Visvesvaraya Technological University

Belagavi, Karnataka

Internship Report

On

"DESIGN & ASSEMBLY OF FLANGE COUPLING BY USING CATIA V5"

An internship report submitted in partial fulfilment for the award of the degree of Bachelor of Engineering in Mechanical Engineering of Visvesvaraya Technological University, Belagavi

Presented by

Jayanth DM
USN:3BR21ME445

Under Guidance of

Mr. K Raghavendra

Assistant Professor.

DEPARTMENT OF MECHANICAL ENGINEERING

BALLARI INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(Recognized by Govt.of Karnataka, approved by AICTE, New Delhi & Affiliated to

Visvesvaraya Technological University, Belagavi)

"Jnana Gangotri" Campus, No.873/2, Ballari-Hospete Road, Allipur,

Ballari-583 104 (Karnataka) (India)

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DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that **Jayanth DM** with **USN: 3BR21ME445,** of 8th semester **of Mechanical Engineering** has satisfactorily presented internship report on the topic entitled **"DESIGN OF FLANGE COUPLING BY USING CATIA V5"** carried out at **"BALLARI INSTITUTE OF TECHNOLOGY AND MANAGEMENT"** as prescribed by **Visvesvaraya Technological University, Belagavi** for partial fulfillment for the award of UG degree during the academic year **2023-2024**

Signature of internship Guide
Mr. K Raghavendra
Assistant Professor.

Signature of the HOD
Dr. V Venkata Ramana

Signature of Principal
Dr. Yadavalli Basavaraj

External Viva

Name of the Examiners: **Signature with Date**

1.

2.

ACKNOWLEDGEMENT

This satisfaction that I feel at the successful completion of Internship, it would be incomplete if I did not mention the names of people, whose noble gesture, affection, guidance, encouragement and support crowned by efforts with success.

I express my deep sense of gratitude to the management of Ballari institute of Technology and Management, Bellary, for providing me the congenial environment in the college.

I express my deep sense of gratitude to **Dr. Yadavalli Basavaraj, Principal**, Ballari Institute of Technology and Management, Bellary, for providing me the congenial environment in the college. I am thankful to for providing the facilities in the department to do this Internship work.

I express my deep sense of gratitude to **Dr. V Venkata Ramana, HOD**, Ballari Institute of Technology and Management, Bellary, for providing me the congenial environment in the college. I am thankful to for providing the facilities in the department to do this Internship work.

I am deeply indebted to my internship guide **Mr. K Raghavendra, Assistant Professor.** Department of ME & for consistently providing me with the required guidance to help me in the timely and successful completion of this internship. In spite of their extremely busy schedules in Department, they were always available to share with me their deep insights, wide knowledge and extensive experience.

I extended sincere thanks to all the teaching staff members of ME Department, for constant support and help during the Internship work.

Jayanth DM

USN:3BR21ME445

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"Jnana Gangotri" Campus, No.873/2, Ballari-Hosapete Road, Near Allipura, Ballari - 583 104 (Karnataka)

In Association with

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CERTIFICATE OF INTERNSHIP

This certificate is awarded to

Name Mr. JAYANTH D M *USN* 3BR21ME445

Student of Mechanical Engineering Department,

Ballari Institute of Technology and Management, Ballari for

Successful Completion of Internship Modules on

CATIA-V5

from 21st February - 2024 *to* 22nd March - 2024

Dr. Yadavalli Basavaraj
Principal
BITM, Ballari.

R. Shankarnarayanan
COO
Harita Techserv Limited

The Training was conducted from 21-02-2024 to 22-03-2024

Sl.No.	Modules	Hours Delivered	Rating (A-D)
1	Drawing Basics	10	A
2	Sketching	15	A
3	Solid Modeling	15	A
4	Drafting and detailing	15	B+
5	Assembly	15	B+
6	Sheet Metal	15	B

All Modules include one or more capstone projects along with lectures

A : Outstanding, B+ : Very Good, B : Good, C : Average, D : Fair

DECLARATION BY THE STUDENT

I am **Jayanth DM** student of B.E in Mechanical Engineering of Ballari Institution of Technology and Management, Ballari, hereby declare that the desertion **"DESIGN OF FLANGE COUPLING BY USING CATIA V5"** embodies the report my internship work carried out by me during 8thsemester

B.E under the Supervision and Guidance of **Mr. K Raghavendra, Assistant Professor,** Mechanical Engineering, BITM. This work has been submitted for the partial fulfilment for the award of the degree of Bachelor of Engineering in Mechanical Engineering during the academic year 2023-2024.

I have not submitted this Internship work to any other university or institution for the award of any degree.

Jayanth DM
USN:3BR21ME445
Signature of Student

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INTRODUCTION TO CATIA

CATIA software (**computer-aided three-dimensional interactive application**) is a multi-platform software suite for COMPUTER AIDED DESIGN (CAD), COMPUTER AIDED MANUFACTURING (CAM), COMPUTER AIDED ENGINEERING (CAE), PLM and 3D, developed by the French company DASSAULT SYSTEMS.

CATIA started as an in-house development in 1977 by French aircraft manufacturer AVIONS MARCEL DASSAULT, at that time customer of the CADAM software to develop Dassault's Mirage fighter jet. It was later adopted by the aerospace, automotive, shipbuilding, and other industries.

Initially named CATI (conception assisted tridimensional interactive – French for interactive aided three-dimensional design), it was renamed CATIA in 1981 when Dassault created a subsidiary to develop and sell the software and signed a non-exclusive distribution agreement with IBM.

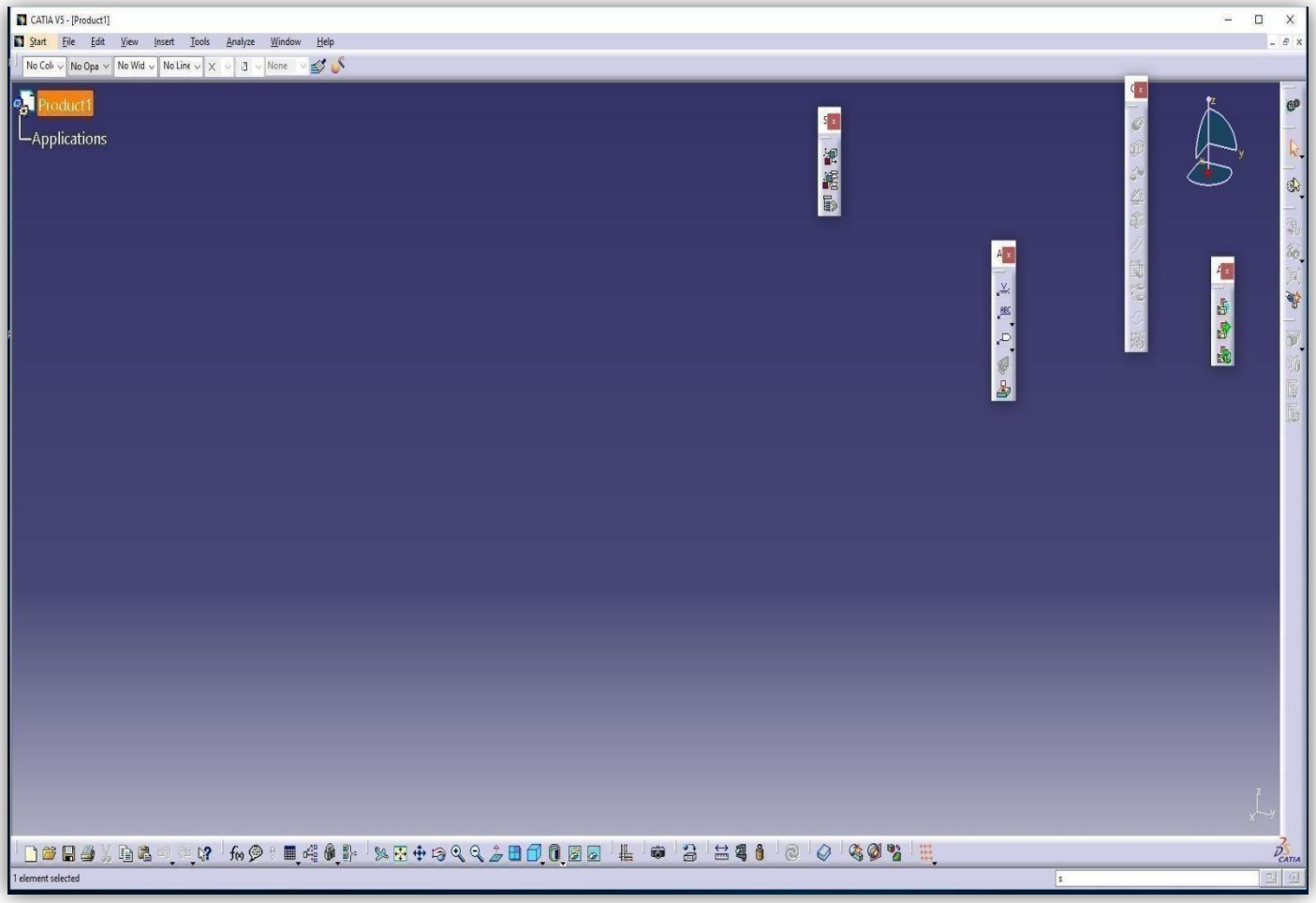
Scope of application

Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAX), including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM).

CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical, fluid and electronic systems design, mechanical engineering and systems engineering.

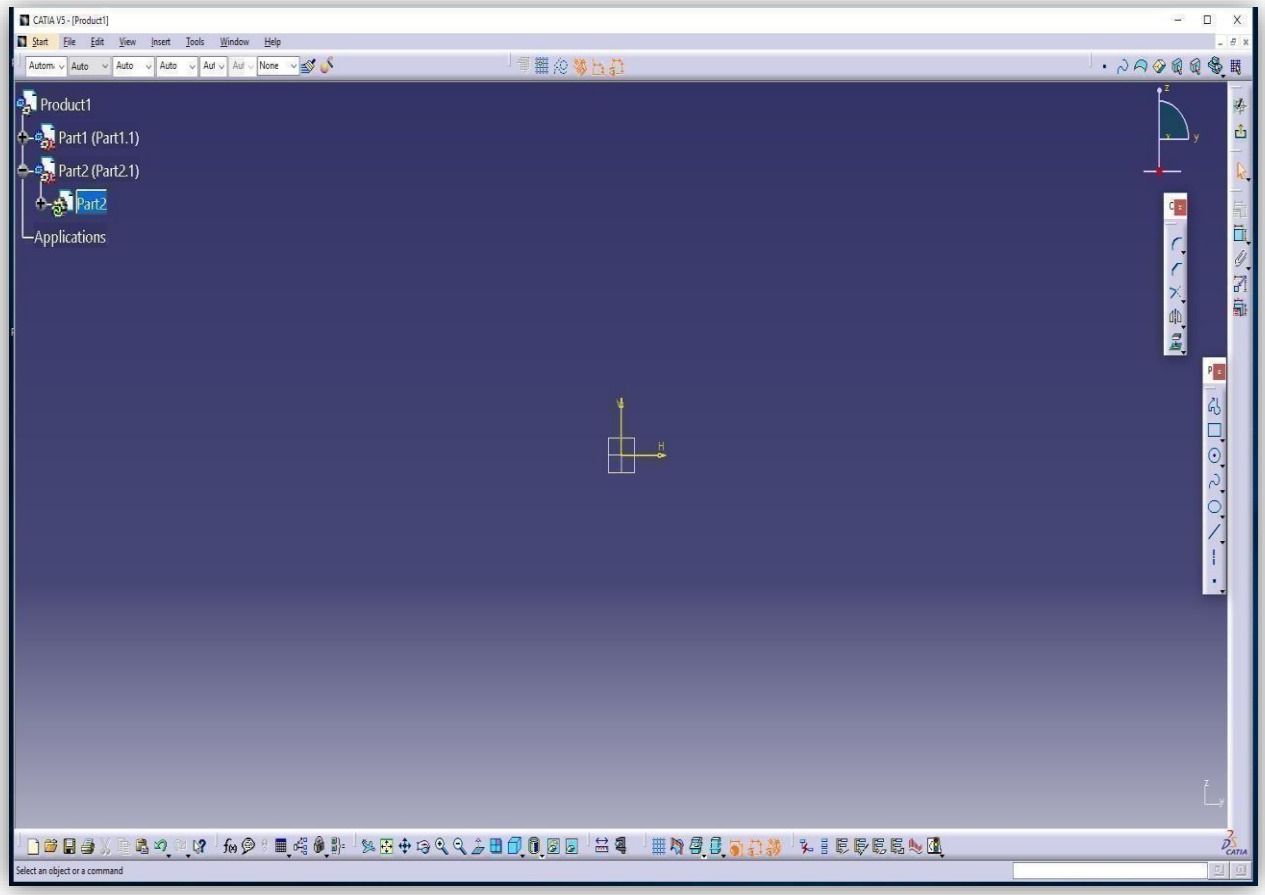
CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

CATIA ENVIRONMENT



CATIA offers a complete engineering toolset within a single working environment. Products for disciplines as diverse as Composites and Electrical design sit side by side with traditional solids, surfacing and drafting workbenches within a seamless and consistent user interface.

The Sketcher Workbench



The Sketcher workbench is a set of tools that helps you create and constrain 2D geometries. Features (pads, pockets, shafts, etc...) may then be created solids or modifications to solids using these 2D profiles. You can access the Sketcher workbench in various ways.

Two simple ways are by using the top pull down menu (Start – Mechanical Design – Sketcher), or by selecting the Sketcher icon. When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or after you select the Sketcher

icon. To exit the sketcher, select the Exit Workbench icon. The Sketcher workbench contains the following standard workbench specific toolbars.

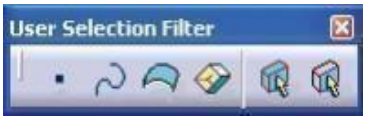
• **Profile toolbar**: The commands located in this toolbar allow you to create simple geometries (rectangle, circle, line, etc...) and more complex geometries (profile, spline, etc...).

• **Operation toolbar**: Once a profile has been created, it can be modified using commands such as trim, mirror, chamfer, and other commands located in the operation toolbar.

• **Constraint toolbar**: Profiles may be constrained with dimensional (distances, angles,) or geometrical (tangent, parallel, etc.) constraints using the commands located in this toolbar ..

• **Sketch tools toolbar**: The commands in this toolbar allow you to work in different modes which make sketching easier

• **User Selection Filter toolbar**: Allows you to activate different selection filters.



• **Visualization toolbar**: Allows you to, among other things to cut the part by the sketch plane and choose lighting effects and other factors that influence how the part is visualized.

• **Tools toolbar**: Allows you to, among others other things, to analyze a sketch for problems, and create a datum. The Sketch tools Toolbar. The Sketch tools toolbar contains icons that activate and deactivate different work modes. These work modes assist you in drawing 2D profiles. Reading from left to right, the toolbar contains the following work modes; (Each work mode is active if the icon is orange and inactive if it is blue.)

• **Grid**: This command turns the sketcher grid on and off.

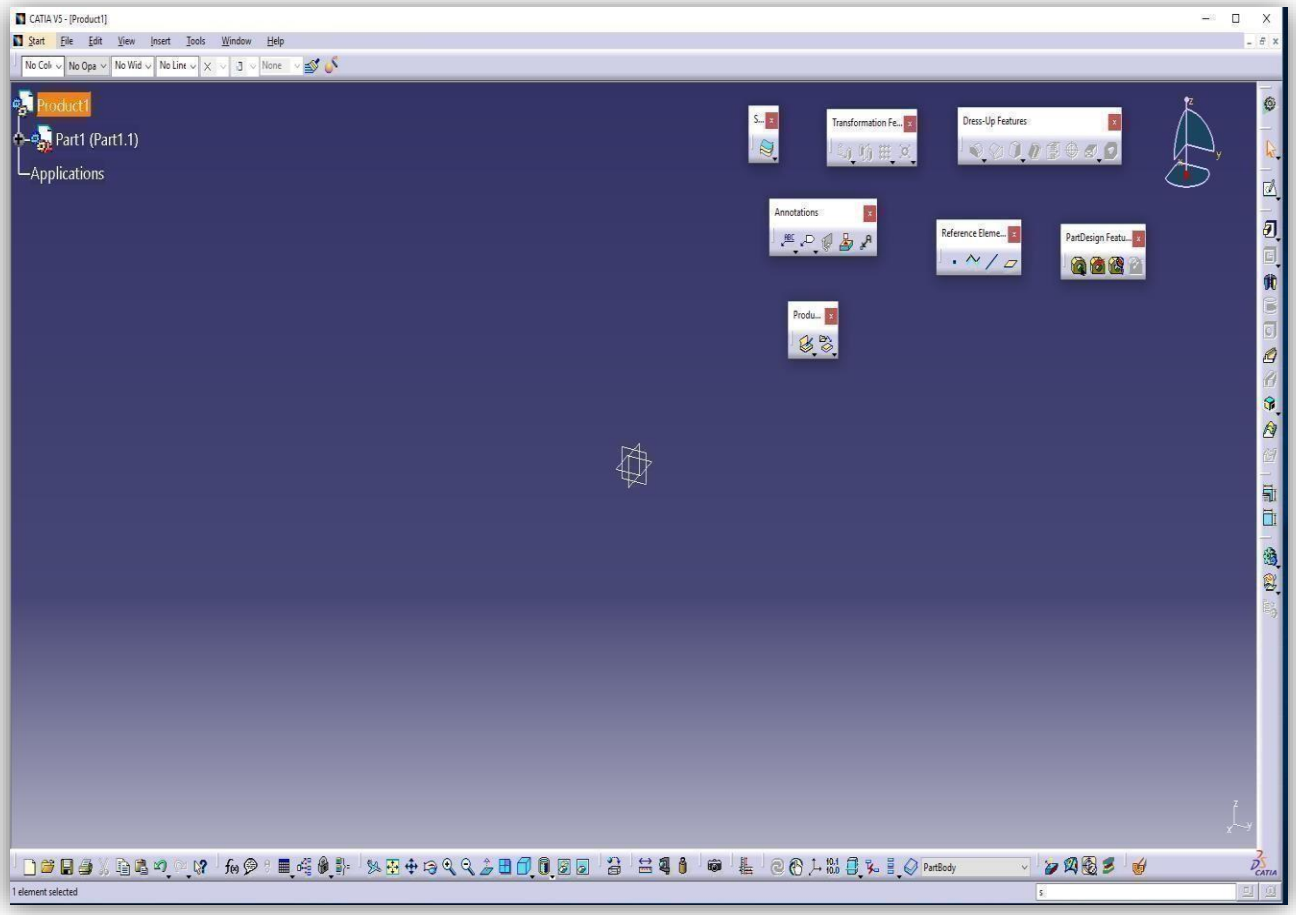
• **Snap to Point:** If active, your cursor will snap to the intersections of the grid lines.

• **Construction / Standard Elements:** You can draw two different types of elements in CATIA a standard element and a construction element. A standard element (solid line type) will be created when the icon is inactive (blue). It will be used to create a feature in the Part Design workbench. A construction element (dashed line type) will be created when the icon is active (orange). They are used to help construct your sketch but will not be used to create features.

• **Geometric Constraints**: When active, geometric constraints will automatically be applied such as tangencies, coincidences, parallelisms, etc...

Dimensional Constraints: When active, dimensional constraints will automatically be applied when corners (fillets) or chamfers are created, or when quantities are entered in the value field. The value field is a place where dimensions such as line length and angle are manually entered.

PART BODY WORKBENCH



The Part Design application makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface, from sketching inan assembly context to iterative detailed design. Part Design application will enable you to accommodate design requirements for parts of various complexities, from simple to advanced.

This application, which combines the power of feature-based design with the flexibility of a Boolean approach, offers a highly productive and intuitive design environment with multiple design methodologies, such as post-design and local 3D parameterization.

As a scalable product, Part Design can be used in cooperation with other current or future companion products such as Assembly Design and Generative Drafting.

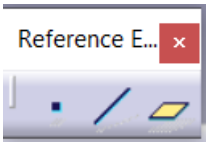
The part design workbench contains the following standard workbench specific toolbars

- Sketch based features – creating geometric features from sketches.

- Dress –up features – creates dress up features for existing geometry.

- Boolean operations – creates geometry from Boolean operation.

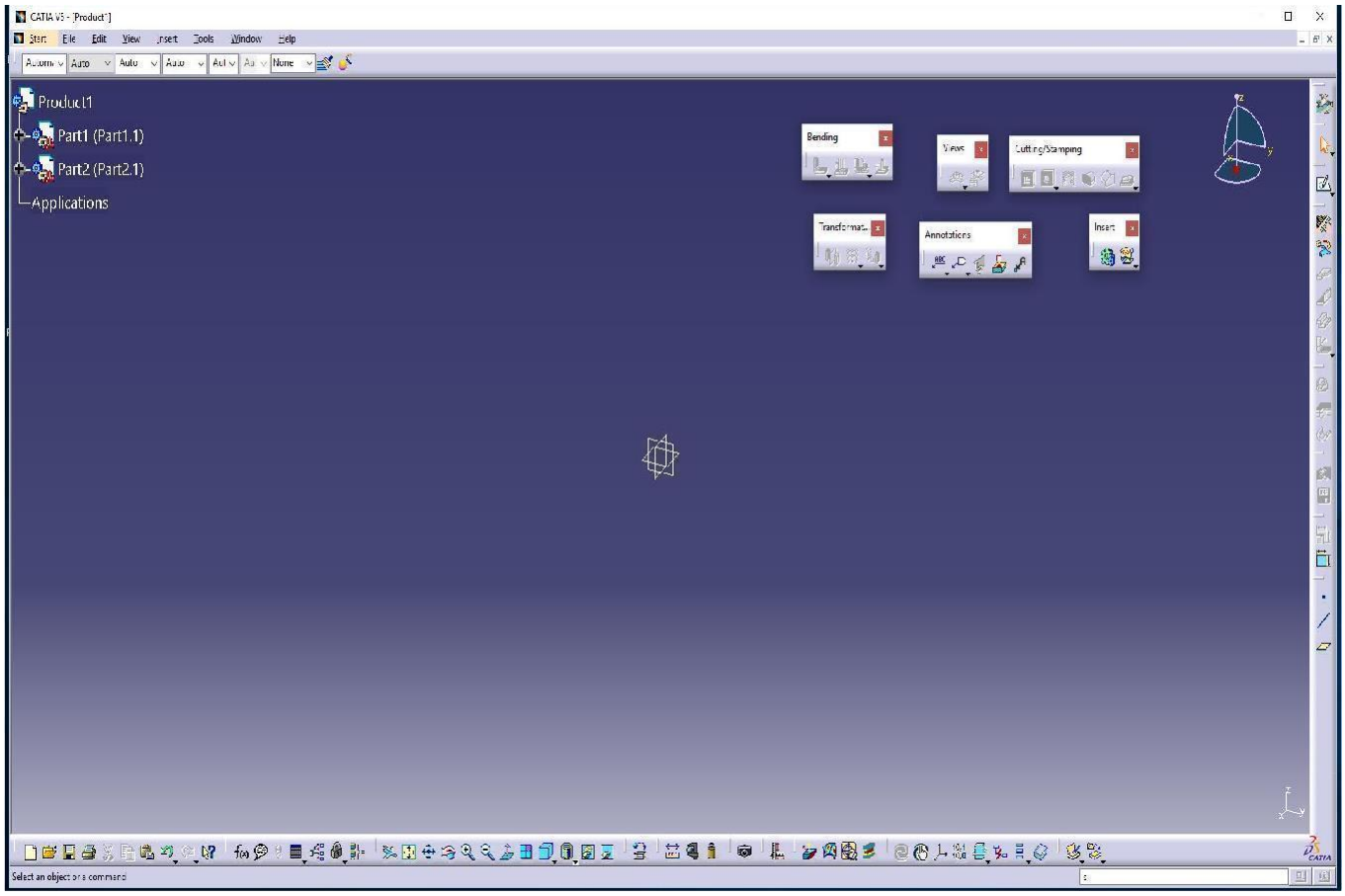
- Reference elements – creates 3D points, surface based features lines and planes.



- Transformations – applies transformation operation to features.

- Surface based features – creates surface-based features.

SHEET METAL WORKBENCH

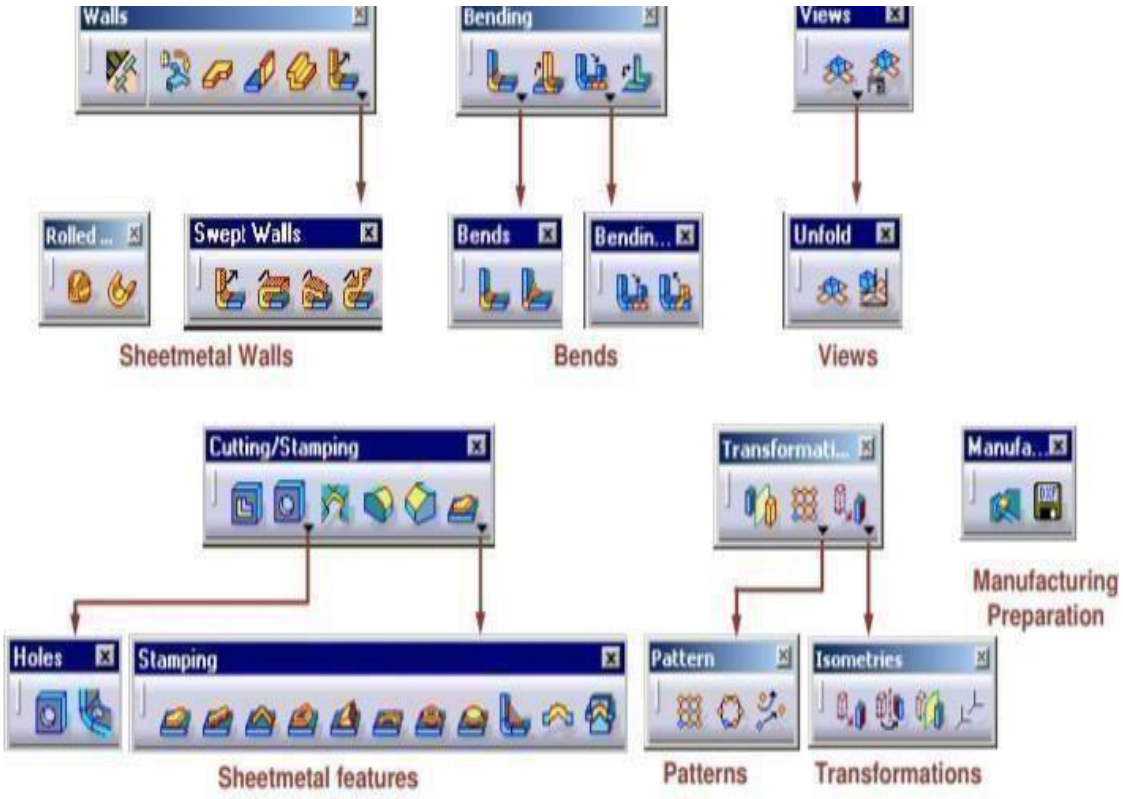


The Generative Sheetmetal Design workbench is a new generation product offering an intuitive and flexible user interface. It provides an associative feature- based modeling, making it possible to design sheet metal parts in concurrent engineering between the unfolded or folded part representation.

Generative Sheetmetal Design offers the following main functions:

- Associative and dedicated sheet metal feature-based modeling
- Concurrent engineering between the unfolded or folded part representation
- Access to company-defined standards tables
- Dedicated drawing capability including unfolded view and specific settings.

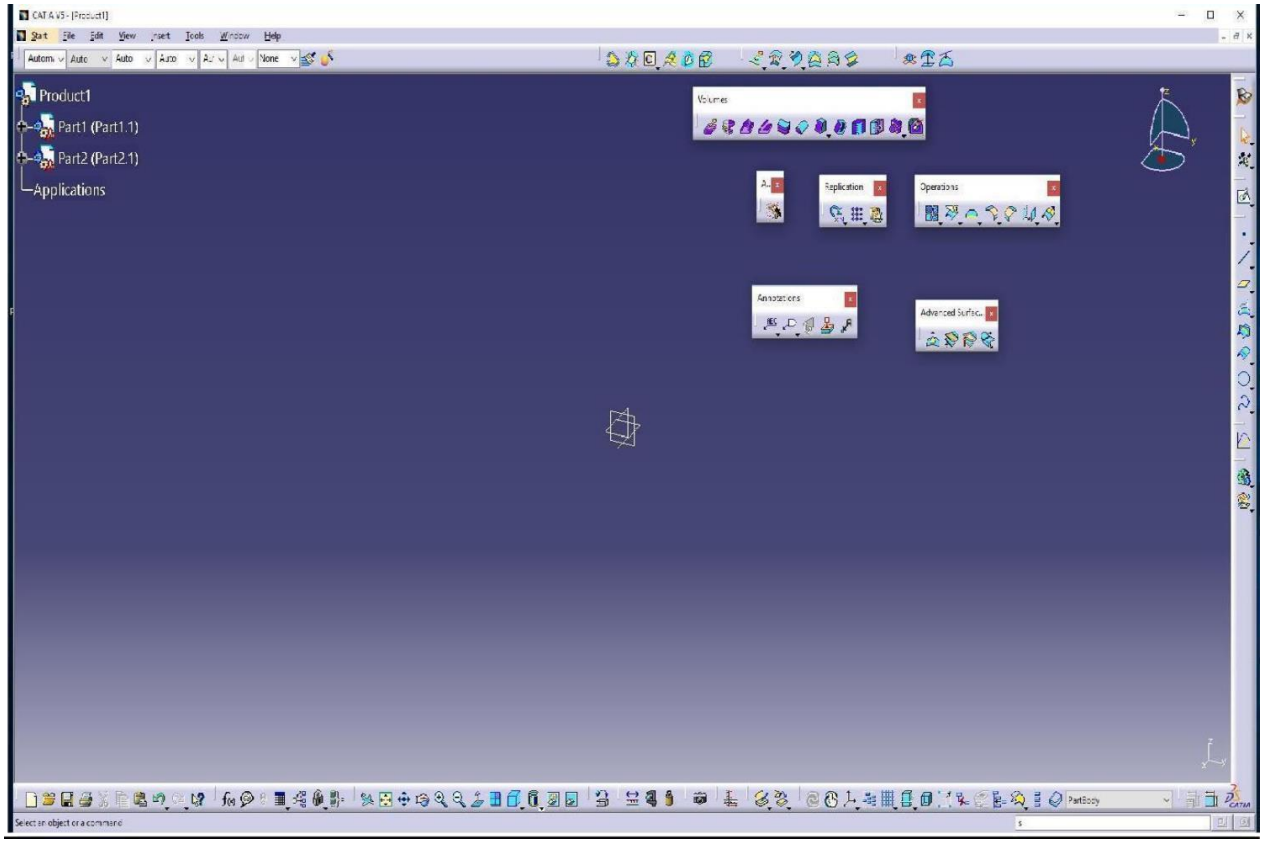
In the "Generative Sheetmetal Design" workbench, several toolbars provide logical grouping of functions for enhanced accessibility. These toolbars are shown below:



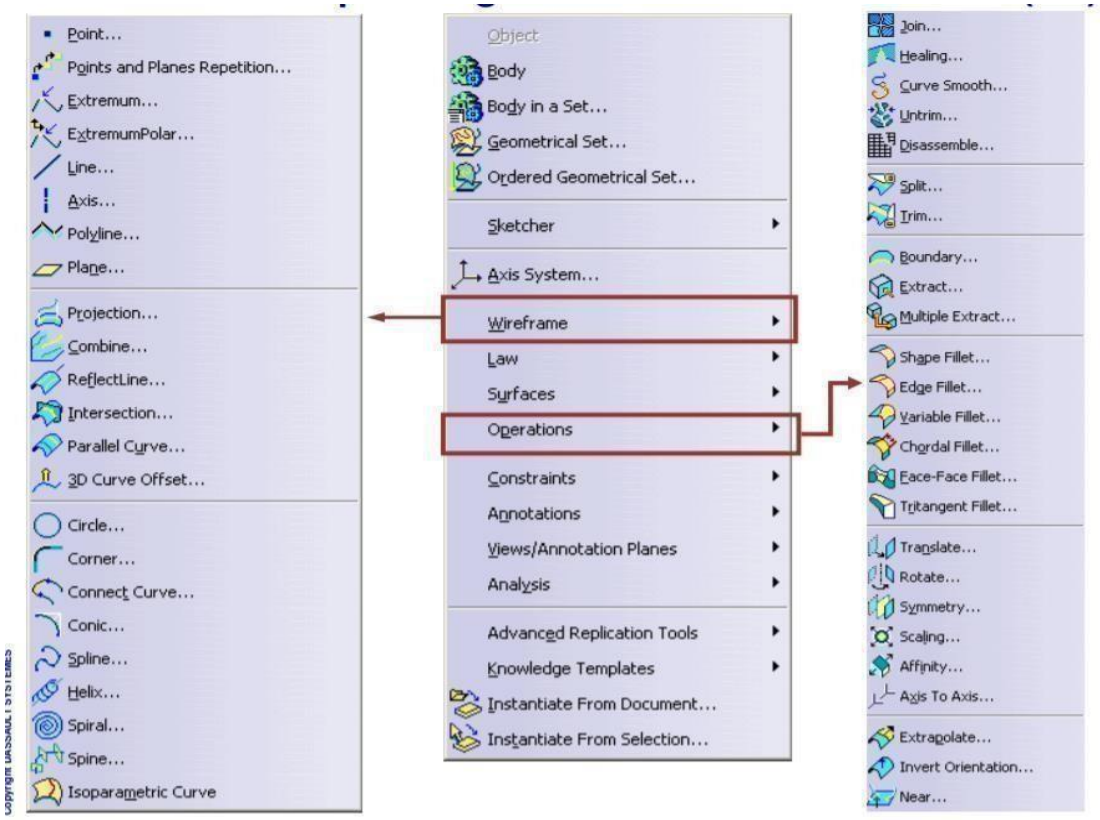
Here are some more tools in Generative Sheetmetal Design workbench.

SURFACING WORKBENCH

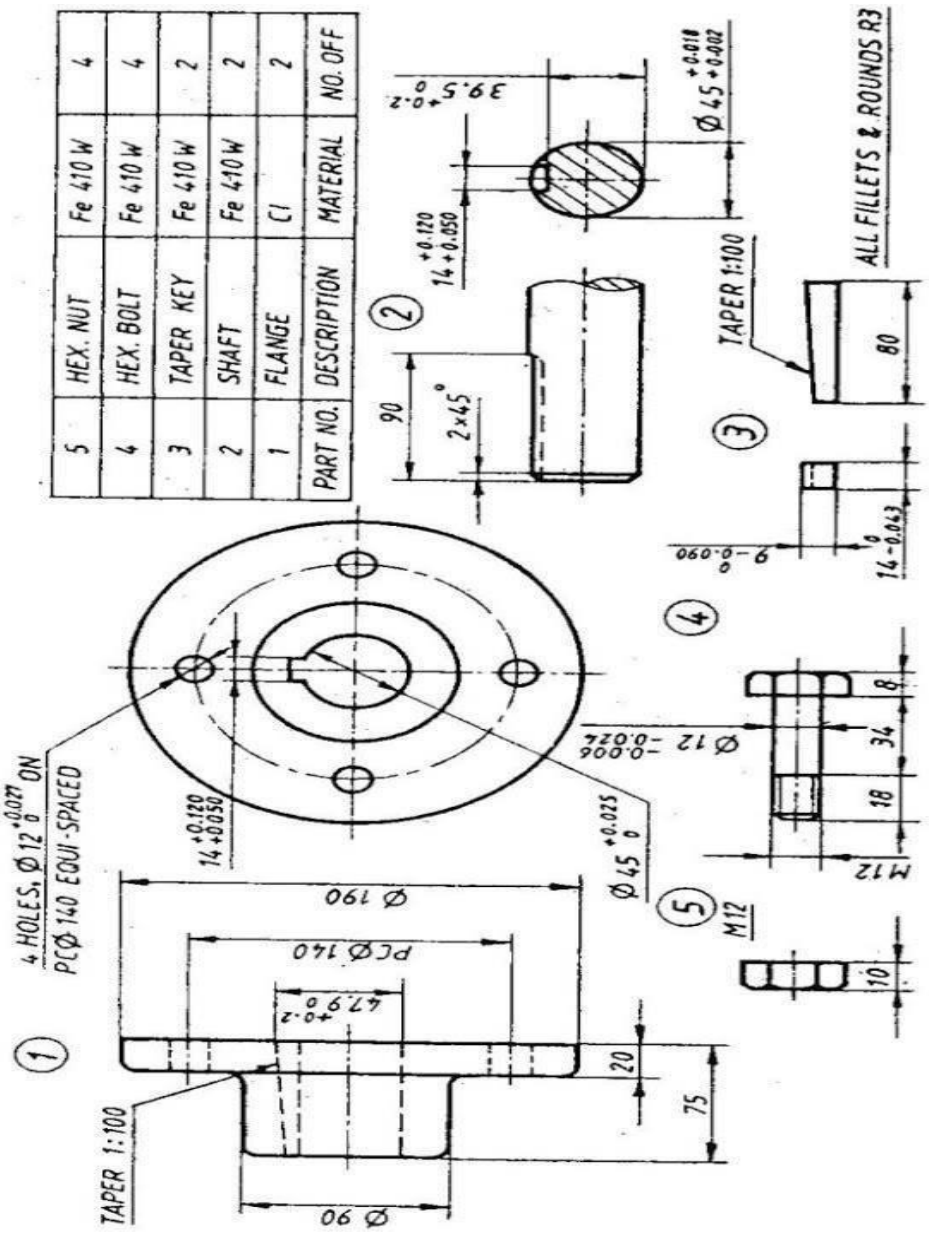
Within the CATIA tool set, we have various manners in which we can create surfaces. CATIA's an powerful tool and it affords us all sorts of ways surfaces can be generated. I'm going to go under the Start pull down and this is where we can see all our workbenches. Here you can see Generative Shape Design, Freestyle, Quick Surface Reconstruction. These are here based off the licenses that I have. Generative Shape Design which is what we're going to be talking about today is a workbench for parametrically generating surfaces. This can create pretty much any shape that you want and for most engineers they're going to be using Generative Shape Design and the reason behind that is is because you can do mechanical surfaces, you can do some styling parametrically based as well & it has some very powerful analysis tools for surfaces which we will be talking about some. When we go into Freestyle, Freestyle is more of the NURBS modeling. You'll see…



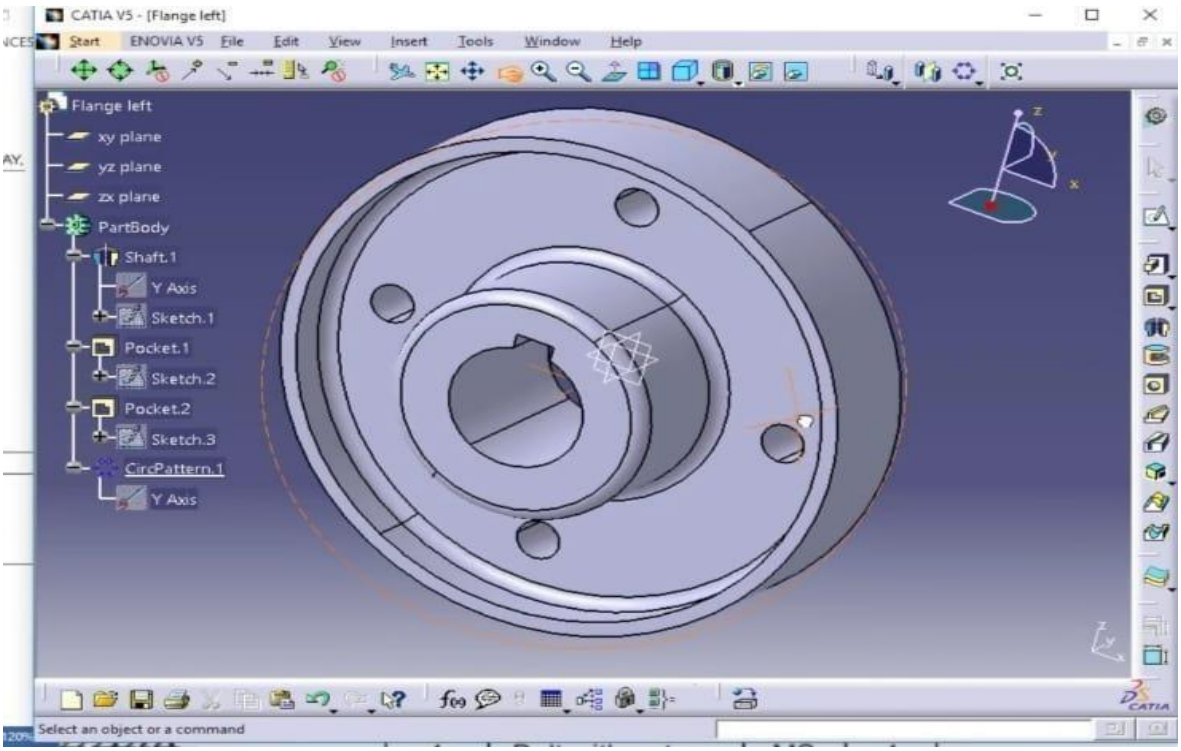
Generative Shape Design Workbench User Interface.



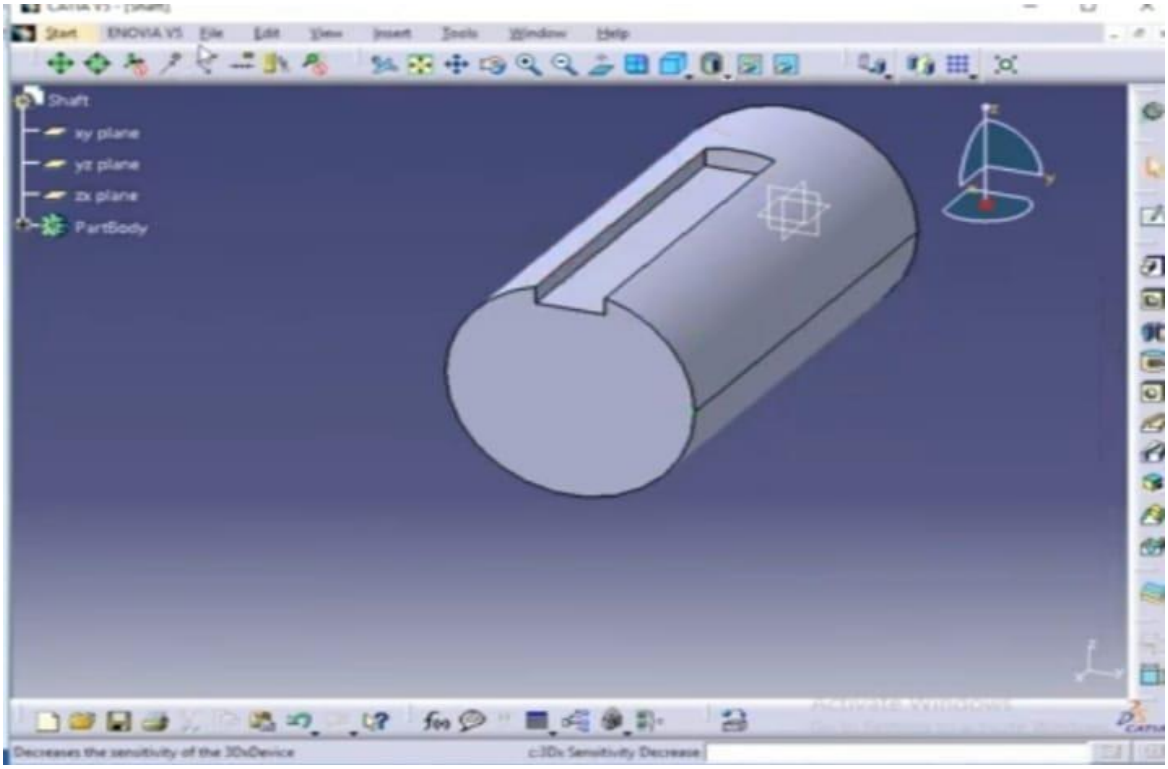
2D Projection



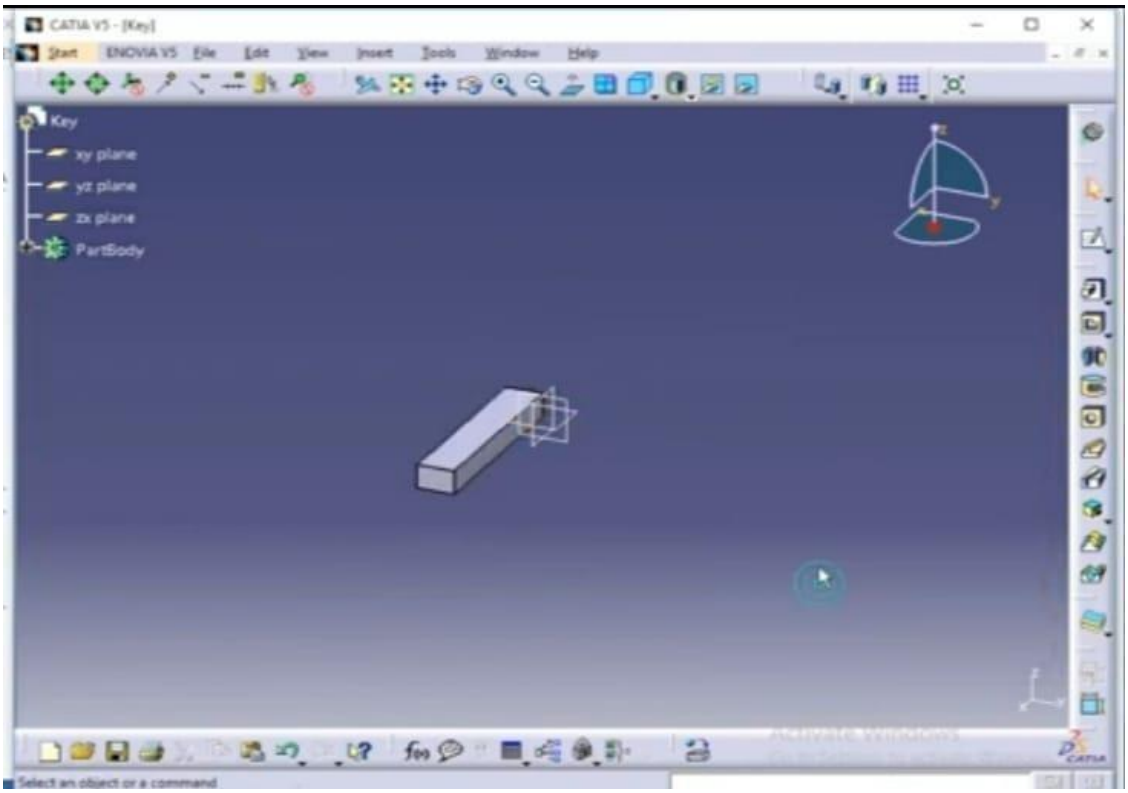
Part Design



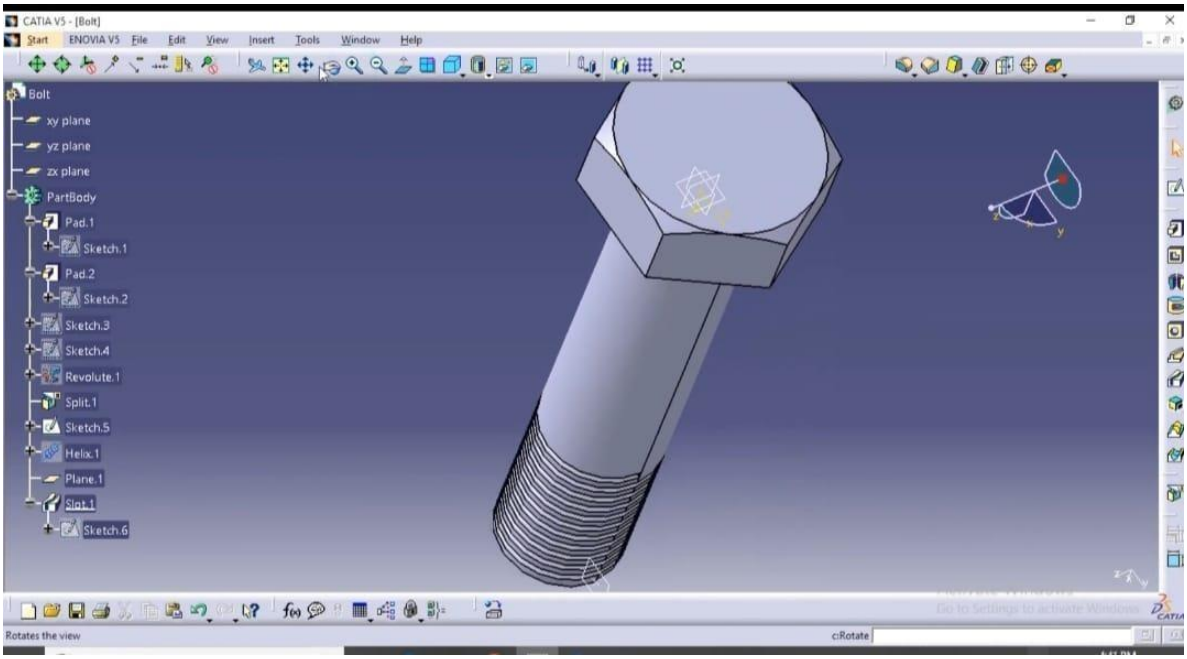
FLANGE



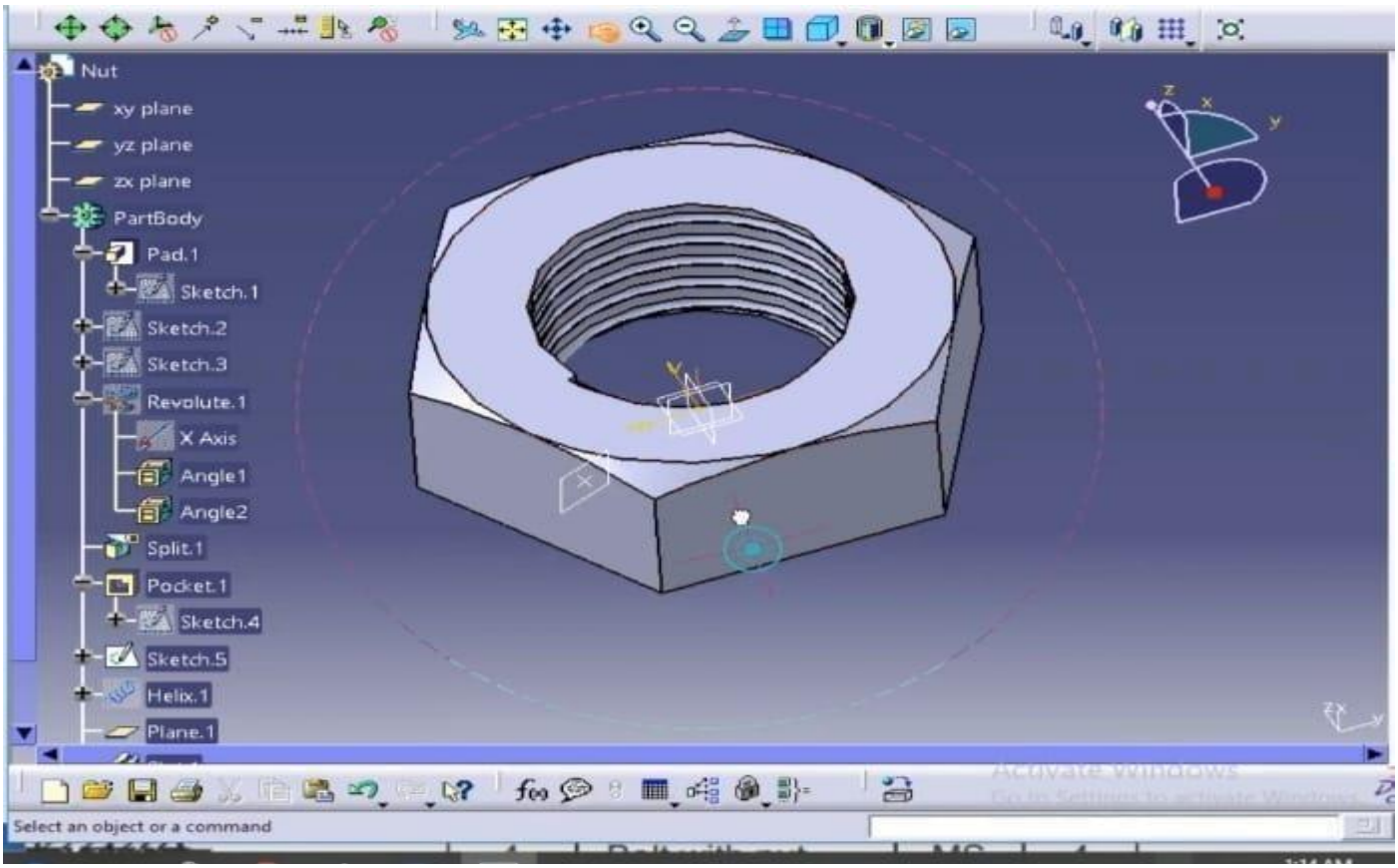
SHAFT



KEY

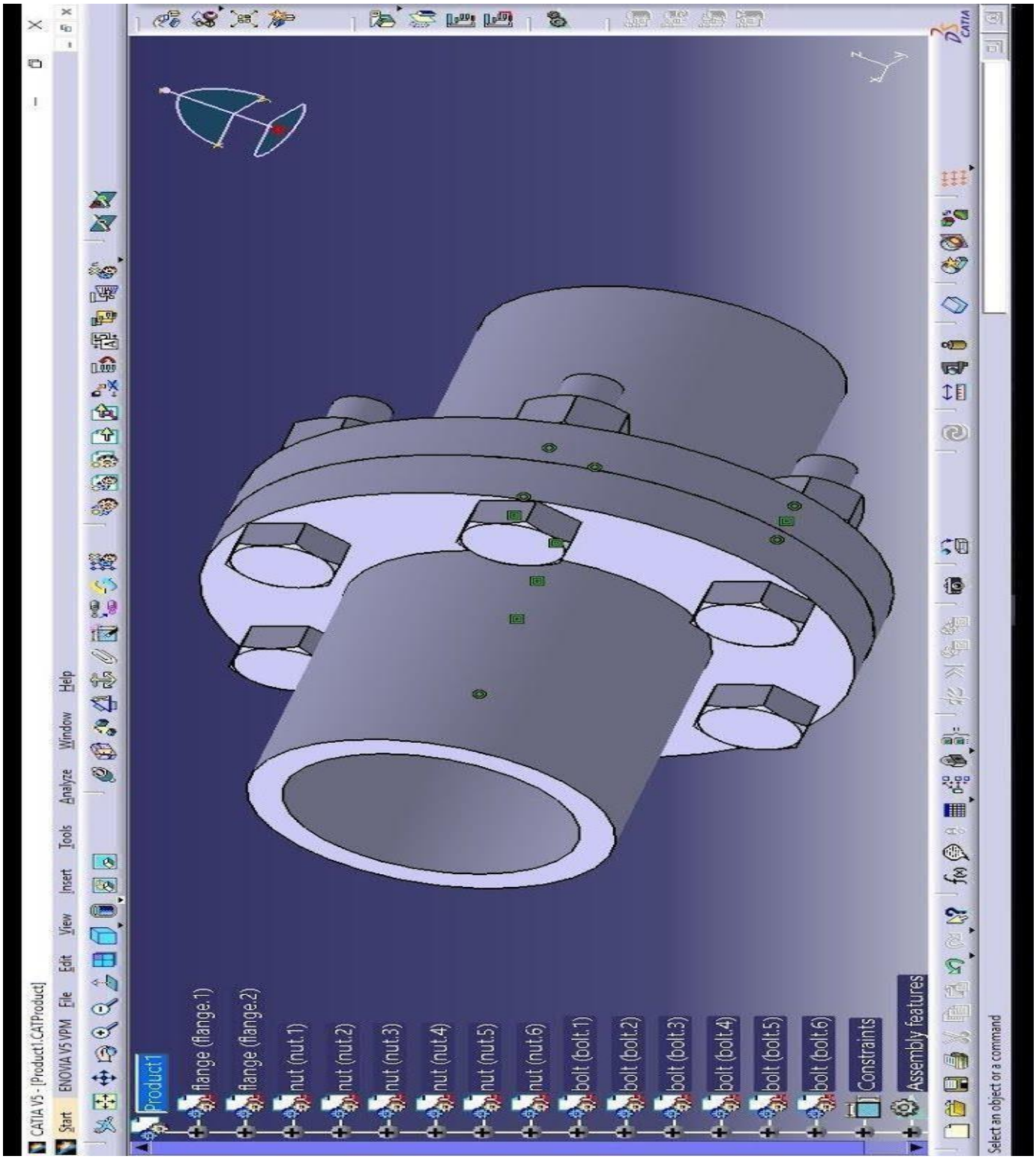


HEXAGONAL BOLT

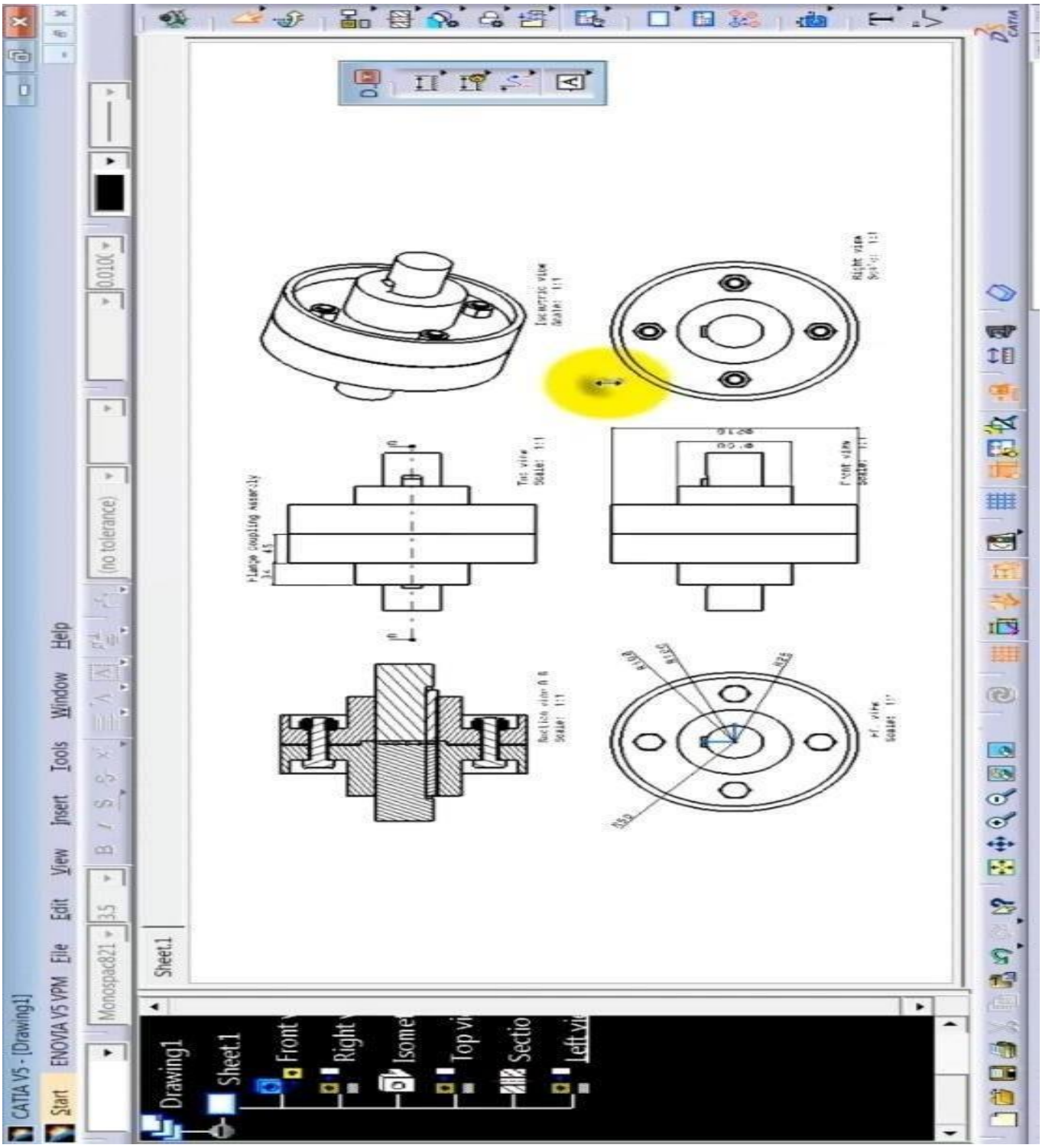


HEXAGONAL NUT

Assembly


CATIA V5 - [Product1.CATProduct]
Start
ENOVIA V5 VPM
File
Edit
View
Insert
Tools
Analyze
Window
Help
Product1
flange (flange.1)
flange (flange.2)
nut (nut.1)
nut (nut.2)
nut (nut.3)
nut (nut.4)
nut (nut.5)
nut (nut.6)
bolt (bolt.1)
bolt (bolt.2)
bolt (bolt.3)
bolt (bolt.4)
bolt (bolt.5)
bolt (bolt.6)
Constraints
Assembly features
Select an object or a command

DRAFTING


CATIA V5 - [Drawing1]
Start
ENOVIA V5 VPM File Edit View Insert Tools Window Help
Drawing1
Sheet.1
Sheet.1

CONCLUSION

In conclusion, **CATIA's power is in dealing with large equipment and assemblies, and its interface helps create complex parametric models**. Its surface modeling's tools are an add on to its features.