Generally, the surface models are a combination of various surfaces. You need to join, trim, split, or translate the surfaces to manage multiple surfaces. CATIA provides a number of such operation tools that can be used on the surfaces created using the tools discussed earlier in this chapter. Some of these operations are discussed in the following section.



Fig 4.2 Operation on Shape Geometry Toolbar

4.3.1 Joining Surfaces: The Join tool is used to join two adjacent surfaces or two adjacent curves. Choose Join button from the operation toolbar, the Join Definition dialog box is displayed. You are prompted to select the elements (curves or surfaces) to be joined. Select the elements that you need to join. Remember that there should not be a large gap between the entities to be joined. Choose the OK button from the Join Definition dialog box to complete the join operation. The surfaces or curves that are joined together will behave as a single entity.

CATIA provides a number of such operation tools that can be used on the surfaces created using the tools discussed earlier in this chapter. Some of these operations are discussed in the following section.

4.3.2 Splitting Surfaces: The Split tool is used to split a surface or a wireframe element using a cutting element. A wireframe element can be split using a point, another wireframe element, or a surface. A surface can be split using another surface or a wireframe element. Consider a case in which two intersecting surfaces are created. Choose the Split button from the Operations toolbar to invoke the Split Definition dialog box; you are prompted to select the curve or surface to split. Select the cylindrical surface and then select the other surface, as the cutting element. The side of the cylindrical surface that is to be removed will be displayed as transparent. You can choose the other side buttons from the Split Definition dialog box to reverse the side of the surface to be removed. You can also retain both sides of the split surface by selecting the Keep both Sides check box from the split definition dialog box. Choose the OK button to split the cylindrical surface. The transparent side will be removed.

4.3.3 Trimming Surfaces: The trim tool allows you to trim two intersecting surfaces or curves with respect to each other. When you invoke this tool, the trim definition dialog box will be displayed, Choose a surface as the first element and the other surface as the second element. One side of each surface will be displayed as transparent. You can choose the other side button to reverse the sides of surfaces to be trimmed.

4.3.4 Extrapolate: This command is used to increase the surface length in any direction.

4.3.5 Boundary: Boundary feature is used to define the boundary of a sheet metal part. This feature essentially sets the outer limits or extents of the sheet metal component, which is crucial for defining its overall shape and

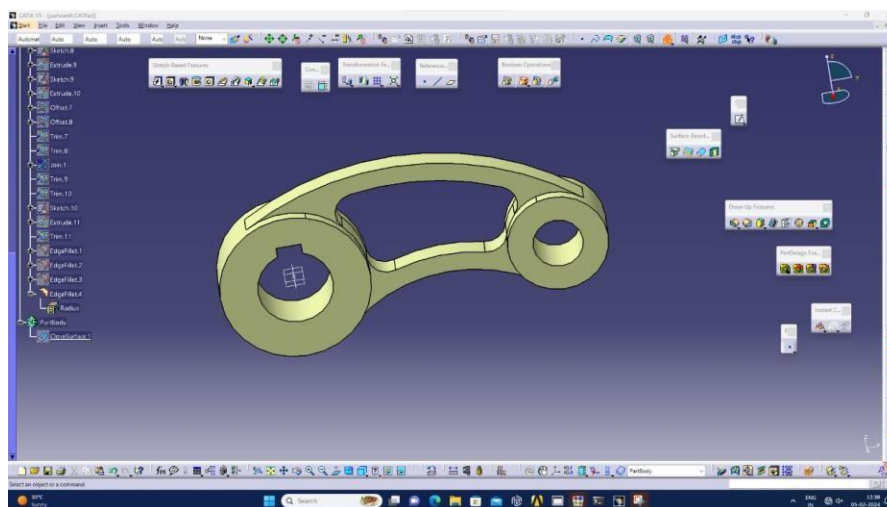


Fig 4.3 Surface Designed using Surface Design Tools

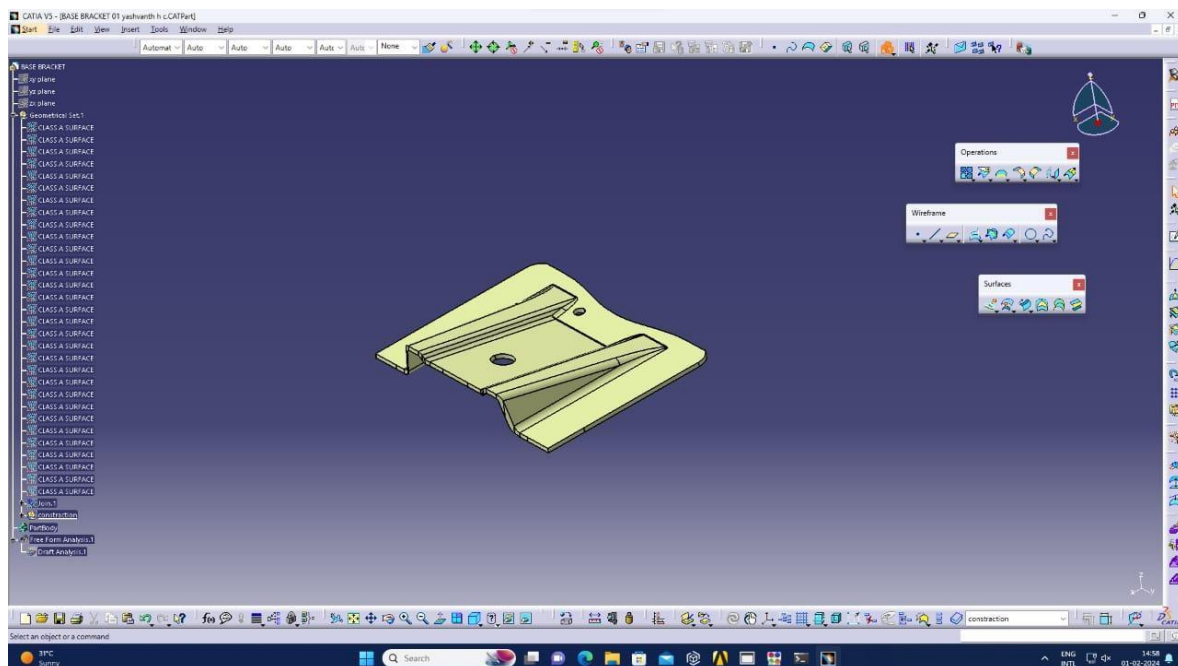


Fig 4.4 Class B and Class C Surface Design for Base Bracket

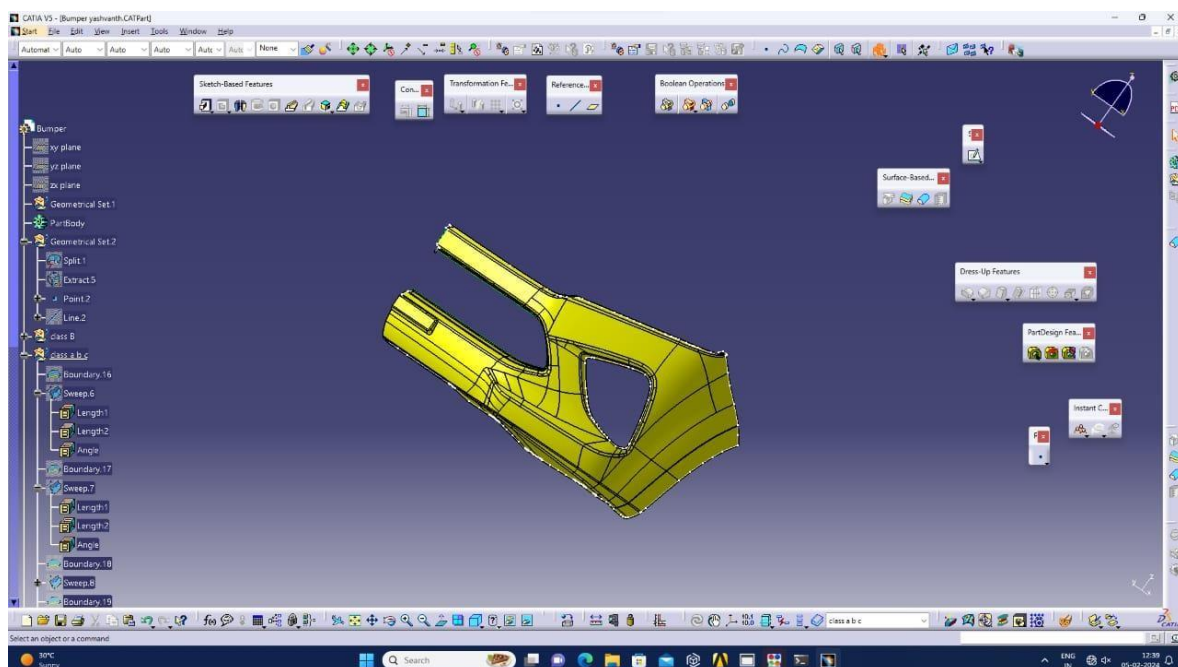


Fig 4.5 Class B and Class C Surface Design for Bumper

CHAPTER – 5

ASSEMBLY

5.1 Introduction:

This module is responsible for assembling the parts previously produced in Part Design, and it is most important for those who work in the field of machinery design or design in general, because it is the one who shows the inter-relationships between the parts of the machine or any mechanical establishment. Assembly modelling 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.

5.2 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.

5.2.1 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 file. Then the product 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.

5.2.2 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.

Then the product 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.

5.3 Product Structure Tools Toolbar

This first task shows how to insert new component, new part and existing component.



Fig 5.1 Product Structure Tools

5.3.1 Inserting a New Component: This task will show you how to insert a component into an existing assembly. This command lets you: create an instance from the reference chosen component use a context-specific representation inside it.

5.3.2 Inserting a New Part: This task will show you how to insert a part in an existing assembly.

1. In the specification tree, select Managing Components01 and click the New Part icon. If geometry exists in the assembly, the New Part: Origin Point dialog box is displayed, proposing two options to locate the part:

- Click Yes to locate the part origin point on a selected point, on another component for example.
- Click No to define the origin point of a component according to the origin point of the parent component

2. For the purposes of this task, click No to locate the part origin according to the Product! Origin point.

5.3.3 Inserting Existing Components: This task will show you how to import one or more components into an existing assembly. You can specify the order in which the files will be imported when you select the files in the Insert an Existing Component dialog box.

This task consists in replacing a component. Using the Replacement Component command means replacing one component with another.

5.3.4 Replacing a Component: This task consists in replacing a component. Using the Replacement Component command means replacing one component with another.

5.4 Constraint toolbar:

Constraints can either be dimensional or geometrical. Dimensional constraints are used to constrain the length of an element, the radius or diameter of an arc or circle, and the distance or angle between elements. Geometrical constraints are used to constrain the orientation of one element relative to another. Other common geometrical constraints include parallel, tangent, coincident, concentric, etc... Reading from left to right:



Fig 5.2 Constraint Toolbar

5.4.1 Creating a Coincidence Constraint: Coincidence type constraints are used to align elements. Depending on the selected elements, you may obtain concentricity, coaxially or coplanarity. This task consists in applying a constraint between two faces.

5.4.2 Creating a Contact Constraint:

Contact type constraints can be created between two planes or faces. The common area between the two planes can be a plane (plane contact), a line (line contact) or a point (point contact).

5.4.3 Creating an Angle Constraint: Angle type constraints fall into three categories:

1. Angle
2. Parallelism (angle value equals zero),
3. Perpendicularity (angle value equals 90°)

When setting an angle constraint, you will have to define an angle value. Note that this angle value must not exceed 900. The tolerance i.e. the smallest angle that can be used to differentiate two elements is set at 10^{-6} radians.

5.4.4 Fixing a Component: Fixing a component means preventing this component from moving from its parent component during the update operation. Note that it does not mean fixing its position according to the geometrical origin of the assembly. To fix the component location according to the geometrical origin of the assembly, check Fix in space in the constraint Properties dialog box.

5.4.5 Fixing Components Together: The Fix Together command attaches selected elements together. You can select as many components as you wish, but they must belong to the active component. This task consists in fixing two components together. Using the Auto constraint Command

The Auto constraint mode allows you to create the first possible constraint as specified in the priority list. This task consists in using the Auto constraint command to create two constraints.

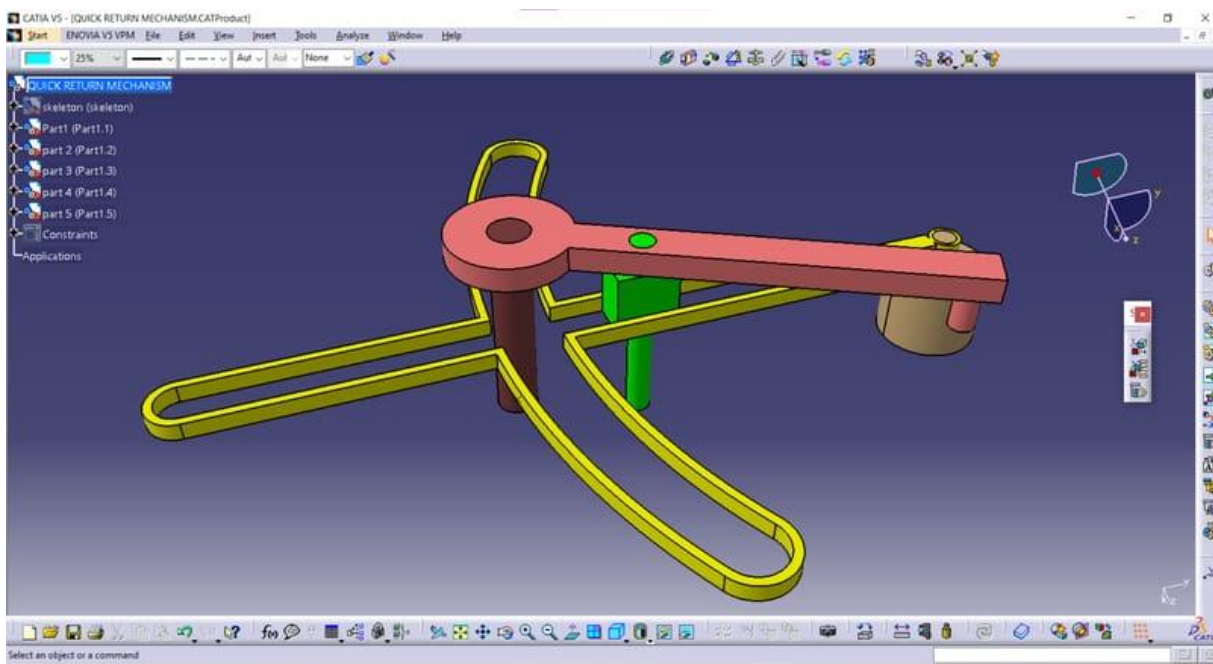


Fig 5.3 Assemble of Quick Return Mechanism

CHAPTER-6

ASSEMBLY

6.1 Knuckle Joint Design

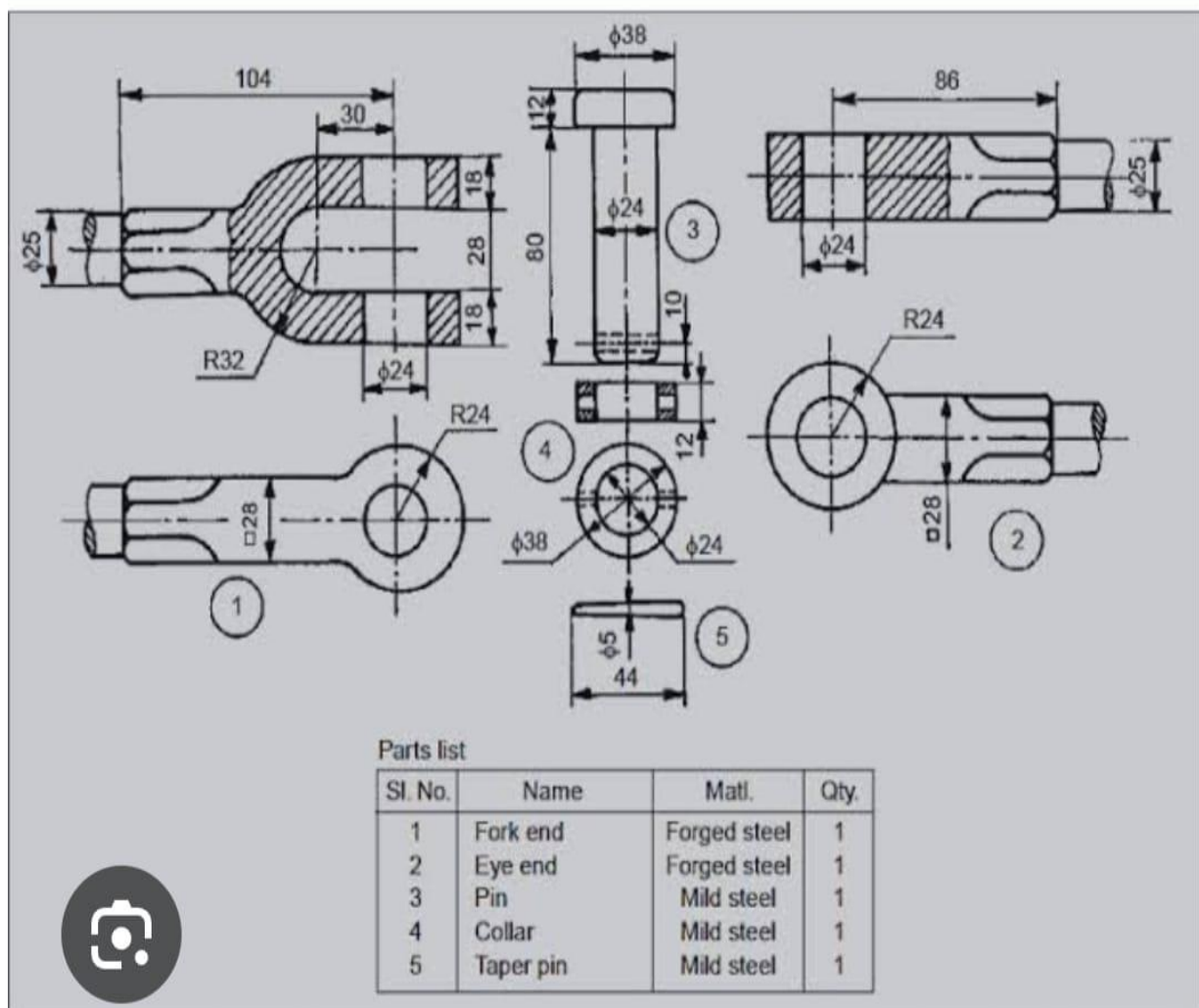


Fig 6.1: 2D drawing of a knuckle joint

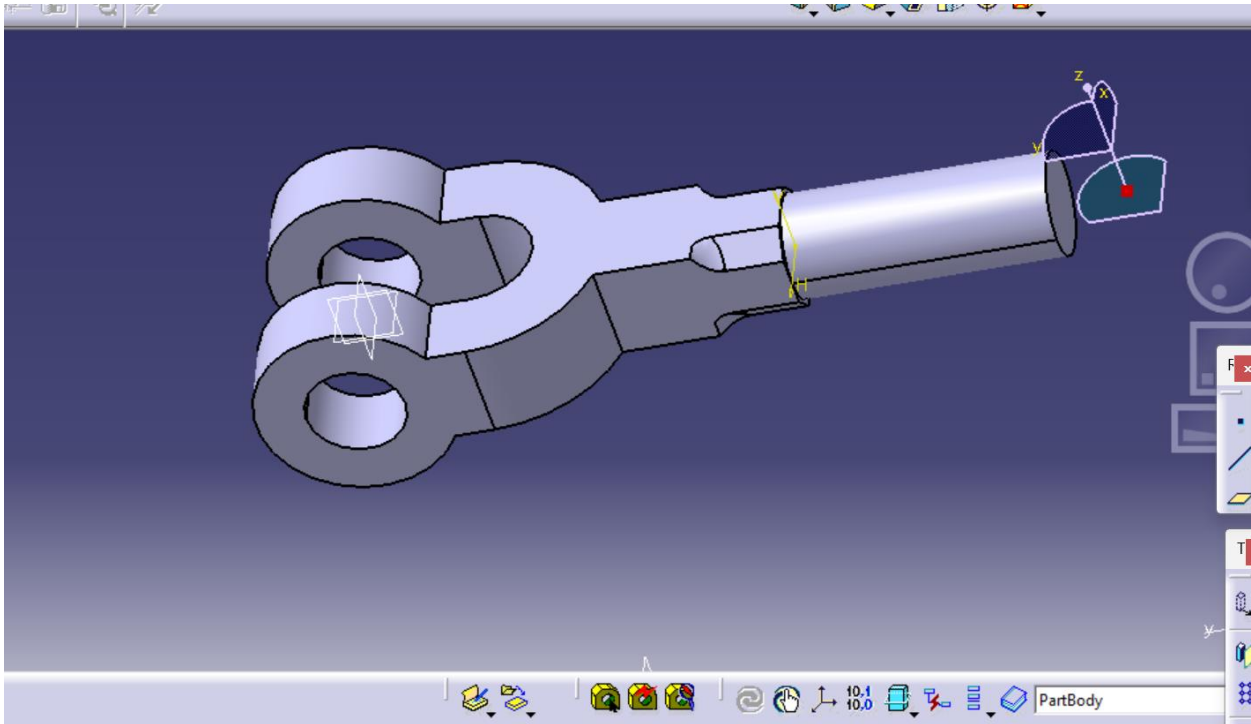


Fig 6.2 Eye end

6.2 Step-by-Step Instructions:

6.2.1 Launch CATIA and Create a New Part

- Open CATIA V5.
- Go to: File > New > Part
- Name it: Knuckle_Joint_Key and click OK.

6.2.2 Start a New Sketch

- Go to Part Design workbench.
- Select Sketch.
- Choose the YZ plane or the appropriate plane (depending on your assembly orientation).

6.2.3 Sketch the Profile of the Key

- Use the Rectangle Tool to draw a basic rectangular cross-section.
- Use Smart Dimension to define:
 - **Length** = e.g., $1.5 \times \text{pin diameter}$
 - **Width** = $1/4 \times \text{pin diameter}$
 - **Height/Thickness** = $1/4 \times \text{pin diameter}$

(Adjust based on the actual size of your knuckle joint)

6.2.4 Exit Sketch and Pad (Extrude)

- Click Exit Sketch.
- Use the **Pad** tool to extrude the key along its thickness (if you haven't already).
- Enter the **thickness** value or length of the key.

6.2.5 Apply Fillets

- Use the Edge Fillet tool to smooth sharp edges if required.

6.2.6 Add Chamfers

- Select edges that need to be beveled.
- Use the Chamfer tool and specify angle & length.

6.2.7 Optional: Assemble It

To insert this into the full Knuckle Joint Assembly:

- Switch to Assembly Design Workbench.
- Import the fork, eye, pin, and key.
- Use Constraints (coincidence, contact, offset) to assemble correctly.

6.2.8 Modify as Needed:

You can update key dimensions using the **Sketch** and **Pad** features anytime.

Exit Sketch and Pad (Extrude)

- Click Exit Sketch.
- Use the Pad tool to extrude the key along its thickness (if you haven't already).
- Enter the thickness value or length of the key.
- Select edges that need to be beveled.
- Use the Chamfer tool and specify angle & length

6.2.9 Exit Sketch and Pad (Extrude)

- Click Exit Sketch.
- Use the Pad tool to extrude the key along its thickness (if you haven't already).
- Enter the **thickness** value or length of the key.
- Click Exit Sketch.
- Use the Pad tool to extrude the key along its thickness (if you haven't already).
- Enter the thickness value or length of the key.
- Select edges that need to be beveled.
- Use the Chamfer tool and specify angle & length

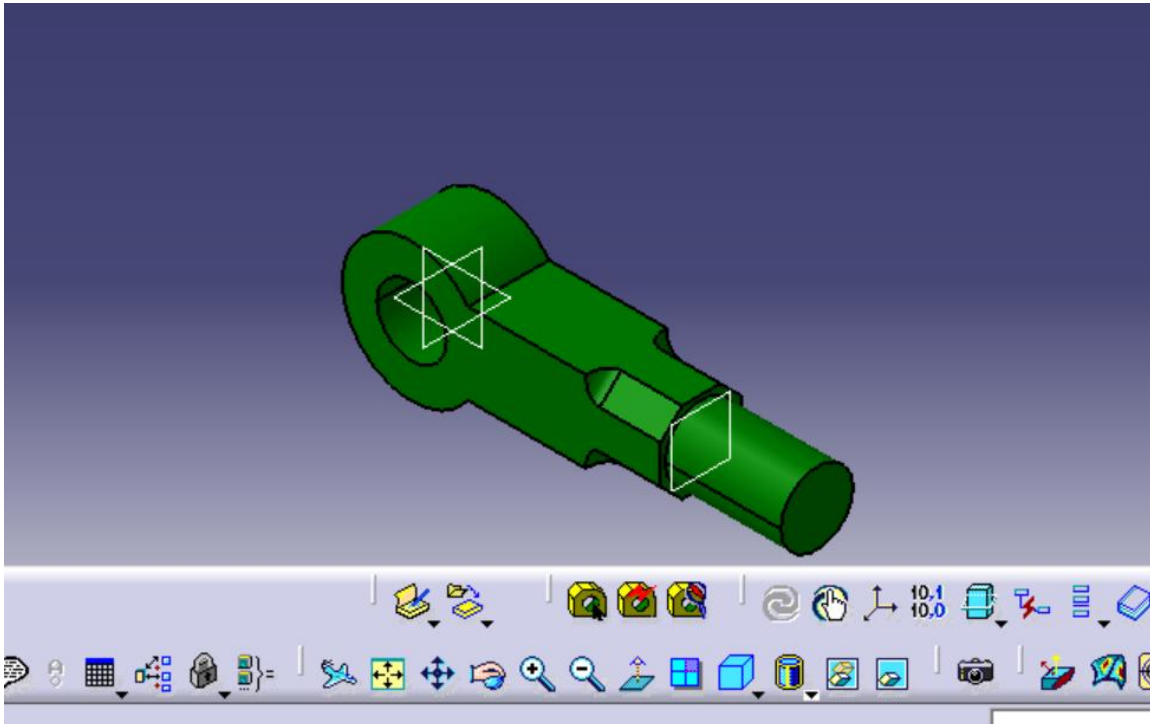


Fig 6.3 Fork End

6.3 Create a New Part

- Open CATIA V5.
- Click on File > New > Part.
- Name the part: Fork_End.

6.3.1 Create the Base Block

- Go to the Part Design workbench.
- Select the XY plane.
- Click on Sketch.

6.3.2 In Sketch:

- Draw a **rectangle** with dimensions:
 - **Width**: equal to the outer width of fork
 - **Height**: equal to the fork's overall height
- Use **Dimension** tool to define size (example: 60 mm × 100 mm).
- **Exit Sketch**.
- Use the **Pad** tool to extrude (thickness of fork body, say 20 mm).

6.3.3 Create the Fork Legs (Side Projections)

- Select the front face.
- Create a **new sketch**.
- Draw two side rectangular blocks (legs of the fork).
- Dimension the legs:

- Leg thickness (say 10 mm)
- Gap between legs = equal to the eye end thickness (say 20 mm)
- Use **Pad** to extrude the legs **forward** (example: 30 mm).

6.3.4 Make the Hole for Pin

- Select **one inner face** of the fork leg.
- Click **Sketch**, draw a **circle** centered between the two legs.
 - Position: centered vertically, and at appropriate height from base (say 50 mm)
 - Diameter: same as pin (say 12 mm)
- **Exit Sketch**.
- Use **Pocket** to cut through **both legs** (select "Through All").

6.3.5 Add Fillets or Chamfers (Optional)

- Use **Edge Fillet** to round off sharp corners.
- Or use **Chamfer** where applicable.

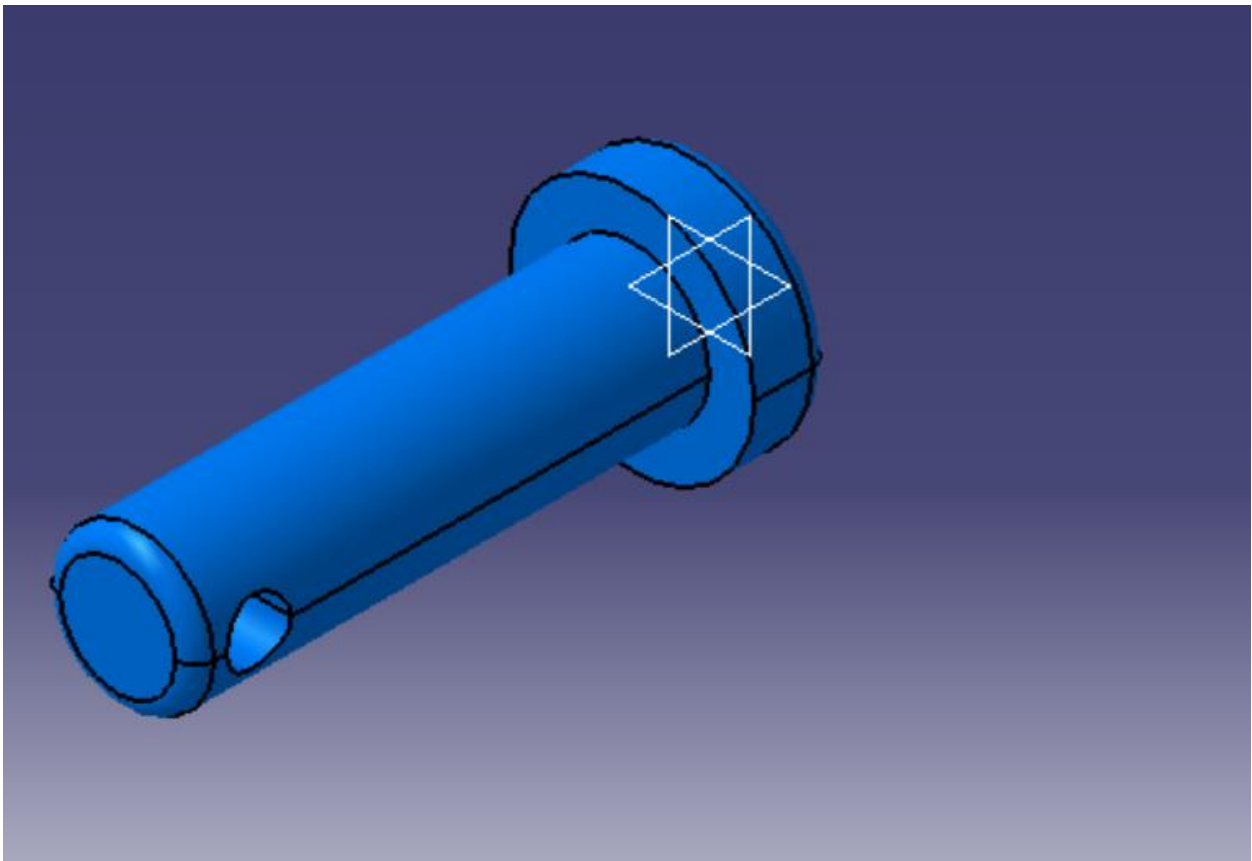


Fig 6.4 Model Pin

6.4 Create a New Part

- Open CATIA V5.
- Click File > New > Part.

- Name it: `Knuckle_Joint_Pin`.

6.4.1 Start a New Sketch

- Switch to Part Design Workbench.
- Click Sketch.
- Choose the YZ plane (side view for revolving a pin).

6.4.2 Sketch Half-Section Profile of Pin

Use the Profile Tool to draw half of the pin's cross-section (for revolution):

- Draw a vertical centerline (axis of revolution).
- Draw the profile of the pin:
 - **Shaft diameter:** typically same as hole (say 12 mm → draw 6 mm from centerline)
 - **Length:** total length including head and collar (e.g., 60 mm)
 - Add a **collar or head** (optional):
 - Head diameter: larger than shaft (e.g., 18 mm)
 - Head thickness: 5–8 mm
 - Collar (opposite end) to prevent over-insertion (optional)

Use **Constraint** tool to dimension the sketch fully.

6.4.3 Exit Sketch and Use Shaft (Revolve)

- Exit Sketch.
- Use the **Shaft** tool (Insert > Shaft).
- Select the vertical axis as Axis of Revolution.
- The profile will revolve 360° to create the 3D pin.

6.4.4 Add Features (Optional)

- Create a sketch on the shaft where you want the groove.
- Use **Slot or Rectangle** profile and cut it using **Pocket**.

6.4.5 Chamfer or Fillet

- Use **Fillet** tool on shaft edges for smooth finish.
- Use **Chamfer** on head if required.

Add a collar or head (optional):

- Use **Radius** on head if required. Head diameter: larger than shaft (e.g., 18 mm)
- Head thickness: 5–8 mm
- Collar (opposite end) to prevent over-insertion (optional)

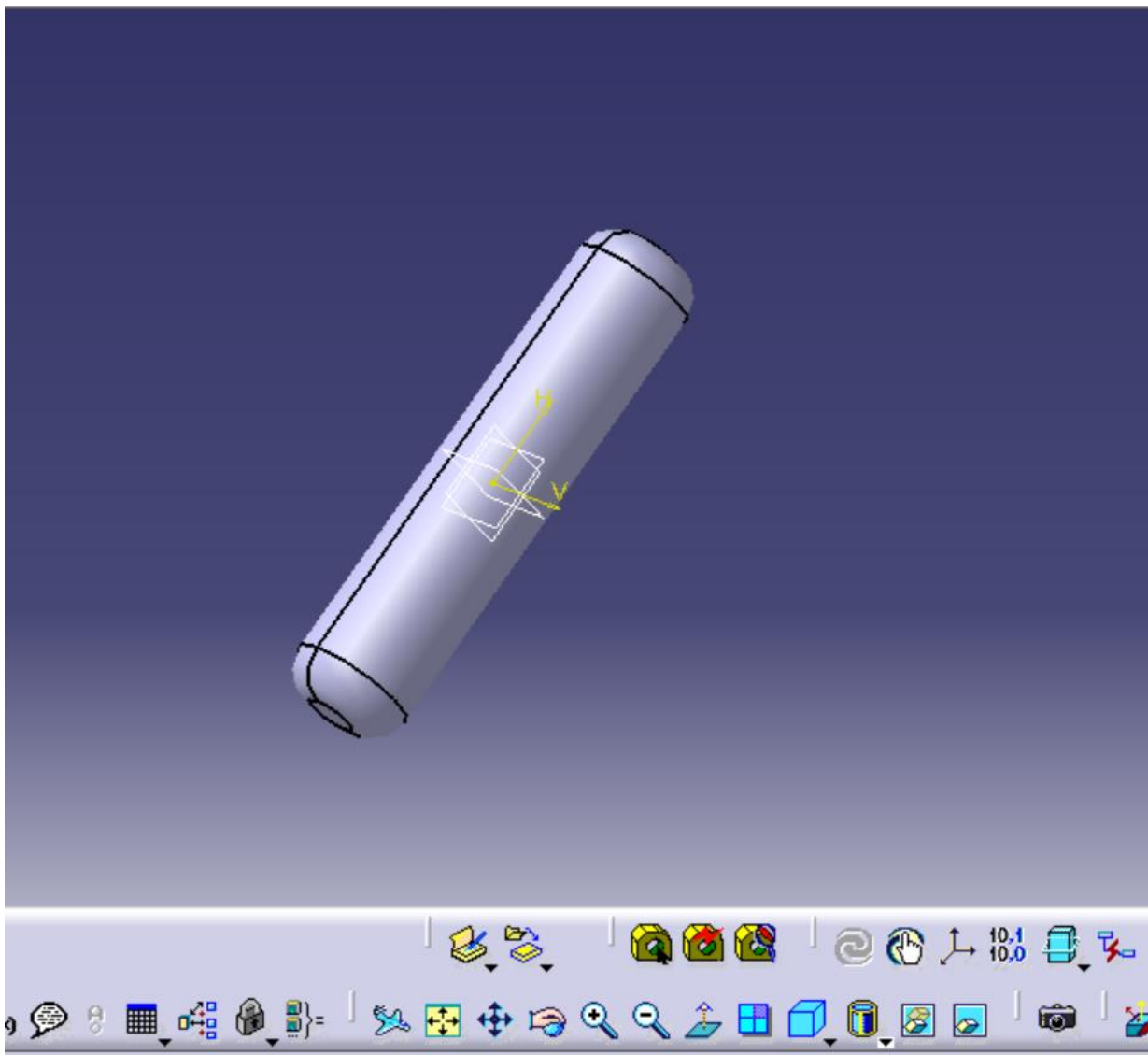


Fig 6.5 Taper Pin

6.5 Open a New Part

- Open CATIA V5.
- Go to File > New > Part.
- Name it: Taper_Pin.

6.5.1 Start a New Sketch

- Go to Part Design workbench.
- Click Sketch.
- Choose the YZ plane (side view for revolving a pin).

6.5.2 Draw Half-Section Profile of Taper Pin

- Draw a **vertical centerline** (this will be the axis of revolution).
- Draw a **tapered triangle** representing half of the pin:
 - Bottom point starts at origin.
 - Height (length of pin): 50 mm
 - Bottom width (small end): 4.5 mm from center (to make 9 mm dia)
 - Top width (large end): 5 mm from center (to make 10 mm dia)

Use the **Constraint** tool:

- Set height = 50 mm
- Set base and top widths = 9 mm and 10 mm total diameter respectively (i.e., 4.5 mm and 5 mm from center)

6.5.3 Exit Sketch and Use Shaft

- Click **Exit Sketch**.
- Go to Insert > Shaft (or click Shaft icon).
- Select the vertical centerline as axis of revolution.
- Revolve the sketch **360°** to create a full solid taper pin.

6.5.4 Add Fillets or Chamfers (Optional)

- Use the **Fillet** tool for smoothing sharp edges.
- Use the **Chamfer** tool to bevel edges if required.

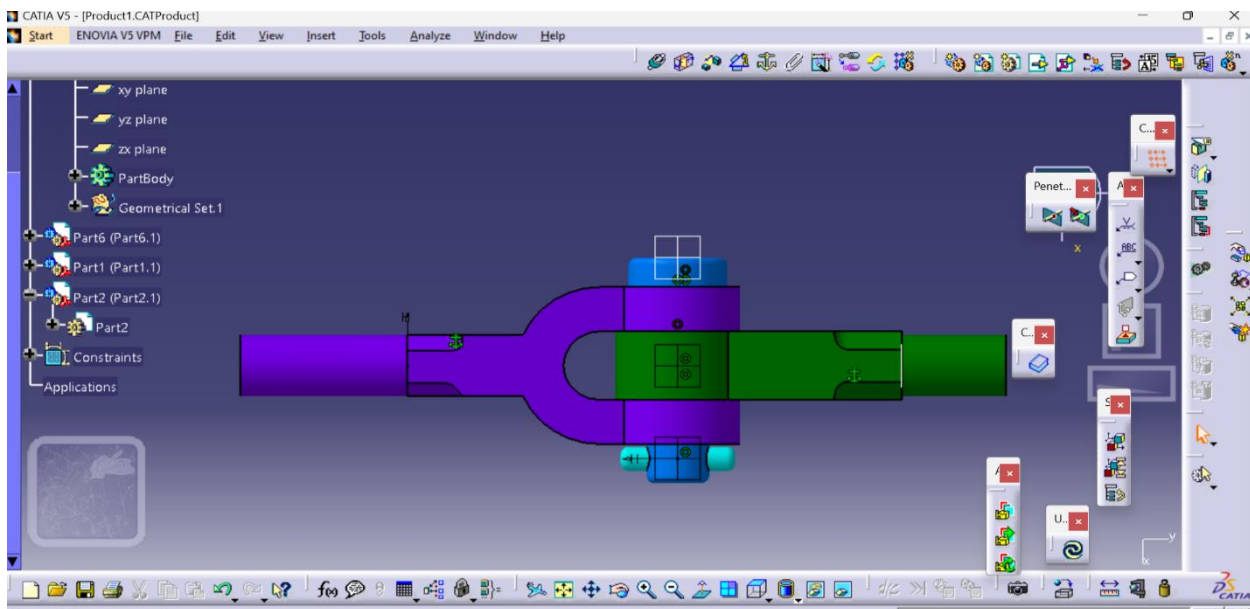


Fig 6.6 Assemble of knuckle joint

CHAPTER-07

Conclusion:

CATIA (Computer-Aided Three-dimensional Interactive Application) is a powerful and versatile software suite developed by Dassault Systèmes. It integrates CAD, CAM, and CAE capabilities, making it a go-to tool for product design, engineering, and manufacturing across industries like aerospace, automotive, and architecture.

The design and assembly of the **Knuckle Joint** using **CATIA V5 Part Design Workbench** provided a comprehensive understanding of 3D modeling, precision sketching, and mechanical assembly techniques. Each component—**fork end, eye end, pin, and taper key**—was modeled with accurate dimensions and design intent, showcasing the parametric power of CATIA.

Through this project, we explored:

- Creating individual mechanical parts with constraints and dimensions
- Using advanced features like **Pad, Pocket, Shaft, Boolean operations, and Assembly constraints**
- Understanding the functional relationships between parts in a real-world mechanical joint

This exercise not only strengthened CAD modeling skills but also highlighted the importance of **design accuracy, part fitment, and mechanical functionality** in virtual prototyping. CATIA's intuitive interface and robust toolset made it possible to visualize and verify the knuckle joint mechanism before actual manufacturing.

In conclusion, the knuckle joint design in CATIA successfully demonstrated the integration of theoretical knowledge with practical design tools—forming a foundation for more complex mechanical system modeling.

A **knuckle joint** is a mechanical joint used to connect two rods under tensile load, allowing limited angular movement. It's simple in design, easy to assemble and disassemble, and commonly used in applications like tie rods, bicycle chains, and valve mechanisms.

In short:

It's a strong, flexible, and efficient joint for transmitting axial force where slight movement is needed.

- It's ideal for **axial tensile force transmission**.
- Offers **slight flexibility** for angular motion.
- Commonly used in **bicycle chains, tie rods, and valve mechanisms**.
- Preferred where **quick disconnection or adjustment** is needed.

REFERENCE:

Useful Links for Knuckle Joint Design in CATIA

1. [**Design and Finite Element Analysis of Knuckle Joint Using CATIA and ANSYS Workbench**](#)
A detailed paper that walks through modelling in CATIA and stress analysis in ANSYS.
2. [**Knuckle Joint Assembly in CATIA V5 – LearnVern Tutorial**](#)
A beginner-friendly video tutorial that explains how to assemble a knuckle joint step-by-step.
3. [**Knuckle Joint Assembly in CATIA V5 – GrabCAD**](#)
Downloadable 3D models and parts of a knuckle joint created in CATIA.
4. [**YouTube Tutorial – How to Design and Assemble a Knuckle Joint in CATIA**](#)
A visual guide for beginners showing the full design and assembly process.