INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

<table>
<tr><th colspan="10">COMPUTER AIDED AIRCRAFT PRODUCTION DRAWING LABORATORY</th></tr>
<tr><td colspan="10">V Semester: AE</td></tr>
<tr><td>Course Code</td><td>Category</td><td colspan="3">Hours / Week</td><td>Credits</td><td colspan="3">Maximum Marks</td></tr>
<tr><td rowspan="2">AAEC21</td><td rowspan="2">Core</td><td>L</td><td>T</td><td>P</td><td>C</td><td>CIA</td><td>SEE</td><td>Total</td></tr>
<tr><td>0</td><td>0</td><td>3</td><td>1.5</td><td>30</td><td>70</td><td>100</td></tr>
<tr><td>Contact Classes: Nil</td><td>Tutorial Classes: Nil</td><td colspan="3">Practical Classes: 45</td><td colspan="4">Total Classes: 45</td></tr>
<tr><td colspan="9">Prerequisite: Laboratory</td></tr>
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I. COURSE OVERVIEW:

This course will also provide the Computer aided design laboratory provides a strong foundations of computer aided designing tool and students will learn the implementation of solid modelling using CATIA. It enables students to master the fundamentals of advanced modelling techniques, sketcher tools, base features, drafting, sheet metal and surface design workbenches. This course focuses on giving the foundations of engineering design and making it very useful for getting the student ready for product manufacturing industry.

II. COURSES OBJECTIVES:

The students will try to learn

1. The principles of isometric and orthographic conversions to create CAD models using CATIA software.
2. To create profiles and subsequently generating three dimensional entities from the generated profiles.
3. The fundamentals of geometric are dimensioning and tolerances and representing those using designing software's.
4. The build various aircraft parts by selecting workbenches appropriate for designing those components.

III. COURSE OUTCOMES:

At the end of the course students should be able to:

CO1	Choose appropriate tools and profiles for developing the required sketch using the Sketcher workbench.
CO2	Make use of wireframe elements, surfaces, trim elements and power copies for constructing the complex surfaces.
CO3	Utilize different geometric and dimensioning symbols and industry standards for the preparation of technical mechanical drawings.
CO4	Select appropriate tools available in assembly workbench for creating three-dimensional assemblies incorporating multiple solid models.
CO5	Build components using sketch Based features, perform sheet metal operations and correctly organize the tree for having maximum compatibility for editing or modifying the model.
CO6	Develop a model from drawing provided and draw conclusions for designing various aircraft components by utilizing different workbenches.

IV. COURE SYLLABUS:

EXERCISES FOR COMPUTATIONAL AIDED AIRCRAFT PRODUCTION DRAWING LABORATORY

Note: Students are encouraged to bring their own laptops for laboratory practice sessions.

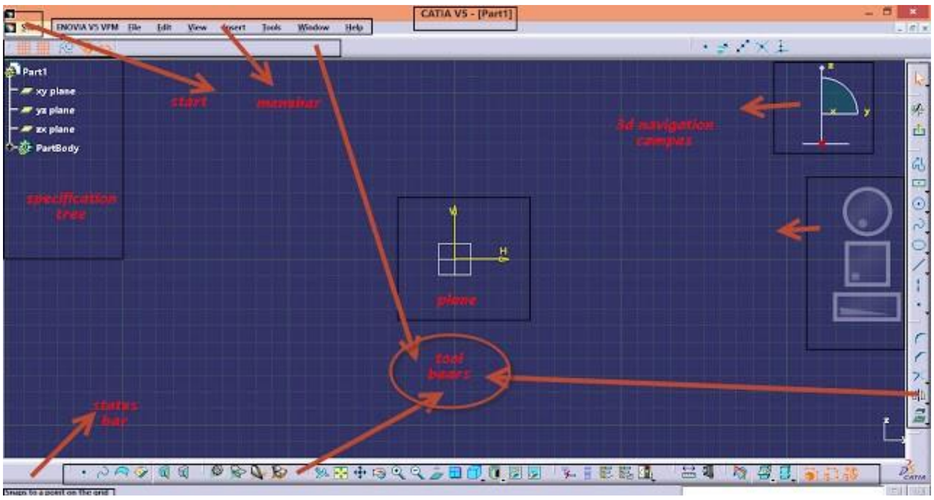
INTRODUCTION TO CATIA V5

CATIA (an acronym of **computer aided three-dimensional interactive application)** developed by the French company Dassault Systems

CATIA is a powerful tool for designing complex parts and assemblies. It allows mechanical engineers to create 3D models and generate 2D drawings with accuracy and precision. It provides tools for sketching, modelling, and draf2ting that enable the design of components with intricate details.

- Installation of CATIA V5
- Understand the CATIA interface
- Interface and Tools
- Precision and Accuracy
- 2D and 3D Modelling
- Collaboration and Sharing
- Customization
- Industry Usage
- Versions and Licensing

CATIA INTERFACE (GUI)



View Toolbar

- Display Commands
  - Fly Through
  - Fit View
  - Layer control
  - Pan
  - Rotate
  - Zoom
  - Normal View
  - Standard Views
  - View Types: Shaded/ Hidden Line/ Wireframe/ User Defined
- Hide/ Show
  - Hide
  - Swap Visible Space
- Properties
  - Display Characteristics for an object are set by selecting the entity, then pressing the right mouse button and selecting **Properties** from the menu

Other Commonly used tools

- Copy/ Paste

  - Geometry entities can be copied and pasted from one part to another.

  - Paste Special allows you to:
    - Paste a complete copy with history
    - Paste a linked copy
    - Paste the result without linking
- Undo/ Redo

  - Allows you to undo previous actions

  - Redo repeats an action that has been undone
- Hide/ Show

  - Allows you to temporarily hide entities from the display

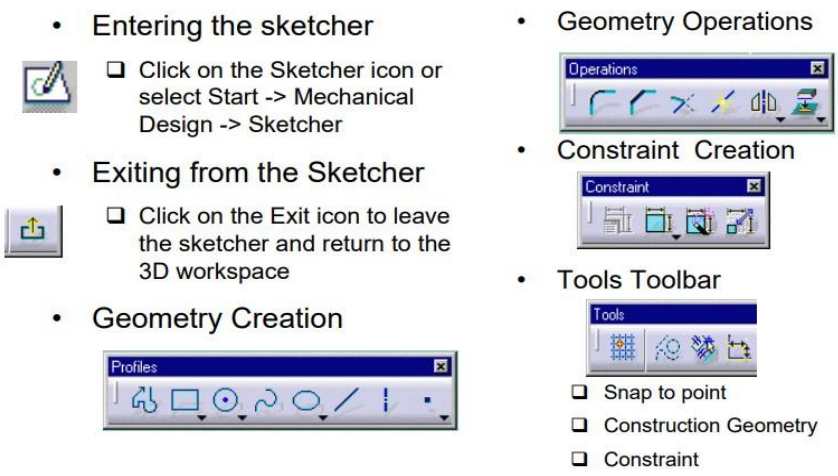
  - Hidden entities can be recovered by clicking on the “Swap visible space” icon, and then selecting the entity to make visible
- Update

  - Used to update the part after modification

Week 1: Getting Started with 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.

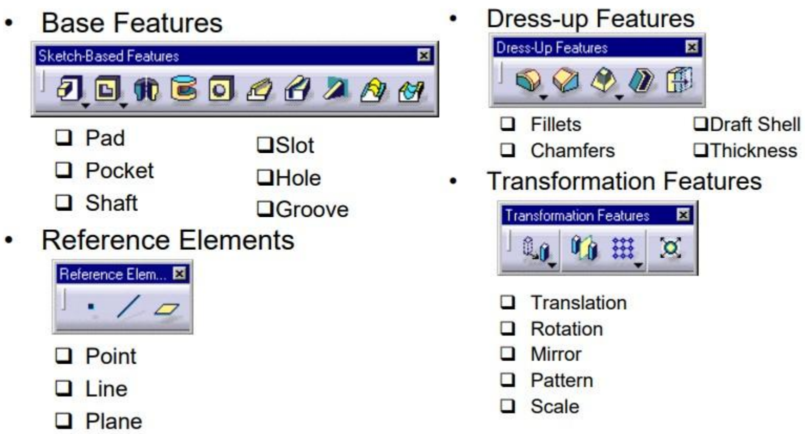
The Sketcher workbench contains the following standard workbench specific toolbars.



- **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, etc…) or geometrical (tangent, parallel, etc…) constraints using the commands located in the Constraint toolbar.

Week 2: Part Design

Part design is used to create 3D models from the basic 2D sketches created in sketcher environment.



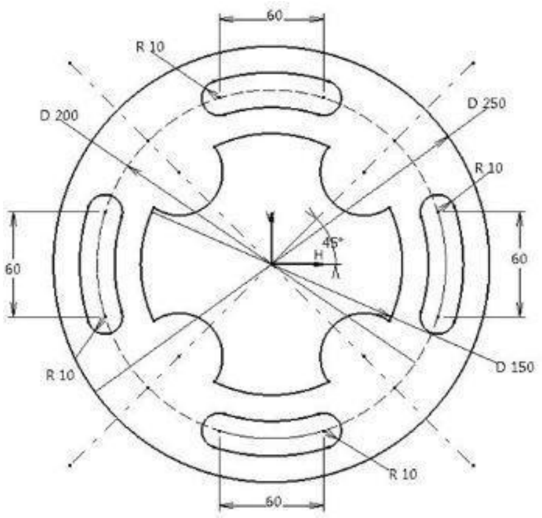
Some of the commands in workbench explained below:

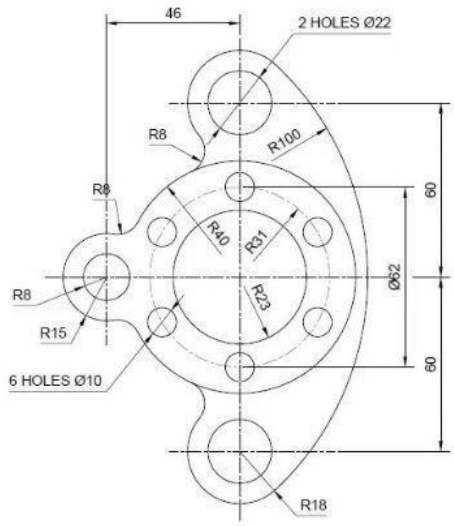
- **PAD command:** This command adds material in the third direction, a direction other than the sketch.
- **POCKET command:** The POCKET command somehow the opposite of PAD command. It simply helps remove geometry belonging to an already create part.
- **SHAFT command:** SHAFT command is mostly used to make shaft like parts. It requires an axis, around which the sketch will be revolved.

- **RIB command**: This command adds material along a guide curve. RIB is used to make components like springs, pipes etc.
- **SLOT command:** SLOT removes the material along a guide curve. This ensures that the cross section will be uniform throughout.

Exercises on Part Design

Part Design -1: Sketched based features, exit workbench, Dress up features

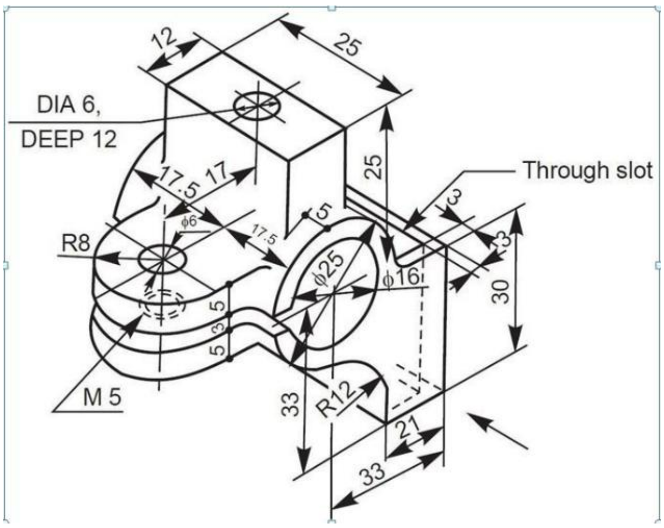




Try:

1. Create the "Swivel. CAT Part" Using Multiple Sketches

Part Design -2:



Try:

1. Modify the dimensions and design the sketch using CATIA

Week 3: Boolean Operations

Boolean operation is an important feature in CATIA V5.

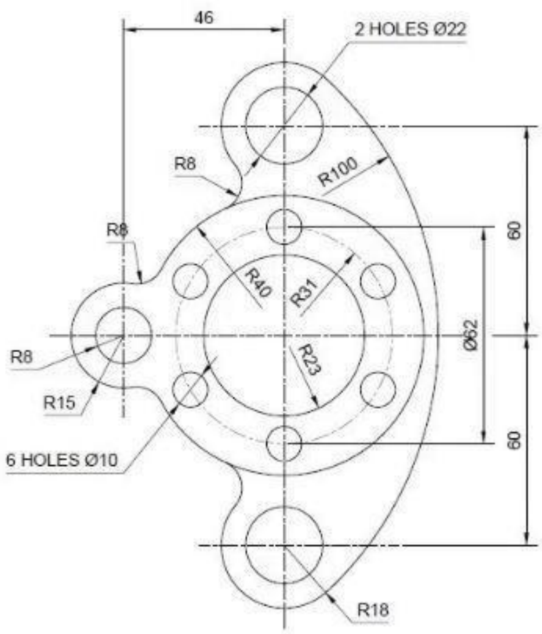
Types of Boolean operations include Assemble, Add, Remove, Intersect, Union Trim, and Remove Lump.

- **Assemble**: The Assemble command basically works considering the polarity of the solid bodies.
- **Add**: Add operation removes only the intersected portion between two bodies so that both parts act as a single body irrespective of the polarity of the bodies. Here, the bodies may be with positive polarity or negative polarity or a combination of both.
- **Remove**: Remove operation removes the selected body first and then is merged with the second body irrespective of the polarity of the bodies.
- **Intersect**:To derive a single body from two different bodies, Intersect Boolean operation is used. Basically, the intersected portion is the output which is displayed as a single body. There is no effect of polar bodies here.
- **Union Trim**: Unwanted material can be removed from any two bodies by using Union Trim operation which results in a single desired body.
- **Remove Lump**: Remove Lump Boolean operation removes the lump inside the body.

During the operation, a tab will give the option to the user to keep/remove any face as required. Using this Boolean operation, a user can remove N number of lump face areas.

Exercises on Boolean Operations

Boolean Operations:



Try:

1.Creating the Bottom U-Joint using Boolean Geometry.

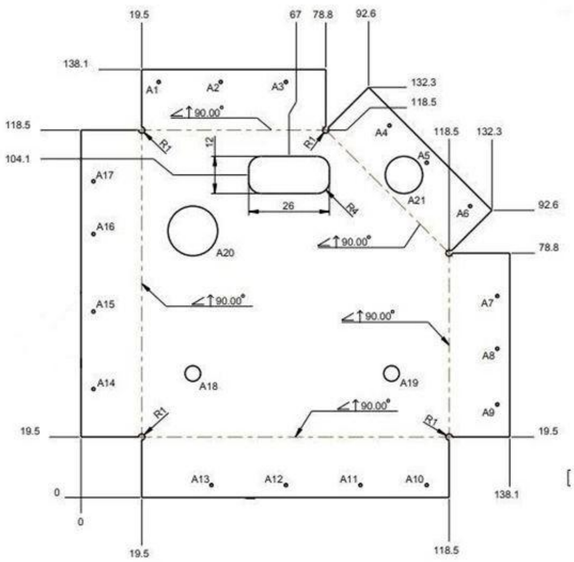
Week 4: Sheet Metal Design

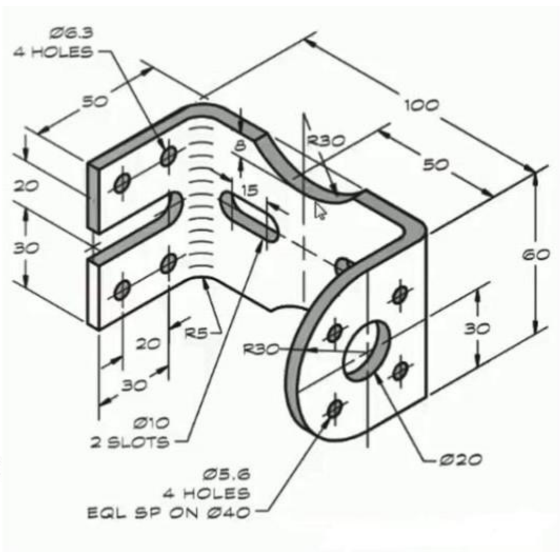
Sheet metal design is used for modifying sheet metal parts created, not to be used to create new sheet metal parts.

- **Walls on Edge:** This command creates a wall from edges of an existing part.
- **Cut-outs:** Additional possibilities are now available when creating a cutout: We can choose a direction for the cutout that is different from, or equal to, the normal direction.
- **Stamping:** Stamping also called pressing involves placing flat sheet metal, in either coil or blank form, into a stamping press. In the press, a tool and die surface form the metal into the desired shape.
- **Bending**: This command bends based on a line or by giving an Angle
- **Rolled Walls**: This command shows how to create rolled walls (such as pipes etc).

Exercises on Sheet Metal Design

Model a Metal Bracket:





Try:

1. Change the number of holes, location and find the maximum stress concentration, strain and displacement.

2. Fix the left side of plate and apply compressive loads on other three sides and find the maximum stress concentration, strain and displacement.

Week 5: Surface Design

Surface features introduce additional flexibility to a model. Surfaces can be used to define a complete part or to integrate a complex shape into the solid part (in the Part Design workbench).

- **Surfaces:** The product and industrial designers these days are giving importance to product styling and providing a unique shape to components. Generally, this is done to make sure that the product looks attractive and presentable to the customer. The shape of products are managed using the surface modelling techniques. Surface models are three-dimensional models with no thickness and unlike solid models, they do not have mass properties. CATIA V5 provides several surface modelling tools to create complex three-dimensional surface models.
- **Operations:** 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.
- **Wireframe**: The wireframe construction elements aid in creating surfaces. The sketches drawn in sketcher workbench can also be used to create surfaces.
- **Replication**

Exercises on Surface Design

Try:

1. Use the aluminium as material and find the natural frequency for same beam.

2. Take a simply supported beam made of mild steel and find the natural frequency.

Week 6: Assembly

Assembly environment is used to provide mating to two or more-part models to from complete assembly

- Product Structure Tools

  - ❑ Insert New Component
  - ❑ Insert New Product
  - ❑ Insert New Part
  - ❑ Insert Existing Component
  - ❑ Replace Component
  - ❑ Reorder Tree
  - ❑ Generate Numbers
  - ❑ Load Components
  - ❑ Unload Components
  - ❑ Manage Representations
  - ❑ Multi-Instantiation

- Move Toolbar

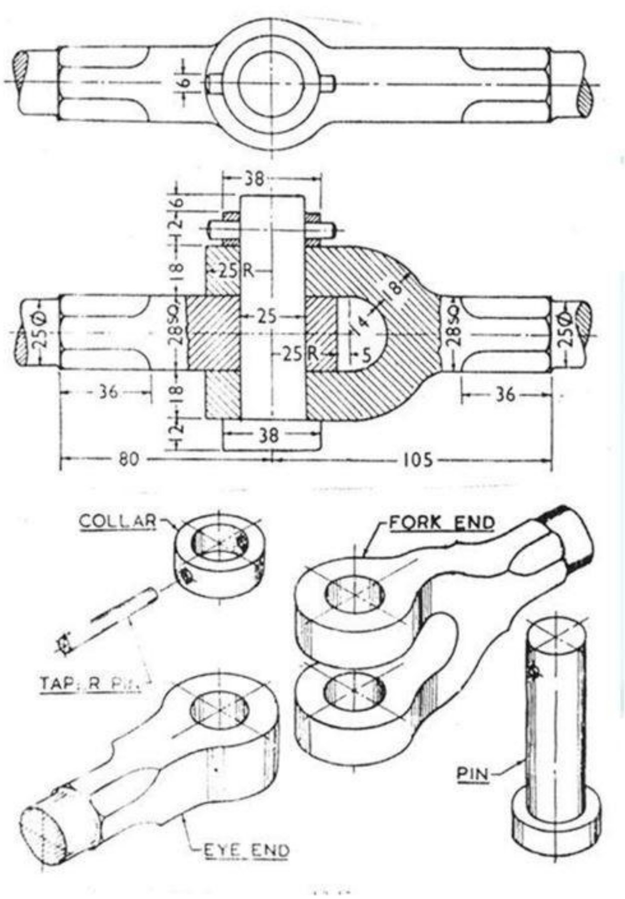
  - ❑ Manipulate
  - ❑ Snap
  - ❑ Explode and Assembly

- Constraints Toolbar

  - ❑ Coincidence
  - ❑ Contact
  - ❑ Offset
  - ❑ Angular
  - ❑ Anchor
  - ❑ Fix Together

Exercises on Assembly

Assemble a Knuckle Joint



Try:

1. Instead of temperature apply heat flux on the sides of square and find the temperature distribution.

2. Introduce a hole at the centre of square and find the temperature distribution

Week 7: GD & T

In this geometric dimensioning and tolerancing (GD&T), a unique set of GD&T symbols are used to define the relationships between part features and measurement references. Designers and engineers utilize this international language on their drawings to accurately describe part features on the basis of size, form, orientation and location.

Exercises on GD & T

GD & T (Cantilever Beam):

A cantilever beam with length of 550 cm and cross section area of 100 mm2 subjected to nonlinear load. Find the deflection and nonlinear behaviour.

Try:

1. Use a cantilever beam with same length as stated in above problem but with circular cross section and find its nonlinear behaviour.

2. Use the same cantilever beam subjected to torque and find its nonlinear behaviour

Week 8: Drafting

Drafting is a process of generating 2D machine drawing for the 3D part models to send it to the manufacturers.

Catia drafting is of two types

- Interactive Drafting
- Generative Drafting

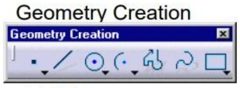
- Views Toolbar

  - ❑ Create a Front View (other views available underneath icon)
  - ❑ Create a section view
  - ❑ Create a detail view
  - ❑ Create a Clipping View
  - ❑ Create Views Via Wizard

- Automatic Dimension Creation

  - ❑ Auto-dimension
  - ❑ Semi-Automatic Dimensions

- Geometry Creation



  - Point
  - Line
  - Circle
  - Arc
  - Profile
  - Curve
  - Pre-Define Profiles
- Transformations Toolbar

  - Translate, Rotate, Scale, Mirror
- Relimitations Toolbar

  - Corner
  - Chamfer
  - Trim
  - Break
- Annotation

  - Text
  - Symbols
- Dimensions Toolbar

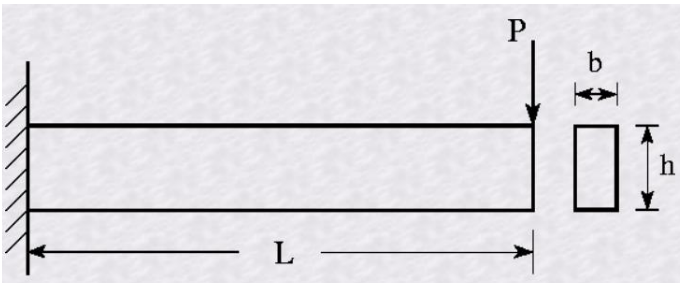
  - Create Dimension
  - Create Tolerance
- Dress up Toolbar

  - Centreline
  - Thread
  - Axis
  - Fill
  - Arrow

Exercises on GD & T

Drafting (Cantilever Beam):

A cantilever beam made of mild steel as following specifications L=150 cm, b=10cm, h=10cm shown in Figure below is subjected to a periodic force, which is mathematically represented below. The amplitude of the force is 1000 N. F = 1000k sin (pi*t/4). Find the deflection of beam.



Try

1. Repeat the above analysis for simply supported beam.

2. Change the magnitude of periodic load and repeat the same analysis.

Week 9 :Design of Aircraft Wing

Exercises on Design of Aircraft Wing

Design of Aircraft Wing

Dynamic Analysis (Model analysis of wing)

Perform model analysis of an aircraft rectangular wing of a span 100 cm and the chord length of wing is 10 cm and cross-sectional area is 100 cm2. Calculate the natural frequency of wing.

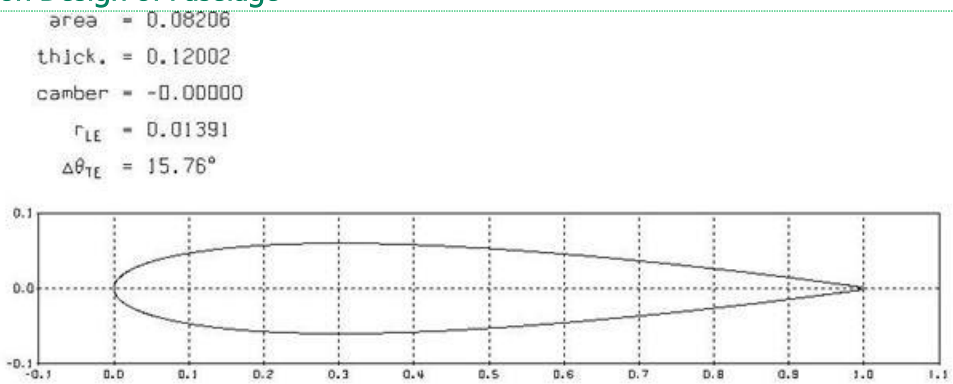
Try:

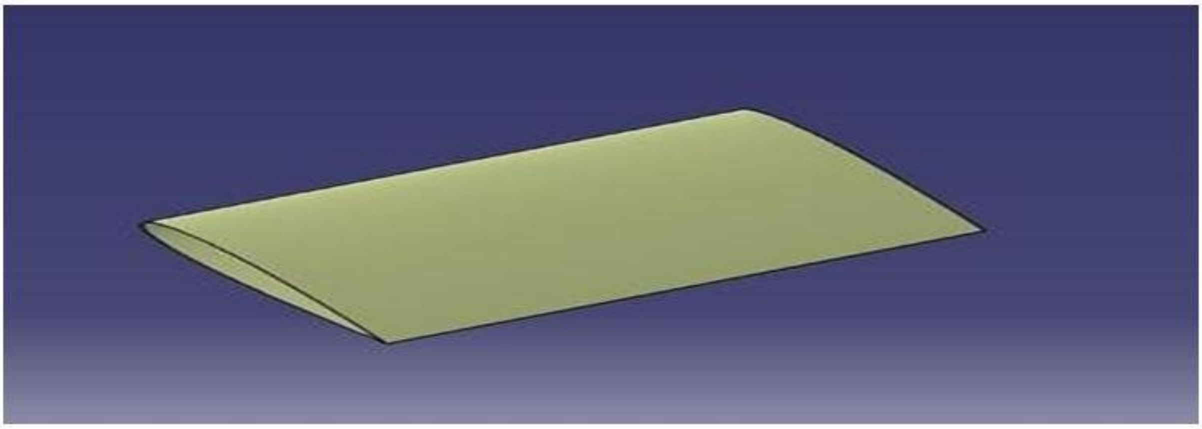
1. Repeat the above analysis for a tapered wing.

2. Repeat the above analysis for tapered wing with uniformly varying load

Week 10: Design of Fuselage

Exercises on Design of Fuselage





Calculate the deformation of the aluminium fuselage section under the application of internal load of 100000 Pa. The radius of fuselage is 0.15m and thickness is 2 mm.

Try

1. Repeat the above analysis for a monocoque fuselage.

2. Repeat the above analysis for a monocoque fuselage with both internal and external load.

Week 11: Dynamic Analysis (Model analysis of Fuselage)

Calculate the natural frequency of the aluminium fuselage section under the application of internal load of 100000 Pa. The radius of fuselage is 0.15m and thickness is 2 mm.

Try

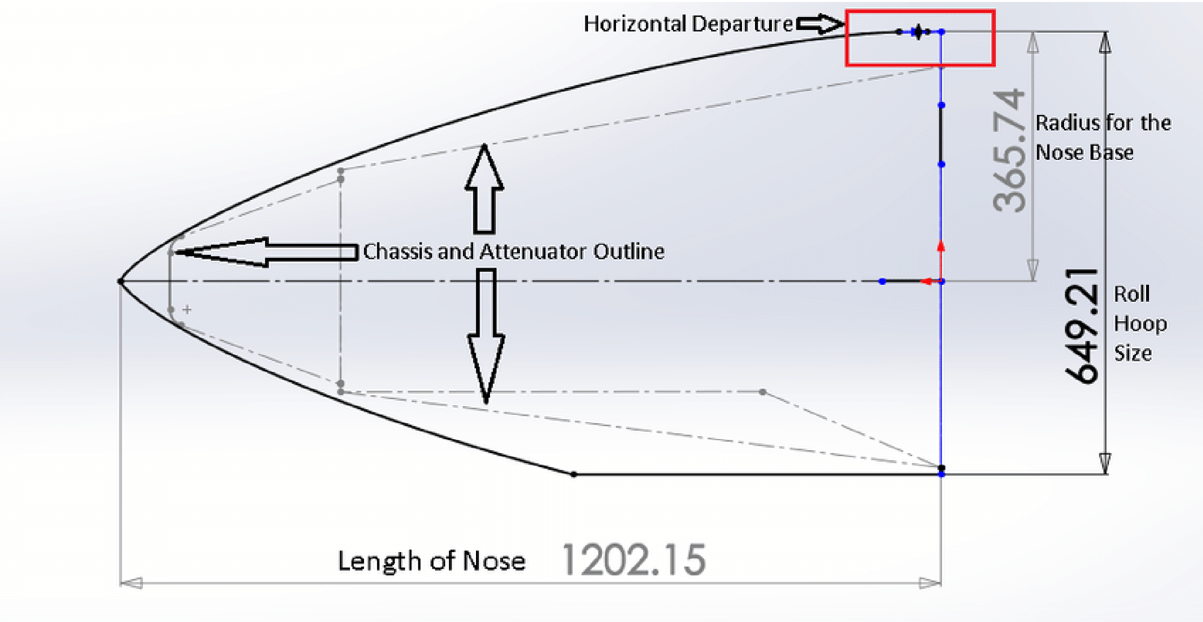
1. Repeat the above analysis for a monocoque fuselage.

2. Repeat the above analysis for a monocoque fuselage with both internal and external load.

Week 12: Design of Nose Cone

Exercises on Design of Nose Cone

A simple retractable landing gear subjected to a load of 10000 N. Find the deformation and stress developed in the landing gear



Try

1. Repeat the above analysis for a landing gear subjected to compression load.

Week 13: Dynamic Analysis (Model analysis of Landing gear)

Perform model analysis of A simple retractable landing gear subjected to a load of 10000 N. Calculate the natural frequency of wing.

Try:

1. Repeat the above analysis for a landing gear subjected to compression load.

2. Repeat the above analysis for tapered wing with uniformly varying load.

Exercises on Design of Landing Gear

Design of Landing Gear

Ansys Composite PrepPost (ACP) find the Interfacial delamination between fiber and resin, Rupture of fibers in a Carbon fiber reinforced polymer composite having 10 layers and subjected to tensile load of 100N.

Try:

1. Repeat the above analysis for Ceramic matrix composite.

2. Repeat the above analysis for Carbon fiber reinforced polymer composite subjected to shear.

V. TEXT BOOKS:

1. isbn9781477689028.indd (e-onsia.com) pdf
2. (PDF) CATIA V5 Workbook Releases 8 & 9 | Andrea Tomos - Academia.edu
3. CATIA V5-6R2021 for Designers, 19th Edition Sham Tickoo
4. CATIA V5 Surface Design with Applications Jaecheol Koh

VI. ELECTRONICS RESOURCES:

1. https://www.iare.ac.in/sites/default/files/lab1/IARE Computer Aided Aircraft Production Drawing laboratory.pdf
2. https://www.dbit.ac.in/ce/syllabus/computer-aided-aircraft-production-drawing-lab.pdf
3. https://edudravida.in/22301-2/

VII. MATERIALS ONLINE:

1. Course Description
2. Laboratory manual