HARMONIC ANALYSIS OF A 3 BLADE MARINE PROPELLER BY USING FEA

A Project Report Submitted

in partial fulfilment of the requirements for the award of the Degree of

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

by

S.GANGADHAR	315126520251
K.JAGADESH KUMAR	315126520280
G.MOHAN	315126520255
P.PRANEETH	315126520242
B.NAVEEN	315126520262

Under the guidance of

Sri. V.RAVITEJA BEHERA, M.E (Ph.D)

Asst. Professor

PRAGNANAM BRAHMA

ANITS

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES (A)

(Affiliated to A.U., approved by AICTE, Accredited by NBA and NAAC with 'A' grade)

SANGIVALASA, VISAKHAPATNAM-531162

2015 – 2019

HARMONIC ANALYSIS OF A 3 BLADE MARINE PROPELLER BY USING FEA

A Project Report Submitted

in partial fulfilment of the requirements for the award of the Degree of

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

by

S.GANGADHAR	315126520251
K.JAGADESH KUMAR	315126520280
G.MOHAN	315126520255
P.PRANEETH	315126520242
B.NAVEEN	315126520262

Under the guidance of

Sri. V.RAVITEJA BEHERA, M.E (Ph.D)

Asst. Professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES (A)

(Affiliated to A.U., approved by AICTE, Accredited by NBA and NAAC with 'A' grade)

SANGIVALASA, VISAKHAPATNAM-531162

2015 – 2019

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES (A)

(Affiliated to A.U., approved by AICTE, Accredited by NBA and NAAC with 'A' grade)

SANGIVALASA, VISAKHAPATNAM-531162

PRAGNANAM BRAHMA

ANITS

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the project entitled "**HARMONIC ANALYAIS OF A 3 BLADE MARINE PROPELLER BY USING FEA**" describes the bonafied work done by S.Gangadhar (315126520251), K.Jagadesh kumar (315126520280), G.Mohan (315126520255), P.Praneeth (315126520242), B.Naveen (315126520262) under the esteemed guidance of **Sri. V.RAVITEJA BEHERA,** in partial fulfilment of the requirements of Degree of B.Tech in Mechanical Engineering .

APPROVED BY

PROJECT GUIDE

13.4.19

Dr.B.NAGA RAJU

Head of the Department

V.RAVITEJA BEHERA

Asst. Professor

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa 531 162 VISAKHAPATNAM Dist. A.P.

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER: K. Srinivasan 13/4/19

EXTERNAL EXAMINER: Bharath 13/4/19

ACKNOWLEDGEMENT

We express immensely our deep sense of gratitude to **Sri V. RAVITEJA BEHERA,** Assistant Professor, Department of Mechanical Engineering,Anil Neerukonda Institute of Technology & Sciences, Sangivalasa, Bheemunipatnam Mandal, Visakhapatnam district for his valuable guidance and encouragement at every stage of work for the successful fulfilment of students.

We are also very thankful to **Prof. T.V.HANUMANTHA RAO** Dean (Academics), **Prof. T. SUBRAHMANYAM**, Principal and **Prof. B.NAGA RAJU,** Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for providing this opportunity.

We express our sincere thanks to the non-teaching staff of Mechanical Engineering Department for their kind co-operation and support to carry on work.

Lastly we are grateful to one and all who have contributed directly or indirectly in the completion of the project.

INDEX

List of contents **page no**

Abstract 1

Chapter 1

1.Introduction 3

1.1.Introduction to propeller 3

1.2.Propeller materials 5

1.3.Propeller construction 5

1.4.Types of propeller blades 5

1.4.a.Classification by number of blades attatched 5

1.5.Propeller applications 9

1.6.Cavitation and its effect 10

1.7.Introduction to ANSYS 10

1.8. Description about ANSYS 12

1.9. Specific Capabilities of ANSYS 13

1.10.Introduction to CATIA 15

Chapter 2

2. Literature review 19

Chapter 3

3. Design of propeller 21

3.1.Steps involved in design of propeller 22

Chapter 4

4.Introduction to Finite Element Analysis (FEA) 27

4.1..Basic types of element used in Finite Element Analysis (FEA) 28

4.2.Common element types used in ANSYS 29

4.3 Harmonic Analysis 30

4.4.Steps for analysis 31

4.5.Selection of element 32

4.6.Material properties 33

4.7.Mesh generation 36

Chapter 5

5.Boundary conditions and Loads 38

5.1.Solution stage 39

Chapter 6

6.Results and Discussions 41

6.1 Harmonic Analysis Of NAB Propeller 41

6.2.Harmonic analysis of composite Propeller 43

Chapter 7

7.Comparsion of results of NAB and GFRP 46

Chapter 8

Conclusions 48

List of figures

page no.

Fig.1 propeller nomenclature 4

Fig.2 3 blade propeller 6

Fig.3 4 blade propeller 7

Fig.4 5 blade propeller 8

Fig.5 6 blade propeller 8

Fig.6 Screw propeller 9

Fig.7 Reverse pitch propeller 9

Fig.8 ANSYS environment 11

Fig.9 Structrural analysis 13

Fig.10 Thermal analysis 14

Fig.11 CATIA environment 15

Fig.12 Design parameters 21

Fig.13 Initial stage of design 22

Fig.14 Secon stage of design 23

Fig.15 Design of single blade 23

Fig.16 Final stage of design 24

Fig.17 Final stage of design with hub 25

Fig.18 Importing model from workbench to APDL 31

Fig.19 Element selection 32

Fig.20 Steps to generate me 36

Fig.21 Mesh generation 36

Fig.22 Boundary conditions and loads 38

Fig.23 Solution stage 39

Fig.24 X-component of displacement vs frequency response graph of NAB 41

Fig.25 Y-component of displacement vs frequency response graph of NAB 42

Fig.26 Z-component of displacement vs frequency response graph of NAB 42

Fig.27 X-component of displacement vs frequency response graph of GFRP 43

Fig.28 Y-component of displacement vs frequency response graph of GFRP 44

Fig.29 Z-component of displacement vs frequency response graph of GFRP 44

List of Tables

Page no.

Table.1 Nickel Aluminium Bronze Properties 33

Table.2 Aramid fibre properties 34

Table.3 Glass Fibre Reinforced Plastic 35

Table.4 Comparison between NAB and GFRP 46

Abstract

Current work intends on harmonic analysis to design a propeller of underwater vehicle with a composite material and to analyze its displacements and natural frequencies using ANSYS software. Harmonic Analysis is performed to evaluate the suitability of composite material for underwater vehicle propeller over NAB (Ni-Al-Bronze alloy) propeller. A propeller is a complex geometry which requires high end modelling software. The solid model of propeller is developed in CATIAV5. A solid mesh is generated for the model. The FE model is then given the necessary boundary conditions and loads. Static analysis and harmonic analysis are carried on both NAB and GFRP propeller (Glass Fibre Reinforced Plastic) in ANSYS software. A comparison analysis is done on metallic and composite propeller and the response graphs for the displacements and frequencies were plotted and analysed.

CHAPTER - 1

1.Introduction

Ships and UWV's as submarines, torpedoes and submersibles etc., uses propeller as propulsion. The Propeller blade geometry and its design is more complex involving many controlling parameters. The strength analysis of such complex 3D blades with conventional formulas will give less accurate values. In such cases numerical analysis (Finite Element Analysis) gives comparable results with experimental values. The conventional UWV Propellers are made up of Nickel Aluminium Bronze (NAB) present the work aims to replace the propeller blade material from Nickel Aluminium Bronze (NAB) metal to a fiber reinforced composite material (FRP). This complex analysis can be solved easily by finite element method techniques. The Harmonic analysis is done for the three bladed solid NAB as well as Composite propeller. The Harmonic analysis includes the evaluation of Displacement and frequency analysis for the propeller blades. The goal of this work is to design and evaluate the performance of the composite Propeller with that of the NAB propeller.

1.1. Introduction to Propeller

A propeller is a rotating fan like structure which is used to propel the ship by using the power generated and transmitted by the main engine of the ship. The transmitted power is converted from rotational motion to generate a thrust which imparts momentum to the water, resulting in a force that acts on the ship and pushes it forward.

Propeller nomenclature:

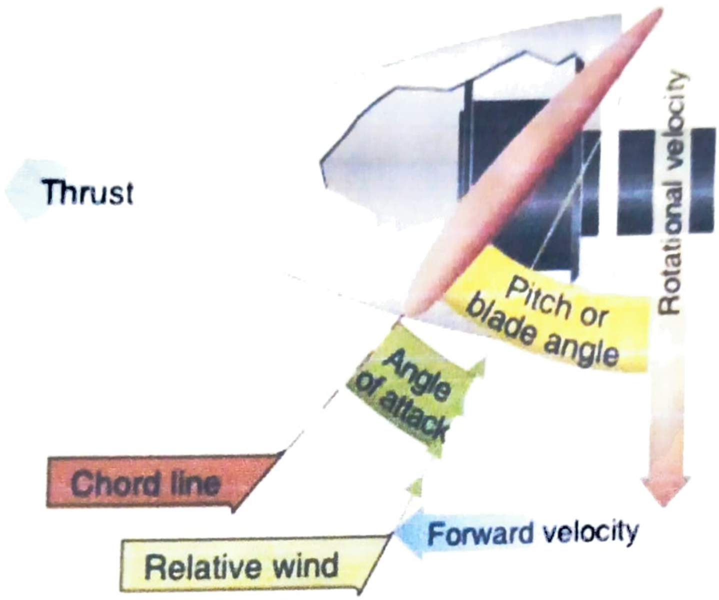

Fig.1 NOMENCLATURE

A ship propels on the basis of Bernoulli's principle and Newton's third law. A pressure difference is created on the forward and aft side of the blade and water is accelerated behind the blades.

The thrust from the propeller is transmitted to move the ship through a transmission system which consists of a rotational motion generated by the main engine crank shaft, intermediate shaft and its bearings, stern tube shaft and its bearing and finally by the propeller itself.

A ship can be fitted with one, two and rarely three propellers depending upon the speed and manoeuvrings requirements of the vessel.

1.2. Propeller Materials

Marine propellers are made from corrosion resistant materials as they are made operational directly in sea water which is a corrosion accelerator. The materials used for making marine propeller are alloy of aluminum and stainless steel. Other popular materials used are alloys of nickel, aluminum and bronze which are 10~15 % lighter than other materials and have higher strength.

In recent days composites , especially glass fibre reinforced plastic and carbon fibre reinforced plastic play a vital role in manufacturing of marine propellers due to their key advantages of corrosion resistance , chemical resistance , high performance at elevated temperatures, durability etc.

1.3. Propeller construction

The construction process of the propeller includes attaching a number of blades to the hub or boss by welding or forging in one piece. Forged blades are highly reliable and have greater strength but are expensive as compared to welded ones. A marine propeller is constructed by sections of helicoidal surfaces acting together to rotate through water with a screw effect , sideways thrust which results from the propeller rotation and affects the steering of a ship. It is most noticeable when close to the quay or in a narrow channel.

1.4. Types of propeller blades

Propellers are be classified on the basis of several factors

1.4.a.Classification by number of blades attatched

Propeller blades may vary from 3 blade propeller to 4 blade propeller and sometimes even 5 blade propeller. However, the most commonly used are 3 blades and 4 blade propellers.

The propeller efficiency will be highest for propeller with minimum number of blades i.e. 2 blade propeller. But to achieve strength factor and considering the heavy loads subjected by the ship, sea and weather two blade propellers are not used for merchant ships.

3 bladed propeller

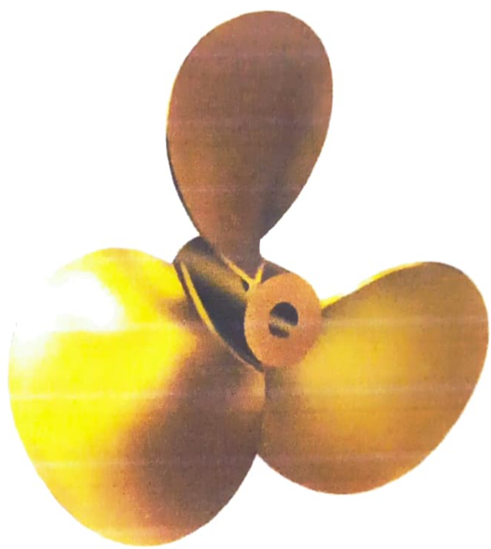

Fig.2 3 Bladed propeller

3 blade propeller has following characteristics:

- The manufacturing cost is lower than other types.
- Are normally made up of aluminium alloy.
- Gives a good high speed performance.
- The acceleration is better than other types.
- Low speed handling is not much efficient.

4 bladed propeller

A 4 bladed propeller has following characteristics:

Fig.3. 4 Bladed propeller

- The manufacturing cost is higher than 3 blade propellers.
- 4 blade propellers are normally made up of stainless steel alloys.
- Have better strength and durability.
- Gives a good low speed handling and performance.
- Has a better holding power in rough seas.
- 4 blade propeller provides a better fuel economy than all the other types.

5 bladed propeller

A 5 blade propeller has following characteristics:

- Manufacturing cost is higher of all.
- Vibration is minimal from all the other types.
- 5 blade propellers have better holding power in rough seas.

Fig .4. 5 Bladed propeller

6 bladed propeller

- Manufacturing cost is high
- Vibration is minimal from all the other types.
- 6 blade propellers have better holding power in rough seas.
- With six blade propeller, the induced pressure field over the propeller decreases

Fig.5, 6 Bladed propeller

1.5.Propeller applications

Shrouded propellers (Screws)

- Marine industry applications

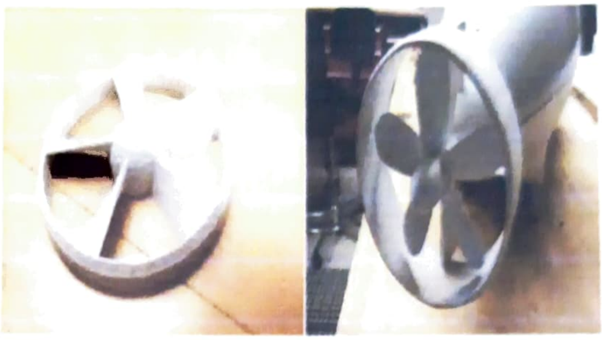

Fig 6. Screw propeller

Reverse pitch propeller

- Used in aerodynamic breaking

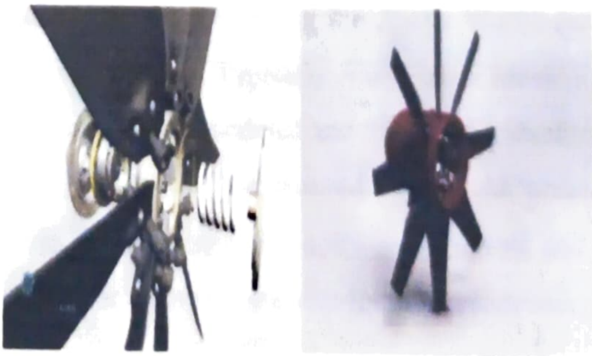

Fig.7 Reverse pitch propeller

1.6.Cavitation and its effect in Propellers:

Cavitation is the formation of vapor bubbles in water near a moving propeller blade in regions of low pressure due to Bernoulli's principle. It can occur if an attempt is made to transmit too much power through the screw, or if the propeller is operating at a very high speed. Cavitation can waste power, create vibration and wear, and cause damage to the propeller. It can occur in many ways on a propeller. The two most common types of propeller cavitation are suction side surface cavitation and tip vortex cavitation.

In General, Cavitation prevails more in GFRPs as the thickness is relatively lower in GFRPs. In order to avoid this cavitation effect in GFRPs, aramid fibres are added as layers . By adding the aramid fibres ,the thickness is increased and thereby the cavitation can be controlled. The aramid fibres are selected as they are resistant enough towards various chemical and corrosive environments.

1.7.Introduction to ANSYS

ANSYS Software:

Ansys develops and markets finite element analysis software used to simulate engineering problems. The software creates simulated computer models of structures, electronics, or machine components to simulate strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. Ansys is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, Ansys software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety.

Most Ansys simulations are performed using the Ansys Workbench software, which is one of the company's main products. Typically Ansys users break down larger structures into small components that are each modeled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the Ansys software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time.

Ansys also develops software for data management and backup, academic research and teaching. Ansys software is sold on an annual subscription basis.

ANSYS Environment

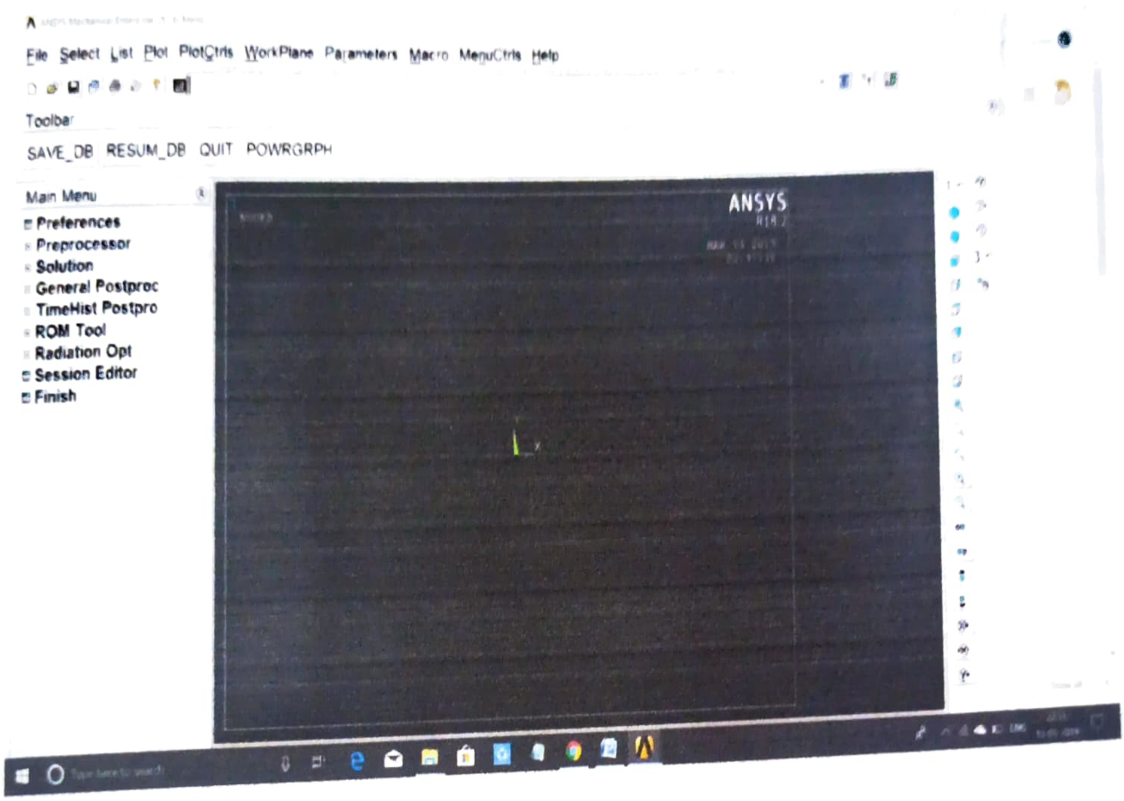

Fig.8 ANSYS Environment

Introduction to ANSYS Mechanical APDL:

This course is recommended for anyone who wishes to perform Finite Element Analysis (FEA) of mechanical parts and has little or no ANSYS Mechanical APDL (MAPDL) software experience. The ANSYS Mechanical APDL workflow, graphical user interface, along with the APDL command syntax, will be introduced to users. The focus will be on linear static structural analyses, although the concepts presented will provide a strong foundation to allow the user to solve other physics, such as thermal analyses, or more complicated analysis procedures with ANSYS Mechanical APDL. After completing the course, attendees should be able to use the ANSYS Mechanical APDL software efficiently to set up, solve, and post process linear static analyses.

1.8.Description about ANSYS

ANSYS is commercial Finite Element Analysis (FEA) software package. The software implements equations that govern the behavior of these elements and solves them all. It creates a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to solve analytically .Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc. ANSYS has the capability to solve complex engineering problems by mathematically simulating the exact behavior of the structures.

ANSYS, Inc. Family of Products include the following

- ANSYS – Advanced nonlinear mechanical and multiphysics FEA solution capabilities
- ANSYS Workbench – Complete environment for geometry modeling, mesh manipulation, structural/thermal analysis, and optimization,
  Which is tightly integrated with CAD packages
- CFX – State-of-the-art CFD solvers, including the coupled, parallel *CFX-5* solver
- ICEM CFD – Powerful meshing tools with general pre- and post-processing features, including ICEM CFD for generating complex CFD grids and AI Environment creating with sophisticated structural FEA meshes

ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering. Partial listing of it`s capabilities:

1) Structural

- Linear
- Nonlinear

  – Material, Geometric, Contact

2) Dynamics

- Spectrum Modal ,Harmonic ,Transient Dynamic ,, Random Vibration

- Explicit Dynamics with ANSYS LS-DYNA

3) Thermal

- Steady State and Transient

4) Fluid (CFD, Acoustics, and other fluid analyses)
5) Low and High Frequency Electromagnetics
6) Coupled field analysis

A partial list of industries in which ANSYS is used:

– Aerospace

– Automotive

– Biomedical

– Bridges & Buildings

– Electronics & Appliances

– Heavy Equipment & Machinery

– MEMS - Micro Electromechanical Systems

1.9.Specific Capabilities of ANSYS:

Structural

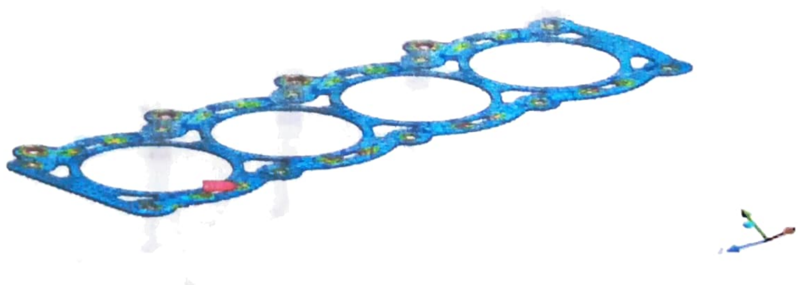

Figure 09: Structural Analysis

Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

Static Analysis:

Static analysis is performed to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses.

Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Transient Dynamic Analysis:

Transient Dynamic analysis is used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

Buckling Analysis:

Buckling Analysis is used to calculate the buckling loads and determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analyses are possible.

In addition to the above analysis types, several special-purpose features are available such as Fracture mechanics, Composite material analysis and Fatigue analysis.

Thermal

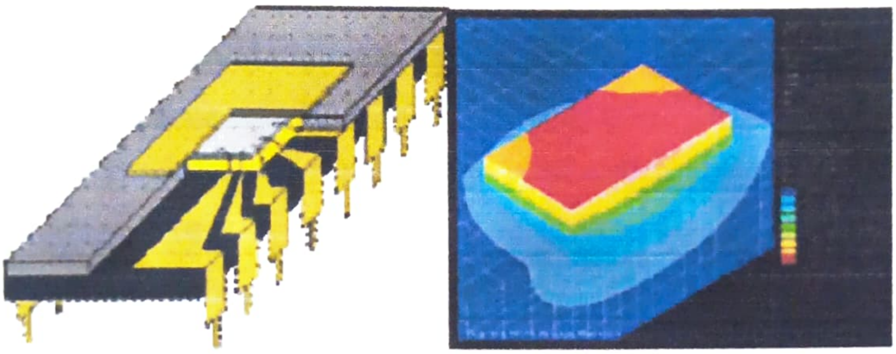

Figure 10: Thermal Analysis

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions. Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time.

Such loads include the following:

- Convection
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material vary with temperature. This temperature dependency being appreciable, the analysis becomes nonlinear. Radiation boundary conditions also make the analysis nonlinear. Transient calculations are time dependent and ANSYS can both solve distributions as well as create video for time incremental displays of models.

1.10. Introduction to CATIA

CATIA(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 French company Dassault Systems.

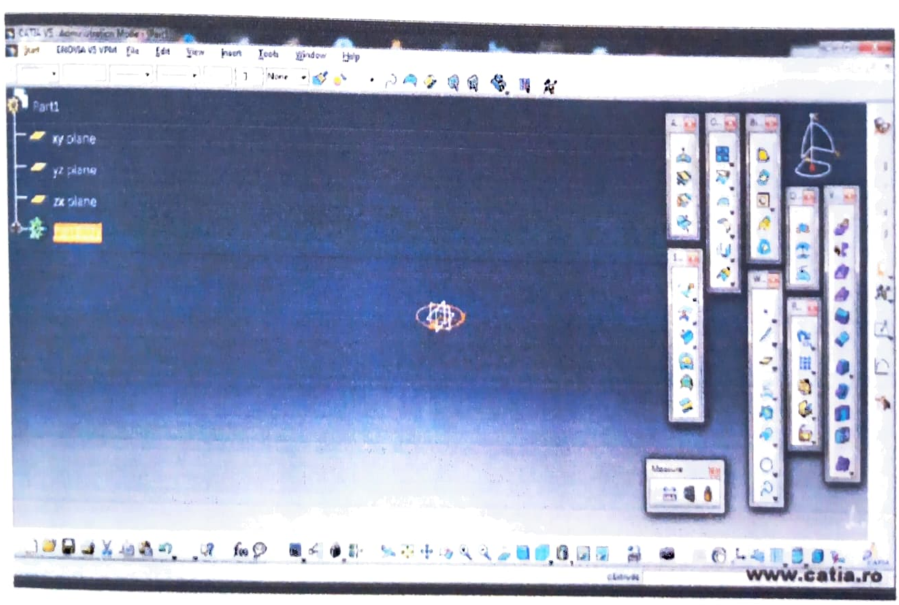

Fig.11 CATIA Environment

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.

Mechanical engineering:

CATIA enables the creation of 3D parts, from 2D sketches, sheetmetal, composites, molded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing & BIW. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mold & die. In the case of Aerospace engineering an additional module named the aerospace sheetmetal design offers the user combine the capabilities of generative sheetmetal design and generative surface design.

Design:

CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches(blueprints).

Systems engineering:

The CATIA Systems Engineering solution delivers a unique open and extensible systems engineering development platform that fully integrates the cross-discipline modeling, simulation, verification and business process support needed for developing complex 'cyber-physical' products. It enables organizations to evaluate requests for changes or develop new products or system variants utilizing a unified performance based systems engineering approach. The solution addresses the Model Based Systems Engineering (MBSE) needs of users developing today's smart products and systems and comprises the following elements: Requirements Engineering, Systems Architecture Modeling, Systems Behavior Modeling & Simulation, Configuration Management & Lifecycle Traceability, Automotive Embedded Systems Development (AUTOSAR Builder) and Industrial Automation Systems Development (Control Build).

CATIA uses the open Modelica language in both CATIA Dynamic Behavior Modeling and Dymola, to quickly and easily model and simulate the behavior of complex systems that span multiple engineering discipline. CATIA & Dymola are further extended by through the availability of a number of industry and domain specific Modelica libraries that enable user to model and simulate a wide range of complex systems – ranging from automotive vehicle dynamics through to aircraft flight dynamics.

Electrical systems:

CATIA v5 offers a solution to formulate the design and manufacturing of electrical systems spanning the complete process from conceptual design through to manufacturing. Capabilities include requirements capture, electrical schematic definition, interactive 3D routing of both wire harnesses and industrial cable solutions through to the production of detailed manufacturing documents including form boards.

Fluid systems:

CATIA v5 offers a solution to facilitate the design and manufacturing of routed systems including tubing, piping, Heating, Ventilating & Air Conditioning (HVAC). Capabilities include requirements capture, 2D diagrams for defining hydraulic, pneumatic and HVAC systems, as well as Piping and Instrumentation Diagram (P&ID). Powerful capabilities are provided that enables these 2D diagrams to be used to drive the interactive 3D routing and placing of system components, in the context of the digital mockup of the complete product or process plant, through to the delivery of manufacturing information including reports and piping isometric drawings.

CHAPTER -2

2.Literature Review

Propeller blades are designed for moderate and deeper depths require minimization of structural weight for increasing payload, performance/speed and operating range for that purpose Aluminum alloy casting is used for the fabrication of propeller blades[1]. In current years the increased need for the light weight structural element with acoustic insulation, has led to use of fiber reinforced multi-layer composite propeller[2].one's attention is restricted to air propellers operating at low Mach numbers (where compressibility effects are negligible) and to water propellers operating without cavitations[3].The objective of this study is to evaluate the strength and vibration characteristics of the Propeller blade design for metal and composite material. Also compare the performance under different operating loading conditions[4].This paper effort is made to reduce frequency of composite propeller so advantage of weight reduction can be obtained[5]. The application of composite materials technology to marine architecture has increased with particular benefits of its light weight, less noise and pressure fluctuation and less fuel consumption[6].GauFenglin[7] carried out stress calculations for fibre reinforced composite thrust blade. Changes to the tensile and flexure properties of marine-grade glass –reinforced polyster, vinylster and resole phenolic composites after exposure to radiant heat are investigated[8].When using the Genetic Algorithm approach, some techniques for parameter setting to provide quick and correct results were discussed along with the influence of these parameters[9].The numerical results are in aggrement with experimental data and the general characteristics of the propeller flow seem to be quite well predicted[10].The metal propellers generally used cause vibration during its operation. In order to avoid it, conventional isotropic materials are replaced with composite materials. Glass fibre reinforced plastics (GFRP) materials are woven with fibre orientation angles 45, –45. Strength analysis is carried out for composite propeller by using different number of layers for composite materials and inter laminar shear stresses are found out[11].

CHAPTER -3

3.Design of propeller

To model a propeller blade of particular series type airfoil points of specific type are required. In present work standard airfoil points are chosen for the modelling. The outline airfoil points and propeller blade are modelled in catia V5. Since the propeller blade consisting of various radii are located through corresponding pitch angles. Then all rotated sectionsare projected onto a right circular cylinder of respective radii as shown in fig below. Then by using multi section surface option, the blade is modelled. The solid model of the propeller blade along with hub is imported and solid mesh is generated for the model.

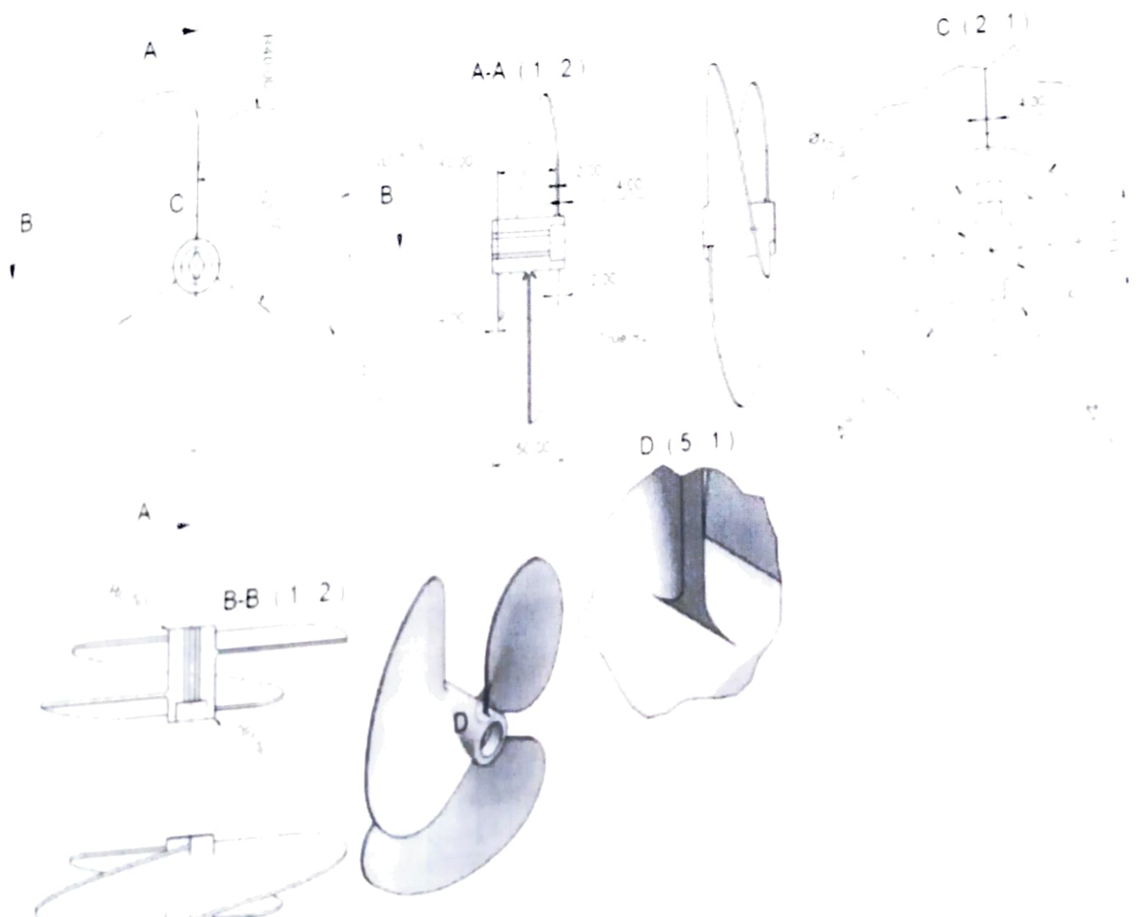

Fig.12 DESIGN PARAMETERS

3.Design of propeller

To model a propeller blade of particular series type airfoil points of specific type are required. In present work standard airfoil points are chosen for the modelling. The outline airfoil points and propeller blade are modelled in catia V5. Since the propeller blade consisting of various radii are located through corresponding pitch angles. Then all rotated sectionsare projected onto a right circular cylinder of respective radii as shown in fig below. Then by using multi section surface option, the blade is modelled. The solid model of the propeller blade along with hub is imported and solid mesh is generated for the model.

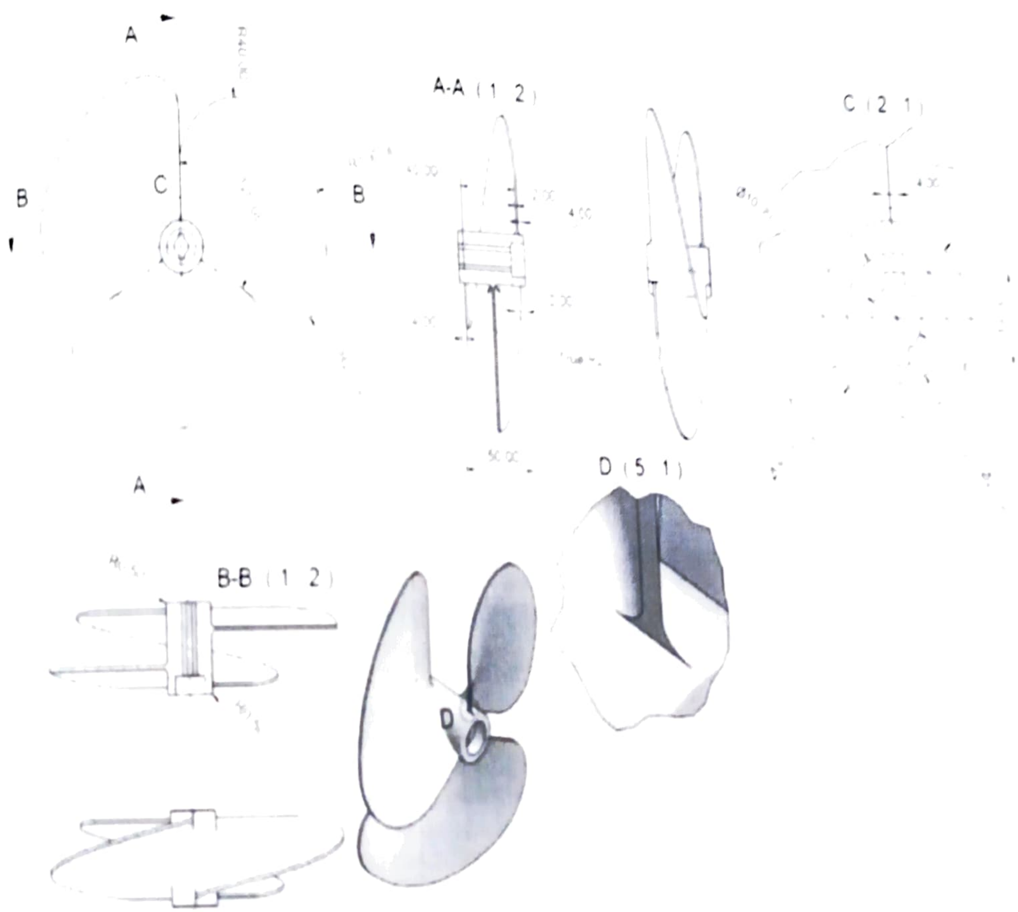

Fig.12 DESIGN PARAMETERS

3.1.Steps involved in design of the propeller

Step1 :

First of all dimensions should be in mm.

We take a circle of diameter equal to the diameter of hub of the propeller

Extrude the circle in vertical plane to a height of equivalent to length of the hub

Take another circle of diameter equal to outer diameter of propeller blade.

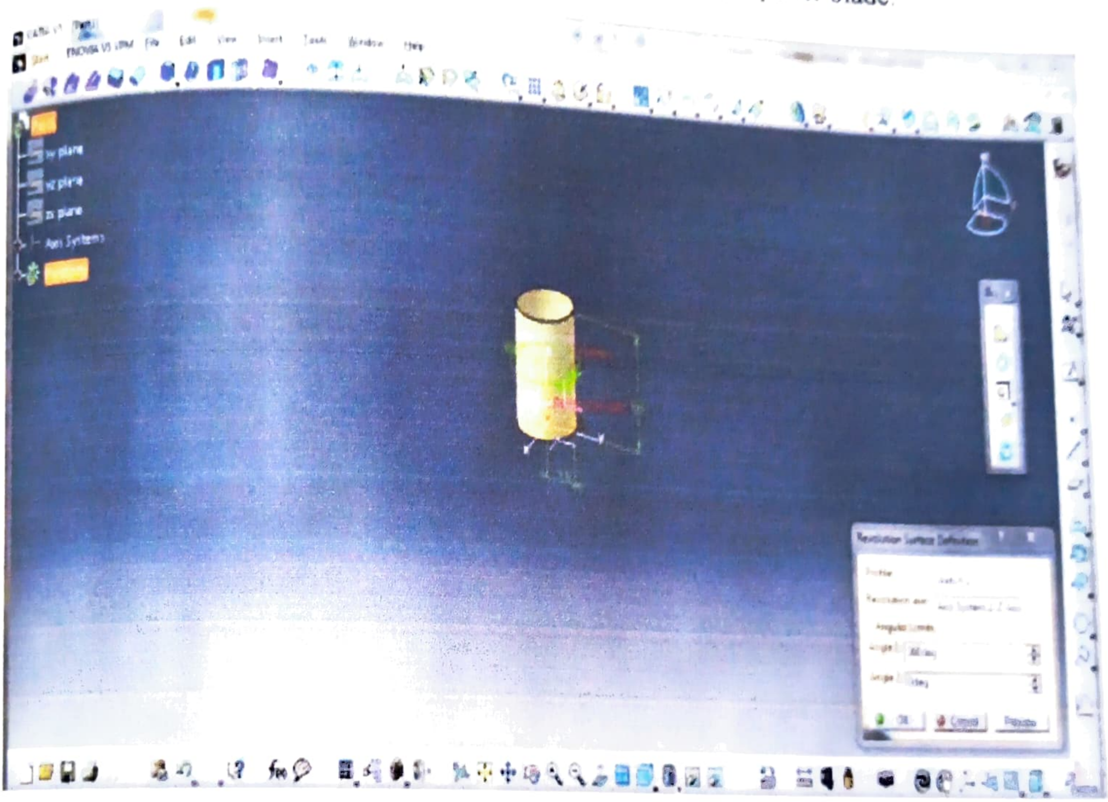

Fig.13 INITIAL STAGE OF DESIGN

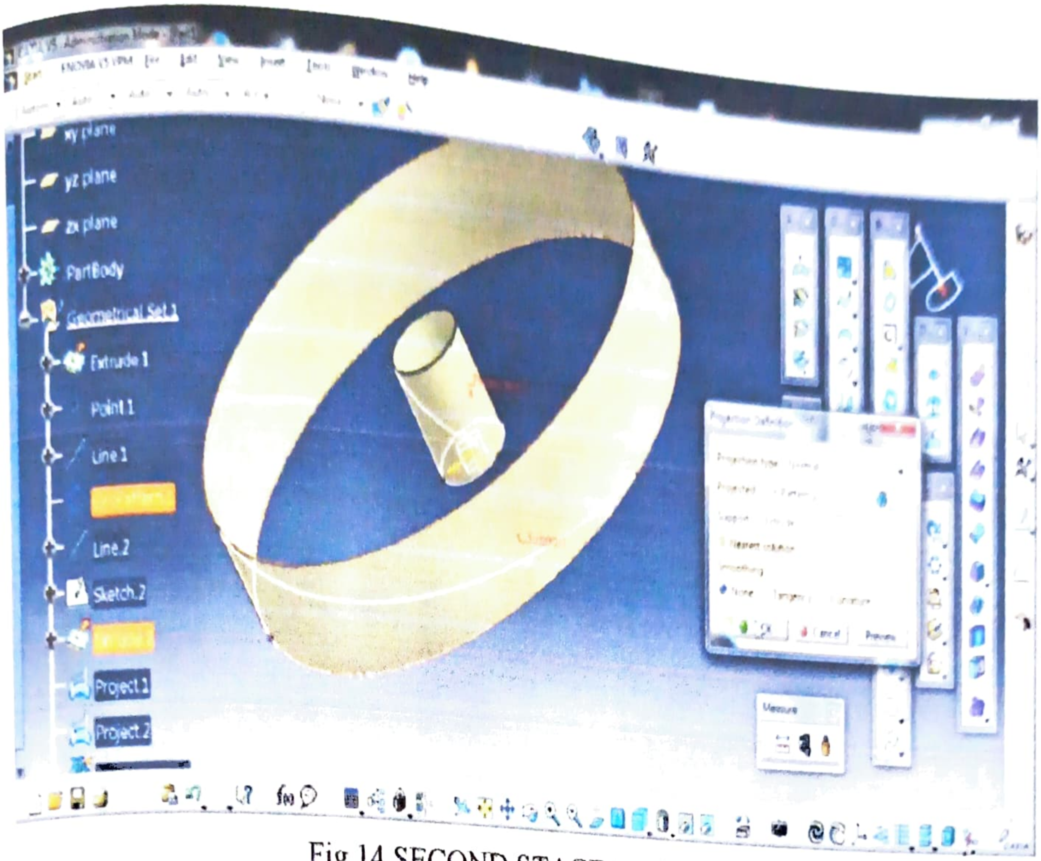

Fig.14 SECOND STAGE OF DESIGN

Step 2: Join the lines from small cylinder to larger cylinder then we get the shape of propeller blade.

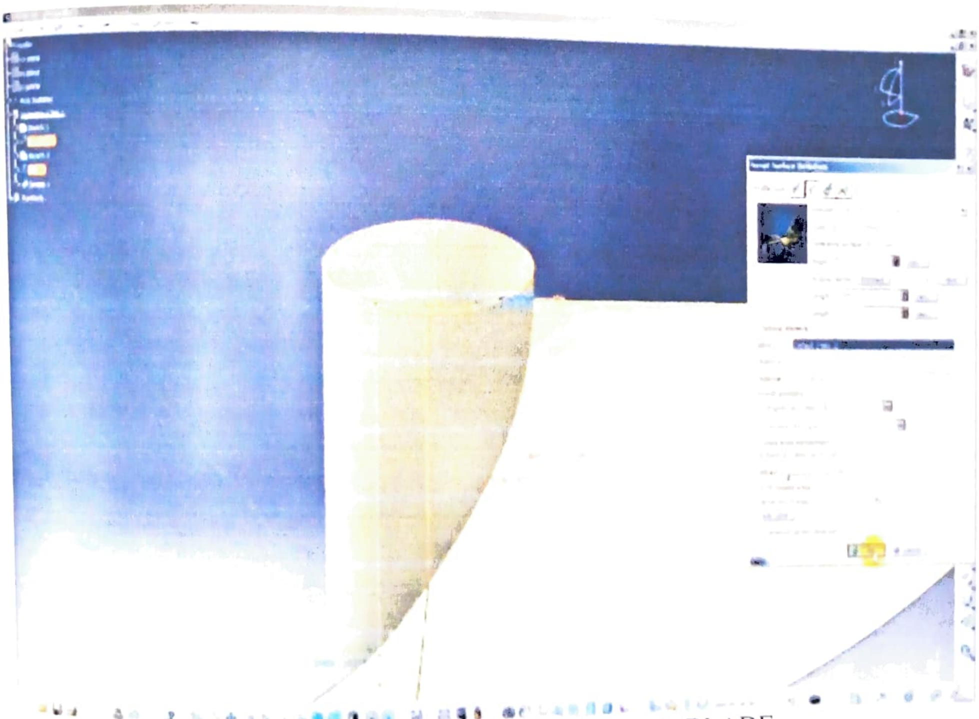

Fig 15 DESIGN OF SINGLE BLADE

Step 3:

After obtaining single blade , by taking circular pattern and give number of blades as 3

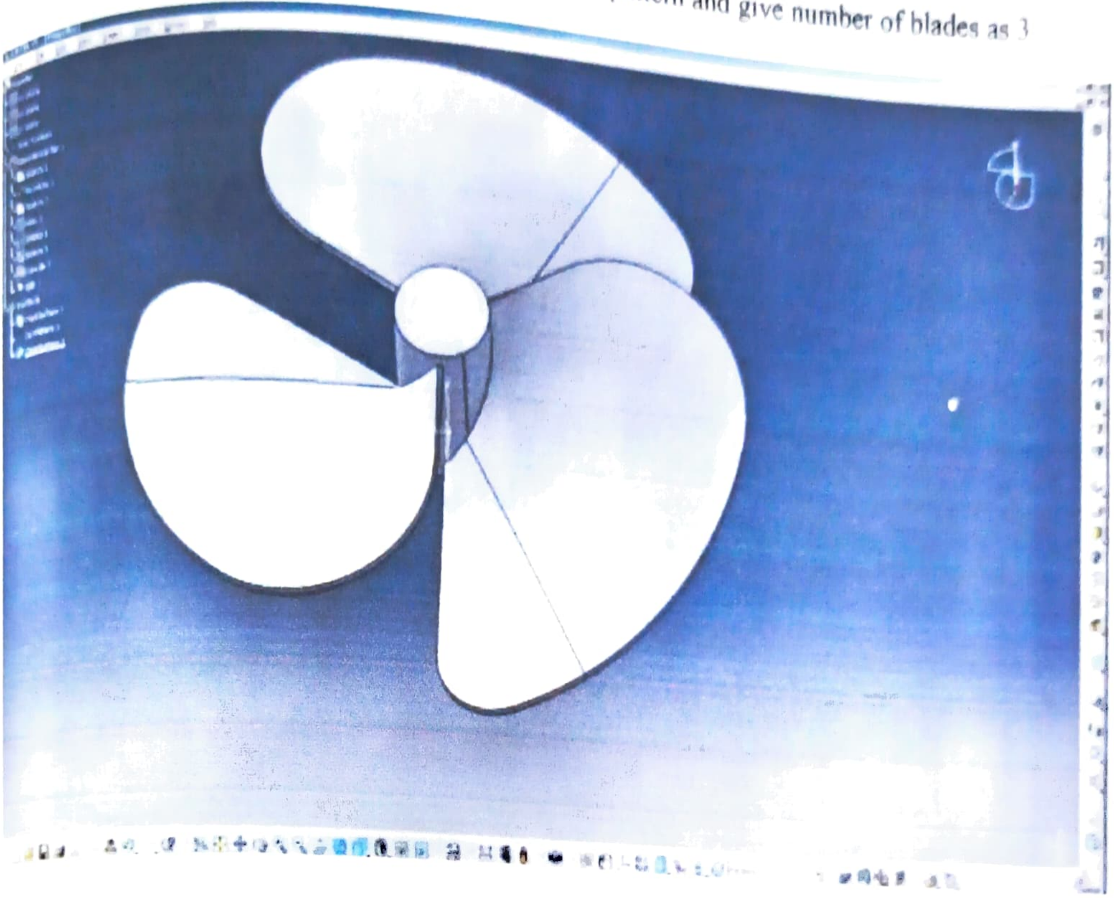

Fig 16 FINAL STAGE OF DESIGN

Stage 4:

Provide a key way to solid hub for mating of driven shaft and hub of propeller.

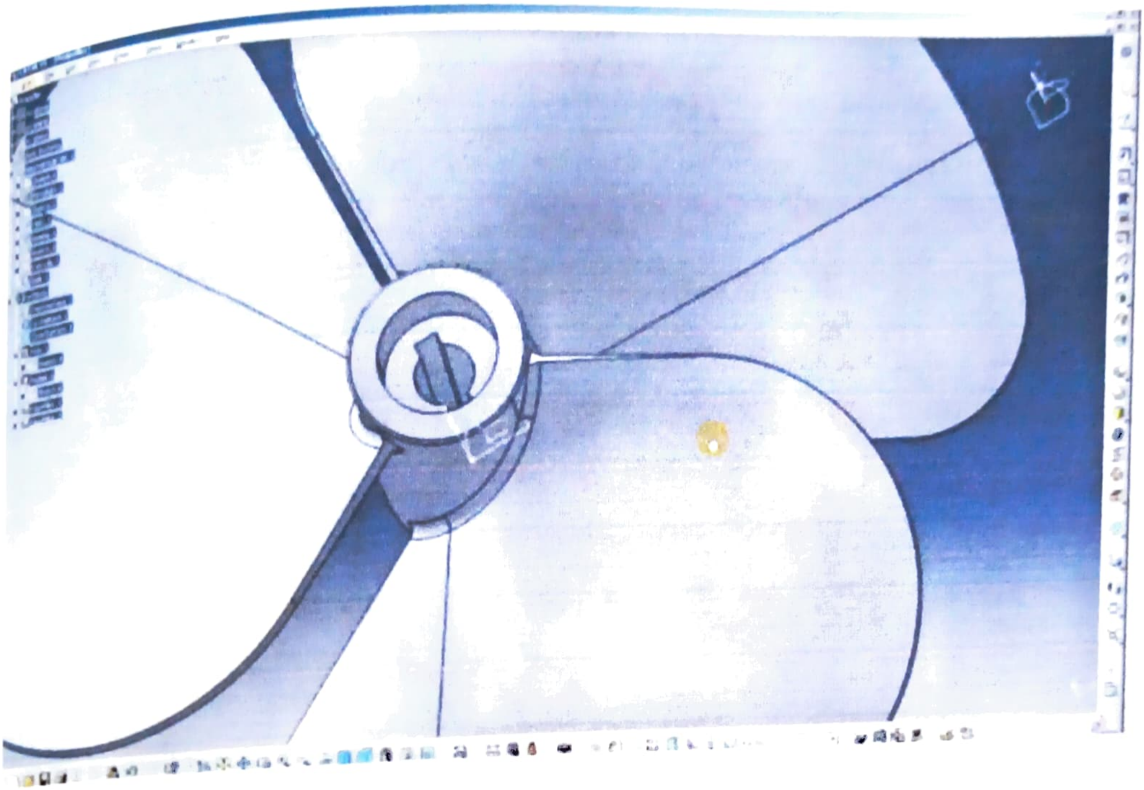

Fig.17: FINAL VIEW OF BLADE WITH A HUB

- Finally save the model in IGES format.

CHAPTER -4

4.Introduction to Finite Element Analysis

Finite Element Analysis uses numerical methods to solve complex problems in Mechanical as well as civil structural problems. It deals with finding out the deflections of structural members, stresses produced due to various loading coming from applications and finding the right size and shape as well as orientation so as to optimize the design of the structure.

Finite Element Analysis(FEA) solves the problem by dividing the problem domain into a small number of parts/cells/elements and for each individual element, the governing equation of problem physics is applied. A final solution is then calculated at the end by summing up effect/solution of each element of the problem domain. The solution means deflection, stress, and strain at the desired location or points in the domain. In FEA, solution largely depends on element types. Each element type has its own characteristics.

Finite Element Analysis consists of the following steps:

1. Preprocessing
   a. Discretization
   b. Apply constraints
2. Processing
3. Postprocessing

This discretization is the process of dividing up the model into elements consisting of nodes. The processing phase solves equations for these nodes and obtains results.Meshing is discretization. It is the most important part of an analysis and can determined the efficiency and effectiveness of an analysis. Therefore, a lot of time is given to meshing of complex models. If you don't mesh, you won't get the finite elements and there won't be a "Finite Element" Analysis.So, meshing is the core of finite element analysis.

Node:

FEM is a numerical method (approximate method of solving equations) which discretizes a continuous domain into finite number of points. These points are called as **nodes**. Equations governing the physics involved in the system are solved at each of these nodes and then solution is interpolated and represented all over the system using an appropriate interpolation function.

Element:

4.1.Basic types of elements used in Finite Element Analysis (FEA)

1D (Line element)

Mainly of three types -

Linear (2 Noded per side) e.g. Beam, truss

Quadratic (3 Noded per side) e.g. Beam

Cubic (4 Noded per side) e.g. Beam

The accuracy of analysis directly depends on the type of element chosen in analysis inputs. Cubic elements give better analysis compared to the quadratic element and quadratic gives a better result than the linear element.

2D (Area element)

Mainly of three types-

Linear (2 Noded per side) e.g. Plane stress

Quadratic (3 Noded per side) e.g. Plane strain

Cubic (4 Noded per side) e.g. Plate and shell

3D (Volume element)

Mainly of two types

Linear

Quadratic

4.2. Common element types used in ANSYS

Element type LINK1:

It is a 2D spar element that supports tension and compression forces in application LINK1 does not support bending force related information, Hence by LINK1 only approximate analysis is possible

This element has only two degrees of freedom (Displacement in X and Y direction only)

Example: Truss elements can be analyzed by LINK1 element

BEAM

2D elastic 3 elements:

This element supports compression, tension and bending capabilities during analysis

2D elastic 3 element has three degrees of freedom (Displacement in X & Y and rotation about 2 axes)

This element defined by 2 nodes, cross-sectional area, the moment of inertia, the height and material property

Element height is used only in the bending and thermal stress calculations

Moment of inertia in the z-direction can be neglected if large deflections are not used.

3D beam element

Element name: Beam 4

This element supports uniaxial compression, tension, and bending capabilities.

It has six degrees of freedom, three in displacement in X, Y, Z direction and remaining six in rotation about the same X, Y, Z axis.

Inputs required to define completely this element are the cross-sectional area, the moment of inertia, the thickness along all directions and shear and strain deflections.

Element Type: Shell

Element name: Elastic 4 node 63

This element supports both membrane and bending capabilities.

Both in-plane and normal loads are permitted for this element type in particular applications

It has six degrees of freedom, three in displacement in X, Y, Z direction and remaining six in rotation about the same axis.

Element Type: Solid/Brick

Element name: Brick 8node 185

This element is used to do Finite element analysis of 3D or solid objects

4.3.Harmonic Analysis

This analysis gives the ability to estimate the sustained dynamic behaviour of structures, thus it enabling to validate the designs will successfully overcome resonance, fatigue and other harmful effects of forced vibrations. Harmonic response analysis is a mode used to find out the steady state response of a linear structure to loads that vary sinusoidal with time. It calculates the propellers response at several frequencies and obtains the graphs of displacement versus frequencies.

4.4.Steps for analysis:

1.Open the ANSYS workbench module.

2.Select geometry under component systems from the tool box.

3.Right click on the geometry and import the model which is saved in IGES form then the model appears in design modeler.

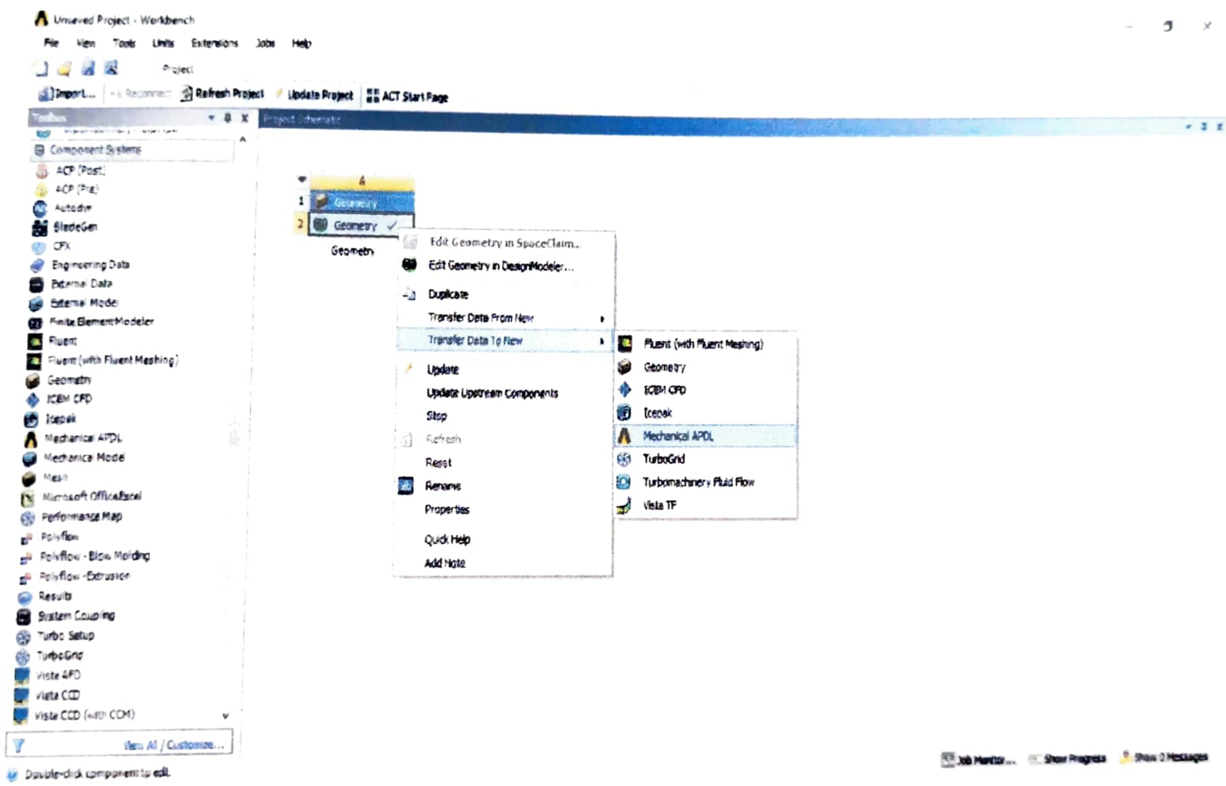

Fig.18: IMPORTING MODEL FROM CATIA TO ANSYS

4.Again right click on the geometry and select **transfer data to new** and select mechanical APDL.and carry out the analysis in APDL.

4.6.Material properties:

TABLE I ARAMID FIBRE

Young's modulus (E)	E_X =58.9Gpa
	E_Y =58.9Gpa
	E_Z = 58.9Gpa
Poisson's ratio (P)	P_{XY} =0.44
	P_{YZ} = 0.44
	P_{ZX} =0.44
Rigidity modulus (G)	G_{XY} =22.09Gpa
	G_{YZ}=22.09Gpa
	G_{ZX} = 22.09Gpa
Density	1440 kg/m^3

TABLE 2. GLASS FIBRE REINFORCED PLASTIC

Young's modulus (E)	E_X =22.925 Gpa
	E_Y =22.925 Gpa
	E_Z = 12.4 Gpa
Poisson's ratio (P)	P_{XY} =0.12
	P_{YZ} = 0.2
	P_{ZX} =0.2
Rigidity modulus (G)	G_{XY} =4.7 Gpa
	G_{YZ} =4.2 Gpa
	G_{ZX} = 4.2 Gpa
Density	2700 kg/m^3

Table.3 NAB MATERIAL PROPERTIES

Young's modulus (E)	7e11 N/m^2
Poisson's ratio	0.29
Density	2700 Kg/m^3

4.7.Mesh generation:

The solid model of the propeller blade along with hub is imported and free mesh is generated for the model.

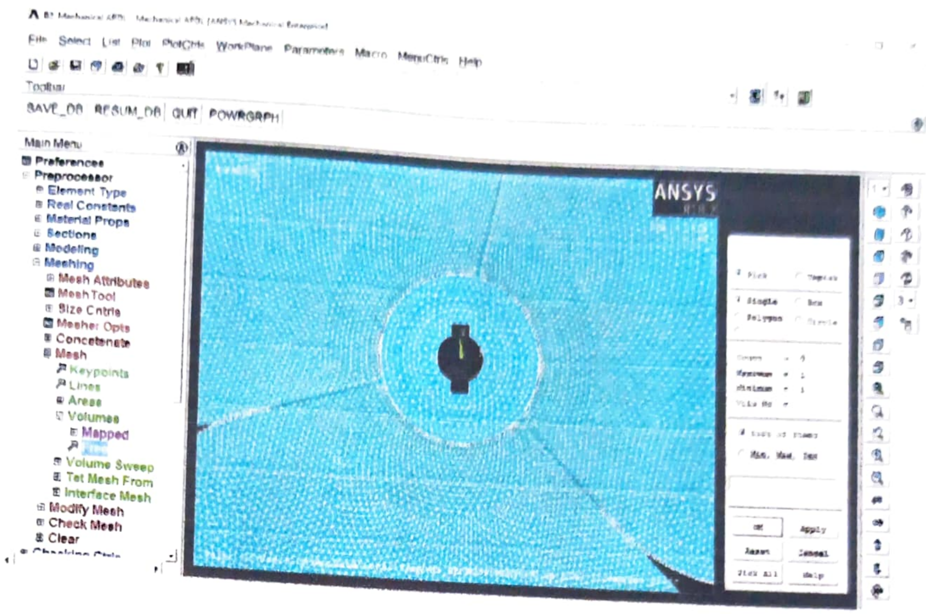

Fig.20. STEPS TO GENERATE MESH

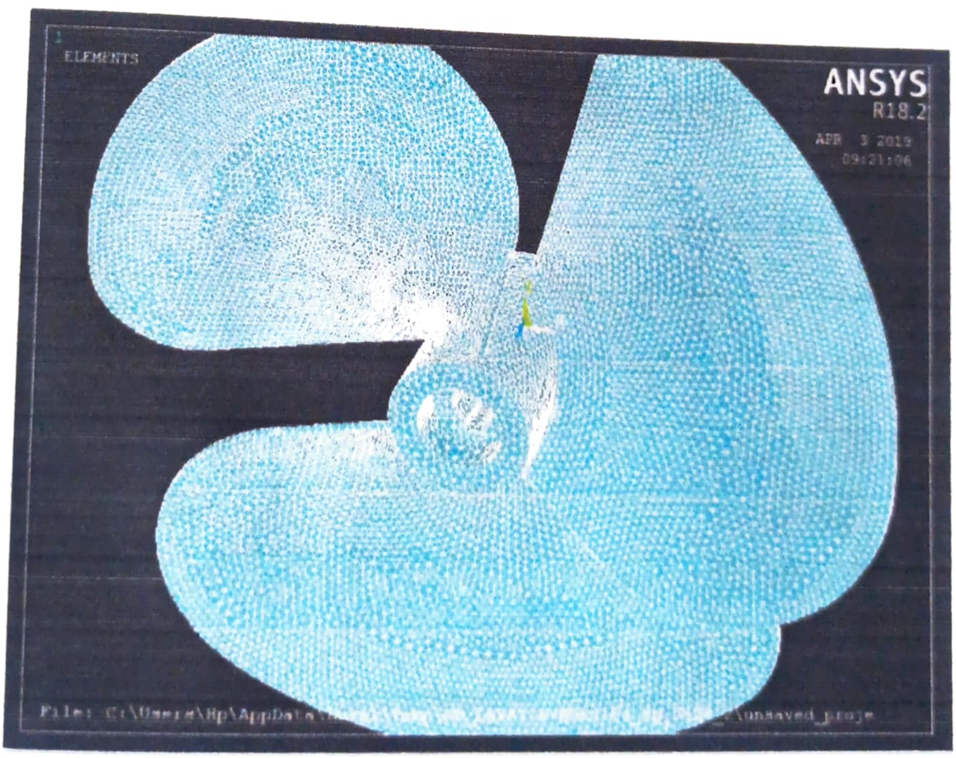

Fig.21. MESH GENERATION

CHAPTER -5

5.BOUNDARY CONDITIONS AND LOADS:

Boundary conditions are applied to meshed model. The contact surface between hub and shaft is fixed in all degrees of freedom. Thrust of 609.1 N is applied.

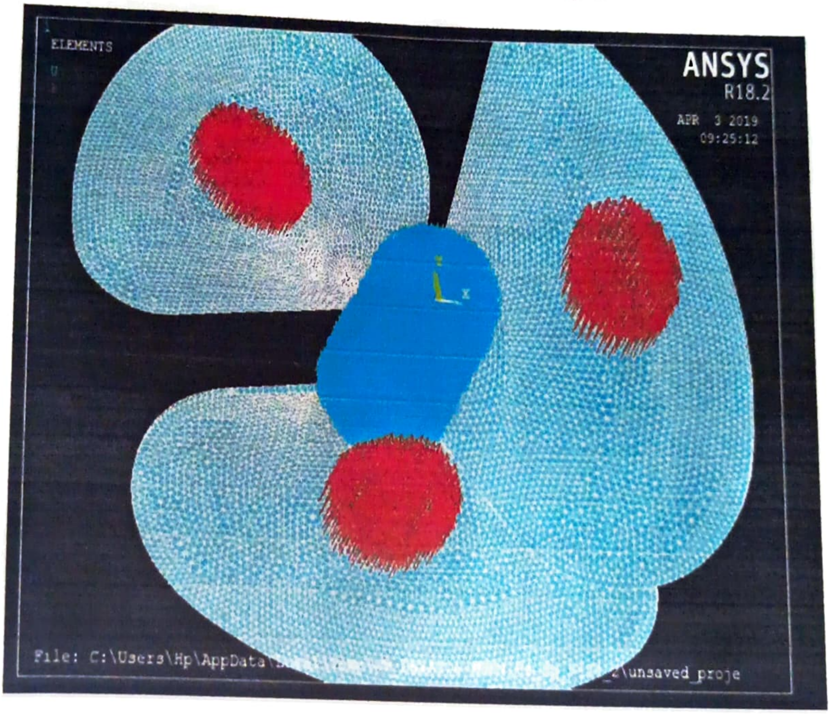

Fig.22: BOUNDARY CONDITIONS AND LOADS

5.1.Solution stage:

After applying required boundary conditions and loads, select the solver to get the solution.

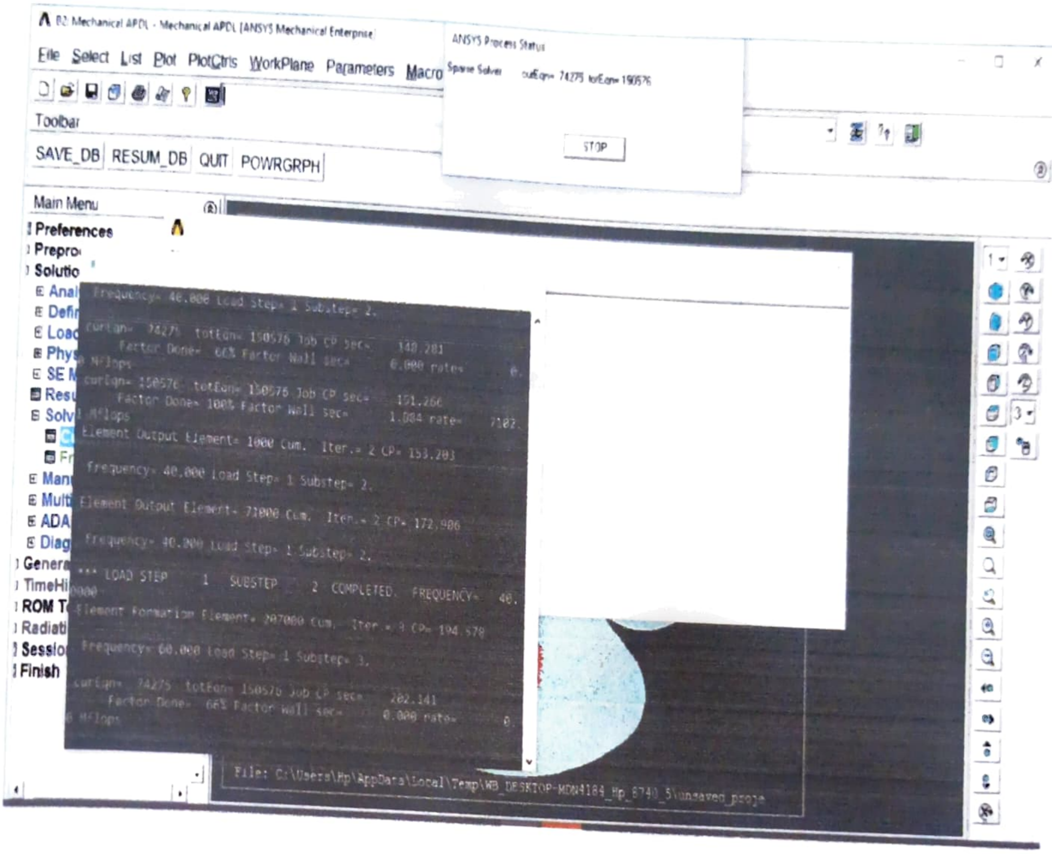

Fig.23: SOLUTION STAGE

CHAPTER -6

6.Results and discussions:

6.1 Harmonic Analysis Of NAB Propeller:

From the harmonic analysis, the displacements of various nodes over the entire frequency range of 0 to 3000Hz were obtained. The natural frequencies of the propeller lies in the same above frequency range. The observed peaks in the frequency response graphs were plotted. Fig 1, Fig 2 and Fig 3 shows the variation of displacement in X, Y and Z directions. The maximum displacement of component in X ,Y and Z directions obtained in harmonic analysis for NAB are shown

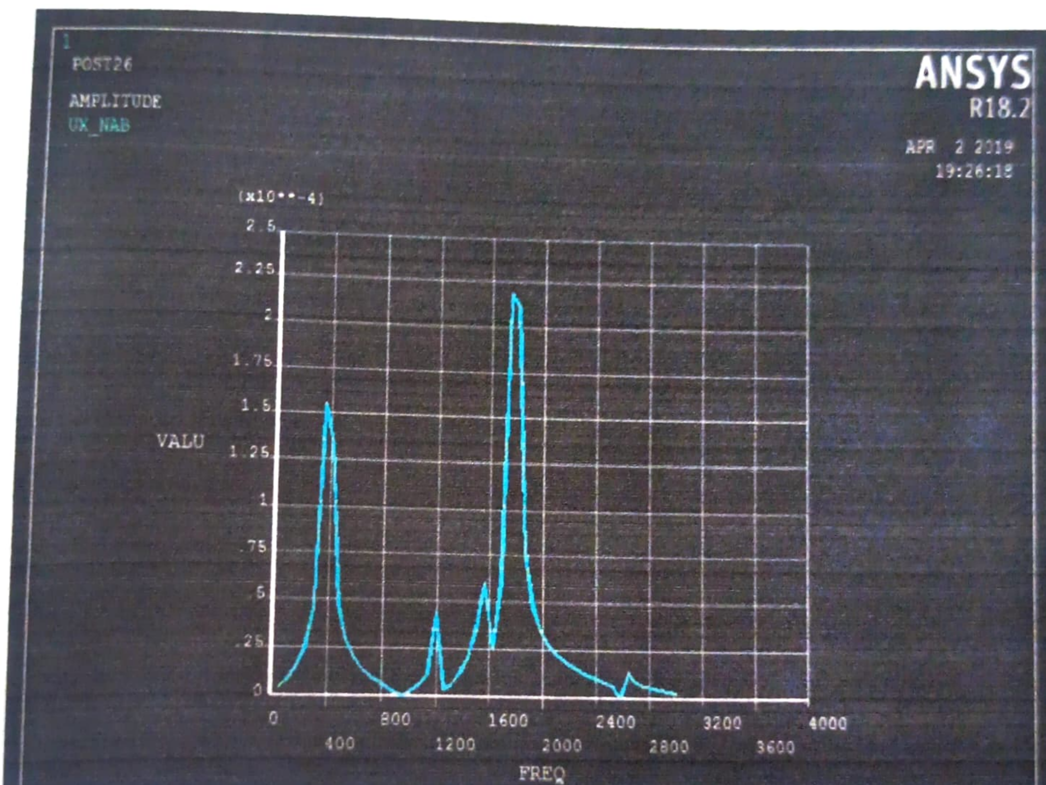

Fig.24. X-Component of displacement Vs frequency response graph of NAB propeller.

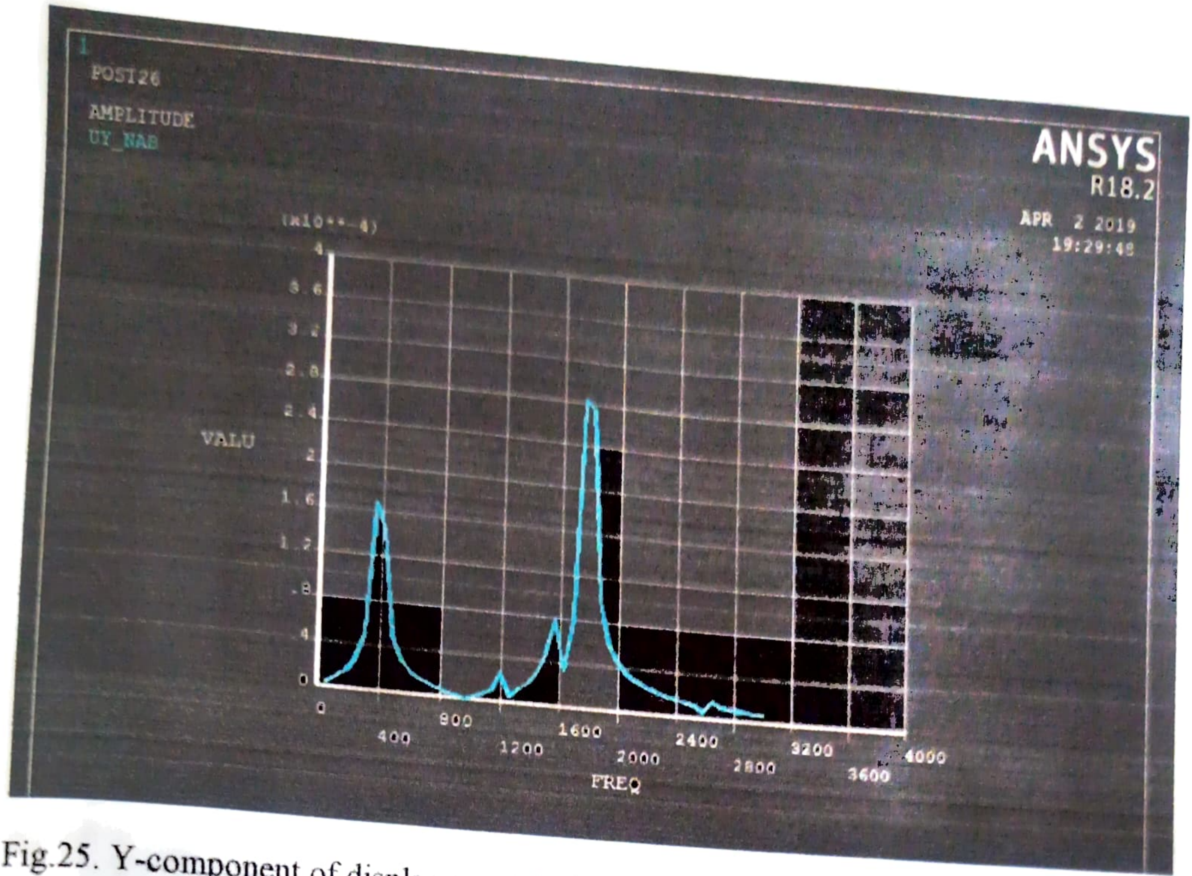

Fig.25. Y-component of displacement Vs frequency response graph of NAB Propeller

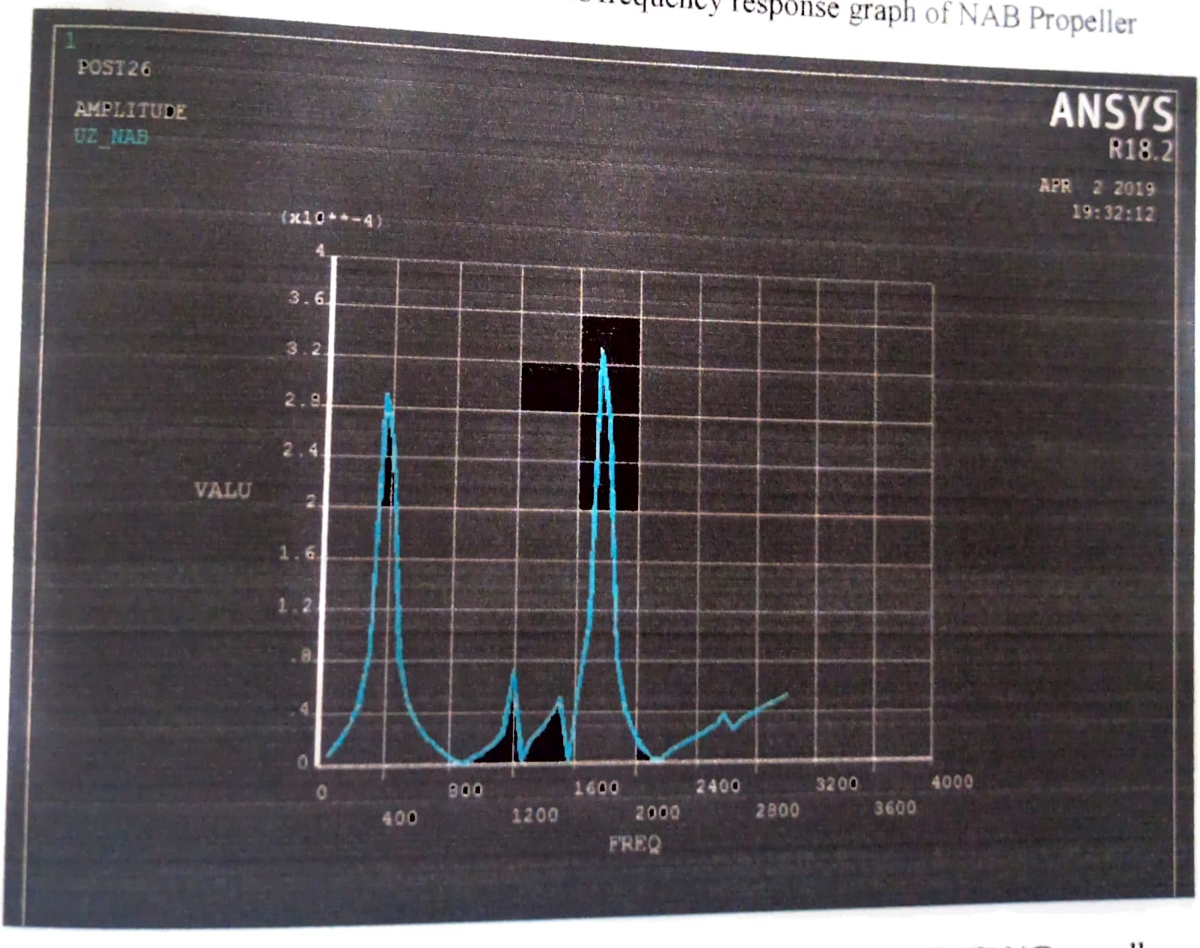

Fig.26. Z-component of displacement Vs frequency response graph of NAB propeller

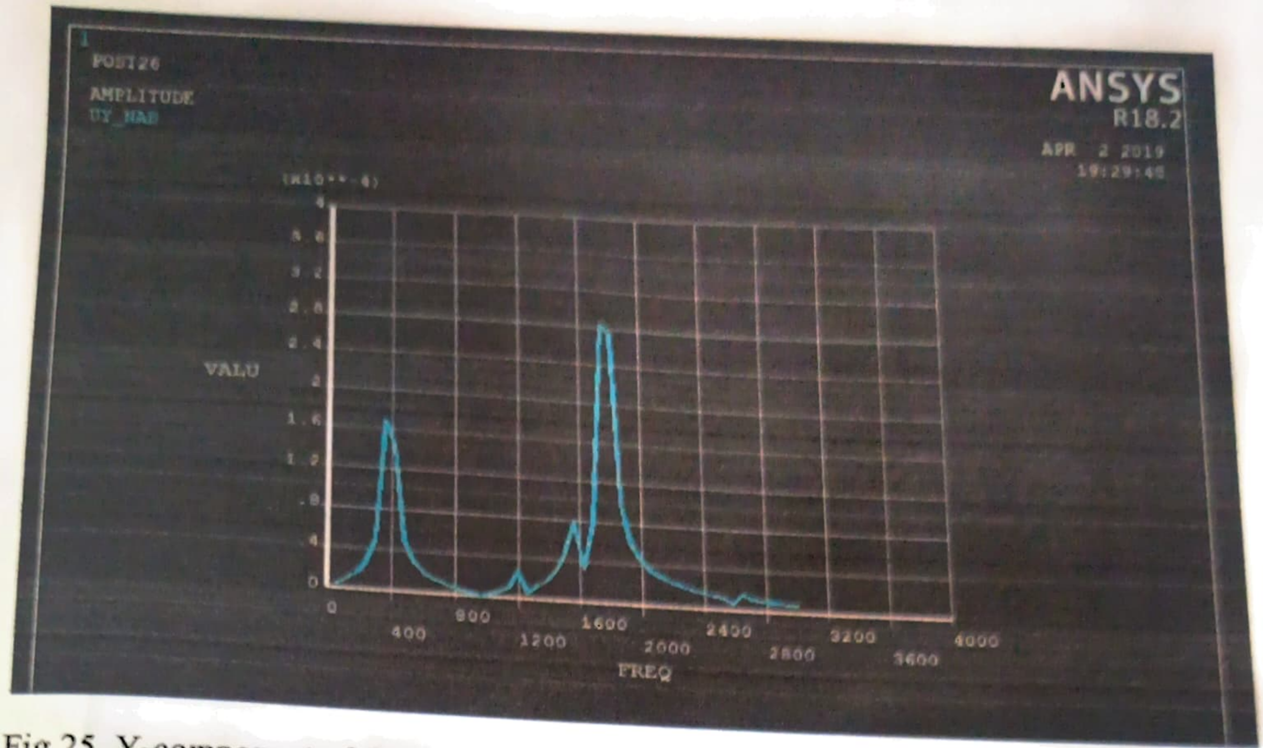

Fig.25. Y-component of displacement Vs frequency response graph of NAB Propeller

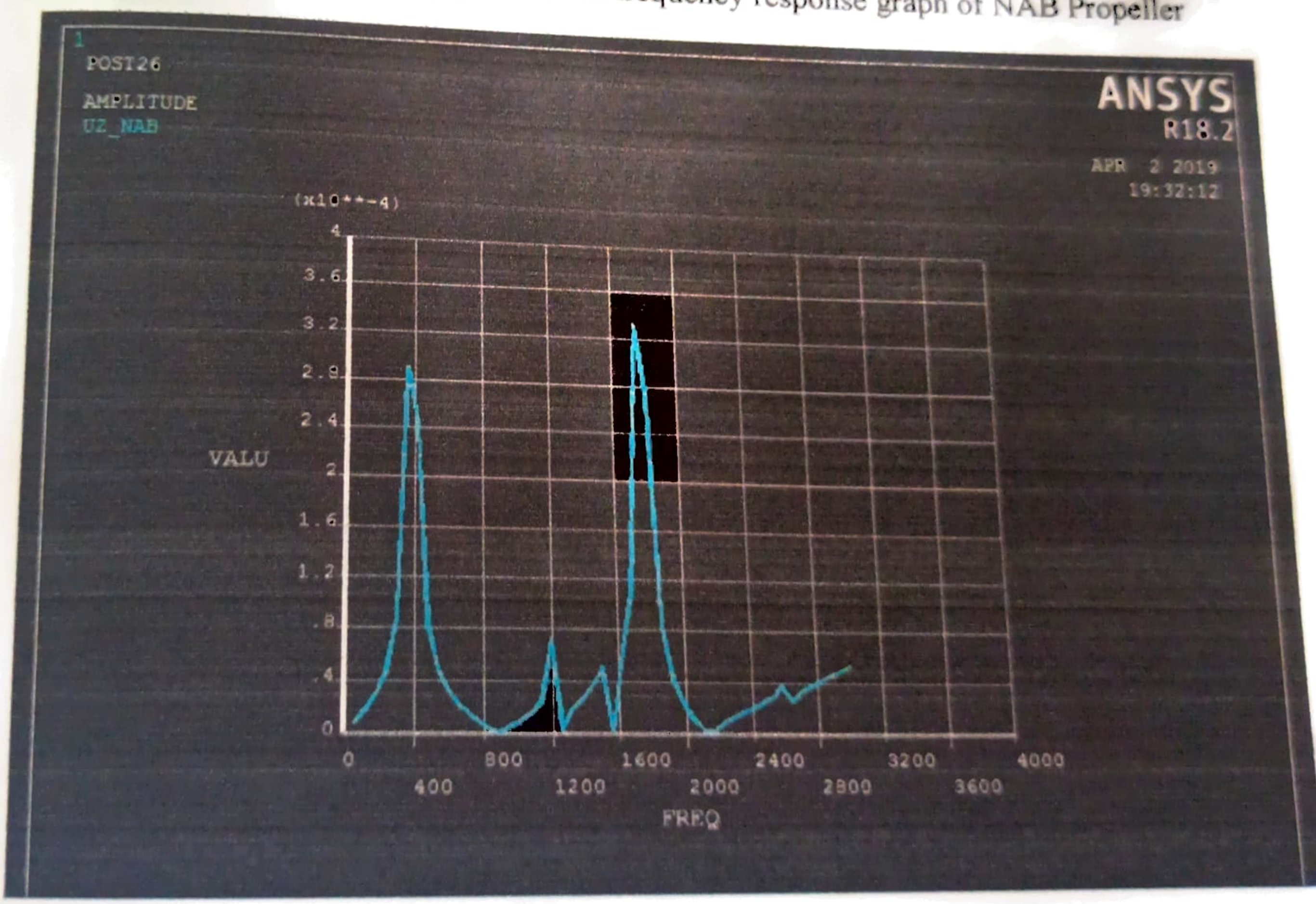

Fig.26. Z-component of displacement Vs frequency response graph of NAB propeller

6.2.Harmonic analysis of composite Propeller:

From the harmonic analysis, the displacements of various nodes over the entire frequency range of 0 to 3000 Hz were obtained. The natural frequencies of the propeller lies in the same above frequency range. The observed peaks in the frequency response graphs were plotted. Fig 4, Fig 5, and Fig6 shows the variation of displacement in X, Y and Z directions. The maximum displacement of component in X ,Y and Z directions obtained in harmonic analysis for composite material were plotted below.

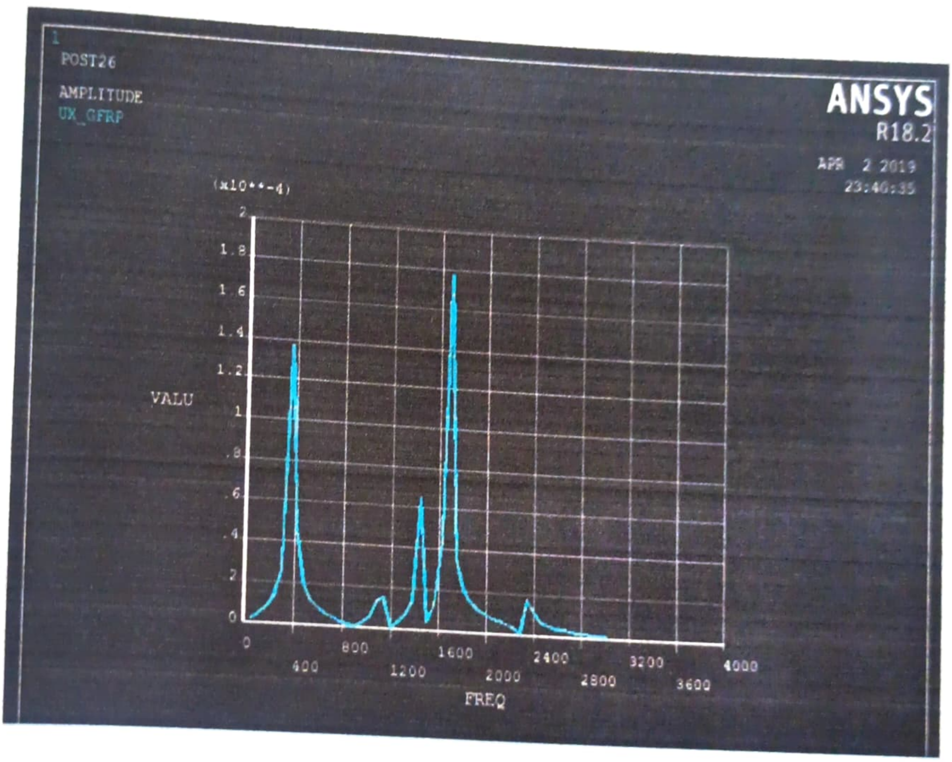

Fig.27. X-component of displacement Vs frequency response graph of GFRP propeller

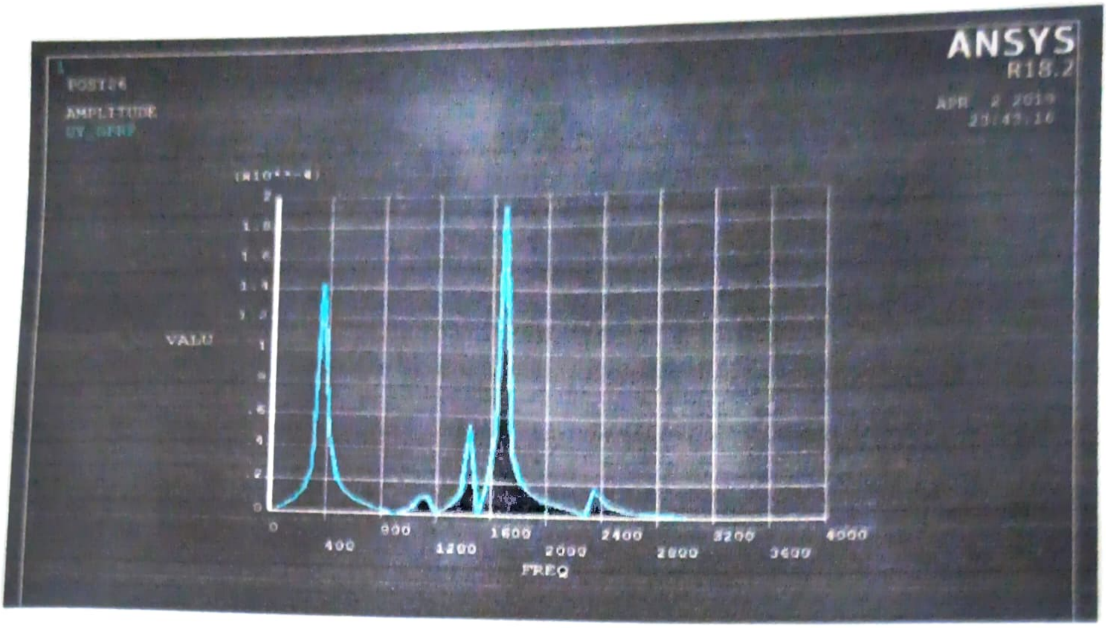

Fig.28. Y-component of displacement Vs frequency response of GFRP Propeller

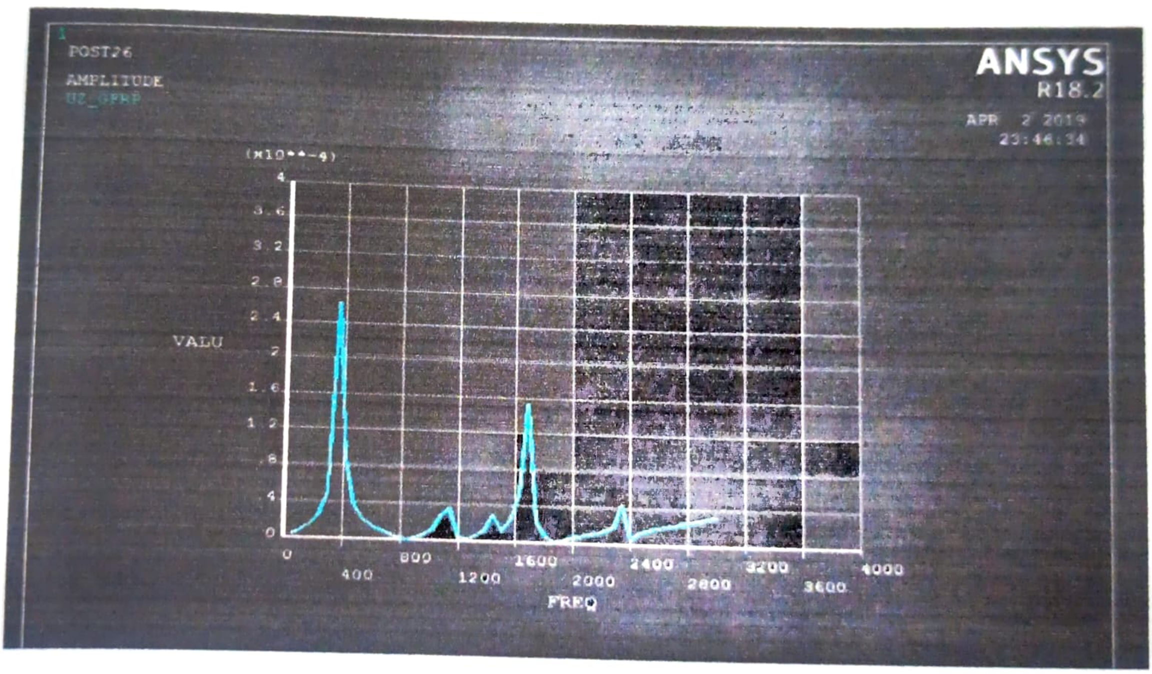

Fig.29. Z-component of displacement Vs frequency response of GFRP Propeller

CHAPTER-7

7.1 COMPARSION OF RESULTS OF NAB AND GFRP :

Table 4. Comparsion of results

	Maximum Displacements of NAB Propeller	Maximum Displacements of Composite Propeller
X-Component	0.22mm	0.178mm
Y-Component	0.285mm	0.192mm
Z-Component	0.34mm	0.265mm

CHAPTER -8

8. CONCLUSIONS:

1. From the results of harmonic analysis, composite propeller is safe against resonance phenomena because the obtained frequency comes in the range of given input i.e 0 to 3000Hz.
2. Harmonic analysis is carried out on both NAB and composite propellers, it was observed maximum displacement for composite propeller is less than the NAB propeller.
3. Due to addition of Aramid fiber layer on the glass fiber cavitation performance of composite propeller is improved.
4. Here by we conclude that weight of the composite propeller is less than that of NAB propeller so that reaction forces can reduce to some extent due to which durability of propeller increases.
5. As weight gets reduced, composite propellers are used to drive high speed under water vehicles.

REFERENCES:

1.Design and analysis of propeller blade.

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 - 0056 Volume: 02 Issue: 06 | Sep-2015

2. Structural and design analysis of propeller blade.

International Journal of Advances in Mechanical and Civil Engineering, ISSN: 2394-2827 Volume-4, Issue-2, Aprl.-2017

3. Modelling & Structural Analysis of Propeller Blade

ISSN: 2455-2631 © July 2017 IJSDR | Volume 2, Issue 7

4. Structural and Vibration Analysis of propeller blade in small aircraft

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056

5. Harmonic analysis of composite propeller for marine applications

International journal of research in engineering and engineering. ISSN 2319-1163.

6. J.F.Tsai, H.J.L.a., Analysis of Underwater Free Vibrations of a Composite Propeller Blade. *Journal of Reinforced Plastics And Composites, 2008. Vol. 27, No. 5:* p. 447-458.

7. Gau-Feng Lin "Three Dimensional Stress Analysis of a Fiber Reinforced Composite Thruster Blade", *the society of naval architects and marine engineers ,1991.*

8.A.P. Mouritz, Z.M, Post fibre mechanical properties of marine polymer composites, *Elseveir science Ltd,2000,47*:p.643-653.

9.Chang j.h, Y.S. Shih, Basic design of series propeller with vibration consideration by genetic algorithm, *Journal of Marine Science and Technology,2007.12(3)*,119-129.

10.Amoraripei M, Predection of marine propeller performance using RANS, Numerical analysis and Applied Mathematics, *International Conference,2008,*771-774.

11. Dynamic analysis of composite propeller of ship using FEA.Int. J. Computer Aided Engineering and Technology, Vol. 10, No. 6, 2018