

Design And Analysis Of A Vertical Axis Wind Turbine Blade

Andyala Siva¹, S. AbzalBasha (M.Tech)²

P.G. Scholar, Assistant Professor

Branch: Machine Design

Geethanjali College of Engineering & Technology, Nannur

Email:- ¹siva10020a043@gmail.com, ²shaikabzalbasha777@gmail.com.

ABSTRACT

One of the most important design parameters for cost-effective VAWT is selection of blade material. VAWT blades must be produced at moderate cost for the resulting energy to be competitive in price and the blade should last during the predicted lifetime (usually between 20 and 30 years). At present, Aluminum blades fabricated by extrusion and bending are the most common type of VAWT materials. The major problem with Aluminum alloy for wind turbine application is its poor fatigue properties and its allowable stress levels in dynamic application decrease quite markedly at increasing numbers of cyclic stress applications. Under this backdrop, an attempt has been made in my project to investigate alternative materials as VAWT blade material.

In my project, required properties of the VAWT Blade Materials are first identified. Then available prospective materials are shortlisted and assessed. Subsequently, comparisons are made between the available materials based on their mechanical properties and costs. Finally, comparisons have been made between the design features of a VAWT with Aluminum and the alternative material blades using one of the prospective airfoils. The results of the design analyses demonstrate the superiority of the alternative blade material over conventionally used Aluminum. Structural and modal analyses have been conducted using advanced finite element methods.

INTRODUCTION

1.1 Introduction of Vertical Axis Wind Turbine

Today, the wind energy market is dominated by horizontal axis wind turbines (HAWTS). HAWTS tend to work best in more open settings, offshore or on land in rural areas where the wind is not disturbed by buildings or trees. In contrast, vertical axis wind turbines (VAWTS) are more suited for built-up urban areas. They have lower wind start-up speeds, can be located nearer to the ground making maintenance easier, work in any wind direction and are relatively quiet.

Vertical axis wind turbine (VAWT) is one of the simplest types of turbo machines which are mechanically uncomplicated. As shown in Figure 1, fixed-pitch VAWT has only three major physical components, namely (a) blade; (b) supporting strut; and (c) central column. One of the most important design parameters for cost-effective VAWT is selection of blade material. SBVAWT blades must be produced at moderate cost for the resulting energy to be competitive in price and the blade should last during the predicted lift-time (usually between 20 and 30 years).

Though horizontal axis wind turbines (HAWTs) work well in rural settings with steady uni-directional winds, VAWTs have numerous advantages over them. At present, Aluminium blades fabricated by extrusion and bending are the

most common type of VAWT materials. The major problem with Aluminium alloy for wind turbine application is its poor fatigue properties and its allowable stress levels in dynamic application decrease quite markedly at increasing numbers of cyclic stress applications. Under this backdrop, an attempt has been made in this paper to investigate alternative materials as VAWT blade material.

1.2 Horizontal versus Vertical Axis Wind Turbines

The HAWT is the most common turbine configuration. The propellers and turbine mechanisms are mounted high above the ground on a huge pedestal. It is a matter of taste as to whether they enhance the landscape. However, there is no denying that the height at which their mechanisms are located is a disadvantage when servicing is required. Also, they require a mechanical yaw system to orient them such that their horizontal axis is perpendicular to and facing the wind. As potential power generation is related to the swept area (diameter) of the rotor, more power requires a larger diameter. The blades experience large thrust and torque forces, so size is limited by blade strength. Figure 1.0 shows GE Wind Energy's 3.6 Megawatt HAWT. Larger wind turbines are more efficient and cost effective.



Figure 1.0. GE Wind Energy's 3.6 Megawatt HAWT.

A VAWT does not need to be oriented into the wind and the power transition mechanisms can be mounted at ground level for easy access. Figure 1.1 shows a picture of an H-Darrius Rotor VAWT.



Figure 1.2. An H-Darrius rotor VAWT.

The perceived disadvantage of the VAWT is that they are not self-starting. However, it could be argued that the HAWT is also not self-starting since it requires a yaw mechanism for orientation. Currently, VAWT are usually rotated automatically until they reach the ratio between blade speed and undisturbed wind speed (Tip Speed Ratio or TSR) that produces a torque large enough to do useful work. Through the use of drag devices and/or variable pitch blade designs, it is hoped that a VAWT will be able to reach the required TSR without the use of a starter.

1.5 Required Properties of the Blade Materials

VAWT blades are exposed to diversified load conditions and dynamic stresses are considerably more severe than many mechanical applications. Based on the operational parameters and the surrounding conditions of a typical VAWT for delivering electrical or mechanical energy, the following properties of the VAWT blade materials are required:

- It should have adequately high yield strength for longer life
- It must endure a very large number of fatigue cycles during their service lifetime to reduce material degradation

- It should have high material stiffness to maintain optimal aerodynamic performance
- It should have low density for reduced amount of gravity and normal force component
- It should be corrosion resistant
- It should be suitable for cheaper fabrication methods
- It must be efficiently manufactured into their final form and
- It should provide a long-term mechanical performance per unit cost

Among all these requirements, fatigue is the major problem facing both HAWTs and VAWTs and an operating turbine is exposed to many alternating stress cycles and can easily be exposed to more than 108 cycles during a 30 year life time. The sources of alternating stresses are due to the dynamics of the wind turbine structure itself as well as periodic variations of input forces.

Prospective Materials

The smaller wind turbine blades are usually made of aluminum, or laminated wood. Metals were initially a popular material because they yield a low-cost blade and can be manufactured with a high degree of reliability, however most metallic blades (like steel) proved to be relatively heavy which limits their application in commercial turbines. In the past, laminated wood was also tried on early machines in 1977. At present, the most popular materials for design of different types of wind turbines are aluminum and fiberglass composites that are briefly discussed below:

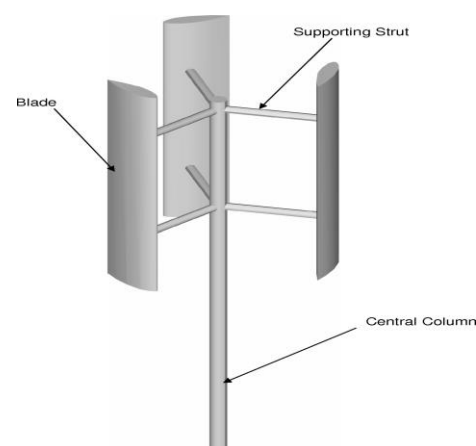
Aluminum:

Aluminum blades fabricated by extrusion and bending are the most common type of VAWT materials. The early blades of Darrieus type VAWTs were made from stretches and formed steel sheets or from helicopter like combinations of aluminum alloy extrusions and fiberglass. It has been reported by Parashivoiu that the former

were difficult to shape into smooth airfoil, while the latter were expensive. The major problem that aluminum alloy for wind turbine application is its poor fatigue properties and its allowable stress levels in dynamic application decreases quite markedly at increasing numbers of cyclic stress applications when compared to other materials such as steel, wood or fiberglass reinforced plastics.

Fibreglass Composites:

Composites constructed with fibreglass reinforcements are currently the blade materials of choice for wind turbine blades of HAWT types. This class of materials is called fibreglass composites or fibre reinforced plastics (FRP). In turbine designs they are usually composed of E-glass in a polyester, vinyl ester or epoxy matrix and blades are typically produced using hand-layup techniques. Recent advances in resin transfer moulding and pultrusion technology have provided the blade manufacturers to examine new procedures for increasing the quality of the final product and reducing manufacturing costs.



The Main Components of a Typical VAWT

1.6 INTRODUCTION TO UNI-GRAPHICS

Overview of Solid Modeling

The Unigraphics NX Modeling application provides a solid modeling system to enable rapid conceptual design. Engineers can incorporate their

requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modeling feature and constraint based solid modeler. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modeling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Blending and Chamfering

- zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modeling Operations

- Profiles can be swept, extruded or revolved to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary
- Fixed and variable radius blends may overlap surrounding faces and extend to a Tapering for modeling manufactured near-net shape parts
- User-defined features for common design elements (Unigraphics NX/User-Defined Features is required to define them in advance)

3.1 General Operation

• Start with a Sketch

Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using Extruded Body or Revolved Body to create a solid or sheet body. You can later refine the sketch to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects.

Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

• Creating and Editing Features

Feature Modeling lets you create features such as holes, slots and grooves on a model. You can then directly edit the dimensions of the feature and locate the feature by dimensions. For example, a Hole is defined by its diameter and length. You can directly edit all of these parameters by entering new values. You can create solid bodies of any desired design that can later be defined as a form feature using User Defined Features. This lets you create your own custom library of form features.

• Associativity

Associativity is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces. See Introduction to Feature Modeling for additional details.

• Positioning a Feature

Within Modeling, you can position a feature relative to the geometry on your model using Positioning Methods, where you position dimensions. The feature is then associated with that geometry and will maintain those associations whenever you edit the model. You can also edit the position of the feature by changing the values of the positioning dimensions.

• Reference Features

You can create reference features, such as Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when needed, or as

construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. You can use a datum plane as a reference plane in constructing sketches, creating features, and positioning features. You can use a datum axis to create datum planes, to place items concentrically, or to create radial patterns.

- **Expressions**

The Expressions tool lets you incorporate your requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, you can define the height of a boss as three times its diameter, so that when the diameter changes, the height changes also.

- **Boolean Operations**

Modeling provides the following Boolean Operations: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

- **Undo**

You can return a design to a previous state any number of times using the Undo function. You do not have to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets you cease worrying about getting it wrong, and frees you to explore more possibilities to get it right.

- **Additional Capabilities**

Other Unigraphics NX applications can operate directly on solid objects created within Modeling without any translation of the solid body. For example, you can perform drafting, engineering analysis, and

NC machining functions by accessing the appropriate application. Using Modeling, you can design a complete, unambiguous, three dimensional model to describe an object.

Parent/Child Relationships

If a feature depends on another object for its existence, it is a child or dependent of that object. The object, in turn, is a parent of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child. A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in your work part, open the Part Navigator.

3.2 Creating A Solid Model

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

3.3 Introduction to Drafting

The Drafting application is designed to allow you to create and maintain a variety of drawings made from models generated from within the Modeling application. Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing. This associatively allows you to make as many model changes as you wish. Besides the powerful associatively functionality, Drafting contains many other useful features.

Updating Models

A model can be updated either automatically or manually. Automatic

updates are performed only on those features affected by an appropriate change (an edit operation or the creation of certain types of features). If you wish, you can delay the automatic update for edit operations by using the Delayed Update option. You can manually trigger an update of the entire model. You might, for example, want to use a net null update to check whether an existing model will successfully update in a new version of Unigraphics NX before you put a lot of additional work into modifying the model. (A net null update mechanism forces a complete update of a model, without changing it.)

The manual methods include:

- The Unigraphics NX Open C and C++ Runtime function, `UF_MODL_update_all_features`, which logs all the features in the current work part to the Unigraphics NX update list, and then performs an update. See the Unigraphics NX Open C and C++ Runtime Reference Help for more information.
- The Playback option on the Edit Feature dialog, which recreates the model, starting at its first feature. You can step through the model as it is created one feature at a time, move forward or backward to any feature, or trigger an update that continues until a failure occurs or the model is complete.

Methods that users have tried in the past that has led to some problems or is tricky to use:

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.

- Another method, attempting to suppress all of the features in a part and then unsuppressed them, can cause updates that are not net null and that will fail. The failures occur because not all features are suppressible; they are left in the model when you try to suppress all features. As the update advances, when it reaches the point where most features were suppressed, it will try to update the features that remain (this is like updating a modified version of the model). Some of the "modifications" may cause the remaining features to fail. For these reasons, we highly recommend that you do not attempt to update models by suppressing all or unsurprising all features. Use the other options described here, instead.

- **Machining of Assemblies**

Assembly parts may be machined using the Manufacturing applications. An assembly can be created containing all of the setup, such as fixtures, necessary to machine a particular part. This approach has several advantages over traditional methods:

- It avoids having to merge the fixture geometry into the part to be machined.
- It lets the NC programmer generate fully associative tool paths for models for which the programmer may not have write access privilege.

It enables multiple NC programmers to develop NC data in separate files simultaneously

PROBLEM DEFINITION AND METHODOLOGY

VAWT blades are exposed to diversified load conditions and dynamic stresses are considerably more severe than many mechanical applications. Based on the operational parameters and the surrounding conditions of a typical VAWT for delivering electrical or mechanical energy, the selection of proper materials for the VAWT blade materials are required. If proper material not choose, it causes to failure.

METHODOLOGY

- Design the vertical axis turbine blade by using Unigraphics software based on 2D dimensions.
- Export .prt file into parasolid (.xt) file and import into Ansys software for analysis.
- Perform static analysis of VATB blade by using Aluminium material to calculate deformation and stresses.
- Perform modal analysis of VATB blade by using Aluminium material to calculate natural frequency.
- Perform static analysis of VATB blade by using E-GLASS/EPOXY material to calculate deformation and stresses.
- Perform modal analysis of VATB blade by using E-GLASS/EPOXY material to calculate natural frequency.
- From analysis results, best material proposed for VATB blade.

3D MODELING OF VERTICAL AXIS WIND TURBINE BLADE

4.1 DESIGN PROCEDURE

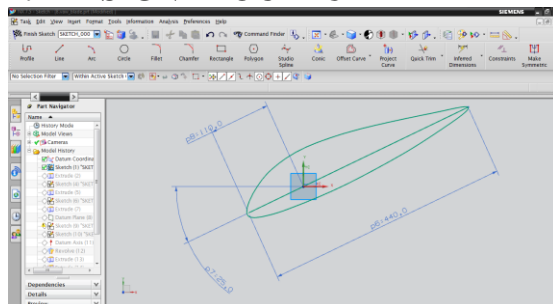


Fig.4.1 2D sketch of blade

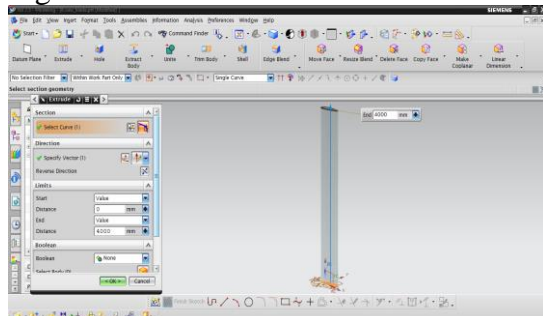


Fig.4.2 Extrude of 2D sketch of blade

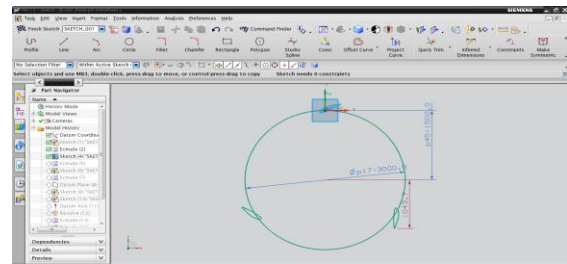


Fig.4.3 2D sketch of blade

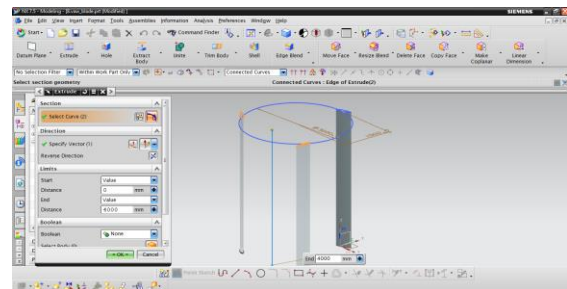


Fig.4.4 Extrude of 2D sketch of blade

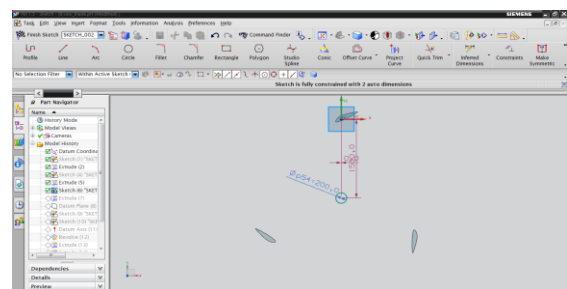


Fig.4.5 2D sketch of blade

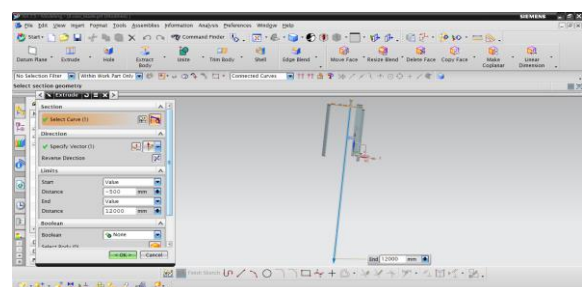


Fig.4.6 Extrude of 2D sketch of blade

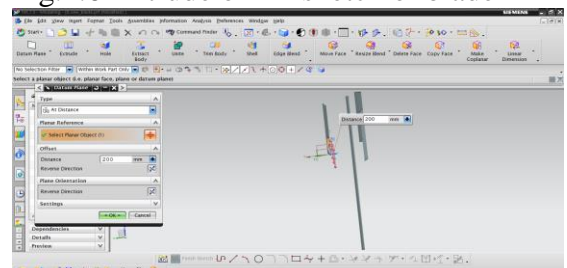


Fig.4.7 Creating datum plane blade

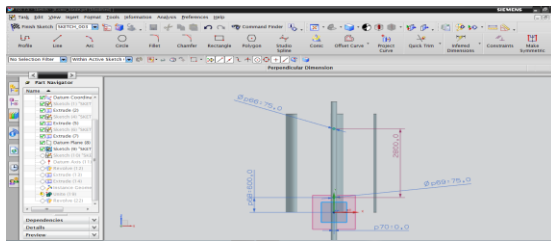


Fig.4.8 2D sketch of blade

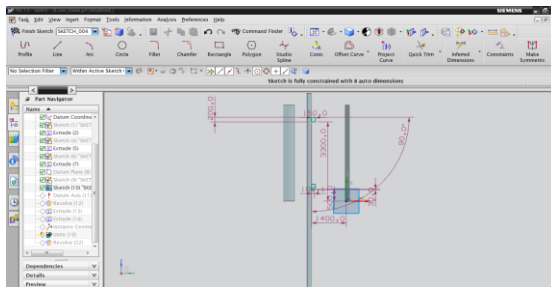


Fig.4.9 2D sketch of blade

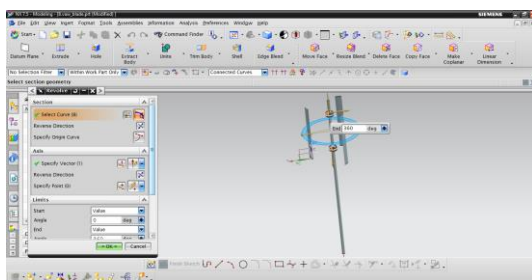


Fig.4.10 Extrude of 2D sketch of blade

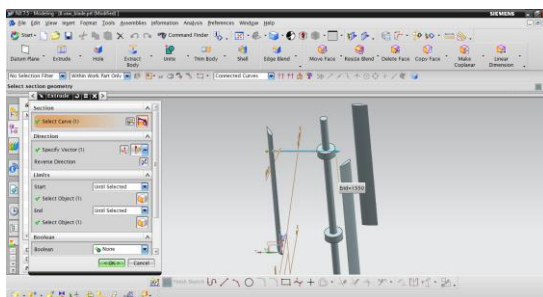


Fig.4.11 Extrude of 2D sketch of blade

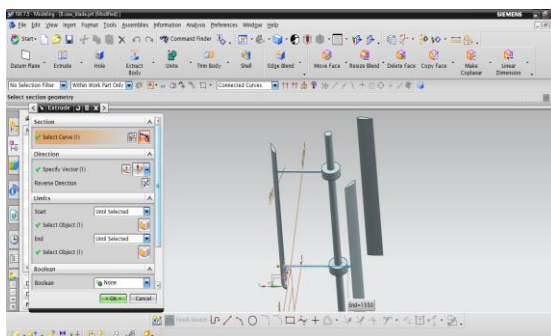


Fig.4.12 Extrude of 2D sketch of blade

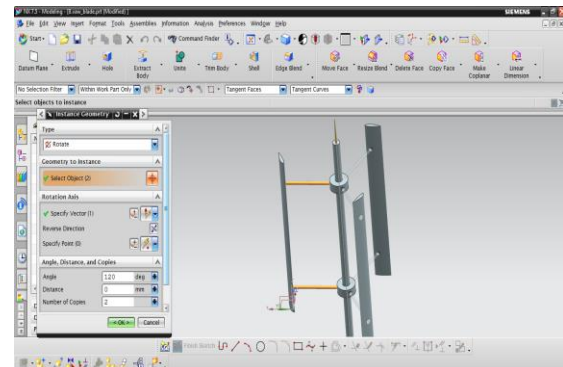


Fig.4.13 Making pattern of blades

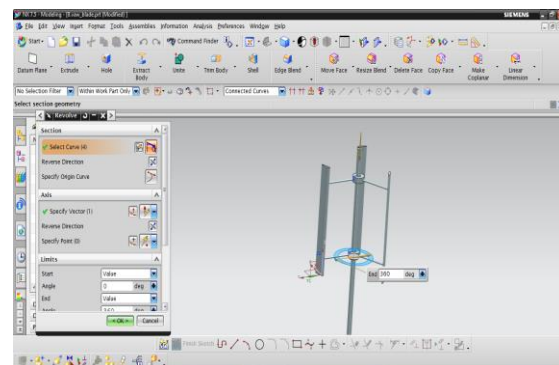


Fig.4.14 Extrude of 2D sketch of blade

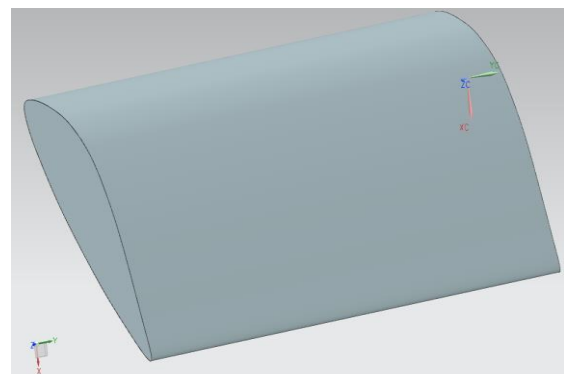


Fig.4.15: 3D model of the vertical axis wind turbine blade

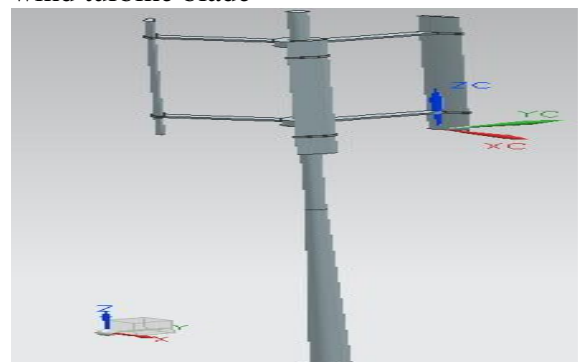


Fig.4.16: 3D model of the vertical axis wind turbine

FINITE ELEMENT METHOD

5.1 INTRODUCTION

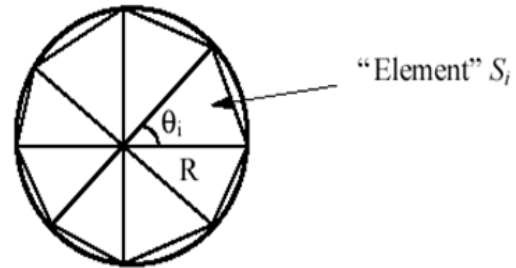
The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called "Finite Elements". The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called "Nodes" or "Nodal Points". Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called "shape functions". This will represent the displacement within the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

The finite element method is a very important tool for those involved in engineering design; it is now used routinely to solve problems in the following areas.

- Structural analysis
- Thermal analysis
- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method for design evaluation. This is because contemporary design problems usually cannot be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in the years gone by, but now it is simply too expensive and time consuming also.



*Basic Concepts: The Finite Element Method is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as engineering. The philosophy of FEA can be explained with a small example such as measuring the area of a circle. Area of one Triangle: $S_i = \frac{1}{2} * R^2 * \sin \theta_i$*

Area of the Circle: $S_N = \frac{1}{2} * R^2 * N * \sin (2 \pi / N) \rightarrow \pi R^2$ as $N \rightarrow \infty$
Where N = total number of triangles (elements)

If one needs to evaluate the area of the circle without using the conventional formula, one of the approaches could be to divide the above area into a number of equal segments. the area of each triangle multiplied by the number of such segments gives the total area of the circle.

4.2 A BRIEF HISTORY OF THE FEM: WHO

The reference credited is to Courant (Mathematician), Turner (air craft industry), Clough (California university), Martin (air craft industry), Argyris (German university),...., However, it was probably established by several pioneers independently.

WHEN

- ✓ Initial idea in mathematical terms was put in 1940s.
- ✓ Application to simple engineering problems in 1950s.
- ✓ Implementation in large computer in 1960s.
- ✓ Development of pre and post processors in 1980s.

- ✓ Analysis of large structural problems in 1990s.

WHERE

Implementation and application were mainly in aircraft industry and automobile sectors (large and fast computers were available only in these industries)

WHAT

Field problems in the form matrix methods of organizing large numbers of algebraic equations are used and matrix equations are solved. Differential equations are transformed into an algebraic form. Blocks with different geometry are hooked together for creating complex geometry of the engineering problem

WHY

The advantage of doing FEM analysis is that it is fairly simple to change the geometry, material and loads recomputed stresses for modified product rather than build and test. The method can be used to solve almost any problem that can be formulated as a field problem. The entire complex problem can be cast as a larger algebraic equation by assembling the element matrices with in the computer and solved.

More about FEA

Finite Element Analysis was first developed for use in the aerospace and nuclear industries where the safety of the structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis. But for a wide range of applications such as steady state and transient temperature distributions, fluid flow simulations and also simulation of manufacturing processes such as injection molding and metal forming.

FEA consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in

new product design, and existing product refinement. A design engineer shall be able to verify the proposed design, which is intended to meet the customer requirements prior to the manufacturing. Things such as, modifying the design of an existing product or structure in order to qualify the product or structure for a new service condition. Can also be accomplished in case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

The Basic Steps Involved in FEA

Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is assumed to be determined by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacements. From this, the equations of equilibrium are assembled in a matrix form which can be easily be programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses and strains can be calculated.

5.3 Basic Steps in FEA

- Discretization of the domain
- Application of Boundary conditions
- Assembling the system equations
- Solution for system equations
- Post processing the results.

Discretization of the domain: The task is to divide the continuum under study into a number of subdivisions called element. Based on the continuum it can be divided into line or area or volume elements.

Application of Boundary conditions: From the physics of the problem we have to apply the field conditions i.e. loads and

constraints, which will help us in solving for the unknowns.

Assembling the system equations: This involves the formulation of respective characteristic (Stiffness in case of structural) equation of matrices and assembly.

Solution for system equations: Solving for the equations to know the unknowns. This is basically the system of matrices which are nothing but a set of simultaneous equations are solved.

Viewing the results: After the completion of the solution we have to review the required results.

The first two steps of the above said process is known as pre-processing stage, third and fourth is the processing stage and final step is known as post-processing stage.

5.4 THEORIES OF FAILURE:

Determining the expected mode of failure is an important first step in analysing a part design. The failure mode will be influenced by the nature of load, the expected response of the material and the geometry and constraints. In an engineering sense, failure may be defined as the occurrence of any event considered to be unacceptable on the basis of part performance. The modes of failure considered here are related to mechanical loads and structural analysis. A failure may include either an unacceptable response to a temporary load involving no permanent damage to the part or an acceptable response which does involve permanent, and sometimes catastrophic, damage. The purpose of theories of failure is to predict what combination of principal stresses will result in failure. There are number of theories to describe failure criteria, of them these are the widely accepted theories.

Maximum principal stress theory (rankine's) σ_1 or σ_2 or σ_3 (which ever is maximum) = σ_y .

According to this theory failure of the material is assumed to have taken place under a state of complex stresses when the value of the maximum principal stress

reaches a value equal to that of the elastic limit stress (yield stress) as found in a simple tensile test.

Maximum shear theory (guest's or coulomb's) $(\sigma_1 - \sigma_2)$ or $(\sigma_2 - \sigma_3)$ or $(\sigma_3 - \sigma_1)$ = σ_y (Which ever is maximum). According to this theory the failure of the material is deemed to have taken place when the maximum shear stress exceeds the maximum shear stress in a simple tension test.

Maximum principal strain theory (St.Venant's)

$\sigma_1 - \nu (\sigma_2 + \sigma_3)$ or $\sigma_2 - \nu (\sigma_3 + \sigma_1)$ or $\sigma_3 - \nu (\sigma_1 + \sigma_2)$ (which ever is maximum) = σ_y . According to this theory, failure of the material is deemed to have taken place when the maximum principal strain reaches a value calculated from a simple tensile test.

Maximum strain energy theory (Beltrami-Haigh's)

$\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - 2\nu (\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)$ = σ_y^2 . According to this theory failure is assumed to take place when the total strain energy exceeds the strain energy determined from a simple tensile test.

Octahedral or distortion energy theory (von mises-hencky)

$\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_3\sigma_1$ = σ_y^2 . According to this theory failure is assumed to take place when the maximum shear strain energy exceeds the shear strain energy in a simple tensile test. This is very much valid for ductile material; in this the energy which is actually responsible for the distortion is taken into consideration.

Soderberg's equations (recommended for ductile materials only):

$$1/n = \sigma_m / \sigma_y + K_f \sigma_a / \sigma - 1$$

$$1/n = \tau_m / \tau_y + K_f \tau_a / \tau - 1$$

Where, σ_m = mean stress

σ_y = yield stress

σ_a = stress amplitude $(\sigma_{\max} - \sigma_{\min}) / 2$

$\sigma - 1$ = endurance limit stress

τ_m = mean shear stress

τ_y = yield shear stress

$1/n$ = factor of safety

Goodman's equations (for brittle materials)

$$1/n = K_t [\sigma_m/\sigma_u + \sigma_a/\sigma - 1]$$

$$1/n = K_t [t_m/t_u + t_a/t - 1]$$

Where, σ_u = ultimate stress

K = stress concentration factor

Choice of the theories of failure:

Well documented experimental results by various authors on the various theories of failure, indicate that the distortion energy theory predicts yielding with greatest accuracy. Compared to this maximum shear stress theory predicts results which are always on safer side. Maximum principal stress theory gives conservative results only if the sign of the two principal stresses is the same (2-D case). Therefore, the use of maximum principal stress theory for pure torsion is ruled out where the sign of the two principal stresses are opposite.

When the fracture of a tension specimen loaded up to rupture is examined, it shows that for ductile materials, failure occurs along lines at angles 45 degrees with the load axis. This indicates a shear failure. Brittle materials on the other hand, rupture on planes normal to the load axis, indicating that maximum normal stress determines failure. Because of the above mentioned observations, it is universally accepted that for a brittle materials, the maximum normal stress theory is the most suitable. For ductile materials, the maximum shear stress theory gives conservative results and it is simpler to use as compared to distortion energy theory, so it is universally accepted as the theory of failure for ductile materials. But, where low weight is desired, the distortion energy theory is recommended.

In brief:

Ductile material

Under combined static loading, the machine parts made of ductile material will fail by yielding. The working or allowable stress is therefore, passed on the yield point stress. The maximum shear stress theory will be used for the design because it is conservative and easy to apply.

Brittle materials

Failure in brittle materials, takes place by fracture. Brittle materials do not have a distinct yield point and so, the ultimate strength is used as the basis for determining the allowable or design stress. Separate design equations should be used in tension and compression, since for materials like cast iron; the ultimate compressive strength is considerably greater than the ultimate tensile strength. The maximum principal stress theory will be used for the design. Due consideration will be given to the sign of principal stresses. If both the principal stresses (2-D case) are of the same sign, the effect of the smaller stress is neglected. If the two principal stresses are of opposite sign, then the maximum principal stress theory does not give conservative results. In that case another equation should be used.

6.1 ORGANIZATION OF THE ANSYS PROGRAM

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The begin level acts as a gateway in to and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When we first enter the program, we are at the begin level. At the processor level, several processors are available; each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor (PREP7) is where we build the model, the solution processor (SOLUTION) is where we apply loads and obtain the solution, and the general postprocessor (POST1) is where we evaluate the results and obtain the solution. An additional postprocessor (POST26), enables we to evaluate solution results at specific points in the model as a function of time.

6.2 PERFORMING A TYPICAL ANSYS ANALYSIS

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

- Build the model
- Apply loads and obtain the solution
- Review the results

The following table shows the brief description of steps followed in each phase.

PRE-PROCESSOR	SOLUTION PROCESSOR	POST-PROCESSOR
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	Load definition	View animated results
Material definition	Solve	
Mesh generation		
Model display		

6.3 PRE-PROCESSOR:

The input data for an ANSYS analysis are prepared using a preprocessor. The general preprocessor (PREP 7) contains powerful solid modeling and mesh generation capabilities, and is also used to define all other analysis data with the benefit of database definition and manipulation of analysis data. Parametric input, user files, macros and extensive online documentation are also available, providing more tools and flexibility.

6.4 SOLUTION PROCESSOR

Here we create the environment to the model, i.e., applying constraints & loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Here three major steps involved:

- Solution type required, i.e. static, modal, or transient etc., is selected

- Defining loads. The loads may be point loads, surface loads; thermal loads like temperature, or fluid pressure, velocity are applied.
- Solve FE solver can be logically divided into three main steps, the pre-solver, the mathematical-engine and post-solver. The pre-solver reads the model created by pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the result. The result returns to the solver and the post solver is used to calculate strains, stresses, etc., for each node within the component or continuum.

6.6 REVIEW THE RESULTS:

Once the solution has been calculated, we can use the ANSYS postprocessor to review the results. Two postprocessors are available: POST1 and POST 26. We use POST 1, the general postprocessor to review the results at one sub step over the entire model or selected portion of the model. We can obtain contour displays, deform shapes and tabular listings to review and interpret the results of the analysis. POST 1 offers many other capabilities, including error estimation, load case combination, calculation among results data and path operations.

We use POST 26, the time history post processor, to review results at specific points in the model over all time steps. We can obtain graph plots of results, data vs. time and tabular listings. Other POST 26 capabilities include arithmetic calculations and complex algebra.

In the solution of the analysis the computer takes over and solves the simultaneous set of equations that the finite element method generates, the results of the solution are

- Nodal degree of freedom values, which form the primary solution

- Derived values which form the element solution

6.7 MESHING:

Before meshing the model and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it.

Compare to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. If we want this type of mesh, we must build the geometry as series of fairly regular volumes and/or areas that can accept a mapped mesh.

FREE MESHING:

In free meshing operation, no special requirements restrict the solid model. Any model geometry even if it is regular, can be meshed. The elements shapes used will depend on whether we are meshing areas or volumes. For area meshing, a free mesh can consist of only quadrilateral elements, only triangular elements, or a mixture of the two. For volume meshing, a free mesh is usually restricted to tetrahedral elements. Pyramid shaped elements may also be introduced in to the tetrahedral mesh for transitioning purposes.

MAPPED MESHING

We can specify the program use all quadrilateral area elements, all triangular area elements or all hexahedra brick volume elements to generate a mapped mesh. Mapped meshing requires that an area or volume be "regular", i.e., it must meet certain criteria. Mapped meshing is not supported when hard points are used. An area mapped mesh consists of either all quadrilateral elements or all triangular elements

For an area to accept a mapped mesh the following conditions must be satisfied:

- The area must be bounded by either three or four lines
- The area must have equal numbers of element divisions specified on opposite sides, or have divisions matching one transition mesh patterns.
- If the area is bounded by three lines, the number of element divisions must be even and equal on all sides
- The machine key must be set to mapped. This setting result in a mapped mesh of either all quadrilateral elements or all triangular elements depending on the current element type and shape key.
- Area mapped meshes shows a basic area mapped mesh of all quadrilateral elements and a basic area mapped mesh of all triangular elements.

6.8 STRUCTURAL STATIC ANALYSIS:

A static analysis calculates the effects of study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed, i.e. the loads and the structure's responses are assumed to vary slowly with respect to time.

ANALYSES OF A VERTICAL AXIS WIND TURBINE

7.1 STATIC ANALYSIS OF VERTICAL AXIS WIND TURBINE USING ALUMINIUM

Element type: Solid187

No. of nodes: 8

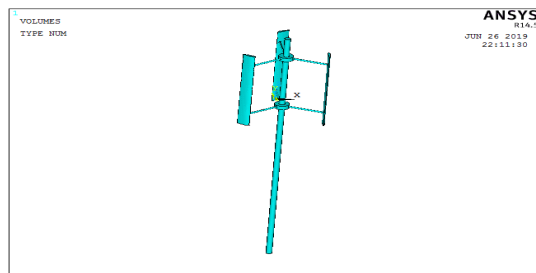
Degrees of freedom: 3 (UX, UY, UZ)

ALUMINIUM:

Mechanical properties:

Young's modulus, E (Gpa) : 71

Poisson's ratio, ν : 0.35
Density (g/cm^3) : 2.65



Imported model

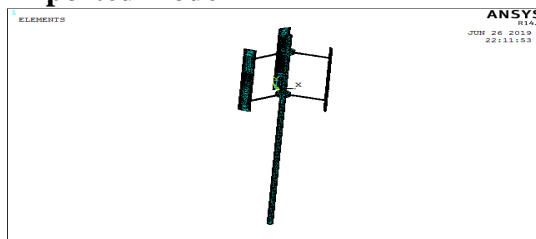


Figure: Finite element model

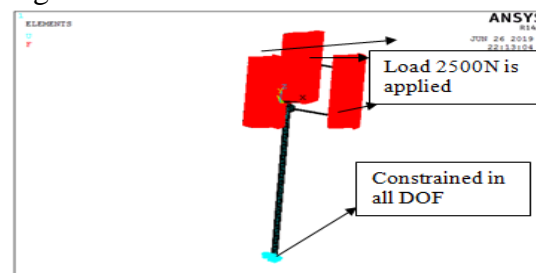
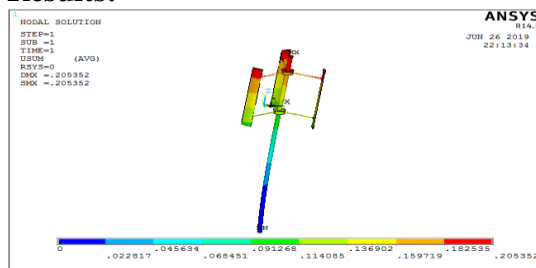
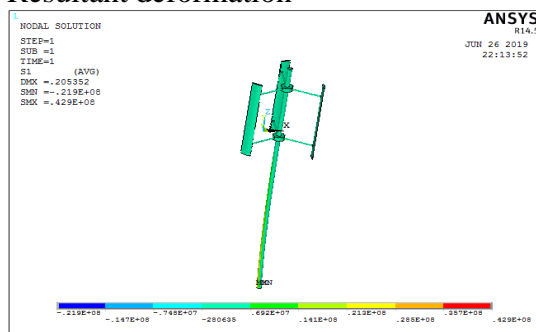


Figure: Loading conditions

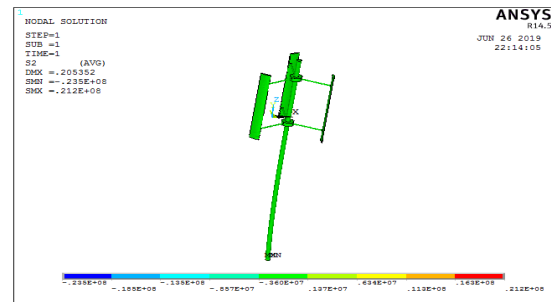
Results:



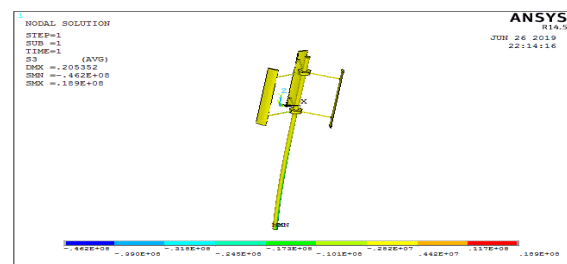
Resultant deformation



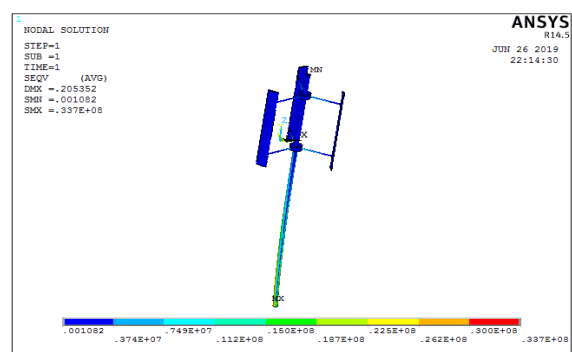
1st Principle stress



2nd Principle stress



3rd Principle stress



Vonmises stress

From above static analysis results, Vonmises stress formed on VAWT is 33.7MPa. Yield strength of Aluminium is 180MPa.

Factor of safety = yield strength/Vonmises stress = $180/33.7 = 5.34$

7.2MODAL ANALYSIS OF VERTICAL AXIS WIND TURBINE USING ALUMINIUM:

Methodology:

- Develop a 3D model.
- The 3D model is created using UNIGRAPHICS-NX software.
- The 3D model is converted into parasolid and imported into ANSYS to do modal analysis.
- Calculate natural frequencies and plot their mode shapes.

Natural Frequency:

Natural frequency is the frequency at which a system naturally vibrates once it has been set into motion. In other words, natural frequency is the number of times a system will oscillate (move back and forth) between its original position and its displaced position, if there is no outside interference.

The natural frequency is calculated from the formula given below. The natural frequencies depend on stiffness of the geometry and mass of the material.

Fundamental Natural Frequency

The fundamental frequency, often referred to simply as the fundamental, is defined as the lowest frequency of a periodic waveform. In terms of a superposition of sinusoids (e.g. Fourier series), the fundamental frequency is the lowest frequency sinusoidal in the sum.

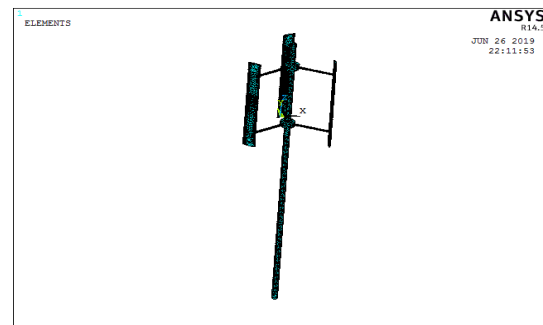
Resonance:

In physics, resonance is the tendency of a system to oscillate with greater amplitude at some frequencies than at others. Frequencies at which the response amplitude is a relative maximum are known as the system's resonant frequencies, or resonance frequencies. At these frequencies, even small periodic driving forces can produce large amplitude oscillations, because the system stores vibration energy.

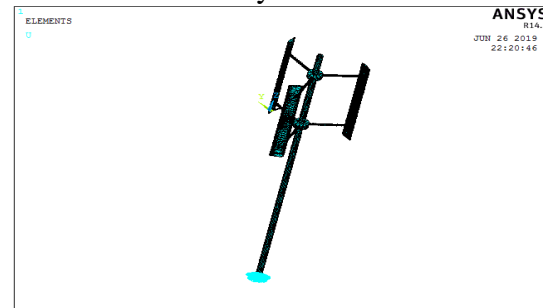
Mode Shapes:

Modal Analysis:

Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.



Mesh model in ansys



Applied fixed loads at end of VAWT

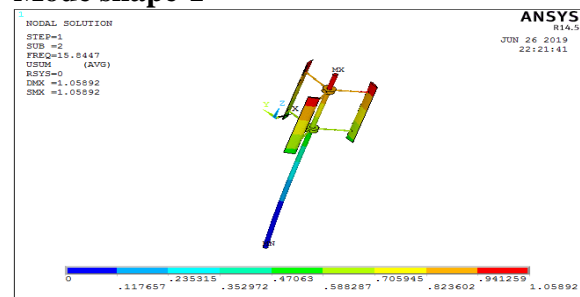
Results

SETLIST Command

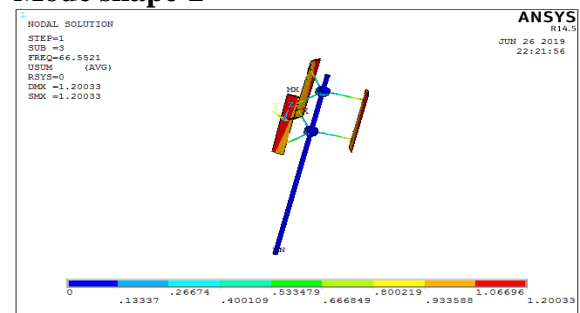
***** INDEX OF DATA SETS ON RESULTS FILE *****				
SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	15.843	1	1	1
2	15.845	1	2	2
3	66.552	1	3	3
4	125.32	1	4	4
5	125.70	1	5	5

Five natural frequencies

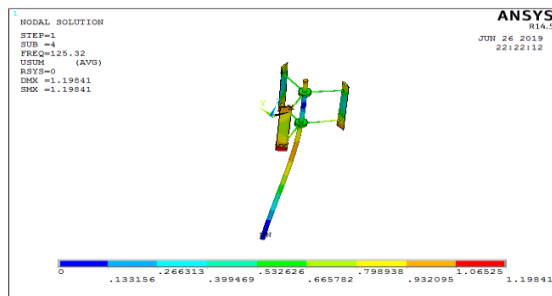
Mode shape-1



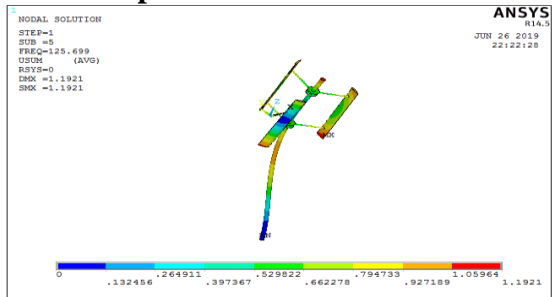
Mode shape-2



Mode shape-3



Mode shape-4



Mode shape-5

7.3 STATIC ANALYSIS OF VERTICAL AXIS WIND TURBINE USING E-GLASS/EPOXY

Element type: Solid185

No. of nodes: 20

Degrees of freedom: 3 (UX, UY, UZ)

EGLASS/EPOXY

Mechanical Properties:

Young's modulus in fiber direction E1 (GPa) : 53.8

Young's modulus in transverse direction, E2 (GPa) : 17.9

Shear modulus, G12 (GPa) : 8.96

Major Poisson's ratio, ν_{12} : 0.25

Minor Poisson's ratio, ν_{21} : 0.08

Strength in the fiber direction, XL (MPa) : 1.03 X 10³

Strength in the transverse direction, XT (MPa) : 27.58

Shear strength, S (MPa) : 41.37

Density (g/cm³) : 2.60

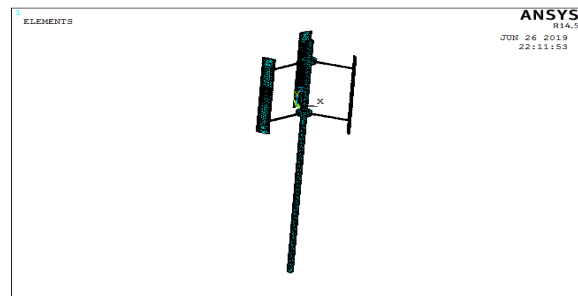
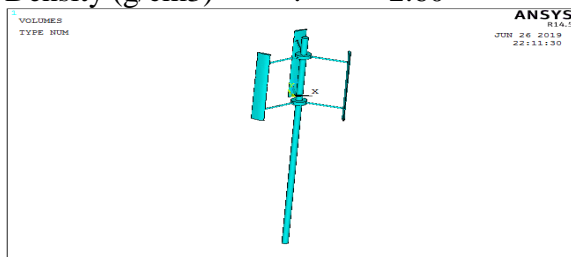


Figure: Finite element model

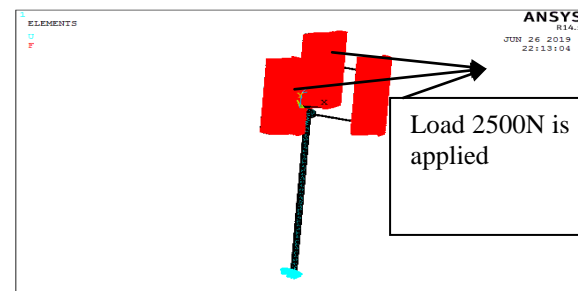
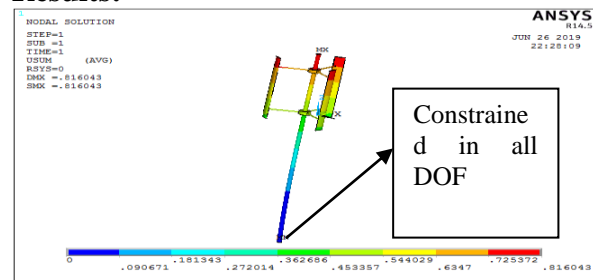
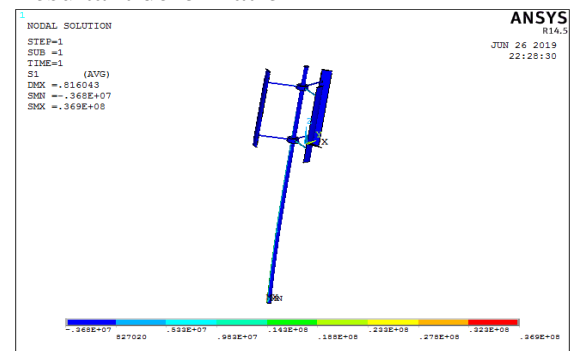


Figure: Loading conditions

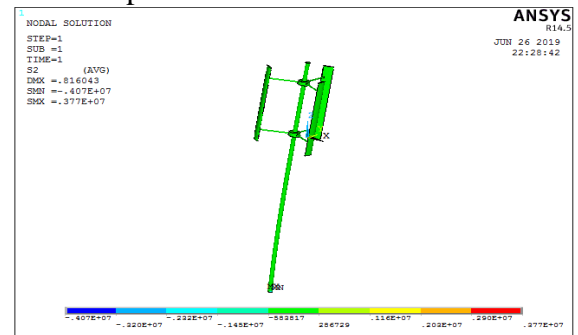
Results:



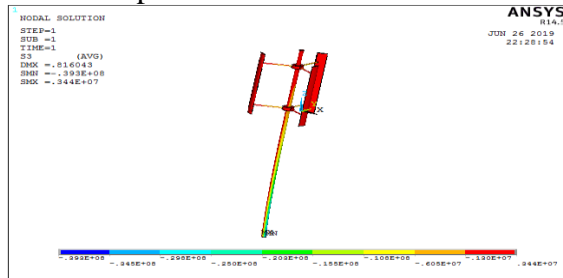
Resultant deformation



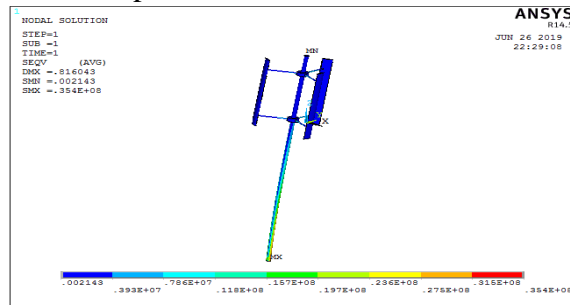
1st Principle stress



2nd Principle stress



3rd Principle stress



Vonmises stress

From above static analysis results, Vonmises stress formed on VAWT is 35.4MPa. Yield strength of E-Glass/Epoxy is 800MPa.

Factor of safety = yield strength/Vonmises stress = $800/35.4 = 22.6$

7.4MODAL ANALYSIS OF VERTICAL AXIS WIND TURBINE USING E-GLASS/EPOXY:

Methodology:

- Develop a 3D model.
- The 3D model is created using UNIGRAPHICS-NX software.
- The 3D model is converted into parasolid and imported into ANSYS to do modal analysis.
- Calculate natural frequencies and plot their mode shapes.

Natural Frequency:

Natural frequency is the frequency at which a system naturally vibrates once it has been set into motion. In other words, natural frequency is the number of times a system will oscillate (move back and forth) between its original position and its displaced position, if there is no outside interference.

The natural frequency is calculated from the formula given below. The natural

frequencies depend on stiffness of the geometry and mass of the material.

Fundamental Natural Frequency

The fundamental frequency, often referred to simply as the fundamental, is defined as the lowest frequency of a periodic waveform. In terms of a superposition of sinusoids (e.g. Fourier series), the fundamental frequency is the lowest frequency sinusoidal in the sum.

Resonance:

In physics, resonance is the tendency of a system to oscillate with greater amplitude at some frequencies than at others. Frequencies at which the response amplitude is a relative maximum are known as the system's resonant frequencies, or resonance frequencies. At these frequencies, even small periodic driving forces can produce large amplitude oscillations, because the system stores vibration energy.

Resonance occurs when a system is able to store and easily transfer energy between two or more different storage modes (such as kinetic energy and potential energy in the case of a pendulum). However, there are some losses from cycle to cycle, called damping. When damping is small, the resonant frequency is approximately equal to the natural frequency of the system, which is a frequency of unforced vibrations. Some systems have multiple, distinct, resonant frequencies.

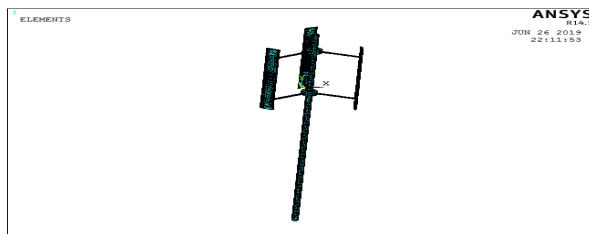
Mode Shapes:

For every natural frequency there is a corresponding vibration mode shape. Most mode shapes can generally be described as being an axial mode, torsional mode, bending mode, or general modes. A crude mesh will give accurate frequency values, but not accurate stress values.

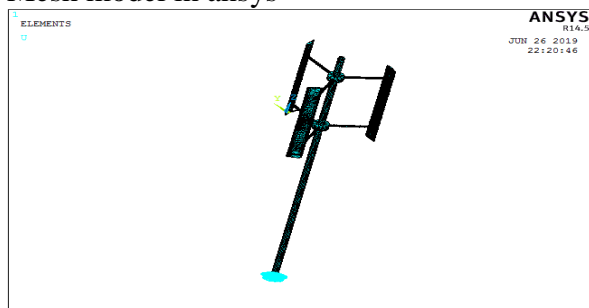
Modal Analysis:

Modal analysis is used to determine the vibration characteristics (natural

frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.



Mesh model in ansys



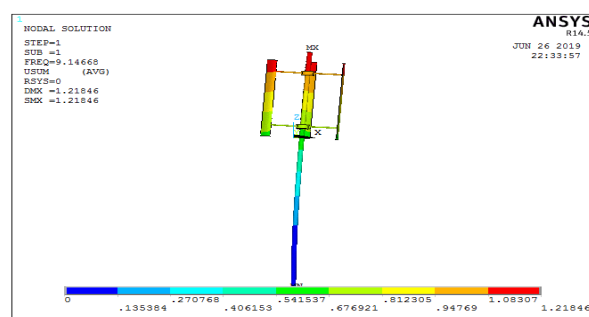
Applied fixed loads at end of VAWT

Results

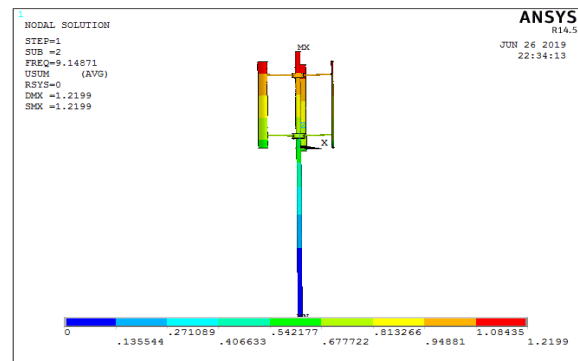
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	9.1467	1	1	1
2	9.1487	1	2	2
3	44.473	1	3	3
4	72.966	1	4	4
5	73.585	1	5	5

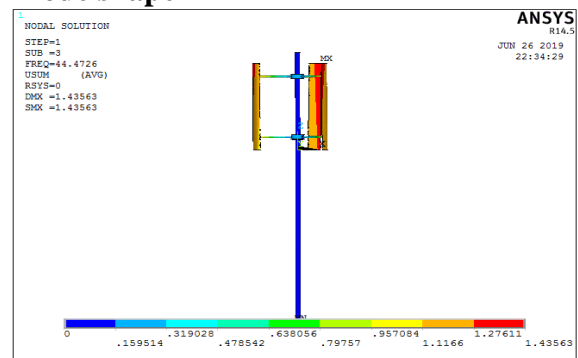
Five natural frequencies



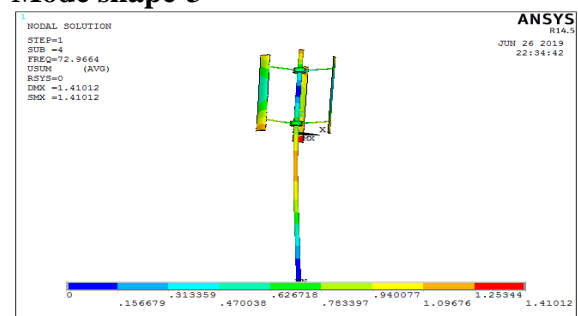
Mode shape-1



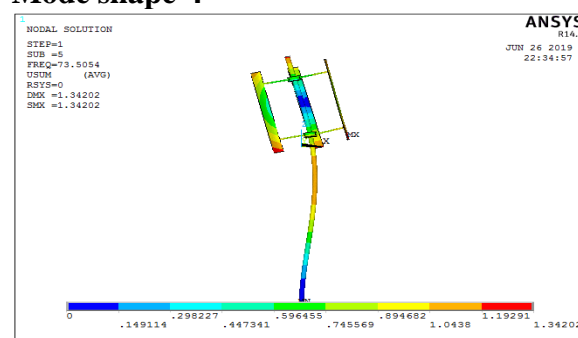
Mode shape-2



Mode shape-3



Mode shape-4



Mode shape-5

RESULTS AND CONCLUSION

	USUM	VON MISES STRESS	NATURAL FREQUENCY RANGE(Hz)	WEIGHT (kg)
ALUMINUM	0.63	5.32	45.65 - 80	284
E-GLASS/EPOXY	2.42	5.28	28.95	179

CONCLUSION:

Vertical axis wind turbine blade has been analyzed and compared at wind speed (2500N approximately), for the different materials. By comparing the results, stresses developed in the blade are almost same in both materials, and these stresses are within the yield strength. E-glass/epoxy blade has very less weight comparing with Aluminum. So, we can conclude that E-glass/epoxy is the best material for the wind turbine blade.

REFERENCES

1. U.S. Department of Energy. "Wind and Hydropower Technologies Program", November, 2005.
2. Chang, Professor L..(2005) "Advanced Topics in Environmental Engineering - Wind Power," University of New Brunswick, 2005.
3. EarthLink (2005). "See How it Flies – A new spin on the perceptions, procedures, and principles of flight.", 2005.
4. Kirke, Brian Kinloch, 1998. "Evaluation of Self-Starting Vertical Axis Wind Turbines for Stand-Alone Applications". Griffith University, Australia, 2005.
5. Sheldahl, Robert E., Klimas, Paul C., 1981. "Aerodynamic Characteristics of Seven Symmetrical Airfoil Sections Through 180-Degree Angle of Attack for Use in Aerodynamic Analysis of Vertical Axis Wind Turbines", Sandia National Laboratories, Albuquerque, NM., USA.
6. Reuss, R.L., Hoffmann, M.J., Gregorek, G.M., December 1995. 'Effects of Surface Roughness and Vortex Generators on the NACA 4415 Airfoil, The Ohio 48 State University, Columbus, Ohio, USA, 2005.
7. Pawsey, N.C.K., 2002. "Development and Evaluation of Passive Variable-Pitch Vertical Axis Wind Turbines", School of

Mechanical and Manufacturing Engineering, The University of South Wales, Australia.

8. Gipe, Paul, 2004. "Wind Power," Chelsea Green Publishing Company, Page 97
9. Johnson, Dr. Gary L. (November 21, 2001) "Wind Turbine Power – Ch 4. Wind Turbine Power, Energy and Torque.", 2005.