



## DESIGN AND ANALYSIS OF AIR CRAFT

**A.Sateesh**

*Master of technology, Gates institute of Technology, Gooty*

### 1. INTRODUCTION

#### 1.1 General Information

Thrust is the force that moves the aircraft through the air. Thrust is generated by the propulsion system of the aircraft. There are different types of propulsion systems develop thrust in different ways, although it usually generated through some application of Newton's Third Law. Propeller is one of the propulsion systems. The purpose of the propeller is to move the aircraft through the air. The propeller consists of two or more blades connected together by a hub. The hub serves to attach the blades to the engine shaft. .

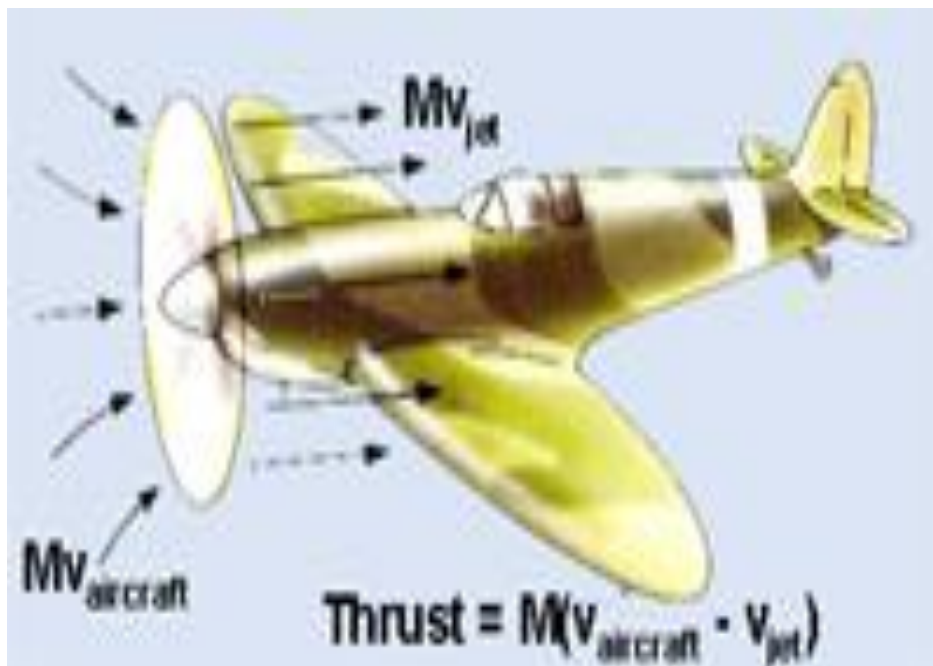


FIGURE 1.1

The blades are made in the shape of an airfoil like wing of an aircraft. When the engine rotates the propeller blades, the blades produce lift. This lift is called thrust and moves the aircraft forward. Most aircraft have propellers that pull the aircraft through the air. These are called tractor propellers. Some aircraft have propellers that push the aircraft. These are called pusher propellers.

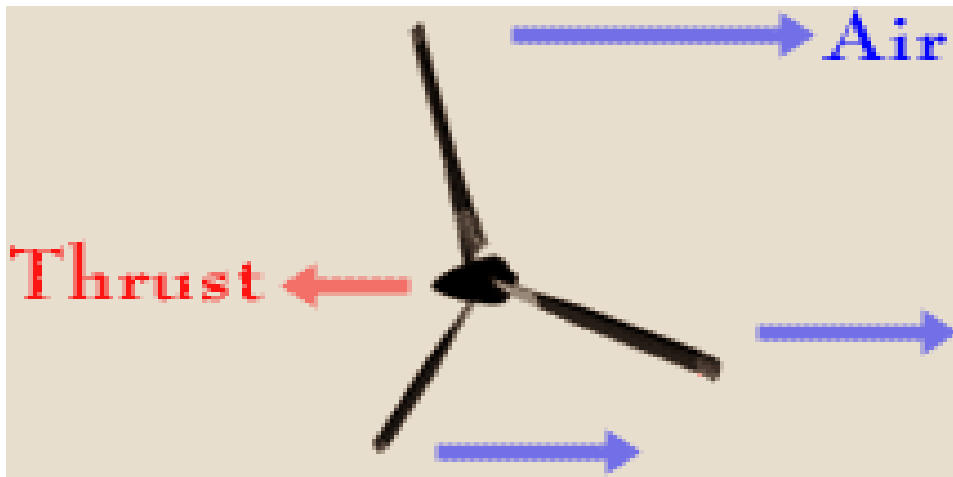


FIGURE 1.2

### 1.2Description

Leading Edge of the airfoil is the cutting edge that slices into the air. As the leading edge cuts the air, air flows over the blade face and the camber side.

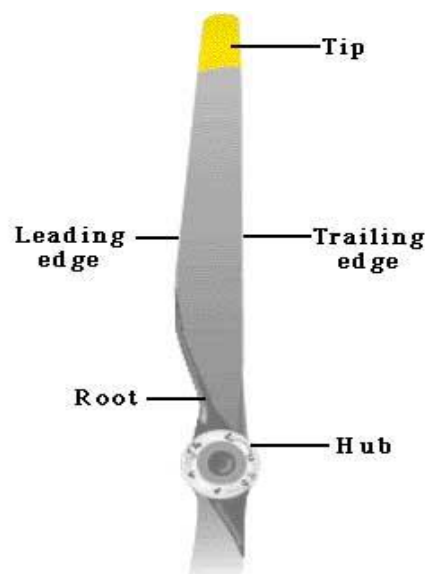


FIGURE 1.3

Blade Face is the surface of the propeller blade that corresponds to the lower surface of an airfoil or flat side, we called Blade Face.

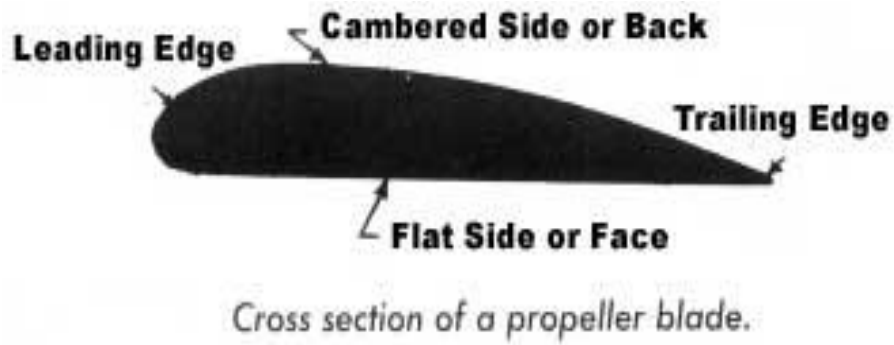


FIGURE 1.4

Blade Back / Thrust Face is the curved surface of the airfoil.

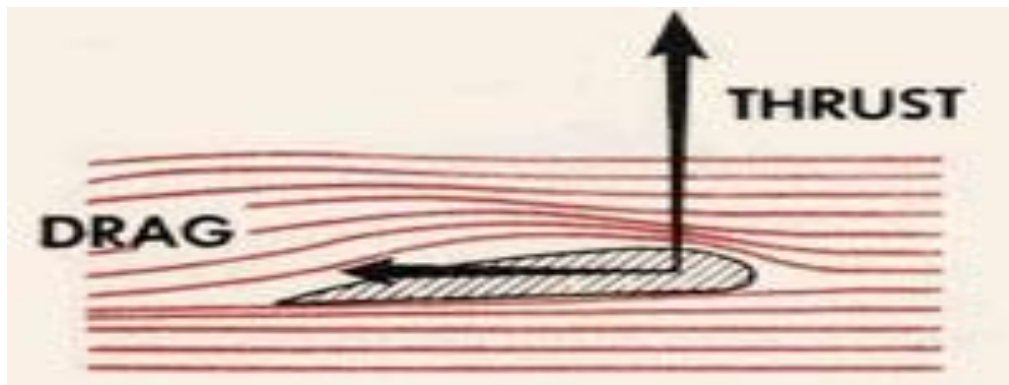


FIGURE 1.5

Blade Shank (Root) is the section of the blade nearest the hub.

Blade Tip is the outer end of the blade farthest from the hub.

Plane of Rotation is an imaginary plane perpendicular to the shaft. It is the plane that contains the circle in which the blades rotate.

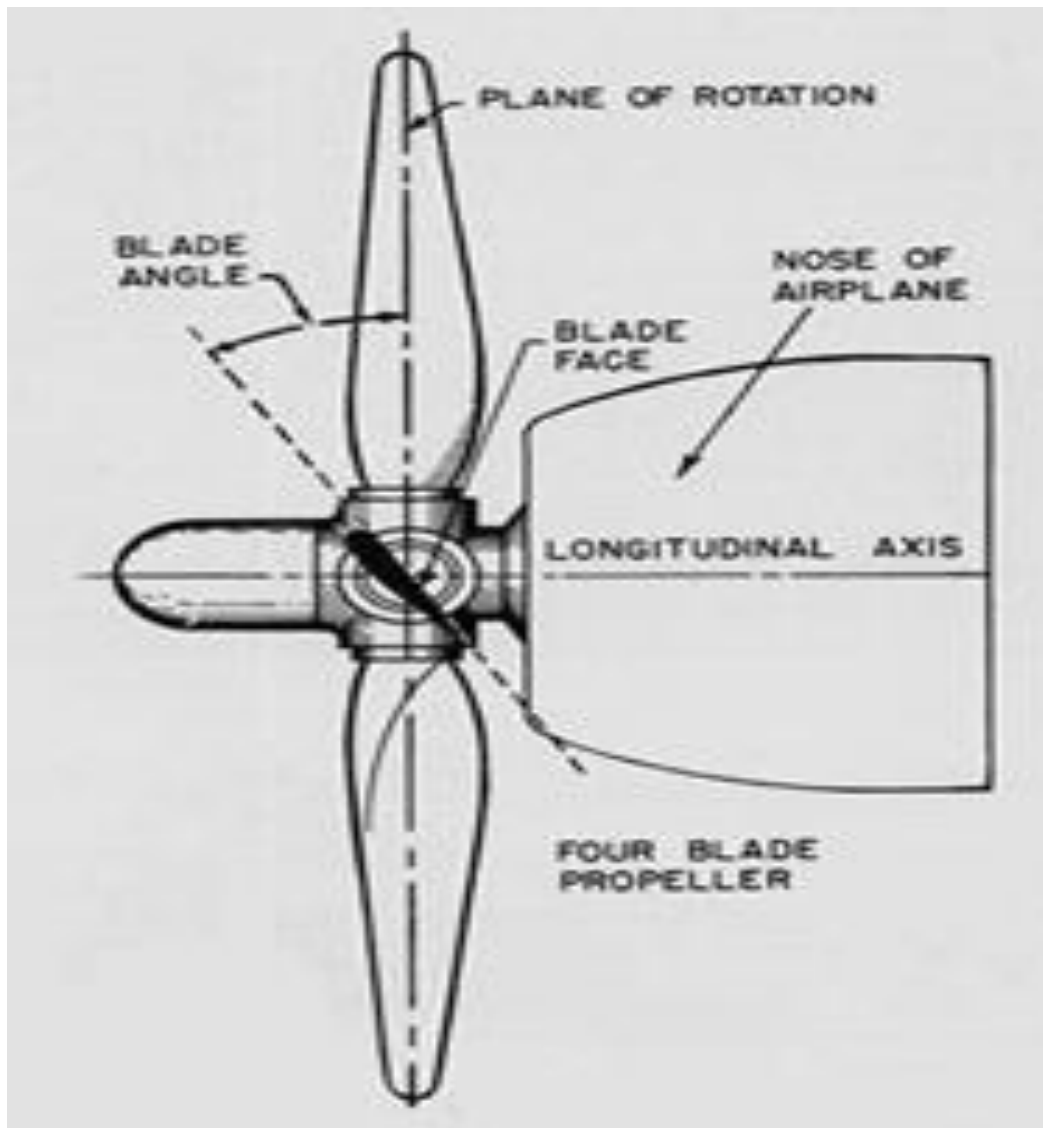


FIGURE 1.6

Blade Angle is formed between the face of an element and the plane of rotation. The blade angle throughout the length of the blade is not the same. The reason for placing the blade element sections at different angles is because the various sections of the blade travel at different speed.

Each element must be designed as part of the blade to operate at its own best angle of attack to create thrust when revolving at its best design speed.

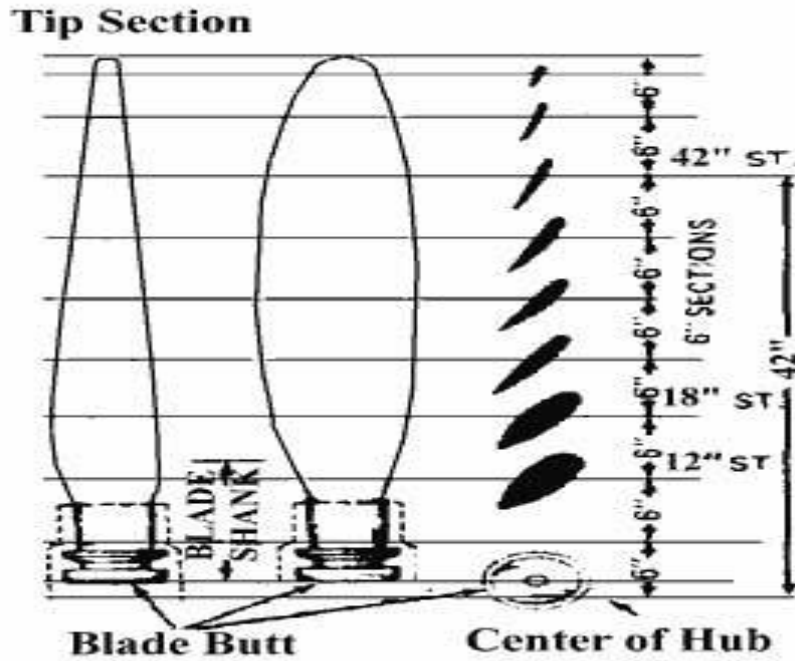
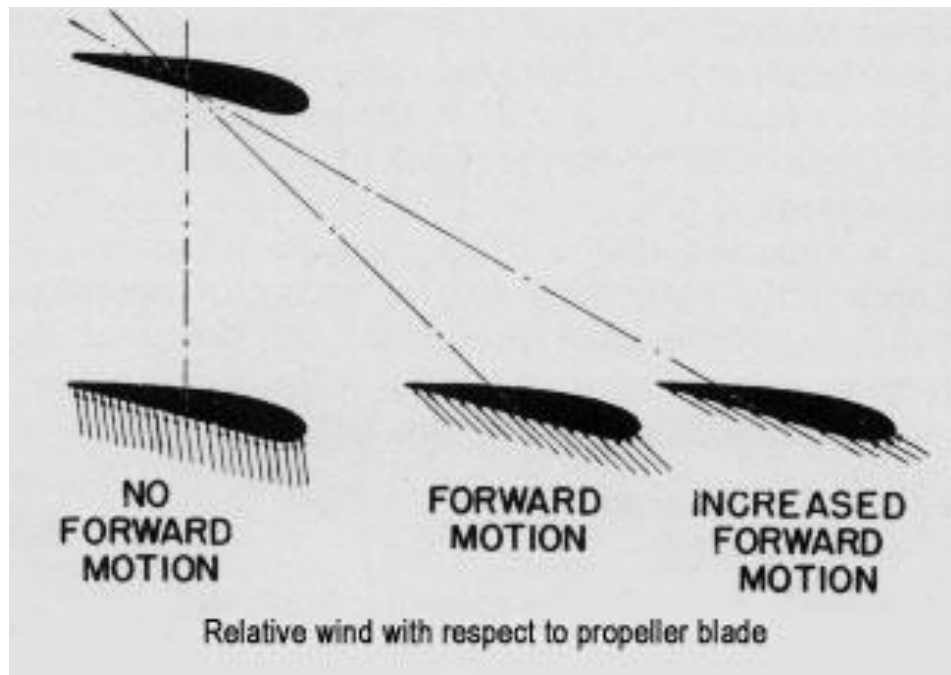


FIGURE 1.7

Blade Element is the airfoil sections joined side by side to form the blade airfoil. These elements are placed at different angles in rotation of the plane of rotation.

The reason for placing the blade element sections at different angles is because the various sections of the blade travel at different speeds. The inner part of the blade section travels slower than the outer part near the tip of the blade. If all the elements along a blade are at the same blade angle, the relative wind will not strike the elements at the same angle of attack.

This is because of the different in velocity of the blade element due to distance from the center of rotation. The blade has a small twist (due to different angle in each section) in it for a very important reason. When the propeller is spinning round, each section of the blade travel at different speed, The twist in the propeller blade means that each section advance forward at the same rate so stopping the propeller from bending. Thrust is produced by the propeller attached to the engine driveshaft. While the propeller is rotating in flight, each section of the blade has a motion that combines the forward motion of the aircraft with circular movement of the propeller. The slower the speed, the steeper the angle of attack must be to generate lift. Therefore, the shape of the propeller's airfoil (cross section) must change from the center to the tips. The changing shape of the airfoil (cross section) across the blade results in the twisting shape of the propeller.

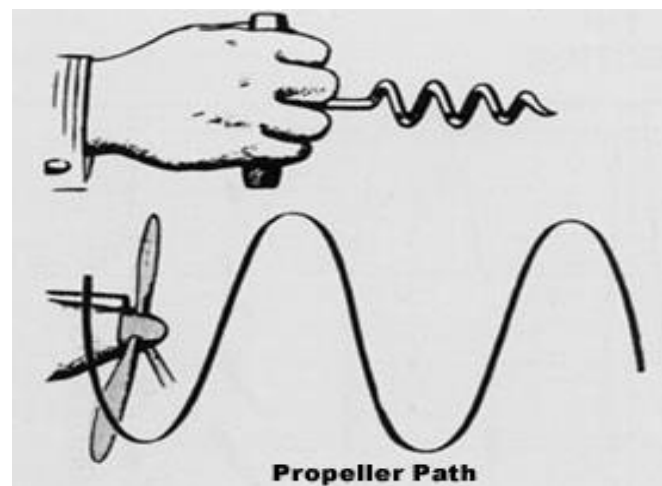


**FIGURE 1.8**

Relative Wind is the air that strikes and passes over the airfoil as the airfoil is driven through the air.

Angle of Attack is the angle between the chord of the element and the relative wind. The best efficiency of the propeller is obtained at an angle of attack around 2 to 4 degrees.

Blade Path is the path of the direction of the blade element moves.



**FIGURE 1.9**

Pitch refers to the distance a spiral threaded object moves forward in one revolution. As a wood screw moves forward when turned in wood, same with the propeller move forward when turn in the air.

Geometric Pitch is the theoretical distance a propeller would advance in one revolution.

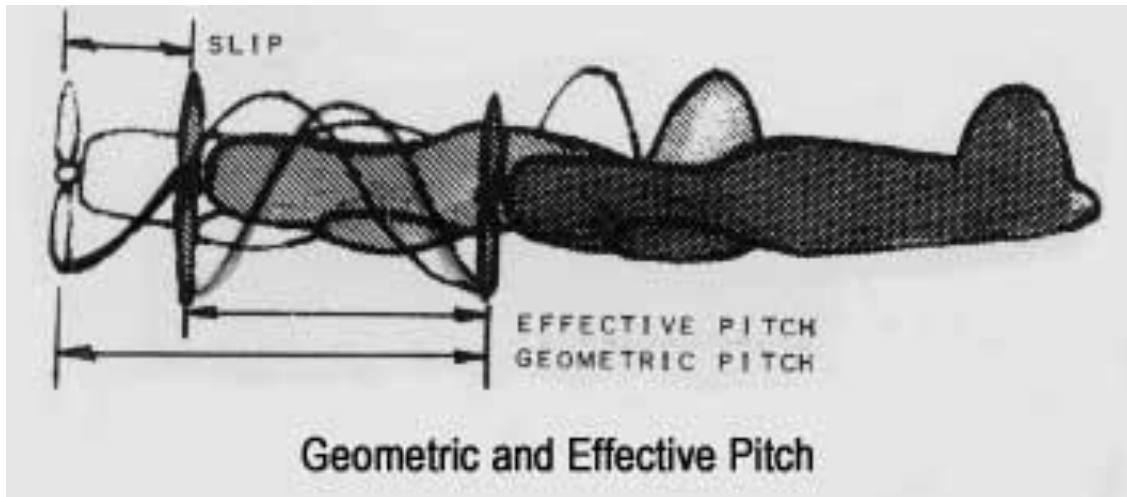


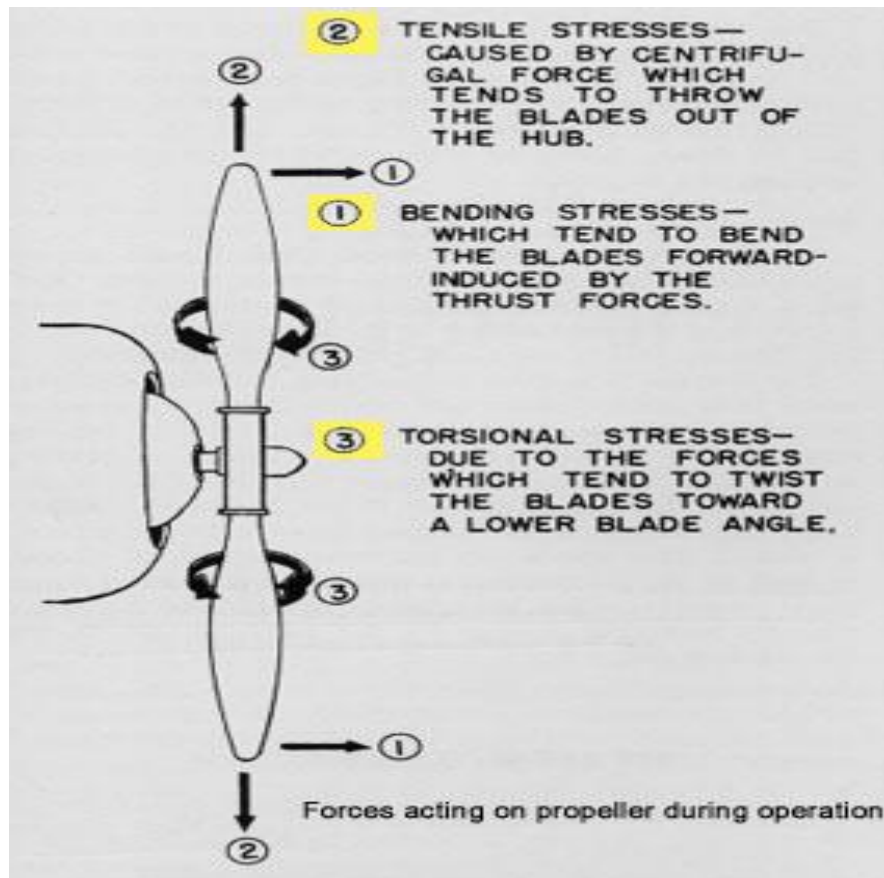
FIGURE 1.10

### 1.3 Forces and Stresses Acting on a Propeller in Flight

Effective Pitch is the actual distance a propeller advances in one revolution in the air. The effective pitch is always shorter than geometric pitch due to the air is a fluid and always slips.

The forces acting on a propeller in flight are:

1. Thrust is the air force on the propeller which is parallel to the direction of advance and induce bending stress in the propeller.
2. Centrifugal force is caused by rotation of the propeller and tends to throw the blade out from the center.
3. Torsion or Twisting forces in the blade itself, caused by the resultant of air forces which tend to twist the blades toward a lower blade angle.



**FIGURE 1.11**

The stress acting on a propeller in flight is:

1. Bending stresses are induced by the thrust forces. These stresses tend to bend the blade forward as the airplane is moved through the air by the propeller.

2. Tensile stresses are caused by centrifugal force.

3. Torsion stresses are produced in rotating propeller blades by two twisting moments. One of these stresses is caused by the air reaction on the blades and is called the aerodynamic twisting moment. Another stress is caused by centrifugal force and is called the centrifugal twisting moment.



---

## 2. LITERATURE SURVEY

**Benjamin viney et. Al.** [1] has presented the flow around the enshrouded marine propellers operating in the wake of an axisymmetric body is rotational and tridimensional. An inverse method based on the model of in viscid and rotational fluid and coupling two complementary steps (axisymmetric computation +3D panel method) is proposed for the design of the marine propellers. The maridional flow computation leads to the determination of axisymmetric stream sheets as well as the approximate camber surface of the blades and gives a good estimation of the surface of the free vertex wake.

**J.E Connolly et al** [2]: Address the problem of wide blades tried to combine both theoretical and experimental investigations, the author carried out measurements of deflection and stress on models blades subjected to simulated loads, with an aim to develop a theoretical model calibrated against the laboratory experiments, the model was then validated by measurements of pressure and stress distribution on the blade of a full-scale ship propeller at sea. based on the experimental results it was concluded that wide blades subject to tensile stress strength on the face and compression stress of similar magnitude in the back was pointed out that the accuracy of the prediction from the modal depends on the accuracy of the working load determined.

**Tergesontvcdt et al** [3] has focused on the application of finite element methods for frequency response and improve to the frozen type of hydrodynamic loading The thin shell element of the triangular type and the super parametric shell element are used in the finite element model it presents the realistic an dynamic stresses in marine propeller blades. Stresses and deformations calculated for ordinary geometry and highly skewed propellers are compared with experimental results.

**Chang suppler et al** [4] have investigated the main sources of propeller blade failures and resolved problem systematically. An FEM analysis is carried out to determine the blade strength in model and full-scale condition and range of safety factor for the propeller under study is determined.

**S.javedjalali and farid Taheri et al** [5] Carbon fiber reinforced plastics properties were taken from journal of composite materials. A new test method for the simultaneous evaluation of the longitudinal and the shear module of CFRP was introduced under the proposed method, specimens with different span to depth ratios are subjected to three point bending method. Therefore, we name the method the varying span method. The method builds on the inherent low shear modulus characteristics of CFRP. This characteristics leads to a flexural modules which is a function of L/H.

**Charles A. Harper et al** [6] Aluminum material property taken from the hand book of material and process. The non-ferrous metals and alloys offer a wide variety of physical and mechanical properties for using the many industries. Aluminum and its alloys posse's properties which find wide use in the many industries. Favorable physical properties good strength weight properties, good corrosion resistance, and low density. Combined with economy in material cost and fabrication cost, make this alloy family a basic material of construction for mechanical assemblies.

---

## 3. MATERIALS USED FOR DESIGN OF THE PROPELLER BLADE

In this project materials used for the design of propeller blade are jute and graphite.

### 3.1 JUTE:

Jute is a long, soft, shiny vegetable fiber that can be spun into coarse, strong threads. It is produced primarily from plants in the genus *Corchorus*, which was once classified with the family Tiliaceae, and more recently with Malvaceae. The primary source of the fiber is *Corchorusolitorius*, but it is considered inferior to *Corchoruscapsularis*. "Jute" is the name of the plant or fiber used to make burlap, hessian or gunny cloth.

Jute is one of the most affordable natural fibers, and second only to cotton in the amount produced and variety of uses. Jute fibers are composed primarily of the plant materials cellulose and lignin. It falls into the bast fiber category (fiber collected from bast, the phloem of the plant, sometimes called the "skin") along with kenaf, industrial hemp, flax (linen), ramie, etc. The industrial term for jute fiber is raw jute. The fibers are off-white to brown, and 1–4 meters (3–13 feet) long. Jute is also called the golden fiber for its color and high cash value.

### 3.2 Features of jute:

- Jute fiber is 100% bio-degradable and recyclable and thus environmentally friendly.
- Jute has low pesticide and fertilizer needs.
- It is a natural fiber with golden and silky shine and hence called The Golden Fiber.
- It is the cheapest vegetable fiber procured from the bast or skin of the plant's stem.

- It is the second most important vegetable fiber after cotton, in terms of usage, global consumption, production, and availability.
- It has high tensile strength, low extensibility, and ensures better breathability of fabrics. Therefore, jute is very suitable in agricultural commodity bulk packaging.
- It helps to make top quality industrial yarn, fabric, net, and sacks. It is one of the most versatile natural fibers that has been used in raw materials for packaging, textiles, non-textile, construction, and agricultural sectors. Bulking of yarn results in a reduced breaking tenacity and an increased breaking extensibility when blended as a ternary blend.
- The best source of jute in the world is the Bengal Delta Plain in the Ganges Delta, most of which is within the borders of Bangladesh.
- Advantages of jute include good insulating and antistatic properties, as well as having low thermal conductivity and a moderate moisture regain. Other advantages of jute include acoustic insulating properties and manufacture with no skin irritations.
- Jute can be blended with other fibers, both synthetic and natural, and accepts cellulosic dye classes such as natural, basic, vat, sulfur, reactive, and pigment dyes. As demand for natural comfort fibers increases, demand for jute and other natural fibers that can be blended with cotton will increase. To meet this demand, some manufacturers in the natural fiber industry plan to modernize processing with the Rieter's Elitex system. Resulting jute/cotton yarns produce fabrics with a reduced cost of wet processing treatments. Jute can also be blended with wool. By treating jute with caustic soda, crimp, softness, pliability, and appearance is improved, aiding in its ability to be spun with wool. Liquid ammonia has a similar effect on jute, as well as the added characteristic of improving flame resistance when treated with flame proofing agents.
- Some noted disadvantages include poor drapability and crease resistance, brittleness, fiber shedding, and yellowing in sunlight. However, preparation of fabrics with castor oil lubricants result in less yellowing and less fabric weight loss, as well as increased dyeing brilliance. Jute has a decreased strength when wet, and also becomes subject to microbial attack in humid climates. Jute can be processed with an enzyme to reduce some of its brittleness and stiffness. Once treated with an enzyme, jute shows an affinity to readily accept natural dyes, which can be made from marigold flower extract. In one attempt to dye jute fabric with this extract, bleached fabric was mordanted with ferrous sulphate, increasing the fabric's dye uptake value. Jute also responds well to reactive dyeing. This process is used for bright and fast colored value-added diversified products made from jute.

### 3.3 PROPERTIES OF JUTE:

PROPERTY	VALUE
Density	1.4g/cm <sup>3</sup>
Young's Modulus(MPa)	30000
Poisson's ratio	0.34

**Table 3.1: Properties of jute**

### 3.4 GRAPHITE:

Graphite archaically referred to as plumbago, is a crystalline form of the element carbon with its atoms arranged in a hexagonal structure. It occurs naturally in this form and is the most stable form of carbon under standard conditions. Under high pressures and temperatures it converts to diamond. Graphite is used in pencils and lubricants. Its high conductivity makes it useful in electronic products such as electrodes, batteries, and solar panels.

### 3.5 USES:

- Graphite (carbon) fiber and carbon nanotubes are also used in carbon fiber reinforced plastics, and in heat-resistant composites such as reinforced carbon-carbon (RCC). Commercial structures made from carbon fiber graphite composites include fishing rods, golf club shafts, bicycle frames, sports car body panels, the fuselage of the Boeing 787 Dream liner and pool cue sticks and have been successfully employed in reinforced concrete. The mechanical properties of carbon fiber graphite-reinforced plastic composites and grey cast iron are strongly influenced by the role of graphite in these materials. In this context, the term "(100%) graphite" is often loosely used to refer to a pure mixture of carbon reinforcement and resin, while the term "composite" is used for composite materials with additional ingredients.

- Modern smokeless powder is coated in graphite to prevent the buildup of static charge.
- Graphite has been used in at least three radar absorbent materials. It was mixed with rubber in Sumpf and Schornsteinfeger, which were used on U-boat snorkels to reduce their radar cross section. It was also used in tiles on early F-117 Nighthawk stealth strike fighters.
- Graphite composites are used as absorber for high-energy particles (e.g. in the LHC beam dump).

### 3.6 PROPERTIES OF GRAPHITE:

PROPERTY	VALUE
Density	1.65g/cm <sup>3</sup>
Young's Modulus(MPa)	10000
Poisson's ratio	0.2

**Table 3.2: Properties of Graphite**

## 4. MODELING OF AIRCRAFT PROPELLER BLADE USING CREO

### 4.1 INTRODUCTION TO CREO:

CREO is a powerful software which is a combination of various designing tools. It is popularly used by many leading manufacturing companies across the globe. It was designed by PTC (Parametric Technology Corporation). CREO is one of the most popular and powerful designing tool used in the market. With the popularity of CREO the demand of CREO certification has also increased in the market.

There are various versions of CREO. The most updates version of CREO is CREO 4.0 and CREO certification with this version is sought in the market. Even with this increased demand, many design engineers still do not have complete understanding about this tool. Hence this blog will be focused on delivering the basic information about CREO. CREO is the updated version of Pro-E software. Every CREO application serves unique purpose of product development. Hence one CREO software can handle every aspect of product design like concept development, designing and analysis. It also supports your communication with your clients, manufacturers or technical publication.

CREO works with combination of various applications like

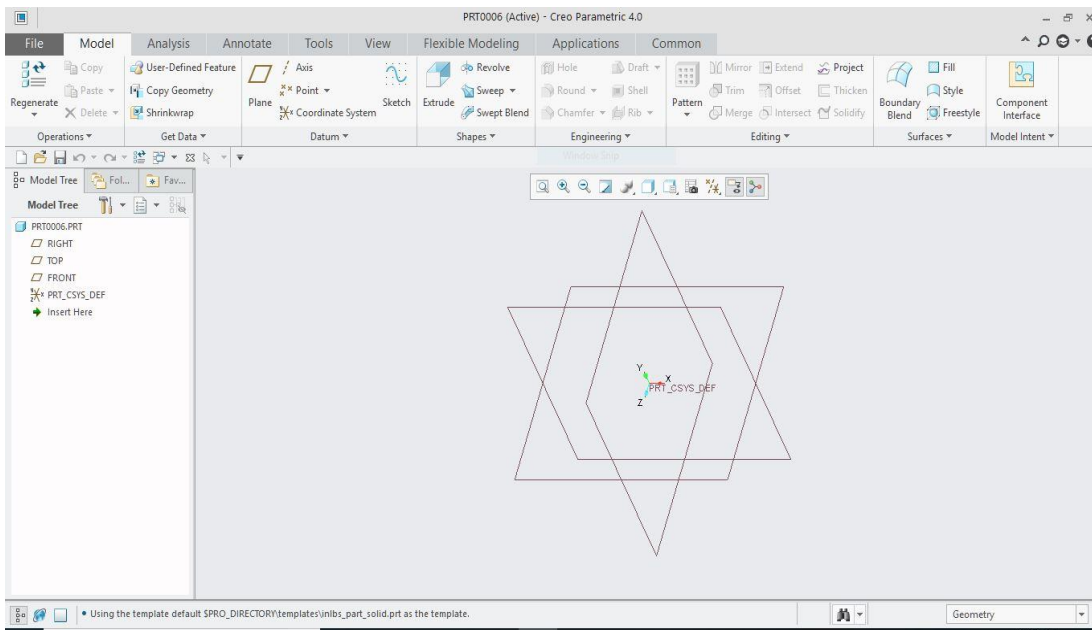
- Creo Parametric
- Creo Simulate
- Creo Direct
- Creo Layout
- Creo Options Modeler

### 4.2 FEATURES OF CREO:

CREO works on various stages of product development. Hence CREO is useful in every department of the company. Adding to it, there are four prominent features which defines the usability of this software. They are

1. **Flexibility:** CREO offers scalable access to designers who are involved in any part of product development. On the other hand, specific CREO applications help developers who are a part of only one particular process. So CREO can also be customized to serve single process and also be used, with the same efficiency, to serve the needs of the entire process.

2. **Interoperability:** As every CREO application is designed under the same roof, the communication between every application is very smooth. There are no data lags when any application is interacting with other application, which saves ample amount of designers time. Hence you can pass the work from one process to other and the same designed will be modified to be made perfect.
3. **Combined Benefits Of Both CAD Modeling Approach:** Working on CREO means using both the CAD modeling approaches i.e. parametric and direct modeling. Hence you achieve two goals with a single software. Designers can enjoy the control provides by parametric modeling on one hand and on the other hand, they can also enjoy the speed and flexibility of direct modeling.
4. **Operating On Multi-CAD Data:** CREO can easily work on any CAD data source. Hence designers can save their lot of time and efforts while using CREO and operating on various platforms. It also eliminates the minor possibility of human errors while redesigning the same design. Thus it is helpful for both the designers and the organization



**FIGURE 4.1: HOME SCREEN OF CREO PARAMETRIC 4.0**

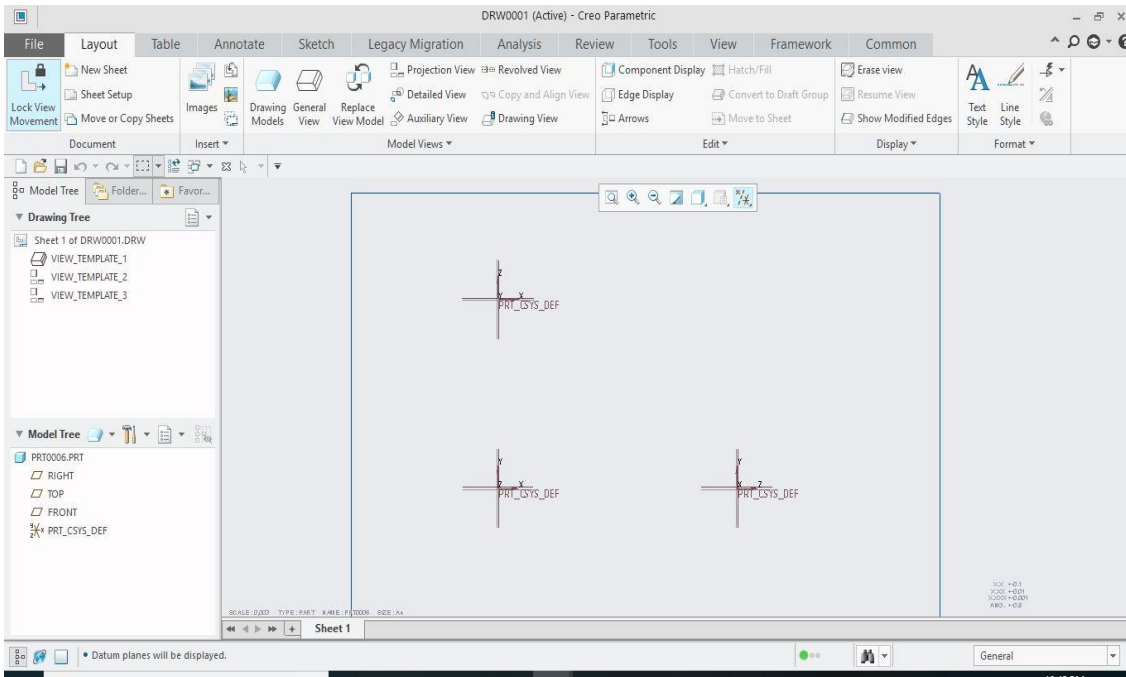


FIGURE 4.2: DRAFTING SCREEN OF CREO PARAMETRIC 4.0

#### 4.3 AIRCRAFT PROPELLER BLADE DESIGN:

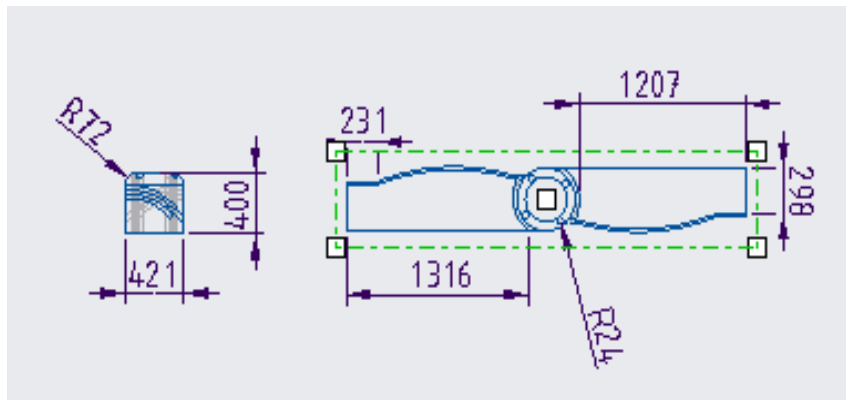
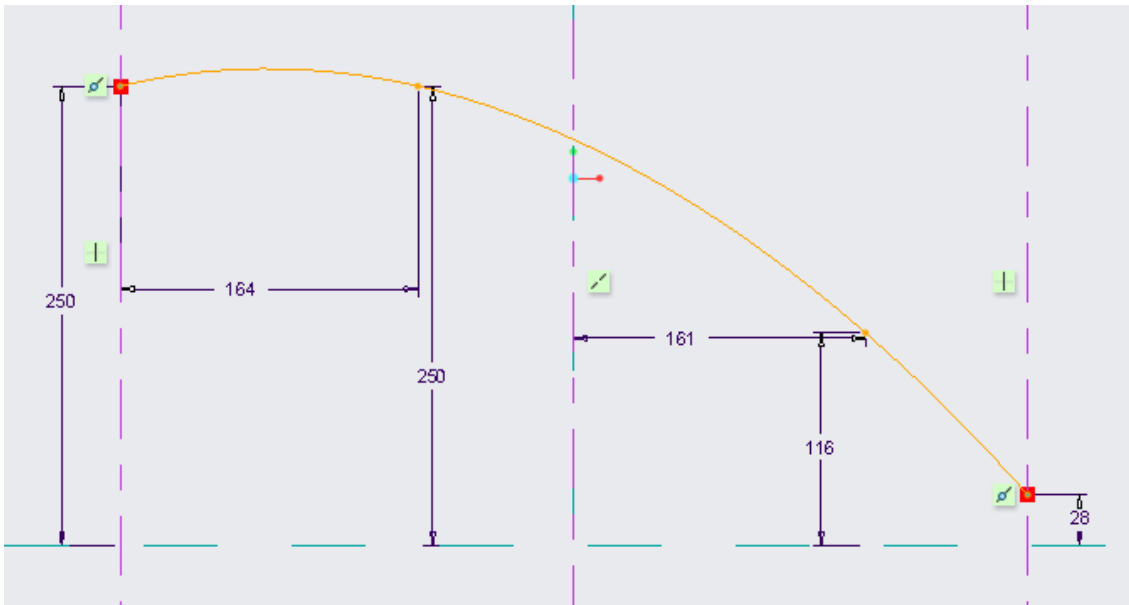
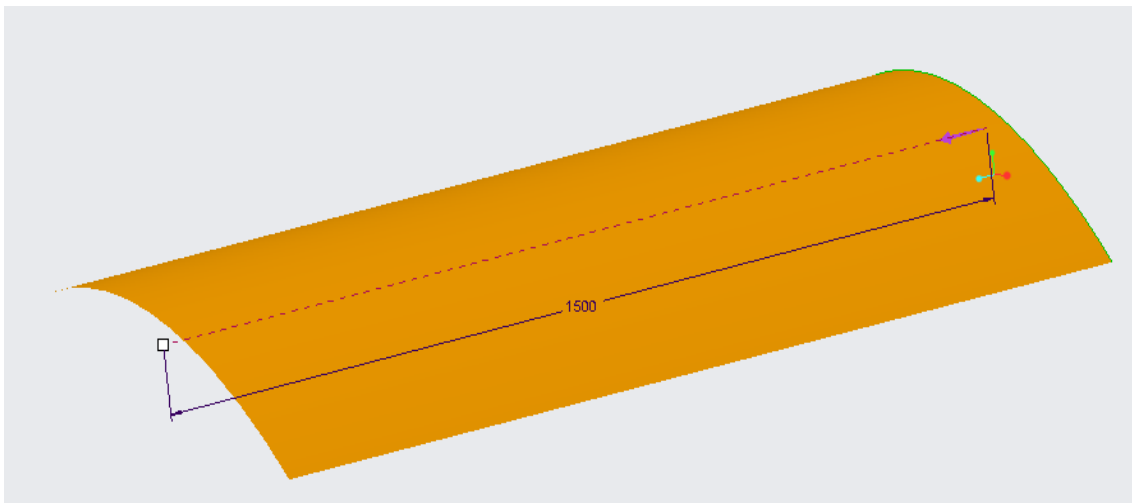


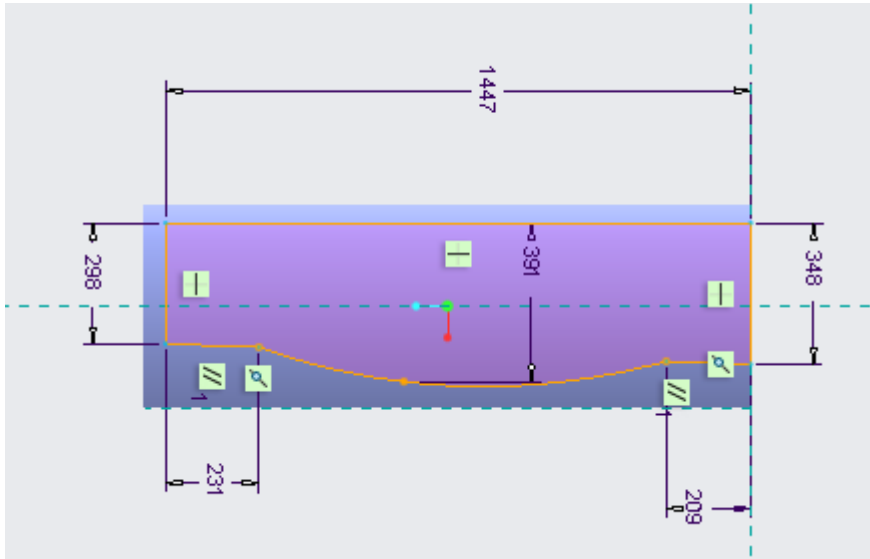
FIGURE 4.3: DIMENSIONS OF AIRCRAFT PROPELLER BLADE

#### 4.4 STEPS FOR DESIGNING THE AIRCRAFT PROPELLER BLADE:

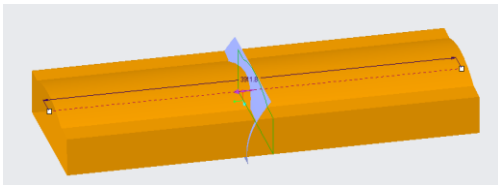
**STEP 1:** First double click on the PTC CREO PARAMETRIC 4.0 icon located on the desktop, then click on the FILE menu and take a NEW part file. Then go to MODEL TREE, in which FRONT VIEW is selected and then click on EXTRUDE it turns in to 2D format and then place the centre lines and draw the curve with spline and extrude it as surface.

**FIGURE 4.4: SPLINE SKETCH****FIGURE 4.5: SURFACE EXTRUDE**

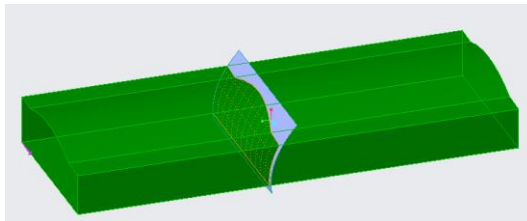
**STEP 2:** Take sketch and select the surface and draw a shape as shown and extrude it symmetrically, take trim option and trim the material and keep the shape as it is and apply thickness and pattern it with a angle of 180degrees.



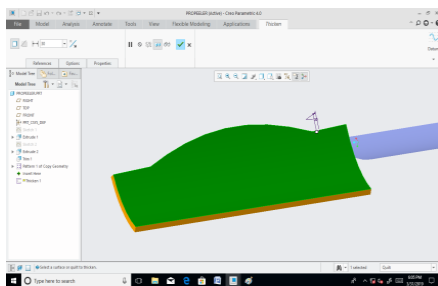
**FIGURE 4.6: DIMENSIONS OF BLADE**



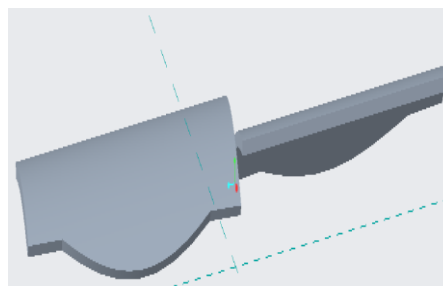
**FIGURE 4.7: EXTRUDED PART**



**FIGURE 4.8: TRIM**

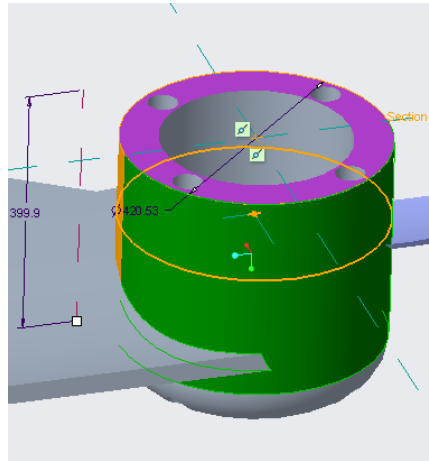


**FIGURE 4.9: THICKNESS**

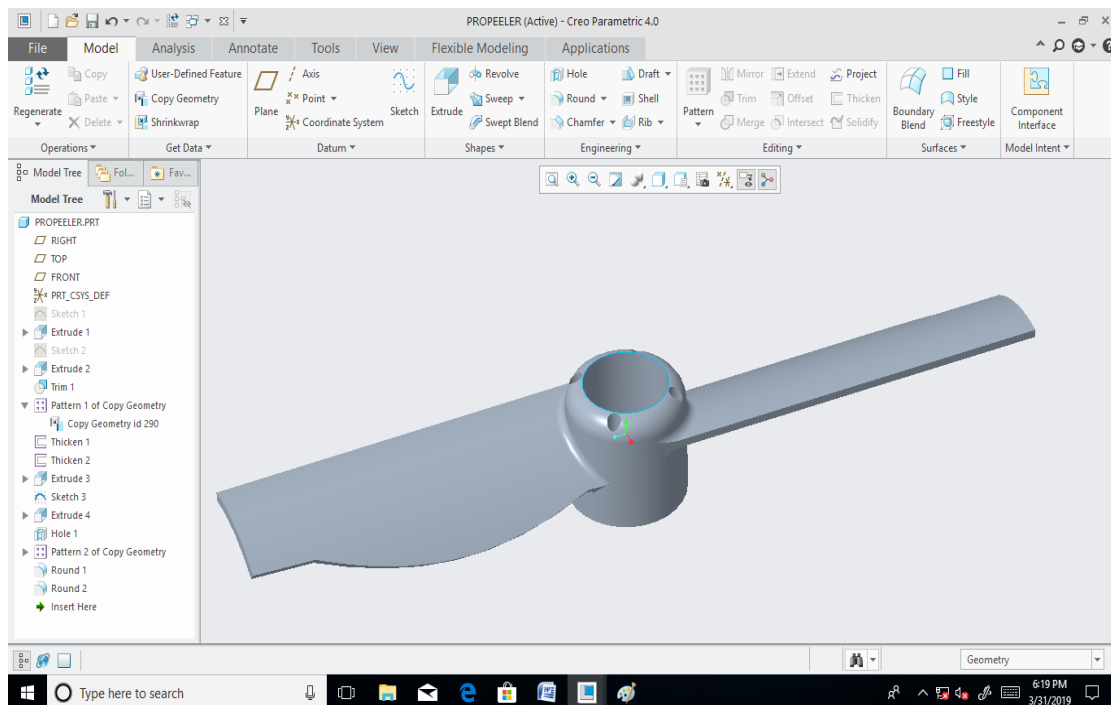


**FIGURE 4.10: PATTERN**

**STEP 3:** Draw the hub and make on the top surface and pattern it into 4 holes



**FIGURE 4.11: HUB WITH HOLES**



**FIGURE 4.12: FINAL DESIGN OF AIRCRAFT PROPELLER BLADE**

## 5. FINITE ELEMENT ANALYSIS

### 5.1 INTRODUCTION TO FEA:

The finite element analysis was the first developed in 1943 by R. Courant (Mathematician), who utilized the Ritz method on numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 M.J. Turner (air craft industry), R.W. Clough (California University), H.C. Martin (air craft industry) and L.J. Top established a broader definition of numerical analysis. The paper entered on the stiffness and deflection of complex structures.

Finite element analysis is a numerical technique in which a complex geometric problem is broken up into a system of more manageable problems. The process of breaking up the problems base on geometry. The geometric complexity of the original problem is such that a solution cannot be obtained using conventional methods. By breaking the shape up in to small shapes that can be solved, a system of equation can be established. Finite element analysis is a general technique that can be applied to a number of phenomena.

We will be using stress and displacement as the phenomena of interest however, other problems such as heat transfer can also be solved using FEA. By the early 70's FEA was limited to expensive main frame computers generally owned by the aeronautics, automotive, defense



and nuclear industries. Since the rapid decline the cost of computers and phenomenal increase in computer power, FEA has been developed to an incredible precision present day super computers are now able to produce accurate result for all kind of parameters. FEA is numerical method. As with any numerical methods, result must be validated.

## 5.2 INTRODUCTION TO ANSYS SOFTWARE:

Ansys was founded in 1970 by John Swanson. Swanson sold his interest in the company to venture capitalists in 1993. Ansys went public on NASDAQ in 1996. In the 2000s, Ansys made numerous acquisitions of other engineering design companies, acquiring additional technology for fluid dynamics, electronics design, and other physics analysis.

It develops and markets engineering simulation software. Ansys software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties. 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.

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.

## 5.3 HISTORY:

The first commercial version of Ansys software was labeled version 2.0 and released in 1971. At the time, the software was made up of boxes of punch cards, and the program was typically run overnight to get results the following morning. In 1975, non-linear and thermo-electric features were added. The software was exclusively used on mainframes, until version 3.0 (the second release) was introduced for the VAX station in 1979. Version 3 had a command line interface like DOS.

In 1980, Apple II was released, allowing Ansys to convert to a graphical user interface in version 4 later that year. Version 4 of the Ansys software was easier to use and added features to simulate electromagnetism. In 1989, Ansys began working with CompuFlo. CompuFlo's Flotran fluid dynamics software was integrated into Ansys by version 5, which was released in 1993. Performance improvements in version 5.1 shortened processing time two to four-fold, and was followed by a series of performance improvements to keep pace with advancements in computing. Ansys also began integrating its software with CAD software, such as Autodesk.

In 1996, Ansys released the Design Space structural analysis software, the LS-DYNA crash and drop test simulation product, and the Ansys Computational Fluid Dynamics (CFD) simulator. Ansys also added parallel processing support for PCs with multiple processors. The educational product Ansys was introduced in 1998. Version 6.0 of the main Ansys product was released in December 2001. Version 6.0 made large-scale modeling practical for the first time, but many users were frustrated by a new blue user interface. The interface was redone a few months later in 6.1. Version 8.0 introduced the Ansys multi-field solver, which allows users to simulate how multiple physics problems would interact with one another. Version 8.0 was published in 2005 and introduced Ansys' fluid-structure interaction software, which simulates the effect structures and fluids have on one another. Ansys also released its Probabilistic Design System and Design Explorer software products, which both deal with probabilities and randomness of physical elements. In 2009 version 12 was released with an overhauled second version of Workbench. Ansys also began increasingly consolidating features into the Workbench software.

Version 15 of Ansys was released in 2014. It added a new feature for composites, bolted connections, and better meshes tools. In February 2015, version 16 introduced the AIM physics engine and Electronics Desktop, which is for semiconductor design. The following year, version 17 introduced a new user interface and performance improvement for computing fluid dynamics problems. In January 2017, Ansys released version 18. Version 18 allowed users to collect real-world data from products and then incorporate that data into future simulations. The Ansys Application Builder, which allows engineers to build, use, and sell custom engineering tools, was also introduced with version 18.

Here for the design of Aircraft propeller blade we had used ANSYS WORKBENCH 15.0 for modal analysis

**5.4 PROCEDURE FOR ANALYSIS:** Static analysis is used to determine the displacements stresses, strains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity impose (non-zero) displacements, temperatures (for thermal strain). A static analysis can be either linear or non linear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps

- Building the model
- Obtaining the solution

- Reviewing the results

#### 5.4.1 BUILD OR IMPORT THE MODEL:

In this step we specify the job name and analysis title use PREP7 to define the element types, real constants, material properties and model geometry element type both linear and non linear structural elements are allowed. The ANSYS elements library contains over 80 different element types. A unique number and prefix identify each element type.

Eg. BEAM 94, PLANE711, SOLID 96, and PIPE 16

#### 5.4.2 MATERIAL PROPERTIES:

Young's Modulus (EX) must be defined for a static analysis. If we plan to apply inertia loads (such as gravity) we define mass properties such as density (DENS). Similarly, if we plan to apply thermal loads (temperatures) we define coefficient of thermal expansion (APLX).

#### 5.4.3 SOLUTION:

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases:

- Pre-processor phase
- Solution phase
- Post-processor phase

**Pre-processor phase:** Preprocessor has been developed so that the same program is available on micro, mini, super mini and mainframe computer system. This allows easy transfer of models one system to other.

The following algorithm shows the brief description of steps followed in each phase:

Preprocessor is an interactive model builder to prepare the FE (Finite element) model and input data. The solution phase utilizes the input data developed by the processor, and prepares the solution according to the problem definition. It creates input files to the temperature etc. on the screen in the form of contours.

- ✓ **Geometrical definitions:** There are four different geometric entities in preprocessor namely key points, lines, area and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.
- ✓ **Model generations:** Two different methods are used to generate a model:
  - Direct generation
  - Solid modeling

With solid modeling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then insert ANSYS program to generate all the nodes and elements automatically. By contrast, with the direct generation method, we determine the location of every node and size shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc) the direct generation method essentially a hands on numerical method that require us to keep track of all the node numbers as we develop the finite element mesh. This detailed book keeping can become different for large models, giving scope for modeling errors. Solid modeling is usually more powerful and versatile than direct generation and is commonly preferred method of generating model.

- **Mesh generation:** In the finite element analysis, the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as Mesh.
- **Finite element generation:** The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Preprocessor allows the user to generate nodes and elements and automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities. The elements developed by various automatic element generation capabilities of preprocessor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion index etc. Generally, and automatic mesh generating capabilities of preprocessor are use rather than defining the nodes individually. if require nodes can be defined easily by defining the allocations or by translating the existing nodes. Also one can plot, delete, or search nodes.
- **Boundary conditions and loading:** after completion of the finite element model it has to constrain an load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are designed set ID. This helps user to keep track of load cases.

- **Model display:** During the construction and verification stages of the model it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Preprocessor offer these capabilities.
- **Model deflections:** All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements there is no indication of thickness. This thickness can be given as element property. Property tables for a particular property set 1-D have to be input. Different types of elements have different properties. For e.g. Beams: Cross sectional area, moment of inertia et.

**SOLUTION:** The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements are stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non linear static analysis, transient dynamic analysis, etc.

**POST-PROCESSOR:** It is a powerful user friendly post processing program using interactive colour graphics. It has extensive plotting features for displaying the results obtained from the finite element analysis. One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds what would take an engineer hour to assess from a numerical output, say in tabular form. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data.

Employing state of art image enhancement techniques, facilities viewing of:

- Contours of stresses, displacements, temperatures, etc.
- Deform geometric plots
- Animated deform shapes
- Time-history plots
- Solid sectioning
- Hidden line plot
- Light source shaded plot
- Boundary line plot

The entire range of post processing options of different types of analysis can be accessed through the command/menu mode there by giving the user added flexibility and convenience.

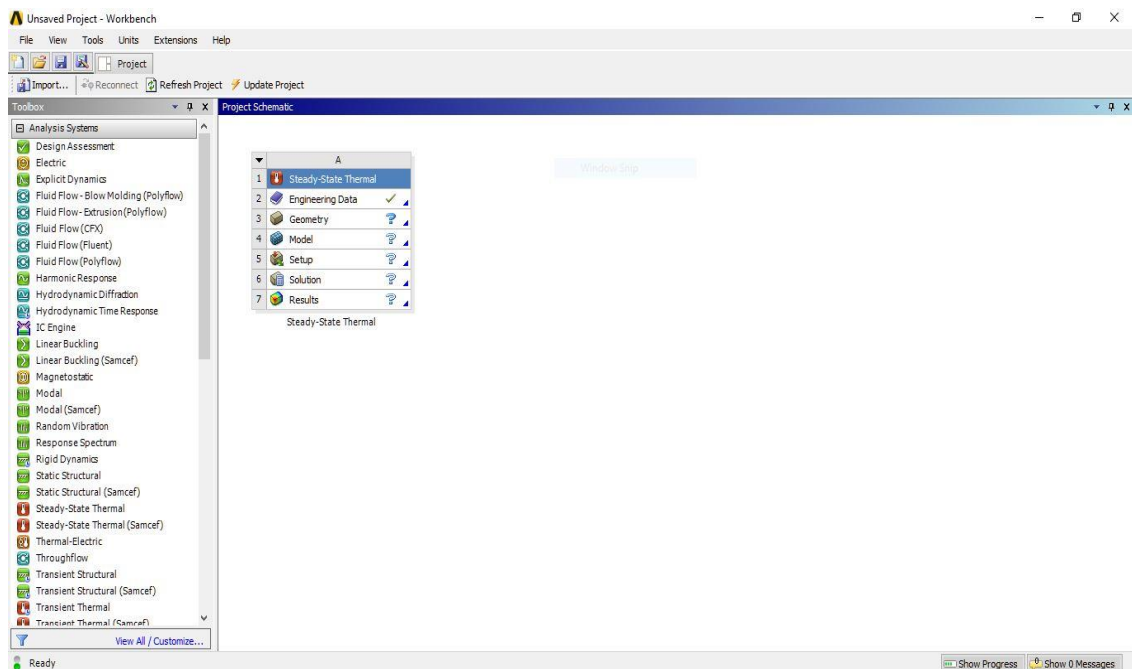


FIGURE 5.1: ANSYS WORKBENCH 15.0 HOME SCREEN

## 5.5 ANSYS WORKBENCH

The following types of cells are available in ANSYS Workbench how you work with them is explained below.

- Engineering Data
- Geometry
- Model/Mesh
- Setup
- Solution
- Results

**ENGINEERING DATA:** Use the engineering data cell with mechanical systems or the engineering data component system to define or access material models for use in an analysis. Double-click the engineering data cell or right-mouse click and choose Edit from the context menu to display the engineering data workspace to define material data.

**GEOMETRY:** Use the geometry cell to import, create, edit or update the geometry model used for analysis. Right-mouse clicks to display the context menu to access these functions. The right mouse button options are context sensitive and will change as the state of your geometry changes. All geometry-specific options are described here; not all will be available at all times.

**New Geometry:** Launches Design Modeler, where you can build a new geometry.

**Import Geometry:** Select Browse to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files.

**Edit:** After you have attached geometry to your system by choosing either New Geometry or Import Geometry, click Edit to open the model in Design Modeler to modify it.

**Replace Geometry:** Select Browse to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files to replace the currently specified file.

**Update from CAD:** Generates an existing CAD geometry using the parameter values as defined in the CAD system.

**Refresh:** Reads in all modified upstream data but does not regenerate the geometry. Enabled when the geometry cell is in the refresh required state.

**Properties:** Displays a properties pane where you can select basic and advanced geometry properties. For a detailed description of the options available from the properties pane, see preferences in the mechanical application help.

**MODEL/MESH:** The model cell in the mechanical application analysis systems or the mechanical model component system is associated with the model branch in the mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition. When linking two systems, you cannot create a share between the model cells of two established systems. You can generate a second system that is linked at the model cell of the first system, but you cannot add a share after the second system has been created. Likewise, you cannot delete a link between the model cells of two systems. The mesh cell in fluid flow analysis systems or the mesh component system is used to create a mesh using the meshing application. It can also be used to import an existing mesh file.

**Edit:** Launches the appropriate model or mesh application (the mechanical application, meshing, etc).

**SETUP:** Use the setup cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS Workbench, including connections between systems. For the mechanical application systems, you will see the following setup options, in addition to the common options

**Edit:** Launches the mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the mechanical application.

**Import Case:** Imports an existing case file containing physics data, region and mesh information for your analysis.

**SOLUTION:** From the solution cell, you can access the solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, you will see the solution cell in a pending state until the remote process completes. For the mechanical application systems, you will see the following setup options, in addition to the common options described earlier:

**Edit:** Launches the mechanical application open to the solution branch

**Delete:** Deletes the Solution and results cell. Deleting the solution cell makes the system a setup-only system, meaning the system will generate only an input file. It will not solve or post results. The solution object and below are removed from the mechanical application tree.

**RESULTS:** The results cell indicates the availability and status of the analysis results (commonly referred to as post-processing). From the results cell, you cannot share data with any other system.

## 6. MODAL ANALYSIS

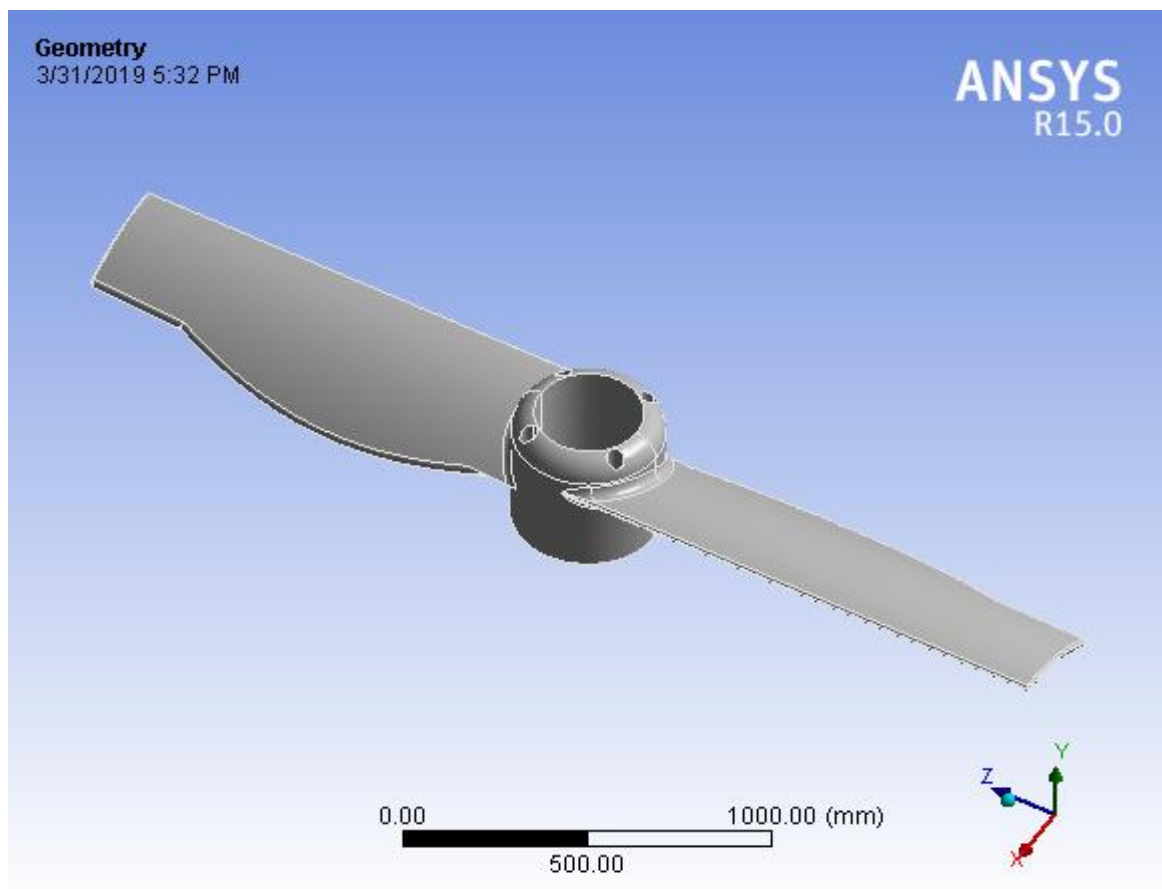
- Modal analysis is the study of the dynamic properties of systems in the frequency domain.
- A modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic analysis, or a spectrum analysis.
- The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration
- . It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analyzed can have arbitrary shape and the results of the calculations are acceptable.
- The types of equations which arise from modal analysis are those seen in eigen systems. The physical interpretation of the Eigen values and eigenvectors which come from solving the system are that they represent the frequencies and corresponding mode shapes. Sometimes, the only desired modes are the lowest frequencies because they can be the most prominent modes at which the object will vibrate, dominating all the higher frequency modes.
- It is also possible to test a physical object to determine its natural frequencies and mode shapes. This is called an Experimental Modal Analysis. The results of the physical test can be used to calibrate a finite element model to determine if the underlying assumptions made were correct (for example, correct material properties and boundary conditions were used).
- Here, for designing the Aircraft propeller blade we had gone with the modal analysis.

### 6.2 PROCEDURE FOR MODALANALYSIS

- First double click on the ANSYS15.0 icon located on the desk top. You will get a tool box opened with types analysis, from we should select modal and then click on it.
- You will be directed to the project schematic window in which it consists of engineering data, geometry, model, setup & solution.
- Right click on the engineering data in which you will be having an edit option in which you can give values to the properties of the materials. Properties like Young's Modulus, Density and poisson's ratio are inserted.
- After that, right click on the geometry in which you will be getting import geometry. By clicking on it, we can import the model which we have designed in the PTC CREO PARAMETRIC which is saved in the form of iges.
- Eg. Aircraft propeller blade.iges
- 5.Click on model and edit it by doing meshing to the model. Meshing is done as follows
  - Sizing → Use advance size function
  - Size function → On proximity & curvature
  - Relevance center → Fine
  - Initial size seed → Active assembly
  - Smoothing → High
  - Transition → Fast
  - Span angle center → Fine
- By, clicking on the update the data given to the model will updated.
- Click on set up to give boundary conditions to the model, i.e Fixed support.
- Click solve to find out the frequency for the modes
- Select all the frequency values and right click on it and select create mode shape results. thus the results are generated

**MECHANICAL PROPERTIES OF THE MATERIALS:**

PROPERTY	Graphite	Jute
Density	1.65g/cm <sup>3</sup>	1.4g/cm <sup>3</sup>
Young's modulus	10000Mpa	30000Mpa
Poison's ratio	0.2	0.34

**Table 6.1: Mechanical properties of the materials****MODAL BOUNDARY CONDITIONS:****FIGURE 6.1: IGES file**

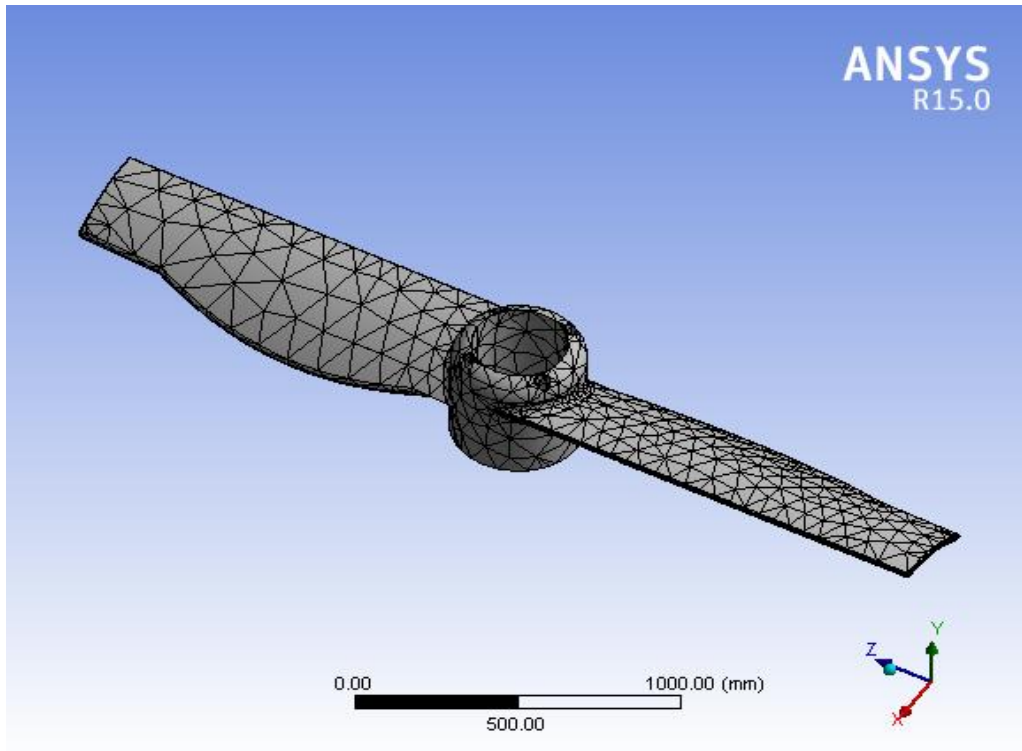


FIGURE 6.2: MESHED MODEL

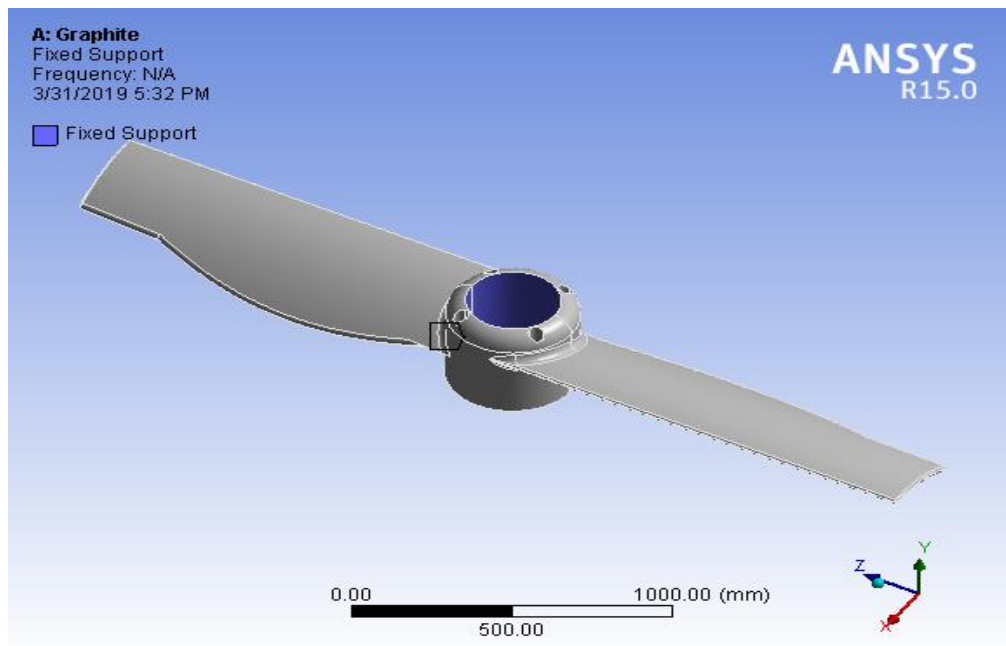


FIGURE 6.3: FIXED SUPPORT

---

## 7. MODAL ANALYSIS

### 7.1.1 DEFORMATION AT DIFFERENT MODES:

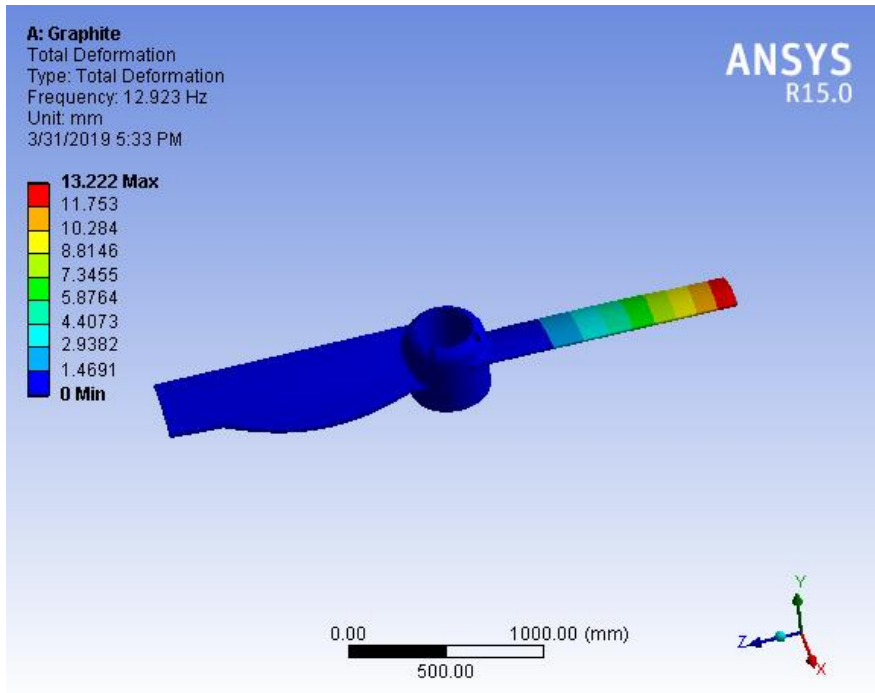


FIGURE 7.1: GRAPHITE AT MODE 1

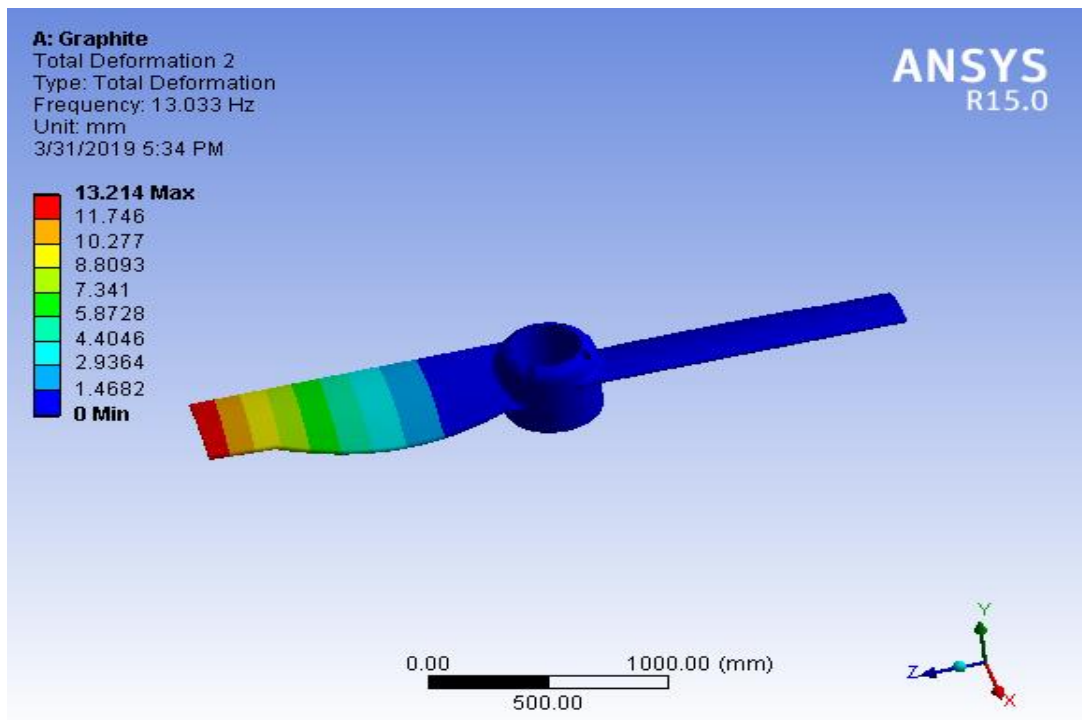


FIGURE 7.2: GRAPHITE AT MODE 2



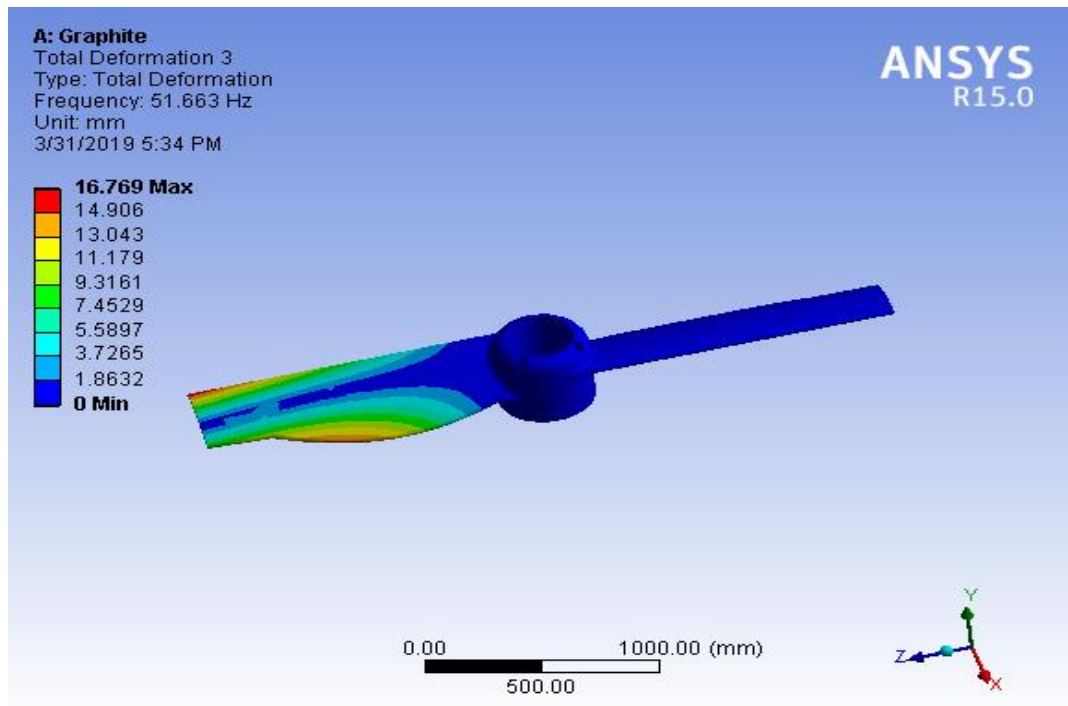


FIGURE 7.3: GRAPHITE AT MODE 3

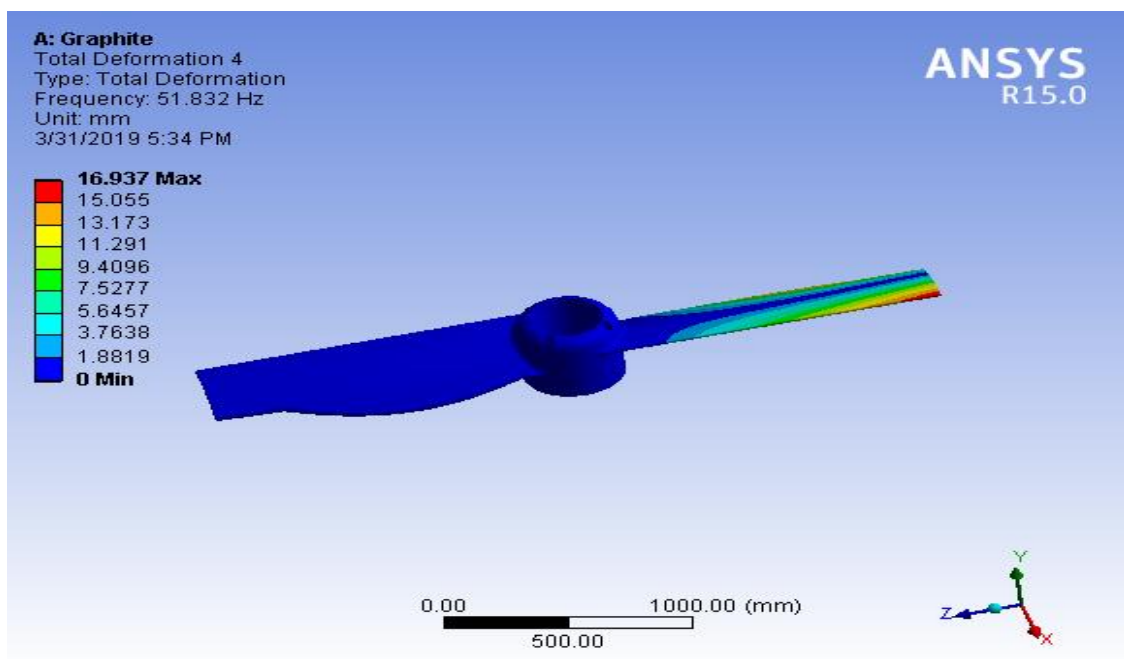


FIGURE 7.4: GRAPHITE AT MODE 4

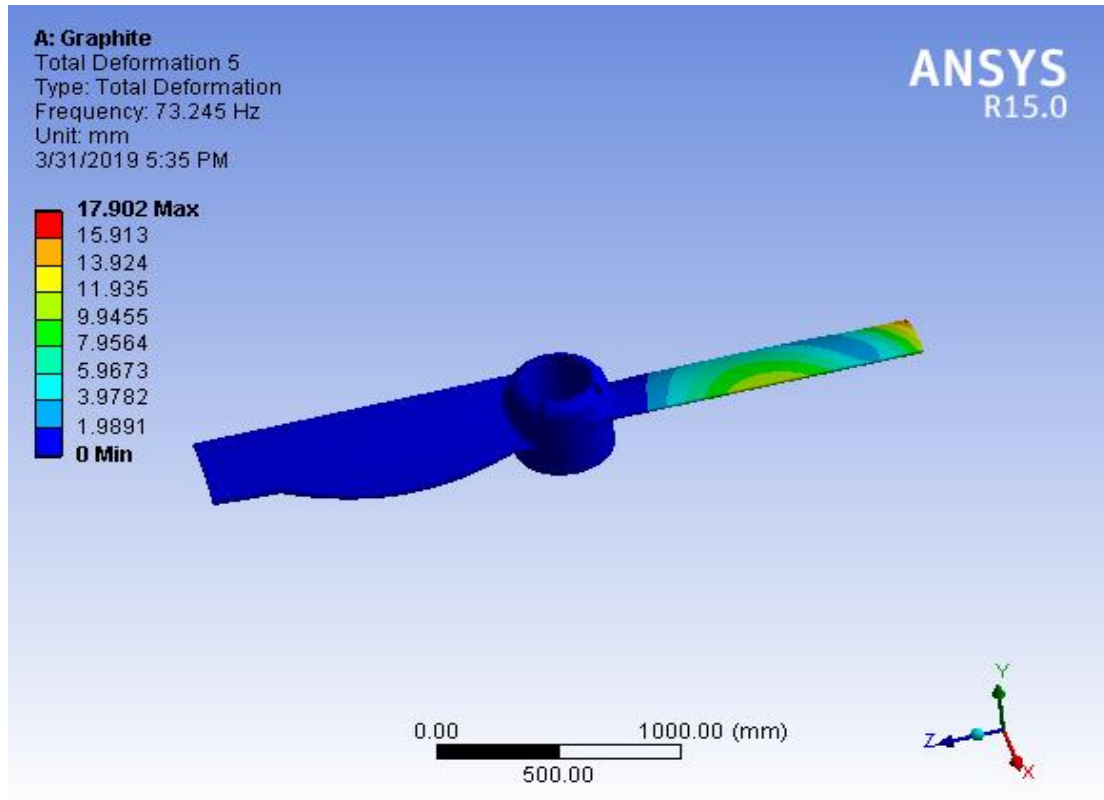


FIGURE 7.5: GRAPHITE AT MODE 5

RESULTS FOR STRUCTURAL ANALYSIS:

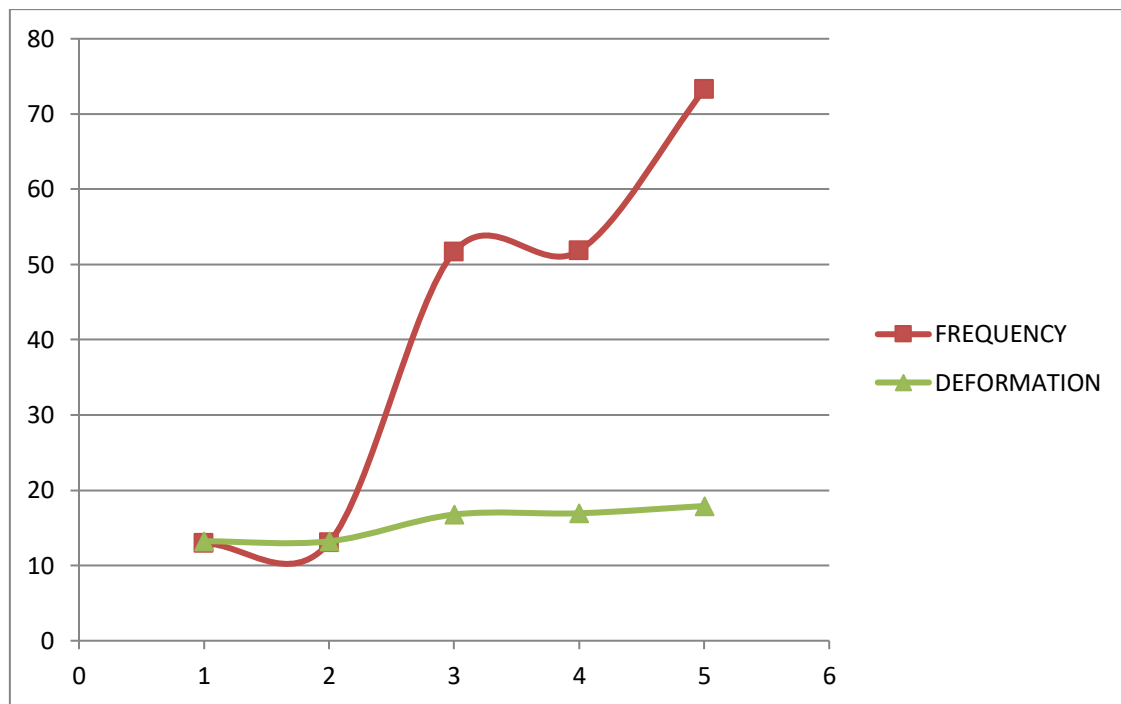
MATERIAL	MODE	FREQUENCY(HZ)	TOTAL DEFORMATION (mm)
GRAPHITE	1	12.923	13.222
	2	13.033	13.214
	3	51.663	16.769
	4	51.832	16.937
	5	73.245	17.902

Table 8.1: Results for Structural analysis

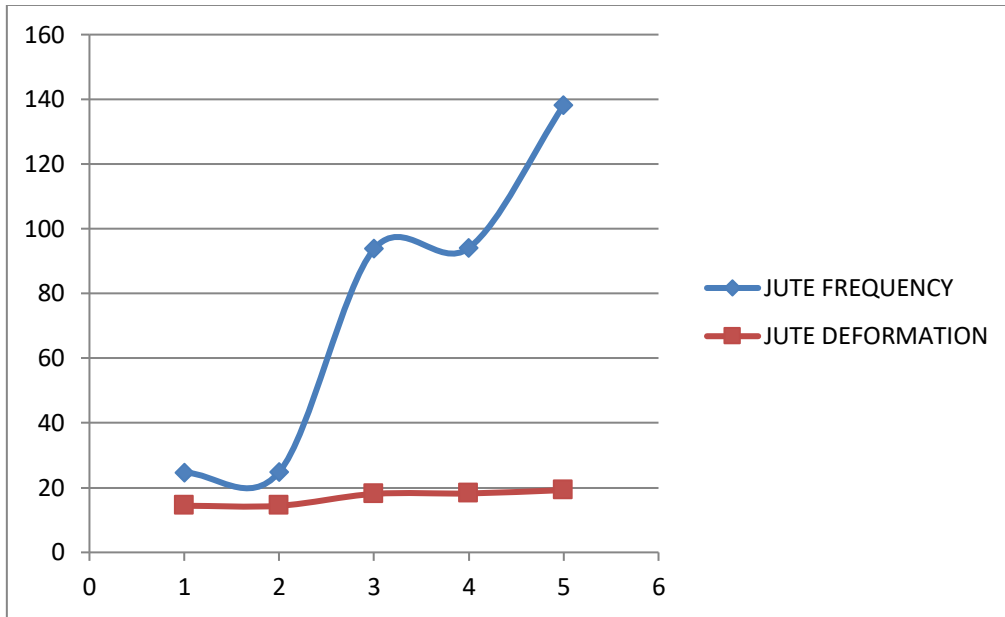
RESULTS FOR STRUCTURAL ANALYSIS:

MATERIAL	MODE	FREQUENCY(HZ)	TOTAL DEFORMATION (mm)
JUTE	1	24.616	14.41
	2	24.816	14.405
	3	93.704	18.068
	4	94.067	18.212
	5	138.16	19.262

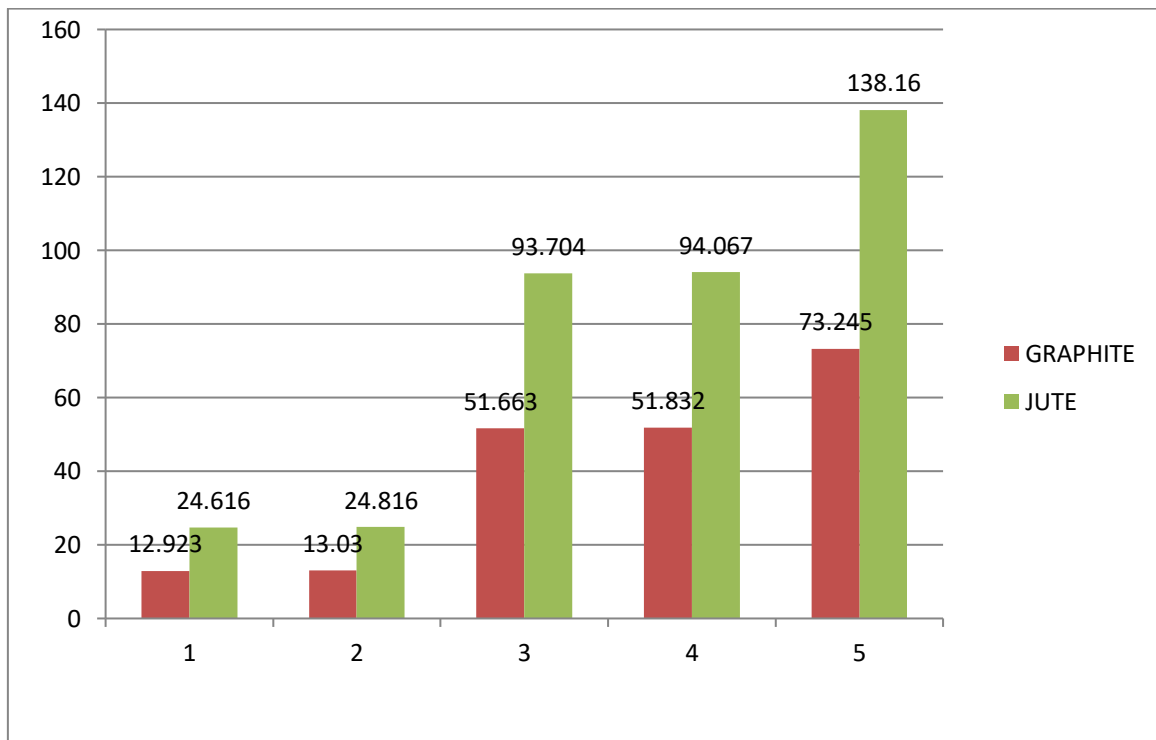
Table 8.2: Results for Structural analysis



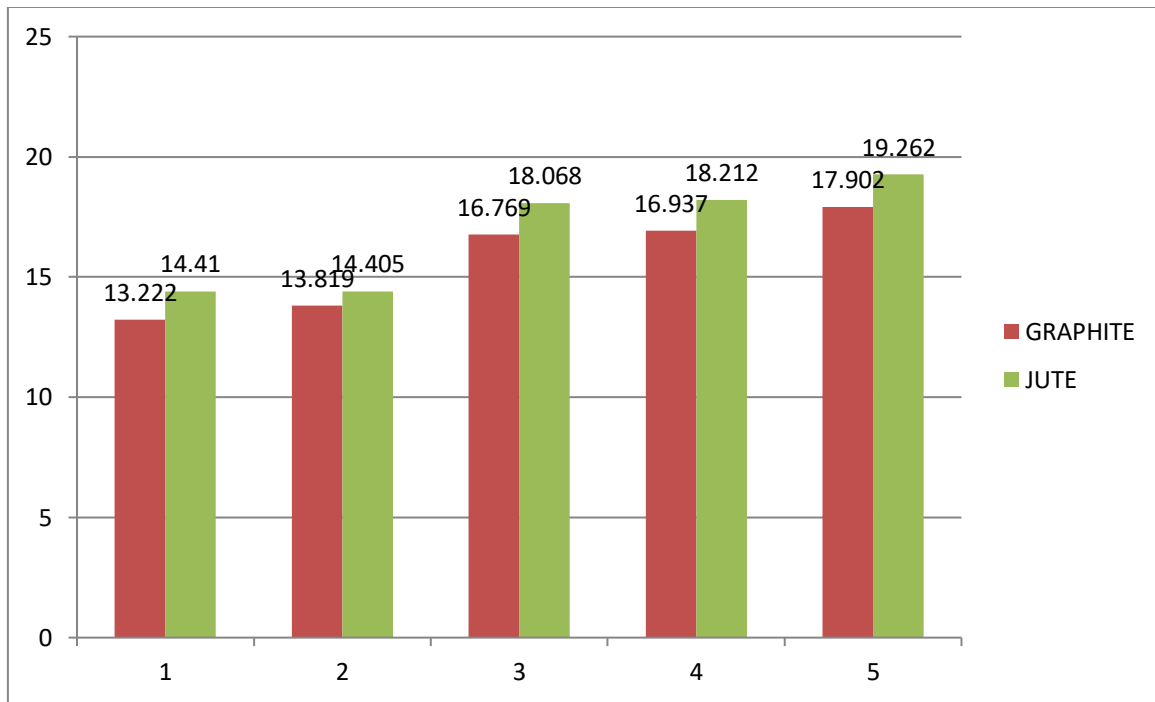
GRAPH-1: FREQUENCY AND DEFORMATION OF GRAPHITE



GRAPH-2: FREQUENCY AND DEFORMATION OF JUTE



GRAPH-3: FREQUENCIES OF GRAPHITE AND JUTE



**GRAPH-4: DEFORMATION OF GRAPHITE AND JUTE**

## 8. CONCLUSION AND FUTURE SCOPE

### 8.1 CONCLUSIONS

- Model analysis is carried out on both graphite and jute propellers it was observed maximum displacement for jute propeller is less than the graphite propeller.
- The natural frequencies of graphite and composite propeller were compared.
- The natural frequencies of graphite propeller were found 2 times more than the jute composite propeller.
- Frequency obtained from FEA analysis graphite=73.245 Hz Frequency obtained from FEA analysis jute Composite=138.16 Hz
- From this project, I have concluded that graphite propeller is the best material when compared to jute composite propeller.

### 8.2 FUTURE SCOPE

- In this project, only the modal analysis on aircraft propeller blade has been performed by the use of the software ANSYS15.0.
- This work can be extended to study the effect of load on the propeller blade under dynamic conditions.
- Experimental stress analysis can be used to compare the different values obtained.
- The study can be extended to dynamic analysis and transient structural analysis of the propeller blade.

## REFERENCES

- [1] ESDU. Propeller slipstream modeling for incidence and sideslip. Item No. 06013, Engineering Sciences Data Unit, London, 2006
- [2] Wald QR. The aerodynamics of propellers. Progress in Aerospace Sciences 42, 2006.
- [3] Kuiper, Wageningen Propeller Series. Marin publication 92-001.
- [4] J.S.Carlton, "Marine Propellers & Propulsion" Butter Worth-Heinemann Limited 1994, Oxford 1994
- [5] Donald L.Blourt and RodertJ.Bartee,"Design of Propulsion System for High Speed Craft", Marine Technology, 1987.

- 
- [6] D. Gay and S.V. Hoa, "Composite materials, Design and application", CRC Press, second edition, ISBN: 978-1-4200-4519-2, (2007).
- [7] Taylor, D.W, "The speed and power of ships", Washington, 1933.
- [8] Gau-Feng Lin "Three Dimensional Stress Analysis of a Fiber Reinforced Composite Thruster Blade", the society of naval architects and marine engineers, 1991.
- [9] Toshio Yamatogi, Kazuro Kageya, Study on Cavitation Erosion of Composite Materials for Marine Propeller. 2010.
- [10] Jonkman JM. Modeling of the UAE Wind Turbine for Refinement of FAST\_AD. NREL Technical Report [NREL/TP-500-34755], 2003.
- [11] Thor I Fossen. Nonlinear Output Feedback Control of Underwater Vehicle Propellers., Mogens Blanke - 2000 - IEEE JOURNAL OF OCEANIC ENGINEERING..
- [12] J.M. Berthelot, "Matériaux composites : Comportement mécanique et analyse des structures", 4ème édition , Paris : Editions Tec & Doc, ISBN: 978-2743007713, (2005).
- [13] H.J.Lin, W.M.L.a.Y.M.K., effect of stacking sequence nonlinear hydro elastic behavior composite propeller Journal of Mechanics, 2010. Vol. 26, No. 3: p. 293-29.